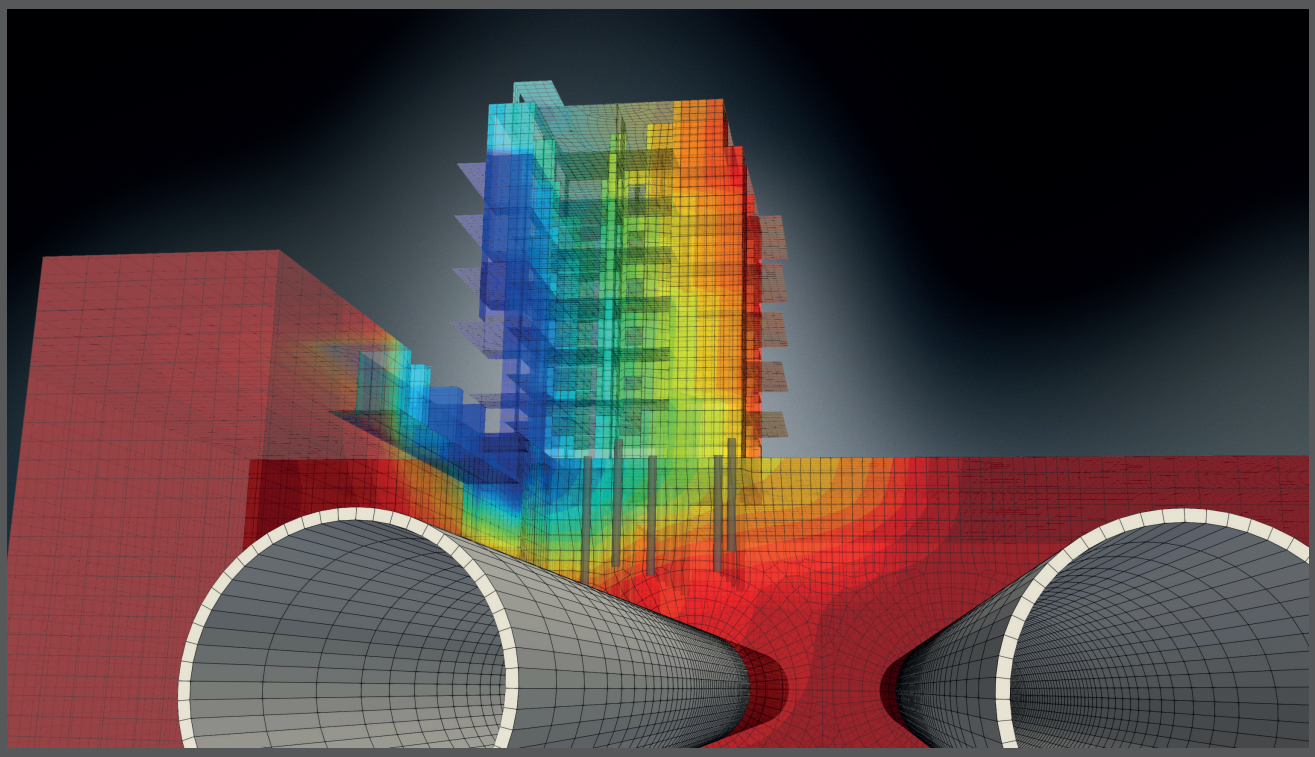
ZSoil

for geotechnics & structures

# User manual
# DATA PREPARATION



## Soil, rock and structural mechanics in dry or partially saturated media

© 1985 - 2022
ZACE Services Ltd

© 2022 - 2026
GeoDev. Sàrl

zsoil.com

ZSoil

for geotechnics & structures

# DATA PREPARATION

## User manual

ZSoil®2026

**A. Truty Th. Zimmermann K. Podleś R. Obrzud**
**with contribution by A. Urbański and S. Commend**

GeoDev.
PO Box CH-1001 Lausanne
Switzerland
https://zsoil.com

## WARNING

ZSoil is regularly updated for minor changes. We recommend that you send us your e-mail, as ZSoil owner, so that we can inform you of latest changes. Otherwise, consult our site regularly and download free upgrades to your version.

Latest updates to the manual are always included in the online help, so that slight differences with your printed manual will appear with time; always refer to the online manual for latest version, in case of doubt.

ZSoil 2026 manual:
1. Data preparation
2. Tutorials and benchmarks
3. Theory

©1985-2022 by ZACE Services Ltd
©2022- - by GeoDev Sarl. All rights reserved.

# END-USER LICENSE AGREEMENT FOR GeoDev's ZSoil® SOFTWARE

## Applicable to all V2026 versions: professional & academic, single user & networks, under Windows 10, 11.

---

**Read carefully this document, it is a binding agreement between you and GeoDev Sarl (GeoDev) for the software product identified above. By installing, copying, or otherwise using the software product identified above, you agree to be bound by the terms of this agreement. If you do not agree to the terms of this agreement, promptly return the unused software product to the place from which you obtained it for full refund of price paid. GEODEV SARL OFFERS A 60 DAYS MONEY-BACK GUARANTEE ON ZSOIL.**

---

### ZSOIL (the Software & associated hotline services when applicable) SOFTWARE PRODUCT LICENSE:

**ZSOIL** Software is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The **ZSOIL** software product is licensed, not sold.

1. GRANT OF LICENSE

   A: **GeoDev Sarl** grants you, the customer, a non-exclusive license to use $\mathbf{N}_{\text{bought}}$ (= the number of licenses bought) copies of **ZSOIL**. You may install copies of **ZSOIL** on an unlimited number of computers, provided that you use only $\mathbf{N}_{\text{bought}}$ copies at the time.

   B: You may make an unlimited number of copies of documents accompanying **ZSOIL**, provided that such copies shall be used only for internal purposes and are not republished or distributed to any third party.

   C: Duration of the agreement may be limited or unlimited, depending on license purchased. Installation of time unlimited licenses of ZSOIL V2026 will be supported for a period of 3 years starting from date of purchase. This support is limited to ZSOIL V2026 upgrades, under Windows 10 and 11.

2. COPYRIGHT
   All title and copyrights in and to **the Software** product (including but not limited to images, photographs, text, applets, etc.), the accompanying materials, and any copies of **ZSOIL** are owned by **GeoDev Sarl**. **ZSOIL** is protected by copyright laws and international treaties provisions. Therefore, you must treat **ZSOIL** like any other copyrighted material except that you may make copies of the software for backup or archival purposes or install the software as stipulated under section 1 above.

3. OTHER RIGHTS AND LIMITATIONS

   A: Limitations on Reverse Engineering, Decompilation, Disassembly. You may not reverse engineer, decompile, or disassemble **the Software**.

   B: No separation of components. **ZSOIL** is licensed as a single product and neither **the Software's** components, nor any upgrade may be separated for use by more than $\mathbf{N}_{\text{bought}}$ user(s) at the time.

   C: Rental. You may not lend, rent or lease the software product.

   D: Software transfer. You may permanently transfer all of your rights under this agreement and within the territory (country of purchase and delivery), provided you do not retain any copies, and the recipient agrees to all the terms of this agreement.

   E: Termination. Without prejudice to any other rights, **GeoDev Sarl** may terminate this agreement if you fail to comply with the conditions of this agreement. In such event, you must destroy all copies of **the Software**.

---

## WARRANTIES & LIMITATIONS TO WARRANTIES

1. DISCLAIMER
   **ZSOIL**, developed by **GeoDev Sarl** is a finite element program for the analysis of above- and underground structures in which soil/rock & structural models are used to simulate the soil, rock and/or structural behaviour. The **ZSOIL** code and its soil/rock & structural models have been developed with great care. Although systematic testing and validation have been performed, it cannot be guaranteed that the **ZSOIL** code is free of errors. Moreover, the simulation of geotechnical and/or structural problems by means of the finite element method implicitly involves some inevitable numerical and modelling errors. **ZSOIL** is a tool intended to be used by trained professionals only and is not a substitute for the user's professional judgment or independent testing. The accuracy at which reality is approximated depends highly on the expertise of the user regarding the modelling of the problem, the understanding of the soil and structural models and their limitations, the selection of model parameters, and the ability to judge the reliability of the computational results. Hence, **ZSOIL** may only be used by professionals that possess the aforementioned expertise. The user must be aware of his/her responsibility when he/she uses the computational results for geotechnical design purposes. **GeoDev Sarl** cannot be held responsible or liable for design errors that are based on the output of **ZSOIL** calculations. The user is solely responsible for establishing the adequacy of independent procedures for testing the reliability, accuracy and completeness of any output of **ZSOIL** calculations.

2. LIMITED WARRANTY
   **GeoDev Sarl** warrants that **ZSOIL** will a) perform substantially in accordance with the accompanying written material for a period of 90 days from the date of receipt, and b) any hardware accompanying the product will be free from defects in materials and workmanship under normal use and service for a period of one year, from the date of receipt.

3. CUSTOMER REMEDIES
   **GeoDev Sarl** entire liability and your exclusive remedy shall be at **GeoDev's** option, either a) return of the price paid, or b) repair or replacement of the software or hardware component which does not meet **GeoDev**'s limited warranty, and which is returned to **GeoDev Sarl**, with a copy of proof of payment. This limited warranty is void if failure of **the Software** or hardware component has resulted from accident, abuse, or misapplication. Any replacement of software or hardware will be warranted for the remainder of the original warranty period or 30 days, whichever is longer.

---

## NO OTHER WARRANTIES.

YOU ACKNOWLEDGE AND AGREE THAT **ZSOIL** IS PROVIDED ON AN "AS IS" AND "AS AVAILABLE" BASIS AND THAT YOUR USE OF OR RELIANCE UPON **ZSOIL** AND ANY THIRD PARTY CONTENT AND SERVICES ACCESSED THEREBY IS AT YOUR SOLE RISK AND DISCRETION. **GeoDev Sarl** AND ITS AFFILIATES, PARTNERS, SUPPLIERS AND LICENSORS HEREBY DISCLAIM ANY AND ALL REPRESENTATIONS, WARRANTIES AND GUARANTIES REGARDING **ZSOIL** AND THIRD PARTY CONTENT AND SERVICES, WHETHER EXPRESS,IMPLIED OR STATUTORY. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, **GeoDev Sarl** DISCLAIMS ALL OTHER WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE, WITH REGARD TO THE SOFTWARE PRODUCT, AND ANY ACCOMPANYING HARDWARE.

## NO LIABILITY FOR CONSEQUENTIAL DAMAGES.

TO THE MAXIMUM EXTENT PERMITTED BY LAW, IN NO EVENT SHALL **GeoDev Sarl** HAVE ANY LIABILITY (DIRECTLY OR INDIRECTLY) FOR ANY SPECIAL INCIDENTAL, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, WITHOUT LIMITATION, DAMAGES FOR LOSS OF BUSINESS, PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, OR ANY OTHER PECUNIARY LOSS) ARISING OUT OF THE USE OF OR INABILITY TO USE ZSOIL, EVEN IF **GeoDev Sarl** HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

## OTHER PROVISIONS.

**SUPPORT**: If included in license price, assistance will be provided by **GeoDev Sarl**, by e-mail exclusively, during the first year following purchase. This service excludes all forms of consulting on actual projects. Installation support is limited to initially supported OS and a four year duration.
**PROFESSIONAL VERSIONS** of **ZSOIL** are meant to be used in practice & in research centers.
**ACADEMIC VERSIONS** of **ZSOIL** are meant to be used exclusively for teaching and research in academic institutions.
**ACADEMIC WITH CONSULTING VERSIONS** of **ZSOIL** are meant to be used exclusively for teaching, research and consulting in academic institutions.
The terms of this agreement may be amended in the future, by **GeoDev Sarl**, when necessary. In such cases the revised agreement will be resubmitted for user approval on the software's front screen.

## APPLICABLE LAW AND JURISDICTION THIS AGREEMENT IS GOVERNED BY THE (SUBSTANTIVE) LAWS OF SWITZERLAND, ALL DISPUTES ARISING OUT OF OR IN CONNECTION WITH THIS AGREEMENT OR THE USE OF ZSOIL SHALL EXCLUSIVELY BE SETTLED BY THE ORDINARY COURTS OF CANTON DE VAUD (ARRONDISSEMENT DE LAUSANNE)

**LAUSANNE 15.11.2025**

**©2022- GeoDev Sarl, Lausanne, Switzerland**

# Contents of Data Preparation


**System requirements** 9

**Getting started** 11

**PREFACE** 13

**1 INTRODUCTION** 15

1.1 SIGN CONVENTION . . . 16

1.2 DEFINITIONS . . . 17

1.2.1 UNIT WEIGHTS . . . 18

1.2.2 ELASTICITY CONSTANTS . . . 19

1.3 UNITS TABLE . . . 20

**2 DATA PREPARATION AND POSTPROCESSING DESCRIPTION** 21

2.1 FILES . . . 23

2.1.1 PARAMETRIC ANALYSIS . . . 24

2.2 CONTROL . . . 26

2.2.1 PROJECT PRESELECTION . . . 27

2.2.2 ANALYSIS & DRIVERS . . . 28

2.2.2.1 PROBLEM TYPE AND DRIVERS . . . 31

2.2.2.1.1 SINGLE PHASE (DEFORMATION) ANALYSIS . 33

2.2.2.1.1.1 INITIAL STATE . . . 36

2.2.2.1.1.2 TIME DEPENDENT . . . 40

2.2.2.1.1.3 STABILITY . . . 45

2.2.2.1.2 DEFORMATION COUPLED WITH FLOW . . . 49

2.2.2.1.2.1 INITIAL STATE . . . 52

2.2.2.1.2.2 TIME DEPENDENT . . . 54

2.2.2.1.2.3 STABILITY . . . 58

2.2.2.1.3 FLOW (STEADY STATE AND TRANSIENT) . . 62

2.2.2.1.3.1 INITIAL STATE . . . 65

2.2.2.1.3.2 TIME DEPENDENT . . . . . 66
2.2.2.1.4 HEAT TRANSFER . . . . . 68
2.2.2.1.4.1 INITIAL STATE . . . . . 71
2.2.2.1.4.2 TIME DEPENDENT . . . . . 72
2.2.2.1.5 HUMIDITY TRANSFER . . . . . 74
2.2.2.1.5.1 INITIAL STATE . . . . . 76
2.2.2.1.5.2 TIME DEPENDENT . . . . . 77
2.2.2.1.6 TRANSIENT DYNAMICS . . . . . 79
2.2.2.1.7 EIGENVALUES AND EIGENMODES . . . . . 86
2.2.2.1.8 PUSHOVER . . . . . 89
2.2.2.2 ANALYSIS TYPES . . . . . 93
2.2.2.3 ASSOCIATED PREPROCESSED PROJECTS . . . . . 94
2.2.3 CONTROL . . . . . 96
2.2.4 DYNAMICS . . . . . 98
2.2.5 PUSHOVER . . . . . 100
2.2.6 CONTACT ALGORITHM . . . . . 101
2.2.7 LINEAR EQUATION SOLVERS . . . . . 103
2.2.8 UNITS . . . . . 104
2.2.9 FINITE ELEMENTS . . . . . 108
2.2.10 RESULTS CONTENT . . . . . 111
2.3 ASSEMBLY . . . . . 114
2.3.1 PREPROCESSING . . . . . 115
2.3.1.1 USER INTERFACE, MAIN MENU AND BASIC TOOLS . 116
2.3.1.1.1 Main preprocessor window . . . . . 117
2.3.1.1.2 Popup/ribbon menu . . . . . 118
2.3.1.1.2.1 FILE . . . . . 120
2.3.1.1.2.2 WINDOWS . . . . . 121
2.3.1.1.2.3 ASSEMBLY . . . . . 124
2.3.1.1.2.4 UNDO . . . . . 125
2.3.1.1.2.5 VIEW . . . . . 126
2.3.1.1.2.6 EDGES/FACES SELECTION . . . . . 127
2.3.1.1.2.7 GLOBAL SELECTION TOOLS . . . . . 130
2.3.1.1.2.8 EDIT . . . . . 134
2.3.1.1.2.9 PARAMETERS . . . . . 138

2.3.1.1.2.10 DRAW . . . 139
2.3.1.1.2.11 MESH . . . 146
2.3.1.1.2.12 TOOLS . . . 149
2.3.1.1.2.13 OVERLAID MESHES . . . 150
2.3.1.1.2.14 FREQUENTLY USED TOOLS . . . 152
2.3.1.1.2.15 SHOW CONSTRUCTION/EXCAVATION STEPS 154
2.3.1.1.2.16 SETTINGS . . . 155
2.3.1.2 MACRO-MODEL . . . 157
2.3.1.2.1 POINT . . . 159
2.3.1.2.2 OBJECTS . . . 161
2.3.1.2.3 2D mesh mapping . . . 171
2.3.1.2.4 Extrusion direction . . . 174
2.3.1.2.5 Subdomain . . . 175
2.3.1.2.6 SEEPAGE . . . 217
2.3.1.2.7 HEAT EXCHANGER . . . 220
2.3.1.2.8 CONVECTION/RADIATION . . . 226
2.3.1.2.9 VISCOUS DAMPER . . . 229
2.3.1.2.10 INTERFACE . . . 232
2.3.1.2.11 PRESSURE BC . . . 238
2.3.1.2.12 TEMPERATURE BC . . . 242
2.3.1.2.13 HUMIDITY BC . . . 245
2.3.1.2.14 FLUID FLUX . . . 248
2.3.1.2.15 HEAT FLUX/POWER . . . 251
2.3.1.2.16 HUMIDITY FLUX . . . 255
2.3.1.2.17 PILE . . . 258
2.3.1.2.18 NAIL . . . 264
2.3.1.2.19 SURFACE LOAD . . . 271
2.3.1.2.20 POINT LOAD . . . 279
2.3.1.3 FE MODEL . . . 282
2.3.1.3.1 COMMON METHODS FOR ALL FE MODEL COMPONENTS . . . 283
2.3.1.3.2 NODE . . . 291
2.3.1.3.3 BEAM . . . 298
2.3.1.3.4 TRUSS/ANCHOR . . . 314
2.3.1.3.5 CONTINUUM 2D . . . 324

2.3.1.3.6 CONTINUUM 3D . . . . . 330
2.3.1.3.7 SHELL . . . . . 337
2.3.1.3.8 SHELL-ONE LAYER . . . . . 344
2.3.1.3.9 MEMBRANE . . . . . 351
2.3.1.3.10 INFINITE ELEMENT . . . . . 357
2.3.1.3.11 TENDON EMBEDDED IN CONTINUUM/SHELL 362
2.3.1.3.12 INTERFACE (SMALL DEFORMATION) . . . . . 365
2.3.1.3.13 INTERFACE (LARGE DEFORMATION) . . . . . 379
2.3.1.3.14 SEEPAGE . . . . . 383
2.3.1.3.15 CONVECTION/RADIATION . . . . . 389
2.3.1.3.16 VISCOUS DAMPER . . . . . 394
2.3.1.3.17 SHELL HINGES . . . . . 398
2.3.1.3.18 BOUNDARY CONDITIONS . . . . . 401
2.3.1.3.18.1 SOLID BC . . . . . 402
2.3.1.3.18.2 TEMPERATURE BC . . . . . 409
2.3.1.3.18.3 HUMIDITY BC . . . . . 414
2.3.1.3.18.4 PRESSURE BC . . . . . 415
2.3.1.3.18.5 PERIODIC BC . . . . . 422
2.3.1.3.19 LOADS . . . . . 426
2.3.1.3.19.1 BODY LOAD . . . . . 427
2.3.1.3.19.2 NODAL LOAD . . . . . 429
2.3.1.3.19.3 SURFACE LOAD . . . . . 431
2.3.1.3.19.4 BEAM LOAD . . . . . 438
2.3.1.3.19.5 POINT LOAD . . . . . 440
2.3.1.3.19.6 LINE LOAD . . . . . 445
2.3.1.3.19.7 SURFACE LOAD . . . . . 450
2.3.1.3.20 ADDED MASSES . . . . . 455
2.3.1.3.20.1 NODAL ADDED MASSES . . . . . 456
2.3.1.3.20.2 DISTRIBUTED ADDED MASSES . . . . 458
2.3.1.3.20.3 FILTERING OF ADDED MASSES . . . . 460
2.3.1.3.20.4 POINT MASS ON ELEMENTS . . . . . 462
2.3.1.3.20.5 LINE MASS ON ELEMENTS . . . . . 467
2.3.1.3.20.6 SURFACE MASS ON ELEMENTS . . . . 472
2.3.1.3.21 DISTRIBUTED FLUX . . . . . 477

2.3.1.3.21.1 FLUID FLUX . . . . . 478
2.3.1.3.21.2 HEAT FLUX . . . . . 481
2.3.1.3.21.3 HUMIDITY FLUX . . . . . 482
2.3.1.3.22 INITIAL CONDITION . . . . . 483
2.3.1.3.22.1 INITIAL PRESSURE . . . . . 484
2.3.1.3.22.2 INITIAL TEMPERATURE . . . . . 488
2.3.1.3.22.3 INITIAL HUMIDITY . . . . . 491
2.3.1.3.22.4 INITIAL STRESS . . . . . 494
2.3.1.3.22.5 INITIAL STRESS ON ELEMENT(S) . . . 498
2.3.1.3.22.6 INITIAL STRAIN . . . . . 502
2.3.1.3.22.7 INITIAL STRAIN ON ELEMENT(S) . . . 505
2.3.1.3.22.8 INITIAL DISPLACEMENT/VELOCITY . . 507
2.3.1.3.22.9 IMPOSED STRAINS ON BEAMS/SHELLS . . . . . 510
2.3.1.3.23 DRM DOMAINS (DOMAIN REDUCTION METHOD) 512
2.3.1.3.24 KINEMATIC CONSTRAINTS . . . . . 515
2.3.1.3.25 NODAL LINK . . . . . 517
2.3.1.3.26 MESH TYING . . . . . 520
2.3.1.4 PUSHOVER CONTROL NODE . . . . . 522
2.3.1.5 BOREHOLES . . . . . 524
2.3.1.6 MOVEMENT PATHS . . . . . 529
2.3.2 MESH INFO . . . . . 533
2.3.3 MATERIALS . . . . . 534
2.3.3.1 USER MATERIAL DATA BASE . . . . . 536
2.3.3.2 CONTINUUM / STRUCTURE TYPE . . . . . 537
2.3.3.2.1 BEAMS AND AXISYMMETRIC SHELLS . . . . . 539
2.3.3.2.1.1 BEAM MODEL . . . . . 540
2.3.3.2.1.2 AXISYMMETRIC SHELL MODEL . . . . 550
2.3.3.2.2 CONTACT . . . . . 556
2.3.3.2.3 PILE INTERFACE . . . . . 563
2.3.3.2.4 PILE TIP INTERFACE . . . . . 568
2.3.3.2.5 NAIL INTERFACE . . . . . 571
2.3.3.2.6 FIXED ANCHOR ZONE INTERFACE . . . . . 575
2.3.3.2.7 CONTINUUM . . . . . 579
2.3.3.2.7.1 ELASTIC . . . . . 580

2.3.3.2.7.2 UNIT WEIGHT . . . 581
2.3.3.2.7.3 FLOW . . . 583
2.3.3.2.7.4 CREEP . . . 587
2.3.3.2.7.5 INITIAL K0 STATE . . . 591
2.3.3.2.7.6 HEAT . . . 594
2.3.3.2.7.7 HUMIDITY . . . 597
2.3.3.2.7.8 LOCAL STABILITY . . . 598
2.3.3.2.7.9 DAMPING . . . 599
2.3.3.2.7.10 CONTINUUM: ELASTIC . . . 600
2.3.3.2.7.11 CONTINUUM: AGING CONCRETE . . . 601
2.3.3.2.7.12 CONTINUUM: CAP . . . 603
2.3.3.2.7.13 CONTINUUM: MODIFIED CAM-CLAY . . . 606
2.3.3.2.7.14 CONTINUUM: DRUCKER–PRAGER . . . 608
2.3.3.2.7.15 CONTINUUM: HOEK–BROWN (M-W) . . . 610
2.3.3.2.7.16 CONTINUUM: MOHR–COULOMB (M-W) . . . 612
2.3.3.2.7.17 CONTINUUM: MULTILAMINATE . . . 614
2.3.3.2.7.18 CONTINUUM: RANKINE (M-W) . . . 616
2.3.3.2.7.19 CONTINUUM: MOHR-COULOMB . . . 617
2.3.3.2.7.20 CONTINUUM: HS-small . . . 619
2.3.3.2.7.21 CONTINUUM: Densification model . . . 624
2.3.3.2.7.22 CONTINUUM: HOEK–BROWN (true) . . . 629
2.3.3.2.7.23 CONTINUUM: Plastic damage model for concrete . . . 633
2.3.3.2.8 CONTINUUM FOR STRUCTURES . . . 636
2.3.3.2.9 HEAT CONVECTION . . . 637
2.3.3.2.10 HEAT RADIATION . . . 638
2.3.3.2.11 HUMIDITY CONVECTION . . . 639
2.3.3.2.12 INFINITE MEDIA . . . 640
2.3.3.2.13 MEMBRANES . . . 641
2.3.3.2.13.1 FIBER MODEL . . . 642
2.3.3.2.13.2 PLANE STRESS MODELS . . . 644
2.3.3.2.13.3 MEMBRANE - FABRIC MODELS . . . 646
2.3.3.2.14 SEEPAGE . . . 648
2.3.3.2.15 VISCOUS DAMPERS . . . 649

2.3.3.2.16 MODEL FOR SHELLS . . . . . 650
2.3.3.2.17 TRUSS (ANCHOR) MODEL . . . . . 657
2.3.3.2.18 BEAM NONLINEAR HINGES . . . . . 660
2.3.3.2.19 SHELL NONLINEAR HINGES . . . . . 663
2.3.3.3 NON-STANDARD MATERIAL DATA . . . . . 665
2.3.3.4 MATERIAL DATA VALUES . . . . . 668
2.3.4 EXISTENCE FUNCTION . . . . . 671
2.3.5 LOAD FUNCTION . . . . . 673
2.3.6 EVOLUTION FUNCTION . . . . . 677
2.3.7 GRAVITY . . . . . 679
2.3.8 SEISMIC INPUT . . . . . 681
2.3.9 TABLE OF REINFORCEMENT SETS FOR BEAMS . . . . . 683
2.4 ANALYSIS . . . . . 684
2.4.1 RESTART . . . . . 685
2.4.2 BATCH PROCESSING . . . . . 686
2.5 RESULTS . . . . . 687
2.5.1 POSTPROCESSING . . . . . 688
2.5.1.1 HOW DO I..... . . . . 689
2.5.1.2 HOW TO USE MACRO . . . . . 691
2.5.1.3 HOW TO VISUALIZE STRESS RESULTANT ENVELOPES FOR STRUCTURAL ELEMENTS . . . . . 693
2.5.1.4 WORKING OUT RESULTS FROM PARAMETRIC ANALYSIS 695
2.6 EXTRAS . . . . . 699
2.7 SYSTEM CONFIGURATION . . . . . 700
2.8 HELP . . . . . 701

**3 TROUBLESHOOTING 703**

3.1 CALCULATION MODULE . . . . . 704
3.1.1 ERRORS AND WARNINGS HANDLING . . . . . 705
3.1.2 FREQUENTLY ASKED QUESTIONS . . . . . 711

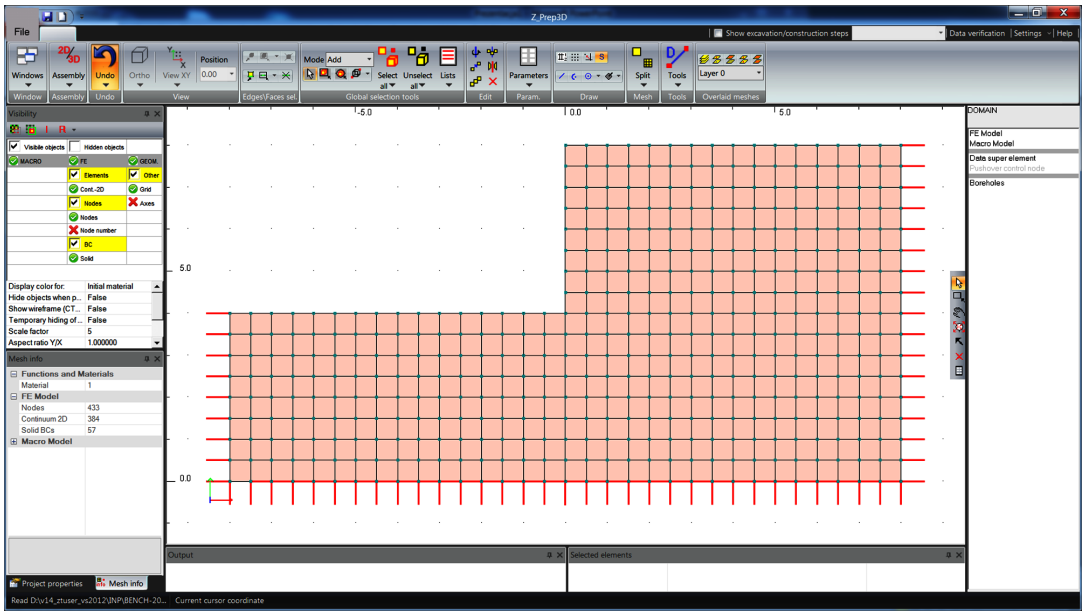
# System requirements

## ZSoil® 2026- Requirements:

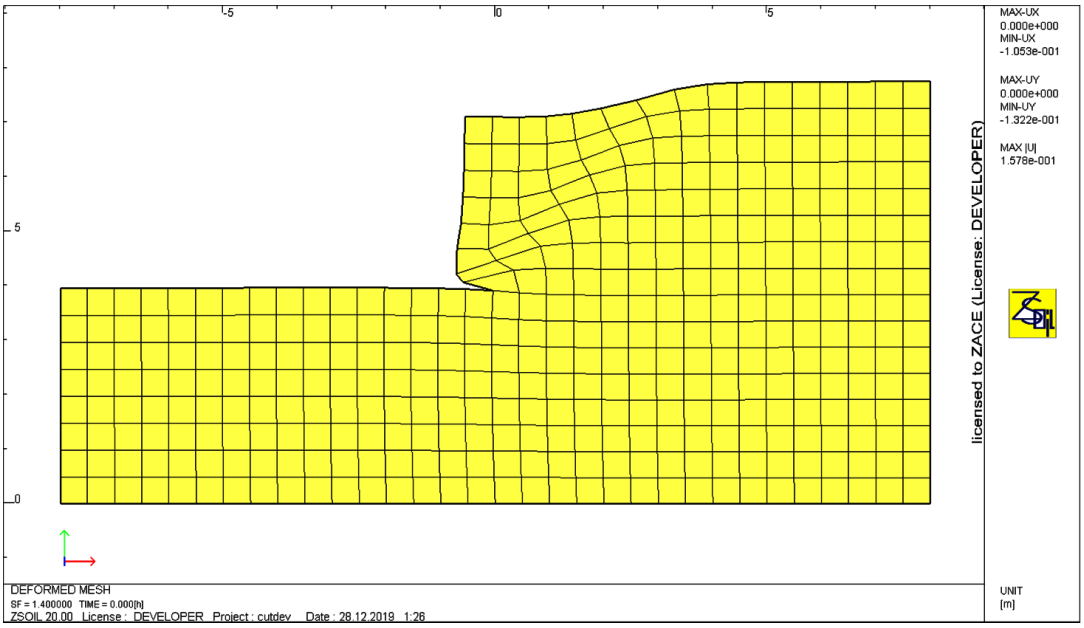
| | |
|---|---|
| Processor : | i5, i7, i9 or Intel XEON; AMD not supported |
| System 64 bit: | Windows: Windows 10, 11 |
| RAM : | 8-64 GB of RAM recommended |
| Hard-disk space : | 500 GB |
| Graphical resolution : | higher than 1280 x 1024 (1024 x 768 supported) |
| Problem size limitations : | hardware-dependent |
| Max. test problem size : | 1 500 000 dofs |
| | average problem size in daily practice |
| | 200 000 ÷ 500 000 dofs |

# Getting started

This little tutorial is here in order to get you acquainted with the ZSoil® environment.



- Move to `File/Open` (which means choose the Open option in the File menu)
- In the "inp/DataPrep" directory, choose the "cutdev.inp" file and open it. This example includes stability analysis of a vertical cut
- Move to `File/Save As ...` and give a new name (for instance "myfile") so that you'll be able to modify freely the parameters of this analysis without destroying the initial data set
- Move to `Assembly/Preprocessing`. You will see the finite element mesh in ZSoil® graphical preprocessor
- To quit the preprocessor move to `File/Exit without saving`
- Then, back in the ZSoil® environment, move to `Analysis/Run Analysis`. This will start the safety analysis for the vertical cut. The analysis will fail for a safety factor of 1.4
- Move to `Results/Postprocessing` option. You will enter ZSoil® graphical postprocessor. Move to `Results/Deformed Mesh` to see the failure mechanism. Choose `Exit` to get back to the ZSoil® application




- You may now move to `Assembly/Materials` and try to change the soil parameters. Then, move again to `Analysis/Run Analysis` to see the influence of your changes on the safety factor and to `Results/Postprocessing` to visualize failure pattern

# PREFACE

Document *DATA PREPARATION MANUAL* provides a help for the options offered in the program. In particular chapter:

**INTRODUCTION** gives preliminary information on sign convention, units and other issues.

**DATA PREPARATION DESCRIPTION** gives definition and explanation of data editing technique for all items present in the program. As often as possible, a simple example is used to illustrate the case.

**TROUBLESHOOTING** gives hints how to proceed in cases when error or warning messages are issued.

More complicated examples of entering computational model data, related to practical problems may be found in part: *TUTORIALS*.

The quickest approach to data preparation consists in loading an existing file, saving it under a different name (option SAVE AS in FILES) and then modifying it. Otherwise define a new mesh and load system with the graphical preprocessor (see e.g. Tutorial 1).

For the theoretical background see *THEORETICAL MANUAL*.

# Chapter 1

# INTRODUCTION

Chapter *INTRODUCTION* gives preliminary information on:

**Sign convention** for stresses, strains, and fluid pressure

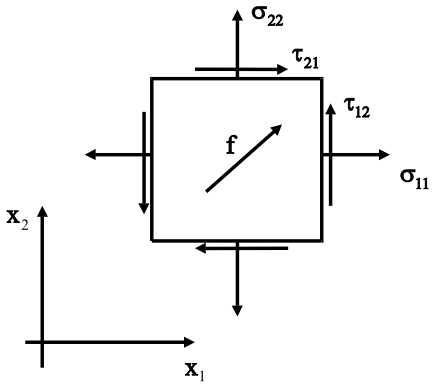
**Definition** e.g. cross–reference between elasticity constants, specific weight, etc.

**Units table** for data and result presentation

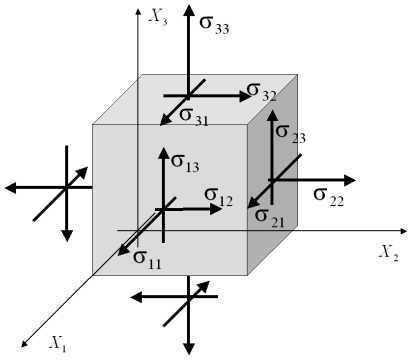
## 1.1 SIGN CONVENTION

**Stresses** Positive sign convention for the stress field, defined with respect to a cartesian coordinate system $x_1$, $x_2$, is shown in Window 1-1:

**Window 1-1: Sign convention for stresses**




Positive stresses (2D case)




Positive stresses (3D case)

Window 1-1

**Remark**: **Tensile stresses are positive, but underlined variables ($\underline{\sigma}$ e.g.) are positive in compression.**

**Strains** Sign convention for strains is adopted consistently with stress field convention, i.e. tensile strains are positive

**Fluid pressures** Sign convention for fluid pressures is adopted consistently with the stress field convention, i.e.:

- positive pressures ($p > 0$) are suction pressures
- negative pressures ($p < 0$) are compressive ones

## 1.2 DEFINITIONS

The following sections give information on:

**Specific weight**

**Elasticity constants**

### 1.2.1 UNIT WEIGHTS

The items related to the definition of unit weight are summarized in the Window 1-2:

**Window 1-2: Unit weight definition**

- $\gamma = \gamma_D + nS\gamma_F$, unit weight for a two–phase, **partially saturated** media,
- $\gamma_{\text{sat}} = \gamma_D + n\gamma_F = \gamma_B + \gamma_F$, unit weight for a two phase, **fully saturated** media,
- $\gamma_D = (1-n)\gamma_{\text{s}}$, unit weight for a single phase, **unsaturated** i.e. dry, porous media,
- $S = \dfrac{V_{\text{F}}}{V_{\text{p}}}$ – saturation ratio
- $V = V_{\text{s}} + V_{\text{p}}$, $V$ – for volume,
- $e = \dfrac{V_{\text{p}}}{V_{\text{s}}}$, void ratio, $e = \dfrac{n}{1-n}$ where:
- $n = \dfrac{V_{\text{p}}}{V}$, porosity, $n = \dfrac{e}{1+e}$ ,

with: $(.)_{\text{s}}$ – for solid skeleton, $(.)_{\text{p}}$ – for voids, $(.)_{\text{sat}}$ – for saturated soil, $(.)_D$ – for dry, $(.)_F$ – for pore fluid (water), $(.)_b$ – for buoyant soil

Window 1-2

## 1.2.2 ELASTICITY CONSTANTS

The following elasticity constants are used. The cross–references are given in the Window 1-3:

**Window 1-3: Relations between elasticity constants**

| expressed by: | $\lambda, \mu$ | $E, \nu$ | $K, \mu$ |
|---|---|---|---|
| Lame modulus $\lambda$ | $\lambda$ | $\dfrac{\nu E}{(1+\nu)(1-2\nu)}$ | $\dfrac{3K-2\mu}{3}$ |
| Kirchhoff modulus $G=\mu$ | $\mu$ | $\dfrac{E}{2(1+v)}$ | $\mu$ |
| Young modulus $E$ | $\dfrac{\mu(3\lambda+2\mu)}{\lambda+\mu}$ | $E$ | $\dfrac{9K\mu}{3K+\mu}$ |
| Oedometric m. $E_{oed}$ | — | $\dfrac{E(1-\nu)}{(1+\nu)(1-2\nu)}$ | — |
| Poisson ratio $\nu$ | $\dfrac{\lambda}{2(\lambda+\mu)}$ | $\nu$ | $\dfrac{3K-2\mu}{2(3K+\mu)}$ |
| Bulk modulus $K$ | $\dfrac{3\lambda+2\mu}{3}$ | $\dfrac{E}{3(1-2\nu)}$ | $K$ |

Window 1-3

## 1.3 UNITS TABLE

Units listed in the table are default basic units

| force | length | time | temperature | angle |
|---|---|---|---|---|
| kN | m | sec | $^{o}$C | deg |

**Remarks**:

1. Different sets of basic units can be used as well.
2. Moreover during data edition temporary units can be used.

For details of handling the units, see the subsection *data preparation: UNITS*

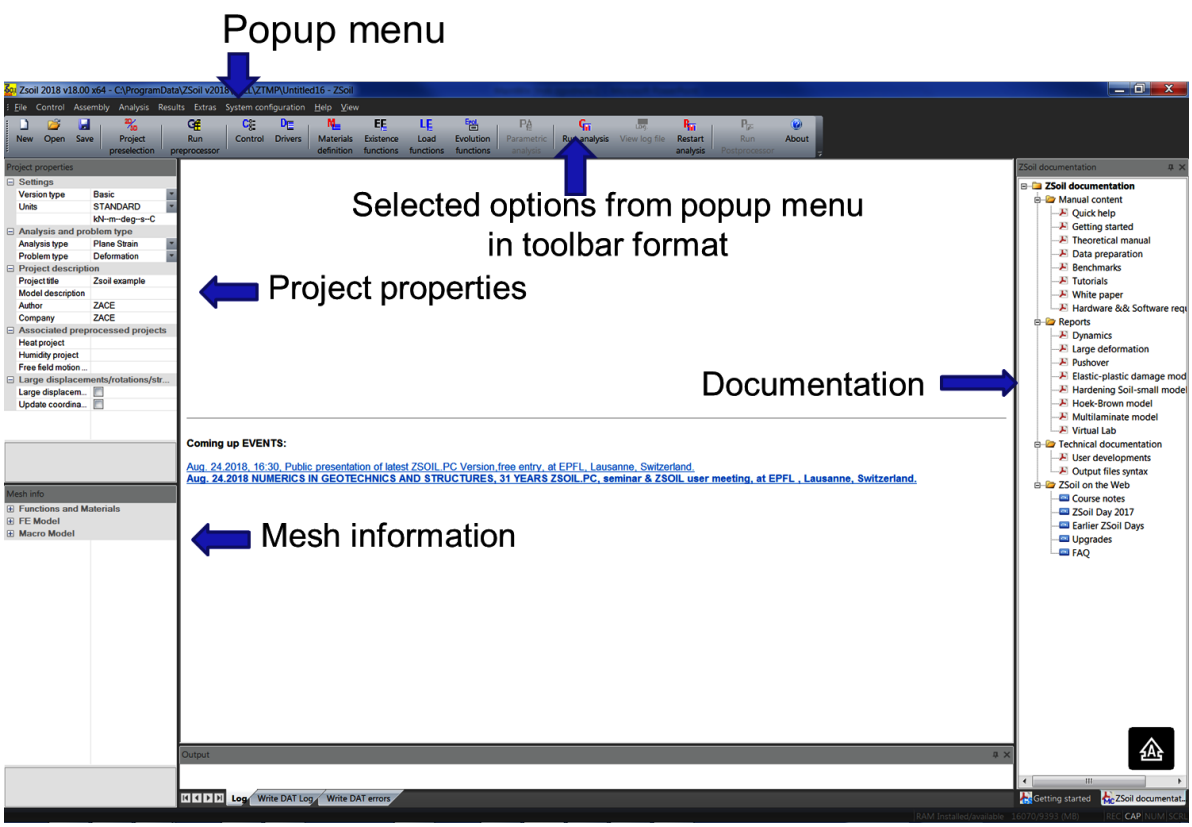
# Chapter 2

# DATA PREPARATION AND POSTPROCESSING DESCRIPTION

The structure of this chapter corresponds as strictly as possible to the structure of the user interface as shown in the Window 2-1

**Window 2-1: ZSoil®menu: contents of popup menu**

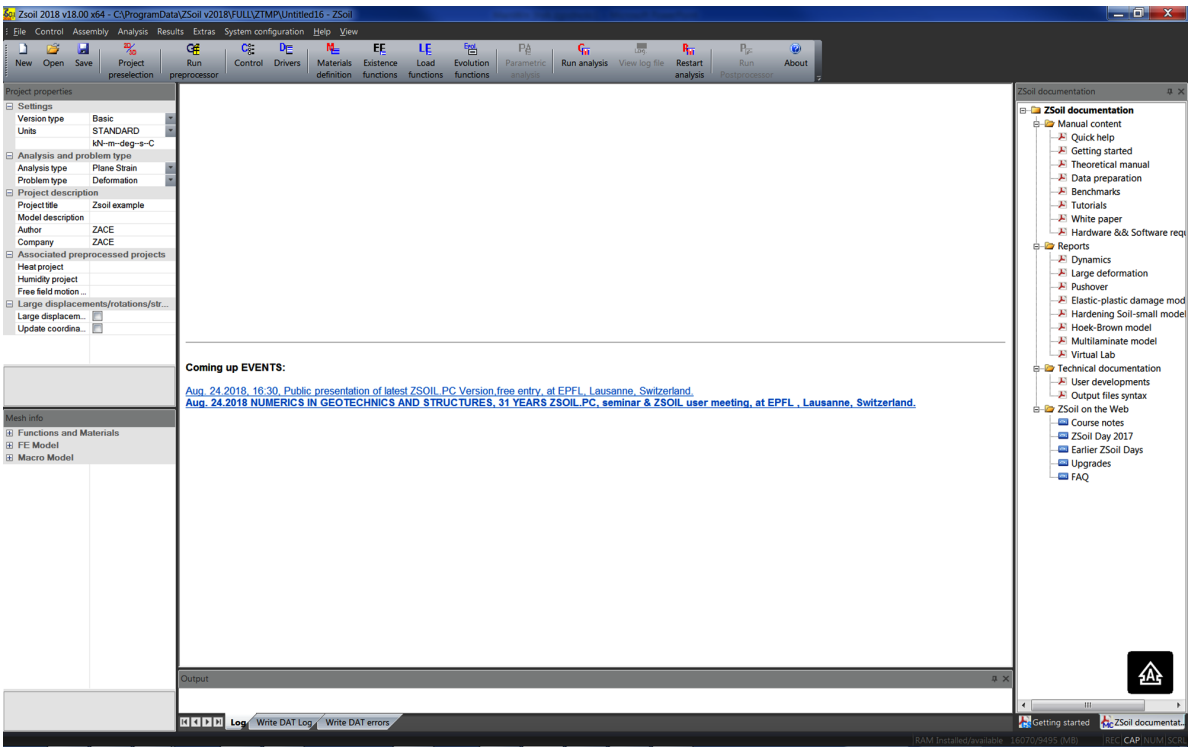
| File | Control | Assembly | Analysis | Results | Extras | System | Help |
|---|---|---|---|---|---|---|---|




Window 2-1

## Window 2-2: ZSoil®main menu toolbar

All toolbars consist of an icon and text description put below the icon.




Window 2-2

## 2.1 FILES

**Window 2-3: Files management**

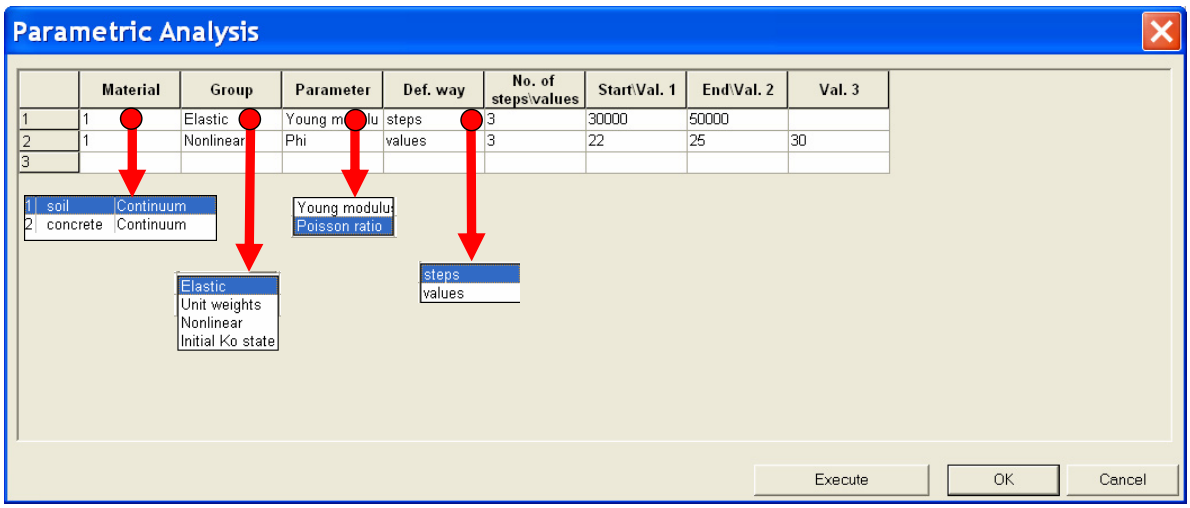
| | |
|---|---|
| `New` | Creates new project under the name "untitled *i*" where *i* is a subsequent number of the project during the session. Calls the dialog `Preferences` to select initial settings of the project |
| `Open` | Opens previously created project. Possible modification or edition of results. |
| `Open ZSoil examples` | Opens of the prepared ZSoil examples delivered with the program |
| `Save` | Calls standard MS–Windows dialog to store created data and results (if exist). |
| `Save As ...` | Calls standard MS–Windows dialog to make a copy of created data and results under a new name. |
| `Convert *.inp to ..` | Converts given data file to another ZSoil version format. |
| `Import data..` | Allows the user to import selectively some data parts from another project (like materials, load time functions etc..) |
| `Delete files` | Deletes ZSoil files (temporary data and results), but preserves data file **.inp* |
| `Batch processing` | Creates a list of data files to be executed in the batch mode and saves it as a *.bat file. Also runs existing *.bat files |
| `Parametric analysis` | Allows to generate in an automatic way a list of computational models, with few selected material properties to be varied |
| `Exit` | Calls the dialog to decide if to store the project at current stage and exits the system. |

Moreover, list of last 4 executed data files is stored. Clicking on any file from the list will open indicated project, closing (and saving) the recent one.

Window 2-3

## 2.1.1 PARAMETRIC ANALYSIS

**Window 2-4: Parametric analysis**




- To add a new property to be varied

  1. Click by mouse on first free row under column `Material` and select one of the materials already defined in the data
  2. Click by mouse next right column (under `Group`) in the same row and select subgroup of properties from which the property will be selected
  3. Click by mouse next right column (under `Parameter`) in the same row and select one of the properties to be varied
  4. Click by mouse next right column (under `Def.way`) in the same row and select one of the options `Steps` or `Values`; if you select `Steps` then the definition is completed by setting number of steps (in column `No.of steps/values`) and the two limiting values; if you select definition through `Values` then the definition is completed by setting number of values (in column `No.of steps/values`) and list of these values placed in next columns; in the example shown in the figure Young modulus will be equal to $E = 30000, 40000, 50000$, kPa while friction angle to $\phi = 22^o, 25^o, 30^o$ respectively

- To delete property from the list put cursor in the first column (the whole row will be highlighted) and press button `Delete`

- To run all automatically generated data files press button **Execute**;

**Remark**:

1. By pressing **Execute** program prepares all data files (*.inp files) that include all possible combinations of parameters to be varied and runs them immediately; each data file inherits a name of the template data file and an automatic suffix is added to it; this suffix consists of a chain of numbers (as many as number of properties to be varied) separated by underscore character; if we take the data file **foot** for instance and will perform parametric analysis

of settlements assuming that Young modulus is parametrized by `Steps` (first row in the table shown in the Figure) and friction angle is parametrized by `Values` (second row in the table shown in the Figure) then the set of 9 data files will be created; the data file labeled as **foot_2_3** corresponds to the template data file **foot** second step of the first parameter ($E$ modulus) and third value of the second parameter ($\phi$);

2. Plotting selected nodal (displacements / pressures) or element (stresses / forces) results vs one of the varied parameters can be made with aid of the graphical postprocessor (refer to section 2.5.1.4)

Window 2-4

## 2.2 CONTROL

Following options allow to control computation and results content

**Window 2-5: Options in menu** `Control`

Type of analysis (Plane strain, axisymmetry or 3D) and problem type (Deformation, Deformation+Flow, Seepage, Heat conduction, Humidity transfer) can be set in window `Project properties` placed at left top corner of ZSoil main application.

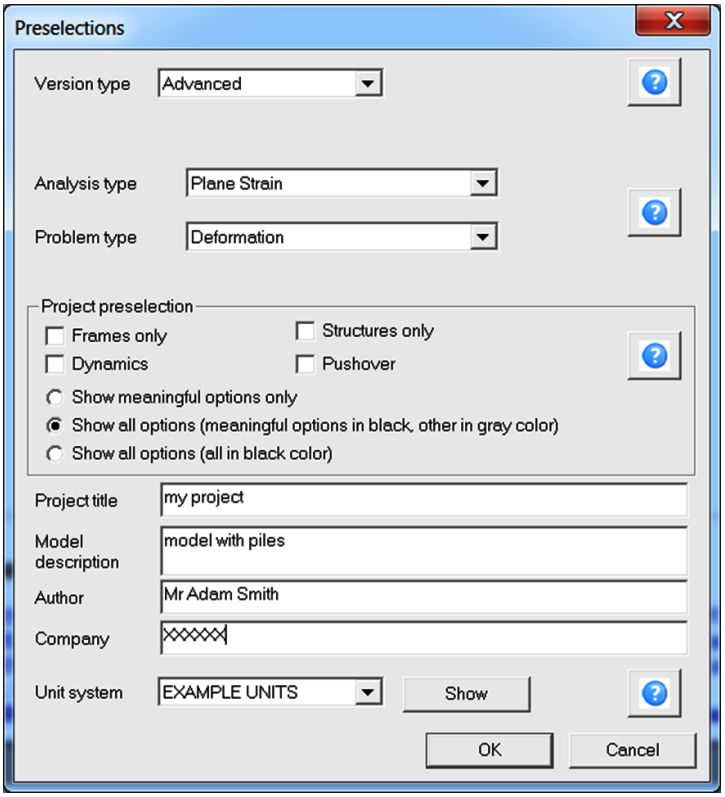
| | |
|---|---|
| `Drivers` | Creates a list of drivers for selected analysis type. |
| `Project preselection` | Selects main options like type of problem, units etc... |
| `Control` | Sets convergence tolerances, type of nonlinear solver and frequency of restart and results storage. |
| `Dynamics` | Sets type and parameters for the time history analysis algorithm and damping factors for dynamic drivers. |
| `Pushover` | Sets parameters for Pushover driver |
| `Contact Algorithm` | Sets up algorithm control data for treatment of contact. |
| `Linear equations solvers` | Selects the linear equation solver |
| `Units` | Handles units system |
| `Results content` | Handles content of the resulting output files |
| `Finite Elements` | Selects type of finite elements to be used |

Window 2-5

## 2.2.1 PROJECT PRESELECTION

**Window 2-6: Project preselection**

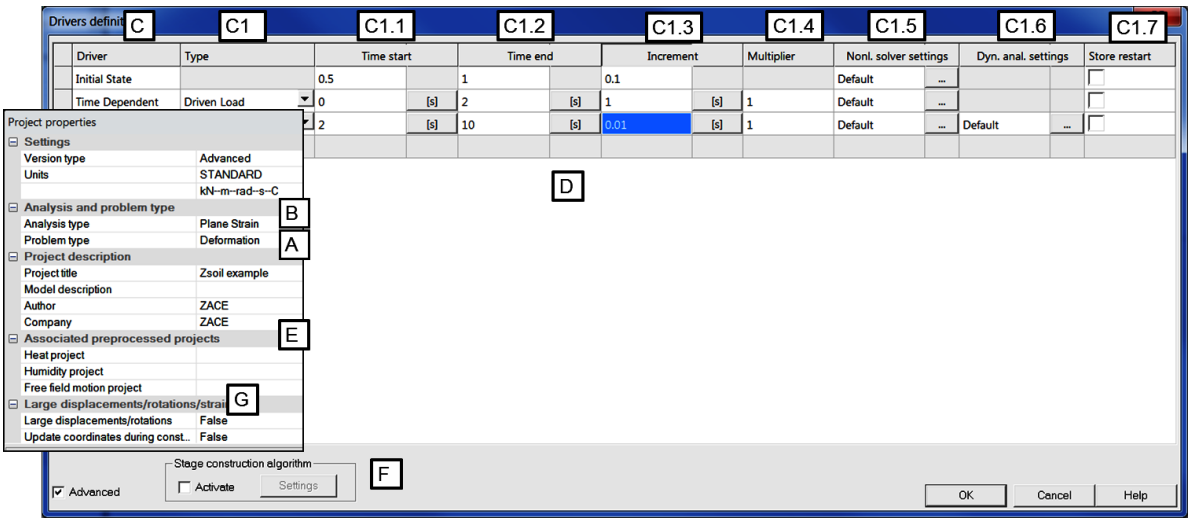
Project preselection dialog box allows the user to edit major project properties (to be set also under `Project properties` in the main ZSoil application window but also at the graphical preprocessor level), to define extra information that will be present on all output screens and customize the preprocessor user interface.




Window 2-6

## 2.2.2 ANALYSIS & DRIVERS

**Window 2-7: Analysis control**




| Key: | Option: | Comment: | Remarks: |
|---|---|---|---|
| **A.** `Problem` | `Deformation:`<br>`Deformation+Flow:`<br>`Flow:`<br>`Heat:`<br>`Humidity:` | | 2,3,4 |
| **B.** `Analysis` | `3D`<br>`Plane Strain`<br>`Axisymmetric` | | |
| **C.** `Driver` | | The definition will differ according to the active `Problem` | 1,7,8 |
| **D.** `Drivers list` | | Displays the sequence of drivers | 5,6 |
| **E.** `Associated...` | | Attach projects with computed nonstationary temperature, humidity and free field motion fields, if relevant | 3 |

| | | | |
|---|---|---|---|
| **F.** Stage construction algorithm | | Each stage construction step is treated incrementally according to the internal setup defined by pressing button **Settings** once ☒ **Activate** option is set ON | 10 |
| **G.** Large displacements / rotations | | Activate geometrical nonlinearities by setting ☒ **Activate** to ON | 9 |

Window 2-7

**Remarks**:

1. Some additional time steps will be automatically enforced by load and existence time functions, at each their characteristic time instances
2. The Problem Type must remain unchanged between restarts.
3. Humidity or Heat Transfer analysis should precede mechanical analysis if strains imposed by nonstationary temperature and/or humidity fields are considered
4. A switch from Deformation + Flow or Deformation to Flow requires recalculation of the problem
5. Sequences of drivers can be prescribed which will engender automatic restarts. Otherwise hand restart is possible.
6. While performing Restart, changes introduced to driver control parameters may concern only these drivers which are not completed yet. Those, which are completed **must** remain unchanged.
7. Note the difference in the meaning of the control parameter like Start, End, Increment, Multiplier (used to increase/decrease time increment) for different Drivers.
8. Although for most of the cases leaving Nonl. solver Settings as default is satisfactory, for some cases it may prove convenient to associate specific control setting with each Driver in the sequel. If needed, use button Set under **C1.5** to enter Control dialog box and to set algorithm control data in a manner specific for given driver. Predefined control data sets can be used as well by selection of its label from the listbox **C1.5**. See section 2.2.3 CONTROL for details. In the current version of ZSoil an automatic switch of nonlinear solver is activated once the divergence of the iterative procedure occurs. However this option will slow down the computation speed. Optionally at the end of the driver one may activate automatic storage of the current stage of computation in a separate sub-directory (C1.7)

9. Switching from geometrically linear to geometrically nonlinear mode does not require any extra action from the user; the only exception concerns standard contact elements which cannot handle large deformations; in that case standard contact elements must be deleted and large deformation contact (node-segment) has to be added to the model; in addition user must take the decision whether special fill algorithm designed for large deformations is to be used (for detailed description refer to the dedicated report on handling large deformations within Z_Soil); in the current development displacements and rotations can arbitrarily be large while strains remain small

10. During stage construction analysis new layers of soil are usually added in a single incremental step; by activating this option each construction step can be subdivided into few incremental steps, to avoid numerical problems (like lack of convergence or divergence of iterative procedure), according to the setup to be defined by pressing button **Settings**; in addition each new layer of soil in the first time step will behave as incompressible (assuming $\nu = 0.499999$) and later on an original value of Poisson's ratio is used

In the following sections see the details of setting:

**PROBLEM TYPE AND DRIVERS**

**ANALYSIS TYPE**

**ASSOCIATED PREPROCESSED PROJECTS**

### 2.2.2.1 PROBLEM TYPE AND DRIVERS

Detailed explanation and examples of application of different drivers are given in next paragraphs for the following `Problem types`:

**SINGLE PHASE (DEFORMATION) ANALYSIS**

**DEFORMATION COUPLED WITH FLOW**

**FLOW (STEADY STATE AND TRANSIENT)**

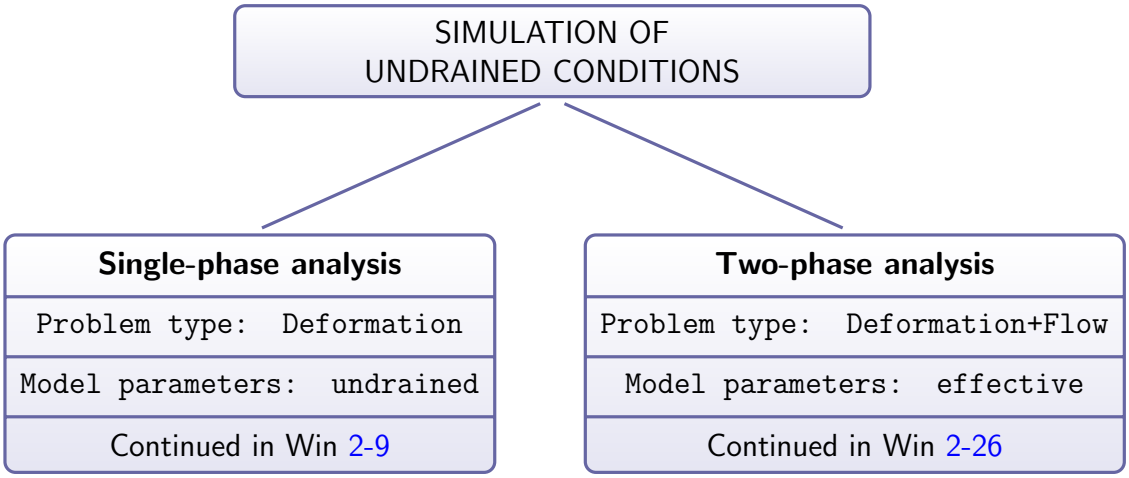
**HEAT TRANSFER (STEADY STATE AND TRANSIENT)**

**HUMIDITY TRANSFER (STEADY STATE AND TRANSIENT)**

## Related Topics

- *Theory: ALGORITHMS*
- *LOCAL STABILITY CONTROLED AT THE MATERIAL LEVEL*

**Window 2-8: Approaches for simulating undrained conditions**




**Remarks:**

- In the **single-phase analysis**, as long as undrained parameters are considered, shear resistance in the "undrained" conditions is defined with undrained shear strength $c_u$ being stress independent (constant). A variable profile for $c = c_u$ may be imposed in material models for which parameter definition through the *Super Element* is allowed.

- The simplified approach for undrained simulation by means of the **single-phase analysis** is possible with the following models: Mohr-Coulomb, Drucker-Prager, Cap and Hardening Soil. The parameter setup sequence for the Hardening Soil model is provided in the report on Hardening Soil model. Note that the single-phase approach for simulating "undrained" conditions is not possible with the Modified Cam-Clay.

- The **two-phase analysis (recommended approach)** allows to analyze the problem considering undrained shear resistance which is stress dependent as it is observed in natural soils (shear resistance will be obtained as the function of the current effective stress, effective strength parameters including overconsolidation). This approach also provides the user with a possibility of analyzing the problem in perfectly "undrained" or partially drained conditions.

Window 2-8

### 2.2.2.1.1 SINGLE PHASE (DEFORMATION) ANALYSIS

**Unknowns:** displacement field $\mathbf{u}(\mathbf{x})$

For this `Problem Type` necessary data consist of:

- solid BC (fixities with possibly applied imposed displacements, rotations),
- solid loads (gravity, surface, nodal)
- mechanical and strength parameters, and unit weights set for each applied material model

**Remarks**:

1. Presence of the underground water, in cases when pressure field is known *a priori*, can be taken into account using two equivalent approaches: (here $\gamma_D, \gamma_F$ means for dry soil and fluid unit weights and $n$ for porosity)
   - **effective stresses** (no pressure applied, buoyant weight $\gamma_B = \gamma_D + (n-1)\gamma_F$ used for saturated media)
   - **total stress** (initial pressure applied, saturated weight $\gamma_{SAT} = \gamma_D + n\gamma_F$ used)
2. Strains induced by changes of temperature or humidity can be taken into account, see `Associated projects`

**Window 2-9: Single phase analysis for simulation of "undrained" conditions**

**SINGLE PHASE ANALYSIS**

`Problem type: Deformation`

`Driver: Time dependent`

`Driver type: Driven Load`

**Total stress analysis**

`Unit weight:`
$\gamma$ (apparent) above water table
$\gamma_{SAT}$ (saturated) below water table
`Example: DataPrep/footwt.inp`

`Initial state:`
$K_o$ must be specified
**Initial pore pressure** should be imposed (use *Initial pressure*)

`Model parameters: undrained`
$\phi = \phi_u = 0^o$
$\psi = 0^o$ (to satisfy $\psi \leq \phi$)
$c = c_u$
$\nu = \nu_u = 0.499$

**Effective stress analysis**

`Unit weight:`
$\gamma_D$ (dry) above water table
$\gamma_B$ (buoyant) below water table
`Example: DataPrep/footes.inp`

`Initial state:`
$K_o$ must be specified

`Model parameters: undrained`
$\phi = \phi_u = 0^o$
$\psi = 0^o$ (to satisfy $\psi \leq \phi$)
$c = c_u$
$\nu = \nu_u = 0.499$

**Remarks:**

- These two methods are equivalent considering the presence of the ground water conditions in a single-phase, deformation analysis.
- A variable profile for $c = c_u$ may be imposed in material models for which parameter definition through the *Super Element* is allowed.
- $K_o$ has to be implicitly specified; otherwise $K_o$ will be calculated as $\nu/(1-\nu)$ taking the specified $\nu_u = 0.499$.
- In the case of **total stress analysis** and an existing ground water table, the initial pore pressure should be imposed in order to obtain an effective stress field in the `Initial State` analysis.
- Considering that the undrained conditions imply $\varepsilon_1 = \varepsilon_3$, undrained stiffness will be automatically computed accounting for the undrained Poisson's coefficient $\nu_u = 0.499$.

Window 2-9

Description of `Drivers` available for `Problem type Deformation` is given in sections:

## INITIAL STATE

## TIME DEPENDENT

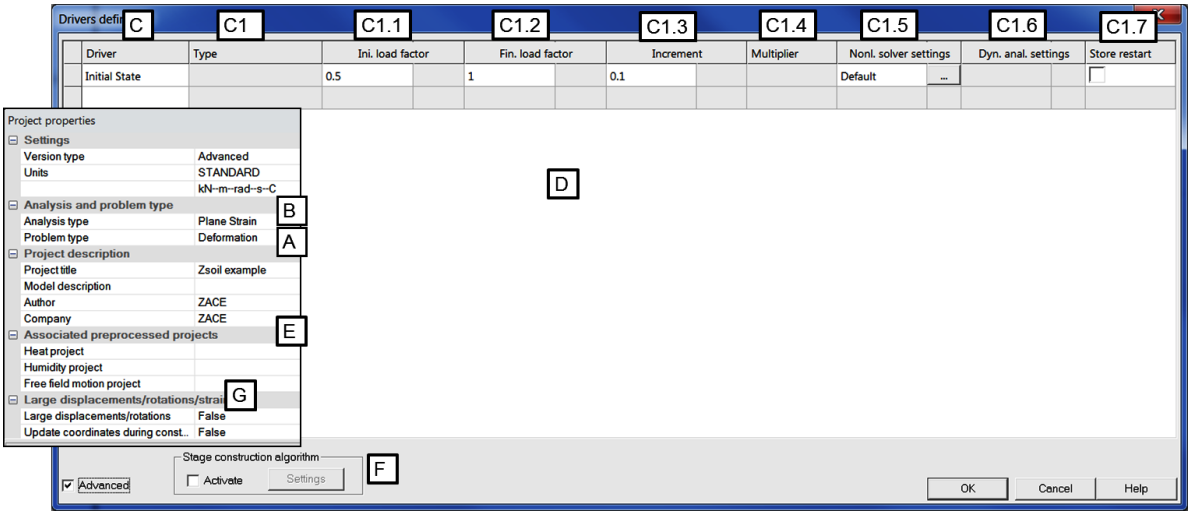
## STABILITY

## 2.2.2.1.1.1 INITIAL STATE

**Window 2-10: Initial state**

**A.** `Problem type: Deformation`
**B.** `Analysis type: any`
**C.** `Driver: Initial State`

This driver is used to define *in situ* stress state induced by gravity as well as other loads acting at time instance $t = 0$.




Control data of the driver are:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1.1** | | Initial Load Factor | (0.5) | 4 |
| **C1.2** | | Final Load Factor | (1.0) | |
| **C1.3** | | Load Factor Increment | (0.1) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-10

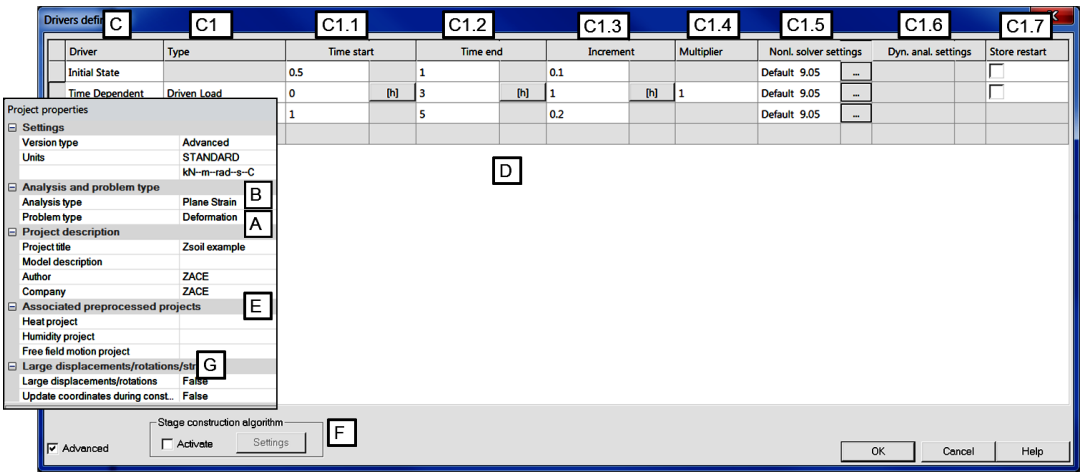
**Remarks**:

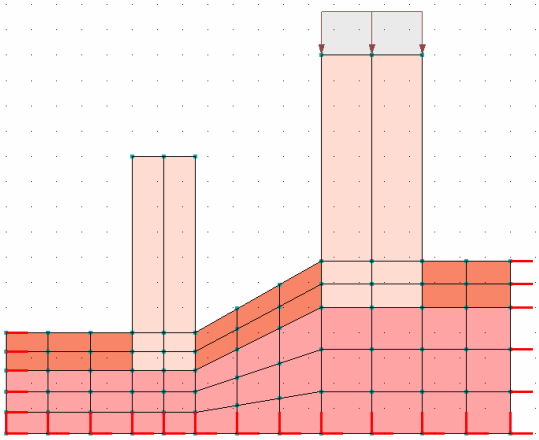
1. `Driver Initial State`: This driver controls the amplitude of loads which are non–zero at time $t = 0$ (i.e. the corresponding load time function are non-zero at $t = 0$). User prescribes: 0.5 for the Initial Load Factor; 0.1 for the Load Factor Increment and 1.0 for Final value of the Load Factor; loads will be applied progressively as $0.5g, 0.6g..., 1.0g$.
2. An initial state with zero deformation and non–zero stress-state will be generated.
3. Gravity is usually the main load to be considered for the initial state; set unit weights for all materials and gravity vector (in load menu) equal to $(0, -1, < 0 >)$ for downwards vertical gravity.
4. Load is usually gravity and/or other deadweight loads. Any `Initial Load Factor` value greater than 0.0 and less or equal to 1.0 is acceptable. The initial value equal to 0.5 is usually used.
5. During an `Initial State` analysis, time is fixed to $t = 0$. As the consequence:
   - only these system components will be taken into account, which exist permanently or have existence status set `ON` at $t = 0$,
   - loads associated with the initial state must have a non-zero load function at $t = 0$; if the load function is equal to zero at $t = 0$, then the load is to be applied later.
   - in case of time dependent material parameters, their values computed at $t = 0$ will be taken into account.
6. An `Initial State` driver can be run only once and must be the first in the list of drivers
7. Omitting `Initial State` and lack of the implicitly set initial effective stresses yields stress free and zero deformation state as the initial condition.
8. Alternatively, if an initial stress field is known, it can be introduced using `Initial Conditions Initial Stress` super elements in the `Preprocessing` session. Setting the initial guess for initial effective stresses (using Initial conditions/Initial stresses option in the preprocessor) is recommended for meshes in which size of elements varies significantly.
9. Each incremental step is computed in two runs due to algorithmic reasons.
10. After convergent run of an `Initial State` deformation is nullified. The only nonzero output concerns the stress state.
11. In case of the divergence deformation can be visualized in the `Postprocessing`. It helps to identify source of the instability. This may happen due to surface instability caused by zero cohesion, improperly set boundary conditions or highly distorted (with high aspect ratios) finite elements placed close to the free boundary
12. Some soil mechanics problems are characterized by a stress state which remains on the yield surface, i.e., on the limit of instability. It is therefore important to adopt appropriate material data to avoid triggering instability.

## Window 2-11: Initial state example – dry medium

Data: TUTORIALS/ENV.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Initial State`



**Goals:**

- To define an initial state with 2 existing buildings
- To define a safety factor with respect to an increase in height of the right building (represented by an increasing load)

**Remarks**:

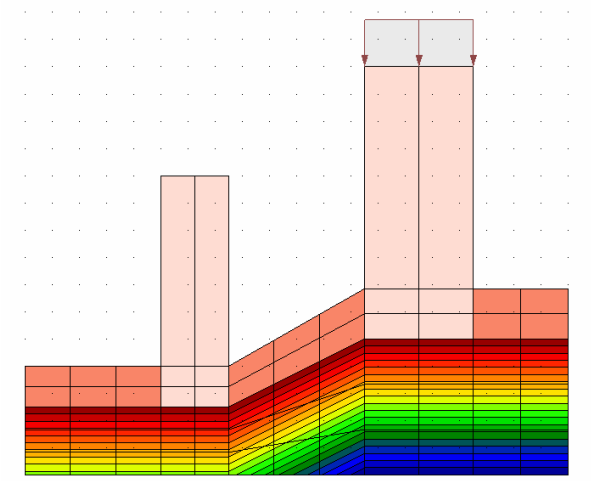
1. `Driver Initial State`: this driver controls the amplitude of loads which are non–zero at time $t = 0$ (i.e. the corresponding load time function in non-zero at time zero); in this case gravity only will be activated since the load applied on the right building is multiplied by a load function which is zero at time $t = 0$. User prescribes: 0.5 the starting gravity multiplier; gravity will be applied progressively as $0.5g, 0.6g..., 1.0g$.
2. As underground water is not taken into account in this example, in material definition $\gamma_{D1} = \gamma_{D2} = 20\text{kN/m}^3$ is specified.

Window 2-11

**Window 2-12: Initial state example – static water table (total stress analysis)**

Data: DATAPREP/ENVSWT.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Initial state`



Initial pressure field

**Remarks**:

1. The data set in this case is the same as for the dry case,except for the $\gamma$
2. The presence of the water table requires:
   - assuming that material 2 is fully saturated and that $\gamma_{D2} = 20\text{kN/m}^3$, then $\gamma_{\text{SAT2}} = \gamma_{\text{D2}} + n\gamma_F = 25\text{kN/m}^3$ (with $e = 1$ and $n = 0.5$),
   - initial pressures have to be set using `Initial Stress` super-elements. Corresponding surface loads must be specified on submerged free surfaces
3. The same problem can be solved in effective stresses.

Window 2-12

**Window 2-13: Initial state example – static water table (effective stress analysis)**

Data: DATAPREP/ENVES.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Initial State`

**Remarks**:

1. No water pressure is specified in this case.
2. Specify $\gamma_D$ (dry) above the water table and $\gamma_B$ (buoyant) $\gamma_B = \gamma_{SAT} - \gamma_F$ ($= 15\text{kN/m}^3$ for material 2)

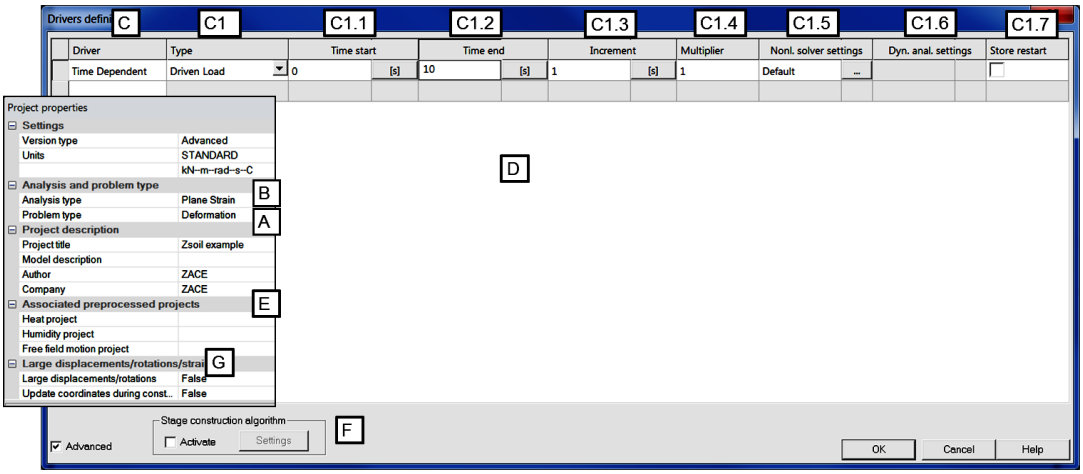
Window 2-13

## 2.2.2.1.1.2 TIME DEPENDENT

**Window 2-14: Time Dependent: Driven load**

This driver can be used for solving a sequence of incremental boundary value problems, evolving with respect to time varying loads, the domain, boundary conditions, material data and creep. In particular load bearing capacity analysis can be performed as `Time Dependent`,`Driven Load`.

**A.** `Problem type: Deformation`
**B.** `Analysis type: any`
**C.** `Driver: Time dependent`
**C1.** `Driver type: Driven load`




Control data of the driver are:

| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,4,5 |
| **C1.1** | `Time start` | Initial time value | (0.0) | 2 |
| **C1.2** | `Time end` | Final time value | (1.0) | 2 |
| **C1.3** | `Time increment` | Initial time step | (1.0) | 6 |
| **C1.4** | `Multiplier` | Time step multiplier | (1.0) | |
| **C1.5** | `Non. Solver` | Control data for nonlinear solver | | |
| **C1.7** | `Store restart` | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-14

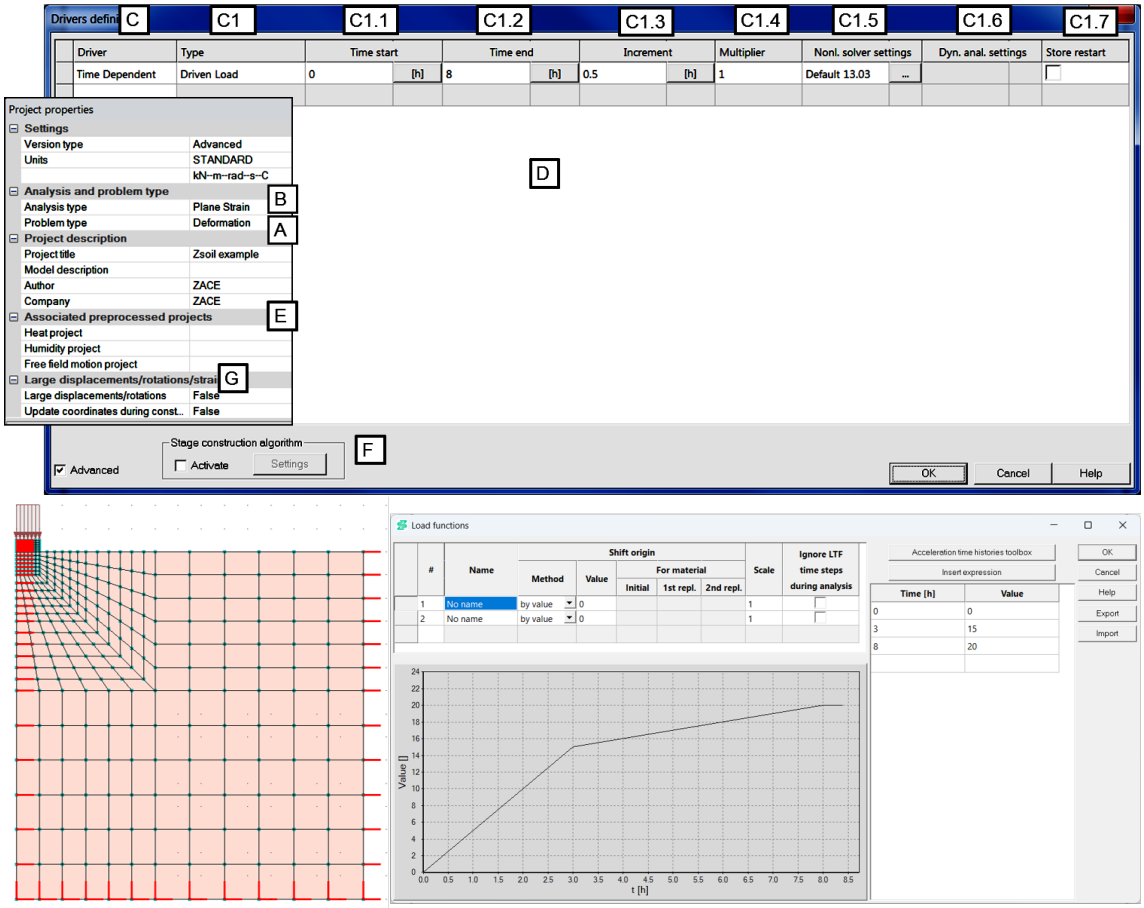
**Remarks**:

1. `Driver type`: `Driven Load` is the only available in this case.
2. Time is treated as a physical parameter (consistently with material data) only if `Creep` is activated (at the material level). In all other cases time may be treated as a fictitious parameter, used mainly to manage loading process
3. Time parameter is used as a common argument of all existence and load functions.
4. A sequence of `Time Dependent Drivers` can be defined, with different time step specification.
5. Within a sequence of `Time Dependent`, `Driven Load Drivers`, an intermediate `Stability Driver` can be added. It does not interfere the solution of `Driven Load` process coming next, in any way.
6. Additional time steps will be automatically enforced by load and existence time functions, at all their characteristic points
7. Strains due to nonstationary temperature and/or humidity fields can be taken into account, see `Associated Preprocessed Projects`

## Window 2-15: Driven load example – Surface footing (problem statement)

Data: BENCHMARKS/FOOT.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load`




**Goals:**

- To estimate ultimate limit load applied downwards (in vertical direction) to the footing. The evolution of the loading process is controlled through load time function. Time in this case is a fictitious parameter.
- To detect failure mechanism

**Results**:

Failure mechanism appears at time $t = 3.8$. The corresponding ultimate limit load is computed by multiplying the amplitude (equal to 1.0) of applied load by the value of the `Load Function` at time 3.6 (last converged step) which yields:

$$q_{\text{ult}} = 1.0 \left[ 15 + \frac{20 - 15}{8 - 3} \, (3.6 - 3) \right] = 15.6 \text{ kN/m}^2.$$

**Remark**:

A preliminary `Initial State` analysis is unnecessary as gravity is ignored in this problem and no other loads are present at time $t = 0$.

**Window 2-16: Driven load example – Surface footing with weighting soil**

Data: DATAPREP/FOOTGR.INP
**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load`

Specify:
- `Unit Weight` (`Assembly`→`Materials`) $\gamma_D = 13$,
- `Directional Gravity` multiplier (`Assembly`→`Gravity`) = {0, -1};

add an `Initial State` driver under `Control`→`Analysis & Drivers` with
- Initial Load Factor $= 0.5$,
- Final Load Factor $= 1.0$
- Load Factor Increment $= 0.1$

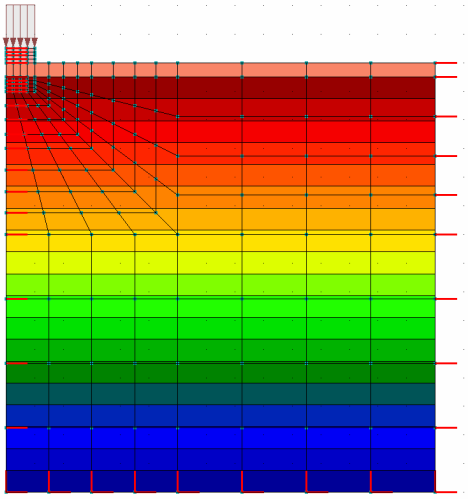
**Results**:

The distributed load with amplitude $q = 5$ kN/m$^2$ is applied downwards. The ultimate limit load of value $q_{ult} = 77$ kN/m$^2$, at the last converged step ($t = 3.4$), is detected.

Window 2-16

**Window 2-17: Driven load example – Surface footing with a water table (total stress)**

Data: DATAPREP/FOOTWT.INP
**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load`



Hydrostatic distribution of prescribed initial pressures

**Material:**
- $\gamma_D = 13$kN/m$^3$ (dry) above the water table,
- $\gamma_{SAT} = 18$kN/m$^3$ (saturated) below the water table.

Window 2-17

**Window 2-18: Driven load example – Surface footing (effective stress solution)**

Data: DATAPREP/FOOTES.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load`

Specify:

- specification of $\gamma_{\mathrm{D}} = 13\mathrm{kN/m}^2$ (dry) above the water table as before,
- specification of $\gamma_{\mathrm{B}} = 8\mathrm{kN/m}^2$ (buoyant) under the water table level.

**Results**:

The same ultimate load as in the previous case are obtained.

**Remark**:

Two methods of considering the presence of underground water in single phase, deformation analysis are equivalent:

- **effective stresses** (no pressure applied, buoyant weight $\gamma_B$ used for saturated media)
- **total stress** (initial pressure applied, saturated weight $\gamma_{SAT}$ used )

Window 2-18

**Window 2-19: Driven load example – Axisymmetric footing (effective stress)**

Data: DATAPREP/FOOTAES.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Axisymmetry`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load`

Change the `Analysis Type` to `Axisymmetry` under (`Control`→`Analysis & Drivers`). In material model data (Drucker–Prager type), switch `Size Adjustment` to `Intermediate` with $\xi = 0$.

**Results**:

The ultimate limit load $q_{\mathrm{ult}} = 95$ kN/m$^2$ is achieved.

Window 2-19

**Remark**:

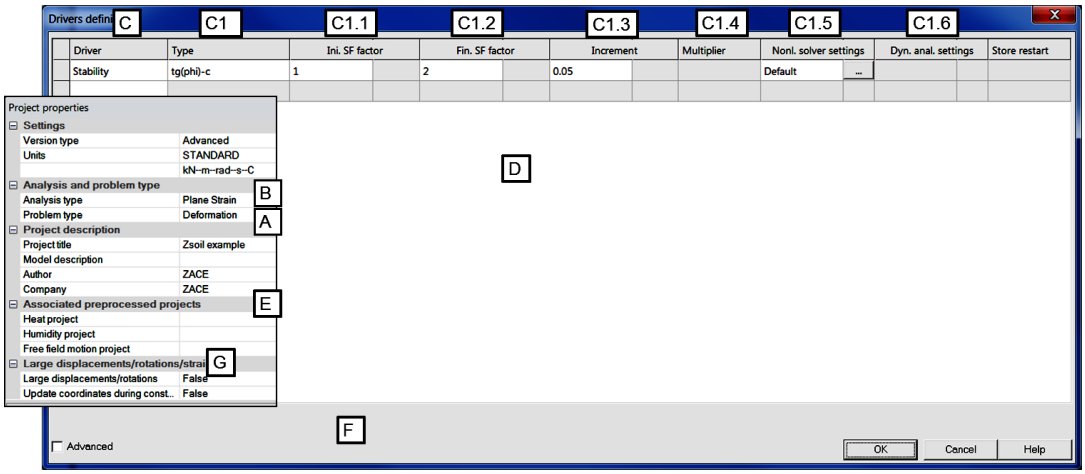
The above examples can be run under displacement control instead of load control.

## 2.2.2.1.1.3 STABILITY

**Window 2-20: Stability**

**A.** `Problem type: Deformation`
**B.** `Analysis type: any`
**C.** `Driver: Stability`
**C1.** `Driver type:` see the window 2-21

This driver can be used to carry out stability analysis. Elastoplastic material models like Mohr–Coulomb, Drucker–Prager, etc... have to be used. The $c - \varphi$ reduction algorithm (pioneered in ZSoil® v.1.), applicable to yield criteria defined through cohesion and friction angle, is used to reduce strength parameters $c$ and $tan(\phi)$, until global instability is detected. Another two reduction algorithms can be choosen: Stress Level reduction (SL) (applicable to material models for which yield surface is defined not by $c$ neither $\phi$) and reduction of Cohesion only.




Driver parameters are as follows:

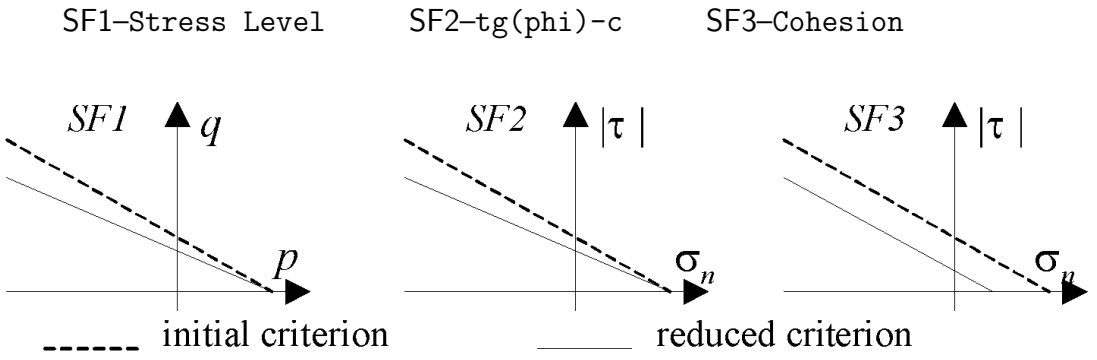
| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 5,6 |
| **C1.1** | | - Initial SF factor | (1.0) | 1,4 |
| **C1.2** | | - Final SF factor | (2.0) | 1,2 |
| **C1.3** | | - Increment of SF factor | (0.02) | 1 |
| **C1.5** | | - Algorithm control data | | |

Window 2-20

**Remarks**:

1. User prescribes Initial value of the safety factor (SF), its final value and increment. In the example shown in Win.2-20 safety analysis will be carried out for SF = 1.0,1.1,1.2 up to 3.0 or SF factor value at which an instability is detected
2. Restart from the last converged step is always possible. During restart user may adjust the final SF value and the increment to obtain solution with assumed accuracy
3. Stability driver can be used at any time, may be declared more than once (also in row) in the list **D** of drivers; it does not affect results of next drivers
4. Stability driver can also be controlled at the material level to avoid early instabilities caused by flat and tiny sliding
5. Results (i.e. safety factor and failure pattern) are usually sensitive to the choice of `Reduction method`(**C1**) (see Window 2-21)
6. Deformation field visualized by the postprocessor at SF value corresponding to diverged safety analysis step represents a **failure pattern**. If the failure pattern cannot be easily detected it is recommended to visualize incremental deformation between last two safety analysis steps.

**Window 2-21:** `Driver types` **for** `Stability` **analysis**

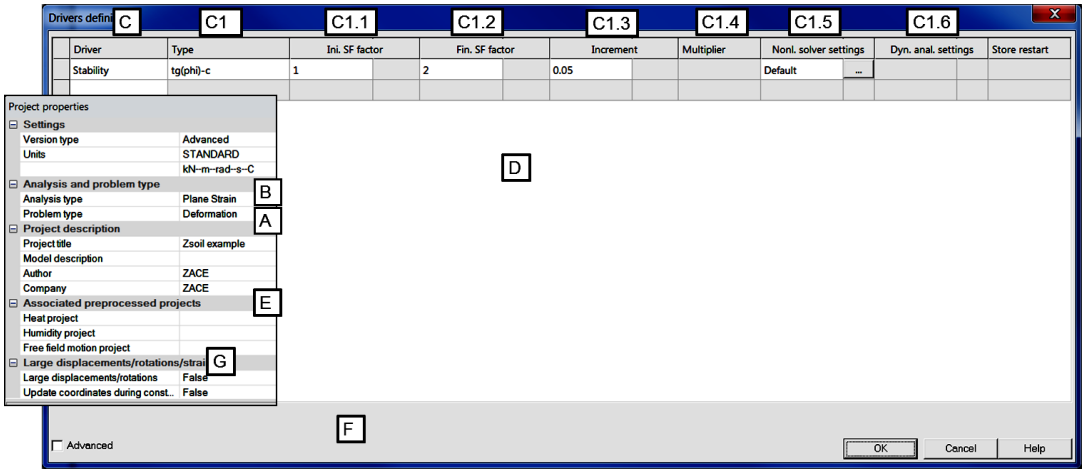



Three reduction methods for strength parameters

Window 2-21

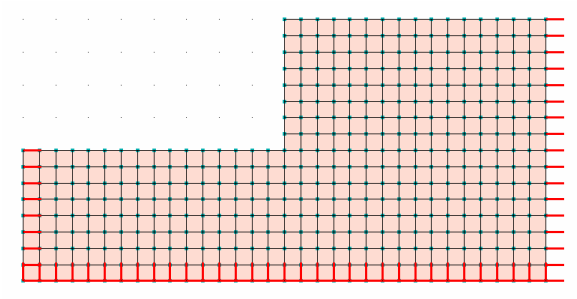
## Window 2-22: CUT (single phase) example – problem outlook

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Stability`




**Goal:**
To estimate safety factor SF2 through $c - \phi$ reduction method.

**Material:**
homogeneous Drucker–Prager material with ($c = 16$ kPa, $\varphi = 30°$, $\gamma_D = 13\text{kN/m}^3$).



FE mesh common for **CUT** examples

Window 2-22

## Window 2-23: Stability example – analysis with $c - \phi$ reduction (SF2)

Data: TUTORIALS/CUT13.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Stability`

**Results**:
Instability occurs for SF2 $= 1.8$.

Window 2-23

**Window 2-24: Stability example – analysis with preliminary initial state**

Data: DATAPREP/CUTIS.INP
**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Stability`

Specify a new driver (Initial State) with Initial Load Factor 0.5, Final Load Factor 1.0 and load increment 0.1.

**Results**:

Instability occurs for SF2 $= 1.8$.

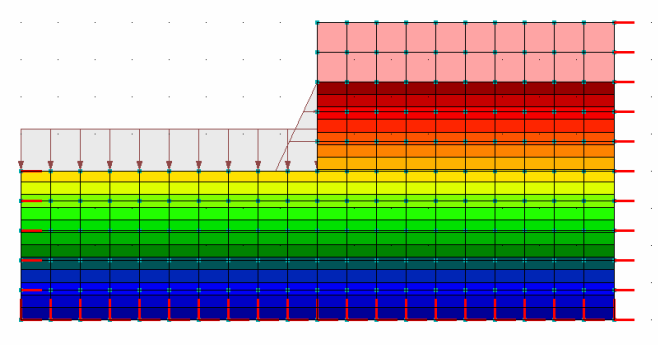
**Remark**:

Notice that the preliminary initial state analysis influences the deformation state but has no influence on the safety factor (SF2 $= 1.8$).

Window 2-24

**Window 2-25: Stability example – analysis with water table**

Data: DATAPREP/CUTWT.INP
**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane strain`
**C.** `Driver: Stability`



Distribution of (initial) water pressures and equivalent surface load on submerged boundaries
**Results**:
The safety factor SF2 $= 2.1$ is obtained.

**Remarks**:
1. Water pressures must be specified as initial conditions and submerged parts of the free boundaries must be loaded by corresponding distributed loads.
2. Different unit weights must be specified above ($\gamma_{\text{D}} = 13\text{kN/m}^3$) and below ($\gamma_{\text{SAT}} = 18\text{kN/m}^3$) water table.

Window 2-25

## Related Topics

*THEORETICAL MANUAL. STABILITY ANALYSIS*

### 2.2.2.1.2 DEFORMATION COUPLED WITH FLOW

**Unknowns:**

**displacement field** $\mathbf{u}(\mathbf{x})$,

**fluid pressure field** $p(\mathbf{x})$

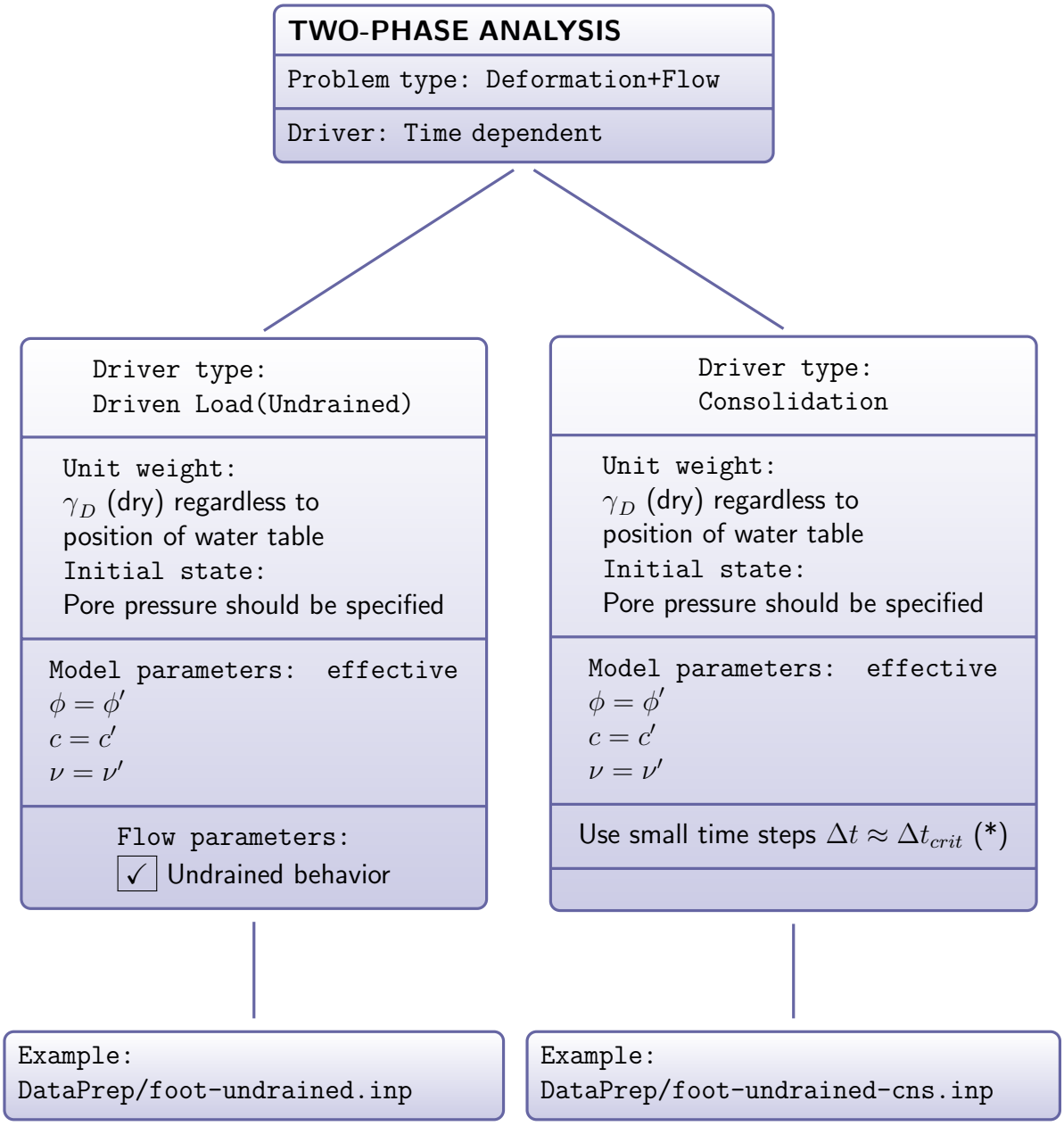
For this `Problem Type` necessary data consists of:

- solid BC (fixities with possibly applied imposed displacements, rotations),
- fluid pressure BC (nodal fluid pressures, fluid heads, seepage surfaces),
- solid loads (gravity, surface, nodal)
- mechanical, seepage, strength parameters and unit weights, set for each applied material model

**Remarks**:

1. Presence of underground water is taken into account in an automatic way (uncoupled or coupled total stress analysis) corresponding to chosen `Driver Type`:
   - **Driven Load+Steady state flow** -each solution step consists of the two substeps; steady state fluid flow problem is solved first for unknown $p(\mathbf{x})$ and then resulting pressure field is taken as an explicit input into the mechanical analysis substep (through extended Bishop's effective stress principle).
   - **Driven Load+Transient flow** -each solution step consists of the two substeps; transient fluid flow problem is solved first for unknown $p(\mathbf{x})$ and then resulting pressure field is taken as an explicit input into the mechanical analysis substep (through extended Bishop's effective stress principle).
   - **Driven Load (undrained)** -a coupled system of equations including equilibrium and fluid flow continuity (without divergence of Darcy velocity term) are solved for unknowns $(\mathbf{u}(\mathbf{x}), \Delta p_{und}(\mathbf{x}))$. Note that in that case pressure in the fluid is composed of two terms i.e. $p$ (nodal pressures in the fluid coming from uncoupled/coupled deformation+flow analysis) and undrained excess pore pressure $\Delta p_{und}$ (at center of element). This driver cannot be followed by **Driven Load+Steady state flow**, **Driven Load+Transient**, **Consolidation**.
   - **Consolidation** -a coupled system of equations including equilibrium and fluid flow continuity are solved for unknowns $(\mathbf{u}(\mathbf{x}), p(\mathbf{x}))$. In this case deformations may influence variation of the pressure field and vice versa.
2. Unit weight of fully or partially saturated two-phase media is internally evaluated as a function of the computed saturation ratio through $\gamma_S = \gamma_D + nS\gamma_F$, with $\gamma_D, \gamma_F$ meaning for dry soil and fluid unit weights, porosity $n = \frac{e}{1+e}$ and saturation ratio $S = S(S_r, \alpha, \gamma_F, p)$
3. Strains induced by changes of temperature or humidity can be taken into account, see `Associated preprocessed projects`

**Window 2-26: Two-phase analysis for simulation of "undrained" conditions**




**Remarks:**

- In the two-phase analysis, a stress dependent "undrained" shear resistance is obtained as the function of the current stress and effective stress parameters, whereas "undrained" stiffness will result from fluid incompressibility.
- `Driven Load(Undrained)` can be run merely at the end of the analysis.
- The "undrained" conditions apply to the material if ✓ Undrained behavior (consider unchecking for a well-permeable material for which "undrained" behavior is unlikely).
- These two approaches are equivalent assuming perfectly "undrained" conditions. In the case of `Driven Load(Undrained)` driver, the perfectly "undrained" conditions are always preserved. Both perfectly "undrained" conditions and partially drained conditions can be obtained with the `Consolidation` driver depending on the permeability coefficient and duration of actions (suggested for real time simulations).

- *The required minimal time step for the `Consolidation` analysis can be obtained for the assumed coefficient of permeability $k$ from the equation for the critical time step $\Delta t_{\text{crit}}$:

$$\Delta t_{\text{min}} \geq \Delta t_{\text{crit}} = \frac{\gamma^{\text{F}} h^2}{E_{\text{oed}} \theta k} \left( \frac{1}{4} + \frac{1}{6} E_{\text{oed}} c \right) \tag{1}$$

where:

| | | |
|---|---|---|
| $c$ | – | compressibility of fluid $= n/\beta_{\text{F}}$ ; $n$-porosity, $\beta_{\text{F}}$-fluid bulk modulus |
| $k$ | – | coefficient of permeability |
| $\gamma^{\text{F}}$ | – | fluid specific weight |
| $\theta$ | – | integration coefficient (in ZSoil $\theta = 1$) |
| $E_{\text{oed}}$ | $=$ | $\dfrac{E(1-\nu)}{(1+\nu)(1-2\nu)}$ oedometric stiffness modulus |
| $h$ | – | element size |

Window 2-26

Definition of the `Driver`(s)

**INITIAL STATE**

**TIME DEPENDENT**

**STABILITY**

for DEFORMATION COUPLED WITH FLOW are presented in following paragraphs.

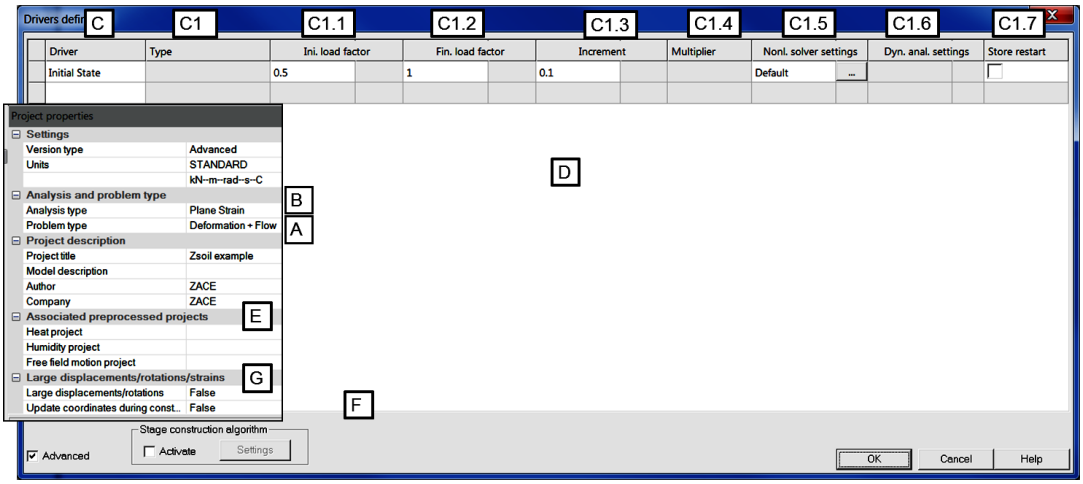
## 2.2.2.1.2.1 INITIAL STATE

**Window 2-27: Initial state**

**A.** `Problem type: Deformation + Flow`
**B.** `Analysis type: any`
**C.** `Driver: Initial State`

For `Problem Type Deformation + Flow`, `Initial State Driver` controls solution of the following problem:

1. `Steady State Flow` problem with pressure BC and fluid fluxes given at time $t = 0$
2. `Initial State` deformation problem taking into account body forces induced by pressure gradients, resulting from `Steady State Flow` solution, and other loads.




Control data of the driver are:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2,6,8,9, 10,11, |
| | | | | 12,13,14,15 |
| **C1.1** | | Initial Load Factor | (0.5) | 4, 7 |
| **C1.2** | | Final Load Factor | (1.0) | 5 |
| **C1.3** | | Load Factor Increment | (0.1) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-27

**Remarks**:

1. Steady State `Flow` problem, with pressure and flux boundary conditions set at time $t = 0$, is solved first
2. During solution of the Steady State `Flow` problem location of the free surface is iteratively adjusted without any guess given by the user
3. Resulting pressure field serves as an initial condition for subsequent `Time Dependent` steady or transient flow analyses

4. This driver controls the amplitude of loads which are non–zero at time $t = 0$ (i.e. the corresponding load time function are non-zero at $t = 0$). User prescribes: 0.5 for the Initial Load Factor; 0.1 for the Load Factor Increment and 1.0 for Final value of the Load Factor; loads will be applied progressively as $0.5g, 0.6g..., 1.0g$. In addition extra body loads resulting from water content and pressure gradients are automatically computed and consistently (by means of incremental approach) treated during solution.
5. An initial state with zero deformation and non–zero stress-state will be generated.
6. Gravity is usually the main load to be considered for the initial state; set unit weights for all materials and gravity vector (in load menu) equal to $(0, -1, < 0 >)$ for downwards vertical gravity.
7. Any `Initial Load Factor` value greater than 0.0 and less or equal to 1.0 is acceptable. The initial value equal to 0.5 is usually used.
8. During an `Initial State` analysis, time is fixed to $t = 0$. As the consequence:
   - only these system components will be taken into account, which exist permanently or have existence status set `ON` at $t = 0$,
   - loads associated with the initial state must have a non-zero load function at $t = 0$; if the load function is equal to zero at $t = 0$, then the load is to be applied later.
   - in case of time dependent material parameters, their values computed at $t = 0$ will be taken into account.
9. An `Initial State` driver can be run only once and must be the first in the list of drivers
10. Omitting `Initial State` and lack of the implicitly set initial effective stresses yields stress free and zero deformation state as the initial condition.
11. Alternatively, if an initial stresses field is known, it can be introduced using `Initial Conditions Initial Stress` super elements in the `Preprocessing` session
12. Each incremental step of mechanical analysis is computed in two runs due to algorithmic reasons.
13. After convergent run of an `Initial State` deformation is nullified. The only nonzero output concerns the stress state and fluid pressure, and velocities
14. In case of the divergence deformation can be visualized in the `Postprocessing`. It helps to identify source of the instability. This may happen due to surface instability caused by zero cohesion, improperly set boundary conditions or highly distorted (with high aspect ratios) finite elements placed close to the free boundary; it may also happen if on submerged surfaces equivalent load is not applied
15. Some soil mechanics problems are characterized by a stress state which remains on the yield surface, i.e., on the limit of instability. It is therefore important to adopt appropriate material data to avoid triggering instability.

## Related Topics

- *THEORETICAL MANUAL*
  - ⋆ *GEOTECHNICAL ASPECT: INITIAL STATE*
  - ⋆ *NUMERICAL IMPLEMENTATION: INITIAL STRESS*
  - ⋆ *NUMERICAL IMPLEMENTATION: INITIAL STATE*

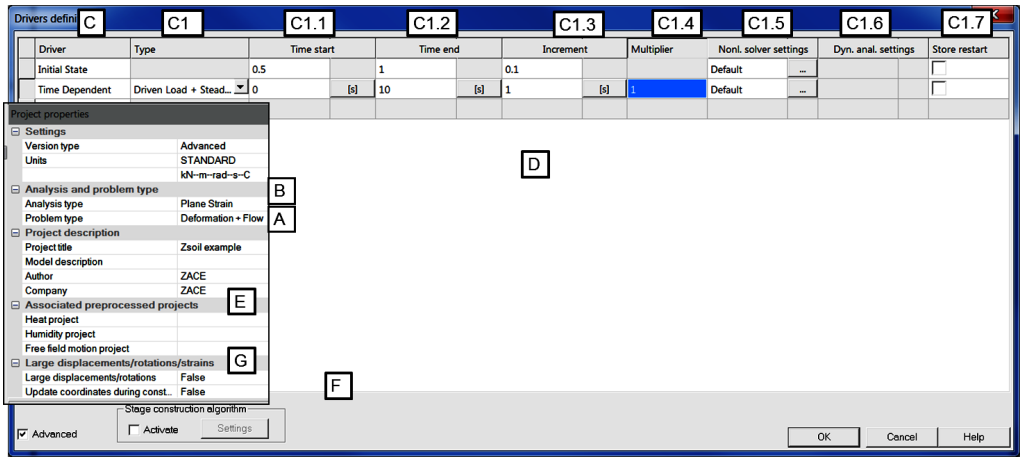
## 2.2.2.1.2.2 TIME DEPENDENT

**Window 2-28: Time Dependent: Driven load+Steady state**

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load+Steady state`

This driver can be used to carry out an uncoupled total stress analysis for a two-phase fully or partially saturated medium. Each time step consists of the two substeps:

- Substep 1: `Steady State Flow` for pressure BC at given time instance $t_{N+1}$
- Substep 2: `Deformation` problem at given time instance $t_{N+1}$ with an explicit pore pressure distribution obtained from Substep 1




Control data for the second driver in list **D** is as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2, 3,4 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (1.0) | |
| **C1.3** | | Initial time step | (1.0) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-28

**Remarks**:

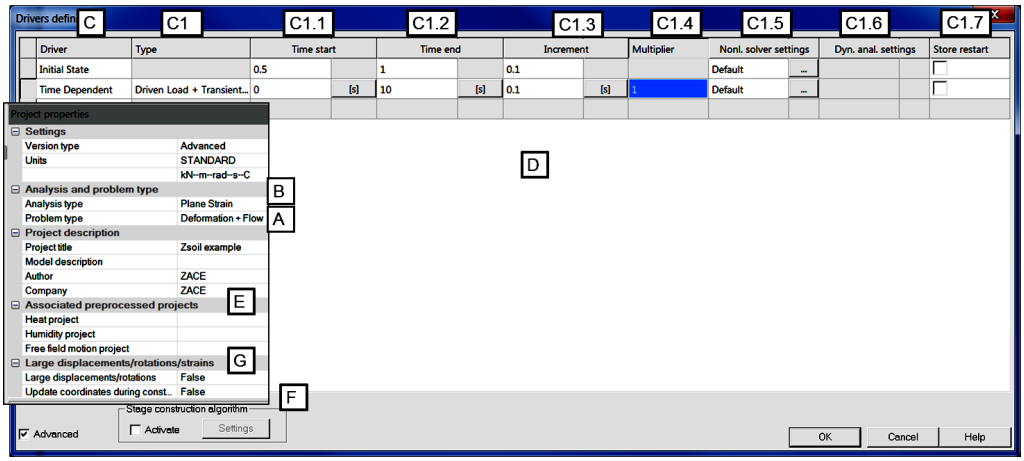
1. Volumetric strain and transient term related to fluid compressibility, and effect of partial saturation, are cancelled from mass balance for the fluid
2. A weak coupling between fluid flow and deformation is due to presence of the fluid pressure weighted by current saturation ratio appearing (Bishop's effective stress principle) in the overall equilibrium of solid-fluid mixture
3. This kind of an approach is essentially limited for soil media with very stiff or incompressible skeletons
4. `Time` is just a sequencing parameter unless creep is activated

## Window 2-29: Time Dependent: Driven load+Transient flow

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Driven Load+Transient flow`

This driver can be used to carry out a weakly coupled total stress analysis for a two-phase fully or partially saturated medium. Each time step consists of the two substeps:

- Substep 1: `Transient Flow` for pressure BC at given time instance $t_{N+1}$
- Substep 2: `Deformation` problem at given time instance $t_{N+1}$ with an explicit pore pressure distribution obtained from Substep 1




Control data for the second driver in list **D** is as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2, 3,4 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (10.0) | |
| **C1.3** | | Initial time step | (0.1) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-29

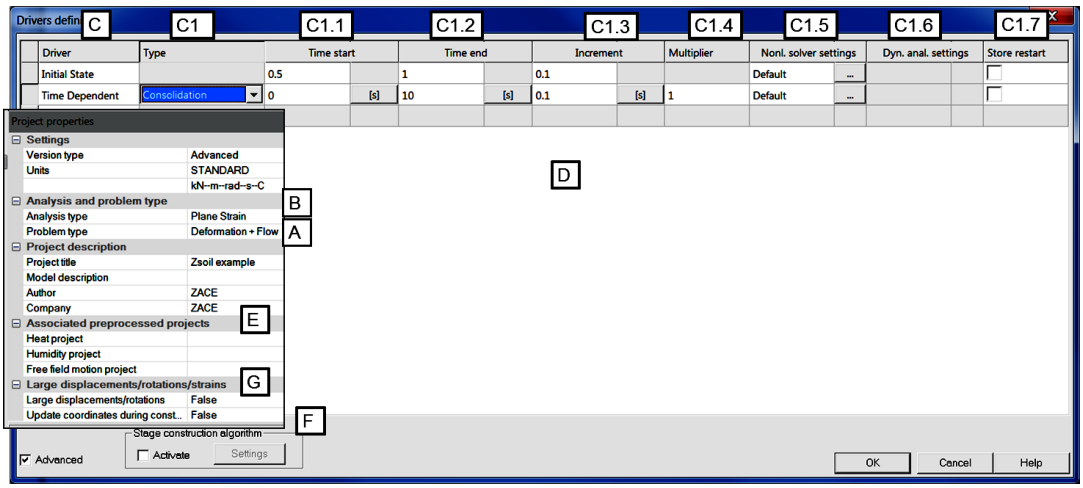
**Remarks**:

1. Volumetric strain is canceled from mass balance for the fluid
2. A weak coupling between fluid flow and deformation is due to presence of the fluid pressure weighted by current saturation ratio appearing (Bishop's effective stress principle) in the overall equilibrium of solid-fluid mixture
3. This kind of an approach is essentially limited for soil media with very stiff or slightly compressible skeletons
4. `Time` is a real time parameter, time stepping procedure is activated

## Window 2-30: Time Dependent: Consolidation

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Consolidation`

This driver is used to carry out a coupled total stress analysis of fully or partially saturated media. Its activation is usually preceded by an appropriate uncoupled analysis (`Initial State` or `Time Dependent-Driven Load`) which gives the initial condition.




Control data of the second driver is as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1, 2, 3, |
| | | | | 4, 5, 6 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (1.0) | |
| **C1.3** | | Initial time step | (1.0) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

**Remarks**:

1. Time integration procedure of coupled system is performed
2. All transient terms in the mass balance for the fluid are taken into account
3. Unit weight of soil in partially saturated zone varies in time due to variation of current saturation ratio; it is automatically computed
4. `Time` is a real parameter
5. `Consolidation` driver is more general than the uncoupled `Driven Load+Transient`; however, applied time steps must be greater than the critical time step value to avoid numerical oscillations in the pressure field[1]:

$$\Delta t \geq \frac{1}{6}\frac{h^2}{\theta C_v} \qquad [\text{s}]$$

   In Z_SOIL, for the mixed displacement–pore pressure formulation the coefficient $1/6$ given by Vermeer is replaced by $1/4$. With:

$$\begin{aligned} C_v &= \frac{E_{\text{oed}} k}{\gamma_F} \qquad [\text{m}^2/\text{s}] \\ E_{\text{oed}} &= \frac{E(1-v)}{(1+v)(1-2v)} \qquad [\text{N}/\text{m}^2] \end{aligned}$$

   where $h$ is maximum element size, $C_v$ is the coefficient of consolidation, $k$ the Darcy coefficient [m/s], $\gamma_F$ the fluid specific weight [N/m$^3$], and $\theta$ is a numerical integration parameter set to 1.0 by default.
6. As an alternative, a stabilized approach can be activated (under `Control-Finite Elements`) eliminating spurious pressure oscillations for any time step

## Related Topics

- *THEORETICAL MANUAL*
  - ⋆ *NUMERICAL IMPLEMENTATION – ULTIMATE LOAD ANALYSIS*
  - ⋆ *NUMERICAL IMPLEMENTATION – LOAD FUNCTIONS AND TIME STEPPING*

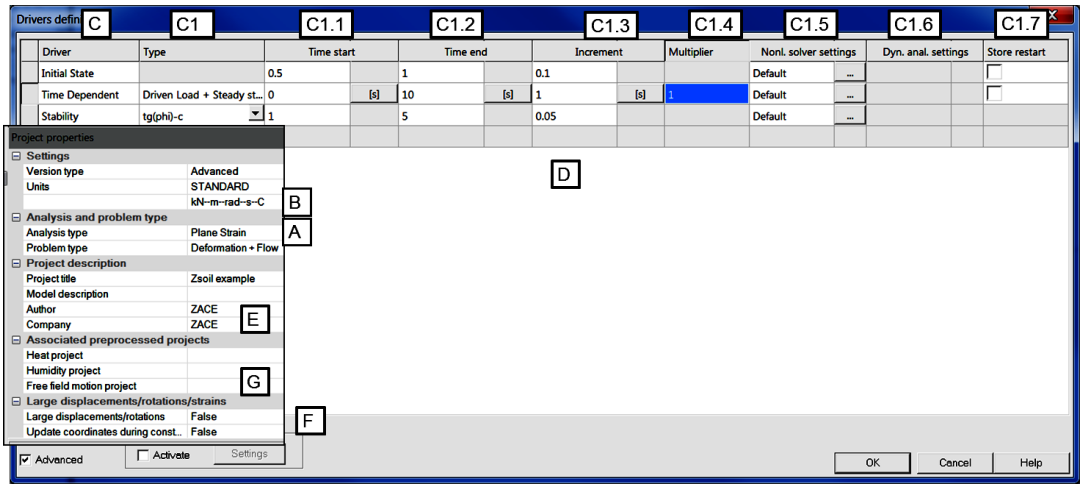
[1] [Ver, 1981]

## 2.2.2.1.2.3 STABILITY

**Window 2-31: Stability**

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type: any`
**C.** `Driver: Stability`
**C1.** `Driver type:` Stress level (see figure below)

This driver can be used to carry out stability analysis. Elastoplastic material models like Mohr–Coulomb, Drucker–Prager, etc. have to be used. The $c - \varphi$ reduction algorithm (pioneered in ZSoil® v.1.), applicable to yield criteria defined through cohesion and friction angle, is used to reduce strength parameters $c$ and $tan(\phi)$, until global instability is detected. Another two reduction algorithms can be choosen: Stress Level reduction (SL) (applicable to material models for which yield surface is defined not by $c$ neither $\phi$) and reduction of Cohesion only. Body forces resulting from the additional water content and pore pressure gradients are taken into account.




Control data for the stability driver is as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2,3,4,5,6 |
| **C1.1** | | Initial SF factor | (1.0) | |
| **C1.2** | | Final SF factor | (2.0) | |
| **C1.3** | | Increment of SF factor | (0.05) | |
| **C1.5** | | Algorithm control data | | |

Window 2-31

**Remarks**:

1. User prescribes Initial value of the safety factor (SF), its final value and increment. In the example shown in Win.2-31 safety analysis will be carried out for SF = 1.0,1.05,1.1 up to 2.0 or SF factor value at which an instability is detected
2. Restart from the last converged step is always possible. During restart user may adjust the final SF value and the increment to obtain solution with assumed accuracy
3. Stability driver can be used at any time, may be declared more than once (also in row) in the list **D** of drivers; it does not affect results of next drivers

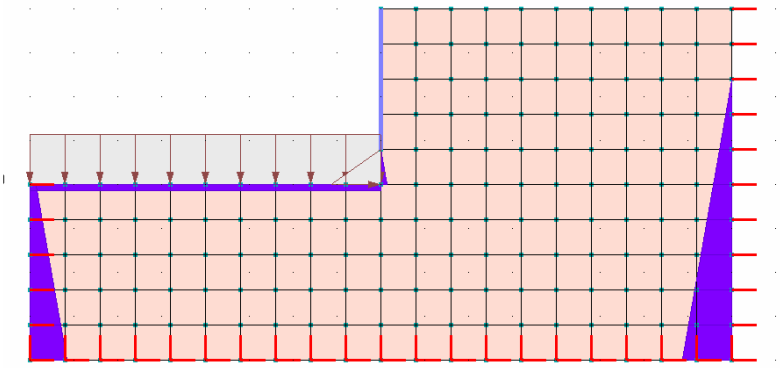
4. Stability driver can also be controlled at the material level to avoid early instabilities caused by flat and tiny sliding
5. Results (i.e. safety factor and failure pattern) are usually sensitive to the choice of `Reduction method`(**C1**) (see Window 2-21)
6. Deformation field visualized by the postprocessor at SF value corresponding to diverged safety analysis step represents a **failure pattern**. If the failure pattern cannot be easily detected it is recommended to visualize incremental deformation between last two safety steps

**Window 2-32: Example of stability analysis with permanent flow**

Data: DATAPREP/CUTFLOW.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Stability`

The same cut as analyzed under option `Deformation` is analyzed with a permanent flow using analysis option `Deformation + Flow`. This type of analysis requires specification of fluid boundary conditions and corresponding loads applied to the solid-fluid mixture, (see Window 2-32):



CUTFLOW – Water pressure boundary conditions, seepage surfaces and equivalent loads on submerged surfaces

**Material data**:

- for the fluid : ($\gamma_F = 10\text{kN/m}^3$, $k = 10e^{-6}\text{m/h}$, $e_o = 1$, $\beta_F = 10^{10}\text{kN/m}^2$, $S_r = 0, \alpha = 2$). $e_o$ and $\beta_F$ are not meaningful for permanent (steady state) flow.
- $\gamma_D$ must be specified for the solid; $\gamma$ will be computed internally as: $\gamma = \gamma_D + nS\gamma_F$,with $n$ (function of $e_o$), the porosity and $S$, current saturation ratio.

**Results**:
The instability occurs at SF=1.80.
**Remark**:
Seepage elements are specified on the vertical wall of the cut as we do not know *apriori* where the free water table intersects with it. These elements automatically switch non–flow flux boundary condition (BC) to the pressure BC below water table

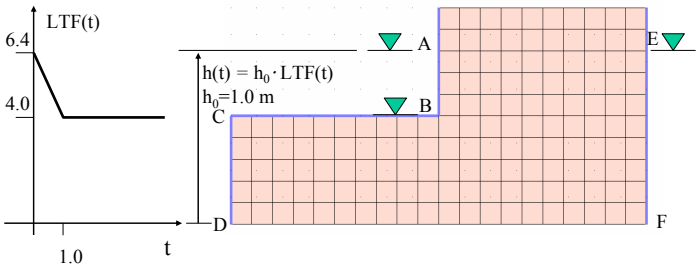
Window 2-32

**Window 2-33: Example of stability analysis with transient seepage boundary conditions**

Data: DATAPREP/CUTTRBC.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Stability`

Stability of the vertical cut can also be analyzed under transient flow conditions. In this test we analyze safety factors at certain time instances due to drawdown of the water table along the boundary A-B-C-D. This example is similar to CUTWT and CUTFLOW. A rapid drawdown of the water table from level A to level B is simulated




CUTTRBC – Transient boundary conditions

The meaningful material data are: $\gamma_F = 10\text{kN/m}^3$, $k = 10e^{-6}\text{m/h}$, $e_o = 1$, $\beta_F = 10^{10}\text{kN/m}^2$, $S_r = 0, \alpha = 2$. Value $\gamma_D$ must be specified for the unit weight of dry soil; ( $\gamma = \gamma_D + nS\gamma_F$) is automatically computed.

Window 2-33

**Remarks**:

- Although pressure boundary conditions are varying in time (transient) solution of the fluid flow is a set of multi-steady states (a real transient behavior can be activated using `Consolidation` or `Driven Load+Transient flow` driver instead of `Driven Load+Steady state flow` one
- Seepage elements on boundary parts AB, BC, CD, EF are geneated; introduction of seepage elements is obligatory for these parts of the boundary where fluid boundary conditions (BC) are defined via `Fluid Head`.
- Fluid boundary conditions are defined via `Fluid Head` and governed by the load time function; actual water elevation, measured from the origin of the global coordinate system in positive $Y$ direction is evaluated as $h(t) = h_o \cdot LTF(t)$; in the dialog box for the `Fluid Head` definition value 1.0 m for the water height $h_o$ is input and its evolution in time is driven by the load time function (recommended way of setting this data); nodal water pressure BC $p = \gamma_F \cdot (h(t) - y_{\text{node}}(t))$ will be generated automatically if node is placed below water elevation (i.e. when $h(t) > y_{\text{node}}$ where $y_{\text{node}}$ is its vertical coordinate); at nodes placed above the water elevation (i.e. $h(t) < y_{\text{node}}$) pressure is set to $p = 0$. The latter case combined with seepage elements will generate impermeable boundary Note, that `Fluid Head` boundary condition is defined separately along boundaries AB,BC,CD (it is varying in time) and boundary EF (here water pressure remains constant)
- On submerged boundaries AB,BC,CD equivalent distributed forces acting on solid-fluid mixture are applied using `Pressure via Fluid Head` option; the same load time function as the one used for the pressure boundary condition is used; surface load $q = \gamma_F \cdot (h(t) - y)$

will be generated automatically at any time instance if the element boundary, to which this load is applied, is submerged

- Load time function for water elevation on AB,BC,CD parts is as follows:

$$t_1 = 0 \qquad h_1 = \texttt{LTF}(t_1) = 6.4$$
$$t_2 = 1 \qquad h_2 = \texttt{LTF}(t_2) = 4.0$$

  On part EF constant LTF should be applied
- Problem type `Deformation+Flow` is mandatory. Drivers `Initial State`, `Driven Load+Steady state flow`, and `Stability` are activated successively with the following parameters:
  - ⋆ `Initial State`(0.5,1;0.1;)
  - ⋆ `Stability`(use SF2) (1.0; 3.0; 0.05; 1.0)
  - ⋆ `Driven Load+Steady state flow`$(0.0, 1.0, 1.0, 1.0)$
  - ⋆ `Stability`(use SF2) (1.0, 3.0, 0.05, 1.0)

## Related Topics

- *STABILITY ANALYSIS*
- *DEFORMATION COUPLED WITH FLOW*

## 2.2.2.1.3 FLOW (STEADY STATE AND TRANSIENT)

**Unknown:** fluid pressure field $p(\mathbf{x})$

For `Flow` problem type necessary data consists of:

- fluid pressure BC (nodal fluid pressures, fluid heads, seepage surfaces),
- fluid fluxes (distributed fluxes)
- seepage parameters and unit weights

The two possible drivers for the fluid flow:

**INITIAL STATE**
**TIME DEPENDENT**

are described in next paragraphs.
**Remarks**:

1. Tracking of the free surface is handled internally
2. For `Time Dependent` driver the two options are available:
   - **STEADY STATE**
     - for solving series of steady state problems (transient term concerning fluid compressibility and effect of partial saturation is canceled from fluid mass balance), evolutionary with respect to time varying pressure boundary conditions, fluxes, domain geometry and material data.
   - **TRANSIENT FLOW**
     - for solving transient flow problem, evolutionary with respect to time varying pressure boundary conditions, fluxes, domain geometry and material data.

### MATERIAL DATA

For both steady and transient flow material data should be specified for all materials models, except those marked as impermeable. Impermeable material zones are excluded from the flow analysis (finite elements in this zone do not have pressure degrees of freedom).

Continuum models (see `Flow` and `Unit Weight` data groups):

| | |
|---|---|
| $\gamma_F$ | $\{[\text{kNm}^{-3}]; [> 0]; [10]\}$<br>fluid specific weight (`Unit Weight`) |
| $k_{()}$ | $\{\text{ms}^{-1}]; [> 0]; [1.0]\}$<br>Darcy coefficients in principal directions of permeability (`Flow`)<br>**Remark:**<br>See `Assembly`→`Materials`→`Data Group`→`Flow` for detailed information on how to set orientation of principal permeability directions |
| $S_r$ | $\{[-]; [0, 1]; [0]\}$<br>residual saturation (`Flow`) |
| $\alpha$ | $\{[\text{m}^{-1}]; [> 0]; [2.0]\}$<br>parameter related to the air entry suction (`Flow`) |
| $n$ | $\{[-]; [> 1]; [2.0]\}$<br>measure of the pore-size distribution (`Flow`) |

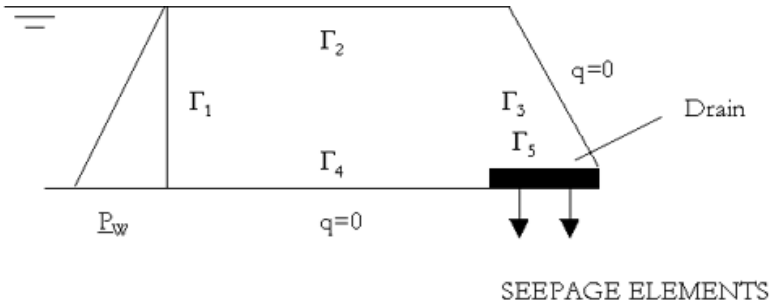
`Transient Flow` requires additional specification of:

| | |
|---|---|
| $e_o$ | $\{[-.]; [0, 10]; [0.0]\}$<br>(initial) void ratio (`Unit Weight`) |
| $\beta_F$ | $\{[\text{kNm}^{-2}]; [> 0]; [10^{+38}]]\}$<br>fluid compressibility (`Flow`) |

NB. Since v.2012 one may cancel the gravity term in Darcy law.

## FLUX AND PRESSURE TYPE BOUNDARY CONDITIONS

| | |
|---|---|
| Fluid pressures | $[\text{N/m}^2]$ |
| Flux | [m/s]<br>selected part of the boundary |
| Drains or seepage surface | Seepage surface element |




Flow boundary conditions

**Remarks**:

1. `Pressure` BC can be prescribed directly on nodes or element edges (faces in 3D) while `Flux` are prescribed on element edges (faces in 3D)
2. Lack of pressure BC at certain part of the boundary means that this boundary is impermeable
3. Seepage elements are directional and must be oriented with the arrow pointing outward, see the Window, otherwise the outflow sign will be changed. Constant or time–dependent pressure boundary conditions must be associated with the Seepage elements; by default, zero outsideexternal pressure is assumed. The element will switch automatically from a pressure boundary condition to no flux boundary condition when no flow through the surface takes place.
4. The impermeable boundary $\Gamma_2$, $\Gamma_3$ could alternatively be modeled with Seepage elements associated with zero external pressure (default value). $\Gamma_1$ could be modeled with Seepage elements (directed outwards), while maintaining the applied pressure. On $\Gamma_4$, $q = 0$, must be prescribed and on $\Gamma_5$, Seepage elements must be present.

## INITIAL CONDITIONS

The initial condition for flow problem corresponds to a prescribed pressure field at all nodal points.

**Remarks**:

1. The `Steady State` driver and the `Initial State` driver will both yield a solution of `Steady State Flow` problem (initial pressure field).

2. For transient analysis, an `Initial State` driver should be run first or, alternatively, an `Initial Pressure` state has to be defined by the user(with aid of initial pressure super-elements)

## Related Topics

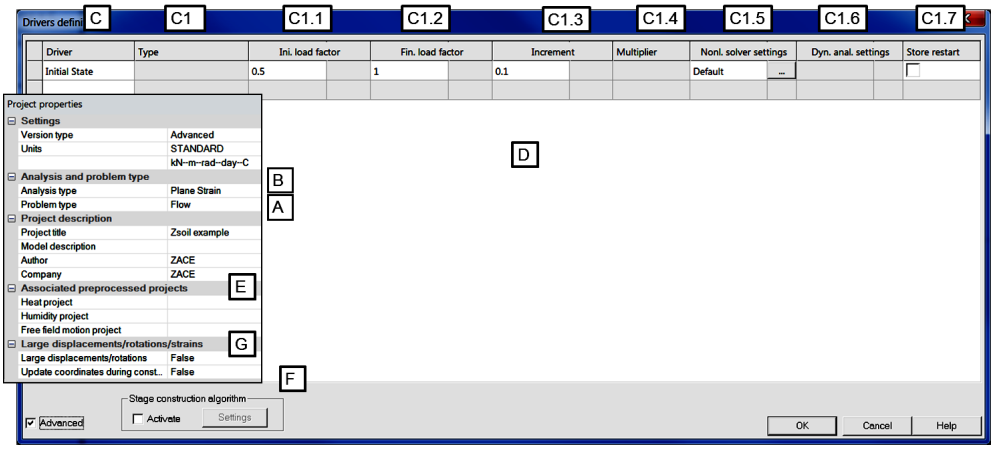
- *DRIVERS AND DRIVERS TYPES*
- *SEEPAGE SURFACE MODEL*

## 2.2.2.1.3.1 INITIAL STATE

**Window 2-34: Initial state**

**A.** `Problem type: Flow`
**B.** `Analysis type: any`
**C.** `Driver: Initial State`

For the `Problem Type Flow`, `initial state Driver` controls solution `Steady State Flow` problem with pressure BC given at the `time`=0.0




`Initial Load Factor`, `Final Load Factor` and `Load Factor Increment` are not meaningful in this case

Window 2-34

**Remarks**:

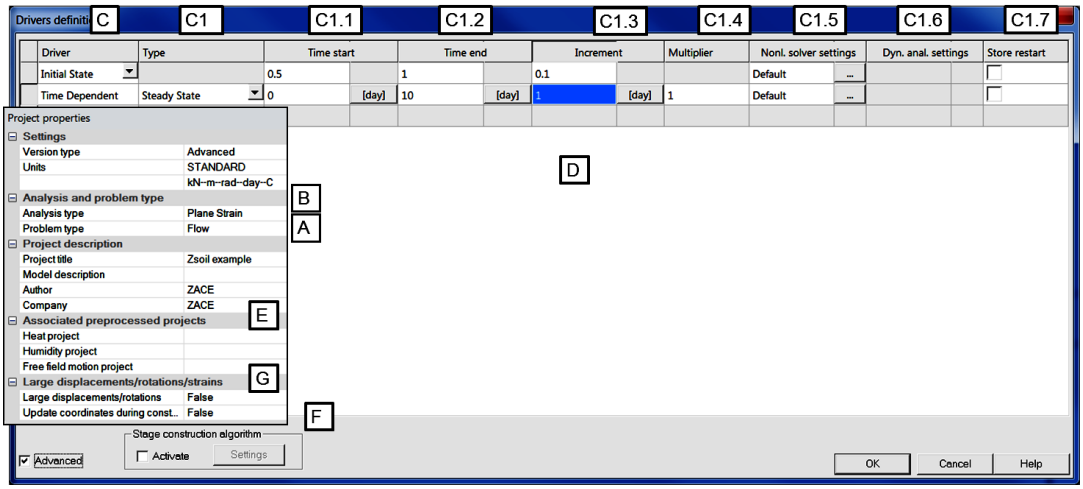
1. Steady State `Flow` problem, with pressure and flux boundary conditions set at time $t = 0$ is solved
2. During the solution an initial location of the free surface is iteratively sought, no user guess is needed.
3. Resulting pressure field serves a reference or an initial condition for subsequent time dependent steady state or transient flow drivers
   During an `Initial State` analysis, time is fixed to $t = 0$. As the consequence:
   - only these system components will be taken into account, which exist permanently or have existence status set `ON` at $t = 0$,
   - loads associated with the initial state must have a non-zero load function at $t = 0$; if the load function is equal to zero at $t = 0$, then the load is to be applied later.
   - in case of time dependent material parameters, their values computed at $t = 0$ will be taken into account.
4. An `Initial State` driver can be run only once and must be the first in the list of drivers; it is usually run to get the initial condition for subsequent transient drivers
5. Alternatively, if an initial pressure field is known apriori, it can be introduced through `Initial Pressure` super-elements during `PrePro` session.
6. If the `Initial state` is skipped and initial pressure field is not defined explicitly then the initial pore pressures are set to zero; this may lead to non-physical results.

## 2.2.2.1.3.2 TIME DEPENDENT

**Window 2-35: Time dependent: Steady state**

**A.** `Problem type: Flow`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Steady State`

This driver is used for solving series of steady state problems (transient term concerning fluid compressibility and effect of partial saturation is cancelled from fluid mass balance), evolutionary with respect to time varying pressure boundary conditions, fluxes, domain geometry and material data.




Control data of the driver are:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2 |
| **C1.1** | | - Initial time value | (0.0) | |
| **C1.2** | | - Final time value | (1.0) | |
| **C1.3** | | - Initial time step | (1.0) | |
| **C1.4** | | - Time step multiplier | (1.0) | |
| **C1.5** | | - Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-35

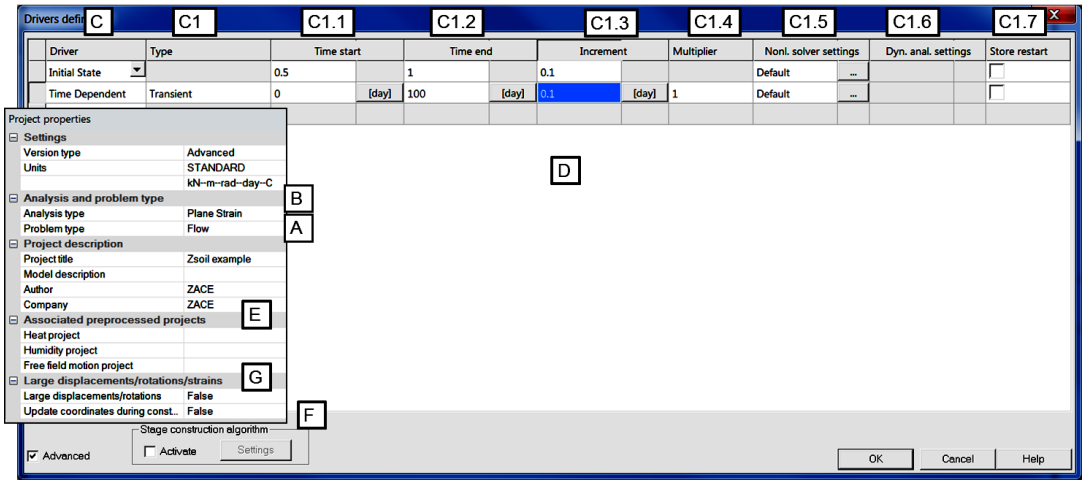
**Remarks**:

1. Iterative solution for detection of a free surface position is performed at each step, no user guess is needed.
2. Time is a fictitious parameter for steady state case

## Window 2-36: Time dependent: Transient

**A.** `Problem type: Flow`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Transient`

This driver is used for solving transient fluid flow problems, evolutionary with respect to time varying pressure boundary conditions, fluxes, domain geometry and material data.




Control parameters for the second driver in list **D** are as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2,3,4 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (1.0) | |
| **C1.3** | | Initial time step | (1.0) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-36

**Remarks**:

1. Time integration procedure is performed starting from the values of initial pressures evaluated during `Initial state` analysis, or alternatively set as `initial pressure` superelements.
2. `Transient`driver requires setting the initial condition for pressure field; lack of `Initial State` driver or user predefined initial pressures will result in zero pressure initial condition; this may lead to non-physical results and divergence of the iterative process
3. A meaningful value must be set for $e_o$ parameter, otherwise all transient effects will be cancelled
4. Time is a real parameter

## 2.2.2.1.4 HEAT TRANSFER

**Unknown:** temperature field $T(\mathbf{x}, t)$

For `Heat` problem type necessary data consists of:

- temperature BC (nodal temperatures), convection BC
- heat fluxes (distributed fluxes)
- heat internal sources (due to hydration process in young concrete)
- heat transfer parameters in material models for continuum (single or two-phase) and heat exchangers
- fluid velocity field taken from the external seepage project to activate advection

The two possible drivers:

**INITIAL STATE**
**TIME DEPENDENT**

are presented in next paragraphs.

**Remarks**:

1. `Heat Transfer` analysis can be carried out to solve:
   - `Heat Transfer` problem treated as *standalone*,
   - `Heat Transfer` problem to preprocess temperature field to be used later on for subsequent `Deformation` analysis. This temperature field yields imposed thermal strains in the medium due to thermal dilatancy; Thermal dilatancy coefficient $\alpha$ has to be set at the material level; `Heat Transfer` and `Deformation` (+`Flow`) analyses **are defined and run as separate tasks**. Then preprocessed temperature field has to be linked to `Deformation` or `Deformation`+Flow problem through →`Associated preprocessed projects` in the `Analysis & Drivers` dialog box ( see `Associated preprocessed projects`).
2. For `Time Dependent` driver can be specialized to `Steady State` or `Transient`
   - **STEADY STATE**
     - is used for solving series of steady state problems (transient term concerning heat capacity and internal source term are cancelled), evolutionary with respect to time varying temperature boundary conditions, fluxes, domain geometry and material data.
   - **TRANSIENT**
     - is used for solving transient heat problems, evolutionary with respect to time varying temperature, convection boundary conditions, fluxes, domain geometry and material data.

### MATERIAL DATA

For continuum elements, any material model may be used and the only meaningful data are set under `Heat` group and contain:

| | |
|---|---|
| solid thermal conductivity $\lambda_s$ | $\{[\text{kN}/(\cdot\,^\circ\text{K}\cdot\text{day})], [>0], [2.4]\}$ (obligatory) |
| solid heat capacity $c_s^*$ | $\{[\text{kN}/\text{m}^2\cdot\,^\circ\text{K})], [>0], [2000]\}$ (obligatory for `Transient` analysis) |
| fluid thermal conductivity $\lambda_f$ | $\{[\text{kN}/(\cdot\,^\circ\text{K}\cdot\text{day})], [>0], [0.6]\}$ (obligatory) |
| fluid heat capacity $c_f^*$ | $\{[\text{kN}/\text{m}^2\cdot\,^\circ\text{K})], [>0], [2000]\}$ (obligatory for `Transient` analysis) |

Fluid thermal parameters are used when the preprocessed seepage project is associated with the thermal one. In that case advection term is added to the heat transfer equation and thermal parameters for the porous medium are computed as weighted average of solid and fluid ones, taking into account current saturation ratio.

Additional data for heat source due to concrete hydration may be defined

## FLUX AND TEMPERATURE BOUNDARY CONDITIONS

- `Temperature` $[^oK], [^oC]$ may be set on nodes.
- `Heat Flux` $[J/(m^2 day)]$ is defined on the boundary.
- `Heat Convection` may be specified on selected part of the boundary by means of `Heat Convection` elements with specialized material model. `Heat Convection` BC data are:
  - ⋆ `Ambient Temperature` $[^oK], [^oC]$ (may be time dependent and controlled by a `Load Time Function`)
  - ⋆ `heat convention coefficient` $h_T [kN/(m \cdot^o K \cdot day)]$

**Remarks**:

1. If no `Temperature`, `Heat Flux`, nor `Heat Convection` is specified on the boundary it is equivalent to setting zero heat flux i.e. adiabatic conditions.
2. Setting `Temperature` at given part of the boundary excludes setting `Heat Flux` or `Heat Convection` and vice versa. `Heat Flux` and `Heat Convection` may coexist.
3. A `Heat Source`, can be optionally defined. In particular, a `Heat Source` due to concrete hydration is available (within `Heat` data of material model)

## INITIAL CONDITIONS

The initial conditions for `Heat Transfer` problem correspond to a prescribed temperature field at all nodal points.

**Remarks**:

1. The `Initial State` driver is equivalent to a `Steady State` problem at time $t = 0$
2. For transient analyses, an `Initial State` analysis should be performed first to serve for the initial condition or, alternatively, initial temperatures can be set with aid of initial temperature super-elements

## Related Topics

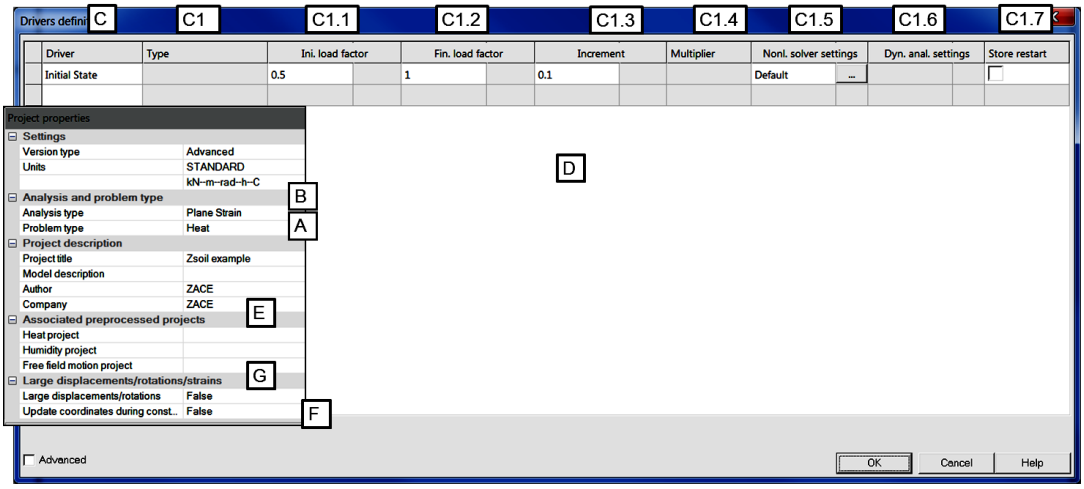
- *Data Preparation:*
  - ⋆ *DRIVERS AND DRIVERS TYPES*
  - ⋆ *ASSOCIATED PREPROCESSED PROJECTS*
  - ⋆ *CONVECTION SURFACE ELEMENTS*
  - ⋆ *HEAT MATERIAL DATA*
  - ⋆ *CONVECTION MODEL*
- *Theory:*
  - ⋆ *PROBLEM STATEMENT*
  - ⋆ *HEAT TRANSFER*

## 2.2.2.1.4.1 INITIAL STATE

**Window 2-37: Initial State**

**A.** `Problem type: Heat`
**B.** `Analysis type: any`
**C.** `Driver: Initial State`

The `Initial State Driver` solves steady state problem for temperature BC set at time $t = 0$.




Driver parameters are not meaningful (solution is obtained in one step).

Window 2-37

**Remarks**:

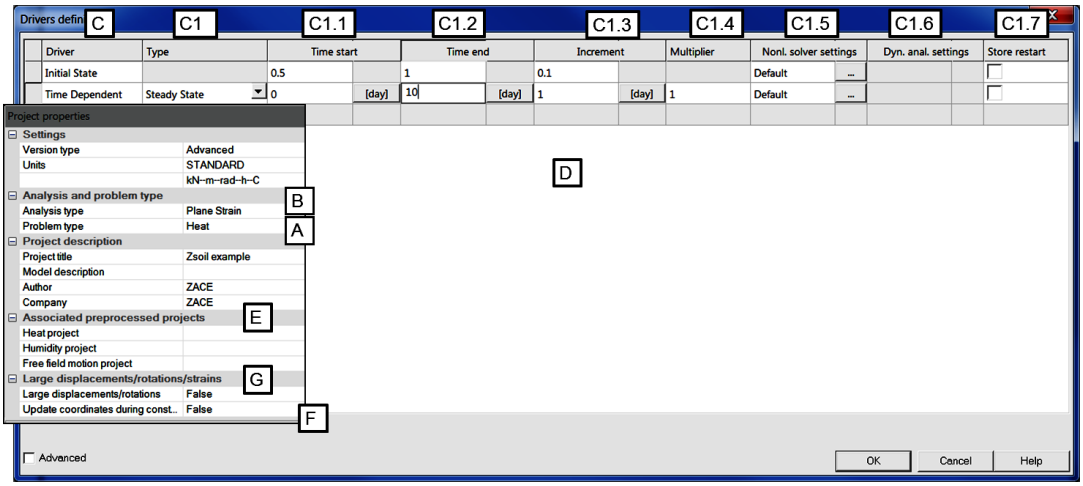
1. Steady State `Heat` problem, with temperature, convection and flux boundary conditions set at time $t = 0$ is solved
2. Resulting temperature field serves a reference or an initial condition for subsequent time dependent steady state or transient heat transfer drivers
   During an `Initial State` analysis, time is fixed to $t = 0$. As the consequence:
   - only these system components will be taken into account, which exist permanently or have existence status set `ON` at $t = 0$, this is determined by the existence functions (=1)
   - fluxes associated with the initial state must have a non-zero load function at $t = 0$; if the load function is equal to zero at $t = 0$, then the flux is to be applied later.
   - in case of time dependent material parameters, their values computed at $t = 0$ will be taken into account.
3. An `Initial State` driver can be run only once and must be the first in the list of drivers; it is usually run to get the initial condition for subsequent transient drivers
4. Alternatively, if an initial temperature field is known apriori, it can be introduced through `Initial Temperature` super-elements during `Preprocessing` session.
5. If the `Initial State` is skipped and initial temperature field is not defined explicitly then the initial temperatures are set to zero; this may lead to non-physical results.

## 2.2.2.1.4.2 TIME DEPENDENT

**Window 2-38: Time dependent: Steady state**

**A.** `Problem type: Heat`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Steady State`

This driver is used for solving series of steady state problems (transient term concerning heat capacity and internal source term are cancelled ), evolutionary with respect to time varying temperature or convection boundary conditions, fluxes, domain geometry and material data.




Control data of the driver are:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2,3 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (1.0) | |
| **C1.3** | | Initial time step | (1.0) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-38

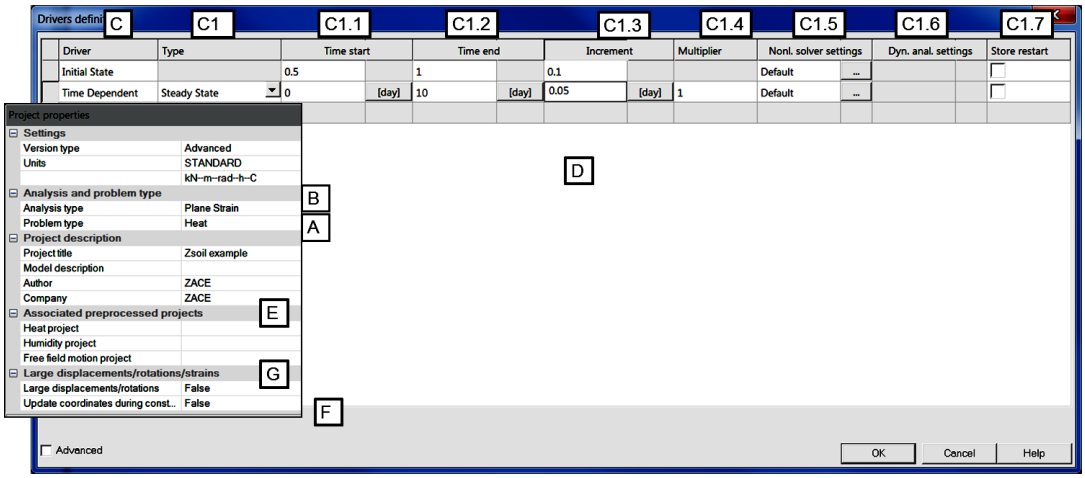
**Remarks**:

1. Heat source term is not accessible for `Steady State` case
2. Time is a fictitious parameter
3. If `Initial State` is skipped and initial temperature field is not defined explicitly then initial temperatures are set to zero

## Window 2-39: Time Dependent: Transient

**A.** `Problem type: Heat`
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Transient`

This driver is used for solving transient thermal problems for varying in time, temperature, convection and heat flux boundary conditions, and internal heat sources.




Control parameters for the second driver are as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2,3,4 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (1.0) | |
| **C1.3** | | Initial time step | (1.0) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-39

**Remarks**:

1. Time integration procedure is performed starting from the values of initial temperatures evaluated during `Initial State` analysis, or alternatively set as `Initial Temperature` with aid of super-elements.
2. `Transient` case must be selected to handle internal heat source term
3. If `Initial State` is skipped and initial temperature field is not defined explicitly then at the first time step zero temperatures will be taken as initial ones. This may lead to non-physical results
4. Time is a real parameter

## 2.2.2.1.5 HUMIDITY TRANSFER

**Unknowns**[2]**:** relative humidity field i.e. moisture potential, $W(\mathbf{x}, t), 0 < W < 1$

For `Humidity` transfer problem type necessary data consists of:
- humidity BC (nodal humidities), convection BC
- humidity fluxes (distributed fluxes)
- humidity transfer parameters in material models

The two possible drivers:
**INITIAL STATE**
**TIME DEPENDENT**
are presented in next paragraphs.

**Remarks**:
1. `Humidity Transfer` analysis can be carried out to solve:
   - `Humidity Transfer` problem treated as *standalone*,
   - `Humidity Transfer` problem to preprocess humidity field to be used later on for subsequent `Deformation` analysis. This humidity field yields imposed hygral strains in the medium due to hygral dilatancy; Hygral dilatancy coefficient $\beta$ has to be set at the material level; `Humidity Transfer` and `Deformation` (+`Flow`) analyses **are defined and run as separate tasks**. Then preprocessed humidity field has to be linked to `Deformation` or `Deformation`+Flow problem through →`Associated preprocessed projects` in the `Analysis & Drivers` dialog box ( see `Associated preprocessed projects`).
2. For `Time Dependent` driver can be specialized to `Steady State` or `Transient`
   - **STEADY STATE**
     - is used for solving series of steady state problems , evolutionary with respect to time varying humidity boundary conditions, fluxes, domain geometry and material data.
   - **TRANSIENT**
     - is used for solving transient humidity transfer problems, evolutionary with respect to time varying humidity, convection boundary conditions, fluxes, domain geometry and material data.

### MATERIAL DATA

For continuum elements, any material model may be used and the only meaningful data are set under `Humidity` group and contain:

| | |
|---|---|
| $a$ | $\{[-], [> 0], [0.05]\}$ |
| $D_1$ | $\{[\mathrm{m}^2/\mathrm{day}], [> 0], [10^{-7}]\}$ |
| $W_1$ | $\{[-], [0 < W_1 < 1], [0.75]]\}$ |

**Remark**:

The data are used to relate diffusion conductivity $D$ with respect to the moisture potential $W$ as a function:

$$D = D(W) = D_1 \left( a + \frac{1-a}{1 + \left( \frac{1-W}{1-W_1} \right)^4} \right)$$

[2]concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT**

### FLUX AND HUMIDITY BOUNDARY CONDITIONS

- `Humidity` (relative) $[-]$ may be set on nodes.
- `Humidity Flux` $[1/(m^2 \cdot day)]$ through part of the boundary.
- `Humidity Convection` may be specified on selected part of the boundary by means of `Humidity Convection` elements with specialized material model. `Humidity Convection` BC data are:
  - ⋆ `Ambient humidity` $[-]$ (may be time dependent, controlled by `Load Time Function`)
  - ⋆ `humidity convention coefficient` $h_W[1/(m^2 \cdot day)]$

**Remarks**:

1. As primary unknown i.e. relative humidity $W$ is, by its nature, delimited by: $0 \leq W \leq 1$ thus the same limitations **must** be fulfilled by relative humidity values set through boundary or initial conditions
2. If no `Humidity`, `Humidity Flux`, nor `Humidity Convection` is specified on the boundary it is equivalent to setting zero humidity flux i.e. perfect isolation conditions.
3. Setting `Humidity` at given part of the boundary excludes setting `Humidity Flux` or `Humidity Convection` and vice versa. `Humidity Flux` and `Humidity Convection` may coexist.

### INITIAL CONDITIONS

The initial conditions for `Humidity Transfer` problem correspond to a prescribed humidity field at all nodal points.

**Remarks**:

1. The `Initial State` driver is equivalent to a `Steady State` problem at time $t = 0$
2. For transient analyses, an `Initial State` analysis should be performed first to serve for the initial condition or, alternatively, initial humidities can be set with aid of initial humidity super-elements

## Related Topics

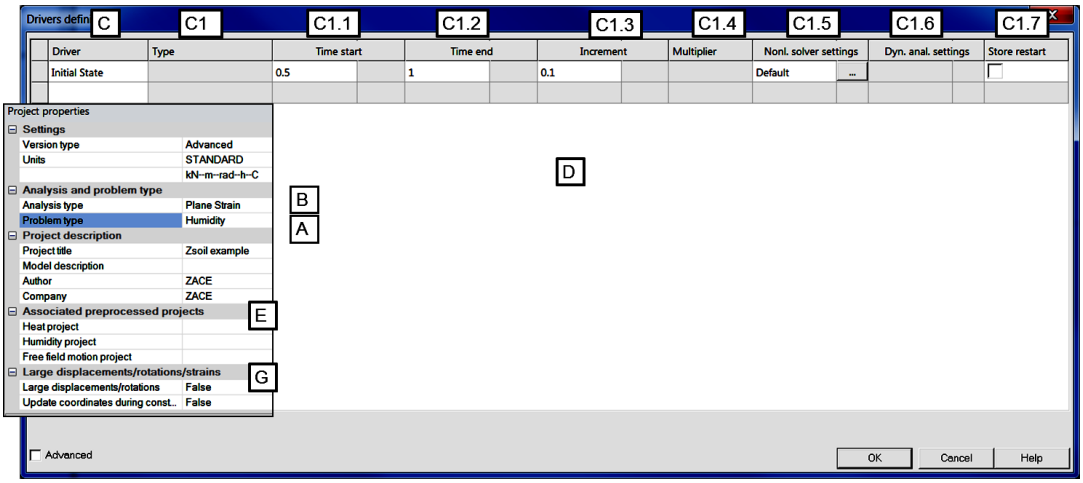
- *Data Preparation:*
  - ⋆ *DRIVERS AND DRIVERS TYPES*
  - ⋆ *ASSOCIATED PROJECTS*
  - ⋆ *CONVECTION SURFACE ELEMENTS*
  - ⋆ *HUMIDITY MATERIAL DATA*
  - ⋆ *CONVECTION MODEL*
- *Theory:*
  - ⋆ *PROBLEM STATEMENT*
  - ⋆ *HUMIDITY TRANSFER*

## 2.2.2.1.5.1 INITIAL STATE

**Window 2-40: Initial state**

**A.** `Problem type: Humidity`[3]
**B.** `Analysis type: any`
**C.** `Driver: Initial State`

The `Initial State Driver` solves steady state problem for humidity BC set at time $t = 0$.




Driver parameters are not meaningful (solution is obtained in one step).

Window 2-40

**Remarks**:

1. Steady State `Humidity` problem, with humidity, convection and flux boundary conditions set at time $t = 0$ is solved
2. Resulting humidity field serves a reference or an initial condition for subsequent time dependent steady state or transient humidity transfer drivers
3. During an `Initial State` analysis, time is fixed to $t = 0$. As the consequence:
   - only these system components will be taken into account, which exist permanently or have existence status set `ON` at $t = 0$,
   - fluxes associated with the initial state must have a non-zero load function at $t = 0$; if the load function is equal to zero at $t = 0$, then the flux is to be applied later.
   - in case of time dependent material parameters, their values computed at $t = 0$ will be taken into account.
4. An `Initial State` driver can be run only once and must be the first in the list of drivers; it is usually run to get the initial condition for subsequent transient drivers
5. Alternatively, if an initial humidity field is known apriori, it can be introduced through `Initial Humidity` super-elements during `Preprocessing` session.
6. If the `Initial State` is skipped and initial humidity field is not defined explicitly then the initial humidities are set to zero; this may lead to non-physical results.

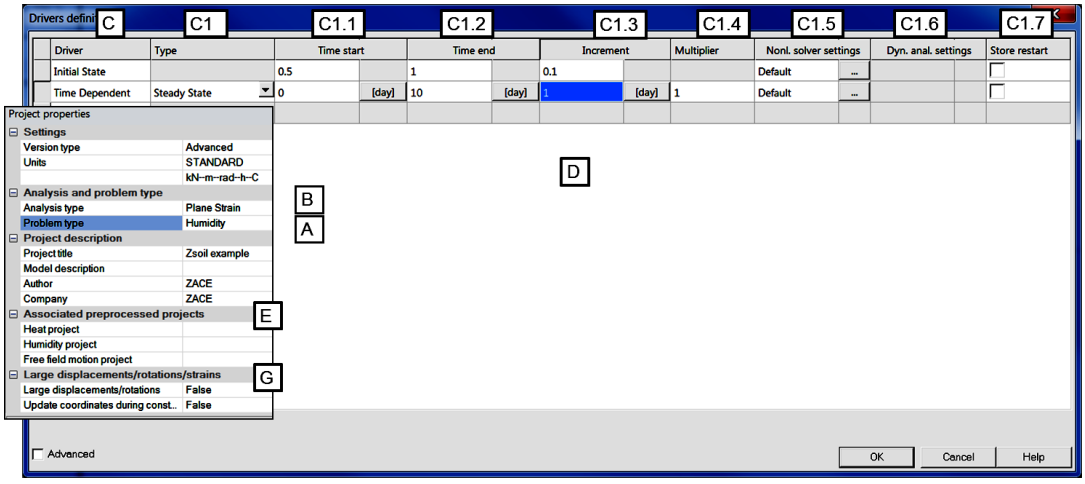
[3] concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT**

## 2.2.2.1.5.2 TIME DEPENDENT

**Window 2-41: Time Dependent: Steady state**

**A.** `Problem type: Humidity`[4]
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Steady State`

This driver is used for solving series of steady state problems evolutionary with respect to time varying temperature or convection boundary conditions, fluxes, domain geometry and material data.




Control parameters for the second driver are as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| | | | | |
| **C1** | | | | 1,2 |
| **C1.1** | | Initial time value | (0.0) | |
| **C1.2** | | Final time value | (10.0) | |
| **C1.3** | | Initial time step | (1.0) | |
| **C1.4** | | Time step multiplier | (1.0) | |
| **C1.5** | | Algorithm control data | | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-41

**Remarks**:

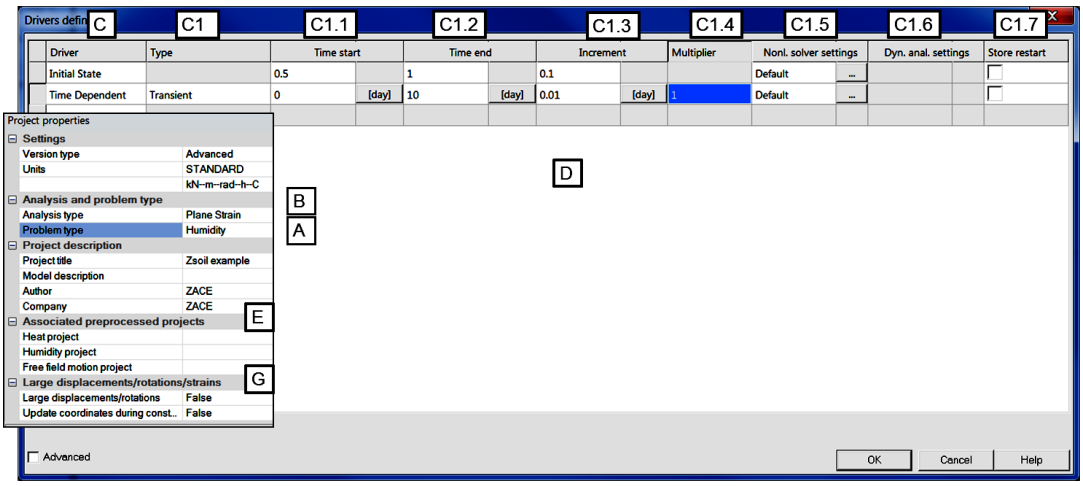
1. Time is a fictitious parameter
2. If `Initial State` is skipped and initial humidity field is not defined explicitly then initial humidities are set to zero

[4] concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT**

## Window 2-42: Time Dependent: Transient

**A.** `Problem type: Humidity`[5]
**B.** `Analysis type: any`
**C.** `Driver: Time Dependent`
**C1.** `Driver type: Transient`

This driver is used for solving transient humidity transfer problems for varying in time, humidity, convection and humidity flux boundary conditions.




Control parameters for the second driver are as follows:

| *Key* | | *Option* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,2,3 |
| **C1.1** | | - Initial time value | (0.0) | |
| **C1.2** | | - Final time value | (1000.0) | |
| **C1.3** | | - Initial time step | (1.0) | |
| **C1.4** | | - Time step multiplier | (1.05) | |
| **C1.7** | | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-42

**Remarks**:

1. Time integration procedure is performed starting from the values of initial humidity evaluated during `Initial State` analysis, or alternatively set as `Initial Humidity` with aid of super-elements.
2. If `Initial State` is skipped and initial humidity field is not defined explicitly then at the first time step zero humidities will be taken as initial ones. This may lead to non-physical results
3. Time is a real parameter

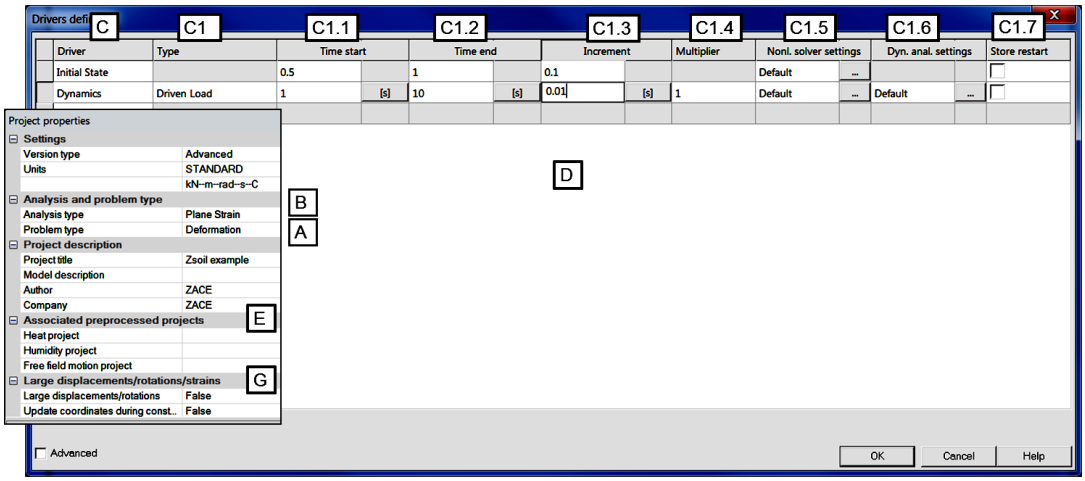
[5] concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT**

## 2.2.2.1.6 TRANSIENT DYNAMICS

**Window 2-43: Dynamics for single-phase soil-structure interaction problems**

**A.** `Problem type: Deformation`
**B.** `Analysis type: any`
**C.** `Driver: Dynamics`
**C1.** `Driver type: Driven load`

This driver can be used for solving a transient dynamic soil-structure interaction problems for single-phase media. The domain reduction method (DRM) can be used to reduce size of the computational model by setting DRM boundary layer of elements in the preprocessor and declaring the free field motion project in the edit field under `Associated preprocessed projects`




Control data of the driver are:

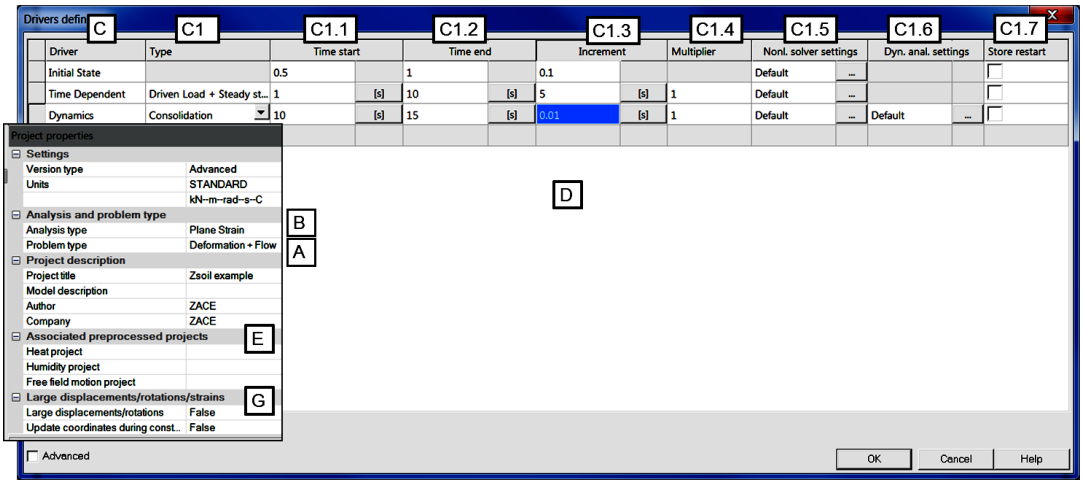
| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,,3,5,6 |
| **C1.1** | `Time start` | Initial time value | (0.0) | 2,3 |
| **C1.2** | `Time end` | Final time value | (10.0) | 2,3 |
| **C1.3** | `Time increment` | Initial time step | (0.01) | 3, 8 |
| **C1.4** | `Multiplier` | Time step multiplier | (1.0) | |
| **C1.5** | `Nonl. Solver` | Control data | | |
| **C1.6** | `Dyn. Solver` | Control data | | |
| **C1.7** | `Store restart` | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-43

## Window 2-44: Dynamics for two-phase soil-structure interaction problems

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type:Plane Strain, 3D`
**C.** `Driver: Dynamics`
**C1.** `Driver type: Consolidation`

This driver can be used for solving a coupled transient dynamic soil-structure interaction problems for two-phase media. In the current version the domain reduction method (DRM) for the two-phase problems is supported.




Control data of the driver are:

| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,,3,5,6 |
| **C1.1** | `Time start` | Initial time value | (10.0) | 2,3 |
| **C1.2** | `Time end` | Final time value | (15.0) | 2,3 |
| **C1.3** | `Time increment` | Initial time step | (0.01) | 3, 8 |
| **C1.4** | `Multiplier` | Time step multiplier | (1.0) | |
| **C1.5** | `Nonl. Solver` | Control data | | |
| **C1.6** | `Dyn. Solver` | Control data | | |
| **C1.7** | `Store restart` | Store restart files in separate subdirectory at end of the driver | OFF | |

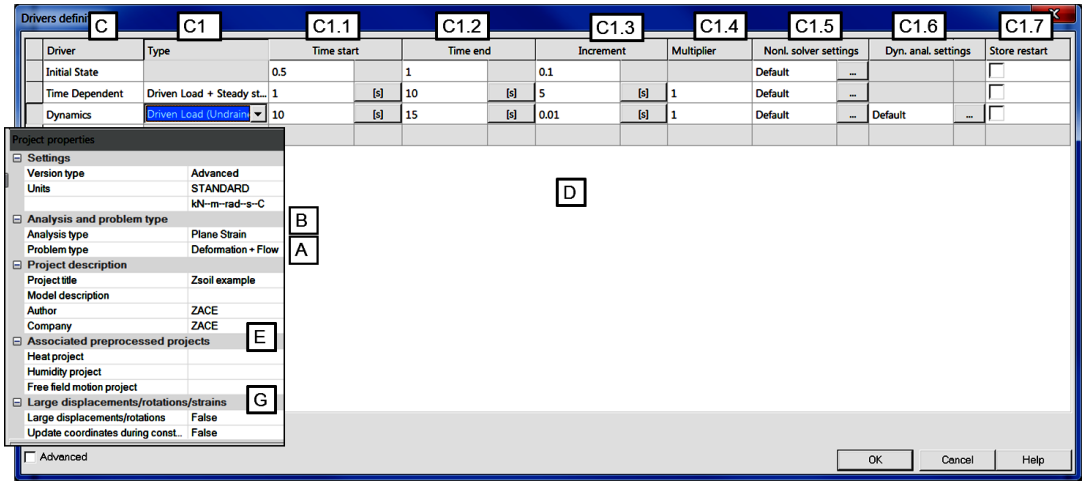
Window 2-44

## Window 2-45: Dynamics for undrained two-phase soil-structure interaction problems

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type:Plane Strain, 3D`
**C.** `Driver: Dynamics`
**C1.** `Driver type: Driven Load(undrained)`

This driver can be used for solving a coupled transient dynamic soil-structure interaction problems for two-phase media assuming that relative motion of fluid particles with respect

to the solid skeleton can be neglected. In the current version the domain reduction method (DRM) for this option is supported. Using this option the excess pore pressure is computed only at the element center using penalty method. The penalty factor is set up assuming the ratio between fluid bulk modulus (in undrained state) and elastic bulk modulus of the skeleton $\frac{K^F}{K} = 10^6$ (as default). In adition user may put a limit value for the total suction pore pressure. The above data can be set in group `Flow`. It has to be emphasized that the total pore pressure can only be checked in the element as it consists of the pressure field (kept an nodes) resulting from the previous non-undrained drivers and undrained excess pressure that is computed at the element center.




Control data of the driver are:

| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 1,,3,5,6 |
| **C1.1** | `Time start` | Initial time value | (10.0) | 2,3 |
| **C1.2** | `Time end` | Final time value | (15.0) | 2,3 |
| **C1.3** | `Time increment` | Initial time step | (0.01) | 3, 8 |
| **C1.4** | `Multiplier` | Time step multiplier | (1.0) | |
| **C1.5** | `Nonl. Solver` | Control data | | |
| **C1.6** | `Dyn. Solver` | Control data | | |
| **C1.7** | `Store restart` | Store restart files in separate subdirectory at end of the driver | OFF | |

Window 2-45

**Remarks**:

1. `Driver type`: `Driven Load` is the only available driver for single-phase media, for two-phase both `Consolidation` and `Driven Load(undrained)` are available (the later one does not seem to be menaingful)
2. Time is treated as a physical parameter (consistently with material data)
3. Time unit expressed in seconds ([s]) must be used
4. Time parameter is used as a common argument of all existence and load time functions.
5. A sequence of `Dynamic Drivers` can be defined, with different time step specification.
6. Within a sequence of `Dynamic`, `Driven Load Drivers`, an intermediate `Time Dependent` (i.e. concerning statics) or `Stability Driver` can be added.

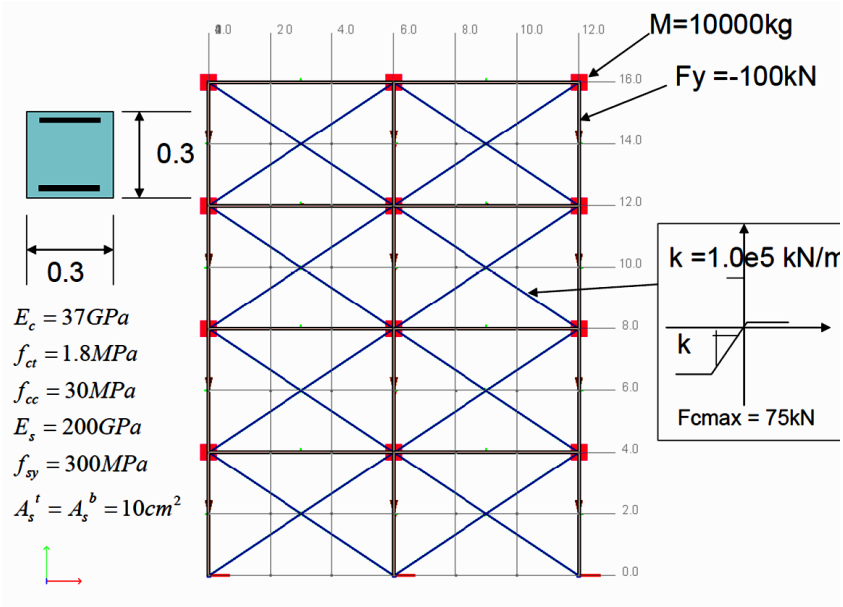
7. In the sequence of drivers the state of the structure at the start of subsequent driver takes into account results (e.g. plastic status, stresses, deformation) of previous dynamic driver(s).
8. Additional time steps will be automatically enforced by load and existence time functions, at all their characteristic points
9. There are two possible methods of imposing seismic input to the structure:
   A: by setting time dependent displacement /velocity / acceleration at fixed node(s) and solving for total motion
   B: as an imposed global acceleration $\mathbf{a}_g = \ddot{\mathbf{u}}_g$ of the ground, common to the whole structure, and solving for relative motion (seismic input option)
   C: Detailed description of the DRM method is given in a dedicated report on transient dynamics

**Window 2-46:** `Dynamics`. **Frame under seismic excitation – problem statement**

Data: BENCHMARKS/ InFillFrame.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C.** `Driver: Dynamic`
**C1.** `Driver type: Driven Load`

In-filed frame of the three-storey building. Cross-bracing (elasto-plastic truss) represents stiffness and limited strength of the masonry infill.




**Goals:** To trace time histories of:

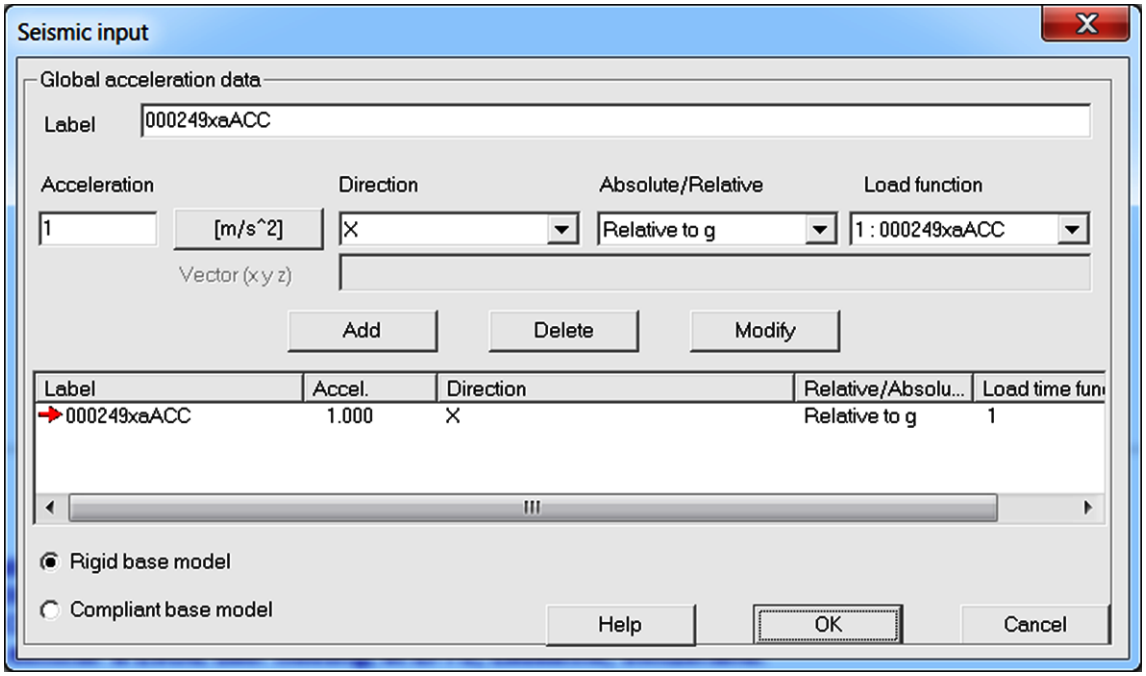
- deformation,
- internal forces

of the frame submitted to seismic action. Seismic load consist of global acceleration record from a real earthquake event taken from public database (e.g. European Strong-Motion Databank http://www.isesd.cv.ic.ac.uk/ESD/frameset.htm)

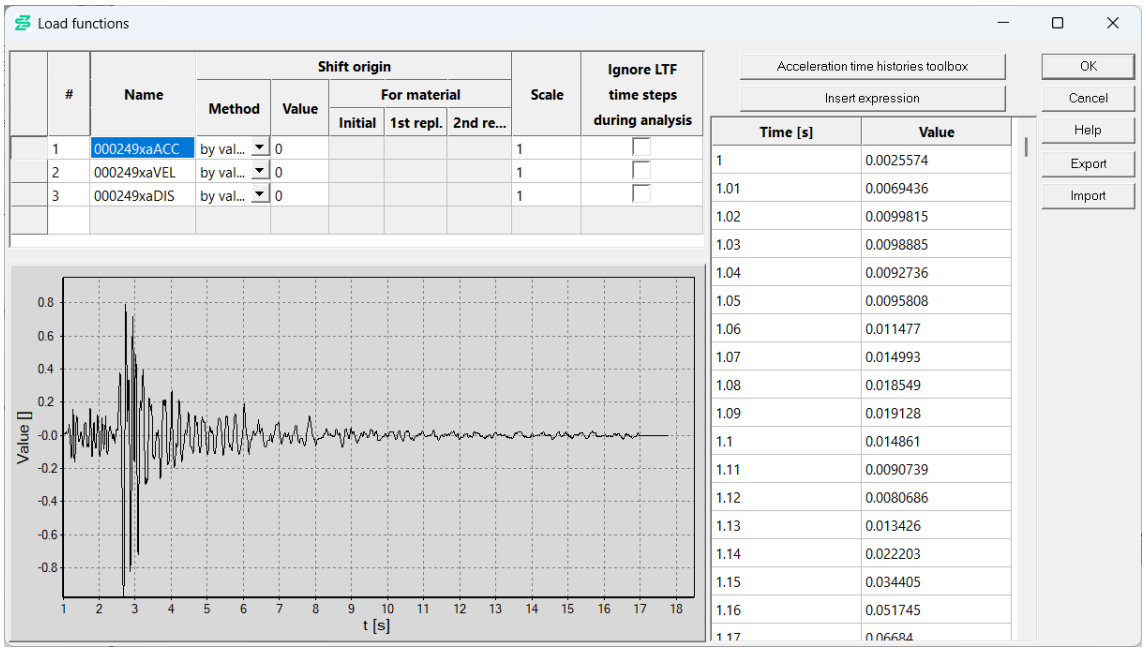
Window 2-46

**Window 2-47:** `Dynamics`**. Frame under seismic excitation – input**

Setting global acceleration as seismic input (use `Menu` / `Assembly` / `Seismic input`)

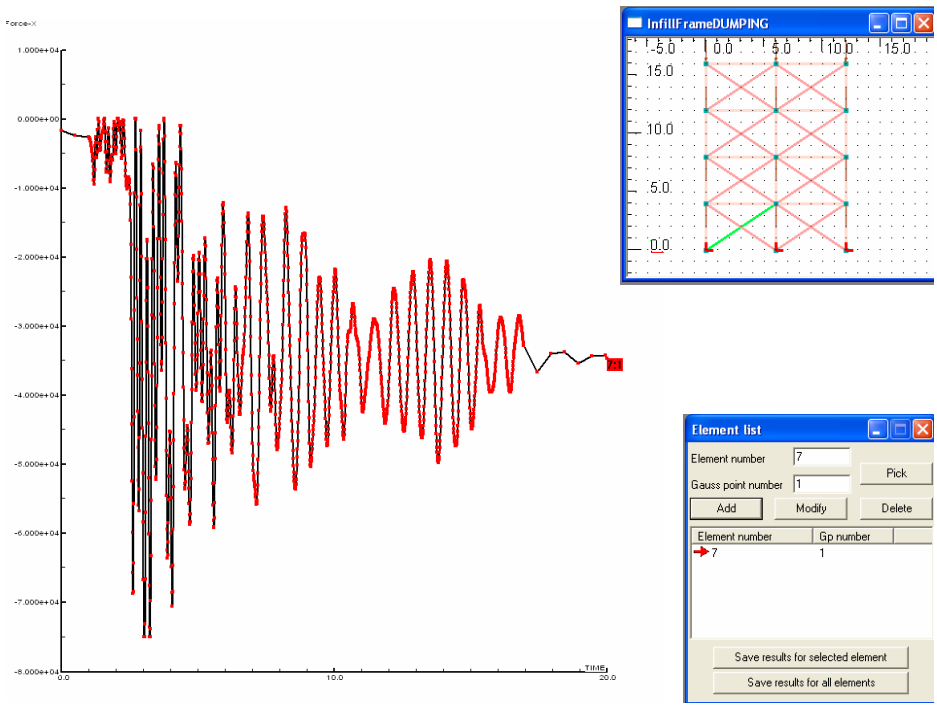



Load time function for acceleration record

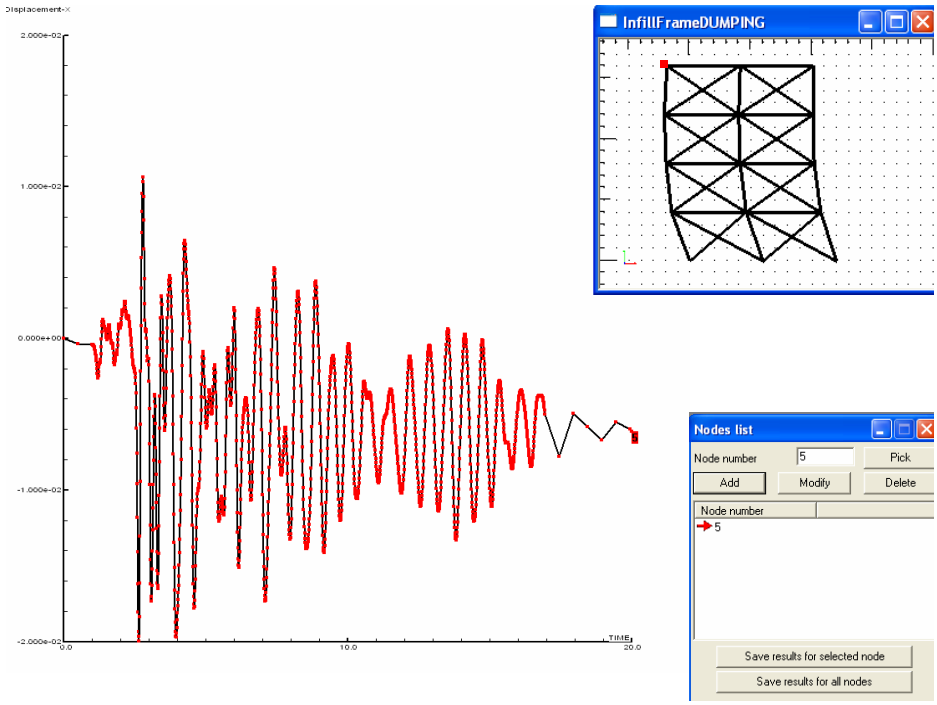


**Window 2-48:** `Dynamics`. **Frame under seismic excitation – results**

History of force in the bottom shear-wall




Top displacement history (relative to the rigid base)




Window 2-48

## Related Topics

*Data preparation*

- DYNAMICS
- SEISMIC INPUT

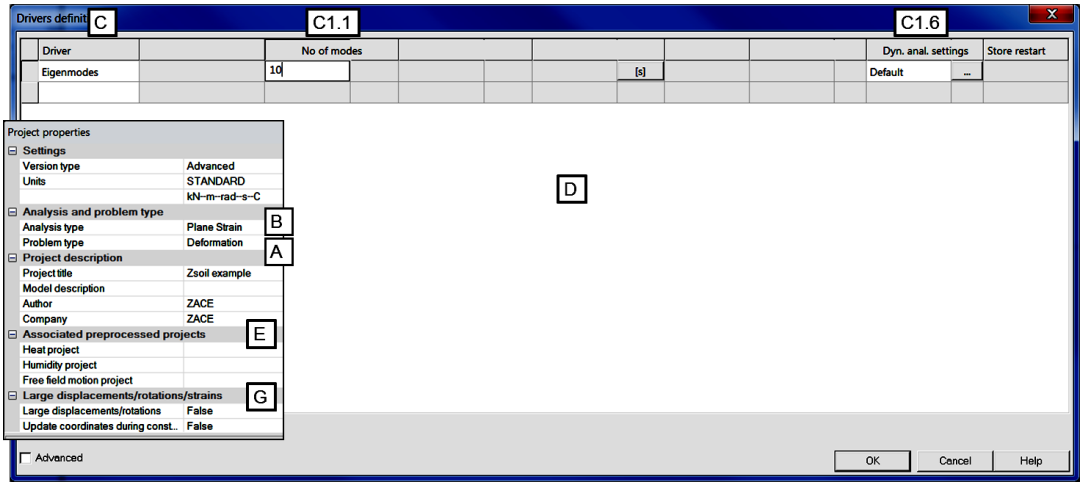
‘

## 2.2.2.1.7 EIGENVALUES AND EIGENMODES

**Window 2-49: Eigenvalues and eigenmodes**

**A.** `Problem type: Deformation`
**B.** `Analysis type: any`
**C.** `Driver: Eigenvalues`
**C1.** `Driver type:`

In the `Deformation` mode this driver can be used for detection of declared number of eigenvalues and corresponding eigenmodes for single-phase media including structural elements.




