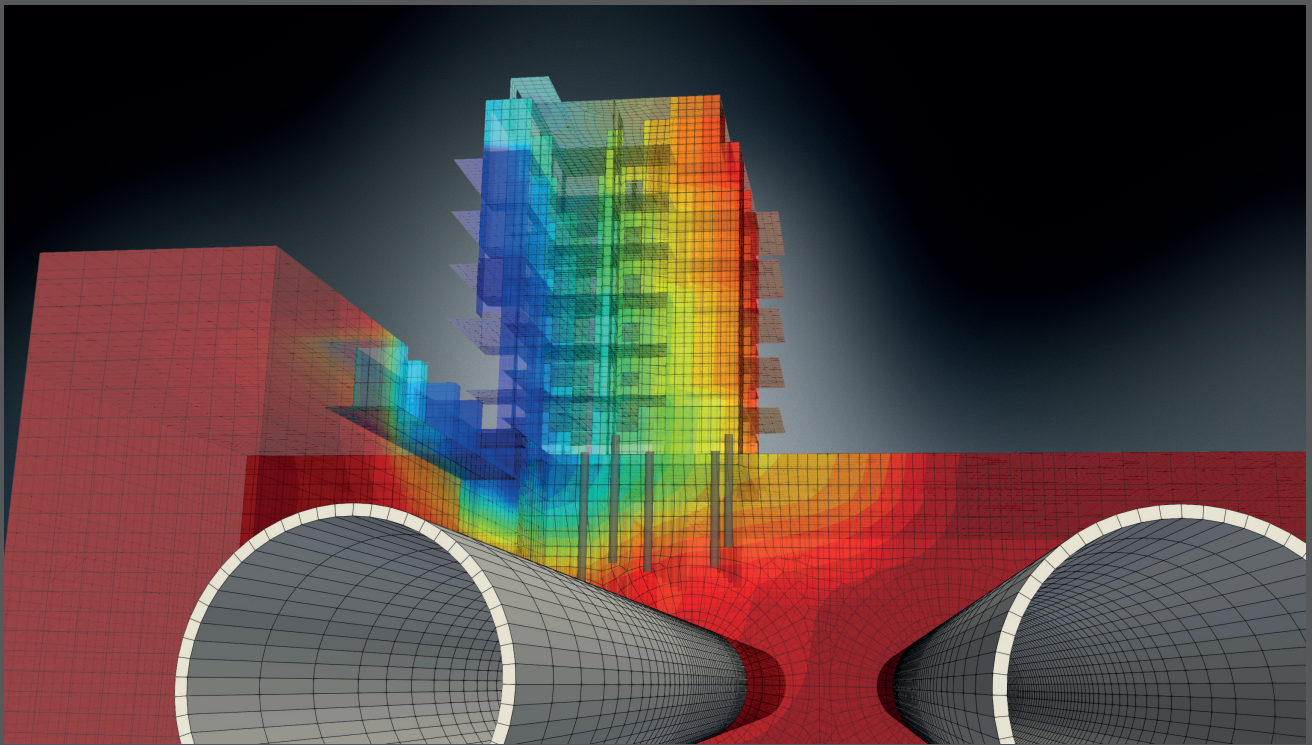




# ZSoil

for geotechnics & structures

## User manual QUICK HELP



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



for geotechnics & structures

---

## QUICK HELP

### User manual

ZSoil®2023

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 2023 manual:

1. Data preparation
2. Tutorials and benchmarks
3. Theory

ISBN 2-940009-08-2

©1985-2022 by ZACE Services Ltd  
©1985-2023 by GeoDev Sarl. All rights reserved.

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

**Applicable to all V2023 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 **N<sub>bought</sub>** (= 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 **N<sub>bought</sub>** 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 V2023** will be supported for a period of 3 years starting from date of purchase. This support is limited to **ZSOIL V2023** 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 **N<sub>bought</sub>** 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 1.11.2022**

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

# Chapter 1

## MENU CONTENTS DESCRIPTION

The ZSoil<sup>®</sup> main screen contains the following menu items:

- File** — Standard MS–Windows rules of handling files
- Control** — Control of computations, set up units, result contents
- Assembly** — Build up the computational model
- Analysis** — Run the analysis
- Results** — Visualize results in graphic or text form
- Extras** — Edit data files in the ASCII form, get an access to databases
- System Configuration** — Set up editors for text editing, preview help files
- Help** — Get an access to the online help

## 1.1 FILE

Standard MS–Windows rules of handling files are adopted. In particular:

New	Creates new job under the name "untitled <i>i</i> " where <i>i</i> is a subsequent number of the job during the session. Calls the dialog to select <b>Analysis Type</b> of the job
Open	Opens previously created job. Possible modification or edition of the result.
Save	Calls standard MS–Windows dialog to store created data and result (if exists).
Save As ...	Calls standard MS–Windows dialog to make a copy of created data and result under a new name.
Delete	Deletes Z_SOIL files (temporary data and result), but preserves job data file *.inp
Batch processing	Creates a file list for batch processing and stores it as a *.bat file. Also runs existing *.bat files
Exit	Calls the dialog to decide if to store the job at current stage and exits the system.

Moreover, file names list of up to 4 previous jobs is stored. Clicking on any file from the list will open indexed job, closing (and saving) the recent one.

## Related Topics

- *data preparation: Batch Processing*

## 1.2 CONTROL

Following options allow to control computational analysis

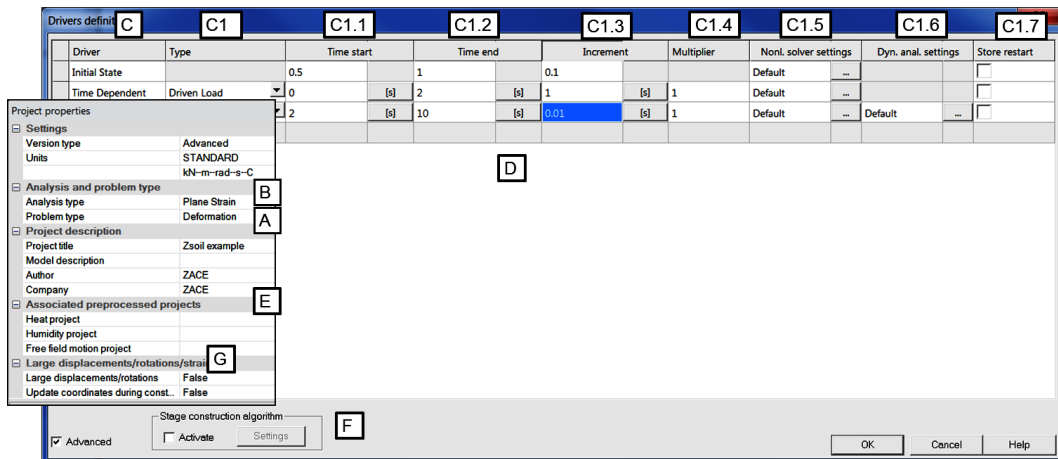
<b>Analysis &amp; Drivers ▶</b>	Sets a sequence of drivers to be run during computation
<b>Control ▶</b>	Sets type of the iterative scheme and corresponding convergence tolerances, frequency of restart and output files storage
<b>Dynamics ▶</b>	Creates list of settings for time history analysis to be associated with dynamic driver under <b>Analysis &amp; Drivers</b> menu
<b>Pushover ▶</b>	Creates list of settings to be associated with pushover drivers under <b>Analysis &amp; Drivers</b> menu
<b>Contact Algorithm ▶</b>	Sets algorithm control data for treatment of contact
<b>Linear equations solvers ▶</b>	Selects the linear equation solver
<b>Units ▶</b>	Sets unit system for input and output
<b>Result contents ▶</b>	Sets content of the resulting output files
<b>Finite elements ▶</b>	Sets type of the continuum elements to be used in computation

### Related Topics

- *data preparation: Analysis and drivers*
- *data preparation: Control*
- *data preparation: Dynamics*
- *data preparation: Pushover*



## 1.2.1 ANALYSIS & DRIVERS



	Key	Options	Comment
<b>A</b>	<b>Problem Type</b>	<b>Deformation</b> <b>Deformation+flow</b> <b>Flow</b> <b>Heat</b> <b>Humidity</b>	for single phase analysis for 2 phase analysis for single phase flow analysis for thermal diffusion analysis for humidity diffusion analysis
<b>B</b>	<b>Analysis</b>	3D Plane strain Axisymmetry	
<b>C</b>	<b>Driver</b>		Depends on <b>A</b>
<b>C1</b>	<b>Driver Type</b>		Depends on <b>C</b>
<b>C1.1</b>	<b>Start</b>		Context depends on <b>C1</b>
<b>C1.2</b>	<b>End</b>		Context depends on <b>C1</b>
<b>C1.3</b>	<b>Increment</b>		Context depends on <b>C1</b>
<b>C1.4</b>	<b>Multiplier</b>		Context depends on <b>C1</b>
<b>C1.5</b>	<b>Nonlinear solver settings</b>		Set algorithm , convergence tolerances, storage frequency
<b>C1.6</b>	<b>Dynamic anal.settings</b>		Set parameters for dynamic driver (algorithm, damping coeff.)
<b>C1.6</b>	<b>Pushover anal.settings</b>		Set parameters for pushover driver (force pattern, direction)
<b>D</b>	<b>Drivers list</b>		Displays the sequence of drivers
<b>E</b>	<b>Associated projects</b>		browse calculated thermal/hygral or free field motion project, to be used in reduced domain reduction model, if relevant

## DRIVERS DATA for Problem Type Deformation

Use for single phase deformation analysis.

See also [Data Preparation: SINGLE PHASE \(DEFORMATION\) ANALYSIS](#)

	Key	Options	Comment
<b>C</b>	Driver	<b>Initial State</b>	Use to compute initial stresses in soils or rocks including $K_o^{insitu}$ effect (it is run at time $t=0$ and all loads acting at that time are taken into account)
<b>C1</b>	Driver Type	none	
<b>C1.1</b>	Start		Initial value of the load factor (default 1.0, must be greater than 0.0)
<b>C1.2</b>	End		Final value of the load factor (default 1.0)
<b>C1.3</b>	Increment		Load factor increment
<b>C1.4</b>	Multiplier		Multiplier of load factor increment
<b>C</b>	Driver	<b>Time dependent</b>	Use to compute problems with variable in time geometries (excavations, stage construction), loads and boundary conditions; time can be a real parameter, if creep is enabled, or fictitious
<b>C1</b>	Driver Type	Driven load	
<b>C1.1</b>	Start		Time at which driver is switched on
<b>C1.2</b>	End		Time at which driver is switched off
<b>C1.3</b>	Increment		Initial time step increment
<b>C1.4</b>	Multiplier		Multiplier of time step increment (meaningful mainly for creep analysis)

	Key	Options	Comment
<b>C</b>	<b>Driver</b>	<b>Stability</b>	Use to compute safety factor (SF) and failure pattern at present state (time) through successive reduction of the yield surface
<b>C1</b>	<b>Driver Type</b>	$C - \tan(\phi)$	Define SF as divisor of both cohesion and friction angle
		<b>Stress level</b>	Define SF as the maximum possible multiplier of deviatoric stress
		$C$	Define SF as divisor of cohesion exclusively
<b>C1.1</b>	<b>Start</b>		Initial value of safety factor
<b>C1.2</b>	<b>End</b>		Final value of safety factor
<b>C1.3</b>	<b>Increment</b>		Safety factor increment
<b>C1.4</b>	<b>Multiplier</b>		not meaningful (kept equal to 1.0)
<b>C</b>	<b>Driver</b>	<b>Dynamics</b>	Use to carry out time history analysis of structures (continuum elements cannot be used !)
<b>C1</b>	<b>Driver Type</b>	<b>Driven load</b>	
<b>C1.1</b>	<b>Start</b>		Time at which driver is switched on
<b>C1.2</b>	<b>End</b>		Time at which driver is switched off
<b>C1.3</b>	<b>Increment</b>		Initial time step increment
<b>C1.4</b>	<b>Multiplier</b>		Multiplier of time step increment
<b>C</b>	<b>Driver</b>	<b>Pushover</b>	Use to estimate bearing capacities of structures (continuum elements cannot be used !) subject to seismic risk
<b>C1</b>	<b>Ctrl node label</b>	<b>user def.node</b>	Use to indicate a node (obligatory) to check target displacement
<b>C1.1</b>	<b>Start</b>		Initial displacement increment
<b>C1.2</b>	<b>End</b>		Target displacement
<b>C1.3</b>	<b>Increment</b>		Displ. increment reduction factor
<b>C1.4</b>	<b>Multiplier</b>		Number of reduction steps

#### Remarks:

1. Uncoupled effective and total stress analyses are possible within this mode for situations with a horizontal water table; to enable effective stress analysis it is sufficient to set soil unit weight to  $\gamma = \gamma_{dry}$  above and  $\gamma = \gamma_{sat} - \gamma^F$  below the water table; for total stress analysis it is necessary to set soil unit weight to  $\gamma = \gamma_{dry}$  above and  $\gamma = \gamma_{sat}$  below the water table and to specify the initial pore pressure in the fully saturated zone using **Initial conditions-Initial pressure** option in the preprocessor

2. Pushover driver automatically reduces (**C1.4**) times the load step (multiplying it by the factor **C1.3**) in case when divergence of the iterative procedure is achieved; after (**C1.4**) trials computation is stopped
3. The incremental treatment of selected stage construction steps can be activated by switching ON the option ☐ **Activate** in the **Stage construction algorithm** groupbox (**F**) (parametrization of this option is similar to the **Initial state** driver)
4. Large displacements/rotations option (**G**) enables geometrically nonlinear analysis using so-called corrotational formulation once the ☐ **Activate** is set ON; this option is fully automatic and requires redefinition of the contact interfaces only (if present); in the large deformations node-segment approach is used to model frictional interfaces; in the large deformations mode it is also possible to activate updating of the coordinates for stage construction algorithm (see the dedicated report on large deformations for more details)

## DRIVERS DATA for Problem Type Deformation+Flow

Use for two phase deformation analysis.