Control data of the driver are:

| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1.1** | `No of modes` | | | 1, 2 |
| **C1.6** | `Dyn. Solver` | Control data | | |

**Remarks**:

1. Number of detected eigenvalues is limited by total number of free degrees of freedom
2. For each eigen-mode mass participation factors are computed for all directions

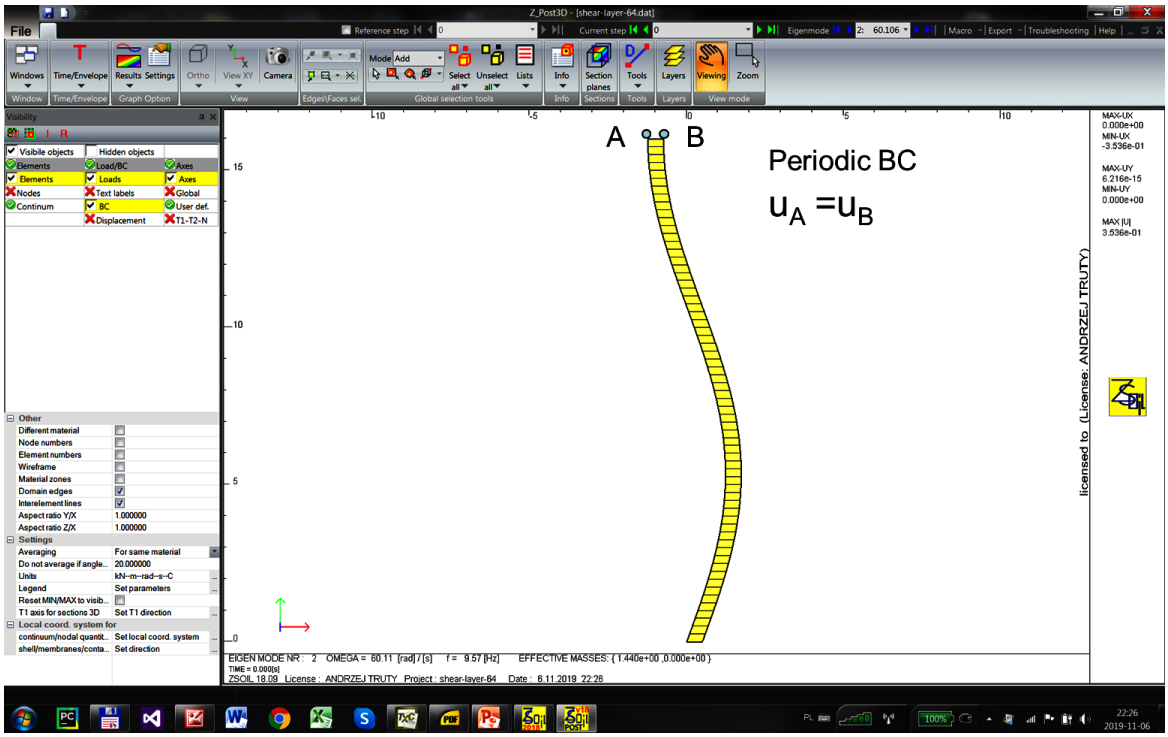
Window 2-49

## Window 2-50: Eigenvalues and eigenmodes: Vibration modes for shear layer

Data: BENCHMARKS/DYNAMICS/EIGENMODES/shear-layer-64.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: Plane Strain`
**C1.** `Driver type: Eigenvalues`

This example concerns eigenvalue analysis of shear layer. To analyze such a problem one can set periodic boundary conditions that helps to tie selected degrees of freedom on two opposite vertical boundaries. In the considered case we assume that $\mathbf{u_A} = \mathbf{u_B}$. Computed eigenvalues, eigenmodes and mass participation factors are visualized in the postprocessor. They can also be exported to the Excel file from main ZSoil menu using option `Results/Eigenvectors`.

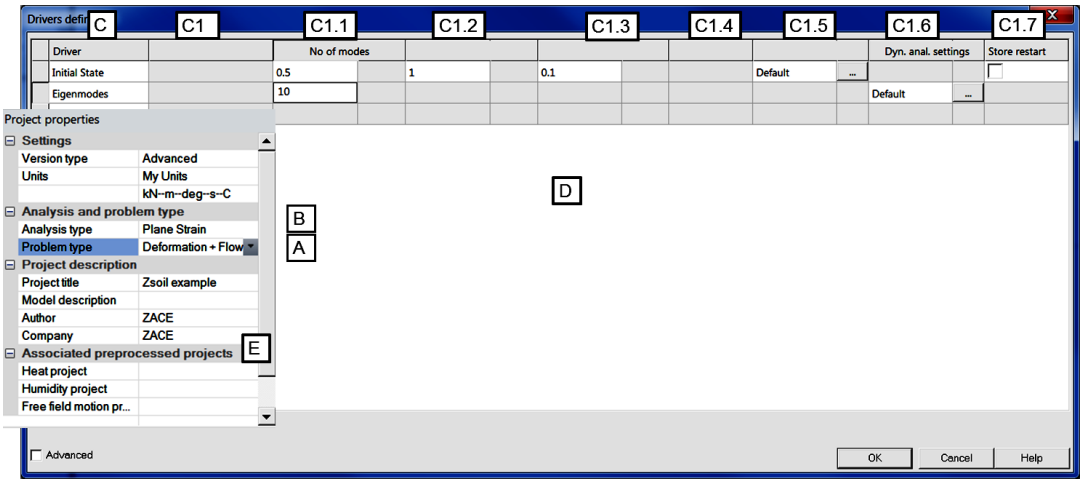



Window 2-50

## Window 2-51: Eigenvalues and eigenmodes

**A.** `Problem type: Deformation+Flow`
**B.** `Analysis type: any`
**C.** `Driver: Eigenvalues`
**C1.** `Driver type:`

In the `Deformation+Flow` mode this driver can be used for detection of declared number of eigenvalues and corresponding eigenmodes for two-phase media including structural elements. Contrary to the `Deformation` mode the mass matrix of continuum elements includes water content corresponding to the current saturation ratio. Hence it is recommended to run the `Initial state` driver first and then continue with the `Eigenvalue` one.




Control data of the driver are:

| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1.1** | `No of modes` | | | 1, 2 |
| **C1.6** | `Dyn. Solver` | Control data | | |

**Remarks**:

1. Number of detected eigenvalues is limited by total number of free degrees of freedom
2. For each eigen-mode mass particicpation factors are computed for all directions

Window 2-51

## Related Topics

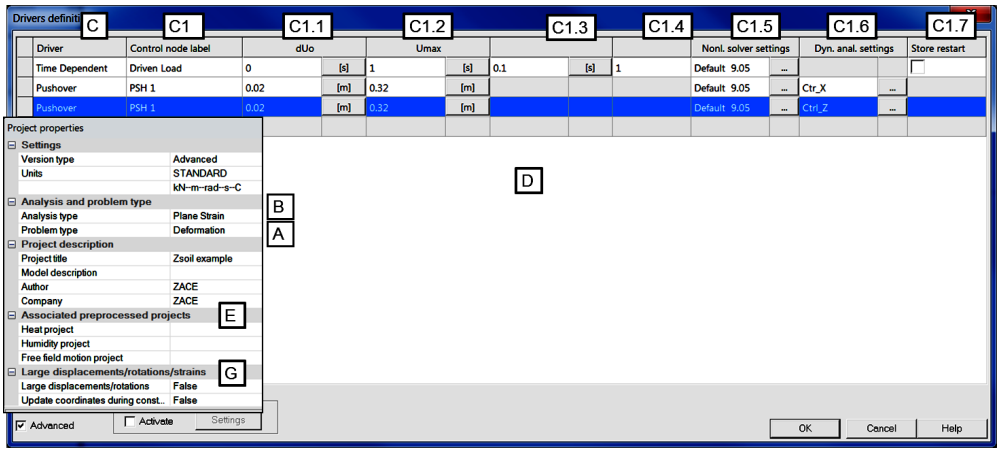
*Data preparation*

- DYNAMICS
- SEISMIC INPUT

## 2.2.2.1.8 PUSHOVER

**Window 2-52: Pushover**

**A.** `Problem type: Deformation`
**B.** `Analysis type:Plane Strain, 3D` (structures only)
**C.** `Driver: Pushover`




Control data of the driver are:

| *Key* | *Option* | *Comment* | *Default* | *Remarks* |
|---|---|---|---|---|
| **C1** | | | | 2,4,5 |
| **C1.1** | $\mathrm{d}U_o$ | Initial value of displ. increment $\Delta U_o$ | (0.01 [m]) | 3 |
| **C1.2** | $U_{max}$ | Final value of $U$ | (0.2[m]) | 3 |
| **C1.5** | `Nonl. Solver` | Control data for nonlinear solver | | |
| **C1.6** | `Dyn. Solver` | Control data for `pushover` direction | | |

Window 2-52

Driver `Pushover` handles a simplified static analysis of structures subject to seismic excitations. It is recommended by several design codes (EC8, FEMA) for building and bridge structures. In recent version only structural elements can be used in the model. The model has to include masses (element and/or nodal). It consists of the following steps:

- solution of an eigen-problem leading evaluation of the 1-st eigenvalue (free vibration period) and 1-st eigen-mode in the assumed direction of seismic excitation.
- nonlinear static procedure (via arc length procedure) in which structure is incrementally loaded by a set of forces $\lambda \mathbf{f}_o$ where $\lambda$ is a load multiplier and $\mathbf{f}_o$ is a load pattern representing the inertia forces acting on the structure during the earthquake. Nonlinear static analysis yields a force-displacement **pushover curve (capacity curve)**
- considered load patterns correspond to inertia forces resulting from:
  - ⋆ actual masses *uniform unit multiplier
  - ⋆ actual masses *triangular distribution
  - ⋆ actual masses *first mode shape
- assessment of the structure seismic capacity basing on the relation between **capacity curve** of equivalent single DOF system and demand spectrum specific to given design code.
- number of reduction steps (in case of divergence of computation) is set up under nonlinear solver settings

**Remarks**:

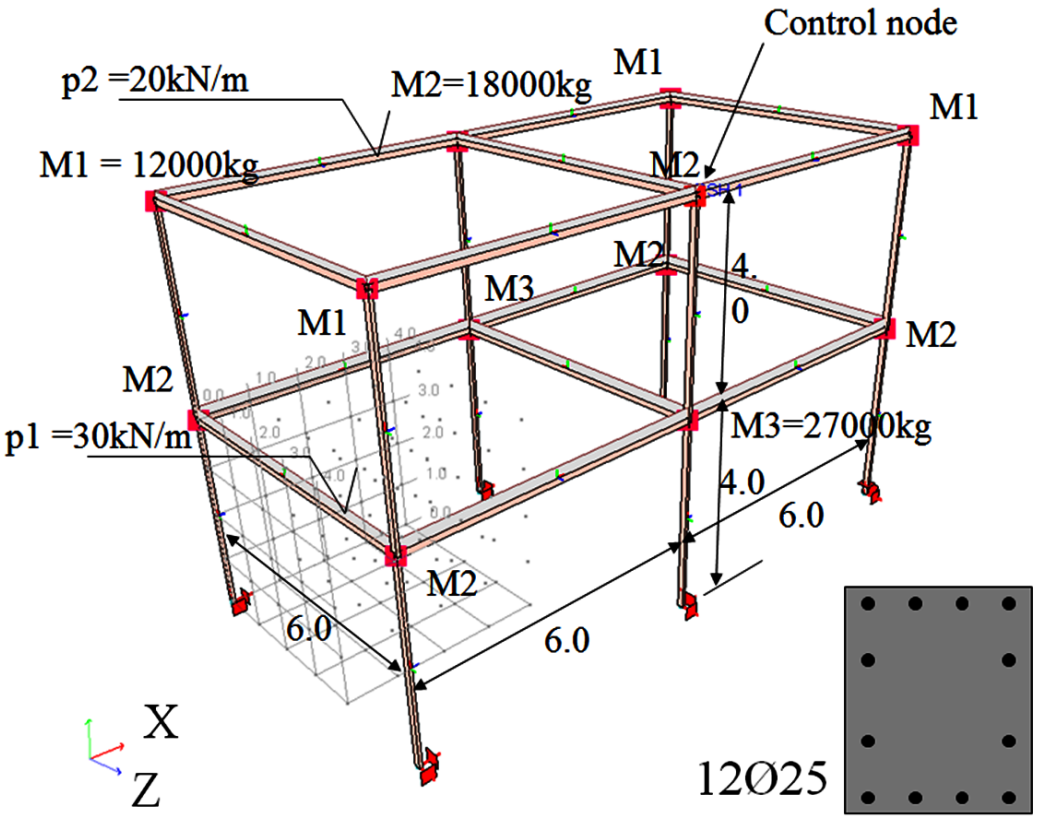
1. Model must have nonzero mass (masses are defined as element and/or nodal)
2. `Driver type: Control node` **must** be set during `preprocessing`.
3. Recommended value of $U_{max}$ is 4% of structure height roughly. To make pushover curve sufficiently smooth use d$U_o < U_{max}/10$
4. A sequence of `Pushover Drivers` can be defined, with different `Control` and `Pushover Settings` specification.
5. `Pushover` can be run within a sequence of `Time dependent`, `Driven Load Drivers`
6. The state of the structure at start of a `pushover` driver takes into account results (e.g. plastic status, stresses, deformation) ) of previous `Time dependent` driver(s).
7. Changes of the structure state (stresses, deformation , plastic status) induced by `pushover` driver are **disregarded** during subsequent drivers of any type
8. Step reduction procedure (combined with arc length method) is automatically performed in case when applying initially set $\Delta U$ causes divergence.
9. When convergence is not reached after applying displacement increment reduced `Red. steps` times from its initial value $\Delta U_o$, execution of the static procedure is terminated before reaching final value $U_{max}$.
10. Large deformation analysis may be activated.
11. After running `pushover` driver use `Postprocessing/Pushover Result` to perform seismic demand assessment automatically.

**Window 2-53:** `Dynamics` – **Pushover analysis of a frame**

Data: BENCHMARKS/ xFrame3D.INP

**A.** `Problem type: Deformation`
**B.** `Analysis type: 3D`
**C.** `Driver: Pushover`
**C1.** `Driver type: none`

R.C. frame of the one-storey building. Stiffness and strength of in-filling walls neglected.



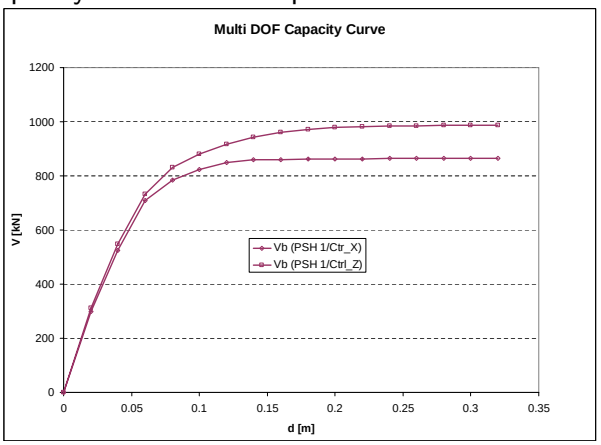

**Goals:**

- To build pushover capacity curve(s) relating top displacement- total horizontal base force
- Having established seismic demand (target displacement), to asses structural performance at demanded load level.

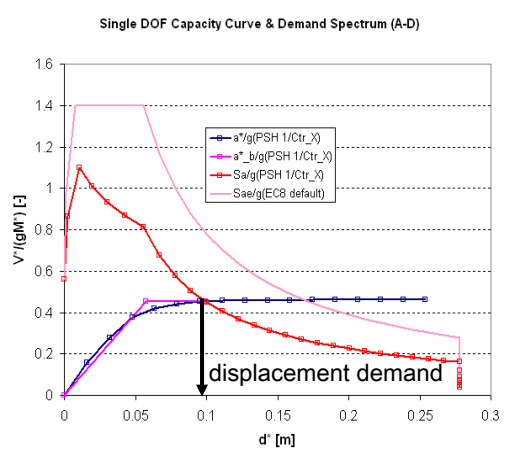
Window 2-53

## Window 2-54: Pushover – Results

Original (multi-DOF) capacity curves for two pushover runs




Seismic demand assessment using EC8 design spectrum (under `Menu` / `Results` / `Pushover results`)




| | | Pushover analysis repor |
|---|---|---|
| Item | Unit | PSH 1/Ctr_X |
| MDOF Free vibr. period.....T | [s] | 0.5714 |
| SDOF Free vibr. period.....T* | [s] | 0.7148 |
| SDOF equivalent mass.....M* | [kg] | 151997 |
| Mass participation factor Gamma | - | 1.2593 |
| Bilinear yield force value..Fy* | [kN] | 676.96 |
| Bilinear displ. at yield....Dy* | [m] | 0.0576 |
| Target displacement.....Dm* | [m] | 0.0995 |
| SDOF displacement demand....Dt | [m] | 0.0995 |
| Energy.....Em* | [kN*m] | 47.854 |
| Reduction factor.....qu | - | 1.7266 |
| Demand ductility factor.....mi | - | 1.7266 |
| Capacity ductility factor...miC | - | 1.7265 |
| MDOF displacement demand.....D | [m] | 0.1253 |

**Remarks**:

1. The demand of the building is given by the design spectrum provided by the design codes transformed to ADRS format.
2. In order to compare the capacity curve to the demand curve given by the design spectrum, the nonlinear pushover curves of the multi-DOF system are first transformed to equivalent single-DOF SDOF, then are approximated by elastic-perfectly plastic (or bilinear) curves
3. After evaluation of target displacement during seismic demand assessment automatic procedure, all static analysis results (including deformation, internal forces), giving hints on performance of the structure under given seismic event, can be traced at the demanded level of control displacement

Window 2-54

## Related Topics

*Data Preparation:*

- *DRIVERS AND DRIVERS TYPES*
- *PUSHOVER*

## 2.2.2.2 ANALYSIS TYPES

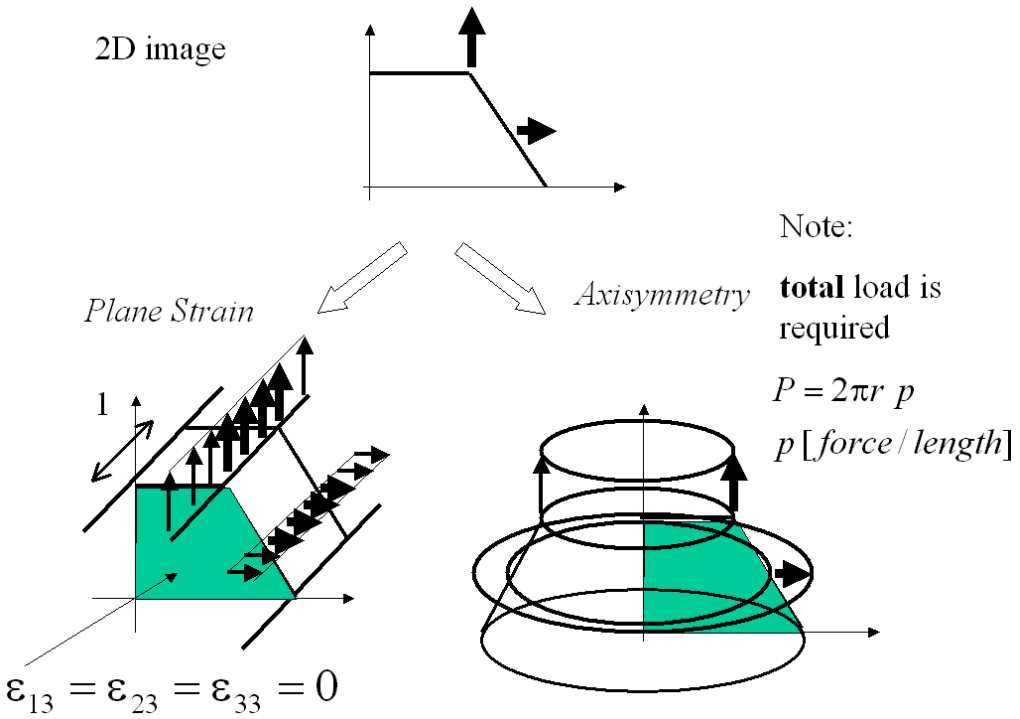
ZSoil offers the following set of analysis types:

3D - three-dimensional analysis without any simplifying assumptions.

`Plane Strain` analysis - a unit "thickness" is assumed along the third direction; three strain components are assumed to be equal to zero: $\varepsilon_{33} = \varepsilon_{13} = \varepsilon_{23} = 0$.

`Axisymmetry` analysis - concerns a model characterized by symmetry in the sense of geometry, material properties, boundary conditions and loads. Axis of axisymmetry is assumed to coincide with $x_2$ (vertical axis), and radial coordinate is equivalent to $x_1$ (NB: always $x_1 \geq 0$). Note that certain loads definition may be different compared to plane strain case (see Window 2-55), also some geometrical data defined at the material level (reinforcement for instance) can be different

**Window 2-55: Plane strain and axisymmetry**




Nodal load in 2D `Plane Strain` and `Axisymmetry` .

Window 2-55

**Remarks**:

1. `Plane Strain`, `Axisymmetry` computational models are built in 2D geometrical space;
2. preselected `Analysis Type` should not be changed during data preparation session unless knowledgable of effects
3. `Analysis Type` is unique for all drivers

## Related Topics

*Theory:*

- PROBLEM STATEMENT
- ELEMENTS

### 2.2.2.3 ASSOCIATED PREPROCESSED PROJECTS

Decoupled thermo-mechanical or moisture-mechanical problems can be solved in two steps. The thermal/hygral problem is solved first. Results of `Heat Transfer` and `Humidity Transfer` projects can then be reused as an input for subsequent `Deformation` analyzes run to compute stresses caused by imposed thermal and/or hygral strains.

The dynamic time history analysis to be carried out with aid of the Domain Reduction Method (DRM) has to be run in two steps. In the first one the free field motion is found by running time history analysis of a simple model (shear layer for instance). Results of this simple model (displacements, velocities and accelerations) are then reused by the DRM method to find the response of the soil-structure interaction model that includes relatively low number of degrees of freedom.

Weakly coupled hydro-thermal problems can be solved in two steps. The seepage problem is solved first. Results of the seepage analysis (fluid velocities and saturation ratio in fact) can then be reused as an input for subsequent `Heat Transfer` analyses in order to include advection.

- `Heat Transfer` project, if associated with `Deformation` (+`Flow`), includes time dependent temperature field used to compute thermal strains in computational domain. Thermal dilatancy coefficient is specified at the material level (group `Heat`) of deformation project
- `Humidity Transfer` project, if associated with `Deformation` (+`Flow`), includes time dependent humidity field used to compute hygral strains in computational domain. Hygral dilatancy coefficient is specified at the material level (group `Humidity`) of deformation project
- `Free field motion` project includes displacement, velocity and acceleration time histories to be applied to the set of continuum elements selected as a DRM boundary layer.
- Seepage project, if associated with `Heat Transfer`, includes time dependent fluid velocities field used in the advection term.

**Remarks**:

- Computational models for `Heat` and `Humidity` analyses may include exclusively continuum and convection elements (in case of heat analysis convection/radiation elements can also be used) (truss, membrane, shell, beam elements may appear in the model but will be ignored during computation)
- Meshes used for `Heat Humidity` and later on for `Deformation`(+`Flow`) analyses can be all different
- Time scale for `Heat Humidity` and later on `Deformation`(+`Flow`) analyses must be coherent
- Increments of imposed strains are computed based on temperature and/or humidity increment computed for two time instances $t_{N+1}$ and $t_N$; for that reason a single steady state thermal solution will yield zero thermal strains (as the reference temperature is equivalent to the current one, hence no temperature change is detected)
- To take an effect of variation of thermal/or hygral strains in structural elements like beams or shells a layered approach **must** be activated (for corresponding material models `Non--Linear` checkbox in the material dialog box must be activated); corresponding temperatures and/or humidities will be projected from `Heat` and/or `Humidity` mesh to any integration point in beam or shell cross section

- First running of `Deformation`(+`Flow`) problem with preprocessed thermal and/or humidity fields takes a considerable amount of time due to the fact that each integration point in continuum, shell, beam, membrane or truss element must be projected onto `Heat` and/or `Humidity` mesh
- To learn on how to build DRM models for dynamic soil-structure interaction problems, please refer to the dedicated report

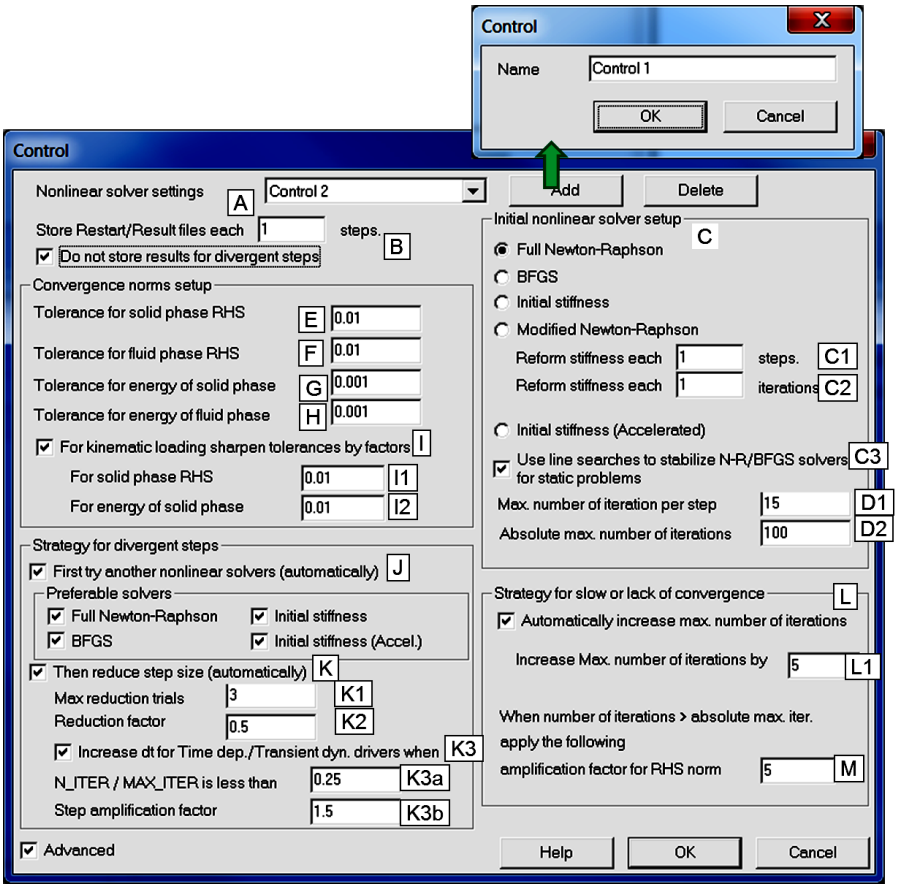
## Related Topics

- *Data Preparation:*
  - ⋆ *HEAT TRANSFER*
  - ⋆ *HUMIDITY TRANSFER*
  - ⋆ *HEAT TRANSFER DATA GROUP*
  - ⋆ *HUMIDITY TRANSFER DATA GROUP*
- *Theory:*
  - ⋆ *PROBLEM STATEMENT*
  - ⋆ *HEAT TRANSFER*

## 2.2.3 CONTROL

Sets convergence tolerances, type of nonlinear solver and frequency of restart and results storage.

**Window 2-56: Control dialog box**




Default control parameters are set by the program automatically. These settings cannot be modified by the user. In order to set user control parameters a new control must be created and labeled (use button **Add**).

A. used to select user defined control parameters
B. sets frequency storage for restart and result files (storing diverged steps is optional)
C. switches between full Newton-Raphson (NR), BFGS, Modified NR, Initial stiffness method and accelerated initial stiffness (modified Thomas method)
C1. sets step frequency of stiffness reevaluation for modified NR nonlinear solver
C2. sets iteration frequency of stiffness reevaluation for modified NR nonlinear solver
C3. activates line searching for NR and BFGS solvers in static problems
D1. default maximum number of iterations per step (will be adjusted automatically during computation unless divergence occurs)
D2. if number of iterations exceeds this value computation is stopped (only when automatic nonlinear solver switch is not possible and step reduction cannot be made)
E. used to stop iterative solution based on right hand-side force residuals for kinematic degrees of freedom (DOFs)
F. same as **E** but for fluid pressure DOFs
G. same as **E**, based on energy of solid phase

H. same as **G**, based on energy of fluid phase
I. contrary to the standard load control most cases kinematic loading programs require sharper convergence tolerances; therefore tolerance for the RHS norm as well as the one for the solid energy have to be decreased
I1. sets reduction factor for the RHS tolerance to be used in kinematic loading programs (standard tolerance defined in (E) will be multiplied by the factor given in (I1))
I2. sets reduction factor for the energy tolerance to be used in kinematic loading programs (standard tolerance defined in (G) will be multiplied by the factor given in (I2))
J. allows to try to use different nonlinear solvers once the assumed one fails to converge (certain drivers require specific nonlinear solvers; therefore user setting will be changed in the calculation module automatically; for instance steady state seepage analysis requires damped NR method to be used); the general rule is such that for 3D models we do always try to start from BFGS nonlinear solver while for 2D ones we do start from full NR by default; it is also possible to to setup active nonlinear solvers
K. flag that activates automatic step reduction in case of divergence of a given step; this option is meaningful for `Initial state`, `Stability`, `Time Dependent` drivers and `Transient dynamics`
K1. sets maximum number of step reduction trials
K2. value of the factor that reduces step size ($\Delta t$, $SF$ or increment of gravity load for the `Initial state`)
K3. activates adaptive time step (for `Time dependent` or `Transient dynamics` drivers) amplification (if previously the step was reduced)
K3a. time step size can be amplified when ratio of number of iterations needed to achieve the convergence state to the declared max. number of iterations per step (D1) is less than given value
K3b. time step size, if possible, will be amplified by a given factor (larger than 1.0 but less than 1.0 divided by the reduction factor K2)
L. flag that allows the calculation module to increase maximum number of iterations once the iterative scheme is converging slowly
L1. once the flag (L) is set on maximum number of iterations will be increased by a given value
M. once the absolute maximum number of iterations (D2) is exceeded the RHS tollerance is amplified by a given factor

**Remarks**:
1. No user input mandatory, the screen is initialized with default values
2. In case of convergence problems, especially for softening materials, use Initial stiffness stiffness method
3. Activating line searches for NR and BFGS solvers may help to stabilize convergence
4. During run one may trace the convergence status interactively through the calculation module dialog box (one may modify convergence tolerances and enforce the calculation module to skip to the next step, or skip to the next nonlinear solver)

Window 2-56

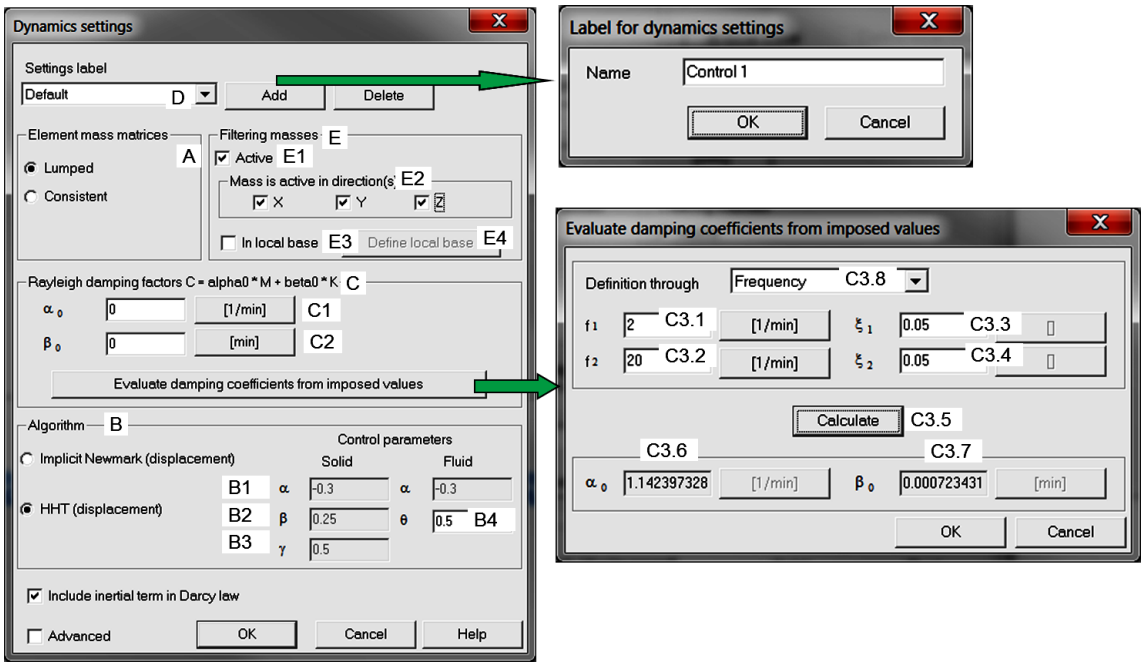
## Related Topics

*Theory*
- ALGORITHMS

## 2.2.4 DYNAMICS

Sets type and parameters of the time history analysis algorithm and damping factors for drivers `Dynamics`.

**Window 2-57: Dynamics dialog box**




**A.** sets which type of the mass matrix is to be used; only lumped masses are supported
**B.** sets the time history analysis algorithm type (Newmark or Hilber-Hughes-Taylor (HHT))
**B1.** sets $\alpha$ parameter for HHT algorithm (default -0.3), $\alpha = 0$ corresponds to Newmark alg.
**B2.** sets $\beta$ parameter for HHT (depends on $\alpha$) and Newmark (default 0.25)
**B3.** sets $\gamma$ parameter for HHT (depends on $\alpha$) and Newmark (default 0.50)
**B4.** sets $\theta$ parameter for integration scheme for pore pressure (default 0.50)
**C.** sets Rayleigh damping factors $\alpha_o$ (applies to the mass) and $\beta_o$ (applies to the stiffness)
**C1.** sets directly Rayleigh damping factor $\alpha_o$
**C1.** sets directly Rayleigh damping factor $\beta_o$
**C3.** runs Rayleigh damping factors calculator and puts results in **C1.** and **C2.** edits
**C3.1** sets 1-st angular frequency $\omega_1$, frequency $f_1$, or period $T_1$ depending on the state of the combo-box C3.8
**C3.2** sets 2-nd angular frequency $\omega_2$, frequency $f_2$, or period $T_2$ depending on the state of the combo-box C3.8
**C3.3** sets percentage of the critical damping for $\omega_1$, $f_1$ or $T_1$
**C3.4** sets percentage of the critical damping for $\omega_2$, $f_2$ or $T_2$
**C3.5** runs $\alpha_o$ and $\beta_o$ calculation
**C3.6** shows calculated $\alpha_o$
**C3.7** shows calculated $\beta_o$
**C3.8** switch from angular frequency to frequency, or period
**D.** adds current dynamic settings to the list of control settings to be used later in conjunction with a selected driver `Dynamics` in `Analysis & Drivers` dialog box
**E.** sets filtering of mass matrix for selected global or local directions
**E1.** switches ON/OFF filtering of mass matrix

**E2.** selects active directions for mass matrix
**E3.** selects local or global coordinate system to filter mass matrix along selected directions
**E4.** activates a dialog box to set up the local coordinate system for mass filtering (note that for local systems defined through the vector in 3D only filtering in the local X direction (vector direction) is meaningful as the other two directions are not defined in a unique manner)

**Remark**:
Assumed damping factors apply to all materials unless overwritten locally in `Materials`

Window 2-57

## Related Topics

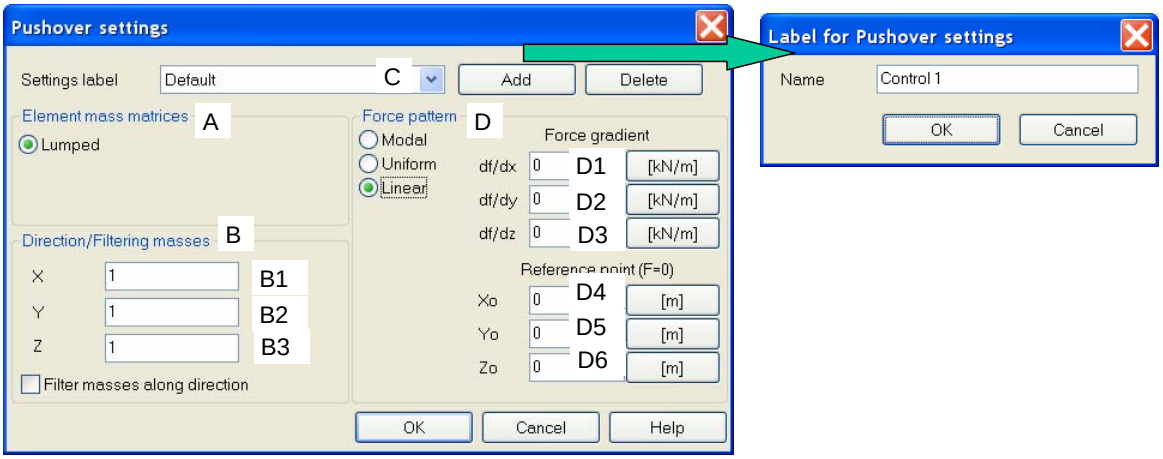
*Theory*

- ALGORITHMS

## 2.2.5 PUSHOVER

Sets parameters for drivers: `Pushover`

**Window 2-58: Pushover dialog box**




A. sets which type of the mass matrix (to extract the first mode) is to be used; only lumped masses are supported
B. activates mass matrix filtering according to the direction vector (masses on that direction will only be active) (this can be necessary to isolate properly the first mode)
B1. sets $x$ component of the direction vector
B1. as above but $y$ direction
B3. as above but $z$ direction (meaningful for 3D problems)
C. adds current pushover settings to the list of control settings to be used later in conjunction with a selected driver `Pushover` in `Analysis & Drivers` dialog box
D. selects pushover force pattern (Modal/Uniform/Linear) (**Modal** is equivalent to the shape of the first eigenmode)
D1. force gradient for the **Linear** pattern in X direction
D2. as above but for Y direction
D2. as above but for Z direction
E1. X coordinate of the reference point (where F=0) for the **Linear** pattern
E2. as above but Y coordinate
E2. as above but Z coordinate

Window 2-58

## Related Topics

*Theory:*

- ALGORITHMS

## 2.2.6 CONTACT ALGORITHM

This option is **meaningful** exclusively for computational models including small or large deformation **contact elements**[6]. It applies both to the contact interfaces working in the continuity mode and true frictional contact interfaces. A common problem of the standard penalty approach used for problems of contacting elasto-plastic media is that the resulting over-penetration is too large and contact stresses may be underestimated. Using high values for penalty stiffness we may observe loss of convergence and oscillatory contact stress distributions. In the case of soil-structure interaction the stress resultants in structural elements (beams, shells) may be underestimated as well. To handle this deficiency an Augmented Lagrangian Approach can be used. In the current contact formulation (segment to segment approach for small deformations and node-segment for large deformations) each time the state of global static equilibrium is achieved, penalty stiffness is increased (by default by factor of 2).

The following control parameters (group A) have to be set once the Augmented Lagrangian approach is activated (see Window 2-59)

1. Maximum number of augmentations (do not exceed 5 as each augmentation increases computational cost)
2. Maximum acceptable value of over-penetration
3. Penalty stiffness multiplier (default is 2.0); at each augmentation step contact stiffness is amplified by this factor in order to speedup the convergence (if the initial contact stiffness is relatively large this parameter must be kept equal to 1.0)
4. Maximum allowed value of the amplification factor for contact penalty stiffness (default is 100.0); if we would assume the stiffness multiplier equal to 10.0 then after the second augmentation the total amplification factor would be equal to 100.0; hence after second augmentation stiffness would not be amplified

Contact frictional interfaces may frequently cause lack of convergence or divergence of iterative schemes. This usually happens when oscillatory switches from stick/sliding to opening occur. In order to damp this effect one may activate extra numerical damping (group C of parameters in the dialog) in which current tangent stiffness is perturbed by the elastic one using the following scheme (value of $\alpha$ is usually problem dependent)

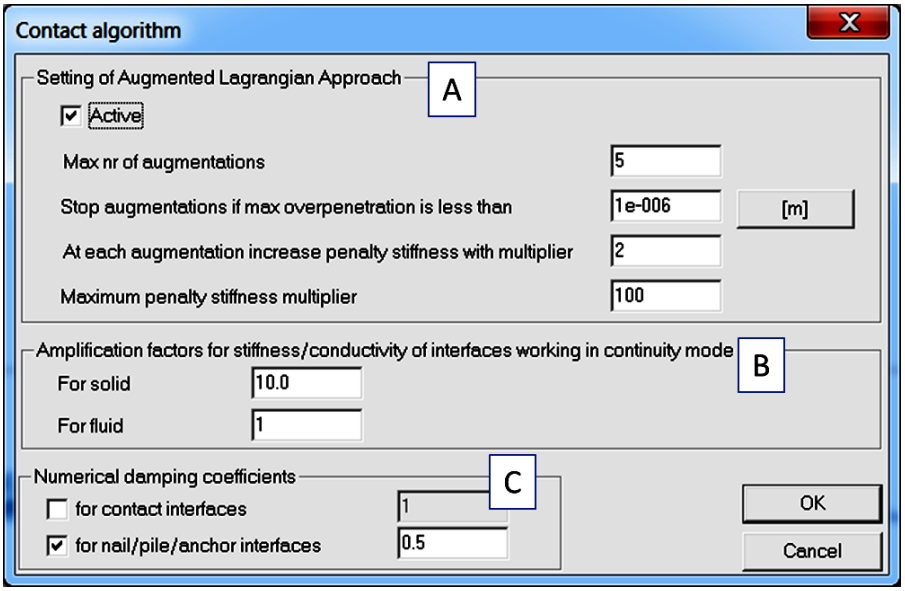
$$\begin{aligned} \mathbf{K} &= (1-f)\mathbf{K}^{\mathbf{ep}} + f\mathbf{K}^{\mathbf{e}} \\ f &= exp(-\alpha \text{ iter}^2) \end{aligned}$$

For contact interfaces working in the continuity mode we use the same computational method as for standard frictional ones but Coulomb's friction law and tensile cut-off are neglected. In some situations we may observe that the displacement or pressure continuity condition is not satisfied to an assumed level of accuracy. To correct this defficiency we may amplify automatically set (by the calculation module) penalty stiffness (for continuity mode interfaces default amplification factors are $10^2$ times larger than for the frictional ones) by a given

[6] concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT** only

coefficient. This can be made independently for the solid phase (continuity of displacements) and fluid one (continuity of pore water pressures) (group B of parameters in dialog box).

**Window 2-59: Control parameters for handling true frictional contact/continuity interfaces**




Control data for true frictional contact/continuity interfaces

Window 2-59

For the detailed explanation of the above settings see the theory under:

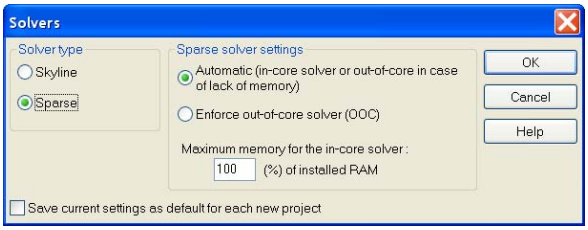
## Related Topics

*Theory:*

- INTERFACES
- CONTACT
- AUGMENTED LAGRANGIAN APPROACH

## 2.2.7 LINEAR EQUATION SOLVERS

**Window 2-60: Selection of linear equation solver**




The two linear equations solvers can be used:

- `Skyline` – based on Cholesky's decomposition; this solver is recommended for 2D problems as it has very good diagnostics for detection of singularities
- `Sparse` – based on Gauss elimination; this powerful solver is recommended for large 2D and 3D computations; however it does not have extended diagnostic tools as skyline does; this solver may be run in core or out of core mode; the latter version is recommended when solving large problems on machines with insufficient RAM size.

**Remark**:

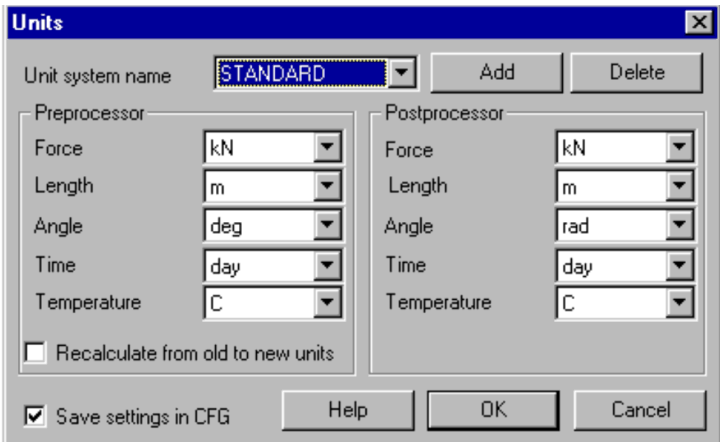
This setting is saved with a given project but can also be used as a global setting for all new projects (use `Save settings in CFG` button in the dialog box).

Window 2-60

## 2.2.8 UNITS

Units system offers a simple conversion tool assuming basic units for **force, length, time, angle, temperature**. Any value entered during data preparation has a specific unit button placed next to the edit field. When creating a new project Z_Soil offers 5 predefined sets of basic units: EXAMPLE UNITS, STANDARD, STANDARD 1, STANDARD 2 and UNDEFINED to be selected from the initial project preselection dialog box. One set of units can be used during computation and another one to visualize results (during postprocessing phase user may use several combinations of basic units). Change of the basic units can also be made later on by activating a dialog box to handle units through menu `Control` ← `Units`.

**Window 2-61: Setting basic units**




Remarks:

| | |
|---|---|
| A | 1,2,3 |
| B | 4,5 |
| C | 1 |
| D | 2 |

**Remarks**:

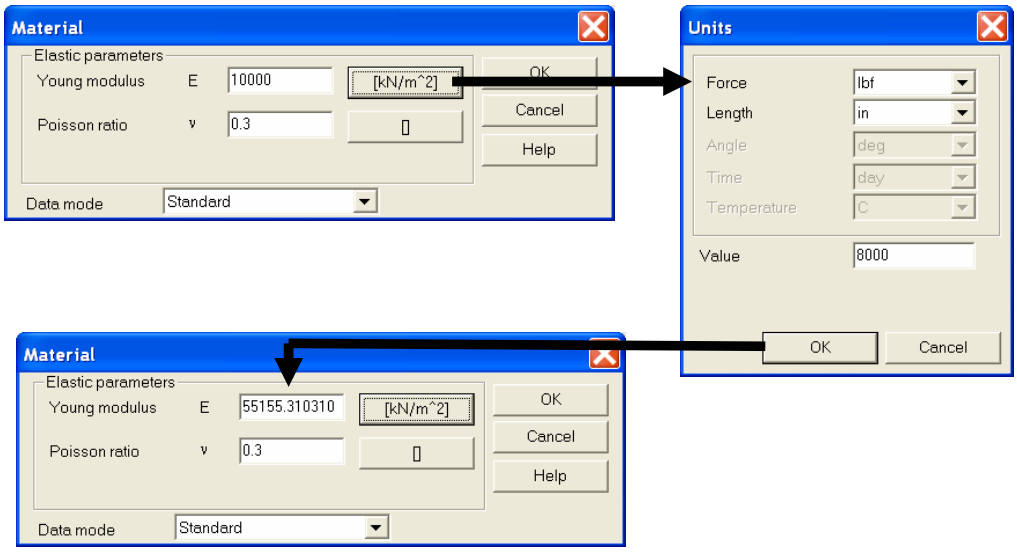
1. Setting basic units can be made on opening of a new project but also during data preparation; in the later case user must decide whether all entered values are to be recalculated to be coherent with the new units or are to be left unchanged; first option is activated when `Recalculate from old to new units` checkbox is set ON and the second if it is set to OFF.
2. Newly defined system of units can be memorized with a distinct label by using option **Add** with the checkbox `Save settings in CFG` set to ON
3. As units system was introduced since version 5.02, older data files will have units system set as UNDEFINED; user may work with this setting but can also apply one of the 5 predefined unit sets or may create its own, as described above
4. Angles defined at the material level (like friction or dilatancy, anisotropy directions) are always given in [deg]
5. Imposed rotation angles are always defined in [rad]
6. Units for mass and mass density are treated in an exceptional way; these values are specified using mass unit kg or lb (for mass density length unit is taken from standard setting) and then recalculated to the assumed basic units

Window 2-61

**Window 2-62: Example of units conversion**

Let assume that basic units are: [kN],[m],[s],[deg],[C]. Suppose that value for Young modulus $E$ is equal to $8000$ [psi] (i.e. [lb/in$^2$]) and we want to have it in kN/m$^2$. To convert the value proceed as follows:

1. Open `Materials`→`Elastic` dialog. A default value of 100000 [kN/m$^2$] appears in `Young modulus` edit field.
2. Click on the unit button next to the $E$ edit field
3. In the dialog box for units conversion set temporary units for force to [lbf] and for length to [in]
4. Enter value $8000.0$
5. Close the dialog with button **OK**
6. Value of $55155.31$ [kN/m$^2$] will appear in `Young modulus` edit field




Window 2-62

**Window 2-63: Available basic units**

Collection of basic units for **force, length, time, angle, temperature** is listed in the table.

| Item | Available basic units | | |
|---|---|---|---|
| *force* | gramforce | 1g | = 9.806e-6 kN |
| | newton | 1N | = 0.001 kN |
| | poundforce | 1lbf | = 0.004448 kN |
| | kilogramforce | 1kp | = 0.009806 kN |
| | decanewton | 1daN | = 0.01 kN |
| | **kilonewton** | **1kN** | - |
| | ton | 1t | = 9.806 kN |
| | Meganewton | 1MN | = 1.0e3 kN |
| | Giganewton | 1GN | = 1.0e6 kN |
| *length* | milimeter | 1mm | = 0.001 m |
| | centimeter | 1cm | = 0.01 m |
| | inch | 1in | = 0.0254 m |
| | foot | 1ft | = 0.3048 m |
| | yard | 1yd | = 0.9144 m |
| | **meter** | **1m** | - |
| *time* | millisecond | 1ms | = 0.001 s |
| | **second** | **1s** | - |
| | minute | 1min | = 60 s |
| | hour | 1h | = 3600 s |
| | day | 1day | = 86400 s |
| | week | 1week | = 604800 s |
| | month | 1month | = 2629800 s(remark 1) |
| | year | 1year | = 31557600 s(remark 1) |
| *angle* | **degree** | **1deg** | - |
| | grad | 1gr | = 9/10 deg |
| | radian | 1rad | = 180/$\pi$ deg |
| *temperature* | **Celsius** | **1C** | - |
| | Kelvin | 1K | = 1C(remark 2) |
| | Fahrenheit | 1F | = 5/9 C(remark 2) |
| *mass* | **kilogram** | **1kg** | - |
| | pound | 1lb | = 0.4536 kg |
| *energy* | Joule | **1J** | = 1Nm |
| *power* | Watt | **1W** | = 1J/s |

**Remarks**:

1. Multipliers for time units are derived assuming:
   1 year=365.25 days (as leap-years are taken into account)
   1 year=12 months
   1 week=7 days
2. Multiplicative conversions between Celsius and Fahrenheit or Kelvin temperature units, as given in the table below, are possible only when they concern increments (or ratio) but not absolute temperatures. For absolute temperatures $X$ given in °F or °K conversions to °C are given by following rules:
   $X[°C] = 9/5 * X[°F] + 32[°C]$
   $X[°C] = (X + 273.15)[°K]$

Window 2-63

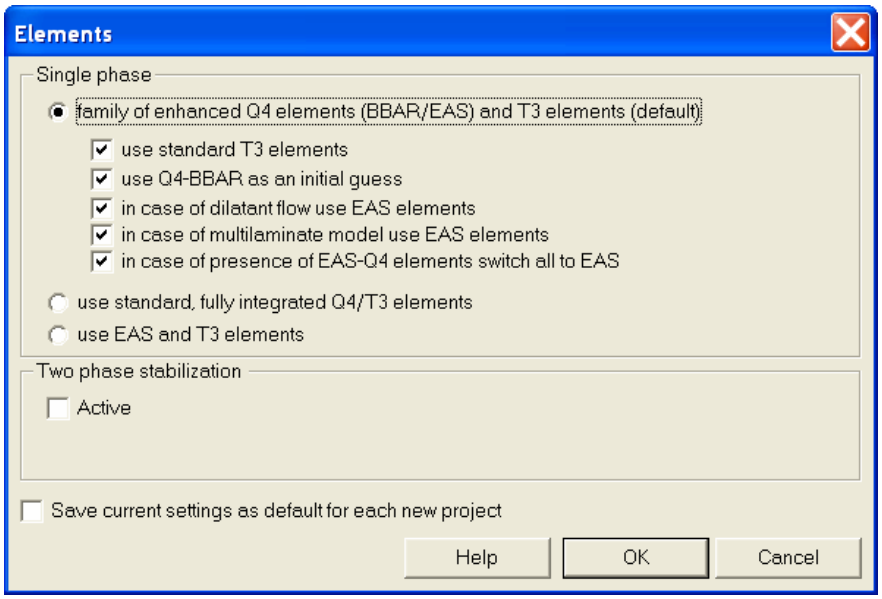
**Window 2-64: Compound units**

- Compound units consist of products or quotients of basic units. For instance, when force unit is set to [N] and length unit is set to [mm] then the resulting unit for stresses will be [N/mm$^2$].
- Commonly used stress compound units and their representation through basic units are as follows:
  Pascal: 1 Pa=1 N/m$^2$
  kiloPascal: 1 kPa=1 kN/m$^2$=10$^3$ N/m$^2$
  MegaPascal: 1 MPa=1 MN/m$^2$=10$^3$ kN/m$^2$=10$^6$N/m$^2$
  Pound force per square inch: 1 psi=6895 Pa=6.895 kPa

Window 2-64

## 2.2.9 FINITE ELEMENTS

**Window 2-65: Selection of finite elements for continuum**




**Remarks**:

- The above settings remain valid exclusively for `Deformation` and `Deformation+Flow` problems
- In ZSoil®only low order elements are available (4-node quadrilateral and 3-node triangle in 2D and 8-node brick and 6-node wedge in 3D)
- Three different formulations can be used:
  - ⋆ standard displacement (not recommended due to volumetric locking deficiency)
  - ⋆ mean-dilatational $\overline{\mathrm{B}}$ (aplicable to quasi-incompressible elasticity and plasticity with deviatoric plastic flow ($\psi = 0^o$)
  - ⋆ enhanced assumed strain (EAS) (aplicable to quasi-incompressible elasticity and plasticity with deviatoric or dilatant plastic flow rule)

Window 2-65

## Window 2-66: Handling volumetric locking effect

The following tables contain an YES or NO assessment of different elements and formulations with respect to the following three sources of locking phenomena:

- A - quasi incompressible elasticity ($\nu \leftarrow 0.5$) or deviatoric plastic flow ($\psi = 0^o$)
- B - dilatant plastic flow ($\psi > 0^o$)
- C - plastic anisotropy (multilaminate model)

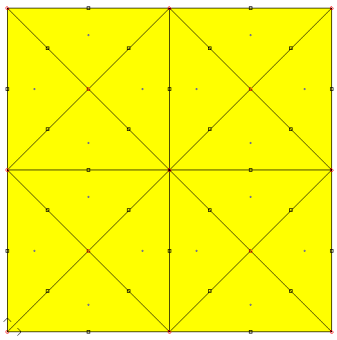
| Elements | Plane-strain | | | Axisymmetry | | |
|---|---|---|---|---|---|---|
| | A | B | C | A | B | C |
| Q4-$\overline{\text{B}}$ | Y | N | N | Y | N | N |
| Q4-EAS | Y | Y | Y | Y | Y | Y |
| Q4 | N | N | N | N | N | N |
| T3* | Y* | Y* | N | N | N | N |

*– only for cross-diagonal patches (see Figure below)

| Elements | 3D | | |
|---|---|---|---|
| | A | B | C |
| B8-$\overline{\text{B}}$ | Y | N | N |
| B8-EAS | Y | Y | Y |
| B8 | N | N | N |
| W6 | N | N | N |

**Remarks**:

- Q4 and B8 $\overline{\text{B}}$ elements are computationally more stable than corresponding EAS elements
- Triangular elements can be used in practical computations but exclusively for plane strain small deformation case using so-called cross-diagonal patches (see Figure below)



Window 2-66

**Window 2-67: Default strategy for selection of continuum elements**

The following default strategy is used for selection of optimal finite elements for continuum:

- First all Q4, B8 elements are set as $\overline{\text{B}}$ elements
- If exists at least one material which exhibits dilatant plastic flow rule then all Q4, B8 elements related to that material are switched from $\overline{\text{B}}$ to EAS
- If exists at least one material which corresponds to multilaminate model for rocks then all Q4, B8 elements related to that material are switched from $\overline{\text{B}}$ to EAS
- If at least one Q4 or B8 element was switched from $\overline{\text{B}}$ to EAS switch all elements to EAS

**Remarks**:

1. These settings can be used for current project (☒ **Save current settings for..** is set to OFF) but one can use them for all new projects if checkbox ☒ **Save current settings for..** is set to ON
2. In order to avoid spurious pore pressure spatial oscillations for two-phase consolidation problems (`Deformation+Flow` or `Dynamics`) and to avoid limitation put on the minimum time step a stabilization procedure can be activated through ☒ **Two phase stabilization** checkbox

Window 2-67

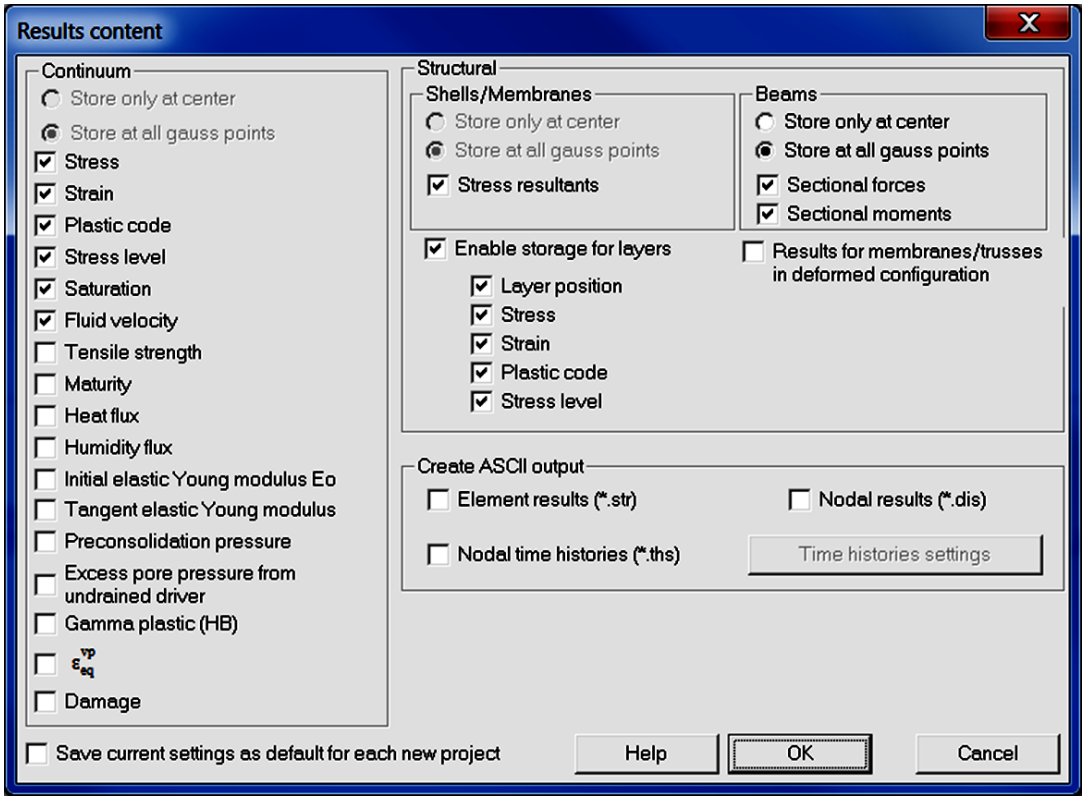
## Related Topics

*Theory:*

- INCOMPRESSIBLE AND DILATANT MEDIA

## 2.2.10 RESULTS CONTENT

**Window 2-68: Setting results content**




**Remarks**:

- The content of results can separately be set for continuum, shell/membranes, beams/truss elements
- Results are stored in form of binary files
- In ZSoil®storing results at center of continuum, shell and membrane elements is obligatory due to application of a robust Superconvergent Patch Recovery (SPR) method used to postprocess element results like stresses, strains, fluid velocities etc..
- Option *Results for membranes/trusses in deformed configuration* is obsolete
- With option *Create ASCII output* user may store element and/or nodal results in form of text files to be analyzed later on using standard text editors (notepad.exe, wordpad.exe etc..); however, activation of this option may generate an excessive output
- All settings made in this dialog box are distinct for current project; to reuse these settings for new projects set checkbox ☒ `Save settings in CFG` to ON

Window 2-68

**Window 2-69: Element results**

- **Continuum** (file .s01)

| Item | Deform.$^{(1)}$ | D+F$^{(1)(2)}$ | Flow$^{(2)}$ | Heat$^{(3)}$ | Humid.$^{(3)}$ |
|---|---|---|---|---|---|
| *Stresses* | + | + | - | - | - |
| *Strains* | + | + | - | - | - |
| *Plastic Code* | + | + | - | - | - |
| *Saturation ratio* | - | + | + | - | - |
| *Fluid velocity* | - | + | + | - | - |
| *Tensile strength* | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Maturity* | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Heat flux* | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Humidity flux* | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Initial modulus* $E_o$ | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Unl.-rel. modulus* $E_{ur}$ | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Preconsolidation pressure* $p_c$ | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Undrained excess pore pressure* $\Delta p_{undr}$ | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Equivalent plastic strain* | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Equivalent viscoplastic strain* $\varepsilon_{eq}^{vp}$ | +$^{(4)}$ | +$^{(4)}$ | - | - | - |
| *Damage level* $D$ | +$^{(4)}$ | +$^{(4)}$ | - | - | - |

- **Shells/membranes**(file *.s03)

| Item | Deformation$^{(1)}$ | D+F$^{(1)}$ | Flow$^{(2)}$ | Heat | Humidity |
|---|---|---|---|---|---|
| *Membrane forces* | + | + | - | - | - |
| *Bending moments* | + | + | - | - | - |
| *Shear forces* | + | + | - | - | - |
| *Strains (membranes)* | + | + | - | - | - |

- **Beams**(results can be stored in all integration points or only at the center in file *.s04)

| Item | Deformation$^{(1)}$ | D+F$^{(1)}$ | Flow$^{(2)}$ | Heat | Humidity |
|---|---|---|---|---|---|
| *Sectional forces* | + | + | - | - | - |
| *Sectional moments* | + | + | - | - | - |

**Remark**:Flexibility based beam elements always produce results at all integration points

**Window 2-70: Nodal results**

- **Nodal results**(file *.s00)

| Item | Deformation$^{(1)}$ | D+F$^{(1)}$ | Flow$^{(2)}$ | Heat | Humidity |
|---|---|---|---|---|---|
| *Displacements* | + | + | - | - | - |
| *Rotations* | + | + | - | - | - |
| *Pore pressures* | + | + | + | - | - |
| *Temperatures* | - | - | - | + | - |
| *Humidities* | - | - | - | - | + |

- **Nodal residuals/reactions**(file *.r00)

| Item | Deformation$^{(1)}$ | D+F$^{(1)}$ | Flow$^{(2)}$ | Heat | Humidity |
|---|---|---|---|---|---|
| *Forces* | + | + | - | - | - |
| *Moments* | + | + | - | - | - |
| *Fluid fluxes* | - | + | + | - | - |
| *Heat fluxes* | - | - | - | + | - |
| *Humidity fluxes* | - | - | - | - | + |

- **Nodal velocities**(file *.v00)

| Item | Deformation$^{(1)}$ | D+F$^{(1)}$ | Flow$^{(2)}$ | Heat | Humidity |
|---|---|---|---|---|---|
| *Translational veloc.* | + | + | - | - | - |

- **Nodal accelerations**(file *.a00)

| Item | Deformation$^{(1)}$ | D+F$^{(1)}$ | Flow$^{(2)}$ | Heat | Humidity |
|---|---|---|---|---|---|
| *Translational accel.* | + | + | - | - | - |

Window 2-70

## 2.3 ASSEMBLY

Computational model is build using the following set of the following model components options are useful to build up computational model:

| | |
|---|---|
| Preprocessing | Building geometrical model |
| Mesh Info | Short report on model components |
| Materials | Definition of material data |
| Existence function | Definition of existence functions |
| Load function | Definition of load time functions |
| Evolution function | Definition of evolution functions |
| Gravity | Control of body forces and $K_o$ state |
| Seismic Input | Definition of base accelerograms |
| Table of reinforcement sets for beams | Definition of reinforcements for beams |

### 2.3.1 PREPROCESSING

The detailed description of all tools to be used during geometrical preprocessing are documented in this section. Each tool is described first in the 2D and then in the 3D modeling context. Geometrical data preprocessing can be made on two basic levels:

- Macro-modeling level
- Finite Element model level

Macro-modeling is a set of CAD type tools which help the user to create geometrical model using basic objects like points, lines, polylines,arcs, circles, 2D/3D surfaces and 3D volumes. Using these objects as a skeleton user may then compose the geometrical model as a set of subdomains. Loads and boundary conditions can easily be applied to the macro-model. Meshing operation is treated here as a background action and it can be run several times for a given subdomain without loosing exact model geometry.

Computational model can also be built directly on the Finite Element level where elements, nodes and other model components are generated and/or updated one by one and control of the real model geometry is maintained by the resulting finite element mesh.

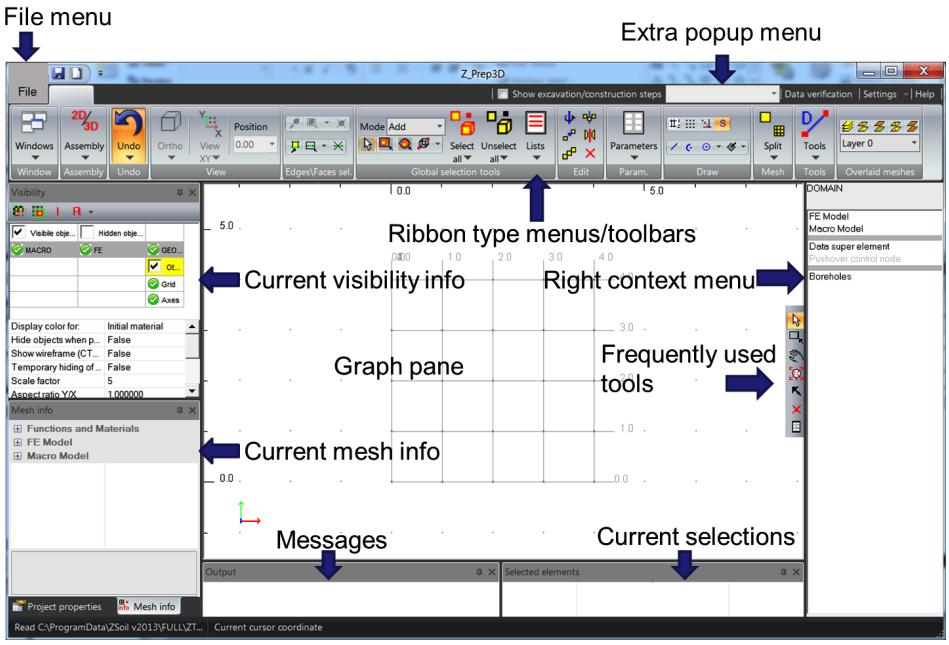
### 2.3.1.1 USER INTERFACE, MAIN MENU AND BASIC TOOLS

User interface consists of the following items:

| | |
|---|---|
| Main preprocessor window | Preprocessor screen including ribbon type menus combined with toolbars, extra menus placed at top right corner and context menu placed in the right panel |

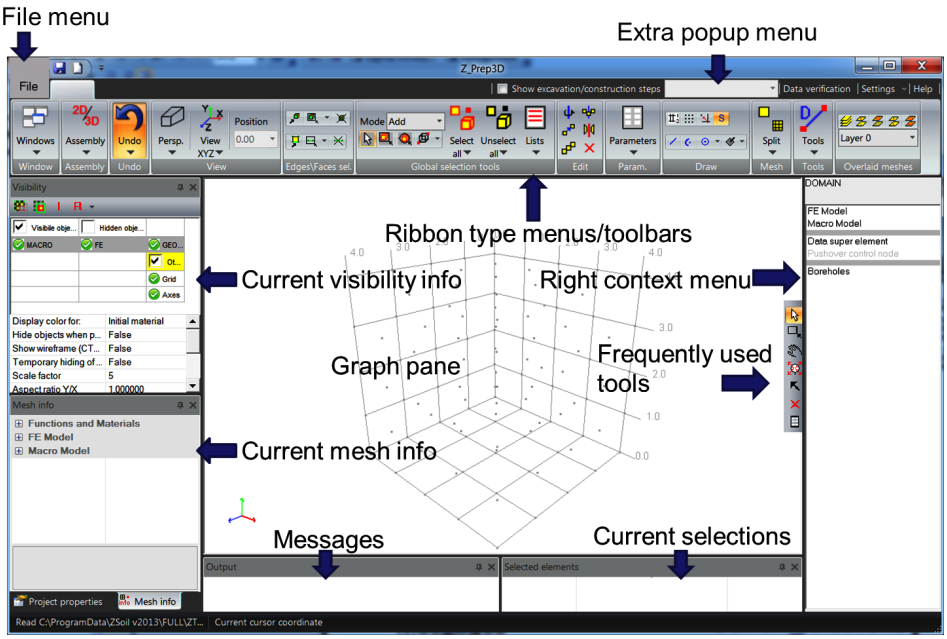
### 2.3.1.1.1 Main preprocessor window

**Window 2-71: User interface for 2D model**




Window 2-71

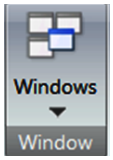
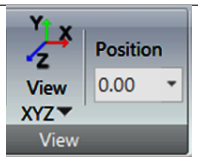
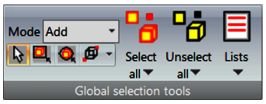
**Window 2-72: User interface for 3D model**

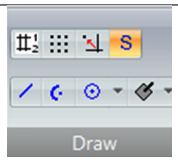
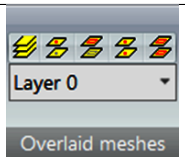



Window 2-72

### 2.3.1.1.2 Popup/ribbon menu

Popup/ribbon menu consists of the following items:

| | | |
|---|---|---|
| File |  | Contains options to handle input data files, import/export the data, merge two existing models |
| Windows |  | Puts focus on Visibility, Mesh info, Output, Selected elements or Project properties window; Project properties and Mesh Info occupy same position on the screen |
| Assembly |  | Options: Project preselection, Materials, Existence function, Load function, Evolution function, Drivers and Control; these are the options that appear under `Assembly` menu in the main ZSoil application |
| Undo |  | Undo/Redo options |
| View |  | Set of options to set up type of camera (perspective/orthogonal), view type (XYZ, XZ, XY, YZ) and working plane (X=..,Y=..,Z=..) (in 3D) |
| Edges/Faces selection |  | Set of methods to select edges and faces |
| Global selection tools |  | Set of methods to select/deselect single or selected set of finite element mesh components by picking the objects, using zoom or closed contour selection tools; logical operations on sets of selections is possible according to the assumed mode (add to selection, subtract, common part or replace) |
| Edit |  | Set of methods to perform geometrical operations (translation, rotation), copying of objects with translation/rotation/mirror projection and deleting a single or selected set of objects |

| | | |
|---|---|---|
| Draw |  | Set of graphical tools to draw segment lines, polylines, arcs, circles, create auxiliary planes, setting construction lines, graphical grid, snap options |
| Mesh |  | Set of methods to split/refine finite element mesh, create virtual mesh and convert virtual to real mesh at macro-element level |
| Tools |  | Set of methods to show coordinates of a selected point/node, to measure distance between selected points, find a normal vector to the face/edge, find vector joining two selected points/nodes |
| Overlaid meshes |  | Set of methods to handle overlaid mesh models (move elements/nodes between mesh layers) |
| Frequently used tools |  | Set of most frequently used tools placed at the right part of the graph pane; it includes zoom, standard selection cursor to pick objects, cursor to continue complex graphical selection that requires an intermediate moving or model rotation, the hand tool to rotate the model using trackball method, view all toolbar, delete the object(s) and setting finite element/subdomain (at macro level) properties |
| Show construction/excavation steps | | Option that allows to trace evolution of the computational model in time |
| Data verification | | Option that allows to trace all potential mistakes in the final computational model. Potential errors are listed in form of a listbox, in a special dialog box. In ost cases clicking on the listed error enforces selection of those objects that cause the error. |
| Settings | | Option that allows the user to set up colors, line thickness and other graphical preprocessor/postprocessor options but also the way warnings and errors during data generation are presented to the user |
| Help | | Load ZSoil help files |

### 2.3.1.1.2.1 FILE

**Window 2-73: Options in menu:`File`**

Any option in the menu `File` (see table below) can be invoked by

- selecting it from menu `File`
- pushing a toolbar icon placed on the taskbar
- shortcut

| Option | Toolbar | Shortcut | Description |
|---|---|---|---|
| `New` |  | | Create a new data set under preprocessor |
| `Open` |  | Ctrl+O | Open an existing project |
| `Save` |  | Ctrl+S | Save under current name |
| `Import geometrical model (*.inp)` | | | Import existing model and add it to the current one; the imported model can be translated by a given vector; nodes located at the same positions as nodes of the current model can be merged |
| `Import mesh` | | | Import nodes and elements from user prepared external text file (option needed to transfer data from another FE packages) |
| `Save model and return to main menu` |  | | Save current model and return to the main ZSoil application |
| `Exit without saving` | | | Quit without saving |

Window 2-73

### 2.3.1.1.2.2 WINDOWS

**Window 2-74: Options in ribbon menu:** `Windows`

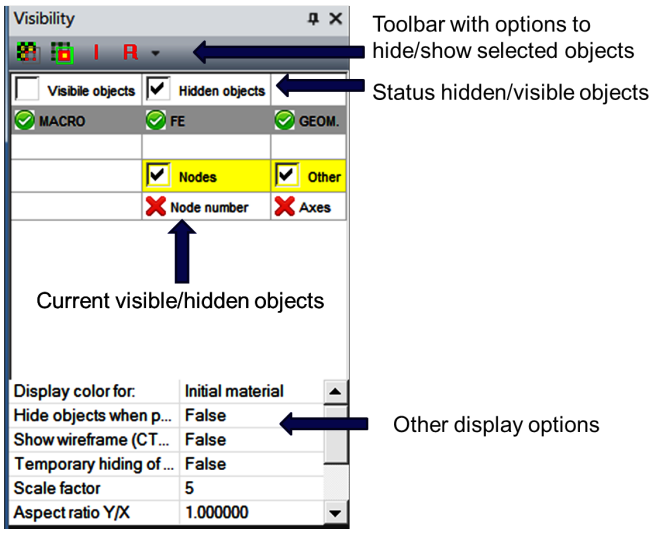
During model generation object of different classes are added. This may happen both at the macro but also at the finite element level. To trace all these model modifications five special dialogs/list boxes can be displayed on the screen. Only four of them can be visualized simultaneously. To put focus on a given set of information user can click on one of the five menu items listed in the following table

| Option | Toolbar | Description |
|---|---|---|
| Visibility |  | Puts focus on grid type dialog Visibility located at the top left corner of the preprocessor window |
| Output | | Puts focus on listbox Output placed at the bottom part of the preprocessor window; this listbox shows all the messages sent from preprocessor; every operation sends a message to that listbox |
| Mesh Info |   | Puts focus on Mesh Info dialog box placed at the left bottom corner of the preprocessor window; this grid type dialog shows all types of macromodel as well as finite element components present in the model |
| Project properties | | Puts focus on Project properties dialog box placed at the left bottom corner of the preprocessor window; this grid type dialog shows major project preselections like version type (basic/advanced), units system, analysis and problem types |
| Selected elements | | Puts focus on Selected elements listbox placed at the right bottom corner of the preprocessor window; this listbox shows all currently selected finite element components (nodes, elements etc..); |

Window 2-74

## Window 2-75: Dialog `Visibility`

In complex FE models (especially 3D) there is a strong need to hide some selected parts of the model. To enable that operation, apart from selecting tools, the following grid type dialog was designed.



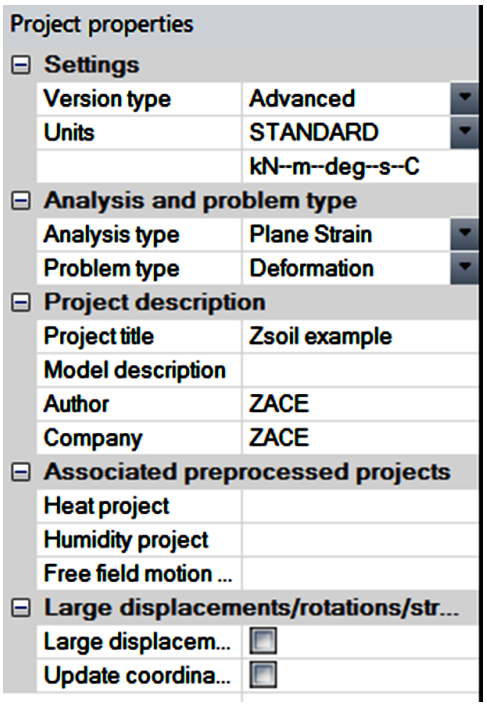

| Option toolbar | Description |
|---|---|
|  | This toolbar will activate hiding all objects that are currently selected |
|  | This toolbar will enforce displaying only these objects that are currently selected |
|  | Invert current visibility; objects that are hidden will be shown while visible will be hidden |
|  | Restore hidden objects; this operation can be made on all objects or selectively on specific classes of objects (only shell elements for instance) when clicking the right triangular button next to the toolbar icon |

**Remarks**:

1. Certain classes of finite elements (or macro-elements) can be displayed with colors that corespond to the `Initial material`, `Existence function`, `Unloading function` or `1-st/2-nd replacement material`; this option can be activated by using 1-st option in combo box placed in `Other display options`
2. Some finite element components (loads for instance) are associated with another objects (finite elements for instance) called here as parent ones; therefore displaying them can be set as coupled or uncoupled; this option can be activated by using 2-nd combo box placed in `Other display options`
3. To hide selected objects temporarily use shortcut (Ctrl-T); status of this operation is shown in dialog `Other display options`

**Window 2-76: Dialog** `Project properties`

The following grid type dialog shows basic project properties




**Remarks**:

1. Same dialog can be accessed from the main ZSoil application
2. To associate with a given project the thermal/hygral or free field motion project click on the edit field placed next to the text item in section `Associated...`

Window 2-76

## 2.3.1.1.2.3 ASSEMBLY

**Window 2-77: Options in ribbon menu:** `Assembly`

This menu includes most frequently used options from `Control` and `Assembly` menus of main ZSoil application. The curent `Assembly` menu includes following options

| Option | Toolbar | Description |
|---|---|---|
| `Project preselection` |  | Set major project options, type of problem and analysis, customizes graphical preprocessor options (see Win.2-6) |
| `Materials` |  | Edit materials properties (see Win.2-817) |
| `Existence functions` |  | Edit existence functions (see Win.2-919) |
| `Load functions` |  | Edit load time functions, filter acceleration records (Butterworth filtering+baseline correction) (see Win.2-921) |
| `Drivers` |  | Set list of drivers (see sec.2.2.2.1) |
| `Control` |  | Set up nonlinear solvers to be used in the computation, results storage frequency (see Win. 2-56) |
| `Gravity` |  | Set control of body forces and earth pressure (Ko) state (see Win. 2-924) |
| `Seismic input` |  | Set base accelerograms for shaking table approach (see Win. 2-925) |
| `Table of beam reinforcements` |  | Edit table of defined reinforcement sets for beams (see Win. 2-926) |

Window 2-77

### 2.3.1.1.2.4 UNDO

**Window 2-78: Options in ribbon menu:** Undo

| Option | Toolbar | Description |
|---|---|---|
| Undo |  | Undo option that allows the user to cancel maximum last 3 generation steps |
| Redo |  | Redo option that allows the user to cancel maximum last 3 undo steps |
| ☒ Undo active |  | Checked menu enabling undo operations (default is ON) |

**Remarks**:

1. Undo option is time consuming during data generation therefore for large 3D models, at some stages of the model generation, this option can be switched OFF

Window 2-78

## 2.3.1.1.2.5 VIEW

**Window 2-79: Options in ribbon menu:** `View`

Any option in menu `View` (see table below), which allows to control view and position of the camera with respect to the model can be invoked by

- selecting it from menu `View`
- shortcut

| Toolbar | Shortcut | Action |
|---|---|---|
|  | | Use orthogonal view |
|  | | Use perspective view |
|  | Ctrl+2 | Set view in XY plane |
|  | Ctrl+3 | Set view in XZ plane |
|  | Ctrl+4 | Set view in YZ plane |
|  | Ctrl+1 | Set axonometric view |
|  | | Set view in XY plane at **Position** [ ] |
|  | | Set view in XZ plane at **Position** [ ] |
|  | | Set view in YZ plane at **Position** [ ] |

**Remarks**:

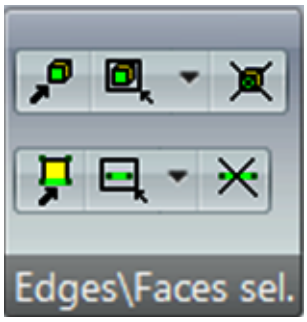
1. To see whole model on the screen use shortcut Ctrl+F or use mouse scroll
2. To rotate camera along X axis use key X or Shift+X to rotate in the opposite direction
3. To rotate camera along Y axis use key Y or Shift+Y to rotate in the opposite direction
4. To rotate camera along Z axis use key Z or Shift+Z to rotate in the opposite direction
5. To rotate camera use right toolbar (hand); to translate camera use toolbar (hand) and press Ctrl key (simultaneously)

Window 2-79

### 2.3.1.1.2.6 EDGES/FACES SELECTION

**Window 2-80: Options in ribbon menu:** `Edges/Faces selection`

Methods for edges and faces selection are organized in form of toolbars placed in two rows (see figure below). Only most frequently used options are visible. To activate more options a small button placed in 3-rd column has to be clicked.




Window 2-80

**Window 2-81: Faces selection toolbars**

| Toolbar | Description |
|---|---|
|  | Select/unselect one finite element face |
|  | Select finite element faces which are fully covered by the zoom box (infinite rectangular prism in 3D) once zoom is dragged from the left to the right side, or select all faces which are at least partially covered by the zoom box once zoom is dragged from the right to the left side |
|  | Deselect all selected faces |
|  | Select finite element faces which are fully covered by the zoom circle (infinite cylinder in 3D) once zoom is dragged from the left to the right side, or select faces which are at least partially covered by the zoom circle once zoom is dragged from the right to the left side |
|  | Select finite element faces which are fully covered by the irregular contour |
|  | Select finite element faces which are fully covered by the box designed by 2 picked nodes; box is always aligned with global XYZ axes |
|  | Select finite element faces through `Selection lists`; handling this selection list is very similar to the one for selection of elements (Window 2-84) or nodes (Window 2-85) |
|  | Select finite element faces which belong to the surface defined by 4 vertices constituting a macro-quadrilateral element; surface is interpolated with aid of standard bilinear shape functions used for Q4 element; face belongs to the surface if distance of any face node and its closest projection on the surface remains very small compared to the characteristic size of the macro-quadrilateral; after activation of this method 4 nodes must be picked by the graphical cursor; it is sufficient to pick any point close to the node to be selected in the element |
|  | Select finite element faces which belong to the surface spanned by 3 vertices constituting a macro-triangular element; surface is interpolated with aid of standard linear shape functions used for T3 element; face belongs to the surface if distance of any face node and its closest projection on the surface remains very small compared to the characteristic size of the macro-triangle; after activation of this method 3 nodes must be picked by the graphical cursor; it is sufficient to pick any point close to the node to be selected in the element |
|  | Select visible faces only; this option can be very useful if we want to select irregular surface of a given soil/rock layer on which some objects (nodes, points) are to be projected |

**Remarks**:

1. Selection by zoom box, zoom circle, irregular contour, by box, by 3 nodes, by 4 nodes undergoes the `Mode` set in ribbon menu `Global selection tools` (add to selection, subtract etc...)
2. To select a face inside of the computational domain (in a unique way) one of two elements adjacent to the face must be hidden
3. Selection of a face inherits the information to which finite element this face belongs; this information is of primary importance when generating seepage elements (3D) and/or applying surface loads (3D)

Window 2-81

**Window 2-82: Edges selection**

| Toolbar | Description |
|---|---|
|  | Select/unselect one finite element edge |
|  | Select finite element edges which are fully covered by the zoom circle (infinite cylinder in 3D) once zoom is dragged from the left to the right side, or select all objects which are at least partially covered by the zoom circle once zoom is dragged from the right to the left side |
|  | Deselect all selected edges |
|  | Select finite element edges which are fully covered by the zoom circle (infinite cylinder in 3D) once zoom is dragged from the left to the right side, or select edges which are at least partially covered by the zoom circle once zoom is dragged from the right to the left side |
|  | Select finite element edges which are fully covered by the irregular contour |
|  | Select element edges along the line section defined by two picked nodes |
|  | Select finite element edges which are fully covered by the irregular contour |
|  | Select finite element edges through `Selection lists`; handling this selection list is very similar to the one for selection of elements (see Window 2-84) or nodes (see Window 2-85)) |

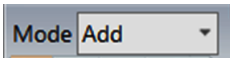
**Remarks**:

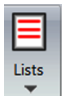
1. Selection by zoom box, zoom circle, irregular contour, inside box and between 2 nodes undergoes the rule set in the `Selection mode`
2. To select an edge inside of the computational domain (in a unique way) one of two elements adjacent to the edge must be hidden
3. Selection of an edge inherits an information to which finite element this edge belongs; this information is of primary importance when generating seepage elements (2D) and/or applying surface loads (2D)

Window 2-82

## 2.3.1.1.2.7 GLOBAL SELECTION TOOLS

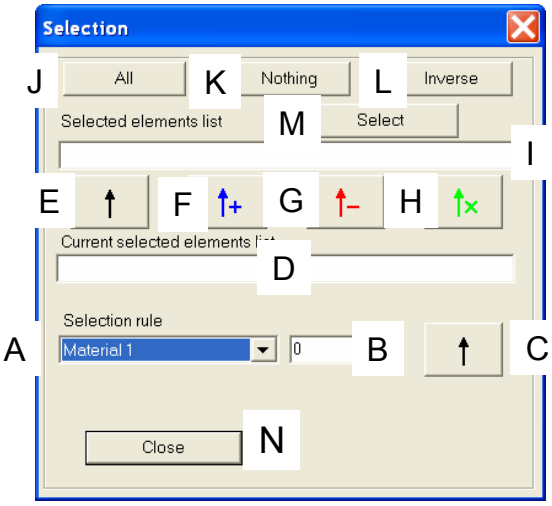
**Window 2-83: Options in ribbon menu:** `Global selection tools`

| Option | Toolbar | Description |
|---|---|---|
| `Mode` |   | • `Replace selection`: object which is not selected after picking it becomes selected and vice versa<br>• `Add to selection`: picked object(s) is (are) added to the set of selected objects<br>• `Subtract from selection`: picked object(s) become unselected and subtracted from set of selected objects<br>• `Common part`: objects which were already selected and are currently selected become selected |
| `Standard selection cursor` |  | This cursor allows to select any visible macro- or finite element component |
| `Selection by zoom box` |  | This option allows to select any visible macro- or finite element component by drawing a zoom on the graph pane; if the zoom is dragged from the left to the right then object must be fully covered by the zoom box in order to be selected; if the zoom is dragged from right to the left then it is enough that only small part of the object is covered by the zoom box |
| `Selection by zoom circle` |  | This option allows to select any visible macro- or finite element component by drawing a circle on the graph pane; if the circle is dragged from the left to the right then object must be fully covered by the circle zoom in order to be selected; if the circle is dragged from right to the left then it is enough that only small part of the object is covered by the circle zoom |
| `Selection within box` |  | This option allows to select objects that are inside of the box defined by two picked nodes |
| `Selection by contour` |  | This option allows to select any visible macro- or finite element component by drawing a closed contour on the graph pane; |

| | | |
|---|---|---|
| Selection of nodes between 2 selected nodes |  | This option allows to select nodes that are located along a segment line defined by two picked nodes |
| Selection by spatial polygon on 4 nodes |  | This option allows to select any finite element component that belongs to the spatial polygon defined by 4 nodes |
| Selection of all elements with material | | This option allows to select finite elements of any class with a given material number |
| Selection of all elements with existence function | | This option allows to select finite elements of any class with a given existence function |
| Selection of all elements with unloading function | | This option allows to select finite elements of any class with a given unloading function |
| Selection of element with a number from *.dat file | | This option allows to select finite element with a number indicated by a calculation module (useful when a certain element, during calculation, generates an error) |
| Selection of all objects |   | This option allows to select all visible macro- or finite element components in one action; in addition by clicking the small button below the icon one may use extra filters to select all objects of a given class (for instance continuum 2D or seepage elements etc..) |
| Unselecting all objects |   | This option allows to unselect all selected macro- or finite element components in one action; in addition by clicking the small button below the icon one may use extra filters to unselect all objects of a given class (for instance continuum 2D or seepage elements etc..) |
| by Lists |   | For detailed explanation on how to use lists to select finite elements or subdomains refer to Window (2-84) and for nodes/points to Window (2-85) |

**Remark**:For options: By zoom box, By zoom circle switching to orthogonal camera (in ribbon menu View) is strongly recommended

**Window 2-84: Unified dialog box for selecting finite elements/subdomains**




On the opening of the dialog box currently selected elements are all listed in field (I).

- To select all visible elements: press button `All` (J), then button `Select` (M) and `Close` (N)
- To unselect all visible elements: press button `Nothing` (K), then button `Select` (M) and `Close` (N)
- To inverse current selection: press button `Inverse` (L), then button `Select` (M) and `Close` (N)

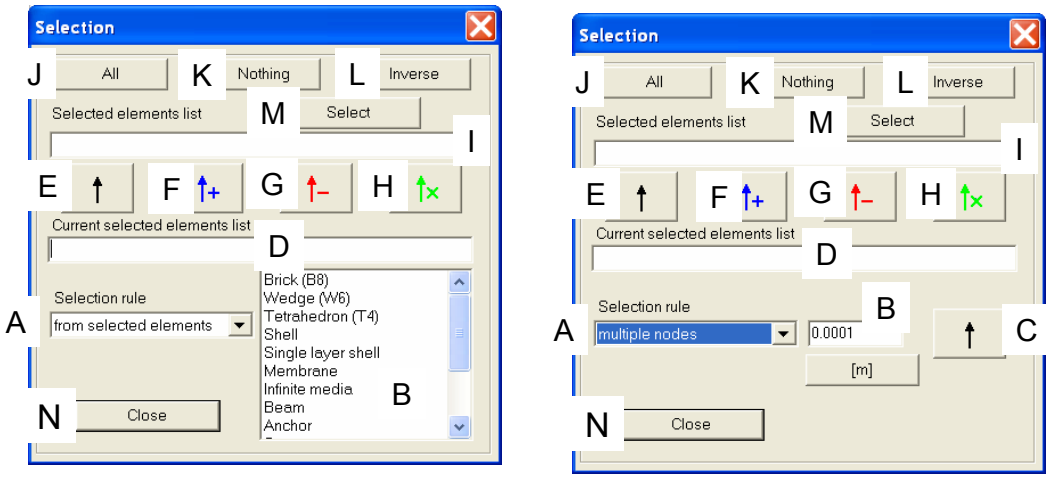
To perform more advanced selection follow the steps:

1. Select one of the available `Selection rules` from combo-box (A)
2. If this selection rule requires an additional customization select criterion from combo-box (B) or type proper data into the edit field (B) (note that depending on the selection rule field (B) may become and edit field (to input data) or a combo-box (to select criterion))
3. Use button (C) to find all objects which satisfy the selection rule and additional criterion; the newly selected elements will be placed in the field (D)
4. At this moment list of newly selected elements (D):
   - can replace (by pressing button (E)) the initial selection list (I)
   - can be added (by pressing button (F)) to the initial selection list (I)
   - can be subtracted (by pressing button (G)) from the initial selection list (I)
   - can be confronted with the initial selection list (I) (by pressing button (H)) and elements appearing in both lists will exclusively be selected
5. Using button `Select` (M) only elements appearing in the list (I) will be shown as selected in the graph pane
6. To make an inverse selection use button (L)) and then select the inverted list

**Remarks**:

1. The above steps can be repeated as many times as needed and this dialog becomes active until the `Close` button (N) is pressed
2. It is possible to edit both edit fields (D) and (I) by hands; The following list of elements (1,2,3,5,7,9,11) will appear in fields (D) and/or (I) as follows 1a3 5a11p2 (letter "a" is used here to define the range from-to while letter "p" defines the step in chain of numbers)

**Window 2-85: Unified dialog box for selecting nodes/points**




On the opening of the dialog box currently selected nodes are all listed in field (I).

- To select all visible nodes: press button `All` (J), then button `Select` (M) and `Close` (N)
- To unselect all visible nodes: press button `Nothing` (K), then button `Select` (M) and `Close` (N)
- To inverse current selection: press button `Inverse` (L), then button `Select` (M) and `Close` (N)

To perform more advanced selection follow the steps:

1. Select one of the available `Selection rules` from combo-box (A)
2. Select an additional criterion from combo-box (B) or type proper data into the edit field (B) (note that depending on the selection rule field (B) may become and edit field (to input data) or a combo-box (to select criterion))
3. Once the criterion is selected use button (C) (if this button is invisible action will be automatically run (see the above figure (a)) to find all objects which satisfy the selection rule and additional criterion; the newly selected nodes will be placed in the field (D)
4. At this moment list of newly selected nodes (D):
   - can replace (by pressing button (E)) the initial selection list (I)
   - can be added (by pressing button (F)) to the initial selection list (I)
   - can be subtracted (by pressing button (G)) from the initial selection list (I)
   - can be confronted with the initial selection list (I) (by pressing button (H)) and nodes appearing in both lists will exclusively be selected
5. Using button `Select` (M) only nodes appearing in the list (I) will be shown as selected in the graph pane
6. To make an inverse selection use button (L)) and then select the inverted list

**Remarks**:

1. The above steps can be repeated as many times as needed and this dialog becomes active until the `Close` button (N) is pressed
2. It is possible to edit both edit fields (D) and (I) by hands; The following list of nodes (1,2,3,5,7,9,11) will appear in fields (D) and/or (I) as follows 1a3 5a11p2 (letter "a" is used here to define the range from-to while letter "p" defines the step in chain of numbers)

### 2.3.1.1.2.8 EDIT

**Window 2-86: Options in ribbon menu:**`Edit`

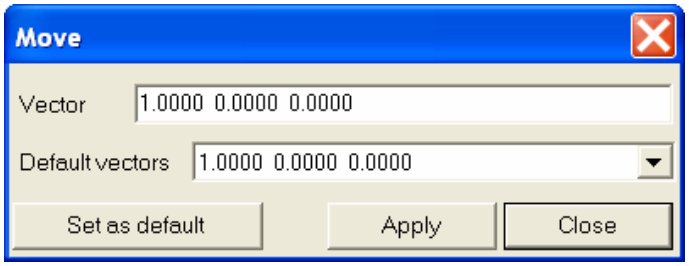
| Option | Toolbar | Description |
|---|---|---|
| `Delete selected` |  | This option is used to delete a single picked object or group of selected objects (use shortcut Del) |
| `Move selected` |  | This option is used to move a single picked object or group of selected objects by a given vector (see Window (2-87)) |
| `Rotate selected` |  | This option is used to rotate a single picked object or group of selected objects by a given angle along given axis (see Window (2-88)) |
| `Copy by translation` |  | This option is used to copy a single picked object or group of selected objects with a given vector ( see Window (2-89)) |
| `Copy by rotation` |  | This option is used to copy a single picked object or group of selected objects with a given vector ( see Window (2-90)) |
| `Copy by symmetry` |  | This option is used to copy a single picked object or group of selected objects by mirror projection ( see Window (2-91)) |

Window 2-86

**Window 2-87:** `Edit:Move selected`

This option applies to any type of the finite element, its node(s) but also to objects being part of the geometrical model. Translation vector is registered in the following dialog box and can be set in two ways:

- By clicking two points, defining the vector, in graph pane
- By direct input in the edit field `Vector` in the dialog



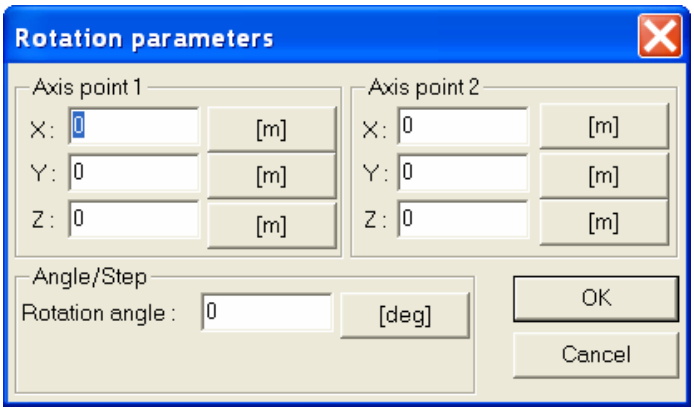

**Remarks**:

1. Current translation vector can be memorized and used as a default for next move actions by pressing the button `Save as default`
2. To accept the vector press button `Apply`
3. Last 5 vectors are kept for further usage and are accessible from combo box `Default vectors`

Window 2-87

**Window 2-88:** `Edit:Rotate selected`

This option applies to any type of the finite element, its node(s) but also to objects being part of the geometrical model. Rotation axis is defined by two points `Axis Point 1` and `Axis Point 2`. Rotation angle should be set as positive in one wants to rotate object(s) with the rotation vector oriented from point `Axis point 1` to `Axis point 2`




Window 2-88

**Window 2-89:** `Edit:Copy by translation`

This option applies to any type of the finite element, its node(s) and also to objects being part of the geometrical model.

- Copying operation is carried out along the vector `Direction` defined by three coordinates (`X, Y, Z`)
- Distance between two subsequent copies of object(s) is equal to `Step size` if `Use normalized` checkbox is set ON or to `Step size` times length of the `Direction` vector if `Use normalized` checkbox is set OFF
- Objects are copied `Number of steps` times
- Assuming `Regression` parameter equal to 1.0 (default) distance between any two subsequent copies is constant
- If `Regression` parameter is not equal to 1.0 then distance between subsequent copies and source object(s) is computed as a product of geometrical expansion with factor equal to the value of `Regression` parameter

Window 2-89

**Window 2-90:** `Edit:Copy by rotation`

This option applies to any type of the finite element, its node(s) and also to objects being part of the geometrical model.

- Copying operation is carried with rotation along the axis defined by two points `Axis point 1`, `Axis point 2`
- Angular distance between two subsequent copies of object(s) is equal to `Rotation angle`
- Objects are copied `Number of steps` times
- Rotation angle should be set as positive in one wants to copy objects with the rotation vector oriented from point `Axis point 1` to `Axis point 2`

Window 2-90

**Window 2-91:** `Edit:Copy by symmetry`

This option applies to any type of the finite element, its node(s) and also to objects being part of the geometrical model.

- To run this option a fictitious plane must be created first
- After this operation orientation (if applies) of copied objects is reversed
- To initiate the action plane of symmetry must be pointed first

Window 2-91

### 2.3.1.1.2.9 PARAMETERS

**Window 2-92: Options in ribbon menu:**`Parameters`

| Option | Toolbar | Description |
|---|---|---|
| `Parameters` |  | This option is used to set `Initial material`, `1-st replacement material`, `2-nd replacement material`, `Existence function`, `Unloading function` for a single picked element or group of selected elements (if all are of the same class) |

Window 2-92

## 2.3.1.1.2.10 DRAW

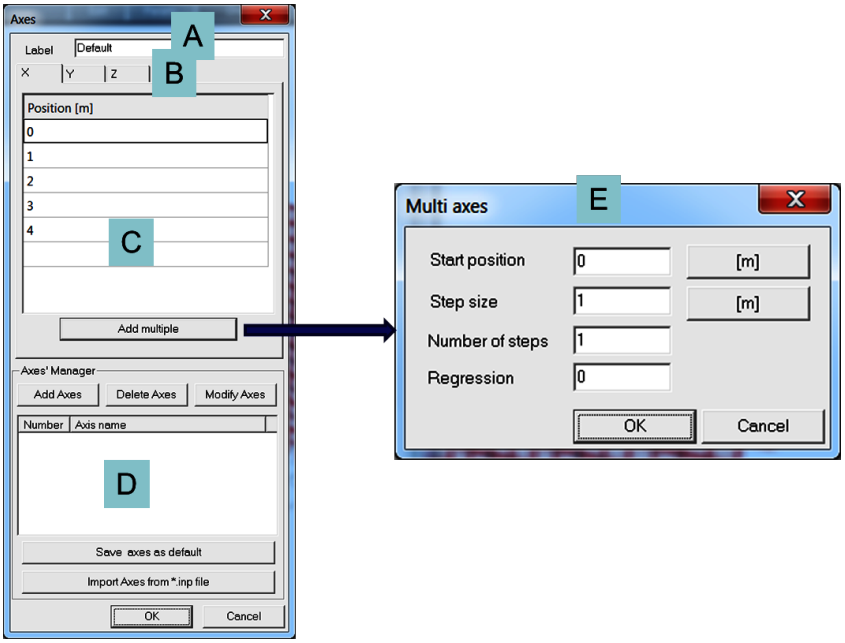
**Window 2-93: Options in ribbon menu:**`Draw`

| Option | Toolbar | Description |
|---|---|---|
| `Construction lines` |  | Set construction axes to simplify accurate drawing (through snap option) (see Window (2-94)) |
| `Grid` |  | Set grid density that may help to draw objects (see Window (2-95)) |
| `Snap options` |  | Set snap options to simplify accurate drawing (see Window (2-96)) |
| `Snap` |  | Set current snap status (ON/OFF) |
| `Point` |  | Create a point through graphical action or setting coordinates in the dialog box; this option is equivalent to the one invoked from right context menu `Macro-model/Point/Create/Point` (see Window (2-121)) |
| `Line` |  | Create a line segment or polyline through graphical action or setting coordinates in the dialog box; this option is equivalent to the one invoked from right context menu `Macro-model/Objects/Create/Line` (see Window (2-122)) |
| `Arc` |  | Create an arc through graphical action and/or setting data in the dialog box; this option is equivalent to the one invoked from right context menu `Macro-model/Objects/Create/Arc` (see Window (2-123)) |
| `Circle` |  | Create circle through graphical action and/or setting data in the dialog box; this option is equivalent to the one invoked from right context menu `Macro-model/Objects/Create/Circle` (see Window (2-124)) |
| `Rectangle` |  | Create rectangle through graphical action; this action is a kind of macro that creates set of four line segments |
| `Cube` |  | Create cube through graphical action; this action is a kind of macro that creates set of twelve line segments |
| `Auxiliary plane` |  | Create auxiliary planes (see Window (2-97)) |

Window 2-93

## Window 2-94: Draw:Construction lines

Using this dialog box one may define construction lines which are frequently used in the standard technical drawing. Construction lines (especially in the 3D) help the user to simplify drawing. Intersection points of construction lines are used to snap graphic cursor.



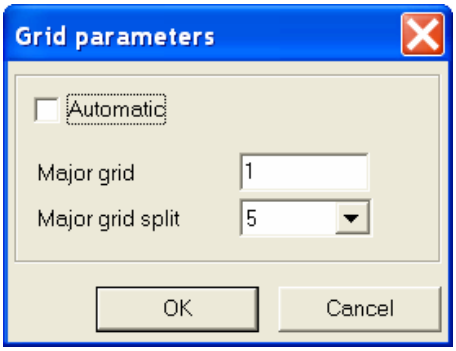

- To define construction lines along global X,Y,Z axis:
  - ⋆ Select an axis for edition (B)
  - ⋆ To create construction lines along selected axis in an automatic manner click a button `Add multiple` and set the data for generation in dialog box (E), otherwise introduce them one by one by setting the value in grid (C)
- To delete single construction line select it from list (C) and then press button `Delete`

**Remarks**:

1. User can keep several sets of construction lines in one model although only one set can be used (A)
2. To add new set of construction lines to the axes manager use button `Add axes`
3. To delete set of construction lines from axes manager use button `Delete axes`
4. To update selected (in list (D)) set of construction in axes manager use button `Modify axes`
5. To import set of axes from another project use button `Import axes from *.inp file`
6. To assume selected set of construction lines as a default one for all new projects use button `Set axes as default`

**Window 2-95:** `Draw:Grid`

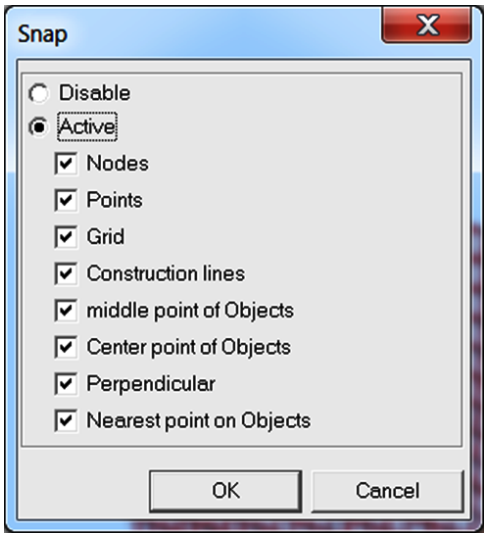
Grid option is available exclusively for 2D models or 3D ones but restricted to 2D view (in plane X-Y,Y-Z,X-Z). It is visible in the graph pane as gray dots to be used mainly for snap purposes. The dialog box for grid setup is shown in figure below. In the edit field `Major grid` user may define size of the grid expressed in length unit. Minor grid can be set in the edit field `Major grid split`.




Window 2-95

**Window 2-96:** `Draw:Snap options`

Position of the point when creating graphical objects like points, lines, polylines etc.. can be snapped to the grid, construction lines, existing points or existing finite element mesh nodes. All possible snap options can be set in the following dialog box shown below




Window 2-96

**Window 2-97:** Draw: Auxiliary plane

Auxiliary planes are frequently used for projection purposes. Standard operations on planes like moving, copying, rotating, deleting can be made by using toolbars from ribbon menu Edit.

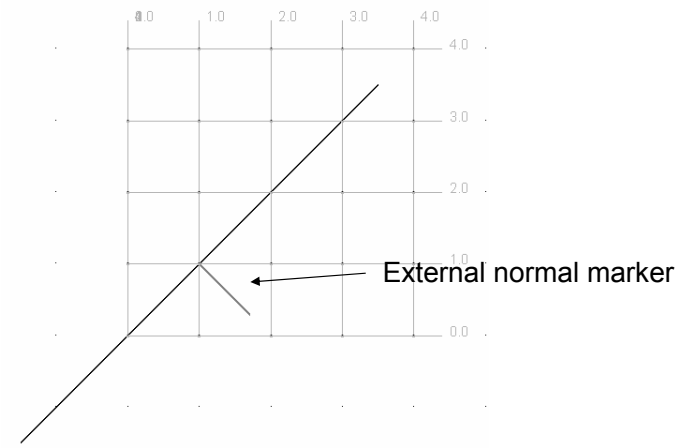
| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes/points | — | Window 2-98 |
| | 2 nodes | — | Window 2-99 |
| | From dialog | — | Window 2-102 |
| | 3 nodes/points | — | Window 2-100 |
| | 3 nodes | — | Window 2-101 |

Window 2-97

**Window 2-98:** `Auxiliary Planes:Create:2 nodes/points`

Using this option one may create an auxiliary plane by picking two points or the two existing nodes. The two picked points will be first snapped to the existing nodes of the real mesh projected on the graph pane and then to the grid points, if snap options are set ON. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create an auxiliary plane between the two existing nodes it is recommended to use the option `Create:2 nodes` described in Window 2-99.

This option is valid only for 2D applications. The newly created plane is drawn as a line segment with an orientation marker indicating the external normal to the plane.




Window 2-98

**Window 2-99:** `Auxiliary Planes:Create:2 nodes`

Using this option one may create an auxiliary plane by picking the two existing nodes. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

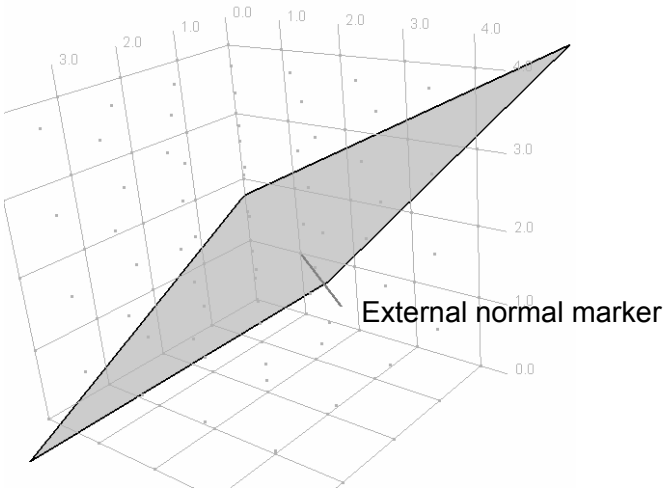
The newly created plane is drawn as a line segment with an orientation marker indicating the external normal to the plane (see Figure in Win.(2-99))

Window 2-99

**Window 2-100:** `Auxiliary Planes:Create:3 nodes/points`

Using this option one may create an auxiliary plane by picking three points or the three existing nodes. The three picked points will be first snapped to the existing nodes of the real mesh projected on the graph pane and then to the grid points, if snap options are set ON. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create an auxiliary plane based on three existing nodes it is recommended to use the option `Create:3 nodes` described in Window 2-101.

This option is valid only for 3D applications. The newly created plane is drawn as a rectangle with an orientation marker indicating the external normal to the plane.




Window 2-100

**Window 2-101:** `Auxiliary Planes:Create:3 nodes`

Using this option one may create an auxiliary plane by picking the three existing nodes. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

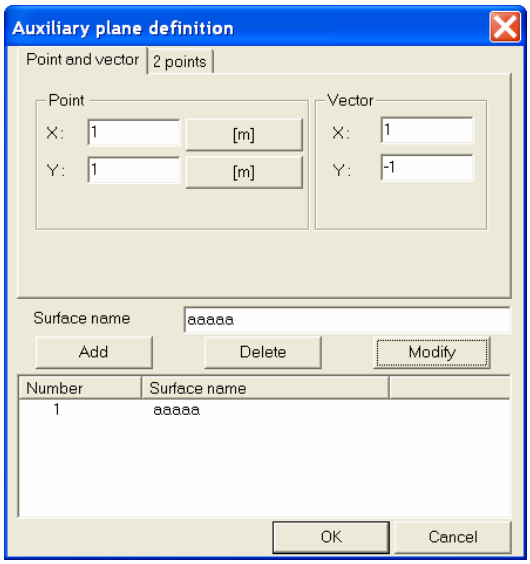
This option is valid only for 3D applications. The newly created plane is drawn as a rectangle with an orientation marker indicating the external normal to the plane (see Figure in Win.(2-101))

Window 2-101

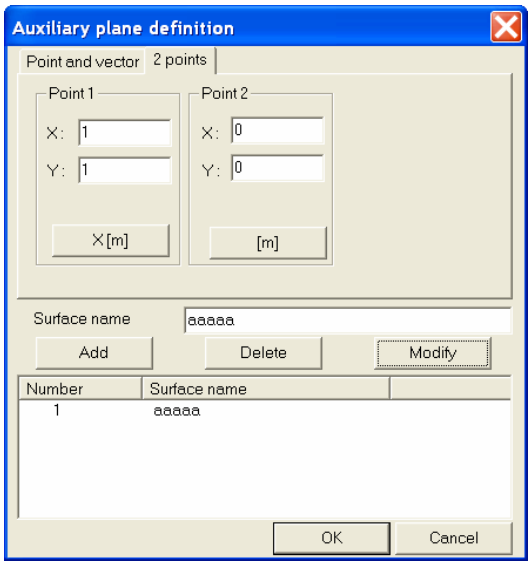
## Window 2-102: Auxiliary Planes:Create:From dialog

Using this option one may create an auxiliary plane by:

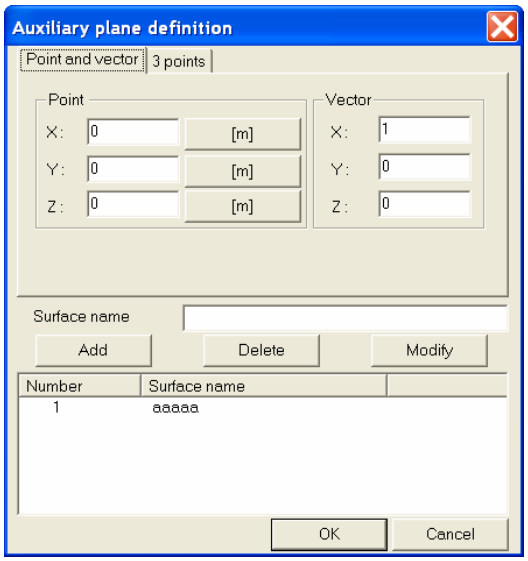
- specifying coordinates of two points (2D only)
- specifying coordinates of a point and vector normal to the plane(2D/3D option)
- specifying coordinates of three points (3D only)



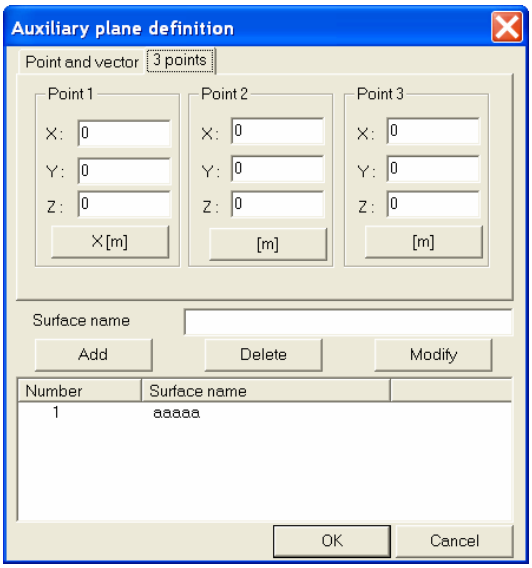

2D case (by point and vector)




2D case (by 2 points)




3D case (by point and vector)




3D case (by 3 points)

### 2.3.1.1.2.11 MESH

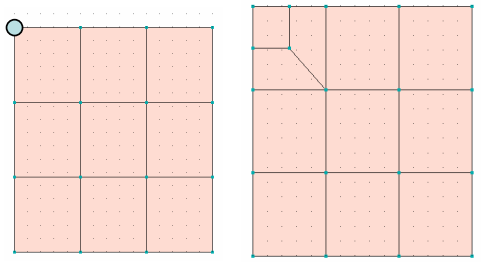
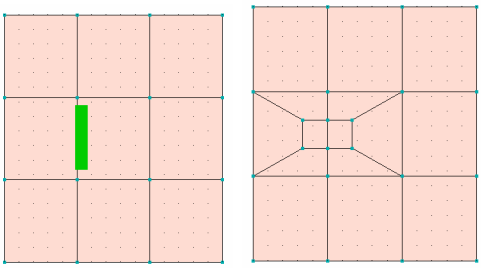
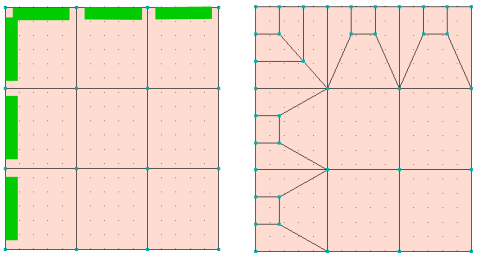
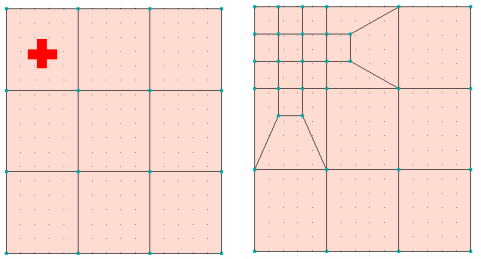
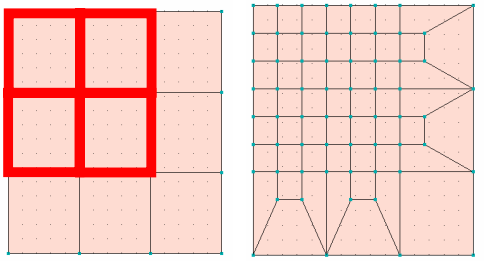
**Window 2-103: Options in ribbon menu:**`Mesh`

| Option | Toolbar | Description |
|---|---|---|
| `Split mesh` |  | Split picked finite element with a template corresponding to the selected element class (example: for Q4 element set split along 2 edges, for beam or anchor element set split value in one direction) |
| `Refine mesh` |  | Split picked or group of selected continuum finite elements(2D/3D), surface (shells/membranes)(3D) and one-dimensional elements (beams, anchors)(2D/3D) with a common split value equal to 3. This split is the only available one which preserves mesh conformity after refinement (for possible mesh refinement templates see Window 2-104 (2D) and Window 2-105 (3D)) |
| `Create virtual mesh` |  | This option allows to generate a virtual mesh in a picked subdomain or macro-beam/truss/membrane elements; it is equivalent to the option `Macro-model/Subdomain/Mesh/Create virtual mesh` (see Window (2-179) (2D) , Window (2-182) (3D) ) |
| `Virtual to real mesh` |  | This option allows to convert virtual mesh in a picked, or group of selected subdomains (macro-elements), to the real finite element mesh (see Window (2-190)) |

Window 2-103

## Window 2-104: Mesh refinement templates in 2D

The following 2D mesh refinement templates can be used for continuum quadrilateral elements.

| node | edge, edge and nodes |
|---|---|
|  |  |
| **selected edges** | **element** |
|  |  |
| **selected elements** | |
|  | |

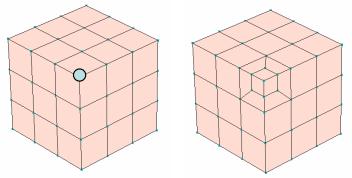
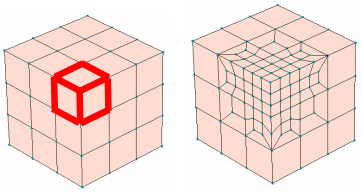
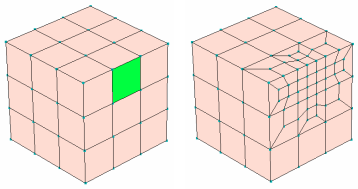
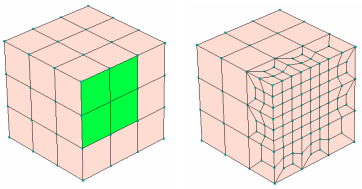
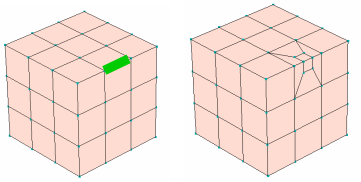
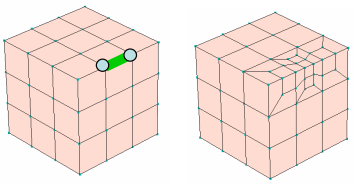
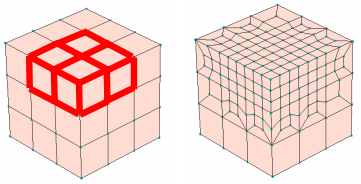
**Remarks**:

1. The above refinement applies exclusively to quadrilateral elements
2. Adjacent beams, anchors or membrane elements will be automatically split to preserve mesh conformity
3. Mesh refinement can exclusively be run for real meshes
4. The basic split value is equal to 3 and cannot be modified

Window 2-104

## Window 2-105: Mesh refinement templates in 3D

The following 3D mesh refinement templates can be used for continuum brick elements.

| node | element |
|---|---|
|  |  |
| face | selected faces |
|  |  |
| edge | edge and nodes |
|  |  |
| selected elements | |
|  | |

**Remarks**:

1. The above refinement applies exclusively to brick elements
2. Adjacent beams, anchors, shell or membrane elements will be automatically split to preserve mesh conformity
3. Mesh refinement can exclusively be run for real meshes
4. The basic split value is equal to 3 and cannot be modified

Window 2-105

## 2.3.1.1.2.12 TOOLS

**Window 2-106: Options in ribbon menu:** `Tools`

| Option | Toolbar | Description |
|---|---|---|
| `Vector` |  | Computes a vector between two nodes or geometrical points; this vector is frequently reused for operations like copying by translation or to set rotation axis |
| `Show distance between 2 nodes` |  | Computes a distance between two nodes or geometrical points |
| `Show vector between nodes` |  | Computes a vector joining two nodes or geometrical points |
| `Show node coordinates` |  | Shows coordinates of a picked node or geometrical point |
| `Show normal to face` |  | Shows coordinates of vector normal to the picked face |
| `Show normal to edge` |  | Shows coordinates of vector normal to the picked edge |
| `Compute area of selected element(s)` |   | Computes area of selected 2D continuum elements or 3D shells/membranes |
| `Compute volume of selected element(s)` |   | Computes volume of selected 3D continuum elements |

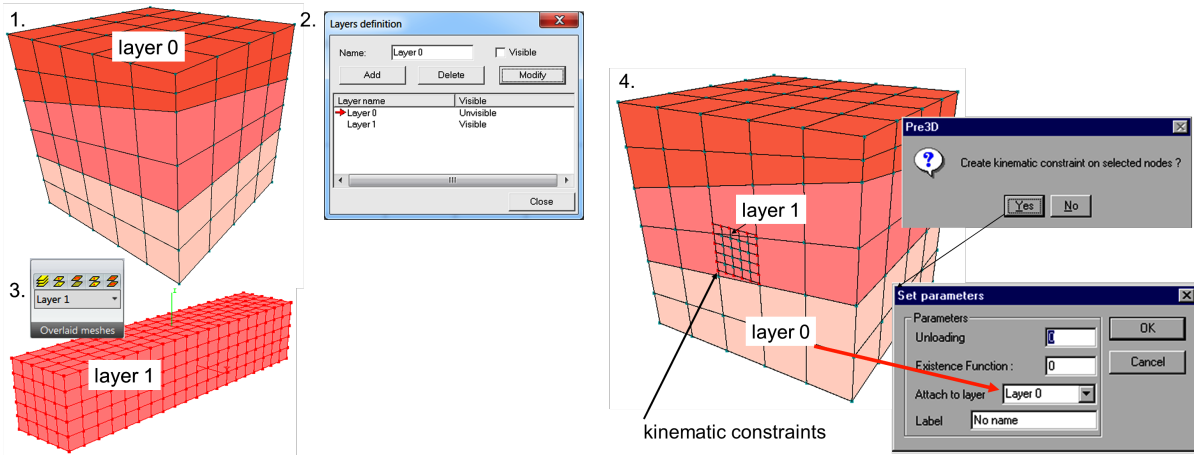
**Remark**:

Output produced by the above methods is shown in the `Output` window

Window 2-106

## 2.3.1.1.2.13 OVERLAID MESHES

**Window 2-107: Example of a model using overlaid meshes**




To generate overlaid mesh:

1. Build a coarse mesh for subsoil/rock mass. By default this mesh is attached to the `Layer 0` (step 1 in the figure)
2. Using toolbar `Layers` open layers manager; use `Add` button to create a new layer (labeled as `Layer 1`) and set visibility status for Layer 0 to OFF (while Layer 1 is visible) (steps 2)
3. Build dense mesh for zone of interest (tunnel for instance) (step 3)
4. Select boundary nodes of mesh in Layer 1 and generate kinematic constraints attached to Layer 0 (step 4)

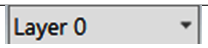
**Remarks**:

1. This option can be very useful for situations when we want to have dense meshes in zones of the structure and small part of adjacent subsoil/rock mass while the remaining part is discretized with a relatively coarse mesh (DSM columns for instance, tunnels etc..)

Window 2-107

**Window 2-108: Options in ribbon menu:** `Overlaid meshes`

All options in this menu help to handle finite element models composed of overlaid meshes.

| Option | Toolbar | Description |
|---|---|---|
| `Active layer` |   | Sets selected layer as an active one; all newly created objects will belong to that layer |
| `Layers` |  | Invokes the dialog box to add new or delete existing mesh layer and/or set up layers visibility status |
| `Move selected nodes on other layer` |  | Moves selected nodes from one layer to another one |
| `Move selected elements on other layer` |  | Moves selected finite elements from one layer to another one |
| `Copy selected nodes to another layer` |  | Copies selected nodes from one layer to another one |
| `Copy selected elements to another layer` |  | Copies selected finite elements from one layer to another one |

**Remark**:

## Related Topics

*HOW TO GENERATE KINEMATIC CONSTRAINT*

*OVERLAID MESHES*

## 2.3.1.1.2.14 FREQUENTLY USED TOOLS

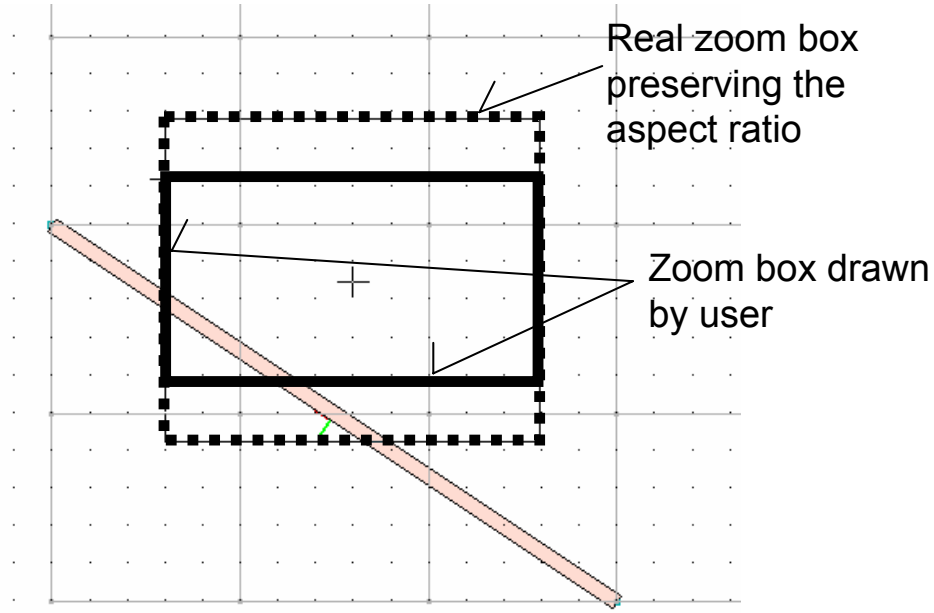
**Window 2-109: Options in ribbon menu:**`Frequently used tools`

| Option | Toolbar | Description |
|---|---|---|
| `Standard selection cursor` |  | Activates standard selection cursor which allows to pick any geometrical object or finite element model component |
| `Zoom` |  | This option allows to focus on a certain detail of the model; see for detailed information in Window (2-110) |
| `Camera manipulator` |  | Use like a track-ball to rotate or move camera ; in the 2D mode only translation is possible; in the 3D to cancel rotation use this toolbar with Ctrl key pressed down |
| `View all` |  | This option is used to view whole model |
| `Continue selection for last action` |  | Some generation methods (initial conditions on superelements for instance) require to pick several objects (points for instance) in order to finish the action; at the same time these objects may be hidden due to current projection; therefore we may need to rotate the camera first, just to see these objects, and then pick them to finish the action; this cursor is designed for picking objects |
| `Delete selected` |  | This option is used to delete a single picked object or group of selected objects (use shortcut Del) (NB. this toolbar appears also in `Edit` ribbon menu) |
| `Parameters` |  | This option is used to set `Initial material`, `1-st replacement material`, `2-nd replacement material`, `Existence function`, `Unloading function` for a single picked element or group of selected elements (if all are of the same class); the same toolbar is available in ribbon menu `Parameters` |

Window 2-109

## Window 2-110: Toolbar: Zoom

During definition of the zoom box the two rectangles are simultaneously drawn (see figure below), the blue (shown by dashed line) and black one (shown with solid line). The blue one is the one defined by the user while the black one represents the zoom which will finally be accepted to preserve the aspect ratio.

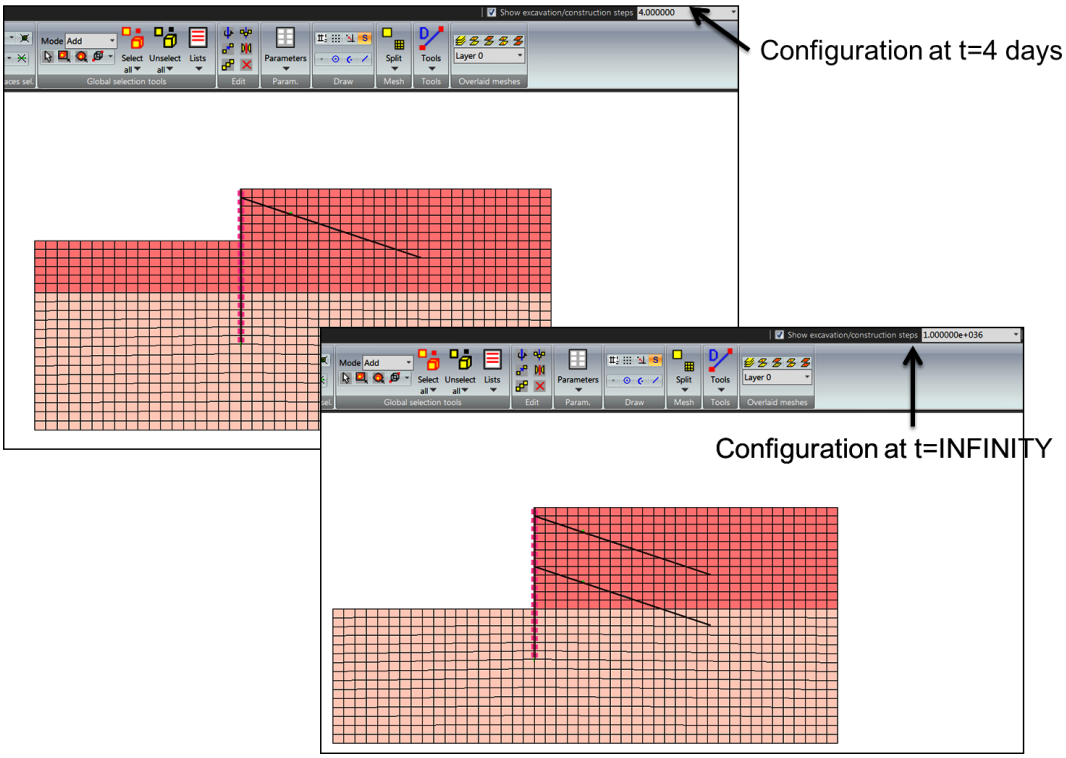



Window 2-110

## 2.3.1.1.2.15 SHOW CONSTRUCTION/EXCAVATION STEPS

This menu item activates verification of assumed construction/excavation stages in the generated computational finite element model. This option is valid for both macro and finite element level. All characteristic time instances are detected automatically from the active existence functions associated with finite elements and/or boundary conditions. An example of model verification is shown in Window (2-111)

**Window 2-111: Application of** `Show construction/excavation steps` **option**




**Remarks**:

1. During verification of construction/excavation steps all options from `Visibility` dialog box can be used

Window 2-111

### 2.3.1.1.2.16 SETTINGS

**Window 2-112: Options in top menu:**`Settings`

| Option | Description |
|---|---|
| `Warning manager` | Define set of warnings (see Window (2-113)) |
| `Preferences` | Customize palette of colors used for visualization of finite element components (see Window(2-114)) |

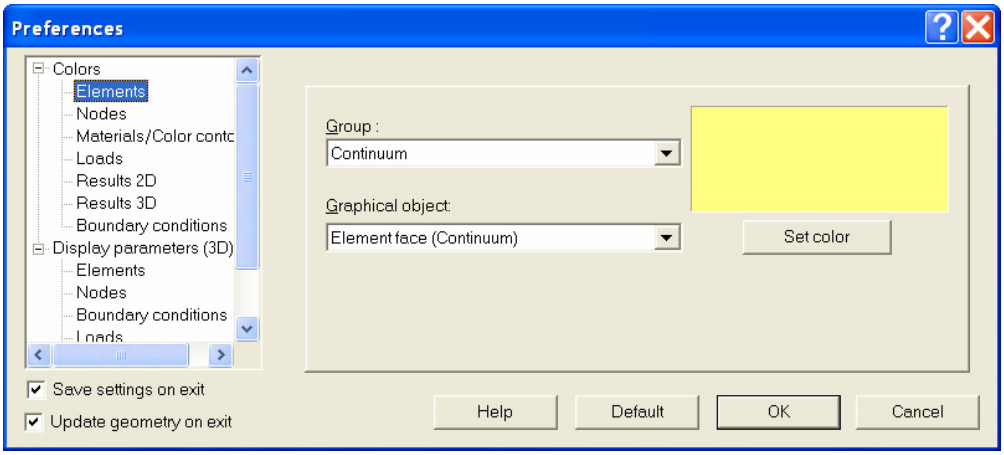
Window 2-112

**Window 2-113:** `Settings:Warnings`

In the invoked dialog box user may cancel certain unwanted warnings that are generated during data preprocessing.

Window 2-113

**Window 2-114:** `Settings:Preferences`




This dialog box (see figure above) allows the user to customize palette of colors used for visualization of different components of the model both in the pre- and postprocessor. In particular the setting may concern:

- different types of elements (continuum, interface, beams, shells etc.)
- different materials ( up to 20)
- boundary condition markers
- loads
- result items
- 20 color palette for color contours in postprocessor

To change predefined setting select an item in the list-box and then push `Set color`.

**Remark**:

All settings made in this dialog box can be saved by the user by switching ON the check-box `Save settings on exit`. All changes apply to any new project.

Window 2-114

### 2.3.1.2 MACRO-MODEL

The right context menu after clicking `DOMAIN/Macro Model` item gives an access to methods that can be used to create model components at the macro level. These components are listed in table shown below

| | |
|---|---|
| Point | Contains all options to create, select, update geometrical points |
| Objects | Contains all options to create, select, update geometrical objects like lines, arcs, circles |
| 2D meshing mapping | Contains all options to mapp any 2D mesh on a new 2D domain |
| Extrusion direction | Contains all options to select objects like lines, arcs to form so-called extrusion direction used later to extrude 3D subdomains from 2D faces, and 3D surface subdomains (shells, membranes) from 3D edges |
| Subdomain | Contains options to generate 2D/3D continuum subdomains, 2D/3D membrane subdomains, 3D shell subdomains, macro-trusses and macro-beams and to create meshes |
| Pile | Contains options to generate 3D piles (beam elements embedded in the continuum) |
| Nail | Contains options to generate nails in 2D/3D (beam elements embedded in the continuum) |
| Seepage | Contains options to generate macro-seepage on 2D/3D continuum subdomain boundaries |
| Heat exchanger | Contains options to generate macro-heat exchangers in 3D on selected geometrical objects |
| Convection | Contains options to generate macro-convection on 2D/3D continuum subdomain boundaries |
| Viscous damper | Contains options to generate macro-viscous damper on 2D/3D continuum subdomain boundaries |
| Interface | Contains options to generate macro-contact interface on 2D/3D continuum subdomain boundaries |
| Pressure BC | Contains options to generate pressure BC on subdomain boundaries |
| Temperature BC | Contains options to generate temperature BC on subdomain boundaries |
| Humidity BC | Contains options to generate humidity BC on subdomain boundaries |
| Fluid flux | Contains options to generate fluid fluxes on subdomain boundaries |
| Heat flux/Power | Contains options to generate heat fluxes on subdomain boundaries or set extracted/injected power from heat exchangers |
| Humidity flux | Contains options to generate humidity fluxes on subdomain boundaries |
| Surface load | Contains options to generate surface loads on subdomain boundaries |
| Point load | Contains options to generate concentrated loads on geometrical points |

### 2.3.1.2.1 POINT

**Window 2-115: Options in menu: Point**

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | Point | — | Window 2-116 |
| Select | all options | — | Window 2-117 |
| Delete | all options | — | Window 2-118 |
| Update | all options | — | Window 2-119 |

Window 2-115

**Window 2-116:** Point:Create:Point

This option activates a graphic cursor with two(in 2D) or three (in 3D) orthogonal lines intersecting at current point position (see Window 2-96 for Snap to.. option).

Window 2-116

**Window 2-117:** Point**:options in menu** Select

| Method | Description |
|---|---|
| Select one object | this option activates graphical cursor in the selection mode and points are selected one by one by pointing them by mouse |
| All | all points are selected; result of this action depends on the status of Selection mode (see Window 2-83) |
| In zoom box | all points inside zoom rectangle are selected |
| In zoom circle | all points inside zoom circle are selected |
| Inside contour | all points inside irregular contour, drawn by mouse, are selected |
| Duplicated | all points occupying the same geometrical position (with a declared tolerance) are selected |
| Single points | all points not attached to any object (like line, circle, arc, polyline) are selected |

Window 2-117

**Window 2-118:** Point**: options in menu** Delete

| Method | Description |
|---|---|
| Delete | if at least a single object is selected then Delete operation removes selected points; otherwise points are removed one by one by pointing them by mouse |
| Delete single points | all points not attached to any object (line, arc, etc..) are removed |

**Remark**:

After deleting points, objects adjacent to these points are removed

Window 2-118

**Window 2-119:** `Point`**: options in menu** `Update`

| Method | Description |
|---|---|
| `Set points's position` | update coordinates of a picked point in the dialog box or click new point position in the graph pane |
| `Move Point` | translate a point or group of selected points by a vector<br>• vector can be input directly in the dialog box by putting 3 coordinates<br>• vector can also be set in a graphical way by clicking two points (`Start` and `End`) |
| `Rotate` | rotate a single point or group of selected points along an axis defined by two points `axis Point 1` and `Axis Point 2` with given `Rotation angle`<br>• Rotation angle should be set as positive in one wants to rotate object(s) with the rotation vector oriented from point `Axis point 1` to `Axis point 2` |
| `Project in plane` | make orthogonal projection of a point or group of selected points on a picked fictitious plane |
| `Project in direction` | make projection of a point or group of selected points on a picked fictitious plane in given direction |
| `Merge multiple points` | merge points occupying the same geometrical position (with a declared tolerance) |

**Remark**:Updating position of a point enforces an update of all objects adjacent to the given point.

Window 2-119

### 2.3.1.2.2 OBJECTS

**Window 2-120: Common methods for objects**

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | Point | — | Window 2-121 |
| | Line |  | Window 2-122 |
| | Arc |  | Window 2-123 |
| | Circle |  | Window 2-124 |
| | Line(s) on edge(s) | — | Window 2-125 |
| | Surface on Q4 skeleton | — | Window 2-126 |
| | Surface on T3 skeleton | — | Window 2-126 |
| | Find Surfaces intersection | — | Window 2-127 |
| | Copy with translation |  | Window 2-128 |
| | Copy with rotation |  | Window 2-129 |
| | Copy by symmetry |  | Window 2-130 |
| | Copy by projection on plane | | Window 2-131 |
| | Copy by projection on plane in direction | | Window 2-132 |
| | Split by plane | | Window 2-133 |
| | Import DXF | | Window 2-134 |
| Select | Options | — | Window 2-135 |
| Delete | Options | — | Window 2-136 |
| Update | Options | — | Window 2-137 |

Window 2-120

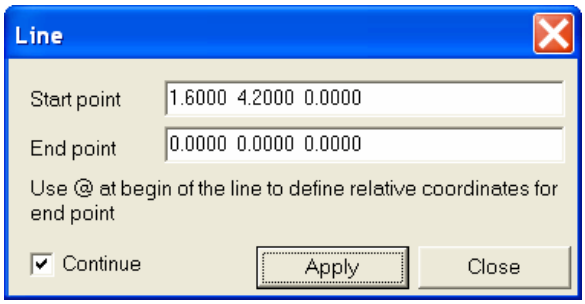
**Window 2-121:** `Objects:Create:Point`

This option activates a graphic cursor with two(in 2D) or three (in 3D) orthogonal lines intersecting at current point position (see Window 2-96 for `Snap to.. option`). Points are automatically created as endpoints for objects like lines and arcs

Window 2-121

**Window 2-122:** `Objects:Create:Line`

This option activates a graphic cursor with two(in 2D) or three (in 3D) orthogonal lines intersecting at current point position (see Window 2-96 for `Snap to.. option`). It may generate single line segment or polyline.




- To create polyline set checkbox ☒ **Continue** to ON; each newly picked point will be the endpoint of a new segment
- To create separated line segments in one run set ☒ **Continue** to OFF; in this case each new segment is created once 2 picked points are registered
- User may enter exact point coordinates in the edit fields for **Start point** [ ] and **End point** [ ]; the edited values are accepted by clicking button **Apply**
- Coordinates of the second point can be defined relative to the first one by putting the symbol @ in front of **End point** [ ];

Window 2-122

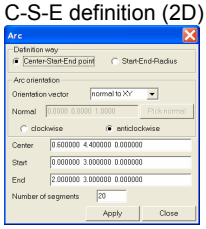
## Window 2-123: `Objects:Create:Arc`

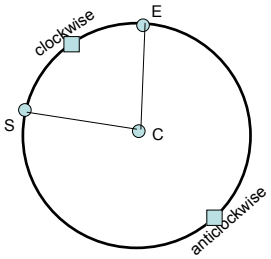
Arc can be defined using the two definitions:

- through center-start-end points (C-S-E), once ◎ **Center-Start-End** radio-button is ON (characteritic points can be picked in the graph pane or defined by exact coordinates in the dialog box)
- through start-end-radius (S-E-R) once ◎ **Start-End-Radius** radio-button is ON (the two endpoints can be picked in the graph pane or defined by exact coordinates, while radius must be explicitly entered in the dialog box)

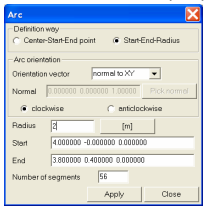
These definitions are supplied with an additional information concerning arc orientation. This information consists of a definition of an auxiliary normal vector $\overline{\mathbf{n}}^*$ plus `Clockwise` or `Anticlockwise` orientation status. For the 2D case the two definitions are illustrated in figures below.

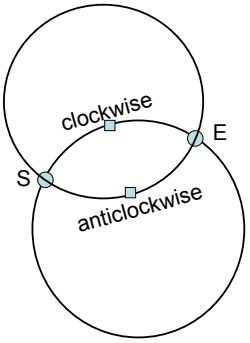
C-S-E definition (2D)





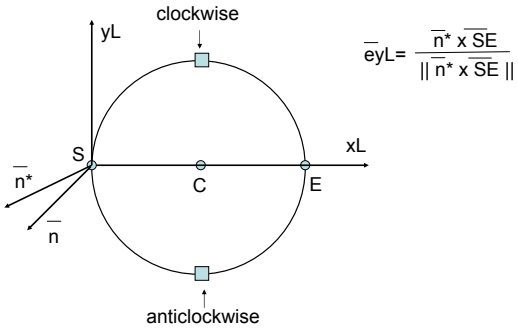

S-E-R definition (2D)





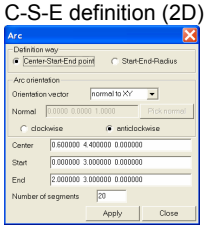

A singular case appears for C-S-E definition when C, S and E points are coaxial. In this case an auxiliary normal vector $\overline{\mathbf{n}}^*$, set in the dialog box in the edit field **Normal** [ ], helps to select proper arc. This case in shown in the figure below (for both 2D and 3D case)

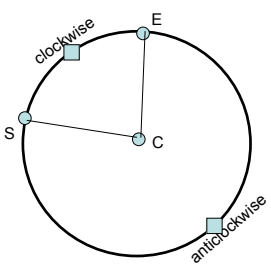
C-S-E definition (2D/3D): C, S, E are coaxial



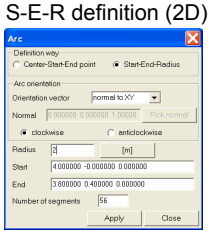

For the 3D case the two definitions are illustrated in figures below.

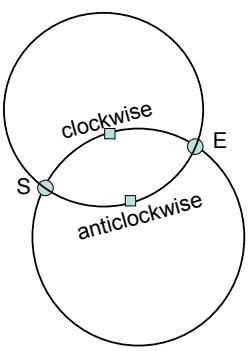
C-S-E definition (2D)



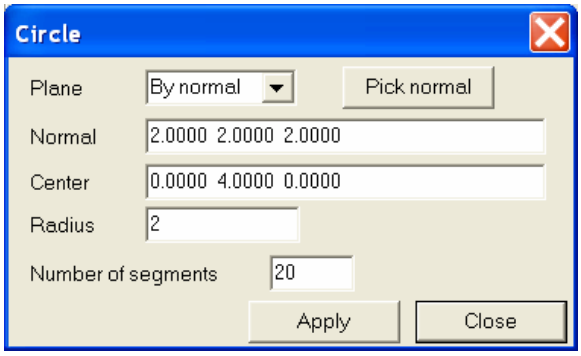



S-E-R definition (2D)





**Window 2-124:** `Objects:Create:Circle`




- Circle is defined by a `Center`, `Radius` and plane (X-Y, X-Z, Y-Z or other defined by a normal vector). (see user interface dialog box)
- User may define center and radius of the circle by drawing it in the graphic pane (valid if circle plane is defined by global X, Y, Z axes) or may define exact coordinates of the center and size of the radius
- `Number of segments` value (default 20) is exclusively used for visualization purposes (as circle is graphically represented by a closed polyline)

Window 2-124

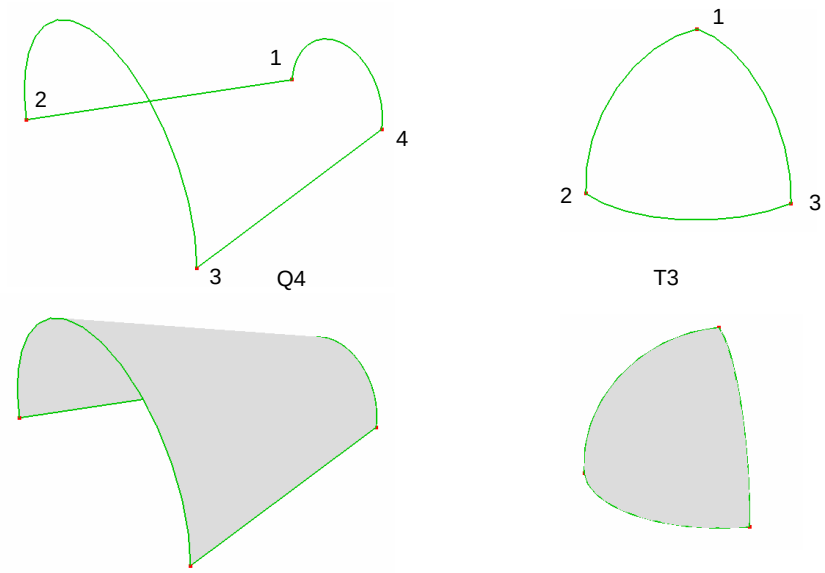
**Window 2-125:** `Objects:Create:Line(s) on edge(s)`

Using this option one may select edges from the existing finite elements and then create line segments on them. Created line segments have the same functionality as objects created through method : `Line`.

Window 2-125

**Window 2-126:** `Objects:Create:Surface on Q4/T3 skeleton`

Using these options one may create an auxiliary 3D surfaces bound by a closed form polygon, consisting of basic objects (lines and arcs), which forms a macro-quadrilateral or a macro-triangle respectively (see figure below)



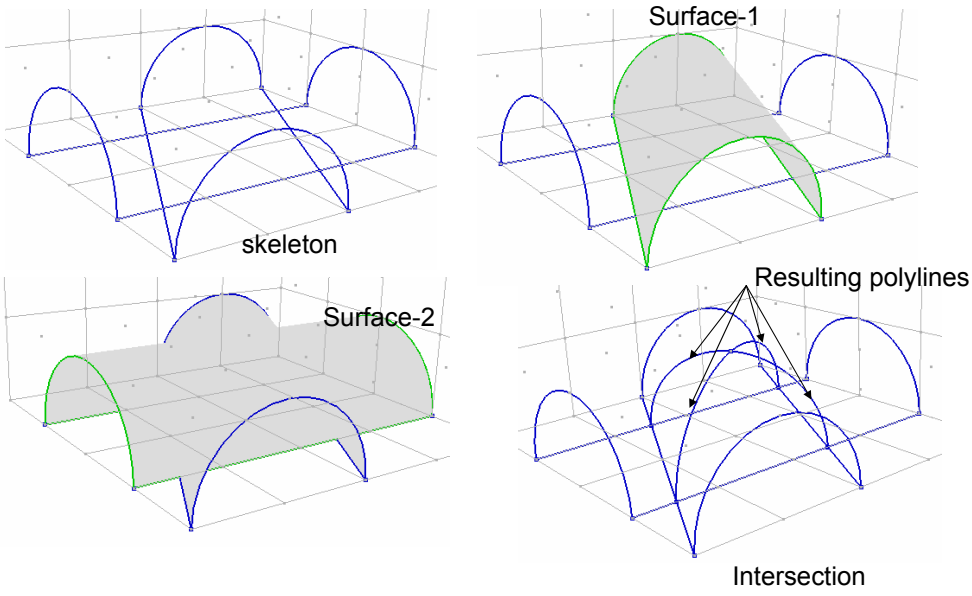

**Remarks**:

1. Conical surfaces created with aid of T3 or Q4 templates have an exact geometrical representation
2. Surfaces are auxiliary objects to be used for finding their intersections; they may also serve as target surfaces for projection of virtual meshes

Window 2-126

**Window 2-127:** `Objects:Create:Find surface intersection`

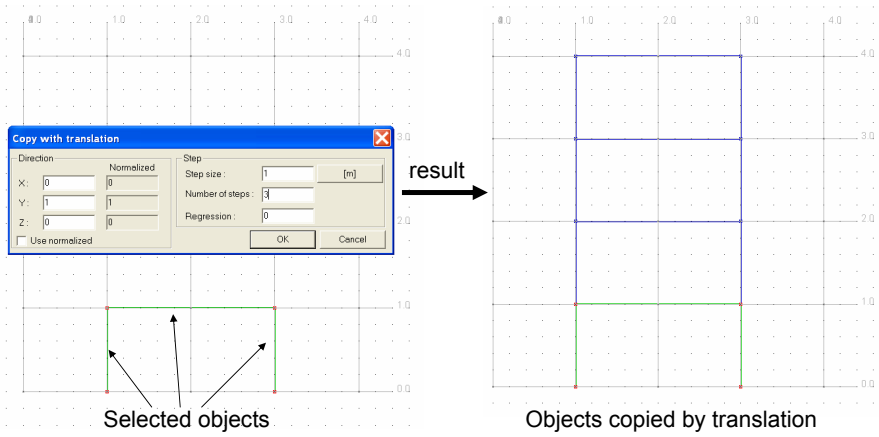
This option allows to obtain an intersection of the two picked T3 or Q4 type surfaces. The intersection takes a form of polylines treated as single objects (see example below).



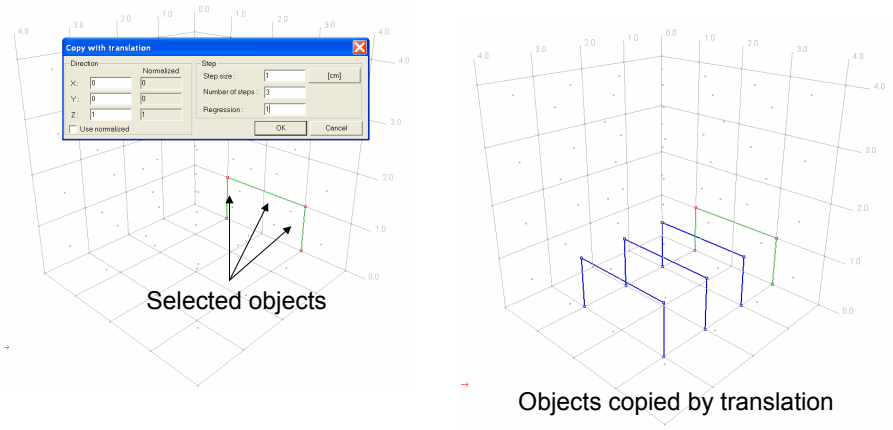

Window 2-127

**Window 2-128:** `Objects:Create:Copy with translation`

Example of `Copy with translation` in 2D




Example of `Copy with translation` in 3D



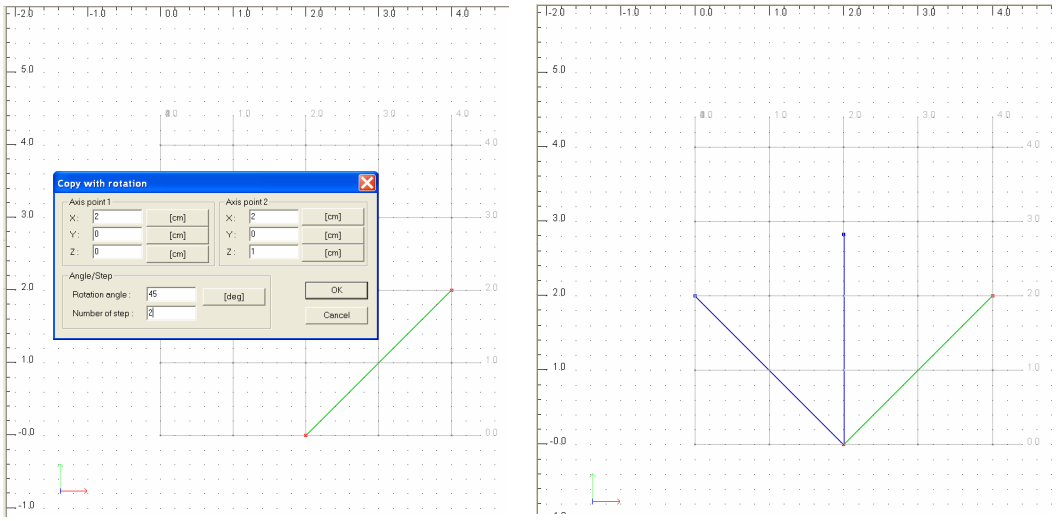

**Remarks**:

- Copying operation is carried out along the vector $\overline{\mathbf{v}}$ defined by three coordinates (`X`, `Y`, `Z`)
- $\overline{\mathbf{v}}$ is normalized (or not) depending on the status of ☒ **Use normalized** check-box
- if ◎ **One step size** is set ON then
  - ⋆ $\Delta L_o = h \quad \| \overline{\mathbf{v}} \|$
- if ◎ **Total size** is set ON then
  - ⋆ $L = h$
  - ⋆ if $s \neq 1$ then $\Delta L_o = \dfrac{L\,(1-s)}{(1-s^n)}$

    else $\Delta L_o = \dfrac{L}{n}$
- $\Delta L_i = s \;\; \Delta L_{i-1} \qquad$ for $i \geq 1$
- $L = \sum_{i=0}^{n-1} \;\; \Delta L_i$
- This option can be applied to a single object (if none of the objects is selected) and to selected objects (if at least a single object, of any type, is selected)

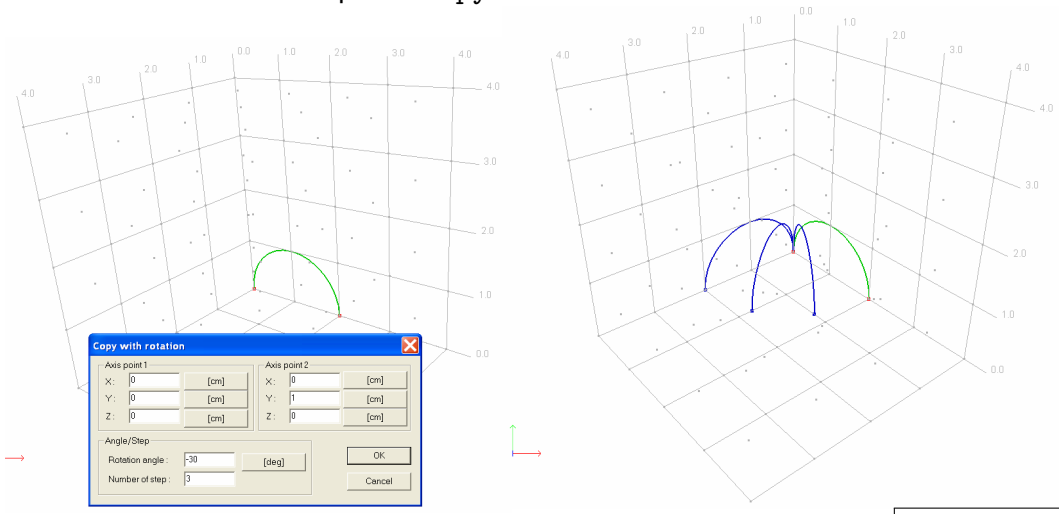
**Window 2-129:** `Objects:Create:Copy with rotation`

Example of `Copy with rotation` in 2D




- This option can be applied to a single object (if none of the objects is selected) and to selected objects (if at least a single object, of any type, is selected)
- Copying operation is carried with rotation along the axis defined by two points `Axis point 1`, `Axis point 2`
- Angular distance between two subsequent copies of object(s) is equal to `Rotation angle`
- Objects are copied `Number of steps` times
- Rotation angle should be set as positive in one wants to copy objects with the rotation vector oriented from point `Axis point 1` to `Axis point 2`

Example of `Copy with rotation` in 3D




Window 2-129

**Window 2-130:** `Objects:Create:Copy by symmetry`

This option can be used to make a copy of a single object (if none of the objects is selected) or group of selected objects by symmetrical projection with respect to the plane created under ribbon menu `Draw:Auxiliary Planes`

- To run this option a fictitious plane must be created first
- After this operation orientation of copied objects is reversed
- To initiate the action plane of symmetry must be pointed first

Window 2-130

**Window 2-131:** `Objects:Create:Copy by projection on plane`

This option can be used to make a copy of a single object (if none of the objects is selected) or group of selected objects by projecting it (them) on the plane created under ribbon menu `Draw:Auxiliary Planes`

- To run this option a fictitious plane must be created first
- To initiate the action the plane must be pointed first and the the object (if not selected)

Window 2-131

**Window 2-132:** `Objects:Create:Copy by projection on plane in direction`

This option can be used to make a copy of a single object (if none of the objects is selected) or group of selected objects by projecting it (them) on the plane in given direction

- To run this option a fictitious plane must be created first
- To initiate the action the plane must be pointed first and the the object (if not selected)

Window 2-132

**Window 2-133:** `Objects:Create:Split by plane`

Using this option user may split objects (lines, arcs, circles) into parts with aid of `Auxiliary Planes`.

- To run this option an `auxiliary plane` must be created first
- Action may be applied to a single object or set of selected objects
- If set of selected objects is empty then `auxiliary plane` is pointed first and then each object to be split one by one
- If set of selected objects is not empty then after pointing `auxiliary plane` all selected objects will be split
- This option is very useful when generating `Subdomains`

Window 2-133

**Window 2-134:** `Objects:Create:Import DXF`

- Objects of type: lines, arcs, circles and splines (converted to polylines) are imported from the *.dxf file
- dxf formats starting from AutoCAD 2000 are supported
- Not meaningful and overlapped objects should be deleted from the *.dxf file to avoid non-unique definitions in the preprocessor

Window 2-134

**Window 2-135:** `Objects`**:options in menu** `Select`

| | |
|---|---|
| `Select one object` | this option activates graphical cursor in the selection mode and objects are selected one by one by pointing them by mouse |
| `All` | all objects are selected; result of this action depends on the status of `Selection mode` (see Window 2-83) |
| `In zoom box` | • if the zoom box is dragged from left to the right side then only fully covered, by the zoom rectangle, objects are selected;<br>• if the zoom box is dragged from right to the left side then even partially covered, by the zoom rectangle, objects are selected |
| `In zoom circle` | • if the zoom circle is dragged from left to the right side then only fully covered, by the circle (cylinder in 3D), objects are selected<br>• if the zoom circle is dragged from right to the left side then even partially covered, by the zoom circle (cylinder in 3D), objects are selected |
| `Select surrounding contour` | with this option user may select groups of objects (lines and arcs) which form a closed polygon; to enable this selection user must pick any point inside of the polygon; this option is valid exclusively for the 2D case |
| `Circles/arcs with radius` | using this option one may easily select all circles and/or arcs with a given radius |

Window 2-135

**Window 2-136:** `Objects`**: options in menu** `Delete`

| | |
|---|---|
| `Delete` | if at least a single object is selected then `Delete` operation removes selected elements; otherwise objects are removed one by one by pointing them by mouse |

**Remark**:

After deleting lines and arcs their vertex points remain not removed; to delete them user can use the option `Delete Point` from menu `Point`

Window 2-136

**Window 2-137:** `Objects`**: options in menu** `Update`

| | |
|---|---|
| `Parameters` | redefines parameters of picked object (does not work for selection) |
| `Move` | translates an object or group of selected objects by a vector<br>• vector can be input directly in the dialog box by putting 3 coordinates and a factor (default=1.0) which scales the vector<br>• vector can also be set in a graphical way by clicking two points (`Start` and `End`) |
| `Rotate` | rotates a single pointed object or group of selected objects along an axis defined by two<br>• points `axis Point 1` and `Axis Point 2` with given `Rotation angle`<br>• rotation angle should be set as positive in one wants to rotate object(s) with the rotation vector oriented from point `Axis point 1` to `Axis point 2` |
| `Split` | splits an object or group of selected objects on parts<br>• number of subdivisions is defined in the edit field `Split`<br>• user may perform split operation with a regression parameter (default=1.0, no regression) defined in the edit field `Regression`<br>• split with regression must be run carefully because depending on the orientation of the object value of `Regression` parameter can be smaller than 1.0 or larger than 1.0<br>• special attention must be paid when making split with regression on group of selected objects as `Regression` parameter applies to all objects |
| `Project on plane`<br><br>`Project on plane in direction` | projects the object on the plane<br>• plane must be created using the `Auxiliary planes` option<br>projects the object on the plane in given direction<br><br>• plane must be created using the `Auxiliary planes` option |
| `Intersection by object(s)` | Finds intersection points of two pointed objects or all intersection points for group of selected objects<br>• after finding intersection points objects are automatically split into parts<br>• this option is crucial for definition of `Subdomains` |
| `Group lines to polylines` | groups all selected lines into one object treated as a polyline |

**Remark**:

After deleting lines and arcs their endpoints remain undeleted; to remove them user can use the option `Delete Point`

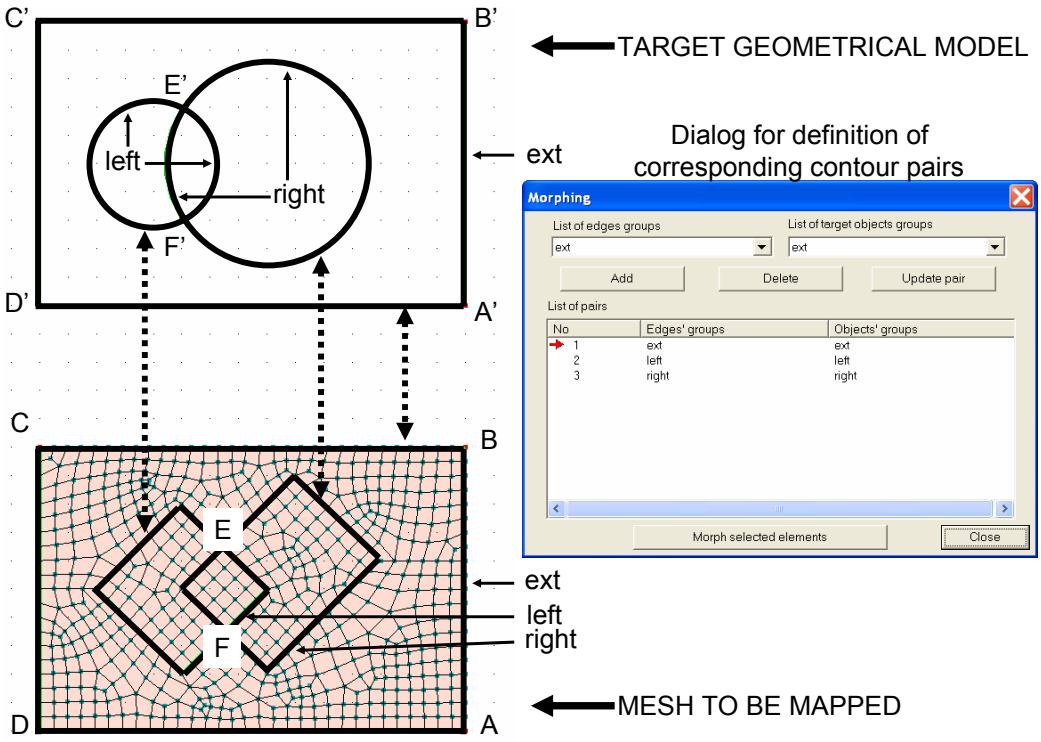
Window 2-137

### 2.3.1.2.3 2D mesh mapping

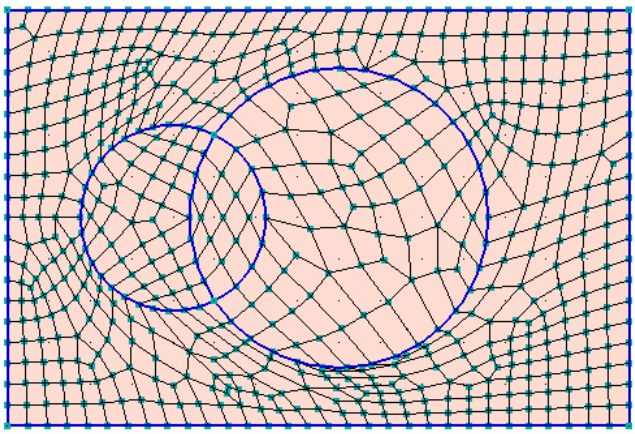
Using this option user can transform an existing 2D mesh onto target geometrical model built by objects of type: lines,arcs, circles. Option is very useful for 3D meshing when volumes between 2D meshes, placed at all characteristic cross sections, are connected. To mapp 2D mesh onto target geometrical model user has to indicate pairs of compatible contours (closed polygons of selected edges from 2D mesh and objects on target geometrical model) and pairs of so-called control points (intersections points, characteristic points).

**Example of method:** Window 2-138
**Options in menu** `Create`**:** Window 2-139
**Options in menu** `Select`**:** Window 2-140

**Window 2-138: 2D mesh mapping: example of method**



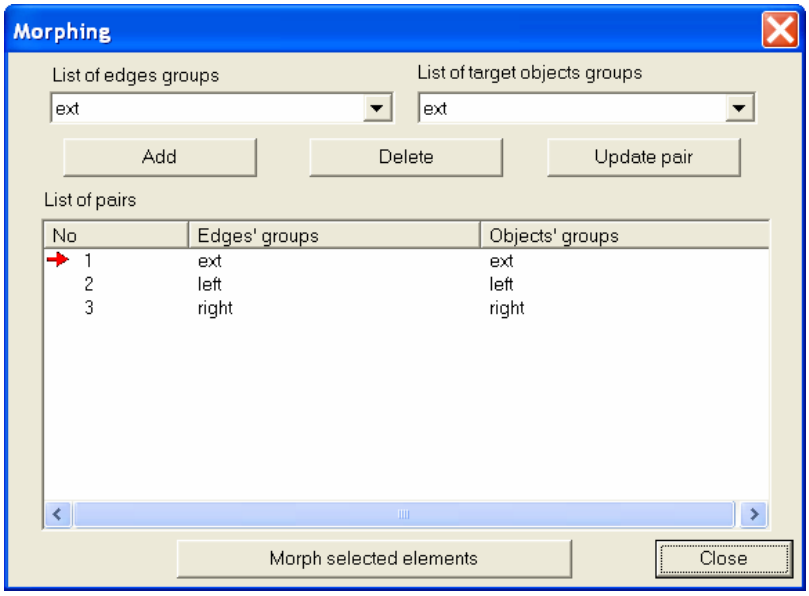

2D mesh mapped on target geometrical model



Window 2-138

**Window 2-139: 2D mesh mapping: options in menu `Create`**

| | |
|---|---|
| `Create edges list` | registers group of edges selected from 2D mesh<br><br>• Edges selected from 2D mesh must form closed polygon<br>• Selected edges cannot be overlapped<br>• In case when edges are placed in the 2D mesh interior it is recommended that elements on one side of the polygon formed by selected edges are hidden first; this is important when using `In zoom box`, `In zoom circle`, `In contour` methods from toolbar for edges selection<br>• When creating a list of selected edges user must label it for further usage |
| `Create target objects list` | registers group of objects which form a target geometrical model<br><br>• Selected objects must form a closed polygon<br>• Before registering a group of selected objects forming a polygon it is recommended to carry out the intersection of all objects; in the example shown in 2-138 the two circles are intersected first and cross points E' and F' are created (each circle is split into two arcs)<br>• Selected edges cannot be overlapped<br>• When creating a list of selected objects forming closed polygon user must label it for further usage |
| `Create control points` | registers pairs of equivalent points in 2D mesh and target geometrical model<br>• All intersections points must be registered (in the example shown in Window 2-138 points E-E' and F-F' must be registered)<br>• All characteristic points must be registered too (in the example shown in Window 2-138 these are the corner points: A-A', B-B', C-C' and D-D') |
| `Run morphing` | performs mapping of the 2D mesh onto target geometrical model<br>• Labeled groups of selected edges are kept under `List of edges groups` combo-box<br>• Labeled groups of selected target geometrical model objects are kept under `List of target objects groups` combo-box<br>• With button `Add` user may add a pair of compatible contours to the `List of pairs`<br>• With button `Delete` user may remove a pair of compatible contours from `List of pairs`<br>• With button `Update` user may modify a pair of compatible contours in `List of pairs`<br>• Morphing can be activated (with button `Morph selected elements`) once pairs of compatible contours in the 2D mesh and target geometrical model are defined, pairs of control points are defined and elements to be mapped from the 2D mesh are selected too(!)<br>• Only selected elements from the 2D mesh will be mapped<br>• After morphing the original mesh will fit boundaries of target geometrical model (see Window 2-138) |




Window 2-139

**Window 2-140: 2D mesh mapping: options in menu `Select`**

| | |
|---|---|
| `Edges with label` | selects all 2D mesh edges with a given label |
| `Target objects with label` | selects all objects of target geometrical model with a given label |
| `Control points with label` | selects pairs of control points (nodal point in the 2D mesh and geometrical point in the target geometrical model) with a given label |

**Remark**:

All selection modes apply (see Window 2-83)

Window 2-140

### 2.3.1.2.4 Extrusion direction

This option allows to create an extrusion direction being an oriented set of selected objects (line segments or arcs). This line is used later on for generation of a 2D (continuum)/3D (continuum, shell, membrane) subdomains. The extrusion direction must have the two distinct endpoints.

**Remark**:

The extrusion direction is a temporary object (it is not memorized in the input data).

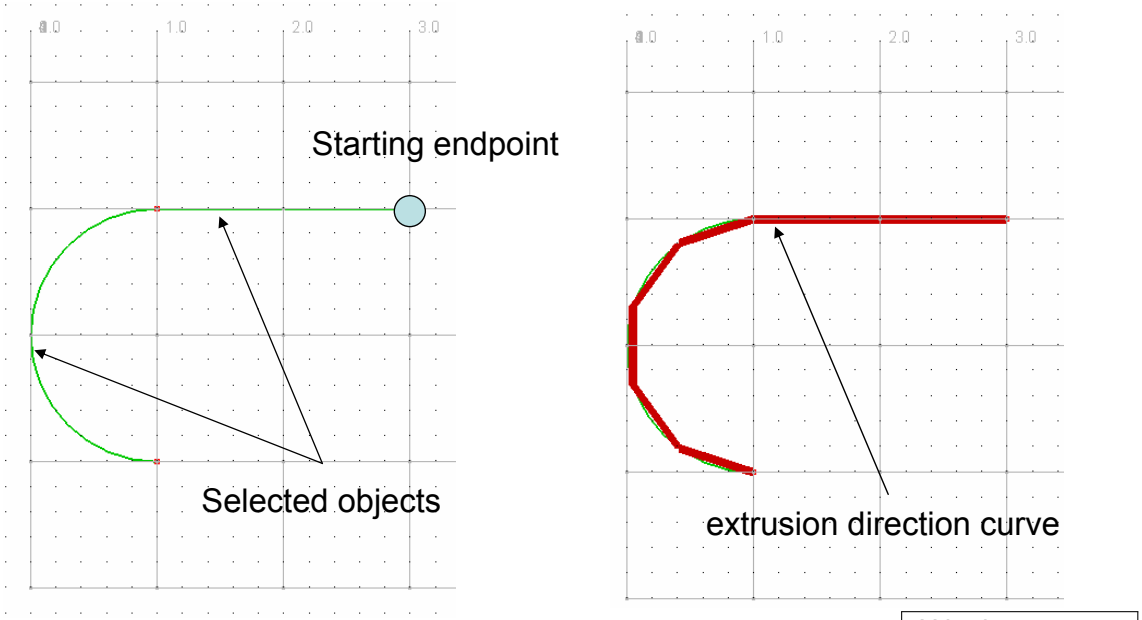
**Window 2-141:** `Subdomain`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | Extrusion dir. on sel.objects | — | Window 2-142 |
| Delete | — | — | Delete picked extrusion direction |

Window 2-141

**Window 2-142:** `Extrusion direction:Create:Extrusion dir. on sel. objects`

Using this option one may create an extrusion direction by selecting line segments or arcs which may form a complex curve in 2D/3D. To assume the orientation pick one of the endpoints to finish the action (see figure below)




Window 2-142

### 2.3.1.2.5 Subdomain

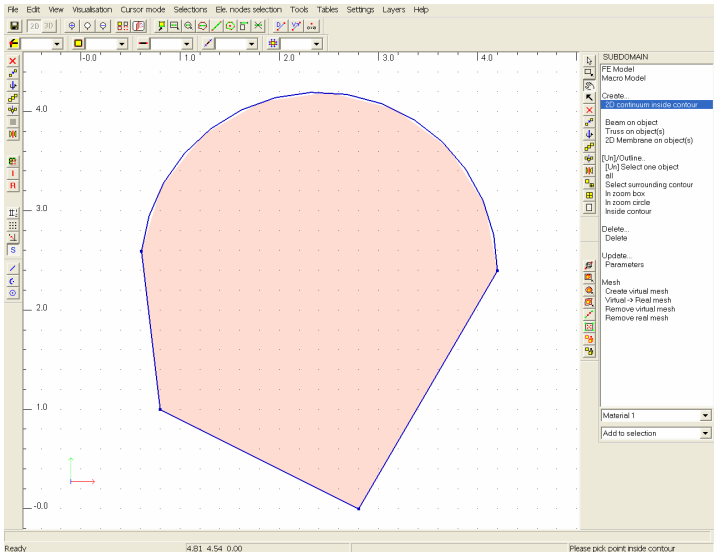
All methods grouped in menu `Subdomain` are designed to create, select and update subdomains of type continuum 2D, continuum 3D, shells, membranes, beams, trusses. In the context of macro-modeling, subdomains are understood as large parts of the computational model. User may define a subdomain, discretize it with a virtual mesh first and then with a real one, then may apply loads, fluxes and certain types of the boundary conditions to the boundary of the subdomain. Meshing procedure is treated here as a background operation, hence user may change the mesh and may update subdomain geometry while existing virtual or real mesh will be adapted to the new geometry.

**Window 2-143:** `Subdomain`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | 2D continuum inside contour | — | Window 2-144 |
| | 2D continuum on contour | — | Window 2-145 |
| | Beam on object(s) | — | Window 2-146 |
| | Truss/Anchor on object(s) | — | Window 2-147 |
| | 2D Membrane on object(s) | — | Window 2-148 |
| | 3D continuum on skeleton | — | Window 2-149 |
| | 3D continuum by face(s) extrusion | — | Window 2-151 |
| | 3D continuum between 2 faces | — | Window 2-150 |
| | 3D shell (one layer) on skeleton | — | Window 2-156 |
| | 3D shell (one layer) on face(s) | — | Window 2-157 |
| | 3D shell (one layer) by edge/object extrusion | — | Window 2-158 |
| | 3D membrane on skeleton | — | Window 2-163 |
| | 3D membrane on face(s) | — | Window 2-164 |
| | 3D membrane by edge/object extrusion | — | Window 2-165 |
| | Copy with rotation | — | Window 2-166 |
| | Copy with translation | — | Window 2-167 |
| | Copy by symmetry | — | Window 2-168 |
| Select | all options | — | Window 2-169 |
| Delete | all options | — | Window 2-170 |
| Parameters | Standard setup | — | Window 2-171 |
| | Setup for beams | — | Window 2-172 |
| | Define excavation front | — | Window 2-173 |
| | Reverse direction | — | Window 2-176 |
| | Reverse X-axis | — | Window 2-177 |
| | Thickness | — | Window 2-178 |
| Mesh | Create virtual mesh (2D) | — | Window 2-179 |
| | Create virtual mesh (3D) | — | Window 2-182 |
| | Virtual → Real mesh | — | Window 2-190 |
| | Remove virtual mesh | — | Window 2-191 |
| | Remove real mesh | — | Window 2-192 |
| | Project subdomain on surface | — | Window 2-193 |

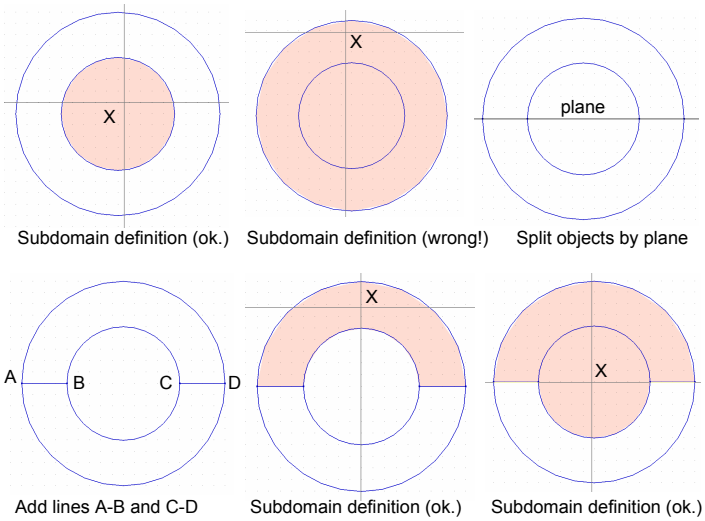
## Window 2-144: Subdomain:Create:2D continuum inside contour

To create a 2D subdomain a skeleton model consisting of basic geometrical objects (lines, arcs, circles) must be defined first. An example of definition of a 2D continuum subdomain is shown in the figure



**Remarks**:

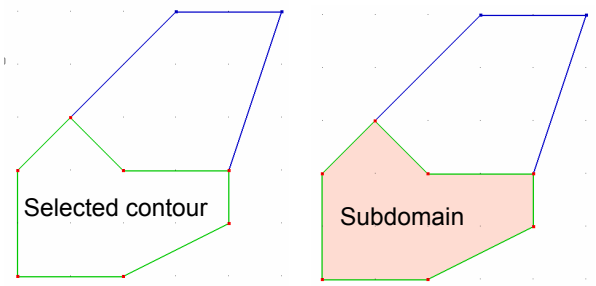
1. Contour must form a closed polygon
2. To create subdomain click any point inside of the closed polygon
3. Any action which updates geometry of any basic object which forms subdomain boundary updates subdomain geometry and its virtual/real mesh too (if defined)
4. Material number, replacement and second replacement material numbers are unique for all elements in subdomain
5. Troubleshooting: point clicked to define a subdomain inside of the contour is denoted by X




- Closed polygons cannot overlapp
- Use methods `Object:Split by plane` or `Object:Intersection by object(s)` to remove overlapping conflict
- If polygon is not closed the subdomain will not be generated and an error message is reported

**Window 2-145:** `Subdomain:Create:2D continuum on contour`

This option allows to create a 2D subdomain bound by the set of selected objects which must form a closed polygon. An example of this definition is shown in the figure




Window 2-145

**Window 2-146:** `Subdomain:Create:Beam on object(s)`

Using this option one can create a beam subdomain on a single picked geometrical object or group of selected objects.

**Remarks**:

1. Definition of the beam local coordinate system and offsets (of centroids with respect to the master nodes) has to be made using option `Update/Parameters` ( see Win.(2-172) for details) for each
2. Definition of material, replacement and second replacement material numbers, existence and unloading function numbers can be made using `Update/Parameters` option

**Window 2-147:** `Subdomain:Create:Truss/Anchor on object`

Using this option one can create a truss/anchor subdomain on a single picked geometrical object or group of selected geometrical objects. A more enhanced option for creating anchors including fixed anchor zones with interfaces is available at the FE level.

**Remark**:

Definition of material, replacement and second replacement material numbers, existence and unloading function numbers can be made using `Update/Parameters` option (see Win.(2-171))

Window 2-147

**Window 2-148:** `Subdomain:Create:2D Membrane on object`

Using this option one can create a 2D membrane subdomain on a single picked geometrical object or group of selected geometrical objects.

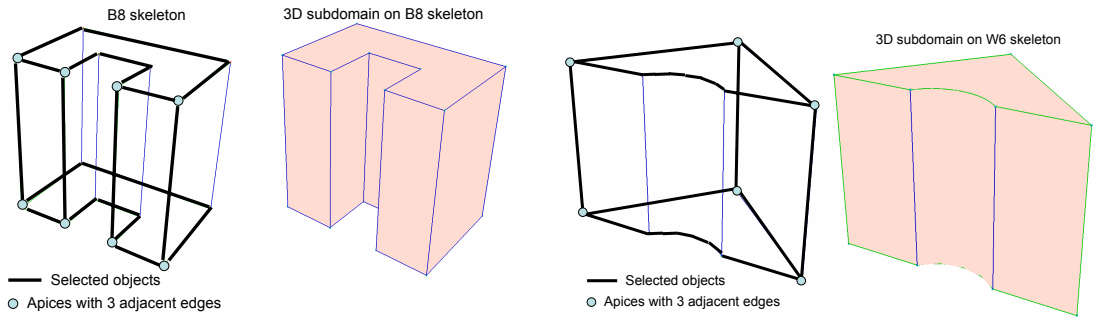
**Remark**:

Definition of material, replacement and second replacement material numbers, existence and unloading function numbers can be made using `Update/Parameters` option (see Win.(2-171))

Window 2-148

**Window 2-149:** `Subdomain:Create:3D continuum on skeleton`

To create a 3D continuum subdomain, using brick B8 or wedge W6 skeleton templates, a geometrical model consisting of basic geometrical objects (lines, arcs) must be defined first (see an example in figure below). In appices of a macro-brick or macro-wedge subdomain (see Figure) only 3 geometrical objects can be adjacent (!). This way 8 (for B8) or 6 (for W6) appices are automatically detected and macro-brick geometry is uniquely defined. Any deviation from that assumption will preclude creation of the subdomain.



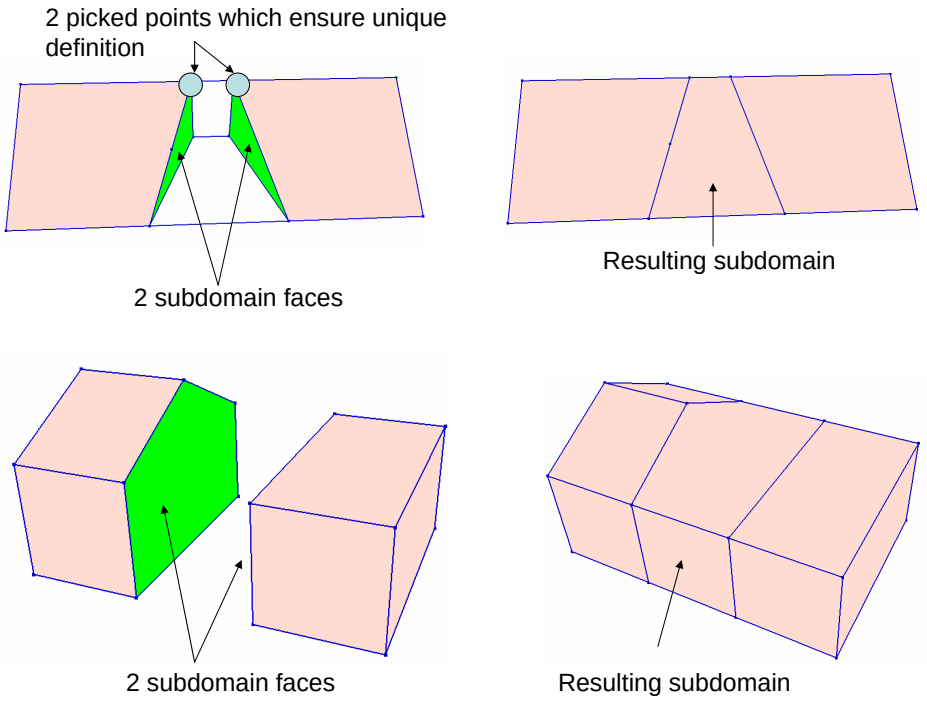

**Remark**:

Objects drawn with thick line can only be selected in the above example

Window 2-149

**Window 2-150:** `Subdomain:Create:3D continuum between 2 faces`

This method allows to create a 3D subdomain between two selected faces of the two existing subdomains (2D continuum, 3D continuum or 3D shell). In the following two figures a new continuum subdomain is created between the two selected faces of the same type (T3-T3) and (Q4-Q4). In the last figure a special case is considered when new 3D subdomain is created between the two unstructured 2D continuum subdomains. However, in this specific case both virtual meshes must fully conform although may be differ by means of the geometry.

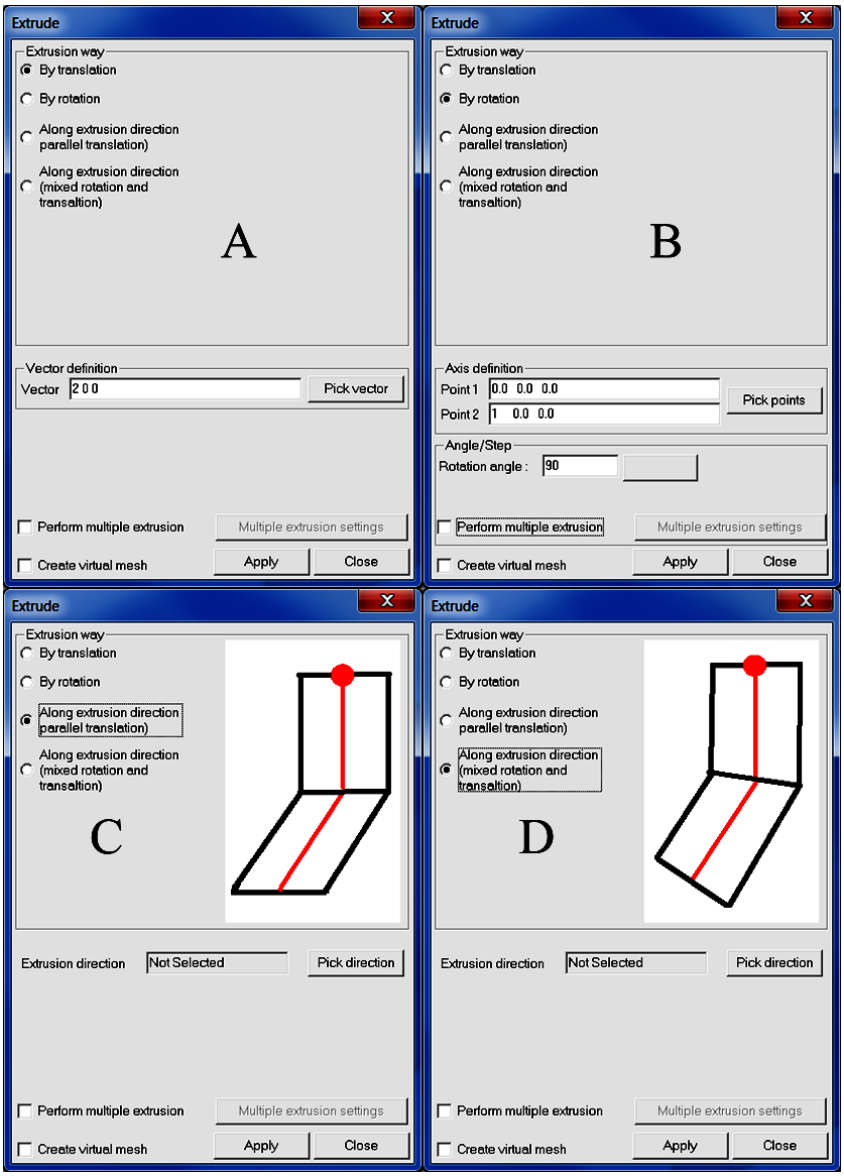



Window 2-150

**Window 2-151:** `Subdomain:Create:3D continuum by face(s) extrusion`

This method allows to create a new 3D subdomain by a single or multiple (along given extrusion direction) extrusion of selected subdomain (of any type) face(s). Four types of extrusion method can be used (see figures and corresponding Windows):

1. Single extrusion by translation along given direction (method A: Window 2-152)
2. Single extrusion by rotation along given axis (method B: Window 2-153)
3. Parallel translation along given direction of extrusion (multiple extrusion) (method C: Window 2-154)
4. Parallel translation combined with rotation along given direction of extrusion (multiple extrusion) (method D: Window 2-155)

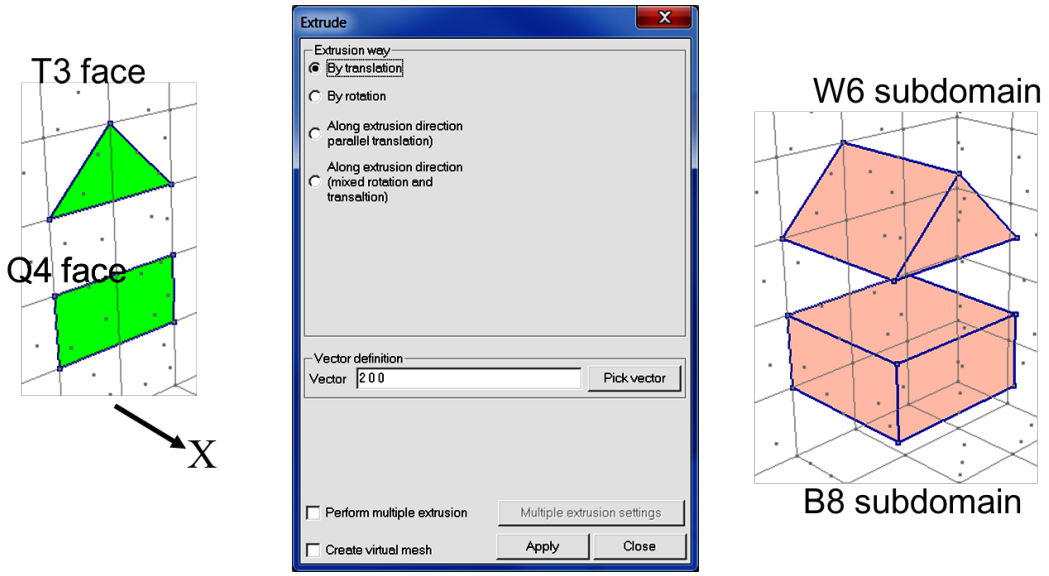



**Remark**:During subdomain generation a virtual mesh can be simultaneously created by setting ☒ **Create virtual mesh** to ON and declaring the split value along the extrusion direction (this edit field is invisible if ☒ **Create virtual mesh** is set to OFF).

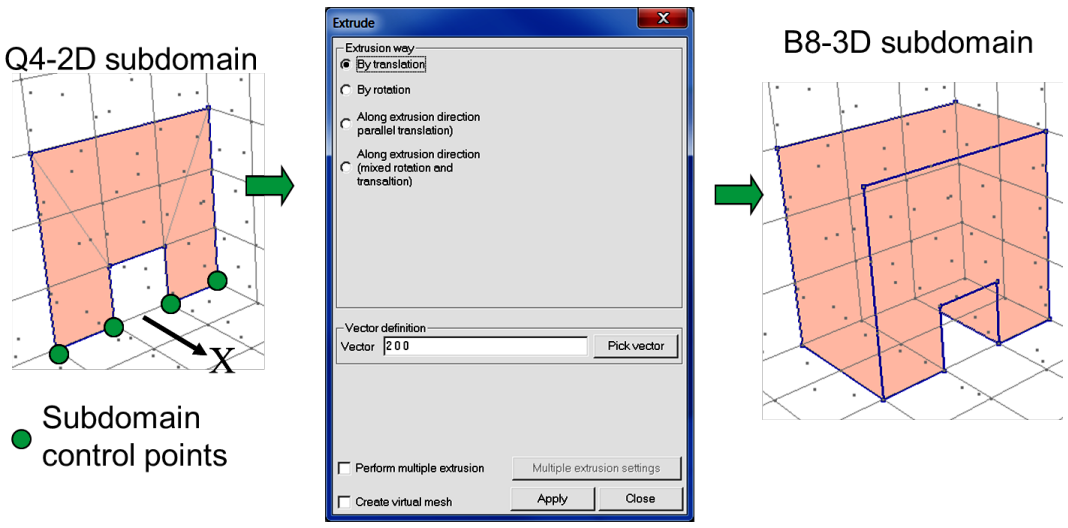
**Window 2-152:** `Subdomain:Create:3D continuum by face(s) extrusion: meth.A`

This method allows to create a new 3D subdomain by a single extrusion of selected subdomain (of any type) face(s) in given direction. Three possible cases have to be considered:

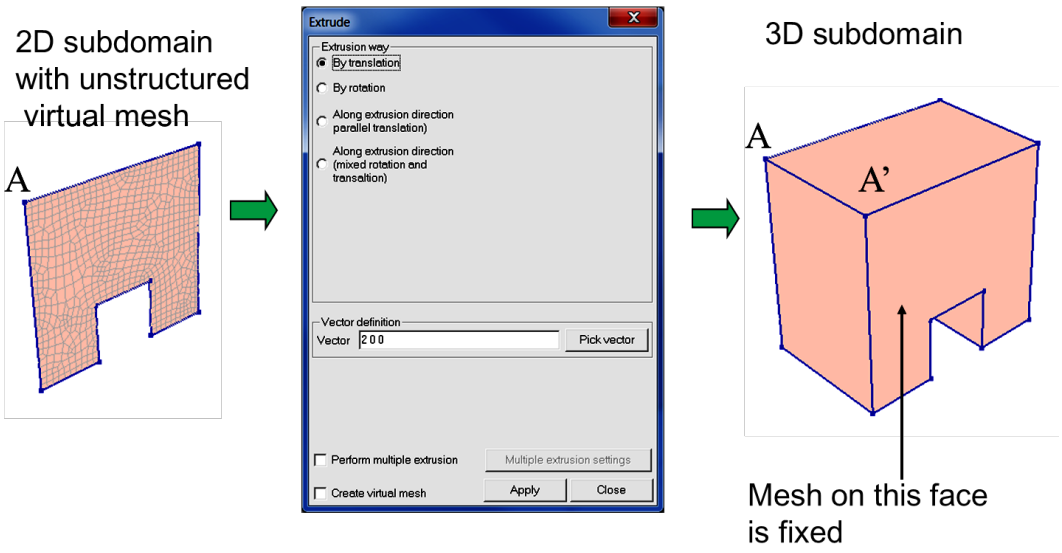
1. Face consists of 3 or 4 vertices and it is not yet virtually discretized; in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 with 8-node brick elements only




2. Face consists of any number of vertices and is already virtually discretized using structured mesh template (T3 or Q4); in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 using 8-node brick elements only




3. Face consists of any number of vertices and is already virtually discretized using unstructured mesh template; in this case a 3D continuum subdomain is created; the only free direction to modify mesh density is the direction of extrusion




**Remarks**:

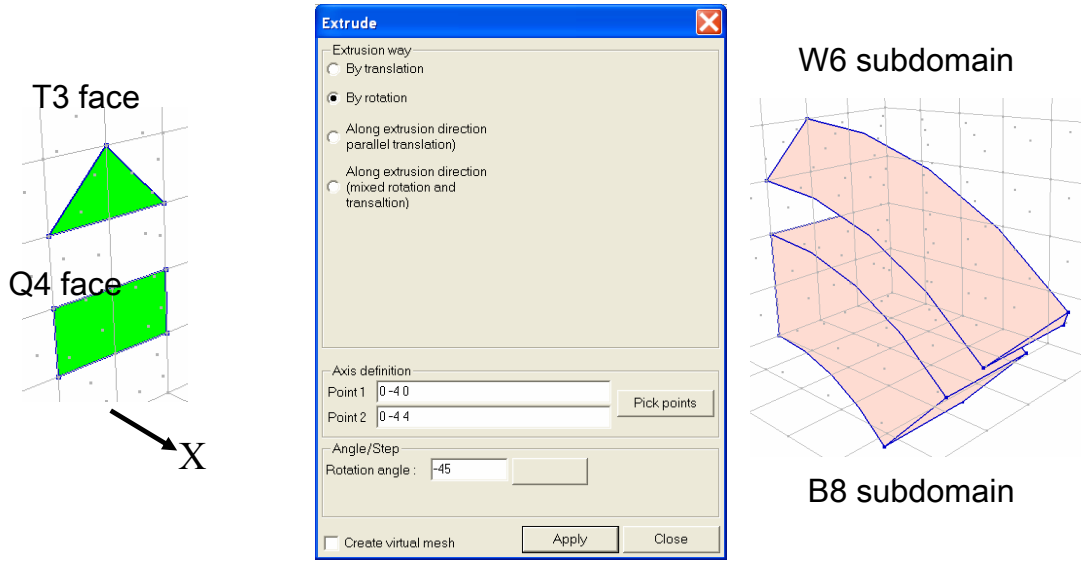
1. In case (1) one may create a structured mesh with any declared split values
2. In case (2) one may also create a structured mesh with any declared split values
3. In case (3) only direction of extrusion can be split with user declared value; mesh in the extruded face remains fixed
4. This operation can be performed on single picked subdomain face or set of selected subdomain faces
5. In all three cases one can perform multiple extrusion operations with given split and extrusion length in each segment

Window 2-152

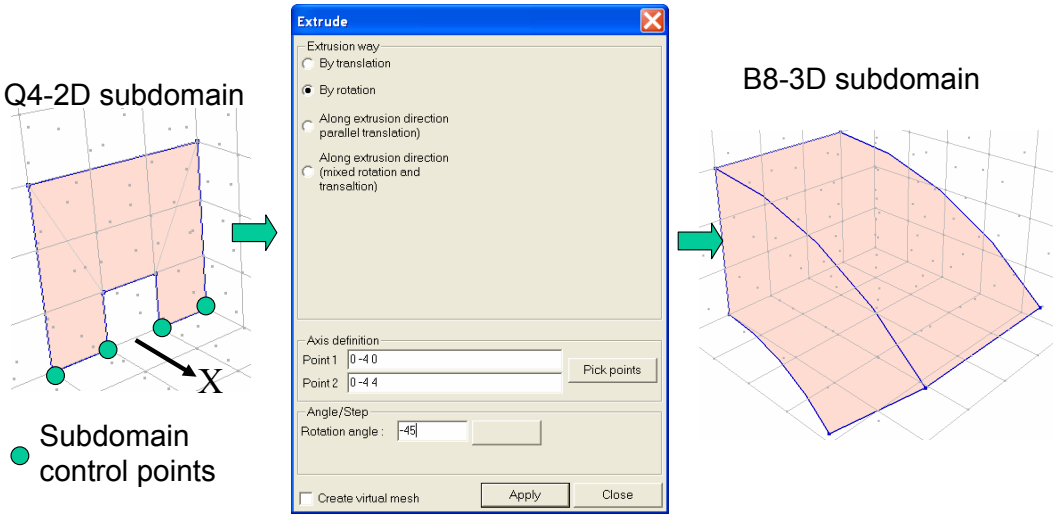
**Window 2-153:** `Subdomain:Create:3D continuum by face(s) extrusion: meth.B`

This method allows to create a new 3D subdomain by a single extrusion of selected subdomain (of any type) face(s) along given rotation axis. Three possible cases have to be considered:

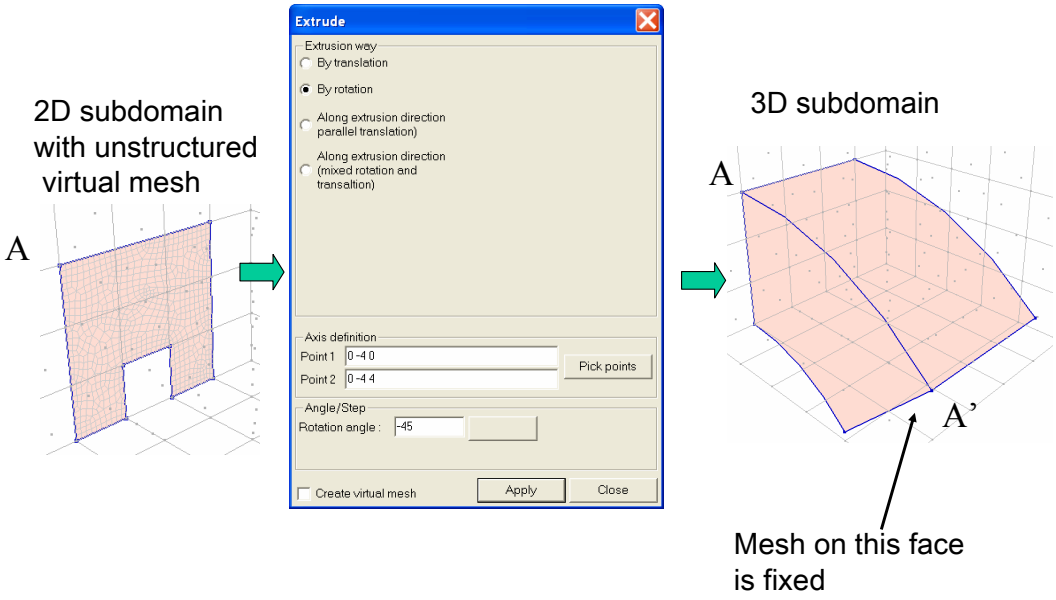
- Face consists of 3 or 4 vertices and it is not yet virtually discretized; in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 with 8-node brick elements only (see first figure below)




- Face consists of any number of vertices and is already virtually discretized using structured mesh template (T3 or Q4); in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 using 8-node brick elements only (see first figure below)




- Face consists of any number of vertices and is already virtually discretized using unstructured mesh template; in this case a 3D continuum subdomain is created; the only free direction to modify mesh density is the direction of extrusion (see second figure below)




**Remarks**:

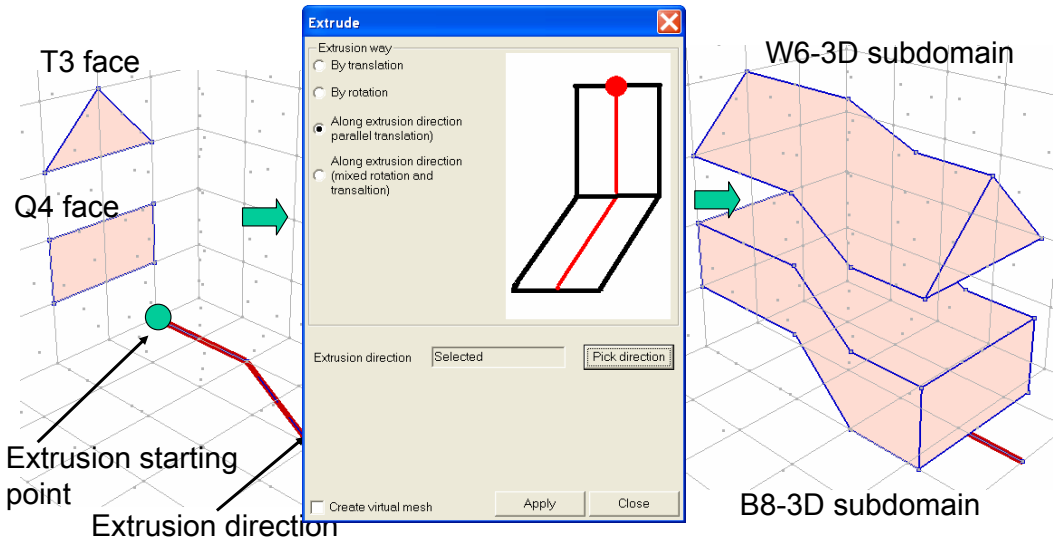
1. In case (1) one may create a structured mesh with any declared split values
2. In case (2) one may also create a structured mesh with any declared split values
3. In case (3) only direction of extrusion can be split with user declared value; mesh in the extruded face remains fixed
4. This operation can be performed on single picked subdomain face or set of selected subdomain faces

Window 2-153

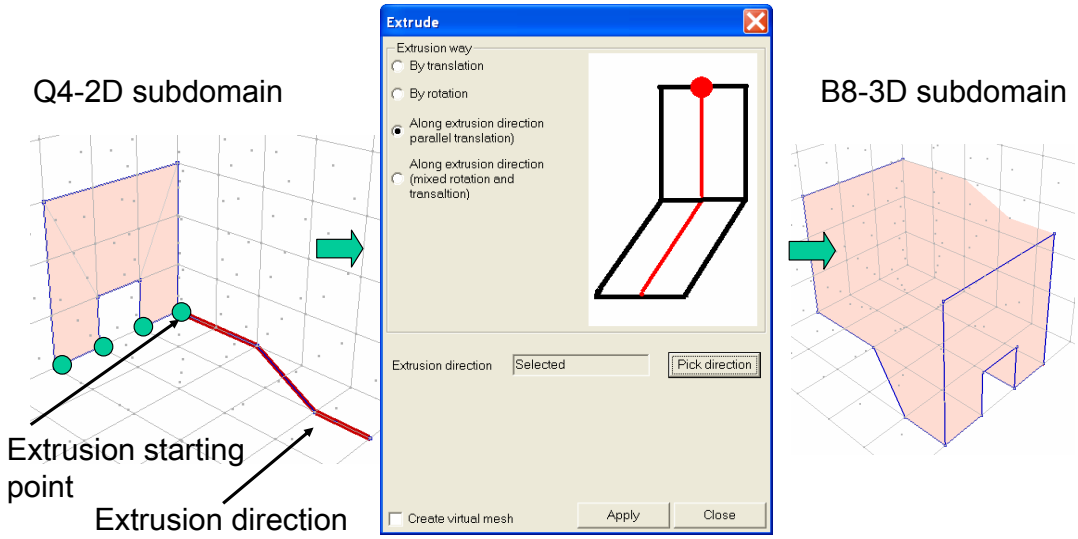
**Window 2-154:** `Subdomain:Create:3D continuum by face(s) extrusion: meth.C`

This method allows to create a new 3D subdomain by a multiple extrusion of selected subdomain (of any type) face(s) along given extrusion direction. All extruded faces are located in parallel planes. Three possible cases have to be considered:

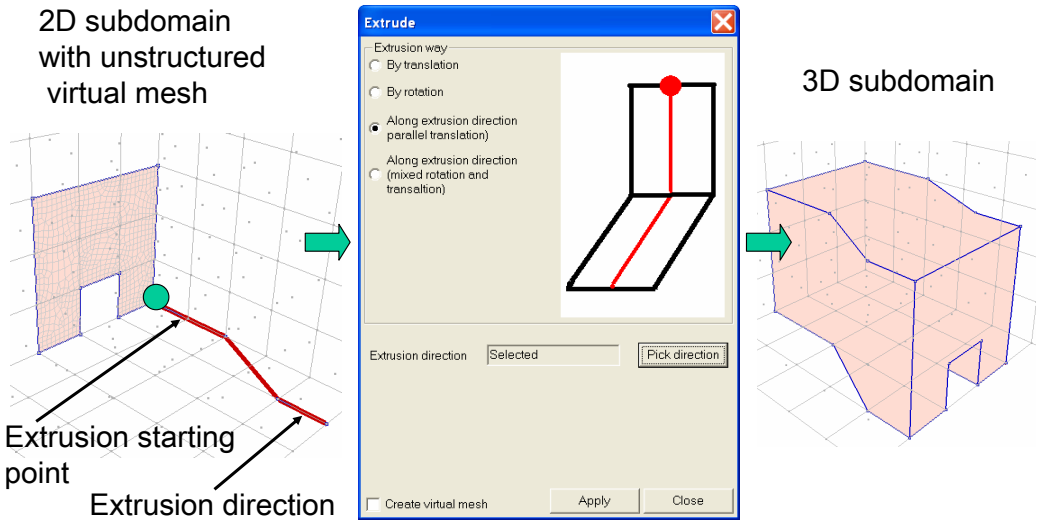
- Face consists of 3 or 4 vertices and it is not yet virtually discretized; in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 with 8-node brick elements only




- Face consists of any number of vertices and is already virtually discretized using structured mesh template (T3 or Q4); in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 using 8-node brick elements only



- Face consists of any number of vertices and is already virtually discretized using unstructured mesh template; in this case a 3D continuum subdomain is created; the only free direction to modify mesh density is the direction of extrusion




**Remarks**:

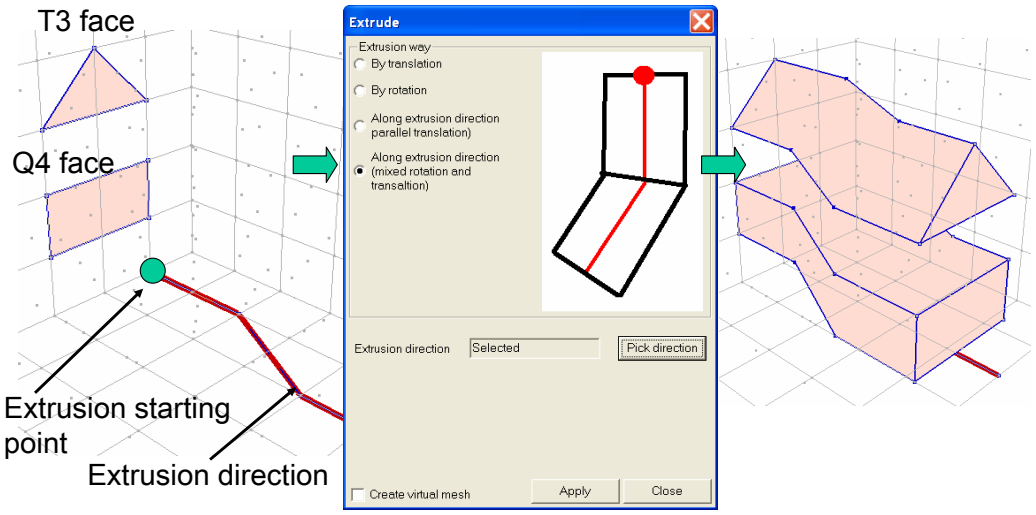
1. In case (1) one may create a structured mesh with any declared split values
2. In case (2) one may also create a structured mesh with any declared split values
3. In case (3) only direction of extrusion can be split with user declared value; mesh in the extruded face remains fixed
4. This extrusion method cannot be used for extrusion directions that include arcs
5. This operation can be performed on single picked subdomain face or set of selected subdomain faces

Window 2-154

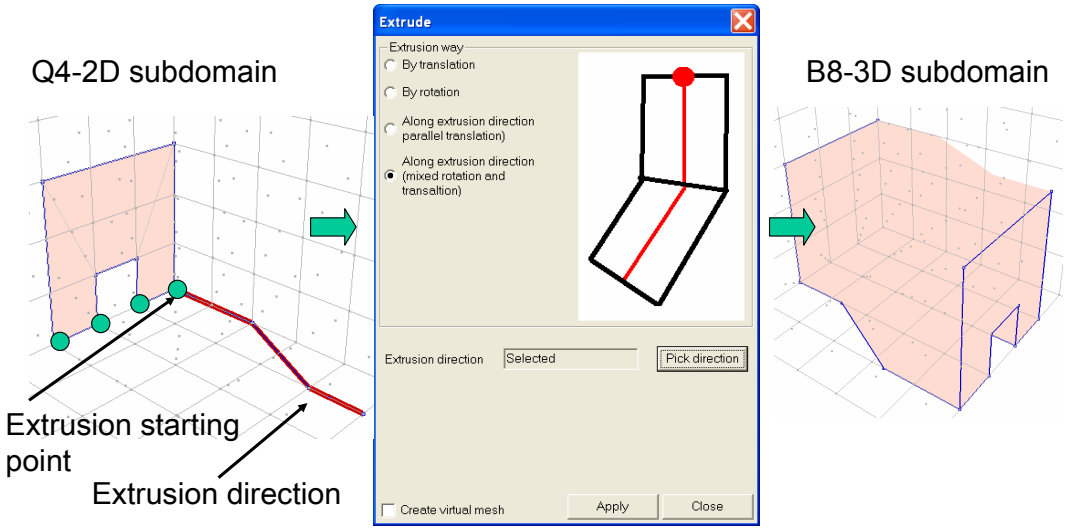
**Window 2-155:** `Subdomain:Create:3D continuum by face(s) extrusion: meth.D`

This method allows to create a new 3D subdomain by a multiple extrusion of selected subdomain (of any type) face(s) along given extrusion direction. Extruded faces are not located in parallel planes in this specific case. This option allows to extrude faces to 3D continuum along arcs. Three possible cases have to be considered:

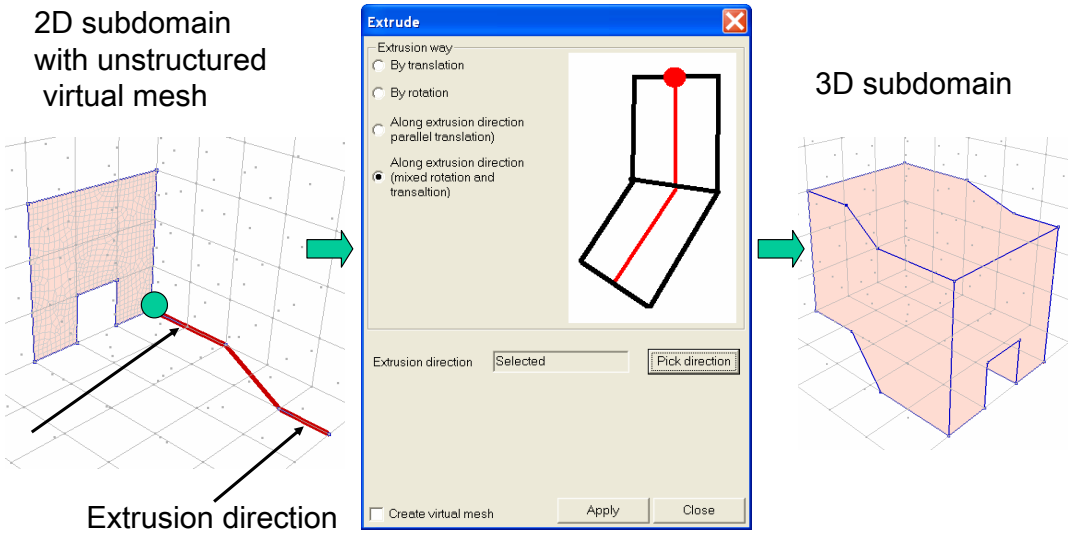
- Face consists of 3 or 4 vertices and it is not yet virtually discretized; in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 with 8-node brick elements only




- Face consists of any number of vertices and is already virtually discretized using structured mesh template (T3 or Q4); in this case a wedge (W6) or brick (B8) type 3D subdomain is created and it can be discretized with structured mesh templates W6 or B8 using 8-node brick elements only



- Face consists of any number of vertices and is already virtually discretized using unstructured mesh template; in this case a 3D continuum subdomain is created; the only free direction to modify mesh density is the direction of extrusion



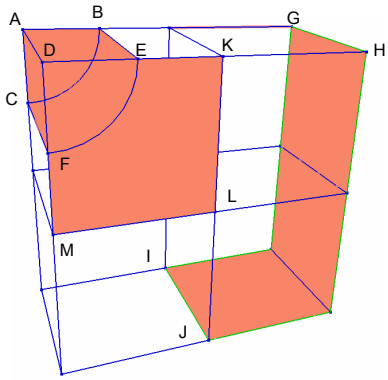

**Remarks**:

1. In case (1) one may create a structured mesh with any declared split values
2. In case (2) one may also create a structured mesh with any declared split values
3. In case (3) only direction of extrusion can be split with user declared value; mesh in the extruded face remains fixed
4. This method is the only meaningful one when extrusion direction includes arcs
5. This operation can be performed on single picked subdomain face or set of selected subdomain faces

Window 2-155

**Window 2-156:** `Subdomain:Create:3D shell (1L) on skeleton`

A 3D one layer shell subdomain is defined on a set of selected basic geometrical objects (lines, arcs) (see Figure below) constituting a closed form polygons.



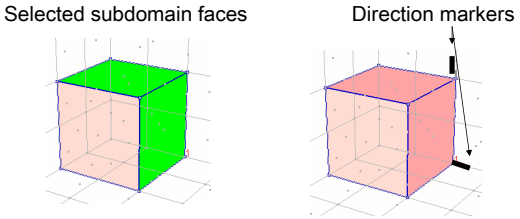

**Remarks**:

1. Surfaces with a single curvature are supported only
2. If all selected objects, constituting a closed form polygon, remain in the plane, the subdomain is created no matter whether it forms macro-triangle (T3) or macro-quad (Q4); if the polygon consists of 3 or 4 vertices then T3 or Q4 type, respectively, subdomain is generated; for polygons with more than 4 vertices decision whether the subdomain is of T3 or Q4 type will have to be taken during virtual mesh generation
3. If all selected objects, constituting a closed form polygon, do not remain in the plane, the subdomain is created according to the following rule:
   - If the polygon consists of 3 or 4 vertices then T3 or Q4 type subdomain is generated
   - For polygons with more than 4 vertices, decision whether the subdomain is of T3 or Q4 type must be taken and vertex points must be picked during subdomain generation; this setup will be used later on during virtual mesh generation
4. In the above example subdomain A-B-C will be of T3 type, B-C-F-E of Q4 type, E-F-L-M-K of a type to be declared later on during virtual mesh generation and I-J-H-G of a Q4 type declared during subdomain generation

Window 2-156

**Window 2-157:** `Subdomain:Create:3D shell (1L) on faces`

A 3D one layer shell subdomain is defined on a set of selected subdomain faces (see Figure below).




**Remarks**:

1. Orientation of external normals can be modified by using method `Update/Direction`

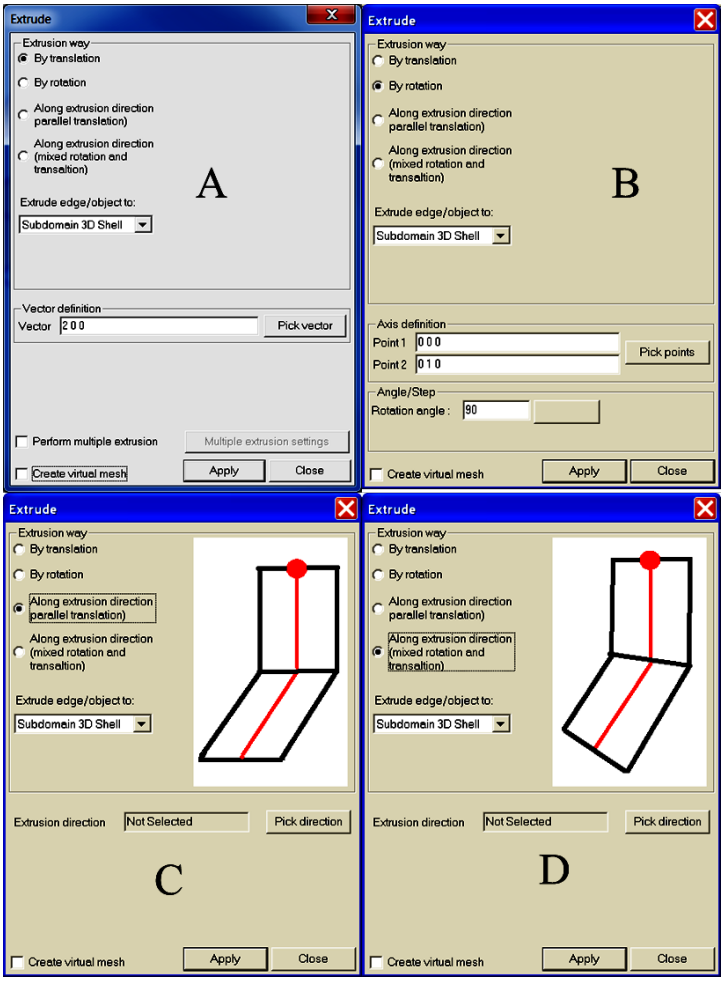
Window 2-157

**Window 2-158:** `Subdomain:Create:3D shell (1L) by edge/object extrusion`

This method allows to create 3D shell (one layer) subdomain by a single or multiple extrusion (along given extrusion direction) of a picked, subdomain edge or group of selected subdomain (of any type) edges, or geometrical objects.

Four types of extrusion method can be used:

1. Single extrusion by translation along given direction (method A, see figure below and Window 2-159)
2. Single extrusion by rotation along given axis (method B, see figure below and Window 2-160)
3. Parallel translation along given direction of extrusion (multiple extrusion) (method C, see figure below and Window 2-161)
4. Parallel translation combined with rotation along given direction of extrusion (multiple extrusion) (method D, see figure below and Window 2-162)



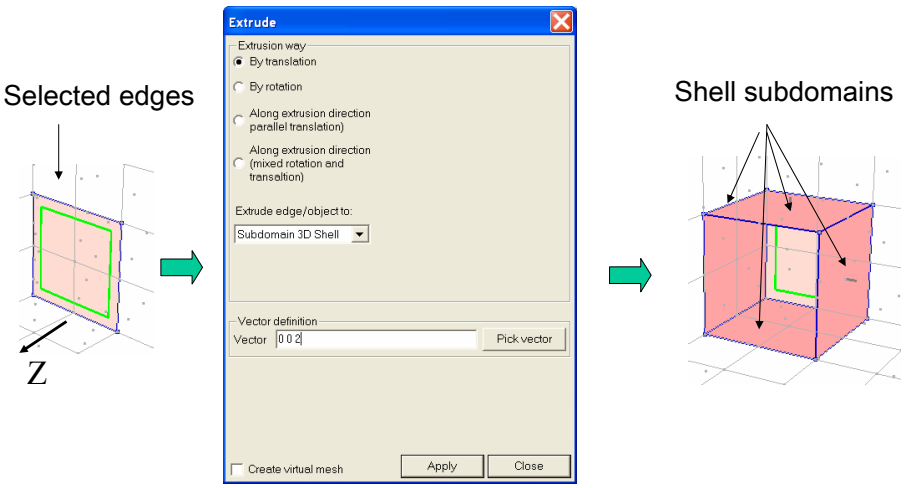

**Remark**:

During subdomain generation a virtual mesh can be simultaneously created by setting ☒ **Create virtual mesh** to ON and declaring the split value along the extrusion direction (this edit field is invisible if ☒ **Create virtual mesh** is set to OFF).

Window 2-158

**Window 2-159:** `Subdomain:Create:3D shell (1L) by edge/object extrusion: meth.A`

This method allows to create a new shell subdomain by a single extrusion of selected subdomain (of any type) edge(s), or geometrical objects, in given direction ( see figure below).



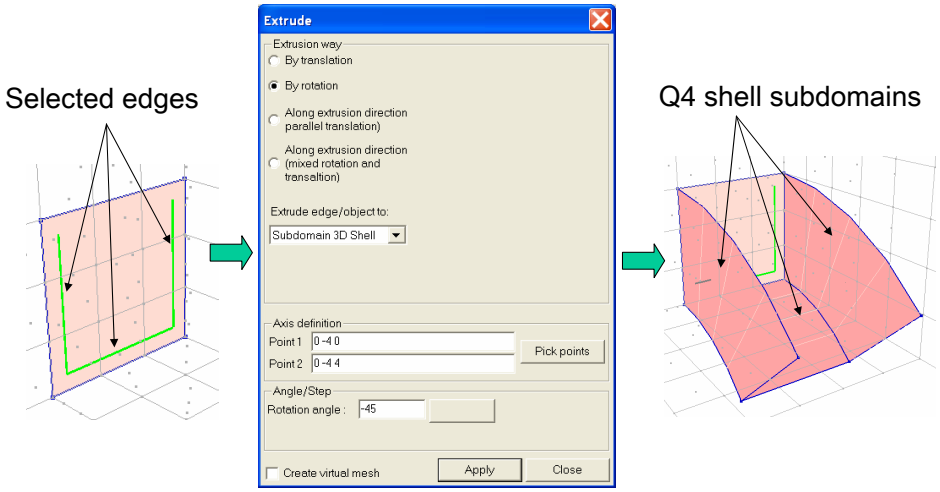

**Remarks**:

1. Q4 subdomain can only be generated by this extrusion method

Window 2-159

**Window 2-160:** `Subdomain:Create:3D shell (1L) by edge/object extrusion: meth.B`

This method allows to create a new shell subdomain by a single extrusion of selected subdomain (of any type) edge(s), or geometrical objects, along given rotation axis ( see figure below).



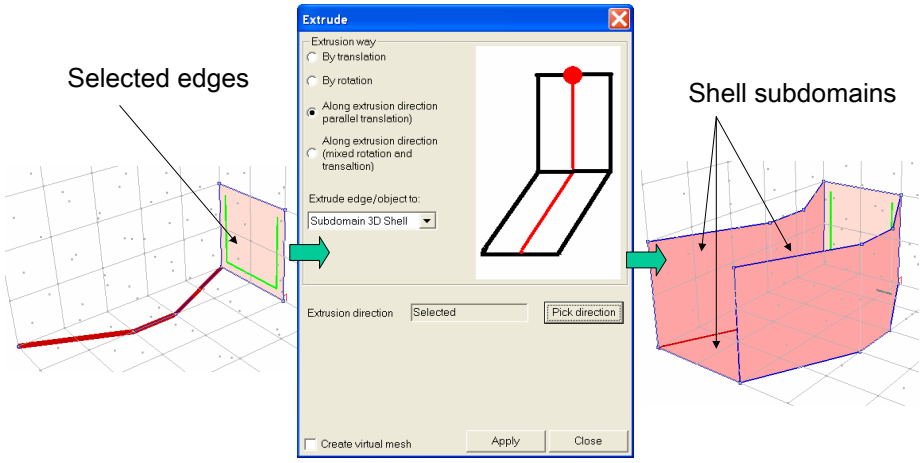

**Remarks**:

1. Both Q4 and T3 subdomains can be generated by this extrusion method; T3 subdomain is created when one of the edge endpoints belongs to the rotation axis

Window 2-160

**Window 2-161:** `Subdomain:Create:3D shell (1L) by edge/object extrusion: meth.C`

This method allows to create a new shell subdomain by a multiple extrusion of selected subdomain (of any type) edge(s), or geometrical objects, along given extrusion direction ( see figure below). All edges of newly created Q4 type shell subdomains, opposite to the extruded ones, are all located in parallel planes. This option precludes shells extrusion from edges for extrusion directions composed of arcs.



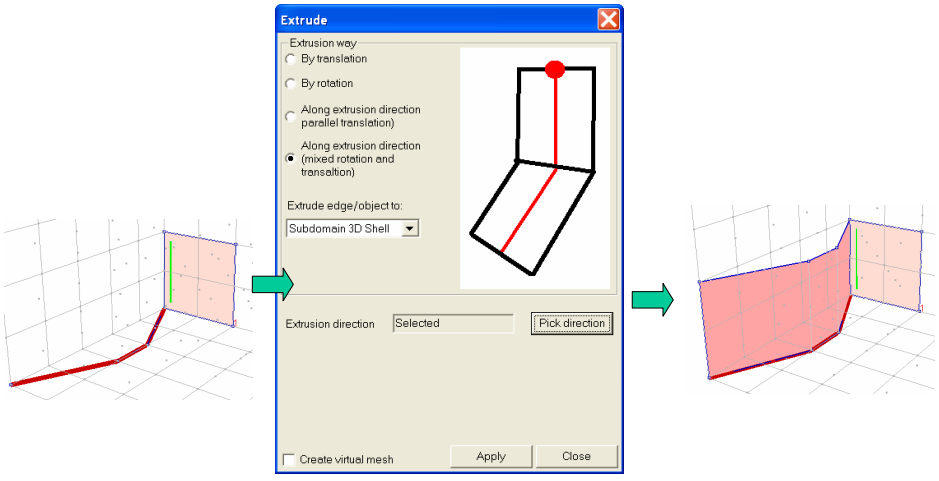

**Remarks**:

1. Q4 subdomains can only be generated by this extrusion method

Window 2-161

**Window 2-162:** `Subdomain:Create:3D shell (1L) by edge/object extrusion: meth.D`

This method allows to create a new shell subdomain by a multiple extrusion of selected subdomain (of any type) edge(s), or geometrical objects, along given extrusion direction ( see figure below). Edges of newly created Q4 type shell subdomains, opposite to the extruded ones, are not all located in parallel planes in this method. This option allows to extrude shells from edges for extrusion directions composed of arcs.




**Remarks**:

1. Q4 subdomains can only be generated by this extrusion method

Window 2-162

**Window 2-163:** `Subdomain:Create:3D membrane on skeleton`

This method is equivalent to the one designed for shell subdomain (see Window 2-156).

Window 2-163

**Window 2-164:** `Subdomain:Create:3D membrane on face(s)`

This method is equivalent to the one designed for shell subdomain (see Window 2-157).

Window 2-164

**Window 2-165:** `Subdomain:Create:3D membrane by edge/object extrusion`

This method is equivalent to the one designed for shell subdomain (see Window 2-158).

Window 2-165

**Window 2-166:** `Subdomain:Create:Copy with rotation`

This option allows to make copies of a selected subdomain(s) with simultaneous rotation. It is fully coherent with the one designed for standard FE components (see.Win.(2-310))

Window 2-166

**Window 2-167:** `Subdomain:Create:Copy with translation`

This option allows to make copies of a selected subdomain(s) with simultaneous translation. It is fully coherent with the one designed for standard FE components (see.Win.(2-311))

Window 2-167

**Window 2-168:** `Subdomain:Create:Copy by symmetry`

This option allows to make a copy of a selected subdomain(s) by symmetrical projection with respect to the picked plane (created under ribbon menu `Draw:Auxiliary Planes`). It is fully coherent with the one designed for standard FE components (see.Win.(2-312))

Window 2-168

**Window 2-169:** `Subdomain`**:Options in menu** `Select`

| | |
|---|---|
| `Select one subdomain` | this option activates graphical cursor in the selection mode and subdomains are selected one by one by pointing them by mouse |
| `All` | all subdomains are selected; result of this action depends on the status of `Selection mode` (see Window 2-83) |
| `In zoom box` | if the zoom box is dragged from left to the right side then only fully covered, by the zoom rectangle, subdomains are selected<br>if the zoom box is dragged from right to the left side then even partially covered, by the zoom rectangle, subdomains are selected |
| `In zoom circle` | if the zoom circle is dragged from left to the right side then only fully covered, by the circle (cylinder in 3D), subdomains are selected<br>if the zoom circle is dragged from right to the left side then even partially covered, by the zoom circle (cylinder in 3D), subdomains are selected |

Window 2-169

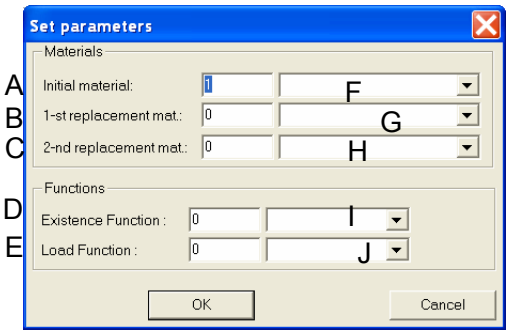
**Window 2-170:** `Subdomain:Delete`

The following basic `Delete` operation can be used: `Delete` : removes picked subdomain or group of selected subdomains

Window 2-170

**Window 2-171:** `Subdomain:Parameters` **(standard setup)**

User interface as shown in figure below is designed for continuum 2D/3D, membranes, anchors (trusses), beams, seepage and convection.



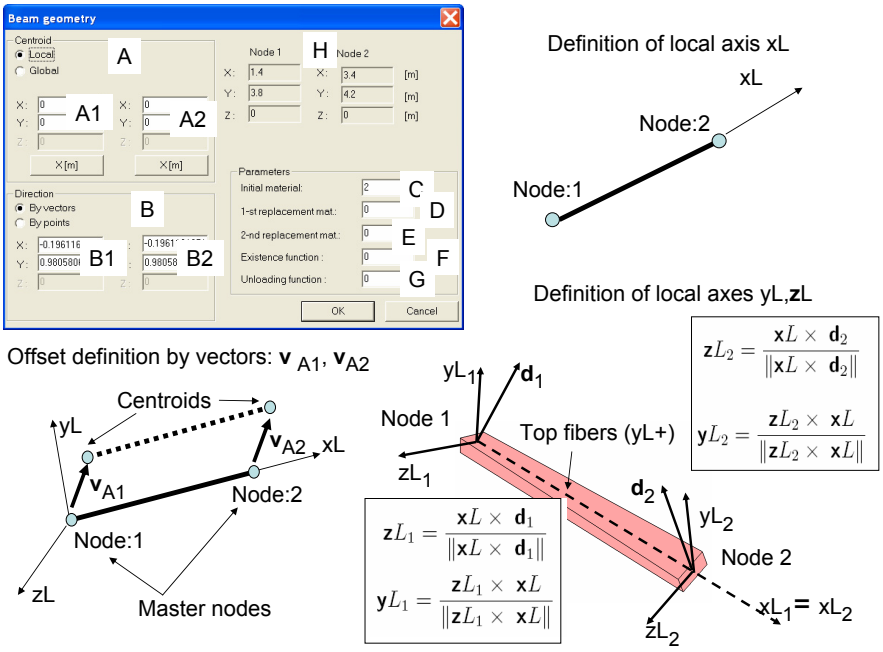

**Remarks**:

1. Material, replacement material and second replacement material, existence function and unloading function can be selectively defined or modified once the proper checkbox (A, B, C, D or E) or few of them are set to ON
2. If list of materials is already prepared all material labels will be visible in combo-boxes (F, G, H) and one may easily apply a proper material
3. If list of existence functions is already prepared then all existence functions labels will be visible in combo-boxes (I) and one may easily apply a proper function
4. If list of load time functions is already prepared then all load time functions labels will be visible in combo-boxes (J) and one may easily apply a proper function

Window 2-171

**Window 2-172:** `Subdomain:Parameters` **(setup for beams)**

User interface for 3D beams, definition of offset vectors and local beam coordinate system are shown in the Figure below.

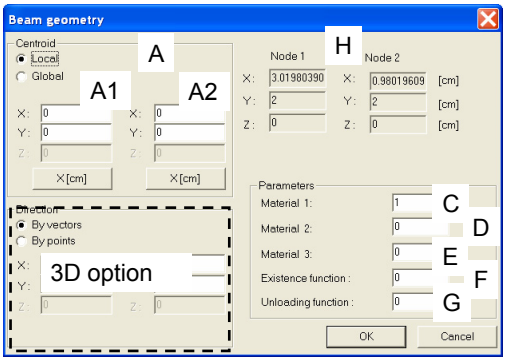



**Remarks**:

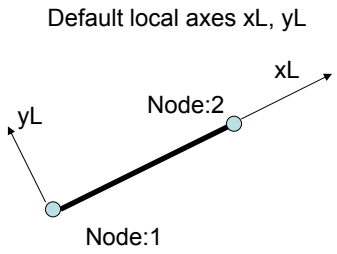
1. The local beam coordinate system is defined separately at both beam endpoints by means of direction vectors $\mathbf{d}_1$ and $\mathbf{d}_2$, hence a moderately twisted beam can be modeled (see Figure)
2. If `Direction` (B) is defined `By vectors` then coordinates of direction vectors $\mathbf{d}_1$ and $\mathbf{d}_2$ are defined directly in edit fields (B1) and (B2) (see Figure)
3. If `Direction` (B) is defined `By points` then coordinates of direction vectors $\mathbf{d}_1$ and $\mathbf{d}_2$ are computed as differences between given points coordinates (defined in edit fields (B1) (B2)) and `Node 1` and Node 2 coordinates given in edit fields (H)
4. Position of the top fibers of the cross section is defined by the positive local $y_L$ axis
5. Offsets of centroid nodes with respect to the master nodes can be given in global or local beam coordinate system
6. During computation real beam coordinates are shifted by given offset vectors
7. Local axes $\mathbf{y}_{L1}$, $\mathbf{x}_L$ and vector $\mathbf{d}_1$ remain in the same plane (the same rule applies to $\mathbf{y}_{L2}$, $\mathbf{x}_L$ and vector $\mathbf{d}_1$)
8. Local axes $y_L$ and $z_L$ at both beam endpoints are defined according to the formula

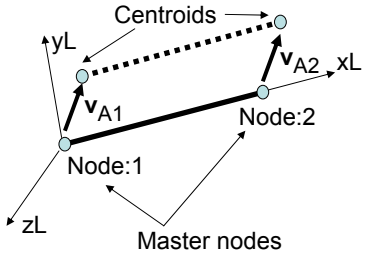
$$\mathbf{z}L_1 = \frac{\mathbf{x}L \times \mathbf{d}_1}{\|\mathbf{x}L \times \mathbf{d}_1\|} \quad \mathbf{y}L_1 = \frac{\mathbf{z}L_1 \times \mathbf{x}L}{\|\mathbf{z}L_1 \times \mathbf{x}L\|} \quad \mathbf{z}L_2 = \frac{\mathbf{x}L \times \mathbf{d}_2}{\|\mathbf{x}L \times \mathbf{d}_2\|} \quad \mathbf{y}L_2 = \frac{\mathbf{z}L_2 \times \mathbf{x}L}{\|\mathbf{z}L_2 \times \mathbf{x}L\|}$$

User interface for 2D beams, definition of offset vectors and local beam coordinate system are shown in the Figure below.




Offset definition by vectors: $\mathbf{v}_{A1}$, $\mathbf{v}_{A2}$




**Remarks**:

1. By default the local $x_L$ axis of the beam is directed towards Node1-Node2 while local $y_L$ axis is rotated by $90^o$ with respect to the $x_L$ in anticlockwise direction
2. Position of the top fibers of the cross section is defined by the positive local $y_L$ axis
3. Offsets of centroid nodes with respect to the master nodes can be given in global or local beam coordinate system
4. During computation real beam coordinates are shifted by given offset vectors
5. Local $x_L$ and $y_L$ axes can be reversed

Window 2-172

**Window 2-173:** `Subdomain:Define excavation front` **(standard setup)**

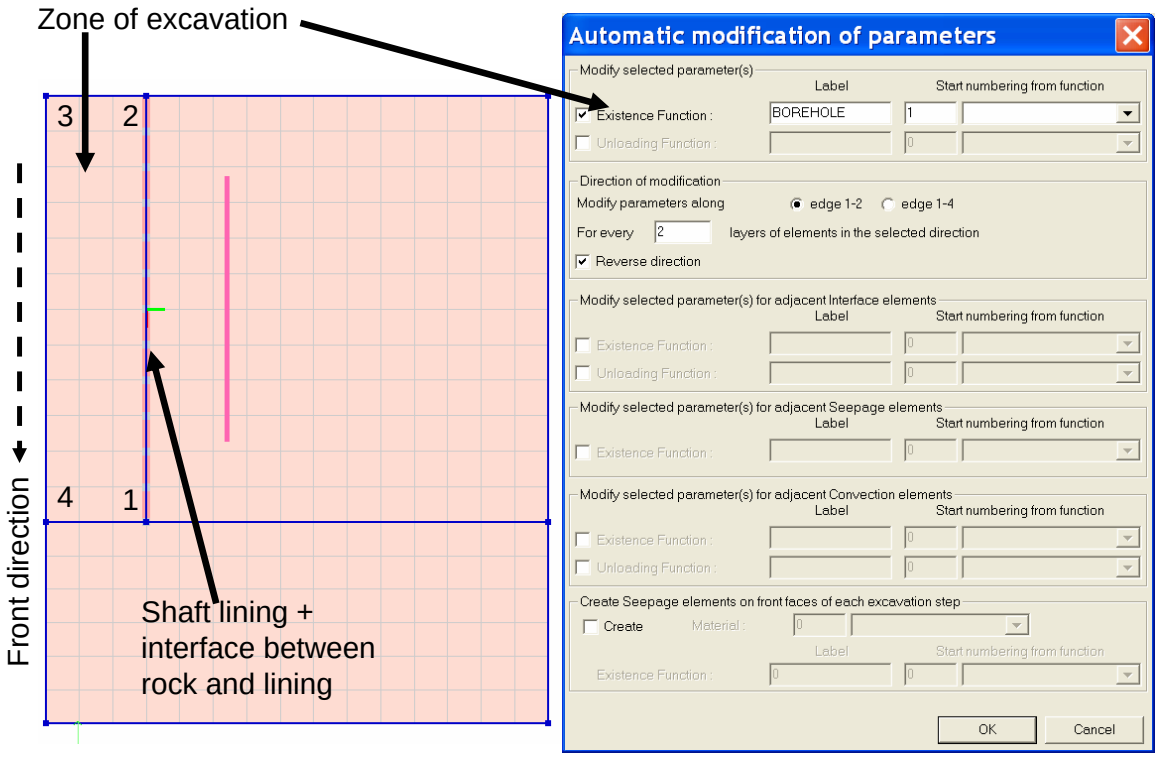
This option allows an automatic definition of the existence functions in 2D/3D continuum, shell, membrane or beam subdomains along selected subdomain edge. This definition can be made both on virtual or real mesh of a selected subdomain. However, the effect of this option is visible exclusively in the real mesh. This definition requires the direction of the excavation/construction front and the modification frequency to be selected. The modification frequency establishes how many elements (along the front) get the same existence function. To speedup data setting user may simultaneously apply distinct existence functions to adjacent seepage, interface and/or convection subdomains. Moreover seepage and/or convection subdomains can be automatically added on the excavation front.

For better understanding see 2D example in Win.2-174 and 3D example in Win.2-175.

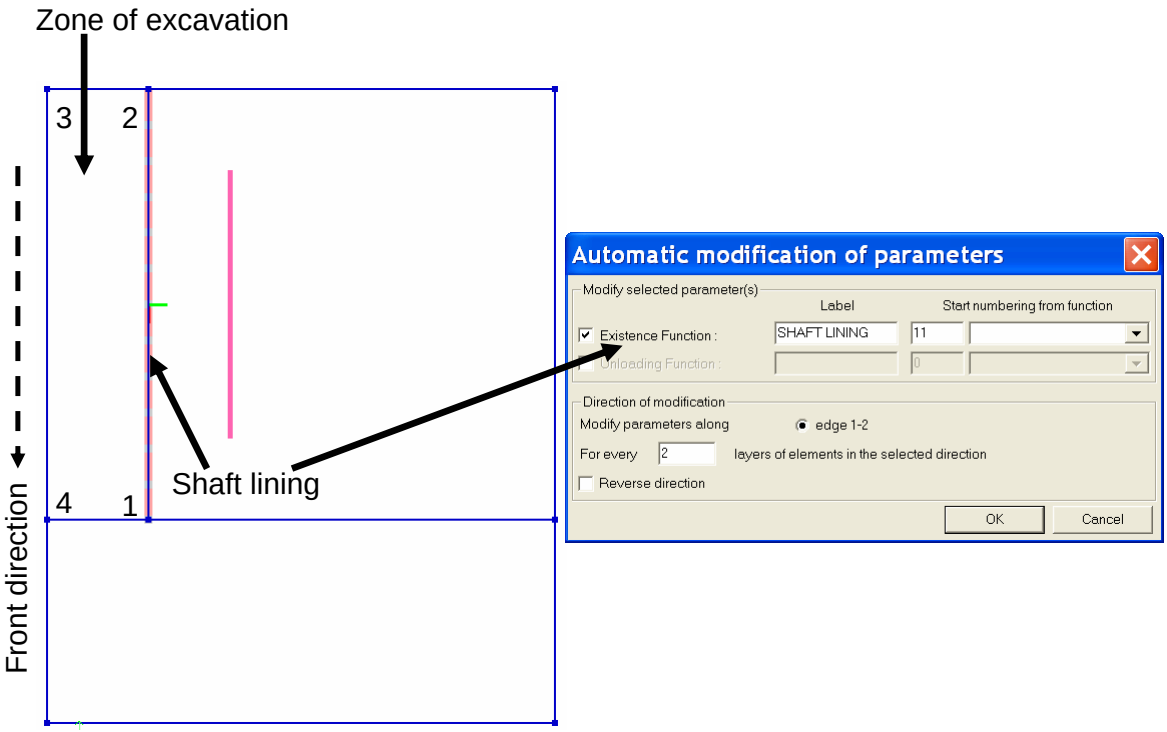
Window 2-173

**Window 2-174:** `Subdomain:Define excavation front` **(2D example)**

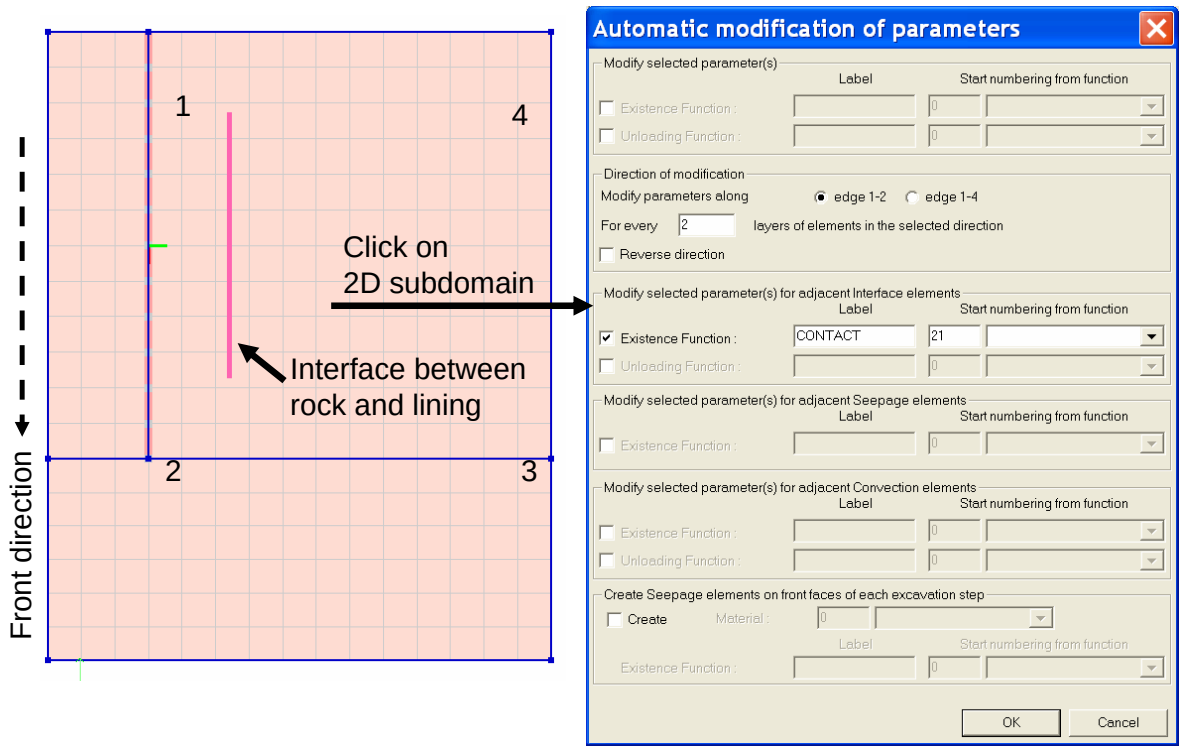
- Define excavation front frequency in the rock



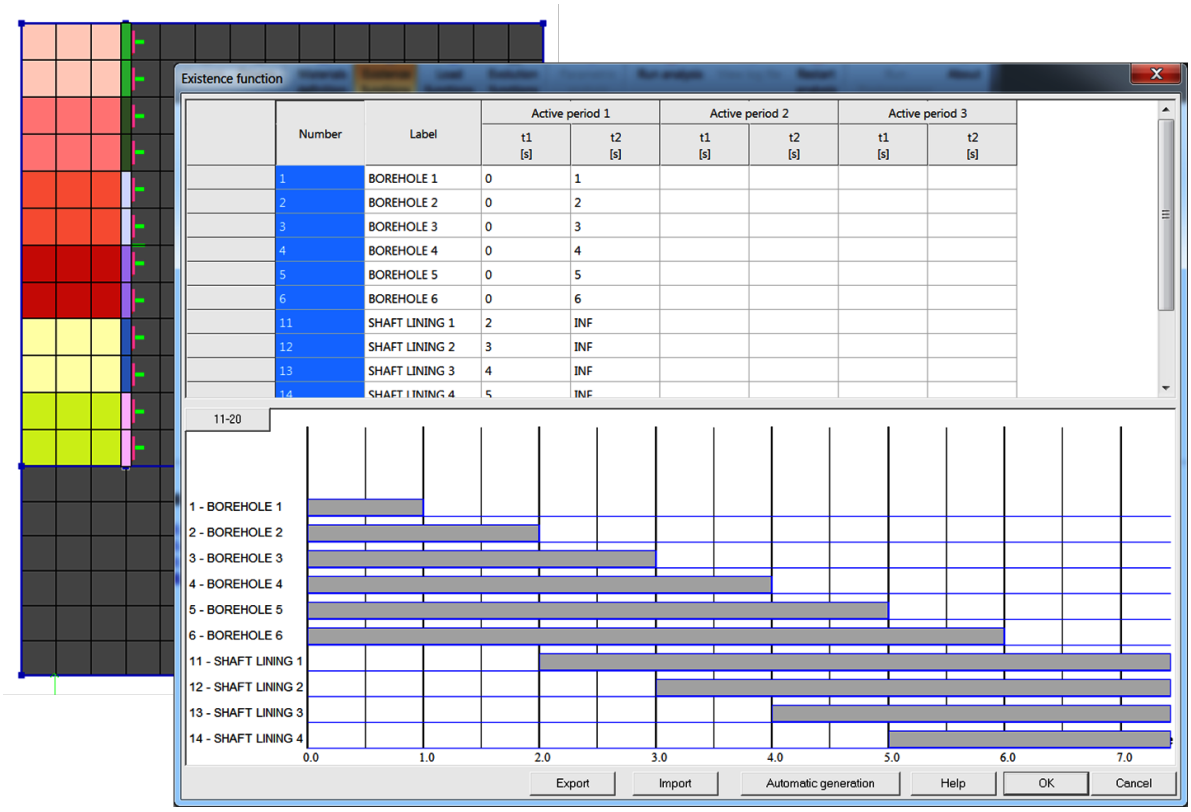
- Define excavation front in the shaft lining.




- Define excavation front in the interface between shaft lining and rock.



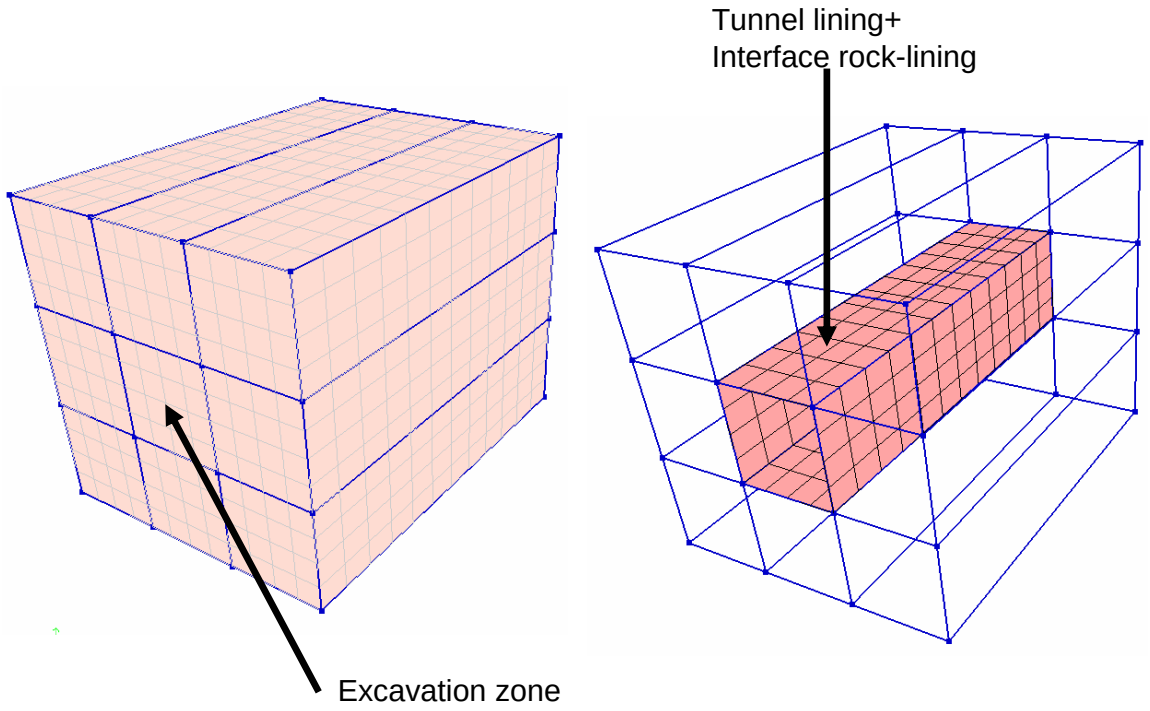
- Generate existence functions using option `Automatic generation`.



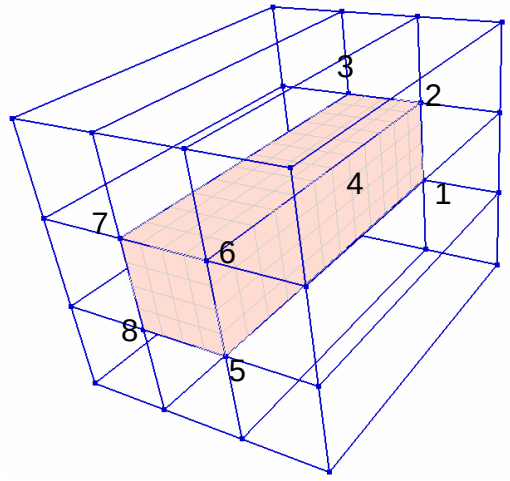

Window 2-174

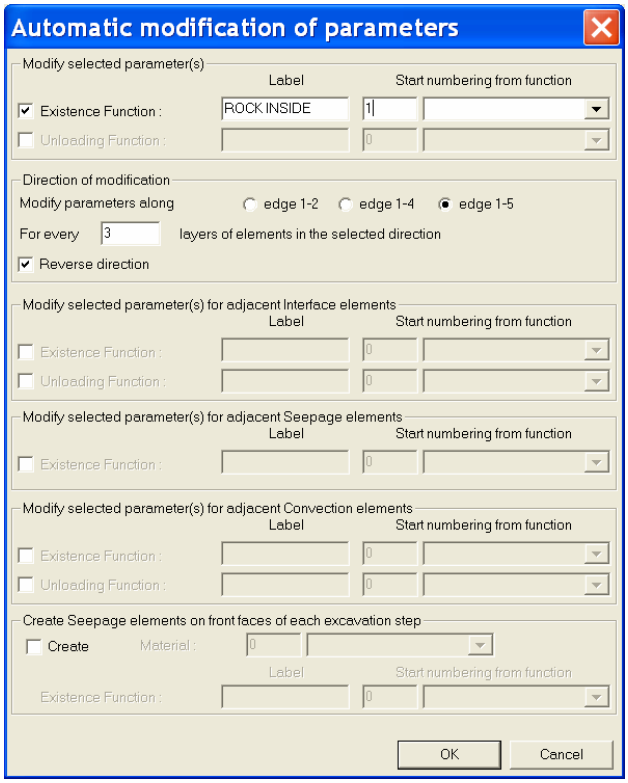
**Window 2-175:** `Subdomain:Define excavation front` **(3D example)**

- Example of an excavation followed by installation of tunnel lining

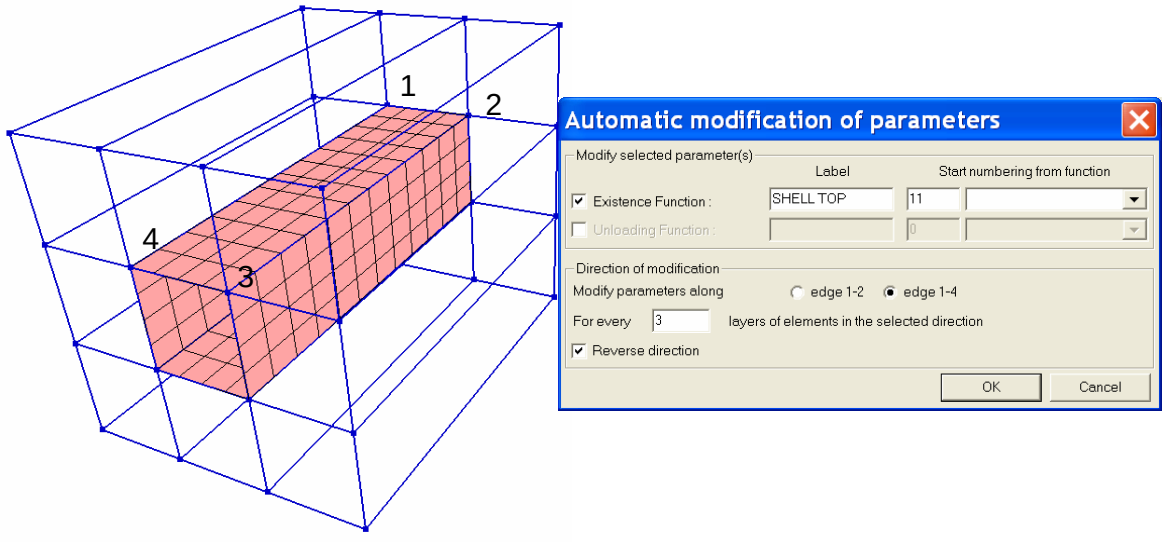



- Define excavation front in the excavated rock subdomain)

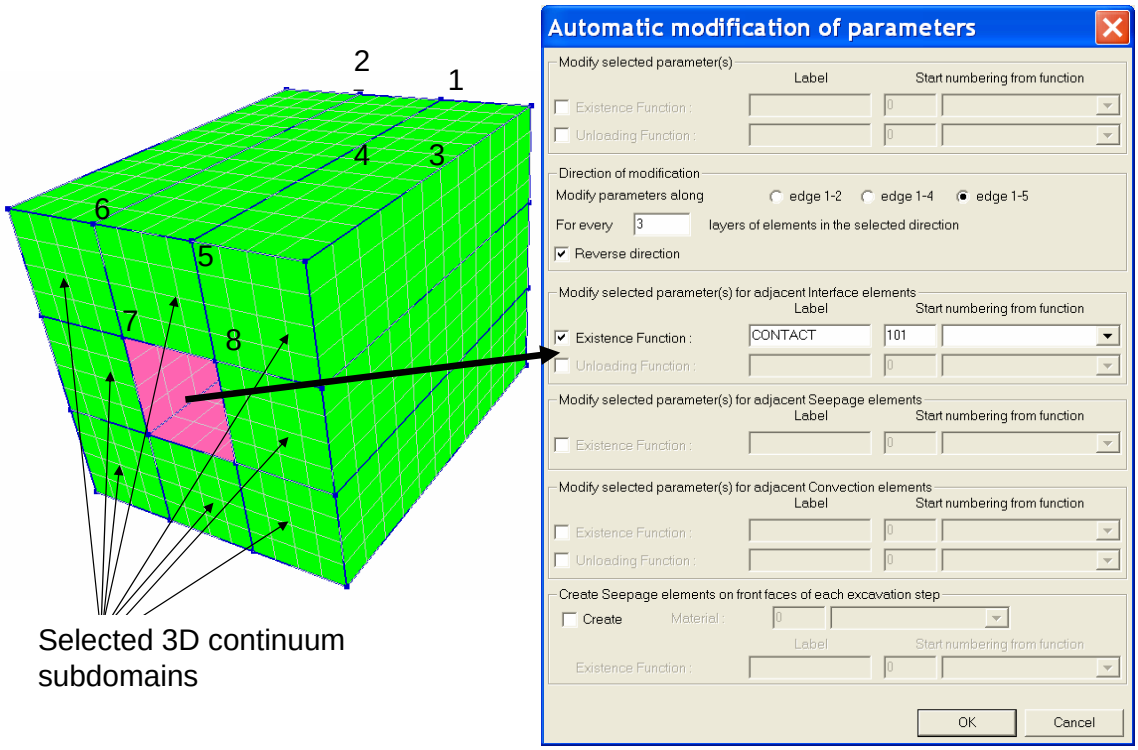


- Define excavation front in shell subdomain representing top part of the tunnel lining (repeat the operation on all shell subdomains)

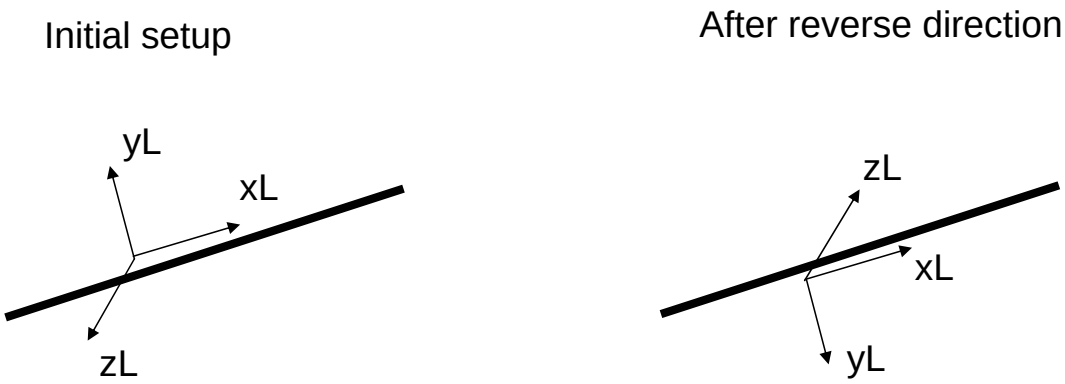



- Define excavation front in the interface between rock and tunnel lining



**Window 2-176:** `Subdomain:Update:Reverse direction`

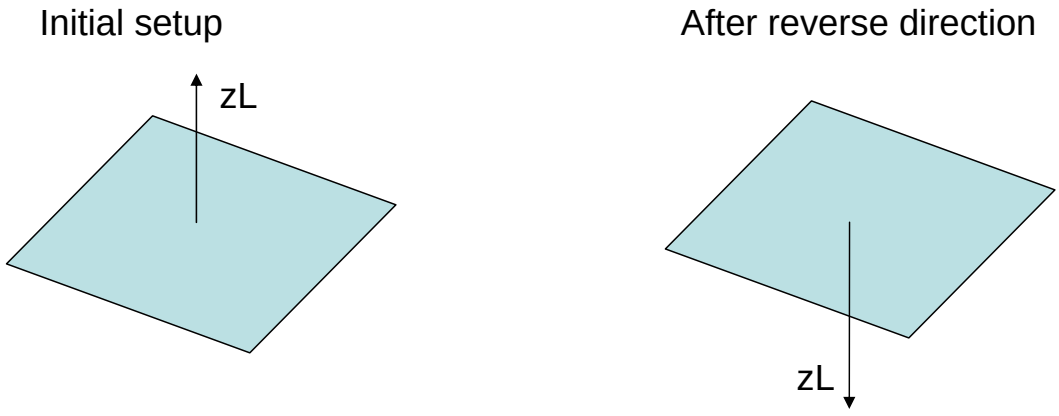
This option applies to macro-beams (2D/3D), macro-membranes (2D) and shell/membrane subdomains. For macro-beams, macro-membranes and trusses/anchors it is used to flip the local axis $y_L$ and in consequence local axis $z_L$ while $x_L$ remains unchanged (see figure below)




**Remarks**:

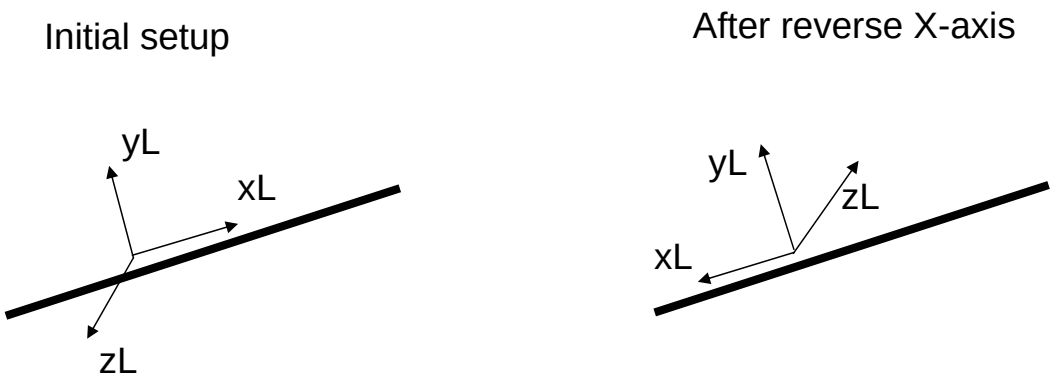
- The detailed explanation on positioning of local beam axes $(x_L, y_L, z_L)$ is given in Window 2-172
- Top fibers of beam cross section are located on positive side of $y_L$ axis while bottom ones on negative
- Proper setting of local beam axes is very important for nonsymmetric cross sections or cross sections of reinforced concrete
- Proper setting of local beam axes is also very important for visualization of normal and shear force diagrams in the postprocessor
- Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

For shell/membrane(3D) subdomains it is used to reverse the direction of the external normal to the midsurface (note that top fibers are placed on the side of the positive external normal).



**Window 2-177:** `Subdomain:Update:Reverse X axis`

This option applies to macro-beams (2D/3D), macro-membranes (2D), and trusses/anchors (2D/3D). This option is used to flip the local axis $x_L$ and in consequence local axis $z_L$ while $y_L$ remains unchanged (see example below).



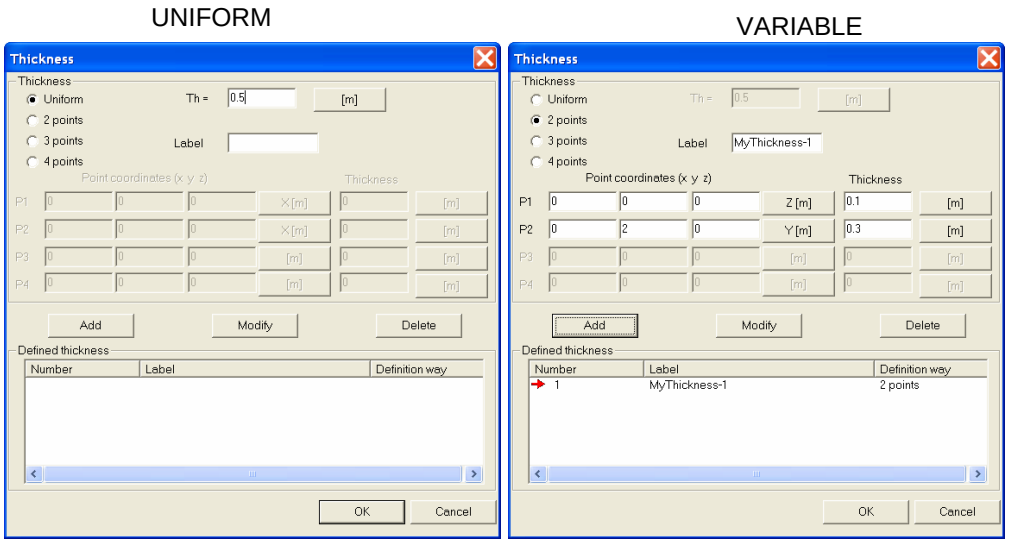

**Remarks**:

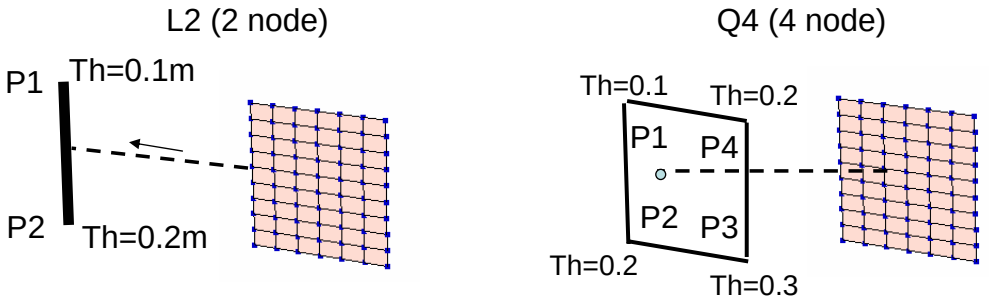
- The detailed explanation on positioning of local beam axes $(x_L, y_L, z_L)$ is given in Window 2-172
- Top fibers of beam cross section are located on positive side of $y_L$ axis while bottom ones on negative
- Proper setting of local beam axes is very important for nonsymmetric cross sections or cross sections of reinforced concrete
- Proper setting of local axes is also very important for visualization of normal and shear force diagrams in the postprocessor
- Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

Window 2-177

## Window 2-178: Subdomain:Update:Thickness

Using this option user may define constant or variable thickness in the single picked shell (one layer) subdomain or group of selected shell subdomains. Setting thickness (which is variable in the shell subdomain) for each resulting shell element in the real mesh is shown in the following figure.



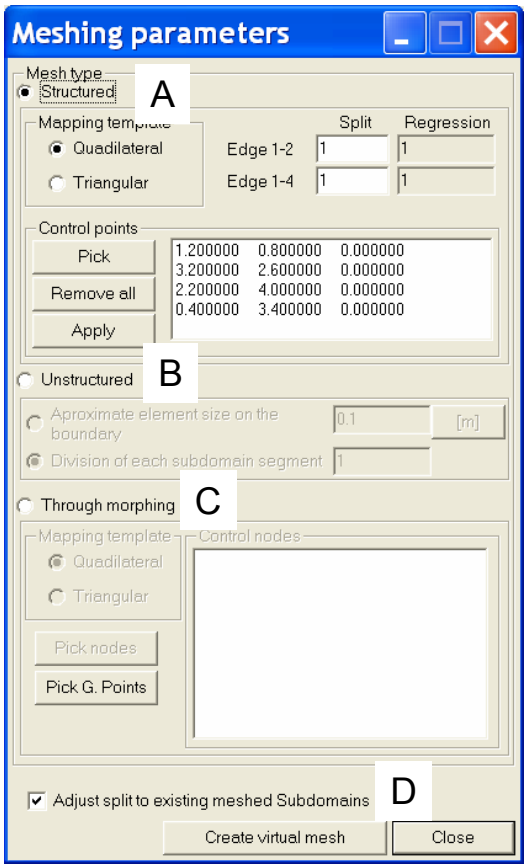

**Remarks**:

1. Constant thickness is set in the edit field **Th.** [ ] once the option ◎ **Uniform** is selected
2. Variable thickness can be defined with aid of 2-node linear , 3-node triangular or 4-node quadrilateral macro-elements; to complete definition of the variable thickness user has to select type of the macro-element (◎ **2-node**, ◎ **3-node**, ◎ **4-node**), define coordinates of macro-element vertices and corresponding thickness at each vertex; thickness at each shell element node is computed by projecting it on the macro-element and then interpolating by means of standard shape functions for L2, T3 or Q4 element (see example)
3. Each thickness definition can be memorized, with a label, for further usage by using button **Add**
4. List of memorized definitions can be modified with buttons **Delete** or **Modify**

**Window 2-179:** `Subdomain:Mesh:Virtual mesh (2D continuum)`

Any continuum 2D subdomain can be virtually discretized with 3 different grid types.

- (A) Structured:
  - ⋆ with quadrilateral (Q4) template (see Window 2-180)
  - ⋆ with triangular (T3) template (see Window 2-181)
- (B) Unstructured (automatic, quad only, meshing using Q-morph algorithm):
  - ⋆ with assumed element size (see Window 2-186)
  - ⋆ with assumed boundary split value (see Window 2-187)
- (C) Through morphing (using Q4 or T3 templates) (see Window 2-188)

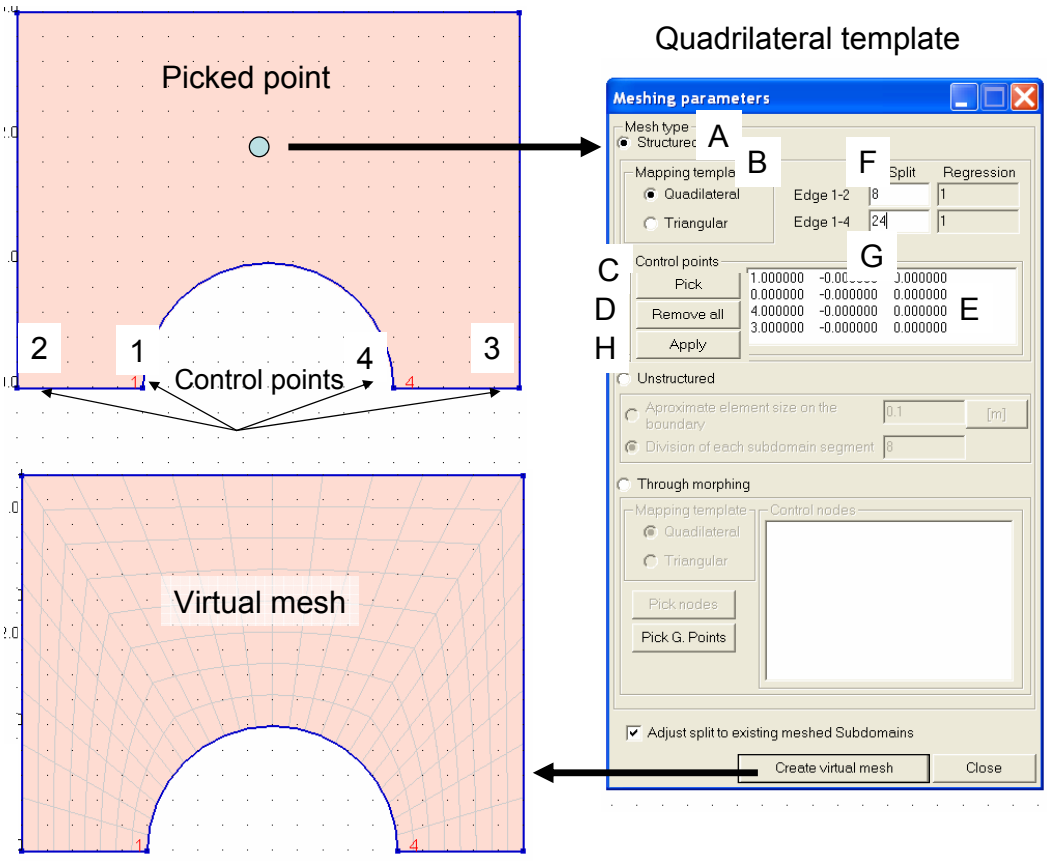



**Remarks**:

1. Structural subdomains modeled with aid of continuum elements should be discretized using `Structured` grids
2. Number of elements along common boundaries of adjacent subdomains can be controlled during discretization once the `Adjust split...` checkbox (D) is set to ON
3. If `Adjust split...` is set to OFF then meshes in adjacent subdomains will not be compatible; special `Mesh tying` elements, available at the finite element level can remedy this problem
4. Virtual mesh is treated here as a guess for the real mesh but subdomain is not yet subdivided into elements.
5. Button **Apply** allows to define control points of the subdomain (if structured mesh is to be generated) without need for setting up the virtual mesh

**Window 2-180:** `Subdomain:Mesh:Virtual mesh(2D cont.):Struct.Q4 template`

To create structured virtual mesh based on quadrilateral template 4 control points must be picked first (using button `Pick` (C)) (list (E) is automatically updated). These 4 points are the corners of a generalized quadrilateral macro-element with possibly curvilinear boundaries.

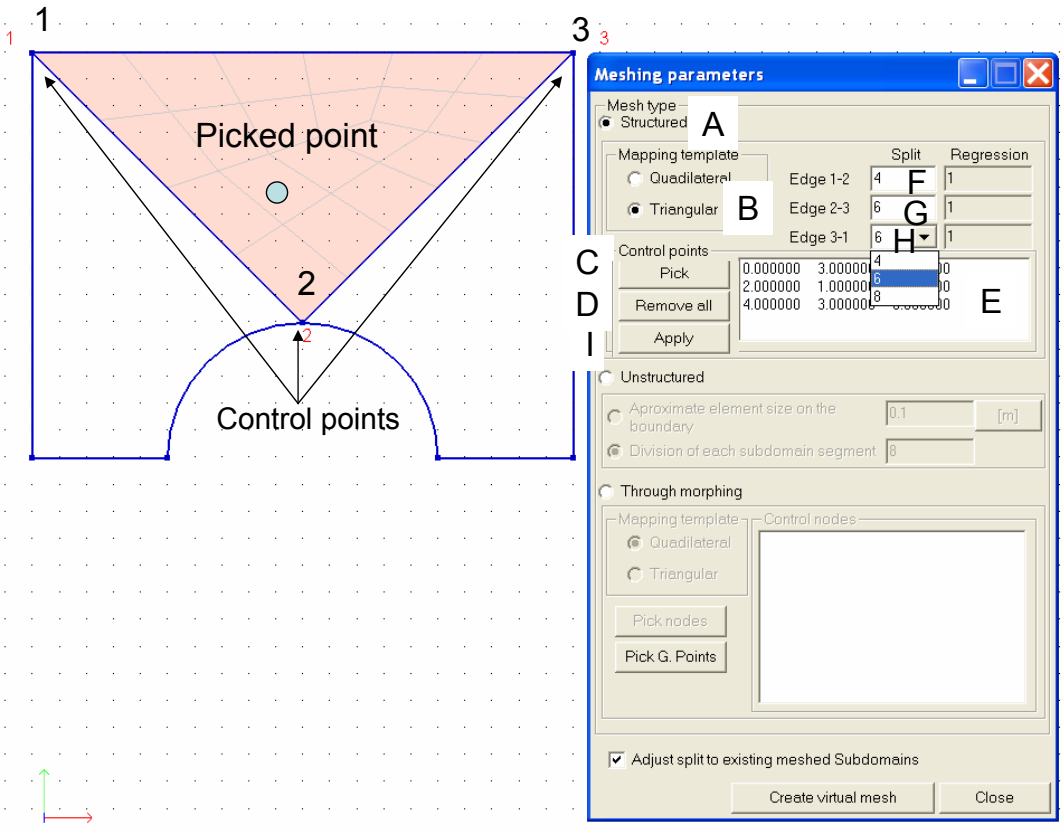



**Remarks**:

1. 4 control points must be picked in a strict order (clockwise or anticlockwise)
2. Modification of the virtual mesh density can be made by changing split values for edges designed by control points 1-2 and 1-4 (edit fields (F) and (G))
3. Control points can be deleted using button `Remove all`
4. To memorize control points without defining virtual mesh press button (H)
5. Additional mesh refinement can be made at the `Finite Element` level using method `Mesh refinement` under menu `Continuum 2D` once the virtual mesh in the subdomain is converted to the real one (with method `Mesh:virtual→real`)

**Window 2-181:** `Subdomain:Mesh:Virtual Mesh(2D cont.):Struct.T3 template`

To create structured virtual mesh, consisting of quadrilateral elements only, and based on triangular template, 3 control points must be picked first (using button `Pick` (C)) (list (E) is automatically updated). These 3 points are the corners of a generalized triangular macro-element with possibly curvilinear boundaries.




**Remarks**:

1. 3 control points must be picked in a strict order (clockwise or anticlockwise)(see Figure)
2. Modification of the virtual mesh density can be made by changing split values for edges spanned on control points 1-2 and 2-3 (edit fields (F) and (G)) while split value for the third edge 1-3 must be selected from list of possible values (H)
3. To memorize control points without defining virtual mesh press button (I)
4. Control points can be deleted using button `Remove all`
5. Additional mesh refinement can be made at the `Finite Element` level using method `Mesh refinement` under menu `Continuum 2D` once the virtual mesh in the subdomain is converted to the real one (with method `Mesh:virtual→real`)

Window 2-181

**Window 2-182:** `Subdomain:Mesh:Virtual mesh (3D continuum)`

Any continuum 3D subdomain can be virtually discretized with 2 different grid types.

- (A) Structured, with brick (B8) (see Window 2-183) or wedge (W6) (see Window 2-184) templates
- (B) Through morphing (using B8 or W6 templates) (see Window 2-189)

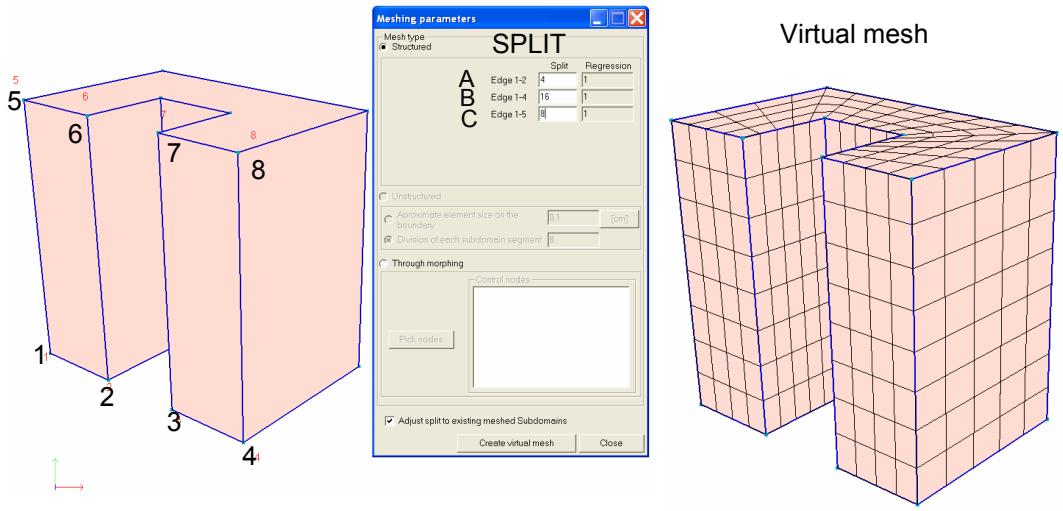
**Remarks**:

1. Structural subdomains modeled with aid of continuum elements should be discretized using `Structured` grids
2. Number of elements along common boundaries of adjacent subdomains can be controlled during discretization once the `Adjust split...` checkbox (D) is set to ON
3. If `Adjust split...` is set to OFF then meshes in adjacent subdomains will not be compatible; special `Mesh tying` elements, available at the finite element level can remedy this problem
4. Virtual mesh is treated here as a guess for the real mesh but subdomain is not yet subdivided into elements.



**Window 2-183:** `Subdomain:Mesh:Virtual mesh(3D cont.):Struct.B8 template`

To create structured virtual mesh for 3D continuum subdomain, created on B8 skeleton, it is enough to pick the subdomain. Definition of control points (8 appices) and macro-brick boundaries is inherited from definition of the subdomain.

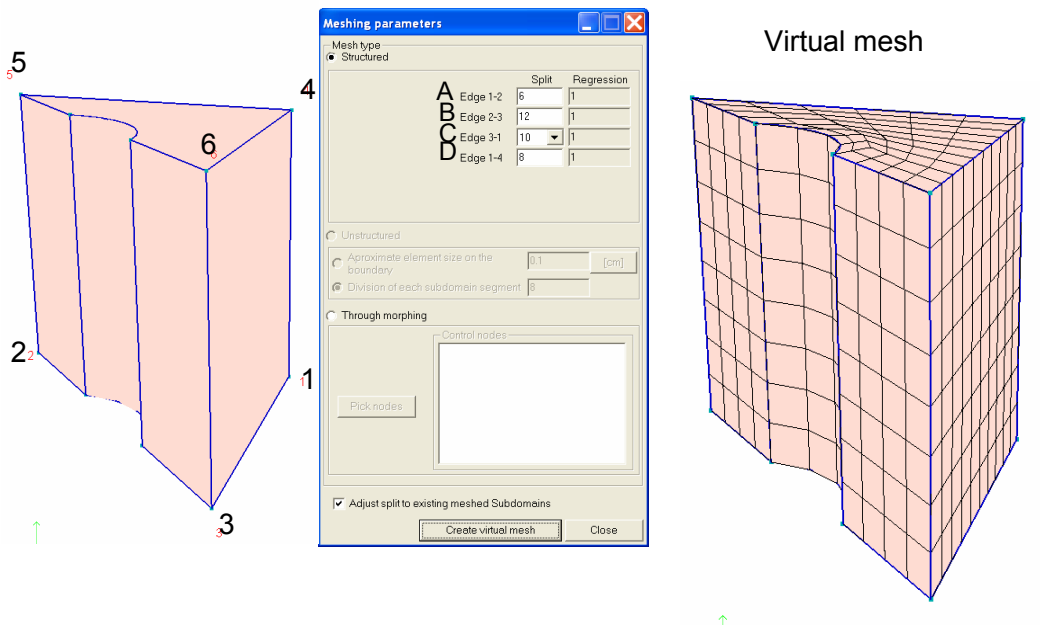



**Remarks**:

1. Modification of the virtual mesh density can be made by changing split values for edges designed by control points 1-2 and 1-4 (edit fields (F) and (G))
2. Control points cannot be changed
3. Additional mesh refinement can be made at the `Finite Element` level using method `Mesh refinement` under menu `Continuum 3D` once the virtual mesh in the subdomain is converted to the real one (with method `Mesh:virtual→real`)

**Window 2-184:** `Subdomain:Mesh:Virtual mesh(3D cont.):Struct.W6 template`

To create structured virtual mesh for 3D continuum subdomain, created on W6 skeleton, it is enough to pick the subdomain. Definition of control points (6 appices) and macro-wedge boundaries is inherited from definition of the subdomain.




**Remarks**:

1. Modification of the virtual mesh density can be made by changing split values for edges designed by control points 1-2 and 1-4 (edit fields (F) and (G))
2. Control points cannot be changed
3. Additional mesh refinement can be made at the `Finite Element` level using method `Mesh refinement` under menu `Continuum 3D` once the virtual mesh in the subdomain is converted to the real one (with method `Mesh:virtual→real`)

Window 2-184

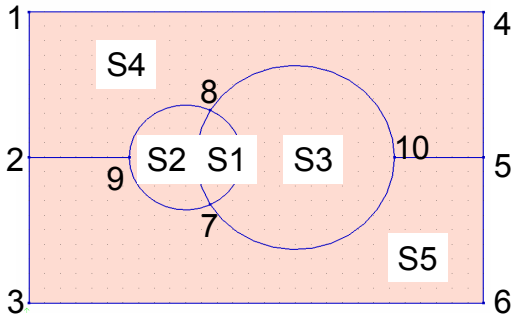
**Window 2-185:** `SubD:Mesh:Virt.mesh(3D shell/memb.:Struct.Q4/T3 template`

The structured virtual mesh for 3D one layer shell or membrane subdomain, created on Q4/T3 skeleton, can be defined in the same manner as structured virtual mesh for the 2D continuum (see Windows 2-180, 2-181)

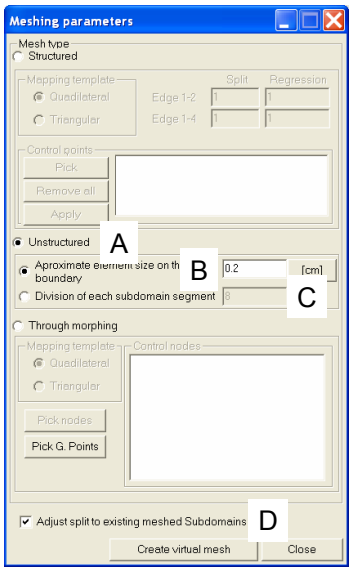
Window 2-185

**Window 2-186:** SubD:Mesh:Virtual Mesh(2D):Unstructured (elem. size)

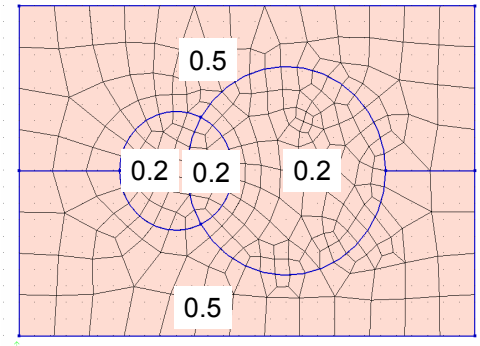
To create unstructured virtual mesh in a selected single subdomain it is enough to pick it and then to select option Unstructured (A) with definition through Approximate element size. For this option the approximate element size, expressed in length unit, must be given in the edit field (B) (see Figure below).




Definition through approximate element size on subdomain boundaries




Distribution of approximate element size



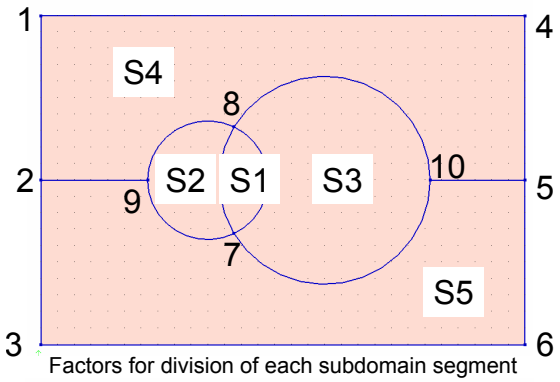

If the same element size is to be used in whole set of subdomains then one may select all subdomains and then run automatic meshing in all of them.

**Remarks**:

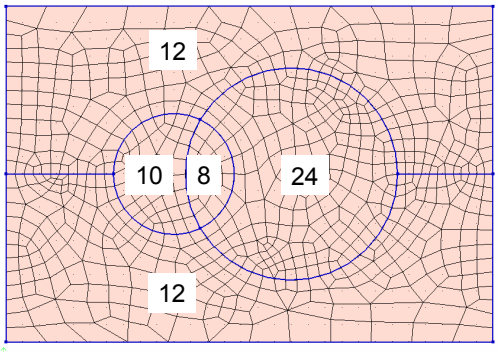
1. In case when mesh is created going from subdomain to subdomain (different element size is to be used for instance) then meshing should begin from internal subdomains to the external ones (in the case shown in the Figure the order: S1, S2, S3, S4, S5 is strongly recommended); this requirement is due to the fact that total number of elements on the whole subdomain boundary **must be an even number** (2,4,....2*N)
2. Generating meshes in subdomains by going from external to internal ones may cause a conflict (in the last subdomain number of elements on the boundary can be an odd number) and mesh will not be created
3. Resulting mesh always consists of quads
4. To preserve mesh conformity among adjacent subdomains the Adjust split to existing... checkbox must be set to ON; in this case boundaries already discretized keep unchanged their current mesh density
5. Different element size can be used
6. Additional mesh refinement can be made at the Finite Element level using method Mesh refinement under menu Continuum 2D once the virtual mesh in the subdomain is converted to the real one (with method Mesh:virtual→real)
7. **This method may also be used to generate virtual meshes on planar 3D subdomains created by method** Subdomain/2D on contour

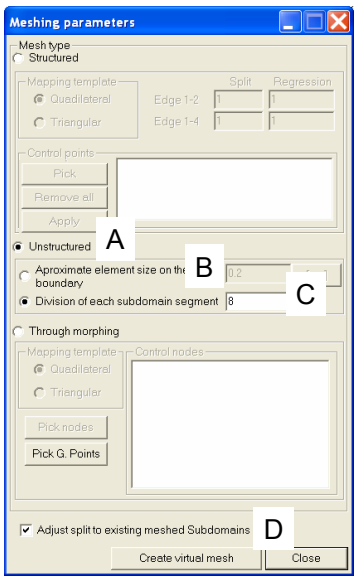
**Window 2-187:** SubD:Mesh:Virtual Mesh(2D):Unstruct.(boundary division)

To create unstructured virtual mesh it is enough to pick the subdomain and then to select option Unstructured (A) with definition through Division of each subdomain segment. For this option the division factor which applies to each subdomain boundary segment (in the example below, subdomain S3, for instance, consists of 3 segments 8-10,10-7,7-8 and 10-7) must be given in the edit field (C).




Factors for division of each subdomain segment



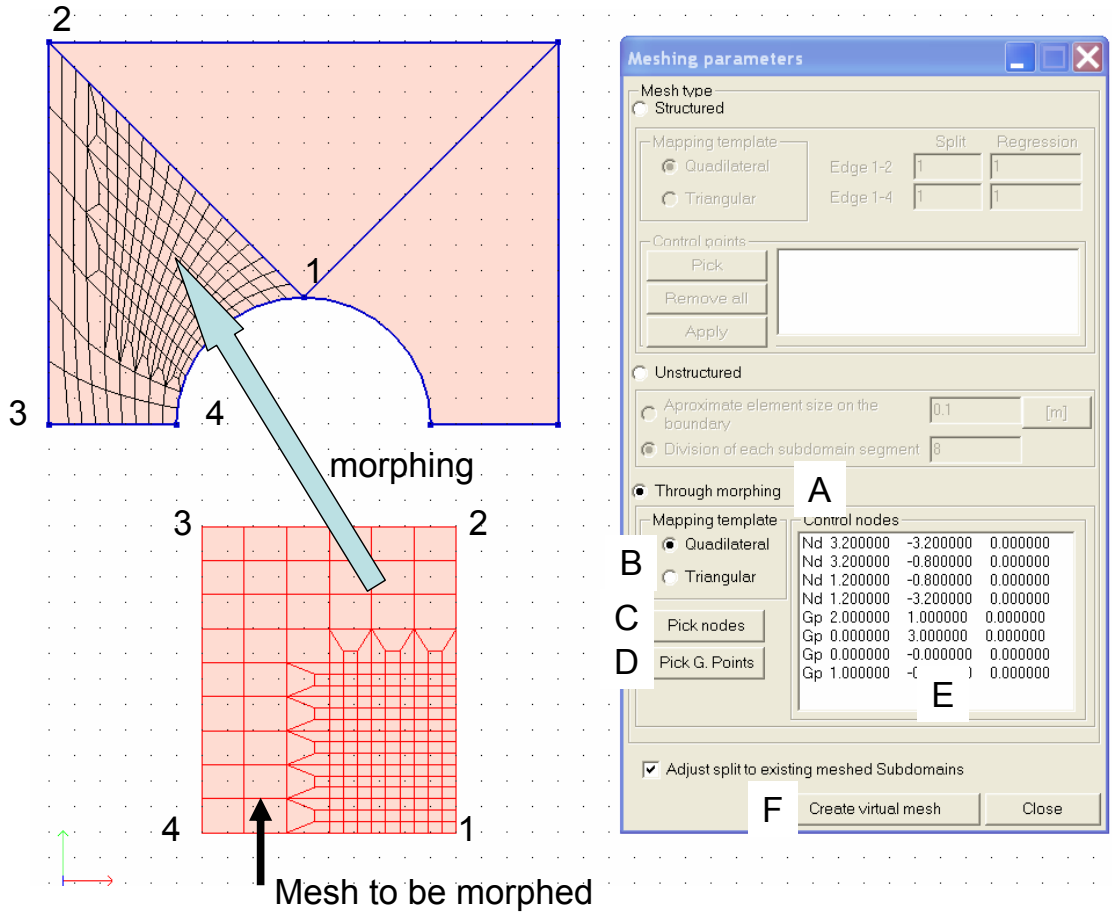




**Remarks**:

1. Creating virtual mesh should always begin by going from internal subdomains to the external ones (in the case shown in the Figure the order: S1, S2, S3, S4, S5 is strongly recommended); this requirement is due to the fact that total number of elements on the whole subdomain boundary **must be an even number** (2,4,....2*N)
2. Generating meshes in subdomains by going from external to internal ones may cause a conflict (in the last subdomain number of elements on the boundary can be an odd number) and mesh will not be created
3. Resulting mesh always consists of quads
4. To preserve mesh conformity among adjacent subdomains the Adjust split to existing... checkbox must be set to ON
5. Different division of subdomain segments can be used
6. Additional mesh refinement can be made at the Finite Element level using method Mesh refinement under menu Continuum 2D once the virtual mesh in the subdomain is converted to the real one (with method Mesh:virtual→real)
7. **This method may also be used to generate virtual meshes on planar 3D subdomains created by method** Subdomain/2D on contour

**Window 2-188:** `Subdomain:Mesh:Virtual Mesh (Cont.2D):Through morphing`

A virtual mesh for continuum 2D subdomain can be created by morphing (A) of an existing 2D mesh on subdomain using triangular or quadrilateral template (B). Three (for triangular template) or four nodes (for quadrilateral template) in the 2D mesh to be morphed must be picked first (with button `Pick nodes` (C)) and then corresponding 3 or 4 points on subdomain contour (with button `Pick G. Points` (D)) (list (E) is automatically updated). The example of this procedure is shown in the figure below.




**Remarks**:

1. 3/4 nodes in the 2D mesh and then 3/4 points in the subdomain contour must be picked in a strict order (clockwise or anticlockwise)
2. Additional mesh refinement can be made at the `Finite Element` level using method `Mesh refinement` under menu `Continuum 2D` once the virtual mesh in the subdomain is converted to the real one (with method `Mesh:virtual→real`)

Window 2-188

**Window 2-189:** Subdomain:Mesh:Virtual Mesh (Cont.3D):Through morphing

A virtual mesh for continuum 3D subdomain can be created by morphing an existing 3D mesh on subdomain using brick (B8) or wedge (W6) template. Eight (for B8 template) or six (for W6) in the 3D mesh to be morphed must be picked first (with button Pick nodes (C)) in order which strictly corresponds to subdomain vertices shown during definition (list (E) is automatically updated). The example of this procedure is shown in the figure below.

**Remark**:

Additional mesh refinement can be made at the Finite Element level using method Mesh refinement under menu Continuum 3D once the virtual mesh in the subdomain is converted to the real one (with method Mesh:virtual→real)

Window 2-189

**Window 2-190:** Subdomain:Mesh:Virtual→Real mesh

This method converts a temporary virtual mesh to the real one.

**Remarks**:

1. During mesh conversion (from virtual to real) real finite elements and nodal points are created
2. Additional mesh refinement can be made at the Finite Element level using method Mesh refinement under menu corresponding to the specific class of elements (like continuum, shell etc..) or by activating the Mesh refinement toolbar

Window 2-190

**Window 2-191:** Subdomain:Mesh:Remove virtual mesh

This method removes a temporary virtual mesh in picked subdomain or group of selected subdomains.

**Remarks**:

1. Subdomain is not deleted
2. Real mesh (if exists) remains undeleted

Window 2-191

**Window 2-192:** Subdomain:Mesh:Remove real mesh

This method removes a real mesh created in picked subdomain or group of selected subdomains

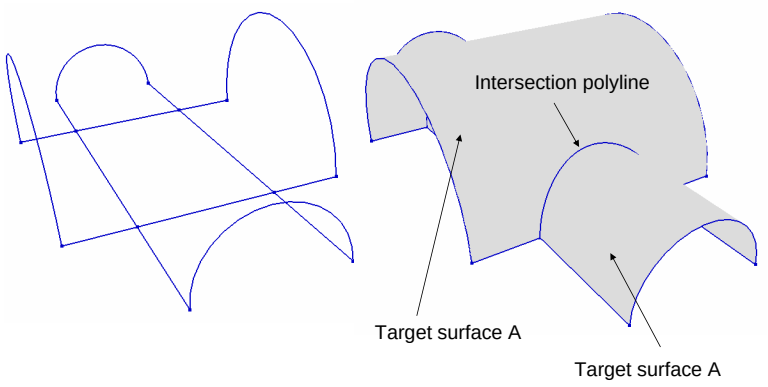
**Remarks**:

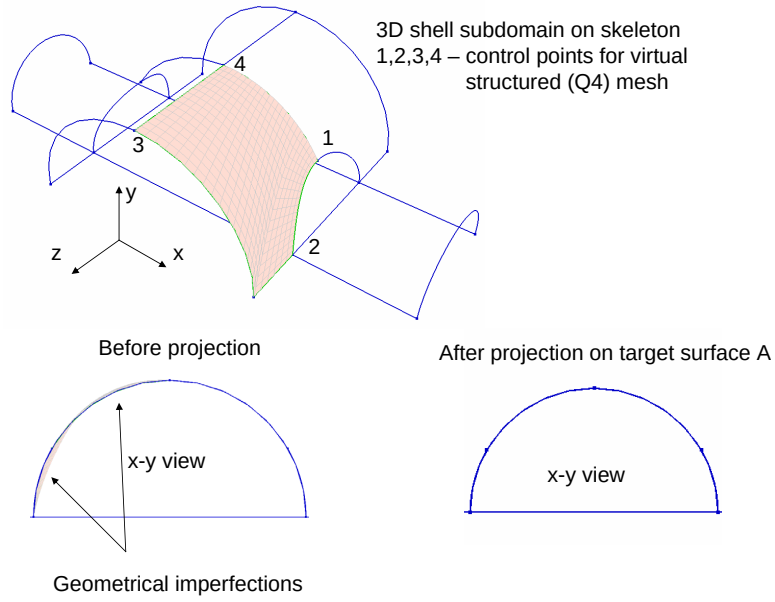
1. Subdomain is not deleted
2. Virtual mesh (if exists) remains undeleted

Window 2-192

**Window 2-193:** `Subdomain:Mesh:Project subdomain on surface`

This option may correct geometry of the 3D shell/membrane subdomains created on curvilinear Q4/T3 skeletons. It projects the virtual mesh on the selected target surface. The typical situation when this method proves useful is shown in the two following figures. The intersection of the two cylindrical surfaces, with two different radiuses, is considered here. As a result we obtain the two polylines, treated as single objects. To obtain meshes of a good quality it is advised to split the objects by two and then to create smaller subdomains. However, we may notice that the resulting surface subdomain (shell for instance) shows a certain level of geometrical imperfection. Projecting it on the selected target surface helps to recover the original geometry.




**Remarks**:

1. Each new creation of the virtual mesh requires a new projection of the virtual mesh on the target surface
2. The level of geometrical imperfection depends strongly on the geometrical complexity of the subdomain boundary
3. The virtual mesh setting (control points) is the optimal for the considered case

### 2.3.1.2.6 SEEPAGE

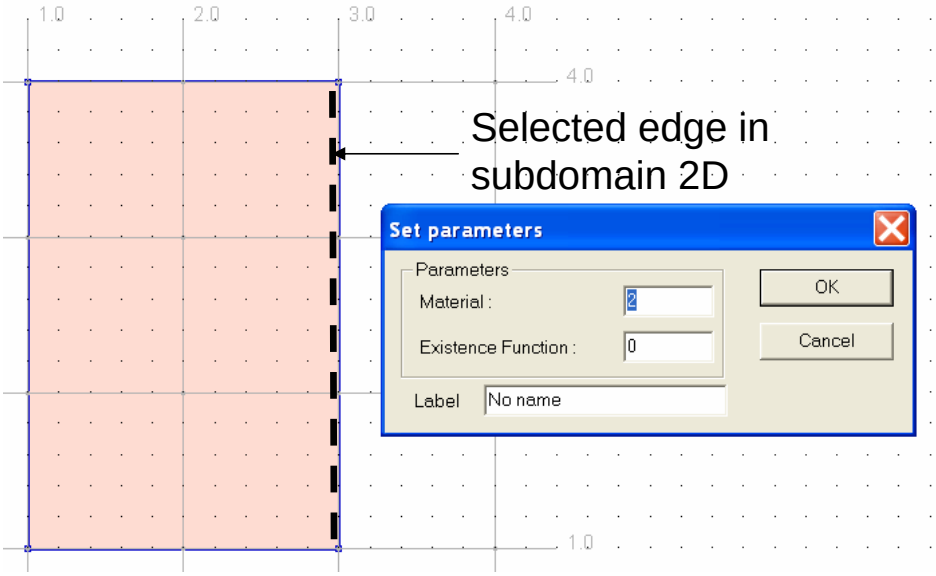
**Window 2-194: Options in menu** Seepage

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On Subdomain edge(s) | — | Window 2-195 |
| | On Subdomain face(s) | — | Window 2-196 |
| Select | All | — | Window 2-197 |
| | With material | — | Window 2-197 |
| | With existence function | — | Window 2-197 |
| | With label | — | Window 2-197 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-198 |
| | Scale | — | Window 2-198 |
| | Show | — | Window 2-198 |
| | Hide | — | Window 2-198 |

Window 2-194

**Window 2-195:** `Seepage:On Subdomain edges`

New 2D seepage macro-element(s) is(are) created on selected subdomain edge(s).



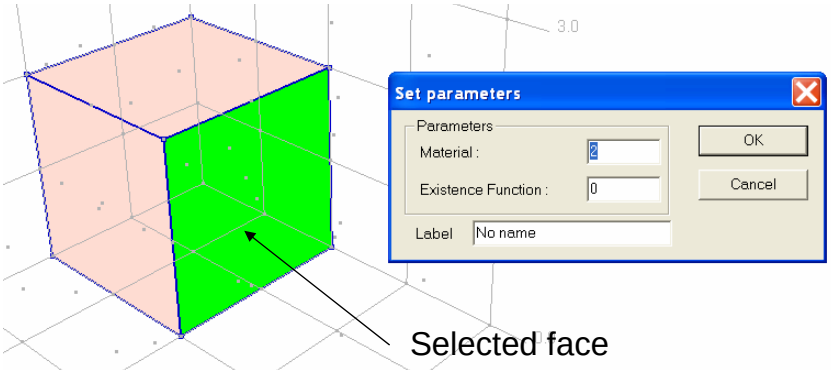

**Remarks**:

1. This macro-element is meaningful for `Flow` and `Deformation+Flow` analyzes
2. Seepage elements will be created on edges of the mesh of continuum elements once the virtual and then the real mesh in the continuum subdomain is created
3. Direction marker helps to identify on which continuum subdomain edge seepage macro-element is created (if two subdomains share the same edge)
4. More information on seepage elements (FE model level) is given in Win.(2-511)
5. Any mesh refinement in the adjacent continuum automatically enforces mesh refinement in the seepage macro-element

Window 2-195

**Window 2-196:** `Seepage:On Subdomain faces`

New 3D seepage macro-element(s) is(are) created by selecting face(s) in the existing continuum 3D subdomains. See remarks given in Win.(2-195).



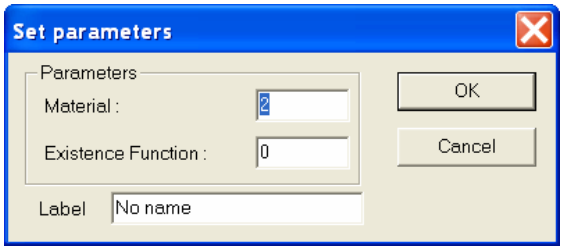

Window 2-196

**Window 2-197:** `Seepage`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible seepage macro-elements. |
| `With material` | select seepage macro-elements with a given material |
| `With existence function` | select seepage macro-elements with a given existence function |
| `With label` | select seepage macro-elements with a given label (all registered labels are given in the combo-box) |

Window 2-197

**Window 2-198:** `Seepage`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | modify the material and existence function ID, and label   |
| `Show` | show direction markers which help to identify on which continuum subdomain edge/face given seepage macro-element(s) is(are) created. |
| `Hide` | hide direction markers |
| `Scale` | enlarge or diminish size of the direction markers |

Window 2-198

### 2.3.1.2.7 HEAT EXCHANGER

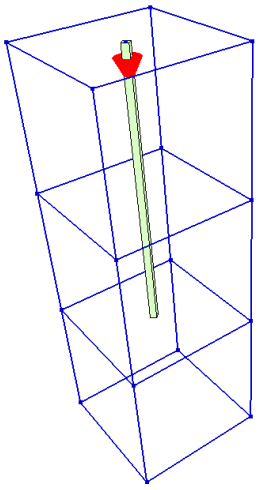
**Window 2-199: Options in menu** `Heat exchanger`

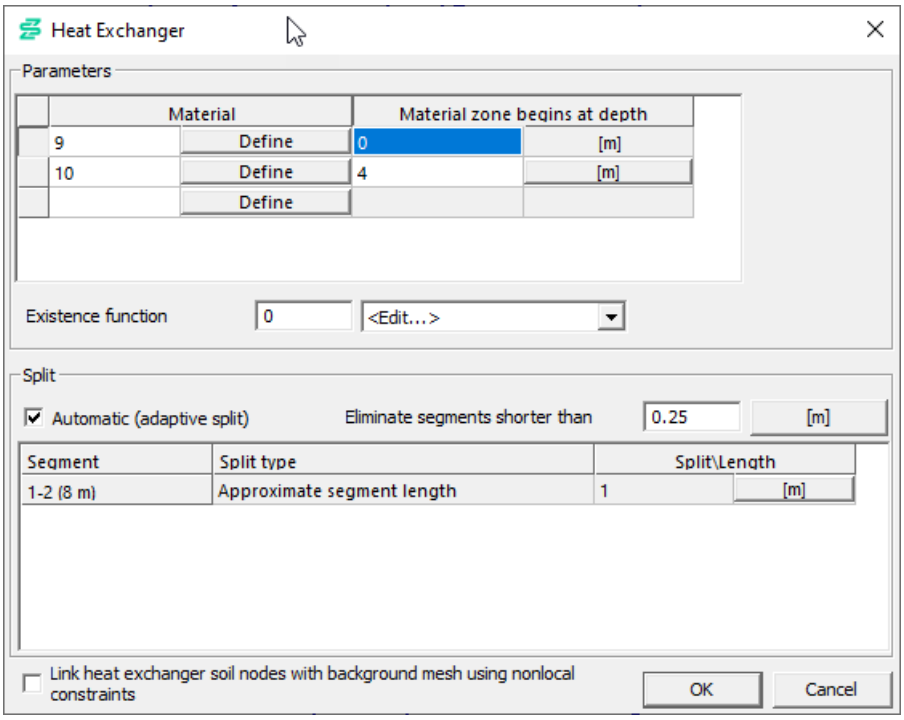
| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | BHE on objects | — | Window 2-200 |
| | Pipes on objects | — | Window 2-201 |
| | Connect pipes | — | Window 2-202 |
| | Copy with translation |  | Window 2-128 |
| | Copy with rotation |  | Window 2-129 |
| | Copy by symmetry |  | Window 2-130 |
| Select | All | — | Window 2-203 |
| | Box with 2 nodes |  | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With material | — | Window 2-203 |
| | With existence function | — | Window 2-203 |
| Delete | Delete |  | Window 2-320 |
| Delete | Delete Pipes link | | |
| Update | Parameters | — | Window 2-204 |
| | Invert orientation | — | Window ?? |
| | Split |  | Window 2-206 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Scale | — | Window 2-208 |

Window 2-199

**Window 2-200:** `Heat exchanger:Create: BHE on objects`

New macro borehole heat exchanger element, with the implicit grout zone (this zone is automatically built in), is created on the selected object(s). It is important to check its orientation which indicates direction in which refrigerant flows in the inlet pipe(s). Type of the borehole heat exchanger is selected at the material level, where user may select 1U, 2U, CXA or CXC configuration and corresponding parameters. **This option for heat exchanger generation is visible only at the macro-modeling level**.






**Remarks**:

1. Grout zone is always circular
2. To flip orientation use option `Update:Reverse orientation`
3. In order to model insulation zone user may split the exchanger onto two (of more if needed) zones; each zone has different exchanger material
4. Split data controls mesh density in the exchanger macro-element (set of exchanger finite elements); it is recommended to avoid too big differences in mesh densities between continuum and heat exchangers; such modeling may lead to high temperature oscillations;

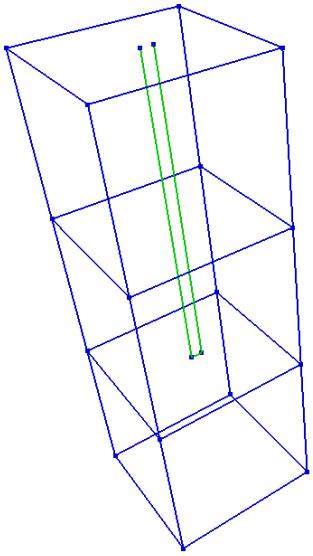
the default split is set to ☒ **Automatic** option in which exact intersection of exchanger geometry and continuum mesh is found; another method is to split exchanger on equal segments by setting split value or single element length at each detected geometrical object (exchanger can be defined on a polyline, in that case each detected segment is listed in the grid at the bottom part of the dialog box; split value may apply to each segment separately)

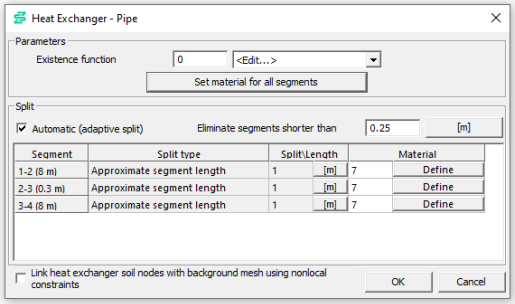
5. To avoid severe mesh dependency it is recommended to use non-local linking exchanger soil nodes with the background continuum mesh; in case this option is set OFF and exchanger mesh does not conform with the continuum mesh local form of constraints is used to connect dissimilar continuum and exchanger meshes

Window 2-200

## Window 2-201: `Heat exchanger:Create: Pipes on objects`

New macro borehole heat exchanger element, with the explicit grout zone (this zone must be discretized using continuum elements), is created on the selected object(s). It is important to check its orientation which indicates direction in which refrigerant flows in generated pipes. Here the exchanger is just a pipe and refrigerant. **This option for heat exchanger generation is visible only at the macro-modeling level**.






**Remarks**:

1. Grout zone can be of any form
2. To flip orientation use option `Update:Reverse orientation`
3. In case of pipes user may apply different material to each geometrical segment; using the option **Set material for all segments** user may set given material number to all segments
4. Split data controls mesh density in the exchanger macro-element (set of exchanger finite elements); it is recommended to avoid too big differences in mesh densities between continuum and heat exchangers; such modeling may lead to high temperature oscillations; the default split is set to ☒ **Automatic** option in which exact intersection of exchanger geometry and continuum mesh is found; another method is to split exchanger on equal segments by setting split value or single element length at each detected geometrical object (exchanger can be defined on a polyline, in that case each detected segment is listed

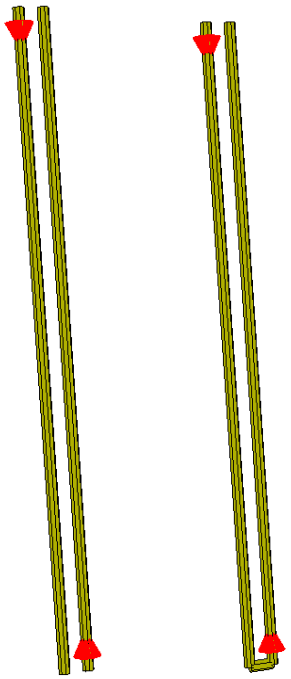
in the grid at the bottom part of the dialog box; split value may apply to each segment separately)

5. To avoid severe mesh dependency it is recommended to use non-local linking exchanger soil nodes with the background continuum mesh; in case this option is set OFF and exchanger mesh does not conform with the continuum mesh local form of constraints is used to connect dissimilar continuum and exchanger meshes
6. In case as shown in the figure there is no need to create short segments joining the two pipes; in that case user may use option `Connect pipes` which is adding a periodic boundary condition enforcing same temperature in both pipes (but not in soil)

Window 2-201

**Window 2-202:** `Heat exchanger:Create: Connect pipes`

This option applies only to pipe type heat exchangers (not the borehole ones). It enables one to connect separated pipes (if distance between them is small) by means of the periodic boundary condition preserving same temperature at linked pipes endpoints.



Window 2-202

**Window 2-203:** `Heat exchanger` **: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | Select all visible heat exchanger macro-elements |
| `With existence function` | Select all visible heat exchanger macro-elements with a given existence functions |
| `With material` | Select all visible heat exchanger macro-elements with a given exchanger material |

Window 2-203

**Window 2-204:** `Heat exchanger:Update:Parameters`

This option enables the user to modify settings for the selected heat exchangers.

Window 2-204

**Window 2-205:** `Heat exchanger:Update:Reverse direction`

This option enables one to define direction of refrigerant flow. It is highly important when combining few pipe segments into one.

Window 2-205

**Window 2-206:** `Heat exchanger:Update:Split`

This option enables one to define split for the exchanger macro-element. Please refer to window 2-200 or 2-201 for more detailed information.

Window 2-206

**Window 2-207:** `Heat exchanger:Update:Link to the background mesh`

This option enables one to define how the resulting exchangers FE mesh is connected to the background continuum mesh. One way is to use conforming meshes but then the background soil mesh grid size must carefully be adjusted with respect to the the grout diameter (for borehole exchangers with the implicit grout zone) or the pipe diameter in case of exchanger with the explicit grout zone. If both meshes are dissimilar (exchanger elements do not share edges with the continuum elements) user can use local or non-local linking exchanger soil nodes with the continuum mesh. **Non-local linking is highly recommended**. In case when the non-local linking option is set ON, no matter whether both meshes are conforming or not, exchnager soil nodes are linked to the background mesh using nonlocal form of temperature constraints. For more details concerning meshing and potential mesh dependency please refer to the dedicated report on heat exchangers.

Window 2-207

**Window 2-208:** `Heat exchanger:Update:Scale`

This option helps to scale the exchanger in the graph pane.

Window 2-208

## 2.3.1.2.8 CONVECTION/RADIATION

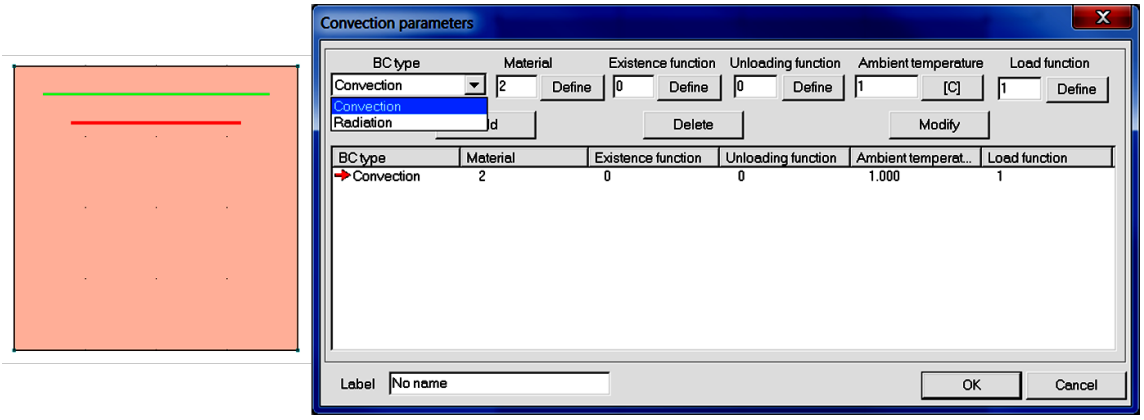
**Window 2-209: Options in menu** `Convection/Radiation`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Subdomain edge(s)` | — | Window 2-210 |
| | `On Subdomain face(s)` | — | Window 2-211 |
| `Select` | `All` | — | Window 2-212 |
| | `With material` | — | Window 2-212 |
| | `With existence function` | — | Window 2-212 |
| | `With label` | — | Window 2-212 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-213 |
| | `Scale` | — | Window 2-213 |
| | `Show` | — | Window 2-213 |
| | `Hide` | — | Window 2-213 |

Window 2-209

## Window 2-210: Convection/Radiation:On Subdomain edges

A new 2D convection and/or radiation macro-element(s) is(are) created on selected subdomain edge(s).



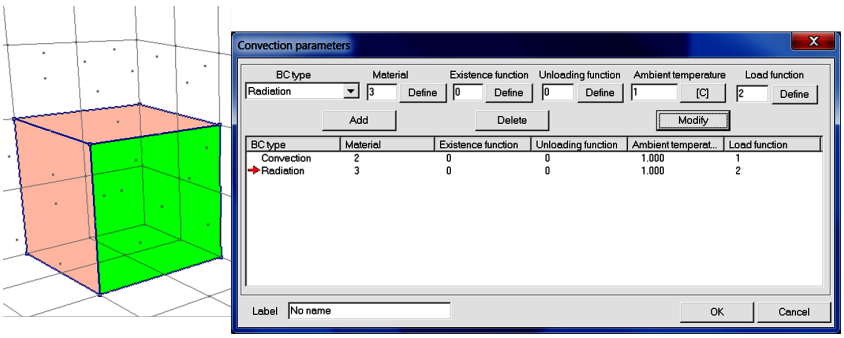

**Remarks**:

1. This macro-element is meaningful for Heat analysis; radiation elements are needed to set properly boundary conditions for fire problems
2. Convection/radiation elements will be created on edges/faces (3D) of the mesh of continuum elements once the virtual and then the real mesh in the continuum subdomain is created
3. More information on convection/radiation elements (FE model level) is given in Win.(2-525)
4. Any mesh refinement in the adjacent continuum automatically enforces mesh refinement in the convection/radiation macro-element

Window 2-210

## Window 2-211: Convection/Radiation:On Subdomain faces

A new 3D convection/radiation macro-element(s) is(are) created by selecting face(s) in the existing continuum 3D subdomains. See remarks given in Win.(2-210).




Window 2-211

**Window 2-212:** `Convection/Radiation`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible convection/radiation macro-elements. |
| `With material` | select convection/radiation macro-elements with a given material |
| `With existence function` | select convection/radiation macro-elements with a given material |
| `With label` | select convection/radiation macro-elements with a given label (all registered labels are given in the combo-box) |

Window 2-212

**Window 2-213:** `Convection/Radiation`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | modify the material, existence and unloading function ID, ambient temperature and associated with it load time function for detailed description see Win.(2-534) |

Window 2-213

## 2.3.1.2.9 VISCOUS DAMPER

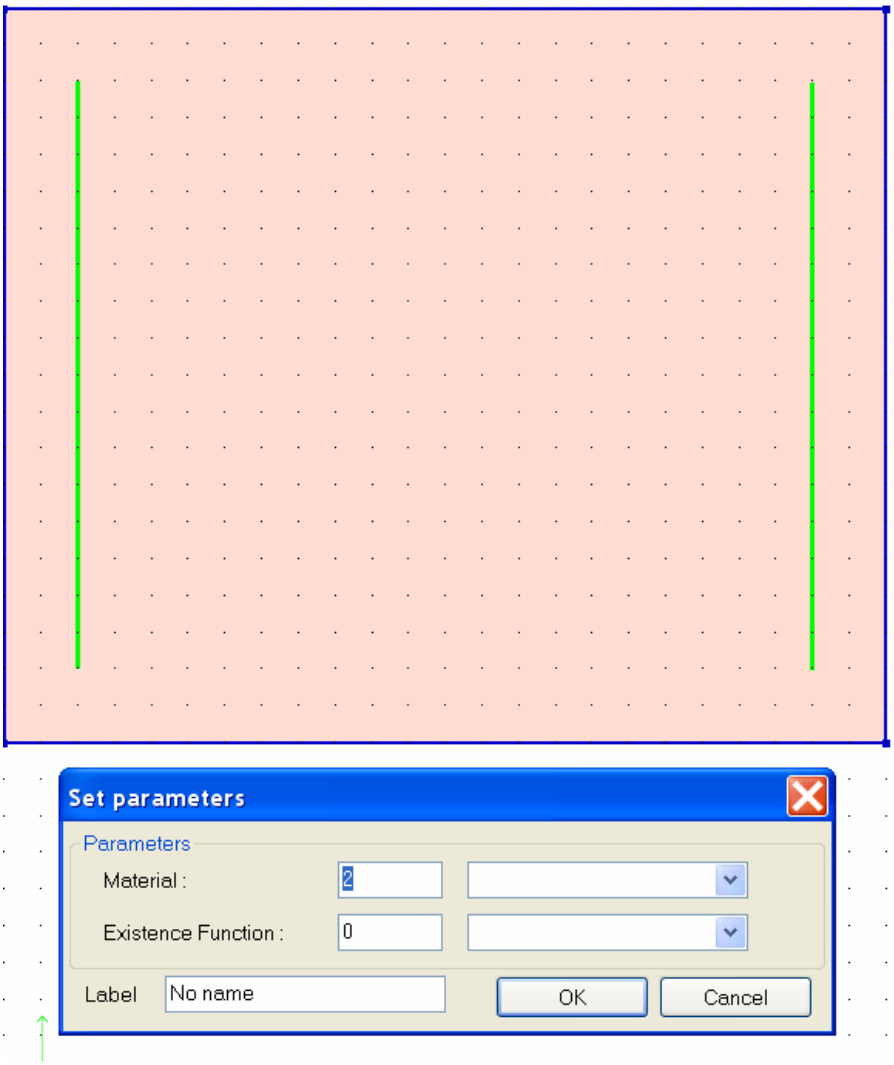
**Window 2-214: Options in menu** `Viscous damper`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Subdomain edge(s)` | — | Window 2-215 |
| | `On Subdomain face(s)` | — | Window 2-216 |
| `Select` | `All` | — | Window 2-217 |
| | `With material` | — | Window 2-217 |
| | `With existence function` | — | Window 2-217 |
| | `With label` | — | Window 2-217 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-218 |
| | `Scale` | — | Window 2-218 |
| | `Show` | — | Window 2-218 |
| | `Hide` | — | Window 2-218 |

Window 2-214

**Window 2-215:** `Viscous damper:On Subdomain edges`

A new 2D viscous damper macro-element(s) is(are) created on selected subdomain edge(s).



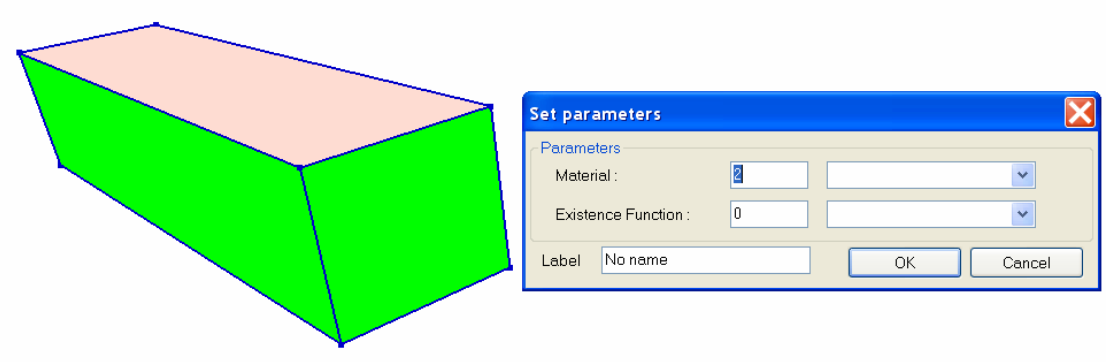

**Remarks**:

1. This macro-element is meaningful for `Deformation` or `Deformation+Flow` analyses when running `Transient dynamics` driver
2. Viscous damper elements will be created on edges of the mesh of continuum elements once the virtual and then the real mesh in the continuum subdomain is created
3. More information on damper elements (FE model level) is given in Win.(2-536)
4. Any mesh refinement in the adjacent continuum automatically enforces mesh refinement in the damper macro-element

Window 2-215

**Window 2-216:** `Viscous damper:On Subdomain faces`

A new 3D viscous damper macro-element(s) is(are) created by selecting face(s) in the existing continuum 3D subdomains. See remarks given in Win.(2-215).




Window 2-216

**Window 2-217:** `Viscous damper`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible damper macro-elements. |
| `With material` | select damper macro-elements with a given material |
| `With existence function` | select damper macro-elements with a given material |
| `With label` | select damper macro-elements with a given label (all registered labels are given in the combo-box) |

Window 2-217

**Window 2-218:** `Viscous damper`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | modify the material and existence function<br>for detailed description see Win.(2-544) |

Window 2-218

### 2.3.1.2.10 INTERFACE

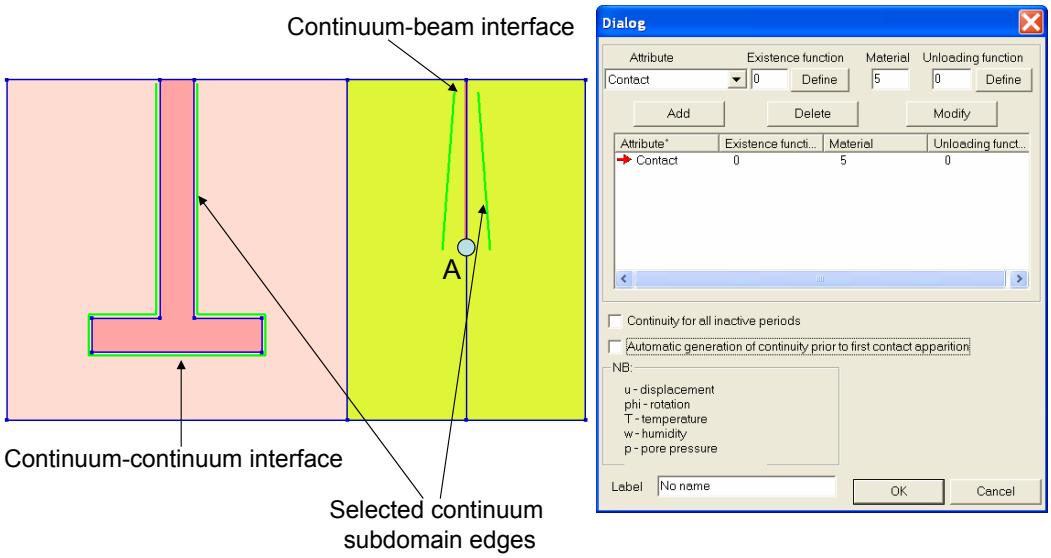
**Window 2-219: Options in menu** `Interface`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Continuum subd.face(s)` | — | Window 2-222 |
| | `On Structural subdomain(s)` | — | Window 2-221,2-223 |
| `Select` | `All` | — | Window 2-224 |
| | `With material` | — | Window 2-224 |
| | `With existence function` | — | Window 2-224 |
| | `With label` | — | Window 2-224 |
| | `With attribute` | — | Window 2-224 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-225 |
| | `Scale` | — | Window 2-225 |
| | `Show direction` | — | Window 2-225 |
| | `Hide direction` | — | Window 2-225 |

Window 2-219

**Window 2-220:** `Interface:On Subdomain edges`

A new 2D contact interface macro-element(s) is(are) created on selected subdomain edge(s).



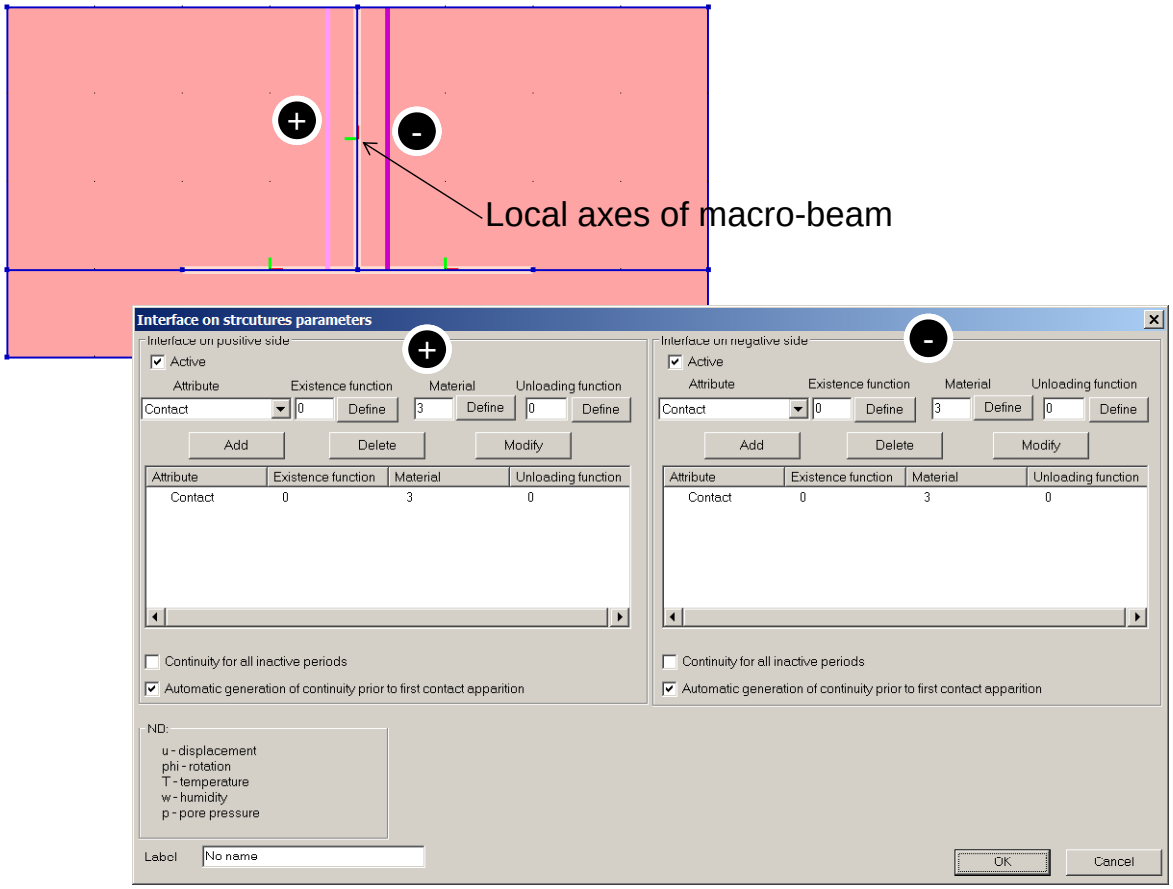

**Remarks**:

1. Interface elements will be created on edges of the continuum elements mesh once the virtual and then the real mesh in the continuum subdomain is created
2. The selected edge must be shared by the two subdomains
3. The interface can be created along continuum-continuum, continuum-beam, continuum-membrane and continuum-truss interfaces
4. Nonstandard situations like connection of the bottom point of the diaphragm wall with soil (point A in the figure; by default beam is separated from the continuum at this point) must be handled at the FE model level (see section 2.3.1.3.12); by default
5. More details on contact interface elements (described for the FE model level) can be found in section 2.3.1.3.12
6. Any mesh refinement in the adjacent continuum or structure automatically enforces mesh refinement in the contact interface

Window 2-220

**Window 2-221:** `Interface:On Structural subdomains (2D)`

To speedup generation of contact interfaces that may be adjacent to both faces of the structural subdomain (beams, membranes) we can define contact interface directly on the structural subdomain. Here we must consider carefuly the orientation of the local axes of the structural subdomain because during generating the interface we will get the access to the definition that concerns positive and negative side. The interface on the positive side will have light magenta color while the one on the negative side dark magenta.



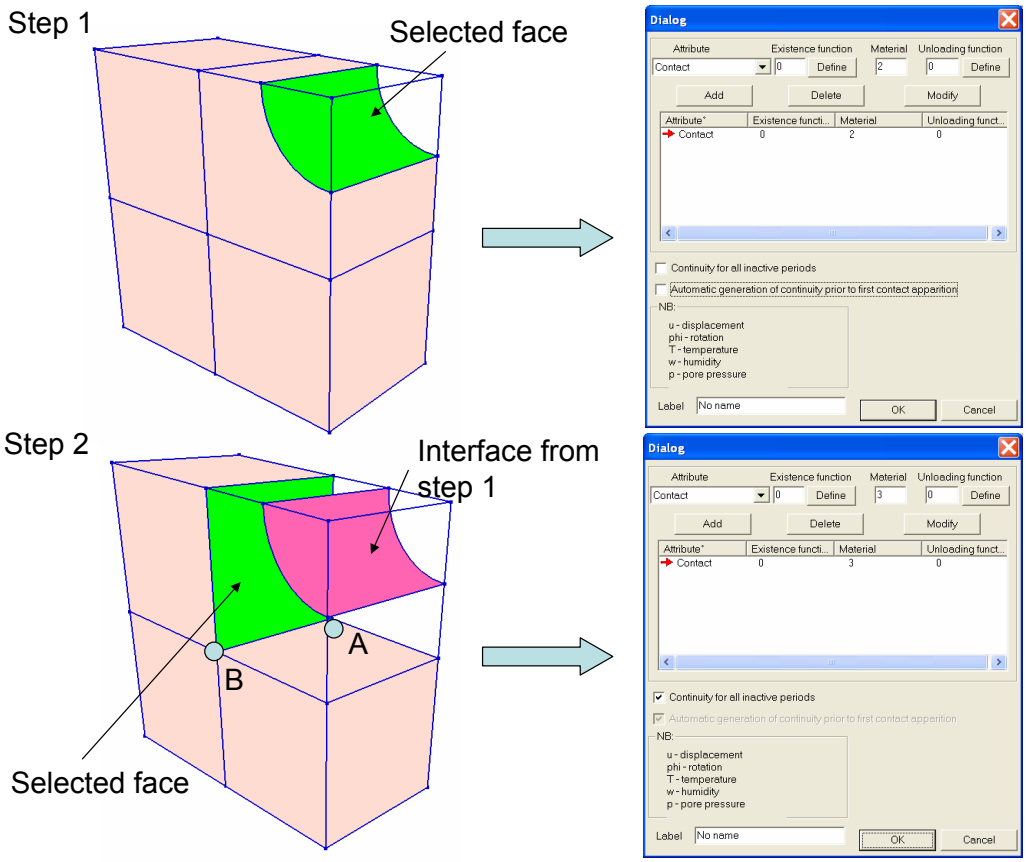

**Remarks**:

1. This definition is equivalent to the method `On Continuum subdomain edges` with this difference that two interfaces are generated in one step. Moreover we do not need to make too many select/show/hide operations as in the method where interface is defined on continuum subdomain edges

Window 2-221

**Window 2-222:** `Interface:On Subdomain faces`

A new 3D contact macro-element(s) is(are) created on selected 3D subdomain face(s).

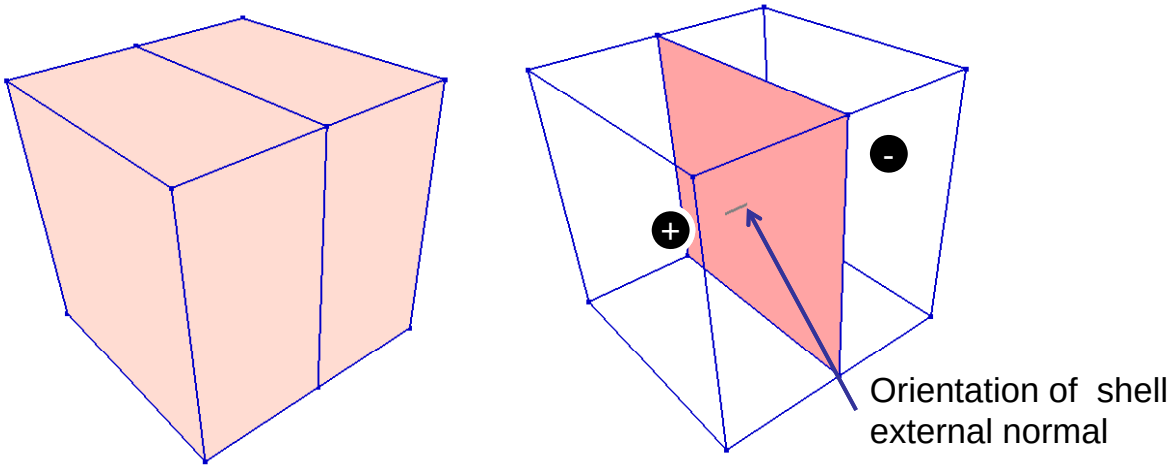


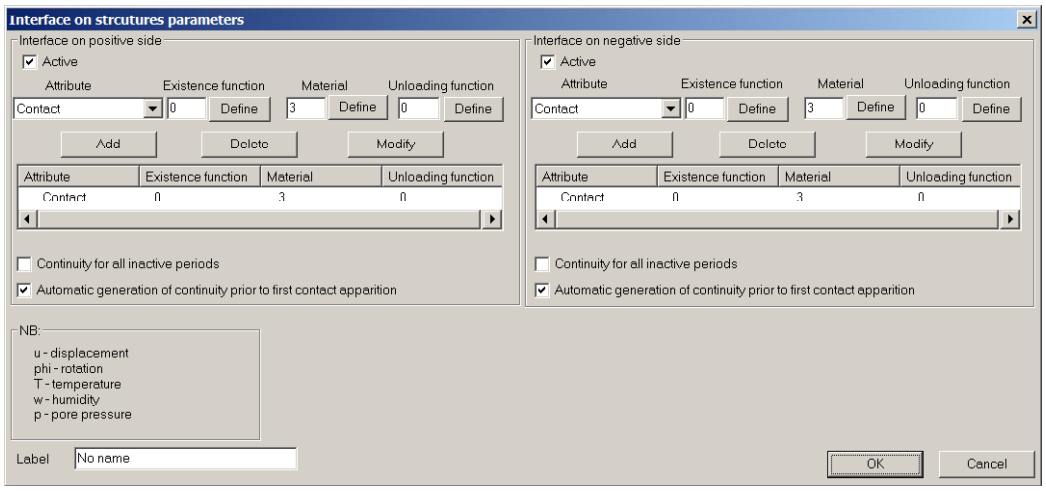

**Remarks**:

1. Interface elements will be created on faces of the continuum subdomain 3D once the virtual and then the real mesh in the continuum subdomain is created
2. The selected subdomain face must be shared by the two subdomains
3. The interface can be created along continuum-continuum, continuum-shell and continuum-membrane interfaces
4. Nonstandard situations like connection of the bottom edge of the diaphragm wall (line A-B in the figure; one-layer shell, placed in such interface, is separated from the continuum along line A-B by default) with soil must be handled at the FE model level (see section 2.3.1.3.12)
5. Any mesh refinement in the adjacent continuum or structure automatically enforces mesh refinement in the contact interface
6. To pick unvisible subdomain faces, subdomains blending the domain of interest must be temporarily hidden (select subdomain to be hidden first and then use Ctrl-T option or a toolbar (see Win.(2-75)))

**Window 2-223:** `Interface:On Structural subdomains (3D)`

To speedup generation of contact interfaces that may be adjacent to both faces of the structural subdomain (shells, membranes) we can define contact interface directly on the structural subdomain. Here we must consider carefuly the orientation of the local axes of the structural subdomain because during generating the interface we will get the access to the definition that concerns positive and negative side. The interface on the positive side will have light magenta color while the one on the negative side dark magenta.




**Remarks**:

1. This definition is equivalent to the method `On Continuum subdomain edges` with this difference that two interfaces are generated in one step. Again we do not need to make too many select/show/hide operations as in the method where interface is defined on continuum subdomain faces.

Window 2-223

**Window 2-224:** `Interface`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible interface macro-elements. |
| `With material` | select interface macro-elements with a given material |
| `With existence function` | select interface macro-elements with a given existence function |
| `With attribute` | select interface macro-elements with an attribute like `Contact`, `Continuity` |
| `With label` | select interface macro-elements with a given label (all registered labels are given in the combo-box) |

Window 2-224

**Window 2-225:** `Interface`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | modify the material, contact attributes, existence function ID, unloading function and label<br>for detailed description see Win.(2-483) |
| `Show direction` | show direction markers which help to identify orientation of the interface |
| `Hide direction` | hide direction markers |
| `Scale` | enlarge or diminish size of the direction markers |

Window 2-225

### 2.3.1.2.11 PRESSURE BC

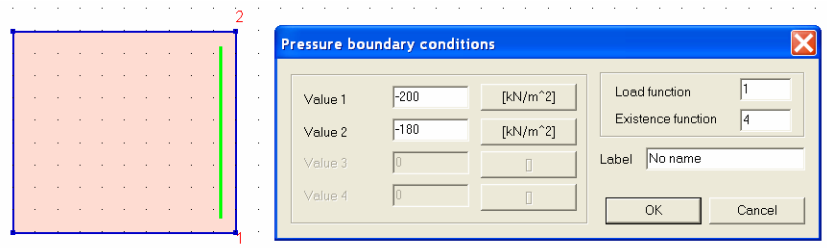
**Window 2-226: Options in menu** `Pressure BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On Subdomain edge | — | Window 2-227 |
| | Pressure head on selected edge(s) | — | Window 2-228 |
| | On Subdomain face | — | Window 2-229 |
| | Pressure head on selected face(s) | — | Window 2-230 |
| Select | All | — | Window 2-231 |
| | With material | — | Window 2-231 |
| | With existence function | — | Window 2-231 |
| | With label | — | Window 2-231 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-232 |
| | Scale | — | Window 2-232 |

Window 2-226

## Window 2-227: Pressure BC:On Subdomain edge

A new pressure boundary condition is created on the picked subdomain edge.



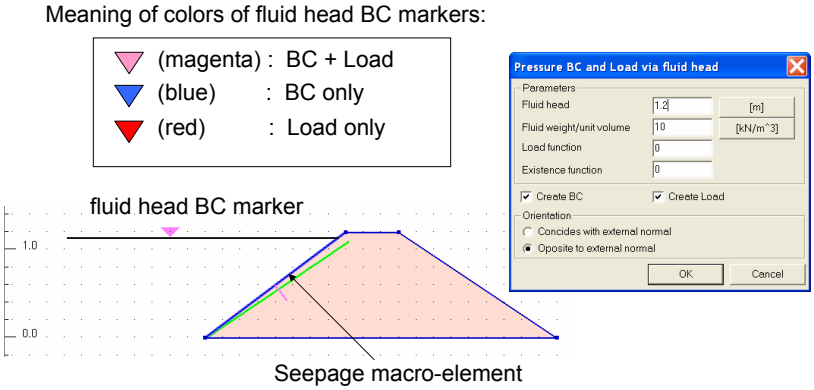

**Remarks**:

1. Imposed pressures are linearly interpolated along the boundary between the two vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed pressures in all boundary nodes of the real mesh

Window 2-227

## Window 2-228: Pressure BC:Pressure head on selected edges

A new, pressure head, type of the boundary condition is created on the selected subdomain edge(s) (see Win.(2-587) for detailed explanation of this type of the boundary condition). Pressure head is expressed by the formula: $h = -\frac{p}{\gamma^F} + y$



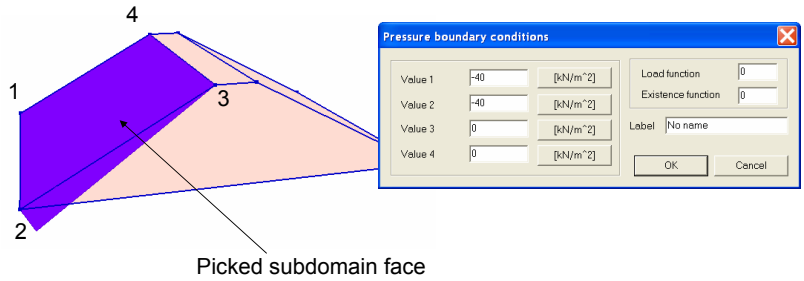

**Remarks**:

1. Imposed pressure head will be applied to all nodes on the boundary designed by selected faces if the checkbox ☒ **Create BC** is set to ON
2. Equivalent pressure load will be applied to the boundary once the checkbox ☒ **Create Load** is set to ON
3. The load time function applies to the pressure head value $h$
4. Mesh refinement is automatically handled and no extra action from the user is needed

Window 2-228

## Window 2-229: Pressure BC:On Subdomain face

A new pressure boundary condition is created on the picked 3D subdomain face. Face must be of T3 or Q4 type (this condition is satisfied for all 3D subdomains created on B8 or W6 skeletons)



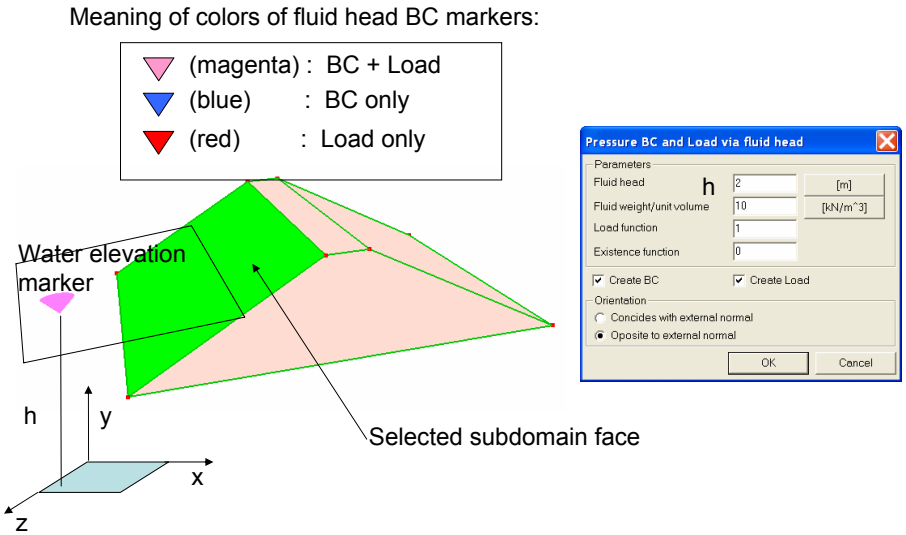

**Remarks**:

1. Imposed pressures are linearly (for T3 face) or bi-linearly (for Q4 face) interpolated using the pressure values set at vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed pressures in all boundary nodes of the real mesh

Window 2-229

## Window 2-230: Pressure BC:Pressure head on Subdomain face(s)

A new pressure head type of the boundary condition is created on selected 3D subdomain faces(s) (see Win.(2-591) for detailed explanation of this type of the boundary condition). Pressure head is expressed by the formula: $h = -\frac{p}{\gamma^F} + y$




**Remarks**:

1. Imposed pressure head will be applied to all nodes on the boundary designed by selected faces if the checkbox ☒ **Create BC** is set to ON
2. Equivalent pressure load will be applied to the boundary if the checkbox ☒ **Create Load** is set to ON
3. The load time function applies to the pressure head value $h$
4. Mesh refinement is automatically handled and no extra action from the user is needed

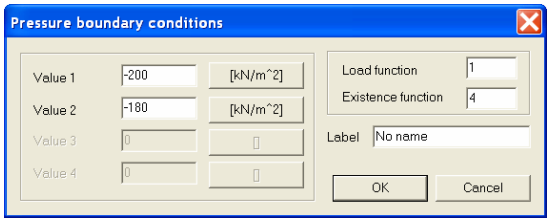
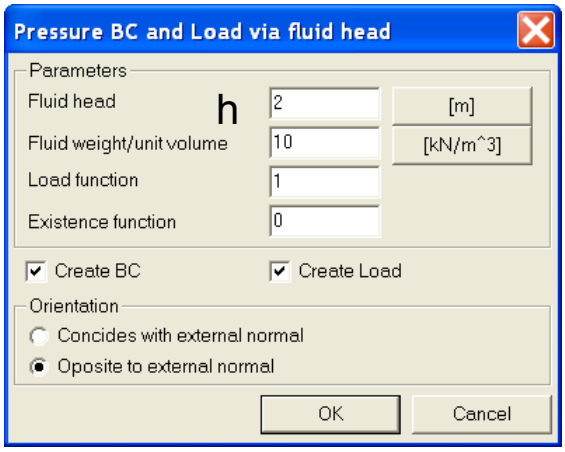
Window 2-230

**Window 2-231:** `Pressure BC`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible Pressure BC macro-objects. |
| `With load time function` | select Pressure BC macro-objects with a given load time function |
| `With existence function` | select Pressure BC macro-objects with a given existence function |
| `With label` | select pressure BC macro-objects with a given label (all registered labels are given in the combo-box) |

Window 2-231

**Window 2-232:** `Pressure BC`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | For pressure BC defined on edges/faces modify values at pressure macro-object vertices, load time function, existence function ID and label   For pressure head BC pick the marker top modify pressure head value, load time and existence functions   |

Window 2-232

### 2.3.1.2.12 TEMPERATURE BC

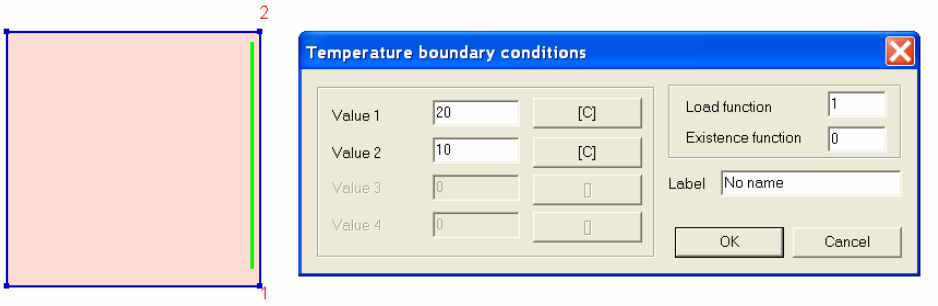
**Window 2-233: Options in menu** `Temperature BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Subdomain edge` | — | Window 2-234 |
| | `On Subdomain face` | — | Window 2-235 |
| `Select` | `All` | — | Window 2-236 |
| | `With material` | — | Window 2-236 |
| | `With existence function` | — | Window 2-236 |
| | `With label` | — | Window 2-236 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-237 |
| | `Scale` | — | Window 2-237 |

Window 2-233

## Window 2-234: Temperature BC:On Subdomain edge

A new temperature boundary condition is created on the picked subdomain edge.



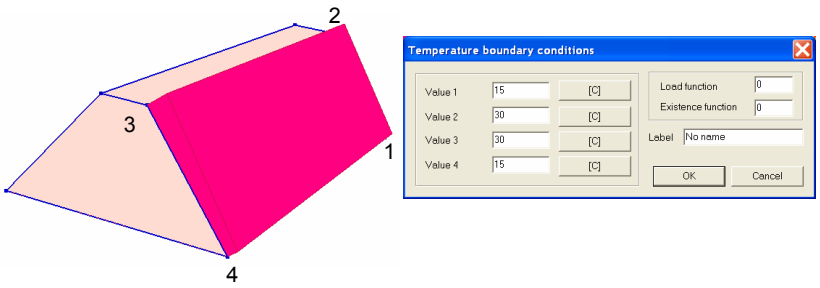

**Remarks**:

1. Imposed temperatures are linearly interpolated along the boundary between the two vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed temperatures in all boundary nodes of the real mesh

Window 2-234

## Window 2-235: Temperature BC:On Subdomain face

A new temperature boundary condition is created on the picked 3D subdomain face. Face must be of T3 or Q4 type (this condition is satisfied for all 3D subdomains created on B8 or W6 skeletons)




**Remarks**:

1. Imposed temperatures are linearly (for T3 face) or bi-linearly (for Q4 face) interpolated using the temperature values set at vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed temperatures in all boundary nodes of the real mesh

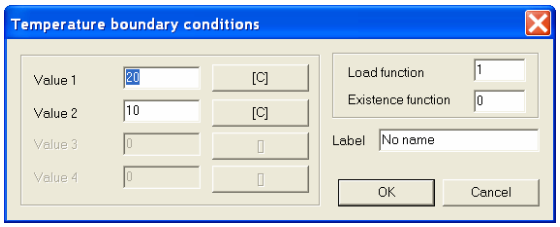
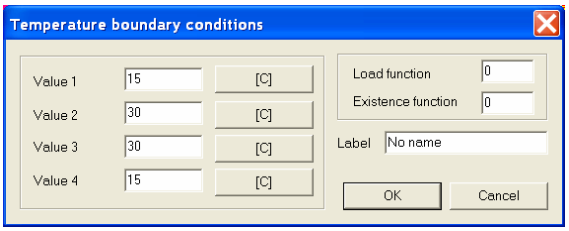
Window 2-235

**Window 2-236:** `Temperature BC`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible temperature BC macro-objects. |
| `With load time function` | select temperature BC macro-objects with a given load time function |
| `With existence function` | select temperature BC macro-objects with a given existence function |
| `With label` | select temperature BC macro-objects with a given label (all registered labels are given in the combo-box) |

Window 2-236

**Window 2-237:** `Temperature BC`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | Modify: temperatures at picked 2D temperature BC macro-object vertices, load time function, existence function ID and label   Modify: temperatures at picked 3D temperature BC macro-object vertices, load time function, existence function ID and label   |

Window 2-237

## 2.3.1.2.13 HUMIDITY BC

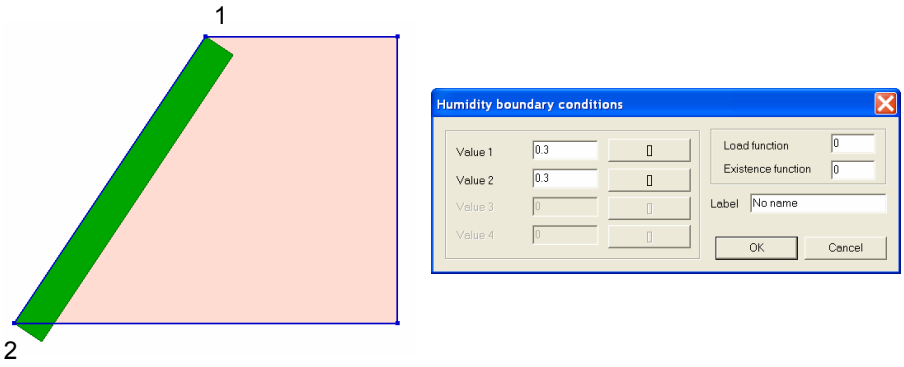
**Window 2-238: Options in menu** `Humidity BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Subdomain edge` | — | Window 2-239 |
| | `On Subdomain face` | — | Window 2-240 |
| `Select` | `All` | — | Window 2-241 |
| | `With material` | — | Window 2-241 |
| | `With existence function` | — | Window 2-241 |
| | `With label` | — | Window 2-241 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-242 |
| | `Scale` | — | Window 2-242 |

Window 2-238

## Window 2-239: Humidity BC:On Subdomain edge

A new humidity boundary condition is created on the picked subdomain edge.



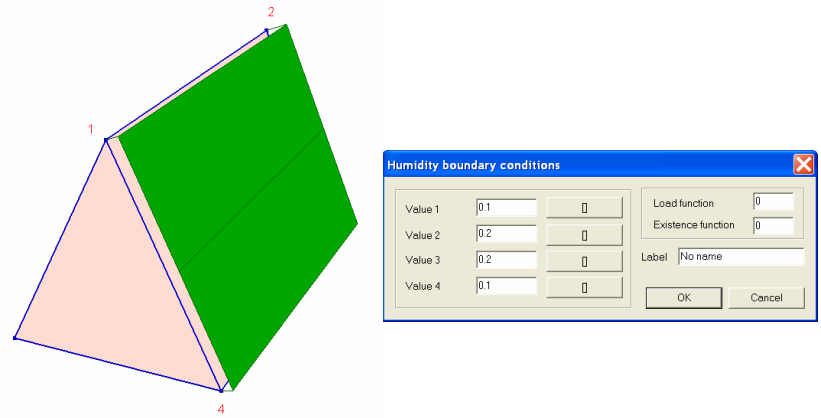

**Remarks**:

1. Imposed humidities are linearly interpolated along the boundary between the two vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed humidities in all boundary nodes of the real mesh

Window 2-239

## Window 2-240: Humidity BC:On Subdomain face

A new humidity boundary condition is created on the picked 3D subdomain face. Face must be of T3 or Q4 type (this condition is satisfied for all 3D subdomains created on B8 or W6 skeletons)




**Remarks**:

1. Imposed humidities are linearly (for T3 face) or bi-linearly (for Q4 face) interpolated using the humidity values set at vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed humidities in all boundary nodes of the real mesh

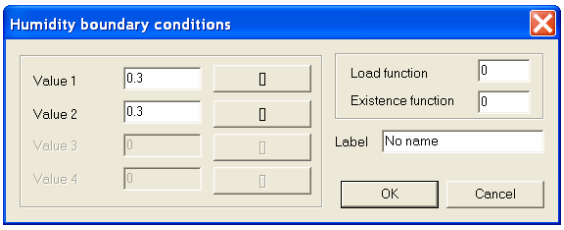
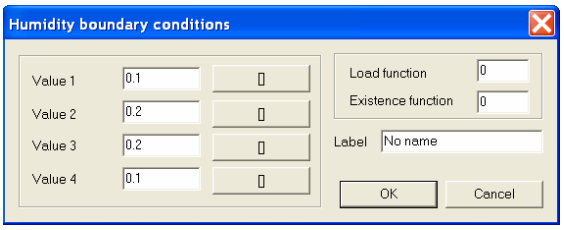
Window 2-240

**Window 2-241:** `Humidity BC`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible humidity BC macro-objects. |
| `With load time function` | select humidity BC macro-objects with a given load time function |
| `With existence function` | select humidity BC macro-objects with a given existence function |
| `With label` | select humidity BC macro-objects with a given label (all registered labels are given in the combo-box) |

Window 2-241

**Window 2-242:** `Humidity BC`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | Modify: humidities at picked 2D humidity BC macro-object vertices, load time function, existence function ID and label   Modify: humidities at picked 3D humidity BC macro-object vertices, load time function, existence function ID and label   |

Window 2-242

### 2.3.1.2.14 FLUID FLUX

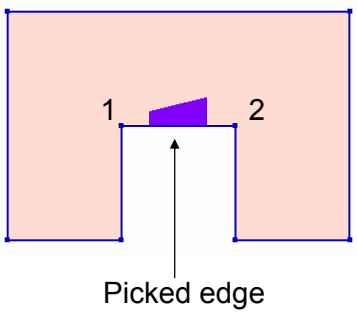
**Window 2-243: Options in menu** `Fluid flux`

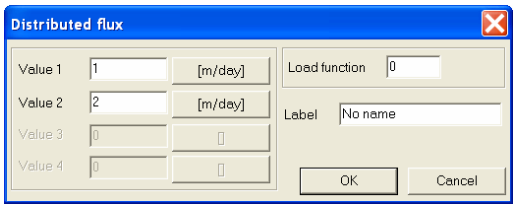
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Subdomain edge` | — | Window 2-244 |
| | `On Subdomain face` | — | Window 2-245 |
| `Select` | `All` | — | Window 2-246 |
| | `With material` | — | Window 2-246 |
| | `With label` | — | Window 2-246 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-247 |
| | `Scale` | — | Window 2-247 |

Window 2-243

## Window 2-244: `Fluid flux:On Subdomain edge`

A new fluid flux boundary condition is created on the picked 2D continuum subdomain edge.



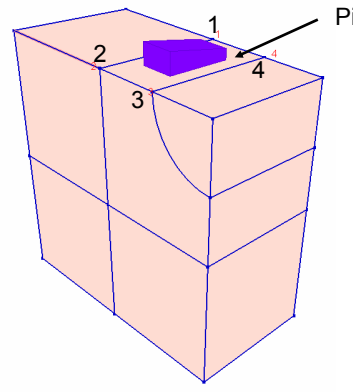

**Remarks**:

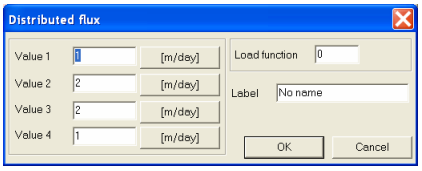
1. Imposed fluid fluxes are linearly interpolated along the boundary bound by the two vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed fluid fluxes in all boundary nodes of the real mesh
3. Fluid flux can be applied exclusively to continuum subdomain edges

Window 2-244

## Window 2-245: `Fluid flux:On Subdomain face`

A new fluid flux boundary condition is created on the picked 3D continuum subdomain face. Face must be of T3 or Q4 type (this condition is satisfied for all 3D subdomains created on B8 or W6 skeletons)




**Remarks**:

1. Imposed fluid fluxes are linearly (for T3 face) or bi-linearly (for Q4 face) interpolated using the fluid flux values set at vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed fluid fluxes in all boundary nodes of the real mesh
3. Fluid flux can be applied exclusively to continuum subdomain faces

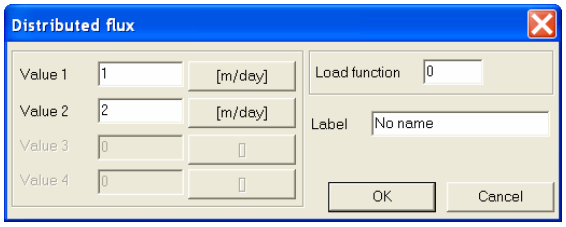
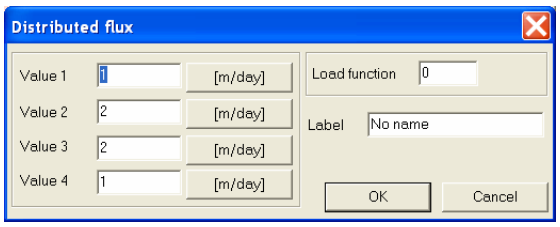
Window 2-245

**Window 2-246:** `Fluid flux`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible fluid flux BC macro-objects. |
| `With load time function` | select fluid flux BC macro-objects with a given load time function |
| `With label` | select fluid flux BC macro-objects with a given label (all registered labels are given in the combo-box) |

Window 2-246

**Window 2-247:** `Fluid flux`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | Modify: fluid fluxes at picked 2D fluid flux BC macro-object vertices, load time function and label   Modify: fluid fluxes at picked 3D fluid flux BC macro-object vertices, load time function and label   |

Window 2-247

## 2.3.1.2.15 HEAT FLUX/POWER

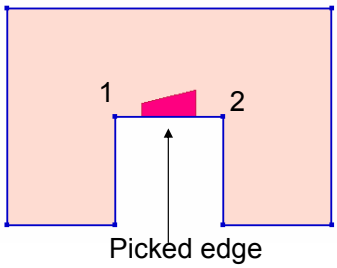
**Window 2-248: Options in menu** `Heat flux/Power`

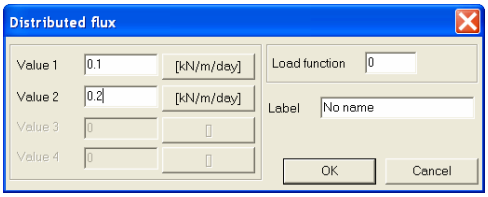
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On Subdomain face` | — | Window 2-250 |
| | `On Heat exchanger(s)` | — | Window 2-251 |
| `Select` | `All` | — | Window 2-252 |
| | `With material` | — | Window 2-252 |
| | `With label` | — | Window 2-252 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-253 |
| | `Scale` | — | Window 2-253 |

Window 2-248

## Window 2-249: Heat flux:On Subdomain edge

A new heat flux boundary condition is created on the picked 2D continuum subdomain edge.



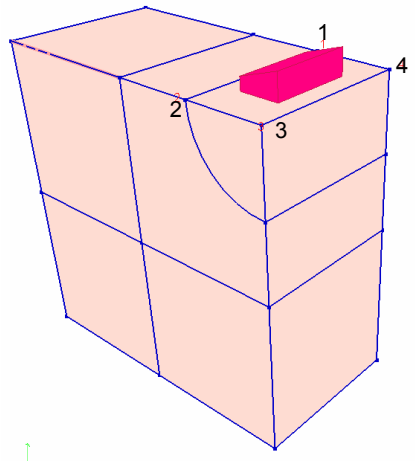

**Remarks**:

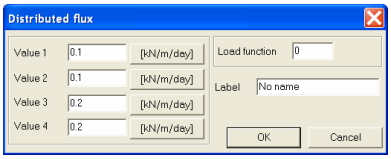
1. Imposed heat fluxes are linearly interpolated along the boundary bound by the two vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed heat fluxes in all boundary nodes of the real mesh
3. Heat flux can be applied exclusively to continuum subdomain edges

Window 2-249

## Window 2-250: Heat flux:On Subdomain face

A new heat flux boundary condition is created on the picked 3D continuum subdomain face. Face must be of T3 or Q4 type (this condition is satisfied for all 3D subdomains created on B8 or W6 skeletons)



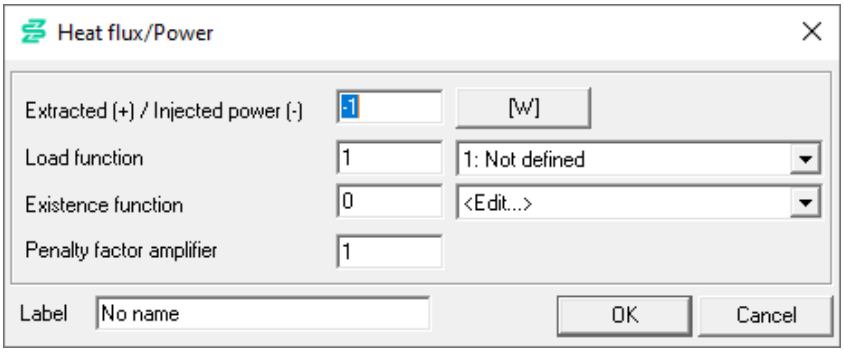

**Remarks**:

1. Imposed heat fluxes are linearly (for T3 face) or bi-linearly (for Q4 face) interpolated using the heat flux values set at vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed heat fluxes in all boundary nodes of the real mesh
3. Heat flux can be applied exclusively to continuum subdomain faces

Window 2-250

**Window 2-251:** `Heat flux:On Heat exchanger(s)`

A heat power boundary condition is created on the picked heat exchanger or selected heat exchangers. Since version 25.04 this condition is handled globally contrary to the previous implementations. It is important that extracted power has sign (+) while the injected one (-) respectively. If this boundary condition is applied to a set of selected heat exchangers then it means that sum of extracted/injected power from all of them is equal to the desired value which may vary in time.