See also [Data Preparation: DEFORMATION COUPLED WITH FLOW](#)

	Key	Options	Comment
<b>C</b>	<b>Driver</b>	<b>Initial State</b>	Use to compute initial pore pressures and initial stresses in soils or rocks including $K_o^{insitu}$ effect (it is run at time $t=0$ and all loads acting at that time are taken into account); initial pressures are computed assuming steady state conditions at time $t=0$
<b>C1</b>	<b>Driver Type</b>	none	
<b>C1.1</b>	<b>Start</b>		Initial value of the load factor (default 1.0, must be greater than 0.0)
<b>C1.2</b>	<b>End</b>		Final value of the load factor (default 1.0)
<b>C1.3</b>	<b>Increment</b>		Load factor increment
<b>C1.4</b>	<b>Multiplier</b>		Multiplier of load factor increment
<b>C</b>	<b>Driver</b>	<b>Time dependent</b>	Use to compute problems with variable in time geometries (excavations, stage construction), loads and boundary conditions; time can be a real parameter, if creep or consolidation is enabled, or fictitious
<b>C1</b>	<b>Driver Type</b>	Driven load+Steady state	Uncoupled total stress analysis; each step consists of the steady state fluid flow evaluation followed then by a computation of deformation
		Driven load+Transient	Weakly coupled total stress analysis; each step consists of the transient fluid flow evaluation followed then by a computation of deformation
		Undrained Consolidation	Undrained total stress analysis Fully coupled total stress analysis
<b>C1.1</b>	<b>Start</b>		Time at which driver is switched on
<b>C1.2</b>	<b>End</b>		Time at which driver is switched off
<b>C1.3</b>	<b>Increment</b>		Initial time step increment
<b>C1.4</b>	<b>Multiplier</b>		Multiplier of time step increment (meaningful mainly for consolidation or creep analysis)

	Key	Options	Comment
<b>C</b>	<b>Driver</b>	<b>Stability</b>	Use to compute safety factor (SF) and failure pattern at present state (time) through successive reduction of the yield surface
<b>C1</b>	<b>Driver Type</b>	$C - \tan(\phi)$	Define SF as divisor of both cohesion and friction angle
		<b>Stress level</b>	Define SF as the maximum possible multiplier of deviatoric stress
		$C$	Define SF as divisor of cohesion exclusively
<b>C1.1</b>	<b>Start</b>		Initial value of safety factor
<b>C1.2</b>	<b>End</b>		Final value of safety factor
<b>C1.3</b>	<b>Increment</b>		Safety factor increment
<b>C1.4</b>	<b>Multiplier</b>		not meaningful (kept equal to 1.0)

**Remark:**

1. For this analysis type only total stress analysis is possible so unit weight of soil is always set as for dry state  $\gamma = \gamma_{dry}$  regardless of the saturation degree (this is automatically handled by the calculation module).
2. The incremental treatment of selected stage construction steps can be activated by switching ON the option ☒ **Activate** in the **Stage construction algorithm** groupbox (**F**) (parametrization of this option is similar to the **Initial state** driver)
3. Large displacements/rotations option (**G**) enables geometrically nonlinear analysis using so-called corrotational formulation once the ☒ **Activate** is set ON; this option is fully automatic and requires redefinition of the contact interfaces only (if present); in the large deformations node-segment approach is used to model frictional interfaces; in the large deformations mode it is also possible to activate updating of the coordinates for stage construction algorithm (see the dedicated report on large deformations for more details)

## DRIVERS DATA for Problem Type Flow

Use for single phase Darcy flow analysis.

See also [Data Preparation: FLOW \(STEADY STATE AND TRANSIENT\)](#)

	Key	Options	Comment
<b>C</b>	Driver	<b>Initial State</b>	Use to compute initial pore pressures assuming steady state conditions at time $t=0$
<b>C1</b>	Driver Type	none	<b>C1.1÷C1.4</b> are all not meaningful
<b>C1.1</b>			
<b>C1.2</b>			
<b>C1.3</b>			
<b>C1.4</b>			
<b>C</b>	Driver	<b>Time dependent</b>	Use to compute problems with variable in time geometries (excavations, stage construction) and boundary conditions; time is a real parameter for transient and fictitious for the steady state driver type
<b>C1</b>	Driver Type	<b>Steady state</b>	Sequence of steady state solutions of Darcy flow
		<b>Transient</b>	Evolution of the fluid pressure and velocity fields in time due to transient boundary conditions
<b>C1.1</b>	Start		Time at which driver is switched on
<b>C1.2</b>	End		Time at which driver is switched off
<b>C1.3</b>	Increment		Initial time step increment
<b>C1.4</b>	Multiplier		Multiplier of time step increment (meaningful mainly for transient case)

## DRIVERS DATA for Problem Type Heat

Use for heat transfer analysis.

See also [Data Preparation: HEAT TRANSFER](#)

	Key	Options	Comment
<b>C</b>	Driver	<b>Initial State</b>	Use to compute temperature field assuming steady state conditions at time $t=0$
<b>C1</b>	Driver Type	none	<b>C1.1÷C1.4</b> are all not meaningful
<b>C1.1</b>			
<b>C1.2</b>			
<b>C1.3</b>			
<b>C1.4</b>			
<b>C</b>	Driver	<b>Time dependent</b>	Use to compute problems with variable in time geometries (excavations, stage construction) and boundary conditions; time is a real parameter for transient and fictitious for the steady state driver type
<b>C1</b>	Driver Type	<b>Steady state</b>	Sequence of steady state solutions of heat transfer
		<b>Transient</b>	Evolution of the temperature field due to transient boundary conditions
<b>C1.1</b>	Start		Time at which driver is switched on
<b>C1.2</b>	End		Time at which driver is switched off
<b>C1.3</b>	Increment		Initial time step increment
<b>C1.4</b>	Multiplier		Multiplier of time step increment (meaningful mainly for transient case)



## DRIVERS DATA for Problem Type Humidity

Use for humidity transfer analysis<sup>1</sup>.

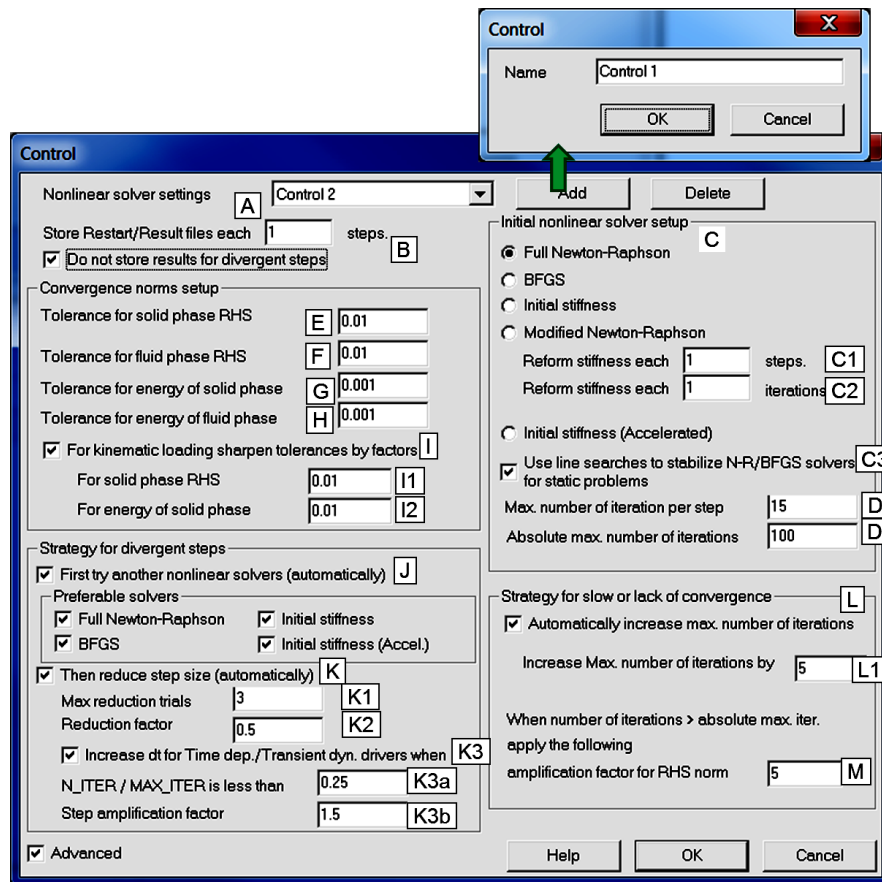
See also [Data Preparation: HUMIDITY TRANSFER](#)

	Key	Options	Comment
<b>C</b>	Driver	<b>Initial State</b>	Use to compute relative humidity field assuming steady state conditions at time $t=0$
<b>C1</b>	Driver Type	none	<b>C1.1÷C1.4</b> are all not meaningful
<b>C1.1</b>			
<b>C1.2</b>			
<b>C1.3</b>			
<b>C1.4</b>			
<b>C</b>	Driver	<b>Time dependent</b>	Use to compute problems with variable in time geometries (excavations, stage construction) and boundary conditions; time is a real parameter for transient and fictitious for the steady state driver type
<b>C1</b>	Driver Type	<b>Steady state</b>	Sequence of steady state solutions of humidity diffusion
		<b>Transient</b>	Evolution of the relative humidity field due to transient boundary conditions
<b>C1.1</b>	Start		Time at which driver is switched on
<b>C1.2</b>	End		Time at which driver is switched off
<b>C1.3</b>	Increment		Initial time step increment
<b>C1.4</b>	Multiplier		Multiplier of time step increment (meaningful mainly for transient case)

<sup>1</sup>concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT** only

## 1.2.2 CONTROL

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



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 tolerance 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)

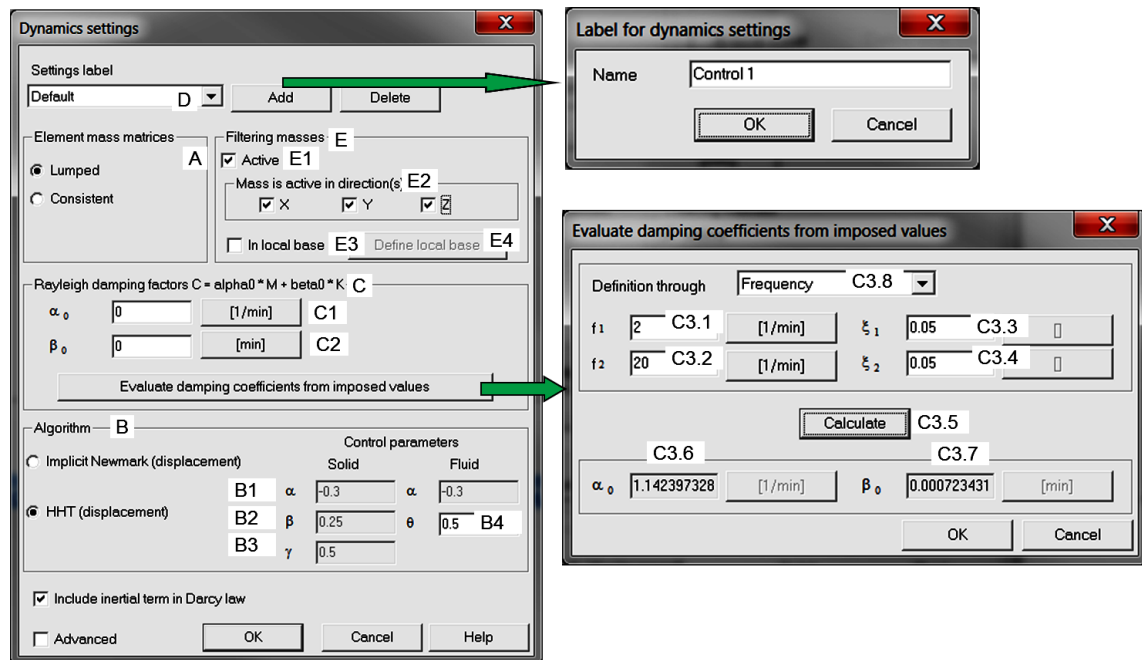
## Related Topics

### Theory

- **ALGORITHMS**

## 1.2.3 DYNAMICS

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



- 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**

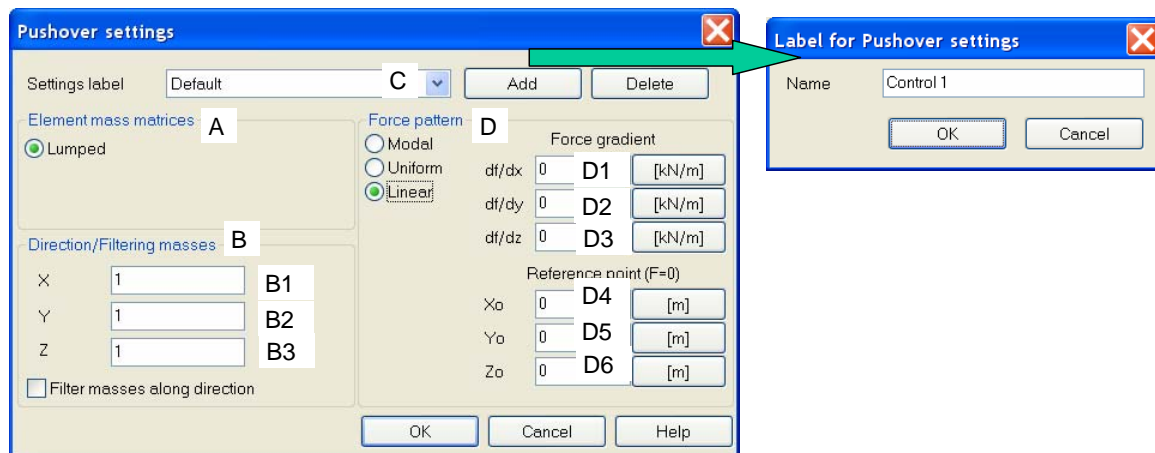
**Related Topics**

*Theory*

- [ALGORITHMS](#)

## 1.2.4 PUSHOVER

Sets parameters for drivers: **Pushover**



- 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
- B2. 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
- D3. as above but for Z direction
- D4. X coordinate of the reference point (where  $F=0$ ) for the **Linear** pattern
- D5. as above but Y coordinate
- D6. as above but Z coordinate

## Related Topics

Theory:

- [ALGORITHMS](#)

## 1.3 ASSEMBLY

Following options allow to build the computational model

November 6, 2023

DataPrep

Theory

Benchmarks

Tutorials

ZSoil<sup>®</sup>-3D-2PHASE v.2023

QH-18

<b>Preprocessing►</b>	<p>Runs unified 2D/3D graphical preprocessor to set model geometry, generate mesh, loads, initial and boundary conditions.</p> <ul style="list-style-type: none"> <li>• <b>INDEX</b></li> <li>• <b>USER INTERFACE</b></li> </ul>
<b>Materials►</b>	<p>Dialog box used to input all the material parameters, and (optionally) to associate with certain properties load time functions (to describe an effect of growing value of the Young's modulus for instance) and data superelements (to describe an effect of variation of a certain parameter in space; Young's modulus variation with depth for instance). <b>Model Data Groups ►</b>, <b>Material Data Base ►</b>, <b>Model Selection ►</b>, <b>Material parameters ►</b></p>
<b>Existence Function►</b>	<p><b>Existence Function</b> is a Heaviside type function (<math>0 \rightarrow</math> non-existence, <math>1 \rightarrow</math> existence) specified at the series of time ranges <math>t_1 \div t_2 \dots t_N \div t_{N+1}</math> (maximum 4 ranges are allowed) to be associated with selected elements, boundary conditions and initial conditions. It is mainly used to handle excavation/construction stages.</p>
<b>Load Function►</b>	<p>Dimensionless functions of time (real or fictitious) to be associated with loads, imposed boundary conditions and certain material properties. They describe the evolution of an amplitude of a given quantity in time. Specify <b>Load Time Function</b> as a list of pairs <math>(t_k, f(t_k))</math>, assuming linear interpolation for intermediate points. Values <math>f(t_k)</math> can be scaled by a given factor and time instances <math>t_k</math> may be shifted by a given time value.</p>
<b>Evolution function►</b>	<p>Functions to be associated with certain material properties. They describe the evolution of a given parameter with respect to the selected state parameter (temperature for instance). Specify <b>Evolution function</b> as a list of pairs <math>(a_k, f(a_k))</math>, assuming linear interpolation for intermediate points.</p>
<b>Gravity►</b>	<p>Specify gravity multipliers in <math>x, y, (z)</math> directions (minus sign downwards) and (optionally) set global value of the <math>K_o^{insitu}</math> coefficient of the lateral earth pressure (at the initial state) for all the materials related to continuum. Specifying <math>K_o^{insitu}</math> at the material level will overwrite the global definition.</p>
<b>Seismic input►</b>	<p>Specify global (applied to all elements) acceleration time history, including direction vector, to be used during execution of dynamic drivers.</p>



### 1.3.1 MODEL DATA GROUPS

Material properties for all type of constitutive models are placed in the following groups:

<b>Elastic</b>	<p>For most models it includes 2 parameters:</p> <ul style="list-style-type: none"> <li>• Young modulus <math>E</math> { [kPa]; [<math>&gt; 0</math>]; 100000 }</li> <li>• Poisons ratio <math>\nu</math> { [-]; [0, 0.49999]; 0.3 }</li> </ul> <p>For some nonlinear continuum models it can be different.</p>
<b>Unit Weight</b>	<p>For all models for solid set:</p> <ul style="list-style-type: none"> <li>• unit weight for the solid <math>\gamma</math> { [kN/m<sup>3</sup>]; [<math>\geq 0</math>]; 0 }</li> </ul> <p>For continuum solid models (dry or saturated) set:</p> <ul style="list-style-type: none"> <li>• unit weight (bouyant, saturated or dry) to be used for single phase problems</li> <li>• dry unit weight to be used for for two phase problems</li> <li>• unit weight for the fluid <math>\gamma_F</math> { [kN/m<sup>3</sup>]; [<math>\geq 0</math>]; 10 }</li> <li>• initial void ratio <math>e_0</math> { [-]; [<math>\geq 0</math>]; 0 }</li> </ul> <p>For continuum models unit weights for both single phase and two-phase problems should be specified to avoid data inconsistency when switching from <b>Deformation</b> to <b>Deformation+Flow</b> analysis. A special application is available to compute both values from specimen data. The specimen data is memorized.</p>
<b>Geom</b>	<p>Used by models attached to structural elements. Defines different geometric characteristics, see detailed description for each model separately</p>
<b>Creep</b>	<p>Set up creep data independently from other attributes and type of constitutive model attached to the continuum or shells (except <b>Aging Concrete</b> and models with nonlinear elasticity). Two-parameter creep laws (exponential, power ,logarithmic and hyperbolic) can be used. Parameters for volumetric and deviatoric creep can be defined separately.</p>
<b>Non--Linear</b>	<p>Set up data to represent nonlinear mechanical behavior of the material; these parameters are distinct for each model</p>
continued on next page	

continued from previous page

## Flow

Set up Darcy flow model parameters (extended by effect of partial saturation) common for all models to be used for continuum

- Darcy coefficients  $k_0$   $\{[\text{m/s}]; [ > 0]; 1.0\}$
- Angle of local  $x'$  to global  $x$  coordinate system for principal permeability direction definition  $\beta$
- Residual saturation ratio  $S_r$   $\{[-]; [0, 1]; 0\}$
- Parameter related to the air entry suction  $\alpha$   $\{[\text{m}^{-1}]; [ > 0]; 2.0\}$
- Measure of the pore-size distribution  $n$   $\{[-]; [ > 1]; 2.0\}$ .
- Permeability function for unsaturated medium  $\{\text{Imay or Mualem; Imay}\}$
- Pore pressure weighting term  $\{S|S_e^{1/(n \cdot m)}; S\}$
- Fluid bulk modulus  $K_F$   $\{[\text{kPa}]; [ > 0]; 1.0e + 38\}$
- Air stiffness bulk modulus at atmospheric pressure  $K_a$   $\{[\text{kPa}]; [ > 0]; 100.0\}$
- Biot coefficient  $\alpha$   $\{[-]; (e_0, 1); 1.0\}$
- Flag that may cancel gravity term in Darcy law
- Flag that may cancel undrained behavior when **Driven load(undrained)** drivers are used
- Cut-off condition for suction pressure generated by undrained drivers (it applies to the term  $S p + \Delta p$ , where  $\Delta p$  is an excess pore pressure produced by undrained drivers)
- Flag that include air compressibility in fluid-gas mixture
- Flag that activates Biot coefficient

### Remarks:

1.  $\alpha$  measures the depth of the transition layer from full ( $S = 1$ ) to residual ( $S = S_r$ ) saturation.
2.  $K_F$  has no meaning for steady state drivers
3.  $S_r > 0$  yields a kind of artificial cohesion which in turn may influence safety factors or bearing capacities, if failure zone is within region of partial saturation

continued on next page

continued from previous page	
<b>Initial State</b>	<p>Sets up data concerning coefficient(s) of the lateral earth pressure <math>K_o</math> in the <i>in situ</i> state.</p> <ul style="list-style-type: none"> <li>• for 2D analysis: <ul style="list-style-type: none"> <li>★ <math>K_{ox}</math> in <math>x'</math> direction</li> <li>★ <math>K_{oz}</math> in <math>z'</math> direction</li> <li>★ inclination angle of <math>x'</math> axis with respect to horizontal axis</li> </ul> </li> <li>• for 3D analysis: <ul style="list-style-type: none"> <li>★ <math>K_{ox}</math> in <math>x_1</math> direction</li> <li>★ <math>K_{oz}</math> in <math>x_3</math> direction</li> <li>★ direction vectors for <math>x_1</math> and <math>x_3</math></li> </ul> </li> </ul> <p><b>Remarks:</b></p> <ol style="list-style-type: none"> <li>1. This data applies exclusively to models for continuum</li> <li>2. This data specified at the material level overwrites (for this material only) global setting made under <b>Gravity</b></li> </ol>
<b>Heat</b>	<p>Set up:</p> <ul style="list-style-type: none"> <li>• Problem type: <b>Heat</b>(models for continuum only !) <ul style="list-style-type: none"> <li>★ thermal conductivity <math>\lambda_T</math> (it can be set as dependent on temperature via evolution function)</li> <li>★ heat capacity <math>c^*</math> (it can be set as dependent on temperature via evolution function)</li> <li>★ hydration heat source characteristics</li> </ul> </li> <li>• Problem type <b>Deformation</b>, <b>Deformation+Flow</b>, (all <b>Solid Phase Models</b>) <ul style="list-style-type: none"> <li>★ thermal dilatancy coefficient <math>\alpha</math> (it can be set as dependent on temperature via evolution function)</li> </ul> </li> </ul>
<b>Humidity<sup>2</sup></b>	<p>Set up:</p> <ul style="list-style-type: none"> <li>• Problem type <b>Humidity</b>(models for continuum only !) <ul style="list-style-type: none"> <li>★ moisture conductivity parameters <math>a, D_1, w_1, \chi</math>.</li> </ul> </li> <li>• Problem type <b>Deformation</b>, <b>Deformation+Flow</b>, (all <b>Solid Phase Models</b>) <ul style="list-style-type: none"> <li>★ hygral dilatancy coefficient <math>\beta</math></li> </ul> </li> </ul>
<b>Stability</b>	<p>Set up control data for stability driver, which for given material overwrites global settings made under <b>Control→Analysis &amp; Drivers</b>.</p>
<b>Damping</b>	<p>Set up damping parameters <math>\alpha_o</math> (applies to the mass) and <math>\beta_o</math> (applies the stiffness) which for this material will overwrite <math>\alpha_o</math> and <math>\beta_o</math> globally defined in <b>Control/Dynamics</b> menu.</p>

Depending on the selected model, **Analysis Type** and **Problem Type**, some of the groups

<sup>2</sup>concerns versions: **ACADEMIC**, **PROFESSIONAL**, **EXPERT** only

are obligatory, some are optional, some will not be activated. Different models may share data for certain groups with other models.

## Related Topics

- *data preparation: Materials*
- *data preparation: Model data groups*
- *data preparation: Material data base*
- *data preparation: Models for continuum*
- *data preparation: Truss (anchor) models*
- *data preparation: Models for beams and axisymmetric shells*
- *data preparation: Models for shells*
- *data preparation: Models for membranes*
- *data preparation: Models for interfaces*
- *data preparation: Elastic*
- *data preparation: Unit weights*
- *data preparation: Geometry*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Nonlinear*
- *data preparation: Initial K0 state*
- *data preparation: Heat*
- *data preparation: Humid*
- *data preparation: Stability*

### 1.3.2 MATERIAL DATA BASE

Data base of material models data can be created and handled with use of the following options

Add Material	Intercept material data from current job and add them to current data base
Modify Material	Intercept material data from current job and set them to current data base under existing item
Remove Material	Delete material from data base
Add from File	Intercept material data from other data base and add them to current data base
Replace from File	Intercept material data from other data base and set them to current data base under existing item
Save Data Base	Store current data base as *.mat file
Take from List	Set data from current data base to current job

#### Related Topics

- *data preparation: Materials*

### 1.3.3 MODEL SELECTION

A list of available models is automatically set by the program, according to the type of elements to which given model is attached during preprocessing stage. For each model, activity of each data group is indicated. *Use standard* means that given group is to be handled in a standard and unified way, as specified in [Model Data Groups](#)

The following models are included in the program:

<b>Models for Continuum</b>  Aging Concrete (adv.) ▶ Modified Cam-Clay (adv.) ▶ Cap (adv.) ▶ Drucker--Prager (simp.) ▶ Elastic (simp.) ▶ Hoek--Brown (M--W) (simp.) ▶ Hardening Soil-small (adv.) ▶ Mohr--Coulomb (M--W) (simp.) ▶ Multilaminate (simp.) ▶ Rankine (M--W) (simp.) ▶ Mohr--Coulomb (simp.) ▶ Plastic damage (concrete) (adv.) ▶ <b>Truss (anchor) model</b> Uni--Axial Model (simp.) ▶	<b>Beams and Axisymmetric Shells</b>  Beam (simp.) ▶ Axisymmetric Shell (simp.) ▶ <b>Models for Shells</b>  Shell Section (simp.) ▶ Shell Component (simp.) ▶ <b>Models for Membrane</b>  Membrane--Fiber (simp.) ▶ Membrane--Plane Stress (simp.) ▶ Membrane--Fabric (simp.) ▶ <b>Models for Infinite m.</b>  Infinite Media (simp.) ▶ <b>Models for Interface</b>  Heat Convection ▶ Humidity Convection <sup>a</sup> ▶ Contact (simp.) ▶ Bond interface (simp.) ▶ Nodal interface (simp.) ▶ Seepage (simp.) ▶
--	---

<sup>a</sup>concerns versions: ACADEMIC, PROFESSIONAL, EXPERT only

#### Remark:

Models are classified as:

- (simp.) -i.e. "simple" if they require only standard data,
- (adv.) -i.e. "advanced" if they require experiments or advanced theoretical background.  
Some simple models have advanced options.