**Remarks**:

1. This boundary condition can be associated with the load time and existence functions
2. To satisfy this boundary condition penalty method is used; therefore one may amplify or diminish the internally set penalty factor (this is done within the calculation module)
3. This boundary condition is visualized as a gray cork on top of the inlet pipe if applied to a distinct heat exchanger or by gray lines connecting selected exchangers

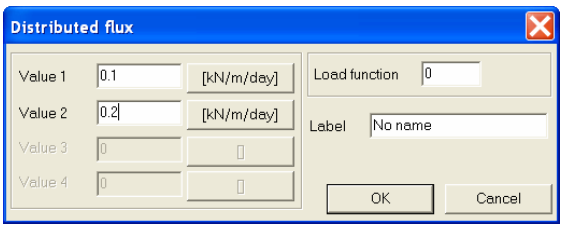
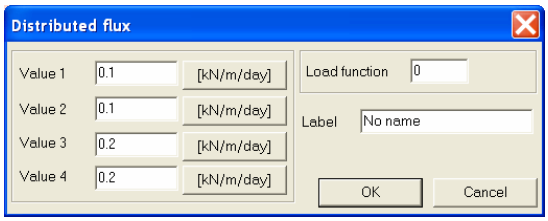
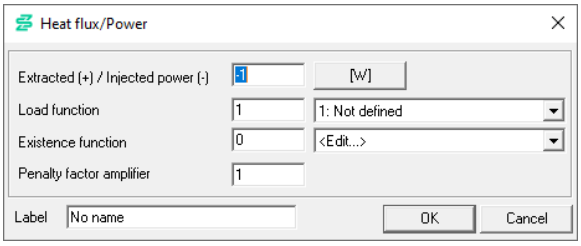
Window 2-251

**Window 2-252:** `Heat flux`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible heat flux/power BC macro-objects. |
| `With load time function` | select heat flux/power BC macro-objects with a given load time function |
| `With label` | select heat flux/power BC macro-objects with a given label (all registered labels are given in the combo-box) |

Window 2-252

**Window 2-253:** `Heat flux`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | Modify: heat fluxes at picked 2D heat flux BC macro-object vertices, load time function and label<br>  <br>Modify: heat fluxes at picked 3D heat flux BC macro-object vertices, load time function and label<br>  <br>Modify: picked heat power BC parameters<br>  |

## 2.3.1.2.16 HUMIDITY FLUX

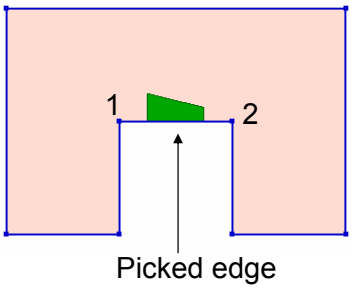
**Window 2-254: Options in menu** `Humidity flux`

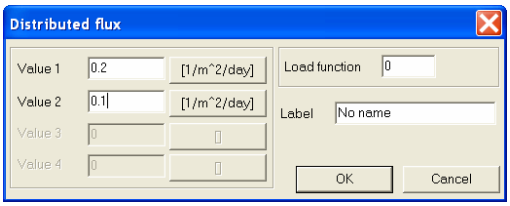
| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On Subdomain edge | — | Window 2-255 |
| | On Subdomain face | — | Window 2-256 |
| Select | All | — | Window 2-257 |
| | With material | — | Window 2-257 |
| | With label | — | Window 2-257 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-258 |
| | Scale | — | Window 2-258 |

Window 2-254

## Window 2-255: Humidity flux:On Subdomain edge

A new heat flux boundary condition is created on the picked 2D continuum subdomain edge.



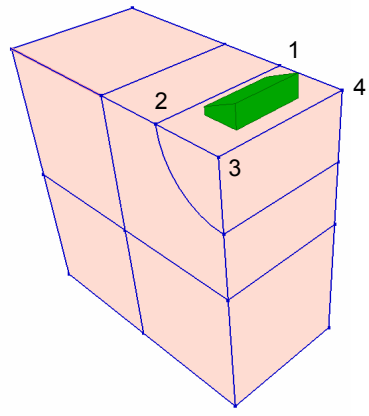

**Remarks**:

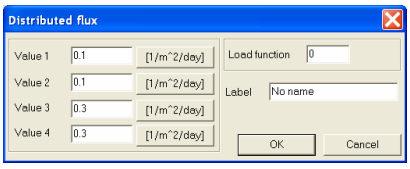
1. Imposed humidity fluxes are linearly interpolated along the boundary bound by the two vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed humidity fluxes in all boundary nodes of the real mesh
3. Humidity flux can be applied exclusively to continuum subdomain edges

Window 2-255

## Window 2-256: Humidity flux:On Subdomain face

A new humidity flux boundary condition is created on the picked 3D continuum subdomain face. Face must be of T3 or Q4 type (this condition is satisfied for all 3D subdomains created on B8 or W6 skeletons)




**Remarks**:

1. Imposed humidity fluxes are linearly (for T3 face) or bi-linearly (for Q4 face) interpolated using the humidity flux values set at vertices
2. Any mesh refinement in the adjacent continuum automatically enforces reevaluation of imposed humidity fluxes in all boundary nodes of the real mesh
3. Humidity flux can be applied exclusively to continuum subdomain faces

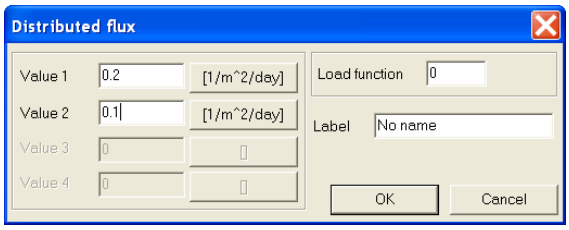
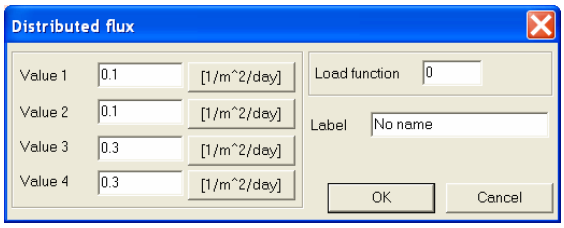
Window 2-256

**Window 2-257:** `Humidity flux`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | select all visible humidity flux BC macro-objects. |
| `With load time function` | select humidity flux BC macro-objects with a given load time function |
| `With label` | select humidity flux BC macro-objects with a given label (all registered labels are given in the combo-box) |

Window 2-257

**Window 2-258:** `Humidity flux`**: options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | Modify: humidity fluxes at picked 2D humidity flux BC macro-object vertices, load time function and label   Modify: humidity fluxes at picked 3D humidity flux BC macro-object vertices, load time function and label   |

Window 2-258

## 2.3.1.2.17 PILE

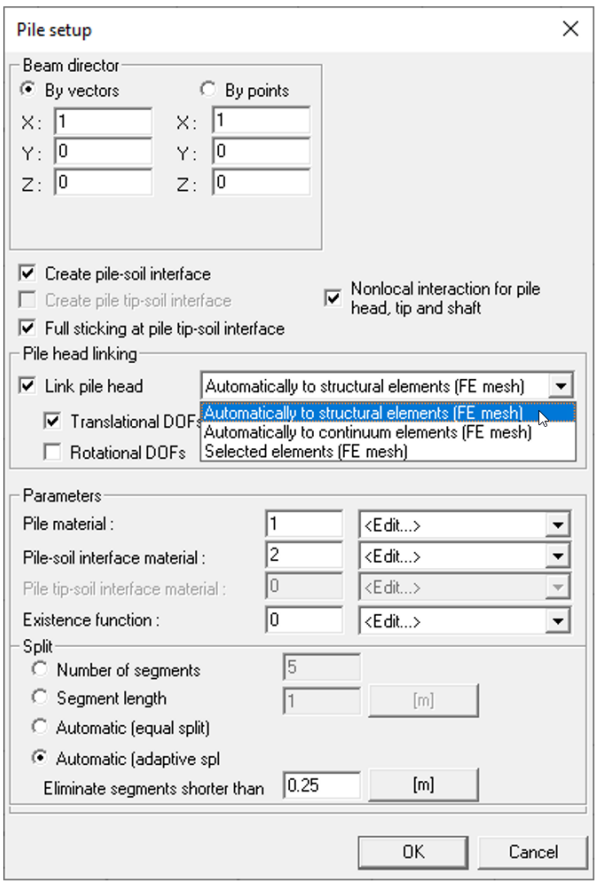
**Window 2-259: Options in menu** `Pile`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes/2 points | — | Window 2-260 |
| | 2 nodes | — | Window 2-261 |
| | Exter XYZ coord. | — | Window 2-262 |
| | On edges | — | Window 2-263 |
| | On objects | — | Window 2-264 |
| | Copy with translation |  | Window 2-128 |
| | Copy with rotation |  | Window 2-129 |
| | Copy by symmetry |  | Window 2-130 |
| Select | All | — | Window 2-265 |
| | Box with 2 nodes |  | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With material | — | Window 2-265 |
| | With existence function | — | Window 2-265 |
| | With unlinked head | — | Window 2-265 |
| | With length | — | Window 2-265 |
| | Overlapped | — | Window 2-265 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-266 |
| | Reverse X axis | — | Window 2-353 |
| | Link pile head | — | Window 2-267 |
| | Reverse X-axis | — | Window 2-269 |
| | Length | — | Window 2-268 |
| | Split |  | Window 2-270 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Scale | — | Window 2-271 |

Window 2-259

**Window 2-260:** `Pile:Create:2 nodes/points`

New pile macro-element (including interfaces) is created by picking the two points in the graph pane. The two picked points are first snapped to the existing nodes of the real mesh projected on the graph pane, then to the grid points, according to the setup under `Snap options`. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create a pile between the two existing nodes it is recommended to use the option `Create:2 nodes` described in Window 2-261. **This option for pile generation is visible only at the macromodeling level**.




**Remarks**:

1. The default orientation of local $y_L$ and $z_L$ axes is set up automatically by the preprocessor as the pile cross section is assumed to be circular; however, one may set it up in the edit fields `Beam director`; setting local pile cross section coordinate system follows the same procedure as for beam elements ( see Window (2-172)
2. To flip orientation of $e_{xL}$ base vector use option `Update:Reverse X-axis`
3. If ☒ **Create pile-soil interface** is set ON then the frictional contact interface will be automatically generated; this interface material will appear in the material menu as `Pile interface`
4. If ☒ **Create pile-soil interface** is set OFF then the full continuity of the displacement field for both pile and continuum will be enforced; this setting cancels the interface between pile tip and continuum
5. ☒ **Link pile head with** option is set ON (by default); top node of the pile will be linked `Automatically to structural elements (FE mesh)` or `Automatically to`

`continuum elements (FE mesh)` or to `Selected elements (FE mesh)`; note that linking of the pile head may be applied to translational (set ON ☒ **Translational DOFs**) and rotational degrees of freedom (set ON ☒ **Rotational DOFs**); this definition made at the macro level will be verified later on during generating the real FE mesh; the linking marker shows only that the linking decision was taken; it will further be verified in data verification stage

6. If ☒ **Create pile tip-soil interface** is set ON then stick-separation interface will be generated; this interface allows to simulate pull-out test and to put limits on the magnitude of compressive stresses under pile tip
7. If ☒ **Create pile tip-soil interface** is set OFF then the pile will freely penetrate the continuum under the pile tip unless ☒ **Full sticking at pile tip-soil interface** is set ON; in the latter case the full continuity of the displacement field for both pile tip and continuum under the pile tip will be enforced
8. If ☒ **Nonlocal interaction for pile head, tip and shaft** is ON then new (since version 2023) nonlocal formulation for 1D inserts embedded in the 3D continuum is enabled which preserves limited mesh dependency; two pile cross sections are handled circular and quadrilateral (for modeling barettes)
9. Split parameter controls mesh density in pile macro-element (set of beam elements and interfaces); it is recommended to avoid too big differences in mesh densities between continuum and piles; such modeling may lead to axial force oscillations in the pile for high strength parameters of subsoil; the two automatic options are available ie. with equal size split or adaptive split, including elimination of too short segments (to be defined by the user); with the other two methods older methods user may set split value or split length
10. To avoid instabilities due to rigid rotation of the pile along its axis the rotation along local pile axis is fixed internally by the preprocessor

Window 2-260

**Window 2-261:** `Pile:Create:2 nodes`

New pile macro-element is created by picking the two existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

The selection rule is the only difference between this option and the one given in Window 2-260 so refer to that Window for remaining information.

Window 2-261

**Window 2-262:** `Pile:Create:XYZ for 2 points`

New pile macro-element element is created by entering X,Y,Z coordinates of its two endpoints in the dialog box. Setting of the pile endpoints is the only difference between this option and the one given in Window 2-260 so refer to that Window for remaining information.

Window 2-262

**Window 2-263:** `Pile:Create:On edge(s)`

New pile macro-element(s) are generated on all selected edges (see figure below). Orientation of each pile macro-element element local $x$ axis is coherent with orientation of the element edge. The remaining pile setup is given in Window 2-260

Window 2-263

**Window 2-264:** `Pile:Create:On object(s)`

New pile macro-element(s) are generated on all selected geometrical objects (see figure below). Orientation of each pile macro-element local $x$ axis is coherent with orientation of the geometrical object. The remaining pile setup is given in Window 2-260

Window 2-264

**Window 2-265:** `Pile`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | Select all visible pile macro-elements |
| `With existence function` | Select all visible pile macro-elements with a given existence functions |
| `With material` | Select all visible pile macro-elements with a given pile material |
| `With unlinked head` | Select all visible pile macro-elements with unlinked heads |
| `With length` | Select all visible pile macro-elements with length selected from the list of all registered pile lengths |
| `Overlapped` | Select all visible pile macro-elements that are copied several times at the same location |

Window 2-265

**Window 2-266:** `Pile:Update:Parameters`

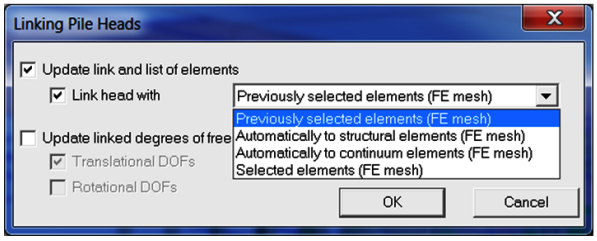
This option allows the user to modify setting for the selected pile or group of piles. In case of group of piles this option is equivalent to the one designed for creating of piles described in Win.( 2-260)

Window 2-266

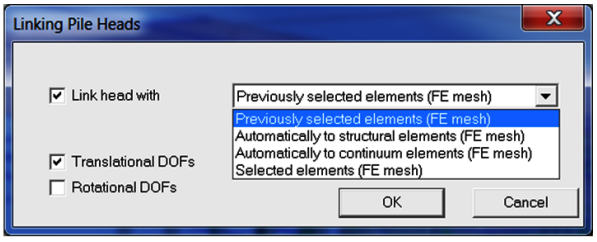
**Window 2-267:** `Pile:Update:Link pile head`

This option allows the user to modify linking of the pile head with other elements of the computational model. The typical situation concerns connection of the pile head with the foundation raft. In such cases we can assume full compatibility of both translational and rotational degree of freedom (if foundation raft is modeled with aid of shell elements), or only translational. By default linking option is set automaticaly to structural elements. The dialog boxes for this setting for a single picked pile or group of selected piles are shown in the figure below. In case of multi-selection one may update linking of the pile head and type of degrees of freedom to be constrained selectively. Note that linking will be analyzed when FE mesh is created (!). Note that in the case of redefining linking definition one additional option is present ie. `Link to previously selected elements`; this happens when the first definition was made with option to `Selected elements`. For more information please refer to Win.(2-260)

Dialog for selection of group of piles




Dialog for selection of one pile




Window 2-267

**Window 2-268:** `Pile:Update:Length`

This option allows the user to modify length of selected pile(s). One may set the length, elongate or shorten the pile by a given value. It is also possible to position, with a given offset, heads or tips of piles with respect to the selected set of faces. The latter option is valid only at the FE level. Effect of each action is presented in the graphical form in the dialog box.

Window 2-268

**Window 2-269:** `Pile:Update:Reverse X axis`

This option helps to control orientation of the pile local X axis. This orientation must be such that vector created between pile head and foot of the pile is the same as orientation of the X local axis. Using this option user may flip the orientation. This option is equivalent to the one designed to control orientation of beam elements (see Win.(2-353))

Window 2-269

**Window 2-270:** `Pile:Update:Split`

This option allows the user to modify pile macro-element(s) mesh density assuming number of segments, segments length or by using an automatic split with cancellation of too short segments. Some recommendations concerning split value are given in Win.(2-260).

Window 2-270

**Window 2-271:** `Pile:Update:Scale`

This option helps to scale size of the pile cross section in the graph pane. Note that the cross section is plot as quadrilateral (instead of cylindrical) to make rendering faster.

Window 2-271

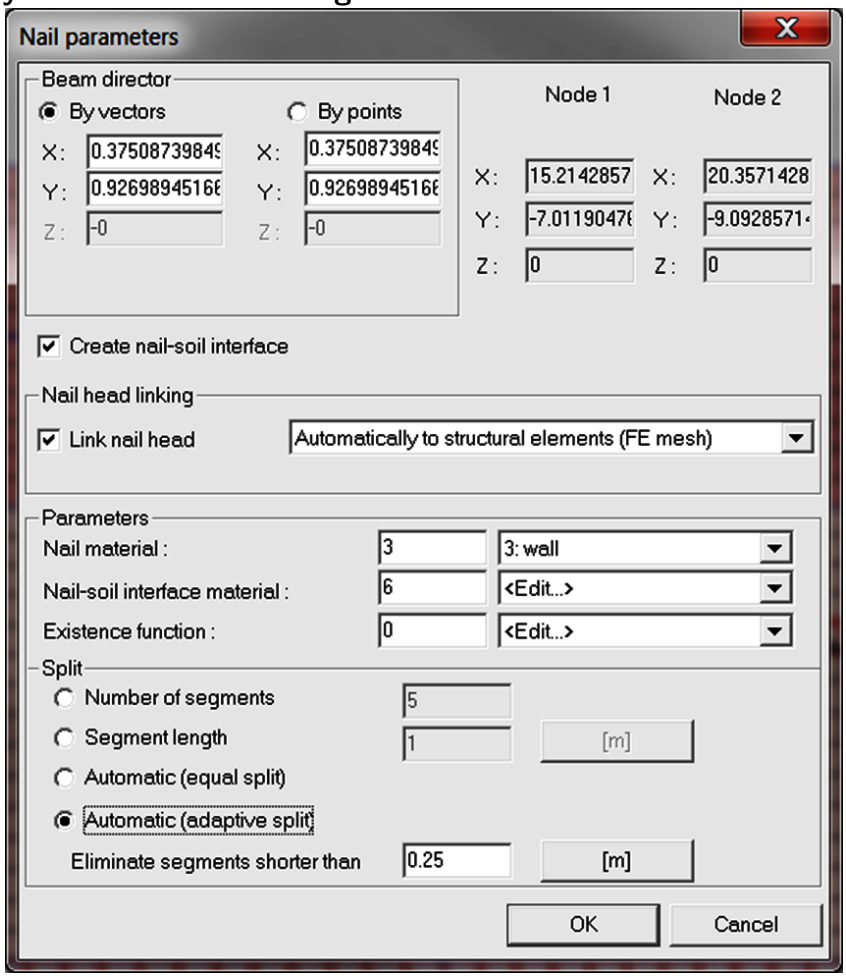
## 2.3.1.2.18 NAIL

**Window 2-272: Options in menu `Nail`**

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes/2 points | — | Window 2-273 |
| | 2 nodes | — | Window 2-274 |
| | Exter XYZ coord. | — | Window 2-275 |
| | On edges | — | Window 2-276 |
| | On objects | — | Window 2-277 |
| | Copy with translation |  | Window 2-128 |
| | Copy with rotation |  | Window 2-129 |
| | Copy by symmetry |  | Window 2-130 |
| Select | All | — | Window 2-278 |
| | Box with 2 nodes |  | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With material | — | Window 2-278 |
| | With existence function | — | Window 2-278 |
| | With unlinked head | — | Window 2-278 |
| | With length | — | Window 2-278 |
| | Overlapped | — | Window 2-278 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-279 |
| | Reverse X axis | — | Window 2-353 |
| | Link nail head | — | Window 2-280 |
| | Length | — | Window 2-281 |
| | Shift by head proj.on edges in dir | — | Window 2-282 |
| | Shift by head proj.on faces in dir | — | Window 2-283 |
| | Shift by head proj.on edges | — | Window 2-284 |
| | Shift by head proj.on faces | — | Window 2-285 |
| | Inclination angle | — | Window 2-286 |
| | Split |  | Window 2-288 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Scale | — | Window 2-289 |

**Window 2-273:** `Nail:Create:2 nodes/points`

New nail macro-element (including interfaces) is created by picking the two points in the graph pane. The two picked points will be first snapped to the existing nodes of the real mesh projected on the graph pane and then to the grid points according to the setup under `Snap options`. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create a nail between the two existing nodes it is recommended to use the option `Create:2 nodes` described in Window 2-274. **This option is visible only at the macromodeling level**.




**Remarks**:

1. The default orientation of local $y_L$ and $z_L$ axes is set up automatically by the preprocessor as the nail cross section is assumed to be circular; however, one may set it up in the edit fields `Beam director`; setting local nail cross section coordinate system follows the same procedure as for beam elements ( see Window (2-172)
2. To flip orientation of $e_{xL}$ base vector use option `Update:Reverse X-axis`
3. If ☒ **Create nail-soil interface** is set ON then the frictional contact interface will be automatically generated; this interface material will appear in the material menu as `Nail interface`
4. If ☒ **Create nail-soil interface** is set OFF then the full continuity of the displacement field for both nail and continuum will be enforced
5. By default ☒ **Link nail head with** is set ON; this way the top node (by default), will be linked with adjacent structural elements; note that linking of the nail head may be applied only to translational degrees of freedom;

6. Split parameter controls mesh density in nail macro-element; it is recommended to avoid too big differences in mesh densities between continuum and nails; such modeling may lead to axial force oscillations in the nail for high strength parameters of subsoil; using automatic split option usually helps to remedy the problem; two automatic split options are available ie. with the equal size or adaptive size (using exact intersection of the nail with background mesh)
7. To avoid instabilities due to rigid rotation of the nail along its axis the rotation along local nail axis is fixed internally by the preprocessor
8. The shear ultimate stress in the nail is set up at the material level (nail interface) and expressed in terms of the stresses rather then forces
9. To obtain meaningful results nails must be added to the computational model via existence functions; activating nails from the very beggining, without the initial state defined, usually results in computation failure or wrong nail forces
10. Nails spacing in 2D is automaticaly inherited from the beam material used to model nail core
11. The diameter of the injected nail zone is defined at the material level for the nail interface

Window 2-273

**Window 2-274:** `Nail:Create:2 nodes`

New nail macro-element is created by picking the two existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

The selection rule is the only difference between this option and the one given in Window 2-273 so refer to that Window for remaining information.

Window 2-274

**Window 2-275:** `Nail:Create:XYZ for 2 points`

New nail macro-element element is created by entering X,Y,Z coordinates of its two endpoints in the dialog box. Setting of the nail endpoints is the only difference between this option and the one given in Window 2-273 so refer to that Window for remaining information.

Window 2-275

**Window 2-276:** `Nail:Create:On edge(s)`

New nail macro-element(s) are generated on all selected edges (see figure below). Orientation of each nail macro-element element local $x$ axis is coherent with orientation of the element edge. The remaining nail setup is given in Window 2-273

Window 2-276

**Window 2-277:** `Nail:Create:On object(s)`

New nail macro-element(s) are generated on all selected geometrical objects (see figure below). Orientation of each nail macro-element local $x$ axis is coherent with orientation of the geometrical object. The remaining nail setup is given in Window 2-273

Window 2-277

**Window 2-278:** `Nail`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | Select all visible nail macro-elements |
| `With existence function` | Select all visible nail macro-elements with a given existence functions |
| `With nail material` | Select all visible nail macro-elements with a given nail material |
| `Without linked head` | Select all visible nail macro-elements without heads that are properly linked (usually to the structural elements) |

Window 2-278

**Window 2-279:** `Nail:Update:Parameters`

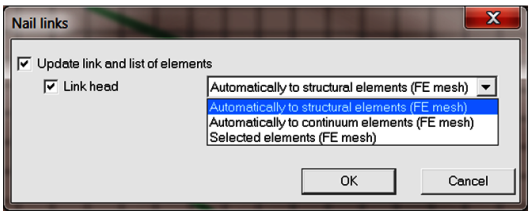
This option allows the user to modify setting for the selected nail or group of nails. In case of group of nails this option is equivalent to the one designed for creating of nails described in Win.( 2-273)

Window 2-279

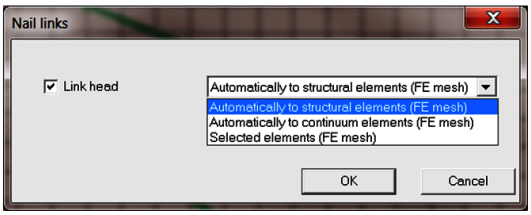
**Window 2-280:** `Nail:Update:Link nail head`

This option allows the user to modify linking of the nail head with other elements of the computational model. The typical situation concerns connection of the nail head with the sheet-pile wall. Three options can be selected ie. linking to structural elements automaticaly, linking to continuum elements automaticaly or linking to selected elements of any class (structural or continuum). This definition will further be verified during data verification stage when the full FE model is generated. Linking marker at the nail head gives only the information that linking decision was already taken. By default when the nail is created its head will be linked to the adjacent structural element. Dialog boxes for this setting for a single picked nail or group of selected nails are shown in the figure below. In case of multi-selection one may update linking of the nail head. Note that in the case of redefining linking definition one additional option is present ie. `Link to previously selected elements`; this happens when the first definition was made with option to `Selected elements`. For more information please refer to Win.(2-273)

Dialog for selection of group of nails




Dialog for selection of one nail




Window 2-280

**Window 2-281:** `Nail:Update:Length`

This option allows the user to update length of selected nail(s).

Window 2-281

**Window 2-282:** `Nail:Update:Shift by head proj.on edges in dir.`

This option allows to shift nail head(s), by preserving nail(s) length and orientation, to the selected set of edges in a given direction.

Window 2-282

**Window 2-283:** `Nail:Update:Shift by head proj.on faces in dir.`

This option allows to shift nail head(s), by preserving nail(s) length and orientation, to the selected set of faces in a given direction.

Window 2-283

**Window 2-284:** `Nail:Update:Shift by head proj.on edges`

This option allows to shift nail head(s), by preserving nail(s) length and orientation, to the selected set of edges by means of a closest distance.

Window 2-284

**Window 2-285:** `Nail:Update:Shift by head proj.on faces`

This option allows to shift nail head(s), by preserving nail(s) length and orientation, to the selected set of faces by means of a closest distance.

Window 2-285

**Window 2-286:** `Nail:Update:Inclination angle`

This option allows the user to modify inclination angle of nail(s) in the vertical plane. During this operation nail head remains unmodified.

Window 2-286

**Window 2-287:** `Nail:Update:Reverse X axis`

This option helps to control orientation of the nail local X axis. This orientation must be such that vector created between nail head and foot of the nail is the same as orientation of the X local axis. Using this option user may flip the orientation. This option is equivalent to the one designed to control orientation of beam elements (see Win.(2-353))

Window 2-287

**Window 2-288:** `Nail:Update:Split`

This option allows the user to modify nail macro-element(s) mesh density. Some recommendations concerning split value are given in Win.(2-273).

Window 2-288

**Window 2-289:** `Nail:Update:Scale`

This option helps to scale size of the nail cross section in the graph pane. Note that the cross section is plot as quadrilateral (instead of cylindrical) to make rendering faster.

Window 2-289

## 2.3.1.2.19 SURFACE LOAD

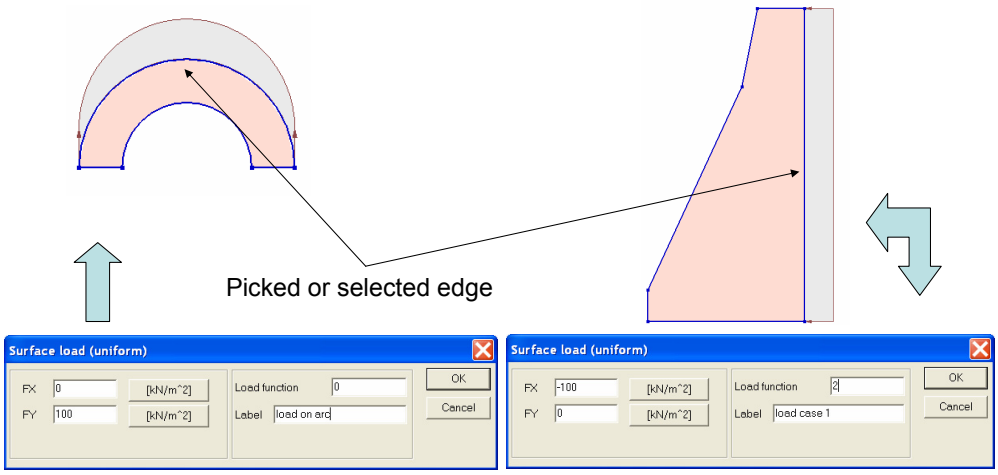
**Window 2-290: Options in menu** `Surface load`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| UNIFORM | On Edge(s)(Uniform) | — | Window 2-291 |
| | On Face(s)(Uniform) | — | Window 2-292 |
| VARIABLE | On Edge (Variable) | — | Window 2-293 |
| | On Face (Variable) | — | Window 2-294 |
| PRESSURE | On Edge (Pressure) | — | Window 2-295 |
| | On selected edges (HP) | — | Window 2-297 |
| | On Face (Pressure) | — | Window 2-296 |
| | On selected faces (HP) | — | Window 2-298 |
| PRESSURE via Fluid Head | On selected edges (SH) | — | Window 2-299 |
| | On selected faces (SH) | — | Window 2-300 |
| Select | All | — | Window 2-301 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With load function | — | Window 2-301 |
| | With label | — | Window 2-301 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-302 |
| | Scale | — | Window 2-302 |

Window 2-290

## Window 2-291: Surface load:UNIFORM:On edge(s) (Uniform)

New uniform surface load is generated on a single picked subdomain edge or group of selected subdomain edges (if current selection list of edges is not empty) (see figure below). This option is valid only in 2D.



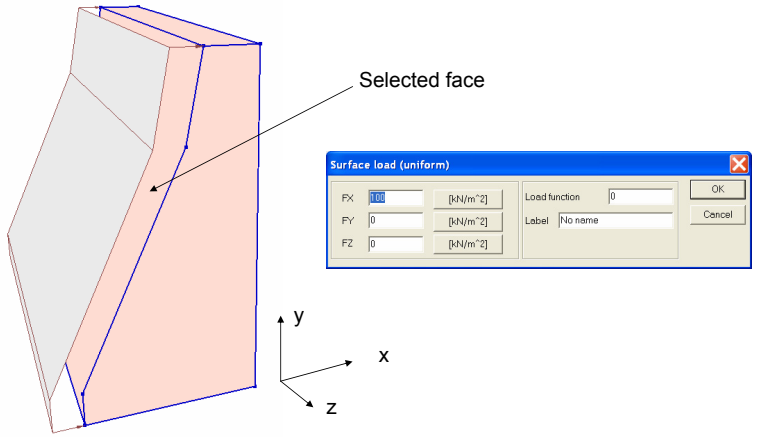

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement
3. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-291

## Window 2-292: Surface load:UNIFORM:On face(s) (Uniform)

New uniform surface load is generated on a single picked subdomain face or group of selected subdomain faces. This option is valid only in 3D.



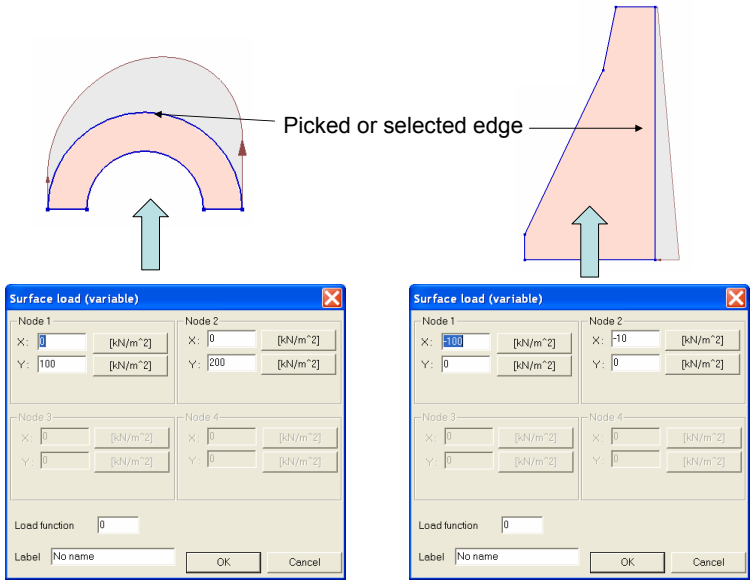

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement
3. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-292

## Window 2-293: Surface load:VARIABLE:On Edge (Variable)

New linearly varying surface load is generated on a picked subdomain edge which can be line segment or an arc. This option is valid only in 2D (see example below).



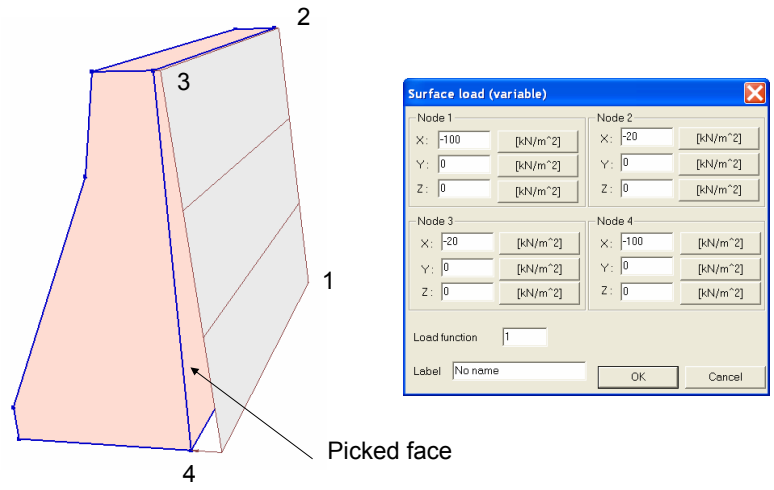

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement
3. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-293

## Window 2-294: Surface load:VARIABLE:On Face (Variable)

New variable surface load is generated on a picked subdomain face. This option is valid only in 3D.



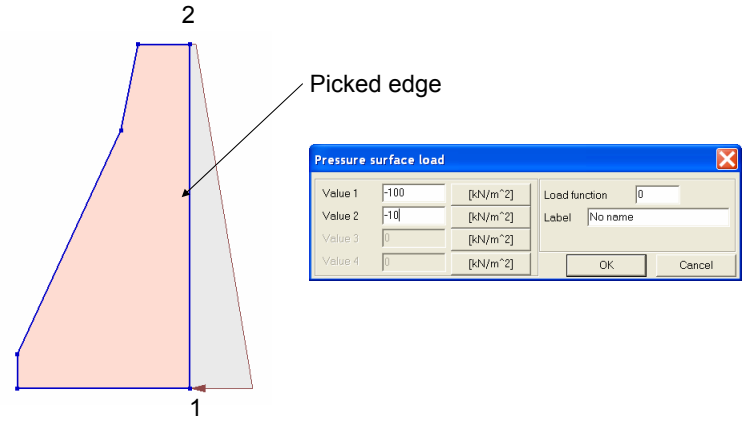

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement
3. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-294

## Window 2-295: Surface load:PRESSURE:On Edge (Pressure)

Using this option one may define a pressure load applied to the picked subdomain edge. The two pressure values have to be set at the edge endpoints, automatically shown in the screen (see example below).



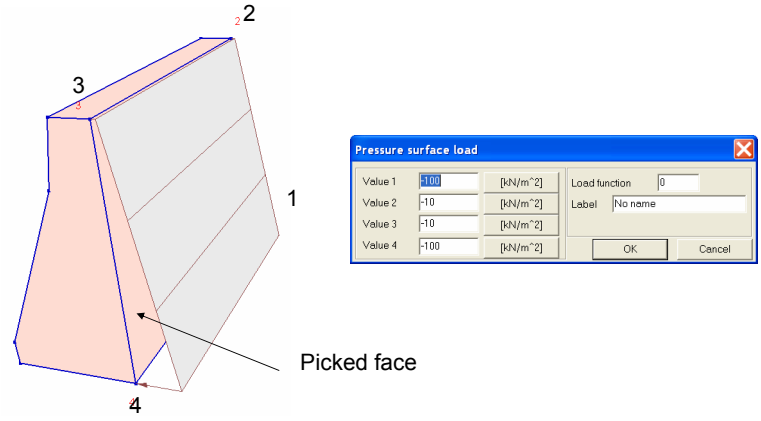

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. Positive pressure is directed towards external normal to the boundary
3. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-295

## Window 2-296: Surface load:PRESSURE:On Face (Pressure)

Using this option one may define a pressure load applied to the picked subdomain face. The three (for T3 face) or four (for Q4 face) pressure values have to be set at the three or four vertices which are automatically shown in the screen (see example below).



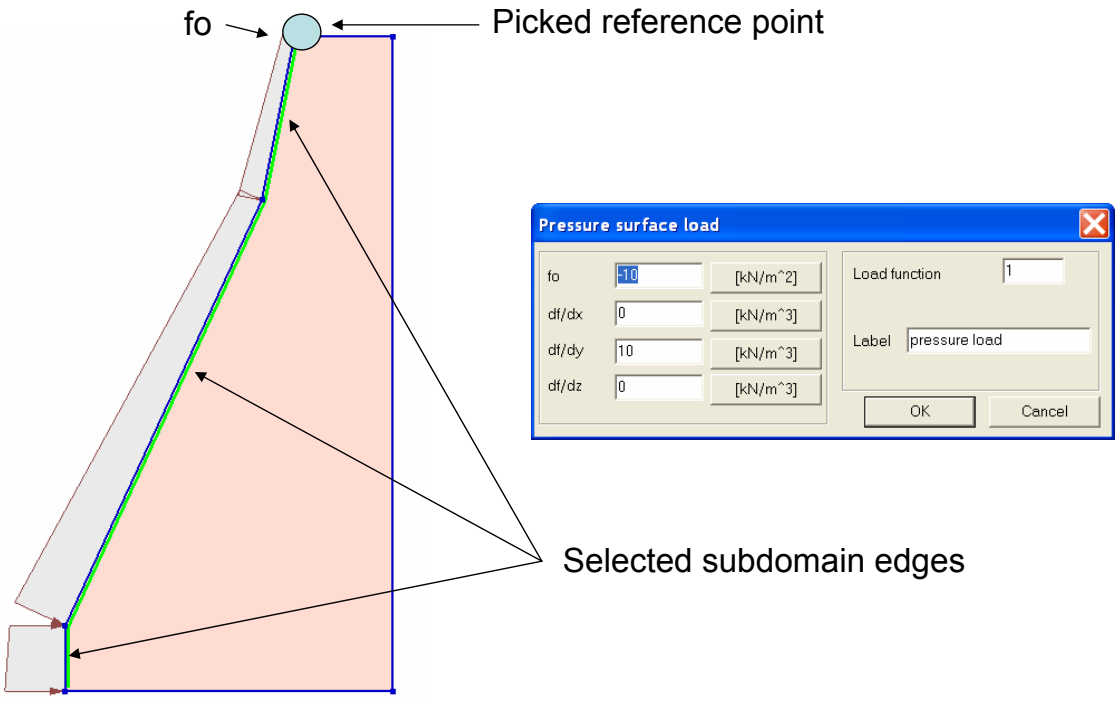

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. Positive pressure is directed towards external normal to the boundary
3. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-296

**Window 2-297:** `Surface load:PRESSURE:On selected edges(HP)`

Using this option one may apply distributed pressure load by picking the point at which value of the pressure $p_o$ is defined, and setting pressure gradients in $x$ and $y$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the pressure at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $p(\mathbf{x}) = f_o + \frac{df}{dx}\cdot(x - x_P) + \frac{df}{dy}\cdot(y - y_P)$. This option is valid only for 2D models.



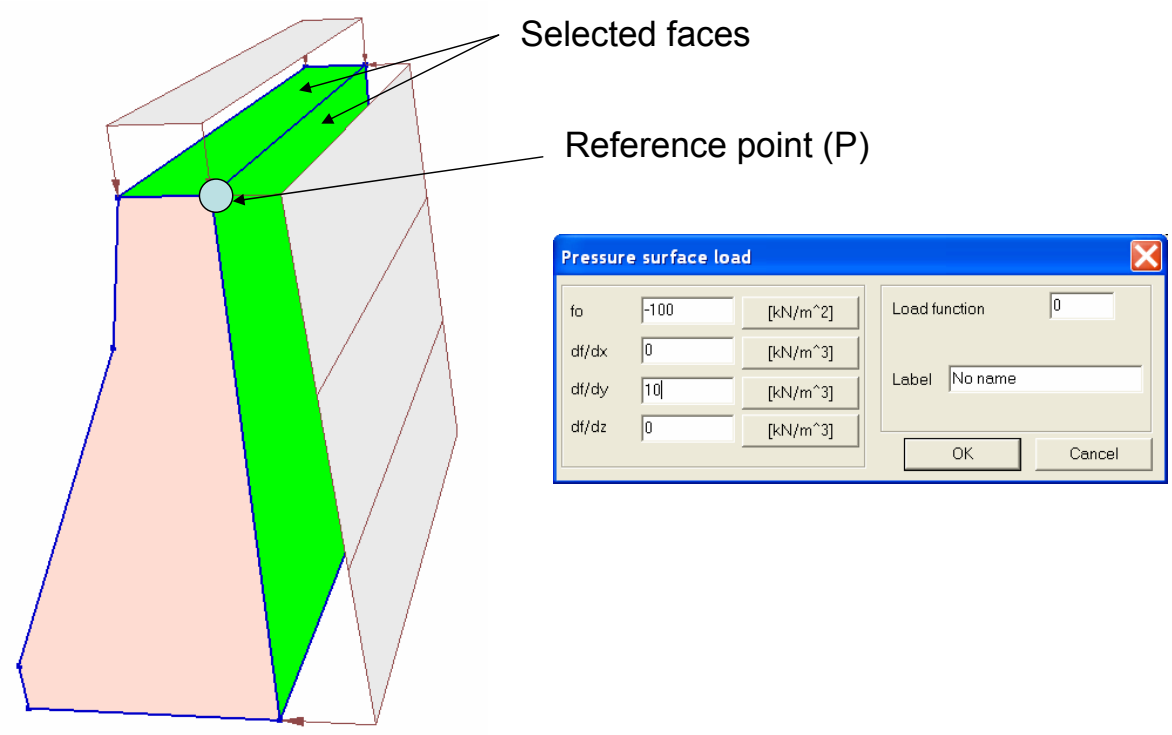

**Remarks**:

1. $f_o$ is a pressure value in the picked reference point $P$
2. For fast selection purposes one may apply a distinct label to this definition
3. Positive pressure is directed towards external normal to the boundary
4. To start the action a reference point must be picked by the mouse
5. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-297

**Window 2-298:** SL: PRESSURE:On selected faces(HP)

Using this option one may apply distributed pressure load by picking the point at which value of the pressure $p_o$ is defined, and setting pressure gradients in $x$, $y$ and $z$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the pressure at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $p(\mathbf{x}) = f_o + \frac{df}{dx} \cdot (x - x_P) + \frac{df}{dy} \cdot (y - y_P) + \frac{df}{dz} \cdot (z - z_P)$. This option is valid only for 3D models.



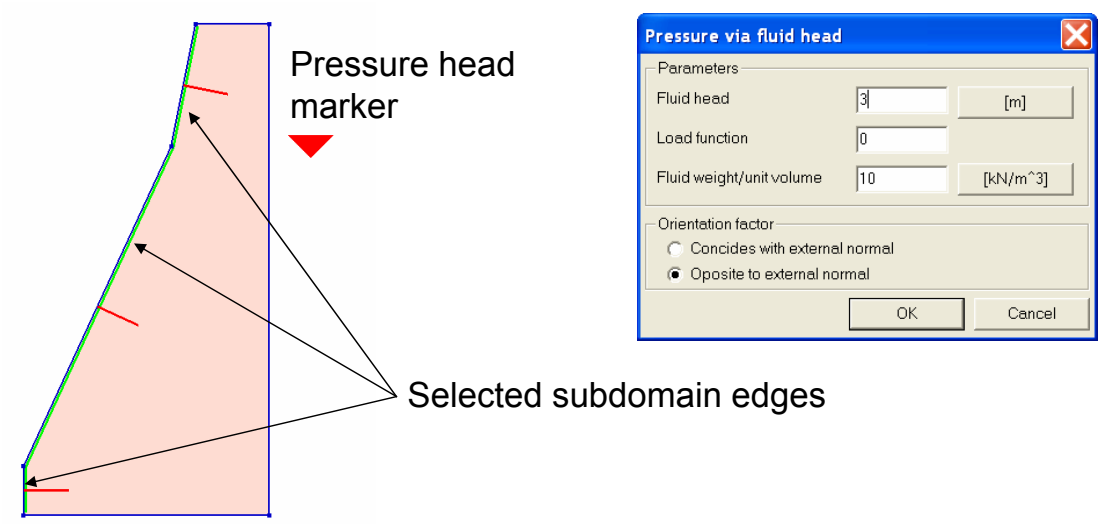

**Remarks**:

1. $f_o$ is a pressure value in the picked reference point $P$
2. For fast selection purposes one may apply a distinct label to this definition
3. Positive pressure is directed towards external normal to the boundary
4. To start the action a reference point must be picked by the mouse
5. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-298

**Window 2-299:** `SL: PRESSURE:On selected edges via Fluid head (SH)`

Using this option one may apply pressure load to the selected edges using definition of the fluid head (potential). Fluid head $h$, is defined as $h = -\frac{p}{\gamma} + y$, where fluid specific weight is denoted by $\gamma$ and vertical coordinate by $y$. Distribution of the pressure applied to the edges (it is automatically computed inside of the calculation module), is derived from the above formula and filtered to cancel the pressure at any point located above elevation $h$. This definition when refining the mesh will automatically be projected on the new mesh and no additional action to be made by the user is needed.



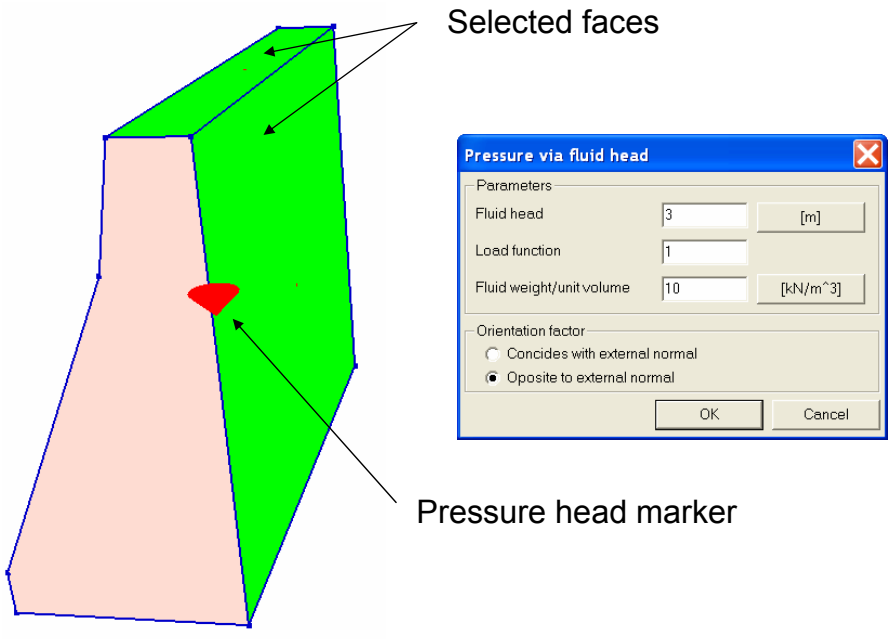

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. It is possible to apply the pressure towards or against the external normal
3. Above the elevation $h$ pressure is set to zero
4. This load can automatically be generated when creating fluid pressure BC for `Deformation+Flow` problems
5. The macromodel should not be deleted once the finite element model is generated (real mesh already exists) otherwise the load will be deleted too

Window 2-299

**Window 2-300:** `Surface load:PRESSURE:On selected faces via Fluid head (SH)`

This option is fully compatible with the one designed for 2D models (`On selected edges via Fluid head (SH)`) except this that the pressure is applied to selected faces (see Window 2-299 for details) instead of edges.




Window 2-300

**Window 2-301:** `Surface load`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | Select all visible surface loads |
| `With load time function` | Select all surface loads with a given load time function |
| `With label` | Select all surface loads with a given label (all registered labels are given in the combo-box) |

Window 2-301

**Window 2-302:** `Surface load`**: some options in menu** `Update`

| Option | Description |
|---|---|
| `Parameters` | This option activates a dialog box to update the applied surface load description. The resulting dialog box, corresponds to the applied load type (see creation methods in Win.(2-290)) |
| `Scale` | Using this option one may enlarge or diminish graphical size of the applied surface loads |

Window 2-302

## 2.3.1.2.20 POINT LOAD

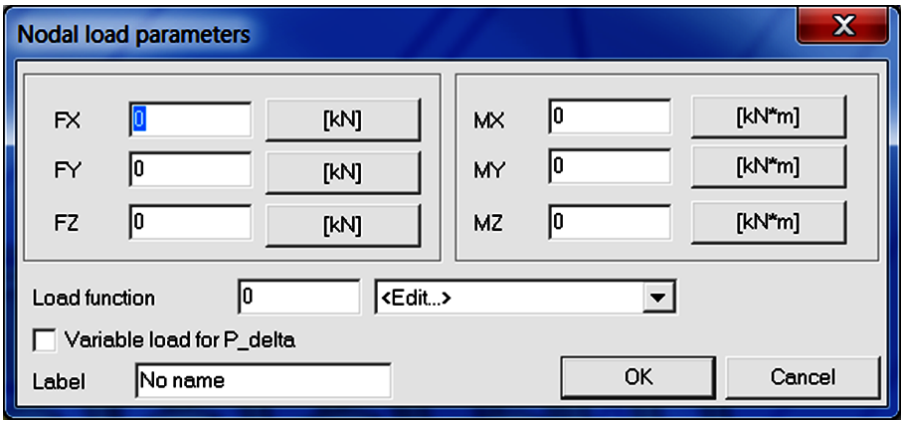
**Window 2-303: Options in menu** `Point load`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On point/pile` | — | Window 2-304 |
| `Select` | `All` | — | Window 2-305 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-306 |
| | `With label` | — | Window 2-306 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-307 |
| | `Scale` | — | Window 2-308 |

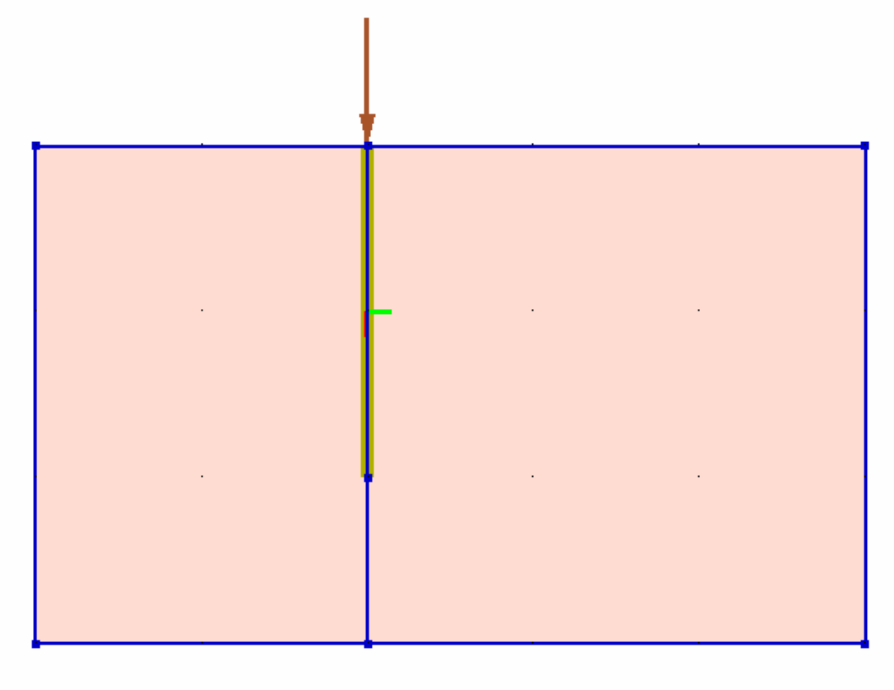
Window 2-303

**Window 2-304:** `Point load:Create:On point/pile`

New concentrated load applied to the translational or rotational degrees of freedom is generated on a picked subdomain vertex(vertices) or pile head(s). The dialog box used for setting these loads is shown in figure below.




In the example shown below, assuming that sheet-pile wall is disconnected from the adjacent continuum by means of the contact interfaces, 3 nodes are created in the real mesh at the geometrical point where the concentrated load is applied. Hence, if we want to apply this load to the sheet-pile-wall we must create it on a beam (sheet-pile wall) subdomain vertex (to do that we can select and then hide the two continuum subdomains first).



**Remarks**:

1. If ☒ **P_Delta** flag is set ON then this load is treated as a variable one in arc-length driver
2. For fast selection purposes one may apply a distinct label to the edited load

Window 2-304

**Window 2-305:** `Point load:Select:All`

This option selects all concentrated loads.

Window 2-305

**Window 2-306:** `Point load:Select:With load f., existence f., label`

Using this option user may select concentrated force with a given load time function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-306

**Window 2-307:** `Point load:Update:Parameters`

This option activates the dialog box to modify the applied concentrated force description, for a single picked point or group of selected points. The detailed description on how to set the data is given in Window 2-304.

**Remark**:When running this option for group of selected points only default parameters will be set in the dialog box.

Window 2-307

**Window 2-308:** `Point load:Update:Scale`

Using this option one may enlarge or diminish size of nodal vectors, representing concentrated forces, or arcs, representing concentrated moments. markers.

Window 2-308

### 2.3.1.3 FE MODEL

The list of all FE model components is listed in the following table.

| | |
|---|---|
| Common methods | Description of common methods to copy, select, delete, update all FE model components (elements, nodes, BC etc..) |
| Node | Methods to create, select, update mesh nodes |
| Beam | Methods to create, select, update beam elements |
| Anchor | Methods to create, select, update anchor (truss) elements |
| Continuum 2D | Methods to create, select, update continuum 2D elements |
| Continuum 3D | Methods to create, select, update 3D continuum elements |
| Shell | Methods to create, select, update shell elements (with double layer of nodes) |
| Shell one layer | Methods to create, select, update shell elements (with single layer of nodes) |
| Membrane | Methods to create, select, update membrane elements |
| Infinite element | Methods to create, select, update infinite elements |
| Tendons | Contains options to generate prestressed tendons embedded in continuum 2D/3D and shells |
| Interface | Methods to create, select, update contact interface elements |
| Interface (large deform.) | Methods to create, select, update large deformation contact |
| Seepage | Methods to create, select, update seepage elements |
| Convection | Methods to create, select, update convection elements |
| Viscous dampers | Methods to create, select, update viscous damper elements |
| Shell hinge | Methods to create, select, update hinges in shell (with single layer of nodes only) elements |
| Boundary conditions | Methods to create, select, update boundary conditions |
| Loads | Methods to create, select, update loads |
| Added masses | Methods to create, select, update added masses needed for dynamic or pushover drivers |
| Distributed flux | Methods to create, select, update distributed fluxes in `Flow`, `Heat` and `Humidity` problems |
| Initial conditions | Methods to create, select, update initial conditions |
| Nodal link | Methods to create, select, update kinematic constraints (node to segment) to be satisfied in an exact way (without penalty) |
| Kinematic constraints | Methods to create, select, update kinematic constraints (node to segment) to be satisfied using penalty method |
| Mesh tying | Methods to create, select, update mesh tying description which allows to connect non-conforming meshes |

### 2.3.1.3.1 COMMON METHODS FOR ALL FE MODEL COMPONENTS

**Window 2-309: Common methods for FE model components**

All options listed in table below are common for all FE model components like nodes, continuum, anchors, beams, membranes etc.. In dedicated menus for each specific FE component user will be referred to windows included in this section.

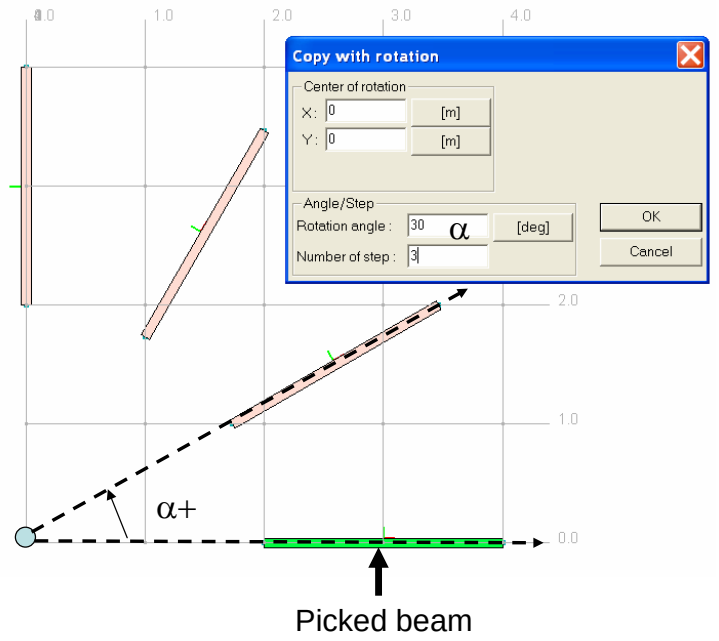
| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| Select | UnSelect one..... |  | Window 2-313 |
| | Between 2 nodes |  | Window 2-314 |
| | Between 4 nodes |  | Window 2-315 |
| | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| | With label | — | Window 2-323 |
| Delete | Delete |  | Window 2-320 |
| Update | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |

Window 2-309

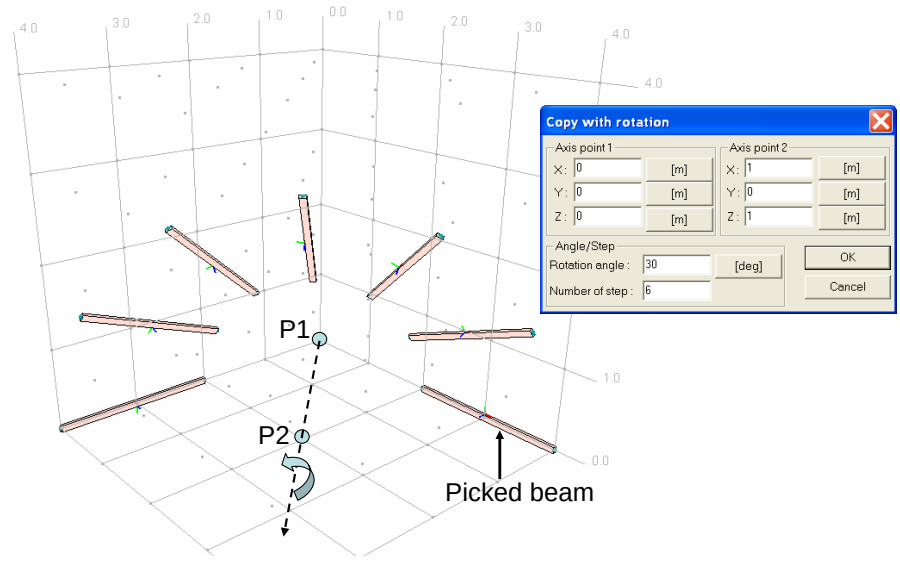
**Window 2-310:** `Create: Copy by rotation`

This option generates new FE model components by copying the picked one (or selected ones) with simultaneous rotation.

(2D case)




(3D case)

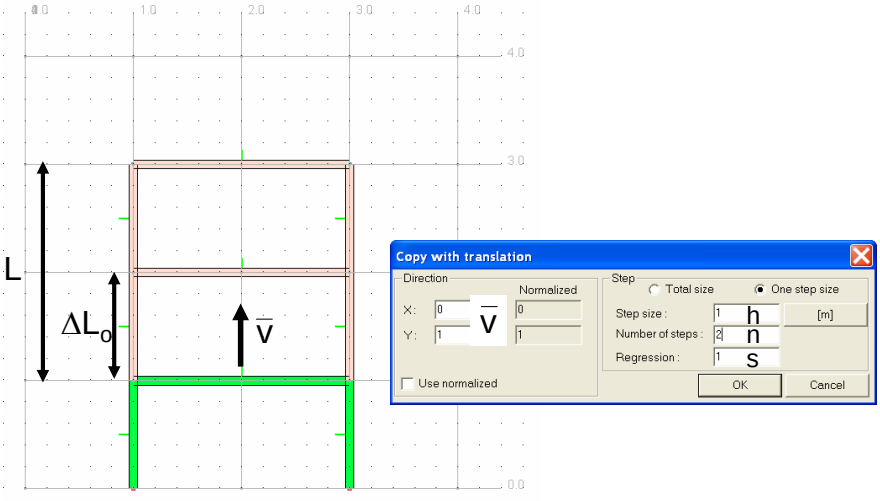



- This option can be applied to a single FE model component (node, beam, anchor, continuum, shell, membrane) (if none of the objects is selected) and to group of selected components (if at least a single component, of any type, is selected)
- In 3D copying operation is carried with rotation along the axis defined by two points `Axis point 1`, `Axis point 2`
- In 2D copying operation is carried with rotation around the point
- Angular distance between two subsequent copies of object(s) is equal to `Rotation angle`
- Objects are copied `Number of steps` times

Window 2-310

**Window 2-311:** `Create: Copy by translation`

Using this option new FE model components are generated by copying, with simultaneous translation, of a picked or selected components (see figure below)




**Remarks**:

- Copying operation is carried out along the vector $\overline{\mathbf{v}}$ defined by three coordinates (`X, Y, Z`)
- $\overline{\mathbf{v}}$ is normalized (or not) depending on the status of `Use normalized` check-box
- if `One step size` is set ON then
  - ⋆ $\Delta L_o = h \ \| \overline{\mathbf{v}} \|$
- if `Total size` is set ON then
  - ⋆ $L = h$
  - ⋆ if $s \neq 1$ then $\Delta L_o = \dfrac{L\,(1-s)}{(1-s^n)}$

    else $\Delta L_o = \dfrac{L}{n}$
- $\Delta L_i = s \ \Delta L_{i-1}$ for $i \geq 1$
- $L = \sum_{i=0}^{n-1} \Delta L_i$

Window 2-311

**Window 2-312:** `Create: Copy by symmetry`

This option can be used to make a copy of a single FE model component (if none of the objects is selected) or group of selected components by symmetrical projection with respect to the plane created under ribbon menu `Draw:Auxiliary Planes`

- To run this option a fictitious plane must be created first
- After this operation orientation of copied components (if applies) is reversed
- To initiate the action plane of symmetry must be pointed first

Window 2-312

**Window 2-313:** `Select: [Un]Select one.....`

By activating this option from menu dedicated for a certain finite element model (like `Node`,`Continuum 2D`, etc..) user may select the component, by picking it in the graphic pane, in the `Replace mode` (selected object becomes unselected and vice versa).

Window 2-313

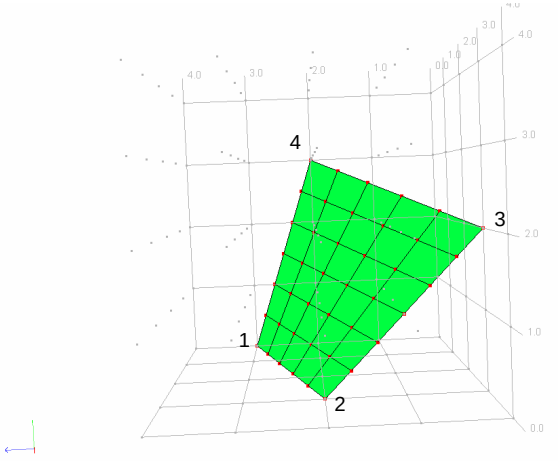
**Window 2-314:** `Select: Between 2 nodes`

This option is used to select (according to the current `Selection mode`) group of FE model components, represented geometrically by points or line segments, belonging to the straight segment defined by the 2 picked points.

Window 2-314

**Window 2-315:** `Select: Between 4 nodes`

This option is used to select (according to the current `Selection mode`) FE components which remain on the surface defined by 4 vertices constituting a macro-quadrilateral element (see figure). Surface is interpolated with aid of standard bilinear shape functions used for Q4 element.



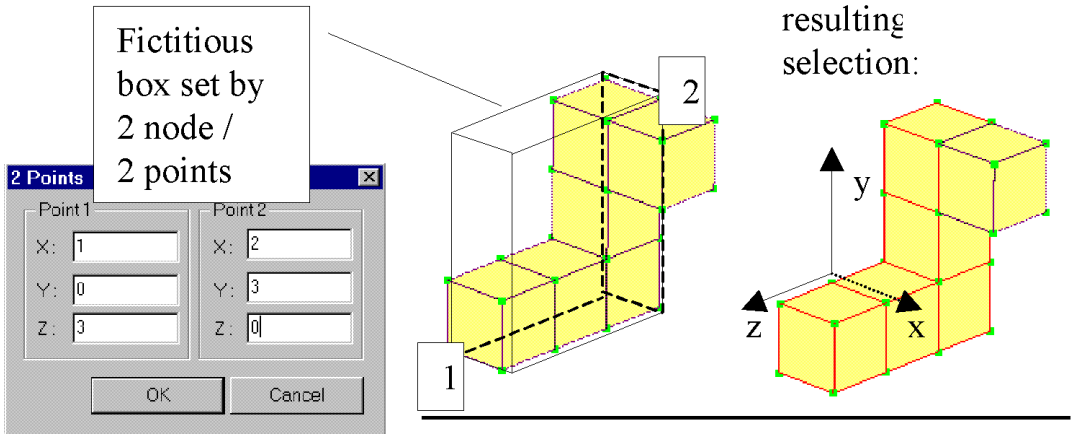

**Remarks**:

1. any FE model component belongs to the surface if distance of its any vertex and its closest projection on the surface remains very small compared to the characteristic size of the macro-quadrilateral
2. after activation of this method 4 nodes must be picked by the graphical cursor
3. it is sufficient to pick a point close to the node but inside of the element
4. this option is supported only for nodes

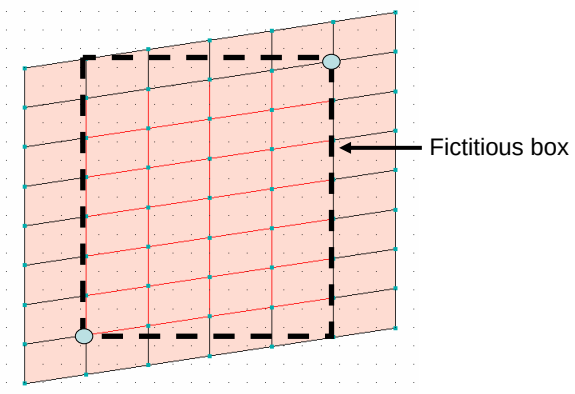
Window 2-315

**Window 2-316:** `Select: Box 2 nodes`

This method is used to select FE model components in the fictitious box aligned with global X,Y,Z directions and defined by the 2 picked points




(3D)

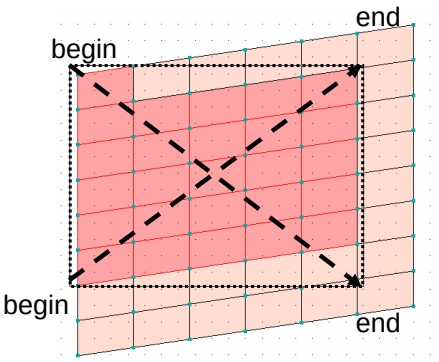



(2D)

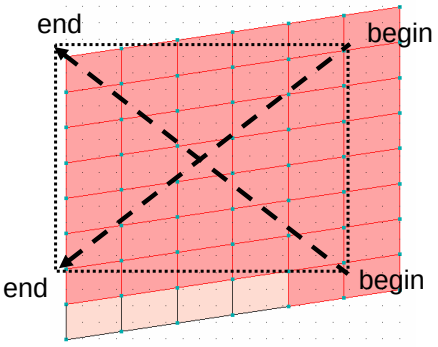
Window 2-316

**Window 2-317:** `Select: In zoom box`

This option is used to select (according to the current `Selection mode`) all FE model components which are covered by the zoom box (infinite rectangular prism in 3D).




Draging box from left → right




Draging box from right→left

**Remarks**:

1. if zoom is dragged from the left to the right side then objects which are fully covered by the box will be selected
2. if zoom is dragged from the right to the left side then all objects which are fully or partially covered by the box will be selected
3. in the 3D it is recommended to switch to orthogonal camera to avoid spurious selections

Window 2-317

**Window 2-318:** `Select: In zoom circle`

This option is used to select (according to the current `Selection mode`) all FE model components which are covered by the zoom circle (infinite cylinder prism in 3D)

**Remarks**:

1. if zoom is dragged from the left to the right side then objects which are fully covered by the box will be selected
2. if zoom is dragged from the right to the left side then all objects which are fully or partially covered by the box will be selected
3. in the 3D it is recommended to switch to orthogonal camera to avoid spurious selections

Window 2-318

**Window 2-319:** `Select: Inside contour`

This option is used to select (according to the current `Selection mode`) all FE model components which are covered by the irregular contour.

**Remarks**:

1. contour must form a closed planar polygon
2. in the 3D it is recommended to switch to orthogonal camera to avoid spurious selections

Window 2-319

**Window 2-320:** `Delete: Delete`

This option is used to:

1. delete single FE model component picked by the mouse if selection list of components of a given class is empty
2. delete group of selected FE model components
3. to pick the component use `Standard selection cursor` (see Window 2-83)

**Remark**:

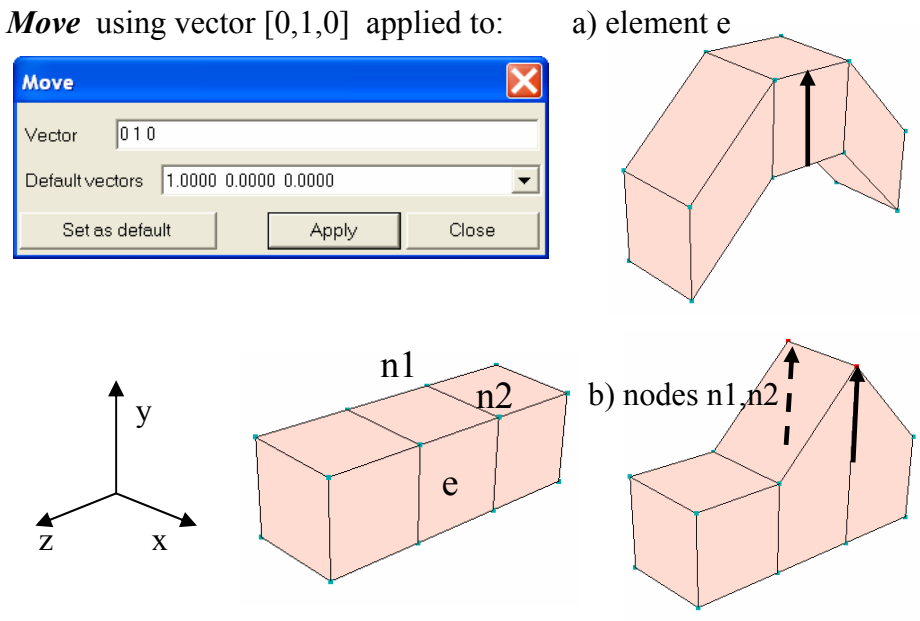
Deletion of nodes will automatically cause deletion of adjacent elements

Window 2-320

**Window 2-321:** `Update: Move`

This option (see figure) is used to carry out translation of:

- single picked FE model component, or group of selected components, by using a toolbar `Move..` (Window 2-86) in ribbon menu `Edit`, or menu `Update/Move` that is dedicated to each FE model component (Node, Anchor etc....)




**Remark**:

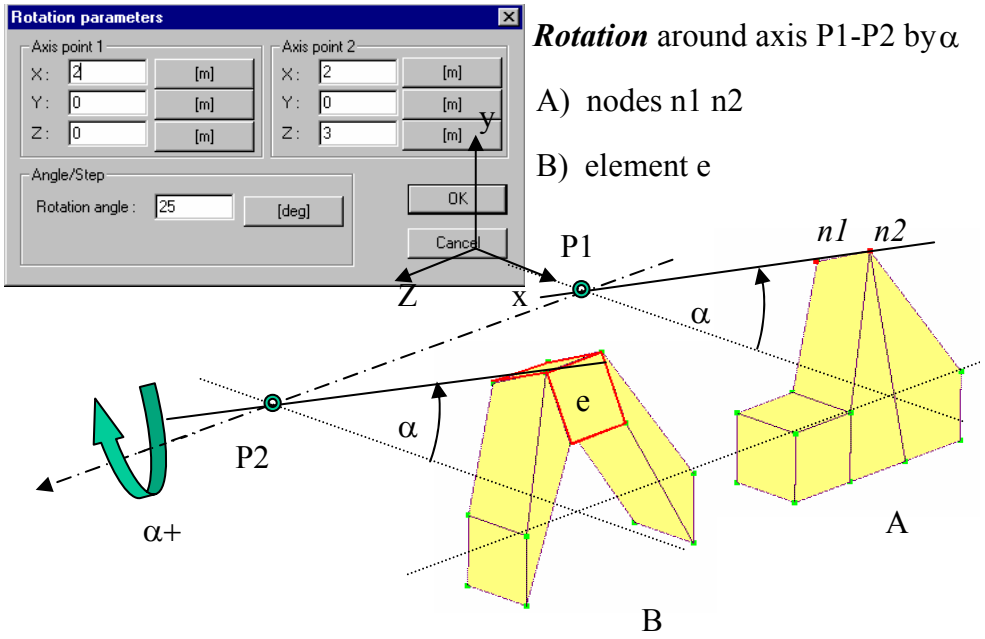
Geometry of components adjacent to the translated one(s) will be automatically updated

Window 2-321

**Window 2-322:** `Update: Rotate`

This option (see figure) is used to carry out rotation of:

- single picked FE model component by using a toolbar (Window 2-86) in ribbon menu `Edit`, or menu `Update/Rotate` that is dedicated to each FE model component (Node, Anchor etc....)




**Remarks**:

1. In the 3D rotation angle should be set as positive in one wants to rotate components with the rotation vector oriented from point `Axis point 1` to `Axis point 2`
2. In the 2D rotation is carried out around a given point
3. Geometry of components adjacent to the rotated one(s) will be automatically updated

Window 2-322

**Window 2-323:** `Select: with label`

Finite element components can be labeled through option `Update/Label`. `Select: with label` option is used to select a single finite element component or group of selected components with the same label (memorized in form of the list).

Window 2-323

### 2.3.1.3.2 NODE

**Window 2-324: Options in menu** `Node`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | Create node | — | Window 2-325 |
| | On Point(s) | — | Window 2-326 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| | Import | — | Window 2-327 |
| Select | [Un]Select one node |  | Window 2-313 |
| | with label | — | Window 2-323 |
| | between 2 nodes |  | Window 2-314 |
| | between 4 nodes |  | Window 2-315 |
| | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| | Nodes Group | | Window 2-329 |
| Delete | Delete |  | Window 2-320 |
| | Delete floating nodes | — | Window 2-330 |
| | Delete Nodes Group | — | Window 2-331 |
| Update | XYZ for one node | — | Window 2-332 |
| | Label | — | Window 2-323 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Homethety | — | Window 2-333 |
| | Project in Plane | — | Window 2-334 |
| | Project in Direction | — | Window 2-335 |
| | Project nodes on selected faces in direction | — | Window 2-337 |
| | Merge multiple nodes | — | Window 2-340 |
| | Export selected nodes to file | — | Window 2-341 |
| | Update coord. from file | — | Window 2-342 |

**Window 2-325:** `Node:Create:Create node`

New node is generated at position picked in the graph pane (if `Snap` is active graphical cursor will be snapped to some of the objects) or through through direct definition of X,Y,Z coordinates in the appearing dialog box.

Window 2-325

**Window 2-326:** `Node:Create:On Point(s)`

New node(s) are generated at positions of selected geometrical points.

Window 2-326

**Window 2-327:** `Node:Create:Import`

This option allows to import nodes from the external text file in which nodes are stored using the following format (all values are separated by spaces):

*node index-1* *X* *Y* *Z*
*node index-2* *X* *Y* *Z*
....
....
*node index-N* *X* *Y* *Z*

NB. *node index* is just an auxiliary counter

Window 2-327

**Window 2-328:** `Node:Create:Nodes Group`

This option allows to memorize selected nodes and label them. It is used mainly to enable a limited output of nodal displacements, velocities or accelerations at few nodal points during dynamic transient analyses (when tuning damping in dynamic models for instance). This option can be activated under `Control/Results content` menu.

Window 2-328

**Window 2-329:** `Node:Select:Nodes Group`

This option allows to select all nodes memorized in `Nodes Group` under given label.

Window 2-329

**Window 2-330:** `Node:Delete floating nodes`

This function removes all nodes which are not attached to any existing finite element, independently on current elements visibility status.

Window 2-330

**Window 2-331:** `Node:Delete Nodes Group`

This option allows to delete selected group of nodes memorized under given label.

Window 2-331

**Window 2-332:** `Node:Update/X Y Z for one node`

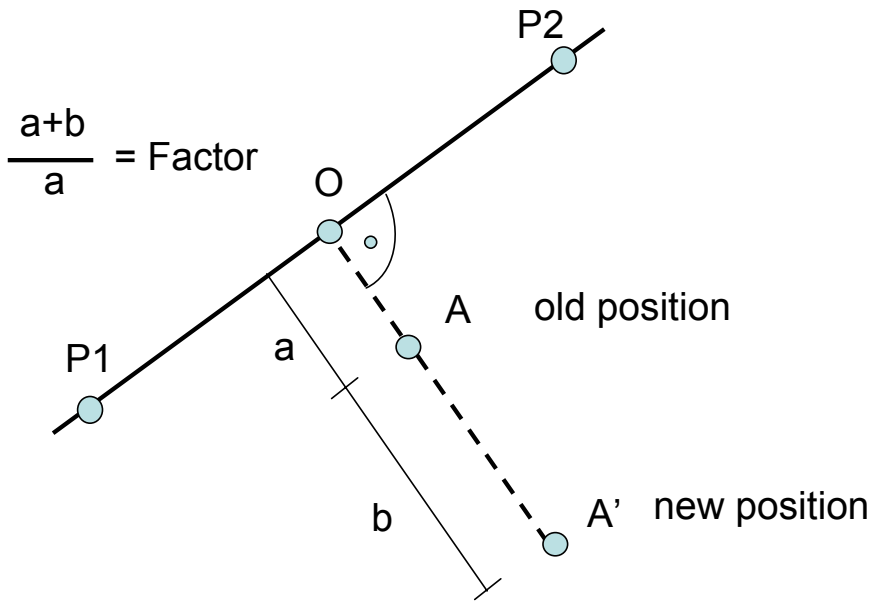
This option allows to modify coordinates of a picked point. It can be done by picking the node to be modified first and then, by picking new node position in the graph pane or by direct definition of new node coordinates in the dialog box as shown below.




Window 2-332

**Window 2-333:** `Node:Update/homothety`

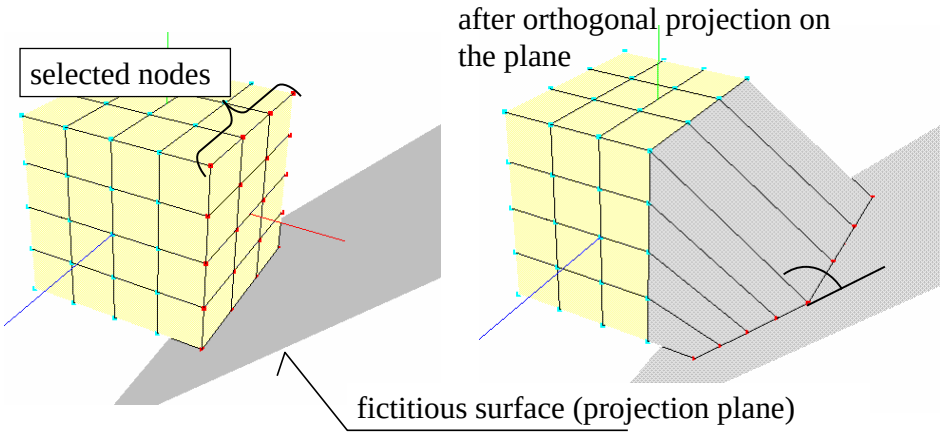
This option allows to perform homothetic transformation of node(s) coordinates with respect to the given axis. The idea of this transformation is shown in figure below.




Window 2-333

**Window 2-334:** `Node:Update/Project on plane`

This option is used for orthogonal projection of a node or group of selected nodes onto picked auxiliary plane.



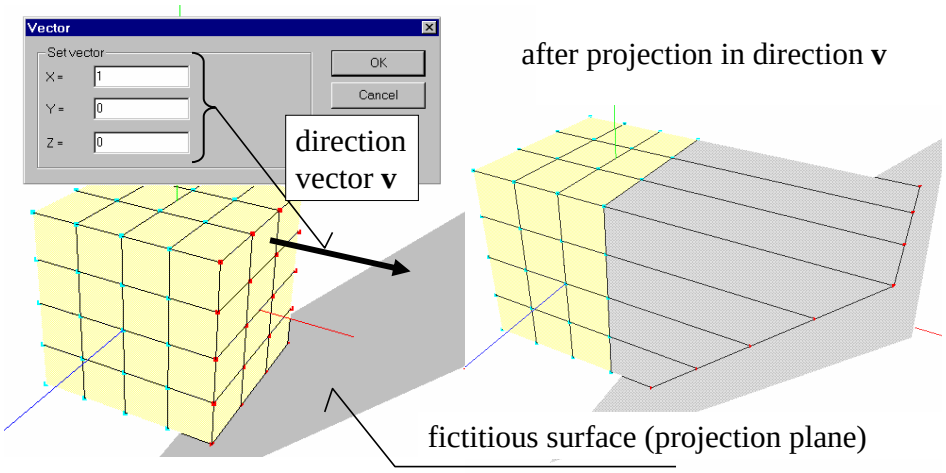

**Remarks**:

1. an auxiliary plane must be created first (ribbon menu `Draw:Auxiliary Planes`)
2. to finalize the action projection plane must be picked with mouse

Window 2-334

**Window 2-335:** `Node:Update/Project on plane in direction`

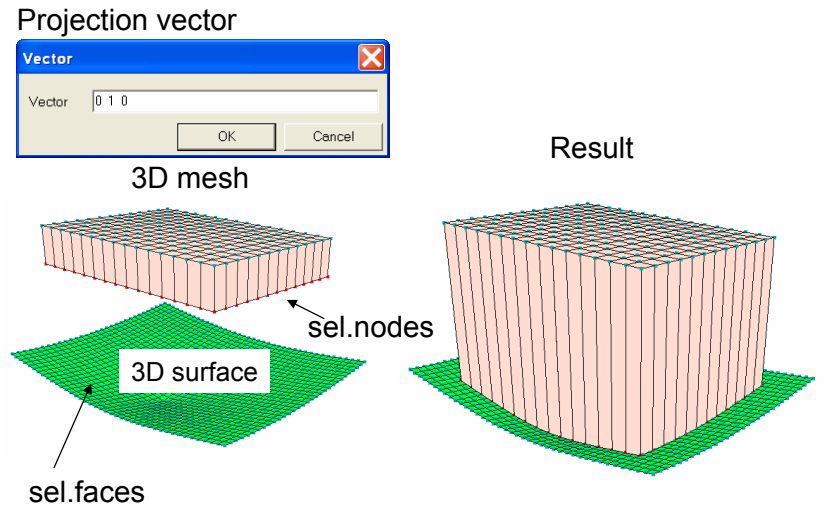
This option is used for projection of node(s) in given direction indicated by vector $\mathbf{v}$ onto picked auxiliary plane.




Window 2-335

**Window 2-336:** `Node:Update/Project nodes on selected faces in direction`

This option is used to project node(s) onto set of selected 3D faces in given direction. It is useful for modeling complex geological layers. In the example given below the 3D surface is represented by a set of quadrilateral elements. Nodes selected at the bottom part of the 3D mesh are projected in the vertical direction (through direction vector) on selected faces of quadrilateral elements (3D surface). After this operation one may split the 3D mesh in the vertical direction to finalize meshing of the 3D volume.



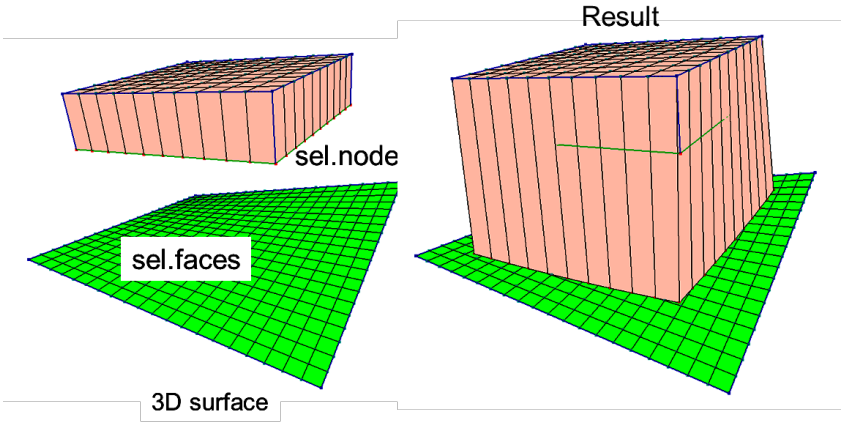

**Remark**:

The auxiliary 3D surface can be generated outside Z_Soil environment and then imported (as list of nodal points and list of elements) through import option under `File`.

Window 2-336

**Window 2-337:** `Node:Update/Project nodes on selected faces`

This option is used to project node(s) onto set of selected 3D faces by means of a closest distance. In the example given below the 3D surface is represented by a set of quadrilateral elements. Nodes selected at the bottom part of the 3D mesh are projected on selected faces of quadrilateral elements (3D surface).




**Remark**:The auxiliary 3D surface can be generated outside Z_Soil environment and then imported (as list of nodal points and list of elements) through import option under `File`.

Window 2-337

**Window 2-338:** `Node:Update/Project nodes on selected subdomain faces in direction`

This option is used to project node(s) onto set of selected subdomain faces in given direction. This option is compatible with the one described in Win.2-336. The only difference is such that selection of faces is replaced by selection of subdomain faces.

Window 2-338

**Window 2-339:** `Node:Update/Project nodes on selected subdomain faces`

This option is used to project node(s) onto set of selected subdomain faces. This option is compatible with the one described in Win.2-337. The only difference is such that selection of faces is replaced by selection of subdomain faces.

Window 2-339

**Window 2-340:** `Node:Update/Merge multiple nodes`

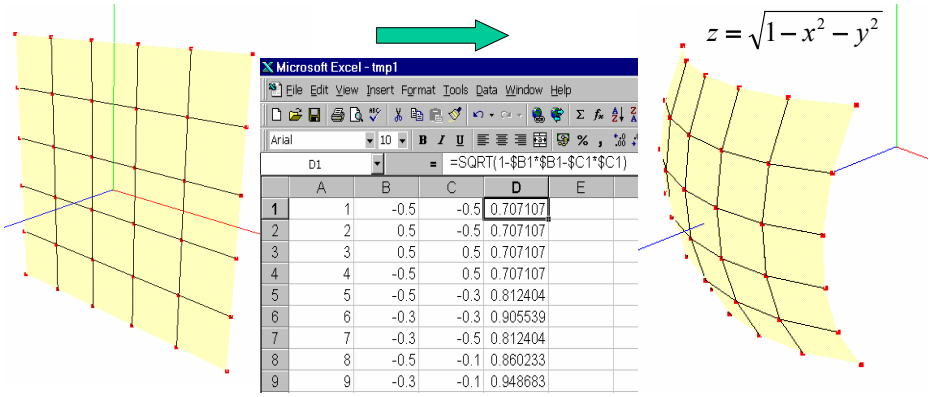
This option is used to detect nodes located approximately at the same position. Such situation may happen due to errors made mainly during data generation of 3D models. By setting the tolerance for the distance user may unify all these nodes.

Window 2-340

**Window 2-341:** `Node:Update/Export selected nodes to file`

The main goal of this option is to export selected nodes coordinates to the external file and then correct them via Excel (or another) application. Nodes are exported using the following format: *Node index* *X* *Y* *Z*
In this case *Node index* cannot be changed.

An example of transformation of a regular plane mesh to the spherical one is shown in figure below.




| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | 1 | -0.5 | -0.5 | 0.707107 | |
| 2 | 2 | 0.5 | -0.5 | 0.707107 | |
| 3 | 3 | 0.5 | 0.5 | 0.707107 | |
| 4 | 4 | -0.5 | 0.5 | 0.707107 | |
| 5 | 5 | -0.5 | -0.3 | 0.812404 | |
| 6 | 6 | -0.3 | -0.3 | 0.905539 | |
| 7 | 7 | -0.3 | -0.5 | 0.812404 | |
| 8 | 8 | -0.5 | -0.1 | 0.860233 | |
| 9 | 9 | -0.3 | -0.1 | 0.948683 | |

Window 2-341

**Window 2-342:** `Node:Update/Update coord. from file`

This option is coupled with option `Node:Update/Export selected nodes to file`. It allows to update coordinates of exported nodes updated externally by Excel or other application. (see Window 2-341)

Window 2-342

### 2.3.1.3.3 BEAM

**Window 2-343: Options in menu** Beam

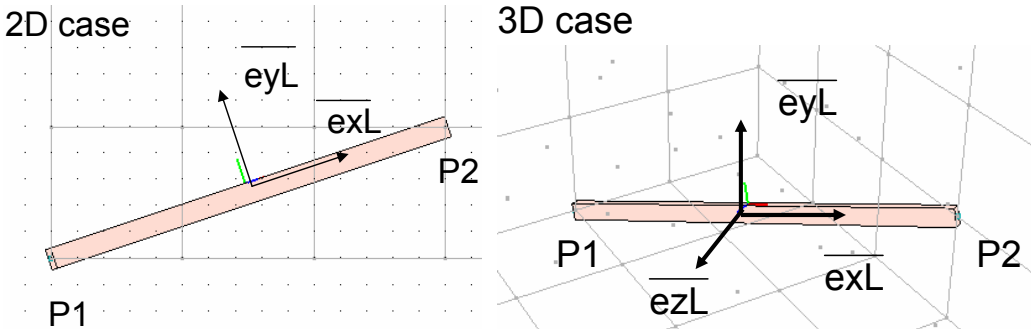
| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 points/nodes | — | Window 2-344 |
| | 2 nodes | — | Window 2-345 |
| | Enter XYZ for 2 nodes | — | Window 2-346 |
| | On edge(s) | — | Window 2-347 |
| | On object(s) | — | Window 2-348 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| | On truss(es) | — | Window 2-349 |
| | On truss(es) remove truss | — | Window 2-350 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-172 |
| | Direction | — | Window 2-351 |
| | Reverse direction | — | Window 2-352 |
| | Reverse X axis | — | Window 2-353 |
| | Centroid | — | Window 2-354 |
| | Hinge | — | Window 2-355 |
| | Reinforcements set | — | Window 2-356 |
| | Tendon | — | Window 2-363 |
| | Split |  | Window 2-368 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Scale | — | Window 2-369 |

Window 2-343

**Window 2-344:** `Beam:Create:2 nodes/points`

New 2-node beam element is created by picking the two points in the graph pane. The two picked points will be first snapped to the existing nodes of the real mesh projected on the graph pane and then to the grid points according to the setup of `Snap options`. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So one wants to create a beam between the two existing nodes it is recommended to use the option `Create:2 nodes` described in Window 2-345.

Position of the local coordinate system is set up by default (see figure below).




**Remarks**:

1. $\overline{e}_{xL}$ base vector is coaxial with vector $\overline{P1P2}$
2. in the 2D case base vector $\overline{e}_{zL}$ is directed along global $z$ axis and base vector $e_{yL}$ is obtained as a vector product $\overline{e}_{yL} = \overline{e}_{zL} \times \overline{e}_{xL}$
3. in the 3D case position of the local $e_{yL}$ axis at both beam endpoints is defined by two direction vectors $\mathbf{d}_1$ and $\mathbf{d}_2$ set at left and right endpoint
4. in the 3D case the default direction vector (common for both beam endpoints) $\overline{d}$ is defined as follows:
   - $\overline{d}_1 = \overline{d}_2 = \overline{d} = \overline{e}_{zG} \times \overline{e}_{xL}$; if $\|\overline{d}\| < \text{TOL}$ then $\overline{d}_1 = \overline{d}_2 = \overline{d} = \overline{e}_{xG} \times \overline{e}_{xL}$
5. the detailed explanation of setting local beam system for the 3D case is given in Window (2-172)
6. To flip orientation of $e_{xL}$ base vector use option `Update:Reverse X-axis`
7. To flip orientation of $e_{yL}$ base vector use option `Update:Reverse direction`

Window 2-344

**Window 2-345:** `Beam:Create:2 nodes`

New 2-node beam element is created by picking the two existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

The selection rule is the only difference between this option and the one given in Window 2-344 so refer to that Window for remaining information.

Window 2-345

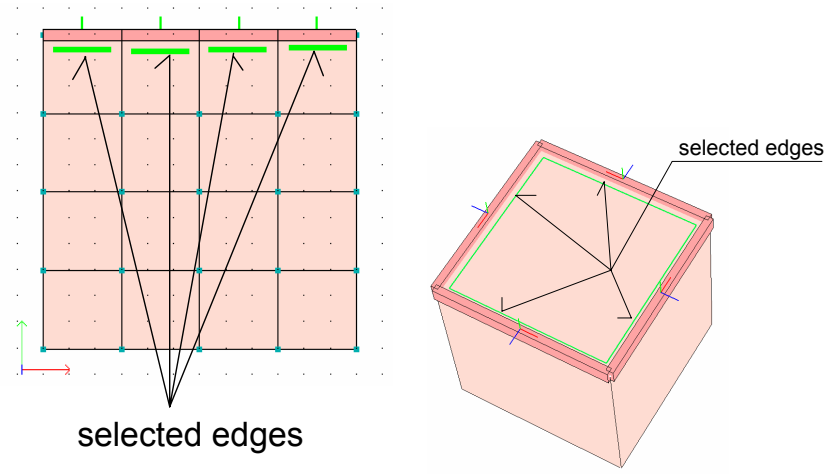
**Window 2-346:** `Beam:Create:XYZ for 2 points`

New beam element is created by entering X,Y,Z coordinates of its two endpoints in the dialog box. Setting default local coordinate system of the beam follows the idea given in Window 2-345

Window 2-346

**Window 2-347:** `Beam:Create:On edge(s)`

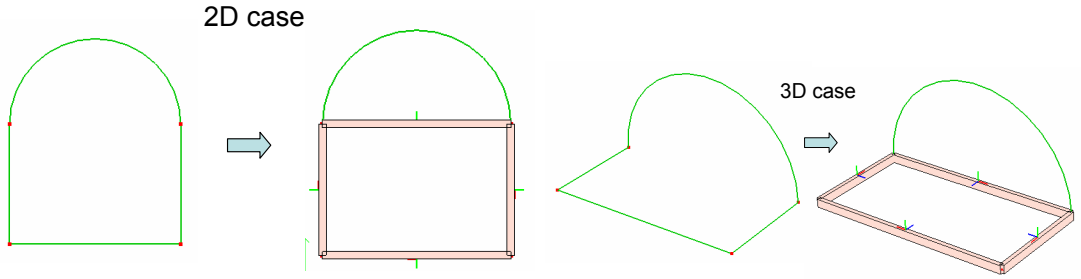
New beam element(s) are generated on all selected edges (see figure below). Orientation of each beam element local $x$ axis is coherent with orientation of the element edge. Setting default local coordinate system of the beam follows the idea given in Window 2-345




Window 2-347

**Window 2-348:** `Beam:Create:On object(s)`

New beam element(s) are generated on all selected geometrical objects (see figure below). Orientation of each beam element local $x$ axis is coherent with orientation of the geometrical object. Setting default local coordinate system of the beam follows the idea given in Window 2-345




**Remark**:

Beams created on arcs are spanned on the arc endpoints hence it is recommended to split the arc first and then to create beams

Window 2-348

**Window 2-349:** `Beam:Create:On truss(es)`

New beam element(s) are generated on all selected truss (anchor) elements without removing them. Orientation of each beam element local $x$ axis is coherent with orientation of the truss element. Setting default local coordinate system of the beam follows the idea given in Window 2-345

Window 2-349

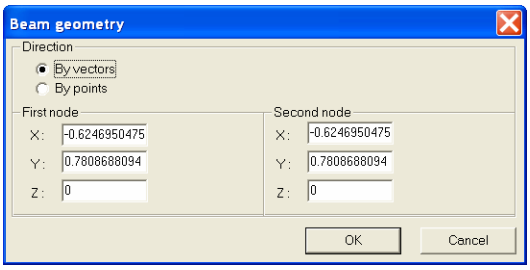
**Window 2-350:** `Beam:Create:On truss(es) (remove truss(es))`

New beam element(s) are generated on all selected truss (anchor) elements. All selected anchor elements are removed once beams generation is finished. Orientation of each beam element local $x$ axis is coherent with orientation of the truss element. Setting default local coordinate system of the beam follows the idea given in Window 2-345

Window 2-350

**Window 2-351:** `Beam:Update:Direction`

This option is used to update exclusively the orientation of beam local coordinate system at both beam endpoints.




**Remarks**:

1. For detailed explanation on how to handle positioning of local beam axes $(x_L, y_L, z_L)$ please refer to Window 2-172
2. Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

Window 2-351

**Window 2-352:** `Beam:Update:Reverse direction`

This option is used to flip the local axis $y_L$ and in consequence local axis $z_L$ while $x_L$ remains unchanged.

**Remarks**:

- The detailed explanation on positioning of local beam axes $(x_L, y_L, z_L)$ is given in Window 2-172
- Top fibers of beam cross section are located on positive side of $y_L$ axis while bottom ones on negative
- Proper setting of local beam axes is very important for nonsymmetric cross sections or cross sections of reinforced concrete
- Proper setting of local beam axes is also very important for visualization of normal and shear force diagrams in the postprocessor
- Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

Window 2-352

**Window 2-353:** `Beam:Update:Reverse X axis`

This option is used to flip the local axis $x_L$ and in consequence local axis $z_L$ while $y_L$ remains unchanged.

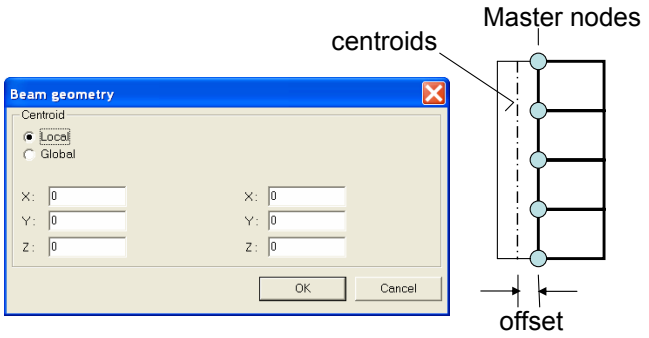
**Remarks**:

- The detailed explanation on positioning of local beam axes $(x_L, y_L, z_L)$ is given in Window 2-172.
- Top fibers of beam cross section are located on positive side of $y_L$ axis while bottom ones on negative
- Proper setting of local beam axes is very important for nonsymmetric cross sections or cross sections of reinforced concrete
- Proper setting of local beam axes is also very important for visualization of normal and shear force diagrams in the postprocessor
- In case when beams form a closed polygon their local $x_L$ axes should be oriented in one direction
- Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

Window 2-353

**Window 2-354:** `Beam:Update:Centroid`

This option is used to update exclusively position of centroids (positions of cross section centers with respect to the master nodes) in the global or local (beam) coordinate system.




**Remark**:

Details on positioning local beam axes $(x_L, y_L, z_L)$ are given Window 2-172.

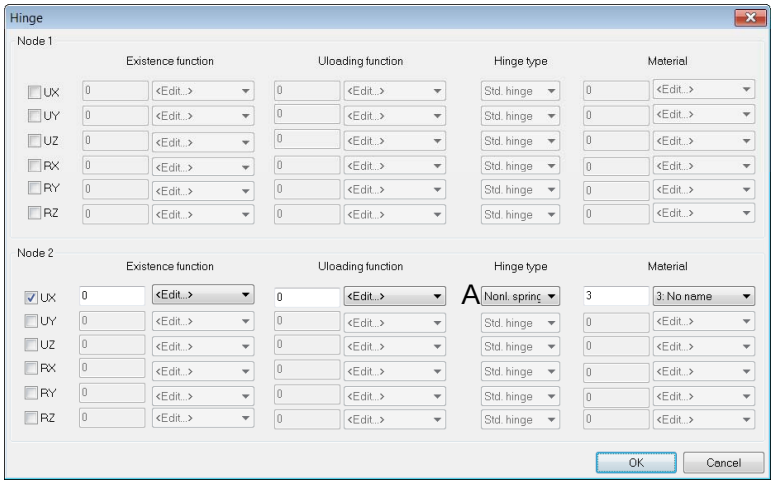
Window 2-354

**Window 2-355:** `Beam:Update:Hinge`

This option allows the user to define at left and/or right beam endpoint

1. an ideal hinge which relaxes some selected degree(s) of freedom
2. nonlinear uncoupled nonlinear spring with any type of the relation between the generalized sectional force (moment) and relative displacement (rotation)
3. nonlinear coupled Janssen nonlinear spring that can be used to model segmental linings

These settings are made in the local basis defined by $x - y_L - z_L$ axes. In addition user may associate an existence function(s) with the hinge/spring at any degree of freedom.




Window 2-355

**Window 2-356:** `Beam:Update:Reinforcements set`

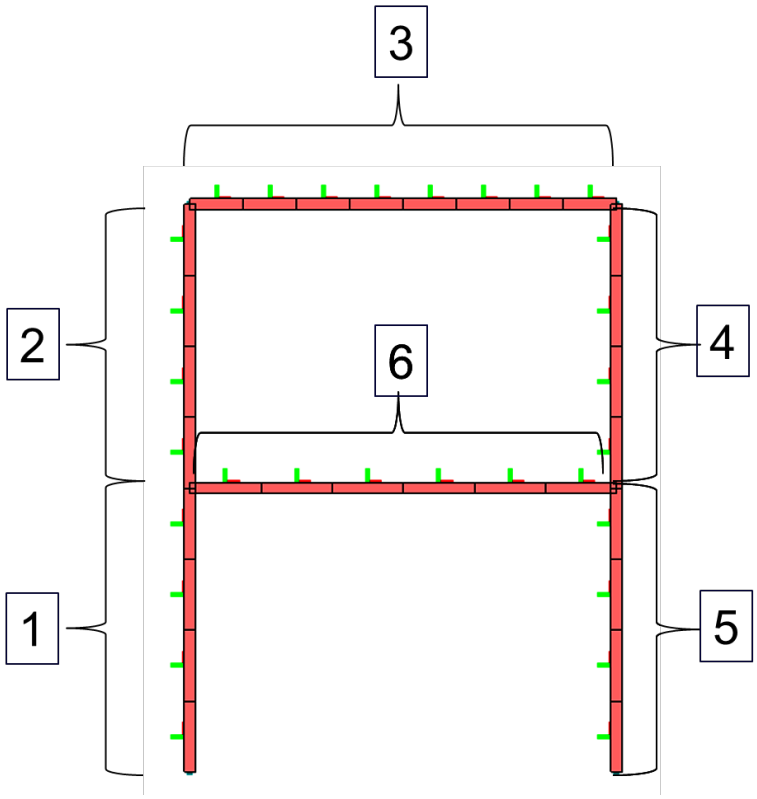
Options enclosed in this submenu allow the user to declare beam reinforcement that is mesh independent. This submenu consists of 5 following options

| Option | Description |
|---|---|
| Define/update on members | Win.2-357 |
| Define/update on beam | Win.2-358 |
| Define split for members | Win.2-359 |
| Remove from member(s) | Win.2-360 |
| Remove from beam(s) | Win.2-361 |
| Verify | Win.2-362 |

Window 2-356

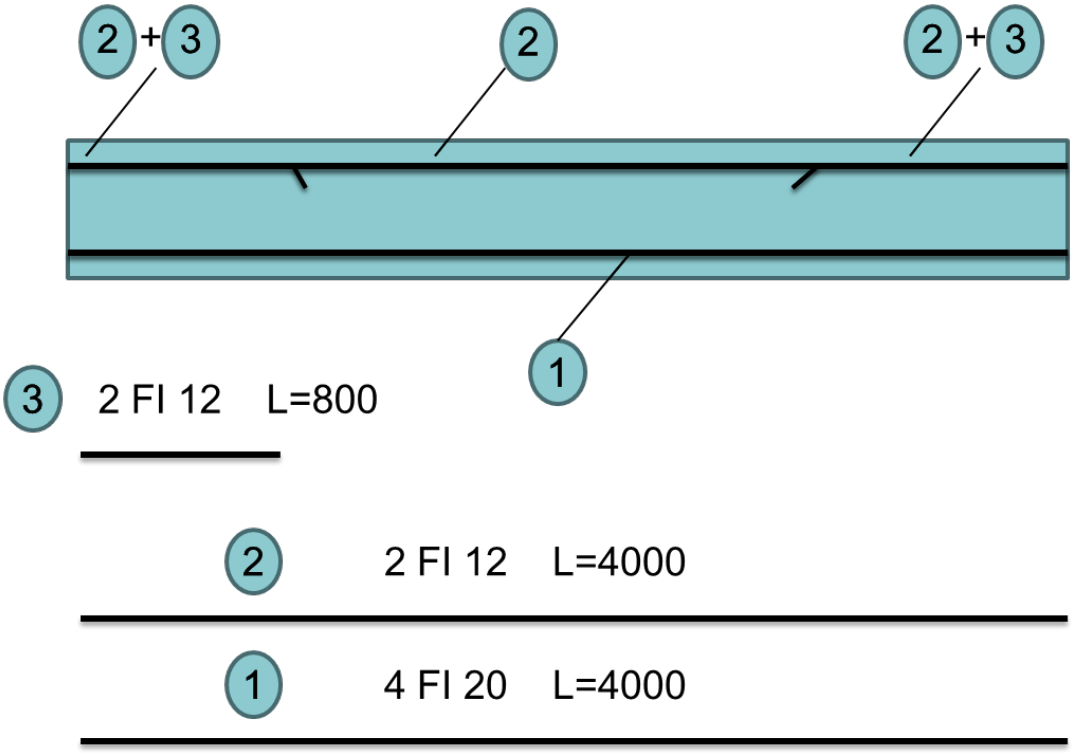
**Window 2-357:** `Beam:Update:Reinforcements set: Define/Update on member`

This definition allows the user to set up reinforcements set on structural beam members. The two situations may happen, in the first one each structural member is discretized with a single beam element (in that case definition is straightforward) while in the second structural member is already split into several beam elements. In the latter case structural members are retrieved from the existing discretization. An example of this procedure is shown in the following figure (here 6 members are detected). Note that any two beam elements may belong to a member once sin of the angle between their cords is below value 0.5 (for instance 3 elements are needed per arc that forms 1/4 of a circle), both have same material label and only these two elements belong to their common nodal point.




Example of structural members recovery

In order to show on how to define reinforcement let us consider a simply supported beam with a rectangular cross section (25cm $\times$ 40 cm) that is reinforced as shown in the figure below. Reinforcing bars (1) and (2) are placed along whole beam length $L$ while bars labeled as (3) have length 0.2 $L$ (relative to the member length $L$).




An example of reinforcement design

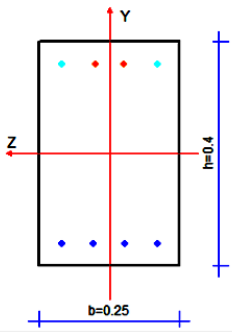
To edit reinforcement in a given member one has to click it first. The table of reinforcement for the above practical example looks as follows

Definition of reinforcements

List of defined reinforcements

| # | Label | Description | Cross section type |
|---|---|---|---|
| 1 | Reinforcement 1 | 4x20 2x16 2x12 2x12 | Plane rectangle |
| 2 | Reinforcement 2 | | Plane rectangle |

OK
Cancel



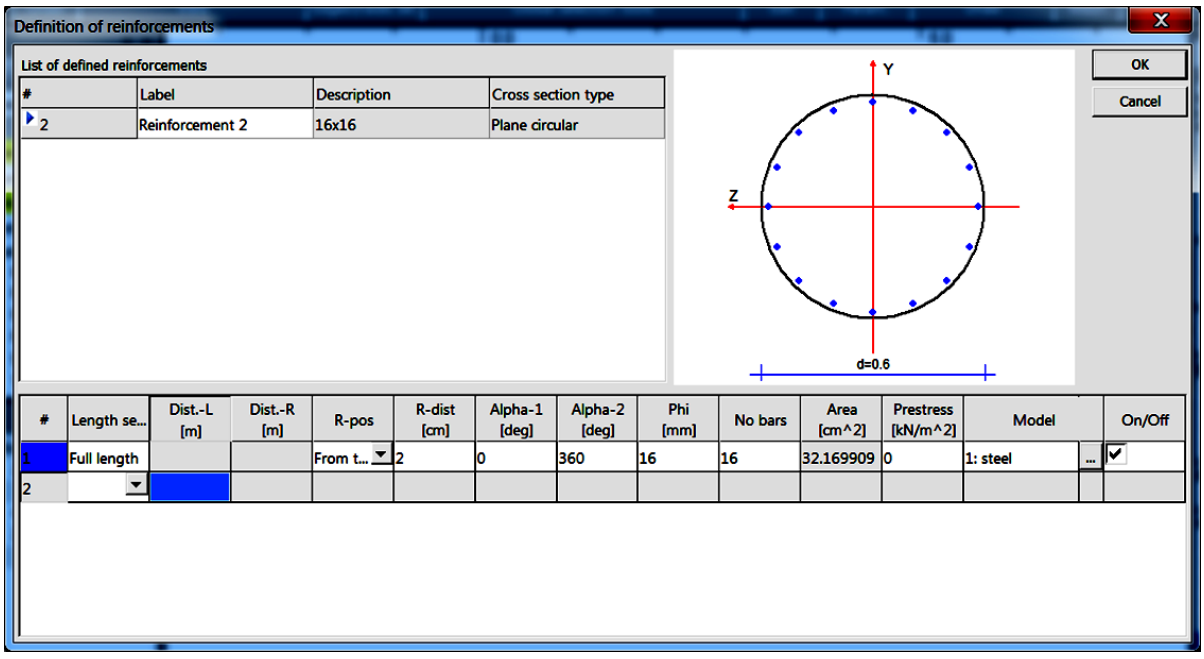

| # | Length se... | Dist.-L [m] | Dist.-R [m] | Y-pos | Y-dist [cm] | Z-offs-L [cm] | Z-offs-R [cm] | Phi [mm] | No bars | Area [cm^2] | Prestress [kN/m^2] | Model | On/Off |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Full length | | | From bot. | 4 | 4 | 4 | 20 | 4 | 12.566371 | 0 | 1: steel | ✓ |
| 2 | Full length | | | From t... | 4 | 4 | 4 | 16 | 2 | 4.021239 | 0 | 1: steel | ✓ |
| 3 | Left side (r... | 0.2 | | From top | 4 | 10 | 10 | 12 | 2 | 2.261947 | 0 | 1: steel | ✓ |
| 4 | Right s... | | 0.2 | From t... | 4 | 10 | 10 | 12 | 2 | 2.261947 | 0 | 1: steel | ✓ |
| 5 | | | | | | | | | | | | | |

Corresponding reinforcement table (setup for cross sections different from circular one)

In this table user may define new set of reinforcing bars or reuse an existing one. Reinforcing bars can be placed along the member using five options (regardless cross section type)

1. `Full length` (columns `Dist-L` and `Dist-R` are not meaningful)
2. `Left side (total)` - reinforcing bar will have total length equal to `Dist-L` and it will be aligned at the left member endpoint
3. `Left side (relative)` - length of the reinforcing bar(s) is defined in column `Dist-L` as percentage of the member length; reinforcing bars are aligned at the left member endpoint
4. Right side (total) - reinforcing bar will have total length equal to `Dist-R` and it will be aligned at the right member endpoint
5. `Right side (relative)` - length of the reinforcing bar(s) is defined in column `Dist-R` as percentage of the member length; reinforcing bars are aligned at the right member endpoint
6. `Partial (total)` - reinforcing bar(s) will be aligned at the middle of the member with distance `Dist-L` and `Dist-R` from the left and right endpoints, respectively

For each reinforcing bar(s) user must specify number of bars (column `No bars`), their diameter (column `Phi`), prestress value (0.0 for inactive) (column `Prestress`) and select/parametrize fiber model. Three possible fiber models can be used: `Elastic`, `Elastic-perfectly plastic` and `User defined` Positioning of reinforcing bars in the cross section depends on its type set in the global material definition for beam. For all shapes, except plane circular and circular tube ones, reinforcing bars offsets are defined in Y and Z directions (see figure above). For circular shapes the positioning of reinforcing bars takes place in the cylindrical coordinate system (anticlockwise direction is positive) while offsets are defined along radial direction (see figure below)




Corresponding reinforcement table (setup for circular cross section)

**Remarks**:

1. Reinforcement defined on member can be modified at the material level using option ☒ **Additional layers** in group `Cross section`(both reinforcements will be considered)

2. Reinforcement defined on a single beam element that belongs to the member replaces the one set up on member
3. Use option `Verify` in this submenu to check if reinforcement matches geometry of the cross section
4. Defined reinforcement can be applied to another member/beam element
5. This definition will not be accepted once local beam bases are not properly ordered. In that case user must use `Reverse X-axis` and/or `Reverse direction` options to adjust local bases.

Window 2-357

**Window 2-358:** `Beam:Update:Reinforcements set: Define/Update on beam(s)`

This option is fully compatible with the one designed for a beam member (see Win.(2-357)). It applies to a selected set of beam elements or to a single picked element.

**Remarks**:

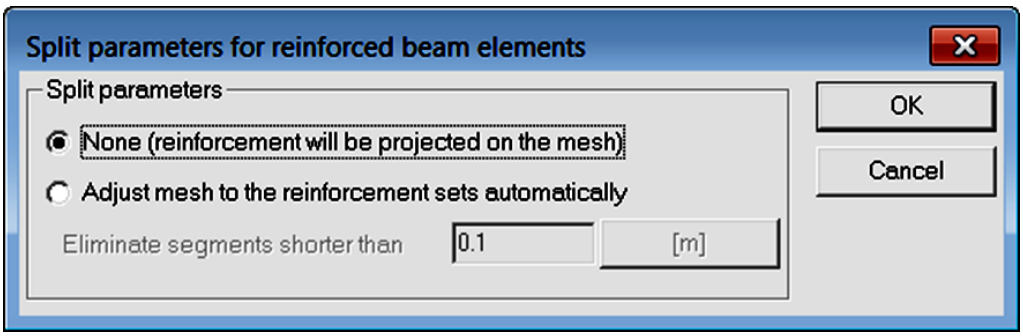
1. Reinforcement defined on a beam replaces the one set up on a member

Window 2-358

**Window 2-359:** `Beam:Update:Reinforcements set: Define split for members`

This option allows to adjust mesh to the definition of reinforcement. The two possibilities are offered to the user

1. Reinforcement is projected on the existing mesh that is frozen
2. Mesh is adjusted to the assumed reinforcement (elements with length shorter than assumed value will not be generated). This option splits these beam elements in which reinforcement is nonuniform. Points where the split will be enforced are shown by line segments perpendicular to the member axis.




**Remarks**:

1. If ◎ **Flexibility based formulation** is activated in group `Main` at the beam material level single beam element per member is sufficient (assuming that the reinforcement is uniform within this beam).
2. For ◎ **Displacement based formulation** activated in group `Main` (at the beam material level) an extra mesh refinement is required to obtain certain level of accuracy

Window 2-359

**Window 2-360:** `Beam:Update:Reinforcements set: Remove from member(s)`

This option allows the user to delete assumed reinforcements set from selected member(s) (it is enough to select one element from a member to apply this action to the whole member). Deleted reinforcements set will remain in the global reinforcements list until it is deleted there.

Window 2-360

**Window 2-361:** `Beam:Update:Reinforcements set: Remove from member(s)`

This option allows the user to delete assumed reinforcements set from selected beam element(s).

**Remarks**:

1. The reinforcements set defined on a member (to which given beam element belongs) becomes active

2. Deleted reinforcements set will remain in the global reinforcements list until it is deleted there.

Window 2-361

**Window 2-362:** `Beam:Update:Reinforcements set: Verify`

This option allows the user to check if declared reinforcement (its length and position) is consistent with the beam element/member geometry and its cross section. This action is automatically invoked during data verification.

Window 2-362

**Window 2-363:** `Beam:Update:Tendon`

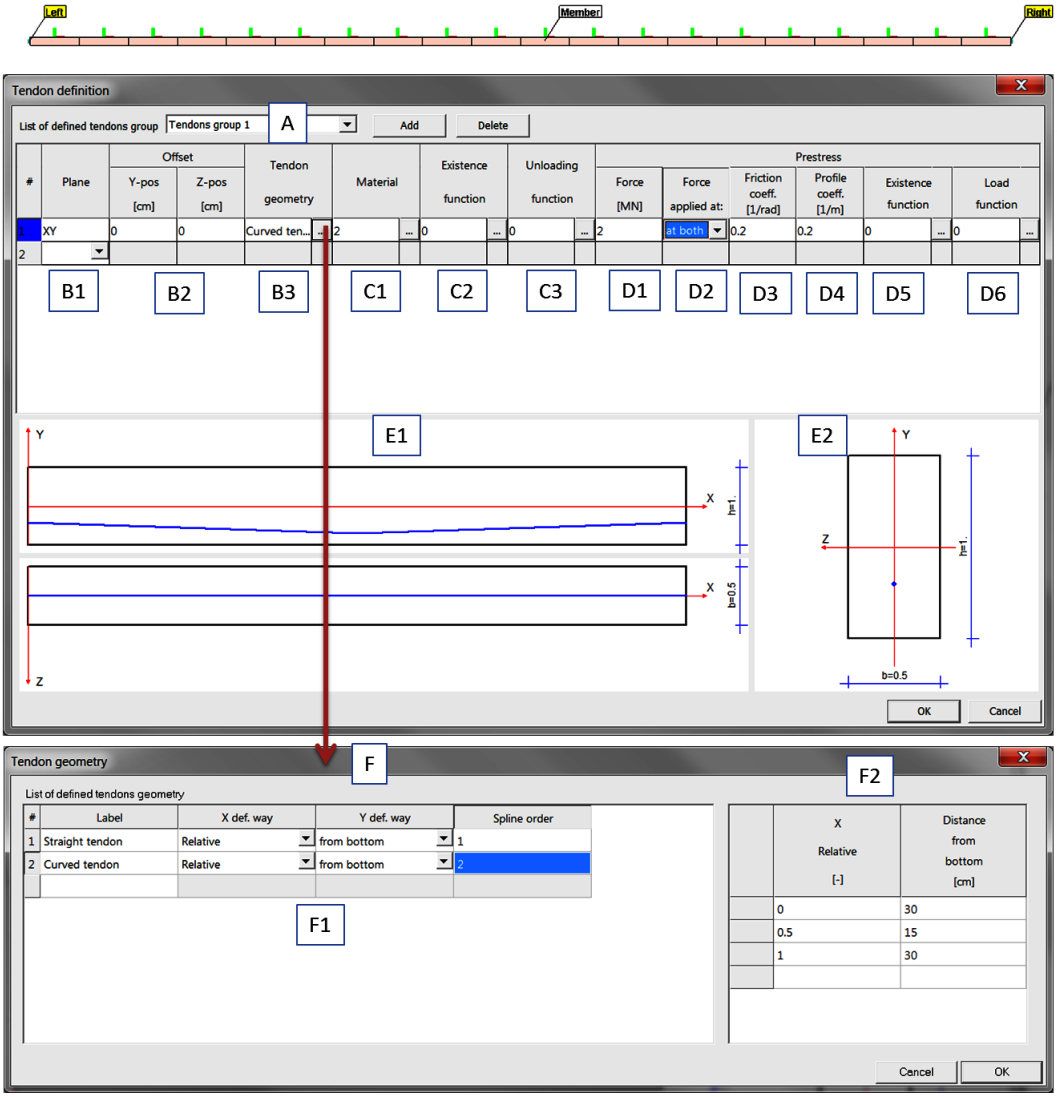
Options enclosed in this submenu allow the user to add prestressed tendons to the selected set of beam elements. This submenu consists of 4 following options

| Option | Description |
|---|---|
| Define on members | Win.2-364 |
| Define on sel. beams | Win.2-365 |
| Update | Win.2-366 |
| Remove | Win.2-367 |

Window 2-363

**Window 2-364:** `Beam:Update:Tendon:Define on members`

This option allows to generate prestressed tendons coupled with beam elements forming chain of elements (called as a member). Precise definition of the member is given in Win.2-357. To add a new tendons group click button **Add**. After adding new tendons group one can edit geometry of each (in group) tendon ie. plane (B1), offsets in Y, Z local beam directions (B2), trajectory (B3), tendon material (as for truss elements) (C1), existence function (C2), unloading function (C3) and prestress application data. The prestress data consists of the force value (D1), application way (D2) (at left endpoint, at right endpoint or at both endpoints), friction (D3) and profile (D4) coefficients, prestress existence function (D5) and loading function (D6). Existence function (D5) applied to the prestressing enforces full force control in the assumed period of activity. After that period full compatibility between beam nodes and tendon nodes is assumed. The load function (D6) allows modification of the prestress force in time in the active time period declared in the existence function. Location of the tendon is shown in 3 projections in local beam coordinate axes (E1, E2).




**Remarks**:

1. One member may have only one tendons group (in one group any number of tendons can be set)

2. Tendons can be straight or curved (via spline); to modify list of fit points use button ... in (B3); position X of the tendon is set as relative with respect to the member length while Y and Z as absolute (in cm)

3. Dialog box (F) includes list of all defined tendons

4. Tendon is split into set of truss elements connected with the beam axis via rigid artificial beams; this operation is hidden to the user

In more complex cases when tendons are added progressively to the member one may add them precisely by selecting beams. Completing remaining part of the data is the same as for the option on member (see Win.2-364).

Window 2-365

**Window 2-366:** `Beam:Update:Tendon:Update`

To update defined tendons click on the group label in the screen. See Win.2-364 for details.

Window 2-366

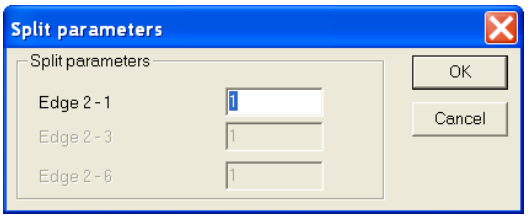
**Window 2-367:** `Beam:Update:Tendon:Remove`

This option allows to delete given tendons group by clicking the label in the screen.

Window 2-367

**Window 2-368:** `Beam:Update:Split`

By using this option user may split picked beam element or group of selected beam elements setting split value in the dialog box as shown below. It is worth to mention that in pure structural applications, once the beam cross section is uniform along the length, single beam element is enough to represent a structural member. To use this type of modeling activate ◎ **Flexibility based formulation** option under parameters group `Main`at the material level. ◎ **Flexibility based formulation** yields highly accurate results for both linear and nonlinear cases. For soil-structure interaction problems structural members have to be discretized to preserve mesh conformity between structure, interface and adjacent continuum. Therefore standard ◎ **Displacement based** formulation is sufficient. This split option does not take into account definition of reinforcements.



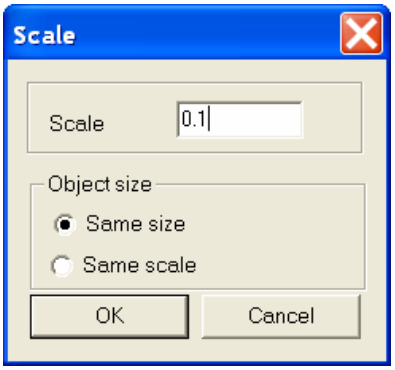

**Remark**:

Every new beam element inherits all the parameters from the parent element

Window 2-368

**Window 2-369:** `Beam:Update:Scale`

Using this option user may scale the size of the beam cross section and size of the local beam axes.




**Remark**:

◎ **Same size** and ◎ **Same scale** switches have no meaning here

Window 2-369

### 2.3.1.3.4 TRUSS/ANCHOR

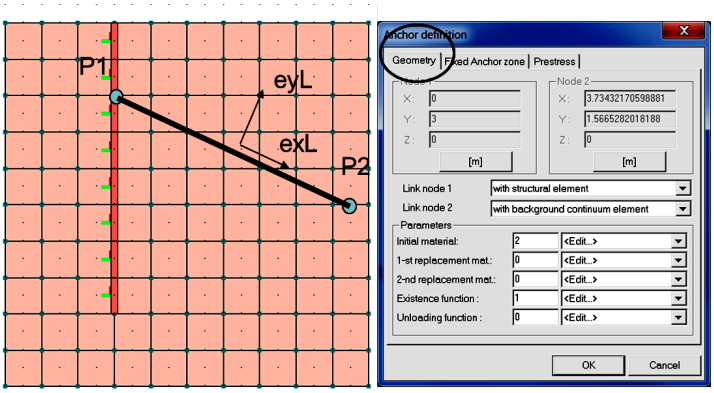
**Window 2-370: Options in menu** `Anchor`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes/points | — | Window 2-374 |
| | 2 nodes | — | Window 2-375 |
| | Dialog | — | Window 2-371 |
| | On edge(s) | — | Window 2-376 |
| | On object(s) | — | Window 2-377 |
| | On beam(s) | — | Window 2-378 |
| | On beam(s) (remove beam(s)) | — | Window 2-379 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| | Point+Vector+Length | — | Window 2-381 |
| | Import (*.2n, *.nvd) | — | Window 2-381 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| | Sel. elements linked to sel. anchors | — | Window 2-382 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-383 |
| | Reverse direction | — | Window 2-387 |
| | Reverse X axis | — | Window 2-388 |
| | Prestress | — | Window 2-373 |
| | Link to elements | — | Window 2-384 |
| | Length | — | Window 2-389 |
| | Inclination angle | — | Window 2-390 |
| | Split |  | Window 2-391 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Scale | — | Window 2-392 |

Window 2-370

**Window 2-371:** `Anchor:Create:Dialog` **option** `2 points`

New anchor element is created by setting coordinates X, Y, Z of the two endpoints.



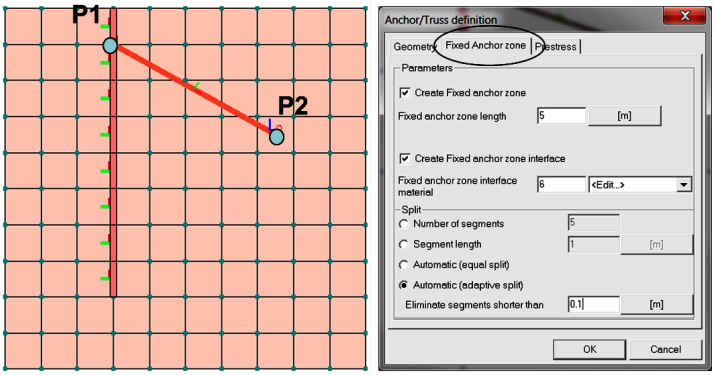

**Remarks**:

- Anchor nodes can be attached to any point inside the background continuum element
- By setting ON ☒ **Link..** checkbox user may connect both endpoints of the anchor with structural elements (beams, shells) or continuum
- The default local axes system $x_L, y_L, z_L$ is defined in the same manner as for beam elements (Window 2-345)
- To flip orientation of $e_{xL}$ base vector use option `Update:Reverse X-axis`
- To flip orientation of $e_{yL}$ base vector use option `Update:Reverse direction`
- User cannot modify the default element basis except flipping axes orientation
- Position of local axes $e_{yL}$ and $e_{zL}$ has absolutely no effect on the results and is required exclusively for the visualization of axial forces in the postprocessor
- After picking 2 points an additional data, required for description of the anchor element, must be set up in the dialog box as shown in the figure
- A fixed anchor zone can automatically be generated through option `Fixed anchor zone` in the main dialog box (Window 2-372)
- Prestress expressed in terms of the axial stress or force can be set under option `Prestress` in the main dialog box (Window 2-373)

Window 2-371

**Window 2-372:** `Anchor:Create:Dialog` **option** `Fixed anchor zone`

Option `Fixed anchor zone` is designed to generate automatically the fixed anchor zone at a certain length on the side of the second endpoint P2 (see figure below). The additional split parameter is needed to smear the interaction with the adjacent soil elements.



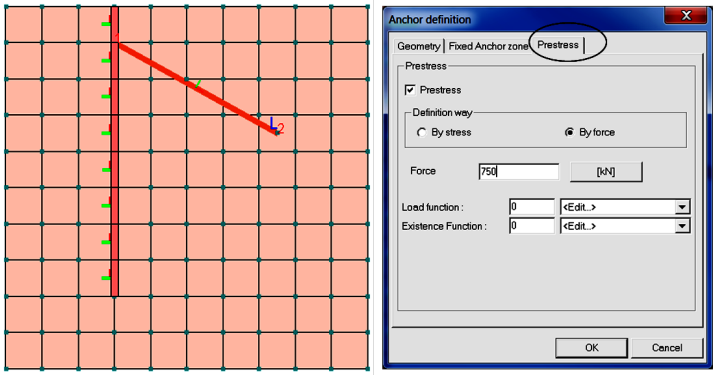

**Remarks**:

- The fixed anchor zone can be created once the state of the ☒ **Create fixed anchor zone** is ON
- If the state of ☒ **Create fixed anchor zone** is set OFF then full continuity of the displacement field for both fixed anchor and subsoil will be preserved (pullout test cannot be properly reproduced)
- If the state of the ☒ **Create fixed anchor zone interface** is ON user may generate an adhesive interface between the subsoil and fixed anchor; this interface is of segment-segment type, where the master segment is linked to the subsoil via `Nodal link` (automaticaly by the preprocessor) while the slave segment is equivalent to the truss element in the fixed anchor zone; the diameter of the injected zone for fixed anchor is set up at the material level for the interface material dedicated to the fixed anchor zone
- Material index for the interface is set up in the **Fixed anchor zone material** [ ]
- To properly reproduce the distribution of the shear stresses in the interface a certain mesh density is needed in that zone; to split this zone user may declare number of segments in the **Number of segments** [ ] or segment length in the **Segment length** [ ] or use one of the two automatic split methods generating equal size split oraz split is enforced by an exact intersection with the background mesh (too short segments should be eliminated)

Window 2-372

**Window 2-373:** `Anchor:Create:Dialog` **option** `Prestress`

Option `Prestress` is designed to apply prestress force to the anchor. Its value expressed in terms of the stress can be driven by the load time function and existence function. In active periods of the existence function prestress is kept under control and current stress will always be equal to the assumed prestress one; during inactive periods current stress in the anchor may deviate from the assumed value.




Window 2-373

**Window 2-374:** `Anchor:Create: 2 Nodes/points`

A 2-node anchor element is created by picking the two geometrical points or nodes in the graphical pane. The two picked points will be first snapped to the existing nodes of the real mesh, then to the grid points. All other parameters required to complete anchor definition are described in Window 2-371

Window 2-374

**Window 2-375:** `Anchor:Create: 2 Nodes`

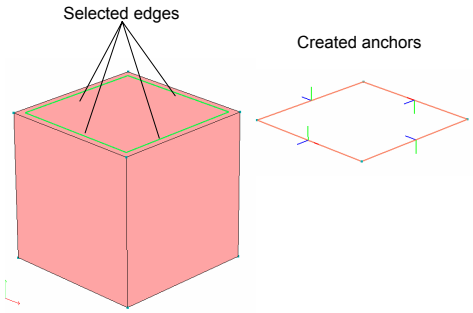
New 2-node anchor element is created by picking the two existing nodes in the graphical pane. It is strongly recommended to not to try to pick the node exactly (it is a difficult action) but rather to select it by picking the existing element next to the one of its vertices that coincides with the node.

All other parameters required to complete anchor definition are described in Window 2-371

Window 2-375

**Window 2-376:** `Anchor:Create:On edge(s)`

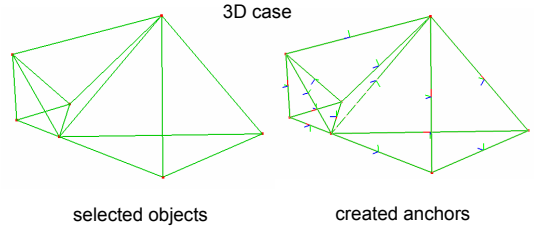
New anchor element(s) are generated on all selected element edges (see figure below). Orientation of each anchor element local $x$ axis is coherent with orientation of the element edge. Setting default local coordinate system of the anchor follows the idea designed for beams, given in Win.(2-345)).




Window 2-376

**Window 2-377:** `Anchor:Create:On object(s)`

New anchor element(s) are generated on all selected geometrical objects (see figure below). Orientation of each anchor element local $x$ axis is coherent with orientation of the geometrical object. Setting default local coordinate system of the anchor follows the idea designed for beam elements, given in Window 2-345




**Remark**:Anchors created on arcs are spanned on the arc endpoints hence it is recommended to split the arc first and then to create anchor elements

Window 2-377

**Window 2-378:** `Anchor:Create:On beam(s)`

New anchor element(s) are generated on all selected beam elements without removing them. Orientation of each anchor element local $x$ axis is coherent with orientation of the parent beam element. Setting default local coordinate system of the anchor follows the idea designed for beam elements and described in Win.(2-345)

Window 2-378

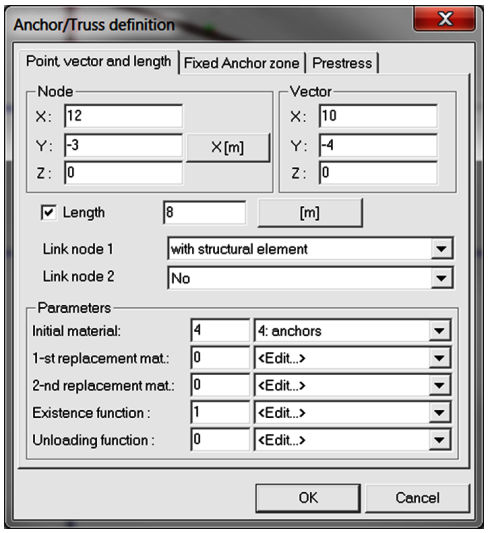
**Window 2-379:** `Anchor:Create:On beam(s) (remove beam(s))`

New anchor element(s) are generated on all selected beam elements. All selected beam elements are removed once anchors generation is finished. Orientation of each anchor element local $x$ axis is coherent with orientation of the parent beam element. Setting default local coordinate system of the anchor follows the idea designed for beam elements and given in Window 2-345

Window 2-379

**Window 2-380:** `Anchor:Create:Point+Vector+Length`

New anchor element is generated by picking the existing node in the graph pane and then by continuing the definition in the dialog box (coordinates of the node can be modified in edit fields **Node** [ ]), setting direction vector (in the edit fields **Vector** [ ]) and setting the anchor length in edit field **Distance** [ ].




**Remarks**:

1. Setting default local coordinate system of the anchor follows the idea designed for beams, given in Window 2-345.
2. For `Sealing` definition see Window 2-372
3. For `Prestress` definition see Window 2-373

Window 2-380

**Window 2-381:** `Anchor:Create:Import (*.2n, *.nvd)`

Anchor elements can be imported from the external ascii file with the two possible templates:

1. for ***.2N*** (2 nodes) file :
   $InitMatNr, MatNr1, MatNr2, ExfNr, UnlFNr, X1, Y1, Z1, X2, Y2, Z2$
   (format: $5 * int$ , $6 * float$)
2. ***.nvd*** (node, vector, distance) file:
   $InitMatNr, MatNr1, MatNr2, ExfNr, UnlFNr, X1, Y1, Z1, Vx, Vy, Vz, D$
   (format: $5 * int$ , $7 * float$)

where:

- $InitMatNr$ is an `Initial material` number
- $MatNr1$ is an `1-st replacement material` number
- $MatNr2$ is an `2-nd replacement material` number
- $ExfNr$ is an `Existence Function` number
- $UnlFNr$ is an `Unloading Function` number
- $Xi, Yi, Zi$ are node coordinates,
- $Vx, Vy, Vz$ are coordinates of an anchor direction vector (does not need to be normalized)
- $D$ is the anchor length

Window 2-381

**Window 2-382:** `Anchor:Select:Sel. elements linked to sel. anchors`

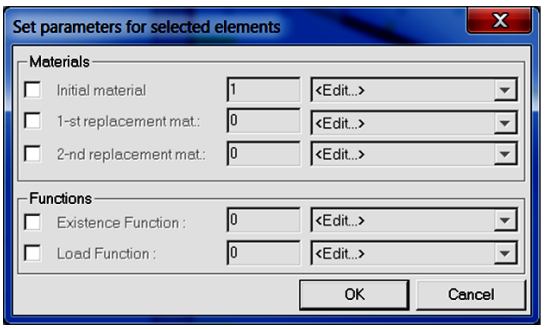
This option is used to select all continuum/structural elements to which nodes of currently selected anchors are linked (through option ☒ **Link to elements**)

Window 2-382

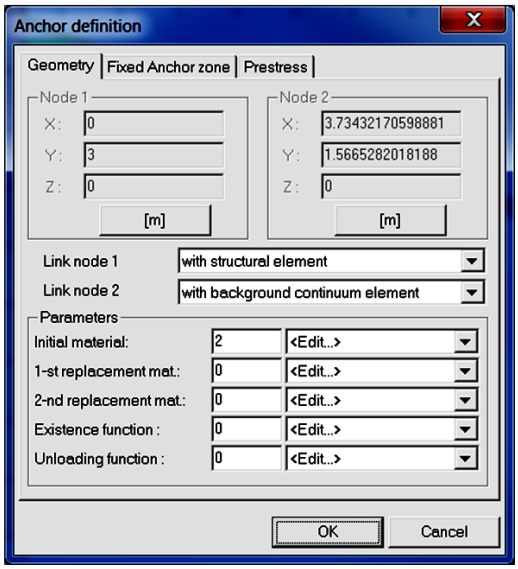
**Window 2-383:** `Anchor:Update:Parameters`

In case when few anchors are selected this option is used to modify `Initial material`, `1-st replacement` and `2-nd replacement materials` IDs, `Existence function`, `Unloading function` exclusively. In case of a single picked anchor, all parameters, including fixed anchor zone, prestress, can be modified.

Few selected anchors




Single selected anchor



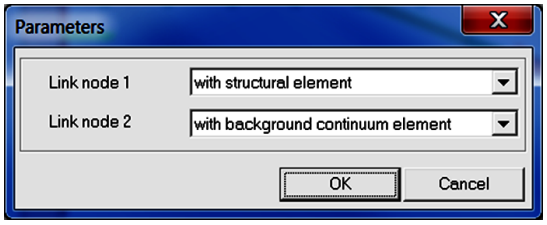

**Remark**:

For `Prestress` definition see Window 2-373

Window 2-383

**Window 2-384:** `Anchor:Update:Link to elements`

This option is used to modify the setup for linking anchor element endpoints to the FE mesh.



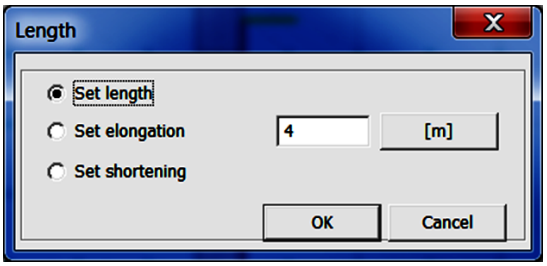

**Remark**:

In the above example first node will be connected to the structural (beam) element (connection does not need to be set necessarily at the node) while the second endpoint will be connected to the continuum element.

Window 2-384

**Window 2-385:** `Anchor:Update:Fixed anchor zone length`

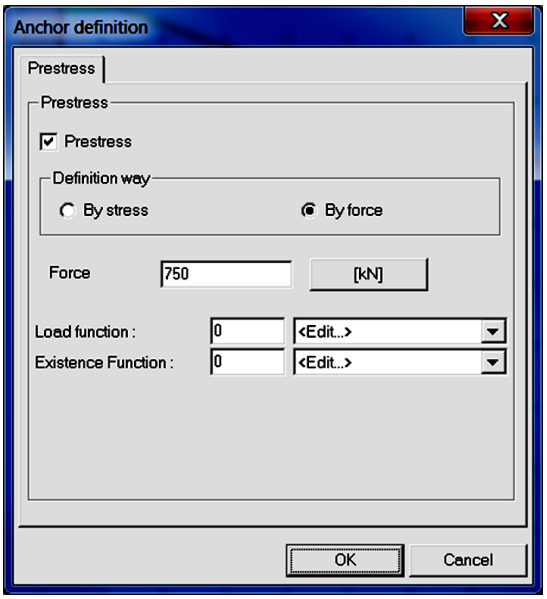
This option allows the user to adjust fixed anchor zone length.




Window 2-385

**Window 2-386:** `Anchor:Update:Prestress`

This option is exclusively used to modify the prestress setup.




**Remark**:

Prestress is defined in terms of the stress or force and can be driven by the load time function and existence function. In active periods of the existence function prestress is kept under control and current axial stress/force will always be equal to the assumed value; during inactive periods current stress/force in the anchor may deviate from the assumed value.

Window 2-386

**Window 2-387:** `Anchor:Update:Reverse direction`

This option is used to flip the local axis $y_L$ and in consequence local axis $z_L$ while $x_L$ remains unchanged.

**Remarks**:

- Proper setting of local anchor axes is important for visualization of normal force diagrams in the postprocessor
- Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

Window 2-387

**Window 2-388:** `Anchor:Update:Reverse X axis`

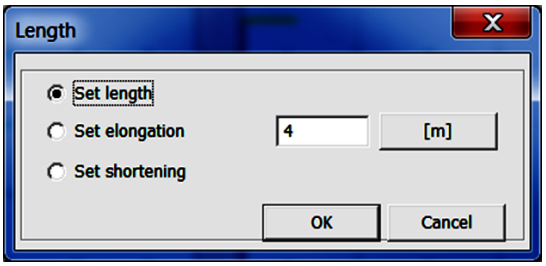
This option is used to flip the local axis $x_L$ and in consequence local axis $z_L$ while $y_L$ remains unchanged.
**Remarks**:

- Proper setting of local anchor axes is very important for visualization of normal force diagrams in the postprocessor
- In case when anchors form a closed polygon their local $x_L$ axes should be oriented in one direction
- Local axis $x_L$, by default, is drawn with red color, $y_L$ with green color and $z_L$ with a blue color

Window 2-388

**Window 2-389:** `Anchor:Update:Length`

This option allows the user to adjust anchor length.




Window 2-389

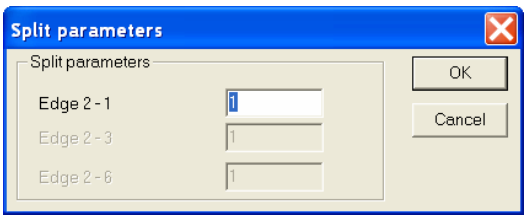
**Window 2-390:** `Anchor:Update:Inclination angle`

This option allows the user to adjust anchor inclination angle in the vertical plane.

Window 2-390

**Window 2-391:** `Anchor:Update:Split`

Using this option user may split picked anchor element or group of selected anchor elements with a split value set in the dialog box.



**Remark**:

Every new anchor element inherits all the parameters from the parent element

Window 2-391

**Window 2-392:** `Anchor:Update:Scale`

Using this option user may scale size of local anchor axes.

**Remark**:

◎ **Same size** and ◎ **Same scale** switches have no meaning here

Window 2-392

### 2.3.1.3.5 CONTINUUM 2D

**Window 2-393: Options in menu** `Continuum 2D`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | T3 on 3 nodes/points | — | Window 2-394 |
| | T3 on 3 nodes | — | Window 2-395 |
| | Q4 on 4 nodes/points | — | Window 2-394 |
| | Q4 on 4 nodes/points | — | Window 2-395 |
| | T3 with 3[x,y,z] | — | Window 2-396 |
| | Q4 with 4[x,y,z] | — | Window 2-396 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| | Extrude edge(s) to continuum |  | Window 2-312 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-398 |
| | Parameters in front | — | Window 2-399 |
| | Split |  | Window 2-400 |
| | Split Q4 to 4 T3 | — | Window 2-400 |
| | Split T3 to 3 Q4 | — | Window 2-400 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Mesh refinement | — | Window 2-402 |

Window 2-393

**Window 2-394:** `Continuum-2D:Create:T3 on 3 nodes/pts, Q4 on 4 nodes/pts`

New T3 or Q4 continuum 2D element is created by picking the three/four points in the graph pane. The picked points will be snapped first to the existing nodes of the real mesh projected on the graph pane and then to the grid points, if snap options are set ON. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create a continuum 2D element using the existing nodes it is recommended to use the option `Create:T3 on 3 nodes/Create:Q4 on 4 nodes` described in Window 2-395.

**Remark**:Points/nodes can be picked in any order

Window 2-394

**Window 2-395:** `Continuum-2D:Create:T3 on 3 nodes, Q4 on 4 nodes`

New T3/Q4 continuum 2D element is created by picking the three/four existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

**Remark**:Nodes can be picked in any order

Window 2-395

**Window 2-396:** `Continuum-2D:Create:T3 with 3 [x,y,z], Q4 with 4 [x,y,z]`

New T3 or Q4 continuum 2D element is created by setting exact coordinates of 3 (for T3 element) or 4 points (for Q4 element) in the dialog box.

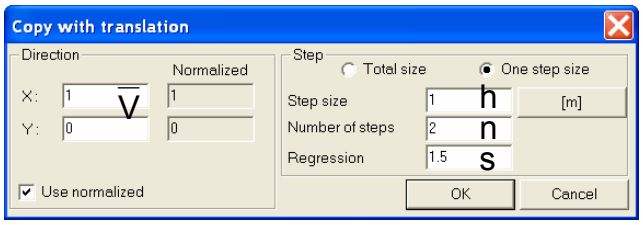
**Remark**:

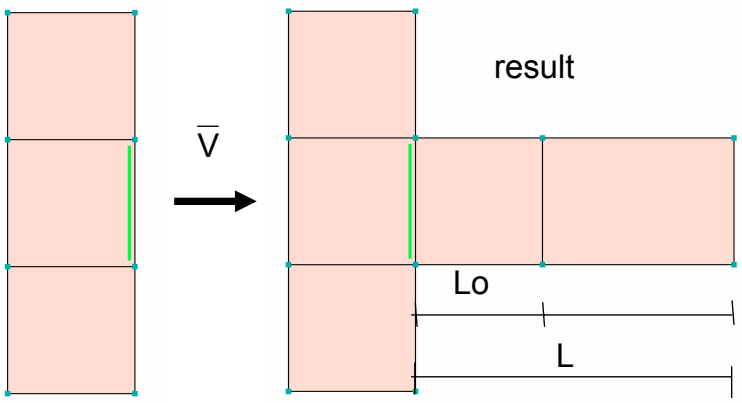
Points can be entered in any order

Window 2-396

**Window 2-397:** `Continuum-2D:Create:Extrude edge(s) to continuum`

New Q4 continuum element(s) are generated by extrusion method on a single picked edge or all selected edges (see figure below).




**Remarks**:

1. Copying operation is carried out along the vector $\overline{\mathbf{v}}$ defined by two coordinates (`X, Y`)
2. $\overline{\mathbf{v}}$ is normalized (or not) depending on the status of ☒ **Use normalized** check-box
3. If ◎ **One step size** is set ON then
   - $\Delta L_o = h \ \| \overline{\mathbf{v}} \|$
4. If ◎ **Total size** is set ON then
   - $L = h$
   - if $s \neq 1$ then $L_o = \dfrac{L\,(1-s)}{(1-s^n)}$

     else $L_o = \dfrac{L}{n}$
5. $L_i = s \ L_{i-1}$ for $i \geq 1$
6. $L = \sum_{i=0}^{n-1} L_i$
7. All parameters in newly generated elements are inherited from the one adjacent to the extruded edge
8. In the example $L_o = 1, L_1 = 1.5, L = 2.5$

Window 2-397

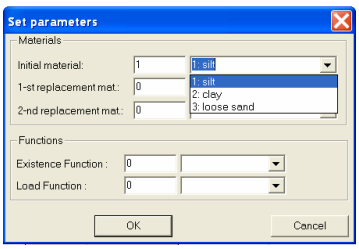
**Window 2-398:** `Continuum-2D:Update:Parameters`

Using this option user may set the IDS of:

**Initial material** [ ],

**1-st replacement material** [ ] and (**2-nd replacement material** [ ],

**Existence function** [ ] and **Unloading function** [ ].



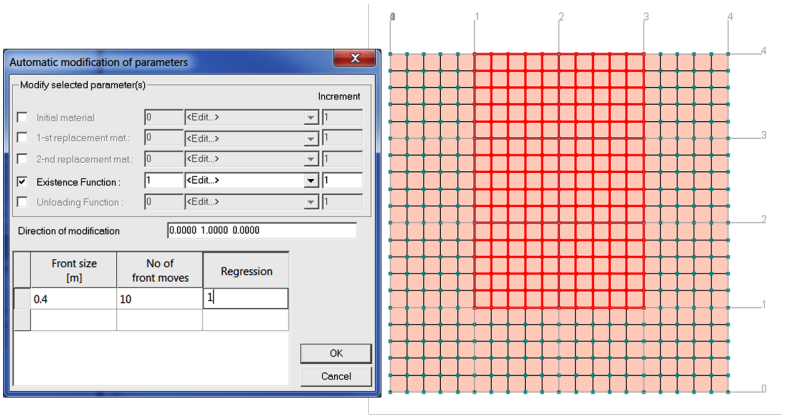

**Remarks**:

1. It is recommended to predefine materials, existence functions and unloading functions first because later on these are visible next to the materials, existence functions etc.. IDs (in the figure list of materials is shown)
2. Only items with ☒ set to ON will be updated; this way user may change only material number without changing other parameters (like existence function, unloading function, first replacement material or second replacement material)

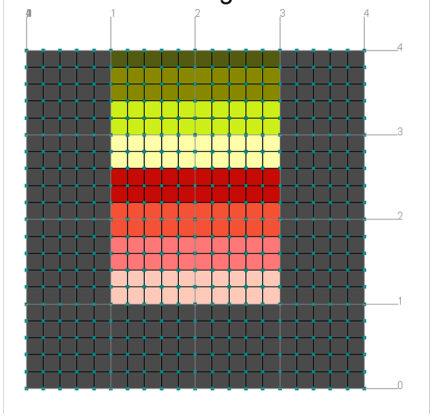
Window 2-398

**Window 2-399:** `Continuum-2D:Update:Parameters in front`

By using this option user may set spatial distribution of the initial materials, replacement materials, existence functions and unloading functions for the selected set of elements (see example shown below). Front size is understood as a zoom size in a given direction that is moving given times.



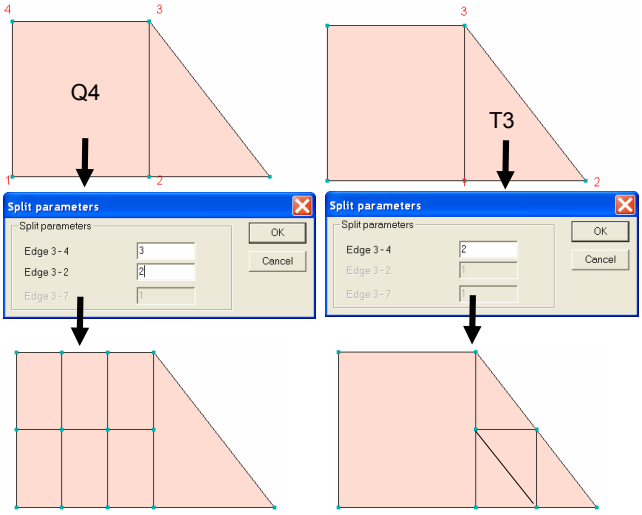

Distribution of resulting existence functions



Window 2-399

**Window 2-400:** `Continuum-2D:Update:Split`

Using this option user may split picked continuum element or group of selected continuum elements by setting the split value in one (for triangles) or two (for quadrilaterals) directions (see example below).



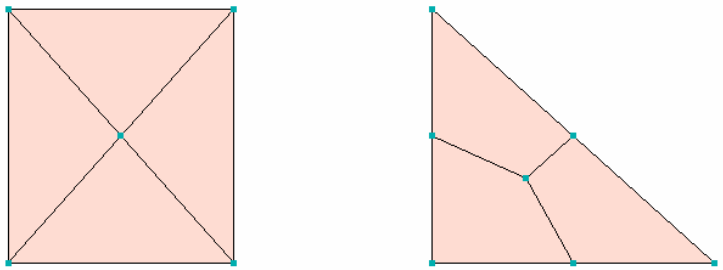

**Remarks**:

1. Every new element inherits all the parameters from the parent element
2. If split is run for group of selected elements then the template split set for the first picked element will be transferred to all adjacent elements to preserve mesh conformity

Window 2-400

**Window 2-401:** `Continuum-2D:Update:Split Q4 to 4 T3, Split T3 to 3 Q4`

Using this option user may split picked continuum element or group of selected continuum elements by setting the split value in one (for triangles) or two (for quadrilaterals) directions (see example below).



**Remarks**:

1. Every new element inherits all the parameters from the parent element
2. Triangular elements can only be used in exceptional situations due to their inherent volumetric locking defect

Window 2-401

**Window 2-402:** `Continuum-2D:Update:Mesh refinement`

For detailed description of this option see Win.2-104.

Window 2-402

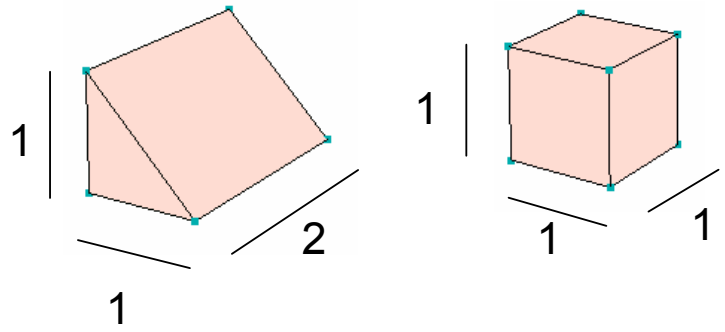
## 2.3.1.3.6 CONTINUUM 3D

**Window 2-403: Options in menu** `Continuum 3D`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | W6 template | — | Window 2-404 |
| | B8 template | — | Window 2-404 |
| | W6 6 nodes | — | Window 2-405 |
| | B8 8 nodes | — | Window 2-405 |
| | 2 Faces | — | Window 2-406 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| | Extrude faces(s) with translation | — | Window 2-407 |
| | Extrude faces(s) with rotation | — | Window 2-408 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-409 |
| | Parameters in front | — | Window 2-410 |
| | Split |  | Window 2-411 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Magnify | — | Window 2-413 |
| | Merge 2B8, 2 W6 | — | Window 2-412 |
| | Mesh refinement |  | Window 2-104, 2-105 |

**Window 2-404:** `Continuum-3D:Create:W6 template, B8 template`

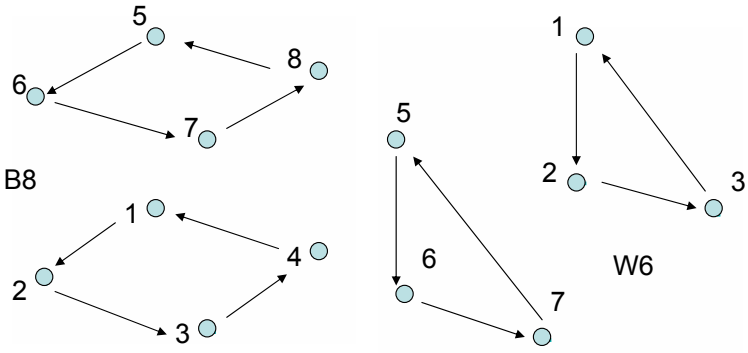
New 6-node wedge (W6) or 8-node brick (B8) continuum 3D element is created with dimensions shown in the figure.




Window 2-404

**Window 2-405:** `Continuum-3D:Create:W6 6 nodes, B8 8 nodes`

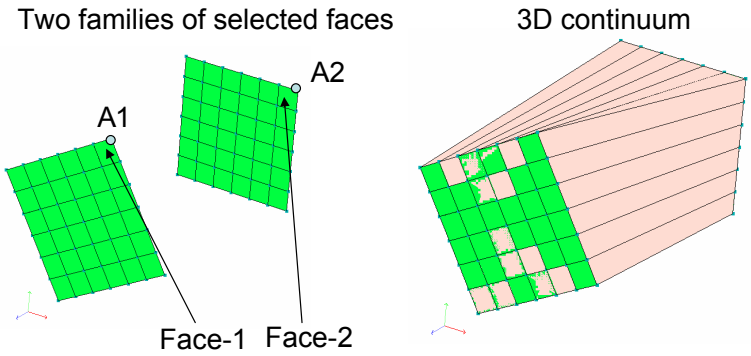
New 6-node wedge (W6) or 8-node brick (B8) continuum 3D element is created by picking 6 or 8 nodes in order shown in the figure.




Window 2-405

**Window 2-406:** `Continuum-3D:Create:2 Faces`

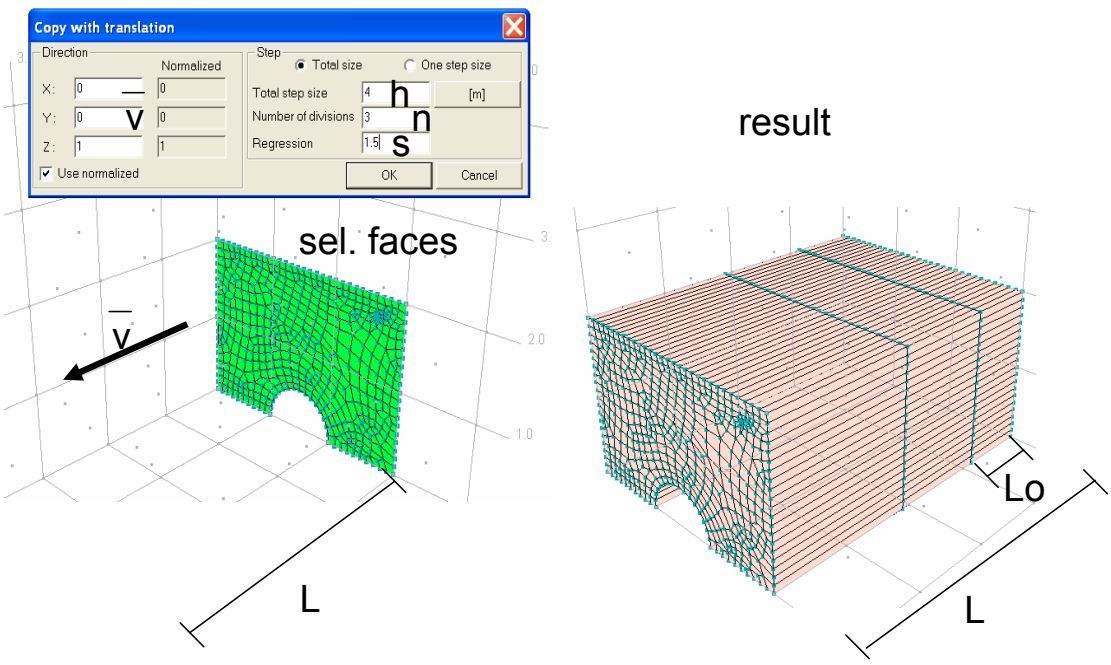
This option allows to create a 3D mesh between the two sets of selected faces. To avoid unexpected results the set of selected faces should form the two disconnected families of faces. To make the definition unique user is asked to pick the two faces to be connected (Face-1, Face-2), first, and then the 2 nodes to be connected (A1, A2)(see Figure below).




Window 2-406

**Window 2-407:** `Continuum-3D:Create:Extrude faces with translation`

New W6, B8 continuum element(s) are generated by extrusion method of a single picked face or all selected faces (see figure below).



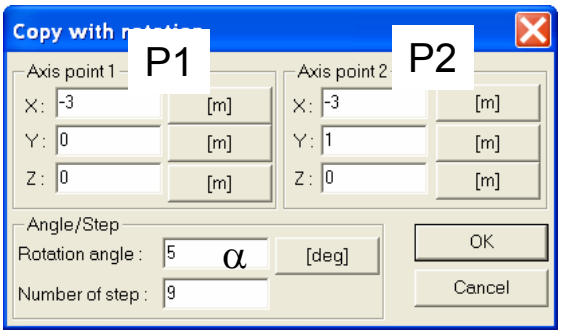

**Remarks**:

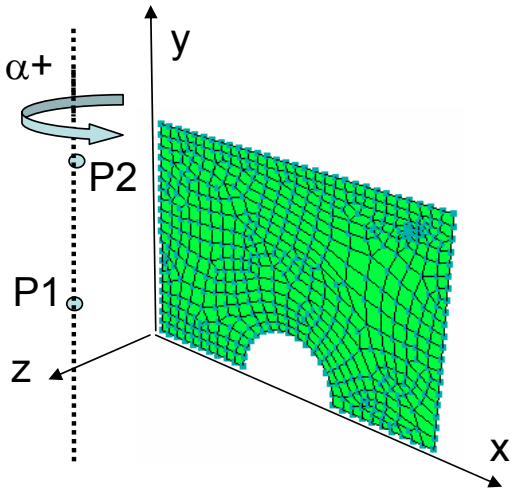
1. Extrusion operation is carried out along the vector $\overline{\mathbf{v}}$ defined by 3 coordinates (`X, Y, Z`)
2. $\overline{\mathbf{v}}$ is normalized (or not) depending on the status of ☒ **Use normalized** check-box
3. If ◎ **One step size** is set ON then
   - $L_o = h \parallel \overline{\mathbf{v}} \parallel$
4. If ◎ **Total size** is set ON then
   - $L = h$
   - if $s \neq 1$ then $L_o = \dfrac{L\,(1-s)}{(1-s^n)}$

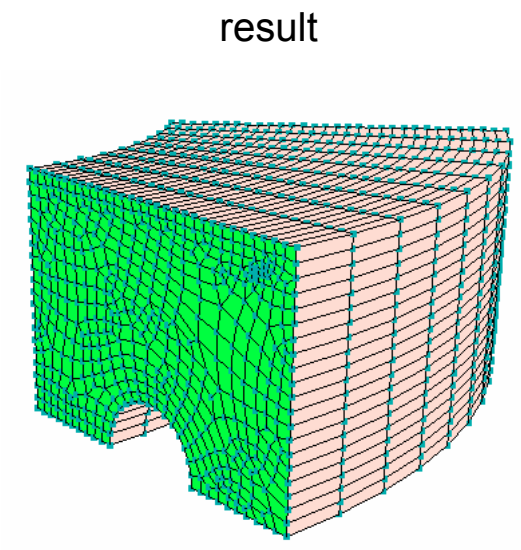
     else $L_o = \dfrac{L}{n}$
5. $L_i = s\ L_{i-1}$ for $i \geq 1$
6. $L = \sum_{i=0}^{n-1} L_i$
7. All parameters in newly generated elements are inherited from one(s) adjacent to the extruded faces

**Window 2-408:** `Continuum-3D:Create:Extrude faces with rotation`

New W6, B8 continuum element(s) are generated by extrusion method of a single picked face or all selected faces (see figure below) with simultaneous rotation.




**Remarks**:

1. Extrusion operation is carried with rotation along the axis defined by two points $P1$ and $P2$
2. Angular distance between two subsequent copies of extruded faces is equal to $\alpha$
3. Faces are copied `Number of steps` times
4. Positive rotation angle is shown in the figure
5. All parameters in newly generated elements are inherited from one(s) adjacent to the extruded faces

Window 2-408

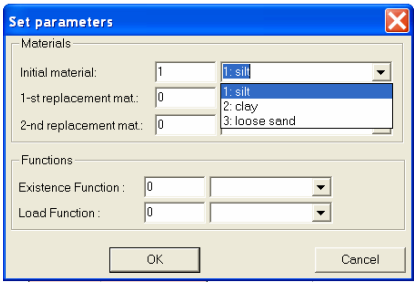
**Window 2-409:** `Continuum-3D:Update:Parameters`

Using this option user may set the IDs of:

**Initial material** [ ],

**1-st replacement material** [ ] and **2-nd replacement material** [ ],

**Existence function** [ ] and **Unloading function** [ ].



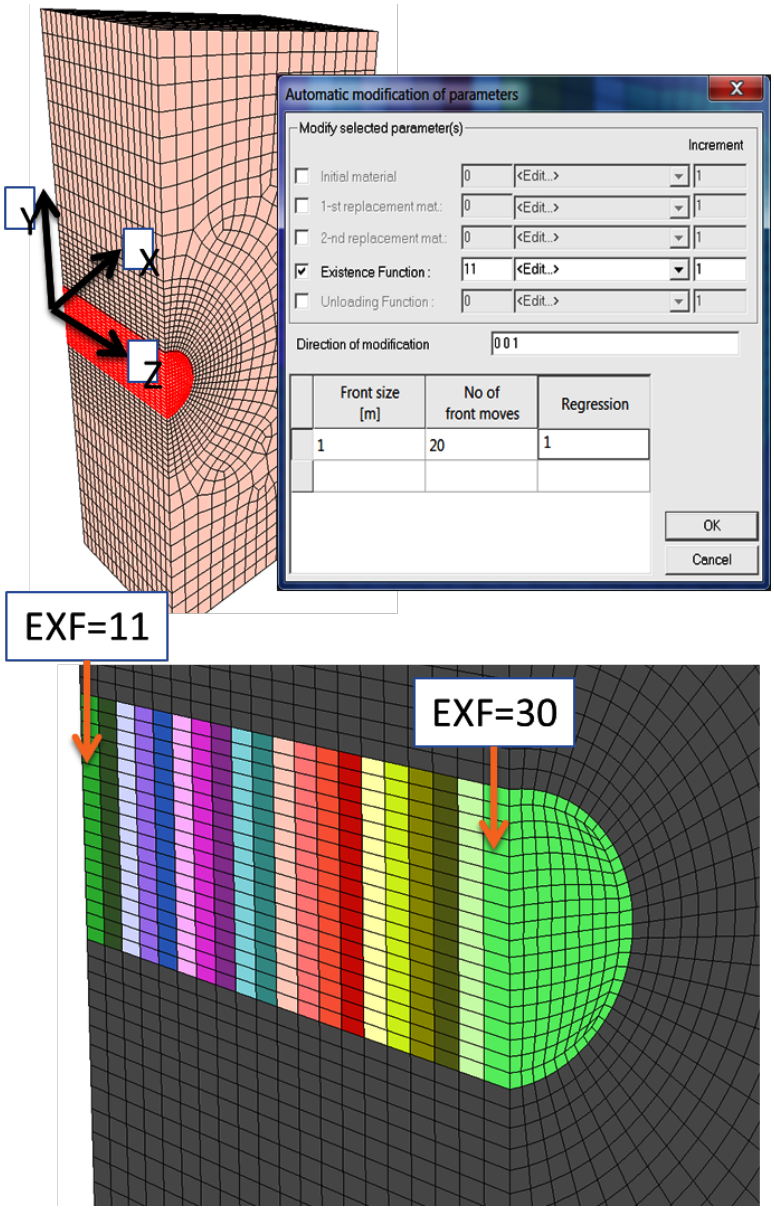

**Remarks**:

1. It is recommended to predefine materials, existence functions and unloading functions first because later on these are visible next to the materials, existence functions etc.. IDs (in the figure list of materials is shown)
2. Only items with ☒ set to ON will be updated; this way user may change only material number without changing other parameters (like existence function, unloading function, first replacement material or second replacement material)

Window 2-409

**Window 2-410:** `Continuum-3D:Update:Parameters in front`

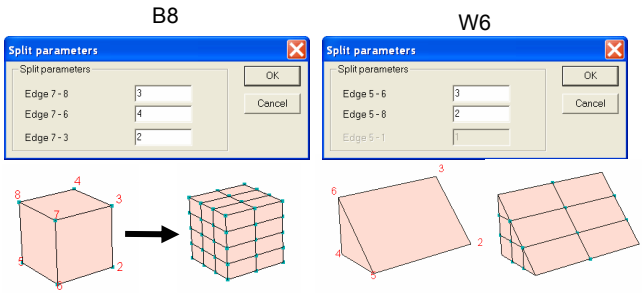
By using this option user may set spatial distribution of the initial materials, replacement materials, existence functions and unloading functions for the selected set of elements (see example shown below). Front size is understood as a zoom size in a given direction that is moving given times. Setting automatic distribution of existence functions for tunnel excavation is shown in the figure.




Window 2-410

**Window 2-411:** `Continuum-3D:Update:Split`

Using this option user may split picked continuum element or group of selected continuum elements by setting the split value in two (for wedges W6) or three (for bricks B8) directions (see example below).



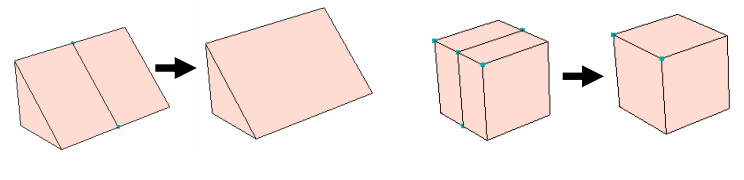

**Remarks**:

1. Every new element inherits all the parameters from the parent element
2. If split is run for group of selected elements then the template split set for the first picked element will be transferred to all adjacent elements to preserve mesh conformity

Window 2-411

**Window 2-412:** `Continuum-3D:Update:Merge 2 B8, 2 W6`

Using this option user can merge two adjacent bricks or wedges by picking them with graphical cursor (see example below).



**Remark**:

The new continuum element will inherit all the parameters from the first picked element

Window 2-412

**Window 2-413:** `Continuum-3D:Update:Magnify`

Using this option user may magnify the element in three (X, Y, Z) directions independently.

**Remark**:

Center of the element remains unchanged

Window 2-413

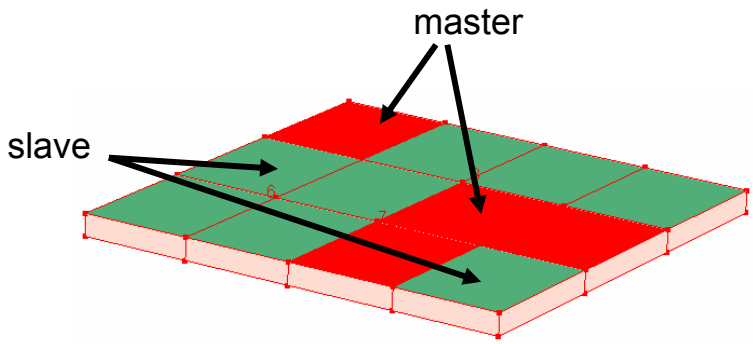
### 2.3.1.3.7 SHELL

**Window 2-414: Options in menu** `Shell`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | Q4 Shell on B8 continuum (leave B8) | — | Window 2-415 |
| | Q4 Shell on B8 continuum (remove B8) | — | Window 2-416 |
| | B8 continuum on Q4 Shell (leave Q4) | — | Window 2-417 |
| | B8 continuum on Q4 Shell (remove Q4) | — | Window 2-418 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| | Extrude faces(s) with translation | — | Window 2-419 |
| | Extrude faces(s) with rotation | — | Window 2-420 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-421 |
| | Parameters in front | — | Window 2-422 |
| | Split |  | Window 2-423 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Mesh refinement |  | Window 2-104, 2-105 |
| Master-Slave | Show | — | Window 2-424 |
| | Hide | — | Window 2-425 |
| | Flip | — | Window 2-426 |
| | Set master on face(s) | — | Window 2-427 |

**Window 2-415:** `Shell:Q4 Shell on B8 continuum (leave B8)`

New Q4 shell element (with double layer of nodes located on master and slave faces) is created without removing the source continuum B8 element.




**Remarks**:

1. Note that B8 continuum and Q4 shell will occupy the same volume
2. Master faces are paint with red color while slave ones with green
3. Master and slave faces must be arranged in such a way that any master face can be adjacent only to another master face and slave to slave accordingly (in the example master and slave faces are not properly arranged)
4. For Deformation+Flow analyzes this shell element can be set as fully impermeable (when ☒ **Flow** group is OFF), fully permeable or permeable with certain seepage coefficients defined as for the interface (contact) elements
5. Thickness of the shell is derived from the B8 type geometry

Window 2-415

**Window 2-416:** `Shell:Q4 Shell on B8 continuum (remove B8)`

This option is equivalent to the `Q4 Shell on B8 continuum (leave B8)` with only one exception that source continuum B8 element is removed (see Window 2-415)

Window 2-416

**Window 2-417:** `Shell:B8 continuum on Q4 shell (leave Q4)`

New B8 continuum element(s) is(are) created in place of picked shell element or group of selected shell elements without removing source shell element(s).

Window 2-417

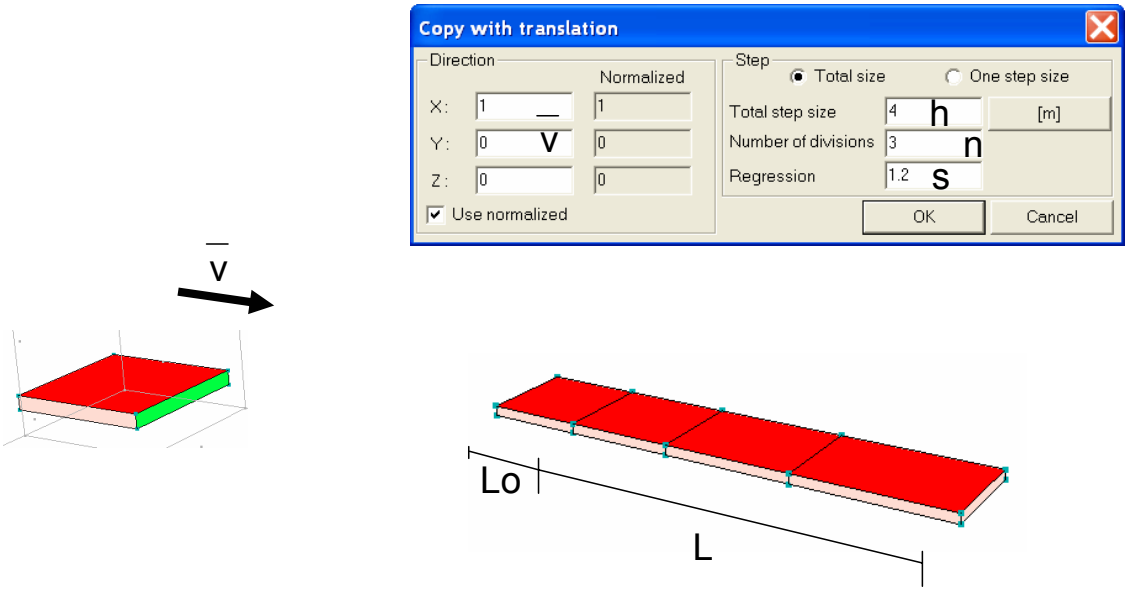
**Window 2-418:** `Shell:B8 continuum on Q4 shell (remove Q4)`

This option is equivalent to the `B8 continuum on Q4 shell (leave Q4)` with the exception that source shell Q4 element is removed (see Window 2-417)

Window 2-418

**Window 2-419:** `Shell:Create:Extrude faces with translation`

New shell element(s) are generated by extrusion method from a single picked face or set of selected faces (see figure below).




**Remarks**:

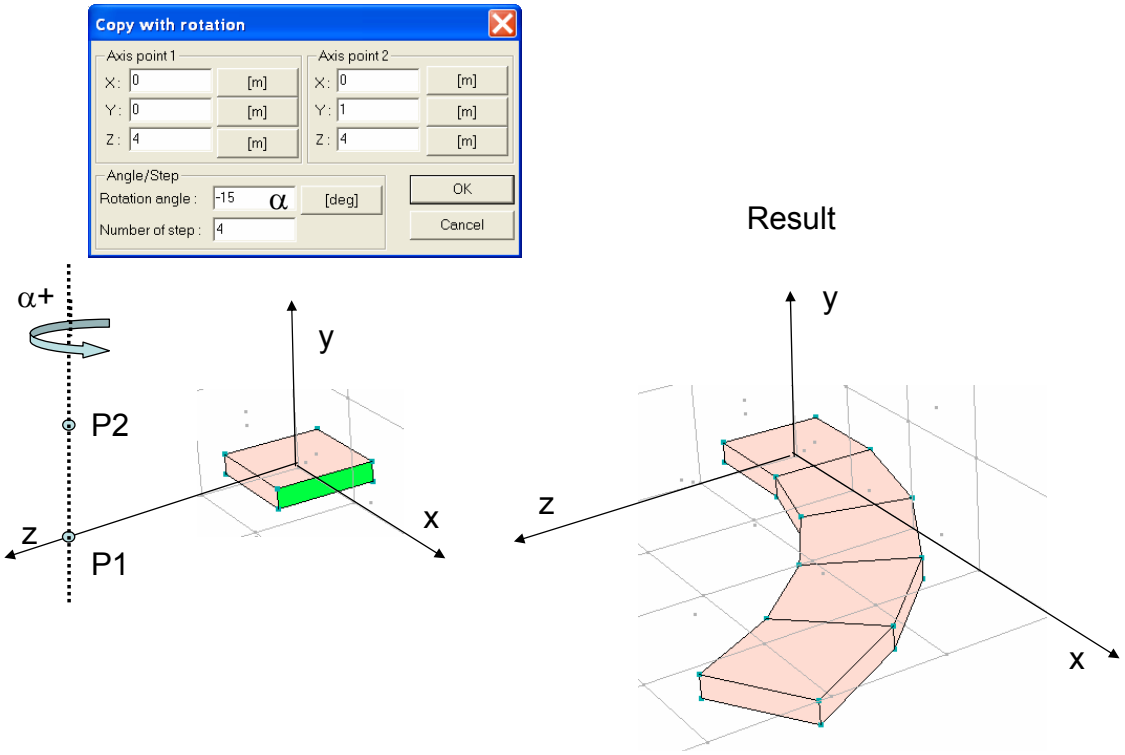
1. Extrusion operation is carried out along the vector $\overline{\mathbf{v}}$ defined by 3 coordinates (`X, Y, Z`)
2. $\overline{\mathbf{v}}$ is normalized (or not) depending on the status of ☒ **Use normalized** check-box
3. If ◎ **One step size** is set ON then
   - $L_o = h \; \| \overline{\mathbf{v}} \|$
4. If ◎ **Total size** is set ON then
   - $L = h$
   - if $s \neq 1$ then $L_o = \dfrac{L\,(1-s)}{(1-s^n)}$

     else $L_o = \dfrac{L}{n}$
5. $L_i = s \;\; L_{i-1}$ for $i \geq 1$
6. $L = \sum_{i=0}^{n-1} \; L_i$
7. All parameters in newly generated elements are inherited from one(s) adjacent to the extruded faces; in case when shell is extruded from continuum element faces, material attached to newly created shells must be corrected by the user
8. Master-slave setting must be verified after the action

Window 2-419

**Window 2-420:** `Shell:Create:Extrude faces with rotation`

New shell elements are generated by extrusion method from a single picked face or set of selected faces (see figure below) with simultaneous rotation.




**Remarks**:

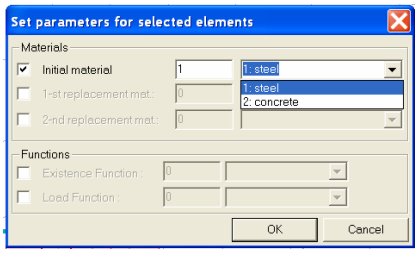
1. Extrusion operation is carried with rotation along the axis defined by two points $P1$ and $P2$
2. Angular distance between two subsequent copies of extruded faces is equal to $\alpha$
3. Faces are copied `Number of steps` times
4. Positive rotation angle is shown in the figure
5. All parameters in newly generated elements are inherited from one(s) adjacent to the extruded faces; in case when shell is extruded from continuum element faces material attached to newly created shells must be corrected by the user

Window 2-420

## Window 2-421: Shell:Update:Parameters

Using this option user may set the IDs of:

**Initial material** [ ], **1-st replacement material** [ ] and **2-nd replacement material** [ ], **Existence function** [ ] **Unloading function** [ ].



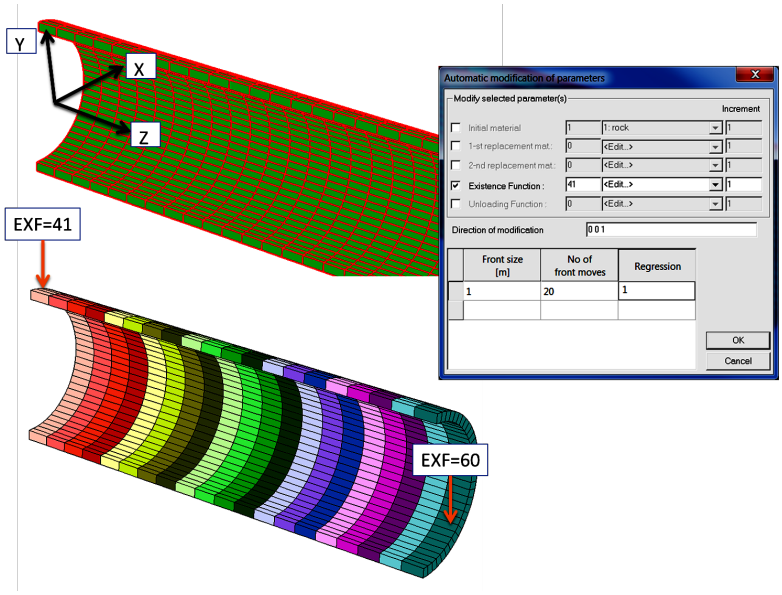

**Remarks**:

1. It is recommended to predefine materials, existence functions and unloading functions first because later on these are visible next to the materials, existence functions etc.. IDs (in the figure list of materials is shown)
2. Only items with ☒... set to ON will be updated; this way user may change only material number without changing other parameters (like existence function, unloading function, first replacement material or second replacement material)

Window 2-421

## Window 2-422: Shell:Update:Parameters in front

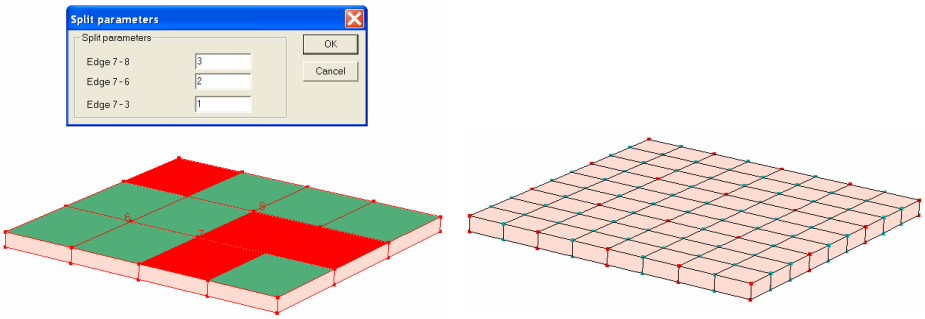
By using this option user may set spatial distribution of the initial materials, replacement materials, existence functions and unloading functions for the selected set of elements (see example shown below). Front size is understood as a zoom size in a given direction that is moving given times. Setting automatic distribution of existence functions for the tunnel lining is shown in the figure.



Window 2-422

**Window 2-423:** `Shell:Update:Split`

Using this option user may split picked shell element or group of selected shell elements by setting the split value in two directions (see example).




**Remarks**:

1. Every new element inherits all the parameters from the parent element
2. If split is run for group of selected elements then the template split set for the first picked element will be transferred to all adjacent elements to preserve mesh conformity (see example)

Window 2-423

**Window 2-424:** `Shell:Master-Slave:Show`

Using this option user may visualize position of master (paint with red color) and slave faces (paint with green color)

**Remarks**:

1. Flipping master-slave faces can be made for a single picked shell element or group of selected shell elements by using option `Update:Flip`
2. Setting masters on a picked shell element face or group of selected shell elements faces can be made through option `Update:On face(s)`

Window 2-424

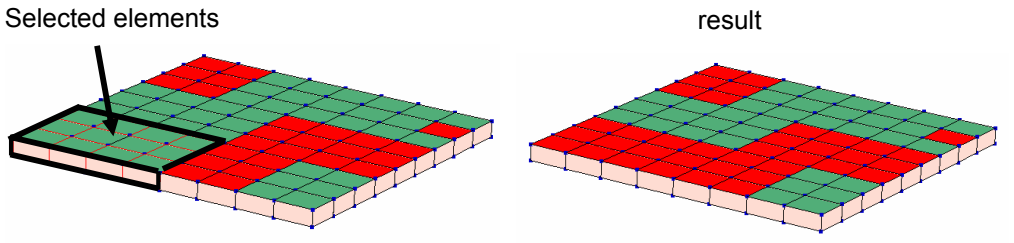
**Window 2-425:** `Shell:Master-Slave:Hide`

This option cancels visualization of master-slave faces.

Window 2-425

**Window 2-426:** `Shell:Master-Slave:Flip`

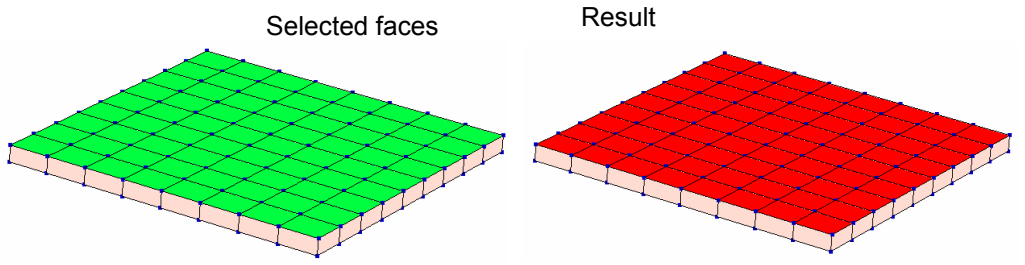
Using this option user may flip position of the master face(s) in a single picked shell element or group of selected shell elements (see figure)




Window 2-426

**Window 2-427:** `Shell:Master-Slave:Set masters on faces`

Using this option user may set master face on a single picked shell element face or on group of selected shell element faces (see figure).




Window 2-427

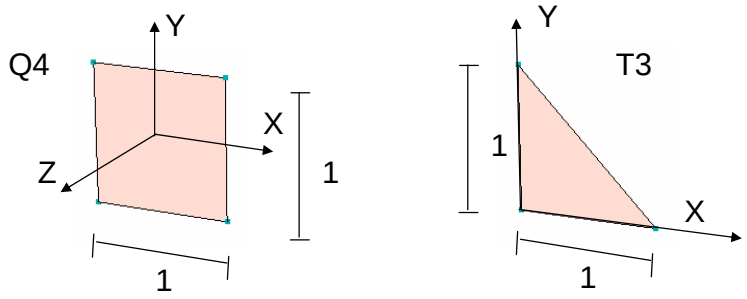
### 2.3.1.3.8 SHELL-ONE LAYER

**Window 2-428: Options in menu** `Shell one layer`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | Template Q4 | — | Window 2-429 |
| | Template T3 | — | Window 2-429 |
| | T3 on 3 nodes/points | — | Window 2-430 |
| | T3 on 3 nodes | — | Window 2-431 |
| | Q4 on 4 nodes/points | — | Window 2-431 |
| | Q4 on 4 nodes | — | Window 2-430 |
| | On face(s) | — | Window 2-432 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-434 |
| | Parameters in front | — | Window 2-435 |
| | Thickness | — | Window 2-433 |
| | Split |  | Window 2-436 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Mesh refinement |  | Window 2-104,2-105 |
| Direction | Show | — | Window 2-437 |
| | Hide | — | Window 2-438 |
| | Reverse | — | Window 2-439 |
| | Scale | — | Window 2-440 |
| Thickness | Hide/Show color map | — | Window 2-441 |
| | Hide/Show volume | — | Window 2-442 |

**Window 2-429:** `Shell one layer:Template Q4, Template T3`

New 3-node triangular (T3) or 4-node quadrilateral (Q4) one layer shell element is created with dimensions shown in the figure.




Window 2-429

**Window 2-430:** `Shell one layer:T3 on 3 nodes/points, Q4 on 4 nodes/points`

New T3 or Q4 shell element is created by picking the three/four points in the graph pane. The picked points will be snapped first to the existing nodes of the real mesh projected on the graph pane and then to the grid points, if snap options are set ON. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create a shell element using the existing nodes it is recommended to use the option `Create:T3 on 3 nodes/Create:Q4 on 4 nodes` described in Window 2-431.

**Remark**:Points/nodes can be picked in any order

Window 2-430

**Window 2-431:** `Shell one layer:T3 on 3 nodes, Q4 on 4 nodes`

New T3/Q4 shell element is created by picking the three/four existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

**Remark**:Nodes can be picked in any order

Window 2-431

**Window 2-432:** `Shell one layer:On faces`

New shell elements are created on a single picked face or group of selected faces.

**Remarks**:

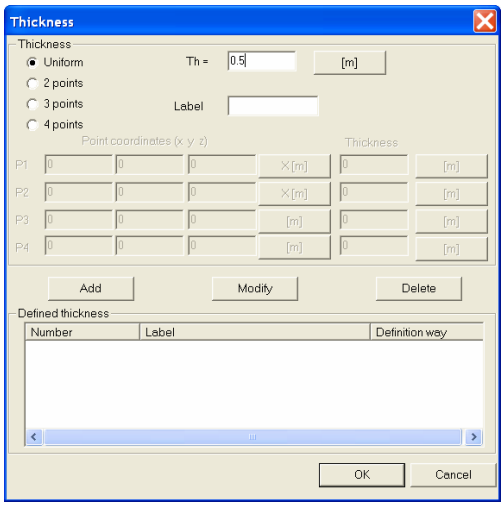
1. Shell elements are usually created on selected continuum element faces
2. Direction of the external shell normal vector must be verified before running computation
3. Top shell fibers are pointed by the positive normal direction

Window 2-432

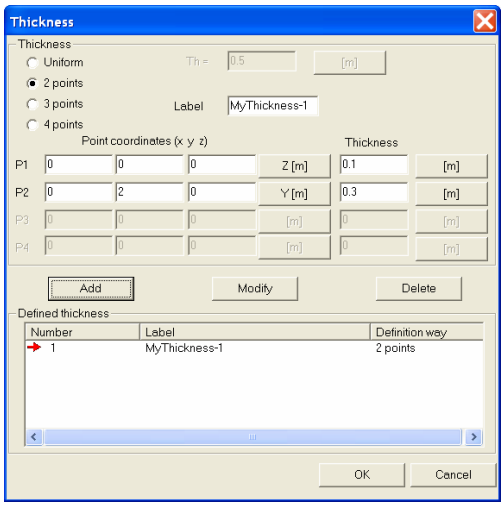
**Window 2-433:** `Shell one layer:Update:Thickness`

Using this option user may define constant or variable thickness in the single picked shell element or group of selected elements.

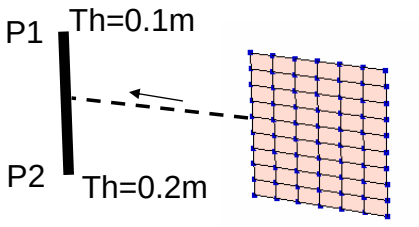
UNIFORM



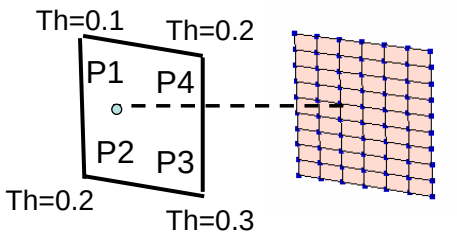

VARIABLE




L2 (2 node)




Q4 (4 node)




**Remarks**:

1. Constant thickness is set in the edit field **Th.** [ ] once the option ◎ **Uniform** is selected
2. Variable thickness can be defined with aid of 2-node linear , 3-node triangular or 4-node quadrilateral macro-elements; to complete definition of the variable thickness user has to select type of the macro-element (◎ **2-node**, ◎ **3-node**, ◎ **4-node**), define coordinates of macro-element vertices and corresponding thickness at each vertex; thickness at each shell element node is computed by projecting it on the macro-element and then interpolating by means of standard shape functions for L2, T3 or Q4 element (see example)
3. Each thickness definition can be memorized, with a label, for further usage by using button **Add**
4. List of memorized definitions can be modified with buttons **Delete** or **Modify**

Window 2-433

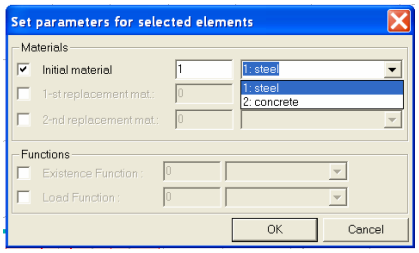
**Window 2-434:** `Shell one layer:Update:Parameters`

Using this option user may set the IDs of:

**Initial material** [ ],

**1-st replacement material** [ ] and **2-nd replacement material** [ ],

**Existence function** [ ] **Unloading function** [ ].



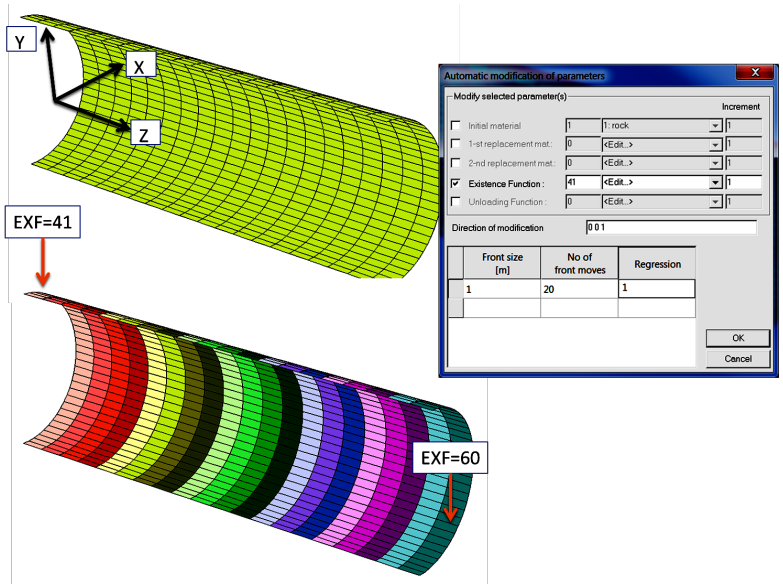

**Remarks**:

1. It is recommended to predefine materials, existence functions and unloading functions first because later on these are visible next to the materials, existence functions etc.. IDs (in the figure list of materials is shown)
2. Only items with ☒ ... set to ON will be updated; this way user may change only material number without changing other parameters (like existence function, unloading function, first replacement material or second replacement material)

Window 2-434

**Window 2-435:** `Shell one layer:Update:Parameters in front`

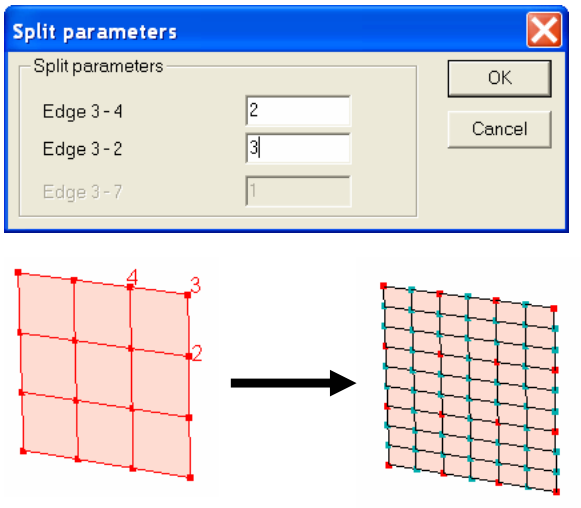
By using this option user may set spatial distribution of the initial materials, replacement materials, existence functions and unloading functions for the selected set of elements (see example shown below). Front size is understood as a zoom size in a given direction that is moving given times. Setting automatic distribution of existence functions for the tunnel lining is shown in the figure.



Window 2-435

**Window 2-436:** `Shell one layer:Update:Split`

Using this option user may split picked shell element or group of selected shell elements by setting the split value in two directions (see example).



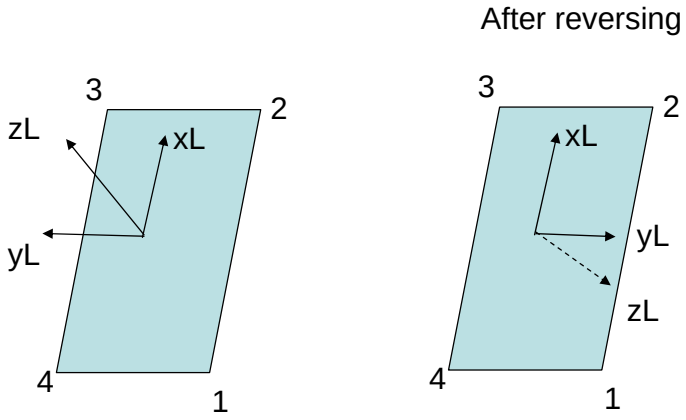

**Remarks**:

1. Every new element inherits all the parameters from the parent element
2. If split is run for group of selected elements then the template split set for the first picked element will be transferred to all adjacent elements to preserve mesh conformity (see example)

Window 2-436

**Window 2-437:** `Shell one layer:Direction:Show`

Using this option user may visualize orientation of the shell external normals. Definition of the resulting local shell coordinate system is shown in figure below.




**Remark**:To modify direction one may use an option `Direction:Reverse`

Window 2-437

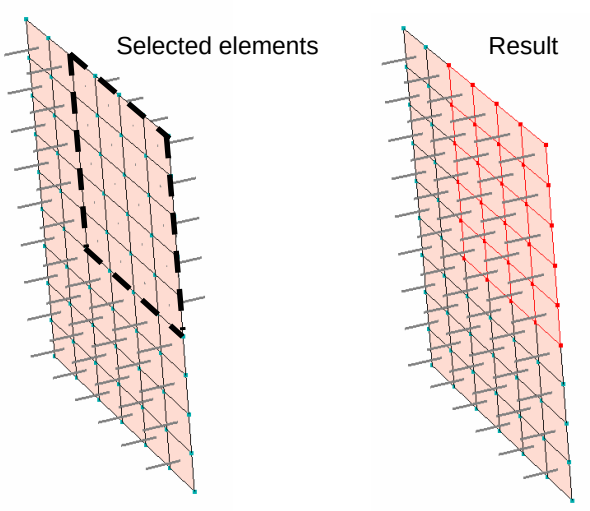
**Window 2-438:** `Shell one layer:Direction:Hide`

This option cancels visualization of shells external normals.

Window 2-438

**Window 2-439:** `Shell one layer:Direction:Reverse`

Using this option user may reverse orientation of a single picked shell element or group of selected shell elements (see figure)




Window 2-439

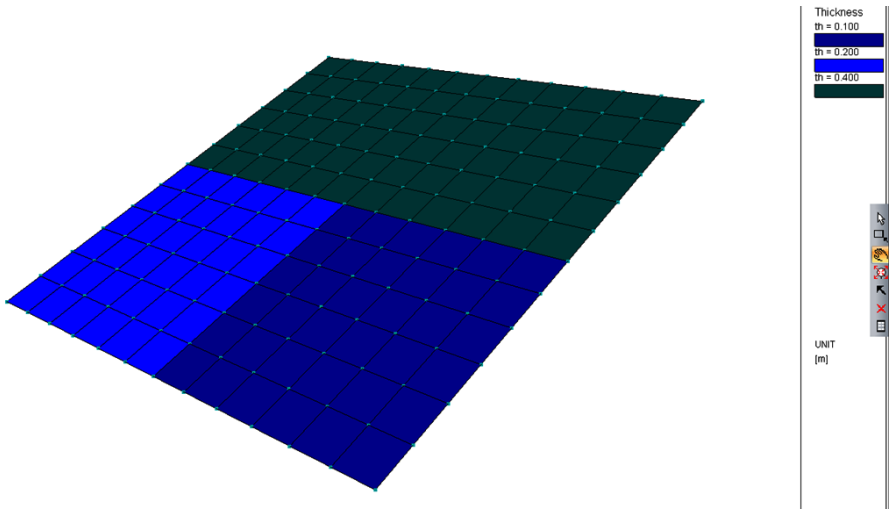
**Window 2-440:** `Shell one layer:Direction:Scale`

Using this option user may enlarge or diminish size of the external normal vectors.

Window 2-440

**Window 2-441:** `Shell one layer:Thickness:Hide/Show color map`

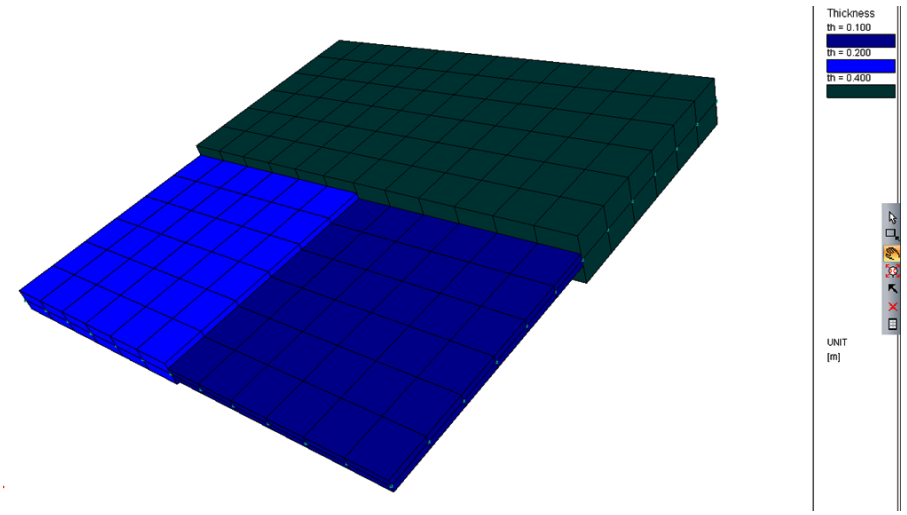
Using this option user may visualize thickness of shell elements in form of color map suplemented by a legend.



Window 2-441

**Window 2-442:** `Shell one layer:Thickness:Hide/Show volume`

Using this option user may visualize thickness of shell elements by visualizing them as solid elements. The third dimension represents the true thickness of the shell element.




Window 2-442

## 2.3.1.3.9 MEMBRANE

**Window 2-443: Options in menu** `Membrane`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes/points | — | Window 2-444 |
| | 2 nodes | — | Window 2-445 |
| | On edge(s) | — | Window 2-446 |
| | T3 on 3 nodes/points | — | Window 2-447 |
| | T3 on 3 nodes | — | Window 2-448 |
| | Q4 on 4 nodes/points | — | Window 2-447 |
| | Q4 on 4 nodes | — | Window 2-448 |
| | On face(s) | — | Window 2-449 |
| | Copy with rotation |  | Window 2-310 |
| | Copy with translation |  | Window 2-311 |
| | Copy by symmetry |  | Window 2-312 |
| Select | Box with 2 nodes |  | Window 2-316 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-450 |
| | Parameters in front | — | Window 2-451 |
| | Split |  | Window 2-452 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |
| | Mesh refinement |  | Window 2-104 , 2-105 |
| Direction | Show | — | Window 2-453 |
| | Hide | — | Window 2-454 |
| | Reverse | — | Window 2-455 |
| | Reverse X-axis | — | Window 2-456 |
| | Scale | — | Window 2-457 |

**Window 2-444:** `Membrane:2 nodes/points`

New 2-node membrane element is created by picking the two points in the graph pane. The two picked points will be first snapped to the existing nodes of the real mesh projected on the graph pane and then to the grid points, if snap options are set ON. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if we want to create an membrane between the two existing nodes it is recommended to use the option `Create:2 nodes` described in Window 2-445.

1. This definition is valid exclusively for the 2D models
2. Positioning of the local axes follow the procedure designed for anchors and described in Window 2-371
3. Membrane elements are usually created on selected continuum element edges
4. Direction of the external membrane normal vector should be verified before running computation to avoid visualization problems (tensile force in the chain of membrane elements can jump from top to the bottom side)
5. To reverse the normal use option `Direction:Reverse`

Window 2-444

**Window 2-445:** `Membrane:2 nodes`

New 2-node membrane element is created by picking the two existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node. For more information concerning setting local membrane axes see Window 2-444.

Window 2-445

**Window 2-446:** `Membrane:On edges`

New 2D membrane elements are created on a single picked edge or group of selected edges.

**Remarks**:

1. Membrane elements are usually created on selected continuum element edges
2. Direction of the external membrane normal vector should be verified before running computation to avoid visualization problems (tensile force in the chain of membrane elements can jump from top to the bottom side)
3. To reverse the normal use option `Direction:Reverse`

Window 2-446

**Window 2-447:** `Membrane:T3 on 3 nodes/points, Q4 on 4 nodes/points`

New T3 or Q4 membrane element is created by picking the three/four points in the graph pane. The picked points will be snapped first to the existing nodes of the real mesh projected on the graph pane and then to the grid points, if snap options are set ON. This option must be used with care because close to the picked point in the graph pane there may appear several projected nodes/grid points and it may not be so easy to control the selection. So if one wants to create a membrane element using the existing nodes it is recommended to use the option `Create:T3 on 3 nodes/Create:Q4 on 4 nodes` described in Window 2-448.

**Remark**:

Points can be picked in any order

Window 2-447

**Window 2-448:** `Membrane:T3 on 3 nodes, Q4 on 4 nodes`

New T3/Q4 membrane element is created by picking the three/four existing nodes in the graph pane. It is strongly recommended to not to try to pick the node exactly (it is rather difficult action) but rather to select it by picking the existing element close to one of its vertices that coincides with the node.

**Remark**:

Nodes can be picked in any order

Window 2-448

**Window 2-449:** `Membrane:On faces`

New 3D (Q4 or T3) membrane elements are created on a single picked face or group of selected faces.

**Remarks**:

1. Membrane elements are usually created on selected continuum element faces
2. Direction of the external membrane normal vector should be verified before running computation to avoid visualization problems (tensile force in the cross section can jump from top to the bottom side)

Window 2-449

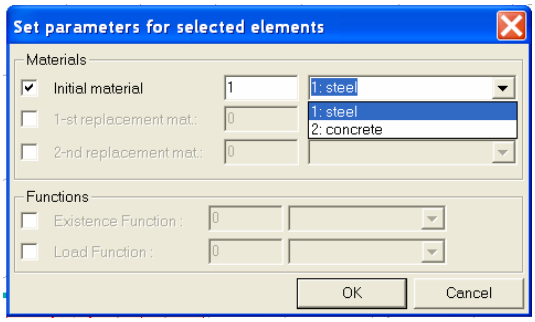
## Window 2-450: Membrane:Update:Parameters

Using this option user may set the IDs of:

**Initial material** [ ],

**1-st replacement material** [ ] and **2-nd replacement material** [ ],

**Existence function** [ ] **Unloading function** [ ].



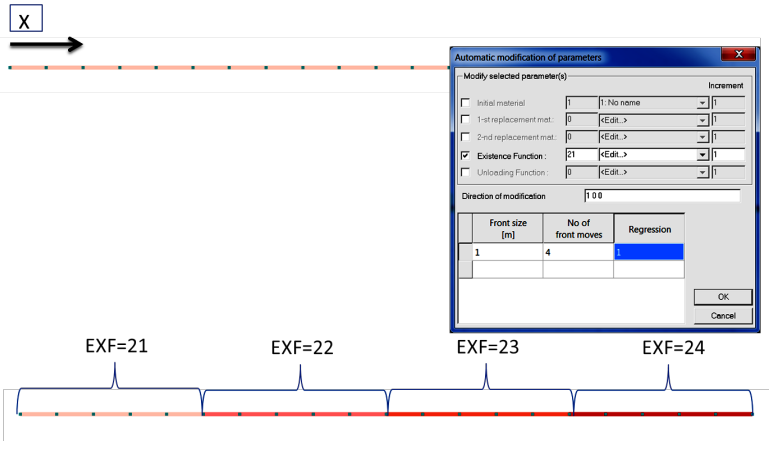

**Remarks**:

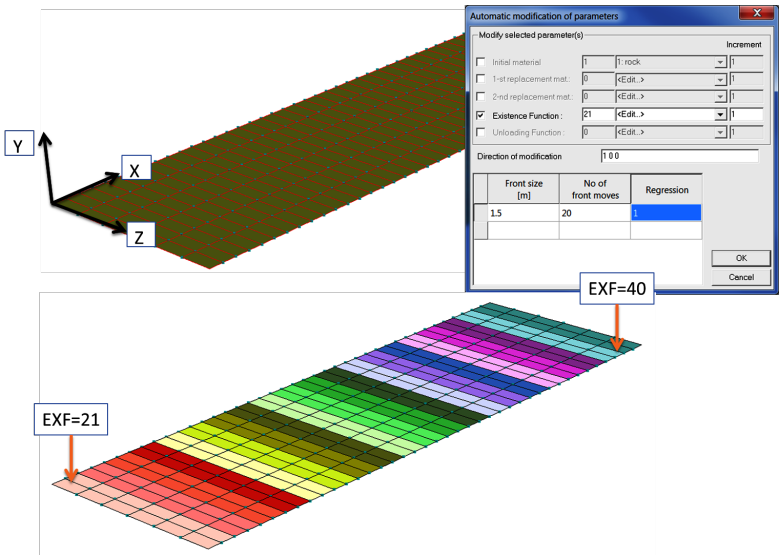
1. It is recommended to predefine materials, existence functions and unloading functions first because later on these are visible next to the materials, existence functions etc.. IDs (in the figure list of materials is shown)
2. Only items with ☒... set to ON will be updated; this way user may change only material number without changing other parameters (like existence function, unloading function, first replacement material or second replacement material)

Window 2-450

## Window 2-451: Membrane:Update:Parameters in front

By using this option user may set spatial distribution of the initial materials, replacement materials, existence functions and unloading functions for the selected set of elements (see example shown below). Front size is understood as a zoom size in a given direction that is moving given times. Setting automatic distribution of existence functions for the membrane is shown below.






Window 2-451

**Window 2-452:** `Membrane:Update:Split`

In the 2D split of membrane element is same as for the anchor one ( see Window 2-391). In the 3D split of membrane element is same as for the shell single layer ( see Window 2-436).

Window 2-452

**Window 2-453:** `Membrane:Direction:Show`

In the 2D this option activates visualization of the local base $xL - yL$ in the same manner as for the anchor element. In the 3D external normals are shown following the procedure designed for single layer shell element (see Window 2-437).

Window 2-453

**Window 2-454:** `Membrane:Direction:Hide`

This option cancels visualization of membranes external normals.

Window 2-454

**Window 2-455:** `Membrane:Direction:Reverse`

Using this option user may reverse orientation of a single picked membrane element or group of selected membrane elements.

For the 2D case this method is same as for the anchor element (see Window 2-387). For the 3D case this method is same as for the shell single layer element (see Window 2-439).

Window 2-455

**Window 2-456:** `Membrane:Direction:Reverse X-axis`

This option is valid only for the 2D cases and is equivalent to the one designed for anchor element (see Window 2-388).

Window 2-456

**Window 2-457:** `Membrane:Direction:Scale`

Using this option user may enlarge or diminish size of the external normal vectors.

Window 2-457

## 2.3.1.3.10 INFINITE ELEMENT

`Infinite elements` representing infinite medium surrounding a domain of interest are developed using mapped infinite element concept by Zienkiewicz, Emson and Bettes, 1983. **These elements do not represent radiation BC in case of dynamic analyses carried out in time domain**.

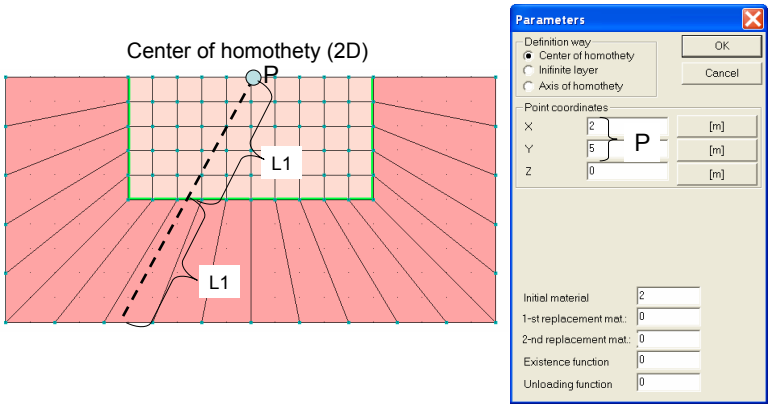
**Window 2-458: Options in menu** `Infinite element`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On edges(s)` | — | Window 2-459, 2-460 |
| | `On face(s)` | — | Window 2-461, 2-463, 2-462 |
| `Select` | `Box with 2 nodes` |  | Window 2-316 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| | `Delete All` |  | Window 2-464 |
| `Update` | `Parameters` | — | Window 2-465 |

Window 2-458

## Window 2-459: Infinite element:On edges (2D)(Center of homothety)

New 2D infinite elements are created on a single picked edge or group of selected edges, through homothetic transformation.



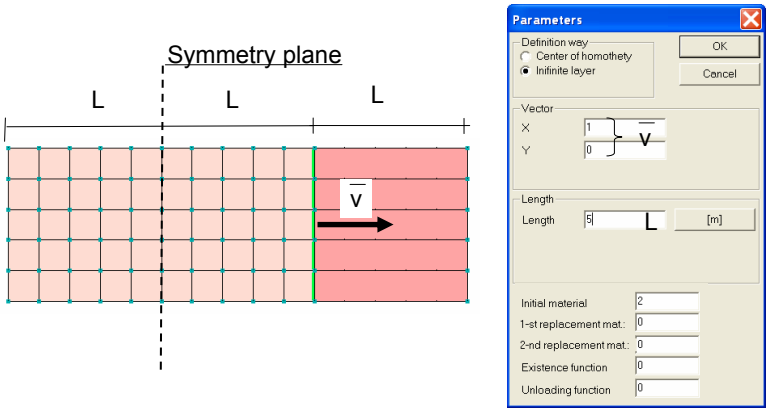

**Remarks**:

1. Infinite mapped elements cannot handle gravity and for that reason special treatment is needed in the initial state computation Window
2. An average length of each infinite mapped element is equal to the distance between center of homothety and center of the edge

Window 2-459

## Window 2-460: Infinite element:On edges (2D)(Infinite layer)

New 2D infinite elements are created on a single picked edge or group of selected edges, to simulate an infinite layer.



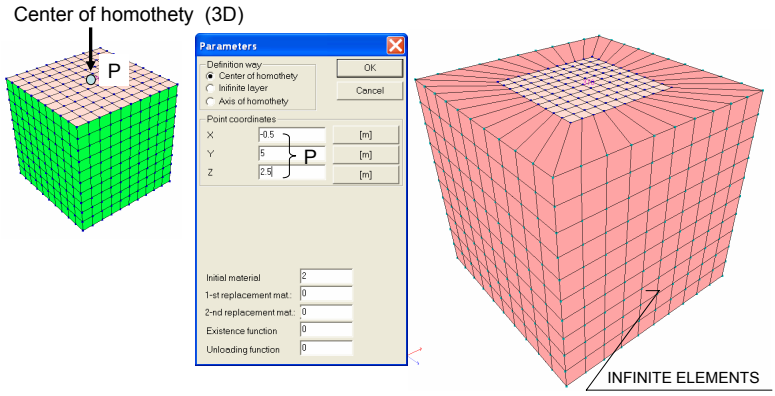

**Remarks**:

1. Infinite mapped elements cannot handle gravity and for that reason special treatment is needed in the initial state computation Window
2. Length of each infinite mapped element is equal to the declared value in field **Length** [ ]

Window 2-460

## Window 2-461: Infinite element:On faces (3D)(Center of homothety)

New 3D infinite elements are created on a single picked face or group of selected faces, through homothetic transformation with a given center (point) of homothety.



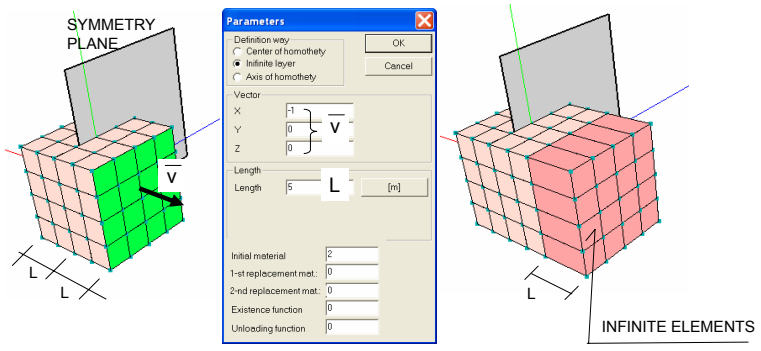

**Remarks**:

1. Infinite mapped elements cannot handle gravity and for that reason special treatment is needed in the initial state computation Window
2. An average length of each infinite mapped element is equal to the distance between center of homothety and center of the face

Window 2-461

## Window 2-462: Infinite element:On faces (3D)(Infinite layer)

New 3D infinite elements are created on a single picked face or group of selected faces, to simulate an infinite layer.



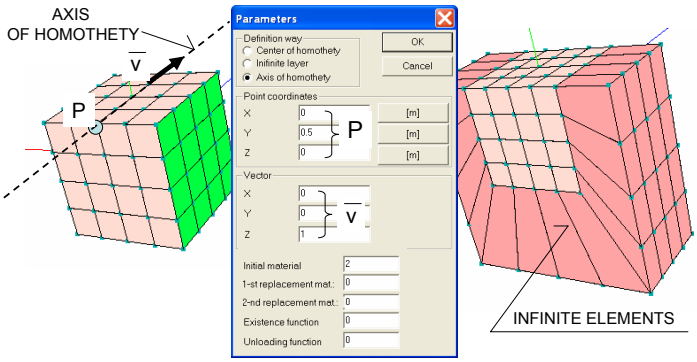

**Remarks**:

1. Infinite mapped elements cannot handle gravity and for that reason special treatment is needed in the initial state computation Window
2. Length of each infinite mapped element is equal to the declared value in field **Length** [ ]

Window 2-462

**Window 2-463:** `Infinite element:On faces (3D)(Axis of homothety)`

New 3D infinite elements are created on a single picked face or group of selected faces, through homothetic transformation with a given axis.




**Remarks**:

1. Infinite mapped elements cannot handle gravity and for that reason special treatment is needed in the initial state computation Window
2. Length of each infinite mapped element is equal to the declared value in field **Length** [ ]

Window 2-463

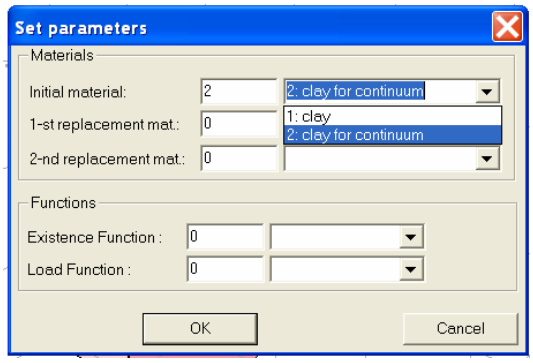
**Window 2-464:** `Infinite element:Delete:Delete all`

Using this option user may delete all generated infinite elements.

Window 2-464

**Window 2-465:** `Infinite element:Update:Parameters`

Using this option user may set the IDs of: **Initial material** [ ], **1-st replacement** [ ] and **2-nd replacement material** [ ], **Existence function** [ ]) and **Unloading function** [ ].



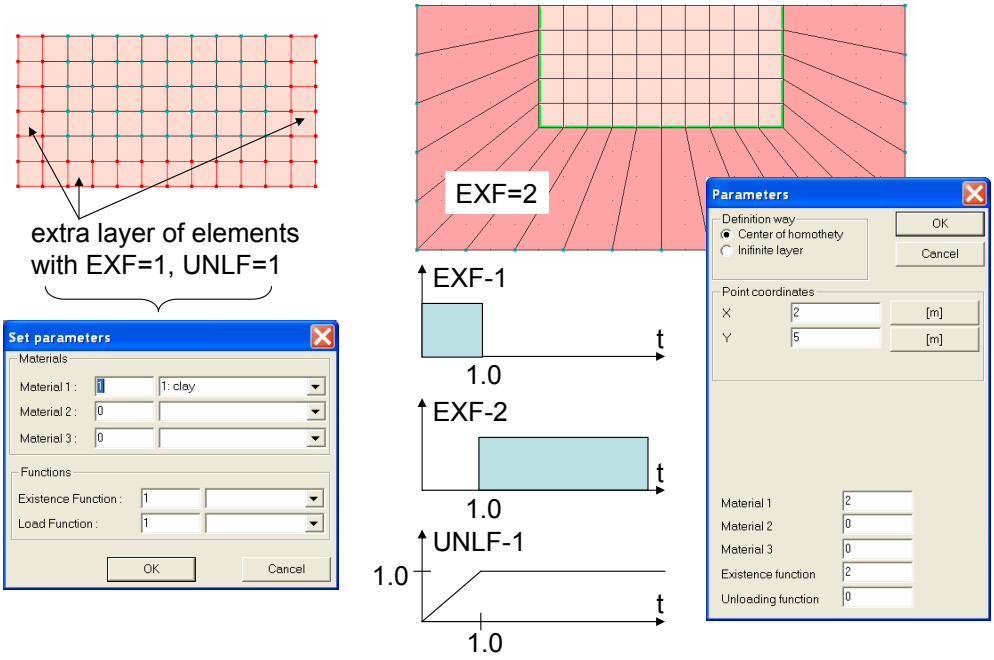

**Remarks**:

1. It is recommended to predefine materials, existence functions and unloading functions first because later on these are visible in the user interfaces next to the materials, existence functions etc.. IDs (in the figure below a list of materials is shown)
2. Only elastic material model can be used to model infinite medium

Window 2-465

**Window 2-466:** `Infinite element:Handling initial state`

As infinite mapped elements cannot handle gravity their presence must be cancelled in the initial state computation. Hence the easiest way to manage that is as follows (see figure)




1. Add additional layer of elements along the boundary at which infinite elements are to be generated
2. Apply box-shaped boundary conditions and assume existence function to all fixities, same as for the additional layer of elements
3. Apply a distinct existence function ID and unloading function ID for the additional layer of elements
   - Existence function for additional layer of elements must be such that in the initial state ($t = 0$) it is ON and then at time when infinite elements are activated it is switched OFF
   - Unloading function for additional layer of elements starts at $t = 0$ from value zero (can be any) and at time when the layer is switched OFF the function must be equal to 1.0 up to infinity (this way all excavation forces will remain unchanged)
4. Apply a distinct existence function ID for infinite elements (inactive at $t = 0$)

Window 2-466

## 2.3.1.3.11 TENDON EMBEDDED IN CONTINUUM/SHELL

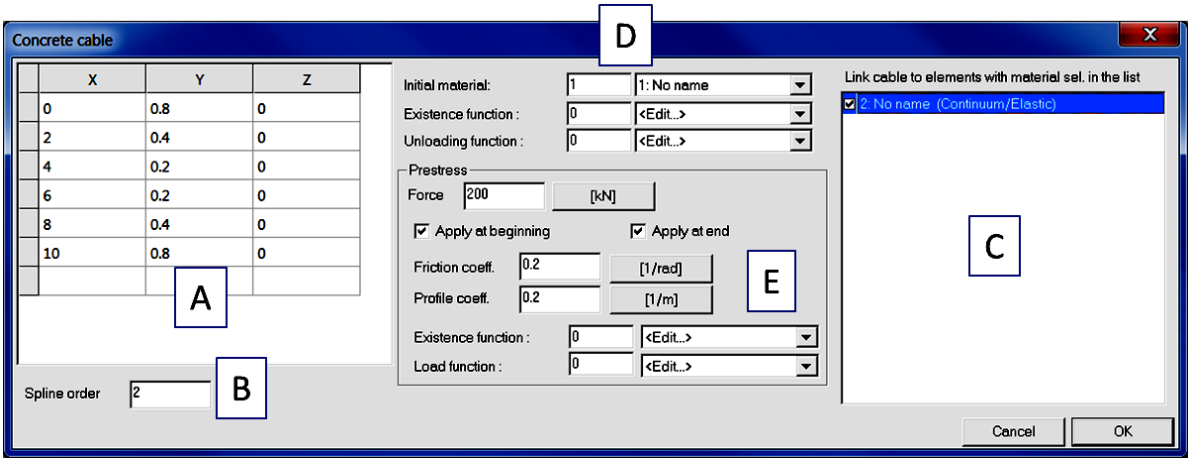
**Window 2-467: Options in menu** `Tendon in continuum/shells`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | From dialog | — | Window 2-468 |
| | By points | — | Window 2-469 |
| | Copy with translation |  | Window 2-128 |
| | Copy with rotation |  | Window 2-129 |
| | Copy by symmetry |  | Window 2-130 |
| Select | All | — | Window 2-470 |
| | Box with 2 nodes |  | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With material | — | Window 2-470 |
| | With existence function | — | Window 2-470 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-471 |
| | Move |  | Window 2-321 |
| | Rotate |  | Window 2-322 |

Window 2-467

**Window 2-468:** `Tendon in continuum/shells:Create:From dialog`

A prestressed tendon is generated based on a given set of fit points put in list (A) of the dialog. Tendon passes exactly through all these points. Trace of the tendon is designed with aid of the 2-nd or 3-rd order (most often) splines. Therefore number of fit points must be larger than the spline order plus one. Spline order is defined int the edit **Spline order** [ ]. Material id, existence and unloading functions are defined in sector (D) of the dialog. Whole set of the data concerning prestress is given is sector (E) of the dialog. Prestress data consists of the value (in terms of force), two flags defining the way it is realized (from one left/right endpoint or from both endpoints), friction $\mu$ and profile $k$ coefficients (these coefficients influence loss of prestress along tendon length) but also the load function and existence function. The latter one defines time period in which prestress is equal to the assumed force value at one or two endpoints. After real FE mesh generation tendon is converted to the set of truss elements (by an exact intersection with the FE mesh) that are linked to the background set of continuum/shell elements via `Nodal link` option (this is done automatically). Selecting background elements, in which tendon is embedded, is defined by means of selected materials set in the list (C).




Window 2-468

**Window 2-469:** `Tendon in continuum/shells:Create:By points`

A prestressed tendon is generated by picking set of fit points (cable passes through these points). This graphical operation after clicking few points can be ended up by clicking right mouse cursor and then by selecting menu item `Enter`. The remaining part of the tendon definition is the same as in option `From dialog` (see Window 2-468). In 2D one may pick any point on the screen once the snap option is set OFF but in 3D tendon fit points will always be snapped to the grid, construction lines or other characteristic points.

Window 2-469

**Window 2-470:** `Tendon in continuum/shells`**: some options in menu** `Select`

| Option | Description |
|---|---|
| `All` | Select all visible concrete cables |
| `In box 2 nodes/points` | Select all visible tendons inside box selected by 2 nodes/points |
| `In zoom box` | Select all visible tendons inside rectangular zoom |
| `In zoom circle` | Select all visible tendons inside circular zoom |
| `Inside contour` | Select all visible tendons inside contour |
| `With material` | Select all visible tendons with given material |
| `With existence function` | Select all visible tendons with a given existence functions |

Window 2-470

**Window 2-471:** `Tendon in continuum/shells:Update:Parameters`

This option allows the user to modify setting for the selected tendon (see Win.2-468).

Window 2-471

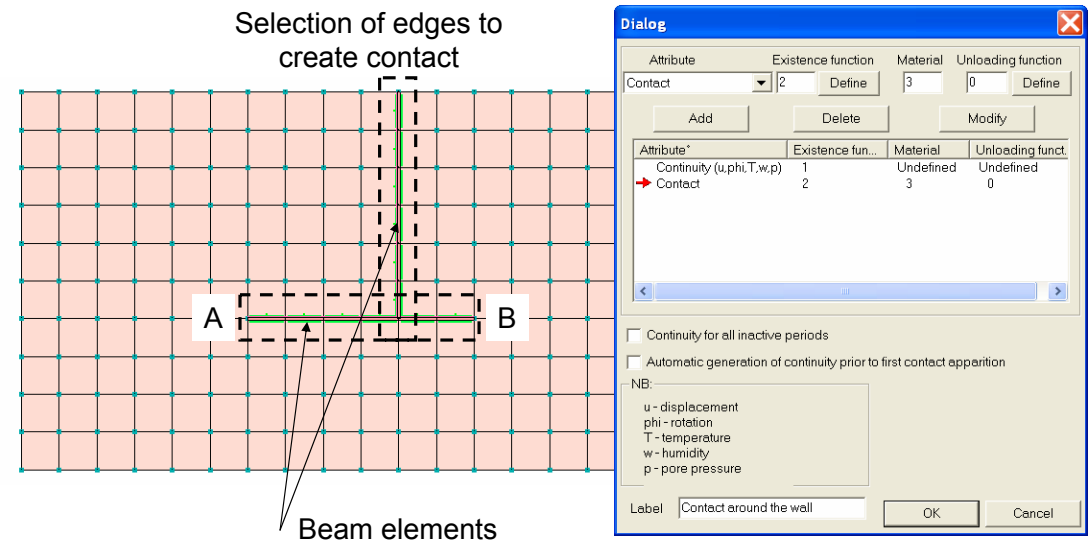
## 2.3.1.3.12 INTERFACE (SMALL DEFORMATION)

**Window 2-472: Options in menu** `Interface`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On Cont.elem.edges (2D) | — | Window 2-473 |
| | On Cont.elem.face(s)(3D) | — | Window 2-475 |
| | On Structural elements | — | Window 2-474 ,2-477 |
| Select | All | — | Window 2-478 |
| | Box with 2 nodes |  | Window 2-316 |
| | With material | — | Window 2-479 |
| | With existence function | — | Window 2-479 |
| | With unloading function | — | Window 2-479 |
| | With label | — | Window 2-479 |
| | With attribute | — | Window 2-479 |
| | With initial gap | — | Window 2-480 |
| | On selected edges (2D) | — | Window 2-481 |
| | On selected faces (3D) | — | Window 2-482 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-483 |
| | Parameters in front | — | Window 2-484 |
| | Scale | — | Window 2-485 |
| | Show direction | — | Window 2-486 |
| | Hide direction | — | Window 2-487 |
| | Standard view | — | Window 2-488 |
| | Control view | — | Window 2-489 |
| | Select singular interface nodes | — | Window 2-490 |
| | Link interface node(s) | — | Window 2-491 |
| | Link selected shell node(s) | — | Window 2-492 |
| | Delete links | | Window 2-493 |
| | Delete all links | — | Window 2-494 |
| | Exclude elements | — | Window 2-495 |
| | Disable excluded elements | — | Window 2-496 |
| | Show excluded elements | — | Window 2-497 |
| Initial gap | Set/update on sel. elements | — | Window 2-498 |
| | Delete all | — | Window 2-499 |
| | Delete | — | Window 2-500 |

**Window 2-473:** `Interface:On edges`

New 2D interface elements are created by selecting edges of continuum elements in the existing real mesh (see example).




A, B – singular points

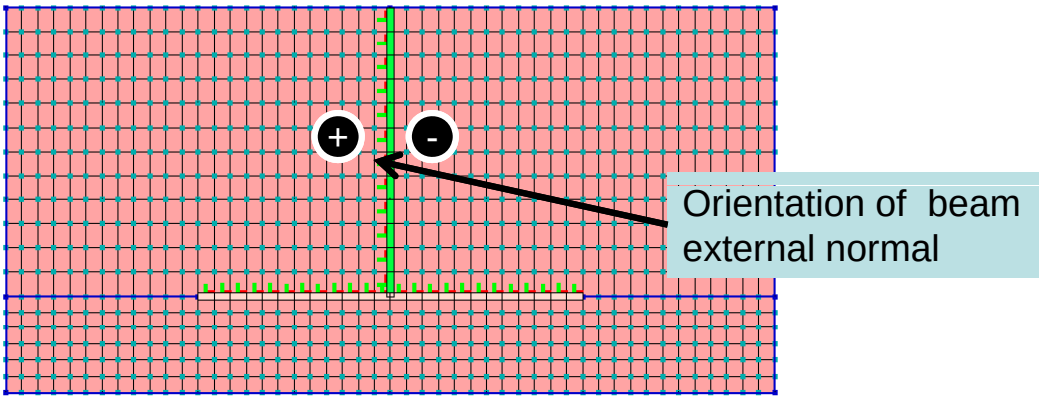
**Remarks**:

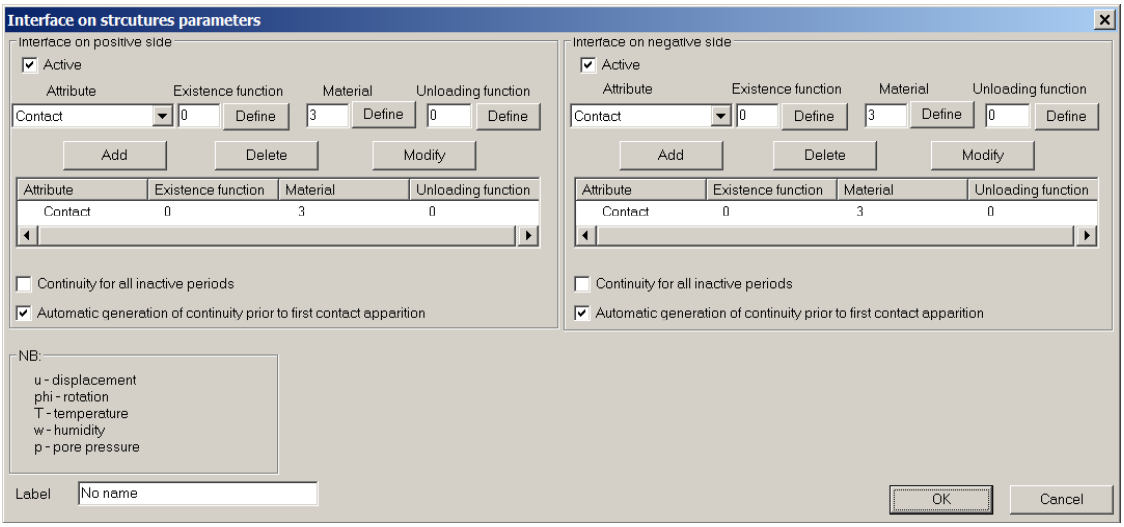
1. Note that beam elements will always be separated from the continuum at points A and B (as contact is ended in the inside of the domain); to enforce continuity in these points use option `Update:Link nodes`
2. In cases like in this example it is recommended to create contact elements by parts to be able to apply a distinct label and to be able to apply different existence function, unloading function and material ID, otherwise contact elements have to be selected from selected edges (2D) and then updated through `Update:Parameters`
3. Contact elements, disconnecting the real mesh along indicated segments, are generated during generation of the input file for calculation module (*.DAT file)
4. During contact generation user may set type of the contact behavior and an additional data in the dialog box (see Window 2-483) for details)

Window 2-473

**Window 2-474:** `Interface:On Structural elements (2D)`

New 2D interface elements are created by selecting structural elements (beams, membranes) in the existing real mesh (see example). In this definition contact interface may be created on any side of the structural element being in contact with continuum. It is important to carefuly set contact properties concerning orientation of the structural element local coordinate system. The positive interface is located in contact zone continuum-top beam/membrane fibers.



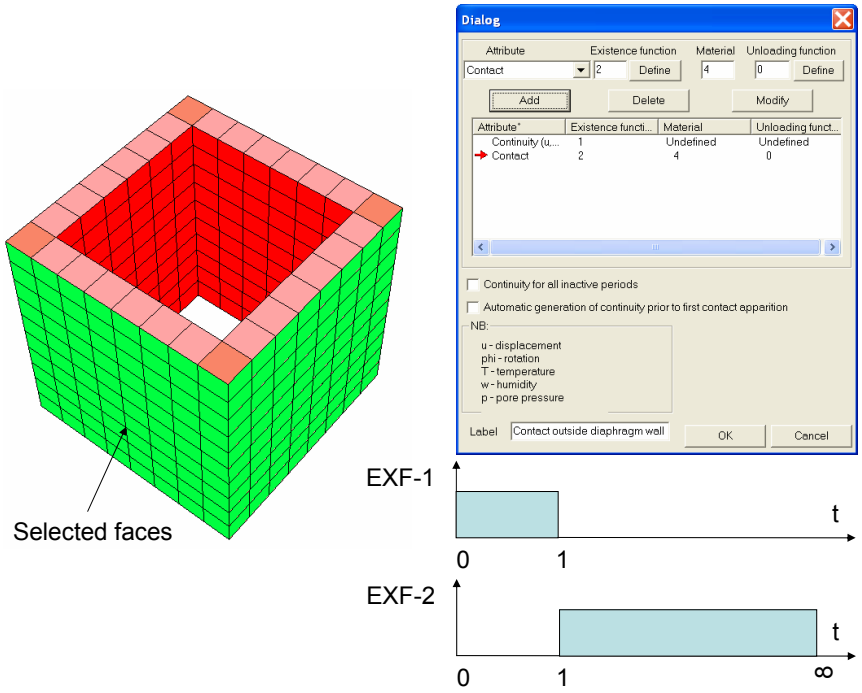

**Remarks**:

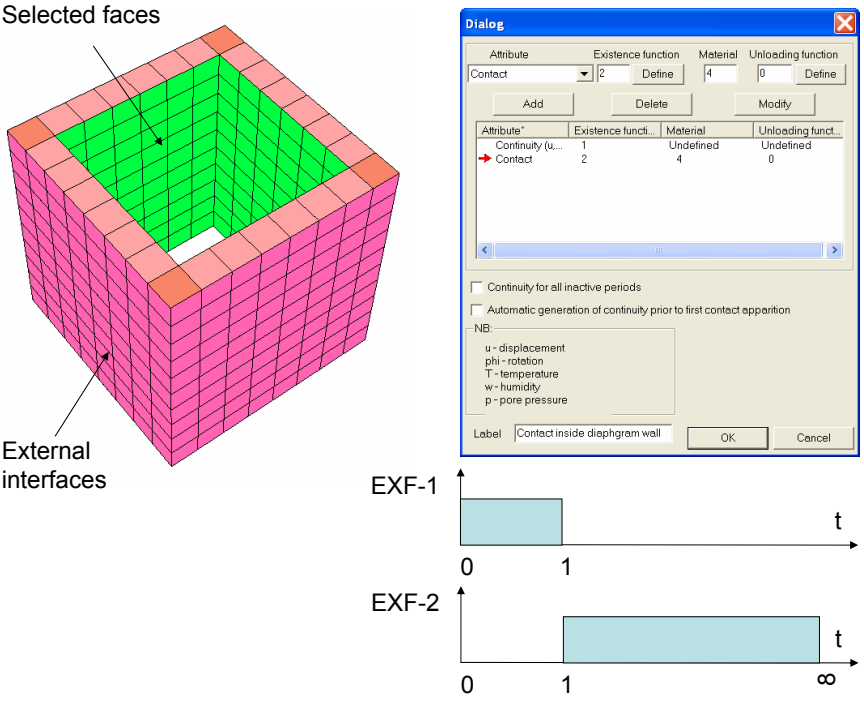
1. This definition may be useful when modeling reinforced soils using geotextiles; in such cases this definition is optimal
2. One may create or not the interface on positive/negative side of the structural element
3. Remarks in Win.(2-473) are valid for this option

Window 2-474

## Window 2-475: Interface:On faces

New 3D interface elements are created by selecting faces in the existing real mesh (see example).

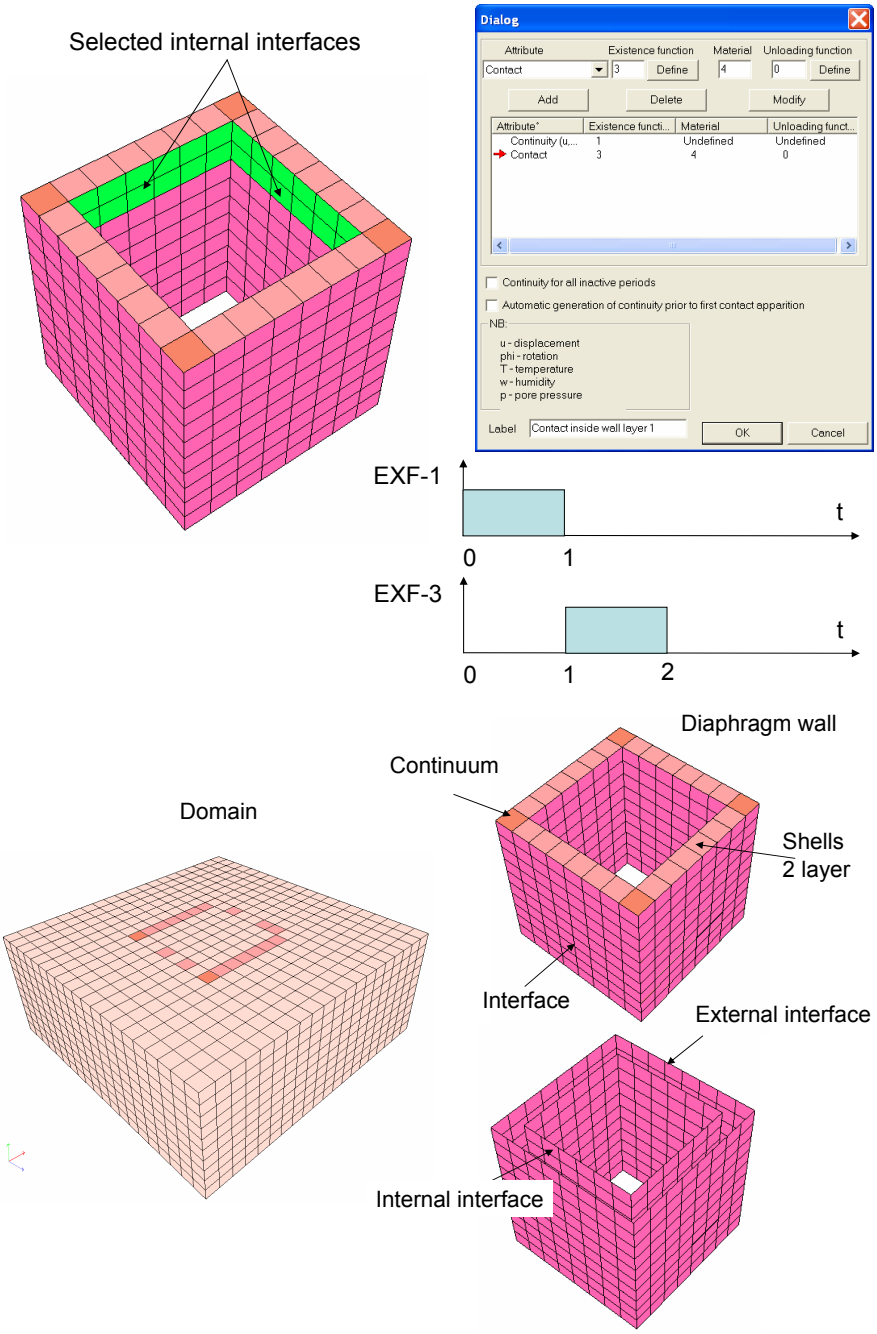



**Remark**:See Window 2-476 for continuation

## Window 2-476: Interface:On faces (continuation)

New 3D interface elements are created by selecting faces in the existing real mesh (see example).

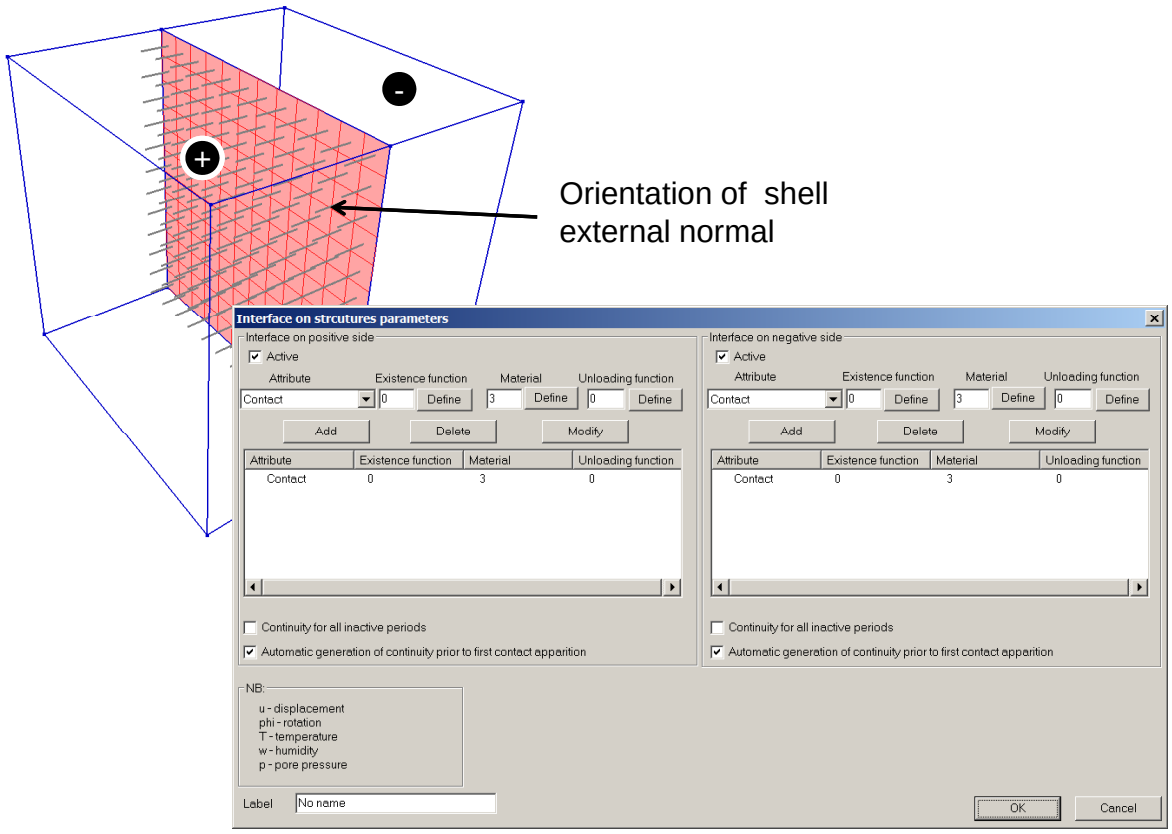



**Remarks**:

1. In case when diaphragm wall is modeled with aid of one layer shell elements the bottom edge of the wall will not be attached to the continuum and for that reason all these nodes must be attached (if needed) through option `Update:Link 2 nodes`
2. During contact generation user may set type of the contact behavior and an additional data in the dialog box (see Window 2-483) for details)

**Window 2-477:** `Interface:On Structural elements (3D)`

New 3D interface elements are created by selecting structural elements (shells, membranes) in the existing real mesh (see example). In this definition contact interface may be created on any side of the structural element being in contact with continuum. It is important to carefuly set contact properties concerning orientation of the structural element external normal (use option `Direction` to modify that orientation if needed). The interface on the positive side is located in contact zone continuum-top shell/membrane fibers.




**Remarks**:

1. This definition is optimal when modeling reinforced soils using geotextiles
2. One may create or not the interface on positive/negative side of the structural element
3. Remarks in Win.(2-475) are valid for this option

Window 2-477

**Window 2-478:** `Interface:Select:All`

Using this option user may select all contact interfaces.

Window 2-478

**Window 2-479:** `Interface:Select: with material, existence/unloading function`

Using this option user may select contact interfaces with a given material, existence function, unloading function ID or a given attribute or label. In case of a label or an attribute one may select a single item from combo-box.

Window 2-479

**Window 2-480:** `Interface:Select: with initial gap`

Using this option user may select contact interfaces with an initial nonzero gap (initial opening). Interface elements can be selected by a distinct value of the initial gap shown in the combo box.

Window 2-480

**Window 2-481:** `Interface:Select:On selected edges`

Using this option user may select 2D contact elements which are overlapping with selected edges.

Window 2-481

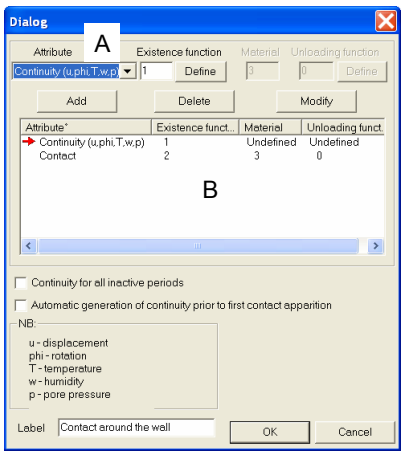
**Window 2-482:** `Interface:Select:On selected faces`

Using this option user may select 3D contact elements which are overlapping with selected faces.

Window 2-482

## Window 2-483: Interface:Update:Parameters

This dialog box is designed to define on how the contact interface should be handled in certain time periods. It may enforce full continuity condition for all types of degree of freedom (DOF) (displacement, pore pressure, temperature, humidity) on both segments of the interface element, full continuity in terms of displacement, humidity, temperature DOF but discontinuity in terms of the pressure and finally true contact behavior. In the later case decision on how to handle non-kinematic DOF (pressure, temperature, humidity) in the interface can be set at the material level (switching ON/OFF and editing groups of parameters: ☒ **Flow**, ☒ **Heat** and ☒ **Humidity**).




**Remarks**:

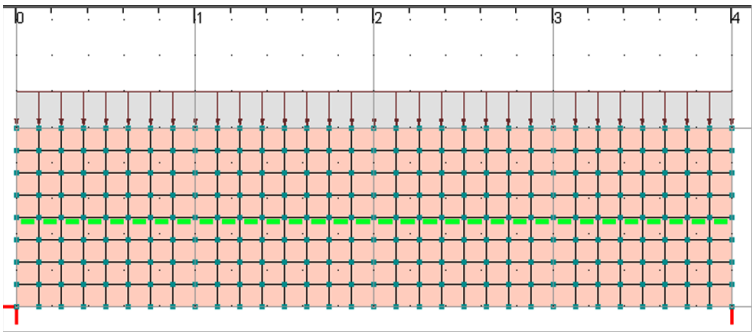
1. The three aforementioned contact type behaviors can be selected from combo-box (A)
2. For contact type: `Continuity...` only existence function is meaningful
3. The existence function, unloading function and material ID can be set in the edit fields **Existence function** [ ], **Unloading function** [ ] **Material** [ ] or selected from list of predefined ones by pressing button **Define** placed next to the appropriate edit fields
4. Modification of the list of different contact type behavior (B) can be made with aid of three buttons **Add**, **Delete** and **Modify**
5. To easily select certain contact elements it is recommended to put a distinct label in the edit field **Label** [ ]
6. The ☒ **Continuity for all inactive periods** check-box set to ON will enforce automatic generation of contact elements with full continuity attribute in all inactive periods of true contact behavior;
7. The ☒ **Automatic generation of continuity prior to first contact apparition** check-box set to ON will enforce automatic generation of contact elements with full continuity attribute only in the first inactive period for true contact behavior; using this option user does not need to add to the list (B) a contact with attribute full continuity as it will be automatically generated (if true contact appears after time $t = 0$)

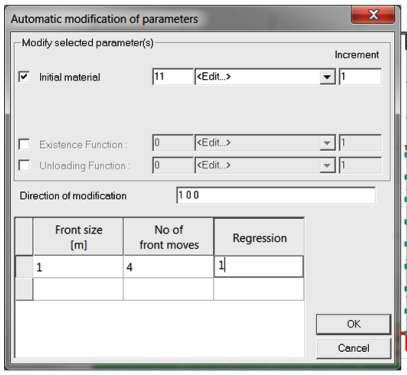
Window 2-483

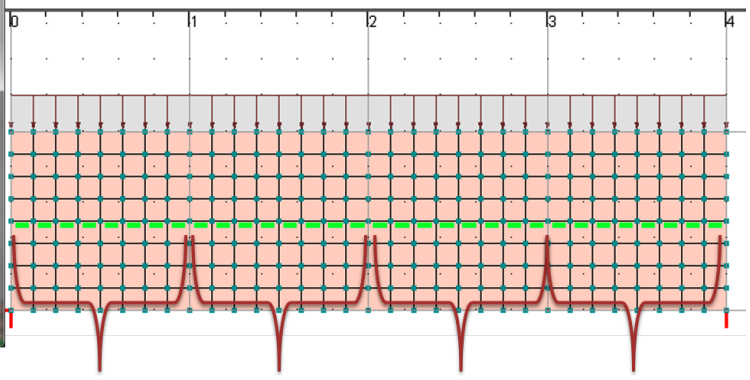
## Window 2-484: Interface:Update:Parameters in front

By using this option user may set spatial distribution of the initial materials, existence functions and unloading functions for the selected set of elements (see example shown below).

Front size is understood as a size of a zoom that is moving in a given direction as many times as declared.




Mat=11 Mat=12 Mat=13 Mat=14

Window 2-484

**Window 2-485:** `Interface:Update:Scale`

Using this option user may enlarge or diminish size of the external normal vectors and opening of the interface in the `Control view`.

Window 2-485

**Window 2-486:** `Interface:Update:Show direction`

Using this option user visualize direction of external normal vectors to faces on which contact elements are created.

Window 2-486

**Window 2-487:** `Interface:Update:Hide direction`

Using this option user may hide external normal vectors to faces on which contact elements are created.

Window 2-487

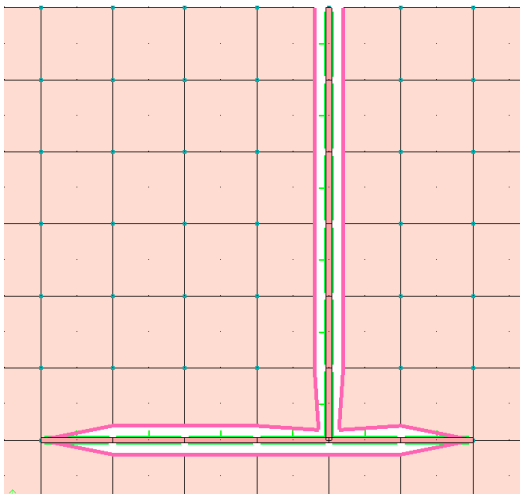
**Window 2-488:** `Interface:Update:Standard view`

With this option standard visualization of contact elements is activated.

Window 2-488

**Window 2-489:** `Interface:Update:Control view`

With this option special visualization of contact elements is activated. It makes easier recognition whether contact is properly set (see example below).



Window 2-489

**Window 2-490:** `Interface:Update:Select singular interface nodes`

In case when structural elements are in contact with soil and their endpoints are located inside of the soil domain (bottom of the diaphragm wall for instance) the structural element endpoint will be disconnected from soil by default when contact interface is created. Option `Select singular interface nodes` enables one to select all nodes in which this effect is detected. To link these separated nodes use option `Link interface nodes` (Window 2-491). This situation is illustrated in the figure in Window 2-473. To enable correct modeling of the pullout test one may add hinges in beam/shell elements with a given compressive resistance but zero tensile one.

Window 2-490

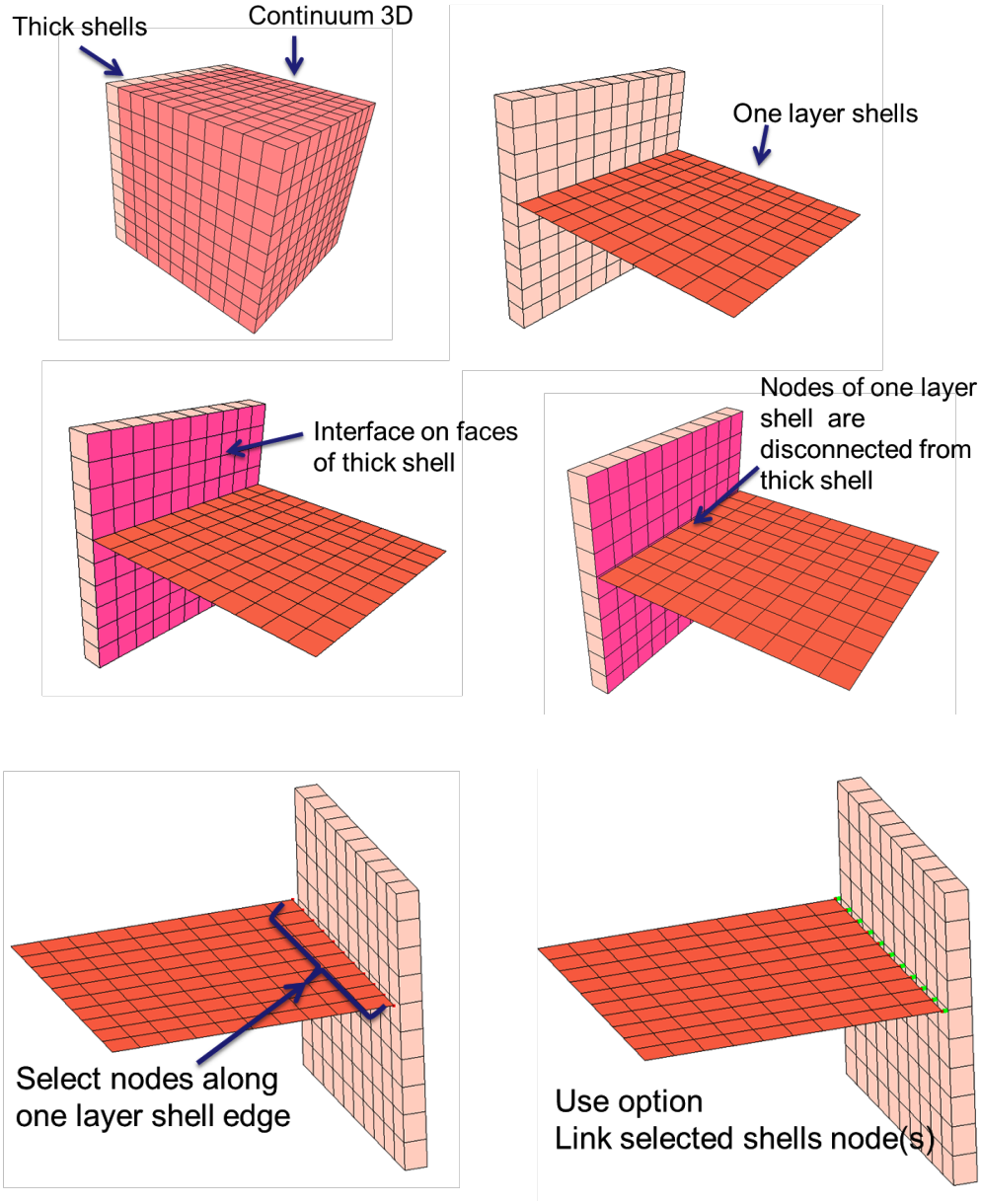
**Window 2-491:** `Interface:Update:Link interface nodes`

This option enables one to link (enforce continuity) nodes separated by contact interface. In most cases it is needed to handle the situation at points A, B as shown in the figure in Window 2-473. Detection of such singular points can be made using the option `Select singular interface nodes` (Window 2-490). Created links between two nodes are plot as light-green bars.

Window 2-491

**Window 2-492:** `Interface:Update:Link selected shell nodes`

This option is explained in the following figures. Let us assume that we want to generate contact interface between the layer of thick shells and the remaining continuum while inside the continuum a layer of thin shell elements exists. Contact interface algorithm will disconnect continnum 3D/thick shells from the thin shell elements. If we need them to be connected shell element nodes have to be selected first and then connected back to the continuum 3D/thick shells using option `Link selected shell nodes`. This option speeds up linking disconnected nodes. Such linking could also be made using option `Link interface nodes` by going from node to node. **It is important that selected nodes will be linked to visible elements (!)**.




Created links between each pair of nodes are plot as light-green bars.

Window 2-492

**Window 2-493:** `Interface:Update:Delete link`

Using this option one may cancel a single picked link of group of links at selected nodes. Hence to delete group of links selection of nodes is requested.

Window 2-493

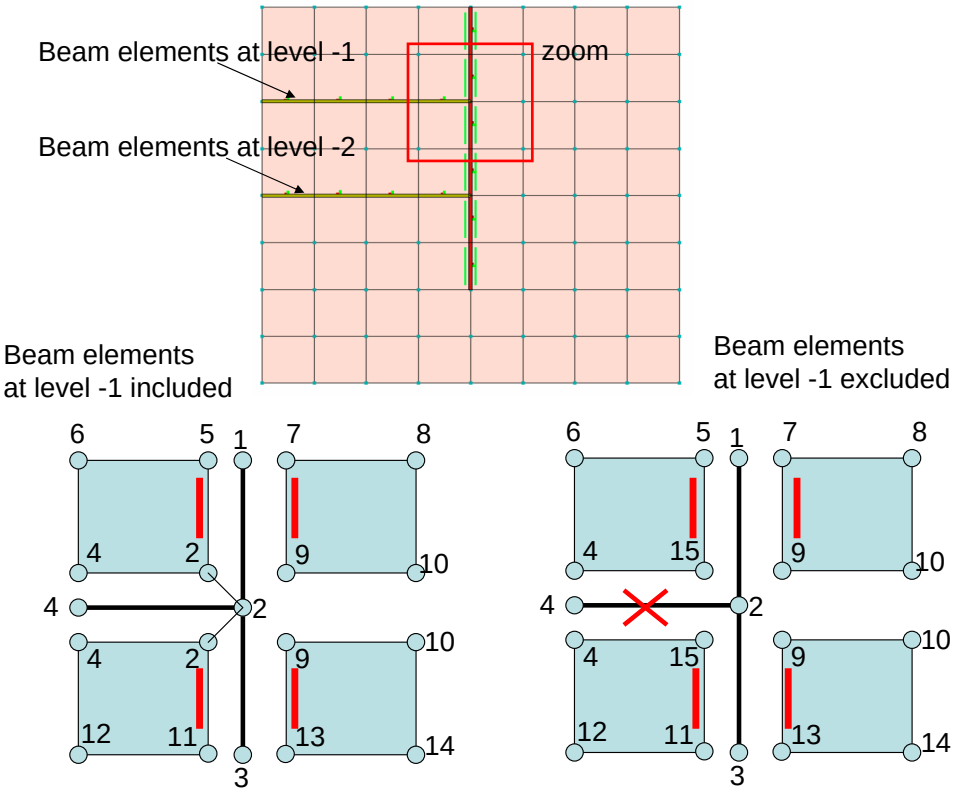
**Window 2-494:** `Interface:Update:Delete all links`

Using this option one may delete all links.

Window 2-494

**Window 2-495:** `Interface:Update:Exclude elements from interface`

When modelling diapraghm walls stiffened by partial floors one may frequently face with the problem of wrong discretization of contact interface between the wall and subsoil. The goal of the option is visualized in the two figures given below. Suppose that we have two vertical beam elements connected to the horizontal one at node 2 on the left side of the model. If we declare the contact interface between subsoil and the two vertical beam elements (by selecting vertical continuum element faces) then the standard contact algorithm will not split the mesh at node 2. To get the correct mesh decomposition one has to select the horizontal beam element and set it as excluded from the interface. With this setting the contact algorithm will not recognize the horizontal beam as being adjacent to node 2 and the two vertical beams will be disconnected from subsoil adjacent to node 2 on the left side of the model. Same problem appears in the 3D when wall and partial floors are modeled using shell elements.




Window 2-495

**Window 2-496:** `Interface:Update:Disable excluded elements`

This option deactivates selection of elements to be excluded in contact interface generation.

Window 2-496

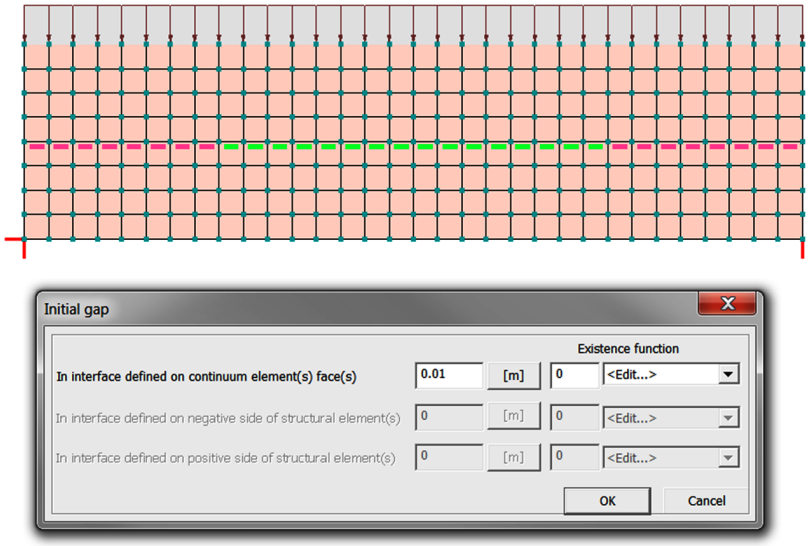
**Window 2-497:** `Interface:Update:Show excluded elements`

This option allows to select elements to be excluded from generation of contact interface.

Window 2-497

**Window 2-498:** `Interface:Initial gap:Set/update on sel. elements`

This option enables one to set an initial opening in the contact interface. This option should be used with care as it may generate instabilities when the interface is open. In case of soil-structure interaction problems to big initial opening may also result in generating spurious slip lines in soil which may preclude global convergence. Existence function is needed to distinguish to which interface elements the initial gap is to be applied (due to excavation/construction steps).




Window 2-498

**Window 2-499:** `Interface:Initial gap:Delete all`

This option enables one to delete all initial gaps defined in any interface element.

Window 2-499

**Window 2-500:** `Interface:Initial gap:Set/Delete`

This option enables one to delete an initial gap in a single selected interface element or group of selected elements.

Window 2-500

## 2.3.1.3.13 INTERFACE (LARGE DEFORMATION)

Large deformation contact is based on node-to-segment approach in which node is called as `Contactor` while the segment belongs to the `Target surface`. To generate large deformation contact interfaces mesh must be disconnected along the interface line (2D) or surface (3D).

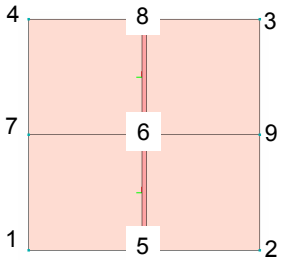
**Window 2-501: Options in menu** `Interface (large deformation)`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Create new nodes on sel. nodes` | — | Window 2-502 |
| | `Contactors on sel. nodes` | — | Window 2-503 |
| | `Target surface on sel. edges/faces` | — | Window 2-504 |
| | `Interface` | — | Window 2-505 |
| `Select` | `With material` | — | Window 2-506 |
| | `With existence function` | — | Window 2-506 |
| | `With unloading function` | — | Window 2-506 |
| | `With label` | — | Window 2-506 |
| | `With attribute` | — | Window 2-506 |
| | `Contactors with label` | — | Window 2-507 |
| | `Target surface with label` | — | Window 2-508 |
| `Delete` | `Delete Contactors/Target surfaces` | — | Window 2-509 |

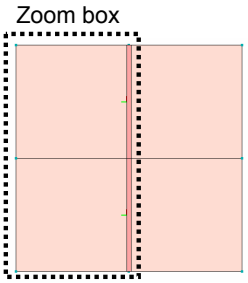
Window 2-501

**Window 2-502:** `Interface (large def.):Create new nodes on sel. nodes`

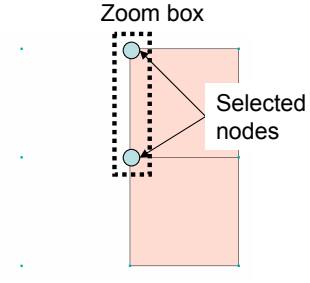
Using this option one may create new nodes on selected ones but exclusively in currently visible, elements, adjacent to the selected nodes. This action is explained by the figure below.



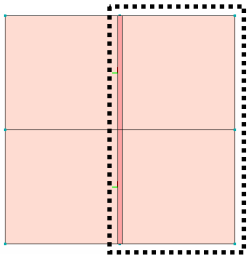

(a) Standard mesh



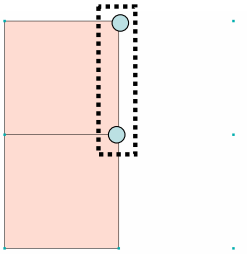

(b) Select continuum on the left + beams and then temporarily hide them



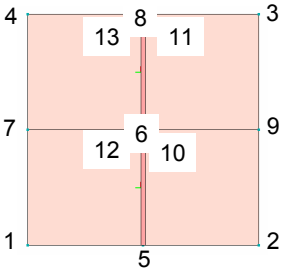

(c) Create new nodes on selected ones in visible elements



(d) Select continuum on the right + beams and then temporarily hide them



(e) Create new nodes on selected ones in visible elements




(f) Final connectivity

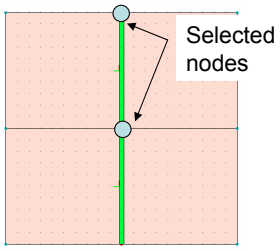
**Remarks**:

1. To avoid unpredictable results elements in which connectivity is to be preserved must be temporarily hidden
2. Same strategy is used in 3D models

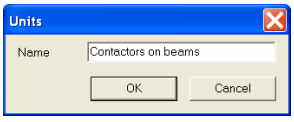
Window 2-502

**Window 2-503:** `Interface (large def.):Create contactors on sel. nodes`

Using this option selected nodes can be set as contactors for node-to-segment contact approach.




(a) Select beams first and then nodes from selected beams




(b) Create contactors (with label) on selected nodes

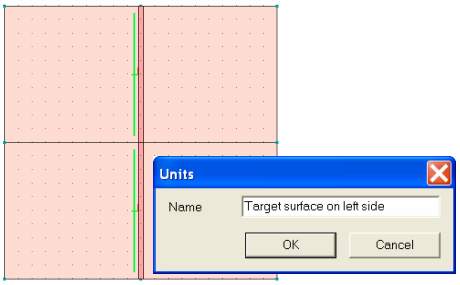
**Remark**:

In the considered case i.e. when contact is to be generated on both sides of the wall, beam nodes can exclusively be set as contactors

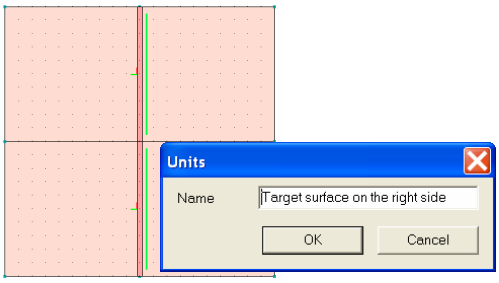
Window 2-503

**Window 2-504:** `Interface (large def.):Create target on sel. edges/faces`

Using this option selected edges(2D)/faces(3D) can be set as target surface for node-to-segment contact approach.




(a) Create target surface on the left side of the wall (on selected edges)




(b) Create target surface on the right side of the wall (on selected edges)

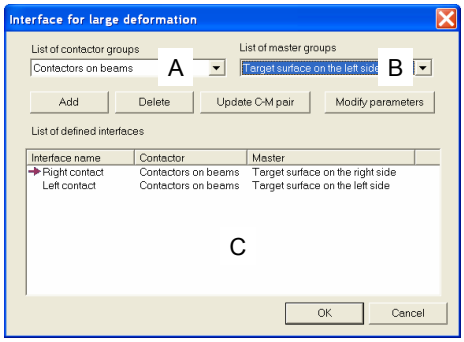
**Remark**:

In the considered case i.e. when contact is to be generated on both sides of the wall, adjacent to the wall, continuum element edges can exclusively be set as target surfaces

Window 2-504

**Window 2-505:** `Interface (large def.):Interface`

This option defines interfaces by combining pairs of contactors and target surfaces in the following dialog box.




**Remarks**:

1. Combo boxes (A) and (B) contain predefined sets of contactors and target surfaces accordingly
2. New established interface can be added to the list (C) of interfaces through button **ADD**
3. Erroneous interface definition can be removed from list (C) with button **Delete** or modified with button **Update**
4. Setting type of the contact behavior (same as for small deformation contact) can be activated with button **Modify Parameters** (see Window 2-483) for detailed description)
5. Interfaces do not need to be redefined after mesh refinement

Window 2-505

**Window 2-506:** `Interface (large def.):Select:Material, exist./unl.f., ...`

This option allows to visualize certain interfaces identified by a given material, existence function, unloading function ID or a given attribute or label. Selected interface(s) are shown by highlighting target surface edges(2D)/faces(3D) and contactor nodes.

Window 2-506

**Window 2-507:** `Interface (large def.):Select:Contactors with label`

This option selects contactor nodes with a given label.

Window 2-507

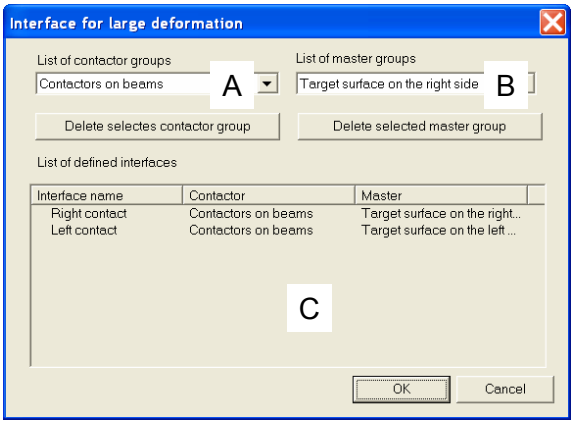
**Window 2-508:** `Interface (large def.):Select:Target surface with label`

This option selects target surface (by highlighting target surface edges(2D)/faces(3D)) with a given label.

Window 2-508

**Window 2-509:** `Interface (large def.):Delete:Contactors/Target surface`

This option allows to delete some sets of contactor nodes or target surfaces through the following dialog box.




1. Combo boxes (A) and (B) contain predefined sets of contactors and target surfaces accordingly
2. By deleting selected contactors set (from (A)) all the interfaces with this contactors set are cancelled from list (C)
3. By deleting selected target surface (from (B)) all the interfaces with this target surface are cancelled from list (C)

Window 2-509

## 2.3.1.3.14 SEEPAGE

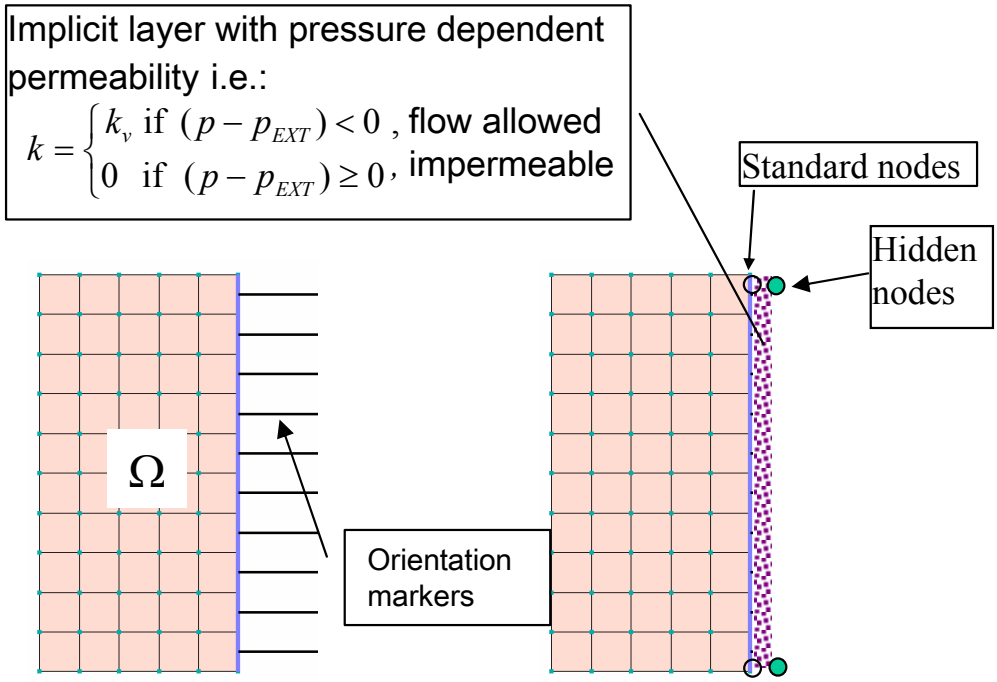
**Window 2-510: Options in menu** Seepage

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes | — | Window 2-512 |
| | On edge(s) | — | Window 2-511 |
| | On Exc./Constr. front | — | Window 2-515 |
| | 3 nodes | — | Window 2-514 |
| | 4 nodes | — | Window 2-514 |
| | On face(s) | — | Window 2-513 |
| Select | All | — | Window 2-516 |
| | Box with 2 nodes |  | Window 2-316 |
| | With material | — | Window 2-517 |
| | With existence function | — | Window 2-517 |
| | With unloading function | — | Window 2-517 |
| | With label | — | Window 2-517 |
| | On selected edges (2D) | — | Window 2-518 |
| | On selected faces (3D) | — | Window 2-519 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-520 |
| Direction | Show direction | — | Window 2-521 |
| | Hide direction | — | Window 2-522 |
| | Scale | — | Window 2-523 |

Window 2-510

**Window 2-511:** `Seepage:On edges`

New 2D seepage elements are created by selecting continuum element edges in the existing real mesh.




**Remarks**:

1. Seepage elements are meaningful for `Flow` and `Deformation+Flow` analyses
2. Seepage elements are to be generated on edges/faces of continuum elements
3. These elements are frequently used to simulate drains and to handle pressure/flux type of the boundary condition in zones where free water surface intersects with domain boundary (see example); the penalty factor $k_v$ is automatically set by the calculation module and can be modified by the user at the material level
4. An additional layer of external nodes is created automatically by the program and all pressure boundary conditions imposed on the seepage boundary are transferred to these nodes
5. If seepage elements are conforming with the adjacent continuum elements then mesh refinement in the continuum will cause seepage refinement automatically
6. Direction marker helps to identify on which continuum element face given seepage element in created; this is very important during stage construction analyses

Window 2-511

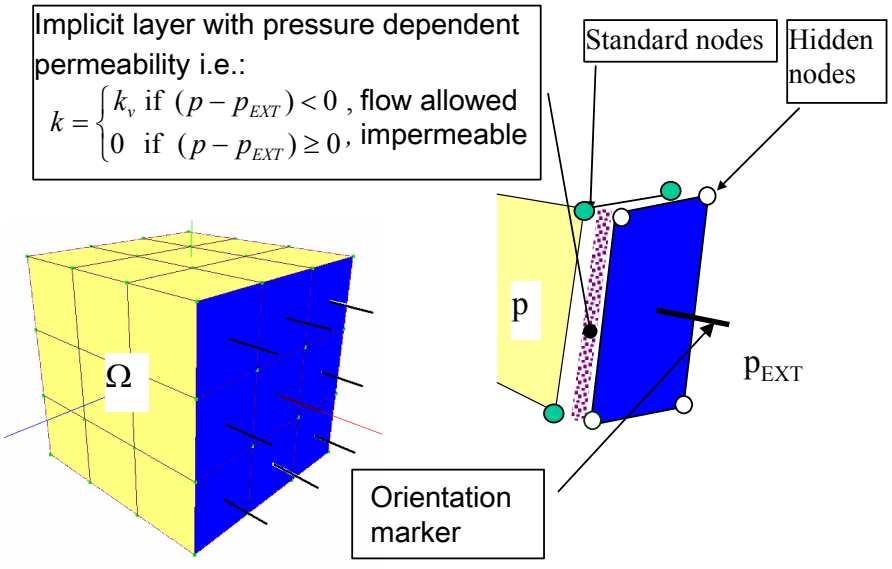
**Window 2-512:** `Seepage:2 nodes`

New 2D seepage elements are created along edges belonging to the line section defined by the 2 picked nodes in the existing real mesh. This option, except the definition, is equivalent to the one described in Window 2-511.

Window 2-512

**Window 2-513:** `Seepage:On faces`

New 3D seepage elements are created by selecting faces in the existing real mesh (see example below). See also remarks given in Win.(2-511).




Window 2-513

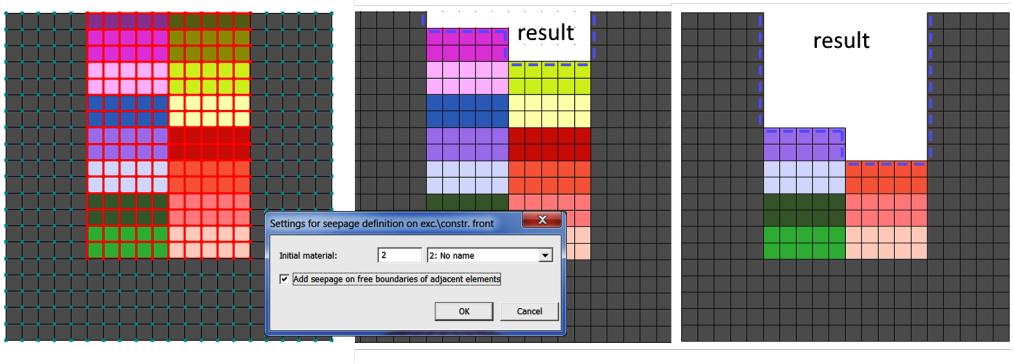
**Window 2-514:** `Seepage:3 nodes, 4 nodes`

New 3D seepage elements are created on faces which belong to the 3D surface defined by the 3 or 4 picked nodes in the existing real mesh (see example). Once faces are detected the generation procedure follows the steps described in Win.(2-513).

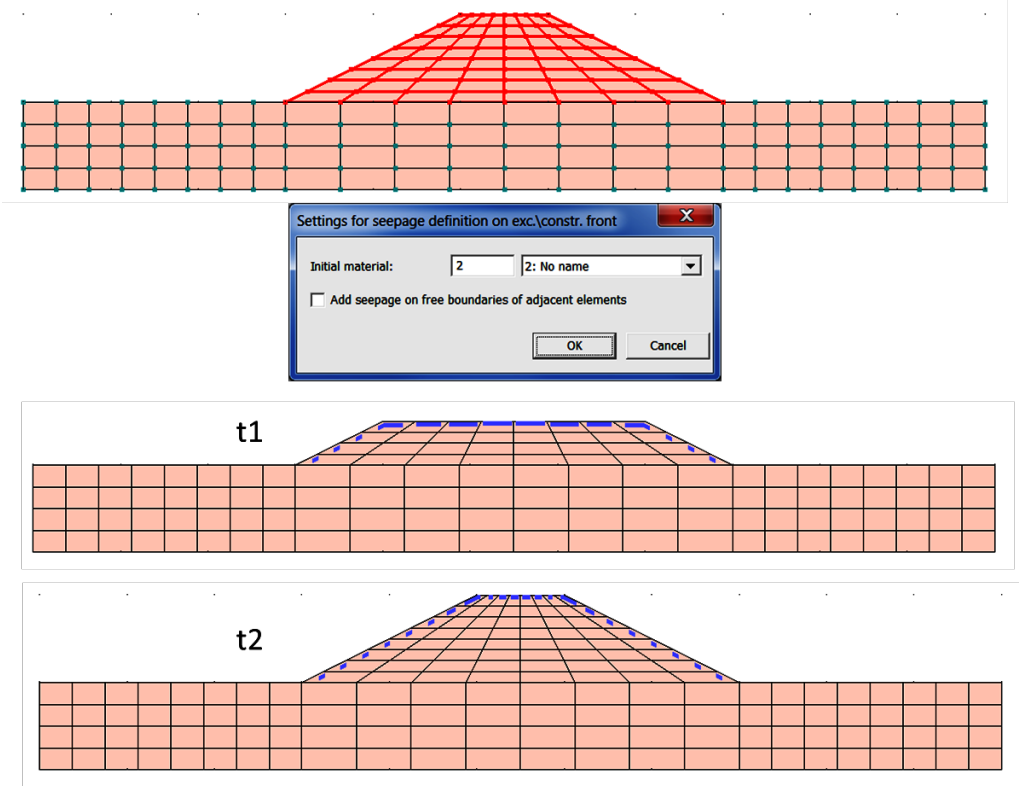
Window 2-514

**Window 2-515:** Seepage:On Exc./Constr. front

This option allows the user to generate automatically seepage elements on external edges/faces (of continuum elements) that form current free boundary during excavation or construction stages. Example of the excavation stage and induced generating seepage elements on the free boundary is shown in the figure below. Another example concerns construction of an embankment. In this case it may be needed to place seepage elements on the current free boundary. In both cases one may select elements that will be excavated or constructed and run the procedure. It is also possible to select all continuum elements and perform this operation for the whole mesh in one step. It may happen that some elements will have to be deleted therefore this operation must be done using standard delete option from menu Seepage. Before running this operation existence functions must be set for the continuum elements first. This option works in the same manner in the 3D cases. Seepage elements on the left/right/top boundaries in these models must be generated using standard tools.




Example of excavation stage




Example of construction stage

**Window 2-516:** `Seepage:Select:All`

Using this option user may select all visible seepage elements.

Window 2-516

**Window 2-517:** `Seepage:Select:With material, existence f., label`

Using this option user may select seepage elements with a given material or existence function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-517

**Window 2-518:** `Seepage:Select:On selected edges`

Using this option user may select 2D seepage elements which are overlapping with selected edges.

Window 2-518

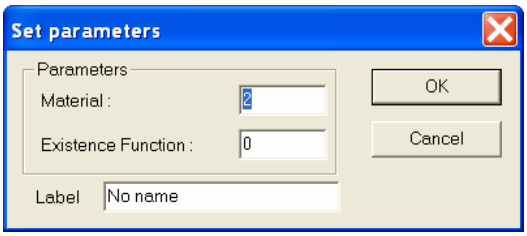
**Window 2-519:** `Seepage:Select:On selected faces`

Using this option user may select 3D seepage elements which are overlapping with selected faces.

Window 2-519

**Window 2-520:** `Seepage:Update:Parameters`

Using this option user can modify the material and existence function ID, and label.




Window 2-520

**Window 2-521:** `Seepage:Direction:Show`

This option activates visualization of direction marker which helps to identify on which continuum element face given seepage element is created.

Window 2-521

**Window 2-522:** `Seepage:Direction:Hide`

This option cancels visualization of seepage direction marker.

Window 2-522

**Window 2-523:** `Seepage:Direction:Scale`

Using this option user may enlarge or diminish size of the direction markers.

Window 2-523

## 2.3.1.3.15 CONVECTION/RADIATION

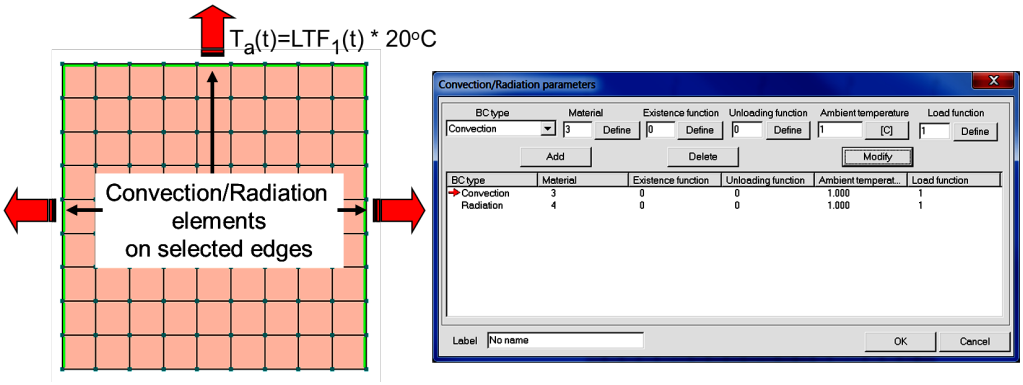
**Window 2-524: Options in menu** `Convection`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 2 nodes | — | Window 2-526 |
| | On edge(s) | — | Window 2-525 |
| | 3 nodes | — | Window 2-528 |
| | 4 nodes | — | Window 2-528 |
| | On face(s) | — | Window 2-527 |
| | On Constr./Exc. front | — | Window 2-529 |
| Select | All | — | Window 2-530 |
| | Box with 2 nodes |  | Window 2-316 |
| | With material | — | Window 2-531 |
| | With existence function | — | Window 2-531 |
| | With unloading function | — | Window 2-531 |
| | With label | — | Window 2-531 |
| | On selected edges (2D) | — | Window 2-532 |
| | On selected faces (3D) | — | Window 2-533 |
| | In zoom box |  | Window 2-317 |
| | In zoom circle |  | Window 2-318 |
| | Inside contour |  | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-534 |

Window 2-524

## Window 2-525: Convection/Radiation:On edges

New 2D convection/radiation elements are created by selecting continuum element edges in the existing real mesh.




**Remarks**:

1. Convection/Radiation elements are meaningful for Heat transfer analyses
2. Convection/Radiation elements can be generated on continuum element edges (2D)/faces (3D)
3. These elements are used to simulate convection type of the boundary condition (BC) and/or radiation ones (in case of analysis of fire conditions); convection/radiation elements remove the requirement of imposing temperature BC at node (at least one)
4. The external temperature is defined through ambient temperature which may vary in time
5. If convection/radiation elements are conforming with the adjacent continuum elements then mesh refinement in the continuum will cause convection/radiation refinement automatically
6. For detailed explanation on how to set parameters describing convection/radiation elements see Window 2-534

Window 2-525

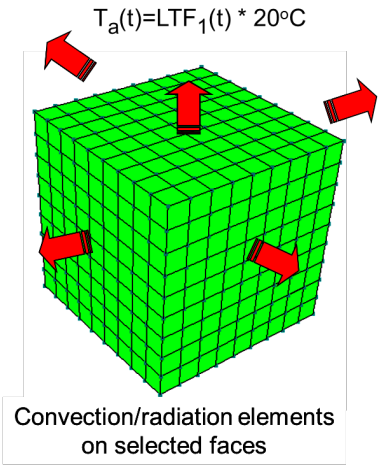
## Window 2-526: Convection/Radiation:2 nodes

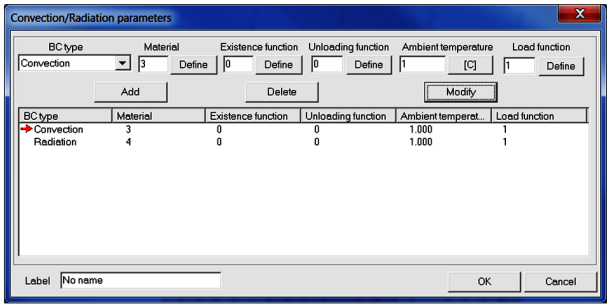
New 2D convection/radiation elements are created along edges belonging to the line section defined by the 2 picked nodes in the existing real mesh. This option, except the definition, is equivalent to the one described in Window 2-525.

Window 2-526

**Window 2-527:** `Convection/Radiation:On faces`

New 3D convection/radiation elements are created by selecting continuum element faces in the existing real mesh (see example below). See also remarks given in Window 2-525.




Window 2-527

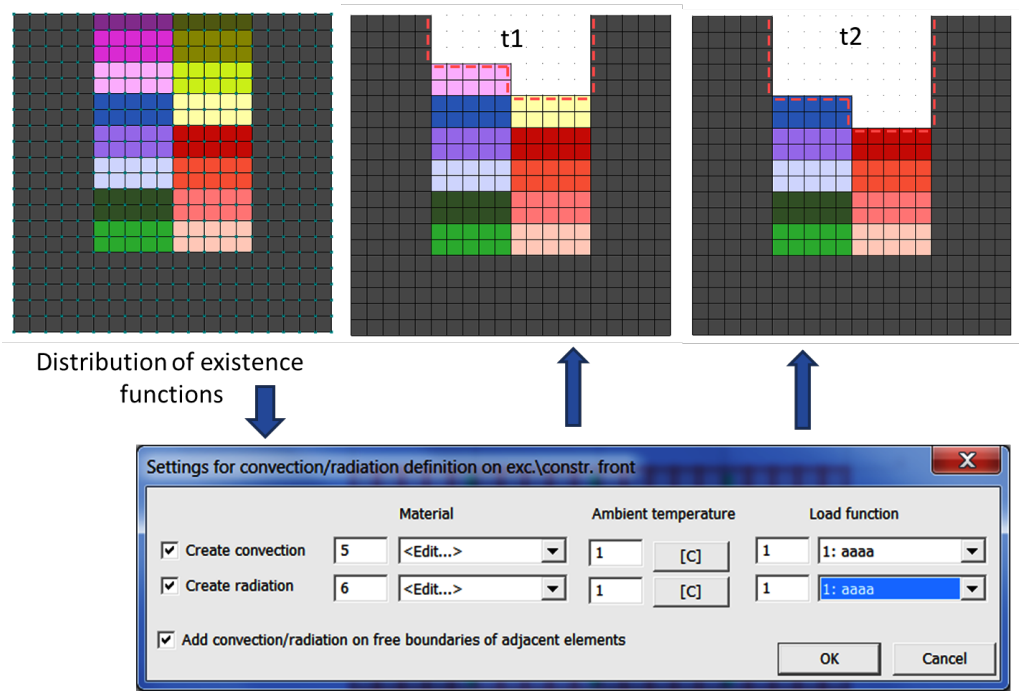
**Window 2-528:** `Convection/Radiation:3 nodes, 4 nodes`

New 3D convection/radiation elements are created on faces which belong to the 3D surface defined by the 3 or 4 picked nodes in the existing real mesh (see example). Once faces are detected the generation procedure follows the steps described in Window 2-527.

Window 2-528

**Window 2-529:** `Convection/Radiation:On Exc./Constr. front`

This option allows the user to generate automatically convection and/or radiation elements on external edges/faces (of continuum elements) that form current free boundary during excavation or construction stages. Example of the excavation stage and induced generating convection/radiation elements on the free boundary is shown in the figure below. Convection/radiation elements on the left/right/top boundaries, in this model, (if needed) must be generated using standard tools. This option is compatible with the corresponding one designed for seepage elements (see Win.2-515).




Window 2-529

**Window 2-530:** `Convection/Radiation:Select:All`

Using this option user may select all visible convection/radiation elements.

Window 2-530

**Window 2-531:** `Convection/Radiation:Select:With material, existence f., label`

Using this option user may select convection/radiation elements with a given material or existence function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-531

**Window 2-532:** `Convection/Radiation:Select:On selected edges`

Using this option user may select 2D convection/radiation elements which are overlapping with selected edges.

Window 2-532

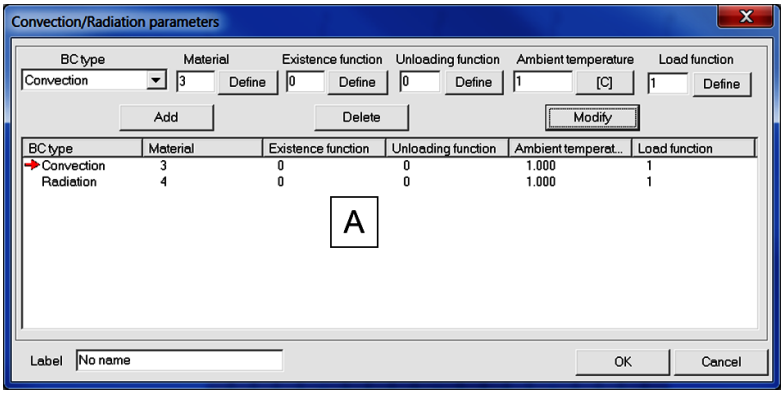
**Window 2-533:** `Convection/Radiation:Select:On selected faces`

Using this option user may select 3D convection/radiation elements which are overlapping with selected faces.

Window 2-533

**Window 2-534:** `Convection/Radiation:Update:Parameters`

Using this option user can modify material, existence function, unloading function (used to regularize an effect of removing convection/radiation elements, mainly for steady state drivers), ambient temperature and associated with it load time function.




**Remarks**:

1. The material, existence function, unloading function ID, ambient temperature and load time function to be associated with ambient temperature can be set in the edit fields **Material** [ ] **Existence function** [ ] **Unloading function** [ ] **Ambient temperature** [ ] and **Load function** [ ] or selected from list of predefined ones by pressing button **Define** placed next to the appropriate edit field
2. Convection/Radiation parameter(s) are defined at the material level under group ☒ **Heat**
3. If ambient temperature is varying in time it is recommended to put ambient temperature equal to 1.0 and reproduce its real time evolution through the load time function
4. Convection/Radiation elements may appear or disappear in certain time periods; to handle this one may duplicate convection/radiation elements (overlapping in time should be avoided) with a specific description, by adding them to the list (A) in the dialog box through button **Add**
5. To modify description of convection/radiation elements select one item from list (A), modify parameters in edit fields and then press button **Modify**
6. To delete one description of convection/radiation elements select one item from list (A) and then press button **Delete**

Window 2-534

## 2.3.1.3.16 VISCOUS DAMPER

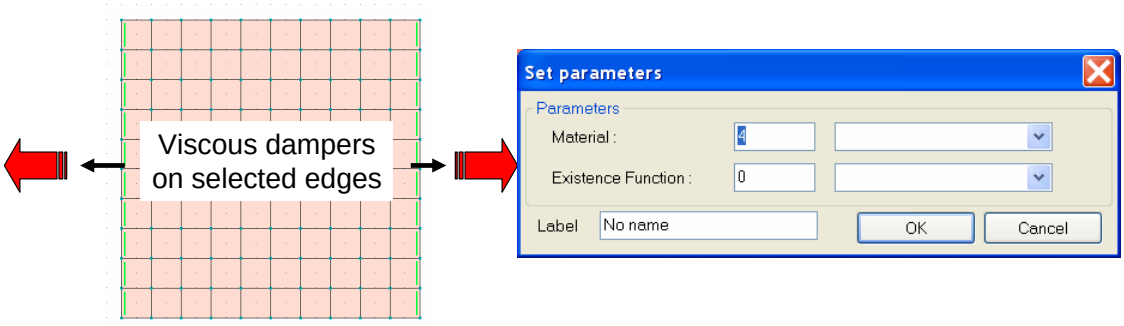
**Window 2-535: Options in menu** `Convection`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `2 nodes` | — | Window 2-537 |
| | `On edge(s)` | — | Window 2-536 |
| | `3 nodes` | — | Window 2-539 |
| | `4 nodes` | — | Window 2-539 |
| | `On face(s)` | — | Window 2-538 |
| `Select` | `All` | — | Window 2-540 |
| | `Box with 2 nodes` |  | Window 2-316 |
| | `With material` | — | Window 2-541 |
| | `With existence function` | — | Window 2-541 |
| | `With label` | — | Window 2-541 |
| | `On selected edges (2D)` | — | Window 2-542 |
| | `On selected faces (3D)` | — | Window 2-543 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-544 |

Window 2-535

**Window 2-536:** `Viscous damper:On edges`

New 2D viscous dampers are created by selecting continuum element edges in the existing real mesh.




**Remarks**:

1. Viscous damper elements are meaningful for `Deformation` and `Deformation+Flow` transient dynamic analyses
2. Viscous dampers can be generated on continuum element edges (2D)/faces (3D)
3. Viscous dampers are usually added on boundaries that are usually fixed for static drivers (during `Initial state` for instance); hence all fixities, in transient dynamic analyses, along the boundaries occupied by dampers must be released by setting proper existence function and unloading function (equal to 1.0 for any time instance); this way the nodal reactions/fluxes will be reapplied as external forces/fluxes to preserve the static equilibrium/flow continuity
4. These elements are used to cancel wave reflections from domain boundaries
5. If viscous dampers are conforming with the adjacent continuum elements then mesh refinement in the continuum will cause their refinement automatically
6. For detailed explanation on how to set parameters describing viscous damper elements see Window 2-544
7. All details concerning modelling transient dynamic problems can be found in the dedicated report on transient dynamics for single- and two-phase media

Window 2-536

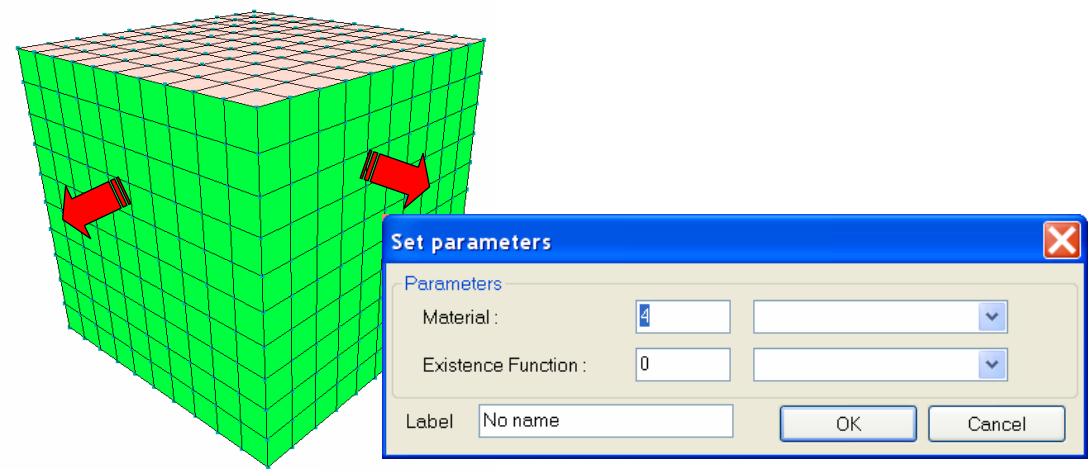
**Window 2-537:** `Viscous damper:2 nodes`

New 2D viscous damper elements are created along edges belonging to the line segment defined by the 2 picked nodes in the existing real mesh. This option, except the definition, is equivalent to the one described in Window 2-536.

Window 2-537

**Window 2-538:** `Viscous damper:On faces`

New 3D viscous damper elements are created by selecting continuum element faces in the existing real mesh (see example below). See also remarks given in Window 2-536.




Viscous damper elements on selected faces

Window 2-538

**Window 2-539:** `Viscous damper:3 nodes, 4 nodes`

New 3D viscous damper elements are created on faces which belong to the 3D surface defined by the 3 or 4 picked nodes in the existing real mesh (see example). Once faces are detected the generation procedure follows the steps described in Window 2-538.

Window 2-539

**Window 2-540:** `Viscous damper:Select:All`

Using this option user may select all visible viscous damper elements.

Window 2-540

**Window 2-541:** `Viscous damper:Select:With material, existence f., label`

Using this option user may select viscous damper elements with a given material or existence function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-541

**Window 2-542:** `Viscous damper:Select:On selected edges`

Using this option user may select 2D viscous damper elements which are overlapping with selected edges.

Window 2-542

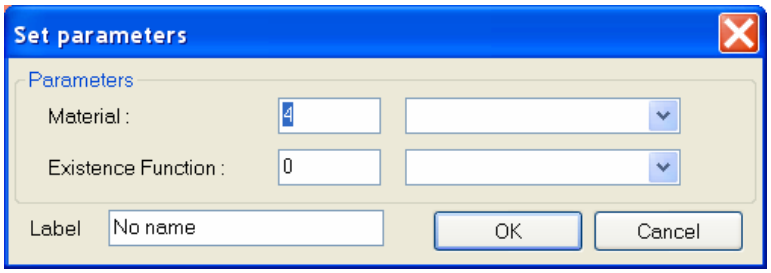
**Window 2-543:** `Viscous damper:Select:On selected faces`

Using this option user may select 3D viscous damper elements which are overlapping with selected faces.

Window 2-543

**Window 2-544:** `Viscous damper:Update:Parameters`

Using this option user can modify material and existence function




**Remarks**:

1. Viscous damper elements can be used in transient dynamic analyses for the `Deformation` and `Deformation+Flow` mode
2. Viscous damper material parameters are defined at the material level under group ☒ **Elastic**, ☒ **Unit weights** and ☒ **Flow**; these material parameters can also be inherited from the adjacent continuum; this setting can be made at the material level

Window 2-544

## 2.3.1.3.17 SHELL HINGES

**Window 2-545: Options in menu** `Shell hinge`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On edge(s)` | — | Window 2-546 |
| `Select` | `All` | — | Window 2-547 |
| | `With unloading function` | — | Window 2-548 |
| | `With existence function` | — | Window 2-548 |
| | `With label` | — | Window 2-548 |
| | `In zoom box` |  | Window 2-317 |
| | `In zoom circle` |  | Window 2-318 |
| | `Inside contour` |  | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-549 |

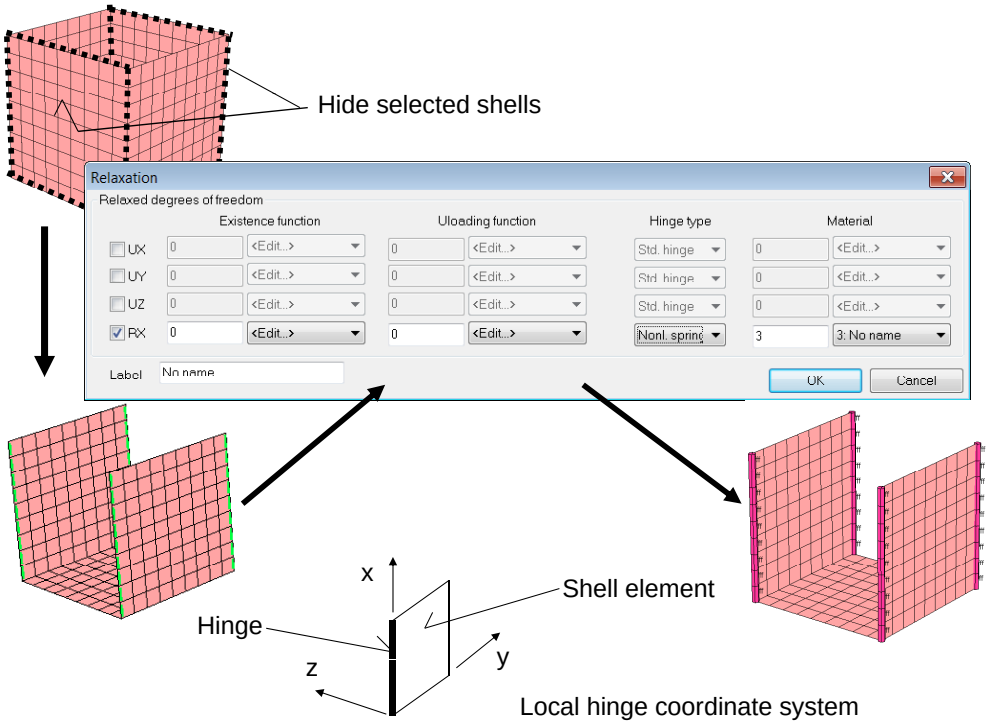
Window 2-545

## Window 2-546: `Shell hinge:On edges`

This option allows the user to define

1. an ideal hinge which relaxes some selected degree(s) of freedom
2. nonlinear uncoupled nonlinear spring with any type of the relation between the generalized sectional force (moment) and relative displacement (rotation)
3. nonlinear coupled Janssen nonlinear spring that can be used to model segmental linings

These settings are made in the local basis defined by $x - y_L - z_L$ axes. In addition user may associate an existence function(s) with the hinge/spring at any degree of freedom.




**Remarks**:

1. Shell hinge elements are meaningful for `Deformation` and `Deformation+Flow` analyses
2. Shell hinge elements can be generated exclusively on single layer shell element edges
3. An additional layer of implicit nodes is created automatically by the program
4. If shell hinges are conforming with the adjacent single layer shell elements then mesh refinement in shells will cause hinges refinement automatically
5. Definition of the local hinge coordinate system is shown in the figure
6. Any translational degree of freedom (in the local system) can be relaxed
7. Rotation along hinge $x$ axis is the only one to be relaxed
8. Each degree of freedom can be treated with a separate existence and unloading function
9. Next to each hinge endpoint special code (describing relaxed or fixed degree of freedom) is put; for instance $fffr$ means that $u_x$, $u_y$, $u_z$ are fixed while $\phi_x$ is relaxed (fixed DOF is is denoted by $f$ while relaxed by $r$)
10. Hinges should not be generated along edges that are common for two visible shell elements, otherwise computation will fail; to avoid that one of the adjacent shells should be temporarily hidden

**Window 2-547:** `Shell hinge:Select:All`

Using this option user may select all visible seepage elements.

Window 2-547

**Window 2-548:** `Shell hinge:Select:With existence f., unloading f., label`

Using this option user may select hinges with a given existence function, unloading function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-548

**Window 2-549:** `Shell hinge:Update:Parameters`

Using this option user can modify current set of relaxed degree of freedom and associated with them existence, unloading functions, and label. See Window 2-546 definition of the local hinge coordinate system.

Window 2-549

## 2.3.1.3.18 BOUNDARY CONDITIONS

The following types of boundary conditions can be created:

| | |
|---|---|
| Solid BC | Create kinematic BC on nodes |
| Temperature BC | Create temperature BC on nodes |
| Humidity BC | Create humidity BC on nodes |
| Pressure BC | Create fluid pressure BC on nodes |
| Periodic BC | Create periodic BC for pairs of nodes |

### 2.3.1.3.18.1 SOLID BC

**Window 2-550: Options in menu** `Solid BC`

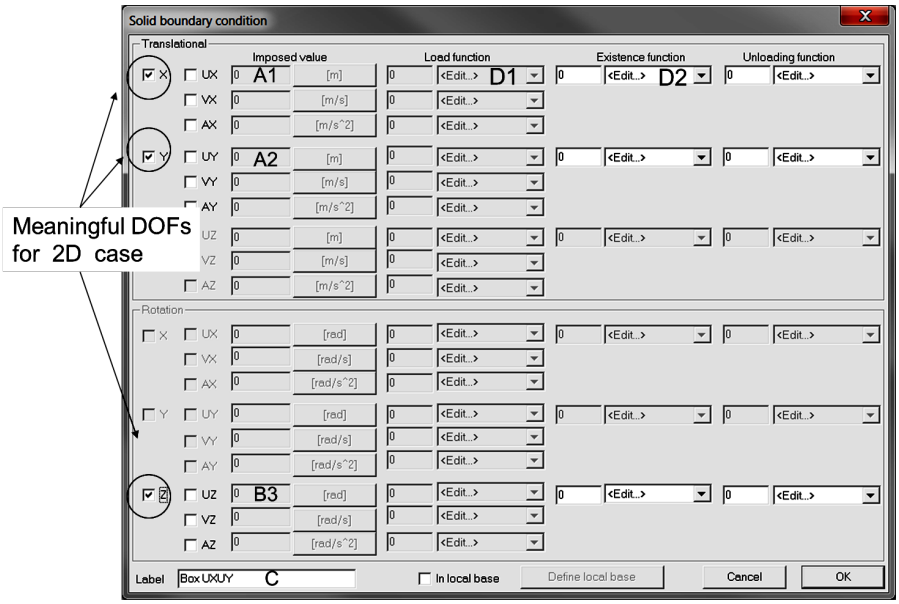
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node` | — | Window 2-551 |
| | `On element node` | — | Window 2-552 |
| | `Add BC` | — | Window 2-553 |
| | `On box` | — | Window 2-554 |
| `Select` | `All` | — | Window 2-561 |
| | `between 2 nodes` | — | Window 2-314 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-562 |
| | `With existence function` | — | Window 2-562 |
| | `With label` | — | Window 2-562 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-563 |
| | `Scale` | — | Window 2-564 |
| | `Export selected BC` | — | Window 2-565 |
| | `Update BC from file` | — | Window 2-566 |

Window 2-550

**Window 2-551:** `Solid BC:Create:On node`

New boundary conditions applied to the translational or rotational degrees of freedom are generated for a single picked node or group of selected nodes (if current selection list for nodes is not empty).

The dialog box used for setting these conditions is shown in figure below.




**Remarks**:

1. Edition of rotational DOF is enabled only if picked node or at least one node in the group of selected nodes is adjacent to the beam or shell element
2. Boundary conditions can be set in the global or local coordinate system; to activate definition in the local one option ☒ **Local base** must be set ON
3. In the 2D local coordinate system can be set through:
   - Angle – Window 2-555
   - Cylindrical system – Window 2-556
   - By vector – Window 2-557
4. In the 3D local coordinate system can be set as:
   - Cartesian local system – Window 2-558
   - Cylindrical system – Window 2-559
   - By vector – Window 2-560
5. Nodes may be fixed in time and the relaxed according to the applied existence function
6. For static type drivers one may impose displacements and /or rotations with a given value and load time function
7. For dynamic driver one may impose also velocities or accelerations at fixed nodes (corresponding displacements are integrated via time integration schemes: Newmark or HHT)
8. If at the fixed node both, displacement and velocity (or acceleration), are imposed then in static drivers imposed displacement is considered while in the dynamic driver velocity (or acceleration) (NB.:**imposed acceleration or velocity has higher priority then imposed displacement**)
9. For fast selection purposes one may apply a distinct label to the edited BC setting (in edit (C))

**Window 2-552:** `Solid BC:Create:On element node`

Using this option one may set the BC on a selected element node(s). This is crucial in case when node belongs to the contact zone and the BC is to be set only at node at one side of the interface. Further steps to define the BC are same as in the standard case (see Window 2-551)

Window 2-552

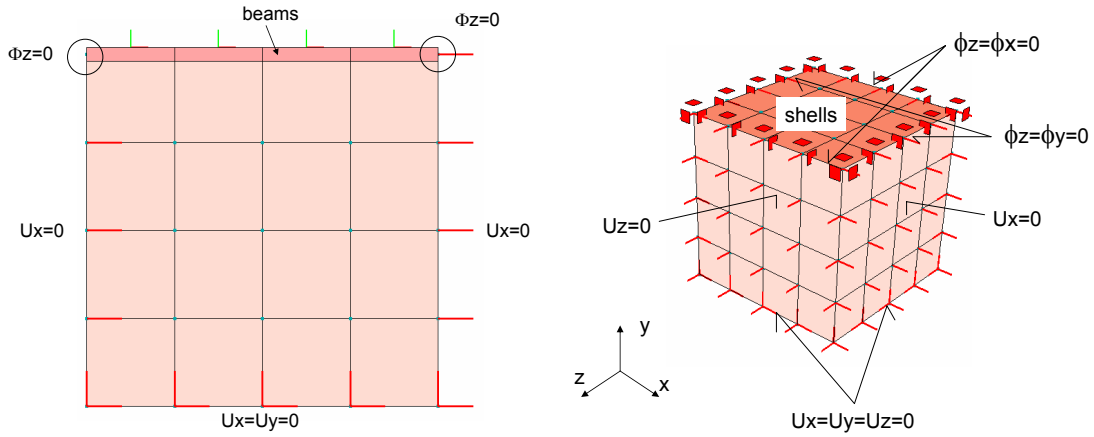
**Window 2-553:** `Solid BC:Create:Add BC`

Using this option one may add to the existing definition the new setting. It is very useful especially for operations on group of selected nodes. The previous and current setting are combined by means of logical summation. It is however forbidden to modify the local base setting. Further steps to define the BC are same as in the standard case (see Window 2-551)

Window 2-553

**Window 2-554:** `Solid BC:Create:On box`

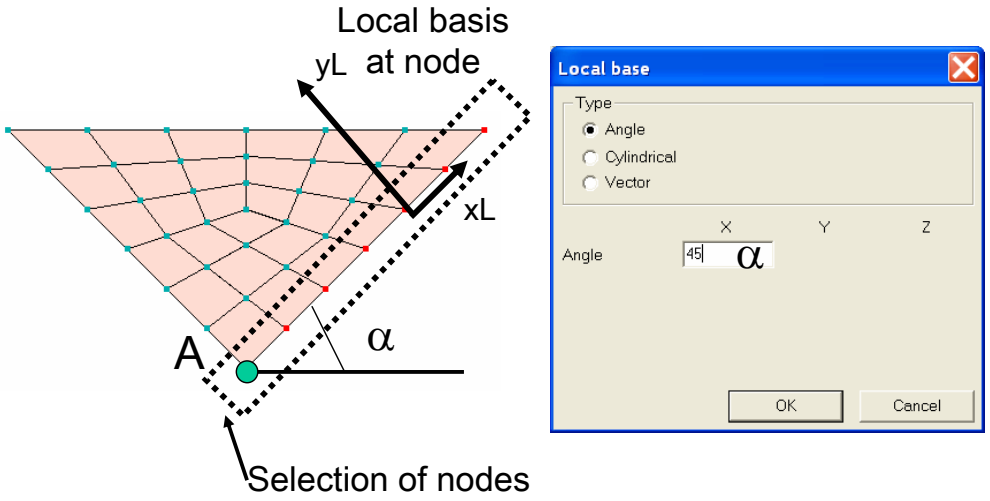
Using this option one may generate standard in the geotechnical practice set of the BC. An example of such setting for 2D and 3D is shown in figure below.




Window 2-554

**Window 2-555:** `Solid BC: Local 2D base def. by angle`

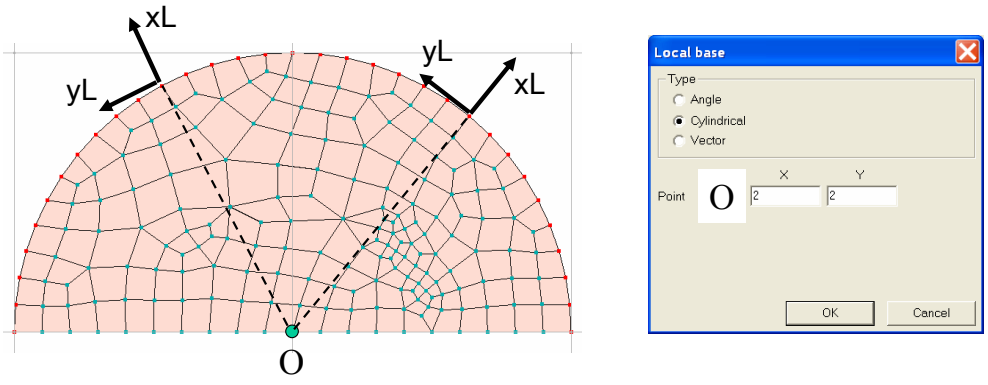
In the considered case the local coordinate system is defined through an angle $\alpha$, measured, in anti-clockwise direction, between global $x$ axis and the local $x_L$ one. The 2D setup is shown in figure below.




Window 2-555

**Window 2-556:** `Solid BC: Cylindrical local 2D base`

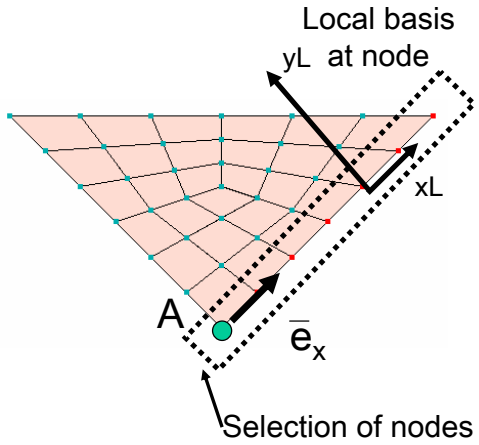
In the considered case the local coordinate system is defined at each selected node by connecting it with point $P$ to set the local $x_L$ axis. The local $z_L$ axis coincides with the global $z$ one and local $y_L$ axis is taken as a vector product of the two remining ones. The 2D setup is shown in figure below.

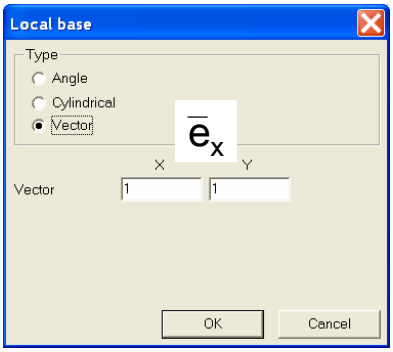



Window 2-556

**Window 2-557:** `Solid BC: Local 2D base def. by vector`

In the considered case the local $x_L$ axis is assumed to be parallel to the given axis vector $\mathbf{v}$ while local $y_L$ axis is perpendicular to $x_L$. The 2D setup is shown in figure below.



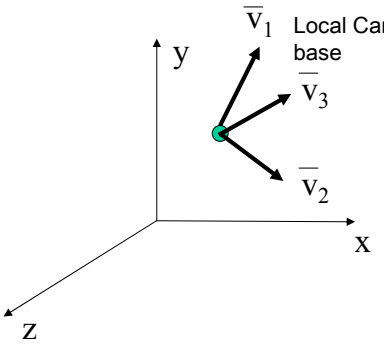

Window 2-557

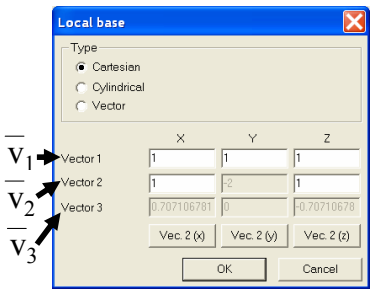
**Window 2-558:** `Solid BC: Cartesian local 3D base`

In the considered case the local coordinate system is defined directly by given two base vectors $\mathbf{v_1}$ and $\mathbf{v_2}$. The third one ($\mathbf{v_3}$) is automatically computed using orthogonality condition. In the second vector user may set only the two coordinates which are independent, the third one is computed using the orthogonality condition. The 3D setup is shown in figure below.

**Remarks**:

1. To disable the first, second or third coordinate of vector $\mathbf{v_2}$ from edition press button **Vec.2 (x)**, **Vec.2 (y)** or **Vec.2 (z)** accordingly

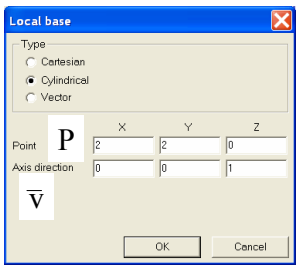


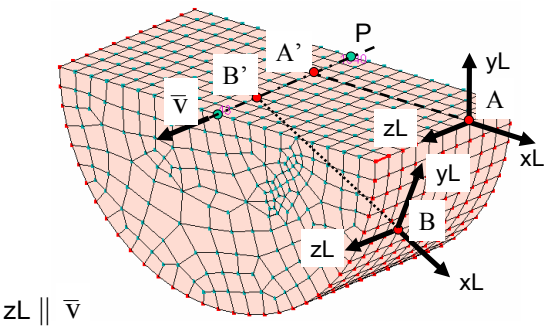




Window 2-558

**Window 2-559:** `Solid BC: Cylindrical local 3D base`

In the considered case the local coordinate system is defined at each selected node by projecting it on the axis $\mathbf{v}$ to set the local $x_L$ axis. The local $z_L$ axis is taken as parallel to the given axis vector $\mathbf{v}$ while the local $y_L$ axis is computed as a vector product of the base vbectors of the two remaining axes. The 3D setup is shown in figure below.

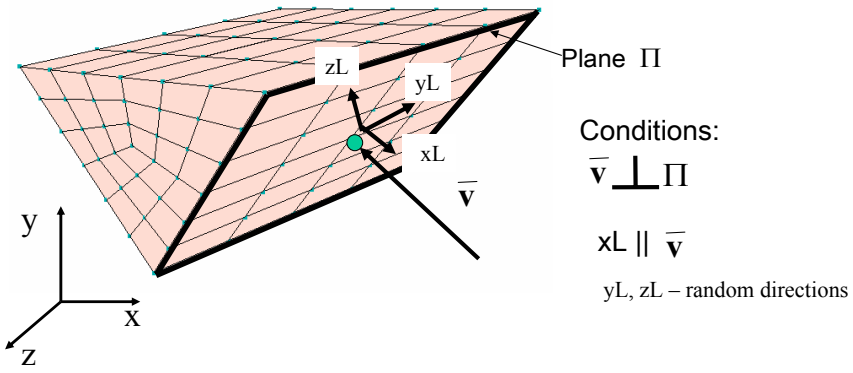



Window 2-559

**Window 2-560:** `Solid BC: Local 3D base def. by vector`

In the considered case the only important direction is the one indicated by a given vector $\mathbf{v}$ and for that reason the local $x_L$ axis coincides with that vector. The two remaining axes are generated in a random manner. The 3D setup is shown in figure below.




**Remark**:The only meaningful direction to be fixed is the one along $x$ axis

Window 2-560

**Window 2-561:** `Solid BC:Select:All`

This option selects all solid BC.

Window 2-561

**Window 2-562:** `Solid BC:Select:With load f., existence f., label`

Using this option user may select solid BC with a given existence function, load time function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-562

**Window 2-563:** `Solid BC:Update:Parameters`

This option activates the dialog box to modify the assumed BC description for a single picked node or group of selected nodes. The detailed description on how to set the data is given in Window 2-551.

**Remark**:Note that running this option for group of selected nodes only default parameters will be set in the dialog box. No information on previous data setting is available.

Window 2-563

**Window 2-564:** `Solid BC:Update:Scale`

Using this option one may enlarge or diminish size of the BC markers.

Window 2-564

**Window 2-565:** `Solid BC:Update:Export sel. BC`

Using this option user may export description of the selected BC to the external text file. This file can be read later on, and possibly modified, with aid of the Excel application.

Window 2-565

**Window 2-566:** `Solid BC:Update/Update BC from file`

This option is coupled with `Solid BC:Update/Export sel. BC` option (see Window 2-565). It allows to update description of exported BC, modified externally by Excel or another application.

Window 2-566

## 2.3.1.3.18.2 TEMPERATURE BC

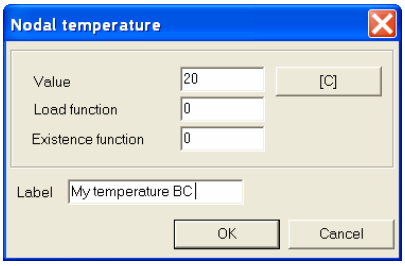
**Window 2-567: Options in menu** `Temperature BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On node | — | Window 2-568 |
| | On element node | — | Window 2-569 |
| | 2 nodes | — | Window 2-570 |
| | On edge(s) | — | Window 2-571 |
| | 3 nodes | — | Window 2-572 |
| | 4 nodes | — | Window 2-572 |
| | On face | — | Window 2-573 |
| | On selected faces | — | Window 2-574 |
| Select | All | — | Window 2-575 |
| | Between 4 nodes | — | Window 2-315 |
| | Box with 2 nodes | — | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With load function | — | Window 2-577 |
| | With existence function | — | Window 2-577 |
| | With label | — | Window 2-577 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-578 |
| | Scale | — | Window 2-579 |

Window 2-567

**Window 2-568:** `Temperature BC:Create:On node`

New boundary conditions applied to the temperature degree of freedom are generated for a single picked node or group of selected nodes (if current selection list for nodes is not empty). The dialog box used for setting these conditions is shown in figure below.




**Remarks**:

1. Temperature can be fixed at nodes according to the existence function
2. The evolution of temperature BC is driven by the load time function($T(t) = T_{BC} \cdot LTF(t)$)
3. One may define a distinct label for current temperature BC definition

Window 2-568

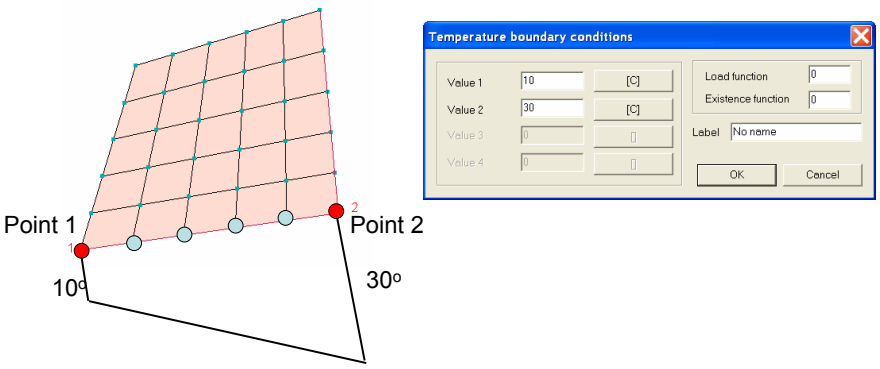
**Window 2-569:** `Temperature BC:Create:On element node`

Using this option one may set the BC on a selected element node(s). This is crucial in case when node belongs to the contact interface zone and the BC is to be set only at node at one side of the interface. Further steps to define the BC are same as in the standard case (see Window 2-568)

Window 2-569

**Window 2-570:** `Temperature BC:Create:2 nodes`

Using this option one may define nodal temperatures by clicking 2 nodes on the boundary of a 2D continuum mesh. These 2 nodes constitute a linear 2-node super-element. The temperature BC is applied at all continuum element edges which belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed.



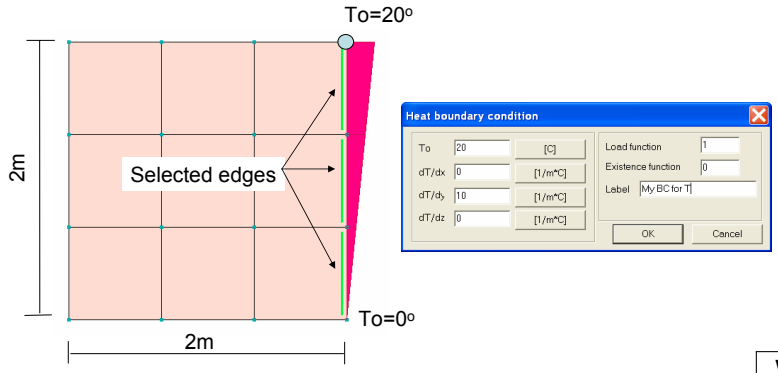

**Remark**:

See remarks in Window 2-568

Window 2-570

## Window 2-571: Temperature BC:Create:On edge(s)

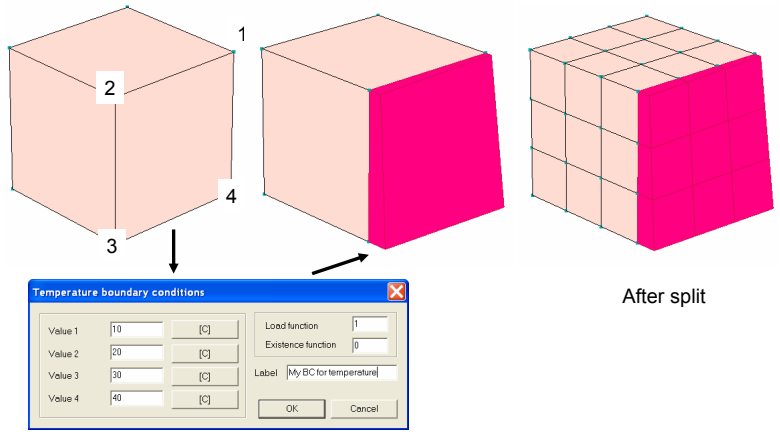
This option has a double meaning depending on whether selection list of edges in empty or not. If none of the edges is selected then user must pick the edge and generation procedure follows strictly the one by 2 nodes described in Window refWin:-ASSEMB-PREPRO-MENU-FE-MODEL-BC-T-CREATE-2N. In all other cases user must pick the point at which he sets value of the temperature $T_o$ and to set temperature gradients in $x$ and $y$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the temperature at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $T(\mathbf{x}) = T_o + dT_x \cdot (x - x_P) + dT_y \cdot (y - y_P)$. This option is valid only for 2D models.




Window 2-571

## Window 2-572: Temperature BC:Create:3 nodes, 4 nodes

Using this option one may define nodal temperatures by clicking 3 or 4 nodes on the boundary of a 3D continuum mesh. These 3/4 nodes constitute a triangular or quadrilateral super-element. The temperature BC is applied at all continuum element faces which belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed.




**Remarks**:

1. Standard linear interpolation rule (linear for triangular and bi-linear for quadrilateral super-element) is used to compute temperature at given node which belongs to the super-element
2. For additional information see Window 2-568

Window 2-572

**Window 2-573:** `Temperature BC:Create:On face`

Using this option one may define nodal temperatures by picking a single face being a part of 3D continuum element. This face is a triangular or quadrilateral surface element and for that reason generation procedure is exactly the same as the one `3 nodes` or `4 nodes` described in Window 2-572.

Window 2-573

**Window 2-574:** `Temperature BC:Create:On selected faces`

Using this option one may set the temperature BC on a selected set of 3D continuum element faces. To complete the definition user must pick the point at which he sets value of the temperature $T_o$ and temperature gradients in $x$, $y$ and $z$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the temperature at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of faces is computed through the formula: $T(\mathbf{x}) = T_o + dT_x \cdot (x - x_P) + dT_y \cdot (y - y_P) + dT_z \cdot (z - z_P)$. This option is valid only for 3D models.

Window 2-574

**Window 2-575:** `Temperature BC:Select:All`

This option selects all temperature BC.

Window 2-575

**Window 2-576:** `Temperature BC:Select:Between 4 nodes`

This option allows to select temperature BC set at all nodes belonging to the 3D surface which forms a quadrilateral super-element defined by 4 picked nodes. This option must be used with caution as the temperature BC may be set using different definitions (like `4 nodes`,`On selected faces`).

Window 2-576

**Window 2-577:** `Temperature BC:Select:With load f., existence f., label`

Using this option user may select temperature BC with a given existence function, load time function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-577

**Window 2-578:** `Temperature BC:Update:Parameters`

This option activates the dialog box to modify the assumed BC description. Each object representing temperature BC inherits the information on how it was initially set up. For that reason updating the temperature BC corresponds strictly to its initial definition described in group of methods under general option `Create`.

Window 2-578

**Window 2-579:** `Temperature BC:Update:Scale`

Using this option one may enlarge or diminish size of the BC markers.

Window 2-579

## 2.3.1.3.18.3 HUMIDITY BC

All options designed to set humidity BC are fully compatible with the ones designed for temperature BC except that temperature must be replaced by humidity. Hence each option is referred to the corresponding one for temperature BC.

**Window 2-580: Options in menu** `Humidity BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On node | — | Window 2-568 |
| | On element node | — | Window 2-569 |
| | 2 nodes | — | Window 2-570 |
| | On edge(s) | — | Window 2-571 |
| | 3 nodes | — | Window 2-572 |
| | 4 nodes | — | Window 2-572 |
| | On face | — | Window 2-573 |
| | On selected faces | — | Window 2-574 |
| Select | All | — | Window 2-575 |
| | Between 4 nodes | — | Window 2-315 |
| | Box with 2 nodes | — | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With load function | — | Window 2-577 |
| | With existence function | — | Window 2-577 |
| | With label | — | Window 2-577 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-578 |
| | Scale | — | Window 2-579 |

Window 2-580

## 2.3.1.3.18.4 PRESSURE BC

In the following pressure BC definitions compressive pore fluid pressures are negative.

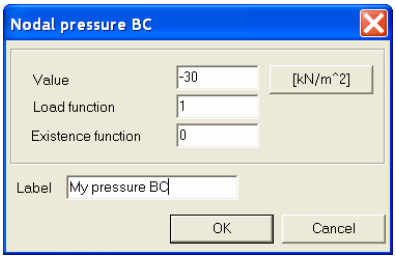
**Window 2-581: Options in menu** `Pressure BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | On node | — | Window 2-582 |
| | On element node | — | Window 2-583 |
| | 2 nodes | — | Window 2-584 |
| | On edge(s) | — | Window 2-585 |
| | Fluid head on selected nodes | — | Window 2-586 |
| | Fluid head on selected edges | — | Window 2-587 |
| | 3 nodes | — | Window 2-588 |
| | 4 nodes | — | Window 2-588 |
| | On face | — | Window 2-589 |
| | On selected faces | — | Window 2-590 |
| | Fluid head on selected faces | — | Window 2-591 |
| Select | All | — | Window 2-592 |
| | Between 4 nodes | — | Window 2-315 |
| | Box with 2 nodes | — | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With load function | — | Window 2-594 |
| | With existence function | — | Window 2-594 |
| | With label | — | Window 2-594 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-595 |
| | Add sel. nodes to fluid head | — | Window 2-596 |
| | Remove sel. nodes from fluid head | — | Window 2-597 |
| | Sel. nodes for fluid head | — | Window 2-598 |
| | Scale | — | Window 2-600 |
| | Scale fluid head | — | Window 2-599 |

Window 2-581

## Window 2-582: Pressure BC:Create:On node

New boundary conditions applied to the pressure degree of freedom are generated for a single picked node or group of selected nodes (if current selection list for nodes is not empty). The dialog box used for setting these conditions is shown in figure below.




**Remarks**:

1. Pressure can be fixed at nodes according to the existence function
2. The evolution of pressure BC is driven by the load time function($T(t) = T_{BC} \cdot LTF(t)$)
3. One may define a distinct label for current pressure BC definition

Window 2-582

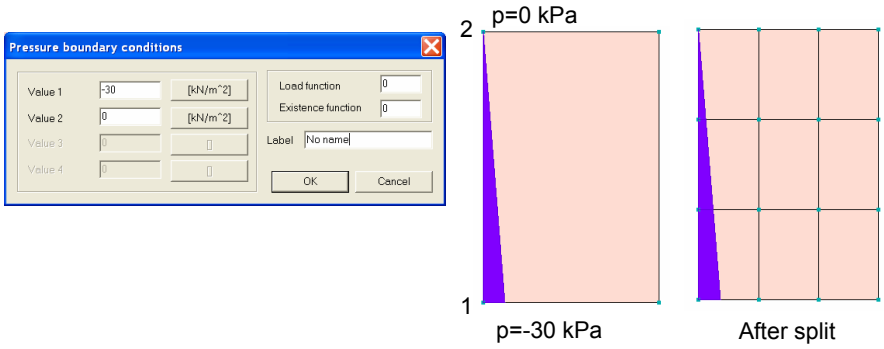
## Window 2-583: Pressure BC:Create:On element node

Using this option one may set the BC on a selected element node(s). This is crucial in case when node belongs to the contact interface zone and the BC is to be set only at node at one side of the interface. Further steps to define the BC are same as in the standard case (see Window 2-582).

Window 2-583

## Window 2-584: Pressure BC:Create:2 nodes

Using this option one may define nodal pressures by clicking 2 nodes on the boundary of a 2D continuum mesh. These 2 nodes constitute a linear 2-node super-element. The pressure BC is applied at all continuum element edges which belong to that super-element. Hence this definition when refining the mesh will automatically be projected on the new mesh and no additional action to be made by the user is needed.



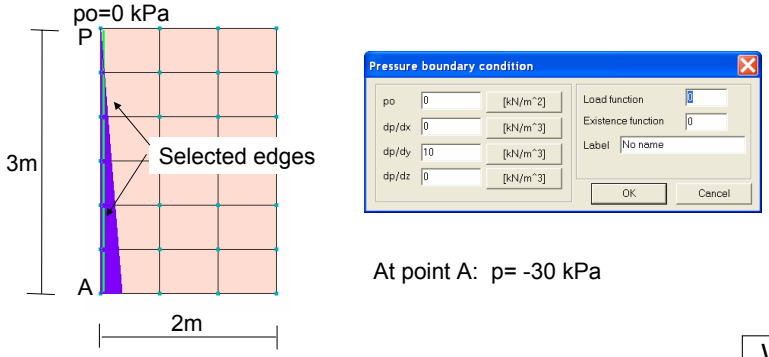

**Remark**:

See remarks in Window 2-582

Window 2-584

## Window 2-585: Pressure BC:Create:On edge(s)

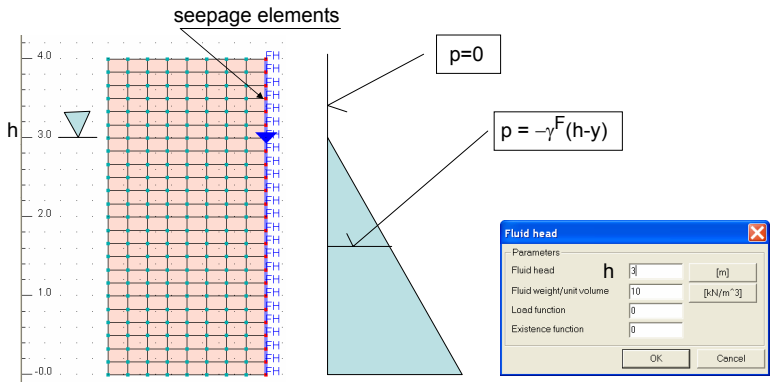
This option has a double meaning depending on whether selection list of edges in empty or not. If none of the edges is selected then user must pick the edge and generation procedure follows strictly the one by 2 nodes described in Window 2-584. In all other cases user must pick the point at which he sets value of the pressure $T_o$ and to set pressure gradients in $x$ and $y$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the pressure at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $T(\mathbf{x}) = T_o + dT_x \cdot (x - x_P) + dT_y \cdot (y - y_P)$. This option is valid only for 2D models.




Window 2-585

## Window 2-586: Pressure BC:Create:Fluid head on sel. nodes

This option is supported only for the back compatibility with earlier Z_Soil versions (it is recommended to use option Create:Fluid head on selected edges instead (see.Win.(2-587))). Using this option user may impose pressure boundary condition on group of selected nodes using so-called fluid head (or pressure head) definition (see figure below). Fluid head is defined as a scalar-valued function $h(p, y)$ such that $h = -\frac{p}{\gamma^F} + y$. This type of the boundary condition should be used in conjunction with seepage elements to enable modeling of filling or drawndown phenomenon.



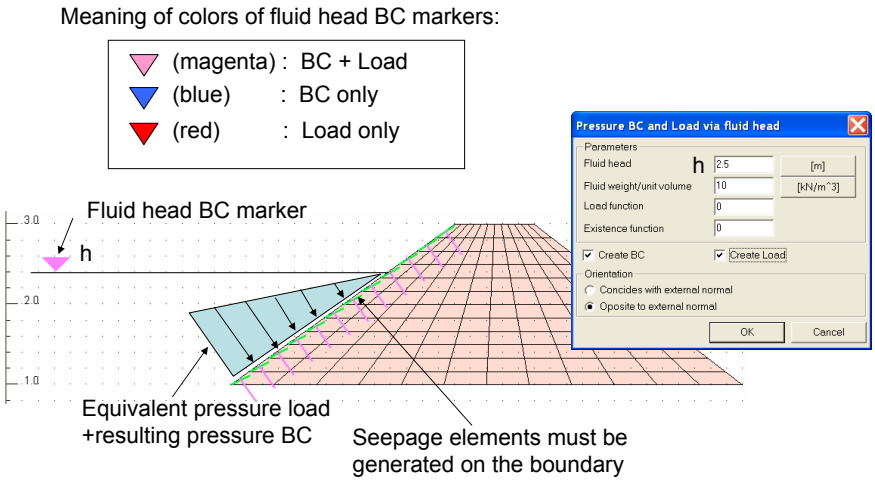

**Remarks**:

1. $h$ is equivalent to the known elevation ($y$ coordinate) of the free water surface
2. setting this boundary condition requires (usually) an application of equivalent pressure load on submerged free boundaries

Window 2-586

**Window 2-587:** `Pressure BC:Create:Fluid head on sel. edges`

Using this option user may impose pressure boundary condition on group of selected edges using so-called fluid head (or pressure head) definition (see figure below). Fluid head is defined as a scalar-valued function $h(p, y)$ such that $h = -\frac{p}{\gamma^F} + y$. This type of the boundary condition should be used in conjunction with seepage elements to enable modeling of filling or drawndown phenomenon. An equivalent pressure load on submerged free solid boundaries can be simultaneously applied.



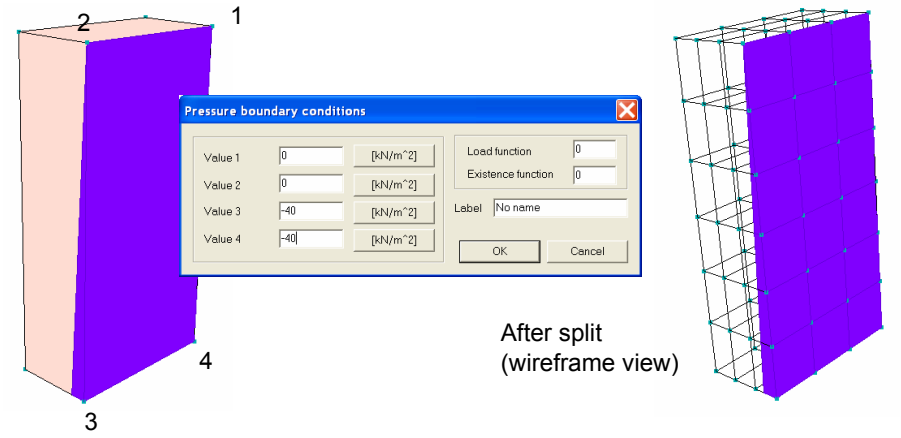

**Remarks**:

1. $h$ is equivalent to the known elevation ($y$ coordinate) of the free water surface
2. Load time function allows to vary the pressure head value in time
3. An existence function may switch ON/OFF the resulting pressure BC
4. Conversion from the imposed pressure head to the prescribed pore pressure value, at all nodal points which belong to the selected edges, is made internally by the code

Window 2-587

**Window 2-588:** `Pressure BC:Create:3 nodes, 4 nodes`

Using this option one may define nodal pressures by clicking 3 or 4 nodes on the boundary of a 3D continuum mesh. These 3/4 nodes constitute a triangular or quadrilateral super-element. The pressure BC is applied at all continuum element faces which belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed.




**Remarks**:

1. Standard linear interpolation rule (linear for triangular and bi-linear for quadrilateral super-element) is used to compute pressure at given node which belongs to the super-element
2. For additional information see Window 2-582

Window 2-588

**Window 2-589:** `Pressure BC:Create:On face`

Using this option one may define nodal pressures by picking a single face being a part of 3D continuum element. This face is a triangular or quadrilateral surface element and for that reason generation procedure is exactly the same as the one `3 nodes` or `4 nodes` described in Window 2-588.

Window 2-589

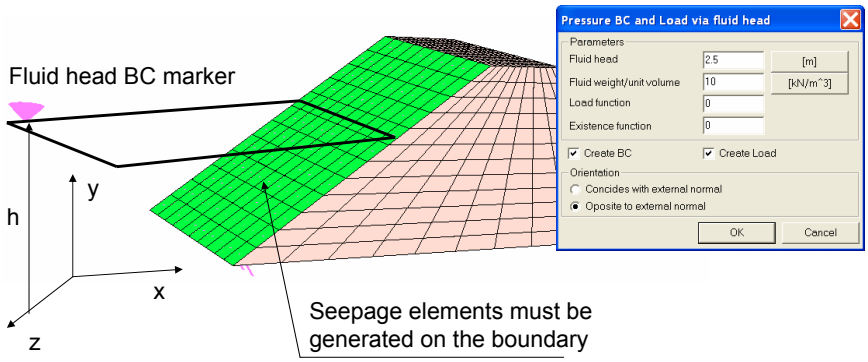
**Window 2-590:** `Pressure BC:Create:On selected faces`

Using this option one may set the pressure BC on a selected set of 3D continuum element faces. To complete the definition user must pick the point at which he sets value of the pressure $T_o$ and pressure gradients in $x$, $y$ and $z$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the pressure at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of faces is computed through the formula: $p(\mathbf{x}) = p_o + dp_x \cdot (x - x_P) + dp_y \cdot (y - y_P) + dp_z \cdot (z - z_P)$. This option is valid only for 3D models.

Window 2-590

**Window 2-591:** `Pressure BC:Create:Fluid head on sel. faces`

Using this option user may impose pressure boundary condition on group of selected faces using so-called fluid head (or pressure head) definition (see figure below). This option is equivalent to the one destined for the 2D models and described in Win.(2-587).




Window 2-591

**Window 2-592:** `Pressure BC:Select:All`

This option selects all pressure BC except `Fluid head...`

Window 2-592

**Window 2-593:** `Pressure BC:Select:Between 4 nodes`

This option allows to select pressure BC set at all nodes belonging to the 3D surface which forms a quadrilateral super-element defined by 4 picked nodes. This option must be used with caution as the pressure BC may be set using different definitions (like `4 nodes`,`On selected faces`).

Window 2-593

**Window 2-594:** `Pressure BC:Select:With load f., existence f., label`

Using this option user may select pressure BC with a given existence function, load time function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-594

**Window 2-595:** `Pressure BC:Update:Parameters`

This option activates the dialog box to modify the assumed BC description. Each object representing pressure BC inherits the information on how it was initially set up. For that reason updating the pressure BC corresponds strictly to its initial definition described in group of methods under general option `Create`.

Window 2-595

**Window 2-596:** `Pressure BC:Update:Add sel. nodes to fluid head`

This option allows to add selected nodes to the fluid head boundary condition represented on the screen as a blue triangle.

Window 2-596

**Window 2-597:** `Pressure BC:Update:Remove sel. nodes from fluid head`

This option allows to delete selected nodes from the fluid head boundary condition represented on the screen as a blue triangle.

Window 2-597

**Window 2-598:** `Pressure BC:Update:Sel. nodes for fluid head`

This option allows to select all nodes associated with the fluid head boundary condition represented on the screen as a blue triangle.

Window 2-598

**Window 2-599:** `Pressure BC:Update:Scale fluid head`

This option allows to scale fluid head boundary condition marker.

Window 2-599

**Window 2-600:** `Pressure BC:Update:Scale`

Using this option one may enlarge or diminish size of the BC markers.

Window 2-600

## 2.3.1.3.18.5 PERIODIC BC

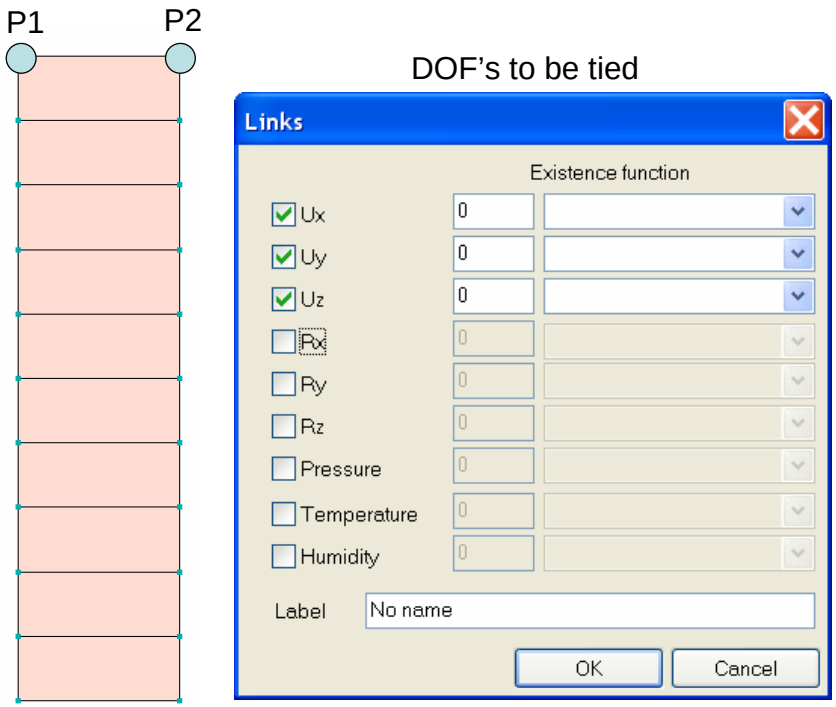
**Window 2-601: Options in menu** `Periodic BC`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `2 nodes` | — | Window 2-602 |
| | `Nodes & plane` | — | Window 2-603 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-604 |
| | `Scale` | — | Window 2-605 |

Window 2-601

**Window 2-602:** `Periodic BC:Create:2 nodes`

This option allows to tie selected degrees of freedom at two picked nodal points. It may concern any type of the degree of freedom (displacement, pressure, temperature, humidity). It is mainly used in dynamic applications to simulate motion of shear layer. The dialog box used for setting these conditions is shown in the figure below.



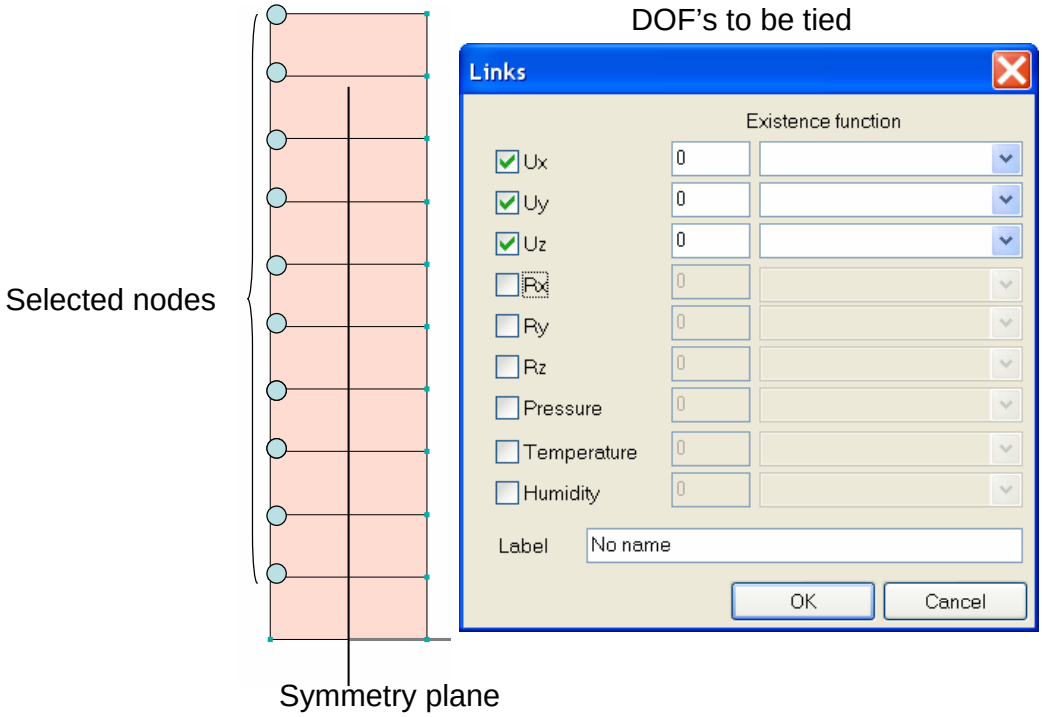

**Remarks**:

1. The tying can be made only in the global coordinate system
2. **This option does not apply to the fixed degree of freedom**
3. Each degree of freedom can be tied according to the declared existence function

Window 2-602

**Window 2-603:** `Periodic BC:Create:Nodes & plane`

This option allows to tie selected degrees of freedom at the two opposite boundaries located symmetrically with respect to the indicated auxiliary plane. This option helps to generate this type of the boundary conditions for large computational models. The dialog box used for setting these conditions is shown in the figure below.




**Remarks**:

1. The tying can be made only in the global coordinate system
2. **This option does not apply to the fixed degree of freedom**
3. Each degree of freedom can be tied according to the declared existence function

Window 2-603

**Window 2-604:** `Periodic BC:Update:Parameters`

This option activates the dialog box to modify the assumed periodic BC description for a single picked node or group of selected nodes. The detailed description on how to set the data is given in Window 2-602.

**Remark:It has to be emphasized that this modification will be made too for all other nodes there were considered during creation of the** `Periodic BC`

Window 2-604

**Window 2-605:** `Periodic BC:Update:Scale`

This option is not yet supported.

Window 2-605

### 2.3.1.3.19 LOADS

The following loads can be applied to the real mesh:

| | |
|---|---|
| Body load | Create body load on selected elements; this load will overwrite the global gravity setting |
| Nodal load | Create concentrated forces on selected nodes |
| Surface load | Create surface loads on selected element faces/edges |
| Beam load | Create surface loads on selected beams |

## 2.3.1.3.19.1 BODY LOAD

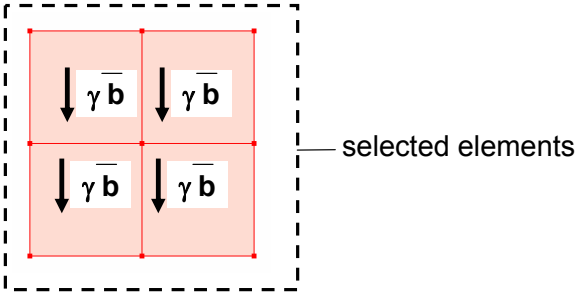
**Window 2-606: Options in menu** `Body load`

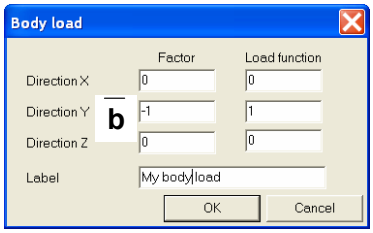
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Apply to element(s)` | — | Window 2-607 |
| `Select` | `Elements with body load` | — | Window 2-608 |
| | `Elements with body load via label` | — | Window 2-609 |
| `Delete` | `Body load for element(s)` | — | Window 2-610 |
| `Update` | `Parameters` | — | Window 2-578 |

Window 2-606

**Window 2-607:** `Body load:Create:Apply to element(s)`

This option applies a body load to a single picked element (continuum or structural) or group of selected elements. The dialog box used for setting the gravity direction and associated load time functions distinct for each global $x$, $y$ and $z$ directions is shown in figure below.




**Remarks**:

1. This setting for body load vector overwrites the global setting declared in main Z_Soil menu under `Gravity` menu
2. The body load component at $i-th$ global direction is computed as $b_i\ (t) = \bar{b}_i\ \gamma\ LTF_i\ (t)$ and unit weight is set up at the material level in group `Unit weight`
3. In the context of dynamic time history analysis this option allows to cancel body loads due to gravity while maintaining element contribution to the global mass matrix

Window 2-607

**Window 2-608:** `Body load:Select:Elements with body load`

This option selects all elements to which body load was applied through option `Apply to element(s)`.

Window 2-608

**Window 2-609:** `Body load:Select:Elem. with body load via label`

This option selects all elements to which body load, identified by a given label, was applied through option `Apply to element(s)`.

Window 2-609

**Window 2-610:** `Body load:Delete:Body load for element(s)`

This option selects all elements to which body load, identified by a given label, was applied through option `Apply to element(s)`.

Window 2-610

**Window 2-611:** `Body load:Update:Parameters`

This option activates the dialog box (see figure in Window 2-607) to update body load direction vector, load time functions and label for a single picked element or group of selected elements.

**Remark**:In case when selection list of elements is not empty initial values in the dialog are filled with default values.

Window 2-611

## 2.3.1.3.19.2 NODAL LOAD

**Window 2-612: Options in menu** `Nodal load`

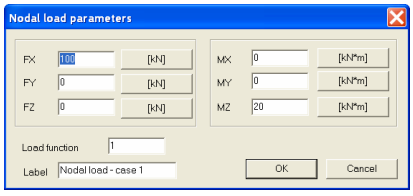
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node` | — | Window 2-613 |
| | `On element node` | — | Window 2-614 |
| `Select` | `All` | — | Window 2-615 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-616 |
| | `With label` | — | Window 2-616 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-617 |
| | `Scale` | — | Window 2-618 |

Window 2-612

**Window 2-613:** `Nodal load:Create:On node`

New nodal load applied to the translational or rotational degrees of freedom is generated for a single picked node or group of selected nodes (if current selection list for nodes is not empty).

The dialog box used for setting these loads is shown in figure below.




**Remarks**:

1. Edition of rotational DOF is enabled only if picked node or at least one node in the group of selected nodes is adjacent to the beam or shell element
2. For fast selection purposes one may apply a distinct label to the edited load

Window 2-613

**Window 2-614:** `Nodal load:Create:On element node`

Using this option one may apply the nodal load to the selected element node(s). This is crucial in case when node belongs to the contact zone and the load is to be set only at node at one side of the interface. Further steps to define the load are same as in the standard case (see Window 2-613)

Window 2-614

**Window 2-615:** `Nodal load:Select:All`

This option selects all nodal loads.

Window 2-615

**Window 2-616:** `Nodal load:Select:With load f., existence f., label`

Using this option user may select nodal load with a given load time function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-616

**Window 2-617:** `Nodal load:Update:Parameters`

This option activates the dialog box to modify the applied nodal load description, for a single picked node or group of selected nodes. The detailed description on how to set the data is given in Window 2-613.

**Remark**:When running this option for group of selected nodes only default parameters will be set in the dialog box.

Window 2-617

**Window 2-618:** `Nodal load:Update:Scale`

Using this option one may enlarge or diminish size of nodal vectors, representing concentrated forces, or arcs, representing concentrated moments. markers.

Window 2-618

## 2.3.1.3.19.3 SURFACE LOAD

**Window 2-619: Options in menu** `Surface load`

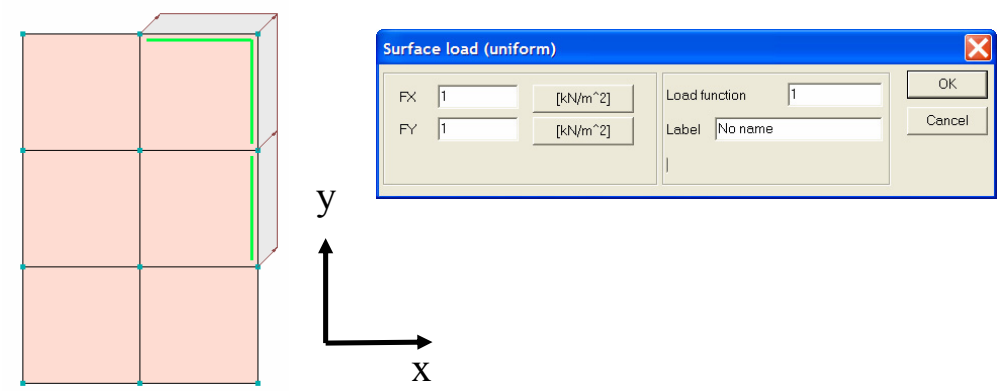
| Option | Method | Toolbar | Description |
|---|---|---|---|
| UNIFORM | On edge(s)(Uniform) | — | Window 2-620 |
| | On face(s)(Uniform) | — | Window 2-621 |
| VARIABLE | 2 nodes (Variable) | — | Window 2-622 |
| | 3 nodes (Variable) | — | Window 2-623 |
| | 4 nodes (Variable) | — | Window 2-623 |
| PRESSURE | 2 nodes (P) | — | Window 2-624 |
| | On selected edges (HP) | — | Window 2-625 |
| | 3 nodes (P) | — | Window 2-627 |
| | 4 nodes (P) | — | Window 2-627 |
| | On selected faces (HP) | — | Window 2-626 |
| PRESSURE via Fluid Head | On selected edges (SH) | — | Window 2-628 |
| | On selected faces (SH) | — | Window 2-629 |
| Select | All | — | Window 2-630 |
| | between 4 nodes | — | Window 2-315 |
| | Box with 2 nodes | — | Window 2-316 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| | With load function | — | Window 2-631 |
| | With label | — | Window 2-631 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-632 |
| | Scale | — | Window 2-633 |

Window 2-619

## Window 2-620: Surface load:UNIFORM:On edge(s) (Uniform)

New uniform surface load is generated on a single picked edge or group of selected edges (if current selection list of edges is not empty). This option is valid only in 2D.

The dialog box used for setting this load is shown in figure below.




**Remarks**:

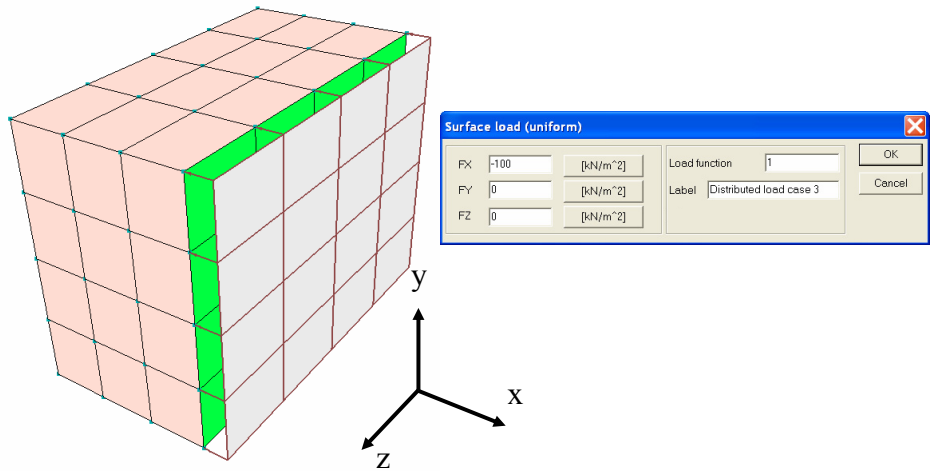
1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement

Window 2-620

## Window 2-621: Surface load:UNIFORM:On face(s) (Uniform)

New uniform surface load is generated on a single picked face or group of selected faces (if current selection list of faces is not empty). This option is valid only in 3D.

The dialog box used for setting this load is shown in figure below.




**Remarks**:

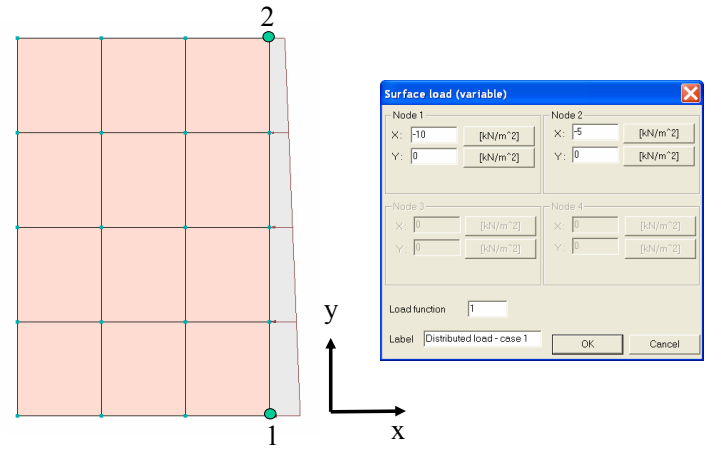
1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement

Window 2-621

**Window 2-622:** `Surface load:VARIABLE:2 nodes (Variable)`

New linearly varying surface load is generated on element edges belonging to the line section designed by the two picked points. This option is valid only in 2D.

An example for this setting is shown in figure below.



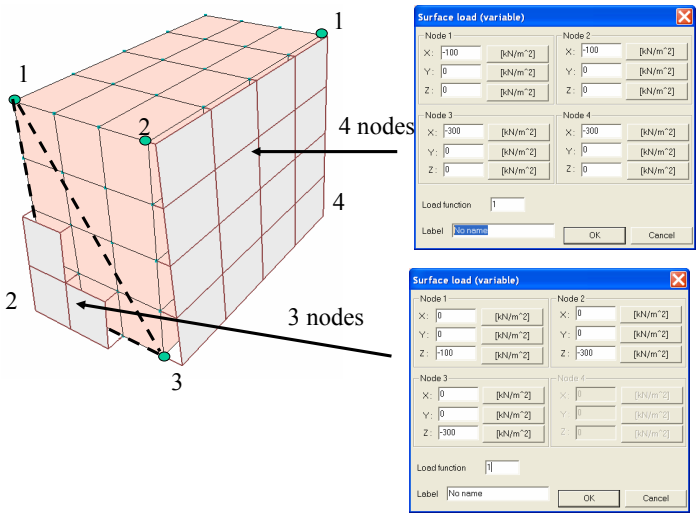

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement

Window 2-622

**Window 2-623:** `Surface load:VARIABLE:3,4 nodes (Variable)`

Using this option one may define variable surface load by clicking 3 or 4 nodes on the boundary of a 3D mesh. These 3/4 nodes constitute a triangular or quadrilateral super-element. The load is applied at all element faces which belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. The dialog box used for setting this load is shown in figure below.



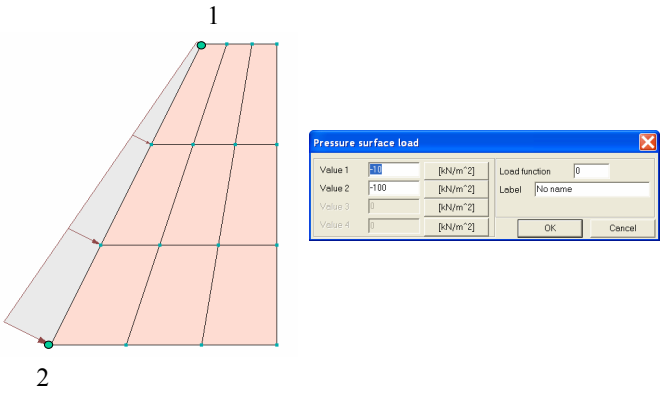

**Remark**:

For fast selection purposes one may apply a distinct label to this definition

Window 2-623

## Window 2-624: Surface load:PRESSURE:2 nodes

Using this option one may define variable surface pressure load by clicking 2 nodes on the boundary of a 2D mesh. These 2 nodes constitute a linear super-element. Pressure is applied at all element edges which belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. The dialog box used for setting this load is shown in figure below.




**Remarks**:

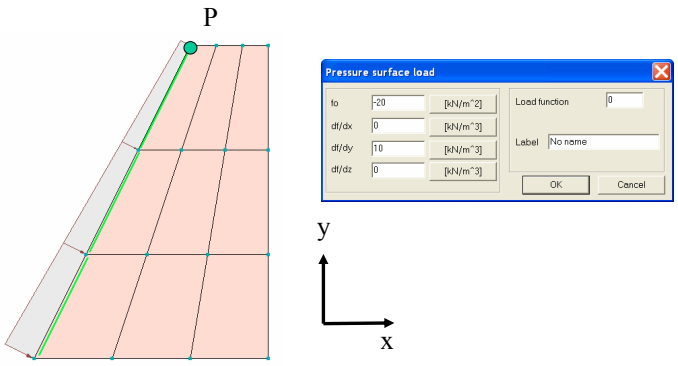
1. For fast selection purposes one may apply a distinct label to this definition
2. Positive pressure is directed towards external element normals

Window 2-624

## Window 2-625: Surface load:PRESSURE:On selected edges(HP)

Using this option one may apply distributed pressure load by picking the point at which value of the pressure $f_o$ is defined, and setting pressure gradients in $x$ and $y$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the pressure at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $p(\mathbf{x}) = f_o + \frac{df}{dx}\cdot(x - x_P) + \frac{df}{dy}\cdot(y - y_P)$.

This option is valid only for 2D models.