### 1.3.3.1 AGING CONCRETE (advanced)

The viscoelastic model of aging concrete <sup>3</sup>

consists of a set of parallel Maxwell units. Each unit is described by a maturity dependent Young modulus  $E_k(M_i) = M_{ki}E$ , and retardation time  $\tau_k = \frac{E_k}{\eta_k}$ . The contribution of each unit depends on maturity measure  $M$  (expressed in time units), by the set of weighting factors  $M_{ik}$ , ( $\sum_k M_{ik} = 1$ ).

Data group	Parameters
<b>Elastic</b>	obligatory <ul style="list-style-type: none"> <li>• <math>nP</math> - number of maturity points for interpolation in time (default 10)</li> <li>• <math>nM</math> - number of Maxwell elements (max=6)</li> <li>• Young moduli <math>E_k</math> [<math>E_k &gt; 0</math> [kPa]] and their distribution factors <math>W_{ki}</math> (<math>\sum_{k=1}^{nM} W_{ik} = 1</math>) over <math>nM</math> Maxwell elements, defined at maturity points (<math>M_i</math> <math>i = 1..nP</math>)</li> <li>• <math>\nu</math> - Poisson ratio same for all Maxwell elements <math>\nu[[-], [ &lt; 0.49999], 0.2]</math></li> <li>• <math>\tau_k</math> - retardation times [<math>h</math>] for each Maxwell element (<math>k = 1..nM</math>)</li> </ul>
<b>Unit Weight</b>	optional; (see more information)
<b>Flow</b>	optional; (see more information)
<b>Non--Linear</b>	needed for output purposes only $f_t(M_i)$ – tensile strengths [kPa] digitized at $nP$ maturity points
<b>Heat</b>	optional; (see more information) <ul style="list-style-type: none"> <li>• for <b>Heat</b> analysis use standard (see more information)</li> <li>• for <b>Deformation</b> and <b>Deformation+Flow</b> analysis it serves parameters needed to integrate maturity <math>M</math> based on nonstationary temperature field given in associated <b>Heat Transfer</b> project; if group <input checked="" type="checkbox"/> <b>Heat</b> is set OFF then maturity is equivalent to the time of existence of a given material:               <ul style="list-style-type: none"> <li>★ <math>\alpha</math> – thermal dilatancy; <math>\alpha</math> [<math>1/^\circ\text{C}</math>], <math>\geq 0</math>, <math>10^{-5}</math>].</li> <li>★ <math>Q/R</math> – hydration heat source constant <math>Q/R = 4000[^\circ\text{K}]</math> use same value as for corresponding material in associated <b>Heat Transfer</b> project</li> <li>★ <math>T_F</math> – reference temp. for hydration heat source <math>T_F = 20[^\circ\text{C}]</math> use same value as for corresponding material in associated <b>Heat Transfer</b> project</li> </ul> </li> </ul>
<b>Damping</b>	optional (see more information)

<sup>3</sup>concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT** only

## Related Topics

- *data preparation: Aging concrete*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Initial  $K_0$  state*
- *data preparation: Heat*
- *data preparation: Humid*



### 1.3.3.2 MODIFIED CAM-CLAY MODEL (advanced)

Data group	Parameters
Elastic	obligatory; only Poisson coefficient must be entered: $\nu(\star), \{[-], [0 < \nu < 0.49999], 0.3\}$
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Non--Linear	obligatory; <ul style="list-style-type: none"> <li>Slope of critical state line in p-q plane <math>M_c</math>, <math>\{[-], [&gt; 0], [&lt; 3], 1.0\}</math></li> <li>Slope of primary consolidation in e-ln(p) <math>\lambda</math>, <math>\{[-], [&gt; 0], 0.1\}</math></li> <li>Slope of secondary consolidation in e-ln(p) <math>\kappa</math>, <math>\{[-], [&gt; 0], \kappa &lt; \lambda, 0.01\}</math></li> <li>Minimum preconsolidation pressure <math>p_{co}</math> (<math>\star</math>), <math>\{[kPa], [&gt; 0], 10\}</math></li> <li>Overconsolidation ratio OCR, <math>\{[-], [\geq 1], 1\}</math></li> <li>Strength anisotropy factor (ratio between tension and compression meridian slopes <math>k = M_e/M_c</math>) can be: <ul style="list-style-type: none"> <li><math>\star</math> disabled (<math>k = 1</math>)</li> <li><math>\star</math> default (<math>k = \frac{3}{3 + M_c}</math>)</li> <li><math>\star</math> prescribed by the user <math>k</math>, <math>\{[-], 0.5 &lt; k &lt; 1.0, 0.75\}</math></li> </ul> </li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

#### Related Topics

- *data preparation: Cam-Clay model*
- *Theoretical Manual: Cam-Clay model*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Initial K0 state*
- *data preparation: Heat*
- *data preparation: Stability*

- *data preparation: Humid*

### 1.3.3.3 CAP MODEL (advanced)

Data group	Parameters
Elastic	obligatory; (see more information)
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Creep	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
Non--Linear	obligatory; <ul style="list-style-type: none"> <li>• Drucker–Prager data</li> <li>• Cap parameters:               <ul style="list-style-type: none"> <li>★ <math>\lambda</math> - slope of primary consolidation line <math>\lambda \{[-], [ &gt; 0], 0.1\}</math> in <math>(e - \ln p)</math></li> <li>★ OCR - overconsolidation ratio; OCR <math>\{[-], [\geq 1.0], 1.0\}</math></li> <li>★ <math>p_{co}</math> and <math>R</math> - initial preconsolidation pressure and initial shape cap surface parameters                   <ol style="list-style-type: none"> <li>for Direct Definition enter:                       <ul style="list-style-type: none"> <li>♦ <math>p_{co}</math> (★) <math>\{\text{kPa}, [ &gt; 0], 10^{38}\}</math></li> <li>♦ <math>R</math> <math>\{[-], [1.1 &lt; R &lt; 2.7], 1.5\}</math></li> </ul> </li> <li>for indirect definition through an Oedometric Test enter:                       <ul style="list-style-type: none"> <li>♦ vertical preconsolidation stress <math>\sigma_{VM}</math>, <math>\{\text{kPa}, [ &gt; 0], 0\}</math></li> <li>♦ coefficient of lateral pressure at normal consolidation state <math>K_0^{NC}</math>, <math>\{[-], [ &gt; 0], 0.5\}</math> (Jaky's formula <math>K_0^{NC} = 1 - \sin(\phi)</math> can be applied)</li> </ul> </li> </ol> </li> </ul> </li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

### Related Topics

- data preparation: Cap model
- data preparation: Drucker–Prager
- Theoretical Manual: Cap model
- data preparation: Elastic

- *data preparation: Unit weight*
- *data preparation: Flow data preparation: Creep*
- *data preparation: Initial  $K_0$  state*
- *data preparation: Heat*
- *data preparation: Stability*
- *data preparation: Humid*

### 1.3.3.4 DRUCKER–PRAGER (simple)

Data group	Parameters
Elastic	obligatory; (see more information)
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Creep	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
Non--Linear	obligatory ; <ul style="list-style-type: none"> <li>• Cohesion <math>C</math> (★), {kPa, [<math>&gt; 0</math>], 20}</li> <li>• Friction angle <math>\varphi</math> (★), {[deg], [0, 89], 30}</li> <li>• Dilatancy angle <math>\psi</math> (★), {[deg], [0, 89], 0} Set <math>\psi = 0</math> for incompressible behavior</li> <li>• Tensile cut-off(★); optional, ignore if not konwlegable of effects</li> <li>• Size adjustment with respect to the Mohr-Coulomb criterion: : External Edges, Internal Edges, Elastic, Plane Strain (default for plane strain), Intermediate (default for axisymmetry)</li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

### Related Topics

- *data preparation: Drucker–Prager*
- *data preparation: Elastic*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Initial  $K_0$  state*
- *data preparation: Stability*
- *data preparation: Heat*

- *data preparation: Humid*

### 1.3.3.5 ELASTIC (simple)

Data group	Parameters
Elastic	obligatory; (see more information)
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Creep	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

### Related Topics

- *data preparation: Elastic model*
- *data preparation: Elastic*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Initial  $K_0$  state*
- *data preparation: Heat*
- *data preparation: Humid*

### 1.3.3.6 INFINITE MEDIA

Infinite medium can be represented exclusively by a linear elastic material.

Data group	Parameters
Elastic	obligatory; <a href="#">see more information</a>

#### Related Topics

- *data preparation: Infinite media*



### 1.3.3.7 HOEK–BROWN (simple /advanced)

Data group	Parameters
Elastic	obligatory; (see more information)
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Creep	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
Non--Linear	obligatory; <ul style="list-style-type: none"> <li>Compressive uni-axial strength <math>f_c</math>, {[kPa], [<math>&gt; 0</math>, <math>&gt; f_t</math>], 15000}</li> <li>Tensile uni-axial strength <math>f_t</math>, {[kPa], [<math>&gt; 0</math>, <math>&lt; f_t</math>], 1000}</li> <li>Biaxial compressive/uniaxial compressive strength ratio <math>f_b/f_c</math>, {[<math>-</math>], [<math>&gt; 1</math>], 1.16}</li> <li><b>Tensile Cut--Off</b> flag ON/OFF first stress invariant <math>I_{1t}</math> (if Cut--Off is ON), {[kPa], [<math>&gt; 0</math>], 0}</li> <li>Plastic flow type Deviatoric, Drucker--Prager, Tensile Meridian, Hoek--Brown (M--W),</li> <li>Dilatancy angle at uniaxial compressive strength <math>\psi_c</math> (if Drucker--Prager or Hoek--Brown (M--W) flow is ON), {[deg], [<math>\arctan\left(\frac{f_t}{\sqrt{2}f_c}\right) \leq 35.3^\circ</math>], 0}</li> <li>Softening flag ON/OFF<sup>4</sup>. This option must be used with caution. If activated, introduce:               <ul style="list-style-type: none"> <li>★ Crack opening at failure <math>w_r</math>, {[m], [<math>&gt; 0</math>], 0.0001}</li> <li>★ Steepness parameter <math>a</math>, {[<math>-</math>], [<math>&gt; 0</math>], 5.0}</li> <li>★ Fictitious number of cracks for compression <math>b</math>, {[<math>-</math>], [<math>&gt; 1</math>], 10.0}</li> </ul> </li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

### Related Topics

- *data preparation: Hoek–Brown*
- *data preparation: Elastic*

- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Initial  $K_0$  state*
- *data preparation: Stability*
- *data preparation: Heat*
- *data preparation: Humid*

### 1.3.3.8 MOHR–COULOMB (M-W)(simple)

Data group	Parameters
Elastic	obligatory; <a href="#">(see more information)</a>
Unit Weight	optional; <a href="#">(see more information)</a>
Flow	optional; <a href="#">(see more information)</a>
Creep	optional; <a href="#">(see more information)</a>
Initial State $K_0$	optional for Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Non--Linear	obligatory; <ul style="list-style-type: none"> <li>• Cohesion <math>c</math>, {[kPa], [<math>&gt; 0</math>], 20}</li> <li>• Friction angle <math>\phi</math>, {[deg], [0, 89], 30}</li> <li>• Cut--Off flag ON/OffFlg</li> <li>• Tensile strength <math>f_t</math> {[kPa], [0, <math>c/\tan(\phi)</math>], 0} (if Cut--Off flag is ON),</li> <li>• Plastic flow type Deviatoric, Drucker--Prager</li> <li>• Dilatancy angle <math>\psi</math>, {[deg], [0, 89], 0} (if Drucker--Prager flow is ON, set <math>\psi = 0</math> for incompressible behavior)</li> <li>• Size adjustment flag ON/OFF</li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Heat	optional for Heat, Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Stability	optional for Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Damping	optional <a href="#">(see more information)</a>

### Related Topics

- [data preparation: Mohr–Coulomb](#)
- [data preparation: Elastic](#)
- [data preparation: Unit weight](#)
- [data preparation: Flow](#)
- [data preparation: Creep data preparation: Initial K0 state](#)
- [data preparation: Stability](#)
- [data preparation: Heat](#)
- [data preparation: Humid](#)

### 1.3.3.9 Hardening Soil

Data group	Parameters
Elastic	<p>obligatory; set</p> <ul style="list-style-type: none"> <li>• <math>E_{ur}^{ref}</math> is the unloading/reloading Young modulus given at the reference stress <math>\sigma^{ref}</math> {[kPa], [<math>&gt; 0</math>], 80000})</li> <li>• Reference stress <math>\sigma^{ref}</math>, {[kPa], [<math>&gt; 0</math>], 100} is the stress value at which the reference stiffness moduli <math>E_{ur}^{ref}</math> is defined</li> <li>• <math>\nu_{ur}</math> is the unloading/reloading Poisson coefficient; it varies from 0.15 to 0.3, hence for sands it is recommended to assume <math>\nu_{ur} = 0.2..0.25</math> and for clays <math>\nu_{ur} = 0.25..0.3</math>, {[<math>-</math>], [0, 0.49999], 0.2}</li> <li>• <math>m</math> is the exponent in stress dependency power law; it varies from <math>m = 0.4</math> to <math>m = 0.9</math>; it is smaller for dense sands and larger for clays</li> <li>• <math>\sigma_L</math> is the minimum allowed stress (<math>\sigma_3</math> or <math>p</math>) value used for evaluation of stiffness moduli {[kPa], [<math>&gt; 0</math>], 10}</li> <li>• stiffness stress dependency type can be one of three following functions <ol style="list-style-type: none"> <li>1. <math>\frac{\sigma_3 + c \cot \phi}{\sigma^{ref} + c \cot \phi}</math></li> <li>2. <math>\frac{\sigma_3}{\sigma^{ref}}</math></li> <li>3. <math>\frac{p}{\sigma^{ref}}</math></li> </ol> <p>the last two functions can be used in simplified undrained analyses when <math>\phi = 0</math> is assumed; the third function is recommended in general but then reference stiffness moduli and power exponent <math>m</math> must be recalculated once they were calibrated assuming <math>\sigma_3</math> type stress dependencies</p> </li> <li>• small strain formulation can be selected from the set of the three possible options <ol style="list-style-type: none"> <li>1. Disabled</li> <li>2. HSs formulation - classical</li> <li>3. Brick formulation</li> </ol> <p>The <b>Brick formulation</b> is superior with respect to the classical HSs.</p> </li> <li>• <math>E_o^{ref}</math> is the Young modulus at very small strains at the reference stress <math>\sigma^{ref}</math>; In case of lack of information on <math>E_o^{ref}</math> one may try to estimate <math>E_o^{ref}</math> based on Alpan's diagram (see external report on Hardening Soil model) assuming <math>E_s = E_{ur}</math></li> </ul>
continued on next page	

continued from previous page

Elastic	obligatory; set <ul style="list-style-type: none"> <li>• <math>\gamma_{0.7}</math> is threshold shear strain at which secant shear modulus <math>G</math> reaches 70 % of its initial value <math>G_o</math> (note that <math>G_o = \frac{E_o}{2(1 + \nu_{ur})}</math>); in the current implementation <math>\gamma_{0.7}</math> is assumed to be constant; in case of lack of information on <math>\gamma_{0.7}</math> 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; <math>\gamma_{0.7} = 0.0001..0.0002</math>, <math>\{[-], [0.0001, 0.0005], 0.0002\}</math></li> </ul>
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
continued on next page	

continued from previous page	
Nonlinear	<p>obligatory;</p> <ul style="list-style-type: none"> <li>• <math>E_{50}^{ref}</math> is the secant Young modulus at 50 % of failure deviatoric stress <math>q_f</math> derived from the <math>q - \varepsilon_1</math> curve in drained triaxial test (see report on Hardening Soil model for more details)</li> <li>• <math>\phi</math> is the friction angle <math>\{[\text{deg}], [ &gt; 0], 30\}</math></li> <li>• <math>\psi</math> is the dilatancy angle <math>\{[\text{deg}], [ &gt; 0], 0\}</math></li> <li>• <math>c'</math> is the effective cohesion <math>\{[\text{kPa}], [ \geq 0], 20\}</math></li> <li>• <math>R_f</math> is the failure ratio <math>\{[-], [0.8 &lt; R_f &lt; 1], 0.9\}</math></li> <li>• <math>f_t</math> is the tensile strength <math>\{[\text{kPa}], [ \geq 0], 0\}</math></li> <li>• <math>e_{max}</math> is the maximum allowed void ratio; if current void ratio exceeds the <math>e_{max}</math> dilatancy angle is switched to <math>\psi = 0</math>, <math>\{[-], [ &gt; 0], 1\}</math></li> <li>• <math>D</math> is the Rowe's dilatancy multiplier in the contractant domain, for standard HS model <math>\{[-], [0 \leq D \leq 1], 0.0\}</math> while for HS-small <math>\{[-], [0 \leq D \leq 1], 0.25\}</math></li> <li>• Cap surface parameter <math>M</math> and hardening parameter <math>H</math> are derived by using a simple calculator which simulates an oedometric test; for given tangent oedometric modulus <math>E_{oed}</math> at a given reference vertical stress <math>\sigma_{oed}^{ref}</math> and for assumed <math>K_o^{NC}</math> parameter (here Jaky's formula can be used) values of <math>H</math> and <math>M</math> are evaluated (press button <b>Evaluate M,H</b>); one may assume <math>E_{oed} = E_{50}^{ref} \left( \frac{\sigma_{oed}^{ref} + c \cot \phi}{\sigma^{ref} + c \cot \phi} \right)^m</math> as a default</li> <li>• Setting the initial state variables <math>\gamma_o^{PS}</math> and <math>p_{co}</math> can be carried out by means of assumed OCR or preoverburden pressure <math>q^{POP}</math>; pairs <math>K_o^{SR}</math> and <math>OCR</math> (<math>OCR \geq 1.0</math>) or <math>K_o^{SR}</math> and <math>q^{POP}</math> are needed to setup the initial position of the cap surface and the initial value of the hardening parameter <math>\gamma^{PS}</math> (please refer to the report on Hardening Soil model for the detailed explanation)</li> <li>• <math>p_{co}^{min}</math> is the minimum allowed value for the initial preconsolidation stress <math>\{[kPa], [ &gt; 0], 10\}</math></li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

**Remarks:**

- Prior using HS or HS-small model it is strongly recommended to study first the external

report on Hardening Soil model

## Related Topics

- *data preparation: HS-small model*
- *Technical report: HS-small model*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Initial K0 state*
- *data preparation: Heat*
- *data preparation: Stability*
- *data preparation: Humid*

### 1.3.3.10 Densification (advanced)

Data group	Parameters
Elastic	<p>obligatory; set</p> <ul style="list-style-type: none"> <li>• <math>E_{ur}^{ref}</math> is the unloading/reloading Young modulus given at the reference stress <math>\sigma^{ref}</math> {[kPa], [<math>&gt; 0</math>], 80000}); note that the elastic stiffness moduli depend on <math>p</math> rather on <math>\sigma_3</math> value in this model</li> <li>• Reference stress <math>\sigma^{ref}</math>, {[kPa], [<math>&gt; 0</math>], 100} is the stress value at which the reference stiffness moduli <math>E_{ur}^{ref}</math> is defined</li> <li>• <math>\nu_{ur}</math> is the unloading/reloading Poisson coefficient; it varies from 0.15 to 0.3, hence for sands it is recommended to assume <math>\nu_{ur} = 0.2..0.25</math> and for clays <math>\nu_{ur} = 0.25..0.3</math>, {[<math>-</math>], [0, 0.49999], 0.2}</li> <li>• <math>m</math> is the exponent in stress dependency power law; for loose sands <math>m \approx 0.5</math></li> <li>• <math>\sigma_L</math> is the minimum allowed reference stress value used for evaluation of stiffness moduli {[kPa], [<math>&gt; 0</math>], 10}</li> <li>• stiffness stress dependency type can be one of three following functions <ol style="list-style-type: none"> <li>1. <math>\frac{\sigma_3 + c \cot \phi}{\sigma^{ref} + c \cot \phi}</math></li> <li>2. <math>\frac{\sigma_3}{\sigma^{ref}}</math></li> <li>3. <math>\frac{p}{\sigma^{ref}}</math></li> </ol> <p>the last two functions can be used in simplified undrained analyses when <math>\phi = 0</math> is assumed; the third function is recommended in general but then reference stiffness moduli and power exponent <math>m</math> must be recalculated once they were calibrated assuming <math>\sigma_3</math> type stress dependencies</p> </li> <li>• small strain formulation can be selected from the set of the three possible options <ol style="list-style-type: none"> <li>1. Disabled</li> <li>2. HSs formulation - classical</li> <li>3. Brick formulation</li> </ol> </li> <li>• <math>E_o^{ref}</math> is the Young modulus at very small strains at the reference stress <math>\sigma^{ref}</math>; note that the current very small strain stiffness modulus <math>E_o</math> depends on <math>p</math> rather than <math>\sigma_3</math> (as in the HS model)</li> </ul>
continued on next page	



continued from previous page

Elastic	obligatory; set <ul style="list-style-type: none"> <li>• <math>\gamma_{0.7}</math> is the threshold shear strain at which secant shear modulus <math>G</math> reaches 70 % of its initial value <math>G_o</math> (note that <math>G_o = \frac{E_o}{2(1 + \nu_{ur})}</math>); in the current implementation <math>\gamma_{0.7}</math> is assumed to be constant; in case of lack of information on <math>\gamma_{0.7}</math> one may use the diagram by Wichtmann and Triantafyllidis included in the report on Hardening Soil model; <math>\gamma_{0.7} = 0.0001..0.0002</math>, <math>\{[-], [0.0001, 0.0005], 0.0002\}</math></li> </ul>
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
continued on next page	

continued from previous page

Nonlinear	<p>obligatory;</p> <ul style="list-style-type: none"> <li>• <math>\phi</math> is the friction angle <math>\{[\text{deg}], [ &gt; 0], 30\}</math></li> <li>• <math>c'</math> is the effective cohesion <math>\{[\text{kPa}], [\geq 0], 1\}</math></li> <li>• <math>f_t</math> is the tensile strength <math>\{[\text{kPa}], [\geq 0], 0\}</math></li> <li>• To activate internal shear plastic mechanism the flag must be set ON</li> <li>• <math>R_f</math> is the failure ratio <math>\{[-], [0.8 &lt; R_f &lt; 1], 0.9\}</math></li> <li>• <math>E_{50}^{ref}</math> is the secant Young modulus at 50 % of failure deviatoric stress <math>q_f</math> derived from the <math>q - \varepsilon_1</math> curve in drained triaxial test (same as in the HS model so please refer to the report on Hardening Soil model for more details)</li> <li>• To activate densification mechanism user must switch ON the flag <input checked="" type="checkbox"/> <b>Densification mechanism</b> and then choose one of the two densification laws, the one proposed by Zienkiewicz or by Sawicki</li> <li>• <math>A, B, \gamma</math> - parameters for densification law by Zienkiewicz</li> <li>• <math>C_1, C_2</math> - parameters for densification law by Sawicki (typical values for <math>C_1</math> are of order <math>0.005 \div 0.009</math> and for <math>C_2</math> of order <math>2 \cdot 10^5</math>) (please refer to the report on Dynamics for more information)</li> <li>• Strain mapping for the accumulated volumetric strain can be selected as <i>Isotropic</i> (here the increment of the accumulated volumetric strain is projected equally on all normal directions) or <i>Extended</i> (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 <i>Extended</i> mapping will put the full accumulated volumetric strain only on the vertical direction</li> <li>• Setting the initial state variable <math>\gamma_o^{PS}</math> can be carried out by means of assumed OCR or preoverburden pressure <math>q^{POP}</math>; pairs <math>K_o^{SR}</math> and <math>OCR</math> (<math>OCR \geq 1.0</math>) or <math>K_o^{SR}</math> and <math>q^{POP}</math> are needed to setup the initial value of the hardening parameter <math>\gamma^{PS}</math> (please refer to the report on Hardening Soil model for the detailed explanation as the same procedure is used in this model)</li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
continued on next page	

continued from previous page	
Damping	optional (see more information)

**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

**Related Topics**

- *data preparation: Densification model*
- *Technical report: Dynamics*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Initial K0 state*
- *data preparation: Heat*
- *data preparation: Stability*
- *data preparation: Humid*

### 1.3.3.11 MULTILAMINATE (simple)

Data group	Parameters
Elastic	obligatory; <a href="#">(see more information)</a>
Unit Weight	optional; <a href="#">(see more information)</a>
Flow	optional; <a href="#">(see more information)</a>
Creep	optional; <a href="#">(see more information)</a>
Initial State $K_0$	optional for Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Non--Linear	<p>obligatory; for each active weakness plane set:</p> <ul style="list-style-type: none"> <li>for 2D models: <ul style="list-style-type: none"> <li>Inclination angle <math>\alpha</math> measured in anti-clockwise direction, <math>\{[\text{deg}], [-90 \leq \alpha \leq 90], 0\}</math></li> </ul> </li> <li>for 3D models: <ul style="list-style-type: none"> <li>Inclination angles <math>\alpha, \beta</math> <math>\{[\text{deg}], [-360 \leq \alpha/\beta \leq 360], 0\}</math>, (if <input type="checkbox"/> <b>Advanced mode</b> is set ON)</li> <li>Vector <b>n</b> normal to the weakness plane (if <input type="checkbox"/> <b>Advanced mode</b> is set OFF)</li> </ul> </li> <li>Cohesion <math>c</math>, <math>\{[\text{kPa}], [&gt; 0], 0\}</math></li> <li>Friction angle <math>\phi</math>, <math>\{[\text{deg}], [0, 89], 0\}</math></li> <li>Dilatancy angle <math>\psi</math>, <math>\{[\text{deg}], [0, 89], 0\}</math>; set <math>\psi = 0</math> for incompressible behavior.</li> <li>Cut--Off flag ON/OFF</li> <li>Tensile strength <math>f_t</math>, <math>\{[\text{kPa}], [0, c/\tan(\phi)], 0\}</math> (if Cut--Off flag is ON)</li> <li>Matrix material model ( elastic if <input type="checkbox"/> <b>Advanced</b> is OFF ) or Mohr--Coulomb (M-W)/ Rankine (M-W)/ Hoek--Brown</li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Heat	optional for Heat, Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Stability	optional for Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Damping	optional <a href="#">(see more information)</a>

### Related Topics

- [data preparation: Multilaminate](#)
- [data preparation: Elastic](#)
- [data preparation: Unit weight](#)
- [data preparation: Flow](#)

- *data preparation: Creep*
- *data preparation: Initial K0 state*
- *data preparation: Stability*
- *data preparation: Heat*
- *data preparation: Humid*

### 1.3.3.12 RANKINE (M-W) (simple)

Data group	Parameters
Elastic	obligatory; <a href="#">(see more information)</a>
Unit Weight	optional; <a href="#">(see more information)</a>
Flow	optional; <a href="#">(see more information)</a>
Creep	optional; <a href="#">(see more information)</a>
Initial State $K_0$	optional for Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Non--Linear	obligatory; set: <ul style="list-style-type: none"> <li>• Tensile strength (uni-axial) <math>f_t</math>, {[kPa], [<math>&gt; 0</math>], 0}</li> <li>• Softening flag; if <input checked="" type="checkbox"/> <b>Softening</b> is set ON define <ul style="list-style-type: none"> <li>★ Crack opening at failure <math>w_r</math>, {[m], [<math>&gt; 0</math>], 0.0001}</li> <li>★ Steepness parameter <math>a</math>, {[<math>-</math>], [<math>&gt; 0</math>], 5.0}</li> <li>★ Fictitious number of cracks for compression <math>b</math>, {[<math>-</math>], [<math>&gt; 1</math>], 10.0}</li> </ul> </li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Heat	optional for Heat, Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Stability	optional for Deformation, Deformation+Flow; <a href="#">(see more information)</a>
Damping	optional <a href="#">(see more information)</a>