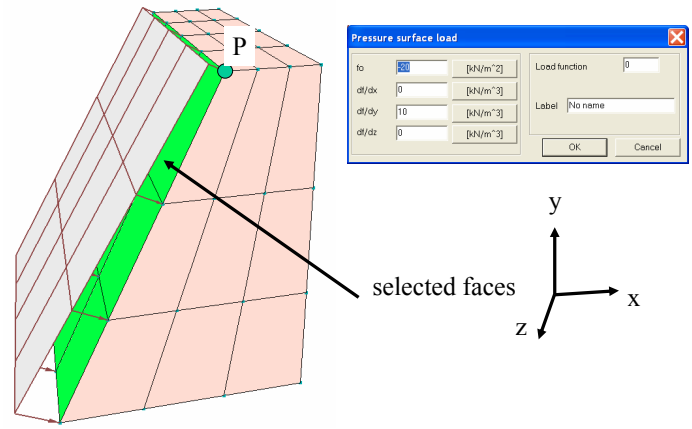

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. Positive pressure is directed towards external normal to the edges

Window 2-625

**Window 2-626:** `Surface load:PRESSURE:On selected faces(HP)`

Using this option one may apply distributed pressure load by picking the point at which value of the pressure $f_o$ is defined, and setting pressure gradients in $x$, $y$ and $z$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the pressure at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $p(\mathbf{x}) = f_o + \frac{df}{dx} \cdot (x - x_P) + \frac{df}{dy} \cdot (y - y_P) + \frac{df}{dz} \cdot (z - z_P)$. This option is valid only for 3D models.



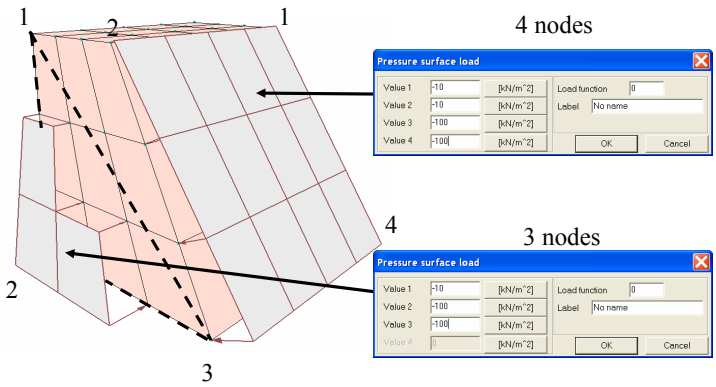

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. Positive pressure is directed towards external normal to the faces

Window 2-626

**Window 2-627:** `Surface load:PRESSURE:3 nodes (P), 4 nodes (P)`

Using this option one may define variable surface pressure load by clicking 3 or 4 nodes on the boundary of a 3D mesh. These 3/4 nodes constitute a triangular or quadrilateral super-element. Pressure is applied at all element faces which fully belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. The dialog box used for setting this load is shown in figure below.



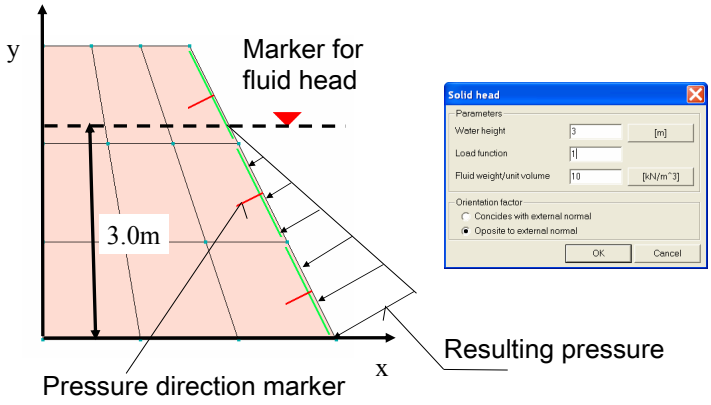

**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. Positive pressure is directed towards external normal to the boundary

Window 2-627

**Window 2-628:** `Surface load:PRESSURE:On selected edges via Fluid head (SH)`

Using this option one may apply pressure load to the selected edges using definition of the fluid head (potential). Fluid head $h$, is defined as $h = -\frac{p}{\gamma} + y$, where fluid specific weight is denoted by $\gamma$ and vertical coordinate by $y$. Distribution of the pressure applied to the edges (it is automatically computed inside of the calculation module), is derived from the above formula and filtered to cancel the pressure at any point located above elevation $h$. This definition when refining the mesh will automatically be projected on the new mesh and no additional action to be made by the user is needed.




**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. It is possible to apply the pressure towards or against the external normal

Window 2-628

**Window 2-629:** `Surface load:PRESSURE:On selected faces via Fluid head (SH)`

This option is fully compatible with the one designed for 2D models (`On selected edges via Fluid head (SH)`) except this that the pressure is applied to selected faces (see Window 2-628 for details) instead of edges.

Window 2-629

**Window 2-630:** `Surface load:Select:All`

This option selects all surface loads.

Window 2-630

**Window 2-631:** `Surface load:Select:With load f., existence f., label`

Using this option user may select surface load with a given load time function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-631

**Window 2-632:** `Surface load:Update:Parameters`

This option activates a dialog box to update the applied surface load description. The resulting dialog box, corresponds to the applied load type.

Window 2-632

**Window 2-633:** `Surface load:Update:Scale`

Using this option one may enlarge or diminish graphical size of the applied surface loads.

Window 2-633

## 2.3.1.3.19.4 BEAM LOAD

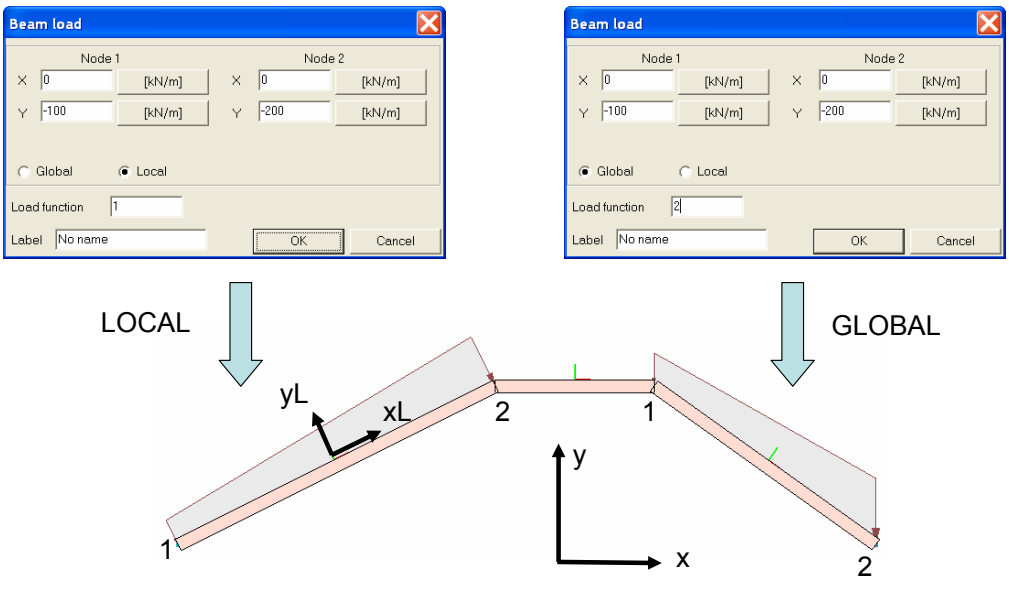
**Window 2-634: Options in menu** `Beam load`

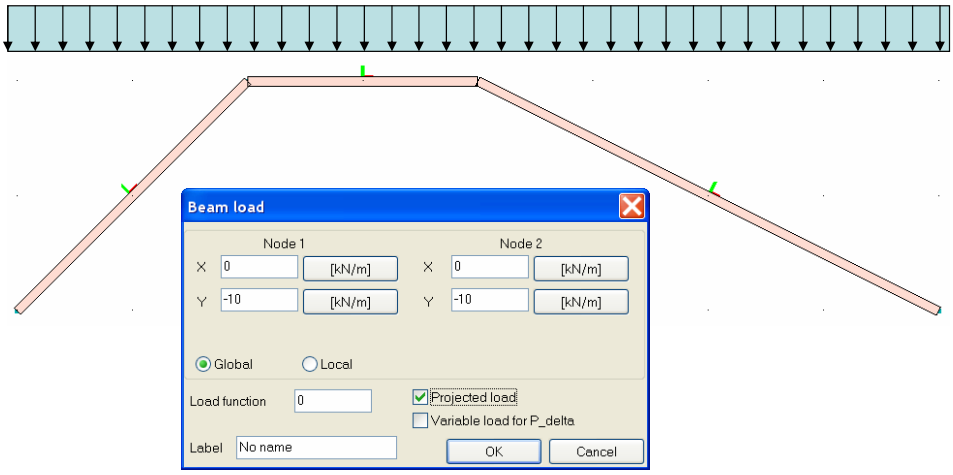
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `UNIFORM` | `On beam` | — | Window 2-635 |
| `Delete` | `Delete` |  | Window 2-320 |
| | `Delete all` | — | Window 2-636 |
| | `Delete for selected beams` | — | Window 2-637 |
| `Update` | `Parameters` | — | Window 2-638 |
| | `Scale` | — | Window 2-639 |

Window 2-634

**Window 2-635:** `Beam load:On beam(s)`

Using this option one may apply a distributed (variable) load to a single picked beam element or group of selected beam elements. This load can be applied in the global coordinate system or local one depending on whether ◎ **Local** or ◎ **Global** option is selected. The 2D setup is shown in the first fig. below (for the 3D $Z$ direction is additionally enabled). It is also possible to set distributed load along the beam as projected one. This setting is possible exclusively in the global coordinate system (see second figure below). It has to be mentioned that drawing of the projected load will look the same as for the standard global setting (!).






**Remarks**:

1. For fast selection purposes one may apply a distinct label to this definition
2. This load is automatically split during mesh refinement

Window 2-635

**Window 2-636:** `Beam load:Delete:Delete all`

The aim of this option is to remove all distributed loads applied to beam elements.

Window 2-636

**Window 2-637:** `Beam load:Delete:Delete for selected beams`

The aim of this option is to remove all distributed loads applied to selected beam elements.

Window 2-637

**Window 2-638:** `Beam load:Update:Parameters`

This option activates a dialog box to update the applied distributed load. The resulting dialog box is the same as for creation method `On beam(s)`.

Window 2-638

**Window 2-639:** `Beam load:Update:Scale`

Using this option one may enlarge or diminish graphical size of the applied distributed loads.

Window 2-639

## 2.3.1.3.19.5 POINT LOAD

**Window 2-640: Options in menu** `Point load on elements`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Point load` | — | Window 2-641 |
| `Select` | `All` | — | Window 2-642 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-643 |
| | `With existence function` | — | Window 2-643 |
| | `With label` | — | Window 2-643 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-644 |
| | `Scale` | — | Window 2-645 |

Window 2-640

**Window 2-641:** `Point load on elements:Create:Point load`

This option enables user to define an arbitrary point force(s)/moment(s) to be applied to set of target elements or their faces. This point load can move along defined movement path with a given constant or variable velocity. Defined point loading is projected on target elements or their faces/edges by means of the closest point projection. The dialog box to input the data is non-modal therefore each modification(s) has to be confirmed by pressing button **Apply**.

In this dialog box user defines three general groups of the data which consists of

- A.1 Generalized `force components` (in 2D $F_x$, $F_y$ and $M_z$ while in the 3D $F_x$, $F_y$, $F_z$ and $M_x$, $M_y$, $M_z$)
- A.2 `Point coordinates` ($x$, $y$ and $z$ (in 2D $z = 0$)); if the load is moving then this point is treated as the initial position
- A.3 Choice of the `target elements` or their edges/faces to which this loading is to be applied; this combo-box includes 8 possibilities

  1. `Automatically to structural element` (shells, beams, membranes, truss elements); here only one group of elements can be chosen therefore it is recommended to make an explicit selection of target elements; **this definition is not always fully safe as it may lead to some ambiguities especially when movement path is significantly shifted with respect to the position of the point load**
  2. `Automatically to shell elements`; here only group of shell elements are considered; **this definition is not always fully safe as it may lead to some ambiguities especially when movement path is significantly shifted with respect to the position of the point load**

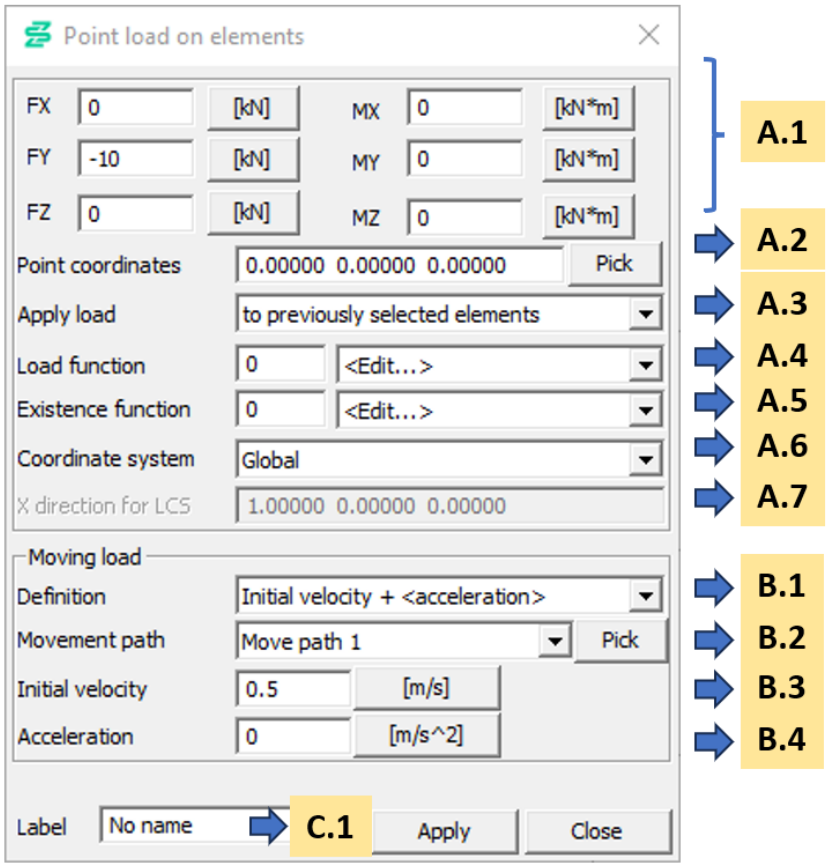



Figure 1: Graphic user interface to define point loads in the 3D

3. `Automatically to beam elements`; here only group of beam elements are considered; **this definition is not always fully safe as it may lead to some ambiguities especially when movement path is significantly shifted with respect to the position of the point load**
4. `Automatically to truss elements`; here only group of truss elements are considered; **this definition is not always fully safe as it may lead to some ambiguities especially when movement path is significantly shifted with respect to the position of the point load**
5. `Automatically to membrane elements`; here only group of membrane elements are considered; **this definition is not always fully safe as it may lead to some ambiguities especially when movement path is significantly shifted with respect to the position of the point load**
6. `Selected elements` (here selected elements of only one of the group are considered; these groups include shell, beam, truss and membrane elements); **this option is highly recommended as it avoids potential ambiguities and by limiting the size of target elements set it reduces CPU time spent by searching algorithms**
7. `to selected edges (2D) or faces(3D)` of continuum elements; **this option is highly recommended as it avoids potential ambiguities and by limiting the size of target elements set it reduces CPU time spent by searching algorithms**
8. `to previously selected elements` (structural) and/or their edges/faces (continuum)

- A.4 `Load time function index`; note that each load component is multiplied by the LTF$(t)$ value; if the load time function index is zero then all force components do not vary in time

- A.5 `Existence function index`; this loading can be activated/deactivated using the existence function; if the existence function index is zero then this loading is present when target elements are active; when the existence function index is nonzero then presence of the loading is controlled by the existence function of target elements as well as the existence function of the loading

- A.6 `Coordinate system` (frame) in which force/moment components are given; the possible choices for this frame are

  1. `Global`; here all components are given in global x-y-z coordinate system
  2. `Local`; in this case force/moment (if apply) components are defined in the element or its edge/face local coordinate system; in case of beam elements local frame is predefined when creating the element; in all other cases (in 3D) we need an additional vector to be set in block A.7; for comprehensive explanation on how the local element frames are defined for beam, truss, membrane, shell elements and continuum element edges(2D)/faces(3D) please refer to the report Rep-Moving_Loads_Mass.pdf
  3. `Local along movement path`; for detailed information on how this local frame is set please refer to the dedicated subsection in report Rep-Moving_Loads_Mass.pdf note that this definition is valid for shell/membrane 3D elements and 3D continuum element faces

- A.7 `X direction for LCS`; this is an auxiliary vector which is needed (in some cases) when we set local frames in 3D; a comprehensive explanation on how the local element frames are defined for truss elements please refer to dedicated subsection in report Rep-Moving_Loads_Mass.pdf, for shells, membranes 3D and continuum 3D faces to the corresponding subsection in report Rep-Moving_Loads_Mass.pdf,

- B.1 `Definition` of the time evolution of the velocity when the load is moving; here the 3 choices are possible

  1. `Inactive`; this means that the load is stationary; in this case data blocks B2, B.3 and B.4 are not meaningful
  2. `Initial velocity + < acceleration >`; if this option is selected then in B.3 please put the initial velocity value $v_0$ and acceleration $a$ in block $B.4$; the current velocity value is computed then using the formula $v(t) = v_0 + a \cdot t^*$; in this formula time $t^* = t - t_{\text{init}}$ where $t_{\text{init}} = 0$, if the load existence function index is 0, or $t_{\text{init}}$ is equal to the time instance at which the load existence function becomes active
  3. `User defined velocity`; if this option is selected then the graphical user interface is slightly modified as shown in the figure below; evolution of the velocity in time is driven now by the selected load time function $v(t) = v_{\text{ref}} \cdot \text{LTF}(t^*)$; as it has been done for the previous option current velocity value is computed in fact for the corrected time value $t^* = t - t_{\text{init}}$

- B.2 `Movement path`; here a predefined movement path has to be selected from the combo-box; this option is inactive when B.1 is set to `Inactive`

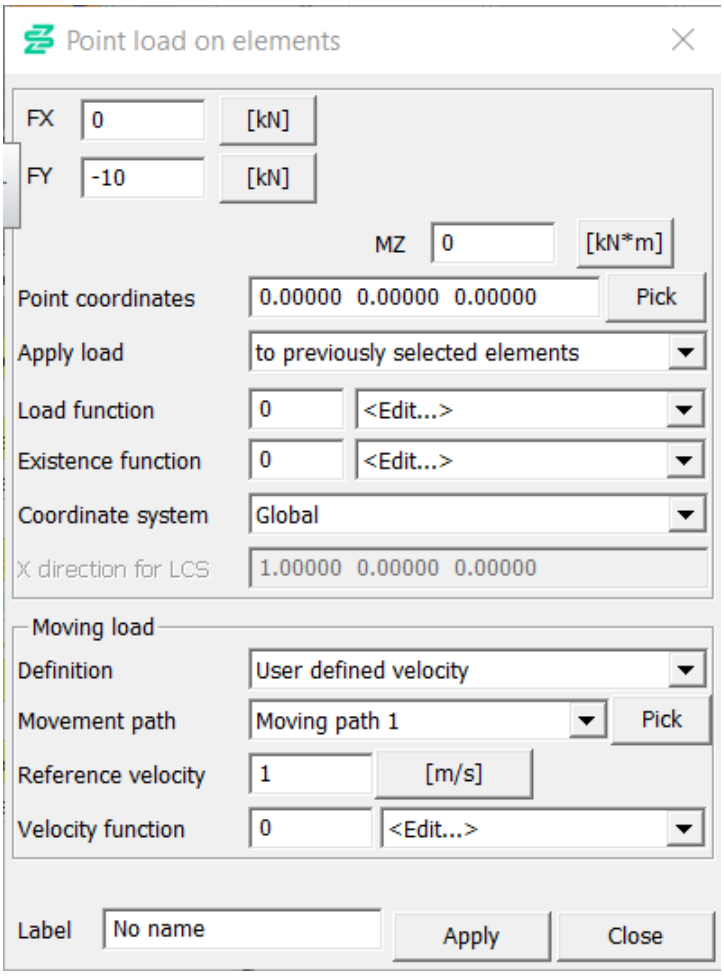



Figure 2: Graphic user interface to define point loads in the 3D when user defined velocity option is selected

- B.3 `Initial/Reference velocity`; this data is unused when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` this data is the initial value of velocity $v_0$; if B.1 is set to `User defined velocity` this declared value plays a role of the reference velocity that is multiplied then by a selected load time function value to be declared in B.4 edit;
- B.4 `Acceleration/Velocity function`; this option is inactive when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` then this data corresponds to the `Acceleration/` value; if B.1 is set to `User defined velocity` then this data corresponds to the `Velocity function` which is the index of the load time function which applies to the reference velocity;
- C.1 `Label`; here we can add a unique label to the point load

Window 2-641

**Window 2-642:** `Point load on elements:Select:All`

This option selects all point loads.

Window 2-642

**Window 2-643:** `Point load on elements:Select:With load f., existence f., label`

Using this option user may select point load with a given load time function ID, existence function ID or a given label.

Window 2-643

**Window 2-644:** `Point load on elements:Update:Parameters`

This option activates the dialog box to modify the applied point load description. It can be used only for a single picked or selected point load. The detailed description on how to set the data is given in Window 2-641.

Window 2-644

**Window 2-645:** `Point load on elements:Update:Scale`

Using this option one may enlarge or diminish size of vectors representing point loads or arcs that represent moments.

Window 2-645

## 2.3.1.3.19.6 LINE LOAD

**Window 2-646: Options in menu** `Line load on elements`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Line load` | — | Window 2-647 |
| `Select` | `All` | — | Window 2-648 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-649 |
| | `With existence function` | — | Window 2-649 |
| | `With label` | — | Window 2-649 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-650 |
| | `Scale` | — | Window 2-651 |

Window 2-646

**Window 2-647:** `Line load on elements:Create:Line load`

This option enables user to define an arbitrary linearly distributed force(s)/moment(s) to be applied to set of target elements or their faces. The line load load can move along defined movement path with a given constant or variable velocity. Defined line loading is projected on target elements or their faces/edges by means of the closest point projection. The dialog box to input the data is non-modal therefore each modification(s) has to be confirmed by pressing button **Apply**.

In this dialog box user defines three general groups of the data which consists of

- A.1 `Distributed force/moments components` at two line segment endpoints (distributed forces $f_{x,1}$, $f_{y,1}$, $f_{z,1}$, $f_{x,2}$, $f_{y,2}$, $f_{z,2}$ and distributed moments $m_{x,1}$, $m_{y,1}$, $m_{z,1}$, $m_{x,2}$, $m_{y,2}$, $m_{z,2}$)
- A.2 `Coordinates of the first point` ($x$, $y$ and $z$ (in 2D $z = 0$)); if the load is moving then these coordinates are treated as the initial position
- A.3 `Coordinates of the second point` ($x$, $y$ and $z$ (in 2D $z = 0$)); if the load is moving then these coordinates are treated as the initial position
- A.4 Choice of the `target elements` or their edges/faces to which this loading is to be applied; this combo-box includes 8 possibilities

  1. `Automatically to structural element` (shells, beams, membranes, truss elements); here only one group of elements can be chosen therefore it is recommended to make an explicit selection of target elements
  2. `Automatically to shell elements`; here only group of shell elements are considered

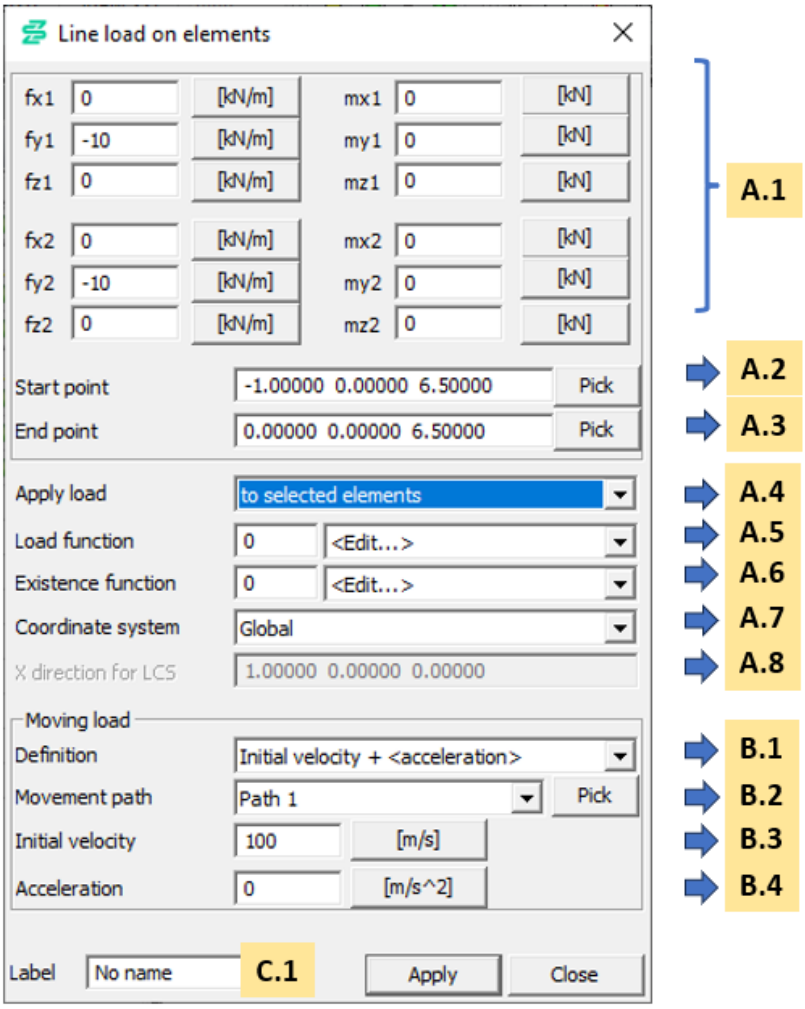



Figure 1: Graphic user interface to define line load in 3D

3. `Automatically to beam elements`; here only group of beam elements are considered
4. `Automatically to truss elements`; here only group of truss elements are considered
5. `Automatically to membrane elements`; here only group of membrane elements are considered
6. `Selected elements` (here selected elements of only one of the group are considered; these groups include shell, beam, truss and membrane elements)
7. `to selected edges (2D) or faces(3D)` of continuum elements
8. `to previously selected elements` (structural) and/or their edges/faces (continuum)

- A.5 `Load time function index`; note that each load component is multiplied by the LTF$(t)$ value; if the load time function index is zero then all force/moment components do not vary in time
- A.6 `Existence function index`; this loading can be activated/deactivated using the existence function; if the existence function index is zero then this loading is present when target elements are active; when the existence function index is nonzero then presence of the loading is controlled by the existence function of target elements as well as the existence function of the loading

- A.7 `Coordinate system` (frame) in which force/moment components are given; the possible choices for this frame are

  1. `Global`; here all components are given in global x-y-z coordinate system
  2. `Local`; in this case force/moment (if apply) components are defined in the element or its edge/face local coordinate system; in case of beam elements local frame is predefined when creating the element; in all other cases (in 3D) we need an additional vector to be set in block A.8;

     for comprehensive explanation on how the local element frames are defined for beam, truss, membrane, shell elements and continuum element edges(2D)/faces(3D) please refer to the report Rep-Moving_Loads_Mass.pdf
  3. `Local along movement path`; for detailed information on how this local frame is set please refer to the dedicated subsection in report Rep-Moving_Loads_Mass.pdf note that this definition is valid for shell/membrane 3D elements and 3D continuum element faces

- A.8 `X direction for LCS`; this is an auxiliary vector which is needed (in some cases) when we set local frames in 3D; a comprehensive explanation on how the local element frames are defined for truss elements please refer to dedicated subsection in report Rep-Moving_Loads_Mass.pdf, for shells, membranes 3D and continuum 3D faces to the corresponding subsection in report Rep-Moving_Loads_Mass.pdf,

- B.1 `Definition` of the time evolution of the velocity when the load is moving; here the 3 choices are possible

  1. `Inactive`; this means that the load is stationary; in this case data blocks B2, B.3 and B.4 are not meaningful
  2. `Initial velocity + < acceleration >`; if this option is selected then in B.3 please put the initial velocity value $v_0$ and acceleration $a$ in block $B.4$; the current velocity value is computed then using the formula $v(t) = v_0 + a \cdot t^*$; in this formula time $t^* = t - t_{\text{init}}$ where $t_{\text{init}} = 0$, if the load existence function index is 0, or $t_{\text{init}}$ is equal to the time instance at which the load existence function becomes active
  3. `User defined velocity`; if this option is selected then the user interface is slightly modified as shown in the figure below; evolution of the velocity in time is driven now by the selected load time function $v(t) = v_{\text{ref}} \cdot \text{LTF}(t^*)$; as it has been done for the previous option current velocity value is computed in fact for the corrected time value $t^* = t - t_{\text{init}}$

- B.2 `Movement path`; here a predefined movement path has to be selected from the combo-box; this option is inactive when B.1 is set to `Inactive`

- B.3 `Initial/Reference velocity`; this data is unused when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` this data is the initial value of velocity $v_0$; if B.1 is set to `User defined velocity` this declared value plays a role of the reference velocity that is multiplied then by a selected load time function value to be declared in B.4 edit;

- B.4 `Acceleration/Velocity function`; this option is inactive when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` then this data corresponds to the `Acceleration/` value; if B.1 is set to `User defined velocity` then

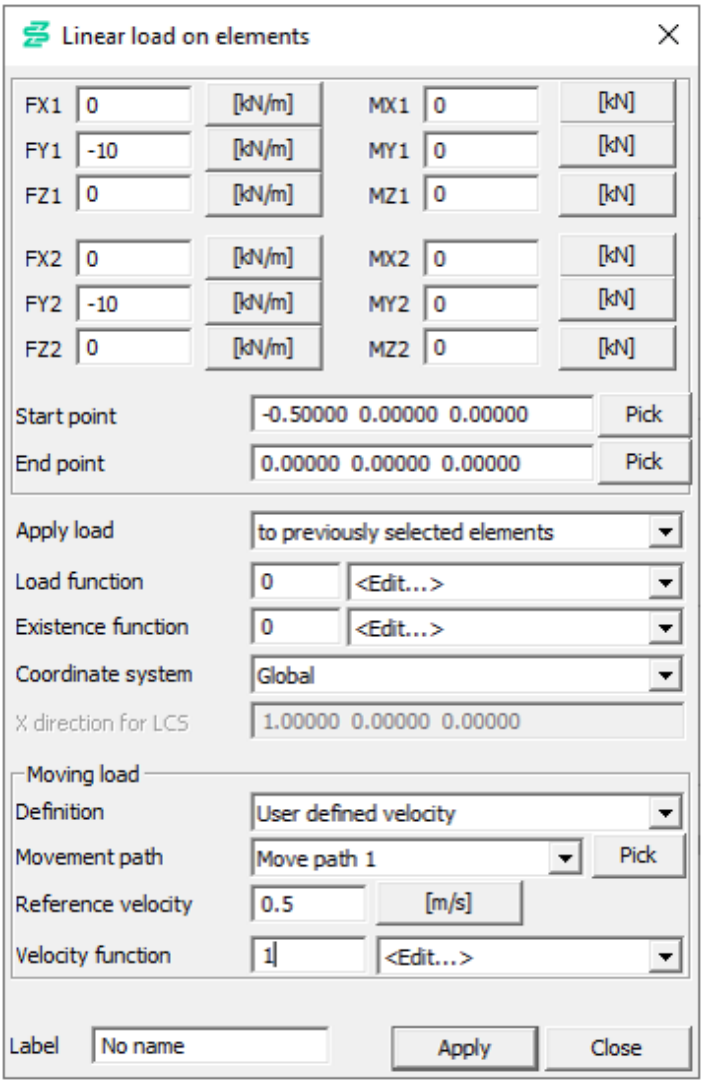



Figure 2: Graphic user interface to define line loads in the 3D when user defined velocity option is selected

this data corresponds to the `Velocity function` which is the index of the load time function which applies to the reference velocity;

- C.1 `Label`; here we can add a unique label to the line load

Window 2-647

**Window 2-648:** `Line load on elements:Select:All`

This option selects all line loads.

Window 2-648

**Window 2-649:** `Line load on elements:Select:With load f., existence f., label`

Using this option user may select line load with a given load time function ID, existence function ID or a given label.

Window 2-649

**Window 2-650:** `Line load on elements:Update:Parameters`

This option activates the dialog box to modify the applied line load description. It can be used only for a single picked or selected line load. The detailed description on how to set the data is given in Window 2-647.

Window 2-650

**Window 2-651:** `Line load on elements:Update:Scale`

Using this option one may enlarge or diminish size of the graphical objects representing line loading.

Window 2-651

## 2.3.1.3.19.7 SURFACE LOAD

**Window 2-652: Options in menu** `Surface load on elements`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Surface load` | — | Window 2-653 |
| `Select` | `All` | — | Window 2-654 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-655 |
| | `With existence function` | — | Window 2-655 |
| | `With label` | — | Window 2-655 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-656 |
| | `Scale` | — | Window 2-657 |

Window 2-652

**Window 2-653:** `Surface load on elements:Create:Surface load`

This option enables user to define an arbitrary load density defined on a quasi-planar polygon to be applied to set of target elements or their faces. The surface load load can move along defined movement path with a given constant or variable velocity. Defined surface loading is projected on target elements or their faces/edges by means of the closest point projection. The dialog box to input the data is non-modal therefore each modification(s) has to be confirmed by pressing button **Apply**.

In this dialog box user defines three general groups of the data which consists of

- A.1 `Selected force component` (only one $X$, $Y$ or $Z$ component can be selected)
- A.2 `Distributed force parameters` ($p_0$, $\frac{\partial p}{\partial x}$, $\frac{\partial p}{\partial y}$ and $\frac{\partial p}{\partial z}$)
- A.3 `Reference point coordinates` ($x$, $y$ and $z$); these coordinates appear in the general formula for the load description $p(x, y, z)$
- A.4 `Set of polygon vertices`; these points can be selected from the existing mesh objects or entered manually; if the load is moving then these points correspond to the initial polygon position; note that this polygon can be located outside the mesh
- A.5 Choice of the `target elements` or their faces to which this load is to be applied; this combo-box includes 8 possibilities

  1. `Automatically to structural element` (shells, beams, membranes, truss elements); here only one group of elements can be chosen therefore it is recommended to make an explicit selection of target elements

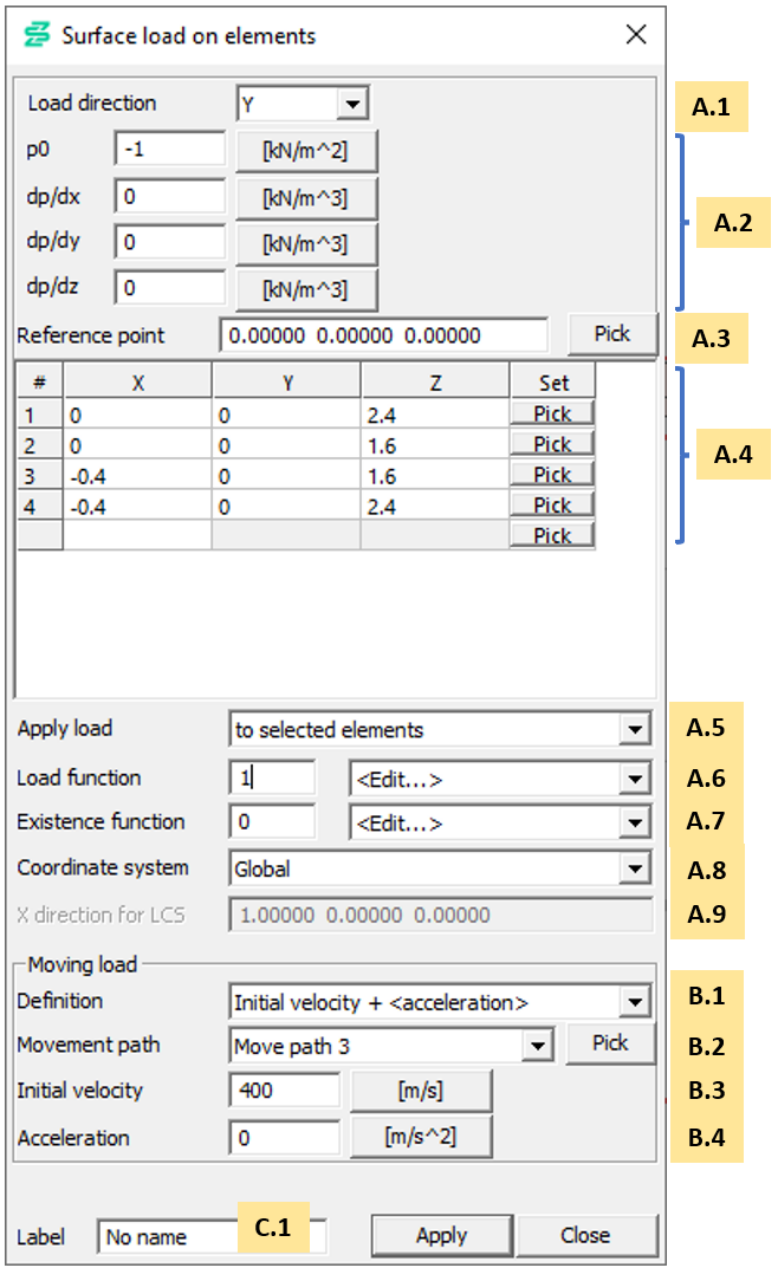



Figure 1: Graphic user interface to define surface load

2. `Automatically to shell elements`; here only group of shell elements are considered
3. `Automatically to beam elements`; here only group of beam elements are considered
4. `Automatically to truss elements`; here only group of truss elements are considered
5. `Automatically to membrane elements`; here only group of membrane elements are considered
6. `Selected elements` (here selected elements of only one of the group are considered; these groups include shell, beam, truss and membrane elements)
7. `to selected faces(3D)` of continuum elements
8. `to previously selected elements` (structural) and/or their faces (continuum)

- A.6 `Load time function index`; note that the load component is multiplied by the LTF($t$) value; if the load time function index is zero then the load does not change in time

- A.7 `Existence function index`; this load can be activated/deactivated using the existence function; if the existence function index is zero then this load is present when target elements are active; when the existence function index is nonzero then presence of the loading is controlled by the existence function of target elements as well as the existence function of the loading

- A.8 `Coordinate system` (frame) in which the load will be applied; the possible choices for this frame are

  1. `Global`; here selected load direction is defined in global x-y-z coordinate system
  2. `Local`; in this case selected force component is defined in the element or its face local coordinate system; in case of beam elements local frame is predefined when creating the element; in all other cases (in 3D) we need an additional vector to be set in block A.9;

     for comprehensive explanation on how the local element frames are defined for beam, truss, membrane, shell elements and continuum element edges(2D)/faces(3D) please refer to the report Rep-Moving_Loads_Mass.pdf
  3. `Local along movement path`; for detailed information on how this local frame is set please refer to the dedicated subsection in report Rep-Moving_Loads_Mass.pdf note that this definition is valid for shell/membrane 3D elements and 3D continuum element faces

- A.9 `X direction for LCS`; this is an auxiliary vector which is needed (in some cases) when we set local frames in 3D;

  a comprehensive explanation on how the local element frames are defined for truss elements please refer to dedicated subsection in report Rep-Moving_Loads_Mass.pdf, for shells, membranes 3D and continuum 3D faces to the corresponding subsection in report Rep-Moving_Loads_Mass.pdf,

- B.1 `Definition` of the time evolution of the velocity when the load is moving; here the 3 choices are possible

  1. `Inactive`; this means that the load is stationary; in this case data blocks B2, B.3 and B.4 are not meaningful
  2. `Initial velocity + < acceleration >`; if this option is selected then in B.3 please put the initial velocity value $v_0$ and acceleration $a$ in block $B.4$; the current velocity value is computed then using the formula $v(t) = v_0 + a \cdot t^*$; in this formula time $t^* = t - t_{\text{init}}$ where $t_{\text{init}} = 0$, if the load existence function index is 0, or $t_{\text{init}}$ is equal to the time instance at which the load existence function becomes active
  3. `User defined velocity`; if this option is selected then the user interface is slightly modified as shown in the figure below; evolution of the velocity in time is driven now by the selected load time function $v(t) = v_{\text{ref}} \cdot \text{LTF}(t^*)$; as it has been done for the previous option current velocity value is computed in fact for the corrected time value $t^* = t - t_{\text{init}}$

- B.2 `Movement path`; here a predefined movement path has to be selected from the combo-box; this option is inactive when B.1 is set to `Inactive`

- B.3 `Initial/Reference velocity`; this data is unused when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` this data is the initial value of velocity $v_0$; if B.1 is set to `User defined velocity` this declared value plays a role of

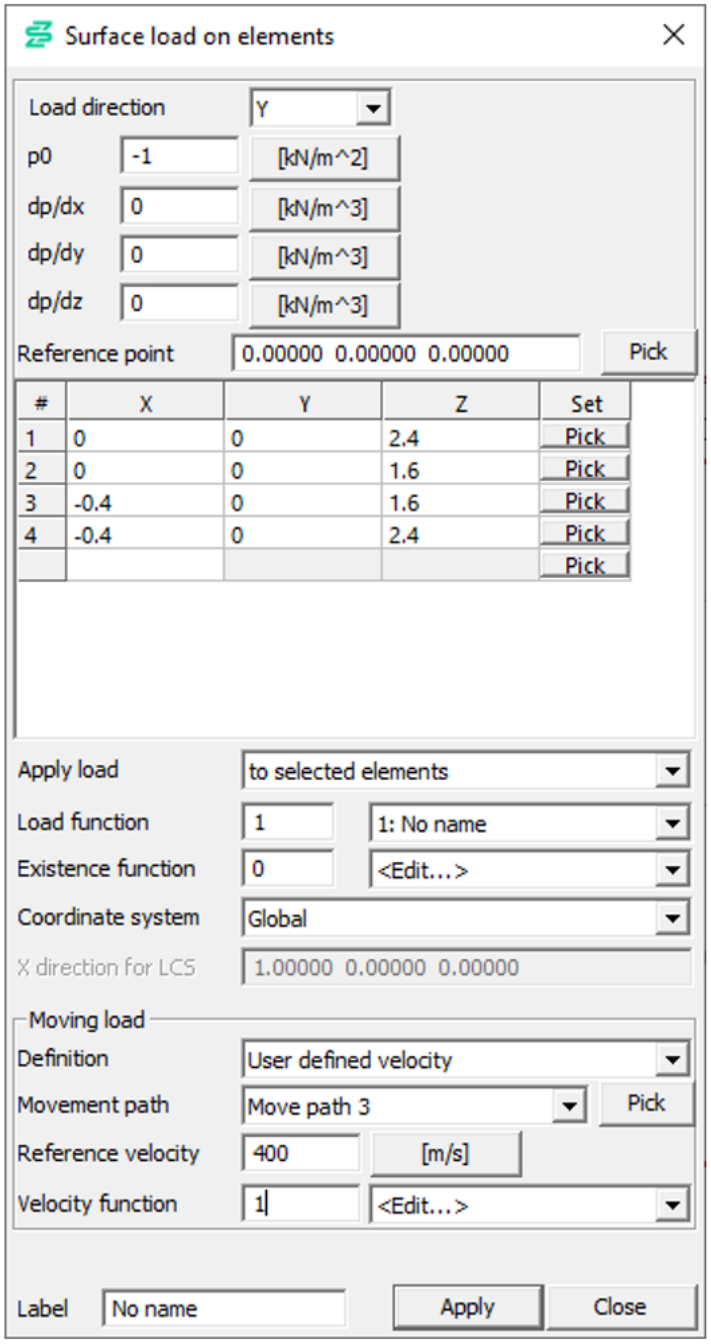



Figure 2: Graphic user interface to define surface loads in the 3D when user defined velocity option is selected

the reference velocity that is multiplied then by a selected load time function value to be declared in B.4 edit;

- B.4 `Acceleration/Velocity function`; this option is inactive when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` then this data corresponds to the `Acceleration/` value; if B.1 is set to `User defined velocity` then this data corresponds to the `Velocity function` which is the index of the load time function which applies to the reference velocity;

- C.1 `Label`; here we can add a unique label to the surface load

Window 2-653

**Window 2-654:** `Surface load on elements:Select:All`

This option selects all arbitrary surface loads.

Window 2-654

**Window 2-655:** `Surface load on elements:Select:With load f., existence f., label`

Using this option user may select line load with a given load time function ID, existence function ID or a given label.

Window 2-655

**Window 2-656:** `Surface load on elements:Update:Parameters`

This option activates the dialog box to modify the applied surface load description. It can be used only for a single picked or selected surface load. The detailed description on how to set the data is given in Window 2-653.

Window 2-656

**Window 2-657:** `Surface load on elements:Update:Scale`

Using this option one may enlarge or diminish size of the graphical objects representing line loading.

Window 2-657

## 2.3.1.3.20 ADDED MASSES

Added masses can be modeled using two options:

| | |
|---|---|
| Nodal added masses | Create masses on selected nodes |
| Distributed added masses | Create masses distributed on element boundaries |
| Point masses | Create moving masses at an arbitrary point |
| Line masses | Create moving masses linearly distributed along an arbitrary line segment |
| Surface masses | Create moving surface masses on arbitrarily defined quasi-planar polygons |

It is possible to filter added masses to eliminate interia terms along selected direction(s). This can be made in the global or local coordinate system (see paragraph FILTERING OF ADDED MASSES )

## 2.3.1.3.20.1 NODAL ADDED MASSES

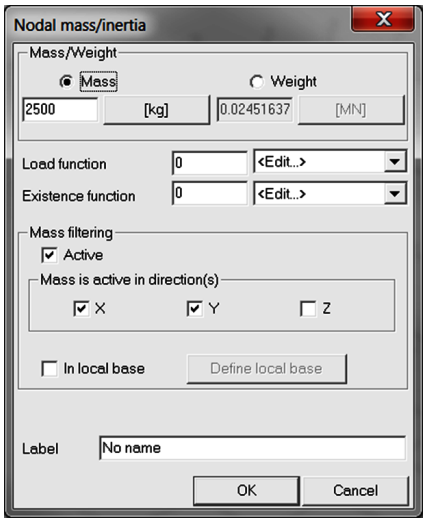
**Window 2-658: Options in menu** `Nodal mass`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node` | — | Window 2-659 |
| | `On element node` | — | Window 2-660 |
| `Select` | `All` | — | Window 2-661 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-662 |
| | `With label` | — | Window 2-662 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-663 |

Window 2-658

**Window 2-659:** `Nodal mass:Create:On node`

Using this option one may add an additional concentrated mass to a single picked node or group of selected nodes (if current selection list for nodes is not empty).




**Remarks**:

1. For fast selection purposes one may apply a distinct label to the edited mass
2. Both load time and existence functions can be associated with the added nodal mass
3. The added mass can be defined directly or through the equivalent weight

Window 2-659

**Window 2-660:** `Nodal mass:Create:On element node`

Using this option one may apply the additional mass to the selected element node(s). This is crucial in case when node belongs to the contact zone and mass is to be set only at node at one side of the interface. Further steps to define the mass are same as in the standard case (see Window 2-659)

Window 2-660

**Window 2-661:** `Nodal mass:Select:All`

This option selects all added nodal masses.

Window 2-661

**Window 2-662:** `Nodal mass:Select:With load f., existence f., label`

Using this option user may select added nodal masses with a given load time function, existence function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-662

**Window 2-663:** `Nodal mass:Update:Parameters`

This option activates the dialog box to modify the applied added nodal masses description, for a single picked node or group of selected nodes. The detailed description on how to set the data is given in Window 2-659.

**Remark**:When running this option for group of selected nodes only default parameters will be set in the dialog box.

Window 2-663

## 2.3.1.3.20.2 DISTRIBUTED ADDED MASSES

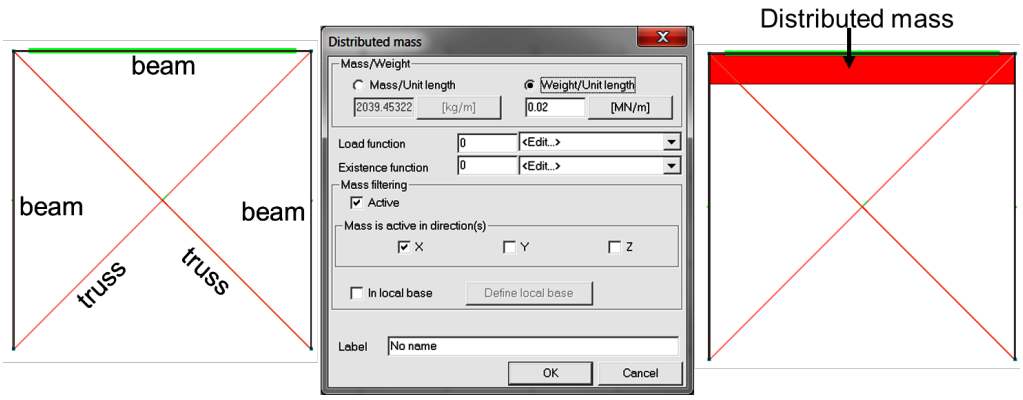
**Window 2-664: Options in menu** `Distributed mass`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On edge(s)` | — | Window 2-665 |
| | `On face(s)` | — | Window 2-666 |
| `Select` | `All` | — | Window 2-667 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-668 |
| | `With label` | — | Window 2-668 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-669 |

Window 2-664

**Window 2-665:** `Distributed mass:Create:On edge(s)`

Using this option one may add an additional distributed mass to a single picked element edge (can be beam, truss or membrane) or group of selected edges (if current selection list for edges is not empty). This option is valid for both 2D and 3D models.



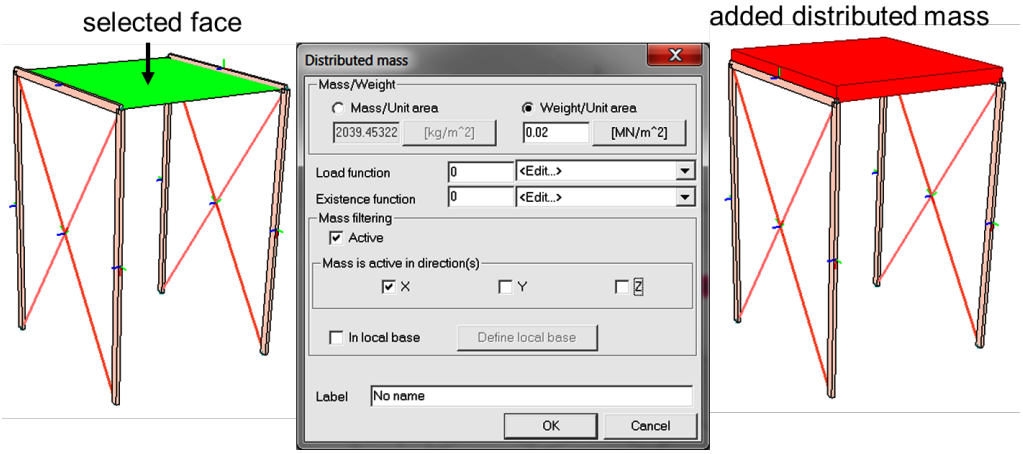

**Remarks**:

1. For fast selection purposes one may apply a distinct label to the edited mass
2. Both load time and existence functions can be associated with the added distributed mass
3. Application of distributed masses along 3D surface elements (shells/membranes) edges or 2D/3D continuum elements is not supported
4. 
5. The density of added mass can be defined directly or through the equivalent weight per unit length/surface

Window 2-665

**Window 2-666:** `Distributed mass:Create:On face(s)`

Using this option one may add an additional distributed mass to a single picked element face (can be shell or membrane in 3D) or group of selected faces (if current selection list for faces is not empty). This option is valid exclusively for 3D models.




**Remarks**:
1. For fast selection purposes one may apply a distinct label to the edited mass
2. Both load time and existence functions can be associated with the added distributed mass
3. Applying distributed masses to faces of 3D continuum elements is not supported

Window 2-666

**Window 2-667:** `Distributed mass:Select:All`

Using this option one may select all added distributed masses.

Window 2-667

**Window 2-668:** `Distributed mass:Select:With load f., existence f., label`

Using this option user may select added distributed masses with a given load time function, existence function ID or a given label. In case of a label one may select a single item from combo-box.

Window 2-668

**Window 2-669:** `Distributed mass:Update:Parameters`

This option activates the dialog box to modify the applied added distributed masses description, for a single picked edge/face or group of selected edges/faces. The detailed description on how to set the data is given in Window 2-665 or Window 2-666.

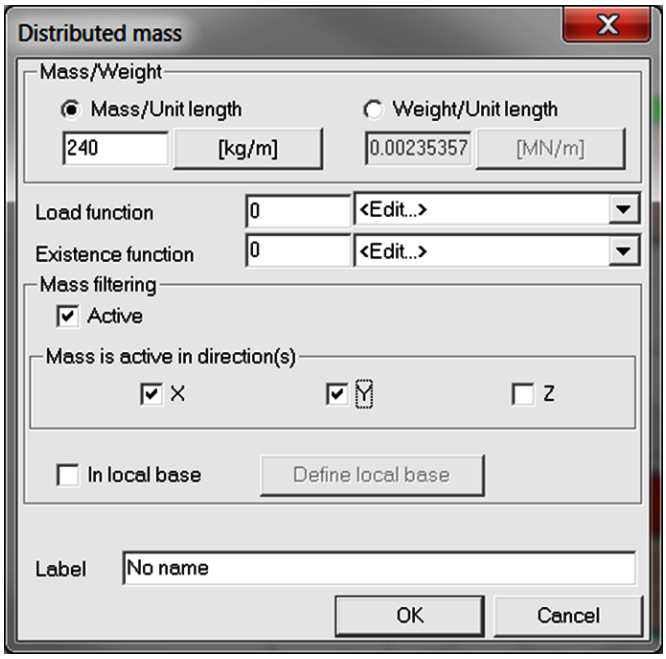
**Remarks**:
1. When running this option for group of selected edges/faces only default parameters will be set in the dialog box
2. If selection list contains added masses defined on both, edges and faces, then the operation can exclusively be run for masses added on edges or on faces

Window 2-669

## 2.3.1.3.20.3 FILTERING OF ADDED MASSES

**Window 2-670:** `Filtering added masses:in global coordinate system`

This setting is common for added masses applied as nodal or distributed along edges or faces. All the required data is isolated by the groupbox `Mass filtering` in dialog boxes for creating of added masses.



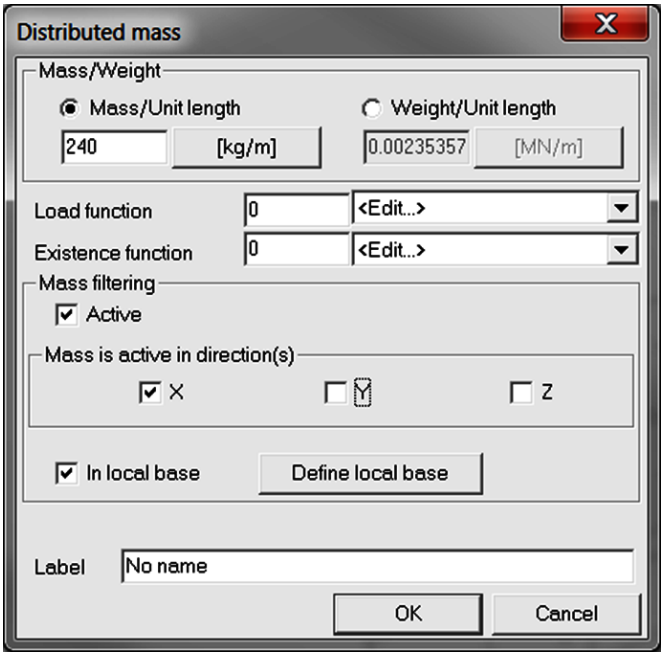

**Remarks**:

1. To filter masses in the global coordinate system set ON the ☒ **Active** while state of the checkbox ☒ **In local system** must be OFF
2. If the state of the checkbox is ON then the mass is present at this selected direction

Window 2-670

**Window 2-671:** `Filtering added masses:in local coordinate system`

This setting is common for added masses applied as nodal or distributed along edges or faces. All the required data is isolated by the groupbox `Mass filtering` in dialog boxes for creating of added masses.




**Remarks**:

1. To filter masses in the local coordinate system set ON state of the checkboxes ☒ **Active** and ☒ **In local system**
2. To filter certain directions user may set up mass multipliers for X,Y and Z (3D) directions independently in edit fields under column `Factors`
3. To set up the local coordinate system press button **Define local base**
4. Local axes can be defined using 4 definitions (used also to create solid BC in rotated local axes):
5. In the 2D local coordinate system can be set through:
   - Angle – Window 2-555
   - Cylindrical system – Window 2-556
   - By vector – Window 2-557
6. In the 3D local coordinate system can be set as:
   - Cartesian local system – Window 2-558
   - Cylindrical system – Window 2-559
   - By vector – Window 2-560; in this case local Y,Z axes are random hence filtering along these directions should be totally canceled (multipliers for Y,Z directions should be equal to 0.0) or left as active (both multipliers equal to 1.0)

Window 2-671

## 2.3.1.3.20.4 POINT MASS ON ELEMENTS

**Window 2-672: Options in menu** `Point mass on elements`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Point mass` | — | Window 2-673 |
| `Select` | `All` | — | Window 2-674 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-675 |
| | `With existence function` | — | Window 2-675 |
| | `With label` | — | Window 2-675 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-676 |
| | `Scale` | — | Window 2-677 |

Window 2-672

**Window 2-673:** `Point mass on elements:Create:Point mass`

This option enables user to define an arbitrary point force(s)/moment(s) to be applied to set of target elements or their faces. This point mass can move along defined movement path with a given constant or variable velocity. Defined point mass is projected on target elements or their faces/edges by means of the closest point projection. The dialog box to input the data is non-modal therefore each modification(s) has to be confirmed by pressing button **Apply**.

In this dialog box, the user defines three general groups of data, consisting of:

- A.1 `Mass value`, which can also be expressed in terms of weight.
- A.2 `Point coordinates` ($x$, $y$, and $z$; in 2D, $z = 0$). If the mass is moving, this point is treated as the initial position.
- A.3 Choice of `target elements` or their edges/faces to which this mass is to be applied. This combo box includes 8 options:

  1. `Automatically to structural element` (shells, beams, membranes, truss elements). Only one group of elements can be chosen here, so an explicit selection of target elements is recommended.
  2. `Automatically to shell elements`; only shell elements are considered.
  3. `Automatically to beam elements`; only beam elements are considered.
  4. `Automatically to truss elements`; only truss elements are considered.
  5. `Automatically to membrane elements`; only membrane elements are considered.

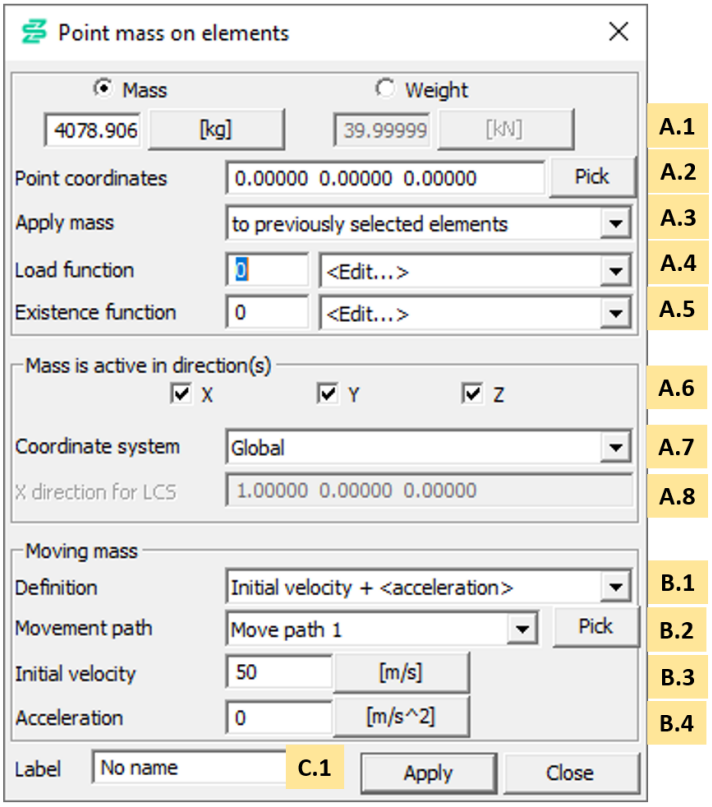



Figure 1: Graphic user interface to define point mass in the 3D

6. `to selected elements`; only selected elements from one group (shell, beam, truss, or membrane elements) are considered.
7. `to selected edges (2D) or faces (3D)` of continuum elements.
8. `to previously selected elements` (structural) and/or their edges/faces (continuum).

- A.4 `Load time function index`. Note that the mass value is multiplied by the LTF$(t)$ value. If the load time function index is zero, the mass value remains constant over time. Otherwise, it may vary, although this option is rarely used in practical cases.
- A.5 `Existence function index`. This mass can be activated/deactivated using the existence function. If the existence function index is zero, the mass is present when target elements are active. If the index is nonzero, the mass's presence is controlled by both the existence function of the target elements and the applied mass.
- A.6 `Active spatial directions for inertial effects`. Here, the user can select the global/local directions in which inertial effects are active.
- A.7 `Coordinate system for mass filtering` (frame), in which mass filtering is defined. This setting is meaningful when at least one direction is inactive. Possible choices for the frame are:

1. `Global`; mass filtering is done in the global x-y-z coordinate system.
2. `Local`; in this case, mass filtering is done in the element or its edge/face local coordinate system. For beam elements, the local frame is predefined during element creation. In all other cases (in 3D), an additional vector must be set in block A.8.

for comprehensive explanation on how the local element frames are defined for beam, truss, membrane, shell elements and continuum element edges(2D)/faces(3D) please refer to the report Rep-Moving_Loads_Mass.pdf

3. `Local along movement path`; for detailed information on how this local frame is set please refer to the dedicated subsection in report Rep-Moving_Loads_Mass.pdf This definition applies to shell/membrane 3D elements and 3D continuum element faces.

- A.8 `X direction for LCS`. This auxiliary vector is needed in some cases when setting local frames in 3D.

  for comprehensive explanation on how the local element frames are defined for truss elements please refer to dedicated subsection in report Rep-Moving_Loads_Mass.pdf, for shells, membranes 3D and continuum 3D faces to the corresponding subsection in report Rep-Moving_Loads_Mass.pdf,

- B.1 `Definition` of the time evolution of the velocity when the mass is moving; here the 3 choices are possible

  1. `Inactive`; this means that the mass is stationary; in this case data blocks B2, B.3 and B.4 are not meaningful
  2. `Initial velocity + < acceleration >`; if this option is selected then in B.3 please put the initial velocity value $v_0$ and acceleration $a$ in block $B.4$; the current velocity value is computed then using the formula $v(t) = v_0 + a \cdot t^*$; in this formula time $t^* = t - t_{\text{init}}$ where $t_{\text{init}} = 0$, if the load existence function index is 0, or $t_{\text{init}}$ is equal to the time instance at which the load existence function becomes active
  3. `User defined velocity`; if this option is selected then the user interface is slightly modified as shown in the figure below; evolution of the velocity in time is driven now by the selected load time function $v(t) = v_0 \cdot \text{LTF}(t^*)$; as it has been done for the previous option current velocity value is computed in fact for the corrected time value $t^* = t - t_{\text{init}}$

- B.2 `Movement path`; here a predefined movement path has to be selected from the combo-box; this option is inactive when B.1 is set to `Inactive`

- B.3 `Initial/Reference velocity`; this data is unused when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` this data is the initial value of velocity $v_0$; if B.1 is set to `User defined velocity` this declared value plays a role of the reference velocity that is multiplied then by a selected load time function value to be declared in B.4 edit;

- B.4 `Acceleration/Velocity function`; this option is inactive when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` then this data corresponds to the `Acceleration/` value; if B.1 is set to `User defined velocity` then this data corresponds to the `Velocity function` which is the index of the load time function which applies to the reference velocity;

- C.1 `Label`. Here, a unique label can be added to the point mass.

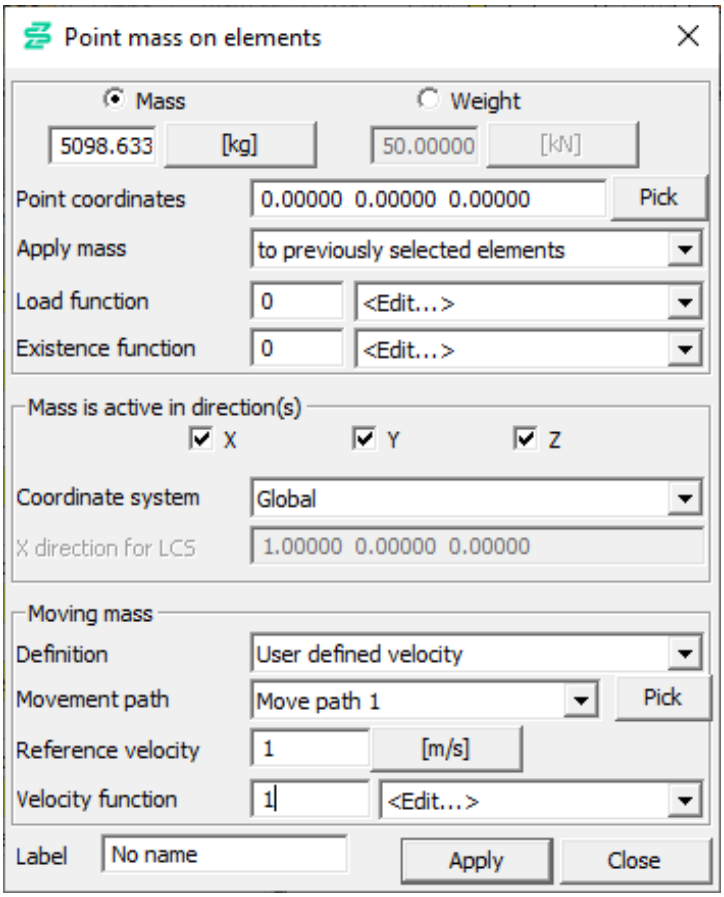



Figure 2: Graphic user interface to define point masses in the 3D when user defined velocity option is selected

**Window 2-674:** `Point mass on elements:Select:All`

This option selects all point masses.

Window 2-674

**Window 2-675:** `Point mass on elements:Select:With load f., existence f., label`

Using this option user may select point mass with a given load time function ID, existence function ID or a given label.

Window 2-675

**Window 2-676:** `Point mass on elements:Update:Parameters`

This option activates the dialog box to modify the applied point mass description. It can be used only for a single picked or selected point mass. The detailed description on how to set the data is given in Window 2-673.

Window 2-676

**Window 2-677:** `Point mass on elements:Update:Scale`

Using this option one may enlarge or diminish size of graphical objects representing point masses.

Window 2-677

## 2.3.1.3.20.5 LINE MASS ON ELEMENTS

**Window 2-678: Options in menu** `Line mass on elements`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Line mass` | — | Window 2-679 |
| `Select` | `All` | — | Window 2-680 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-681 |
| | `With existence function` | — | Window 2-681 |
| | `With label` | — | Window 2-681 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-682 |
| | `Scale` | — | Window 2-683 |

Window 2-678

**Window 2-679:** `Line mass on elements:Create:Line mass`

This option enables user to define a linearly distributed mass density along an arbitrary line segment to be applied to set of target elements or their faces. The line mass can move along defined movement path with a given constant or variable velocity. Defined line mass is projected on target elements or their faces/edges by means of the closest point projection. The dialog box to input the data is non-modal therefore each modification(s) has to be confirmed by pressing button **Apply**.

In this dialog box, the user defines three general groups of data, consisting of:

- A.1 `Mass value at first vertex`, which can also be expressed in terms of weight.
- A.2 `Mass value at the second vertex`, which can also be expressed in terms of weight.
- A.3 `First vertex coordinates` ($x$, $y$, and $z$; in 2D, $z = 0$). If the load is moving, this point is treated as the initial position.
- A.4 `Second vertex coordinates` ($x$, $y$, and $z$; in 2D, $z = 0$). If the load is moving, this point is treated as the initial position.
- A.5 Choice of `target elements` or their edges/faces to which this distributed mass is to be applied. This combo box includes 8 options:

  1. `Automatically to structural element` (shells, beams, membranes, truss elements). Only one group of elements can be chosen here, so an explicit selection of target elements is recommended.

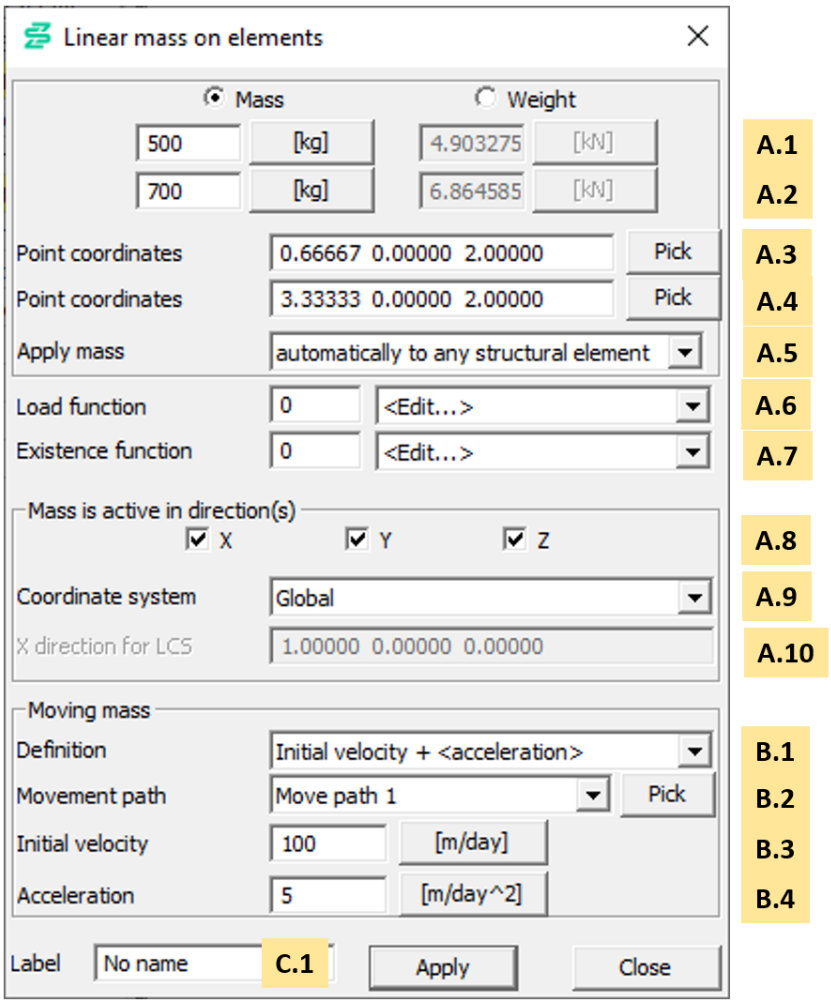



Figure 1: Graphic user interface to define line mass

2. `Automatically to shell elements`; only shell elements are considered.
3. `Automatically to beam elements`; only beam elements are considered.
4. `Automatically to truss elements`; only truss elements are considered.
5. `Automatically to membrane elements`; only membrane elements are considered.
6. `to selected elements`; only selected elements from one group (shell, beam, truss, or membrane elements) are considered.
7. `to selected edges (2D) or faces (3D)` of continuum elements.
8. `to previously selected elements` (structural) and/or their edges/faces (continuum).

- A.6 `Load time function index`. Note that the mass value is multiplied by the LTF$(t)$ value. If the load time function index is zero, the mass value remains constant over time. Otherwise, it may vary, although this option is rarely used in practical cases.

- A.7 `Existence function index`. This mass can be activated/deactivated using the existence function. If the existence function index is zero, the mass is present when target elements are active. If the index is nonzero, the mass's presence is controlled by both the existence function of the target elements and the applied mass.

- A.8 `Active spatial directions for inertial effects`. Here, the user can select the global/local directions in which inertial effects are active.

- A.9 `Coordinate system for mass filtering` (frame), in which mass filtering is defined. This setting is meaningful when at least one direction is inactive. Possible choices for the frame are:

  1. `Global`; mass filtering is done in the global x-y-z coordinate system.
  2. `Local`; in this case, mass filtering is done in the element or its edge/face local coordinate system. For beam elements, the local frame is predefined during element creation. In all other cases (in 3D), an additional vector must be set in block A.10.

     for comprehensive explanation on how the local element frames are defined for beam, truss, membrane, shell elements and continuum element edges(2D)/faces(3D) please refer to the report Rep-Moving_Loads_Mass.pdf
  3. `Local along movement path`; for detailed information on how this local frame is set please refer to the dedicated subsection in report Rep-Moving_Loads_Mass.pdf This definition applies to shell/membrane 3D elements and 3D continuum element faces.

- A.10 `X direction for LCS`. This auxiliary vector is needed in some cases when setting local frames in 3D. for comprehensive explanation on how the local element frames are defined for truss elements please refer to dedicated subsection in report Rep-Moving_Loads_Mass.pdf, for shells, membranes 3D and continuum 3D faces to the corresponding subsection in report Rep-Moving_Loads_Mass.pdf,

- B.1. `Definition` of the time evolution of the velocity when the load is moving; here the 3 choices are possible

  1. `Inactive`; this means that the mass is stationary; in this case data blocks B2, B.3 and B.4 are not meaningful
  2. `Initial velocity + < acceleration >`; if this option is selected then in B.3 please put the initial velocity value $v_0$ and acceleration $a$ in block $B.4$; the current velocity value is computed then using the formula $v(t) = v_0 + a \cdot t^*$; in this formula time $t^* = t - t_{\text{init}}$ where $t_{\text{init}} = 0$, if the load existence function index is 0, or $t_{\text{init}}$ is equal to the time instance at which the load existence function becomes active
  3. `User defined velocity`; if this option is selected then the user interface is slightly modified as shown in the figure below; evolution of the velocity in time is driven now by the selected load time function $v(t) = v_0 \cdot \text{LTF}(t^*)$; as it has been done for the previous option current velocity value is computed in fact for the corrected time value $t^* = t - t_{\text{init}}$

- B.2. `Movement path`; here a predefined movement path has to be selected from the combo-box; this option is inactive when B.1 is set to `Inactive`

- B.3 `Initial/Reference velocity`; this data is unused when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` this data is the initial value of velocity $v_0$; if B.1 is set to `User defined velocity` this declared value plays a role of the reference velocity that is multiplied then by a selected load time function value to be declared in B.4 edit;

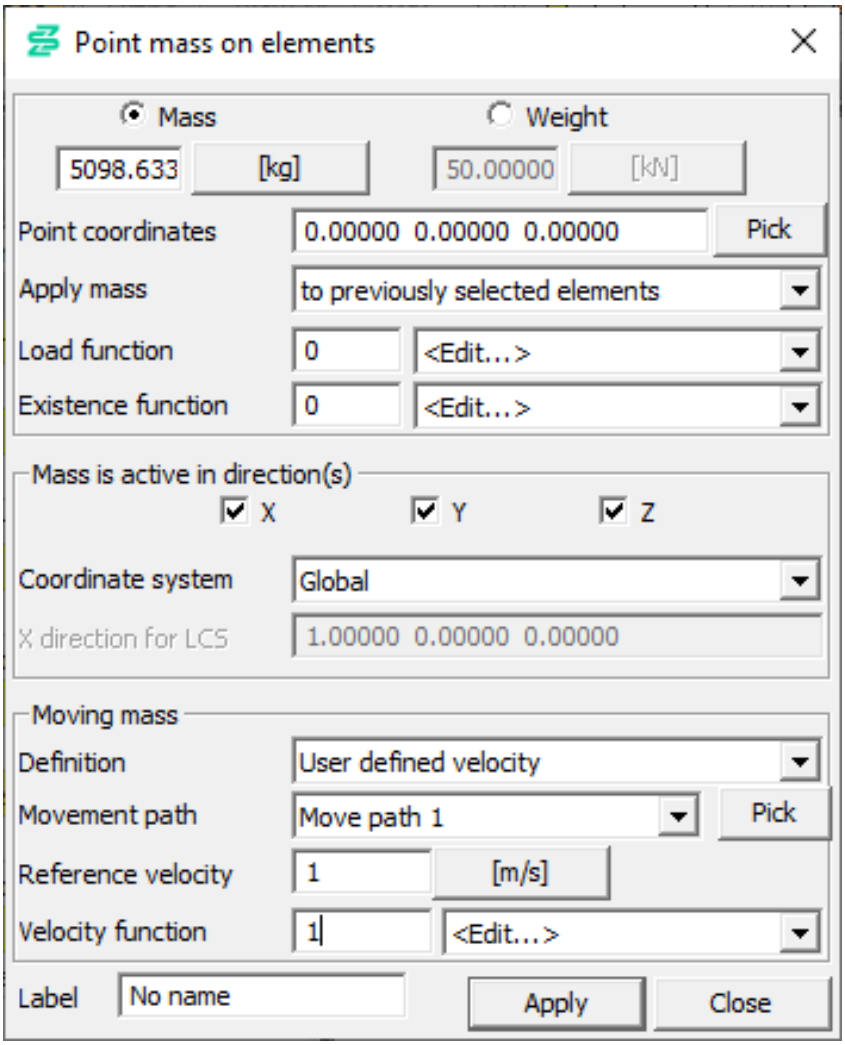



Figure 2: Graphic user interface to define line mass when user defined velocity option is selected

- B.4 `Acceleration/Velocity function`; this option is inactive when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity` + < `acceleration` > then this data corresponds to the `Acceleration/` value; if B.1 is set to `User defined velocity` then this data corresponds to the `Velocity function` which is the index of the load time function which applies to the reference velocity;
- C.1 `Label`. Here, a unique label can be added to the line mass.

Window 2-679

**Window 2-680:** `Line mass on elements:Select:All`

This option selects all line masses.

Window 2-680

**Window 2-681:** `Line mass on elements:Select:With load f., existence f., label`

Using this option user may select line mass with a given load time function ID, existence function ID or a given label.

Window 2-681

**Window 2-682:** `Line mass on elements:Update:Parameters`

This option activates the dialog box to modify the applied line mass description. It can be used only for a single picked or selected line mass. The detailed description on how to set the data is given in Window 2-679.

Window 2-682

**Window 2-683:** `Line mass on elements:Update:Scale`

Using this option one may enlarge or diminish size of graphical objects representing line masses.

Window 2-683

## 2.3.1.3.20.6 SURFACE MASS ON ELEMENTS

**Window 2-684: Options in menu** `Surface mass on elements`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Surface mass` | — | Window 2-685 |
| `Select` | `All` | — | Window 2-686 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-687 |
| | `With existence function` | — | Window 2-687 |
| | `With label` | — | Window 2-687 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-688 |
| | `Scale` | — | Window 2-689 |

Window 2-684

**Window 2-685:** `Surface mass on elements:Create:Surface mass`

This option enables user to define an arbitrary distributed mass density on quasi-planar polygons to be applied to set of target elements or their faces. The surface mass can move along defined movement path with a given constant or variable velocity. Defined surface mass is projected on target elements or their faces/edges by means of the closest point projection. The dialog box to input the data is non-modal therefore each modification(s) has to be confirmed by pressing button **Apply**.

In this dialog box, the user defines three general groups of data, consisting of:

- A.1 `Choice of the input type, by mass density or by an equivalent unit weight`
- A.2 `Distributed mass parameters` ($m_{\text{const}}$, $\frac{\partial m}{\partial x}$, $\frac{\partial m}{\partial y}$ and $\frac{\partial m}{\partial z}$)
- A.3 `Reference point coordinates` ($x$, $y$ and $z$); these coordinates appear in the general formula for the mass density description $m(x, y, z)$; these coordinates are meaningful if all mass density spatial gradients are nonzero
- A.4 `Set of polygon vertices`; these points can be selected from the existing mesh objects or entered manually; if the mass is moving then these points correspond to the initial polygon position; note that this polygon can be located outside the mesh
- A.5 Choice of the `target elements` or their faces to which this mass is to be applied; this combo-box includes 8 possibilities

  1. `Automatically to structural element` (shells, beams, membranes, truss elements). Only one group of elements can be chosen here, so an explicit selection of target elements is recommended.

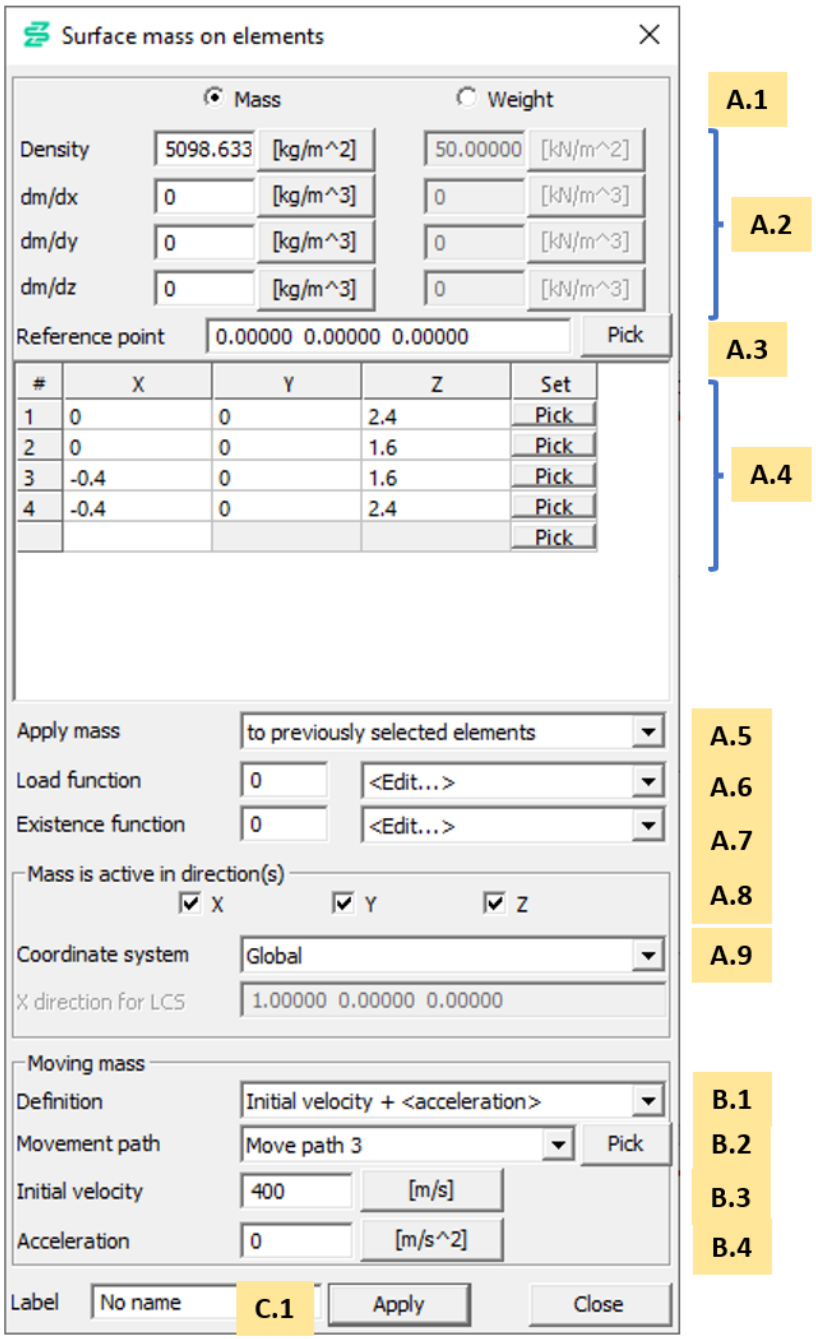



Figure 1: Graphic user interface to define surface mass

2. `Automatically to shell elements`; only shell elements are considered.
3. `Automatically to beam elements`; only beam elements are considered.
4. `Automatically to truss elements`; only truss elements are considered.
5. `Automatically to membrane elements`; only membrane elements are considered.
6. `to selected elements`; only selected elements from one group (shell, beam, truss, or membrane elements) are considered.
7. `to selected edges (2D) or faces (3D)` of continuum elements.
8. `to previously selected elements` (structural) and/or their edges/faces (continuum).

- A.6 `Load time function index`. Note that the mass value is multiplied by the LTF$(t)$ value. If the load time function index is zero, the mass value remains constant over time. Otherwise, it may vary, although this option is rarely used in practical cases.

- A.7 `Existence function index`. This mass can be activated/deactivated using the existence function. If the existence function index is zero, the mass is present when target elements are active. If the index is nonzero, the mass's presence is controlled by both the existence function of the target elements and the applied mass.

- A.8 `Active spatial directions for inertial effects`. Here, the user can select the global/local directions in which inertial effects are active.

- A.9 `Coordinate system for mass filtering` (frame), in which mass filtering is defined. This setting is meaningful when at least one direction is inactive. Possible choices for the frame are:

  1. `Global`; mass filtering is done in the global x-y-z coordinate system.
  2. `Local`; in this case, mass filtering is done in the element or its edge/face local coordinate system. For beam elements, the local frame is predefined during element creation. In all other cases (in 3D), an additional vector must be set in block A.10.

     for comprehensive explanation on how the local element frames are defined for beam, truss, membrane, shell elements and continuum element edges(2D)/faces(3D) please refer to the report Rep-Moving_Loads_Mass.pdf
  3. `Local along movement path`; for detailed information on how this local frame is set please refer to the dedicated subsection in report Rep-Moving_Loads_Mass.pdf This definition applies to shell/membrane 3D elements and 3D continuum element faces.

- A.10 `X direction for LCS`. This auxiliary vector is needed in some cases when setting local frames in 3D. for comprehensive explanation on how the local element frames are defined for truss elements please refer to dedicated subsection in report Rep-Moving_Loads_Mass.pdf, for shells, membranes 3D and continuum 3D faces to the corresponding subsection in report Rep-Moving_Loads_Mass.pdf,

- B.1. `Definition` of the time evolution of the velocity when the load is moving; here the 3 choices are possible

  1. `Inactive`; this means that the mass is stationary; in this case data blocks B2, B.3 and B.4 are not meaningful
  2. `Initial velocity + < acceleration >`; if this option is selected then in B.3 please put the initial velocity value $v_0$ and acceleration $a$ in block $B.4$; the current velocity value is computed then using the formula $v(t) = v_0 + a \cdot t^*$; in this formula time $t^* = t - t_{\text{init}}$ where $t_{\text{init}} = 0$, if the load existence function index is 0, or $t_{\text{init}}$ is equal to the time instance at which the load existence function becomes active
  3. `User defined velocity`; if this option is selected then the user interface is slightly modified as shown in the figure below; evolution of the velocity in time is driven now by the selected load time function $v(t) = v_{\text{ref}} \cdot \text{LTF}(t^*)$; as it has been done for the previous option current velocity value is computed in fact for the corrected time value $t^* = t - t_{\text{init}}$

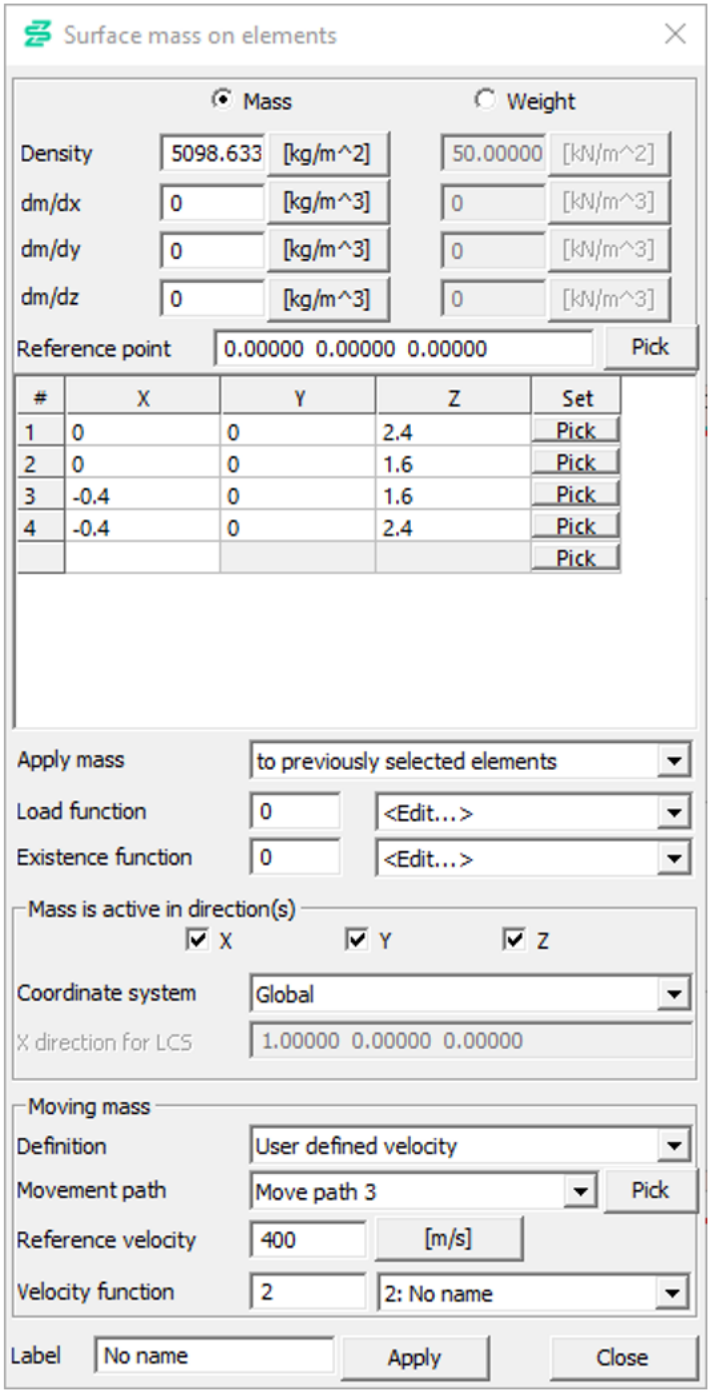



Figure 2: Graphic user interface to define surface mass when user defined velocity option is selected

- B.2. `Movement path`; here a predefined movement path has to be selected from the combo-box; this option is inactive when B.1 is set to `Inactive`

- B.3 `Initial/Reference velocity`; this data is unused when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity` + < `acceleration` > this data is the initial value of velocity $v_0$; if B.1 is set to `User defined velocity` this declared value plays a role of the reference velocity that is multiplied then by a selected load time function value to be declared in B.4 edit;

- B.4 `Acceleration/Velocity function`; this option is inactive when B.1 is set to `Inactive`; if B.1 is set to `Initial velocity + < acceleration >` then this data corresponds to the `Acceleration/` value; if B.1 is set to `User defined velocity` then this data corresponds to the `Velocity function` which is the index of the load time function which applies to the reference velocity;
- C.1 `Label`. Here, a unique label can be added to the surface mass.

Window 2-685

**Window 2-686:** `Surface mass on elements:Select:All`

This option selects all surface masses.

Window 2-686

**Window 2-687:** `Surface mass on elements:Select:With load f., existence f., label`

Using this option user may select surface mass with a given load time function ID, existence function ID or a given label.

Window 2-687

**Window 2-688:** `Surface mass on elements:Update:Parameters`

This option activates the dialog box to modify the applied surface mass description. It can be used only for a single picked or selected surface mass. The detailed description on how to set the data is given in Window 2-685.

Window 2-688

**Window 2-689:** `Surface mass on elements:Update:Scale`

Using this option one may enlarge or diminish size of graphical objects representing surface masses.

Window 2-689

## 2.3.1.3.21 DISTRIBUTED FLUX

Three groups of fluxes can be applied to elements in the real mesh

| | |
|---|---|
| Fluid flux | Create distributed fluid flux on selected elements |
| Heat flux | Create distributed heat flux on selected elements |
| Humidity flux | Create distributed humidity flux on selected elements |

## 2.3.1.3.21.1 FLUID FLUX

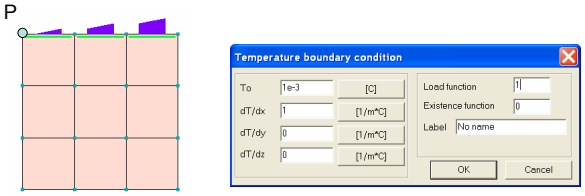
**Window 2-690: Options in menu** `Fluid flux`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On edge(s)` | — | Window 2-691 |
| | `2 nodes` | — | Window 2-692 |
| | `3 nodes` | — | Window 2-694 |
| | `4 nodes` | — | Window 2-694 |
| | `On face(s)` | — | Window 2-693 |
| `Select` | `All` | — | Window 2-695 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-696 |
| | `With label` | — | Window 2-696 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-697 |

Window 2-690

**Window 2-691:** `Fluid flux:Create:On edge(s)`

This option has a double meaning depending on whether selection list of edges in empty or not. If none of the edges is selected then user must pick the edge and generation procedure follows strictly the one by 2 `nodes` described in Window 2-692). In all other cases user must pick the point at which he sets value of the flux $q^F$ and to set flux gradients in $x$ and $y$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the flux at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of edges is computed through the formula: $q^F(\mathbf{x}) = q^F_o + dq^F_x \cdot (x - x_P) + dq^F_y \cdot (y - y_P)$. This option is valid only for 2D models.



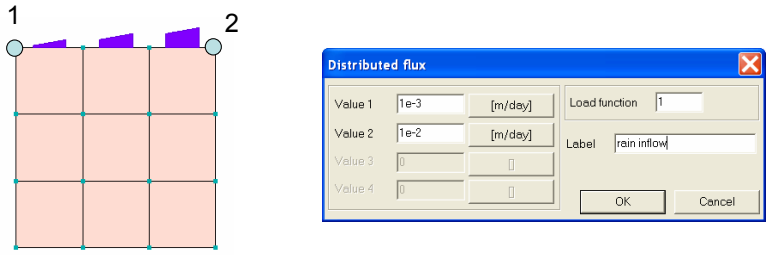

**Remarks**:

1. For fast selection purposes one may apply a distinct label to the edited flux
2. Load time function can be associated with the applied flux
3. For modeling rain inflow (`Flow/Deformation+Flow` problems only) it is recommended to apply the fluid flux on the continuum element edge at which seepage element is added; seepage element is needed to cut-off pore water pressure on the flux part of the boundary
4. Fluxes are automatically refined during mesh refinement
5. Positive flux means fluid inflow into domain while negative an outflow

Window 2-691

## Window 2-692: `Fluid flux:Create:2 nodes`

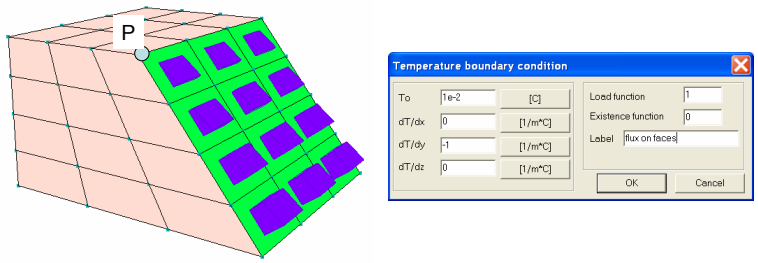
Using this option one may apply a fluid flux to all continuum 2D element edges, belonging to the line section designed by the two picked points. This option is valid only for 2D models and is equivalent to the one `On edge(s)` (see Window 2-691).




Window 2-692

## Window 2-693: `Fluid flux:Create:On face(s)`

This option has a double meaning depending on whether selection list of faces in empty or not. If none of the faces is selected then user must pick the face and generation procedure follows strictly the one by `3 nodes, 4 nodes` described in Window 2-694. In all other cases user must pick the point at which he sets value of the flux $q^F$ and to set flux gradients in $x$, $y$ and $z$ directions. This definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed. Assuming that picked point coordinates are denoted by $\mathbf{x}_P$ the flux at any node (with coordinates denoted by $\mathbf{x}$) belonging to the selected set of faces is computed through the formula: $q^F(\mathbf{x}) = q_o^F + dq_x^F \cdot (x - x_P) + dq_y^F \cdot (y - y_P) + dq_z^F \cdot (z - z_P)$. This option is valid only for 3D models.



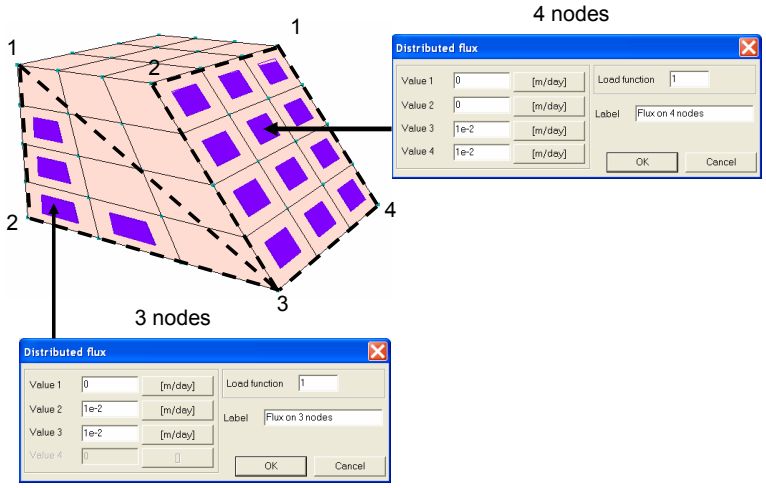

**Remarks**:

1. For fast selection purposes one may apply a distinct label to the edited flux
2. Load time function can be associated with the applied flux
3. For modeling rain inflow (`Flow/Deformation+Flow` problems only) it is recommended to apply the fluid flux on the continuum element face at which seepage element is added; seepage element is needed to cut-off pore water pressure on the flux part of the boundary
4. Fluxes are automatically refined during mesh refinement
5. Positive flux means fluid inflow into domain while negative an outflow

Window 2-693

**Window 2-694:** `Fluid flux:Create:3 nodes, 4 nodes`

Using this option one may apply fluxes by clicking 3 or 4 nodes on the boundary of a 3D continuum mesh. These 3/4 nodes constitute a triangular or quadrilateral super-element. The fluid flux is applied at all continuum element faces which belong to that super-element. Hence this definition when refining the mesh will be automatically projected on the new mesh and no additional action to be made by the user is needed.




Window 2-694

**Window 2-695:** `Fluid flux:Select:All`

Using this option one may select all fluxes.

Window 2-695

**Window 2-696:** `Fluid flux:Select:With load f., label`

Using this option user may select applied fluxes with a given load time function, or a given label. In case of a label one may select a single item from combo-box.

Window 2-696

**Window 2-697:** `Fluid flux:Update:Parameters`

This option activates the dialog box to modify the applied fluid flux description, for a single picked edge/face or group of selected edges/faces. The detailed description on how to set the data is given in Window 2-691 or Window 2-693 .

**Remarks**:

1. When running this option for group of selected edges/faces only default parameters will be set in the dialog box

Window 2-697

## 2.3.1.3.21.2 HEAT FLUX

Handling heat fluxes is fully compatible with handling fluid fluxes (only flux unit is different). Hence all options designed for heat fluxes are described by referring to corresponding Windows on fluid fluxes.

**Window 2-698: Options in menu** `Heat flux`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On edge(s)` | — | Window 2-691 |
| | `2 nodes` | — | Window 2-692 |
| | `3 nodes` | — | Window 2-694 |
| | `4 nodes` | — | Window 2-694 |
| | `On face(s)` | — | Window 2-693 |
| `Select` | `All` | — | Window 2-695 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-696 |
| | `With label` | — | Window 2-696 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-697 |

Window 2-698

## 2.3.1.3.21.3 HUMIDITY FLUX

Handling humidity fluxes is fully compatible with handling fluid fluxes (only flux unit is different). Hence all options designed for humidity fluxes are described by referring to corresponding Windows on fluid fluxes.

**Window 2-699: Options in menu** `Humidity flux`

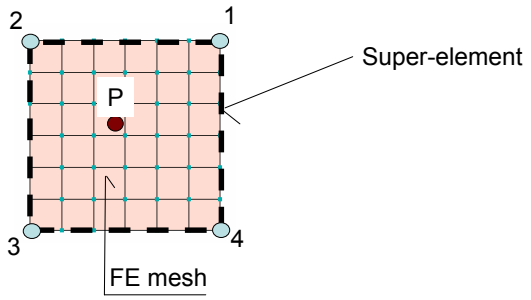
| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On edge(s)` | — | Window 2-691 |
| | `2 nodes` | — | Window 2-692 |
| | `3 nodes` | — | Window 2-694 |
| | `4 nodes` | — | Window 2-694 |
| | `On face(s)` | — | Window 2-693 |
| `Select` | `All` | — | Window 2-695 |
| | `between 4 nodes` | — | Window 2-315 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-696 |
| | `With label` | — | Window 2-696 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-697 |

Window 2-699

## 2.3.1.3.22 INITIAL CONDITION

Initial conditions of any type can be applied with aid of super-elements of Q4 (for 2D models) or B8 (for 3D models) type. The idea is such that a given initial condition (like initial pressures, in situ effective stress components, etc..) is set at super-element vertices and then it is interpolated, by means of standard Q4/B8 interpolation functions, to obtain assumed initial condition at any node or integration point within the element (Gauss point) (see Win.(2-700)).

**Window 2-700:** `Concept of super-elements`




**Remarks**:

1. Any scalar, vectorial or tensorial data can be set at super-element vertices
2. To recover the initial condition at a given node, or integration point, the following algorithm is used
   - Compute vector of local coordinates $\boldsymbol{x}_P$ of a given point P within the super-element by solving the nonlinear equation: $\mathbf{x}_P - \sum_{i=1}^{NEN} N_i(\boldsymbol{\xi}_P)\ \mathbf{x}_i = \mathbf{0}$ (here number of super-element nodes is denoted by $NEN$)
   - Interpolate scalar, vectorial/tensorial (component by component) initial condition using the formula: $\mathbf{f}_P = \sum_{i=1}^{NEN} N_i(\boldsymbol{\xi}_P)\ \mathbf{f}_i$

Window 2-700

The following initial conditions can be set up:

| | |
|---|---|
| Initial pressure | Create initial pore fluid pressure via superelements |
| Initial temperature | Create initial temperatures via superelements |
| Initial humidity | Create initial humidity via superelements |
| Initial stress | Create initial effective stresses in continuum via superelements |
| Initial stresses on element(s) | Create initial stresses in continuum/trusses/membranes |
| Initial strain | Create imposed strains via superelements |
| Initial strains on element(s) | Create imposed strains on continuum/trusses/membranes |
| Initial displacement/velocity | Create initial displacements and/or velocities on selected nodes |
| Imposed strain on beams/shells | Create imposed strains on beam/shell elements |

## 2.3.1.3.22.1 INITIAL PRESSURE

**In the definition of initial pressure compressive pore fluid pressures are negative in compression**.

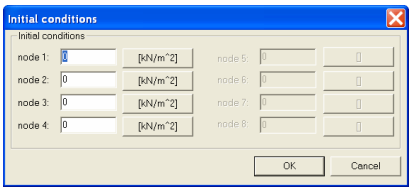
**Window 2-701: Options in menu** `Initial pressure`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `4 nodes` | — | Window 2-702 |
| | `4 nodes/points` | — | Window 2-703 |
| | `8 nodes` | — | Window 2-702 |
| | `8 nodes/points` | — | Window 2-703 |
| | `On bounding box` | — | Window 2-704 |
| | `On bounding box of sel. ele.` | — | Window 2-705 |
| | `On construction front` | — | Window 2-706 |
| | `Extrude 2D to 3D by translation` | — | Window 2-707 |
| `Select` | `All` | — | Window 2-708 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-709 |

Window 2-701

**Window 2-702:** `Initial pressure:Create:4 nodes, 8 nodes`

Using this option one may define initial pore fluid pressure by setting quadrilateral /brick super-element geometry (clicking 4/8 existing nodes) (see Window 2-700) and setting pressure values at 4/8 vertices in the following dialog box

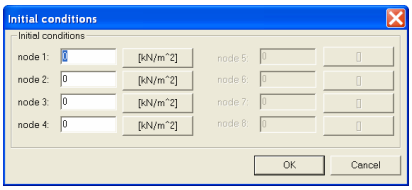



**Remarks**:
1. Super-element vertices must be entered in order that prevents its distortion
2. This initial condition is meaningful for `Deformation`, `Deformation+Flow` and `Flow` problems

Window 2-702

**Window 2-703:** `Initial pressure:Create:4 nodes/points, 8 nodes/points`

Using this option one may define initial pore fluid pressure by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes/points) (see Window 2-700) and setting pressure values at 4/8 vertices in the following dialog box




**Remarks**:

1. This option is compatible with the one described in Win.2-702 but selection of pressure super-element vertices is extended to nodes, geometrical points and grid points

Window 2-703

**Window 2-704:** `Initial pressure:Create:On bounding box`

This option is similar to the one described in Win.2-702 but super-element nodes are automaticaly selected and coincide with current model bounding box vertices. These vertices are labeled and shown in the graphic scene.

Window 2-704

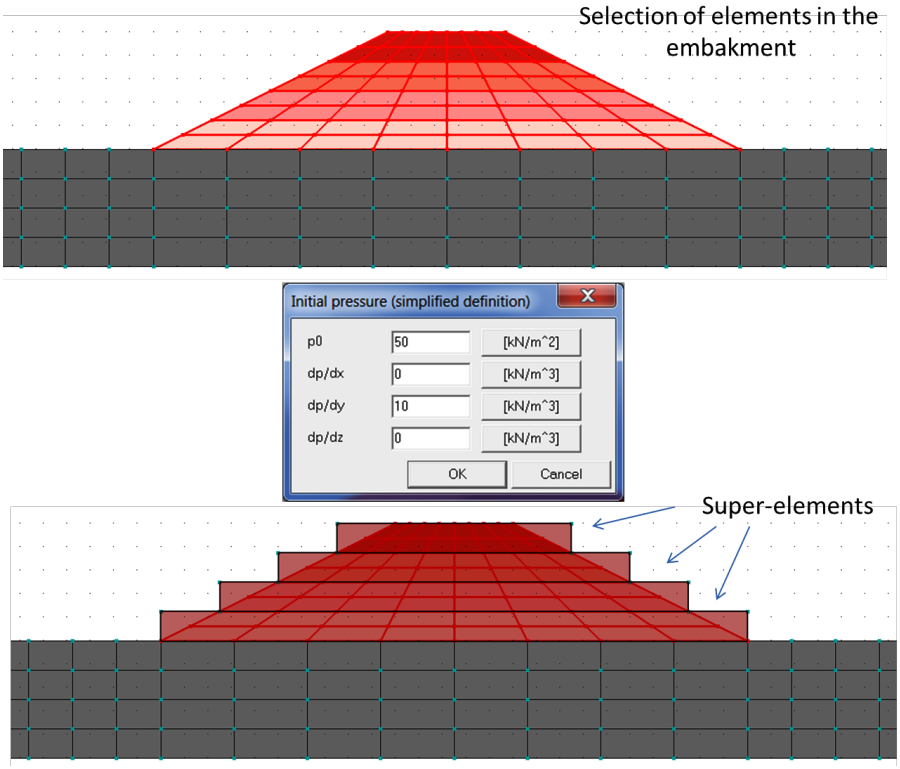
**Window 2-705:** `Initial pressure:Create:On bounding box of sel. ele.`

This option is similar to the one described in Win.2-702 but super-element nodes are automaticaly selected and coincide with the vertices of the bounding box of selected continuum elements.

Window 2-705

**Window 2-706:** `Initial pressure:Create:On construction front`

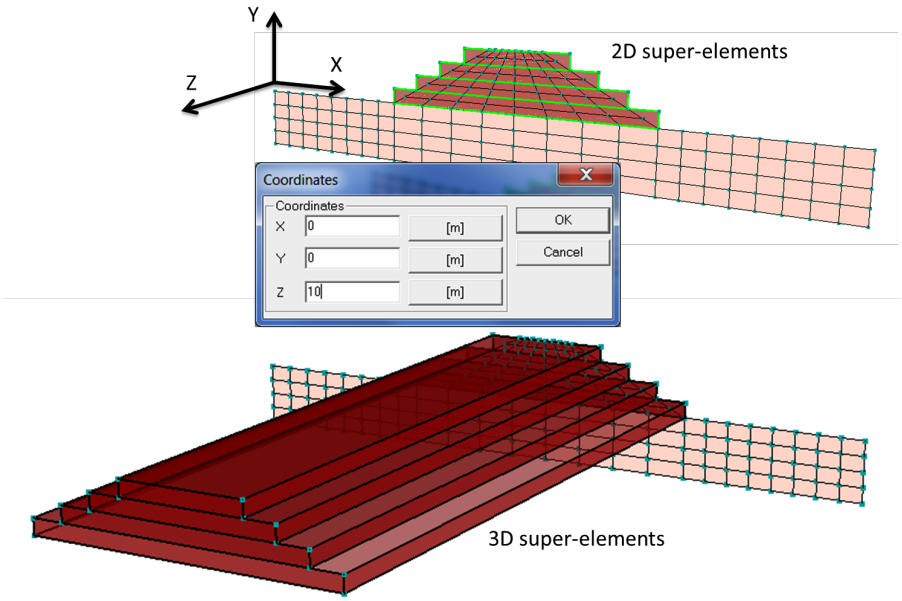
This option allows the user to create, automatically, initial pore pressure super-elements for each newly constructed layer of elements. In the example shown below construction of the embankment is modeled in 4 steps therefore 4 super-elements are created assuming that the pressure at the top is equal to $p_o$ value and its gradient along vertical axis $(y)$ is equal to 10 kPa/m. This option can also be used in 3D.




Window 2-706

**Window 2-707:** `Initial pressure:Create:Extrude 2D to 3D by translation`

This option allows the user to generate 3D initial pressure super-elements using extrusion method applied to the 2D objects.




Window 2-707

**Window 2-708:** `Initial pressure:Select:All`

Using this option one may select all initial pressure super-elements.

Window 2-708

**Window 2-709:** `Initial pressure:Update:Parameters`

This option activates the dialog box (see Window 2-702) to modify the assumed initial pore fluid pressures at super-element vertices.

Window 2-709

## 2.3.1.3.22.2 INITIAL TEMPERATURE

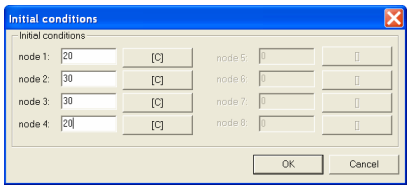
**Window 2-710: Options in menu** `Initial temperature`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 4 nodes | — | Window 2-711 |
| | 4 nodes/points | — | Window 2-712 |
| | 8 nodes | — | Window 2-711 |
| | 8 nodes/points | — | Window 2-712 |
| | On bounding box | — | Window 2-713 |
| | On bounding box of sel. ele. | — | Window 2-714 |
| | On construction front | — | Window 2-715 |
| | Extrude 2D to 3D by translation | — | Window 2-716 |
| Select | All | — | Window 2-717 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-718 |

Window 2-710

**Window 2-711:** `Initial temperature:Create:4 nodes , 8 nodes`

Using this option one may define initial temperature by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes) (see Window 2-700) and setting temperatures at 4/8 vertices in the following dialog box



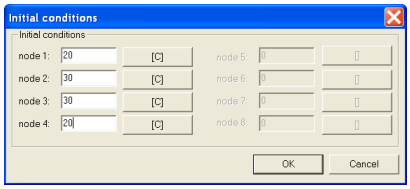

**Remarks**:

1. Super-element vertices must be entered in order that prevents its distortion
2. This initial condition is meaningful for `Heat` problem only

Window 2-711

**Window 2-712:** `Initial temperature:Create:4 nodes/points, 8 nodes/points`

Using this option one may define initial temperatures by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes/points) (see Window 2-700) and setting temperatures at 4/8 vertices in the following dialog box




**Remarks**:

1. This option is compatible with the one described in Win.2-711 but selection of temperature super-element vertices is extended to nodes, geometrical points and grid points

Window 2-712

**Window 2-713:** `Initial temperature:Create:On bounding box`

This option is similar to the one described in Win.2-711 but super-element nodes are automaticaly selected and coincide with current model bounding box vertices. These vertices are labeled and shown in the graphic scene.

Window 2-713

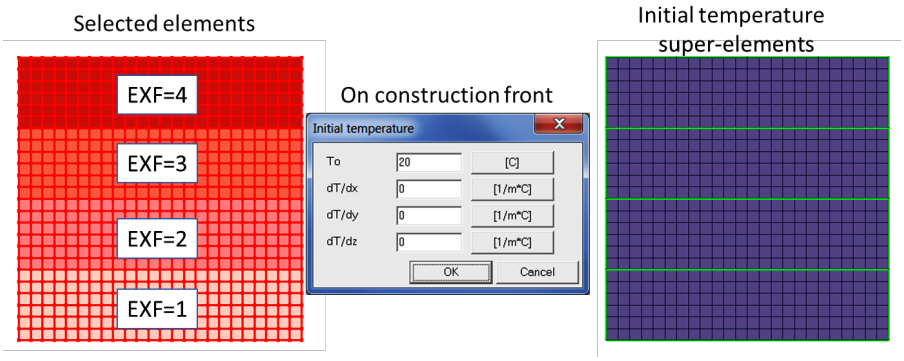
**Window 2-714:** `Initial temperature:Create:On bounding box of sel. ele.`

This option is similar to the one described in Win.2-711 but super-element nodes are automaticaly selected and coincide with the vertices of the bounding box of selected continuum elements.

Window 2-714

**Window 2-715:** `Initial temperature:Create:On construction front`

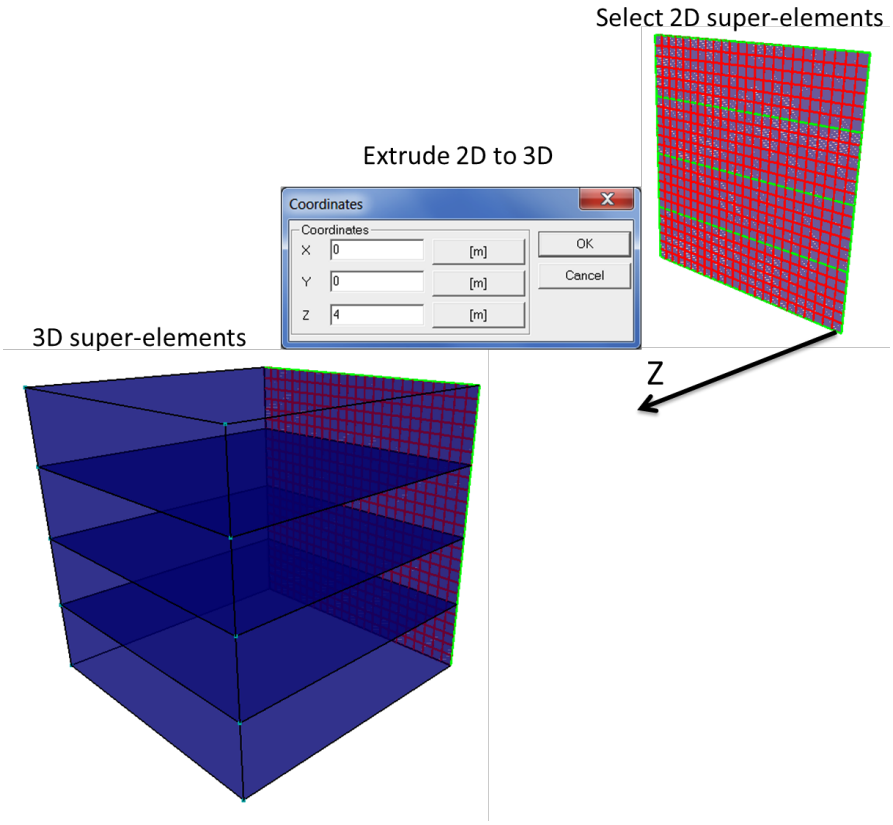
This option allows the user to create, automatically, initial temperature super-elements for each newly constructed layer of elements. In the example shown below construction of massive concrete block is modeled in 4 steps therefore 4 super-elements are created assuming that the initial temperature $T_o$ is set to $20^o$ while all temperature gradients are zero. This option can also be used in 3D.




Window 2-715

**Window 2-716:** `Initial temperature:Create:Extrude 2D to 3D by translation`

This option allows the user to generate 3D initial temperature super-elements using extrusion method applied to the 2D objects.




Window 2-716

**Window 2-717:** `Initial temperature:Select:All`

Using this option one may select all initial temperature super-elements.

Window 2-717

**Window 2-718:** `Initial temperature:Update:Parameters`

This option activates the dialog box (see Window 2-711) to modify the assumed initial temperatures at super-element vertices.

Window 2-718

## 2.3.1.3.22.3 INITIAL HUMIDITY

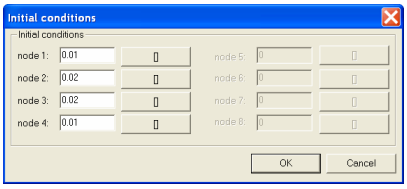
**Window 2-719: Options in menu** `Initial humidity`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `4 nodes` | — | Window 2-720 |
| | `4 nodes/points` | — | Window 2-721 |
| | `8 nodes` | — | Window 2-720 |
| | `8 nodes/points` | — | Window 2-721 |
| | `On bounding box` | — | Window 2-722 |
| | `On bounding box of sel. ele.` | — | Window 2-723 |
| | `On construction front` | — | Window 2-724 |
| | `Extrude 2D to 3D by translation` | — | Window 2-725 |
| `Select` | `All` | — | Window 2-726 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-727 |

Window 2-719

**Window 2-720:** `Initial humidity:Create:4 nodes , 8 nodes`

Using this option one may define initial humidity by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes) (see Window 2-700) and setting humidities at 4/8 vertices in the following dialog box



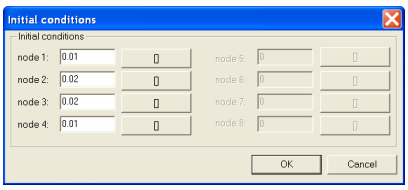

**Remarks**:

1. Super-element vertices must be entered in order that prevents its distortion
2. This initial condition is meaningful for `Humidity` problem only

Window 2-720

**Window 2-721:** `Initial humidity:Create:4 nodes/points, 8 nodes/points`

Using this option one may define initial humidity by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes/points) (see Window 2-700) and setting humidities at 4/8 vertices in the following dialog box




**Remarks**:

1. This option is compatible with the one described in Win.2-720 but selection of humidity super-element vertices is extended to nodes, geometrical points and grid points

Window 2-721

**Window 2-722:** `Initial humidity:Create:On bounding box`

This option is similar to the one described in Win.2-720 but super-element nodes are automaticaly selected and coincide with current model bounding box vertices. These vertices are labeled and shown in the graphic scene.

Window 2-722

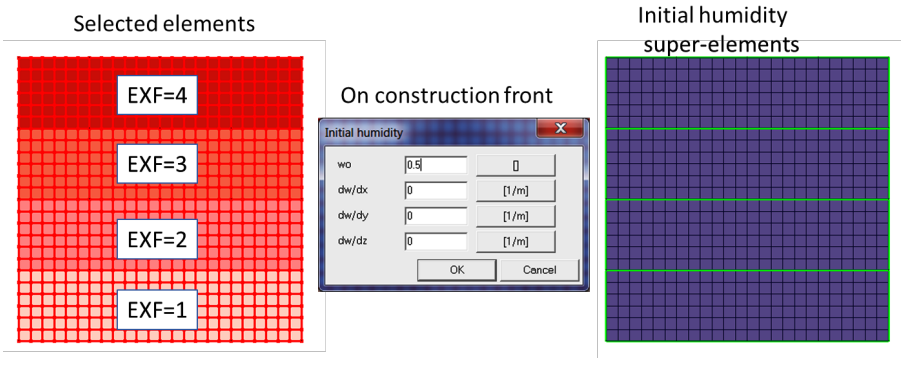
**Window 2-723:** `Initial humidity:Create:On bounding box of sel. ele.`

This option is similar to the one described in Win.2-720 but super-element nodes are automaticaly selected and coincide with the vertices of the bounding box of selected continuum elements.

Window 2-723

**Window 2-724:** `Initial humidity:Create:On construction front`

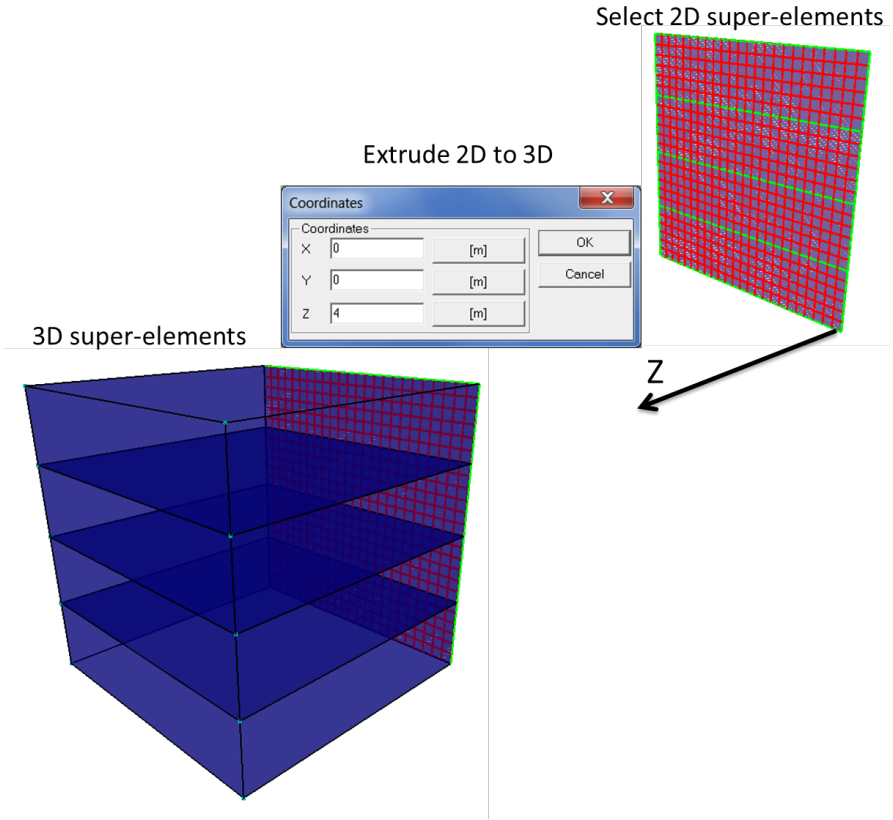
This option allows the user to create, automatically, initial humidity super-elements for each newly constructed layer of elements. In the example shown below construction of the embankment is modeled in 4 steps therefore 4 super-elements are created assuming that the initial humidity $w_o$ is set to 0.8 (80 %) while all humidity gradients are zero. This option can also be used in 3D.



Window 2-724

**Window 2-725:** `Initial humidity:Create:Extrude 2D to 3D by translation`

This option allows the user to generate 3D initial humidity super-elements using extrusion method applied to the 2D objects.




Window 2-725

**Window 2-726:** `Initial humidity:Select:All`

Using this option one may select all initial humidity super-elements.

Window 2-726

**Window 2-727:** `Initial humidity:Update:Parameters`

This option activates the dialog box (see Window 2-720) to modify the assumed initial humidities at super-element vertices.

Window 2-727

## 2.3.1.3.22.4 INITIAL STRESS

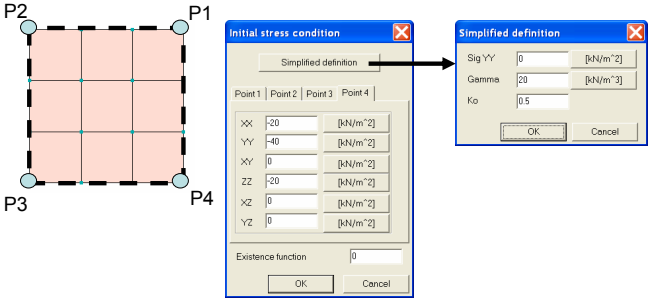
**Window 2-728: Options in menu** `Initial stress`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 4 nodes | — | Window 2-729 |
| | 4 nodes/points | — | Window 2-730 |
| | 8 nodes | — | Window 2-729 |
| | 8 nodes/points | — | Window 2-730 |
| | On bounding box | — | Window 2-731 |
| | On bounding box of sel. ele. | — | Window 2-732 |
| | On construction front | — | Window 2-733 |
| | Extrude 2D to 3D by translation | — | Window 2-734 |
| Select | All | — | Window 2-735 |
| | In zoom box | — | Window 2-317 |
| | In zoom circle | — | Window 2-318 |
| | Inside contour | — | Window 2-319 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-736 |
| Show | XX | — | Window 2-737 |
| | YY | — | Window 2-737 |
| | XY | — | Window 2-737 |
| | ZZ | — | Window 2-737 |
| | XZ | — | Window 2-737 |
| | YZ | — | Window 2-737 |

Window 2-728

**Window 2-729:** `Initial stress:Create:4 nodes , 8 nodes`

Using this option one may define initial effective stresses by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes) (see Window 2-700) and setting all effective stress components at 4/8 vertices in the following dialog box




**Remarks**:

1. The initial stresses can be generated by using a simplified procedure; all shear components are assumed to be zero while horizontal and out of the plane components are related to the vertical one through the $K_o^{\text{in situ}}$ (to be set in the edit field **Ko** [ ]) coefficient

- In the dialog box for simplified definition the value set in the **Sig YY** [ ] ($\sigma_y^*$) is assumed at the super-element vertex located at maximum $y$ coordinate
- At any point $P$ (Gauss point) the vertical stress is computed as: $\sigma_y(\mathbf{x}) = \sigma_y^* + \gamma\ (y_P - y)$ while the two remaining normal components are equal to: $\sigma_x = K_o^{\text{in situ}}\ \sigma_y$, $\sigma_z = K_o^{\text{in situ}}\ \sigma_y$

2. Initial stresses are exclusively computed at the element center and then applied at all standard Gauss points (note that stress generated from gravity is constant in low order continuum finite element)
3. Super-element vertices must be entered in order that prevents its distortion
4. This initial condition is meaningful for `Deformation` and `Deformation+Flow` problems
5. Tensile stresses are positive
6. The existence function is needed here to handle construction algorithm, mainly in the situation when elements in certain zone appear then disappear and appear again (with possibly new material properties); in such cases the initial stress super-element and finite elements, to which these initial stresses are to be applied, must both have compatible existence functions

Window 2-729

**Window 2-730:** `Initial stress:Create:4 nodes/points, 8 nodes/points`

This option is compatible with the one described in Win.2-729 but selection of stress super-element vertices is extended to nodes, geometrical points and grid points

Window 2-730

**Window 2-731:** `Initial stress:Create:On bounding box`

This option is similar to the one described in Win.2-729 but super-element nodes are automaticaly selected and coincide with current model bounding box vertices. These vertices are labeled and shown in the graphic scene.

Window 2-731

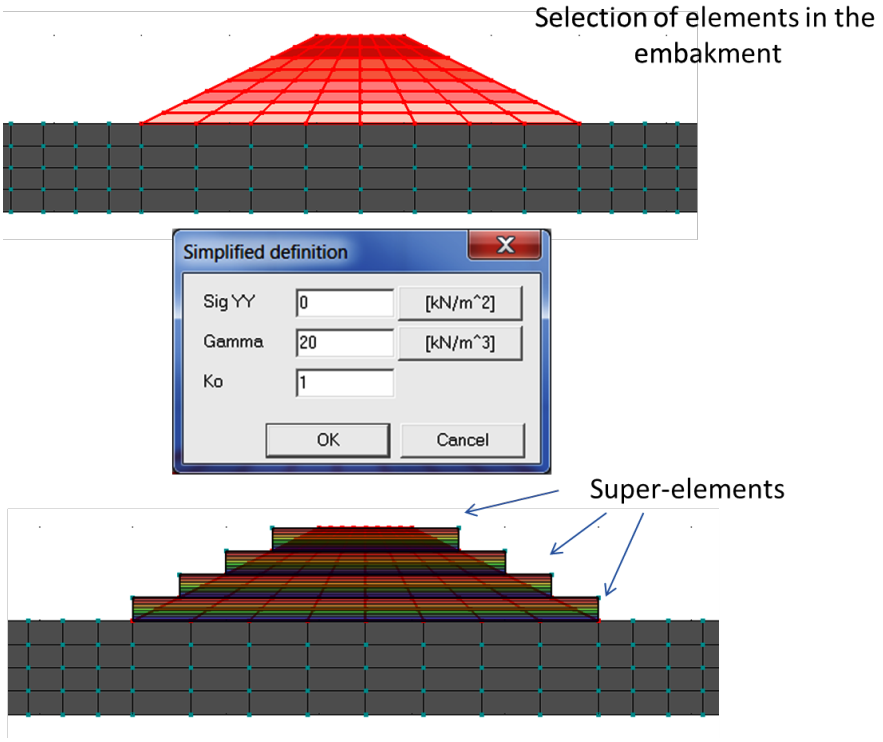
**Window 2-732:** `Initial stress:Create:On bounding box of sel. ele.`

This option is similar to the one described in Win.2-729 but super-element nodes are automaticaly selected and coincide with the vertices of the bounding box of selected elements.

Window 2-732

**Window 2-733:** `Initial stress:Create:On construction front`

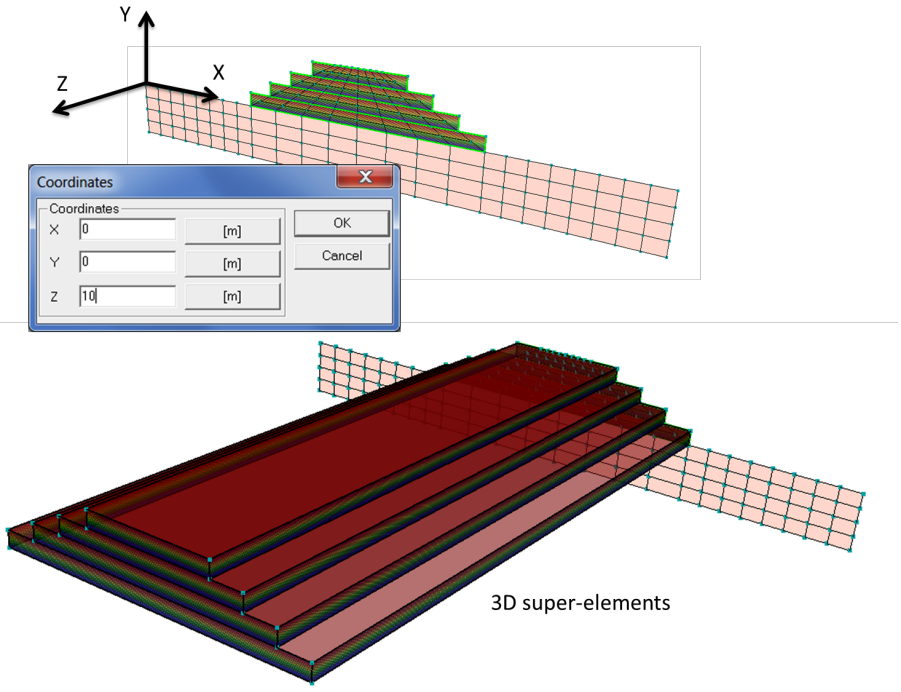
This option allows the user to create, automatically, initial effective stress super-elements for each newly constructed layer of elements. In the example shown below construction of the embankment is modeled in 4 steps therefore 4 super-elements are created assuming that the vertical stress at the top is equal to $\sigma_{yy,o} = 0$ and its gradient along vertical axis ($y$) is equal to 20 kPa/m while $K_o = 1$. This option can also be used in 3D.




Window 2-733

**Window 2-734:** `Initial stress:Create:Extrude 2D to 3D by translation`

This option allows the user to generate 3D initial stress super-elements using extrusion method applied to the 2D objects.




Window 2-734

**Window 2-735:** `Initial stress:Select:All`

Using this option one may select all initial effective stress super-elements.

Window 2-735

**Window 2-736:** `Initial stress:Update:Parameters`

This option activates the dialog box (see Window 2-729) to modify the assumed initial effective stress components at super-element vertices.

Window 2-736

**Window 2-737:** `Initial stress:Show:XX,YY ...`

With this option user may visualize color contours of the selected stress component.

Window 2-737

## 2.3.1.3.22.5 INITIAL STRESS ON ELEMENT(S)

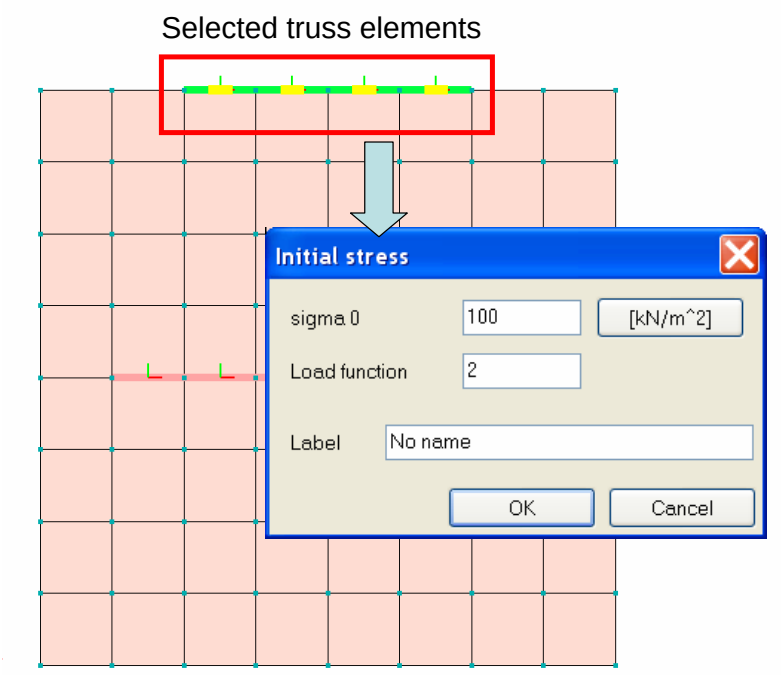
**Window 2-738: Options in menu** `Initial stresses on element(s)`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Apply to trusses` | — | Window 2-739 |
| | `Apply to membranes` | — | Window 2-740 |
| | `Apply to continuum` | — | Window 2-741 |
| `Select` | `Elem. with init.stress` | — | Window 2-742 |
| | `Elem. with init.stress via label` | — | Window 2-743 |
| `Delete` | `Delete init. stress for elements` | | Window 2-744 |
| `Update` | `Parameters` | — | Window 2-745 |

Window 2-738

**Window 2-739:** `Initial stress:Create:Apply to trusses`

Using this option one may define initial axial stress in selected or picked anchor/truss element(s) through the following dialog box



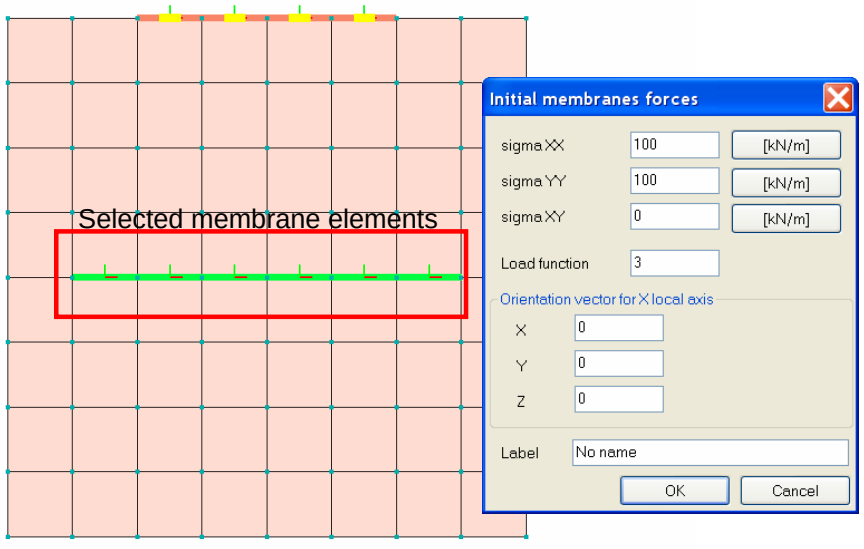

**Remarks**:

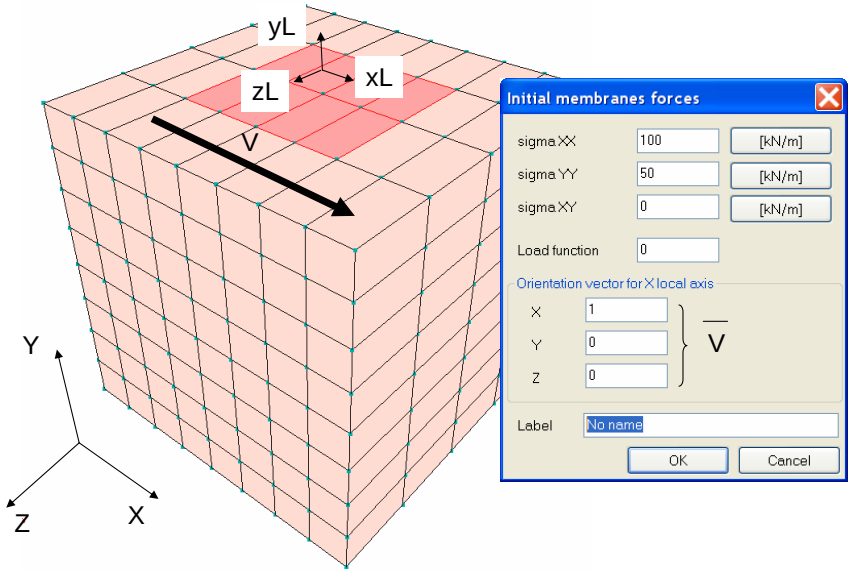
1. This initial condition is meaningful for `Deformation` and `Deformation+Flow` problems
2. Tensile stresses are positive
3. The load time function allows to analyze situations with variable (in time) initial axial stress
4. This option can be useful for modeling cables where initial stresses are needed to start computation

Window 2-739

**Window 2-740:** `Initial stress:Create:Apply to membranes`

Using this option one may define initial stresses in selected or picked membrane element(s) through the following dialog box



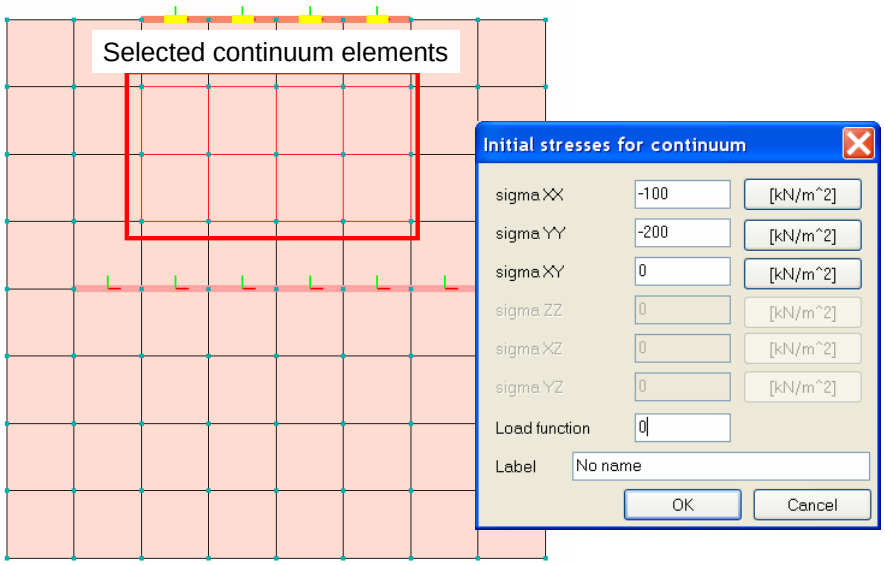



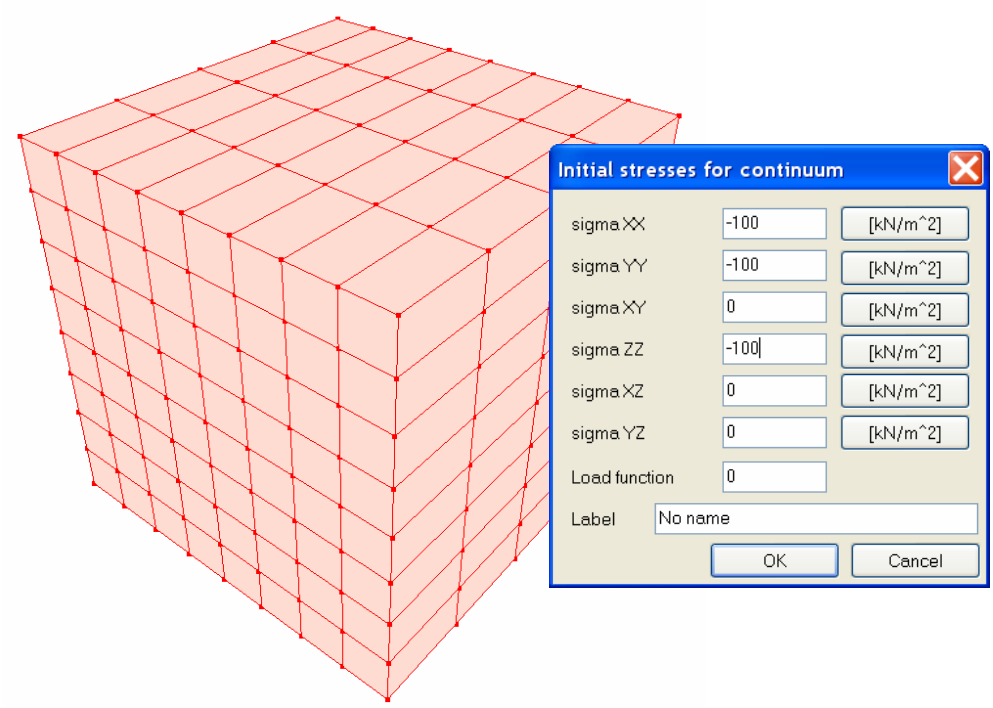

**Remarks**:

1. This initial condition is meaningful for `Deformation` and `Deformation+Flow` problems only
2. Tensile stresses are positive
3. The load time function allows to analyze situations with variable (in time) initial axial stress
4. This option can be useful for modeling structural membranes where initial stresses are needed to start the computation
5. In the 3D, in case of anisotropic initial stress setting, an additional vector must be given to position the local coordinate system (in the membrane) in which this setting is made; projection of this vector on the membrane surface gives the local membrane $xL$ axis

**Window 2-741:** `Initial stress:Create:Apply to continuum`

Using this option one may define initial effective stresses in selected or picked continuum element(s) through the following dialog box




**Remarks**:

1. This initial condition is meaningful for `Deformation` and `Deformation+Flow` problems only
2. Tensile stresses are positive
3. This definition applies to effective stresses
4. The load time function allows to analyze situations with variable (in time) initial stresses

**Window 2-742:** `Initial stress:Select:Elements with initial stress`

This option allows to select all elements with the initial stresses defined under this menu.

Window 2-742

**Window 2-743:** `Initial stress:Select:Elements with initial stress via label`

This option allows to select all elements with the initial stresses defined with a given label (under this general menu).

Window 2-743

**Window 2-744:** `Initial stress:Delete:Initial stress for element(s)`

This option allows to cancel initial stresses for selected elements.

Window 2-744

**Window 2-745:** `Initial stress:Update:Parameters`

This option activates the dialog box (see Window 2-739, Window 2-740, Window 2-741) to modify the assumed initial stress components at selected elements.

Window 2-745

## 2.3.1.3.22.6 INITIAL STRAIN

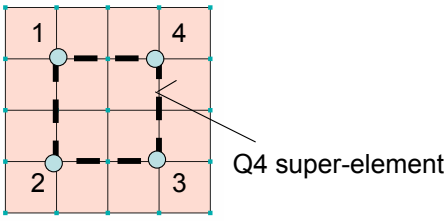
**Window 2-746: Options in menu** `Initial strain`

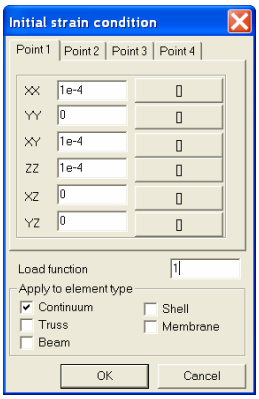
| Option | Method | Toolbar | Description |
|---|---|---|---|
| Create | 4 nodes | — | Window 2-747 |
| | 4 nodes/points | — | Window 2-748 |
| | 8 nodes | — | Window 2-747 |
| | 8 nodes/points | — | Window 2-748 |
| | On bounding box | — | Window 2-749 |
| | On bounding box of sel. ele. | — | Window 2-750 |
| | Extrude 2D to 3D by translation | — | Window 2-751 |
| Delete | Delete |  | Window 2-320 |
| Update | Parameters | — | Window 2-753 |
| Show | XX | — | Window 2-754 |
| | YY | — | Window 2-754 |
| | XY | — | Window 2-754 |
| | ZZ | — | Window 2-754 |
| | XZ | — | Window 2-754 |
| | YZ | — | Window 2-754 |

Window 2-746

**Window 2-747:** `Initial strain:Create:4 nodes , 8 nodes`

Using this option one may define imposed strains by setting quadrilateral /brick super-element geometry (clicking 4/8 nodes) (see Window 2-700) and setting strain components at 4/8 vertices in the following dialog box




**Remarks**:

1. Super-element vertices must be entered in order that prevents its distortion

2. This option is meaningful for `Deformation` and `Deformation+Flow` problems only
3. Tensile strains are positive
4. By activating check-boxes representing different element types user may apply imposed strains only to the selected ones
5. The evolution of imposed strains in time is driven by the load time function
6. Thermal/hygral imposed strains can be introduced here by setting normal strain components only ( $\varepsilon_x^o(x,t) = \varepsilon_y^o(x,t) = \varepsilon_z^o(x,t) = \alpha(T(x,t) - T(x,t_o))$

Window 2-747

**Window 2-748:** `Initial strain:Create:4 nodes/points, 8 nodes/points`

This option is compatible with the one described in Win.2-747 but selection of strain super-element vertices is extended to nodes, geometrical points and grid points

Window 2-748

**Window 2-749:** `Initial strain:Create:On bounding box`

This option is similar to the one described in Win.2-747 but super-element nodes are automaticaly selected and coincide with current model bounding box vertices. These vertices are labeled and shown in the graphic scene.

Window 2-749

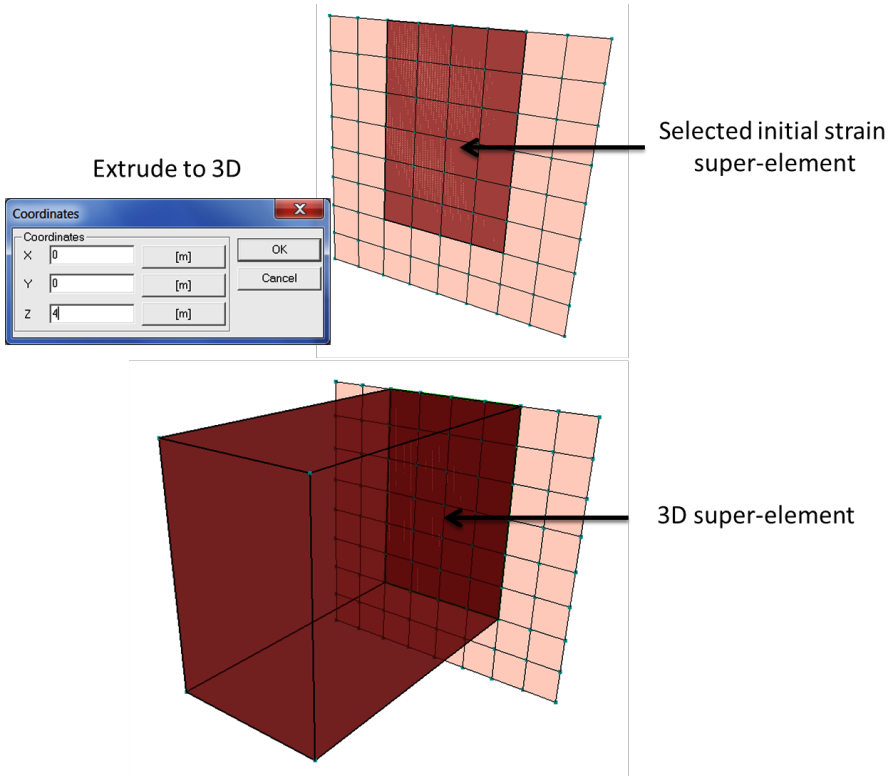
**Window 2-750:** `Initial strain:Create:On bounding box of sel. ele.`

This option is similar to the one described in Win.2-747 but super-element nodes are automaticaly selected and coincide with the vertices of the bounding box of selected elements.

Window 2-750

**Window 2-751:** `Initial strain:Create:Extrude 2D to 3D by translation`

This option allows the user to generate 3D initial strain super-elements using extrusion method applied to the 2D objects.




Window 2-751

**Window 2-752:** `Initial strain:Select:All`

Using this option one may select all initial strain super-elements.

Window 2-752

**Window 2-753:** `Initial strain:Update:Parameters`

This option activates the dialog box (see Window 2-747) to modify the assumed imposed strains components at super-element vertices.

Window 2-753

**Window 2-754:** `Initial strain:Show:XX,YY ...`

With this option user may visualize color contours of the selected strain component.

Window 2-754

## 2.3.1.3.22.7 INITIAL STRAIN ON ELEMENT(S)

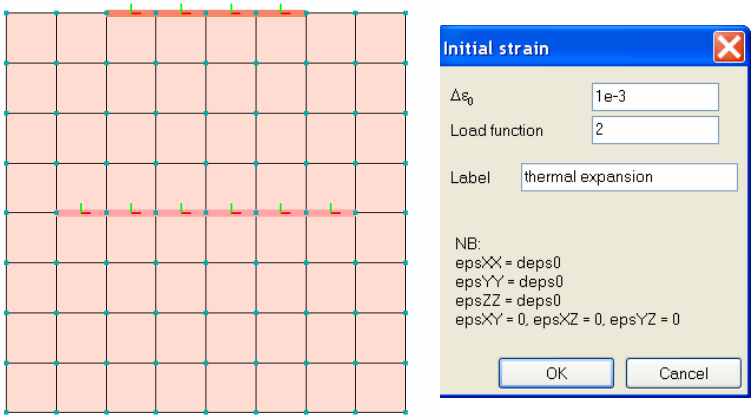
**Window 2-755: Options in menu** `Initial strains on element(s)`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Apply to element(s)` | — | Window 2-756 |
| `Select` | `Elem. with initial strains` | — | Window 2-757 |
| | `Elem. with initial strains via label` | — | Window 2-758 |
| `Delete` | `Delete initial strain for elements` | — | — |
| `Update` | `Parameters` | — | Window 2-756 |

Window 2-755

**Window 2-756:** `Initial strain:Create:Apply to element(s)`

Using this option one may define imposed strains ($\Delta\varepsilon_{xo} = \Delta\varepsilon_{yo} = \Delta\varepsilon_{zo} = \Delta\varepsilon_o$) for picked single element or group of selected elements.




**Remarks**:

1. In this setting thermal type induced strains can be generated only (for continuum or membranes all normal components are equal)
2. This option is meaningful for `Deformation` and `Deformation+Flow` problems only
3. Tensile strains are positive
4. The evolution of imposed strains in time is driven by the load time function

Window 2-756

**Window 2-757:** `Initial strain:Select:Elements with initial strain`

This option selects all elements with defined initial strains (using exclusively the option `Initial strains on element(s)`).

Window 2-757

**Window 2-758:** `Initial strain:Select:Elem.with initial strains via label`

This option selects all elements with defined initial strains (using exclusively the option `Initial strains on element(s)`), and given label.

Window 2-758

**Window 2-759:** `Initial strain:Delete:Initial strains for elements`

This option cancels initial strains, defined through the option `Initial strains on element(s)`, for a single picked element or group of selected elements.

Window 2-759

## 2.3.1.3.22.8 INITIAL DISPLACEMENT/VELOCITY

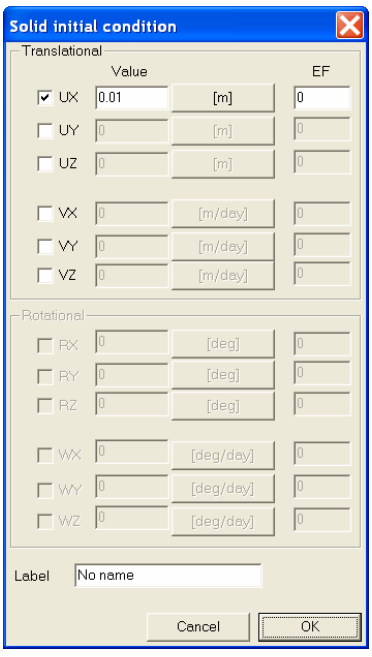
**Window 2-760: Options in menu** `Initial displacement/velocity`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node` | — | Window 2-761 |
| | `On element node` | — | Window 2-762 |
| `Select` | All options | — | Window 2-763 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-765 |
| | `Scale` | — | Window 2-766 |

Window 2-760

**Window 2-761:** `Initial displacement/velocity:Create on node`

Using this option one may define the initial displacement and/or velocity on a single picked nodal point or set of selected nodal points.




**Remarks**:

1. This initial condition is meaningful only for dynamic driver which can be run in `Deformation` mode
2. The existence function is needed to handle the case when dynamic driver is run after `Driven load` driver(s); in such case the initial displacement is understood as an increment of the displacement by which the structure will be excited
3. The applied initial condition may have a label

Window 2-761

**Window 2-762:** `Initial displacement/velocity:Create on element node`

This option is equivalent to the one described in Window 2-761 but in this case the initial condition is applied to a single picked element node. This way in case when at a given geometrical point few nodes appear (due to contact interface for instance) one may apply the initial condition to the proper node.

Window 2-762

**Window 2-763:** `Initial displacement/velocity`**:Options in menu** `Select`

| | |
|---|---|
| `All` | all initial conditions are selected; result of this action depends on the status of `Selection mode` (see Window 2-83) |
| `Between 2 nodes` | all initial conditions defined on nodes belonging to the section line defined by two picked points are selected; result of this action depends on the status of `Selection mode` in ribbon menu `Global selection tools`(see Window 2-83) |
| `Between 4 nodes` | all initial conditions defined on nodes belonging to the surface defined by four picked points are selected; result of this action depends on the status of `Selection mode` in ribbon menu `Global selection tools`(see Window 2-83) |
| `In zoom box` | if the zoom box is dragged from left to the right side then only fully covered, by the zoom rectangle, initial conditions are selected if the zoom box is dragged from right to the left side then even partially covered, by the zoom rectangle, initial conditions are selected |
| `In zoom circle` | if the zoom circle is dragged from left to the right side then only fully covered, by the circle (cylinder in 3D), initial conditions are selected |
| | if the zoom circle is dragged from right to the left side then even partially covered, by the zoom circle (cylinder in 3D), initial conditions are selected |
| `Inside contour` | all initial conditions being inside of the drawn closed contour are selected |
| `With existence function` | all initial conditions with a given existence function are selected |
| `With label` | all initial conditions with a given label are selected |

Window 2-763

**Window 2-764:** `Initial displacement/velocity:Delete:Delete`

Using this option one may delete a single picked initial condition marker or group of selected initial conditions.

Window 2-764

**Window 2-765:** `Initial displacement/velocity:Update:Parameters`

This option activates the dialog box (see Window 2-761) to modify created initial conditions.

Window 2-765

**Window 2-766:** `Initial displacement/velocity:Update:Scale`

With this option one may increase or diminish the size of initial condition markers.

Window 2-766

## 2.3.1.3.22.9 IMPOSED STRAINS ON BEAMS/SHELLS

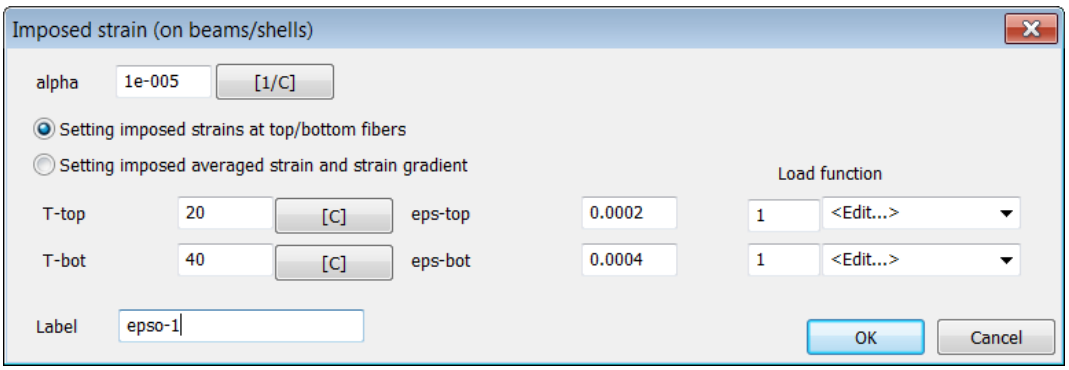
**Window 2-767: Options in menu** `Imposed strains on beams/shells`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Apply to element(s)` | — | Window 2-768 |
| `Select` | `Elem.with imposed strains` | — | Window 2-769 |
| | `Elem.with imposed strains via label` | — | Window 2-770 |
| `Delete` | `Delete imposed strains for elements` | — | Window 2-771 |
| `Update` | `Parameters` | — | Window 2-772 |

Window 2-767

**Window 2-768:** `Imposed strains (on beams/shells):Create:Apply to element(s)`

Using this option one may define imposed strains for beam or shell elements by setting (varying in time) temperatures for top and bottom fibers. In case of beam elements this definition is valid only when the cross section is explicitly defined. Another definition, that can always be used, takes into account an averaged temperature and temperature gradient, both varying in time according to the given load time function.




**Remarks**:

1. This option is meaningful for `Deformation` and `Deformation+Flow` problems only
2. The evolution of temperatures in time is driven by the load time function
3. For given temperatures imposed strains are computed as
   $\varepsilon_{xo}^{top}(t) = \varepsilon_{yo}^{top}(t) = \alpha T^{top} \times LTF^{top}(t)$
   $\varepsilon_{xo}^{bot}(t) = \varepsilon_{yo}^{bot}(t) = \alpha T^{bot} \times LTF^{bot}(t)$

Window 2-768

**Window 2-769:** `Imposed strains on beams/shells:Select:Elem.with imposed strain`

This option selects all beam/shell elements with defined imposed strains.

Window 2-769

**Window 2-770:** `Imposed strains on beams/shells:Select:Elem.with imposed strain via la`

This option selects all beam/shell elements with defined imposed strains under given label.

Window 2-770

**Window 2-771:** `Imposed strains on beams/shells:Delete`

This option allows the user to cancel imposed strains for single picked element (beam/shell) or group of selected elements.

Window 2-771

**Window 2-772:** `Imposed strains on beams/shells:Update:Parameters`

This option activates the dialog box (see Window 2-767) to modify the assumed temperatures at top and bottom fibers, or averaged temperature and temperature gradient.

Window 2-772

## 2.3.1.3.23 DRM DOMAINS (DOMAIN REDUCTION METHOD)

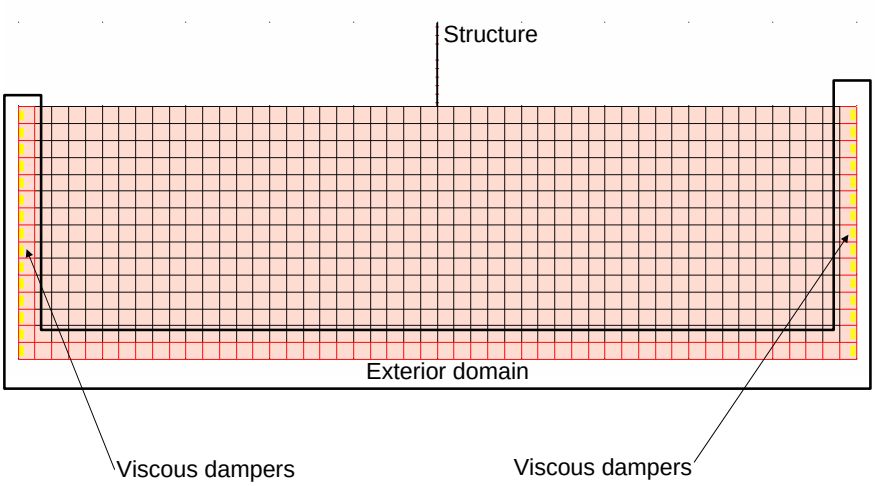
**Window 2-773: Options in menu** `DRM domains`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Exterior domain on element(s)` | — | Window 2-774 |
| | `Boundary domain on element(s)` | — | Window 2-775 |
| `Select` | `Elements in exterior domain` | — | Window 2-776 |
| | `Elements in boundary domain` | — | Window 2-777 |
| `Delete` | `Delete exterior domain` | — | Window 2-778 |
| | `Delete boundary domain` | — | Window 2-779 |

Window 2-773

**Window 2-774:** `DRM domains:Create: Exterior domain on element(s)`

This option allows the user to define the exterior domain in Domain Reduction Method (DRM) for the set of selected continuum elements. This domain must not overlap with the boundary DRM layer.



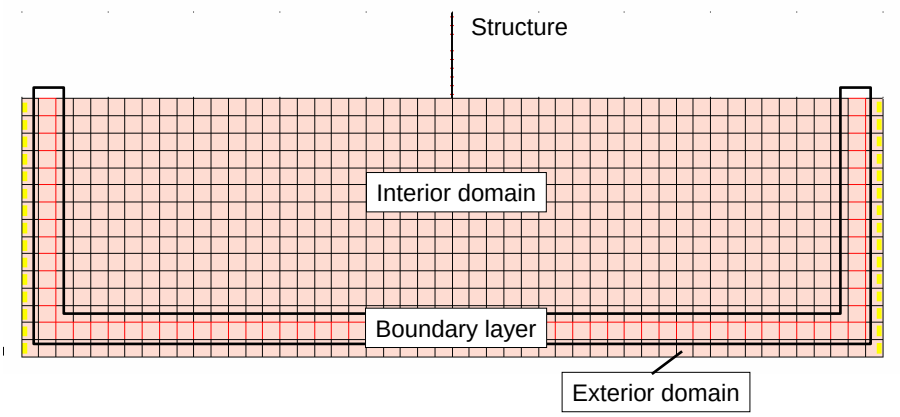

**Remarks**:

1. On the external edges/faces viscous dampers should be generated to avoid spurious reflections from finite boundaries

Window 2-774

**Window 2-775:** `DRM domains:Create: Boundary domain on element(s)`

This option allows the user to define the boundary layer of elements in Domain Reduction Method (DRM) for the set of selected continuum elements. This domain must not overlap with the exterior DRM layer.




**Remarks**:

1. The interior domain is detected automaticaly once the boundary and exterior domains are selected
2. In the boundary domain continuum must behave in the elastic manner; this is enforced by the calculation module in an automatic manner for any constitutive model for continuum

Window 2-775

**Window 2-776:** `DRM domains:Select:Elements in exterior domain`

This option selects all visible continuum elements that were selected in the exterior DRM domain.

Window 2-776

**Window 2-777:** `DRM domains:Select:Elements in boundary domain`

This option selects all visible continuum elements that were selected in the boundary DRM layer .

Window 2-777

**Window 2-778:** `DRM domains:Delete:Elements in exterior domain`

This option removes all continuum elements from the set of elements in the exterior DRM domain.

Window 2-778

**Window 2-779:** `DRM domains:Delete:Elements in boundary domain`

This option removes all continuum elements from the set of elements in the boundary DRM layer .

Window 2-779

## 2.3.1.3.24 KINEMATIC CONSTRAINTS

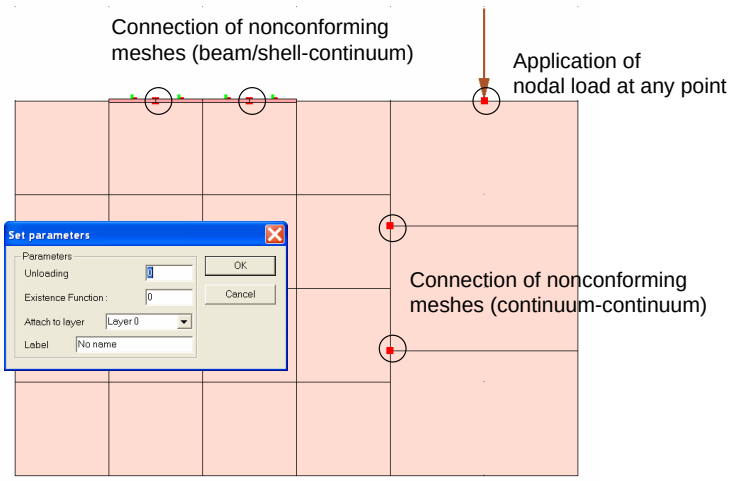
**Window 2-780: Options in menu** `Kinematic constraint`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node` | — | Window 2-781 |
| `Select` | `All` | — | Window 2-782 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With load function` | — | Window 2-783 |
| | `With existence function` | — | Window 2-783 |
| | `With label` | — | Window 2-783 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-784 |

Window 2-780

**Window 2-781:** `Kinematic constraint:Create:On node`

This option allows to connect any nodal point to continuum element (one may apply a nodal load at any point that belongs to the continuum element (2D/3D) too). As the penalty approach is used one may generate nested kinematic constraints without emergency of singularity of the resulting global stiffness matrix although results may not be correct. A definitely more accurate definition can be made using `Nodal link` option that allows to connect any node to any type of finite elements without penalty method but with the limitation that constraints cannot be nested. Hence in some critical situations when meshing is too complex (especially in 3D) and we have to connect nonconforming meshes several times this option may serve as an efficient tool. It proves to be useful also for overlaid meshes approach.



Window 2-781

**Window 2-782:** `Kinematic constraint:Select:All`

This option selects all kinematic constraints.

Window 2-782

**Window 2-783:** `Kinematic constraint:Select:With load f., exist.f., label`

Using this option user may select assumed kinematic constraints with a given load time function ID, existence function or a given label. In case of a label one may select a single item from combo-box.

Window 2-783

**Window 2-784:** `Kinematic constraint:Update:Parameters`

This option activates the dialog box to modify the definition of kinematic constraint at a given node. The detailed description on how to set the data is given in Window 2-781.

Window 2-784

## 2.3.1.3.25 NODAL LINK

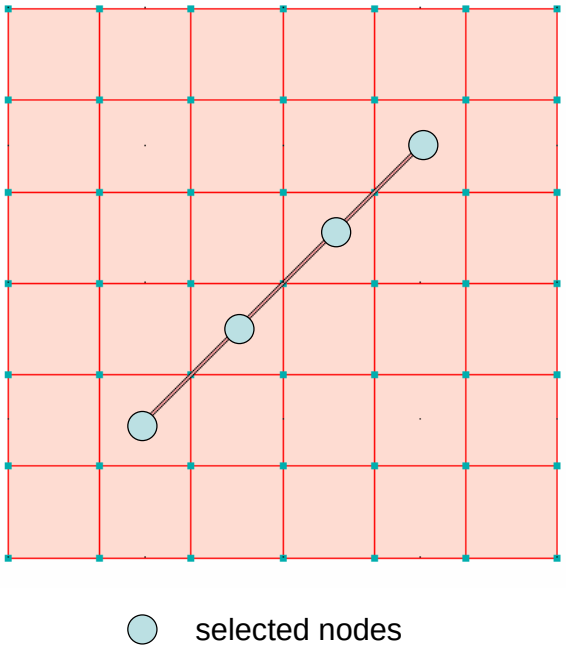
**Window 2-785: Options in menu** `Nodal link`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node(s)` | — | Window 2-786 |
| | `On node(s) aut.sel.elements` | — | Window 2-787 |
| `Select` | `All` | — | Window 2-788 |
| | `Box with 2 nodes` | — | Window 2-316 |
| | `In zoom box` | — | Window 2-317 |
| | `In zoom circle` | — | Window 2-318 |
| | `Inside contour` | — | Window 2-319 |
| | `With existence function` | — | Window 2-789 |
| | `With label` | — | Window 2-789 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-790 |

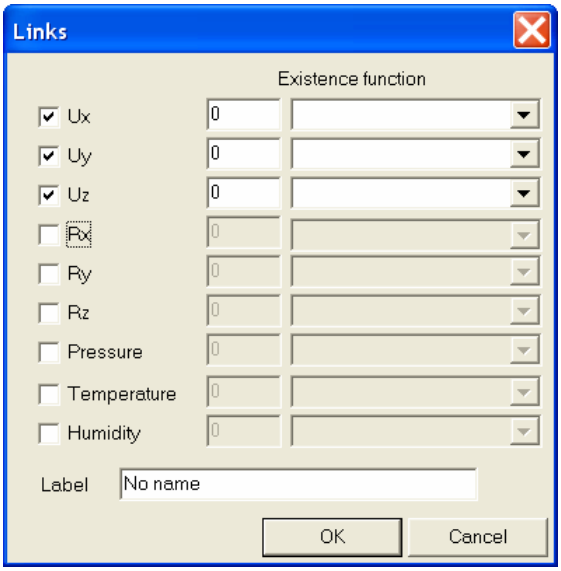
Window 2-785

**Window 2-786:** `Nodal link:Create:On node(s)`

This option, valid for 2D and 3D, allows to link selected node(s) to selected elements (can be continuum, shells, beams, trusses, membranes) in an exact manner (without penalty method used in the option `Kinematic constraint`). It is obvious that a given node must be located in the interior of the element to which it is to be linked. This linking can be applied to selected degrees of freedom (DOFs). It has to be emphasized that linking is possible only for these DOFs which are common for given node and element to be linked with(for instance when beam node is linked to the continuum then only translational DOFs can be constrained, if we link beam node to the shell element then all translational and rotational DOFs can be constrained). Activating this option must be preceded by selection of nodes (the ones to be linked) and background elements. To select DOFs to be constrained user must also select degrees of freedom to be constrained (see picture below).




Selection of degrees of freedom to be constrained

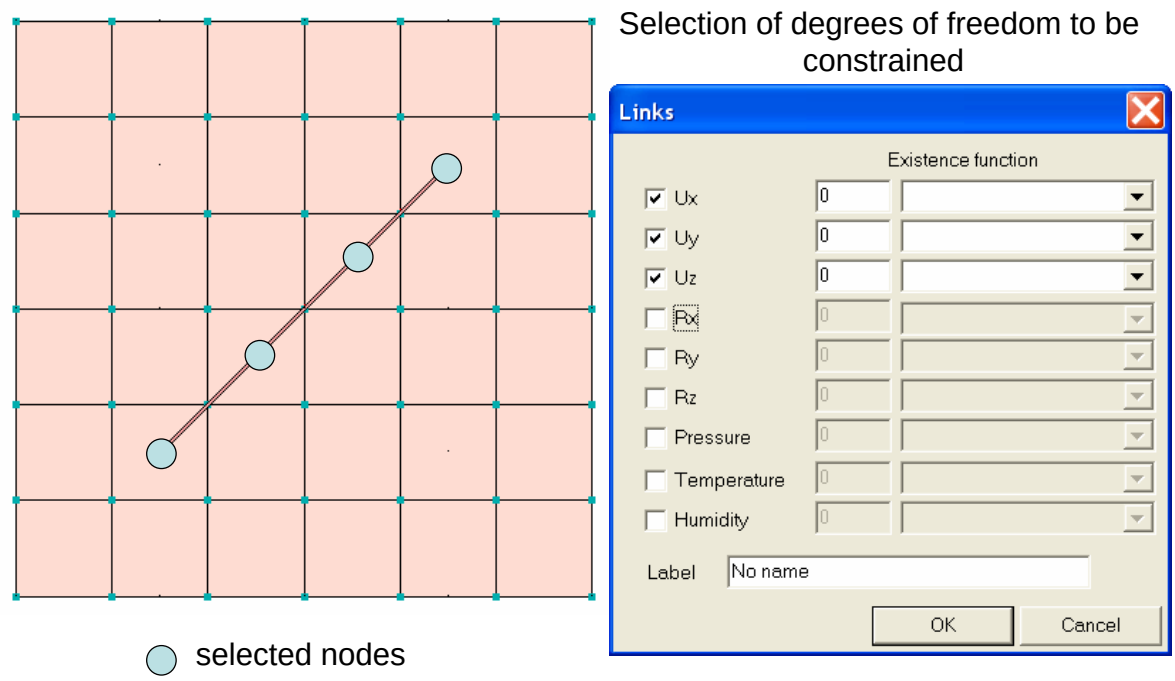



**Remarks**:

1. Linking operation cannot be nested; it means that linking of a node to the element whoese nodes are already linked to another element is forbidden; in such cases to avoid multilevel linking one may use of standard kinematic constraints (option `Kinematic constraint`) enforced with aid of penalty method
2. This option can be used to apply nodal forces at any point (it is useful to apply nodal forces to large foundation rafts at any point)

**Window 2-787:** `Nodal link:Create:On node(s) aut. sel. elements`

This option is equivalent to the one described in Win.2-786 but elements (the ones visible in the graph pane) to which selected nodes are linked are selected automatically by the preprocessor.




Window 2-787

**Window 2-788:** `Nodal link:Select:All`

This option selects all nodal links.

Window 2-788

**Window 2-789:** `Nodal link:Select:With exist.f., label`

Using this option user may select nodal links with a given existence function ID (applied to some constrained DOF) or a given label. In case of a label one may select a single item from combo-box.

Window 2-789

**Window 2-790:** `Nodal link:Update:Parameters`

This option activates the dialog box to modify the applied nodal link definition (degrees of freedom to be constrained), for a single picked nodal link or group of selected nodal links. The detailed description on how to set the data is given in Window 2-786.

Window 2-790

## 2.3.1.3.26 MESH TYING

Mesh tying is a technique to connect two nonconforming continuum element meshes in 2D/3D without penalty factors.

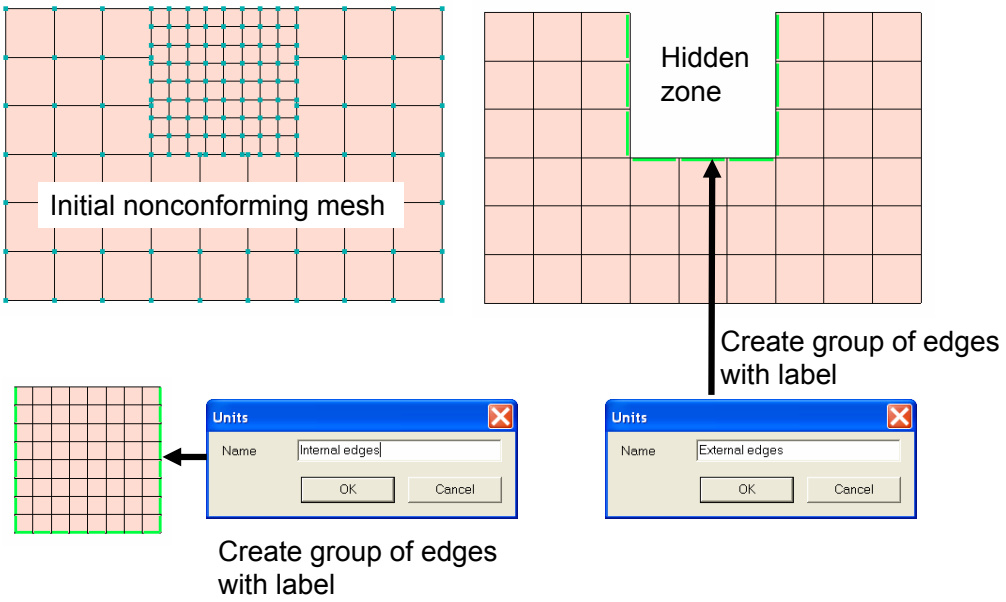
**Window 2-791: Options in menu** `Mesh tying`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `Group of edges` | — | Window 2-792 |
| | `Group of faces` | — | Window 2-793 |
| | `Mortar interface` | — | Window 2-794 |
| `Delete` | `Delete` |  | Window 2-320 |
| `Update` | `Parameters` | — | Window 2-795 |
| | `Scale` | — | Window 2-796 |

Window 2-791

**Window 2-792:** `Mesh tying:Create:Group of edges`

Using this option one may apply a label to the set of selected 2D continuum element edges (see example below) to be used, later on, for definition of 2D mortar interface.




**Remarks**:

1. Edges must not overlap for unique selection
2. One side of the boundary must be fully selected as mortar interface should not be combined of several small fragments
3. Applying boundary conditions at nodes in the mortar interface may cause some singularities or lead to overconstrained problem (too many constraints at single point) and for that reason it should be avoided

Window 2-792

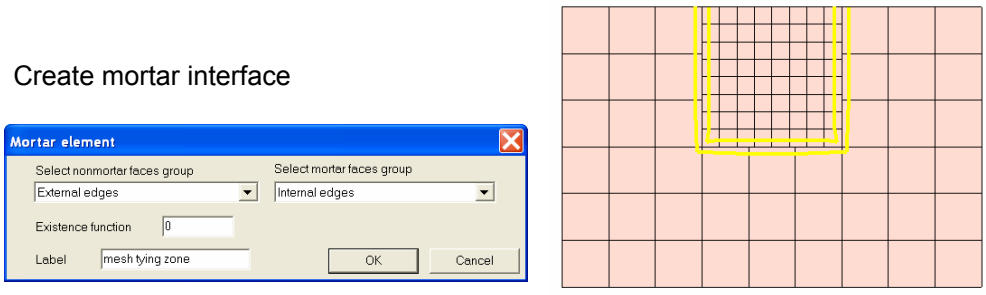
**Window 2-793:** `Mesh tying:Create:Group of faces`

Using this option one may apply a label to the set of selected 3D continuum element faces (see example below) to be used, later on, for definition of 3D mortar interface. All further steps are exactly the same as for the option `Group of edges` (see Window 2-792)

Window 2-793

**Window 2-794:** `Mesh tying:Create:Mortar interface`

This option allows to combine pairs of selected groups of edges/faces to create mortar interface (interface at which full compatibility of any type of degree of freedom is satisfied).




**Remarks**:

1. One may create several mortar interfaces but they cannot interact
2. Mortar interface activity in time is driven by the existence function

Window 2-794

**Window 2-795:** `Mesh tying:Update:Parameters`

This option activates the dialog box to modify definition of mortar interfaces. The description on how to handle this definition is given in Window 2-794.

Window 2-795

**Window 2-796:** `Mesh tying:Update:Scale`

Using this option one may scale graphical representation of mortar interface zone.

Window 2-796

### 2.3.1.4 PUSHOVER CONTROL NODE

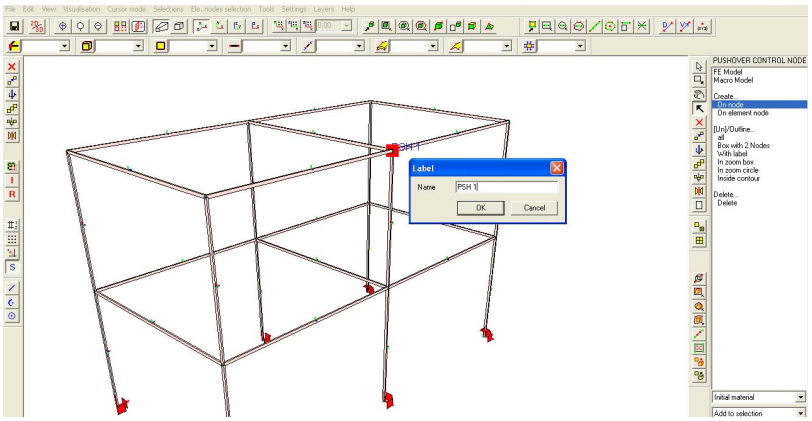
**Window 2-797: Options in menu** `Pushover control node`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On node` | — | Window 2-798 |
| | `On node-reference node` | — | Window 2-799 |
| | `On element node` | — | Window 2-798 |
| `Select` | `All` | — | |
| | `With label` | | |
| `Delete` | `Delete` |  | |

Window 2-797

**Window 2-798:** `Pushover control node:Create:On node`

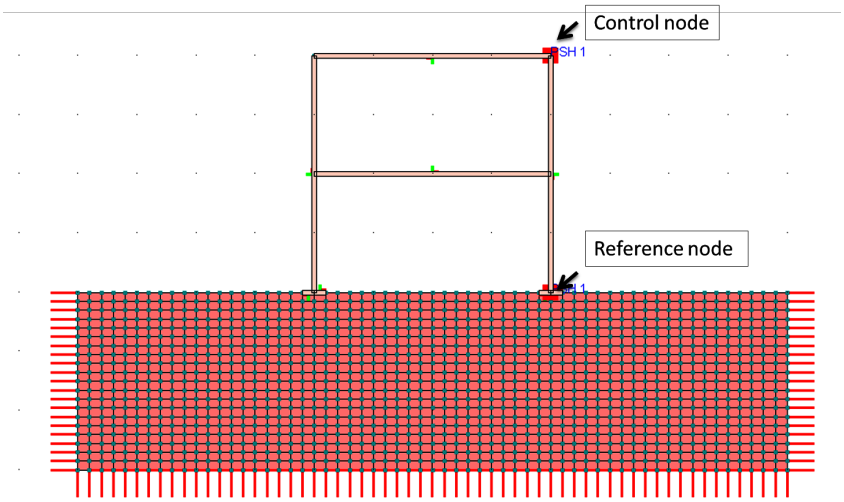
Using this option one may set `Pushover control node` on a single picked node (or element node). This setting is obligatory for running `Pushover driver`. For identification purposes one should apply a distinct label to the `Pushover control node`. One `Pushover control node` can be used by few pushover drivers. The target displacement is traced in this node.




Window 2-798

**Window 2-799:** `Pushover control node:Create:node-reference node`

This option allows to trace target displacement as a differential one between the two selected nodes. This is very important for cases in which structure is embedded in the subsoil.




Window 2-799

### 2.3.1.5 BOREHOLES

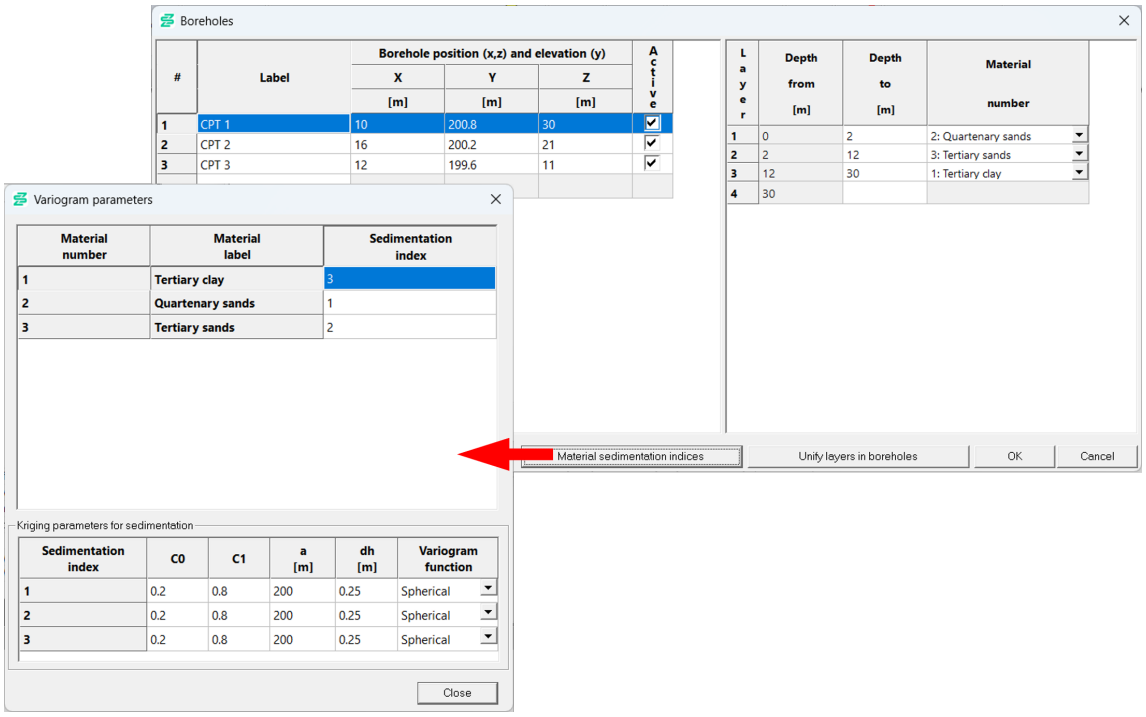
**Window 2-800: Options in menu** `Boreholes`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `From dialog` | — | Window 2-801 |
| | `Import` | — | Window 2-803 |
| `Delete` | `Delete` |  | |
| `Update` | `Parameters` | — | Window 2-804 |
| | `Map materials on visible elements` | — | Window 2-806 |
| | `Map materials on selected elements` | — | Window 2-805 |

Window 2-800

**Window 2-801:** `Boreholes:Create:FromDialog`

This option allows the user to add geological profiles at boreholes and then to project materials from boreholes to the real 3D continuum mesh. This way one may create a uniform 3D mesh avoiding strict generation of all 3D complex geological strata. In addition one may set the order of sedimentation for given set of materials. This helps to avoid locating tertiary soils above quarternary ones for instance. By default all materials have sedimentation index equal to 1. Soils with lower sedimentation index are located below soils that have higher index. Each sedimentation layer may have different variogram setting.