### Related Topics

- [data preparation: Rankine](#)
- [data preparation: Elastic](#)
- [data preparation: Unit weight](#)
- [data preparation: Flow](#)
- [data preparation: Creep](#)
- [data preparation: Initial K0 state](#)
- [data preparation: Stability](#)
- [data preparation: Heat](#)
- [data preparation: Humid](#)

### 1.3.3.13 MOHR–COULOMB (simple)

Data group	Parameters
Elastic	obligatory; (see more information)
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Creep	optional; (see more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
Non--Linear	obligatory; <ul style="list-style-type: none"> <li>Cohesion <math>C</math> (★), {[kPa], [<math>&gt; 0</math>], 0}</li> <li>Friction angle <math>\varphi</math> (★), {[deg], [0, 89], 0}</li> <li>Dilatancy angle <math>\psi</math> (★), {[deg], [0, 89], 0}</li> <li>Set <math>\psi = 0</math> for incompressible behavior</li> <li>Tensile cut-off(Rankine); (optional) <math>f_t</math> (★), {[kPa], [<math>\geq 0</math>], 0}</li> <li>Dilatancy cut-off; (optional) <math>e_{MAX}</math>, {[<math>-</math>], [<math>e_{max} &gt; e_o</math>]}</li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information)
Stability	optional for Deformation, Deformation+Flow; (see more information)
Damping	optional (see more information)

### Related Topics

- *data preparation: Mohr-Coulomb (non-smooth)*
- *data preparation: Mohr-Coulomb (smooth, M-W)*
- *data preparation: Drucker–Prager*
- *data preparation: Elastic*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Initial  $K_0$  state*
- *data preparation: Stability*
- *data preparation: Heat*

- *data preparation: Humid*



## 1.3.3.14 PLASTIC DAMAGE MODEL FOR CONCRETE

Data group	Parameters
Elastic	obligatory; (see more information) 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 more information)
Flow	optional; (see more information)
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 more information)
Initial State $K_0$	optional for Deformation, Deformation+Flow; (see more information)
continued on next page	

continued from previous page

#### Nonlinear

obligatory;

- $\underline{f}_c$  - 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)
- $\underline{f}_{co}/\underline{f}_c$  - initial to peak compressive strength ratio (0.4÷0.8)
- $\underline{f}_{cbo}/\underline{f}_{co}$  - initial biaxial to uniaxial strength ratio (1.1÷1.2)
- $\underline{\sigma}_{c,D}/\underline{f}_c$  - 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)
- $\tilde{\sigma}_c/\underline{f}_c$  - stress level for damage calibration in compression (on post-peak branch)( $\leq 1.0$ )
- $\tilde{D}_c$  - 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)
- $G_c$  - fracture energy in compression ( $50 G_f \leq G_c \leq 100 G_f$ )
- $f_t$  - 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)
- $\tilde{\sigma}_t/f_t$  - stress level for damage calibration (on post-peak branch) ( $< 1$ )
- $\tilde{D}_t$  - damage parameter in uniaxial tension at the assumed reference stress level ( $\geq 0.3$ )
- $G_t$  - 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)
- $s$  - stiffness recovery factor due to crack closure for tension-compression cycles (0.1 ÷ 0.3)
- Dilatancy type - Constant/Variable
- $\alpha_p$  - Dilatancy parameter value for Constant option or dilatancy parameter in uniaxial compression when option Variable is used (0.1 ÷ 0.5)
- $\alpha_{po}$  - dilatancy parameter in uniaxial tension option Variable is used (0.1 ÷ 0.4)
- $\underline{\sigma}_{c,dil}/\underline{f}_c$  - activation of dilatancy in compression when Variable option is used;  $\underline{\sigma}_{c,dil}/\underline{f}_c \geq \underline{f}_{co}/\underline{f}_c$ ; (0.4 ÷ 1.0)
- $\alpha_d$  - smoothing factor for plastic potential; value 1.0 is recommended (0.2 ÷ 2.0)

continued on next page

continued from previous page

Nonlinear	obligatory; <ul style="list-style-type: none"> <li>Char. length (RC) flag - enforced characteristic length flag for reinforced concrete (ON/OFF)</li> <li><math>l_c^{RC}</math> - characteristic length to be used for reinforced concrete when Char. length (RC) flag is ON (<math>&gt; 0</math>)</li> <li>Secant stiffness flag - use secant elastic stiffness (<math>D^e(1-D)</math>) when ON or consistent tangent when OFF (secant stiffness is recommended for cases with poor convergence)</li> </ul>
Humidity	optional for Humidity, Deformation, Deformation+Flow; (see more information)
Heat	optional for Heat, Deformation, Deformation+Flow; (see more information) 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 more information)
Damping	optional (see more information)

## Related Topics

- *data preparation: Plastic damage for concrete*
- *data preparation: Elastic*
- *data preparation: Unit weight*
- *data preparation: Flow*
- *data preparation: Creep*
- *data preparation: Initial K0 state*
- *data preparation: Stability*
- *data preparation: Heat*
- *data preparation: Humid*

### 1.3.3.15 TRUSS (ANCHOR) MODEL (simple)

Elasto-plastic uni-axial model for truss (anchor) and ring (for **Axisymmetry** only) elements. **Analysis Type**, i.e.: **Plane Strain**, **Axisymmetry**, **3D** must be properly set prior to entering the data, due to some differences in data handling.

Model	Use & Set up
<b>Elastic</b>	obligatory; set: Young's elastic modulus, $E$ {[kPa]; [ $> 0$ ]; 100000},
<b>Unit Weight</b>	optional; set: volumic weight of the material $\gamma$ , {[kN/m <sup>3</sup> ]; [ $> 0$ ]; 0}.
<b>Non--Linear</b>	optional; set: uniaxial tensile strength $f_t$ {[kPa], $> 0$ , 1} uniaxial compressive strength $f_c$ {[kPa], $> 0$ , 1}
<b>Geometry</b>	obligatory; set: cross-section area $A$ , {[m <sup>2</sup> ]; [ $> 0$ ]; 1} distance (2D Plane Strain only) $a$ , {[m]; [ $> 0$ ]; 1}
<b>Heat</b>	optional; set: $\alpha$ , {[C <sup>-1</sup> ]; [ $> 0$ ]; $1.0e^{-5}$ } - thermal dilatancy coefficient. In case of active <b>Data Group Heat</b> and active associated <b>Heat Transfer</b> project (if $\alpha > 0$ ), thermal strains will be evaluated.
<b>Damping</b>	optional; set Rayleigh damping factors $\alpha_o$ (applies to the mass) and $\beta_o$ (applies to the stiffness); use dedicated calculator if needed;

### Related Topics

- *data preparation: Truss (anchor) model*

### 1.3.3.16 BEAM MODEL

Two general modeling options are:

1. 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
<b>Elastic</b>	obligatory; set: • Young's elastic modulus $E\{[\text{kPa}]; [> 0]; 100000\}$ • Poisons ratio $\nu\{[-]; [0, 0.49999]; 0.3\}$ . Both will be used to compute shear Kirchoff modulus $G = \frac{E}{2(1 + \nu)}$
<b>Cross section</b>	obligatory; cross-section characteristics, setting dependent on analysis type: • $a\{[\text{m}]; [> 0]; 1\}$ – the distance between members with given section, for <b>Plane Strain</b> • $A\{[\text{m}^2]; [> 0]; 1\}$ – cross-section area, for all <b>Analysis Type</b> • $I_x\{[\text{m}^4]; [> 0]; 1\}$ – torsional momentum of inertia, for 3D only • $I_y\{[\text{m}^4]; [> 0]; 1\}$ — momentum of inertia in bending around local $Y_L$ axis (i.e. in XZ plane), for 3D only • $I_z\{[\text{m}^4]; [> 0]; 1\}$ – momentum of inertia in bending around local $Z_L$ axis (i.e. in XY plane), for all <b>Analysis Type</b> • $A_y = \kappa_y A\{[\text{m}^2]; [> 0]; 1\}$ – shear section area in $Y_L$ direction ( $\kappa_y$ is a shear correction factor), for all <b>Analysis Type</b> • $A_z = \kappa_z A\{[\text{m}^2]; [> 0]; 1\}$ – shear section area in $Z_L$ direction ( $\kappa_z$ is a shear correction factor) for 3D

#### Remarks:

- A: Hand setting of cross-sectional characteristics is possible only if **Type** is set to **Values** (default on entry)
- B: 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.
- C: 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.
- D: Large values of shear correction factor eliminate shear stiffness (switching to Kirchhoff-Bernoulli hypothesis instead of Timoshenko one).

2. 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	<p>optional</p> <p>If <b>Automatic evaluation of shear modulus</b> is set ✓ (default), Kirchhoff modulus <math>G</math> will be evaluated automatically basing on <math>E, \nu</math> of <b>Core material</b>. Otherwise set:</p> <ul style="list-style-type: none"> <li>• Young's elastic modulus <math>E</math> { [kPa]; [<math>&gt; 0</math>]; 100000 },</li> <li>• Poisons ratio <math>\nu</math> { [-]; [0, 0.49999]; 0.3 }.</li> </ul> <p>Both will be used only to create shear Kirchhoff modulus <math>G = \frac{E}{2(1 + \nu)}</math>, independently of <b>Core material</b> data.</p> <p>In any case, elasticity modulus <math>E_X</math> in axial direction has to be entered via <b>Cross section</b> → <b>Core material</b> → <b>Edit material</b></p>
Cross section	<p>obligatory</p> <p>Selection between section <b>Types</b> has to be done. Use alternatively:</p> <ul style="list-style-type: none"> <li>• <b>Profiles</b> - to set a typical manufactured (steel) section choosen form available data base.</li> <li>• <b>User</b> - to enter geometrical data of predefined types.</li> </ul> <p>In both cases cross section area will automatically be divided into layers with uni-axial elastic/elasto-plastic or user defined model marked as <b>Core</b> at each. In addition extra layers (possibly prestressed) can be added to the core material by activating <input checked="" type="checkbox"/> <b>Additional layers</b> option. The latter option can be used to model reinforced concrete, although setting beam reinforcement can be made much easier during preprocessing (under <b>Beam</b> use option <b>Reinforcement sets</b>). 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 <input checked="" type="checkbox"/> <b>Edit material</b> and for extra layers use <input checked="" type="checkbox"/> <b>Edit layers</b>. Moreover, cross sectional characteristics, related to shear or torsion, will be automatically evaluated:</p> <ul style="list-style-type: none"> <li>• <math>I_x</math> { [m<sup>4</sup>]; [<math>&gt; 0</math>]; 1 } – torsional momentum of inertia, for 3D only</li> <li>• <math>A_y = \kappa_y A</math> { [m<sup>2</sup>]; [<math>&gt; 0</math>]; 1 } – shear section area in <math>Y_L</math> direction (<math>\kappa_y</math> is a shear correction factor), for all <b>Analysis Type</b></li> <li>• <math>A_z = \kappa_z A</math> { [m<sup>2</sup>]; [<math>&gt; 0</math>]; 1 } – shear section area in <math>Z_L</math> direction (<math>\kappa_z</math> is a shear correction factor) for 3D</li> </ul> <p>For <b>Plane Strain</b> set:</p> <ul style="list-style-type: none"> <li>• <math>a</math> { [m]; [<math>&gt; 0</math>]; 1 } – the interval between beams with given section (leave <math>a = 1</math> in case of a continues lining).</li> </ul>
continued on next page	

continued from previous page

Heat

optional; set:  
•  $\alpha$ ,  $\{[C^{-1}]; [ > 0 ]; 1.0e^{-5}\}$  - 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.

## Related Topics

- *data preparation: Beam model*

### 1.3.3.17 AXISYMMETRIC SHELL MODEL (simple)

The model may be used in the context of **Axisymmetry Analysis Type** and exclusively for continuum shell type linings.

The following groups of parameters are common for both linear and nonlinear (layered) axisymmetric shell models:

Data	Use & Set up
<b>Unit Weight</b>	optional; <ul style="list-style-type: none"> <li><math>\gamma</math>, {[kNm<sup>-3</sup>]; [&gt; 0]; 0} - unit weight of the shell (averaged over the whole section);</li> <li>Density <math>\rho = \frac{\gamma}{g}</math> is automatically computed from <math>\gamma</math> (and vice versa)</li> <li>Multiplier for <math>\gamma</math> {[−]; [≥ 0]; 1.0}</li> <li>Multiplier for <math>\rho</math> {[−]; [≥ 0]; 1.0}</li> </ul> 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 <b>Dynamics</b> or <b>Pushover</b> is set ON in the project preselection).
<b>Heat</b>	optional $\alpha$ , {[C <sup>-1</sup> ]; [> 0]; 1.0e <sup>-5</sup> } - thermal dilatancy coefficient. In case of active <b>Data Group Heat</b> and active associated <b>Heat Transfer</b> project (if $\alpha > 0$ ) thermal strains will be computed.
<b>Stability</b>	optional Handles material strength parameters reduction, see: <b>LOCAL STABILITY</b> for the details.
<b>Damping</b>	Rayleigh damping factors $\alpha_o$ (applies to the mass) {[s <sup>-1</sup> ]; [≥ 0]; 0.0} and $\beta_o$ (applies to the stiffness) {[s]; [≥ 0]; 0.0}; dedicated calculator can be used if needed;

To complete data definition for linear elastic isotropic shell with constant thickness the following groups of parameters must be set:



Data	Use & Set up
<b>Elastic</b>	obligatory <ul style="list-style-type: none"> <li>Young's elastic modulus, <math>E</math> {[kPa]; [<math>&gt; 0</math>]; 100000},</li> <li>Poisson ratio <math>\nu</math> {[<math>-</math>]; [0, 0.49999]; 0.3}</li> </ul> $E$ and $\nu$ constant for whole section, used for all stiffness components
<b>Cross section</b>	obligatory <ul style="list-style-type: none"> <li>shell thickness <math>h</math> {[m]; [<math>&gt; 0</math>]; 1}</li> <li>shear correction factor <math>\kappa</math> {[<math>-</math>]; [<math>&gt; 0</math>]; 0.83333} The shear stiffness of the section will be then evaluated as: <math>GA_y = \kappa \frac{Eh}{2(1+\nu)}</math></li> </ul> NB. Note that large values of shear correction factor will eliminate shear stiffness (Kirchhoff–Bernoulli model)

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
<b>Elastic</b>	<p>optional</p> <p>If <b>Automatic evaluation of shear modulus</b> is set ✓ (default), Kirchhoff modulus <math>G</math> will automatically be evaluated basing on <math>E, \nu</math> of <b>Core</b> material.</p> <p>Otherwise set:</p> <ul style="list-style-type: none"> <li>• Young's elastic modulus <math>E\{[\text{kPa}]; [ &gt; 0]; 100000\}</math>,</li> <li>• Poisons ratio <math>\nu\{[-]; [0, 0.49999]; 0.3\}</math>.</li> </ul> <p>Both will be used only to compute shear Kirchhoff modulus</p> $G = \frac{E}{2(1 + \nu)}$ <p>independently of <b>Core</b> material data.</p> <p><b>Remark:</b></p> <p>In any case, elastic constants for each component separately, are set in the material data under group ☒ <b>Cross section</b> by activating ☒ <b>Layered cross section</b> option. <i>(to be continued on the next page)</i></p>

Data Group	Use & Set up
Cross section	<p>obligatory</p> <p>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:</p> <ul style="list-style-type: none"> <li>• Set shell thickness <math>h\{[m]; [ &gt; 0 ]; 1\}</math></li> <li>• The section will automatically be split into core material layers with bi-axial elastic or elasto-plastic model</li> <li>• To enter material model data for core use <b>Edit material</b> button. In the grid type dialog box edit elastic and plastic properties of the shell core material.</li> <li>• To activate reinforcing radial/circumferential fibers set ON the option <input checked="" type="checkbox"/> <b>Additional layers</b>; to edit properties of extra layers enter to the dialog box by pressing the button <b>Edit layers</b>; in the dialog define additional layers by setting <ul style="list-style-type: none"> <li>★ <b>Orientation</b> of the reinforcement in given layer switching between <b>Radial</b> and <b>Circumferential</b></li> <li>★ <b>Type</b> of the reinforcement area definition by selecting <b>Density</b> (standard setup in the practice where reinforcement is set with the unit <math>[m^2/m]</math> to preserve certain percentage) (valid for both radial and circumferential direction) or <b>Total</b> (kept mostly for back compatibility with previous versions) (here unit is <math>m^2</math>)(valid only for the radial direction)</li> <li>★ <b>Y-Position</b> of the given layer referred to the top, bottom or the middle of the cross section</li> <li>★ <b>Y-distance</b> that is an offset between selected core fibers defined in column <b>Y-Position</b> and current reinforcement layer</li> <li>★ <b>Area</b> of the reinforcement defined with unit that may depend on the reinforcement <b>Type</b></li> </ul> </li> </ul>

## Related Topics

- *data preparation: Axisymmetric shell model*
- *data preparation: Beam model*

### 1.3.3.18 SHELL SECTION MODEL

Data for linear (elastic) shell model

Data groups	Parameters
<b>Elastic</b>	obligatory (same as for continuum);
<b>Unit Weight</b>	optional (same as for continuum);
<b>Flow</b>	optional (same as for contact interface);
<b>Creep</b>	not supported (switch to nonlinear/layered shell model to activate it)
<b>Heat</b>	optional; $\alpha$ , $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group <input checked="" type="checkbox"/> <b>Heat</b> is active and preprocessed <b>Heat Transfer</b> project is declared thermal strains will be computed (if $\beta > 0$ ).
<b>Humidity</b>	optional; $\beta$ , $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient; In case when group <input checked="" type="checkbox"/> <b>Humidity</b> is active and preprocessed <b>Humidity Transfer</b> project is declared hygral strains will be computed (if $\beta > 0$ ).
<b>Damping</b>	optional (same definition as in the global <b>Control/Dynamics dialog</b> )

**Elastic**, which may be used if the section is homogeneous and shell material is linearly elastic, exclusively. Activation of this case is done after setting **Non--Linear** OFF.

Data	Use & Set up
<b>Elastic</b>	obligatory <ul style="list-style-type: none"> <li>Young's elastic modulus, <math>E</math> <math>\{[\text{kPa}]; [&gt; 0]; 100000\}</math>,</li> <li>Poisson ratio <math>\nu</math> <math>\{[-]; [0, 0.49999]; 0.3\}</math></li> </ul>

Data groups	Parameters
<b>Unit Weight</b>	optional (same as for continuum);
<b>Flow</b>	optional (same as for contact interface);
<b>Non--Linear</b>	obligatory; definition of layered cross section;
<b>Damping</b>	optional (same definition as in the global <b>Control/Dynamics dialog</b> )

### Related Topics

- data preparation: Models for shells*

- *data preparation: Shell section model*
- *data preparation: Shell component model*

### 1.3.3.19 SHELL COMPONENT MODEL

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
<b>Elastic</b>	obligatory; <ul style="list-style-type: none"> <li>Young's elastic modulus <math>E</math> {[kPa]; [<math>&gt; 0</math>]; 100000}</li> <li>Poisons ratio <math>\nu</math> {[<math>-</math>]; [0, 0.49999]; 0.3},</li> </ul>
<b>Creep</b>	optional (same as for continuum) ;
<b>Heat</b>	optional; $\alpha$ , {[ $C^{-1}$ ]; [ $> 0$ ]; $1.0e^{-5}$ } - thermal dilatancy coefficient In case when group <input checked="" type="checkbox"/> <b>Heat</b> is active and preprocessed <b>Heat Transfer</b> project is declared thermal strains will be computed (if $\beta > 0$ ).
<b>Humidity</b>	optional; $\beta$ , {[ $-$ ]; [ $\geq 0$ ]; 0.01} - hygral dilatancy coefficient; In case when group <input checked="" type="checkbox"/> <b>Humidity</b> is active and preprocessed <b>Humidity Transfer</b> project is declared hygral strains will be computed (if $\beta > 0$ ).
<b>Non--Linear</b>	<ul style="list-style-type: none"> <li><b>Elastic</b>, <b>Mohr--Coulomb (M--W)</b>, <b>Hoek--Brown (M--W)</b>, <b>Aging Concrete</b> shells, <b>Plastic damage (concrete)</b> shells – obligatory set data as for continuum models</li> <li><b>Huber--Mises</b> shell – obligatory; set: <ul style="list-style-type: none"> <li>★ uniaxial tensile strength <math>f_t</math>, {[kPa]; [<math>&gt; 0</math>]; 1}</li> </ul> </li> </ul>
<b>Stability</b>	optional for <b>Deformation</b> , <b>Deformation+Flow</b> ;

Material data for reinforcement material model: **Fiber** (shell)

Data groups	Parameters
<b>Elastic</b>	obligatory; Young's elastic modulus, $E$ {[kPa]; [ $> 0$ ]; 100000} 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)
<b>Creep</b>	not supported
<b>Heat</b>	optional; $\alpha$ , {[ $C^{-1}$ ]; [ $> 0$ ]; $1.0e^{-5}$ } - thermal dilatancy coefficient In case when group <input checked="" type="checkbox"/> <b>Heat</b> is active and preprocessed <b>Heat Transfer</b> 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)
<b>Humidity</b>	optional; $\beta$ , {[ $-$ ]; [ $\geq 0$ ]; 0.01} - hygral dilatancy coefficient; In case when group <input checked="" type="checkbox"/> <b>Humidity</b> is active and preprocessed <b>Humidity Transfer</b> project is declared (if $\beta > 0$ ) hygral strains will be computed.
<b>Non--Linear</b>	optional; <ul style="list-style-type: none"> <li>Uniaxial tensile strength <math>f_t</math>, {[kPa]; [<math>f_t &gt; 0</math>]; 1}</li> <li>Uniaxial compressive strength <math>f_c</math>, {[kPa]; [<math>f_c \geq 0</math>]; 1}</li> </ul> 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)
<b>Geometry</b>	obligatory; <ul style="list-style-type: none"> <li>Set <math>X</math>, <math>Y</math>, <math>Z</math> being the coordinates of an arbitrary direction vector in 3D space indicating an orientation of fibers in 3D space;</li> </ul>
<b>Stability</b>	optional for <b>Deformation</b> , <b>Deformation+Flow</b>

## Related Topics

- data preparation: Models for shells*
- data preparation: Shell section model*
- data preparation: Shell component model*

### 1.3.3.20 MEMBRANE–FIBER MODEL (simple)

This model represents a layer of equally spaced reinforcement bars.

Data groups	Parameters
<b>Elastic</b>	obligatory; Young's elastic modulus, $E$ {[kPa]; [ $> 0$ ]; 100000}
<b>Unit Weight</b>	optional; <ul style="list-style-type: none"> <li>• <math>\gamma</math>, {[kNm<sup>-2</sup>]; [<math>&gt; 0</math>]; 0} - unit weight of fiber material</li> <li>• Density <math>\rho = \frac{\gamma}{g}</math> is automatically computed from <math>\gamma</math> (and vice versa)</li> <li>• Multiplier for <math>\gamma</math> {[<math>-</math>]; [<math>\geq 0</math>]; 1.0}</li> <li>• Multiplier for <math>\rho</math> {[<math>-</math>]; [<math>\geq 0</math>]; 1.0}</li> </ul> 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 <b>Dynamics</b> or <b>Pushover</b> is set ON in the project preselection).
<b>Creep</b>	not supported
<b>Heat</b>	optional; $\alpha$ , {[C <sup>-1</sup> ]; [ $> 0$ ]; $1.0e^{-5}$ } - thermal dilatancy coefficient In case when group <input checked="" type="checkbox"/> <b>Heat</b> is active and preprocessed <b>Heat Transfer</b> project is declared (if $\alpha > 0$ ) thermal strains will be computed.
<b>Humidity</b>	optional; $\beta$ , {[ $-$ ]; [ $\geq 0$ ]; 0.01} - hygral dilatancy coefficient; In case when group <input checked="" type="checkbox"/> <b>Humidity</b> is active and preprocessed <b>Humidity Transfer</b> project is declared (if $\beta > 0$ ) hygral strains will be computed.
<b>Non--Linear</b>	optional <ul style="list-style-type: none"> <li>• Uniaxial tensile strength <math>f_t</math>, {[kPa]; [<math>f_t &gt; 0</math>]; 1}</li> <li>• Uniaxial compressive strength <math>f_c</math>, {[kPa]; [<math>f_c \geq 0</math>]; 1}</li> </ul>
<b>Geometry</b>	obligatory <ul style="list-style-type: none"> <li>• For <b>Plane Strain</b>:               <ul style="list-style-type: none"> <li>★ Area per unit length <math>A</math>, {[m<sup>2</sup>/m=m], [<math>&gt; 0</math>], [1]}</li> </ul> </li> <li>• For <b>Axisymmetry</b>:               <ul style="list-style-type: none"> <li>★ Direction: one of: <input checked="" type="radio"/> <b>Circumferential</b> or <input checked="" type="radio"/> <b>Longitudinal</b></li> <li>★ Area per unit length <math>A</math>, {[m<sup>2</sup>/m=m], [<math>&gt; 0</math>], [1]}</li> </ul> </li> <li>• For <b>3D</b>:               <ul style="list-style-type: none"> <li>★ Set <math>X</math>, <math>Y</math>, <math>Z</math> being the coordinates of an arbitrary direction vector in 3D space indicating an orientation of fibers in 3D space;</li> <li>★ Area per unit length <math>A</math>, {[m<sup>2</sup>/m=m], [<math>&gt; 0</math>], [1]}</li> </ul> </li> </ul>
<b>Damping</b>	optional (same definition as in the global <b>Control/Dynamics dialog</b> )



## Related Topics

- *data preparation: Models for membranes*
- *data preparation: Membrane – fiber model*
- *data preparation: Membrane – plane stress models*
- *data preparation: Membrane – fabric models*

### 1.3.3.21 MEMBRANE-PLANE STRESS MEMBER MODELS (simple)

Models are destined to describe structural members under plane stress state condition. Contrary to **Membrane--Fabric** these models require setting of the **Thickness** of a member as a geometrical data.