**Remarks**:

- So far this option is valid for boreholes oriented exclusively towards the vertical (y) axis
- To edit the profile a list of predefined materials must be set up first at the material level
- Location of the borehole in the x-z plane is defined by the two coordinates **X** [ ] and **Z** [ ] while its elevation (y coordinate of the ground surface) is defined in **Y** [ ] edit field
- Minimum 3 boreholes are needed
- The kriging procedure is used for mapping of the materials from boreholes to the real mesh (see Win.(2-802) for details concerning kriging parameters)
- Depths are assumed to be positive numbers with zero value at the top of the borehole
- Number of material layers in the borehole can be any
- It is recommended that all boreholes have the same depth

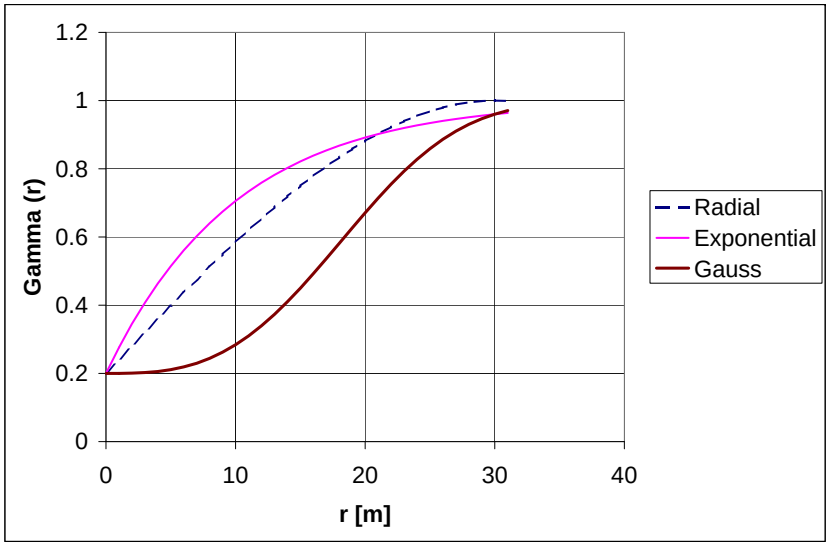
- To update boreholes setting use option `Create/From dialog`

Window 2-801

**Window 2-802:** `Boreholes: setting variogram data for kriging`

The main difficulty in the kriging procedure is to set up the proper shape of the variogram. In the current version one may choose the three types of the variograms

- Radial (spherical)
  $\widetilde{\gamma}(r) = C0 + C1\left(\frac{3}{2}\frac{r}{a} - \frac{1}{2}\frac{r^3}{r^3}\right)$ for $r < a$
- Gauss
  $\widetilde{\gamma}(r) = C0 + C1\left(1 - \exp -\frac{3r^2}{a^2}\right)$
- Exponential
  $\widetilde{\gamma}(r) = C0 + C1\left(1 - \exp -\frac{3\|r\|}{a}\right)$




**Remarks**:

1. The $a$ parameter defines at which distance variogram reaches plateu
2. The $C0$ and $C1$ define the shape of the variogram
3. The $dh$ parameter cancels thin layers in the subsoil that are thiner than the given value

Window 2-802

**Window 2-803:** `Boreholes:Create:Import`

The two formats *.txt and *.csv are supported. Both formats have the same syntax but the only difference between the two is the data separator. In the *.txt file space character is the separator while in the *.csv file the list separator defined in the regional settings of Windows system (usually ;). This format is as follows:

*Borehole-1 label*
*X-borehole-1 Y-borehole-1 Z-borehole-1*
*floor elevation of layer-1 material index*
*floor elevation of layer-2 material index*
....
....
*floor elevation of layer-N-1 material index*
*floor elevation of layer-N material index*
*empty line*

The above sequence must be repeated for each borehole. In the *.txt leave spaces as separators while in the *.csv windows list separator (see regional settings in your system).

Window 2-803

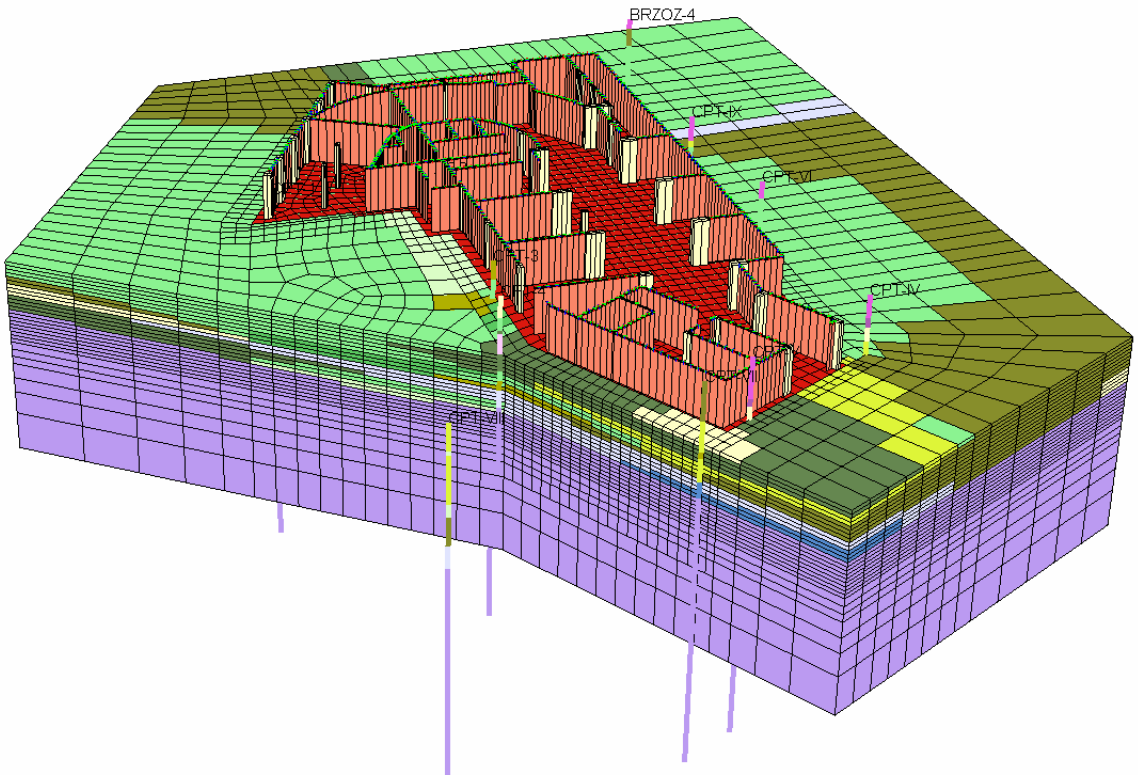
**Window 2-804:** `Boreholes:Update:Parameters`

Using this option one can modify data of the picked borehole. See window 2-801 for details concerning borehole setup.

Window 2-804

**Window 2-805:** `Boreholes:Update:Map on selected elements`

This option performs mapping of geological strata, extrapolated from the set of boreholes, on the real continuum 3D mesh but exclusively for selected set of elements. By using this operation only `Initial material` is updated for selected element. If we consider a set of several boreholes then the result of material mapping will be as shown in the next figure. It has to be mentioned that this result may strongly depend on the type and parameters of the variogram that is used in the kriging procedure.




**Remarks**:

1. If some of the boreholes are outside current computational domain it is recommended to include them in the kriging routine

Window 2-805

**Window 2-806:** `Boreholes:Update:Map on visible elements`

This option works in the same manner as the one given in Win.(2-805) but on all currently visible continuum 3D elements.

Window 2-806

### 2.3.1.6 MOVEMENT PATHS

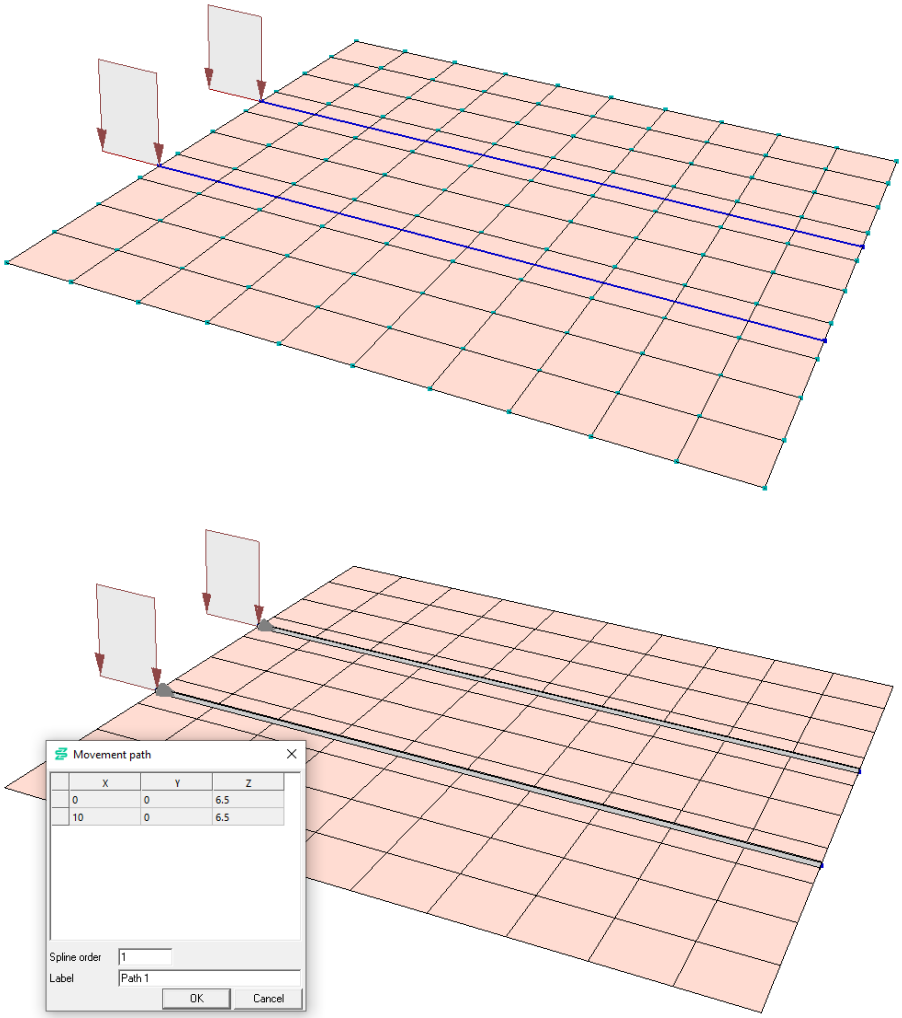
**Window 2-807: Options in menu** `Movement Path`

| Option | Method | Toolbar | Description |
|---|---|---|---|
| `Create` | `On object(s)` | — | Window 2-808 |
| | `On Beams/Trusses/Membrane(2D)` | — | Window 2-809 |
| | `On Edges` | — | Window 2-810 |
| | `By Point/Direction/Distance` | — | Window 2-811 |
| `Delete` | `Delete` |  | |
| | `Delete all` | — | Window 2-813 |
| | | | |
| `Update` | `Parameters` | — | Window 2-814 |
| | `Reverse orientation` | — | Window 2-815 |

Window 2-807

**Window 2-808:** `Movement path:Create:On Object(s)`

This option enables the user to design movement path on selected geometrical objects that can be used to define moving loads and masses. The necessary condition in this definition is such that selected objects create a unique continuous line/polyline or a curve. In case of polylines one may use spline smoothing by selecting proper spline order (from 1 to 3).




**Remarks**:

- In the current version list of characteristic points cannot be modified

Window 2-808

**Window 2-809:** `Movement path:Create:On Beams/Trusses/Membranes (2D)`

This option enables the user to design movement path on selected 1D elements such as beams, trusses or membranes (in 2D). The necessary condition in this definition is such that selected elements must form a unique continuous line or polyline. In case of polylines one may use spline smoothing by selecting proper spline order (from 1 to 3).

Window 2-809

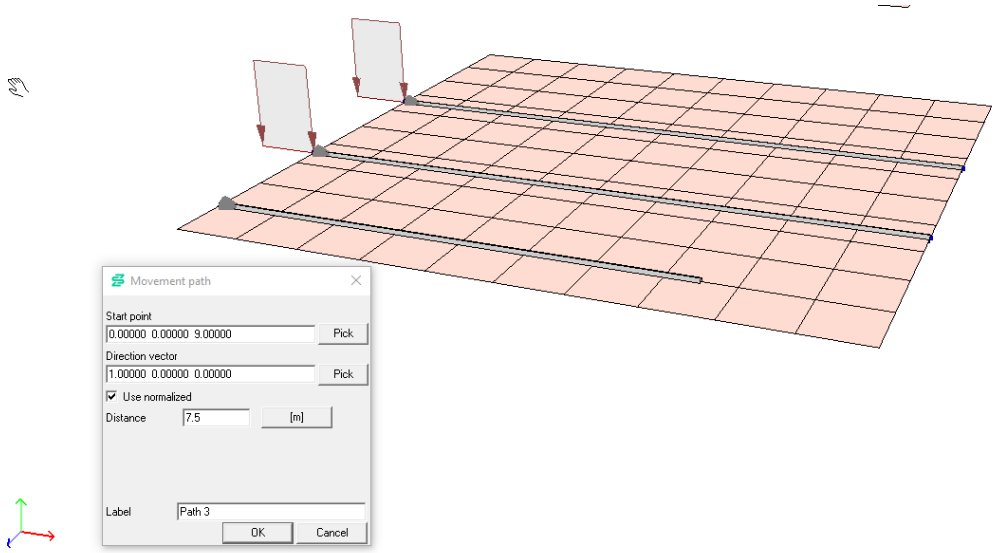
**Window 2-810:** `Movement path:Create:On Edge(s)`

This option enables the user to design movement path on selected edges of continuum elements in 2D, edges of 1D elements like beams, trusses or membranes (2D) or edges of shell elements in 3D. The necessary condition in this definition is such that selected edges must form a unique continuous line or polyline. In case of polylines one may use spline smoothing by selecting proper spline order (from 1 to 3).

Window 2-810

**Window 2-811:** `Movement path:Create:By Point/Direction/Distance`

This option enables the user to design movement path in form of a straight line by setting starting point coordinates, direction vector and a distance.




Window 2-811

**Window 2-812:** `Movement path:Delete:Delete`

This option enables the user to delete picked movement path or selected set of paths.

Window 2-812

**Window 2-813:** `Movement path:Delete:Delete all`

This option enables the user to delete all movement paths.

Window 2-813

**Window 2-814:** `Movement path:Update:Parameters`

This option enables the user to update movement path parameters. In case of definition by point/direction and distance one may redefine each parameter used in the definition. In all other cases only the label and/or spline order can be changed.

Window 2-814

**Window 2-815:** `Movement path:Update:Reverse orientation`

This option enables the user to inverse orientation of the movement path. The orientation marker in form of a gray cone is always located at the starting point.

Window 2-815

## 2.3.2 MESH INFO

**Window 2-816: Summary of computational model**

Displays short reports on current data containing the **number of**:

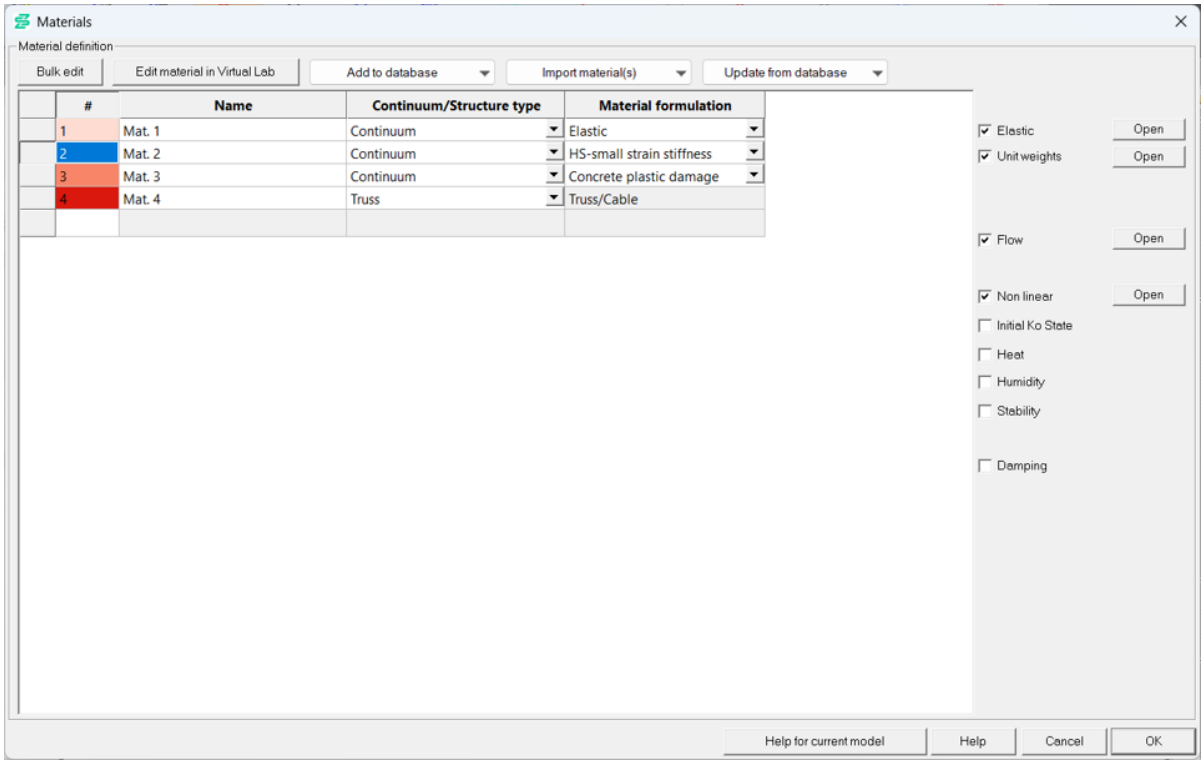
... beam elements
... beams relaxation
... boundary conditions
... contact elements
... convection elements
... different materials
... existence functions
... heat boundary conditions
... humidity boundary conditions
... infinite elements
... initial conditions
... initial heat conditions
... initial humidity conditions
... initial stress conditions
... kinematic constrains
... load time functions
... loads for trusses
... membrane elements
... nodal forces
... nodes
... pressure heads loads
... quad elements
... seepage elements
... single layer shell elements
... shell elements
... surface loads
... truss elements
... volumic elements
... water boundary conditions
... water flux

Window 2-816

## 2.3.3 MATERIALS

**Window 2-817: Definition of materials**

The user interface used to define materials are shown in figure below.




- **To add a new material**
  1. Enter material number in last empty row
  2. Enter material label in `Name` column (recommended)
  3. Choose type of the element class from combo-box `Continuum & structure type` to which introduced constitutive model will be applied
  4. Choose a suitable constitutive model from combo-box `Material formulation`
  5. To apply material from the user data base press button **Import material(s)**, select menu item `Import material from database` and follow steps described in Window 2-818
  6. To apply material from the Virtual Lab data base press button **Import material(s)** and select menu item `Import material from Virtual Lab Database`
- **To delete selected materials** from list use **Delete** key on a keyboard

**Remarks**:

1. Wrong application of a selected material to the finite element(s) in the computational model will cause error report and computation will not be started
2. Material properties are grouped in several categories (see Window 2-819); to edit them in one of these groups press button **Open** placed next to the category check-box
3. List of available categories is dynamically set up according to the selected element class, constitutive model and `Problem type`
4. Material properties can be memorized in the temporary or external databases (see Window 2-818 for the details
5. Materials can be imported from another *.inp file (press button **Import material(s)**, select menu item `Import materials from *.inp`)

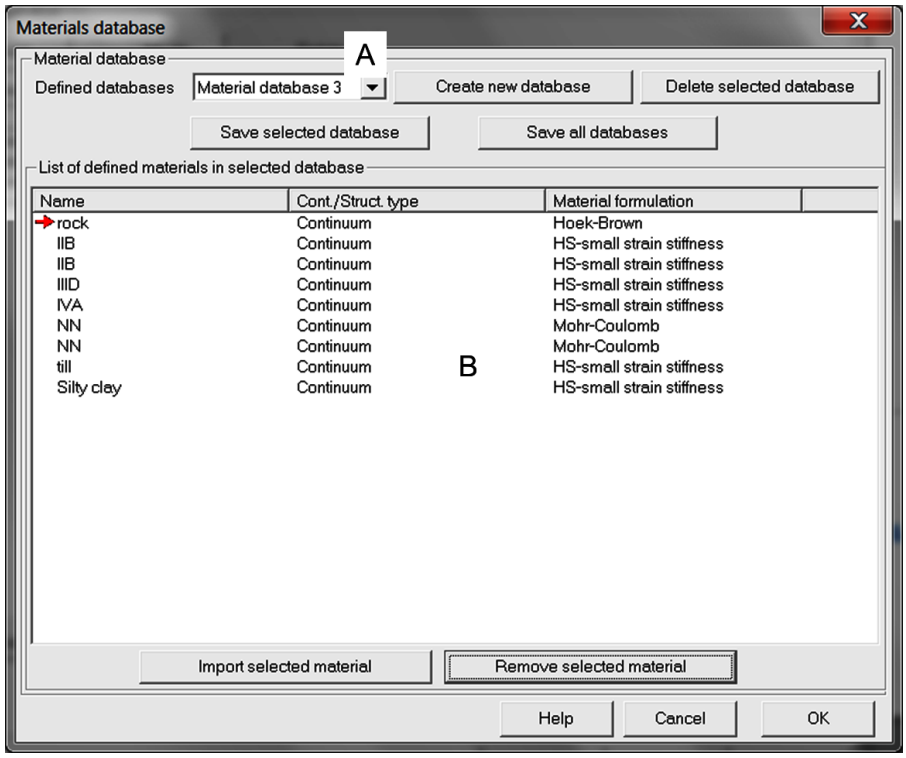
6. Materials can also be imported from the Virtual Lab database (press button **Import material(s)**, select menu item `Import materials from Virtual Lab database`)
7. Properties can be defined as variable in time (see Window 2-915 for details) and space (see Window 2-914 for details)
8. For plastic damage model for concrete used in the continuum or shell elements extension to fire problems (according to the Eurocode) is possible
9. Elastic-plastic 1D model for truss elements can also handle fire problems

Window 2-817

## 2.3.3.1 USER MATERIAL DATA BASE

**Window 2-818: User material database**

Material data base is an user data bank created himself to collect material definitions, with a distinct label, for further use.




1. `Defined databases` combo-box (A) opens an access to the selected database
2. **Create new database** creates an empty and new database with a given label
3. **Delete selected database** removes database selected from combo-box (A)
4. **Save selected database** saves database selected from combo-box (A)
5. **Save all selected databases** saves all databases
6. **Import selected material** indicates which material (from list (B)) should be used by the parent dialog box used (see Window 2-817)
7. **Remove selected material** cancels material selected from list (B)in the selected database indicated in combo-box (A)

Window 2-818

## 2.3.3.2 CONTINUUM / STRUCTURE TYPE

**Window 2-819: Material models, categories of properties**

Obligatory categories of properties to be set are denoted by ✓ while optional by ☐.

| Continuum /Structure | Material Formulation | CREEP | ELASTIC | FLOW | GEOMETRY | HEAT | HUMIDITY | INITIAL K0 | NON LINEAR | STABILITY | UNIT WEIGHTS | DAMPING |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| BEAM | Beam | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| | Axisymmetric Shell | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| CONTACT | Contact | | ✓ | ✓ | | | | | ✓ | ☐ | ✓ | |
| PILE INTERFACE | Pile interface | | ✓ | ✓ | | | | | ✓ | ☐ | ✓ | |
| PILE TIP INT. | Pile tip interface | | ✓ | ✓ | | | | | ✓ | ☐ | ✓ | |
| NAIL INTERFACE | Nail interface | | ✓ | ✓ | | | | | ✓ | ☐ | ✓ | |
| ANCHOR INTERFACE | Fixed anch.zone interf. | | ✓ | ✓ | | | | | ✓ | ☐ | ✓ | |
| CONTINUUM FOR STRUCTURES: | Elastic | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | | ☐ | ✓ | ☐ |
| CONTINUUM | | | | | | | | | | | | |
| linear: | Elastic | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | | ☐ | ✓ | ☐ |
| nonlinear: | Aging Concrete | | ✓ | ✓ | | ☐ | ☐ | ☐ | ☐ | | ✓ | ☐ |
| | Cap | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Modified Cam-Clay | | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Drucker–Prager | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Hoek–Brown (M-W) | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Mohr–Coulomb | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Mohr–Coulomb(M-W) | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Multilaminate | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Rankine(M-W) | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | HS-small | | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Densification | | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Hoek-Brown | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| | Plastic damage... | ☐ | ✓ | ✓ | | ☐ | ☐ | ☐ | ✓ | ☐ | ✓ | ☐ |
| HEAT CONVECTION | Heat Convection | | | | | ✓ | | | | | | |
| HUMIDITY CONVECTION | Humidity Convection | | | | | | ✓ | | | | | |
| INFINITE MEDIA | Elastic infinite media | | ✓ | | | | | | | | | |
| MEMBRANES | | | | | | | | | | | | |
| linear: | Elastic | | ✓ | | ✓ | ☐ | ☐ | | | | ✓ | ☐ |
| nonlinear: | Anisotropic | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| | Fiber | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| | Isotropic | | ✓ | | | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| | Hoek-Brown | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| | Huber-Mises | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| | Rankine | | ✓ | | ✓ | ☐ | ☐ | | ☐ | ☐ | ✓ | ☐ |
| SEEPAGE | Seepage | | | ✓ | | | | | | | | |
| SHELLS | Shell | | ✓ | ☐ | ☐ | ☐ | ☐ | | ☐ | | ✓ | ☐ |
| | Orthotropic shell | | ✓ | ☐ | ✓ | ☐ | ☐ | | ☐ | | ✓ | ☐ |
| TRUSS | Truss/cable | | ✓ | | ✓ | ☐ | | | ☐ | ☐ | ✓ | ☐ |

| Continuum /Structure | Material Formulation | CREEP | ELASTIC | FLOW | GEOMETRY | HEAT | HUMIDITY | INITIAL K0 | NON LINEAR | STABILITY | UNIT WEIGHTS | DAMPING |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| BEAM NONL.HINGE | Uncoupled axial | | | | | | | | ✓ | | | |
| | Uncoupled tangent | | | | | | | | ✓ | | | |
| | Uncoupled torsional | | | | | | | | ✓ | | | |
| | Uncoupled flexural | | | | | | | | ✓ | | | |
| | Coupled flexural (Janssen) | | | | ✓ | | | | ✓ | | | |
| SHELL NONL.HINGE | Uncoupled axial | | | | | | | | ✓ | | | |
| | Uncoupled tangent | | | | | | | | ✓ | | | |
| | Uncoupled flexural | | | | | | | | ✓ | | | |
| | Coupled flexural (Janssen) | | | | ✓ | | | | ✓ | | | |

Window 2-819

### 2.3.3.2.1 BEAMS AND AXISYMMETRIC SHELLS

Material model description, designed for beam or axisymmetric shell element, consists of :

- definition of cross section geometry
- definition of material properties

The above data is specific for current analysis type ( 2D, 3D or axisymmetry), hence it is recommended to select proper problem type in the initial pre-selection dialog box.

The two following models are available:

**BEAM MODEL**

**AXISYMMETRIC SHELL**

## Related Topics

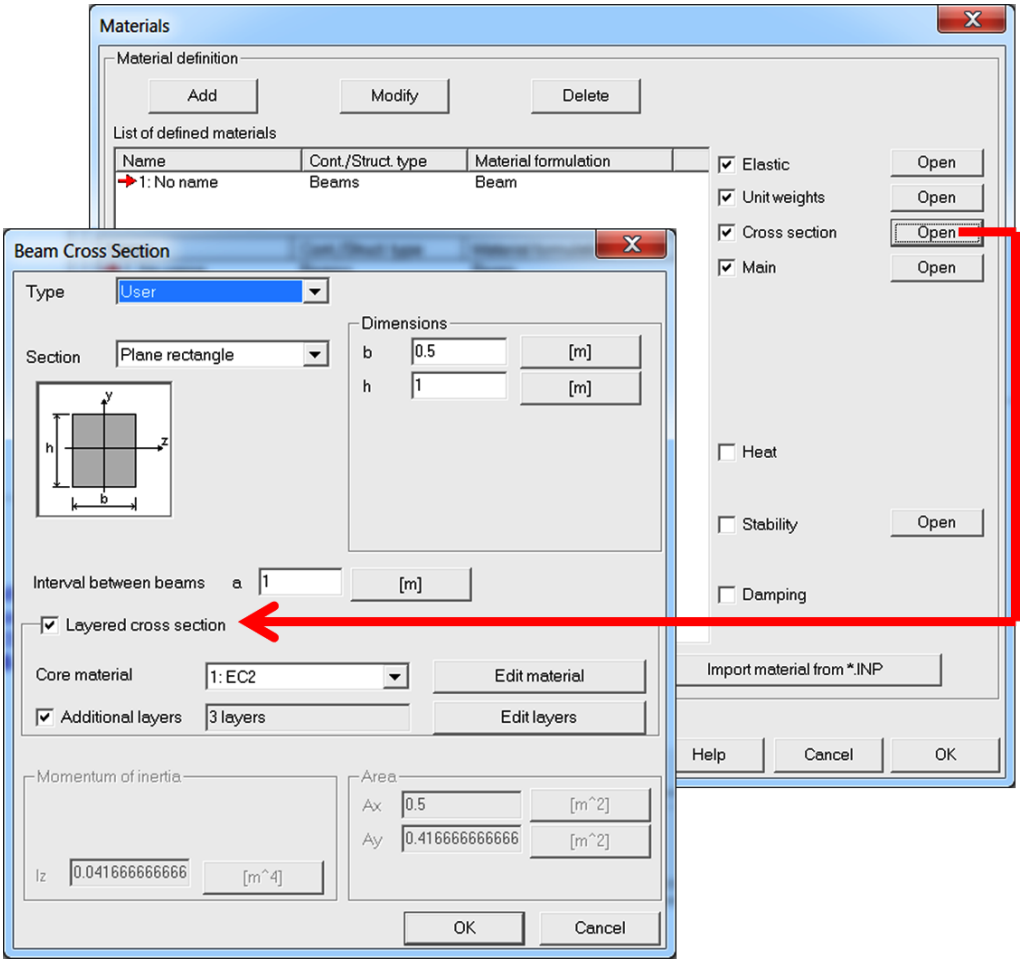
*Theory:*

- BEAMS THEORY

## 2.3.3.2.1.1 BEAM MODEL

**Window 2-820: Selecting linear/nonlinear (layered) beam model**

Activation of linear (integral)/nonlinear (layered) beam model can be made by setting OFF/ON ☒ **Layered cross section** under option ☒ **Cross section** in the main material dialog box (see Fig. below)




**Remarks**:

1. For discrete beams, in plane-strain, geometrical characteristics like cross section area and momentum of inertia are given per beam; the equivalent characteristics (computed for model with thickness of an unit slice) are automatically computed for a given distance $a$ (to be set in group ☒ **Geometry**) between beams (in $z$ direction) as follows:

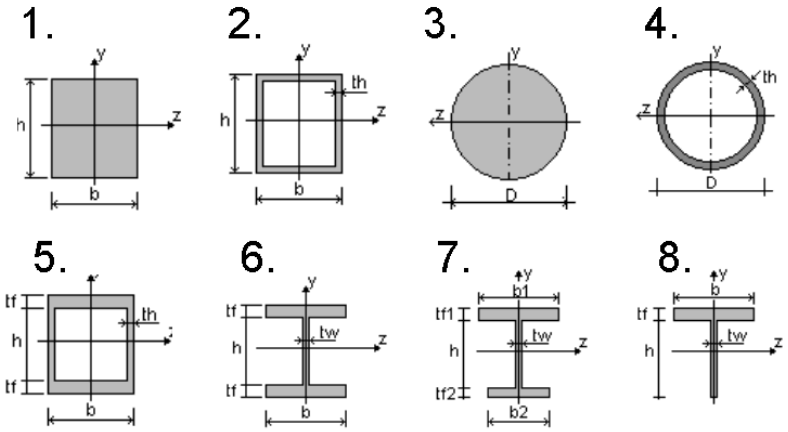
$$A' = A/a; \qquad I' = I/a$$

2. In axisymmetry beam elements are converted to axisymmetric shells so $a = 1$.
3. Cross section can be defined (under group ☒ **Cross section**)
   - as profile from databases (see Win.(2-822))
   - as user defined cross section (see Win.(2-821))
   - by direct values (see Win.(2-823))

Window 2-820

**Window 2-821: Cross section type:** `User defined`

The following predefined cross section shapes are supported:




1. Plane rectangle
2. Rectangular tube
3. Plane circular
4. Circular tube
5. Box
6. I-bisym
7. I-asym
8. T-section

**Remark**:This setting is valid for both linear/nonlinear beam model

Window 2-821

**Window 2-822: Cross section type:** `Profiles`

The following steel profiles databases are available:

- **Aisc.pro** (American)
- **CatPro.pro** (European)
- **CISC.pro** (Canadian)
- **Korea.pro** (Korean)
- **NS_3472.pro** (Norwegian)
- **Polish.pro** (Polish)
- **StalProdukt.pro** (Polish (StalProdukt))
- **Suomi.pro** (Finnish)
- **USTK.pro** (UK)

To select cross section from database

- Set `Type` to `Profiles`
- Select one of the above databases
- Select profile symbol from list `Section`

**Remark**:This setting is valid for both linear/nonlinear beam model

Window 2-822

**Window 2-823: Cross section type:** `Values`

In this specific case direct values of cross section area, shear area, momentum of inertia for bending and in 3D momentum of inertia for torsion must be introduced.

**Remark**:This setting is exclusively valid for linear beam model

Window 2-823

**Window 2-824: Linear/nonlinear beam: common data groups**

| Data | Use & Set up |
|---|---|
| `Unit Weight` | optional;<br>• $\gamma$, $\{[\text{kNm}^{-3}]; [> 0]; 0\}$ - weight of beam cross section per unit length (averaged over the whole section);<br>• Density $\rho = \frac{\gamma}{g}$ is automatically computed from $\gamma$ (and vice versa)<br>• Multiplier for $\gamma$ $\{[-]; [\geq 0]; 1.0\}$<br>• Multiplier for $\rho$ $\{[-]; [\geq 0]; 1.0\}$<br>Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor) while setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if `Dynamics` or `Pushover` is set ON in the project preselection) |
| `Main` | obligatory;<br>• Beam FE formulation type<br>Select one of ◎ **Displacement** or ◎ **Flexibility based**. The first one is a classical displacement formulation in which beams are represented by a 2-node linear elements with a single integration point located at the element center. A single member must be split into several beam elements in order to get sufficient accuracy of results. In this case diagram of bending moment is postprocessed and its linear part is recovered by using shear force value ($M = M_o \pm Q\ L/2$). Other internal forces are plot as constant all over the element. The second formulation is a non-classical one in which any member can be represented by a single 2-node beam element with 5 Gauss-Lobatto integration points. This element is highly accurate (practicaly exact for linear/nonlinear static problems) and recommended for pure structural applications where single beam element is sufficient per structural member. Values of internal forces at any integration point are exact. In case of soil-structure interaction problems (sheet-pile walls, diaphragm walls etc..) beams adjacent to continuum must be split anyway (to preserve mesh conformity and to put contact among them) so standard displacement formulation is sufficient. The flexibility based formulation is more expensive in terms of computational time. |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient. In case of active `Data Group Heat` and active associated `Heat Transfer` project (if $\alpha > 0$), thermal strains will be evaluated at all integration points of the cross section. |
| `Stability` | optional; Handle material strength parameters reduction, see: *LOCAL STABILITY* for the details. |
| `Damping` | optional (same definition as in the global *Control/Dynamics dialog*) |

**Window 2-825: Linear beam: specific material data**

Elastic beam model with an arbitrary cross section, is described by its integral characteristics (area and momentum of inertia) and by elastic constants. In that case the following data is required:

| Data | Use & Set up |
|---|---|
| `Elastic` | obligatory; set:<br>• Young's elastic modulus $E\{[\text{kPa}];[>0];100000\}$<br>• Poisons ratio $\nu\{[-];[0,0.49999];0.3\}$,<br>Both will be used to compute shear Kirchoff modulus $G = \frac{E}{2(1+\nu)}$ |
| `Cross section` | obligatory; cross–section characteristics, setting dependent on analysis type:<br>• $a\{[\text{m}];[>0];1\}$ – the distance between members with given section, for `Plane Strain`<br>• $A$ $\{[\text{m}^2];[>0];1\}$ – cross–section area, for all `Analysis Type`<br>• $I_x\{[\text{m}^4];[>0];1\}$ – torsional momentum of inertia, for 3D only<br>• $I_y$ $\{[\text{m}^4];[>0];1\}$ — momentum of inertia in bending around local $Y_L$ axis (i.e. in XZ plane), for 3D only<br>• $I_z$ $\{[\text{m}^4];[>0];1\}$ – momentum of inertia in bending around local $Z_L$ axis (i.e. in XY plane), for all `Analysis Type`<br>• $A_y = \kappa_y A\{[\text{m}^2];[>0];1\}$ – shear section area in $Y_L$ direction ($\kappa_y$ is a shear correction factor), for all `Analysis Type`<br>• $A_z = \kappa_z A\{[\text{m}^2];[>0];1\}$ – shear section area in $Z_L$ direction ($\kappa_z$ is a shear correction factor) for 3D |

**Remarks**:

1. Hand setting of cross-sectional characteristics is possible only if `Type` is set to `Values` (default on entry)
2. For convenience, use provided utilities for setting cross section characteristics, setting `Type` alternatively to:
   - `Profiles` - to set a typical manufactured (steel) section choosen form available data base.
   - `User` - to enter geometrical data of predefined types.
3. In both cases all necessary cross section characteristics are automatically evaluated and shown in edit fields. After switching `Type` back to `Values` modifications of each data will be again possible.
4. Large values of shear correction factor eliminate shear stiffness (switching to Kirchhoff–Bernoulli hypothesis instead of Timoshenko one).

**Window 2-826: Nonlinear beam: specific material data**

Nonlinear (layered) beam section model is represented by a set of layers, each described by its area and 1D elastic, elastic-perfectly plastic or user defined, model material data. In that case the following data is required:

| Data | Use & Set up |
|---|---|
| `Elastic` | optional<br>If `Automatic evaluation of shear modulus` is set ✓ (default), Kirchoff modulus $G$ will be evaluated automatically basing on $E, \nu$ of `Core` material. Otherwise set:<br>• Young's elastic modulus $E\{[\text{kPa}]; [> 0]; 100000\}$,<br>• Poisons ratio $\nu\{[-]; [0, 0.49999]; 0.3\}$.<br>Both will be used only to create shear Kirchhoff modulus $G = \frac{E}{2(1+\nu)}$, independently of `Core` material data.<br>In any case, elasticity modulus $E_X$ in axial direction has to be entered via `Cross section`→ `Core material` → `Edit material` |

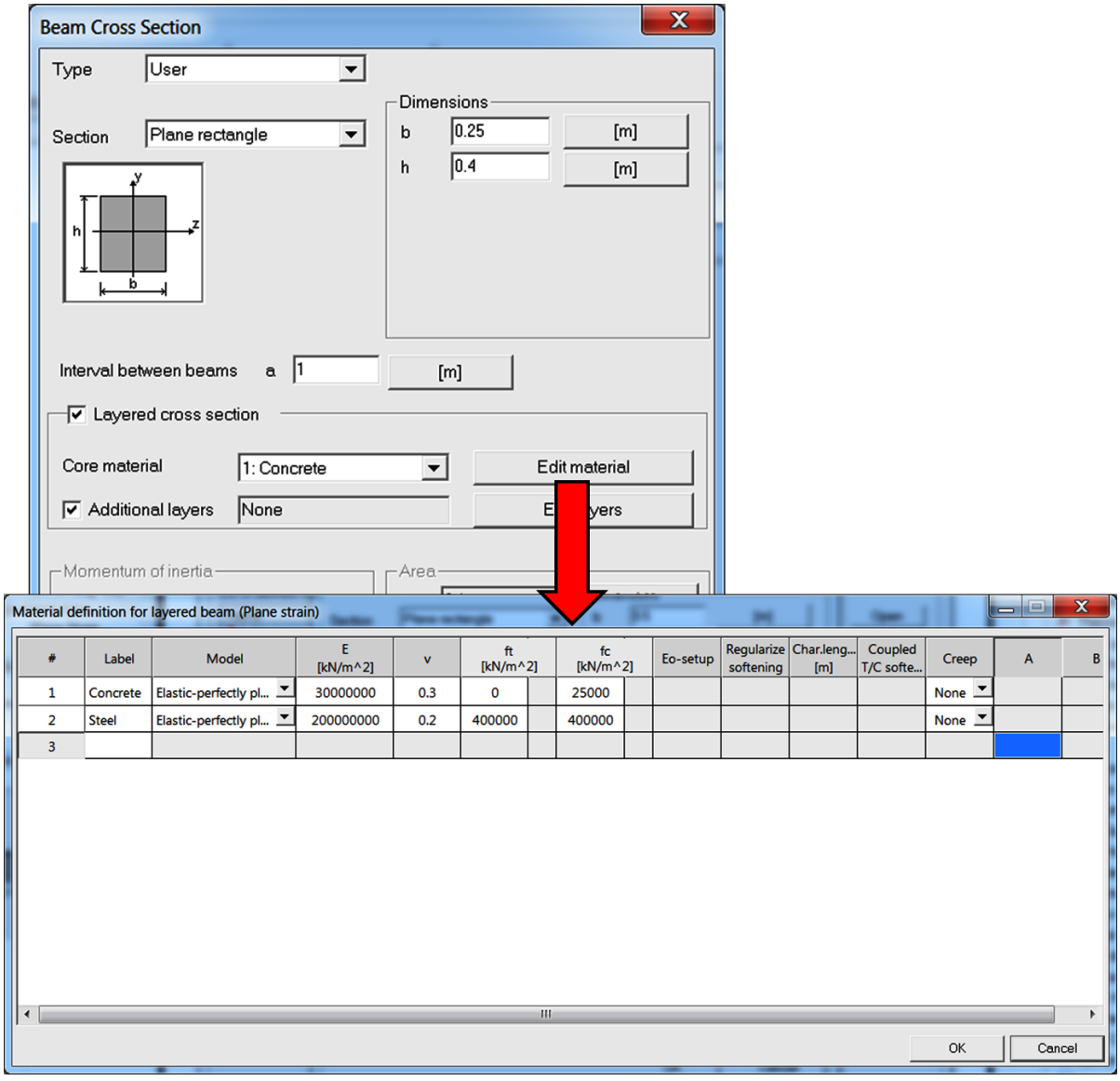
continued on next page

**continued from previous page**

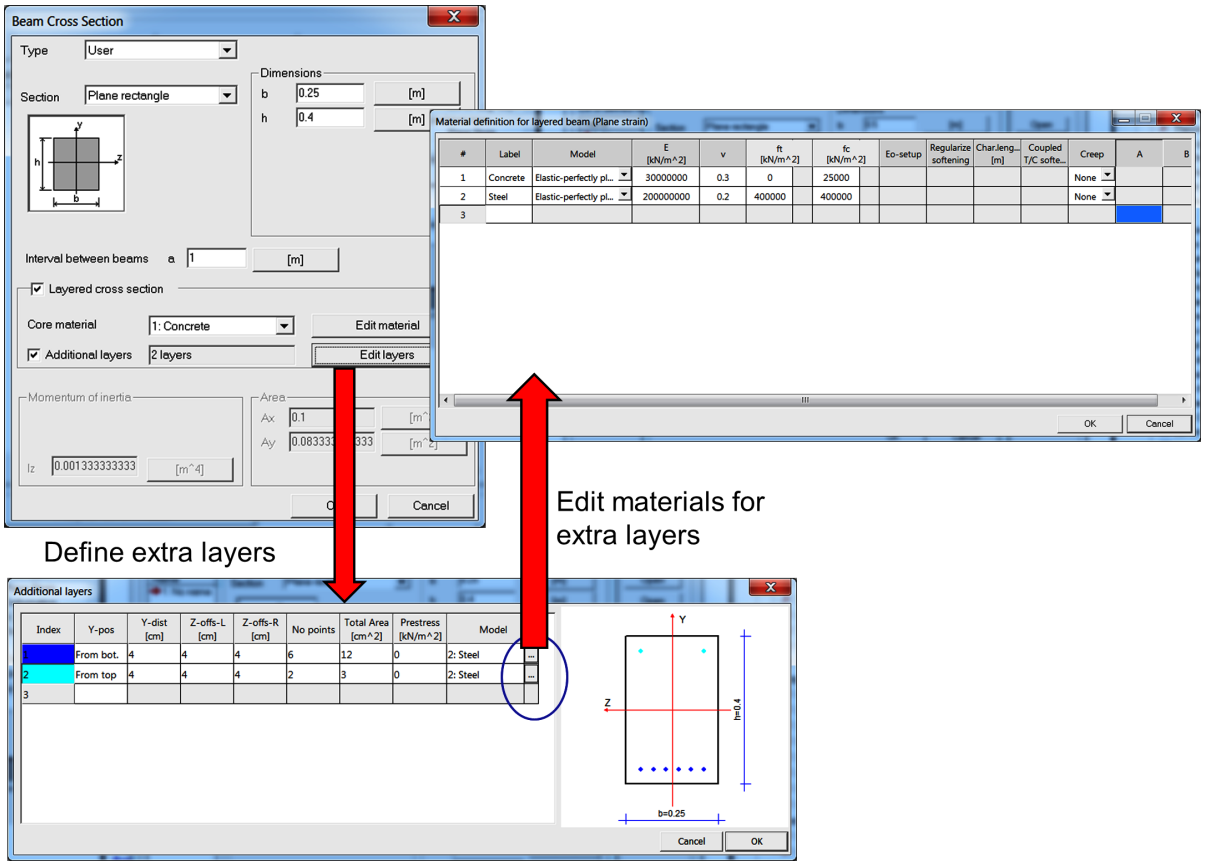
| | |
|---|---|
| Cross section | obligatory<br>Selection between section Types has to be done. Use alternatively:<br>• Profiles - to set a typical manufactured (steel) section choosen form available data base.<br>• User - to enter geometrical data of predefined types.<br>In both cases cross section area will automatically be divided into layers with uni-axial elastic/elasto-plastic or user defined model marked as Core at each. In addition extra layers (possibly prestressed) can be added to the core material by activating ☒ **Additional layers** option. The latter option can be used to model reinforced concrete, although setting beam reinforcement can be made much easier during preprocessing (under Beam use option Reinforcement sets). Reinforcement sets defined at the preprocessing stage and extra layers defined at the material level will be added. To edit data for core material use ☒ **Edit material** and for extra layers use ☒ **Edit layers**. Moreover, cross sectional characteristics, related to shear or torsion, will be automatically evaluated:<br>• $I_x\{[\mathrm{m}^4];[>0];1\}$ – torsional momentum of inertia, for 3D only<br>• $A_y = \kappa_y A\{[\mathrm{m}^2];[>0];1\}$ – shear section area in $Y_L$ direction ($\kappa_y$ is a shear correction factor), for all Analysis Type<br>• $A_z = \kappa_z A\{[\mathrm{m}^2];[>0];1\}$ – shear section area in $Z_L$ direction ($\kappa_z$ is a shear correction factor) for 3D<br>For Plane Strain set:<br>• $a\{[\mathrm{m}];[>0];1\}$ – the interval between beams with given section (leave $a = 1$ in case of a continues lining). |
| Heat | optional; set:<br>• $\alpha$, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient. In case of active Data Group Heat and active associated Heat Transfer project (if $\alpha > 0$), thermal strains will be evaluated at all integration points of the cross section. |

## Window 2-827: Nonlinear beam: setting layered cross section

The layered cross section can be set up under group **Cross section** by activating the option ☒ **Layered cross section**. Once this option becomes active one may select proper model from the existing list of predefined models or define his own one (use **Edit material** button). In addition one may add extra layers (to add reinforcement bars for instance) (use ☒ **Additional layers** checkbox). To define these layers one has to press button **Edit layers**.




Editing core material




Adding extra layers

**Remarks**:

1. Only one core material may appear in the cross section
2. The following three 1D fiber models can be used: linear, elasto-plastic and user defined; for all of them creep can be activated (the exponential, power, logarithmic and hyperbolic laws can be used; if it is sufficient the exponential creep is recommended as it uses a single Kelvin dashpot instead of few ones as for another laws; creep in beam layers cannot be delayed by means of the existence function;)
3. Extra layers can be prestressed
4. Extra layers can be positioned in the cross section using Y, Z offsets (for all cross sections except circular and circular tube ones, for which offset is set in the radial direction); the area of each layer is the total one for all points representing this layer
5. Extra layers are added to the cross section no matter whether the reinforcement is defined, or not, at the beam level (using option `Reinforcement sets` under menu `Beam` in the preprocessor)

Window 2-827

**Window 2-828: Linear/Nonlinear beam: general remarks**

**Remarks**:

1. Terms `Top`, `Bottom` are related to the orientation of local element coordinate system $Y_L$ axis (positive $Y_L$ points on `Top`). This in turns is related to:

   `Direction`(3D) - as direction vector $d$ projected on section plane determine $Y_L$

   node order (all 2D cases), as $Y_L$ is set anticlockwise versus $X_L$ pointing from the 1st to 2nd node of a beam element

2. For torsional momentum of inertia the value displayed in grayed field is the value related to predefined section type evaluated as

$$I_x = \frac{1}{3}b^4\left(n - 0.63 + \frac{0.052}{n^4}\right), \quad n = \frac{h}{b} > 1$$

   for rectangular section. For I section the value is evaluated as a sum of the above for the web and flanges. The default value may be changed by the user.

3. The model describing shear and torsion of the beam is assumed to remain purely elastic and uncoupled from its, in general, nonlinear bending and axial behavior.

4. The area of reinforcement layers will be neglected in evaluation of shear correctors $\kappa$ and in torsional stiffness $I_x$

5. Geometry of layers related to base area of the section (i.e. `Core`) is fixed and can not be changed.

6. For core material or extra layer elastic, elastic-perfectly plastic or user defined model has to be set.

7. For the elastic/elastic-plastic `Material definition` the following data must be entered:
   - $E\{[\text{kPa}];[>0];100000\}$ – Young's elastic modulus of the component material
   - Poisons ratio $\nu\{[-];[0, 0.49999];0.3\}$. This will be used exclusively for `Core` material, only if under `Elastic`→ `Automatic evaluation of shear modulus` is set ✓ (default). Then Kirchoff modulus $G$ for the whole section will be automatically evaluated basing on $E, \nu$ of `Core` material. In all other cases, i.e. for reinforcement layer $\nu$ is ignored.
   - If `Type` is set to `Elastic-perfectly plastic` then additionally enter:
     ⋆ $f_t\{[\text{kPa}], > 0, 1\}$ – uniaxial tensile strength of the component material
     ⋆ $f_c\{[\text{kPa}], > 0, 1\}$ – uniaxial compressive strength of the component material

8. For the `User defined Material definition` the tension and compression 1D characteristics must be defined via evolution functions ($\varepsilon - \sigma$ relations)(stress/strain values are positive for both tensile and compressive branch); for softening laws set characteristic length $L_c$ to represent properly fracture energies (in tension and compression)

9. Material model data once entered will be stored in the job and may be reused for other sections or components.

## Related Topics

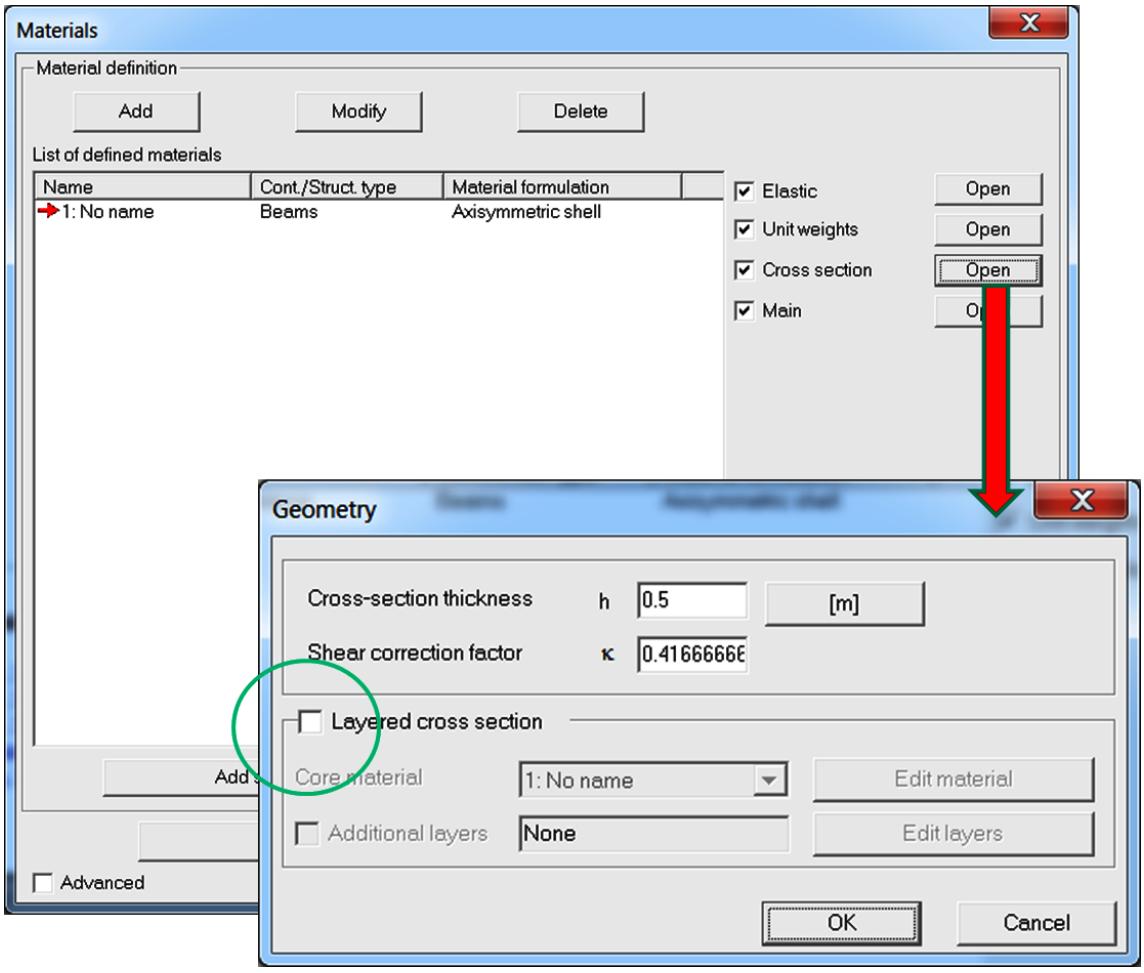
*Theory*

- BEAMS THEORY
- UNI-AXIAL ELASTO–PLASTIC MODEL
- UNI-AXIAL USER DEFINED MODEL

## 2.3.3.2.1.2 AXISYMMETRIC SHELL MODEL

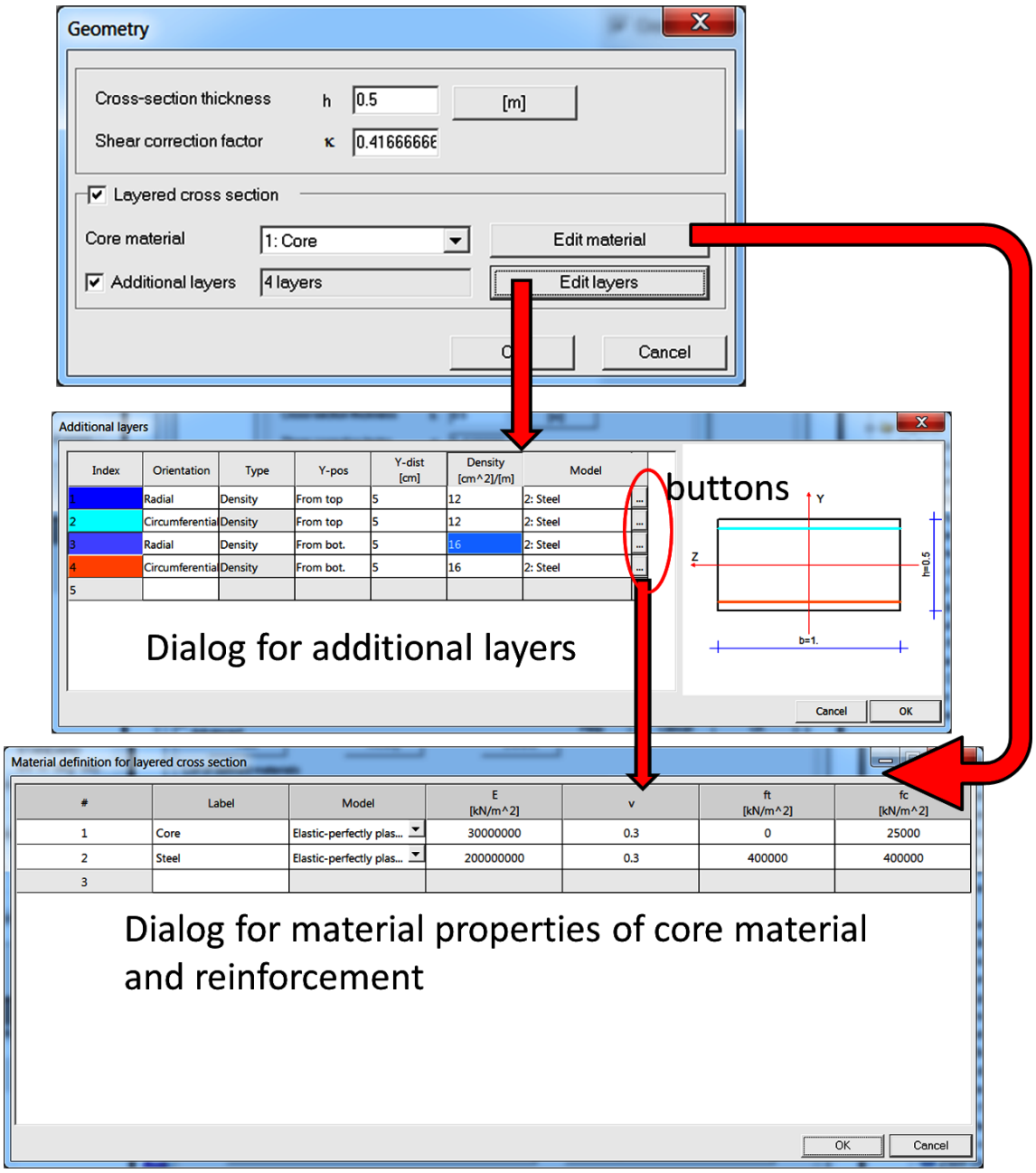
**Window 2-829: Selecting linear/nonlinear(nonlinear) axisymmetric shell model**

Activation of linear/nonlinear (layered) axisymmetric shell model can be made by switching OFF/ON the ☒ **Layered cross section** option under parameters group `Cross section` (see Fig. below) in the main material dialog box. By activating the layered option one has the ability to setup material properties of the core material (elastic or elastic-perfectly plastic one) but also to add additional layers (to model reinforcement for instance) (by setting ON the checkbox ☒ **Additional layers**)). These additional layers are treated as elastic or elastic-perfectly plastic 1D fibers than can be set in the circumferential or in the radial direction, respectively.




Setting shell thickness and activating linear/nonlinear (layered) cross section definition

In the next figure activation procedure for the layered cross section and adding extra layers is shown. Setting material properties for the core material can be accessed after pressing button **Edit material**. By doing this we can edit elastic and plastic properties of the core material. Editing extra layers requires an intermediate step in which these layers are added first and then their properties can be edit by pressing button ... placed in the table in column `Model`.




Editing core material properties and reinforcement layers (and their properties)

Window 2-829

**Window 2-830: Linear/nonlinear(layered) axisymmetric shell model: common data**

The following groups of parameters are common for both linear and nonlinear (layered) axisymmetric shell models:

| Data | Use & Set up |
|---|---|
| `Unit Weight` | optional;<br>• $\gamma$, $\{[\text{kNm}^{-3}];[>0];0\}$ - unit weight of the shell (averaged over the whole section);<br>• Density $\rho = \frac{\gamma}{g}$ is automatically computed from $\gamma$ (and vice versa)<br>• Multiplier for $\gamma$ $\{[-];[\geq 0];1.0\}$<br>• Multiplier for $\rho$ $\{[-];[\geq 0];1.0\}$<br>Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor) while setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if `Dynamics` or `Pushover` is set ON in the project preselection). |
| `Heat` | optional<br>$\alpha$, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient.<br>In case of active `Data Group Heat` and active associated `Heat Transfer` project (if $\alpha > 0$) thermal strains will be computed. |
| `Stability` | optional<br>Handles material strength parameters reduction, see: *LOCAL STABILITY* for the details. |
| `Damping` | Rayleigh damping factors $\alpha_o$ (applies to the mass) $\{[s^{-1}];[\geq 0];0.0\}$ and $\beta_o$ (applies to the stiffness) $\{[s];[\geq 0];0.0\}$; dedicated calculator can be used if needed; |

Window 2-830

**Window 2-831: Linear axisymmetric shell model: specific data**

To complete data definition for linear elastic isotropic shell with constant thickness the following groups of parameters must be set:

| Data | Use & Set up |
|---|---|
| `Elastic` | obligatory<br>• Young's elastic modulus, $E$ $\{[\text{kPa}]; [> 0]; 100000\}$,<br>• Poisson ratio $\nu\{[-]; [0, 0.49999]; 0.3\}$<br>$E$ and $\nu$ constant for whole section, used for all stiffness components |
| `Cross section` | obligatory<br>• shell thickness $h\{[m]; [> 0]; 1\}$<br>• shear correction factor $\kappa\{[-]; [> 0]; 0.83333\}$ The shear stiffness of the section will be then evaluated as: $GA_y = \kappa\frac{Eh}{2(1+\nu)}$<br>NB. Note that large values of shear correction factor will eliminate shear stiffness (Kirchhoff–Bernoulli model) |

Window 2-831

**Window 2-832: Nonlinear axisymmetric shell model: specific data**

To complete data definition for nonlinear isotropic shell (possibly reinforced) with constant thickness the following groups of parameters must be set:

| Data | Use & Set up |
|---|---|
| `Elastic` | optional<br>If `Automatic evaluation of shear modulus` is set ✓ (default), Kirchoff modulus $G$ will automatically be evaluated basing on $E, \nu$ of `Core` material.<br>Otherwise set:<br>• Young's elastic modulus $E\{[\text{kPa}]; [> 0]; 100000\}$,<br>• Poisons ratio $\nu\{[-]; [0, 0.49999]; 0.3\}$.<br>Both will be used only to compute shear Kirchhoff modulus $G = \frac{E}{2(1+\nu)}$ independently of `Core` material data.<br>**Remark**:<br>In any case, elastic constants for each component separately, are set in the material data under group ☒ **Cross section** by activating ☒ **Layered cross section** option. *(to be continued on the next page)* |

| Data Group | Use & Set up |
|---|---|
| `Cross section` | obligatory<br>This option allows to describe a layered axisymmetric shell section model with possibly non–linear behavior for the core material and radial/circumferential reinforcement. The data setting is split into the following phases:<br>• Set shell thickness $h\{[m];[>0];1\}$<br>• The section will automatically be split into core material layers with bi-axial elastic or elasto-plastic model<br>• To enter material model data for core use **Edit material** button. In the grid type dialog box edit elastic and plastic properties of the shell core material.<br>• To activate reinforcing radial/circumferential fibers set ON the option ☒ **Additional layers**; to edit properties of extra layers enter to the dialog box by pressing the button **Edit layers**; in the dialog define additional layers by setting<br>⋆ `Orientation` of the reinforcement in given layer switching between `Radial` and `Circumferential`<br>⋆ `Type` of the reinforcement area definition by selecting `Density` (standard setup in the practice where reinforcement is set with the unit $[m^2/m]$ to preserve certain percentage) (valid for both radial and circumferential direction) or `Total` (kept mostly for back compatibility with previous versions) (here unit is $m^2$)(valid only for the radial direction)<br>⋆ `Y-Position` of the given layer refered to the top, bottom or the middle of the cross section<br>⋆ `Y-distance` that is an offset between selected core fibers defined in column `Y-Position` and current reinforcement layer<br>⋆ `Area` of the reinforcement defined with unit that may depend on the reinforcement `Type` |

**Remarks**:

1. Material data for `Core` as well as all data for additional (i.e. reinforcement) layers has to be entered with use of `Layers editor`. In `List of defined layers` select the layer to `Modify` or `Add` a new one. This will call `Layer definition` dialog.
2. After opening `Material definition` dialog the data can be entered:
   - $E\{[\text{kPa}];[>0];100000\}$ – Young's elastic modulus of the component material
   - Poisons ratio $\nu\{[-];[0,0.49999];0.3\}$. This will be used exclusively for `Core` material. If under `Elastic`→ `Automatic evaluation of shear modulus` is set ✓ (default), then shear Kirchoff modulus $G$ for the whole section will be evaluated automaticaly basing on $E,\nu$ of `Core` material. For reinforcement layer $\nu$ is ignored.
   - If the flag `Nonlinear` is set ✓ , enter:
     ⋆ $f_t\{[\text{kPa}],>0,1\}$ – tensile strength of the component material
     ⋆ $f_c\{[\text{kPa}],>0,1\}$ – compressive strength of the component material
     otherwise layers material will be assumed elastic.

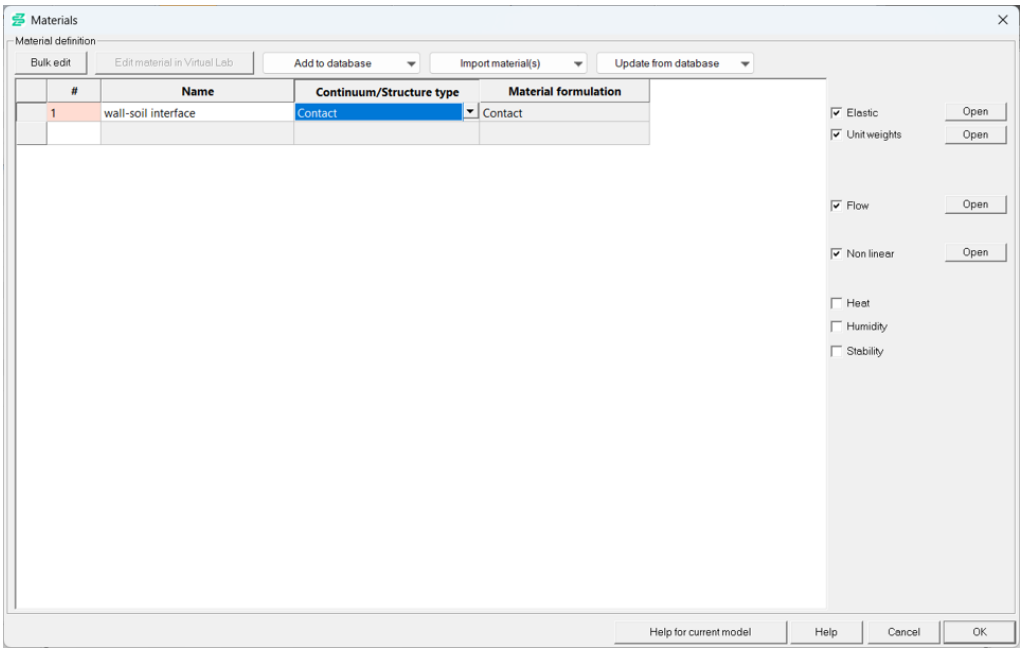
3. In contradiction to `Plane Strain` or `3D` cases, material model for core shell in the axisymmetric case is no longer uni-axial, as it has to deal with 2 normal components (longitudinal and circumferential) of stress and strain tensors. Based on the introduced $f_t, f_c$ values, a smooth Hoek-Brown stress criterion is used to evaluate perfectly plastic constitutive response of a medium. Additional (implicitly assumed) material data are eccentricity $e = 0.52$ (optimal for concrete) and dilatancy angle $\psi$ under uniaxial compression, i.e. an average between minimum and maximum values of $y$ (see Theoretical Manual). In the case when tensile yield stress is close to compressive one ( $f_t > 0.8 f_c$), Huber-Misses criterion will be used instead of Hoek-Brown.

Window 2-832

### 2.3.3.2.2 CONTACT

**Window 2-833: Selecting `Contact` model**

Contact material model can be selected from combo-box `Continuum & structure type`.




Window 2-833

**Window 2-834: Material data for solid phase**

| Data | Use & Set up |
|---|---|

| | |
|---|---|
| `Elastic` | optional<br>• `Direct values of interface stiffness` is set OFF(default). Elastic stiffness of the contact surfaces is set automatically basing on properties of surrounding media. The user may, if necessary, change the default using:<br>⋆ normal stiffness $K_n$ multiplier $\{[-],[>0],[0.01]\}$ i.e. $K_n = \texttt{multiplier} * K_n^{def}$<br>⋆ tangential/normal stiffness ratio $K_t/K_n$ $\{[-],[>0],[0.01]\}$<br>• `Direct values of interface stiffness` is set ON. User must set elastic properties of the interface explicitly specifying:<br>⋆ normal stiffness $K_n$, $\{[kN/m^3],[>0],[10000]\}$<br>$K_n$ is equivalent to $K_n = E/h$ where $h$ is the depth of the very thin weak (interface) layer and $E$ is its elasticity modulus.<br>⋆ tangential stiffness $K_t$, $\{[kN/m^3],[>0],[10000]\}$<br>$K_t$ is equivalent to $K_t = G/h$ where $h$ is the depth of the weak layer and $G$ is its shear modulus. |
| `Flow` | optional<br>User may choose between `Impermeable Surface` of the interface ($k=0$) and `Fully Permeable Surface` with permeability $k$ estimated by the program as large enough to assure pressure compatibility, or as `Permeable Surface` with permeability given by the user (see Win. 2-836).<br>Following data have to be entered only if `Fully Permeable Surface` is set OFF.<br>• permeability in the normal direction -$k'_z$ $\{[1/s],[>0],[0.01]\}$<br>• in-plane permeability -$k'_x, k'_y$.$\{m^2/s, >0, 1\}$<br>If `Anisotropic Flow` in interface is set ON, then in–plane permeabilities $k'_x, k'_y$ have to be set in 2 principal anisotropy directions. The anisotropy direction of $x\prime$ will be evaluated as the projection of `Orientation Vector` for anisotropy $\{v_x, v_y, v_z\}$. If set OFF in-plane flow properties are assumed to be isotropic characterized by permeability $k'_x$. |

| | |
|---|---|
| Non--Linear | obligatory<br>If ◎ **Direct input** is set ON data for Coulomb type frictional law describing contact interface material model are:<br>• cohesion $c$, $\{[\text{kPa}], [\geq 0], 0\}$ (valid only for small deformation contact formulation)<br>• friction angle $\phi$, $\{[\text{deg}], [0, 89], 0\}$<br>• dilatancy angle $\psi$, $\{[\text{deg}], [0, 89], 0\}$ (valid only for small deformation contact formulation)<br>If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may set up multipliers to $\tan(\phi)$, $c$ and $\tan(\psi)$ at each material for `Continuum` or `Continuum for structures`; the current contact strength parameters are inherited from the adjacent continuum elements and modified (if multiplier is different from 1.0) internally by the calculation module; to edit multipliers click on button **Setup**. User may set several tables with scaling factors but only one can be applied to the given interface material.<br>**Remark**:<br>Tensile `Cut--Off` is implicitly activated in the model |

**Remarks**:

1. Elastic normal stiffness in the contact interface is set as: $K_n^{\text{def}} = \min\left(\frac{E_1}{h_1}, \frac{E_2}{h_2}\right) \frac{A}{\sqrt{N_{\text{eq}}\varepsilon}}$
   where: $E_i$ Young moduli of adjacent continuum, $h_i$ adjacent element sizes normal to the interface, $\varepsilon$-machine precision, $A$ experimental tuning factor ($= 10^{-4}$) and $Neq$ is a number of equations in the linear system

2. Penalty elasticity constants were estimated automatically by the program as values excluding significant penetration, based on the properties of surrounding continuum. Modification of penalty factors is however allowed. In this case note that:
   - reduction of $K_n$ is sometime helpful in maintaining convergence,
   - increasing of $K_n$ will diminish penetration but may cause ill-conditioning of the system.

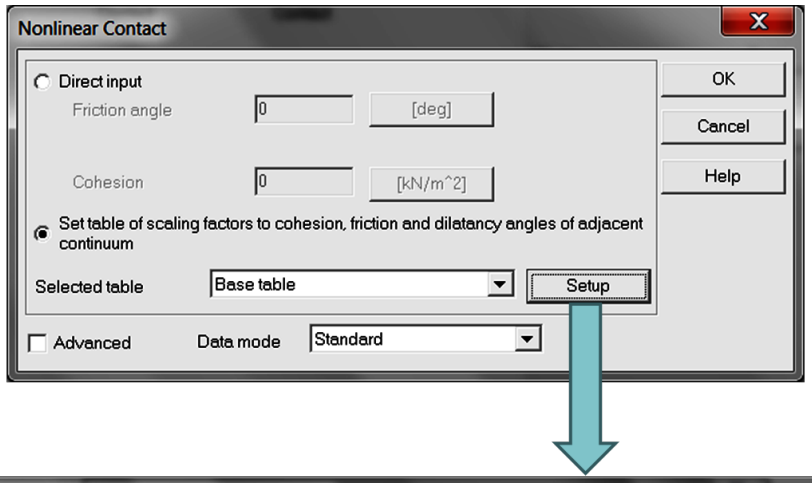
Window 2-834

## Window 2-835: Strength parameters inherited from adjacent continuum

If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may apply a single contact material for all contact elements along the interface, independently on number of different soil layers (for instance sheet-pile wall installed in the layered subsoil). By pressing the button **Setup** one may set plastic parameters in the contact interface by applying multipliers to soils' $\tan(\phi)$, $\tan(\psi)$ and $c$. It has to be emphasized that user may generate several tables with scaling factors, although only one of them can be applied to the given interface material. For all soil/rock materials strength and dilatancy parameters listed in

columns 5 ÷ 7 are common for all tables (!). By default materials, for which $\phi$, $c$ and $\psi$ are intrinsic parameters, are all set as active in aforementioned tables. Other materials for continuum are listed in tables too, however their equivalent plastic properties are all initially set to zero and activity status to the OFF state. The continuum material status active/inactive means that any interface element may setup its $\tan(\phi)$, $\tan(\psi)$ and $c$ parameters based on the corresponding parameters of the adjacent continuum element material.

Typical table is shown in the figure below.




Tables of scaling factors to cohesion, friction and dilatancy angles of adjacent continuum

Selected table: Base table — Add table | Copy table | Delete table

| Index | Material | Model | Active | Original continuum strength parameters | | | Standard Interface data | | | Piles interface data | | | | | Nails/Fixed anchor zone interface data |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Phi [deg] | c [kN/m^2] | Psi [deg] | tg(Phi) mult. | c mult. | tg(Psi) mult. | tg(Phi) mult. | c mult. | tg(Psi) mult. | Tensile bearing capacity qt [kN/m^2] | Compressive bearing capacity qc [kN/m^2] | Ultimate shear stress tau [kN/m^2] |
| 1 | No name | Elastic | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 2 | No name | HS-small strain stiffness | ☑ | 30 | 20 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 3 | No name | Concrete plastic damage | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |

**Remarks**:

1. Multipliers to be set for standard contact interface are highlighted by the yellow color
2. The current values of plastic parameters for models designed for continuum are listed in cells `Original continuum strength parameters`; if a given model is described by means of friction angle, cohesion and dilatancy angle (like M-C model, Cap, D-P or HS-small) then these original strength parameters are placed in non-editable cells (filled by a gray color); for other models (like `Elastic`) user must set the reference strength parameters (in this case cells are white and can be edited)
3. The following rules are applied to setup cohesion $c^i$, friction $\phi^i$ and dilatancy $\psi^i$ angles in the interface, based on the adjacent continuum plastic properties $c^{\text{cont.}}$, $\phi^{\text{cont.}}$, $\psi^{\text{cont.}}$

$$\begin{aligned} \tan(\phi^i) &= \tan(\phi^{\text{cont.}})\, R^i_{\tan\phi} \\ c^i &= c^{\text{cont.}}\, R^i_c \\ \tan(\psi^i) &= \tan(\psi^{\text{cont.}})\, R^i_{\tan\psi} \end{aligned}$$

where $R^i_c$, $R^i_{\tan\phi}$ and $R^i_{\tan\psi}$ are the corresponding scaling factors

4. Setting current contact strength parameters is carried out in the calculation module at each time step (as certain strength parameters for continuum models may vary in time); in all models based on the M-C strength parameters $\phi$, $c$ and dilatancy angle $\psi$ time variation of these parameters will induce time variation of the corresponding interface parameters (factors $R_c^i$, $R_{\tan\phi}^i$ and $R_{\tan\psi}^i$ are kept constant in time)

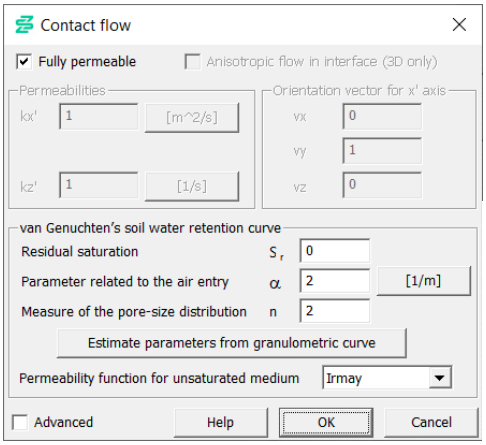
5. First seven columns are common for each table

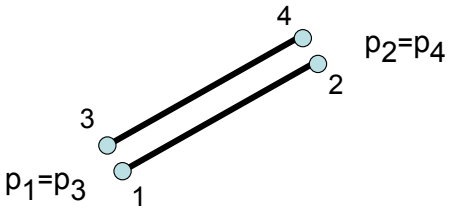
6. Having several tables with different multipliers to soil M-C strength and dilatancy parameters is useful when different pile classes are used in the computation (bored piles, CFA piles etc..)

Window 2-835

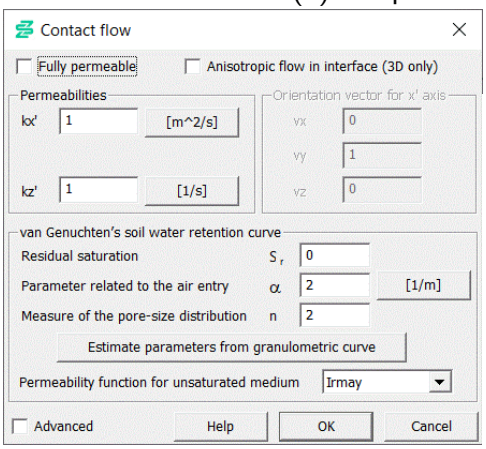
## Window 2-836: Material data for fluid phase

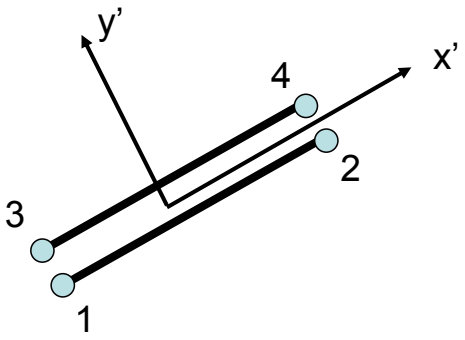
The three possible ways of handling flow through the interface are used:



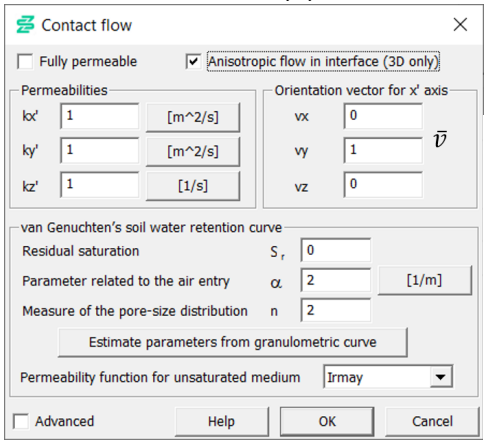



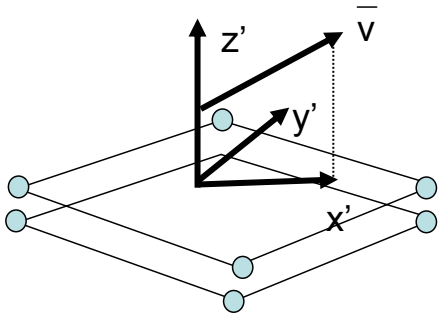

(a) full pressure compatibility




(b) transverse orthotropy (2D)




(c) anisotrophy (3D)

**Remarks**:

1. To disable flow through the interface set ☒ **Flow** check-box OFF
2. Residual saturation $S_r$ and seepage parameter $\alpha$ have the same meaning as for continuum
3. The equivalent isotropic (2D) and (3D) permeabilities can be computed as: $k_{x'} = k\ t; \quad k_{z'} = \frac{k}{t}$ where $t$ is a fictitious interface thickness while $k$ is a standard permeability coefficient

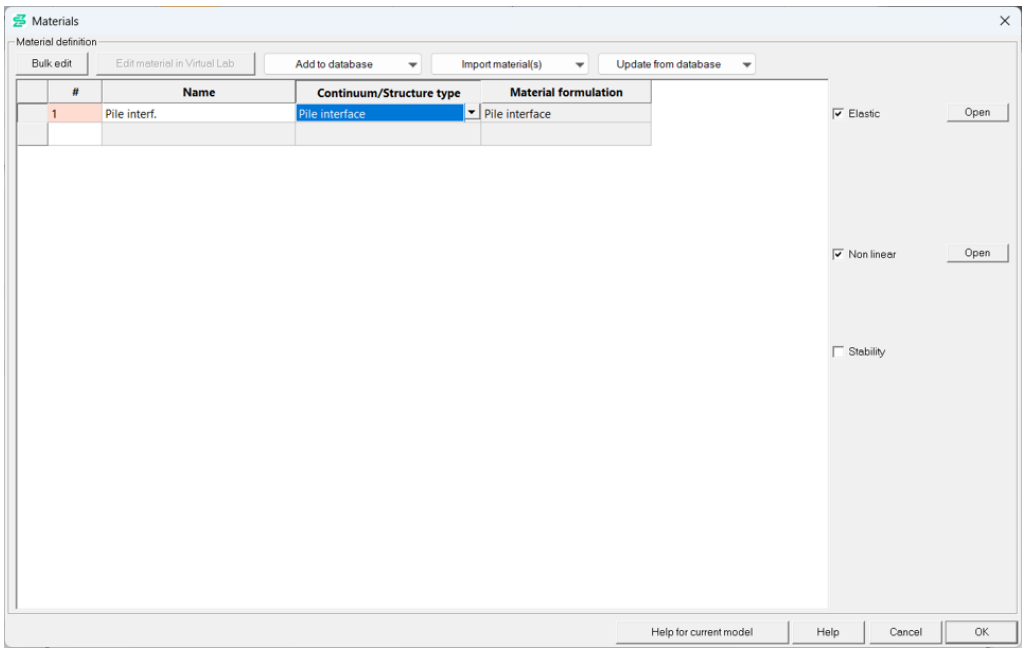
## Related Topics

- *theory: CONTACT*

## 2.3.3.2.3 PILE INTERFACE

**Window 2-837: Selecting `Pile interface` model**

Contact skin interface material dedicated for 3D piles modeled as beam elements embedded in the continuum can be selected from the combo-box `Continuum & structure type`.




Window 2-837

**Window 2-838: Material data**

| Data | Use & Set up |
|---|---|

| | |
|---|---|
| `Elastic` | optional<br>• `Direct values of interface stiffness` is set OFF(default). Elastic stiffness of the contact surfaces is set automatically basing on properties of surrounding media. The user may, if necessary, change the default using:<br>⋆ normal stiffness $K_n$ multiplier $\{[-],[>0],[0.01]\}$ i.e. $K_n = \texttt{multiplier}* K_n^{def}$<br>⋆ tangential/normal stiffness ratio $K_t/K_n$ $\{[-],[>0],[0.01]\}$<br>• `Direct values of interface stiffness` is set ON. User must set elastic properties of the interface explicitly specifying:<br>⋆ normal stiffness $K_n$, $\{[kN/m^3],[>0],[10000]\}$<br>$K_n$ is equivalent to $K_n = E/h$ where $h$ is the depth of the very thin weak (interface) layer and $E$ is its elasticity modulus.<br>⋆ tangential stiffness $K_t$, $\{[kN/m^3],[>0],[10000]\}$<br>$K_t$ is equivalent to $K_t = G/h$ where $h$ is the depth of the weak layer and $G$ is its shear modulus. |
| `Non--Linear` | obligatory<br>If ◎ **Direct input** is set ON data for Coulomb type frictional law describing contact interface material model are:<br>• cohesion $c$, $\{[\text{kPa}],[\geq 0],0\}$ (valid only for small deformation contact formulation)<br>• friction angle $\phi$, $\{[\text{deg}],[0,89],0\}$<br>• dilatancy angle $\psi$, $\{[\text{deg}],[0,89],0\}$ (valid only for small deformation contact formulation)<br>If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may set up multipliers to $\tan(\phi)$, $c$ and $\tan(\psi)$ at each material for `Continuum` or `Continuum for structures`; the current contact strength parameters are inherited from the adjacent continuum elements and modified (if multiplier is different from 1.0) internally by the calculation module; to edit multipliers click on button **Setup**. User may set several tables with scaling factors but only one can be applied to the given interface material.<br>**Remark**:<br>In this type of the interface there is no problem with the separation; the effective normal stress required by the Coulomb's law is retrieved from the continuum element in which the interface is embedded |

**Remarks**:

1. Elastic stiffness of the contact surfaces are set as:

$$K_n^{\text{def}} = \min\left(\frac{E_1}{h_1}, \frac{E_2}{h_2}\right) \frac{A}{\sqrt{N_{\text{eq}}\varepsilon}}$$

   where: $E_i$ Young moduli of adjacent continuum, $h_i$ adjacent element sizes normal to the interface, $\varepsilon$-machine precision, $A$ experimental tuning factor ($= 10^{-4}$) and $Neq$ is a number of equations in the linear system

2. Penalty elasticity constants are estimated automatically by the program. However, user may modify this setting by means of multipliers or direct values of penalty factors. It is worth noting that:

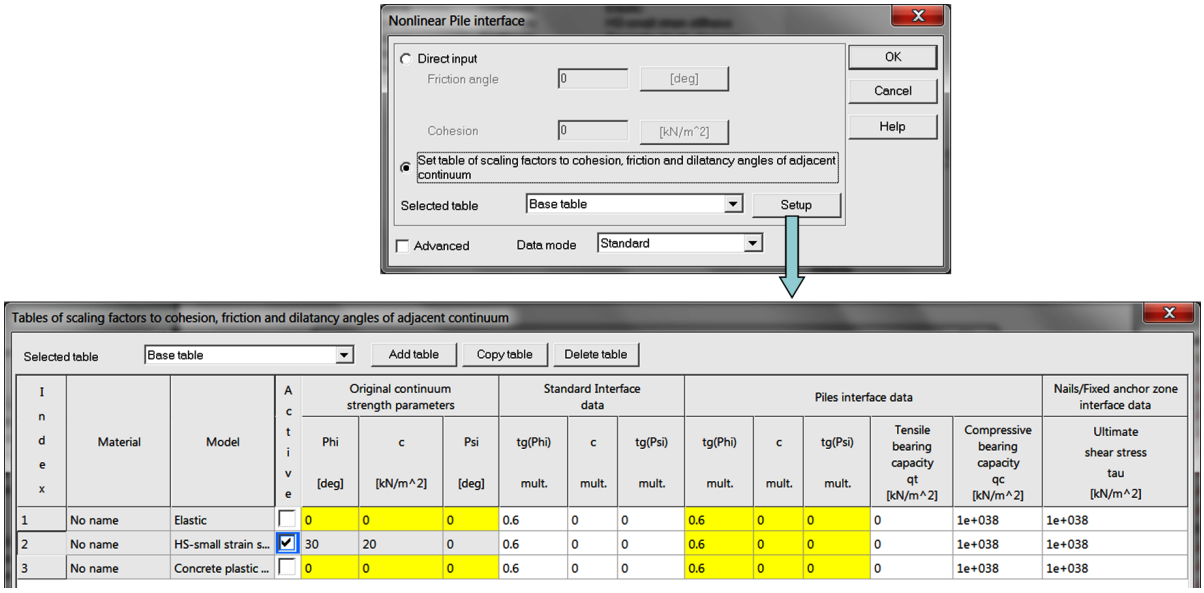
   - Reduction of $K_n$ (and $K_s$ as well), is sometime helpful in maintaining convergence
   - Surface of the pile interface, needed for the integration of the tangential force, is inherited from the adjacent beam elements (pile)

Window 2-838

## Window 2-839: Strength parameters inherited from adjacent continuum

If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may apply a single contact material for all contact elements along the interface, independently on number of different soil layers (for instance pile installed in the layered subsoil). By pressing the button **Setup** one may set plastic parameters in the contact interface by applying multipliers to soils' $\tan(\phi)$, $\tan(\psi)$ and $c$. It has to be emphasized that user may generate several tables with scaling factors, although only one of them can be applied to the given interface material. For all soil/rock materials strength and dilatancy parameters listed in columns $5 \div 7$ are common for all tables (!). By default materials, for which $\phi$, $c$ and $\psi$ are intrinsic parameters, are all set as active in aforementioned tables. Other materials for continuum are listed in tables too, however their equivalent plastic properties are all initially set to zero and activity status to the OFF state. The continuum material status active/inactive means that any interface element may setup its $\tan(\phi)$, $\tan(\psi)$ and $c$ parameters based on the corresponding parameters of continuum element material in which this interface is embedded.

Typical table is shown in the figure below.




| Index | Material | Model | Active | Original continuum strength parameters | | | Standard Interface data | | | Piles interface data | | | | | Nails/Fixed anchor zone interface data |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Phi [deg] | c [kN/m^2] | Psi [deg] | tg(Phi) mult. | c mult. | tg(Psi) mult. | tg(Phi) mult. | c mult. | tg(Psi) mult. | Tensile bearing capacity qt [kN/m^2] | Compressive bearing capacity qc [kN/m^2] | Ultimate shear stress tau [kN/m^2] |
| 1 | No name | Elastic | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 2 | No name | HS-small strain s... | ☑ | 30 | 20 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 3 | No name | Concrete plastic ... | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |

**Remarks**:

1. Multipliers to be set for pile contact interface are highlighted by yellow color
2. The current values of plastic parameters for models designed for continuum are listed in cells `Original continuum strength parameters`; if a given model is described by means of friction angle, cohesion and dilatancy angle (like M-C model, Cap, D-P or HS-small) then these original strength parameters are placed in non-editable cells (filled by a gray color); for other models (like `Elastic`) user must set the reference strength parameters (in this case cells are white and can be edited)
3. The following rules are applied to setup cohesion $c^i$, friction $\phi^i$ and dilatancy $\psi^i$ angles in the skin interface, based on the adjacent continuum plastic properties $c^{\text{cont.}}$, $\phi^{\text{cont.}}$, $\psi^{\text{cont.}}$

$$\begin{aligned} \tan(\phi^i) &= \tan(\phi^{\text{cont.}})\, R^i_{\tan\phi} \\ c^i &= c^{\text{cont.}}\, R^i_c \\ \tan(\psi^i) &= \tan(\psi^{\text{cont.}})\, R^i_{\tan\psi} \end{aligned}$$

where $R_c^i$, $R_{\tan\phi}^i$ and $R_{\tan\psi}^i$ are the corresponding scaling factors

4. Setting current contact strength parameters is carried out in the calculation module at each time step (as certain strength parameters for continuum models may vary in time); in all models based on the M-C strength parameters $\phi$, $c$ and dilatancy angle $\psi$ time variation of these parameters will induce time variation of the corresponding skin interface parameters (factors $R_c^i$, $R_{\tan\phi}^i$ and $R_{\tan\psi}^i$ are kept constant in time)
5. First seven columns are common for each table
6. Having several tables with different multipliers to soil M-C strength and dilatancy parameters is useful when different pile types are used in the model (bored piles, CFA piles etc..)

Window 2-839

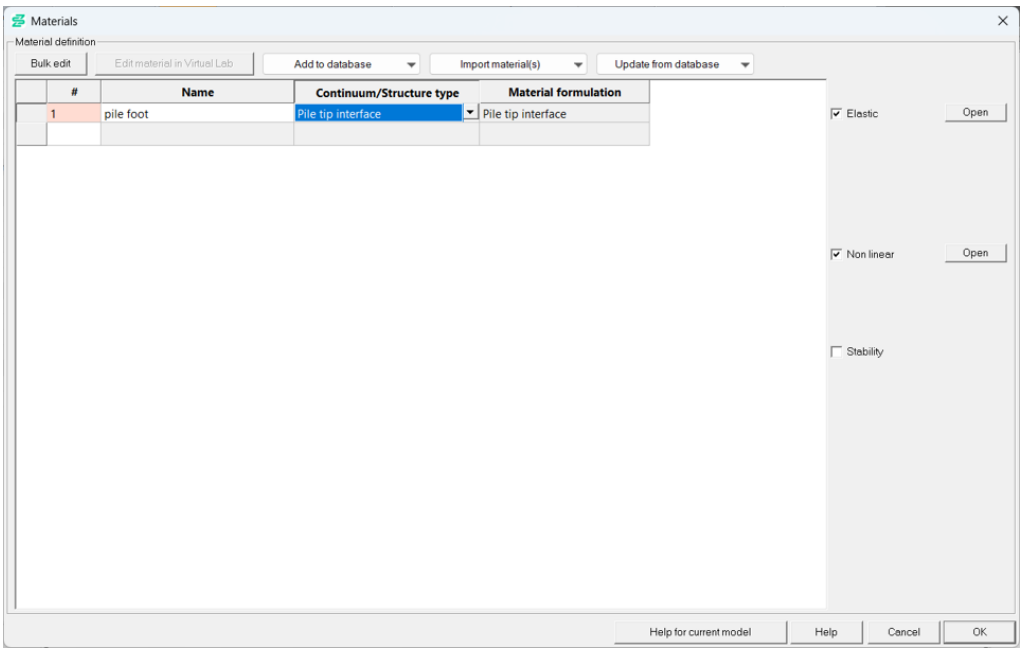
## Related Topics

- *theory: PILE INTERFACE*

### 2.3.3.2.4 PILE TIP INTERFACE

**Window 2-840: Selecting `Pile tip interface` model**

Contact material dedicated for 3D piles to model the interface between tip of the 3D pile and subsoil can be selected from the combo-box `Continuum & structure type`.




Window 2-840

**Window 2-841: Material data**

| Data | Use & Set up |
|---|---|
| `Elastic` | optional<br>• `Direct values of interface stiffness` is set OFF(default). Elastic stiffness of the contact surfaces is set automatically basing on properties of surrounding media. The user may, if necessary, change the default using:<br>⋆ normal stiffness $K_n$ multiplier $\{[-],[>0],[0.01]\}$ i.e. $K_n = \texttt{multiplier} * K_n^{def}$<br>• `Direct values of interface stiffness` is set ON. User must set elastic properties of the interface explicitly specifying:<br>⋆ normal stiffness $K_n$, $\{[kN/m^3],[>0],[10000]\}$ |

| | |
|---|---|
| `Non--Linear` | obligatory<br>If ◎ **Direct input** is set ON data for stick-separation contact material model are:<br>• Tensile bearing capacity $q_t$, $\{[\text{kPa}], [\geq 0], 0\}$<br>• Compressive bearing capacity $q_c$, $\{[\text{kPa}], [\geq 0], 10^{38}\}$<br>If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may set up compressive/tensile bearing capacities at each material for `Continuum` or `Continuum for structures`.<br>User may set several tables with compressive/tensile bearing capacities but only one can be applied to the given interface material.<br>**Remark**:Compressive/tensile bearing capacity is expressed in terms of total stresses |

**Remarks**:

1. Elastic stiffness of the contact surfaces are set as:

$$K_n^{\text{def}} = \min\left(\frac{E_1}{h_1}, \frac{E_2}{h_2}\right) \frac{A}{\sqrt{N_{\text{eq}}\varepsilon}}$$

   where: $E_i$ Young moduli of adjacent continuum, $h_i$ adjacent element sizes normal to the interface, $\varepsilon$-machine precision, $A$ experimental tuning factor ($= 10^{-4}$) and $Neq$ is a number of equations in the linear system

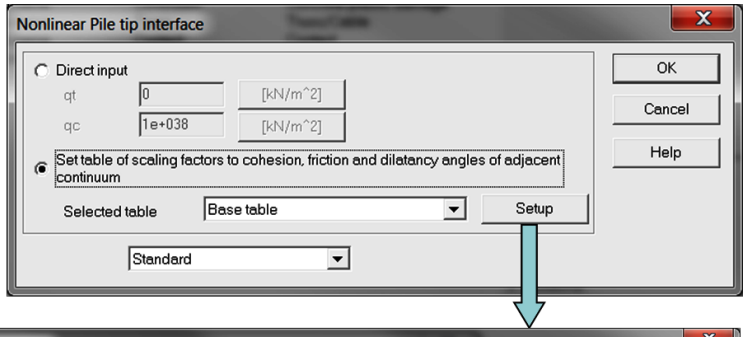
2. Penalty elasticity constant is estimated automatically by the program. However, user may modify this setting by means of multiplier or direct value of penalty factor. It is worth noting that:

   - Reduction of $K_n$ is sometime helpful in maintaining convergence
   - Surface of the pile tip interface, needed for the integration of the normal force, is inherited from the adjacent beam element (pile)

## Window 2-842: Strength parameters inherited from adjacent continuum

If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may apply a single contact material for all pile tip contact elements, independently on number of different soil layers (for instance piles installed in the layered subsoil). By pressing the button **Setup** one may set compressive/tensile bearing capacities for each continuum material. It has to be emphasized that user may generate several tables with these bearing capacities, although only one of them can be applied to the given interface material.

Typical table is shown in the figure below.




Tables of scaling factors to cohesion, friction and dilatancy angles of adjacent continuum

Selected table: Base table | Add table | Copy table | Delete table

| Index | Material | Model | Active | Original continuum strength parameters: Phi [deg] | c [kN/m^2] | Psi [deg] | Standard Interface data: tg(Phi) mult. | c mult. | tg(Psi) mult. | Piles interface data: tg(Phi) mult. | c mult. | tg(Psi) mult. | Tensile bearing capacity qt [kN/m^2] | Compressive bearing capacity qc [kN/m^2] | Nails/Fixed anchor zone interface data: Ultimate shear stress tau [kN/m^2] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | No name | Elastic | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 2 | No name | HS-small strain s... | ☑ | 30 | 20 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 600 | 1e+038 |
| 3 | No name | Concrete plastic ... | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |

**Remarks**:

1. $q_t$ and $q_c$ values to be set for the pile tip interface are highlighted by yellow color; these values are usually available in codes for standard piles design

Window 2-842

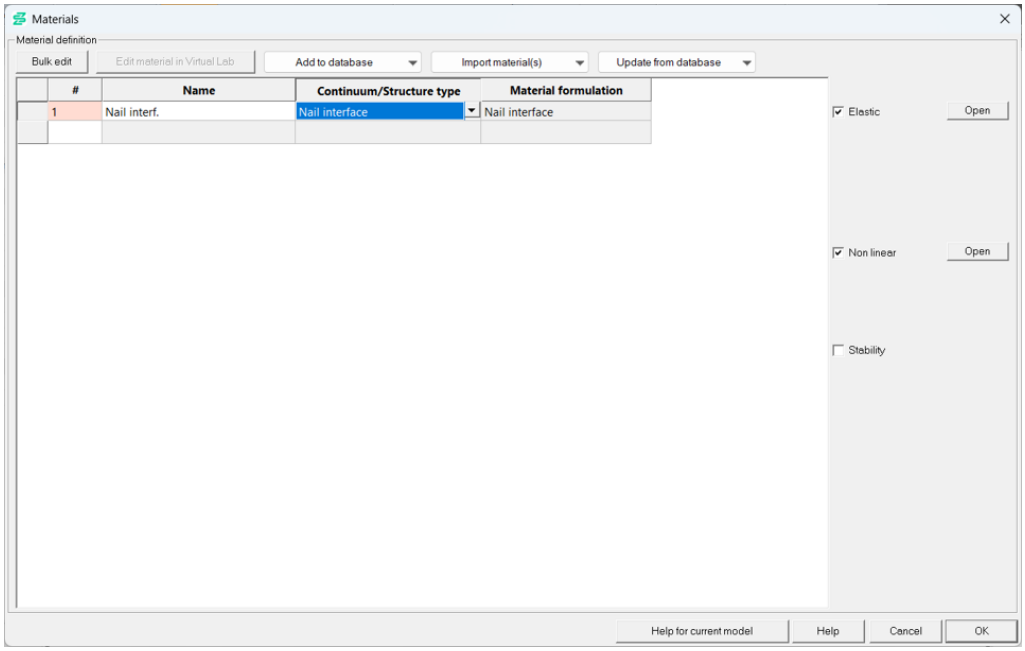
## Related Topics

- *theory: PILE TIP INTERFACE*

## 2.3.3.2.5 NAIL INTERFACE

**Window 2-843: Selecting `Nail interface` model**

Contact interface material dedicated for 2D/3D nails modeled as beam elements embedded in the continuum can be selected from the combo-box `Continuum & structure type`.




Window 2-843

**Window 2-844: Material data**

| Data | Use & Set up |
|---|---|

| | |
|---|---|
| `Elastic` | optional<br>• `Direct values of interface stiffness` is set OFF(default). Elastic stiffness of the contact surfaces is set automatically basing on properties of surrounding media. The user may, if necessary, change the default using:<br>⋆ normal stiffness $K_n$ multiplier $\{[-],[>0],[0.01]\}$ i.e. $K_n = \texttt{multiplier}* K_n^{def}$<br>⋆ tangential/normal stiffness ratio $K_t/K_n$ $\{[-],[>0],[0.01]\}$<br>• `Direct values of interface stiffness` is set ON. User must set elastic properties of the interface explicitly specifying:<br>⋆ normal stiffness $K_n$, $\{[kN/m^3],[>0],[10000]\}$<br>$K_n$ is equivalent to $K_n = E/h$ where $h$ is the depth of the very thin weak (interface) layer and $E$ is its elasticity modulus.<br>⋆ tangential stiffness $K_t$, $\{[kN/m^3],[>0],[10000]\}$<br>$K_t$ is equivalent to $K_t = G/h$ where $h$ is the depth of the weak layer and $G$ is its shear modulus. |
| `Non--Linear` | obligatory<br>If ◎ **Direct input** is set ON data for adhesive law describing contact interface material model are:<br>• contact zone diameter (equivalent to the diamater of the injection zone) $\{[m],[\geq 0],0.1\}$ (valid only for small deformation contact formulation)<br>• ultimate shear stress $\tau$, $\{[\text{kPa}],[\geq 0],0\}$ (valid only for small deformation contact formulation)<br>If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may set up ultimate shear stresses for each material for `Continuum` or `Continuum for structures`. User may set several tables with ultimate shear stresses but only one can be applied to the given interface material. |

**Remarks**:

1. Elastic stiffness of the contact surfaces are set as:

$$K_n^{\text{def}} = \min\left(\frac{E_1}{h_1}, \frac{E_2}{h_2}\right) \frac{A}{\sqrt{N_{\text{eq}}\varepsilon}}$$

where: $E_i$ Young moduli of adjacent continuum, $h_i$ adjacent element sizes normal to the interface, $\varepsilon$-machine precision, $A$ experimental tuning factor ($= 10^{-4}$) and $Neq$ is a number of equations in the linear system

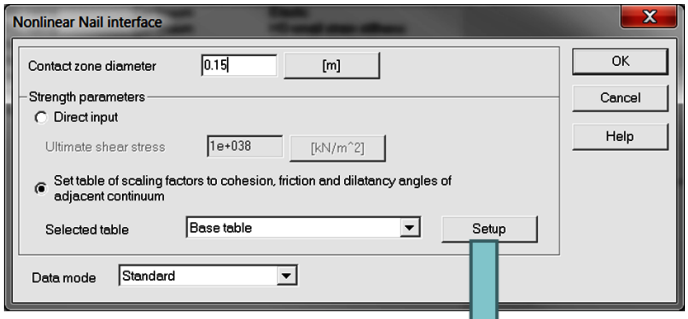
2. Penalty elasticity constants are estimated automatically by the program. However, user may modify this setting by means of multipliers or direct values of penalty factors. It is worth noting that:

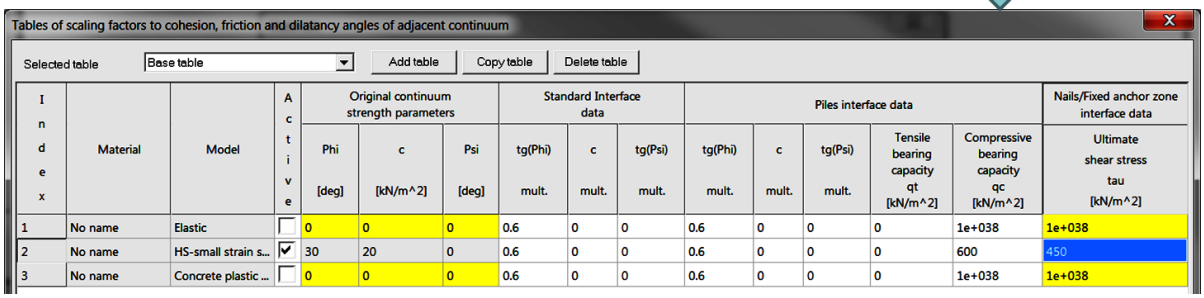
   - Reduction of $K_n$ (and $K_s$ as well), is sometime helpful in maintaining convergence

## Window 2-845: Strength parameters inherited from adjacent continuum

If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may apply a single contact material for all contact elements along the nail interface, independently on number of different soil layers (for instance nail installed in the layered subsoil). By pressing the button **Setup** one may set ultimate shear stress values for each active continuum material in which nail can be embedded.

Typical table is shown in the figure below.




| Index | Material | Model | Active | Original continuum strength parameters | | | Standard Interface data | | | Piles interface data | | | | | Nails/Fixed anchor zone interface data |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Phi [deg] | c [kN/m^2] | Psi [deg] | tg(Phi) mult. | c mult. | tg(Psi) mult. | tg(Phi) mult. | c mult. | tg(Psi) mult. | Tensile bearing capacity qt [kN/m^2] | Compressive bearing capacity qc [kN/m^2] | Ultimate shear stress tau [kN/m^2] |
| 1 | No name | Elastic | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 2 | No name | HS-small strain s... | ☑ | 30 | 20 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 600 | 450 |
| 3 | No name | Concrete plastic ... | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |

**Remarks**:

1. Values to be set for nail interface are highlighted by yellow color
2. The current values of plastic parameters for models designed for continuum are listed in cells `Original continuum strength parameters`; if a given model is described by means of friction angle, cohesion and dilatancy angle (like M-C model, Cap, D-P or HS-small) then these original strength parameters are placed in non-editable cells (filled by a gray color); for other models (like `Elastic`) user must set the reference strength parameters (in this case cells are white and can be edited) and set status `Active` ON
3. One can define several tables with ultimate shear stress values in each continuum material; however only one of them can be used for a given nail interface material

Window 2-845

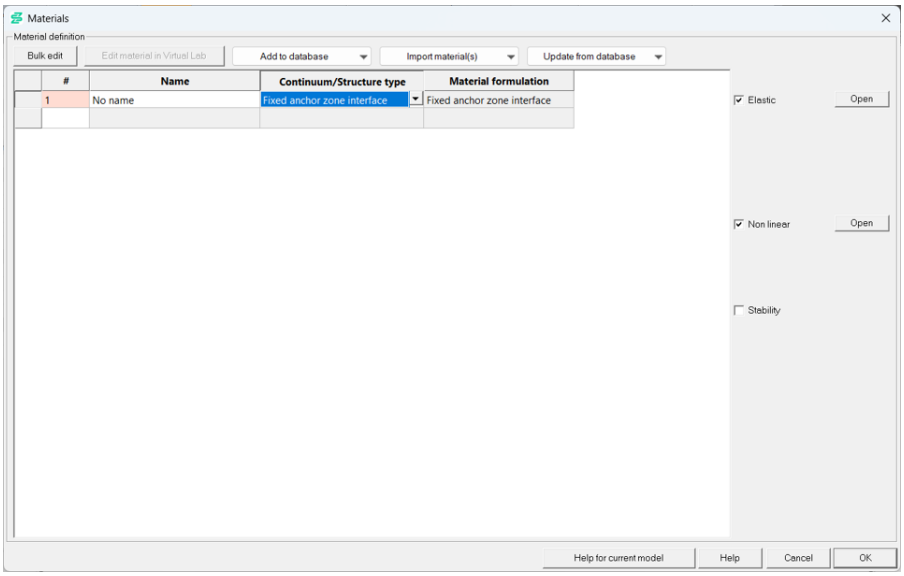
## Related Topics

- *theory: NAIL INTERFACE*

## 2.3.3.2.6 FIXED ANCHOR ZONE INTERFACE

**Window 2-846: Selecting `Fixed anchor zone interface` model**

Contact interface material dedicated for 2D/3D fixed anchor zones, modeled as truss elements embedded in the continuum, can be selected from the combo-box `Continuum & structure type`.




Window 2-846

**Window 2-847: Material data**

| Data | Use & Set up |
|---|---|

| | |
|---|---|
| `Elastic` | optional<br>• `Direct values of interface stiffness` is set OFF(default). Elastic stiffness of the contact surfaces is set automatically basing on properties of surrounding media. The user may, if necessary, change the default using:<br>⋆ normal stiffness $K_n$ multiplier $\{[-],[>0],[0.01]\}$ i.e. $K_n = \texttt{multiplier}* K_n^{def}$<br>⋆ tangential/normal stiffness ratio $K_t/K_n$ $\{[-],[>0],[0.01]\}$<br>• `Direct values of interface stiffness` is set ON. User must set elastic properties of the interface explicitly specifying:<br>⋆ normal stiffness $K_n$, $\{[kN/m^3],[>0],[10000]\}$ $K_n$ is equivalent to $K_n = E/h$ where $h$ is the depth of the very thin weak (interface) layer and $E$ is its elasticity modulus.<br>⋆ tangential stiffness $K_t$, $\{[kN/m^3],[>0],[10000]\}$ $K_t$ is equivalent to $K_t = G/h$ where $h$ is the depth of the weak layer and $G$ is its shear modulus. |
| `Non--Linear` | obligatory<br>If ◎ **Direct input** is set ON data for adhesive law describing contact interface material model are:<br>• contact zone diameter (equivalent to the diamater of the injection zone) $\{[m],[\geq 0],0.1\}$ (valid only for small deformation contact formulation)<br>• ultimate shear stress $\tau$, $\{[\text{kPa}],[\geq 0],0\}$ (valid only for small deformation contact formulation)<br>If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may set up ultimate shear stresses for each material for `Continuum` or `Continuum for structures`. User may set several tables with ultimate shear stresses but only one can be applied to the given interface material. |

**Remarks**:

1. Elastic stiffness of the contact surfaces are set as:

$$K_n^{\text{def}} = \min\left(\frac{E_1}{h_1}, \frac{E_2}{h_2}\right) \frac{A}{\sqrt{N_{\text{eq}}\varepsilon}}$$

where: $E_i$ Young moduli of adjacent continuum, $h_i$ adjacent element sizes normal to the interface, $\varepsilon$-machine precision, $A$ experimental tuning factor ($= 10^{-4}$) and $Neq$ is a number of equations in the linear system

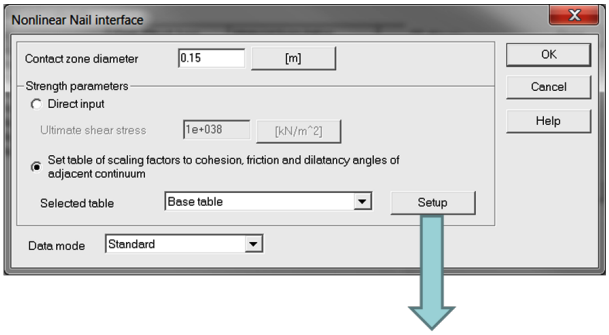
2. Penalty elasticity constants are estimated automatically by the program. However, user may modify this setting by means of multipliers or direct values of penalty factors. It is worth noting that:

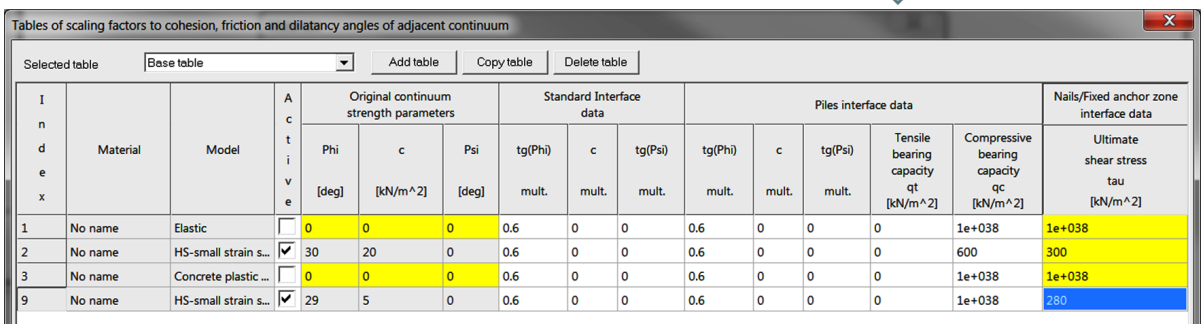
   - Reduction of $K_n$ (and $K_s$ as well), is sometime helpful in maintaining convergence

## Window 2-848: Strength parameters inherited from adjacent continuum

If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may apply a single contact material for all contact elements along the fixed anchor zone interface, independently on number of different soil layers (for instance nail installed in the layered subsoil). By pressing the button **Setup** one may set ultimate shear stress values for each active continuum material in which fixed anchor zone can be embedded.

Typical table is shown in the figure below.




| Index | Material | Model | Active | Original continuum strength parameters | | | Standard Interface data | | | Piles interface data | | | | | Nails/Fixed anchor zone interface data |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | | Phi [deg] | c [kN/m^2] | Psi [deg] | tg(Phi) mult. | c mult. | tg(Psi) mult. | tg(Phi) mult. | c mult. | tg(Psi) mult. | Tensile bearing capacity qt [kN/m^2] | Compressive bearing capacity qc [kN/m^2] | Ultimate shear stress tau [kN/m^2] |
| 1 | No name | Elastic | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 2 | No name | HS-small strain s... | ☑ | 30 | 20 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 600 | 300 |
| 3 | No name | Concrete plastic ... | ☐ | 0 | 0 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 1e+038 |
| 9 | No name | HS-small strain s... | ☑ | 29 | 5 | 0 | 0.6 | 0 | 0 | 0.6 | 0 | 0 | 0 | 1e+038 | 280 |

**Remarks**:

1. Values to be set for anchor interface are highlighted by yellow color
2. The current values of plastic parameters for models designed for continuum are listed in cells `Original continuum strength parameters`; if a given model is described by means of friction angle, cohesion and dilatancy angle (like M-C model, Cap, D-P or HS-small) then these original strength parameters are placed in non-editable cells (filled by a gray color); for other models (like `Elastic`) user must set the reference strength parameters (in this case cells are white and can be edited) and set status `Active` ON
3. One can define several tables with ultimate shear stress values in each continuum material; however only one of them can be used for a given fixed anchor zone interface material

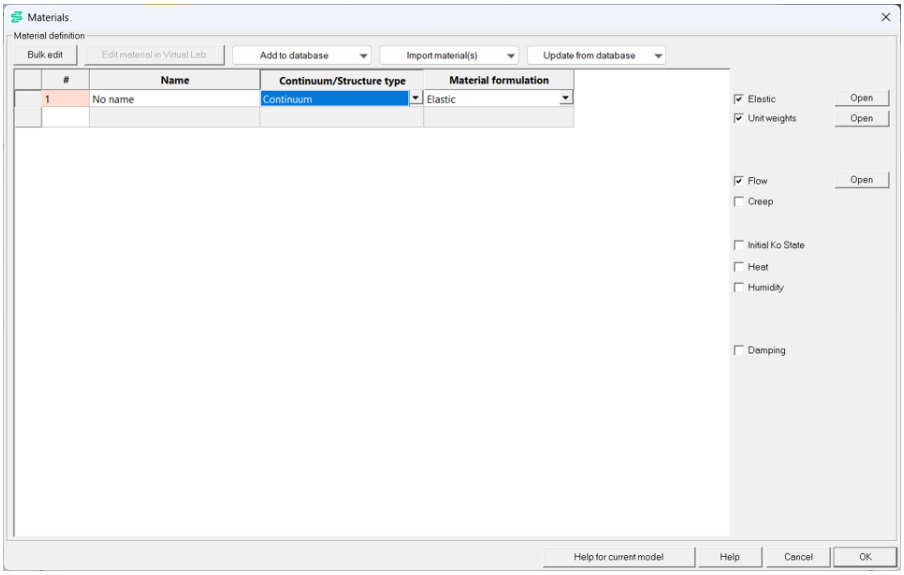
Window 2-848

## Related Topics

- *theory: ANCHOR INTERFACE*

## 2.3.3.2.7 CONTINUUM

**Window 2-849: Models for** `Continuum`




| | Material models for continuum | Data groups |
|---|---|---|
| linear: | `Elastic` | **ELASTIC** |
| nonlinear: | `Aging Concrete` | **UNIT WEIGHTS** |
| | `Cap` | **CREEP** |
| | `Cam-Clay` | **FLOW** |
| | `Drucker--Prager` | NONLINEAR |
| | `Hoek--Brown (M-W)` | **HEAT** |
| | `Mohr--Coulomb` | **HUMID** |
| | `Mohr--Coulomb (M-W)` | **STABILITY** |
| | `Multilaminate` | **DAMPING** |
| | `Rankine(M-W)` | |
| | `HS-small` | |
| | `Densification` | |
| | `Hoek-Brown` | |
| | `Plastic damage for concrete` | |

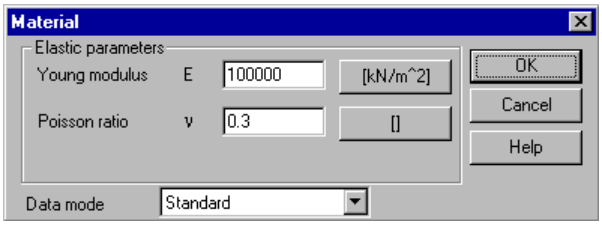
**Remark**:All listed material models for continuum (their names correspond to the constitutive model applied to the solid phase) can be used for `Flow`, `Heat`, `Humidity` analyzes. In these cases material parameters defined in ☒ **Flow**, ☒ **Heat** or ☒ **Humidity** groups are meaningful.

Window 2-849

## 2.3.3.2.7.1 ELASTIC

**Window 2-850: Elastic properties**

Parameters stored in group ☒ **Elastic** characterize elastic behavior for solid phase. In most of the constitutive models with the linear elastic kernel it consists of the two parameters: Young's modulus $E$ and Poisson's coefficient $\nu$




| Symbol | Definition | Remarks |
|---|---|---|
| $E$ | Young's modulus(★),$\{[\text{kPa}]; [> 0]; 100000\}$, | 1 |
| $\nu$ | Poisson's ratio (★), $[-]; [0, 0.49999]; 0.3$ | 1 |

**Remarks**:

1. Parameters marked with (★) can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)

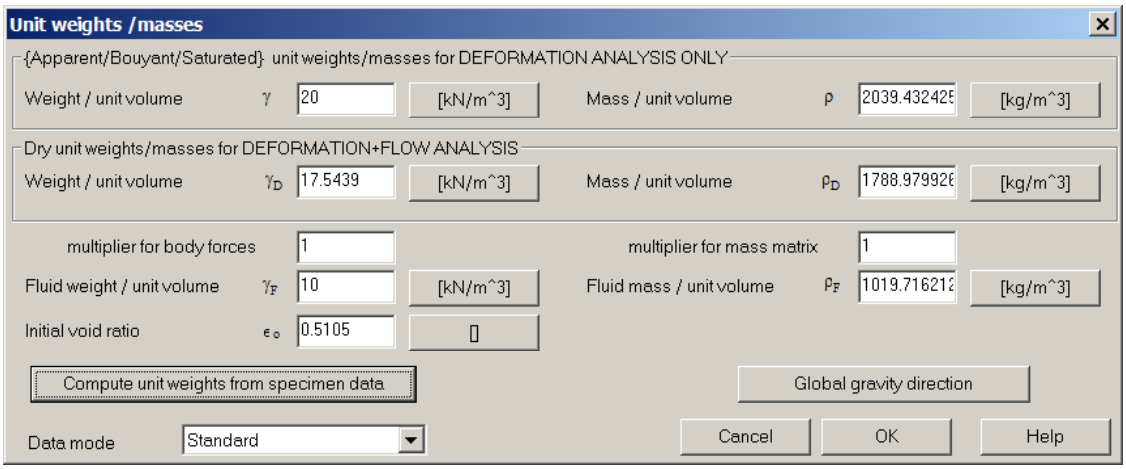
Window 2-850

## Related Topics

- *theory: ELASTICITY*
- *NON-STANDARD MATERIAL DATA*

## 2.3.3.2.7.2 UNIT WEIGHT

**Window 2-851: Unit weights**




| Symbol | Definition | Remarks |
|---|---|---|
| $\gamma$ | specific weight for the dry solid (★), $\{[\text{kNm}^{-3}];[>0];0\}$ | 1, 2, 4, 7, 6 8 |
| $\gamma_F$ | specific weight for the fluid (★), $\{[\text{kNm}^{-3}];[>0];10\}$ | 5, 8 |
| $e_0$ | initial void ratio (★), $\{[-];[0,1];0\}$ | |
| $\gamma$ mult. | multiplier for $\gamma$, $\{[-];[0,1];1.0\}$ | 9 |
| $\rho$ mult. | multiplier for $\rho$, $\{[-];[0,1];1.0\}$ | 10 |

**Remarks**:

1. For `Deformation` (single-phase) total stress analysis specify $\gamma$ as :
   - Unit weight (saturated) $\gamma_{SAT}$ in zone below the water table
   - Apparent weight (with natural moisture content) $\gamma$ or dry $\gamma_D$ in zone above the water table
2. For `Deformation` (single-phase) effective stress analysis specify $\gamma$ as :
   - Unit weight (buoyant) $\gamma_B = \gamma_{SAT} - \gamma_F$ in zone below the water table
   - Apparent weight (with natural moisture content) $\gamma$ or dry $\gamma_D$ in zone above the water table
3. To compute unit weights from specimen data use an application that is invoked by pressing button **Compute unit weights...** (see Win.(2-852))
4. For `Deformation+ Flow` (two-phase) total stress analysis specify $\gamma$ as:
   - Unit weight (dry) $\gamma_D$

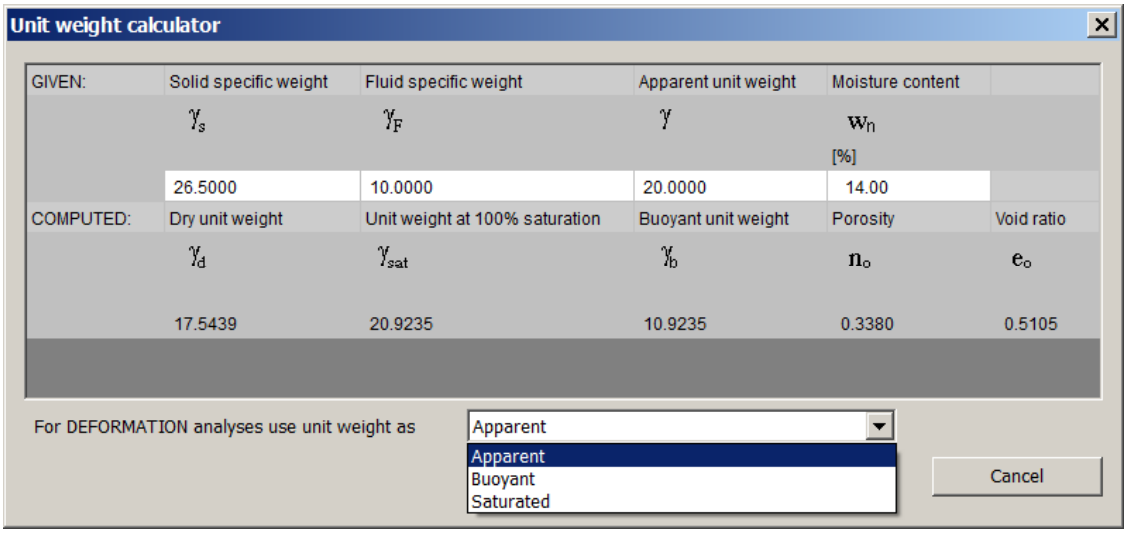
   Current unit weight $\gamma$ is computed as $\gamma = \gamma_D + n \cdot S \cdot \gamma_F$ ( $n = e_0/(1+e_0)$ (initial porosity) and current saturation ratio is denoted by $S$)
5. For `Flow` analysis (fluid-phase exclusively) specify:
   - fluid specific weight $\gamma_F$ (obligatory)
   - initial void ratio $e_0$ (obligatory for transient case only, if left equal to zero will cancel all transient effects)
6. In case of materials for structural members specify $\gamma$ as a standard unit weight (in case of layered sections use an averaged value over the section).
7. Instead of unit weight for solid/fluid one can set density (unit weight will automatically be computed); to cancel body loads, while $\gamma$ is nonzero, press **Global gravity vector**, choose option ☒ **Advanced**, select option `Body load components` and apply zero valued load time functions to the gravity directions

8. Parameters marked with (★) can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)
9. Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor)(this setting is active if `Dynamics` or `Pushover` are active in the project preselection)
10. Setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix in dynamic simulations (this setting is active if `Dynamics` or `Pushover` are active in the project preselection)

Window 2-851

## Window 2-852: Computing unit weights from specimen data

Unit weights for both `Deformation` and `Deformation+Flow` analyses can be set from the given specimen data defined by solid specific weight $\gamma_s$ ($\gamma_s = 26.5 \div 26.7$ kN/m$^3$), fluid specific weight $\gamma_F$ ($\gamma_F = 10$ kN/m$^3$), apparent unit weight $\gamma$ ($\gamma < \gamma_s$) and natural moisture content $w_n$. Using this data the skeleton dry unit weight $\gamma_D$, unit weight at 100 % of saturation, buoyant unit weight $\gamma_b$, void ratio $e_o$ and porosity $n_o$ are computed. To update unit weight to be used in `Deformation` analyses user must decide which unit weight is to be taken as $\gamma$ the apparent, buoyant or saturated one.

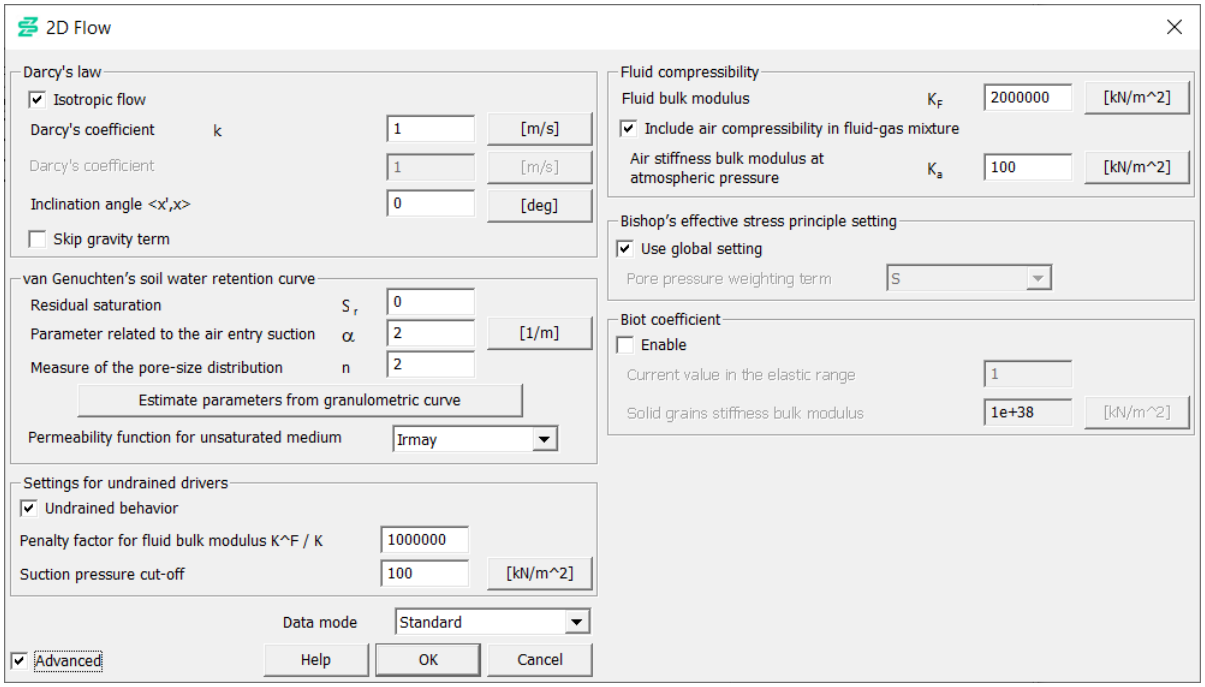



Window 2-852

## 2.3.3.2.7.3 FLOW

This group of parameters is meaningful only for `Flow` or `Deformation + Flow` problems. If group ☒ **Flow** is set OFF then the considered material is treated as impermeable [2].

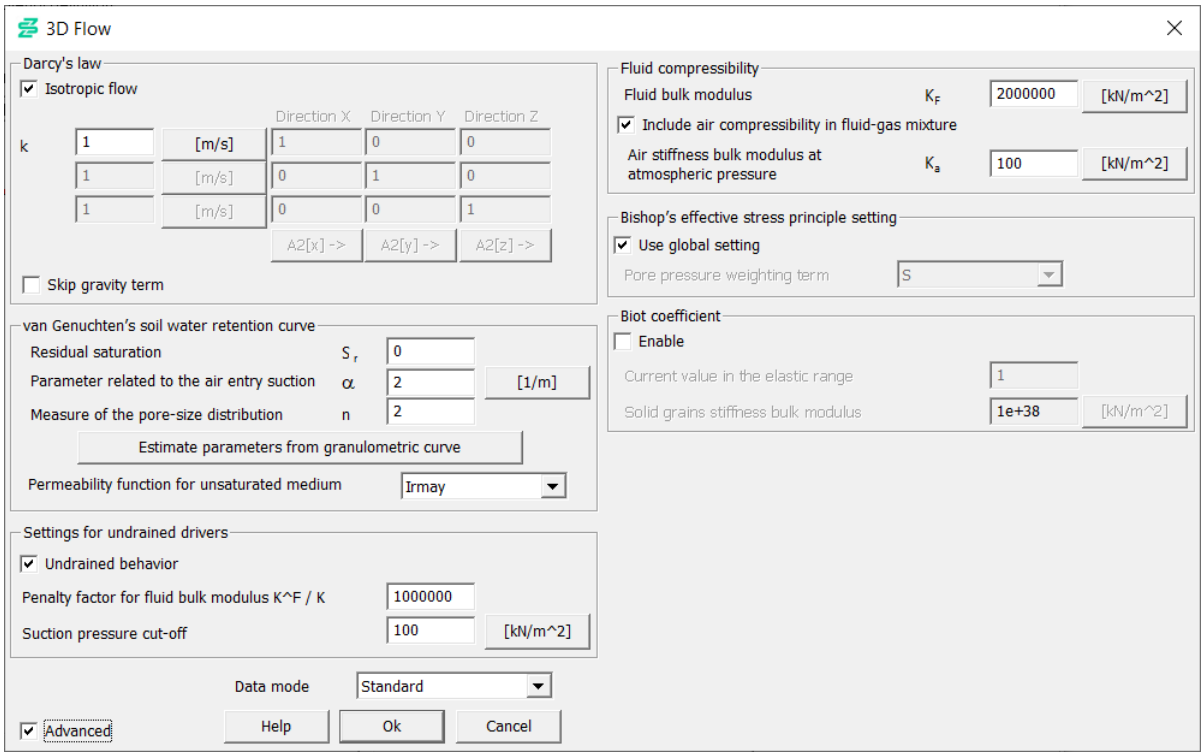
**Window 2-853: Flow properties (2D case)**




| Symbol | Definition | Remarks |
|---|---|---|
| $k_x, k_y$(★) | Permeability coefficients $\{[\text{ms}^{-1}]; [> 0]; 1.0\}$ | 1,4,5 |
| $\beta$ | Inclination angle measured from $x$ axis to $x\prime$ permeability direction $\{[\text{deg}]; [0, 90]; 0\}$ | 5 |
| $S_r$ | Residual saturation $\{[-]; [0, 1]; 0\}$ | |
| $\alpha$ | Parameter related to the air entry suction $\{[\text{m}^{-1}]; [> 0]; 2.0\}$ | 7 |
| $n$ | Measure of the pore-size distribution $\{[-]; [> 1]; 2\}$ | |
| ☒ **Skip ..** | Skip gravity term in Darcy's law {OFF} | 8 |
| ☒ **Undr...** | Enable undrained behavior {ON} | 9 |
| $\frac{K^F}{K}$ | Penalty factor $\{[-]; [10^5 \div 10^7]; 10^6\}$ | 9 |
| $p_{cut-off}$ | Suction cut-off value $\{[kN/m^3]; [\geq 0]; 100\}$ | 9 |
| $K_F$ | Fluid bulk modulus $\{[\text{kPa}]; [> 0]; 1.0e + 38\}$ | 3 |
| ☒ **Inclu ...** | Include air compressibility in fluid-gas mixture {ON} | |
| $K_a$ | Air stiffness bulk modulus at atmospheric pressure $\{[\text{kPa}]; [> 0]; 100.0\}$ | |
| ☒ **Enab...** | Enable Biot coefficient {OFF} | |
| $\alpha$ | Biot coefficient $\{[-]; (e_0, 1]; 1.0\}$ | |

Window 2-853

## Window 2-854: Flow properties (3D case)




| Symbol | Definition | Remarks |
|---|---|---|
| $k_1, k_2, k_3(\bigstar)$ | Permeability coefficients $\{[\text{ms}^{-1}]; [> 0]; 1.0\}$ | 1,4,5 |
| $Direction$ | Base vectors, co-linear with permeability directions $\{x', y'\}$ | 5,6 |
| $S_r$ | Residual saturation $\{[-]; [0, 1]; 0\}$ | |
| $\alpha$ | Parameter related to the air entry suction $\{[\text{m}^{-1}]; [> 0]; 2.0\}$ | 7 |
| $n$ | Measure of the pore-size distribution $\{[-]; [> 1]; 2\}$ | |
| ☒ **Skip ..** | Skip gravity term in Darcy's law {OFF} | 8 |
| ☒ **Undr...** | Enable undrained behavior {ON} | 9 |
| $\frac{K^F}{K}$ | Penalty factor $\{[-]; [10^5 \div 10^7]; 10^6\}$ | 9 |
| $p_{cut-off}$ | Suction cut-off value $\{[kN/m^3]; [\geq 0]; 100\}$ | 9 |
| $K_F$ | Fluid bulk modulus $\{[\text{kPa}]; [> 0]; 1.0e + 38\}$ | 3 |
| ☒ **Inclu ...** | Include air compressibility in fluid-gas mixture {ON} | |
| $K_a$ | Air stiffness bulk modulus at atmospheric pressure $\{[\text{kPa}]; [> 0]; 100.0\}$ | |
| ☒ **Enab...** | Enable Biot coefficient {OFF} | |
| $\alpha$ | Biot coefficient $\{[-]; (e_0, 1]; 1.0\}$ | |

Window 2-854

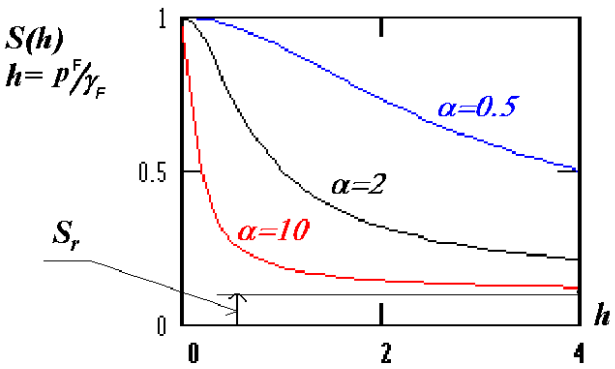
## Window 2-855: General remarks on flow properties

**Remarks**:

1. Parameters marked with (★) can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)
2. Avoid situation when seepage elements are adjacent to impermeable material (☒ **Flow** is set OFF)
3. Large value of $\beta_F$ yields fluid quasi-incompressibility
4. Permeability coefficients are set up in principal directions
5. For isotropic media set $k_x = k_y = k_z$. In that case setting direction is meaningless
6. Direction of $z\prime$ will be evaluated automatically as perpendicular to both $\{x', y'\}$. Setting directions $\{x', y'\}$ as orthogonal vectors in 3D space is assisted in the dialog. Pushing any of **A2[x]**→, **A2[y]**→, **A3[z]**→, will adjust selected component of 2-nd vector to fulfill the orthogonality condition
$$x'_1 y'_1 + x'_2 y'_2 + x'_3 y'_3 = 0$$
7. Large value of $\alpha$ reduces the thickness of the transition zone from the saturated to unsaturated, as shown in Window 2-856. It will also slow down convergence.
8. This flag allows to cancel the gravity term in Darcy law. This option may be useful when modeling nonhydrostatic initial distribution of the presure field.
9. Pure undrained behavior of soils can be modeled by using `Driven load-undrained` driver that is available for `Deformation+Flow` analyses. Any material for continuum that has this flag set to ON will undergo undrained behavior when undrained driver is currently run. The undrained behavior is implemented through penalty approach assuming that the ratio between the artifficial bulk modulus of fluid and soil skeleton (the current elastic bulk modulus) is a relatively large value from range $10^5 \div 10^7$. In addition one may control the evolution of the suction pressure that can be generated during excavations. However, special care must be taken here. If the undrained driver follows any other like `Driven load+Steady state`, `Driven load+Transient` or `Consolidation` then the condition for the suction pressure is verified for pressure $p = S(p_o)\ p_o + \Delta p$ where $p_o$ is the pressure value produced from full two-phase theory, $S(p_o)$ is the saturation ratio corresponding to the $p_o$ pressure (according to the van Genuchten's law) and the undrained excess pore pressure $\Delta p$ is produced exclusively by the undrained driver.

Window 2-855

**Window 2-856: Saturation ratio**




Saturation ratio in partially saturated zone as a function of $h = p^{\mathrm{F}} / \gamma_F$ for different $\alpha$

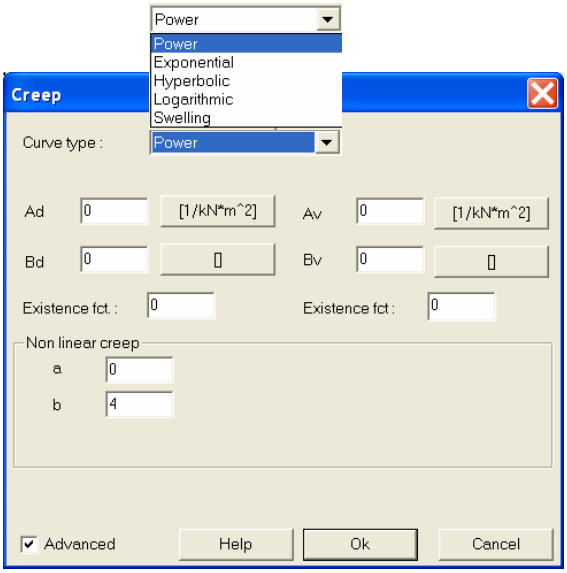
Window 2-856

## Related Topics

- *PROBLEM STATEMENT: TWO-PHASE MEDIA*
- *NUMERICAL IMPLEMENTATION: TWO-PHASE MEDIA*
- *MATERIAL MODELS: DARCY LAW*
- *NON-STANDARD MATERIAL DATA*

## 2.3.3.2.7.4 CREEP

**Window 2-857: Creep data: standard laws**



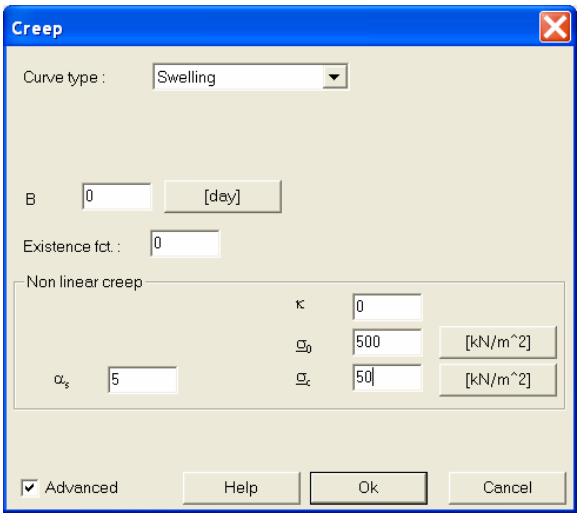

Creep measures
Exponential:
$C(t,t_0) = A\left\{1 - \exp\left[-\frac{1}{B}(t-t_0)\right]\right\}$
Power:
$C(t,t_0) = A\,(t-t_0)^B$
Hyperbolic:
$C(t,t_0) = \dfrac{A\,t}{B+t}$
Logarithmic:
$C(t,t_0) = A\,\ln(1+B\,t)$

| Symbol | Law | Mechanism | Definition |
|---|---|---|---|
| $A_d$ | Power<br>Hyperbolic<br>Logarithmic<br>Exponential | Deviatoric | Amplitude per unit stress $\{[\text{kPa}^{-1}];[\geq 0];0\}$ |
| $A_v$ | Power<br>Hyperbolic<br>Logarithmic<br>Exponential | Volumetric | Amplitude per unit stress $\{[\text{kPa}^{-1}];[\geq 0];0\}$ |
| $B_d$ | Power<br>Hyperbolic<br>Logarithmic<br>Exponential | Deviatoric | Evolution rate $\{[-];[\geq 0];0\}$<br>Evolution rate $\{[-];[\geq 0];0\}$<br>Evolution rate $\{[\text{day}^{-1}];[\geq 0];0\}$<br>Evolution rate $\{[\text{day}];[\geq 0];0\}$ |
| $B_v$ | Power<br>Hyperbolic<br>Logarithmic<br>Exponential | Volumetric | Evolution rate $\{[-];[\geq 0];0\}$<br>Evolution rate $\{[-];[\geq 0];0\}$<br>Evolution rate $\{[\text{day}^{-1}];[\geq 0];0\}$<br>Evolution rate $\{[\text{day}];[\geq 0];0\}$ |
| $a$ | Any | Both | Nonlinear creep parameter $\{[-];[\geq 0];0\}$ |
| $b$ | Any | Both | Nonlinear creep parameter $\{[-];[\geq 0];0\}$ |

**Remarks**:
1. Creep activation is controlled by existence functions independently for volumetric and deviatoric mechanisms
2. Creep may apply to any constitutive model, also nonlinear, but with linear elastic kernel
3. Nonlinear creep amplifies the creep measure $C(t,t_0)$ by term $C_1(\boldsymbol{\sigma}) = 1 + a\ SL^b$ where stress level $SL$ is defined as $SL = \frac{\sqrt{J_2}}{\sqrt{J_{2f}}}$ and $a$ and $b$ are material properties
4. For swelling see Window 2-858

Window 2-857

## Window 2-858: Creep data: swelling

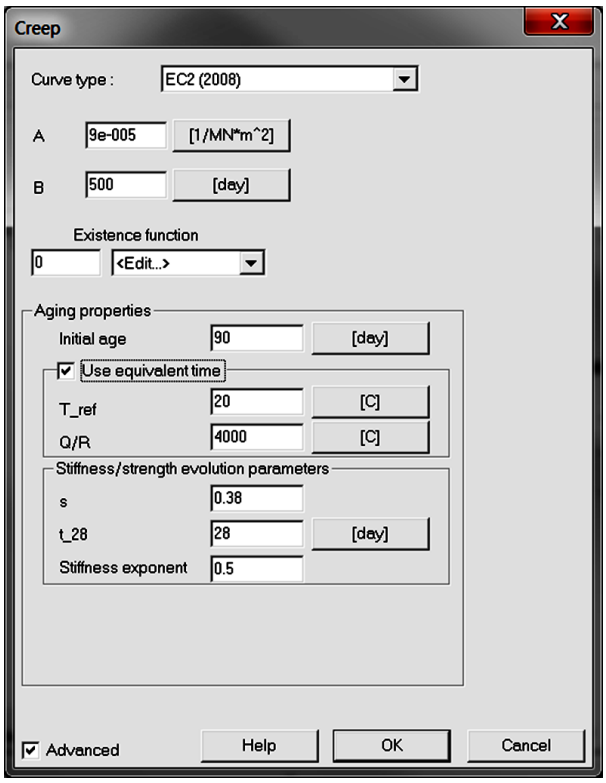



| Symbol | Definition | Remarks |
|---|---|---|
| $B$ | Evolution rate $\{[day];[\geq 0];0\}$ | 4,5 |
| $\alpha_s$ | Parameter related to evolution rate $\{[-];[\geq 0];5\}$ | 5 |
| $\kappa$ | Swelling potential parameter $\{[-];[\geq 0];0\}$ | 3 |
| $\sigma_{oc}$ | Swelling potential parameter $\{[\text{kPa}];[\geq 0];0\}$ | 3 |
| $\sigma_{os}$ | Swelling potential parameter $\{[\text{kPa}];[\geq 0];0\}$ | 3 |

**Remarks**:

1. Swelling activation is controlled by the existence function
2. Swelling may apply to any constitutive model, also nonlinear, but with linear elastic kernel
3. Swelling potential in the oedometric test is defined as $\varepsilon^{SW} = -\kappa \ln\left(\frac{\sigma}{\sigma_o}\right)$ for the vertical stress in the range $\sigma_c \leq \sigma \leq \sigma_o$; hence free swelling strain is limited to the value $\varepsilon^{SW} = -\kappa \ln\left(\frac{\sigma_c}{\sigma_o}\right)$ and by knowing the free swelling strain one may easily derive the $\sigma_c$ parameter; for stress $\sigma \geq \sigma_o$ swelling potential is zero
4. As swelling is implemented in the explicit format hence time stepping must be adjusted with respect to the $B$ parameter (larger $B$ yields smaller swelling rate) ($\Delta t \leq \frac{2B\sigma_{MIN}}{\kappa\, E_{oed}}$ where $\sigma_{MIN}$ is the stress value which yields largest creep potential and $E_{oed} = \frac{E\,(1-\nu)}{(1-2\nu)(1+\nu)}$ is the oedometric stiffness modulus)
5. Parameter $\alpha_s$ is used to slowdown swelling rate in zones of small swelling potential (for $\alpha_s \neq 0$ $B$ is amplified by some function of the stress state); this effect can be cancelled by setting $\alpha_s = 0$

## Window 2-859: Creep data: EC2 creep with aging




| Symbol | Definition | Remarks |
|---|---|---|
| $A$ | Creep amplitude | 3 |
| $B$ | Creep rate | 4 |
| Initial age | Age of analyzed concrete | 5 |
| Equivalent time flag | Flag whether to use temperature adjusted time | 7, 6 |
| $\frac{Q}{R}$ | Ratio between activation energy and universal gas constant | 8 |
| $T_{ref}$ | Reference temperature | |
| $s$ | Strength evolution parameter | 9 |
| $t_{28}$ | Time of 28 days in formula for $\beta_{cc}$ (EC2) | 10 |
| $n$ | Exponent in expression for stiffness modulus | 11 |

**Remarks**:

1. **This creep law is aplicable exclusively to the plastic damage for concrete**
2. The EC2 creep model is implemented in the modified framework of aging visco-elasticity
3. $A = A_1 = \phi_{RH}\beta(f_{cm})/E$ (see EC2 or dedicated report on plastic damage model for concrete)
4. $B = \beta_H$ (see EC2 or dedicated report on plastic damage model for concrete)
5. when age of analyzed concrete is larger than $t_{28}$ then concrete is treated as fully matured
6. $t_T(t) = \int_{t_1}^{t} \exp(-\frac{Q}{R}\left(\frac{1}{273+T(\tau)} - \frac{1}{273+T_{ref}}\right) d\tau$
7. In case of uncoupled thermo-mechanical analysis evolution of creep and aging can be amplified when temperature is larger than $T_{ref}$; this option is valid only when thermal field is defined

8. $Q/R = 4000$ [K]
9. according to EC2 $\beta_{cc} = \begin{cases} \exp(s(1-\sqrt{t_{28}/t})) & \text{for } t \leq t_{28} \text{ days} \\ 1 & \text{for } t > t_{28} \text{ days} \end{cases}$
10. in the EC2 standard $t_{28} = 28$ days
11. in the EC2 standard evolution of Young modulus in time is expressed as $E = E_{28}\ \beta_{cc}^{n}$

Window 2-859

## Related Topics

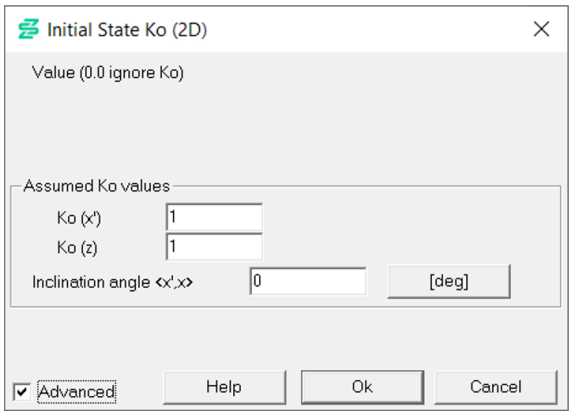
*Theory:*

- CREEP

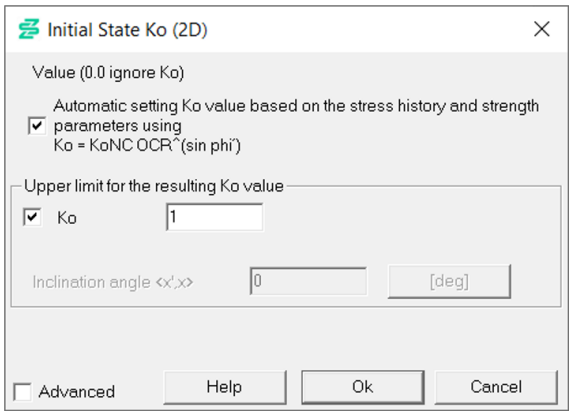
## 2.3.3.2.7.5 INITIAL K0 STATE

**Window 2-860: Initial Ko state data (2D)**

Standard models




HS model




| Symbol | Definition | Remarks |
|---|---|---|
| $K_{ox}$ | Earth pressure coefficient in $x'$ direction $\{[-];[>0];1\}$, | 1,3 |
| $K_{oz}$ | Earth pressure coefficient in $x'$ direction $\{[-];[>0];1\}$, | 1,3 |
| $\beta$ | Angle between $x$ and $x'$ axes | 2 |
| $K_o$ | Upper limit for the resulting Ko value $\{[-];[>0];1.0\}$, | 5 |

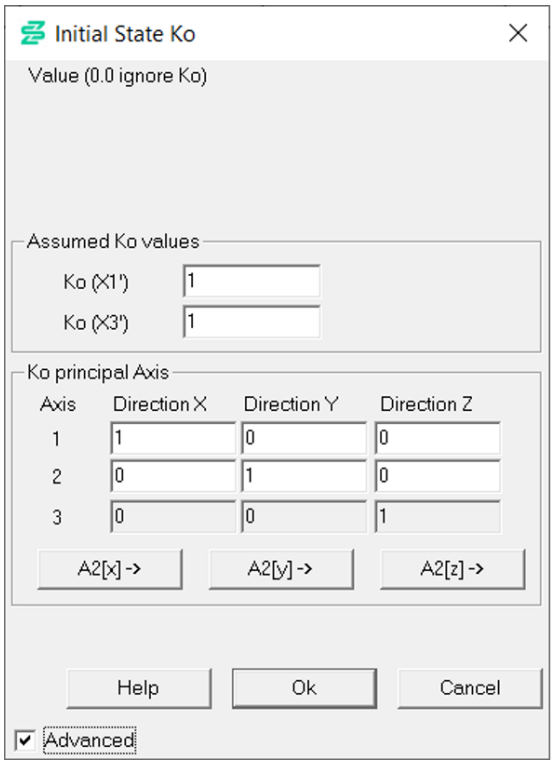
**Remarks**:

1. This setting overwrites the global one (see Window 2-924)
2. In most cases $K_{ox} = K_{oz}$ and inclination angle is equal to zero
3. The assumed $K_o$ values are meaningful only for the initial state analysis
4. In case of the HS model one may reuse the data concerning stress history (OCR or $q^{POP}$); in this case at each element a distinct value of $K_o$ coefficient will be computed (see Win.2-862 for more details); this option is meaningful for for cohesive soils
5. Setting upper limit for Ko is recommended to avoid too high resulting Ko-in situ values at shallow soil layers (concerns mainly the HS-Brick). The resulting Ko will be limited by the passive pressure coefficient Kp anyway.
6. For the 3D see Win.(2-861)

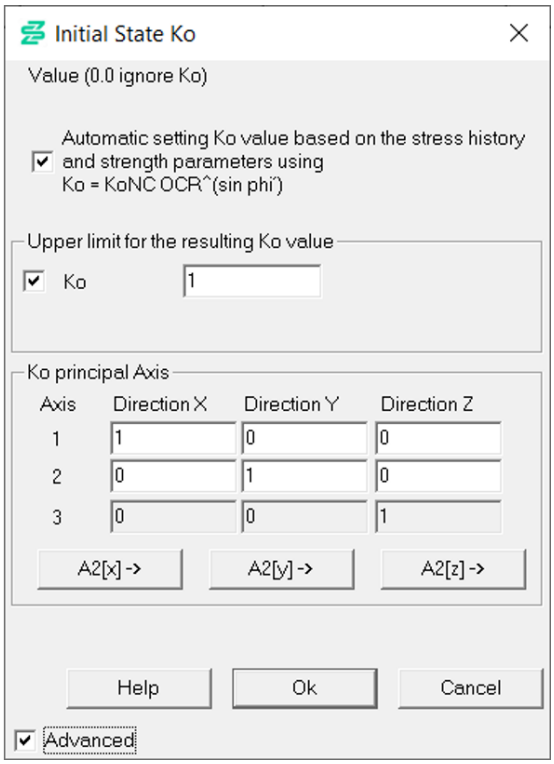
Window 2-860

## Window 2-861: Initial Ko state data (3D)

Standard models




HS model




| Symbol | Definition | Remarks |
|---|---|---|
| $K_{o\,x1'}$ | Earth pressure coefficient in $x_1'$ direction $\{[-];[>0];1\}$, | 1,2, 3 |
| $K_{o\,x3'}$ | Earth pressure coefficient in $x_3'$ direction $\{[-];[>0];1\}$, | 1,2,3 |
| $Direction$ | Base vectors, co-linear with $K_o$ directions $\{x1\prime, x3\prime\}$ | 5 |
| $K_o$ | Upper limit for the resulting Ko value $\{[-];[>0];1.0\}$, | 6 |

**Remarks**:

1. This setting overwrites the global one (see Window 2-924)
2. In most cases $K_{ox} = K_{oz}$ and direction vectors coincide with global basis vectors
3. The assumed $K_o$ values are meaningful only for the initial state analysis
4. In case of HS model one may reuse the data concerning stress history (OCR or $q^{POP}$); in this case at each element a distinct value of $K_o$ coefficient will be computed (see Win.2-862 for more details)
5. Direction of $x3\prime$ will be evaluated automatically as perpendicular to both $\{x1\prime, x2\prime\}$; setting of the directions $\{x1', x2'\}$ as orthogonal vectors in 3D space is assisted by the dialog; pushing any of **A2[x]**→, **A2[y]**→, **A2[z]**, will adjust selected component of 2-nd ( i.e. $y$) vector to fulfill the orthogonality condition $x_1'y_1' + x_2'y_2' + x_3'y_3' = 0$
6. Setting upper limit for Ko is recommended to avoid too high resulting Ko-in situ values at shallow soil layers (concerns mainly the HS-Brick). The resulting Ko will be limited by the passive pressure coefficient Kp anyway.

Window 2-861

## Window 2-862: Automatic setting $K_o$ for HS model

In case of cohesive soils the $K_o$ coefficient is strongly related to the stress history defined by means of the OCR or preoverburden pressure ($q^{POP}$). For shallow layers in subsoil the second definition (via $q^{POP}$) is usually recommended and it yields a variable OCR profile with depth ($y$ coordinate) according to the formula:

$$OCR(y) = \frac{\sigma'_{y,o} + q^{POP}}{\sigma'_{y,o}}$$

The resulting $K_o$ value at a given depth is set using the formula:

$$K_o(y) = K_o^{NC} \, (OCR)^{\sin\phi'}$$

where:

$$K_o^{NC} = 1 - \sin\phi'$$

The $K_o(y)$ value is limited by the passive pressure coefficient $K_p = \frac{1 + \sin\phi'}{1 - \sin\phi'}$

For sands we usually assume OCR=1 and therefore $K_o = K_o^{NC}$.

Window 2-862

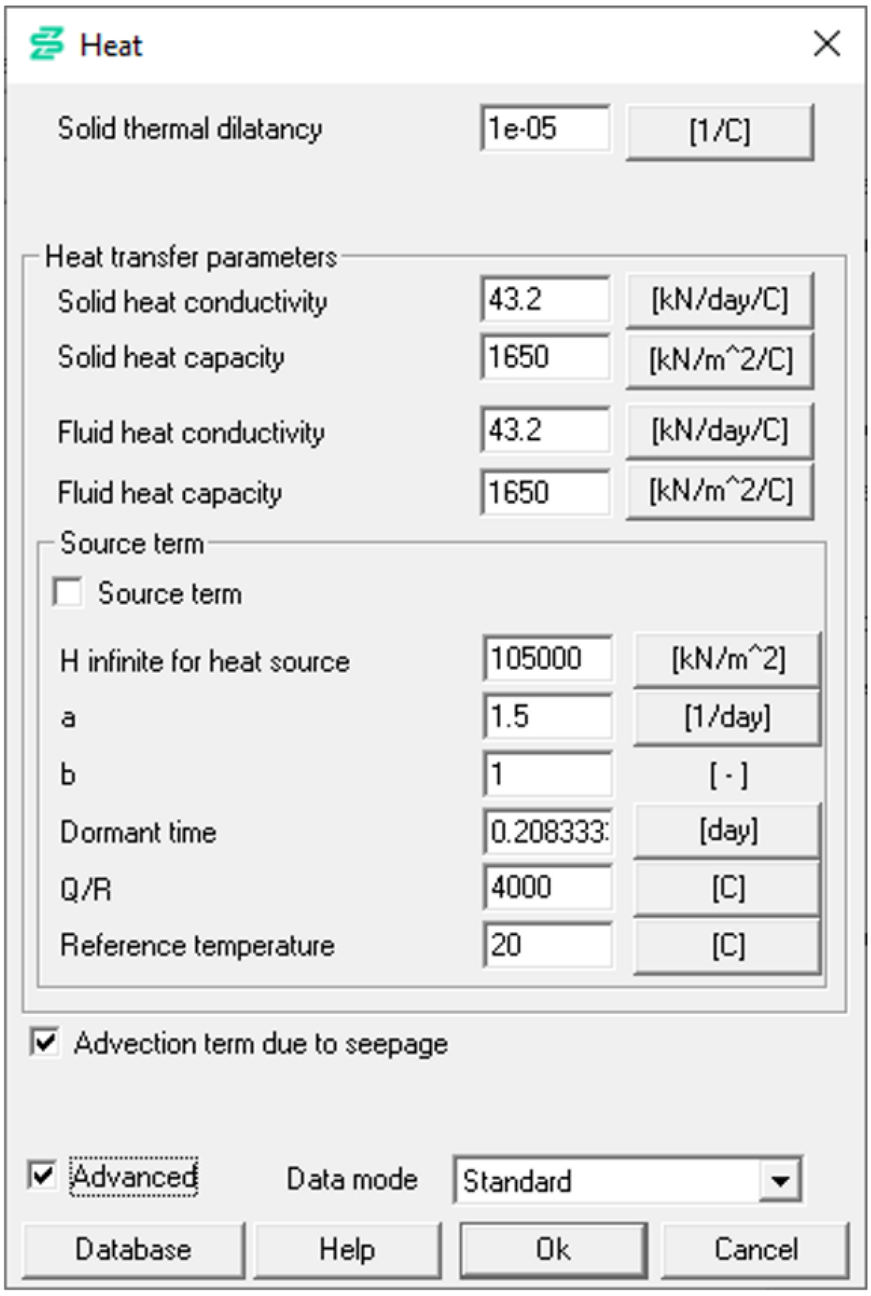
## Related Topics

*Theory:*

- GEOTECHNICAL ASPECT - INITIAL STATE
- NUMERICAL IMPLEMENTATION - INITIAL STRESS

### 2.3.3.2.7.6 HEAT

**Window 2-863: Heat data**




Data used for: `Heat`, `Deformation` and `Deformation+Flow` `Problem Type`.

| Data | Definition |
|---|---|
| `Heat` | • solid thermal dilatancy $\alpha^s$ $\{[1/(°\text{K})], [\geq 0], [10^{-5}]\}$<br>• fluid thermal dilatancy $\alpha^f$ $\{[1/(°\text{K})], [\geq 0], [10^{-5}]\}$<br>• solid thermal conductivity $\lambda^s$ $\{[\text{kN}/(°\text{K day})], [> 0], [203.7]\}$ (in case of a weakly coupled hydro-thermal analysis this parameter corresponds to the dry soil state)<br>• solid heat capacity $c^{s,*}$ $\{[\text{kN}/(\text{m}^2\,°\text{K})], [> 0], [2000]\}$ (in case of a weakly coupled hydro-thermal analysis this parameter corresponds to the dry soil state)<br>• parameters of heat source function $H = H(t,T)$ and maturity growth function $M = M(t,T)$<br>$H_\infty$, $\{[\text{kN}/(\text{m}^2)], [> 0], [105000]\}$ – total value of heat produced by hydration of cement covered in unit volume of the concrete.<br>$a$ – heat source parameter $\{[1/\text{day}], [> 0], [0.8]\}$<br>$b$ – heat source parameter $\{[-], [> 0], [1.0]\}$<br>$t_d$ – dormant time for heat source $\{[\text{day}], [> 0], [0.5]\}$<br>$Q/R$ – constant for heat source $\{[°\text{K}], [> 0], [4000]\}$ activation energy / universal gas constant<br>$T_F$ – reference temperature for heat $\{[°\text{C}], [> 0], [20]\}$. It corresponds to $293°\text{K}$.<br>• ☒ **Advection term due to seepage** flag whether advective term is to be included $\{[-], [True/False/1], [True]\}$ (this flag is meaningful only if a seepage project is associated)<br>• fluid thermal conductivity $\lambda^f$ $\{[\text{kN}/(°\text{K day})], [> 0], [51.84]\}$ (when seepage project is associated with the heat transfer one, a weakly coupled hydro-thermal problem is solved, including advection term; in such cases overall conductivity is computed using formula $\lambda = (1-n)\lambda^s + n\ S\ \lambda^f$ (here $S$ is the current saturation ratio mapped from seepage mesh onto thermal one)); this parameter is meaningful only if the seepage project is associated<br>• fluid heat capacity $c^{f,*}$ $\{[\text{kN}/(\text{m}^2\,°\text{K})], [> 0], [4182]\}$ (when seepage project is associated with the heat transfer one, a weakly coupled hydro-thermal problem is solved, including advection term; in such cases overall heat capacity is computed $c^* = (1-n)c^{s,*} + n\ S\ c^{f,*}$ (here $S$ is the current saturation ratio mapped from seepage mesh onto thermal one)); this parameter is meaningful only if the seepage project is associated |

**Remarks**:

1. `Heat Data Group` set to OFF cancels thermal field from any continuum element (with this material) for `Heat` problem, and thermal stresses at any element (with this material) for `Deformationor Deformation+Flow` problems
2. weakly coupled hydro-thermal analysis can be carried out when the seepage project is associated; in that case overall conductivity, and heat capacity, are computed for given parameters for both soil and fluid
3. if the seepage project is not associated then the overall solid heat conductivity is defined exclusively by the $\lambda^s$ parameter and the heat capacity by the $c^{s,*}$ one

4. $\alpha$ and $\alpha^F$ are needed exclusively for `Deformation` or `Deformation+Flow` problems and apply to any solid material model
5. $\lambda^s$ is obligatory for all heat transfer drivers
6. $c^{s,*}$ is obligatory for the transient heat driver unless the advection term is included; $c^* = c\rho$ (specific heat is denoted by $c$ and density of the medium by $\rho$)
7. hydration source parameters are optional and needed for
   - `Heat` problem if transient driver(s) are used
   - `Deformation` or `Deformation+Flow` problems when, `Aging concrete` model is used in conjunction with supplied nonstationary temperature field declared in `Analysis & Drivers` dialog (based on supplied thermal solution and given heat source parameters so-called maturity growth function $M(T(t), t)$ is integrated in time; this function value is an argument of the current stiffness modulus for `Aging concrete` model
   - heat source is described by the following formula

$$H(t,T) = H_\infty \frac{(aM)^b}{1+(aM)^b}$$

   - $a$ and $b$ parameters need to be optimized for given cement class; for $b = 1$ the $dH/dt(t)$ function is monotonicaly decreasing in time while for $b > 1$ it starts from zero then reaches the maximum and then it is monotonicaly decreasing; for the cement class CEM I 42.5R optimal parameters are $a = 1.645$ [1/day] and $b = 1.605$ [-] while for cement class CEM I 32.5R $a = 1.517$ [1/day] and $b = 1.553$ [-]; for other cement classes best fit procedures must be used
   - $H_\infty$ is computed by multiplying the total heat produced from 1kg of a given cement (for CEM I 42.5R it is 360 kJ/kg and for CEM I 32.5R 350 kJ/kg) by mass of cement in unit concrete volume
8. $\alpha^F$,$\lambda$, $c^*$ and hydration source parameters apply exclusively to material models coherent with continuum finite elements
9. One may use certain sets of parameters from database by pressing **Database** button
10. Parameters $\lambda$, $c^*$,$\alpha$ can be assumed as temperature dependent by using evolution functions (at the material level in group `Heat` use `Evolution function` in `Data mode`)

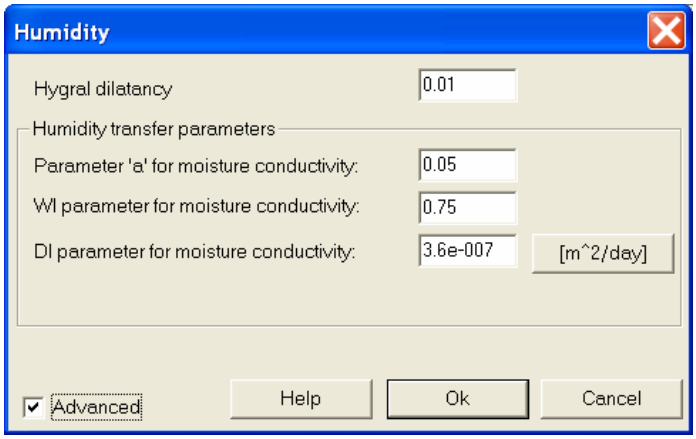
Window 2-863

## Related Topics

- MATERIAL MODELS: HEAT TRANSFER
- PROBLEM STATEMENT: HEAT TRANSFER

## 2.3.3.2.7.7 HUMIDITY

**Window 2-864: Humidity data**




| Data | Definition |
|---|---|
| `Humidity` | • Hygral dilatancy parameter $\beta$, $\{[-],[>0],[0.01]\}$<br>• Parameters for nonlinear function of diffusion coefficient<br>⋆ $a$, $\{[-],[>0],[0.05]\}$<br>⋆ $W_1$, $\{[-],[0 \geq w_1 \leq 1],[0.75]]\}$ - moisture potential at which the diffusion coefficient is equal to $\frac{1}{2}D_1(1+a)$<br>⋆ $D_1$, $\{[m^2/s],[>0],[3.6^{-7}]\}$ - diffusion coefficient for a moisture potential $W=1$ |

**Remarks**:

1. `Humidity Data Group` set to OFF cancels moisture diffusion (in each continuum element with this material) for `Humidity` problem, and stresses due to moisture variation for `Deformation` or `Deformation+Flow` problems
2. $\beta$ is needed exclusively for `Deformation` or `Deformation+Flow` problems and applies to any solid material model
3. Diffusion coefficient is defined as $D(W) = D_1 \left( a + \dfrac{1-a}{1+\left(\dfrac{1-W}{1+W_1}\right)^4} \right)$
4. Constant moisture diffusion coefficient can be obtained assuming $a = 1$

Window 2-864

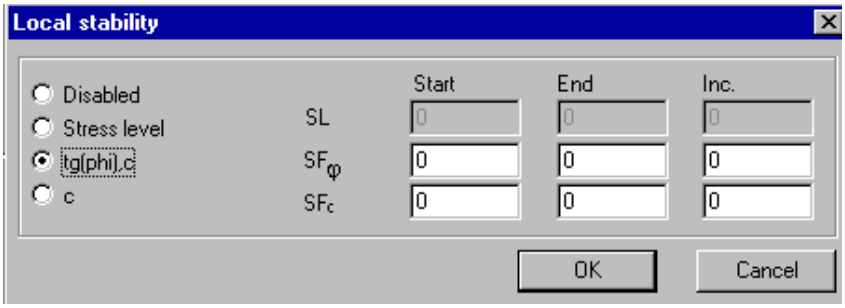
## Related Topics

- *PROBLEM STATEMENT: HUMIDITY TRANSFER*

## 2.3.3.2.7.8 LOCAL STABILITY

**Window 2-865:** `Stability` **data**

This setting overrides global stability driver setup. Elimination of local instabilities (or preventing shallow sliding) is the main aim of this option.




**Remarks**:

1. Four options can be used for strength reduction:
   - `Disabled`: strength reduction is cancelled for this material
   - `Stress Level`: (see Win.(2-21))
   - $c - \tan(\phi)$: (see Win.(2-21))
   - $c$: (see Win.(2-21))
2. This setting is meaningful for stability drivers and `Deformation` or `Deformation+Flow` problem types
3. Example:
   - Suppose that global stability driver is defined as $c - \tan(\phi)$ with $SF_{init} = 1.2$, $SF_{end} = 2.5$ and $\Delta SF = 0.1$
   - Let the local stability setup for given material is defined as: $c - \tan(\phi)$ with $SF_{init} = 1.0$, $SF_{end} = 1.3$ and $\Delta SF = 0.1$
   - Then the following pairs of $SF$ values, at global ($SF_g$) and local ($SF_l$), material, level, are used during stability analysis (($SF_g = 1.2,\ SF_l = 1.0$), ($SF_g = 1.3,\ SF_l = 1.1$), ($SF_g = 1.4,\ SF_l = 1.2$), ($SF_g = 1.5,\ SF_l = 1.3$), ($SF_g = 1.6,\ SF_l = 1.3$), ($SF_g = 1.7,\ SF_l = 1.3$), .... etc until loss of stability is achieved or $SF_g$ reaches its final value $SF_g = 2.5$

Window 2-865

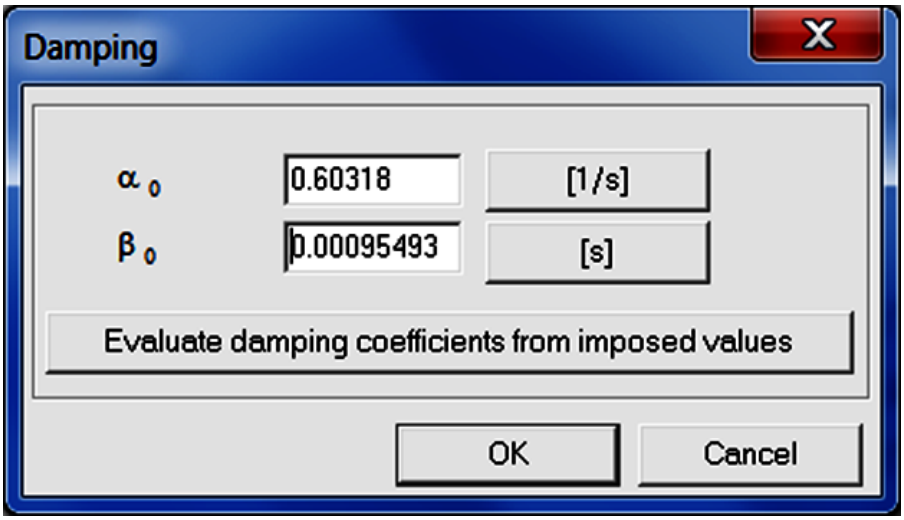
## Related Topics

- DRIVER TYPES
- STABILITY ANALYSIS

## 2.3.3.2.7.9 DAMPING

**Window 2-866: Damping data)**

This setting overrides Rayleigh damping factors $\alpha_o$ (applies to the mass) and $\beta_o$ (applies to the stiffness) (see Win.2-57 for more information on how to compute these coefficients from imposed angular frequency/frequency/period values).




Window 2-866

## 2.3.3.2.7.10 CONTINUUM: ELASTIC

**Window 2-867: Linear `Elastic` model for `Continuum`**

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. For `Flow` problem the only meaningful data are to be set under `Flow` and `Unit Weight` groups
2. For `Heat` problem the only meaningful data are to be set under `Heat` group
3. For `Humidity` problem the only meaningful data are to be set under `Humidity` group

Window 2-867

## Related Topics

*Theory*

- ELASTICITY

## 2.3.3.2.7.11 CONTINUUM: AGING CONCRETE

Aging concrete model is designed to represent time dependent mechanical properties as well as rheological behavior of concrete at early age.

**Window 2-868:** `Aging concrete` **model for** `Continuum`

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory<br>• $nP$ - number of maturity points for interpolation in time (default 10)<br>• $nM$ - number of Maxwell elements (max=6)<br>• Young moduli $E_k$ $[E_k > 0$ [kPa] and their distribution factors $W_{ki}$ $(\sum_{k=1}^{nM} W_{ik} = 1)$ over $nM$ Maxwell elements, defined at maturity points $(M_i\ i = 1..nP)$<br>• $\nu$ - Poisson ratio same for all Maxwell elements $\nu[[-], [< 0.49999], 0.2]$<br>• $\tau_k$ - retardation times $[h]$ for each Maxwell element $(k = 1..nM)$ |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Non--Linear` | needed for output purposes only $f_t(M_i)$ – tensile strengths [kPa] digitized at $nP$ maturity points |
| `Heat` | optional; (see Win.(2-863))<br>• for `Heat` analysis use standard (see Win.(2-863))<br>• for `Deformation` and `Deformation+Flow` analysis it serves parameters needed to integrate maturity $M$ based on nonstationary temperature field given in associated `Heat Transfer` project; if group ☒ **Heat** is set OFF then maturity is equivalent to the time of existence of a given material:<br>⋆ $\alpha$ – thermal dilatancy; $\alpha$ $[[1/°\text{C}], \geq 0, 10^{-5}]$.<br>⋆ $Q/R$ – hydration heat source constant $Q/R = 4000[°\text{K}]$ use same value as for corresponding material in associated `Heat Transfer` project<br>⋆ $T_F$ – reference temp. for hydration heat source $T_F$= $20[°\text{C}]$ use same value as for corresponding material in associated `Heat Transfer` project |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. For `Flow` problem the only meaningful data are to be set under `Flow` and `Unit Weight` groups
2. For `Heat` problem the only meaningful data are to be set under `Heat` group

3. For `Humidity` problem the only meaningful data are to be set under `Humidity` group

Window 2-868

## 2.3.3.2.7.12 CONTINUUM: CAP

**Window 2-869:** `Cap` **model for** `Continuum`

This model is a critical state, multisurface elasto-plastic model with volumetric hardening and nonassociated flow rule, used mainly to reproduce clay behavior.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory;<br>• Drucker–Prager data<br>• Cap parameters:<br>⋆ $\lambda$ - slope of primary consolidation line $\lambda$ $\{[-],[>0],0.1\}$ in $(e-\ln\ p)$<br>⋆ OCR - overconsolidation ratio; OCR $\{[-],[\geq 1.0],1.0\}$<br>⋆ $\underline{p}_{\text{co}}$ and $R$ - initial preconsolidation pressure and initial shape cap surface parameters<br>1. for `Direct Definition` enter:<br>♦ $\underline{p}_{\text{co}}$ (★) {kPa, $[>0], 10^{38}$}<br>♦ $R$ $\{[-],[1.1<R<2.7],1.5\}$<br>1. for indirect definition through an `Oedometric Test` enter:<br>♦ vertical preconsolidation stress $\underline{\sigma}_{\text{VM}}$, { kPa, $[>0],0$}<br>♦ coefficient of lateral pressure at normal consolidation state $K_0^{\text{NC}}$, $\{[-],[>0],0.5\}$(Jaky's formula $K_0^{\text{NC}}=1-\sin(\phi)$ can be applied) |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Parameters marked with (★) can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)
2. In case of stability analysis the elliptic cap is ignored and the algorithm applies only to the Drucker-Prager yield surface

3. Note that the initial preconsolidation pressure is not a material but rather a state parameter; for extended handling the initial preconsolidation pressure via OCR see Window 2-870,2-871

Window 2-869

**Window 2-870: Setting $p_{co}$ via OCR if `Initial state` driver is active**

User defined $\underline{p}_{\text{co}}$ is assumed to be the minimum allowed value of preconsolidation pressure ($\underline{p}_{\text{co-min}}$). During the initial state computation hardening is deactivated and model behaves as an ideal elasto-plastic one. Once the initial state is terminated the code estimates automatically, at each integration point, the initial $p_{co}$, based on computed initial stress state $\boldsymbol{\sigma}_{\mathbf{o}}$, user defined OCR value and $\underline{p}_{\text{co-min}}$. This algorithm is summarized below.

- For given stress state $\boldsymbol{\sigma}_{\mathbf{o}}$ compute the $\underline{p}_{\text{c}}^{*}$ such that $F(\boldsymbol{\sigma},\underline{p}_{\text{c}}^{*}) = 0$ (cap plasticity condition)
- Correct the $\underline{p}_{\text{c}}^{*}$ by current OCR ratio: $\underline{p}_{\text{c}}^{**} = \underline{p}_{\text{c}}^{*}$ OCR
- IF $\underline{p}_{\text{c}}^{**} < \underline{p}_{\text{co-min}}$ set: $\underline{p}_{\text{co}} = \underline{p}_{\text{co-min}}$
  ELSE $\underline{p}_{\text{co}} = \underline{p}_{\text{c}}^{**}$

**Remark**:

To force the code to accept user defined $\underline{p}_{\text{co-min}}$ as an initial value for preconsolidation pressure OCR=1 must be assumed; however if given $\underline{p}_{\text{co-min}}$ value is too small at a given integration point its value will automatically be enlarged to satisfy plastic admissibility condition $F(..) \leq 0$

Window 2-870

**Window 2-871: Setting $p_{co}$ if `Initial state` driver is not active**

In this case the two possibilities are considered by the code to set up the initial $\underline{p}_{co}$ value

1. If the initial guess for the initial stress $\boldsymbol{\sigma}_{o}^{*}$ is defined by the user (through the `Initial stress super-elements`) then during the initialization phase the initial value of the $\underline{p}_{co}$ is set in the same manner as in Window 2-870 assuming $\boldsymbol{\sigma}_{o}$=$\boldsymbol{\sigma}_{o}^{*}$.
2. If the initial guess for the initial effective stress state is not given explicitly by the user then the OCR factor is not interpreted and the initial preconsolidation pressure is set directly to the value of $\underline{p}_{co}$ as declared in the user interface

Window 2-871

## Related Topics

*Theory:*

- CAP MODEL

## 2.3.3.2.7.13 CONTINUUM: MODIFIED CAM-CLAY

**Window 2-872:** `Modified Cam-Clay` **model for** `Continuum`

This model is a critical state single surface elasto-plastic model with volumetric hardening and associated flow rule, used mainly to reproduce normally or overconsolidated clay behavior.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; only Poisson coefficient must be entered: $\nu(\bigstar)$,$\{[-],[0<\nu<0.49999],0.3\}$ |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Non--Linear` | obligatory;<br>• Slope of critical state line in p-q plane $M_c$, $\{[-],[>0],[<3],1.0\}$<br>• Slope of primary consolidation in e-ln(p) $\lambda$, $\{[-],[>0],0.1\}$<br>• Slope of secondary consolidation in e-ln(p) $\kappa$, $\{[-],[>0],\kappa<\lambda,0.01\}$<br>• Minimum preconsolidation pressure $\underline{p_{co}}$ $(\bigstar)$, $\{[\text{kPa}],[>0],10\}$<br>• Overconsolidation ratio OCR, $\{[-],[\geq 1],1\}$<br>• Strength anisotropy factor (ratio between tension and compresion meridian slopes $k=M_e/M_c$) can be:<br>⋆ `disabled` $(k=1)$<br>⋆ `default` $(k=\frac{3}{3+M_c})$<br>⋆ `prescribed` by the user $k$, $\{[-],0.5<k<1.0,0.75\}$ |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Parameters marked with $(\bigstar)$ can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)
2. In case of stability analysis the elliptic cap is ignored and the algorithm applies only to the M-C type yield surface (in p-q plane it coincides with the critical state line)
3. Note that the initial preconsolidation pressure is not a material but rather a state parameter; the extended handling of the initial preconsolidation pressure via OCR is compatible with the one used for Cap model (see Win.(2-870,2-871))
4. **Model cannot be run without setting the initial guess for the in situ stress** as the shear and bulk moduli dependent on the effective mean stress

5. In case of overconsolidated soils problem of mesh dependency may appear and thus the results should be analyzed carefully

Window 2-872

## Related Topics

*Theory*

- NON-STANDARD MATERIAL DATA
- CAM-CLAY MODEL
- MODIFIED-CAM CLAY MODEL – STRESS RETURN ALGORITHM

## 2.3.3.2.7.14 CONTINUUM: DRUCKER–PRAGER

**Window 2-873:** `Drucker-Prager` **model for** `Continuum`

This model is a multisurface (due to optional cut-off condition), perfect, elasto-plastic model with a nonassociated flow rule, adjusted with respect to the Mohr-Coulomb criterion.

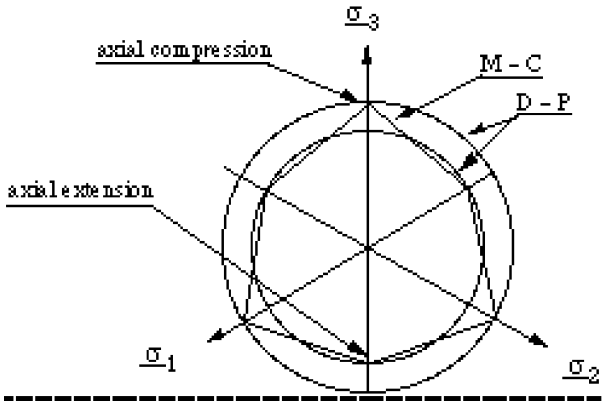
| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory ;<br>• Cohesion $C$ (★), {kPa, $[>0]$, 20}<br>• Friction angle $\varphi$ (★), {[deg], $[0, 89]$, 30}<br>• Dilatancy angle $\psi$ (★), {[deg], $[0, 89]$, 0} Set $\psi = 0$ for incompressible behavior<br>• Tensile cut–off(★); optional, ignore if not konwlegable of effects<br>• Size adjustment with respect to the Mohr-Coulomb criterion: (see Window 2-874: `External Edges`, `Internal Edges`, `Elastic`, `Plane Strain` (default for plane strain), `Intermediate` (default for axisymmetry) |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Parameters marked with (★) can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)
2. $\phi - \psi \geq 30^o$ may yield convergence problems due to strain localization effects (caused by nonassociated flow rule)
3. Huber-Misses criterion is obtained setting $\phi = 0$, $C = 2\sqrt{3}f_y$
4. $\psi = 0$ corresponds to the incompressible plastic flow rule while $\psi > 0$ yields always dilatancy
5. `Tensile Cut--Off` condition reduces the maximum stress ratio: $(\sqrt{J_2}/I_{1t}) \leq 1/\sqrt{3}$ or equivalently $p/q \leq 3$.

Window 2-873

**Window 2-874: Size adjustment of D-P criterion with respect to M-C**




**Remarks**:

1. `Plane Strain` size adjustment of the D-P criterion with respect to the M-C one is the default for plane strain problems; in this particular case limit loads, safety factors are exactly the same no matter if exact Mohr-Coulomb or Drucker-Prager model is used
2. Drucker-Prager model is computationally more efficient than the M-C one
3. `Intermediate` size adjustment of the D-P criterion with respect to the M-C one is the default for axisymmetric case

Window 2-874

## Related Topics

*Theory:*

- DRUCKER-PRAGER CRITERION
- DILATANCY
- NON-STANDARD MATERIAL DATA

## 2.3.3.2.7.15 CONTINUUM: HOEK–BROWN (M-W)

**Window 2-875:** `Hoek-Brown` **model for** `Continuum`

This model is a multi-surface (due to $I_1$ type of the cut-off condition) elasto-plastic model with softening (optional) and nonassociated flow rule, used to reproduce concrete or rock behavior. The yield surface is described by a general Menetrey-Willam criterion with noncircular form in the deviatoric plane and parabolic shape of meridians in the $\sqrt{J_2} - I_1$ plane.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory;<br>• Compressive uni-axial strength $f_c$, $\{[\text{kPa}], [> 0, > f_t], 15000\}$<br>• Tensile uni-axial strength $f_t$, $\{[\text{kPa}], [> 0, < f_t], 1000\}$<br>• Biaxial compressive/uniaxial compressive strength ratio $f_b/f_c$, $\{[-], [> 1], 1.16\}$<br>• `Tensile Cut--Off` flag ON/OFF<br>first stress invariant $I_{1t}$ (if `Cut--Off` is ON), $\{[\text{kPa}], [> 0], 0\}$<br>• Plastic flow type `Deviatoric`, `Drucker--Prager`, `Tensile Meridian`, `Hoek--Brown (M--W)`,<br>• Dilatancy angle at uniaxial compressive strength $\psi_c$<br>(if `Drucker--Prager` or `Hoek--Brown (M--W)` flow is ON), $\{[\text{deg}], [\arctan\left(\frac{f_t}{\sqrt{2}f_c}\right) \leq 35.3°], 0\}$<br>• Softening flag ON/OFF[7]. This option must be used with caution. If activated, introduce:<br>⋆ Crack opening at failure $w_r$, $\{[m], [> 0], 0.0001]\}$<br>⋆ Steepness parameter $a$, $\{[-], [> 0], 5.0]\}$<br>⋆ Fictitious number of cracks for compression $b$, $\{[-], [> 1], 10.0]\}$ |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Typical values of yield surface parameters for concrete and rock are:

| | units | rock | concrete |
|---|---|---|---|
| $f_t$ | kPa | $(2-30)\cdot 10^3$ | $4\cdot 10^3$ |
| $f_c$ | kPa | $(5-400)\cdot 10^3$ | $40\cdot 10^3$ |
| $e$ | – | 0.5-0.6 | 0.52 |

2. Exponential softening law [8] is driven by norm of tensile part of plastic strain tensor as stated below:

$$F(\sigma)-c=0, \quad c=\exp(-a\frac{w}{w_r}), \quad \Delta w = h^e\cdot\Delta q\frac{1}{N(b,\sigma)}, \quad \Delta q = \|<\Delta\epsilon^p>\|$$

where: $c$ is a softening function, $w$ and $q$ are softening parameters, $h^e$ is an element size, $N(b,\sigma)$ is a function describing fictitious number of cracks(=1 for tension, $b$ for compression) and principal tensile plastic strain is denoted by $\epsilon^p$

Window 2-875

## Related Topics

*Theory:*

- HOEK–BROWN CRITERION
- MOHR–COULOMB MODEL
- DILATANCY

[8] concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT** only

## 2.3.3.2.7.16 CONTINUUM: MOHR–COULOMB (M-W)

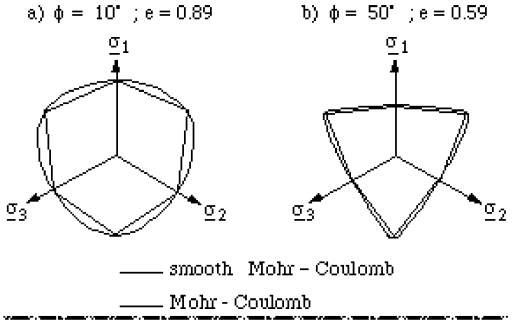
**Window 2-876:** `Mohr-Coulomb (M-W)` **(smooth version) model for** `Continuum`

This model is a multi-surface (due to $I_1$ type of the cut-off condition) elasto-plastic model with nonassociated flow rule, used to reproduce soil or rock behavior. The yield surface is described by a general Menetrey-Willam criterion specialized here to the smooth version of Mohr-Coulomb one.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory;<br>• Cohesion $c$, $\{[\text{kPa}], [> 0], 20\}$<br>• Friction angle $\phi$, $\{[\text{deg}], [0, 89], 30\}$<br>• `Cut--Off` flag ON/OffFlg<br>• Tensile strength $f_t$ $\{\{[\text{kPa}], [0, c/\tan(\phi)], 0\}$ (if `Cut--Off` flag is ON),<br>• Plastic flow type `Deviatoric`, `Drucker--Prager`<br>• Dilatancy angle $\psi$, $\{[\text{deg}], [0, 89], 0\}$ (if `Drucker--Prager` flow is ON, set $\psi = 0$ for incompressible behavior<br>• Size adjustment flag ON/OFF |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Deviatoric sections of smooth M-C surface are shown in fig. below




2. $\phi - \psi \geq 30^o$ may yield convergence problems due to strain localization effects (caused by nonassociated flow rule)
3. Plastic flow is driven by a Drucker–Prager type plastic potential (circular in the deviatoric plane); hence even for $\psi = \phi$ plastic flow is non-associated; $\psi = 0$ results in deviatoric plastic flow while $0 < \psi \leq \phi$ yields dilatancy
4. `Tensile Cut--Off` condition imposes: $I_1 \leq I'_{1t}$
5. To achieve same limit loads and safety factors when applying standard M-C model and M-C (M-W), in the plane strain (exclusively), ☒ **Size adjustment** has to be set ON; for $\phi = 0$ (Huber–Misses criterion) size reduction factor is of order $0.866$;

Window 2-876

## Related Topics

*Theory:*

- MOHR–COULOMB
- DILATANCY

## 2.3.3.2.7.17 CONTINUUM: MULTILAMINATE

**Window 2-877:** `Multilaminate` **model for** `Continuum`

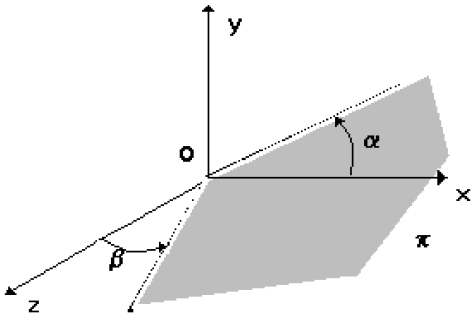
| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory;for each active weakness plane set:<br>• for 2D models:<br>⋆ Inclination angle $\alpha$ measured in anti-clockwise direction, $\{[\text{deg}], [-90 \leq \alpha \leq 90], 0\}$<br>• for 3D models:<br>⋆ Inclination angles $\alpha$, $\beta$ $\{[\text{deg}], [-360 \leq \alpha/\beta \leq 360], 0\}$, (if ☒ **Advanced mode** is set ON)<br>⋆ Vector $\mathbf{n}$ normal to the weakness plane (if ☒ **Advanced mode** is set OFF)<br>• Cohesion $c$, $\{[\text{kPa}], [> 0], 0\}$<br>• Friction angle $\phi$, $\{[\text{deg}], [0, 89], 0\}$<br>• Dilatancy angle $\psi$, $\{[\text{deg}], [0, 89], 0\}$; set $\psi = 0$ for incompressible behavior.<br>• `Cut--Off` flag ON/OFF<br>• Tensile strength $f_t$, $\{[\text{kPa}], [0, c/\tan(\phi)], 0\}$ (if `Cut--Off` flag is ON)<br>• Matrix material model ( elastic if ☒ **Advanced** is OFF ) or `Mohr-Coulomb (M-W)` / `Rankine (M-W)`/`Hoek-Brown` |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

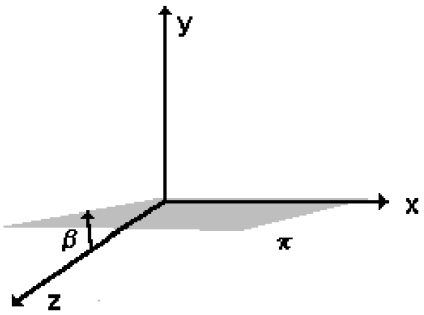
1. To deactivate weakness plane set: $\phi = \psi = c = 0.0$
2. Matrix material is assumed as elastic by default; if ☒ **Advanced** mode is set ON one may apply an elasto-plastic model described by Rankine (M-W) (see Win.(2-879)), Drucker-Prager (see Win.(2-873)) or Mohr-Coulomb (M-W) (smooth) (see Win.(2-876)) yield surface

Window 2-877

**Window 2-878: Meaning of $\alpha$ and $\beta$ angles in 3D**




Standard case




Special case: $\alpha = 0$

Window 2-878

## Related Topics

*Theory:*

MULTILAMINATE MODEL

## 2.3.3.2.7.18 CONTINUUM: RANKINE (M-W)

**Window 2-879:** `Rankine (M-W)` **model for** `Continuum`

This model is a single surface, strain softening (optional) elasto-plastic model with a nonassociated flow rule, derived from general Menetrey-Willam criterion.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory; set:<br>• Tensile strength (uni-axial) $f_t$, $\{[\text{kPa}], [>0], 0\}$<br>• Softening flag; if ☒ **Softening** is set ON define<br>⋆ Crack opening at failure $w_r$, $\{[m], [>0], 0.0001\}$<br>⋆ Steepness parameter $a$, $\{[-], [>0], 5.0\}$<br>⋆ Fictitious number of cracks for compression $b$, $\{[-], [>1], 10.0\}$ |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Plastic flow is governed by a DP type plastic potential (hence flow rule is nonassociated)
2. Strain softening option must be used with caution

Window 2-879

## Related Topics

*Theory:*

- MOHR–COULOMB
- HOEK–BROWN CRITERION

## 2.3.3.2.7.19 CONTINUUM: MOHR-COULOMB

**Window 2-880:** `Mohr-Coulomb` **(exact) model for** `Continuum`

This model is a multi-surface perfect elasto-plastic model with a nonassociated flow rule, used to reproduce soil or rock behavior.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| `Non--Linear` | obligatory; (see dialog in Window 2-880)<br>• Cohesion $C$ (★), $\{[\text{kPa}],\ [>0],\ 0\}$<br>• Friction angle $\varphi$ (★), $\{[\text{deg}],\ [0,89],\ 0\}$<br>• Dilatancy angle $\psi$ (★), $\{[\text{deg}],\ [0,89],\ 0\}$<br>Set $\psi = 0$ for incompressible behavior<br>• Tensile cut–off(Rankine); (optional) $f_t$ (★), $\{[\text{kPa}],[\geq 0],0\}$<br>• Dilatancy cut-off; (optional) $e_{\text{MAX}}$ , $\{[-],[e_{\text{max}} > e_o]\}$ |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

1. Parameters marked with (★) can be defined as variable in time and space (see *NON-STANDARD MATERIAL DATA*)
2. $\phi - \psi \geq 30^o$ may yield convergence problems due to strain localization effects (caused by nonassociated flow rule)
3. $\phi = 0$ converts M-C criterion to Tresca one
4. Plastic flow is driven by a M-C type plastic potential; $\psi = 0$ results in deviatoric plastic flow while $0 < \psi \leq \phi$ yields dilatancy
5. ☒ **Cut-off Rankine** set to ON activates strict Rankine criterion limiting the maximum tensile stress

## Related Topics

*Theory:*

- MOHR–COULOMB
- DILATANCY
- NON-STANDARD MATERIAL DATA

### 2.3.3.2.7.20 CONTINUUM: HS-small

**Window 2-881:** `Hardening Soil-Brick` **model for** `Continuum`

This model belongs to the class of a multi-surface elasto-plastic models. It can properly represent stiffness of soils in the range of small strains. Hence it is recommended for most of the soil structure-interaction problems. Soil behavior in the range of small strains can be reproduced using the classical Hardening Soil-small (HSs) version or the Brick one. The latter one is free of serious drawback of the HSs in which strain history can artificially be reset by small unloading cycles, hence it is highly recommended to all kind of applications. The classical HSs is still present but mainly for the sake of the back compatibility. This model can be used for sands and for clays. All stress paths penetrating the interior of the elastic domain undergo the modified Hardin-Drnevich stress-strain law in the range of small strains, hence the hysteretic damping can easily be reproduced (only by the HS-Brick version) in dynamic applications. All details concerning this model are given in the dedicated report on the Hardening Soil model.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; set<br>• $E_{ur}^{ref}$ is the unloading/reloading Young modulus given at the reference stress $\sigma^{ref}$ $\{[\text{kPa}], [> 0], \ 80000\}$)<br>• $E_{ur,v}^{ref}$ is the vertical unloading/reloading Young modulus given at the reference stress $\sigma^{ref}$ $\{[\text{kPa}], [> 0], \ 80000\}$), (if ☒ **Anisotropy** is set ON)<br>• Reference stress $\sigma^{ref}$, $\{[\text{kPa}], [> 0], \ 100\}$ is the stress value at which the reference stiffness moduli $E_{ur}^{ref}$ is defined<br>• $\nu_{ur}$ is the unloading/reloading Poisson coefficient; it varies from 0.15 to 0.3, hence for sands it is recommended to assume $\nu_{ur} = 0.2..0.25$ and for clays $\nu_{ur} = 0.25..0.3$, $\{[-], [0, 0.49999], \ 0.2\}$<br>• $\nu_{ur,hh}$ is the unloading/reloading Poisson's ratio for horizontal strain due to complementary horizontal strain. $\{[-], [-1.0 < \nu_{ur,hh} < 0.5], 0.2\}$, (if ☒ **Anisotropy** is set ON)<br>• $m$ is the exponent in stress dependency power law; it varies from $m = 0.4$ to $m = 0..9$; it is smaller for dense sands and larger for clays<br>• $\sigma_L$ is the minimum allowed stress ($\sigma_3$ or $p$) value used for evaluation of stiffness moduli $\{[\text{kPa}], [> 0], \ 10\}$<br>• stiffness stress dependency type can be one of three following functions<br>1. $\dfrac{\sigma_3 + c \ \cot\phi}{\sigma^{ref} + c \ \cot\phi}$<br>2. $\dfrac{\sigma_3}{\sigma^{ref}}$<br>3. $\dfrac{p}{\sigma^{ref}}$<br>the last two functions can be used in simplified undrained analyses when $\phi = 0$ is assumed; the third function is recommended in general but then reference stiffness moduli and power exponent $m$ must be recalculated once they were calibrated assuming $\sigma_3$ type stress dependencies<br>• small strain formulation can be selected from the set of the three possible options<br>1. `Disabled`<br>2. `HSs formulation` - classical<br>3. `Brick formulation`<br>The `Brick formulation` is superior with respect to the classical HSs. |

continued on next page

**continued from previous page**

| | |
|---|---|
| `Elastic` | obligatory; set<br>• $E_o^{ref}$ is the Young modulus at very small strains at the reference stress $\sigma^{ref}$; In case of lack of information on $E_o^{ref}$ one may try to estimate $E_o^{ref}$ based on Alpan's diagram (see external report on Hardening Soil model) assuming $E_s = E_{ur}$<br>• $E_{o,v}^{ref}$ is the vertical Young modulus at very small strains at the reference stress $\sigma^{ref}$, (if ☒ **Anisotropy** is set ON)<br>• $\gamma_{0.7}$ is threshold shear strain at which secant shear moduls $G$ reaches 70 % of its initial value $G_o$ (note that $G_o = \frac{E_o}{2(1+\nu_{ur})}$); in the current implementation $\gamma_{0.7}$ is assumed to be constant; in case of lack of information on $\gamma_{0.7}$ one may use the diagram by Vucetic and Dobry for cohesive soils and diagram by Wichtmann and Triantafyllidis for cohesionless ones included in the report on Hardening Soil model; $\gamma_{0.7} = 0.0001..0.0002$, $\{[-], [0.0001, 0.0005], 0.0002\}$<br>• $\alpha_G$ $\{[-], [0.7 \leq \alpha_G \leq 3.0], 2.0\}$, (if ☒ **Anisotropy** is set ON)<br>• $\beta$ $\{[-], [1.0 \leq \beta \leq 2.0], 1.6\}$, (if ☒ **Anisotropy** is set ON)<br>• Tilt angle $\theta$ $\{[\text{deg}], [-90 \leq \theta \leq 90], 0\}$, (if ☒ **Anisotropy** is set ON)<br>• Azimuthal angle $\varphi$ $\{[\text{deg}], [0 \leq \varphi \leq 180], 90\}$, (if ☒ **Anisotropy** is set ON) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |

continued on next page

**continued from previous page**

| | |
|---|---|
| `Nonlinear` | obligatory;<br>• $E_{50}^{ref}$ is the secant Young modulus at 50 % of failure deviatoric stress $q_f$ derived from the $q-\varepsilon_1$ curve in drained triaxial test (see report on Hardening Soil model for more details)<br>• $\phi$ is the friction angle $\{[\text{deg}], [>0],\ 30\}$<br>• $\psi$ is the dilatancy angle $\{[\text{deg}], [>0],\ 0\}$<br>• $c'$ is the effective cohesion $\{[\text{kPa}], [\geq 0],\ 20\}$<br>• $R_f$ is the failure ratio $\{[-], [0.8 < R_f < 1],\ 0.9\}$<br>• $f_t$ is the tensile strength $\{[\text{kPa}], [\geq 0],\ 0\}$<br>• $e_{max}$ is the maximum allowed void ratio; if current void ratio exceeds the $e_{max}$ dilatancy angle is switched to $\psi = 0$, $\{[-], [>0],\ 1\}$<br>• $D$ is the Rowe's dilatancy multiplier in the contractant domain, for standard HS model $\{[-], [0 \leq D \leq 1],\ 0.0\}$ while for HS-small $\{[-], [0 \leq D \leq 1],\ 0.25\}$<br>• Cap surface parameter $M$ and hardening parameter $H$ are derived by using a simple calculator which simulates an oedometric test; for given tangent oedometric modulus $E_{oed}$ at a given reference vertical stress $\sigma_{oed}^{ref}$ and for assumed $K_o^{NC}$ parameter (here Jaky's formula can be used) values of $H$ and $M$ are evaluated (press button **Evaluate M,H**); one may assume $E_{oed} = E_{50}^{ref}\left(\frac{\sigma_{oed}^{ref} + c\ \text{cot}\phi}{\sigma^{ref} + c\ \text{cot}\phi}\right)^m$ as a default<br>• Setting the initial state variables $\gamma_o^{PS}$ and $p_{co}$ can be carried out by means of assumed OCR or preoverburden pressure $q^{POP}$; pairs $K_o^{SR}$ and $OCR$ $(OCR \geq 1.0)$ or $K_o^{SR}$ and $q^{POP}$ are needed to setup the initial position of the cap surface and the initial value of the hardening parameter $\gamma^{PS}$ (please refer to the report on Hardening Soil model for the detailed explanation)<br>• $p_{co}^{min}$ is the minimum allowed value for the initial preconsolidation stress $\{[kPa], [>0],\ 10\}$ |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

**Remarks**:

- Prior using HS or HS-small model it is strongly recommended to study first the external

## Related Topics

*Theory:* Technical report on HS-small model

## 2.3.3.2.7.21 CONTINUUM: Densification model

**Window 2-882:** `Densification model` **for** `Continuum`

This model, derived directly from HS model, is designed to reproduce loose soil deposits behavior subject to cylic loading. Hence it is to be used mainly for dynamic predictions. It is able to reproduce cyclic compaction in single-phase problems and liquefaction in two-phase problems. This model can also be customized to nonlinear Mohr-Coulomb one (with nonlinear elasticity inside the yield surface) by switching off deviatoric plastic mechanism and deactivating densification mechanism. Using the Brick version of the small strain overlay in the HS model is highly recommended.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; set<br>• $E_{ur}^{ref}$ is the unloading/reloading Young modulus given at the reference stress $\sigma^{ref}$ $\{[\text{kPa}], [> 0],\ 80000\}$); note that the elastic stiffness moduli depend on $p$ rather on $\sigma_3$ value in this model<br>• Reference stress $\sigma^{ref}$, $\{[\text{kPa}], [> 0],\ 100\}$ is the stress value at which the reference stiffness moduli $E_{ur}^{ref}$ is defined<br>• $\nu_{ur}$ is the unloading/reloading Poisson coefficient; it varies from 0.15 to 0.3, hence for sands it is recommended to assume $\nu_{ur} = 0.2..0.25$ and for clays $\nu_{ur} = 0.25..0.3$, $\{[-], [0, 0.49999],\ 0.2\}$<br>• $m$ is the exponent in stress dependency power law; for loose sands $m \approx 0.5$<br>• $\sigma_L$ is the minimum allowed reference stress value used for evaluation of stiffness moduli $\{[\text{kPa}], [> 0],\ 10\}$<br>• stiffness stress dependency type can be one of three following functions<br>1. $\dfrac{\sigma_3 + c\ \cot\phi}{\sigma^{ref} + c\ \cot\phi}$<br>2. $\dfrac{\sigma_3}{\sigma^{ref}}$<br>3. $\dfrac{p}{\sigma^{ref}}$<br>the last two functions can be used in simplified undrained analyses when $\phi = 0$ is assumed; the third function is recommended in general but then reference stiffness moduli and power exponent $m$ must be recalculated once they were calibrated assuming $\sigma_3$ type stress dependencies<br>• small strain formulation can be selected from the set of the three possible options<br>1. `Disabled`<br>2. `HSs formulation` - classical<br>3. `Brick formulation`<br>• $E_o^{ref}$ is the Young modulus at very small strains at the reference stress $\sigma^{ref}$; note that the current very small strain stiffness modulus $E_o$ depends on $p$ rather than $\sigma_3$ (as in the HS model) |

continued on next page

| **continued from previous page** | |
|---|---|
| `Elastic` | obligatory; set<br>• $\gamma_{0.7}$ is the threshold shear strain at which secant shear moduls $G$ reaches 70 % of its initial value $G_o$ (note that $G_o = \frac{E_o}{2(1+\nu_{ur})}$); in the current implementation $\gamma_{0.7}$ is assumed to be constant; in case of lack of information on $\gamma_{0.7}$ one may use the diagram by Wichtmann and Triantafyllidis included in the report on Hardening Soil model; $\gamma_{0.7} = 0.0001..0.0002,\ \{[-], [0.0001, 0.0005], 0.0002\}$ |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |
| continued on next page | |

**continued from previous page**

| | |
|---|---|
| `Nonlinear` | obligatory;<br>• $\phi$ is the friction angle $\{[\text{deg}], [> 0],\ 30\}$<br>• $c'$ is the effective cohesion $\{[\text{kPa}], [\geq 0],\ 1\}$<br>• $f_t$ is the tensile strength $\{[\text{kPa}], [\geq 0],\ 0\}$<br>• To activate internal shear plastic mechanism the flag must be set ON<br>• $R_f$ is the failure ratio $\{[-], [0.8 < R_f < 1],\ 0.9\}$<br>• $E_{50}^{ref}$ is the secant Young modulus at 50 % of failure deviatoric stress $q_f$ derived from the $q-\varepsilon_1$ curve in drained triaxial test (same as in the HS model so please refer to the report on Hardening Soil model for more details)<br>• To activate densification mechanism user must switch ON the flag ☒ **Densification mechanism** and then choose one of the two densification laws, the one proposed by Zienkiewicz or by Sawicki<br>• $A$, $B$, $\gamma$ - parameters for densification law by Zienkiewicz<br>• $C_1$, $C_2$ - parameters for densification law by Sawicki (typical values for $C_1$ are of order $0.005 \div 0.009$ and for $C_2$ of order 2*10$^5$) (please refer to the report on Dynamics for more information)<br>• Strain mapping for the accumulated volumetric strain can be selected as *Isotropic* (here the increment of the accumulated volumetric strain is projected equally on all normal directions) or *Extended* (here the increment of the accumulated volumetric strain is projected on those normal directions that are not restrained by the boundary conditions); using the extended mapping is recommended; in the shear layer problem the *Extended* mapping will put the full accumulated volumetric strain only on the vertical direction<br>• Setting the initial state variable $\gamma_o^{PS}$ can be carried out by means of assumed OCR or preoverburden pressure $q^{POP}$; pairs $K_o^{SR}$ and $OCR$ ($OCR \geq 1.0$) or $K_o^{SR}$ and $q^{POP}$ are needed to setup the initial value of the hardening parameter $\gamma^{PS}$ (please refer to the report on Hardening Soil model for the detailed explanation as the same procedure is used in this model) |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |

continued on next page

| continued from previous page | |
|---|---|
| Damping | optional (see Win.(2-866)) |

**Remarks**:

- Prior using Densification model it is strongly recommended to study report on Dynamics where a dedicated section is devoted to that model, the theory and benchmarks

Window 2-882

## Related Topics

*Theory:* Technical report on dynamics

## 2.3.3.2.7.22 CONTINUUM: HOEK–BROWN (true)

**Window 2-883:** `Hoek-Brown` **model for** `Continuum`

This model is a 2002 edition of the true Hoek-Brown one. It can be calibrated based on the given GSI index or directly through the $\sigma_{ci}$, $m_b$, $s$, $a$ parameters. In addition the dilatancy law includes the two versions i.e. with constant and with stress dependent dilatancy. For more detailed information on that model please refer to the dedicated report.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional; (see Win.(2-857)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |

continued on next page

**continued from previous page**

| | |
|---|---|
| `Non--Linear` | obligatory;<br>• GSI index, $\{[-],[0 \leq GSI \leq 100],100\}$ (inactive for Direct definition of $m_b$,$s$,$a$ parameters)(based on GSI $E$ modulus can be defined)<br>• Disturbance factor $D$, $\{[-],[0 \leq D \leq 1],0\}$ (to be set only when definition through GSI index is used) (for undisturbed rock use 0.0 and for highly disturbed one use 1.0) (inactive for Direct definition of $m_b$,$s$,$a$ parameters)<br>• Parameter $m_i$ (ratio of compressive to tensile strength for intact rock) $m_i$, $\{[-],[m_i > 1],10.0\}$ (to be set only when definition through GSI index is used) (inactive for Direct definition of $m_b$,$s$,$a$ parameters)<br>• Intact rock compressive strength $\sigma_{ci}$, $\{[kPa],[\sigma_{ci} > 0],15000.0\}$<br>• Parameter $m_b$, $\{[-],[m_b > 0,10\}$ (to be set only when Direct definition is used)<br>• Parameter $s$, $\{[-],[s > 0.0,1.0\}$ (to be set only when Direct definition is used)<br>• Parameter $a$, $\{[-],[0.5 \leq s \leq 0.67,0.5\}$ (to be set only when Direct definition is used)<br>• Constant dilatancy/Stress dependent dilatancy flag (default is constant)<br>• Dilatancy angle for constant dilatancy $\psi$, $\{[deg],[0.0 \leq \psi \leq 90.0,0.0\}$<br>• Dilatancy cut-off stress $\sigma_\psi$ for stress dependent dilatancy, $\{[kPa],[\sigma_\psi \geq 0.0]\}$<br>• Equivalent shear strain $\gamma_r$ at peak on $q-\varepsilon_1$ curve in uniaxial compression test (this value is needed only for strain dependent dilatancy and active hardening/softening)<br>• Equivalent shear strain $\gamma_{res}$ at reasidual state on $q-\varepsilon_1$ curve in uniaxial compression test (this value is needed only for strain dependent dilatancy and active hardening/softening) |

continued on next page

| **continued from previous page** | |
|---|---|
| | • Weighting factors $f_\psi$ (can be negative in pre-failure stage) at given characteristic equivalent shear strain magnitudes ($\gamma_o$, $\gamma_r$, $\gamma_{res}$)<br>• Pre-peak hardening and hardening law type (standard, parabolic and with stiffening effect)<br>• For standard hardening law parameters $m_{bo}$, $s_o$ and $a_o$ are describing the initial position of the yield surface; (it is strongly recommended to assume $a_o = a$)); certain restrictions are put on values of these parameters; for detailed information see dedicated report on HB model<br>• For parabolic law set $r_o$ parameter $\{[-], [r_o > 0, 1\}$ (see dedicated report for details)<br>• For hardening that includes stiffening effect set two pairs $\xi_1 - r_1$ and $\xi_2 - r_2$ of parameters $\{[-], [r_1, r_2 > 0, 1\}$, $\{[-], [\xi_1, \xi_2 > 0, 1\}$, $\xi_1 < \xi_2$, $r_1 < r_2$ (see dedicated report for details)<br>• Post-peak softening flag<br>• Once softening is activated parameters $m_{bres}$, $s_{res}$ and $a_{res}$ are the values of $m_b$, $s$ and $a$ at residual state; they describe residual strength envelope (it is strongly recommended to assume $a_{res} = a$)<br>• Parameter $\alpha_R$ describes the softening law for tensile stress paths controled by the Rankine yield surface<br>• Softening cut-off stress $\sigma_{sf}$: softening will be cancelled once $\sigma_3 > \sigma_{sf}$, $\{[kPa], [\sigma_{sf} \geq 0.0]\}$<br>• Type of softening regularization: Local/Softening scaling; the later one must be used in practical computations; the local one is used only for single element tests; local softening may lead to lack of convergence or mesh-dependent solutions<br>• Parameter $L_c$ - characteristic length required for softening scaling (must be set up based on experimental data) $\{[m], [L_c \geq 0.0]\}$; it is of order of milimiters; decides whether post-peak path in $\sigma - \varepsilon$ curve is brittle or ductile |
| `Humidity` | optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see Win.(2-864)) |
| `Heat` | optional for `Heat`, `Deformation`, `Deformation+Flow`; (see Win.(2-863)) |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-865)) |
| `Damping` | optional (see Win.(2-866)) |

## Related Topics

*Theory:*

- Technical report on HB model

## 2.3.3.2.7.23 CONTINUUM: Plastic damage model for concrete

**Window 2-884:** `Plastic damage` **model for** `Continuum`

Current implementation of plastic damage model for concrete (CPDM) is based on the formulation proposed by Lee and Fenves with certain improvements introduced by the ZSoil developers. This model couples single surface elasto-plasticity with hardening and an enhanced scalar damage allowing for description of stiffness degradation and stiffness recovery in cyclic tension-compression tests. This model is also able to describe aging, creep (according to the EC2 standard) and damage at early stage of concrete maturing. The recent development (since version 2018) includes model extension to high temperatures. All details concerning this model can be found in the report on Plastic damage model for concrete and Concrete structures at high temperatures.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)); this parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures) |
| `Unit Weight` | optional; (see Win.(2-851)) |
| `Flow` | optional; (see Win.(2-853), Win.(2-854) ) |
| `Creep` | optional (two creep models can be used in this model; the first one is the dedicated creep model following the EC2 standard while the second one describes thermal creep during fire; only one of them can be used during the analysis); (see Win.(2-859)) |
| `Initial State`$K_0$ | optional for `Deformation`, `Deformation+Flow`; (see Win.(2-860)) |

continued on next page

**continued from previous page**

| | |
|---|---|
| `Nonlinear` | obligatory;<br>• $\underline{f}_c$ - uniaxial compressive strength (positive in compression); use according to the EC2 standard; these parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures)<br>• $\underline{f}_{co}/\underline{f}_c$ - initial to peak compressive strength ratio (0.4÷0.8)<br>• $\underline{f}_{cbo}/\underline{f}_{co}$ - initial biaxial to uniaxial strength ratio (1.1÷1.2)<br>• $\underline{\sigma}_{c,D}/\underline{f}_c$ - stress level at which damage starts to occur ($\underline{\sigma}_{c,D}/\underline{f}_c \geq \underline{f}_{co}/\underline{f}_c$)(0.4÷0.9); these parameter is internally modified in fire analyses (see the report on reinforced concrete structures subject to elevated temperatures)<br>• $\tilde{\underline{\sigma}}_c/f_c$ - stress level for damage calibration in compression (on post-peak branch)($\leq 1.0$)<br>• $\tilde{D}_c$ - damage parameter in uniaxial compression at the assumed reference stress level ($\geq 0.3$); these parameter can be modified by the user supplied thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures)<br>• $G_c$ - fracture energy in compression ($50\ G_f \leq G_c \leq 100\ G_f$)<br>• $f_t$ - uniaxial tensile strength (positive in tension); use according to the EC2 standard; these parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures)<br>• $\tilde{\sigma}_t/f_t$ - stress level for damage calibration (on post-peak branch) ($< 1$)<br>• $\tilde{D}_t$ - damage parameter in uniaxial tension at the assumed reference stress level ($\geq 0.3$)<br>• $G_t$ - fracture energy in tension; can be estimated as $73\ f_c^{0.18} * 10^{-6}$ (use MPa as strength unit here) ($50*10^{-6} \div 150*10^{-6}$ MN/m)<br>• $s$ - stiffness recovery factor due to crack closure for tension-compression cycles ($0.1 \div 0.3$)<br>• Dilatancy type - $\mathrm{Constant/Variable}$<br>• $\alpha_p$ - Dilatancy parameter value for $\mathrm{Constant}$ option or dilatancy parameter in uniaxial compression when option $\mathrm{Variable}$ is used ($0.1 \div 0.5$)<br>• $\alpha_{po}$ - dilatancy parameter in uniaxial tension option $\mathrm{Variable}$ is used ($0.1 \div 0.4$)<br>• $\underline{\sigma}_{c,dil}/\underline{f}_c$ - activation of dilatancy in compression when $\mathrm{Variable}$ option is used; $\underline{\sigma}_{c,dil}/\underline{f}_c \geq \underline{f}_{co}/\underline{f}_c$; ($0.4 \div 1.0$)<br>• $\alpha_d$ - smoothing factor for plastic potential; value 1.0 is recommended ($0.2 \div 2.0$) |

continued on next page

| continued from previous page | |
|---|---|
| Nonlinear | obligatory;<br>• Char. length (RC) flag - enforced characteristic length flag for reinforced concrete (ON/OFF)<br>• $l_c^{RC}$ - characteristic length to be used for reinforced concrete when Char. length (RC) flag is ON ($> 0$)<br>• Secant stiffness flag - use secant elastic stiffness ($\mathbf{D}^e(1-D)$) when ON or consistent tangent when OFF (secant stiffness is recommended for cases with poor convergence) |
| Humidity | optional for Humidity, Deformation, Deformation+Flow; (see Win.(2-864)) |
| Heat | optional for Heat, Deformation, Deformation+Flow; (see Win.(2-863)) parameters $\lambda$, $c^*$ and $\alpha$ can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures) |
| Stability | optional for Deformation, Deformation+Flow; (see Win.(2-865)) |
| Damping | optional (see Win.(2-866)) |

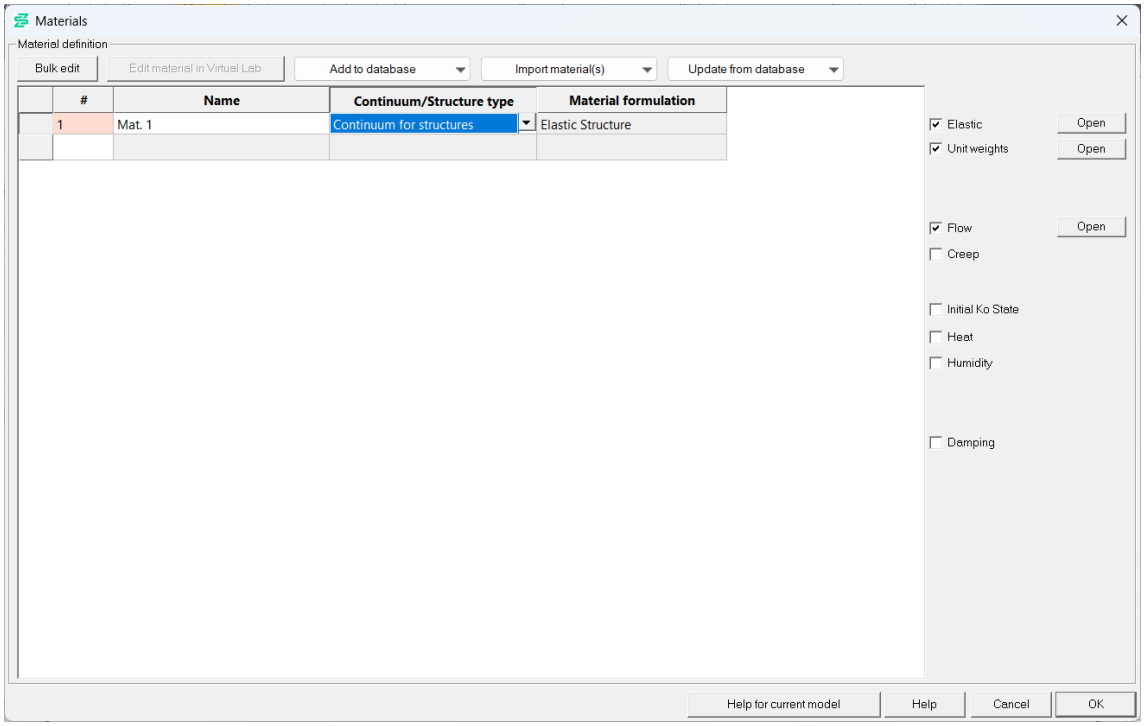


## Related Topics

*Reports:*
Technical report on plastic damage model for concrete
Technical report on concrete structures subject to elevated temperatures

## 2.3.3.2.8 CONTINUUM FOR STRUCTURES

**Window 2-885: Models for** `Continuum for structures`




| Material models for continuum for structures | Data groups |
|---|---|
| linear: `Elastic` | **ELASTIC** |
| | **UNIT WEIGHTS** |
| | **CREEP** |
| | **FLOW** |
| | **INITIAL K0 STATE** |
| | **HEAT** |
| | **HUMIDITY** |
| | **DAMPING** |

**Remarks**:

1. List of material models applicable to continuum for structures (beams, shells) is limited only to the linear elastic model
2. **This material cannot be attached to the triangular continuum elements**
3. All continuum finite elements declared as continuum for structures will be converted to EAS elements with enhanced bending and shear representation (EAS-7 in plane-strain, EAS-5 in axisymmetry and EAS-9 in 3D)
4. **Use more than 1 element across the thickness of beam/shell (to enable very accurate super-convergent patch recovery of stresses and stress resultants)**

Window 2-885

## 2.3.3.2.9 HEAT CONVECTION

**Window 2-886:** `Heat convection` **model**

This model is applicable exclusively to `Heat Problem Type`It is used to describe convective properties of the boundary and surrounding media (with ambient temperature), in order to control heat exchange through the boundary. It may exclusively be attached to `Heat Convection` elements.

| Data group | Parameters |
|---|---|
| `Heat` | obligatory<br>• Heat convection coefficient $h_T$ $\{[\text{kN}/(\text{m}\cdot{}^\circ\text{K}\cdot\text{s})], > 0,10\}$ (see the report on reinforced concrete structures subject to elevated temperatures to set up proper value of this parameter in case of fire analyses) |

Window 2-886

## Related Topics

*Theory:* HEAT TRANSFER
*Reports:* Technical report on concrete structures subject to elevated temperatures

## 2.3.3.2.10 HEAT RADIATION

**Window 2-887:** `Heat radiation` **model**

This model is applicable exclusively to `Heat Problem Type`It is used to describe radiation boundary condition during fire. It may exclusively be attached to `Heat Radiation` elements.

| Data group | Parameters |
|---|---|
| `Heat` | obligatory<br>• configuration factor $\phi$ $\{[-], > 0, 1\}$<br>• surface emissivity of the member $\varepsilon_m$ $\{[-], > 0, 0.8\}$<br>• emissivity of the fire $\varepsilon_f$ $\{[-], > 0, 1.0\}$<br>• Boltzmann constant $\sigma$ $\{[kN/m^3/s/C^4], > 0, 5.67e-11\}$<br>(see the report on reinforced concrete structures subject to elevated temperatures for more detailed information) |

Window 2-887

### Related Topics

*Theory:*

HEAT TRANSFER

*Reports:*

Technical report on concrete structures subject to elevated temperatures

## 2.3.3.2.11 HUMIDITY CONVECTION

**Window 2-888:** `Humidity convection` **model**

The model is applicable exclusively to `Humidity Problem Type` [9].

It is used to describe convective properties of the boundary and surrounding media (with ambient relative humidity), in order to control humidity exchange through the boundary. It may be attached to `Humidity Convection` elements exclusively.

| Data group | Parameters |
|---|---|
| `Humidity` | obligatory;<br>• Humidity convection coefficient $h_W$ $\{[1/(\mathrm{m}^2 \cdot \mathrm{d})], > 0, 10\}$ |

Window 2-888

[9] concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT** only

## 2.3.3.2.12 INFINITE MEDIA

**Window 2-889:** `Infinite media` **model**

Infinite medium can be represented exclusively by a linear elastic material.

| Data group | Parameters |
|---|---|
| `Elastic` | obligatory; (see Win.(2-850)) |

Window 2-889

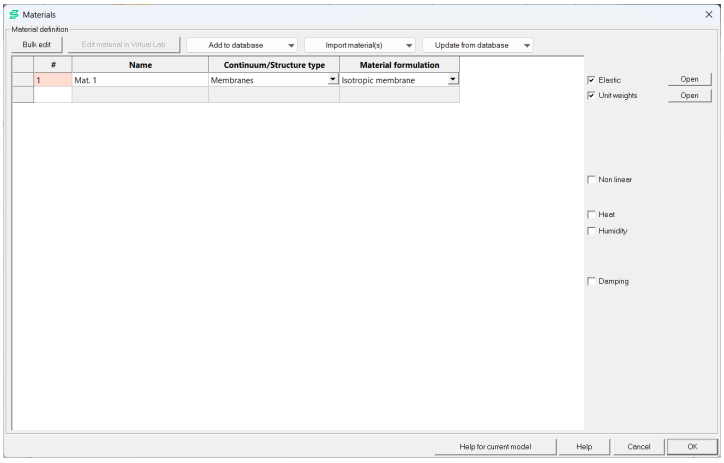
### Related Topics

*Theory:*

FAR FIELD

## 2.3.3.2.13 MEMBRANES

**Window 2-890: Models for** `Membranes`




**Fiber model**

**Plane stress member models**

**Elastic**

**Hoek-Brown**

**Huber-Mises**

**Rankine**

**Fabric models**

**Anisotropic**

**Isotropic**

Window 2-890

## 2.3.3.2.13.1 FIBER MODEL

**Window 2-891: Fiber model for** `Membranes`

This model represents a layer of equally spaced reinforcement bars.

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory;<br>Young's elastic modulus, $E$ $\{[\text{kPa}];[>0];100000\}$ |
| `Unit Weight` | optional;<br>• $\gamma$, $\{[\text{kNm}^{-2}];[>0];0\}$ - unit weight of fiber material<br>• Density $\rho = \frac{\gamma}{g}$ is automatically computed from $\gamma$ (and vice versa)<br>• Multiplier for $\gamma$ $\{[-];[\geq 0];1.0\}$<br>• Multiplier for $\rho$ $\{[-];[\geq 0];1.0\}$<br>Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor) while setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if `Dynamics` or `Pushover` is set ON in the project preselection). |
| `Creep` | not supported |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared (if $\alpha > 0$) thermal strains will be computed. |
| `Humidity` | optional;<br>$\beta$, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared (if $\beta > 0$) hygral strains will be computed. |
| `Non--Linear` | optional<br>• Uniaxial tensile strength $f_t$, $\{[\text{kPa}];[f_t > 0];1\}$<br>• Uniaxial compressive strength $f_c$, $\{[\text{kPa}];[f_c \geq 0];1\}$ |
| `Geometry` | obligatory<br>• For `Plane Strain`:<br>★ Area per unit length $A$, $\{[\text{m}^2/\text{m}{=}\text{m}],[>0],[1]\}$<br>• For `Axisymmetry`:<br>★ Direction: one of: ◎ **Circumferential** or ◎ **Longitudinal**<br>★ Area per unit length $A$, $\{[\text{m}^2/\text{m}{=}\text{m}],[>0],[1]\}$<br>• For 3D:<br>★ Set $X$, $Y$, $Z$ being the coordinates of an arbitrary direction vector in 3D space indicating an orientation of fibers in 3D space;<br>★ Area per unit length $A$, $\{[\text{m}^2/\text{m}{=}\text{m}],[>0],[1]\}$ |
| `Damping` | optional (same definition as in the global *Control/Dynamics dialog*) |

**Remarks**:

In 3D models an error will be reported when `Direction` vector is orthogonal to the membrane element surface at any integration point

Window 2-891

## Related Topics

*Theory:*

- MEMBRANE ELEMENTS
- UNI-AXIAL ELASTO-PLASTIC MODEL
- SETTING THE DIRECTION ON SURFACE ELEMENTS
- SETTING OF THE LOCAL BASE ON A SURFACE ELEMENT

## 2.3.3.2.13.2 PLANE STRESS MODELS

**Window 2-892: Plane stress models for** `Membranes`

The following models are used to represent membrane behavior under the plane-stress conditions:

**Elastic**
**Hoek-Brown (M-W)**
**Huber-Mises**
**Rankine (M-W)**

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory (same as for continuum); see Win.(2-850) |
| `Unit Weight` | optional;<br>• $\gamma$, $\{[\text{kNm}^{-3}];[>0];0\}$ - unit weight<br>• Density $\rho = \frac{\gamma}{g}$ is automatically computed from $\gamma$ (and vice versa)<br>• Multiplier for $\gamma$ $\{[-];[\geq 0];1.0\}$<br>• Multiplier for $\rho$ $\{[-];[\geq 0];1.0\}$<br>Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor) while setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if `Dynamics` or `Pushover` is set ON in the project preselection). |
| `Geometry` | obligatory;<br>Thickness $t$, $\{[\text{m}];[>0];0.1\}$ |
| `Creep` | not supported |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared thermal strains will be computed (if $\beta > 0$). |
| `Humidity` | optional;<br>$\beta$, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared hygral strains will be computed (if $\beta > 0$). |
| `Non--Linear` | set:<br>• For `Huber--Mises` and `Rankine (M--W)` (obligatory):<br>⋆ Uniaxial tensile strength $f_t$, $\{[\text{kPa}];[\geq 0];1\}$<br>• For `Hoek--Brown (M--W)` (obligatory):<br>⋆ Uniaxial tensile strength $f_t$, $\{[\text{kPa}];[\geq 0];1\}$<br>⋆ Uniaxial compressive strength $f_c$, $\{[\text{kPa}];[f_c > f_t \geq 0];1\}$ |
| `Damping` | optional (same definition as in the global *Control/Dynamics dialog*) |

Window 2-892

## Related Topics

*Theory:* MEMBRANES THEORY

## 2.3.3.2.13.3 MEMBRANE - FABRIC MODELS

**Window 2-893: Fabric models for** `Membranes`

The two following models are supported: **Anisotropic** and **Isotropic**

These models can represent different types of geo-textiles, geo-grids and similar products. The important point is that thickness parameter is included within the definition of stiffness and strength parameters. Both isotropic and anisotropic models can be used.

`Anisotropic` and `Isotropic` membrane models: common data.

| Data groups | Parameters |
|---|---|
| `Unit Weight` | optional; (see Win.(2-891)) |
| `Creep` | not supported |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared thermal strains will be computed (if $\beta > 0$). |
| `Humidity` | optional;<br>$\beta$, $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared hygral strains will be computed (if $\beta > 0$). |
| `Damping` | optional (same definition as in the global *Control/Dynamics dialog*) |

`Isotropic` membrane: specific data.

| Data | Use & Set up |
|---|---|
| `Elastic` | obligatory;<br>• Elastic stiffness $K$ (equivalent to $E\times$thickness) $\{[\text{kN/m}]; [> 0]; 100000\}$,<br>• Poisson ratio $\nu$, $\{[-]; [0, 0.49999]; 0.3\}$<br>Resulting elasticity matrix for the plane stress state will take form:<br>$$\mathbf{D}_{(3x3)} = \begin{bmatrix} K & \nu K & 0 \\ \nu K & K & 0 \\ 0 & 0 & \dfrac{K}{2(1+\nu)} \end{bmatrix},$$ |
| `Non--Linear` | optional<br>If not activated elastic membrane with unlimited strength will be considerd<br>• Uniaxial tensile strength $f_t$, $\{[\text{kN/m}]; [> 0]; 1\}$<br>• Uniaxial compressive strength $f_c$, $\{[\text{kN/m}]; [> 0]; 1\}$ |

`Anisotropic` membrane: specific data.

| Data | Use & Set up |
|---|---|
| `Elastic` | obligatory;<br>• Elastic stiffness in anisotropy direction $K_{XX}$, $K_{YY}$, $K_{XY}$ $\{[\text{kN/m}]; [> 0]; 100000\}$,<br>Resulting elasticity matrix for the plane stress state will take form:<br>$$\mathbf{D}_{(3x3)} = \begin{bmatrix} K_{xx} & K_{xy} & 0 \\ K_{xy} & K_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix},$$<br>i.e. no shear stiffness is assumed in the model. |
| `Geometry` | obligatory<br>$X$, $Y$, $Z$ the coordinates of an arbitrary direction vector in 3D space. It determines anisotropy axis $X$ |
| `Non--Linear` | optional<br>If not activated elastic membrane with unlimited strength will be considered<br>• Uniaxial tensile strength in anisotropy direction X,Y: $f_tX$, $f_tY$, $\{[\text{kN/m}]; [\geq 0]; 1\}$<br>• Uniaxial compressive strength in anisotropy direction X,Y: $f_cX$, $f_cY$, $\{[\text{kN/m}]; [\geq 0]; 1\}$ |

**Remarks**:

1. As a particular case no-tension material may be analyzed if $f_c = 0$ (isotropic case) or $f_{cX} = f_{cY} = 0$ (anisotropic case)
2. No shear stiffness is assumed in the anisotropic model

Window 2-893

## Related Topics

*Theory:*

MEMBRANES THEORY

## 2.3.3.2.14 SEEPAGE

**Window 2-894:** Seepage **model**

This fictitious material model is associated with seepage surface elements simulating an automatic switch from zero flux to pressure boundary condition (useful to model drains or filling/drawdown effects). Behavior of this artificial interface is driven by a single parameter $K_v$. Its reference value for the isotropic case is equal to $K_v^{ref} = \frac{k}{\gamma^F \, h^e}$ where $k$ is the seepage coefficient and $h^e$ is the continuum element size adjacent to the seepage surface element. In the standard case $K_v = K_v^{ref} \, \epsilon$ where $\epsilon$ is a large number (penalty method is used), but there are also two other possibilities (since version 2018) to set this value. User may set direct value of $K_v$ parameter or may set his own multiplier to the $K_v^{ref}$.

| Data | Use & Set up |
|---|---|
| `Flow` | obligatory;<br>• `Multiplier` to $K_v^{ref}$ $\epsilon$, $\{[-],[>0],1\}$<br>• Direct value of $K_v$ $\{[m^3/(kN\ s)],[>0],1\}$<br>• `Multiplier` to $K_v^{ref}$, $\{[-],[>0],1\}$<br>Use standard option with multiplier 1.0 unless computed pore pressure fields do not satisfy the assumed pressure boundary conditions with required order of accuracy; in such cases try to enlarge the `Multiplier`; this may however lead to convergence problems. In special cases when modeling drains with limited influx use second (direct value of $K_v$) or third setting (user given multiplier to the reference value $K_v^{ref}$) |

**Remark**:

If continuum elements adjacent to seepage surface have impermeable material (group ☒ **Flow** is set OFF), the error, caused by singularity of the resulting system of equations, will be reported.

Window 2-894

## Related Topics

*Theory:*

- TWO-PHASE PARTIALLY SATURATED MEDIUM

## 2.3.3.2.15 VISCOUS DAMPERS

**Window 2-895:** `Viscous dampers` **model**

This fictitious material model is attached to viscous damping elements that prevent wave reflections from domain boudaries. In most practical cases this data should be equivalent with the one set up for adjacent continuum elements.

By default the flag ☒ **Inherit properties from adjacent continuum** is set ON. However, user may set up the data himself in the same way as for the standard continuum.

Window 2-895

### Related Topics

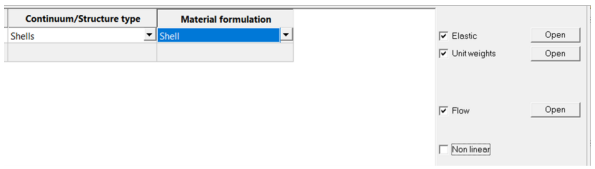
*Theory:*

- ABSORBING BOUNDARIES-SINGLE PHASE
- ABSORBING BOUNDARIES-TWO PHASE

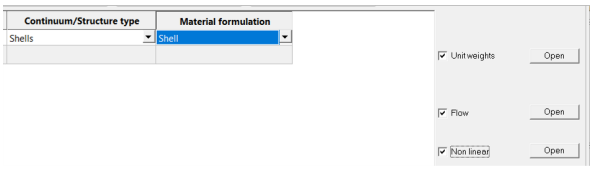
## 2.3.3.2.16 MODEL FOR SHELLS

**Window 2-896: Linear isotropic vs Nonlinear/Layered `Shell` models**

The elastic isotropic or nonlinear (with isotropic elastic kernel) shell model can be used by selecting, at the material level (under `Material formulation`) the `Shell` option. Standard isotropic elastic or nonlinear/layered shell models can be choosen by switching ON/OFF the ☒ **Nonlinear** check-box.

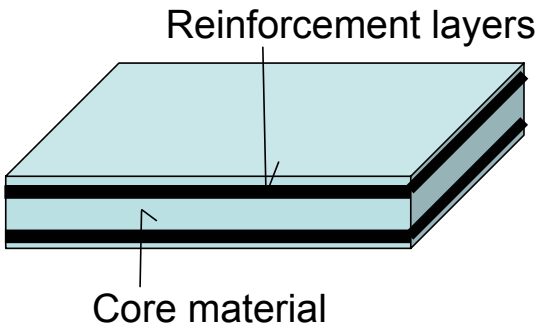



Linear model




Nonlinear/Layered model






**Remarks**:

1. Existing two classes of shell elements i.e.: shell with 2 layers of nodes and shell with 1 layer of nodes use the same material model for description of mechanical (solid phase) behavior; the major difference concerns flow modeling, which is not supported by the shell with 1 layer of nodes; in this case flow can be modeled by means of interface elements adjacent to shell elements
2. Data specific to linear (elastic) shell model is specified in Window 2-897
3. Data specific to nonlinear/layered shell model is specified in Window 2-898
4. Special case of elastic orthotropic shell is described in Win.2-901

Window 2-896

## Window 2-897: Data for linear(elastic) Shell model

Data for linear (elastic) shell model

| Data groups | Parameters |
|---|---|
| Elastic | obligatory (same as for continuum); see Win.(2-850) |
| Unit Weight | optional (same as for continuum); see Win.(2-851) |
| Flow | optional (same as for contact interface); see Win.(2-836) |
| Creep | not supported (switch to nonlinear/layered shell model to activate it) |
| Heat | optional;<br>$\alpha$, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed Heat Transfer project is declared thermal strains will be computed (if $\beta > 0$). |
| Humidity | optional;<br>$\beta$, $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed Humidity Transfer project is declared hygral strains will be computed (if $\beta > 0$). |
| Damping | optional (same definition as in the global *Control/Dynamics dialog)* |

Window 2-897

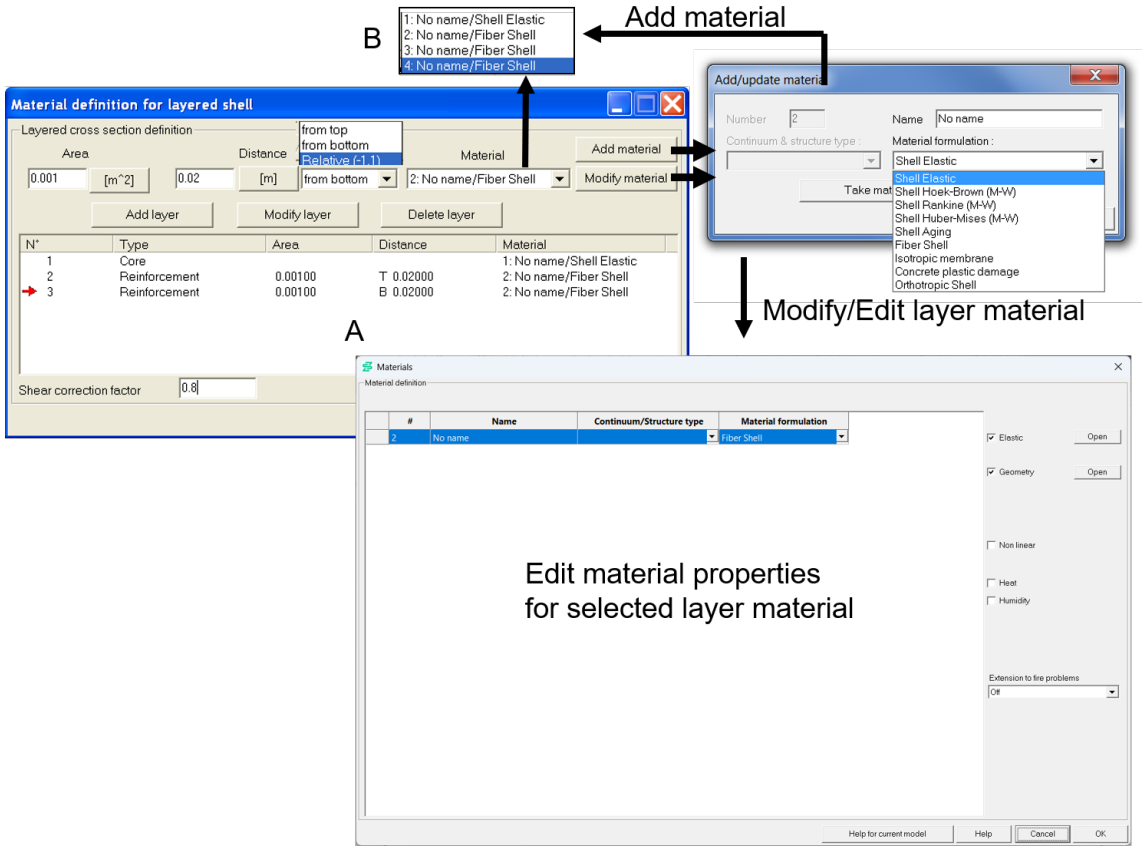
## Window 2-898: Data for nonlinear/layered Shell model

The following data groups are defined at the global level (for the whole cross section)

| Data groups | Parameters |
|---|---|
| Unit Weight | optional (same as for continuum); see Win.(2-851) |
| Flow | optional (same as for contact interface); see Win.(2-836) |
| Non--Linear | obligatory; definition of layered cross section; see Win.(2-899) |
| Damping | optional (same definition as in the global *Control/Dynamics dialog)* |

Window 2-898

## Window 2-899: Materials for layers in nonlinear/layered `Shell` model




To define the layered cross section:

1. Create a temporary data bank (list (B) in the figure) of material models to be used for definition of core and/or reinforcement sets;
   A: With button **Add** add a new material model which can be one of:
   - Elastic (shell) (for **core** material)
   - Mohr–Coulomb (shell) (for **core** material)
   - Hoek-Brown (shell) (for **core** material)
   - Huber-Mises(shell) (for **core** material)
   - Aging concrete(shell) (for **core** material)
   - Plastic damage for concrete (shell) (for **core** material)
   - Fiber (shell) (for **reinforcement**)

   B: Edit its properties (Window ...)

   C: With button **Modify** update, from list (B), selected material model and its properties, if needed
2. For the reinforcement sets use buttons **Add layer** or **Modify layer** to add or update (selected from list A) a layer characterized by the area given in the edit field **Area** [ ], distance of the reinforcement, given in the edit field **Distance** [ ], and measured
   - ◎ **from top** (top shell fibers are indicated by positive $Z$-direction of shell element)
   - ◎ **from bottom** (bottom shell fibers are indicated by negative $Z$-direction of shell element)
   - given as ◎ **relative** value $\{[-], -1 \leq d \leq 1, 0\}$ (+1 if for top, -1 at the bottom and 0 and midsurface)

**Remarks**:

1. The layered shell material consists of a **single** core material (cannot be disabled) and any number of reinforcement sets (optional)
2. Elastic, creep, nonlinear, thermal and hygral data are defined at each layer material level;
3. Number of discretization layers in core material is fixed and equal to 10
4. In any other layered shell model materials temporarily stored in list B will be available
5. Equivalent shear stiffness modulus $G$ (as bending is decoupled from shear) is computed based on elastic properties $E$ and $\nu$ of core material; modification of this stiffness modulus can be easily made with aid of `Shear correction factor` which multiplies $G$ value

Window 2-899

## Window 2-900: Material data for shell layers

Material data for core material models: `Elastic` (shell), `Mohr--Coulomb (M--W)` (shell), `Hoek--Brown (M--W)` (shell), `Plastic damage (concrete)` (shell), `Huber--Mises` (shell), `Aging Concrete` (shell).

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory;<br>• Young's elastic modulus $E\{[\text{kPa}];[>0];100000\}$<br>• Poisons ratio $\nu\{[-];[0,0.49999];0.3\}$, |
| `Creep` | optional (same as for continuum) ; see Win.(2-857) |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared thermal strains will be computed (if $\beta > 0$). |
| `Humidity` | optional;<br>$\beta$, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared hygral strains will be computed (if $\beta > 0$). |
| `Non--Linear` | • `Elastic`, `Mohr--Coulomb (M--W)`, `Hoek--Brown (M--W)`, `Aging Concrete` shells, `Plastic damage (concrete)` shells – obligatory set data as for continuum models<br>• `Huber--Mises` shell – obligatory; set:<br>⋆ uniaxial tensile strength $f_t$, $\{[\text{kPa}];[>0];1\}$ |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; see Win.(2-865) |

Material data for reinforcement material model: `Fiber` (shell)

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory;<br>Young's elastic modulus, $E$ $\{[\text{kPa}];[>0];100000\}$<br>these parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see dedicated report on reinforced concrete structures subject to elevated temperatures) |
| `Creep` | not supported |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared (if $\alpha > 0$) thermal strains will be computed; these parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see dedicated report on reinforced concrete structures subject to elevated temperatures) |
| `Humidity` | optional;<br>$\beta$, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared (if $\beta > 0$) hygral strains will be computed. |
| `Non--Linear` | optional;<br>• Uniaxial tensile strength $f_t$, $\{[\text{kPa}];[f_t > 0];1\}$<br>• Uniaxial compressive strength $f_c$, $\{[\text{kPa}];[f_c \geq 0];1\}$<br>these parameters can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see dedicated report on reinforced concrete structures subject to elevated temperatures) |
| `Geometry` | obligatory;<br>• Set $X$, $Y$, $Z$ being the coordinates of an arbitrary direction vector in 3D space indicating an orientation of fibers in 3D space; |
| `Stability` | optional for `Deformation`, `Deformation+Flow` |

**Remarks**:

1. Fiber direction is the closest to given direction among all directions tangent to the element surface
2. In the case when direction is tangent to element surface, fiber direction will coincide with direction
3. An error will be reported during computation in the case when direction is orthogonal to element surface
4. The `Plastic damage (concrete)` model is the only one that can be used to model

concrete shell core layers subject to high temperatures (fire analyses); see the dedicated report on concrete structures at high temperatures for more details

5. Shell fiber model can be used in fire analyses; thermal evolution functions for stiffness and strength parameters are taken from the EC 2; see the dedicated report on concrete structures at high temperatures for more details

Window 2-900

## Related Topics

*Theory:*

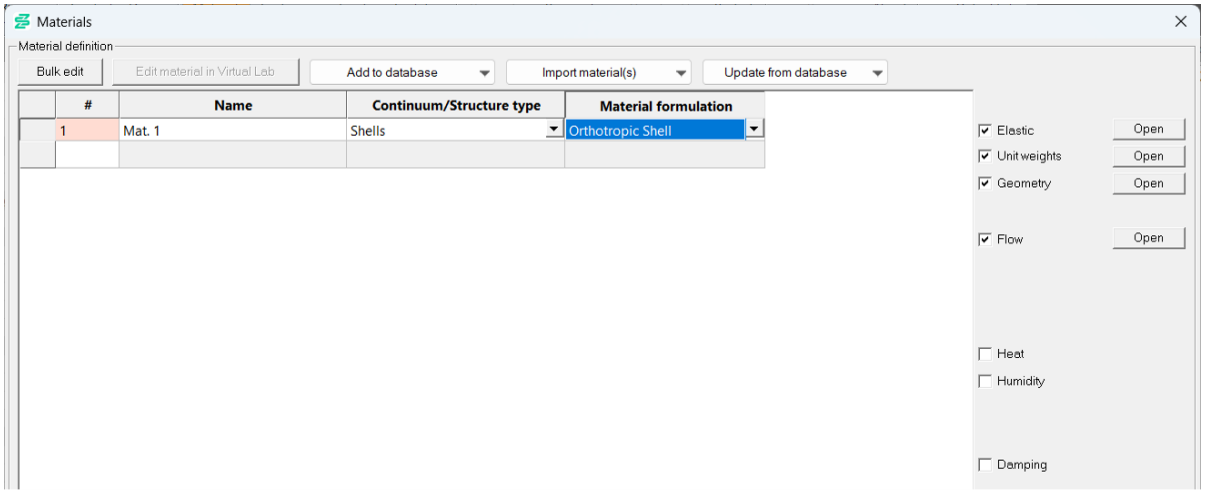
- SHELLS THEORY
- UNI-AXIAL ELASTO-PLASTIC MODEL
- SETTING THE DIRECTION ON SURFACE ELEMENTS
- SETTING OF THE LOCAL BASE ON A SURFACE ELEMENT

*Reports:*

Technical report on concrete structures subject to elevated temperatures

### Window 2-901: Linear orthotropic `Shell` model

The orthotropic elastic shell model can be used by selecting, at the material level (under `Material formulation`) the `Orthotropic shell model`.




**Remarks**:

1. Four elastic constants must be defined in group `Elastic` $E_1$, $E_2$, $G_o$, $\nu_1$; the second Poisson ratio $\nu_2 = \nu_1 \dfrac{E_2}{E_1}$
2. In addition the orientation vector of the first orthotropy direction (corresponding to $E_1$) must be indicated by the vector in the global coordinate system under group `Geometry`; this vector must not be perpendicular to any shell element with orthotropic material

Window 2-901

## Window 2-902: Data for linear(elastic) orthotropic `Shell` model

Data for linear (elastic) orthotropic shell model

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory (same as for continuum);<br>• $E_1$ - Young modulus in the first orthotropy direction<br>• $E_2$ - Young modulus in the second ortothropy direction<br>• $G_o$ - Shear modulus<br>• $\nu_1$ - Poisson ratio in the first ortothropy direction<br><br>NB. $\nu_2 = \nu_1 \frac{E_2}{E_1}$ |
| `Unit Weight` | optional (same as for continuum); see Win.(2-851) |
| `Flow` | optional (same as for contact interface); see Win.(2-836) |
| `Creep` | not supported |
| `Heat` | optional;<br>$\alpha$, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient<br>In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared thermal strains will be computed (if $\beta > 0$). |
| `Humidity` | optional;<br>$\beta$, $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient;<br>In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared hygral strains will be computed (if $\beta > 0$). |
| `Damping` | optional (same definition as in the global *Control/Dynamics dialog)* |

Window 2-902

## 2.3.3.2.17 TRUSS (ANCHOR) MODEL

**Window 2-903: Data for truss model**

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory;<br>• Young's elastic modulus, $E$ $\{[\text{kPa}];[>0];100000\}$,<br>this parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures) |
| `Unit Weight` | optional;<br>• $\gamma$, $\{[\text{kNm}^{-3}];[>0];0\}$ - unit weight<br>• Density $\rho = \frac{\gamma}{g}$ is automatically computed from $\gamma$ (and vice versa)<br>• Multiplier for $\gamma$ $\{[-];[\geq 0];1.0\}$<br>• Multiplier for $\rho$ $\{[-];[\geq 0];1.0\}$<br>Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor) while setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if `Dynamics` or `Pushover` is set ON in the project preselection). |
| `Non--Linear` | optional;<br>• Uniaxial tensile strength $f_t\{[\text{kPa}],>0,1\}$<br>• Uniaxial compressive strength $f_c\{[\text{kPa}],>0,1\}$<br>these parameters can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see the report on reinforced concrete structures subject to elevated temperatures) |
| `Geometry` | obligatory;<br>• For `Plane Strain` :<br>⋆ Cross section area $A$, $\{[\text{m}^2];[>0];1\}$<br>⋆ Interval between anchors $a$, $\{[\text{m}];[>0];1\}$<br>• For `3D` :<br>⋆ Cross section area $A$, $\{[\text{m}^2];[>0];1\}$<br>• For `Axisymmetry` :<br>⋆ Cross section area $A$, $\{[\text{m}^2];[>0];1\}$<br>⋆ Cross section area definition: one of<br>◎ **Total along whole circumference** or<br>◎ **Per unit length of circumference** |
| `Stability` | optional for `Deformation`, `Deformation+Flow`; see Win.(2-865) |
| `Damping` | optional (same definition as in the global *Control/Dynamics dialog*) |

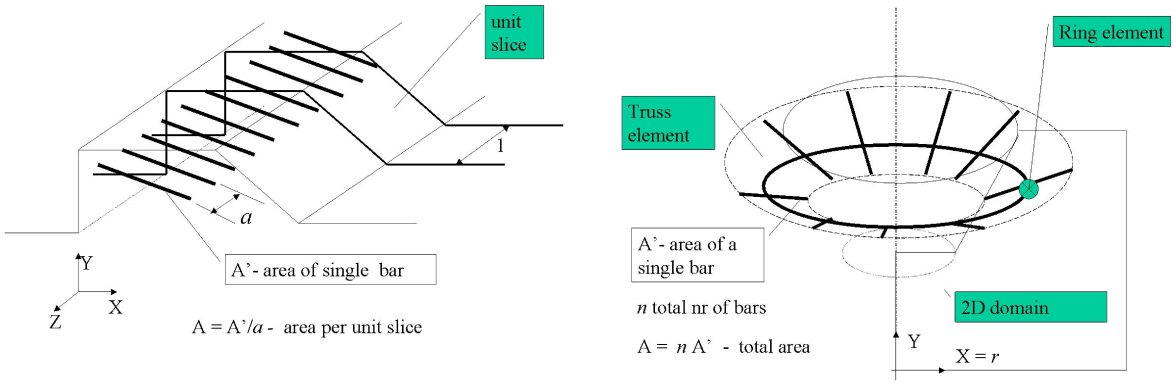
**Remarks**:

1. Perfect elasto-plastic model is used

## Related Topics

*Reports:* Technical report on concrete structures subject to elevated temperatures

**Window 2-904: Interpretation of truss interval and cross section area**




Plane strain

Axisymmetry

Window 2-904

## Related Topics

- *theory: TRUSSES*
- *theory: UNI–AXIAL ELASTO–PLASTIC MODEL*

## 2.3.3.2.18 BEAM NONLINEAR HINGES

**Window 2-905: Data for uncoupled axial hinge model**

| Data groups | Parameters |
|---|---|
| `Non--Linear` | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - if this checkbox is set ON then behavior in tension and compression will be the same (by means of absolute values)<br>• List of pairs: $\{u_i^+, F(u_i^+)\}$<br>• List of pairs: $\{u_i^-, F(u_i^-)\}$<br>(to be defined when ☒ **Common for positive/negative branch** is set ON) |

Window 2-905

**Window 2-906: Data for uncoupled tangent hinge model**

| Data groups | Parameters |
|---|---|
| `Non--Linear` | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - this setting must be ON<br>• List of pairs: $\{u_i, F(u_i)\}$ |

Window 2-906

## Window 2-907: Data for uncoupled flexural hinge model

| Data groups | Parameters |
|---|---|
| Non--Linear | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - if this checkbox is set ON then behavior in tension and compression will be the same (by means of absolute values)<br>• List of pairs: $\{\phi_i^+, M(\phi_i^+)\}$<br>• List of pairs: $\{\phi_i^-, M(\phi_i^-)\}$<br>(to be defined when ☒ **Common for positive/negative branch** is set ON) |

Window 2-907

## Window 2-908: Data for uncoupled torsional hinge model

| Data groups | Parameters |
|---|---|
| Non--Linear | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - this setting must be ON<br>• List of pairs: $\{\phi_{x,i}, M_x(\phi_{x,i})\}$ |

Window 2-908

## Window 2-909: Data for coupled flexural hinge model

| Data groups | Parameters |
|---|---|
| `Elastic` | obligatory;<br>• Young's elastic modulus, $E$ {[kPa]; [$> 0$]; 100000}<br>• Poisson ratio, $\nu$ {[-]; [$0.0 \leq \nu < 0.5$; 0.3}} |
| `Geometry` | obligatory;<br>• Joint depth (joint dimension in plane of bending) $h${[m], $> 0, 0.2$}<br>• Joint width (joint dimension in plane perpendicular to the plane of bending) $b${[m], $> 0, 1.0$} |
| `Main` | obligatory;<br>• ◎ **Janssen type** - so far Janssen theory is enforced |

Window 2-909

- *theory: NONLINEAR BEAM HINGES*

## 2.3.3.2.19 SHELL NONLINEAR HINGES

**Window 2-910: Data for uncoupled membrane hinge model**

| Data groups | Parameters |
|---|---|
| `Non--Linear` | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - if this checkbox is set ON then behavior in tension and compression will be the same (by means of absolute values)<br>• List of pairs: $\{u_i^+, n(u_i^+)\}$<br>• List of pairs: $\{u_i^-, n(u_i^-)\}$<br>(to be defined when ☒ **Common for positive/negative branch** is set ON) |

Window 2-910

**Window 2-911: Data for uncoupled tangent hinge model**

| Data groups | Parameters |
|---|---|
| `Non--Linear` | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - this setting must be ON<br>• List of pairs: $\{u_i, t(u_i)\}$ |

Window 2-911

## Window 2-912: Data for uncoupled flexural hinge model

| Data groups | Parameters |
|---|---|
| Non--Linear | obligatory;<br>• ◎ **Nonlinear elastic** - if this flag is set ON then loading/unloading-reloading paths will follow the same curve<br>• ◎ **Damage type unloading/reloading** - if this flag is set ON then unloading will is linear elastic with secant stiffness modulus, the same behavior will be reproduced during reloading process<br>• ☒ **Common for positive/negative branch** - if this checkbox is set ON then behavior in tension and compression will be the same (by means of absolute values)<br>• List of pairs: $\{\phi_i^+, m(\phi_i^+)\}$<br>• List of pairs: $\{\phi_i^-, m(\phi_i^-)\}$<br>(to be defined when ☒ **Common for positive/negative branch** is set ON) |

Window 2-912

## Window 2-913: Data for coupled flexural hinge model

| Data groups | Parameters |
|---|---|
| Elastic | obligatory;<br>• Young's elastic modulus, $E$ $\{[\text{kPa}]; [> 0]; 100000\}$<br>• Poisson ratio, $\nu$ $\{[\text{-}]; [0.0 \leq \nu < 0.5; 0.3\}$ |
| Geometry | obligatory;<br>• Joint depth (joint dimension in plane of bending) $h\{[\text{m}], > 0, 0.2\}$<br>• Joint width (fixed to 1.0m) |
| Main | obligatory;<br>• ◎ **Janssen type** - so far Janssen theory is enforced |

Window 2-913

- *theory: NONLINEAR SHELL HINGES*

### 2.3.3.3 NON-STANDARD MATERIAL DATA

There are two major extensions of standard definition (i.e. constant valued) of material parameters:

- Explicit dependence on time
- Explicit dependence on position in space

These extensions are available only for some selected data parameters like elastic or strength parameters.

In general, as first two extensions can be simultaneously used, a value $V$ of the material parameter at given point $\mathbf{x}$ (taken at the element center) and at given time instance $t$ can be set as:

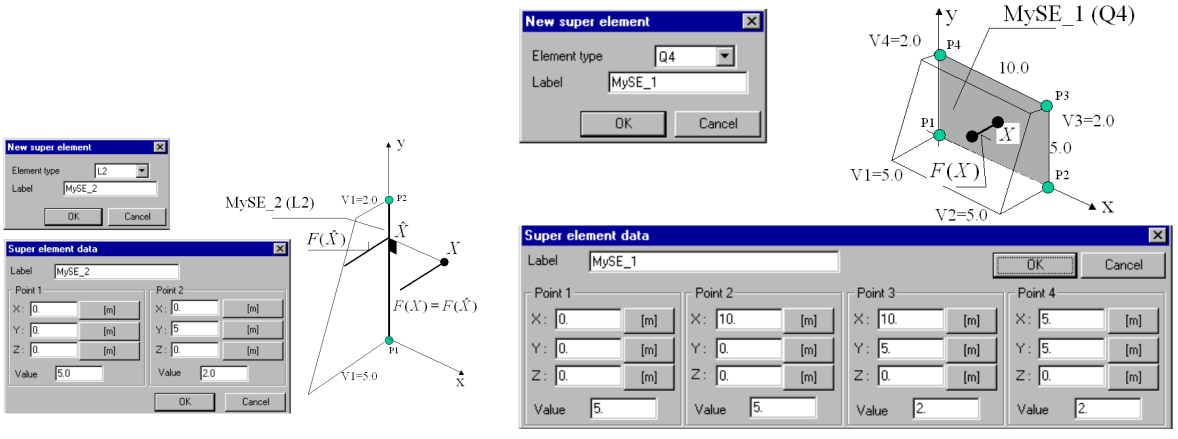
$$V(\mathbf{x},t) = V_{BASE} \times LTF(t) \times F(\mathbf{x}).$$

with $F(\mathbf{x})$ being an interpolation function generated by `Data super elements` and $LTF$ being a `load time function`.

In addition there is an evolution function mechanism available for selected parameters from `Heat` group of properties ($\lambda$, $c^*$, $\alpha$). These parameters can be assumed by the user as temperature dependent via evolution functions. This data mode can only be combined with explicit dependence on position in space option.

The same mechanism is used to analyze fire problems by means of uncoupled thermo-mechanical analysis. The plastic damage model for concrete (in continuum and shell elements only) and 1D elasto-plastic model for truss elements and shell fibers, used to model reinforcement, may have some parameters defined as dependent on temperature. However the corresponding evolution functions are frozen and user cannot edit them directly (see dedicated report on concrete structures at high temperatures for more details).

**Window 2-914: Setting data super-elements**




L2 super-element Q4 super-element

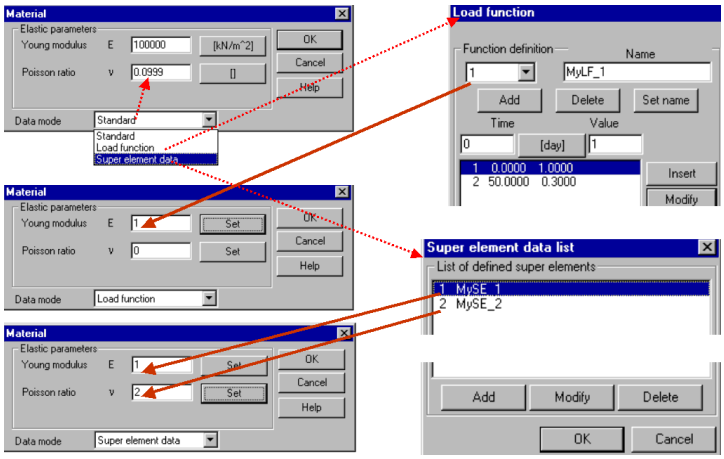
**Remarks**:

1. The super-element nodal data $f_i$, $i = 1, Nen$, $Nen = 2$ or $4$, $(8\ in\ 3D)$ and coordinates $\mathbf{x}_i$ are used to generate an interpolation function $F(\mathbf{x}) = \sum_{i=1}^{Nen} f_i N_i(\mathbf{x})$, where $N_i(\mathbf{x})$ are shape functions related to super element nodes

2. In case of a 4-node (quad) or 8-node (brick)(3D) super-element, the interpolation will be performed only for elements with central point placed inside the super element; for points being outside the super-element, base values of the material data are preserved
3. In case of L2 super-element, center of the finite element is orthogonally projected on the defined line segment, and value of the data multiplier is linearly interpolated based on values defined at segment vertices

Window 2-914

**Window 2-915: Activating nonstandard material data definition**




To set time and space dependent material properties follow the steps:

1. Open the dialog for given material data group. On entry `Data mode` is set as `Standard`.
2. Introduce base values $V_{BASE}$ of parameters
3. If a given parameter is to be defined as time dependent:
   A: `Set Data` mode on `Load function`
   B: Push the button `Set` at the right to the edit field of given data item
   C: In the appearing dialog select existing `Load function` or generate a new one
   D: Close the `Load function` dialog; a non-zero integer number, being an ID of the selected `Load function` will appear in the edit field of given data item
4. Repeat the action for other data items, if needed, or leave 0 ID in edit fields of these data items which remain time independent
5. If a given parameter is to be defined as variable in space:
   A: Set `Data mode` on `Data` super-element
   B: Push the button `Set` at the right to the edit field of given data item
   C: In the appearing dialog select the one from the `List of existing super-elements` (defined previously during preprocessing session ) or generate a new one (using `Add`) entering manually nodal coordinates and values of interpolation function $F(\mathbf{x})$; to modify the existing super-element use button **Modify**
   D: Close the `Data super-element` dialog; a non-zero integer number, being an ID of the selected `Data super-element` will appear in the edit field of given data item
6. Repeat action for another data items, if needed, or leave 0 ID in edit fields of these data items which remain constant

**Remark**:

In case of non-standard data mode, responsibility for the physical correctness of the material data remains totally on the User, as routine check (for example if Young modulus $E > 0$ or if Poisson ratio $v < 0.5$) will not be performed

Window 2-915

## Related Topics

LOAD FUNCTIONS

## 2.3.3.4 MATERIAL DATA VALUES

### Summary of material data

**Window 2-916: Summary of material data**

The following table summarizes the most important material data

| Data concerning | Item | Symbol | Unit |
|---|---|---|---|
| UNIT WEIGHTS | Unit weight for solid | $\gamma$ | [kN/m$^3$] |
| ELASTICITY | Young modulus | $E$ | [kPa] |
| | Poisson ratio | $\nu$ | [-] |
| M-C PLASTICITY | Friction angle | $\phi$ | [°] |
| | Cohesion | $c$ | [kPa] |
| CUT-OFF | Tensile strength | $f_t$ | [kPa] |
| DILATANCY | Dilatancy angle | $\psi$ | [°] |
| CAP | Initial void ratio | $e_o$ | [-] |
| | Yield surface shape parameter | $R$ | [-] |
| | Slope of virgin consolidation line in e-ln(p) | $\lambda$ | [-] |
| | Preconsolidation press. | $p_c$ | [kPa] |
| CREEP | Parameters $A, B$ | | units dependent on creep law |
| CONSOLIDATION & FLOW | Permeability (Darcy coeff.) | $k$ | [m/s] |
| | Fluid bulk modulus | $\beta_F$ | [kPa] |
| | Fluid specific weight | $\gamma_F$ | [k/m$^3$] |
| | Residual saturation | $S_r$ | [-] |
| | Parameter related to the air entry suction | $\alpha$ | [m$^{-1}$] |
| | Measure of the pore-size distribution | $n$ | [-] |
| | Initial void ratio | $e_o$ | [-] |
| HEAT TRANSFER | Heat conductivity | $\lambda$ | [kN h/°K] |
| | Heat capacity | $c^*$ | [kN/(m$^2$°K] |

**Remarks**:

Units are indicated in square brackets [..].

Window 2-916

**Window 2-917: Typical data values for geo-materials**

- **Rock**

| | |
|---|---|
| UNIT WEIGHT | $\gamma = 18 \div 21$ |
| ELASTIC | $E = 5 \div 100 \cdot 10^6$<br>$\nu = 0.2$ |
| M-C PLASTICITY | $c = 5 \div 100$<br>$\phi = 25 \div 45$ |
| CAP | |
| FLOW | $k = 10^{-14} \div 10^{-10}$ |

- **Gravel**

| | |
|---|---|
| UNIT WEIGHT | $\gamma = 21.5 \div 22.5$ |
| ELASTIC | $E = 10^4 \div 10^5$<br>$\nu = 1 - \sin\phi/2 - \sin\phi$ |
| M-C PLASTICITY | $c =$<br>$\phi = 40 \div 45$ |
| CAP | $e_o = 0.36 \div 0.48$<br>$\lambda = 5 \cdot 10^{-3}$ |
| FLOW | $k = 10^{-4} \div 10^{-2}$ |

- **Sand**

| | |
|---|---|
| UNIT WEIGHT | $\gamma = 18.5 \div 19.5$ |
| ELASTIC | $E = 10^4 \div 10^5$<br>$\nu = 1 - \sin\phi/2 - \sin\phi$ |
| M-C PLASTICITY | $c =$<br>$\phi = 30 \div 35$ |
| CAP | $e_o = 0.79 \div 1.0$<br>$\lambda = 5 \cdot 10^{-3}$ |
| FLOW | $k = 10^{-4} \div 10^{-5}$ |

- **Silt**

| | |
|---|---|
| UNIT WEIGHT | $\gamma = 19.0 \div 20.0$ |
| ELASTIC | $E = 10^4 \div 10^5$<br>$\nu = 1 - \sin\phi/2 - \sin\phi$ |
| M-C PLASTICITY | $c = 10 \div 20$<br>$\phi = 25 \div 35$ |
| CAP | $\lambda = 0.4 \div 0.9 \cdot 10^{-2}$<br>$e_o = 0.70 \div 0.89$ |
| FLOW | $k = 10^{-7} \div 10^{-9}$ |

- **Clay**

| | |
|---|---|
| UNIT WEIGHT | $\gamma = 18.0 \div 19.0$ |
| ELASTIC | $E = 10^3$<br>$\nu = 1 - \sin\phi/2 - \sin\phi$ |
| M-C PLASTICITY | $c = 0 \div 10$<br>$\phi = 20 \div 30$ |
| CAP | $\lambda = 0.9 \div 2 \cdot 10^{-3}$<br>$R = 1 \div 1.5$ |
| FLOW | $0.89 \div 1.1$<br>$k = 10^{-7} \div 10^{-9}$ |

Window 2-917

**Window 2-918: Common reference material data**

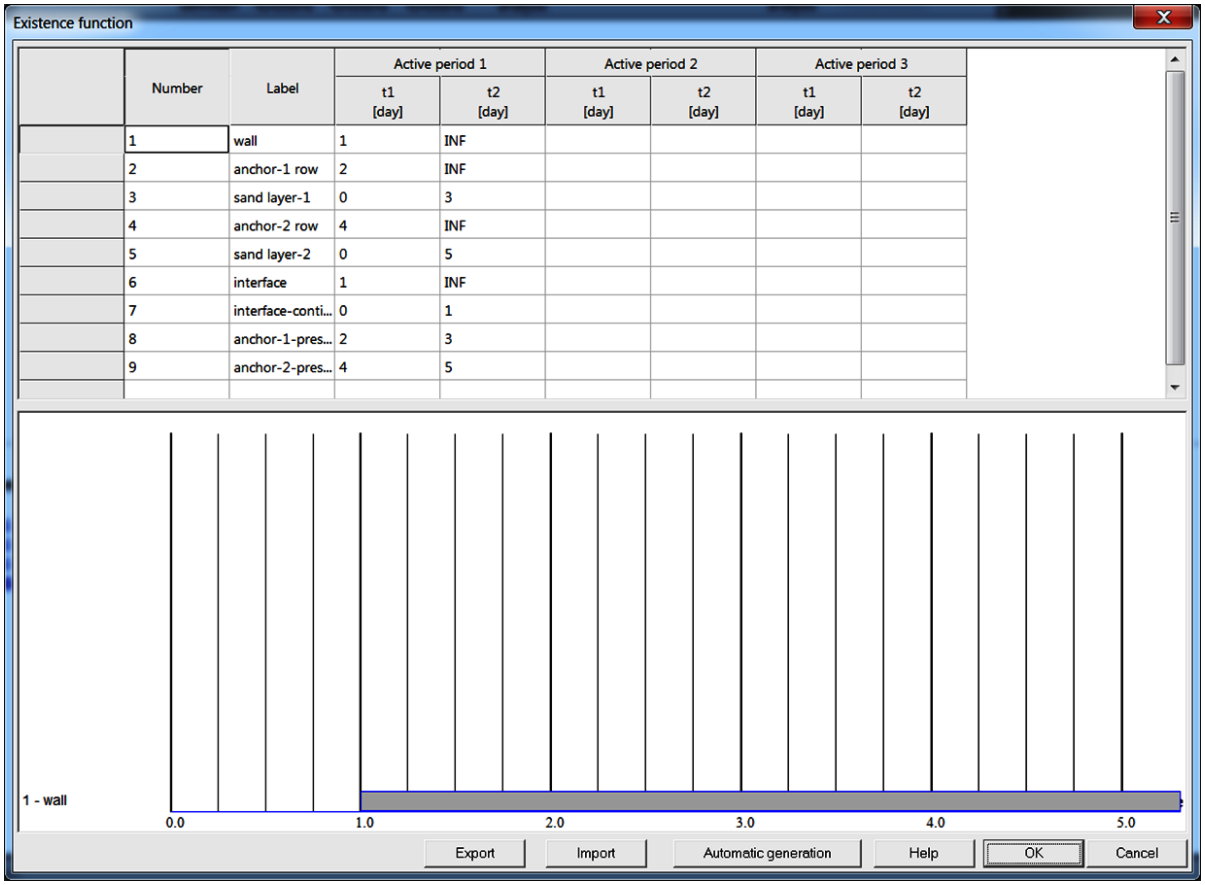
| Item: | | Unit | Steel | Concrete | Soils | Rocks | Water |
|---|---|---|---|---|---|---|---|
| Specific weight | $\gamma$ | kNm$^{-3}$ | 78.0 | 24.0 | $18 \div 22$ | $18 \div 25$ | 10.0 |
| Elastic modulus | $E$ | kPa | $2.1 \cdot 10^8$ | $1 \div 3 \cdot 10^8$ | $10^3 \div 10^5$ | $5 \div 100 \cdot 10^6$ | – |
| Tensile strength | $f_t$ | kPa | – | $4 \cdot 10^3$ | – | $2 \div 30 \cdot 10^3$ | – |
| Compression s. | $f_c$ | kPa | – | $4 \cdot 10^4$ | – | $5 \div 400 \cdot 10^3$ | – |
| Poisson ratio | $\nu$ | – | 0.3 | 0.2 | $0.2 \div 0.5$ | 0.2 | – |
| Bulk modulus | $\beta_F$ | kPa | – | – | – | – | $2 \cdot 10^5$ |
| Heat conductivity | $\lambda$ | W/(mK) | $40 \div 80$ | 2.4 | $1.0 \div 2.0$ | $1.0 \div 2.0$ | 0.060 |
| Heat capacity | $c^*$ | kN/(m$^2$K) | $0.4 \cdot 10^3$ | $2 \cdot 10^3$ | $10^3$ | $1 \div 3 \cdot 10^3$ | $4.2 \cdot 10^3$ |

Window 2-918

## 2.3.4 EXISTENCE FUNCTION

Presence of finite element components like elements or boundary conditions, may be controlled through so-called existence functions. This functions are needed to handle excavation/construction stages. The existence function is the Heaviside like function with value 0 (corresponding to non-existence) or 1 (corresponding to existence) specified at the series of time instances up to $t_1, t_2, \ldots, t_5$.

**Window 2-919: Existence functions: definition**




**Remarks**:

1. Discontinues indexing of existence functions is supported
2. To visualize certain existence function one has to click on corresponding row in the table
3. To show the set of selected existence functions one has to select them by pressing -shift-key and clicking on function that is to be visualized
4. Time instance corresponding to infinity is denoted by **INF** string
5. Maximum 3 periods of activity can be defined
6. The default existence function (it has index 0) is equivalent to permanent existence
7. One may select all cells in the grid and then paste to the Excel sheet; this operation can be made as well in the opposite way
8. All characteristic points appearing in the definition of existence functions will automatically be detected by the computational module; computation at these points will be carried out automaticaly
9. Note that if the given FE model component (like element, BC etc..) has associated existence function that is active from let say $t_1$ =1 month till $t_2$ =3 months then this component will become active just after $t_1 = 1$ month ($t_1$ does not belong to the period of activity)

10. Existence functions can automatically be generated by using the option **Automatic generation** (Window 2-920); these functions can also be directly copied to the Excel using standard copy and paste tools, then modified in the Excel and copied back to ZSoil
11. `Existence Functions` can be exported (using button **Export**) to an external file (with extension *.EXF), for future use in another project or for further processing with use of external software (Excel)
12. Existence functions exported to the *.EXF file can be included into the current project by using an option **Import**; this may be proceed in the three following modes:
    - ◎ **Add** - only new functions will be added.....
    - ◎ **Replace** - current content of existence functions is replaced by the new one.....
    - ◎ **Add only new functions** - functions which do not exist in current list are only added.....

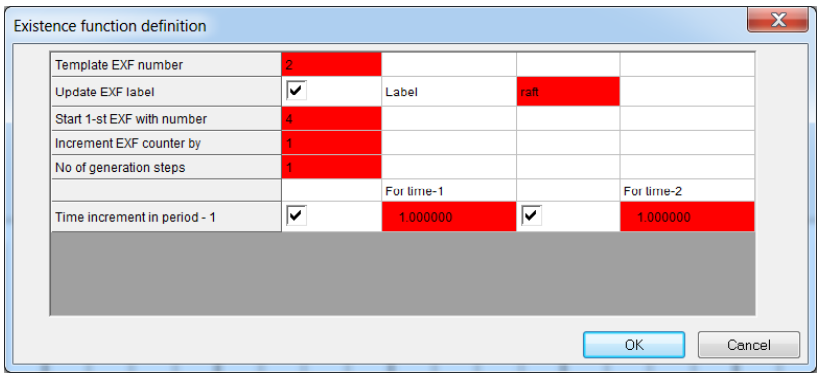
Window 2-919

**Window 2-920: Automatic generation of existence functions**

This option allows to generate series of existence functions by choosing the template function and then copying it $N$ times ( **No of generation steps** [ ] ) with simultaneous updating period of activity.

**Remarks**:

1. If ☒ **Update existence function label** option is set ON then all functions will inherit the basic label given in the edit field **Label** [ ] modified by an extra index (the template function will have extra index 1).
2. The first generated function will have the index indicated in the edit field **Start 1-st existence function with number** [ ]
3. All newly generated functions will be incremented by value **Increment EXF counter by** [ ].
4. By setting ON/OFF the ☒ **Time increment in period...** one can decide if the $t_1$ and/or $t_2$ are incremented




Window 2-920

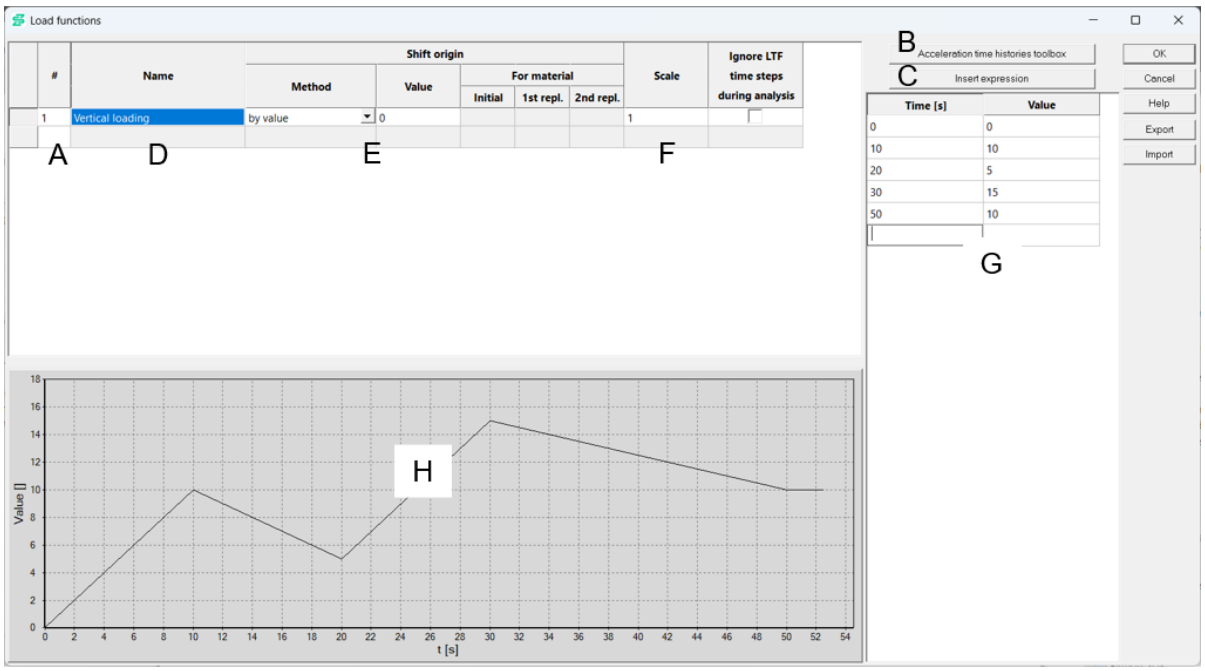
## Related Topics

*Theory:* SIMULATION OF EXCAVATION AND CONSTRUCTION STAGES

## 2.3.5 LOAD FUNCTION

Load time functions are needed to describe evolution in time (real or fictitious) the applied loading, prescribed boundary condition or value of a certain material property.

**Window 2-921: Load time function**




**Remarks**:

1. Each load time function is defined as a set of pairs $(t_k,\ LTF(t_k))$, assuming linear interpolation for intermediate points
2. At any time $t_{n+1}$ current load (for instance) is computed as: $F(t_{n+1}) = F_o \times LTF(t_{n+1})$ where $F_o$ is a reference load amplitude
3. It is recommended to add an unique label to each load time function $LTF$ identified by an index
4. The following general operations can be performed:
   - Enter number in last row to start new load function definition (A)
   - **Delete** - deleting selected functions
   - **CTRL-C** - copying selected functions to cliboard
   - **CTRL-V** - pasting functions from cliboard
5. Content of the selected load time function can be defined by
   - Editing the table (G)
   - Setting the mathematical expression (press button **Insert Expression** (C)); in the definition of the load time function via mathematical expression in the time range `Time begin-Time end`) and with the assumed time increment `Step`; (example: 10*sin(3.14*t)); the following operations and functions are supported:
     - ⋆ arithmetic operators of the type: sum +, subtraction -, multiplication *, division /, parenthesis ( )
     - ⋆ trigonometric functions: SIN, COS, TG, ARCTG
       (arguments of trigonometric functions may be declared *degrees* or *radians*)

- ⋆ square root function SQRT
- ⋆ power function ^

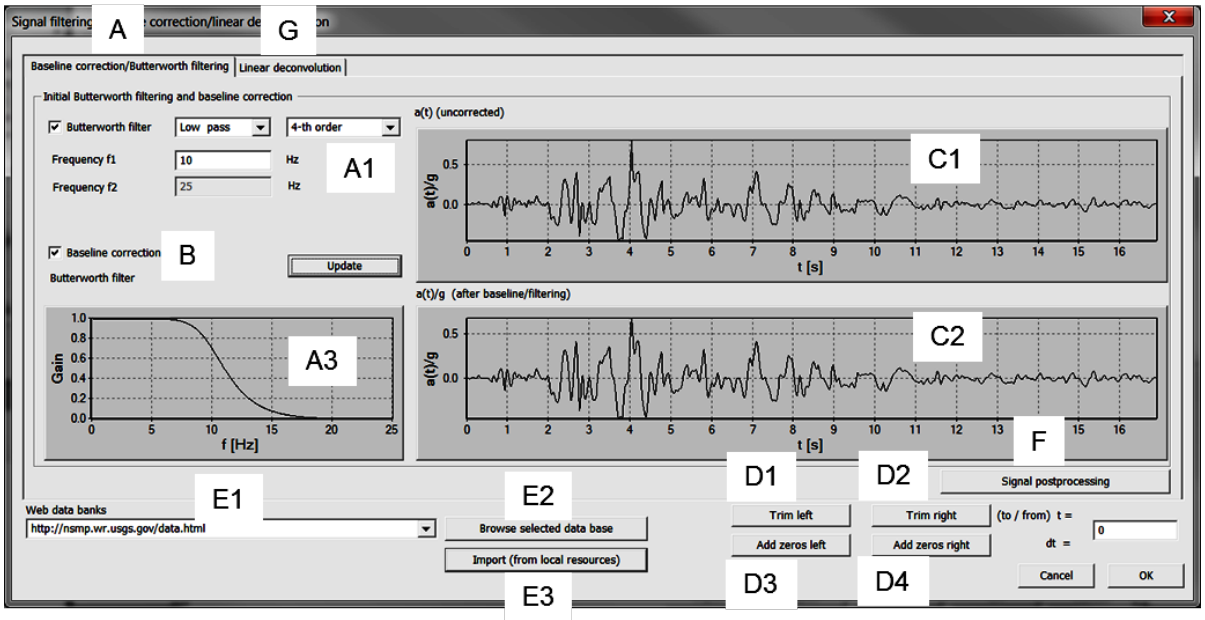
• Importing the acceleration time history from earthquake data bases (press button (B))

6. One may shift the origin of given load time function by a value indicated in edit field **Shift origin** [ ]; this operation can be made by declaring time shift value (`by value`), by associating time shift value with the element activation (for the initial material, 1-st/2-nd replacement one) (this is useful when defining time dependent material properties) or deactivation (for the initial material, 1-st/2-nd replacement one) (this is very useful when defining unloading functions for several soil/rock excavated layers).
7. One may scale the load time function values ($LTF(t)$) by a value indicated in edit field **Scale values by factor** [ ]
8. To inspect the evolution of the function in time one may zoom (with mouse) the diagram (G)
9. Discontinues numbering is supported
10. The load time function that is not defined will yield value 0.0 at any time instance; user will be warned when such undefined function is used
11. For each function, value set at the last point is maintained until infinity
12. During computation extra time steps are generated to trace all characteristic time instances appearing at any load time function definition

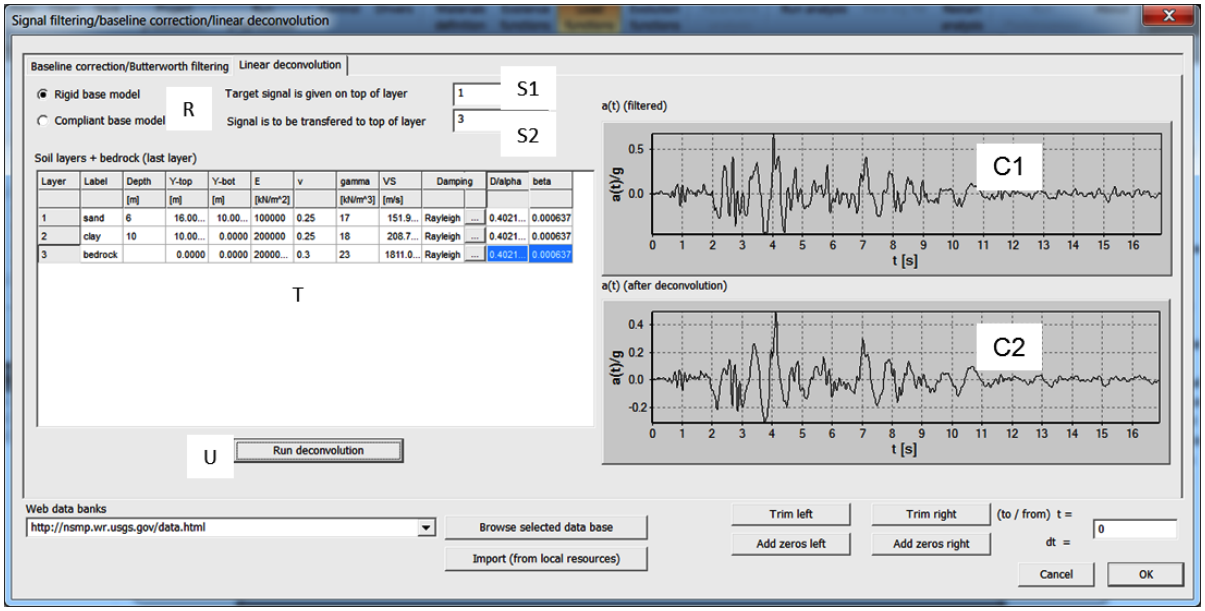
Window 2-921

## Window 2-922: Acceleration time history toolbox

To apply the acceleration time histories as a seismic input, or imposed BC expressed in terms of accelerations, a special toolbox was designed that allows to filter given $a(t)$ signal using one of the high pass, low pass, band pass or band stop (combo box (A)), 2-nd, 3-rd or 4-th order Butterworth filter. This toolbox may carry out so called baseline correction that eliminates the linear drift in the computed displacement time histories but it may also perform the linear deconvolution of the given signal from the control point to the base of the model. It may also visualize and save in the *.csv format uncorrected/corrected acceleration/velocity/displacement time histories, damped elastic response spectra and Fourier spectrum.




Toolbox view for Baseline correction/Butterworth filtering tab active



Toolbox view for Linear deconvolution tab active

**Remarks**:

1. To perform baseline correction press button **Update**; the result of the correction will be shown in diagram (C2)
2. To add an additional signal filtering one may use a Butterworth filter; for low pass filter set value $f_1$ that will cancel frequencies higher than $f_1$ (the gain function (A3) shows how the Fourier amplitudes are scaled); for high pass set again value $f_1$ that will cancel frequencies lower than $f_1$, and for band pass and band stop set values $f_1$ and $f_2$ that give the range of frequencies that are to be maintained or canceled.
3. Three formats of data bases can be imported: the US data bank published at http://nsmp.wr.usgs.gov/data.html, the European one published at http://www.isesd.hi.is/ESD_Local/frameset.htm and the free format where pairs $(t, a(t)/g)$ are stored (E1); user may browse one of the mentioned data banks by pressing button **Browse..**
4. One may trim the acceleration record from the left and right side (use buttons (D1), (D2))
5. One may add quiet zone at the beginning and end of the given record (use buttons (D3), (D4))
6. After downloading the acceleration records user may store them at his own directory; by clicking the button (E3) user may select one of the stored records
7. To export filtered, uncorrected/corrected (through baseline correction) acceleration time histories, and corresponding undamped/damped Fourier spectra, to the Excel, press button (F); for long records this operation is time consuming.
8. To perform deconvolution of the signal switch to the `Linear deconvolution` tab (G), choose proper model (rigid base or compliant base)(R), edit data for soil layers in table (T), define where the target signal is given (S1) and where it should be transfered (S2), run deconvolution with button (U); result will be shown in window (C2); more details and theory can be found in dedicated report on dynamics

Window 2-922

## Related Topics

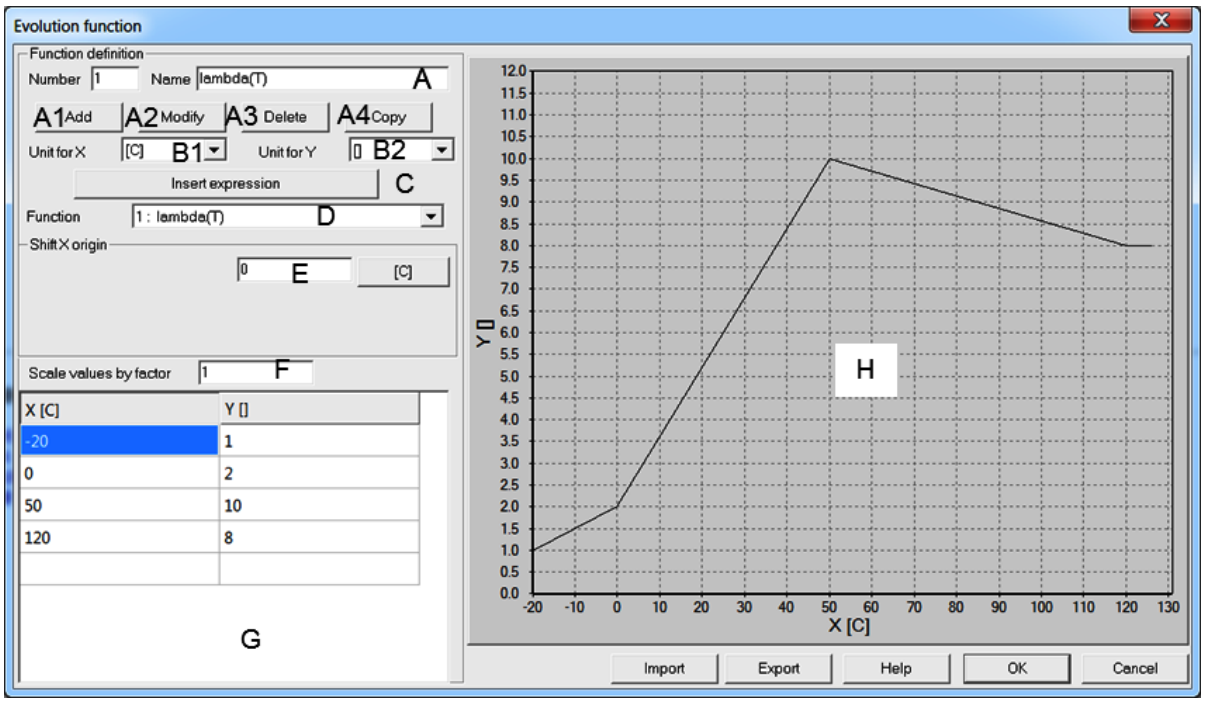
*Theory:*

LOAD FUNCTIONS

## 2.3.6 EVOLUTION FUNCTION

Evolution functions are needed to describe variation of a given material property with respect to parameters like temperature, humidity, pore water pressure etc.. So far this definition can only be applied to heat transfer parameters, mechanical parameters of plastic damage for concrete, truss elastic-plastic model and shell fiber one. These functions are loaded from the internal data base once fire extension is activated at the material level (for the above mentioned models).

**Window 2-923: Evolution function**




**Remarks**:

1. Each evolution function is defined as a set of pairs $(x_k,\ EVF(x_k))$, assuming linear interpolation for intermediate points; global time stepping does not control these functions; therefore one can represent them with very high accuracy
2. At any time $t_{n+1}$ current value of parameter $P$, associated with a given evolution function, is computed as: $x_{n+1} = x(t_{n+1})$
   $P(x_{n+1}) = P_o \times EVF(x_{n+1})$ where $P_o$ is a reference value of the parameter
3. It is recommended to add an unique label to each evolution function identified by an index **Number** [ ]
4. The following general operations can be performed:
   - **Add** (A1) - adding a new function identified by an index (in edit **Number** [ ]) and label(in edit **Label** [ ])
   - **Delete** - (A2) - deleting function selected from combo-box `Function`
   - **Modify** - (A3) updating the content of a selected function from combo-box `Function`
   - **Copy** - (A4) copying selected function as a new one

5. Content of the selected evolution function can be defined by
   - Editing the table (G)
   - Setting the mathematical expression (press button **Insert Expression** (C)); in the definition of the evolution function via mathematical expression in the range $x_o$-$x_{\text{end}}$) and with the assumed increment $\Delta x$; (example: $10 * sin(3.14 * t)$ (use always $t$ as a function argument));
     the following operations and functions are supported:
     - ⋆ arithmetic operators of the type: sum $+$, subtraction -, multiplication *, division $/$, parenthesis ( )
     - ⋆ trigonometric functions: SIN, COS, TG, ARCTG
       (arguments of trigonometric functions may be declared *degrees* or *radians*)
     - ⋆ square root function SQRT
     - ⋆ power function ^
6. One may shift given evolution function in time scale by a value indicated in edit field **Shift time origin** [ ];
7. One may scale the evolution function values ($EVF(t)$) by a value indicated in edit field **Scale values by factor** [ ]
8. To inspect the evolution of the function in time one may zoom (with mouse) the diagram (H)
9. Discontinues numbering is supported
10. The evolution function that is not defined will yield value 0.0 at any time instance; user will be warned when such undefined function is used
11. For each function, value set at the last point is maintained until infinity
12. **None of characteristic instances**, appearing in definition of an evolution function, is traced during computation

Window 2-923

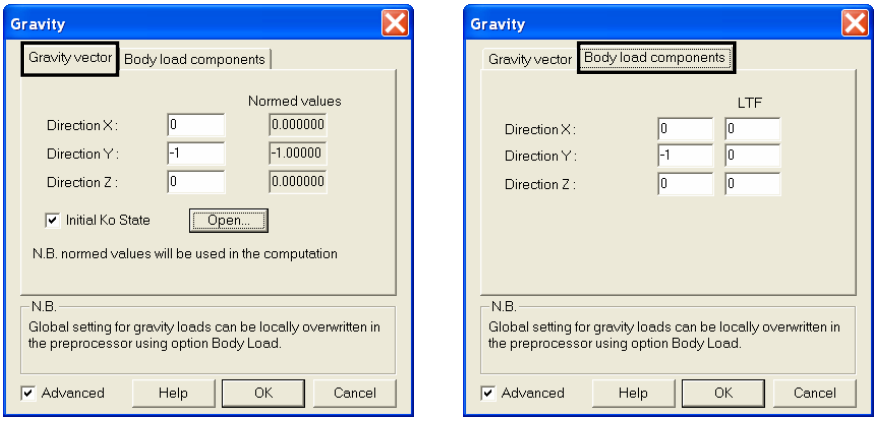
## Related Topics

*Theory:*

EVOLUTION FUNCTIONS

## 2.3.7 GRAVITY

**Window 2-924: Gravity**




**Remarks**:

1. Gravity direction can be set up using two definitions:
   A: `Gravity vector`
   - Using this definition user may declare gravity vector $\overline{\mathbf{b}}$ (default is (0,-1,0))
   - This vector $\overline{\mathbf{b}}$ is used to compute current body load vector $\mathbf{b} = \gamma\ \overline{\mathbf{b}}$ at each finite element (body load definition may be overwritten, at the element level, during preprocessing stage, using option `Body load`)
   - In addition to that option user may set a global initial state $K_o$ coefficient; this global $K_o$ setup can be overwritten at the material level by activating ☒ **Initial state** $K_o$ group of parameters

   B: `Body load components` (only in the ☒ **Advanced** mode)
   - Using this definition user may declare gravity vector $\overline{\mathbf{b}}$ (default is (0,-1,0)) plus load time functions independently for all directions
   - In this case gravity direction vector is not normalized
   - Vector $\overline{\mathbf{b}}$ is used to compute current body load components $b_i = \gamma\ \overline{b_i}\ LTF_i(t)$ at each finite element (body load definition may be overwritten, at the element level, during preprocessing stage, using option `Body load`)
2. For pseudo-seismic stability analysis the option `Body load components` has to be used (for instance in 2D models one may assume $\overline{\mathbf{b}} = (0.1, -1.0)$)
3. Using the option `Body load components` one may cancel gravity loads, but not mass, in the dynamic time history analysis; the easiest way, however, to cancel body loads or mass (in dynamic analyses), is to define the two independent multipliers, one for $\gamma$ and one for $\rho$ at the material level in the setup for `Unit weight`. Setting $\gamma$ multiplier to $0.0$ will cancel body forces in all elements with that material (unless body load definition is made at the element level in the preprocessor) and setting $\rho$ multiplier to $0.0$ will cancel contribution of all elements with that material to the mass matrix.
4. Gravity direction influences also the hydraulic flow potential

## Related Topics

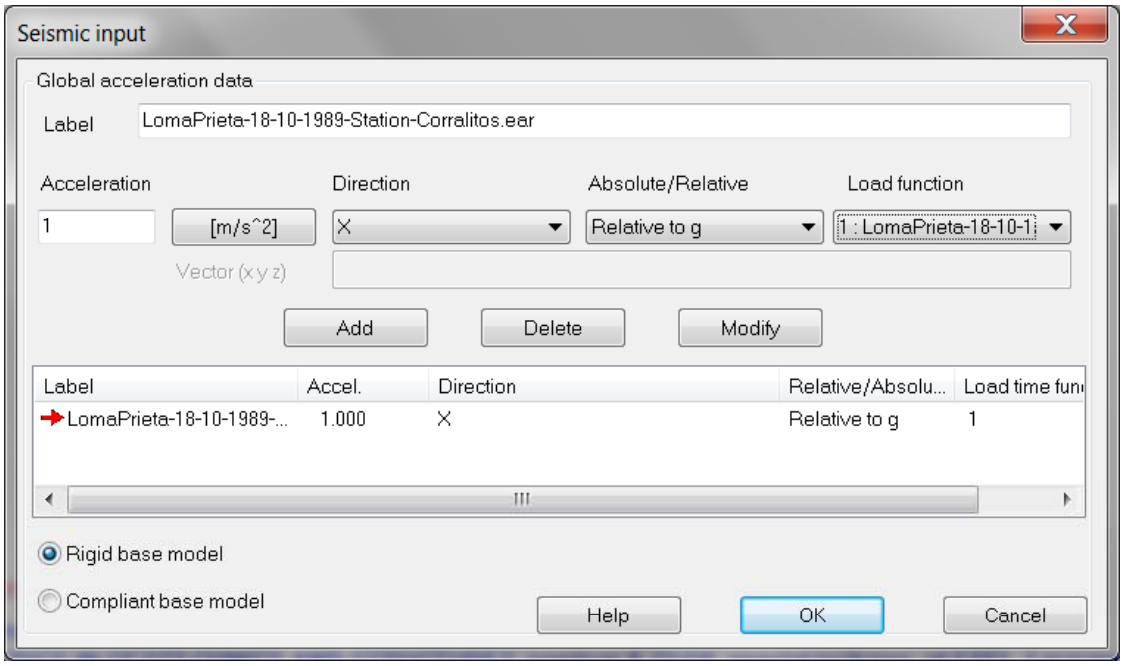
- *data preparation: INITIAL K0 STATE*
- *theory: GEOTECHNICAL ASPECT - INITIAL STATE*
- *theory: NUMERICAL IMPLEMENTATION - INITIAL STRESS*

## 2.3.8 SEISMIC INPUT

**Window 2-925: Seismic input**

Seismic input option allows the user to apply the earthquake in the finite element model

- in the relative format for rigid base model; here seismic input is given as an imposed acceleration $\mathbf{a}_g = \ddot{\mathbf{u}}_g$ of the ground that is global to the whole structure; the corresponding inertia forces are shifted to the right hand side $\mathbf{F}(t) = -\mathbf{M}\mathbf{a}_g(t)$ (set ◎ **Rigid base model** ON)
- in the absolute format for compliant base model; here seismic input is given as an imposed acceleration $\mathbf{a}_g = \ddot{\mathbf{u}}_g$ that is integrated to velocities (via Newmark method) and applied at the base of the model where viscous dashpots must be added; this way the acceleration record is converted to the traction time history and applied to the bottom boundary of the model (the bottom boundary is identified as a line segment (2D) or surface (3D), shared by viscous boundary located at $y = y_{MIN} = const$) (compliant base model circumvents waves reflections) (set ◎ **Compliant base model** ON)




Seismic input dialog window

**Remarks**:

1. Each acceleration record describes a time dependent acceleration vector that can be set in one of global $X$, $Y$ or $Z$ directions, or in the direction defined by a given vector
   - for global $X$ direction : $\mathbf{a}_g(t) = a_o\,(1,0,0)^T \cdot LTF(t)$
   - for global $Y$ direction : $\mathbf{a}_g(t) = a_o\,(0,1,0)^T \cdot LTF(t)$
   - for global $X$ direction : $\mathbf{a}_g(t) = a_o\,(0,0,1)^T \cdot LTF(t)$
   - for given vector $\mathbf{v}$: $\mathbf{a}_g(t) = a_o\mathbf{v} \cdot LTF(t)$

   where $a_o$ is the value set in the edit field **Acceleration** [ ]

2. Acceleration values can be set as:
   A: relative to gravity acceleration $g = 9.81[m/s^2]$
   B: absolute
3. Seismic input may consist of collection of acceleration records. At given time instance global acceleration will be created as a sum of all records
4. Each record may have a different `Load Function`
5. To perform baseline correction/Butterworth filtering or linear deconvolution of the given record use acceleration time history toolbox that can be activated at the `Load time function` level
6. Activate ◎ **Compliant base model** option if seismic signal was transformed by linear deconvolution procedure
7. ◎ **Compliant base model** / ◎ **Rigid base model** switch is common for all applied earthquake records

## Related Topics

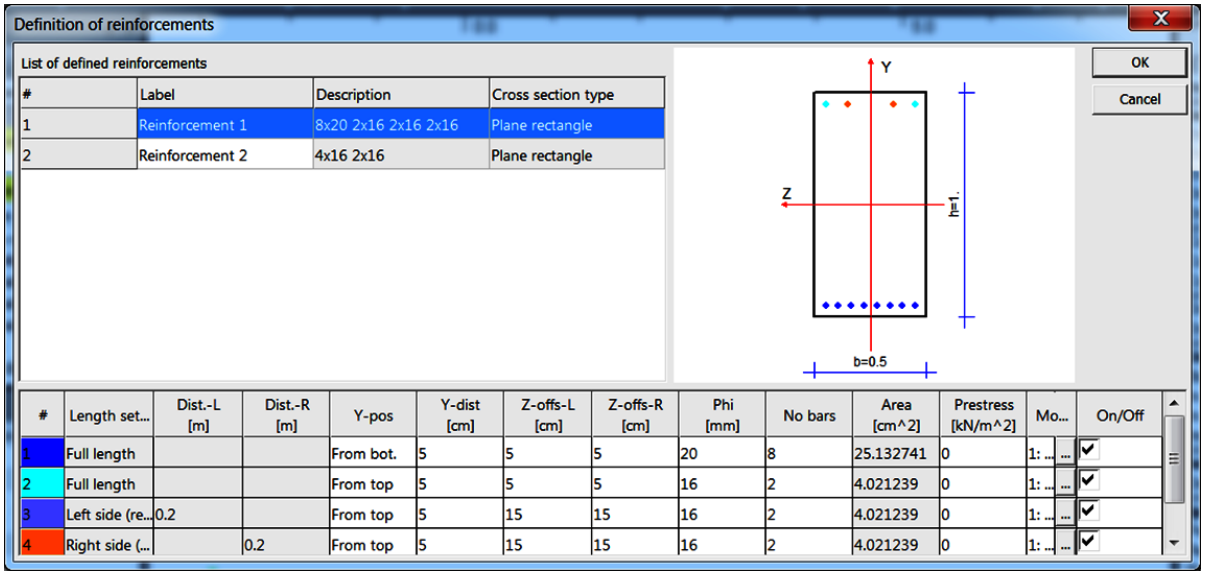
- *data preparation: TRANSIENT DYNAMICS*
- *data preparation: LOAD FUNCTIONS*

Window 2-925

## 2.3.9 TABLE OF REINFORCEMENT SETS FOR BEAMS

**Window 2-926: Table of reinforcement sets for beams**

This option gives an access to the table of reinforcement sets defined by the user (for beam elements) during preprocessing stage. It is shown in the following figure




| # | Label | Description | Cross section type |
|---|---|---|---|
| 1 | Reinforcement 1 | 8x20 2x16 2x16 2x16 | Plane rectangle |
| 2 | Reinforcement 2 | 4x16 2x16 | Plane rectangle |

| # | Length set... | Dist.-L [m] | Dist.-R [m] | Y-pos | Y-dist [cm] | Z-offs-L [cm] | Z-offs-R [cm] | Phi [mm] | No bars | Area [cm^2] | Prestress [kN/m^2] | Mo... | On/Off |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Full length | | | From bot. | 5 | 5 | 5 | 20 | 8 | 25.132741 | 0 | 1: ... | ✓ |
| 2 | Full length | | | From top | 5 | 5 | 5 | 16 | 2 | 4.021239 | 0 | 1: ... | ✓ |
| 3 | Left side (re... | 0.2 | | From top | 5 | 15 | 15 | 16 | 2 | 4.021239 | 0 | 1: ... | ✓ |
| 4 | Right side (... | | 0.2 | From top | 5 | 15 | 15 | 16 | 2 | 4.021239 | 0 | 1: ... | ✓ |

Dialog to edit `table of beam reinforcement sets`

To get acquainted on how to edit reinforcement sets in beams please refer to the Window 2-357 in paragraph 2.3.1.3.3 for detailed explanation. In this table only a limited set of actions can be made no matter whether it is called from the main ZSoil®menu or at the preprocessing stage. This dialog allows the user to delete defined sets or to modify their following items

- Y, Z (or radial/angular for circular shapes) offsets (measured with respect to the cross section)
- number of bars and their diameters
- prestress value
- 1D model to represent $\sigma - \varepsilon$ relationship (elastic, elastic-perfectly plastic and user defined)
- status of activity (ON/OFF)

## Related Topics

- *data preparation: BEAMS*
- *theory: UNIAXIAL STRESS-STRAIN LAWS FOR BEAM FIBERS*

Window 2-926

## 2.4 ANALYSIS

Three computation modes are available:

| | |
|---|---|
| `Run Analysis` | This option consists of the following actions:<br>• Saving the **.inp* file<br>• Generation of the **.dat* file<br>• Running calculation module starting from the first driver in the list declared under `Analysis and Drivers` |
| `Restart` | This option allows to restart computation from the last last converged configuration of equilibrium<br>See: Window 2-927 : General remarks<br>Window 2-928 : Allowed and prohibited model modifications before restart |
| `Batch processing` | This option is designed to allow the User to prepare a list of projects (***.inp** files), to store them in the ***.bat** file and then to execute them in batch mode.<br>See Window 2-929 : Handling batch processing |

### 2.4.1 RESTART

**Window 2-927: Restart: General remarks**

1. In case when at a given time step of the `Time dependent Driver` computation fails to converge user may introduce the following modifications:
   - Change time step to diminish applied load increments
   - Modify `Control` data, i.e. convergence tolerances, nonlinear solver by switching full Newton-Raphson method to the BFGS one (for instance) or the initial stiffness

   and then to restart the computation
2. In case when at a given stability analysis step computation fails to converge, and no clear failure mechanism is detected, user may introduce the following modifications:
   - Change safety factor increment (usually reduction is needed)
   - Modify `Control` data, i.e. convergence tolerances, nonlinear solver by switching full Newton-Raphson method to the BFGS one (for instance) or the initial stiffness

   and then to restart the computation
3. If user did not declare some material property as time dependent (which is recommended) and at certain time he wants to change its value then restart must be enforced. To handle that case the following actions have to be performed:
   - Modification of the material property
   - Insertion of the next driver to continue computation starting from certain time instance
   - Executing computation restart

Window 2-927

**Window 2-928: Restart: Allowed and prohibited model modifications**

The following model modifications are allowed:

- Modification of `Control` data
- Change of `Driver` list (only not yet executed drivers can be deleted)
- Change of `Driver` parameters (time stepping)
- Modification of `Load time function` in time ranges not completed yet, including time points and function values
- Modification of `Existence function` in time ranges not completed yet, including time points and existence status
- Loads (i.e. forces, fluxes) can be added / deleted
- Load values can be modified (reference values as well as load time functions)
- Boundary conditions can be added / deleted
- Material model parameters can be changed

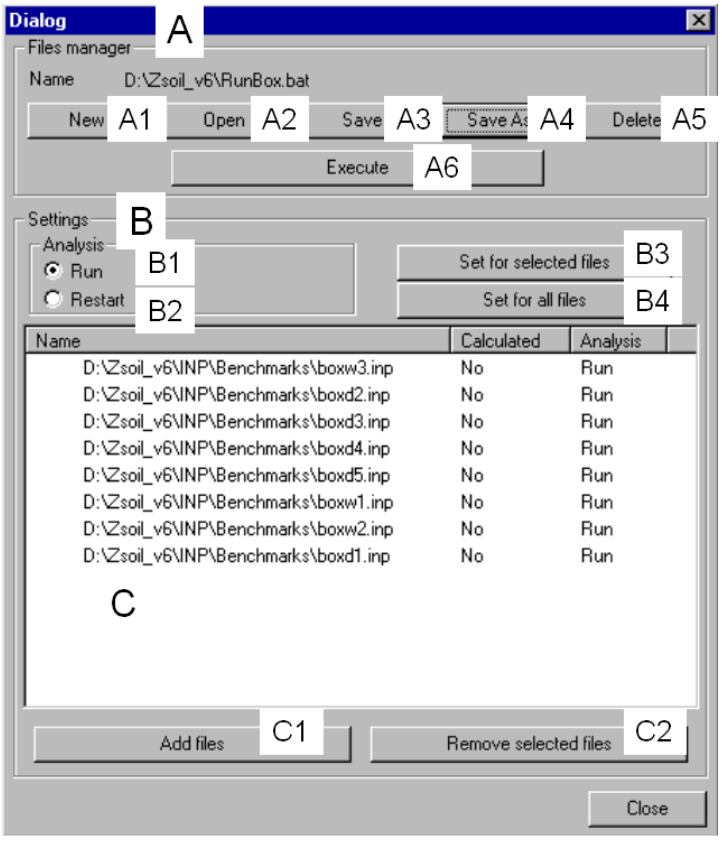
The following model modifications are prohibited:

- Changing `Problem type` in the sequence of `Drivers`
- Changing `Analysis type`
- Adding / deleting nodes
- Adding / deleting elements
- Changing assignment of the material model to the elements
- Changing material formulation for the material

Window 2-928

## 2.4.2 BATCH PROCESSING

**Window 2-929: Batch processing**




- **A**. Files manager actions:
  - ⋆ **A1.** **New** –Create a new (empty) ***.bat** file with **Untitled.bat** file name;list **C** is cleared
  - ⋆ **A2**. **Open** –Open an existing ***.bat** file; After opening list **C** will be filled with ***.inp** files
  - ⋆ **A3**. **Save** –Store current ***.bat** file with the current name
  - ⋆ **A4**. **Save As** – Store current ***.bat** file with a given name
  - ⋆ **A5**. **Delete** – Delete indicated ***.bat** file
  - ⋆ **A6.** **Execute** –Run computation of all projects shown in list **C** (considering declared execution mode for each file)
- **B**. `Settings` – Execution mode to be assumed for any file from list **C**, in particular:
  - ⋆ **B1**. ◎ **Run** – Project will be run from the first declared driver
  - ⋆ **B2**. ◎ **Restart** – Project is run starting from the last saved configuration of equilibrium (see section `Restart` (2.4.1).
- **C.** Actions to be made on list (C) of ***.inp** files are as follows:
  - ⋆ **C1.** **Add** – Add selected ***.inp** files to the list (C)
  - ⋆ **C2**. **Remove** – Delete selected ***.inp** files from list (C)

**Remark**:Selection of files from the list can be performed by pointing on the file line with the mouse. Pressing `Shift` allows for multiple selection of files in a continuous block.

Window 2-929

## 2.5 RESULTS

| | |
|---|---|
| `Postprocessing` | Runs unified 2D/3D graphical postprocessor to visualize results of computation: input data, deformed mesh, plastic zones, stress resultants in form of diagrams or color contours, stress/strain contours, time histories of any selected nodal or element quantities, color contours and diagrams in cross sections, principal stresses in continuum elements, principal membrane forces and/or bending moments in membrane/shell elements, section forces and moments in beams/trusses. |
| `Pushover Results` | Performs automatically seismic demand assessment according to given design spectra (EC8 or custom) Each action will invoke MS-EXCEL spreadsheet for numerical results and graphics. These include: demand spectra in A-D or A-T format, multi-DOF(i.e. original system) capacity curve, SDOF (i.e. equivalent single DOF system)capacity curve and demand spectra reduced by ductility, pushover analysis summary. |
| `Results for piles` | Shows results for piles, modeled as beams embedded in the 3D continuum, in the form of Excel sheet. |
| `Results for fix.anch.zones` | Shows results for fixed anchor zones in the form of Excel sheet. |
| `Eigen vectors` | Exports eigenfrequencies, mass participation factors and eigenvectors to the Excel sheet. |
| `View log file *.log` | Browse the log file (created by calculation module) to trace warnings which may indicate source of potential mistakes in the model. In case of fatal error, computation is stopped and explanation is given at the end of the file including some hints on how to remedy the problem. |
| `View element results` | Browse the file (*.str) containing results stored at Gauss points at each element according to the specification set under `Control/Results content` menu. |
| `View nodal results` | Browse the file (*.dis) containing results stored at nodal points according to the specification set under `Control/Results content` menu. |
| `View echo of data` | Browse the file (*.eda) containing echo of the data (*.dat) file generated by the menu and sent to the calculation module. |

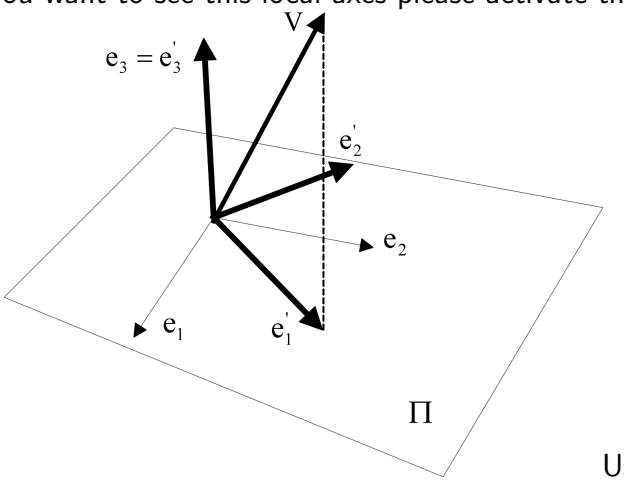
### 2.5.1 POSTPROCESSING

The detailed explanation on how to use graphical postprocessor is given in form of video films recorded for all prepared tutorials. Some specific topics in form *How do I...* are given in next sections.

| | |
|---|---|
| How do I ... | Explanations for some specific options |
| How to use macro | Explanation on how to use macros to create an automatic output |
| Parametric analysis | Explanation on how to plot nodal/element results vs varying material properties at a given time step |

### 2.5.1.1 HOW DO I.....

1. **paste a picture to text editor or Powerpoint**
   There are three possibilities:
   - through File / Save As selecting *.bmp format;
     then under text editor you can import this bitmap,
   - through Ctrl-C and then Ctrl-V (here white background is recommended),
2. **set a local system for nodal results and results for continuum elements**
   Use *Settings/Local coord. system/continuum, nodal quantities menu* to open a dialog. This dialog allows you to define a new X'-Y'-Z' local coordinate system or cylindrical system. For X'-Y'-Z' you have to define the two orthogonal basis vectors (normalization is not required) and the third one will be set as vector product of first two. You may get an error message which says that the orthogonality check has failed. All results in nodes and continuum elements will be transformed to this new system.
   For cylindrical system you have to define a point and orientation vector coinciding with axis of axial symmetry. All nodal results and results for continuum elements will be transformed to this system.
   In all cases you want to change the local system just press *Modify* button and then OK. If you want to see this local basis please activate it *Visibility setup.*
3. **set a local system for shells/membranes/contact**
   Use *Settings/Local coord. system/shells,membranes/contact menu* to open a dialog. This dialog allows you to define a vector $V$ in reference system $X-Y-Z$ used for shell/membrane/contact element local basis setup. The definition of the local coordinate system (where $\Pi$ is a plane tangent to the midsurface) is show in figure below Local axis $X_L$ coincides with $\mathbf{e}_1^{'}$, axis y with $\mathbf{e}_2^{'}$. Note, that axis $X_L$ of that system, in each point separately, is determined as the closest possible to the vector $V$, among all directions tangent to to the element. If $V$ is tangent to the element itself, $X_L$ will coincide with $V$. The detailed procedure of evaluating $\{\mathbf{e}_i'\}$ is given in the Theroretical manual section **??**
   It may happen that for complex shell structures this definition leads to the situation that vector $V$ is parallel to the normal local axis $\mathbf{e}_3$. In that case new local basis setup is nonunique and hence results for these elements will not be displayed. Additionally the following warning is shown : "*No way to define a local system for shell/contact for given vector*".
   To remedy this situation please use mechanism of selections and work on groups of elements.
   If you want to see this local axes please activate them in *Visibility setup.*




User defined shell-membrane-contact coordinate system

4. **set local basis for 3D section of continuum**
   When making a 3D section of continuum element we need to define a local basis for results transformation. A normal vector to the section plane is assumed to be "N" axis while the two axes "T1" and "T2", in the section plane, have to be set using additional information. Through *Settings/T1 axis for sections 3D* we open a dialog box in which we can set a vector, such that "T1" axis coincides always with projection of this vector on section plane $\Pi$.
   If you want to see this local basis please activate it *Visibility setup.*
   It may happen that running *Graph Option/Sectional quantities* you get a warning "The T1 axis will be adjusted automatically so please modify it if necessary". It means that the default vector (for positioning of the T1 axis) was such that normal to the section plane was parallel to this vector. In that case program automatically changes this vector to avoid a conflict but its position is random so please verify it especially if you analyze tensorial/vectorial components with index T1,T2.
5. **save/ read graphical settings for project**
   Please use *Graph Option/Save Settings, Graph Option/Read Settings.*
6. **compute integrals for results shown in Color Maps**
   Please use *Misc./Maps/Integral:INT(rsl) dA or Misc./Maps/Time history of INT(rsl) dA.* This second possibility use with caution as it may take along time.
7. **compute integrals for results shown in Sectional quantities**
   Please use *Misc./Sections 3D/Integral:INT(rsl) dA or Misc./Sections 3D/Time history of INT(rsl) dA.* This second possibility use with caution as it may take along time.

### 2.5.1.2 HOW TO USE MACRO

**Option** `Macro` **is designed to perform the following:**

- to generate a template sequence of plots and store them in external ***.BMP** files
- to generate a template sequence of numerical results and store them in external ***.CSV** files
- to repeat a template action for selected time instants in one job.
- to execute an action from the template previously executed and stored in another job

**Remarks**:

1. Produced output i.e. graphic (***.BMP**) or text (***.CSV**) files can be easily processed in an automated way by any suitable software like MS-Word, PowerPoint, Excel.
2. Note, that using the technique of "linking" but not "copying" objects allows you to generate template documents which will be automatically actualized if only ZSoil® output will change.

**To generate a template action:**

- Activate a macro session by: `Macro`→ `Record macro`,
- Execute any desired control sequence of the plot including:
  - ⋆ `Time` instance,
  - ⋆ its contents (i.e. `Graph. option`),
  - ⋆ detailed `Settings` including `Visibility` setup,
  - ⋆ position on the screen and zoom (under `View`),
  - ⋆ required `Selection`.
- Store current plot status in a macro file (*.**mcr**) by pressing `CTRL-C` or `File`→ `Copy to clipboard`.
- Generate and record next plots.
- To end the macro session use `Macro`→ `Save macro`. Assign any name to the macro (*.**mcr**) file proceeding in a standard MS-Windows way.

**How to generate files resulting from the recorded action**

**Remark**: Any time the macro is played, the plots previously activated by `CTRL-C` during macro recording are stored in ***.BMP** files. Names of these files are automatically created. Each name contains a full information on file contents. It consist of three parts separated by "-" concerning:

- plot contents identified by the **label** identical as the string appearing at the bottom of the PostPro frame
- **job name** (i.e. of ***.INP** file)
- **time instant** in the form "t=x.y" , plot instance (in case of multiple plots of the same origin)

See the example of automatically generated file name:

EFFECTIVE PRINCIPAL STRESSES-MyINPFileName-t=0.1.bmp

The *.BMP files will be created at the same directory where *.**mcr** file is located by the User.

To generate files:

- `Record` and `Save` a macro, see the previous point.

- Use: `Macro→ Play macro`
- In appearing dialog `Open` the ***.mcr** file

**How to repeat a template action for selected time steps**

- Activate a macro session by: `Macro→ Record macro`
- Execute a template sequence of actions for given time instance
- Use `Macro→ Repeat action for selected time step`
- In the appearing listbox select the time steps for which the action has to be performed again (i.e point the time step lines with the mouse, marking them in dark blue).
- Store the sequence of actions using `Macro→ Save`
- Use `Macro→ Play` to generate plots and store them in external files

**To execute an action from the template previously executed and stored in another job**

- Use `Macro → Play macro`
- Select and then `Open` related existing ***.mcr** file. This will generate the plots and/or result files for the recent job with all settings identical as in the template session.

**To execute Z_POST3D with a macro**

To execute any action recorded in a macro stored in the ***.mcr** file, without entering Z_POST3D type a command on a level of operating system or run it as a batch file:

$Z_POST3D$ $[\#@PRJ$ **DatPath**$@\#]$ $[\#@POST3D_MCR$ **McrPath**$@\#]$

where:

- **DatPath** is a full path to the ***.DAT** file containing compiled data
- **McrPath** is a full path to the ***.MCR** file containing pre-recorded macro as described above
- $[\#@PRJ,$ $[\#@POST3D_MCR,$ $@\#]$ are obligatory control sign sequences

### 2.5.1.3 HOW TO VISUALIZE STRESS RESULTANT ENVELOPES FOR STRUCTURAL ELEMENTS

In ZSoil®2026envelopes of any quantity can be visualized for all kind of elements. This option is of major interest in case when results in structural elements are analyzed. In order to visualize envelopes an example of analysis of beam sectional forces is considered. THe following steps lead to the result

1. Create selection list(s) of time instances to be taken for envelopes using option `Envelopes mode` from the ribbon (see fig. 2.1 and fig. 2.2) (using the three buttons one may add a new, modify or delete the existing selection). In the listbox A.1.a of dialog box A.1 time instances that are not taken to the envelope are listed while in the listbox A.1.b only selected ones are considered (see fig. 2.2). In the dialog box A one may set a proper name for a given selection (see fig. 2.2).

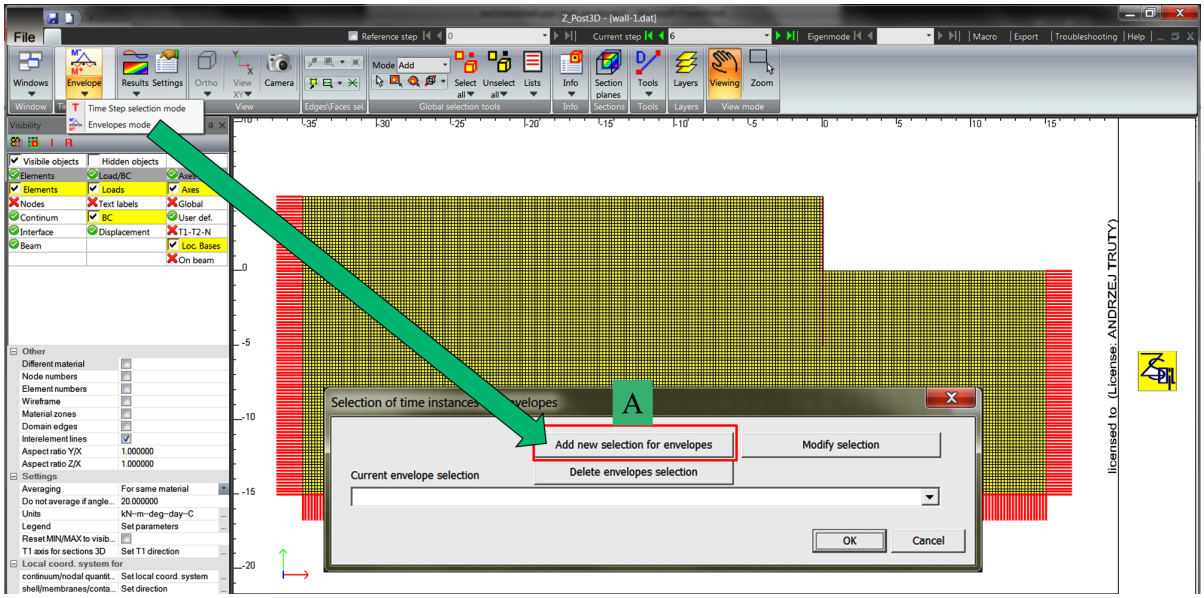



Figure 2.1: Generating selection of time instances for envelopes

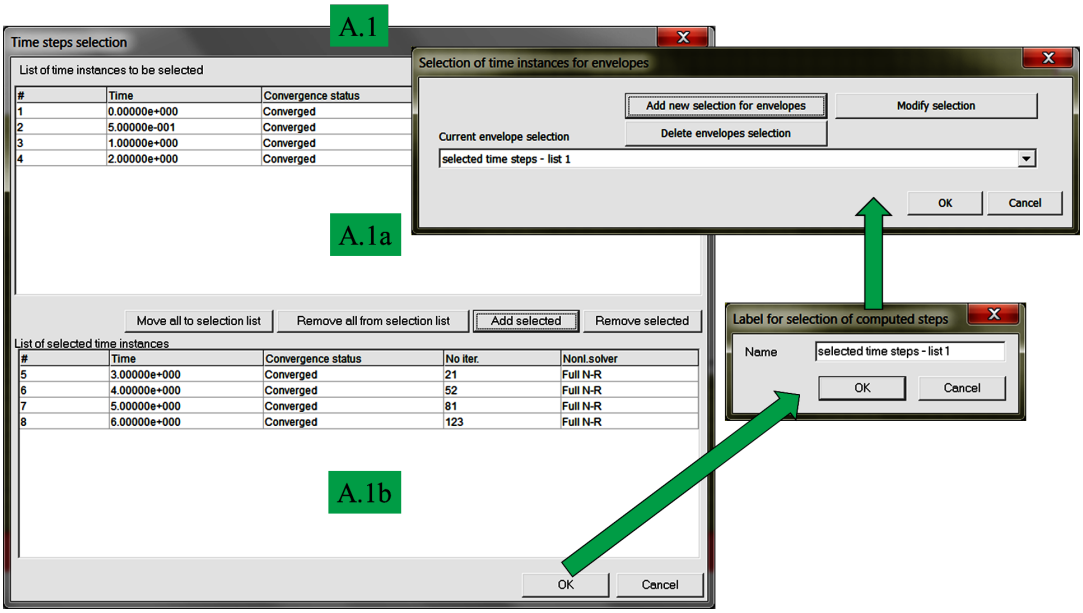



Figure 2.2: Selection of time instances for envelopes

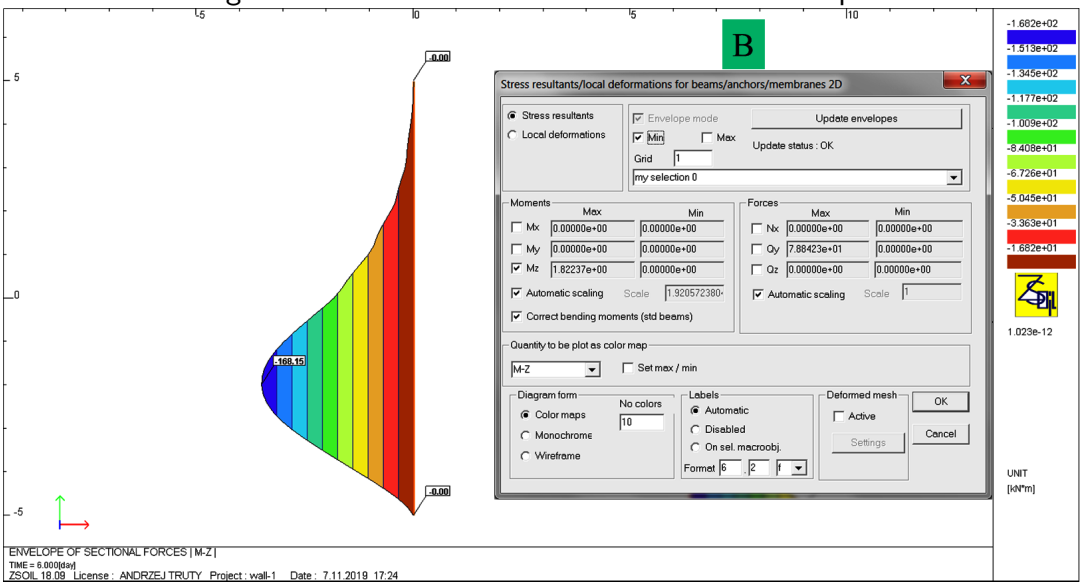



Figure 2.3: Visualization of envelopes

2. Run graphic action `Graph.option/Results in beams/...`

- In the dialog box B (see fig.(2.3)) one may decide whether to show `Min` or `Max` or both envelopes
- To get a better accuracy one may set the grid value greater than 1
- As evaluation of envelopes can be time consuming it is not activated immediately after each modification of the setup; the current envelopes reevaluation status is always shown in the dialog as OK or NOT DONE; to recalculate envelopes one may click on the **Update envelopes**
- Different forms of diagrams and additional labels with values at characteristic points can be set up in dialog box B

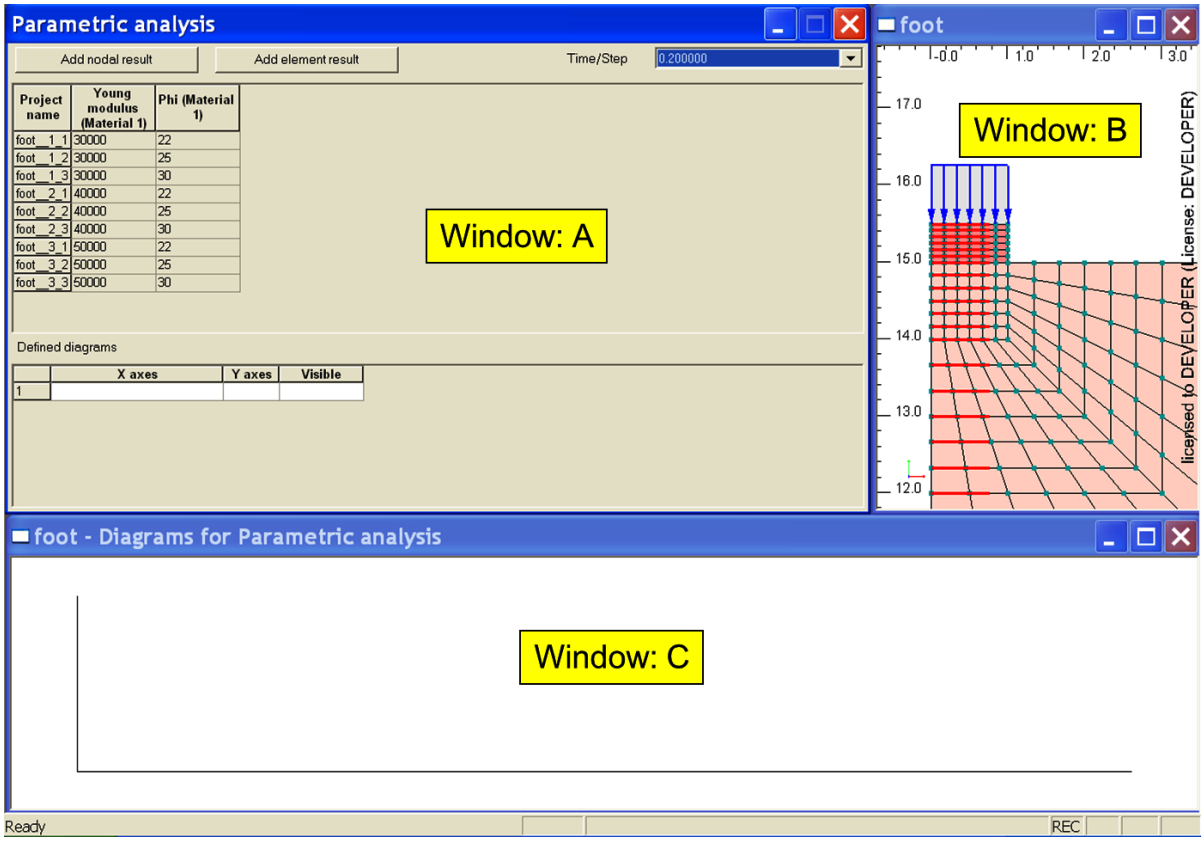
### 2.5.1.4 WORKING OUT RESULTS FROM PARAMETRIC ANALYSIS

Results produced by running of a set of data files automatically generated by using the option `File/Parametric analysis` can be analyzed in a standard manner file by file. However, in the real applications one may want to analyze the influence of a certain varied material property on a selected nodal (settlement, pressure, reaction, temperature etc..), or element (stress, saturation ratio, fluid velocity etc..) quantity at a given time step. This kind of output can be produced in the graphical postprocessor through option `Graph.option/Parametric analysis`. To present on how to use this option let us consider the example of an analysis of bearing capacity of a footing discussed already in section 2.1.1.

**Window 2-930: Main windows**

By running the `File/Parametric analysis` option the following 3 graphical windows are put on the screen:

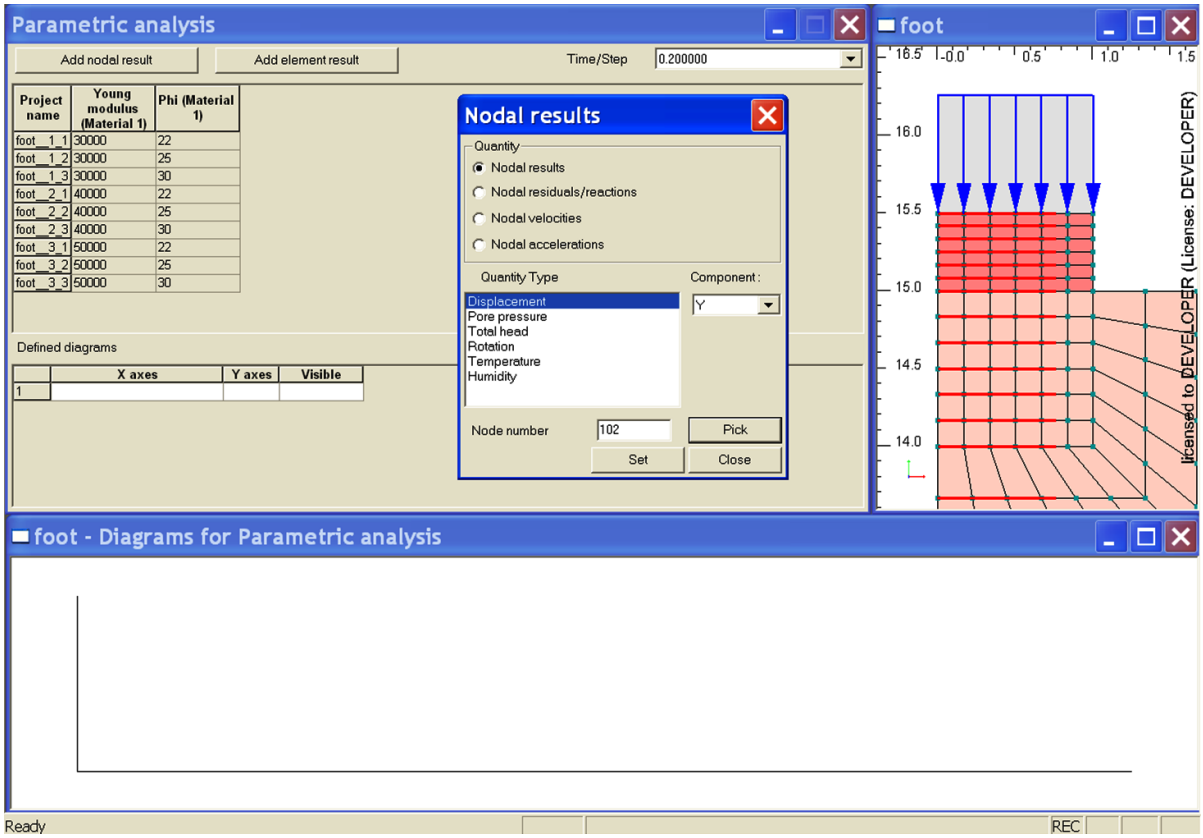
- Window A: this window is used to collect selected nodal/element results which are placed in columns
- Window B: this window is used to pick nodal point or an element
- Window C: this window is used to present diagram set up in Window A




Window 2-930

## Window 2-931: Dialog boxes for selection of a nodal/element result

- Pressing button **Add nodal result** or **Add element results**, in Window A, one gets an access to the dialog box(es) which allow to select a quantity to be analyzed and to activate graphical cursor for selection of a single nodal point or an element
- To activate graphical cursor press button **Pick**), then select a node, or an element, from Window B, and then press button **Set** in the dialog box to accept the choice

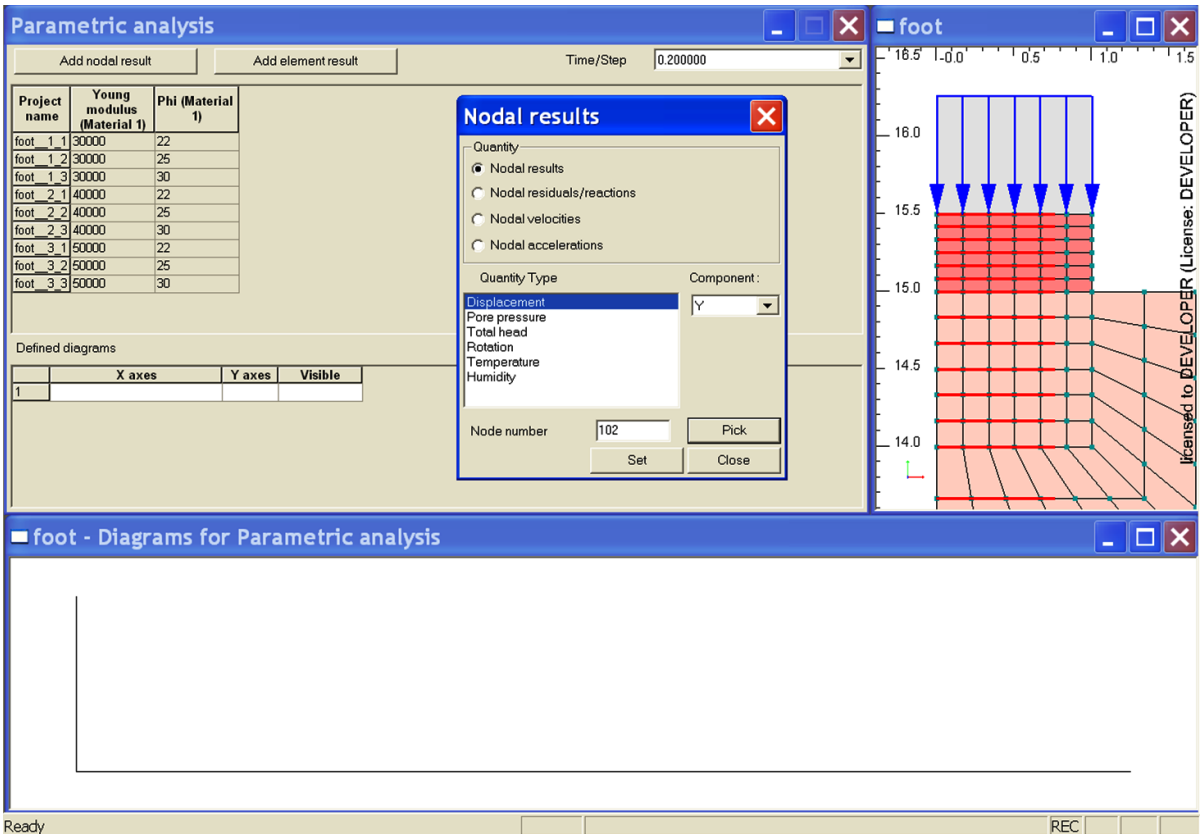



Window 2-931

## Window 2-932: Adding new nodal/element results

To add a new nodal/element result to the list (in Window A) make the following steps:

1. To select quantity and nodal point/element please follow the steps described in Win.(2-931)
2. The newly added result will be placed in the new added column
3. To change the time step unroll the combo box **Time step** [ ] and select demanded value




**Remark**:

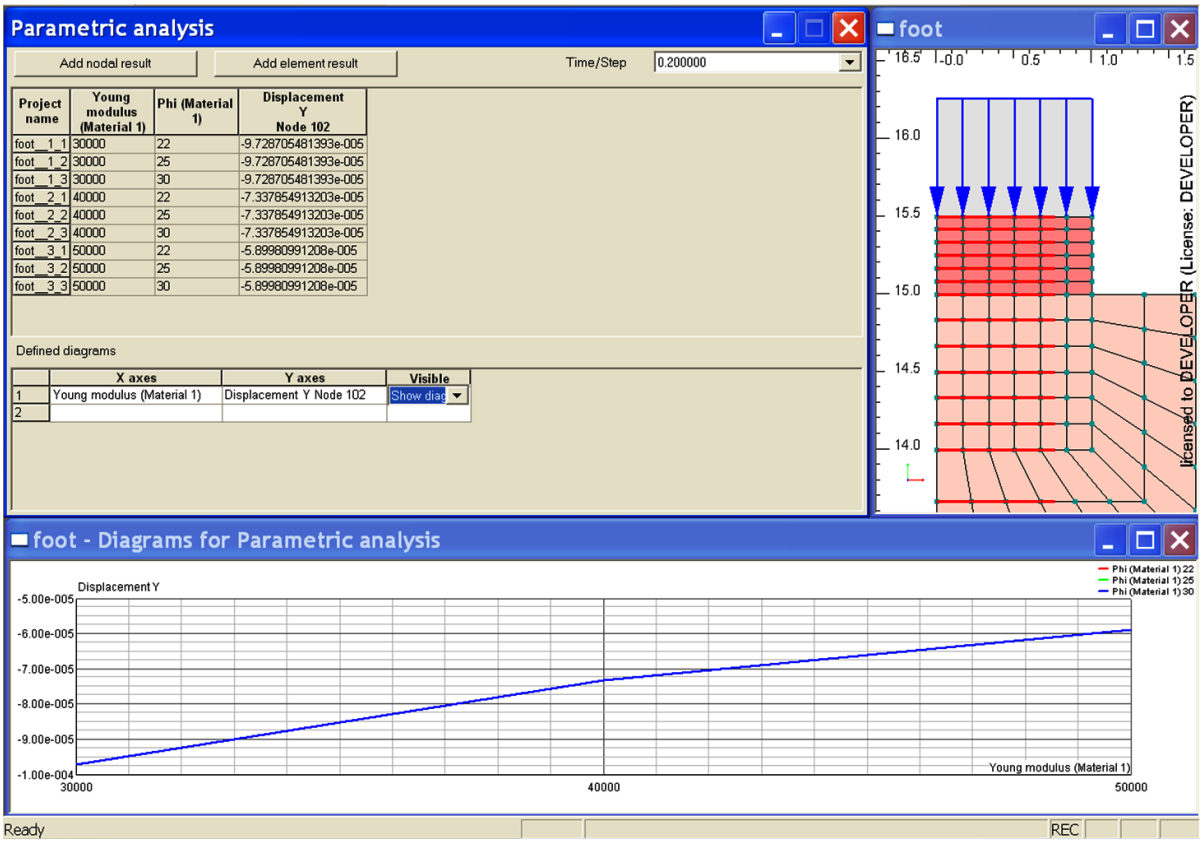
1. To copy the content of the sheet to the excel select columns in Window A and then paste with standard Ctrl+V tool in the Excel application

Window 2-932

## Window 2-933: Creating diagrams

To show the diagram (in Window C) make the following steps:

1. Add demanded results to the sheet (see Win.(2-932))
2. To set up the diagram perform the following steps:
   - Place cursor in the first column of the first free row, unroll the combo box and select material property which will be put in the x-axis
   - Place cursor in the second column of the first free row, unroll the combo box and select nodal/element quantity which will be plot on the y-axis
   - Set the status of diagram by placing cursor in the third column of the first free row and select one of the options (Show/Hide)
   - The new diagram will be plot in Window C




Window 2-933

## 2.6 EXTRAS

| | |
|---|---|
| `Run Analysis` | Calls computational module without storing job file (*.inp) nor creating *.dat file. Use for *.dat file modified by hand |
| `Write *.dat` | Creates explicit data file without calling computation |
| `View *.dat` | Edits *.dat file (only for very advanced users but in case of hand corrections hotline is not foreseen)) |
| `View *.inp` | Edits *.inp user data file (in case of hand corrections hotline is not foreseen) |
| `View log file *.log` | Edits *.log file which contains echo of computation plus detected errors marked as warnings or fatal errors |
| `Units conversion` | Invokes a dialog box to handle units conversion |
| `Databases` | Gives an access to view:<br>• Databases containing steel profiles (in Excel format)<br>• User created material databases (see Window 2-818) |
| `Protection report` | Generates protection report |
| `License` | Handles license upgrades |

## 2.7 SYSTEM CONFIGURATION

| | |
|---|---|
| `Version type` | User may decide whether to use ☒ **Advanced** or ☒ **Basic** version of the program; even in basic version one may switch temporarily (at the material level for instance) to the advanced mode; in addition this setting can be assumed to be a default for all new projects |
| `Help directory` | Using this menu user can attach one of all installed Z_Soil manuals; this menu is invoked automatically when opening the help for the first time |
| `Acrobat Reader path` | Acrobat Reader is assumed as a default *.pdf viewer used in Z_Soil; to change the viewer use this option and indicate path to the PDF viewer application |
| `Text file editor` | Notepad.exe text editor is assumed as a default editor for text files; to change the text files editor use this option and indicate path to the application |
| `*.CSV file editor` | Excel.exe application is assumed as a default editor to handle *.csv files produced by postprocessor; to change the editor use this option and indicate path to the application |
| `User's supplied developments` | This option allows to set paths to all major Z_Soil applications:<br>calculation module, preprocessor and postprocessor; this option is used mainly to manage custom Z_Soil versions and to replace standard applications by delivered ones. This option allows the user to plug his own developed modules for constitutive models for continuum and modules for an additional graphical output |
| `Preferences` | This option allows to customize user interface for the preprocessor and postprocessor, according to the user requirements; see Window 2-114 |

## 2.8 HELP

| | |
|---|---|
| `Quick help` | View Quick help (on line) manual |
| `User manual contents` | View the PDF page with all Z_Soil manuals |
| `Tutorials` | View manual with tutorials |
| `Getting started` | View short example on how to use Z_Soil |
| `White paper` | View the technical paper on abilities of Z_Soil code |
| `Hardware and software` | View hardware and software Z_Soil requirements |
| `Technical documentation` | View technical documents on:<br>• output files syntax<br>• user supplied constitutive models for continuum |
| `About Z_Soil` | View dialog box with information on Z_Soil |

# Chapter 3

# TROUBLESHOOTING

## 3.1 CALCULATION MODULE

Errors and warning handling

Frequently asked questions

### 3.1.1 ERRORS AND WARNINGS HANDLING

All errors and/or warnings reported by the calculation module are stored in the *.log file. After completion of computation one should review this file.

In many situations in the log file you will find that an error occurs for an element with number ..... It is not straightforward to identify this element as there is no correspondence between element numbers in the preprocessor and calculation module. The only way is to identify this element directly in the *.dat file, which is a kind of compiled input file for the calculation module, then to identify its nodes from given list. Having these nodes we can return to the preprocessor and highlight these nodes. As the result we will see the element generating reported error/warning.

To do that please run *Extras/View dat file,* and search for keyword ELEM which opens list of elements. The data syntax after this keyword is as follows:

**Window 3-1: ELEMent syntax in *.DAT file**

'ELEM' nElem
do i =1, nElem
Single Element description line(s)
end do
where:

- Element opening line :
  'ELEM' - Key word
  nElem - total number of elements in the system
- Single Element description line(s):

  ---

  iEle,Etyp,iProp,<'L_REC',nRec>,(Lnodes[i],i=1,Nen) (I10,A5,I10,A5,I10,Nen*I10)
  <ListOfLoads[i],i=1,nRec>(16*I10)

  ---

  where:

  - ⋆ iEle - element nr (this is dummy data)
  - ⋆ Etyp- element class name (like Q4,Q4ES,C_L2,C_Q4,SXQ4 etc...)
  - ⋆ iProp - index of property record attached to this element
  - ⋆ <'L_REC', nRec >
    - optional , put if any load is attached to element otherwise
  - ⋆ leave 10 blanks
  - ⋆ 'L_REC'- key word
  - ⋆ nRec - number of load records attached to this element

★ (Lnodes[i],i=1,Nen) - list of element node numbers where Nen is set by program

after recognizing of Etyp.

★ <ListOfLoads[i],i=1,nRec>(16*I10)

if 'L_REC' was specified then from the beginning of the next line(s) , list of load

record indices should be given . It may be continued, in format (16*I10), in

following lines.

The following errors and warnings can be reported:

- **CFG** *Cannot open configuration file for ZSOIL –>zsoil.cfg*

  The configuration file zsoil.cfg (in the Z_SOIL root directory) has been damaged.

  Check the existence of this file in your Z_SOIL directory and if it does not exists reinstall your Z_SOIL system.

- **DATA** *Data inconsistency has been detected ( see echo of data )*

  Some data inconsistencies have been detected. Please have a look to the *.eda file which covers echo of the input data.

- **DATA** *To this element material model of inconsistent type is attached*

  ..... (here some element number)

  A wrong association of the material model and element class has been detected. You can identify this element in the same way as for negative or zero Jacobian case.

- **DATA** *Nodes of DSC element not found at this beam element:*

  ..... (here beam element number)

  A special element, simulating relaxation/continuity between given beam element and the other, has unattached node(s).

- **DATA** *Inadmissible material data group:*

  .....(here given group string)

  This group of parameters is not allowed for material.

- **DATA** *The seepage element is unattached to any element or is attached to the element with impermeable material; this may result in computation failure; If so delete in the preprocessor all seepage elements attached to elements with impermeable materials.*

  .....(here given element number)

  The explanation is given in the description of the warning.

- **DATA** *This contact element is unattached to any element or is connected with elements with impermeable materials; this may result in computation failure for flow or def.+flow problems. If so make the contact material also impermeable.*

  .....(here given element number). The explanation is given in the description of the warning.

- **DATA** *Following shell node is assigned as slave to more than 1 master*

  *Use shell/direction show and flip in PrePro3D*

  .....(here given node number)

  The explanation is given in the description of the warning.

- **DATA** *Following shell node is assigned as both slave and master. Use shell/direction show and flip in PrePro3D*

  The explanation is given in the description of the warning.....(here given node number)

- **DATA** *There no exists any element attached to this contact element*

  A "flying" contact element has been detected. Please verify the existence of this element with respect to the neighboring elements (continuum/ shells/membranes/beams).

- **DATA** *No direction specified for infinite layer boundary element* Return to Pre-Pro and define correct vector

- **DATA** *For this variable thickness shell element ZERO thickness detected* Return to Pre-Pro and define the thickness correctly

- **DATA** *Too large gap for contact element*

  ..... (here given contact element number)

  A contact segment to segment element has been defined in such a way that corresponding nodes on the opposite faces do not coincide. This is a warning which does not break the computation. A tolerance for the geometrical gap has been assumed to be 0.05 * averaged element size. You can identify this element in the *.dat file (contact elements are denoted as C_L2 in 2D and C_Q4 in 3D) similarly as for case of zero or negative Jacobian.

- **DATA** *The internal node of penalty element is not inside domain*

  An element enforcing compatibility between two nonconforming continuum elements is wrongly defined.

- **DATA** *Impossible to realize assumed prestress (probably Stress >Yield-Stress )*

  The assumed prestress in anchor element is possibly higher than yield stress.

- **DATA** *This element reveal 0 or negative Jacobian*

  ..... (here element number)

  The indicated element reveals zero or negative Jacobian. This may happen only if the element is highly distorted. Please identify the element in the *.dat file, as described in the introductory part, and then return to the preprocessor to correct the data.

- **DATA** *Gauss point radial coordinate <= 0.0 in axisymmetry*

  In the axisymmetric case any x-coordinate has to be greater than zero.

  Return to the preprocessor and verify your data.

- **DATA** *Invalid nr of layers in composite property set no. :*

  Nr of layers exceeds allowable maximum. Try to reduce nr of layers in composite section.

- **DATA** *Fiber direction undetermined in layered shell element no. :*

  .....(here is given element number)

  It is possible that the fiber direction vector is perpendicular to the shell midsurface.

- **DATA** *Unconnected point outside demanded element in link element no. :*

  ..... (here is given element number)

  One of the anchor element endpoints is unconnected.

  Please look to the *.dat file (through View dat file option), find the element in the element list after keyword ELEM. Note that there is no correspondence between element numbers

in the preprocessor and calculation module. The only way is to identify all nodes of the element in the *.dat file and then to select them in the preprocessor.

- **DATA** *For this beam element local base is undetermined due to incorrect position of directional point.*

  .....(here is given element numbers)

  A direction for given beam element is possibly parallel to the element axis. Return to the preprocessor and correct it.

- **I/O** *Cannot read/write file with optimized node/ele sequence*

  This error may appear in the following cases:

  a) you tried to restart the analysis without having at least one converged step,

  b) the auxiliary file *.opt has been deleted

  To eliminate this error again Run Analysis

- **I/O** *Error during file I/O operation*

  ..... (here file no.)

  An I/O error has occurred during read/write operation.

  Your free disk space is possibly exhausted.

- **I/O** *Lack of ZSOIL.TXT file*

  The file with system strings zsoil.txt (in the Z_SOIL root directory) has been damaged.

  Check the existence of this file in your Z_SOIL directory and if it does not exists reinstall your Z_SOIL system.

- **I/O** *Error while writing this binary result*

  ..... (here file number)

  An I/O error has occurred during results storage. Your free disk space is possibly exhausted.

- **PIVOT** *Null pivot detected at :*

  .....(here there should be given node no. and degree of freedom)

  There are following reasons which lead to such a situation:

  a) insufficient number of boundary conditions

  b) separation state along the whole contact interface (note that the effective stress concept in contact holds),

  c) a limit state has been reached,

  d) boundary conditions are set in such a way that rigid body motion is not eliminated. In this case set proper BC.

- **RAM** *Length of equation is greater than length of block*

  Try to enlarge the RAM size to be used by Z_SOIL in System configuration

- **RAM** *Too many blocks in given equation*

  Try to enlarge the RAM size to be used by Z_SOIL in System configuration. Current RAM size is insufficient for skyline block solver.

- **RAM** *Lack of free RAM to run a job (Note : RAM_MAXIMUM >64 kB in zsoil.cfg !!) Try to close other running applications .*

  The amount of the RAM size in System configuration has to be greater than 64 kb. It is possible that the declared RAM size is too large while concurrently using other Windows applications.

- **RAM** *Dynamic memory allocation error; try to reduce the RAM size in zsoil.cfg to the following size :*

  Please reduce your RAM size in System configuration to given value.

- **RAM** *Lack of memory to allocate frontal solver*

  Your working memory space has been exhausted. Try to enlarge RAM size in System configuration.

- **RAM** *Lack of free space detected while allocating this*

  In *System configuration* there is an edit box to define maximum RAM available for the calculation module. However it may happen that for large problems assumed RAM size is insufficient. Please enlarge it through System configuration but never exceed your platform limitation.

- **RUN** *Step not converged due to limitation of the absolute maximum nr of iterations*

  The maximum number of iterations has been overreached. If you want to enlarge this number please do it through Control or try to reduce time step/ increment of safety factor or gravity multiplier.

- **RUN** *Divergence of iteration process - can not continue calculations*

  Calculation process has been stopped due to divergence or lack of convergence.

  Try to reduce gravity multiplier increment in the initial state analysis / time step for time dependent drivers or safety factor increment for stability analysis and then restart.

- **RUN** *Restart is impossible*

  Please Run Analysis as the restart is impossible.

If appearing error message does not belong to the above list, or the message persistently appears despite an attempt to correct it, please contact hotline ZSOILZace.com

### 3.1.2 FREQUENTLY ASKED QUESTIONS

# Index


3D
  analysis and drivers, DP: 27, 28, DP: 93

Analysis
  batch processing, DP: 685
  restart computation, DP: 684
  run computation, DP: 683
  run computation without writing *.dat, DP: 698
Auxiliary planes, DP: 142
Axisymmetry
  analysis and drivers, DP: 27, 28, DP: 93

Beams
  create/select/update/delete elements, DP: 297
  create/select/update/delete subdomain 2D, DP: 178
  create/select/update/delete subdomain 3D, DP: 178
  Reinforcement sets, DP: 682
  reinforcements set, DP: 304, DP: 310
  subdomain 2D parameters, DP: 197
  subdomain 3D parameters, DP: 197
Bearing capacity, DP: 42–44
Boundary conditions
  for humidity, DP: 244, DP: 413
  for humidity on macro-elements, DP: 244
  for pore pressure, DP: 237, DP: 414
  for pore pressure on macro-elements, DP: 237
  for solid phase, DP: 401
  for temperature, DP: 241, DP: 408
  for temperature on macro-elements, DP: 241
  periodic, DP: 421

Construction algorithm
  analysis and drivers, DP: 28
Continuum 2D elements, DP: 323
Continuum 2D macromodel
  Automatic mesh generation, DP: 212, 213
  Mesh morphing, DP: 214
  Semi-automatic mesh generation, DP: 208, 209
  Subdomain generation, DP: 177, 178
  Virtual mesh, DP: 207–210
Continuum 3D elements, DP: 329
Continuum 3D macromodel
  Mesh morphing, DP: 215
  Semi-automatic mesh generation, DP: 210, 211
  Subdomain generation, DP: 179–182, DP: 184, DP: 186, DP: 188
  Virtual mesh, DP: 210, 211
Convection 2D macromodel
  Subdomain generation, DP: 225
Convection 3D macromodel
  Subdomain generation, DP: 225
Convection elements, DP: 388
Creep
  standard properties, DP: 586
  swelling properties, DP: 586

Dynamics
  added masses, DP: 454
  consistent mass matrix, DP: 98
  control parameters, DP: 98
  Domain Reduction Method (DRM), DP: 512
  HHT scheme, DP: 98
  lumped mass matrix, DP: 98
  mass filtering, DP: 100
  Newmark scheme, DP: 98
  Reylaigh damping, DP: 98
  Seismic input, DP: 680

Elasticity
  constants, DP: 19
  material properties, DP: 579

Elasto-plastic 1D
model, DP: 656
Evolution functions, DP: 676
Existence functions, DP: 670

FE model preprocessing
common methods to copy elements/nodes, DP: 282
common methods to delete elements/nodes, DP: 282
common methods to move elements/nodes, DP: 282
common methods to rotate elements/nodes, DP: 282
common methods to select elements/nodes, DP: 282
Finite elements
selection strategy, DP: 108, DP: 110
stabilization of pressure oscillations, DP: 110
volumetric locking, DP: 109
Flow
analysis, DP: 27, 28, DP: 62
flux boundary condition, DP: 63, DP: 247
initial conditions, DP: 63
initial state driver, DP: 62, DP: 65
material data, DP: 62, DP: 582
pressure boundary condition, DP: 63, DP: 237
steady state driver, DP: 62, DP: 66
time dependent drivers, DP: 66
transient driver, DP: 62, DP: 67
Fluxes on elements
fluid, DP: 477
heat, DP: 480
humidity, DP: 481
Fluxes on macromodel
fluid, DP: 247
heat, DP: 250
humidity, DP: 254

Gravity, DP: 678
body load components, DP: 678
direction, DP: 678

Heat
analysis, DP: 27, 28, DP: 68
flux boundary conditions, DP: 70, DP: 250
initial conditions, DP: 70, 71
initial state driver, DP: 68, DP: 71
material properties, DP: 593
steady state driver, DP: 68, DP: 72
temperature boundary conditions, DP: 70, DP: 241
thermal strains, DP: 94
time dependent drivers, DP: 68, DP: 72
transient driver, DP: 68, DP: 73
Heat exchanger 3D macromodel
macro-element generation, DP: 219
Hinges
in beam elements, DP: 303, DP: 659
in shell elements, DP: 397, DP: 662
Humidity
analysis, DP: 27, 28, DP: 74
flux boundary conditions, DP: 75, DP: 254
humidity boundary conditions, DP: 75, DP: 244
hygral strains, DP: 74, DP: 94
initial conditions, DP: 75, 76
initial state driver, DP: 74, DP: 76
material properties, DP: 74, DP: 596
steady state driver, DP: 77
time dependent drivers, DP: 74, DP: 77
transient driver, DP: 78

Infinite elements, DP: 356
Initial condition
for displacements, DP: 506
for humidity, DP: 490, DP: 493
for pore pressure, DP: 483
for solid velocities, DP: 506
for temperature, DP: 487
Initial state Ko
material properties, DP: 590
Interface 2D macromodel
Subdomain generation, DP: 231
Interface 3D macromodel
Subdomain generation, DP: 231
Interface elements, DP: 364
Interface elements for large deformations, DP: 378

Kinematic constraints, DP: 514

Large deformations

analysis and drivers, DP: 28
Linear equation solvers, DP: 103
skyline, DP: 103
sparse, DP: 103
Load time functions, DP: 672
Loads
body, DP: 426
on beam elements, DP: 437
on element surfaces, DP: 430
on elements, DP: 439, DP: 444, DP: 449
on nodes, DP: 428
on subdomain boundaries, DP: 270

Macromodeling
2D mesh mapping, DP: 170
Bore holes, DP: 523
Extrusion direction, DP: 173
Fluid flux, DP: 247
Heat flux, DP: 250
Humidity BC, DP: 244
Humidity flux, DP: 254
Movement path, DP: 528
Nails, DP: 263
Objects, DP: 160
Piles, DP: 257
Point, DP: 158
Point loads, DP: 278
Pressure BC, DP: 237
Subdomain, DP: 174
Subdomain parameters, DP: 196
Surface load, DP: 270
Temperature BC, DP: 241
Virtual to real mesh conversion, DP: 215
Macromodeling objects
Arc, DP: 163
Circle, DP: 164
common methods, DP: 161
Delete methods, DP: 169
DXF import, DP: 168
Line, DP: 162
Line(s) on edge(s), DP: 164
Point, DP: 162
Select methods, DP: 169
Split by plane, DP: 168
Surface intersection, DP: 165, DP: 216
Surface on Q4 skeleton, DP: 165
Surface on T3 skeleton, DP: 165
Update methods, DP: 170
Main Z_Soil menu
Analysis options, DP: 683
Assembly options, DP: 113
Control options, DP: 25, 26
Control:analysis and drivers, DP: 28
Extras, DP: 698
File options, DP: 23
Help, DP: 700
Postpro, DP: 686
System configuration, DP: 9, DP: 699
toolbars, DP: 22
Materials
creep properties group, DP: 586
data groups, DP: 579
databases, DP: 536
elastic properties group, DP: 579
flow properties group, DP: 582
handling, DP: 534
heat properties group, DP: 593
humidity properties group, DP: 596
initial Ko properties group, DP: 590
local stability properties group, DP: 597
models, DP: 536
properties varying in space, DP: 664
properties varying in time, DP: 664
Rayleigh damping parameters group, DP: 598
unit weight properties group, DP: 580
Materials for axisymmetric shell elements
linear elastic model, DP: 549, DP: 553
nonlinear (layered) model, DP: 549, DP: 553
Materials for beam elements, DP: 538
linear elastic model, DP: 539, DP: 543
nonlinear (layered) model, DP: 539, DP: 544, DP: 546
Materials for contact elements, DP: 555
frictional contact model, DP: 556
Materials for continuum elements, DP: 578
Aging concrete model, DP: 600
Cap model, DP: 602
Densification model, DP: 623
Drucker-Prager model, DP: 608
Hoek-Brown model, DP: 628
Hoek-Brown(M-W) model, DP: 609
HS-small model, DP: 618
Linear elastic model, DP: 599
Modified Cam Clay model, DP: 605

Mohr-Coulomb (M-W) model, DP: 611
Mohr-Coulomb model, DP: 616
Multilaminate model, DP: 613
Plastic damage for concrete, DP: 632
Rankine(M-W) model, DP: 615
Materials for continuum elements for structures, DP: 636
Materials for fixed anchor interface elements
adhesive contact model, DP: 575
Materials for fixed anchor zone interface elements, DP: 574
Materials for heat convection elements
convection model, DP: 636
Materials for heat radiation elements
radiation model, DP: 637
Materials for humidity convection elements
convection model, DP: 638
Materials for infinite elements
Linear elastic model, DP: 639
Materials for membrane elements, DP: 640
Anisotropic elasto-plastic model, DP: 645
Fiber elasto-plastic model, DP: 641
Isotropic elasto-plastic model, DP: 645
Plane stress elastic model, DP: 644
Plane stress Hoek-Brown model, DP: 644
Plane stress Huber-Mises model, DP: 644
Plane stress Rankine model, DP: 644
Materials for nail interface elements, DP: 570
adhesive contact model, DP: 571
Materials for pile interface elements, DP: 562
frictional contact model, DP: 563, DP: 568
Materials for pile tip interface elements, DP: 567
Materials for seepage elements, DP: 647
Materials for shell elements, DP: 649
Linear elastic model, DP: 651
Nonlinear (layered) model, DP: 651–653
Orthotropic elastic model, DP: 655, 656
Membrane 2D macromodel
Subdomain generation, DP: 178
Membrane 3D macromodel
Subdomain generation, DP: 194
Membranes, DP: 350
Mesh tying, DP: 519

Nodal link, DP: 516
Node, DP: 290
Nonlinear solvers, DP: 96

Overlaid meshes
generation, DP: 151

Plane strain
analysis and drivers, DP: 27, 28, DP: 93
Postprocessing
envelopes for structural elements, DP: 692
how do I...., DP: 688
using macros, DP: 690
Preferences, DP: 156
Preprocessing
FE model, DP: 115, DP: 281
Macromodelling, DP: 115, DP: 156
Main menu, DP: 116
User interface, DP: 115
Preprocessor
construction lines, DP: 140
copy by rotation selected objects, DP: 137
copy by symmetry, DP: 137
copy by translation selected objects, DP: 136
grid, DP: 141
import geometrical model, DP: 120
move selected objects, DP: 135
rotate selected objects, DP: 136
selection of finite elements, DP: 132
selection of nodes, DP: 133
show distance, DP: 149
show node coordinates, DP: 149
show vector, DP: 149
show volume of continuum element, DP: 149
snap options, DP: 141
Preprocessor menu
Assembly, DP: 123
Draw, DP: 138
Edges/Faces selection, DP: 127
Edit, DP: 134
Files, DP: 119
Frequently used tools, DP: 151

Global selection tools, DP: 130
Mesh, DP: 146
Overlaid meshes, DP: 149
Parameters, DP: 137
Settings, DP: 154
Show construction/excavation steps, DP: 153
Tools, DP: 148
Undo, DP: 124
View, DP: 125
Windows, DP: 120
Preprocessor toolbars
2D mesh refinement, DP: 147
3D mesh refinement, DP: 148
edges selection, DP: 129
faces selection, DP: 128
Pushover
control parameters, DP: 100
results, DP: 687
Pushover control node, DP: 521

Restart
control, DP: 96
Results
content, DP: 111
for beam elements, DP: 112
for continuum elements, DP: 112
for shell/membrane elements, DP: 112
nodal solid accelerations, DP: 113
nodal solid velocities, DP: 113
residuals at nodes, DP: 113
standard nodal results, DP: 113
storage frequency, DP: 96

Seepage 2D macromodel
Subdomain generation, DP: 216
Seepage 3D macromodel
Subdomain generation, DP: 216
Seepage elements, DP: 382
Shell 1L macromodel
Subdomain generation, DP: 190–193
Virtual mesh, DP: 211
Shell elements, DP: 336
Shell elements with one layer of nodes, DP: 343
Single phase
analysis, DP: 27, 28, DP: 33
driven load driver, DP: 40
effective stress analysis, DP: 33
initial state driver, DP: 36
time dependent drivers, DP: 40
total stress analysis, DP: 33, DP: 39, DP: 48
Stability, DP: 45
analysis, DP: 47, 48
driver, DP: 45–48
local material setting, DP: 46, DP: 597
Strains
imposed, DP: 501, DP: 509
thermal, DP: 68
Stress
sign convention, DP: 15
Subdomain
Excavation front, DP: 199
Project subdomain on surface, DP: 216
Swelling
material properties, DP: 586

Tendons
in continuum/shells, DP: 361
Truss 2D macromodel
Subdomain generation, DP: 178
Truss 3D macromodel
Subdomain generation, DP: 178
Truss elements, DP: 313
Two phase
analysis, DP: 27, 28, DP: 49
consolidation driver, DP: 56
driven load+steady state flow driver, DP: 54
driven load+transient flow driver, DP: 55
initial state driver, DP: 52
stability driver, DP: 58, 59
time dependent drivers, DP: 54
undrained driver, DP: 32, DP: 34, DP: 49, 50, DP: 80, DP: 583, 584

Unit weight
definitions, DP: 18
in single phase problems, DP: 33, DP: 39
in two phase problems, DP: 59
material properties, DP: 580
Units, DP: 104
basic, DP: 19, DP: 106
compound, DP: 107
conversion, DP: 105

setting basic units, DP: 104

Viscous damper 2D macromodel
Subdomain generation, DP: 228
Viscous damper 3D macromodel
Subdomain generation, DP: 228
Viscous dampers, DP: 393, DP: 648

Warnings, DP: 155