Data groups	Parameters
<b>Elastic</b>	obligatory (same as for continuum);
<b>Unit Weight</b>	optional; <ul style="list-style-type: none"> <li>• <math>\gamma</math>, {[kNm<sup>-3</sup>]; [&gt; 0]; 0} - unit weight</li> <li>• Density <math>\rho = \frac{\gamma}{g}</math> is automatically computed from <math>\gamma</math> (and vice versa)</li> <li>• Multiplier for <math>\gamma</math> {[−]; [≥ 0]; 1.0}</li> <li>• Multiplier for <math>\rho</math> {[−]; [≥ 0]; 1.0}</li> </ul> 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 <b>Dynamics</b> or <b>Pushover</b> is set ON in the project preselection).
<b>Geometry</b>	obligatory; Thickness $t$ , {[m]; [> 0]; 0.1}
<b>Creep</b>	not supported
<b>Heat</b>	optional; $\alpha$ , {[C <sup>-1</sup> ]; [> 0]; 1.0e <sup>-5</sup> } - thermal dilatancy coefficient In case when group <input checked="" type="checkbox"/> <b>Heat</b> is active and preprocessed <b>Heat Transfer</b> project is declared thermal strains will be computed (if $\beta > 0$ ).
<b>Humidity</b>	optional; $\beta$ , {[−]; [≥ 0]; 0.01} - hygral dilatancy coefficient; In case when group <input checked="" type="checkbox"/> <b>Humidity</b> is active and preprocessed <b>Humidity Transfer</b> project is declared hygral strains will be computed (if $\beta > 0$ ).
<b>Non--Linear</b>	set: <ul style="list-style-type: none"> <li>• For <b>Huber--Mises</b> and <b>Rankine (M--W)</b> (obligatory):               <ul style="list-style-type: none"> <li>★ Uniaxial tensile strength <math>f_t</math>, {[kPa]; [≥ 0]; 1}</li> </ul> </li> <li>• For <b>Hoek--Brown (M--W)</b> (obligatory):               <ul style="list-style-type: none"> <li>★ Uniaxial tensile strength <math>f_t</math>, {[kPa]; [≥ 0]; 1}</li> <li>★ Uniaxial compressive strength <math>f_c</math>, {[kPa]; [<math>f_c &gt; f_t \geq 0</math>]; 1}</li> </ul> </li> </ul>
<b>Damping</b>	optional (same definition as in the global <b>Control/Dynamics dialog</b> )

## Related Topics

- *data preparation: Models for membranes*

- *data preparation: Membrane – fiber model*
- *data preparation: Membrane – plane stress models*
- *data preparation: Membrane – fabric models*

### 1.3.3.22 MEMBRANE–FABRIC MODELS (simple)

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;
Creep	not supported
Heat	optional; $\alpha$ , $\{[C^{-1}]; [ > 0 ]; 1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group <input type="checkbox"/> <b>Heat</b> is active and preprocessed <b>Heat Transfer</b> project is declared thermal strains will be computed (if $\beta > 0$ ).
Humidity	optional; $\beta$ , $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient; In case when group <input type="checkbox"/> <b>Humidity</b> is active and preprocessed <b>Humidity Transfer</b> project is declared hygral strains will be computed (if $\beta > 0$ ).
Damping	optional (same definition as in the global <i>Control/Dynamics dialog</i> )

Isotropic membrane: specific data.

Data	Use & Set up
Elastic	obligatory; <ul style="list-style-type: none"> <li>Elastic stiffness <math>K</math> (equivalent to <math>E \times \text{thickness}</math>) <math>\{[kN/m]; [ &gt; 0 ]; 100000\}</math>,</li> <li>Poisson ratio <math>\nu</math>, <math>\{[-]; [0, 0.49999]; 0.3\}</math></li> </ul>
Non--Linear	optional If not activated elastic membrane with unlimited strength will be considered <ul style="list-style-type: none"> <li>Uniaxial tensile strength <math>f_t</math>, <math>\{[kN/m]; [ &gt; 0 ]; 1\}</math></li> <li>Uniaxial compressive strength <math>f_c</math>, <math>\{[kN/m]; [ &gt; 0 ]; 1\}</math></li> </ul>

Anisotropic membrane: specific data.

Data	Use & Set up
<b>Elastic</b>	obligatory; <ul style="list-style-type: none"> <li>Elastic stiffness in anisotropy direction <math>K_{XX}</math>, <math>K_{YY}</math>, <math>K_{XY}</math> {[kN/m]; [<math>&gt; 0</math>]; 100000},</li> </ul>
<b>Geometry</b>	obligatory $X$ , $Y$ , $Z$ the coordinates of an arbitrary direction vector in 3D space. It determines anisotropy axis $X$
<b>Non--Linear</b>	optional If not activated elastic membrane with unlimited strength will be considered <ul style="list-style-type: none"> <li>Uniaxial tensile strength in anisotropy direction X,Y: <math>f_tX</math>, <math>f_tY</math>, {[kN/m]; [<math>\geq 0</math>]; 1}</li> <li>Uniaxial compressive strength in anisotropy direction X,Y: <math>f_cX</math>, <math>f_cY</math>, {[kN/m]; [<math>\geq 0</math>]; 1}</li> </ul>

## Related Topics

- *data preparation: Models for membranes*
- *data preparation: Membrane – Fiber Model*
- *data preparation: Membrane – Plane Stress Models*
- *data preparation: Membrane – Fabric Models*

### 1.3.3.23 HEAT CONVECTION (advanced)

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 <ul style="list-style-type: none"> <li>Heat convection coefficient <math>h_T</math> {[kN/(m · °K · s)], &gt; 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)</li> </ul>

### Related Topics

- *data preparation: Heat convection*
- *data preparation: Humidity convection*
- *data preparation: Contact*
- *data preparation: Seepage*

### 1.3.3.24 HUMIDITY CONVECTION

The model is applicable exclusively to **Humidity Problem Type** <sup>5</sup>.

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
<b>Humidity</b>	obligatory; <ul style="list-style-type: none"> <li>Humidity convection coefficient <math>h_W</math> <math>\{[1/(\text{m}^2 \cdot \text{d})], &gt; 0, 10\}</math></li> </ul>

#### Related Topics

- *data preparation: Heat convection*
- *data preparation: Humidity convection*
- *data preparation: Contact*
- *data preparation: Seepage*

<sup>5</sup>concerns versions: **ACADEMIC, PROFESSIONAL, EXPERT** only

## 1.3.3.25 CONTACT (simple)

Data	Use & Set up
Elastic	<p>optional</p> <ul style="list-style-type: none"> <li>• <b>Direct values of interface stiffness</b> 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: <ul style="list-style-type: none"> <li>★ normal stiffness <math>K_n</math> multiplier <math>\{[-], [ &gt; 0], [0.01]\}</math> i.e.  <math>K_n = \text{multiplier} * K_n^{def}</math></li> <li>★ tangential/normal stiffness ratio <math>K_t/K_n</math> <math>\{[-], [ &gt; 0], [0.01]\}</math></li> </ul> </li> <li>• <b>Direct values of interface stiffness</b> is set ON. User must set elastic properties of the interface explicitly specifying: <ul style="list-style-type: none"> <li>★ normal stiffness <math>K_n</math>, <math>\{[kN/m^3], [ &gt; 0], [10000]\}</math>  <math>K_n</math> is equivalent to <math>K_n = E/h</math> where <math>h</math> is the depth of the very thin weak (interface) layer and <math>E</math> is its elasticity modulus.</li> <li>★ tangential stiffness <math>K_t</math>, <math>\{[kN/m^3], [ &gt; 0], [10000]\}</math>  <math>K_t</math> is equivalent to <math>K_t = G/h</math> where <math>h</math> is the depth of the weak layer and <math>G</math> is its shear modulus.</li> </ul> </li> </ul>
Flow	<p>optional</p> <p>User may choose between <b>Impermeable Surface</b> of the interface (<math>k = 0</math>) and <b>Fully Permeable Surface</b> with permeability <math>k</math> estimated by the program as large enough to assure pressure compatibility, or as <b>Permeable Surface</b> with permeability given by the user .</p> <p>Following data have to be entered only if <b>Fully Permeable Surface</b> is set OFF.</p> <ul style="list-style-type: none"> <li>• permeability in the normal direction <math>-k'_z</math> <math>\{[1/s], [ &gt; 0], [0.01]\}</math></li> <li>• in-plane permeability <math>-k'_x, k'_y</math>. <math>\{m^2/s, &gt; 0, 1\}</math></li> </ul> <p>If <b>Anisotropic Flow</b> in interface is set ON, then in-plane permeabilities <math>k'_x, k'_y</math> have to be set in 2 principal anisotropy directions. The anisotropy direction of <math>x'</math> will be evaluated as the projection of <b>Orientation Vector</b> for anisotropy <math>\{v_x, v_y, v_z\}</math>.</p> <p>If set OFF in-plane flow properties are assumed to be isotropic characterized by permeability <math>k'_x</math>.</p>



Non--Linear	<p>obligatory</p> <p>If ☉ <b>Direct input</b> is set ON data for Coulomb type frictional law describing contact interface material model are:</p> <ul style="list-style-type: none"> <li>• cohesion <math>c</math>, {[kPa], <math>[\geq 0]</math>, 0} (valid only for small deformation contact formulation)</li> <li>• friction angle <math>\phi</math>, {[deg], [0, 89], 0}</li> <li>• dilatancy angle <math>\psi</math>, {[deg], [0, 89], 0} (valid only for small deformation contact formulation)</li> </ul> <p>If ☉ <b>Set table of scaling factors to cohesion, friction...</b> is set ON then user may set up multipliers to <math>\tan(\phi)</math>, <math>c</math> and <math>\tan(\psi)</math> at each material for <b>Continuum</b> or <b>Continuum for structures</b>; 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 <b>Setup</b>. User may set several tables with scaling factors but only one can be applied to the given interface material.</p> <p><b>Remark:</b> Tensile Cut--Off is implicitly activated in the model</p>
-------------	--

## Related Topics

- *data preparation: Heat convection*
- *data preparation: Humidity convection*
- *data preparation: Contact*
- *data preparation: Seepage*

### 1.3.3.26 SEEPAGE (simple)

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	<p>obligatory;</p> <ul style="list-style-type: none"> <li>• <b>Multiplier</b> to <math>K_v^{ref}</math>, <math>\epsilon</math>, <math>\{[-], [ &gt; 0], 1\}</math></li> <li>• Direct value of <math>K_v</math> <math>\{[m^3/(kN\ s)], [ &gt; 0], 1\}</math></li> <li>• <b>Multiplier</b> to <math>K_v^{ref}</math>, <math>\{[-], [ &gt; 0], 1\}</math></li> </ul> <p>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 <b>Multiplier</b>; this may however lead to convergence problems. In special cases when modeling drains with limited influx use second (direct value of <math>K_v</math>) or third setting (user given multiplier to the reference value <math>K_v^{ref}</math>)</p>

### Related Topics

- *data preparation: Heat convection*
- *data preparation: Humdity convection*
- *data preparation: Contact*
- *data preparation: Seepage*

## 1.4 ANALYSIS

Three modes of computational analysis are possible:

Run Analysis	Standard mode. Saving job file *.inp, generating the *.dat file and running computation driven by the list of declared drivers (under <b>Analysis &amp; Driver</b> ) from the beginning (all previous results are deleted).
Restart	Running computation from the point of last termination with possible modifications like changing material properties, adding new drivers, changing nonlinear solver settings which happens in case of numerical divergence, driver parameters (like reduced time step for instance) etc... <b>At this stage any change to the mesh, types of finite elements, list of drivers, which were already terminated, content of the results, Analysis and Problem Type is not allowed.</b>
Batch processing	Creates a list of input files to be computed in the batch mode; manages stored *.bat files and runs existing *.bat files

## 1.5 POSTPRO

<b>Postprocessing</b>	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.
<b>Pushover Results</b>	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.
<b>Results for piles</b>	Shows results for piles, modeled as beams embedded in the 3D continuum, in the form of Excel sheet.
<b>Results for fix.anch.zones</b>	Shows results for fixed anchor zones in the form of Excel sheet.
<b>Eigen vectors</b>	Exports eigenfrequencies, mass participation factors and eigenvectors to the Excel sheet.
<b>View log file *.log</b>	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.
<b>View element results</b>	Browse the file (*.str) containing results stored at Gauss points at each element according to the specification set under <b>Control/Results content</b> menu.
<b>View nodal results</b>	Browse the file (*.dis) containing results stored at nodal points according to the specification set under <b>Control/Results content</b> menu.
<b>View echo of data</b>	Browse the file (*.eda) containing echo of the data (*.dat) file generated by the menu and sent to the calculation module.

## 1.6 EXTRAS

Dialog for edition of results, data files in the text form (ASCII):

<b>Run Analysis</b>	Calls computational module without storing job file (*.inp) nor creating *.dat file. Use in case when *.dat file is modified manually.
<b>Write *.dat</b>	Creates data file (*.dat) for the computation module without calling it
<b>View *.dat</b>	Edits compiled data file (*.dat) (from *.inp) allowing for its modification (can be used by experienced users only and no technical support is foreseen if errors occur)
<b>View *.inp</b>	Edits standard input file (*.inp) allowing for its modification (risk of complete loss of data)
<b>Units</b>	Activates a tool to convert units
<b>Databases</b>	Gives an access to the steel profiles data base through submenu <b>Database with steel profiles</b> and stored materials database through submenu <b>Database with material data</b>
<b>Protection report</b>	Edits ASCII file containing protection report

## 1.7 SYSTEM CONFIGURATIONS

Version type ►	Set up version type, e.g. basic / advanced
Acrobat Reader path ►	Enter the path to ACROBAT READER to be used to view help files, created in PDF format
ASCII files editor ►	Attach external executable for edition of ASCII files (under <b>Extras</b> ) e.g.: NOTEPAD.EXE
CSV files editor ►	Attach external executable for edition of CSV files (MS-EXCEL default)
Executable files ►	Set up paths to Z_SOIL executable files
Preferences ►	Set up a palette of colors used for visualisation of different components of the model in the graphical pre- and post-processor.

## 1.8 HELP

Invoke help files using Acrobat Reader.

The ZSoil® help consists of following parts:

- [QUICK HELP](#)
- [DATA PREPARATION MANUAL](#)
- [THEORY MANUAL](#)
- [BENCHMARKS](#)
- [TUTORIALS](#)
- [WHITE PAPER](#)
- [SYSTEM REQUIREMENTS](#)