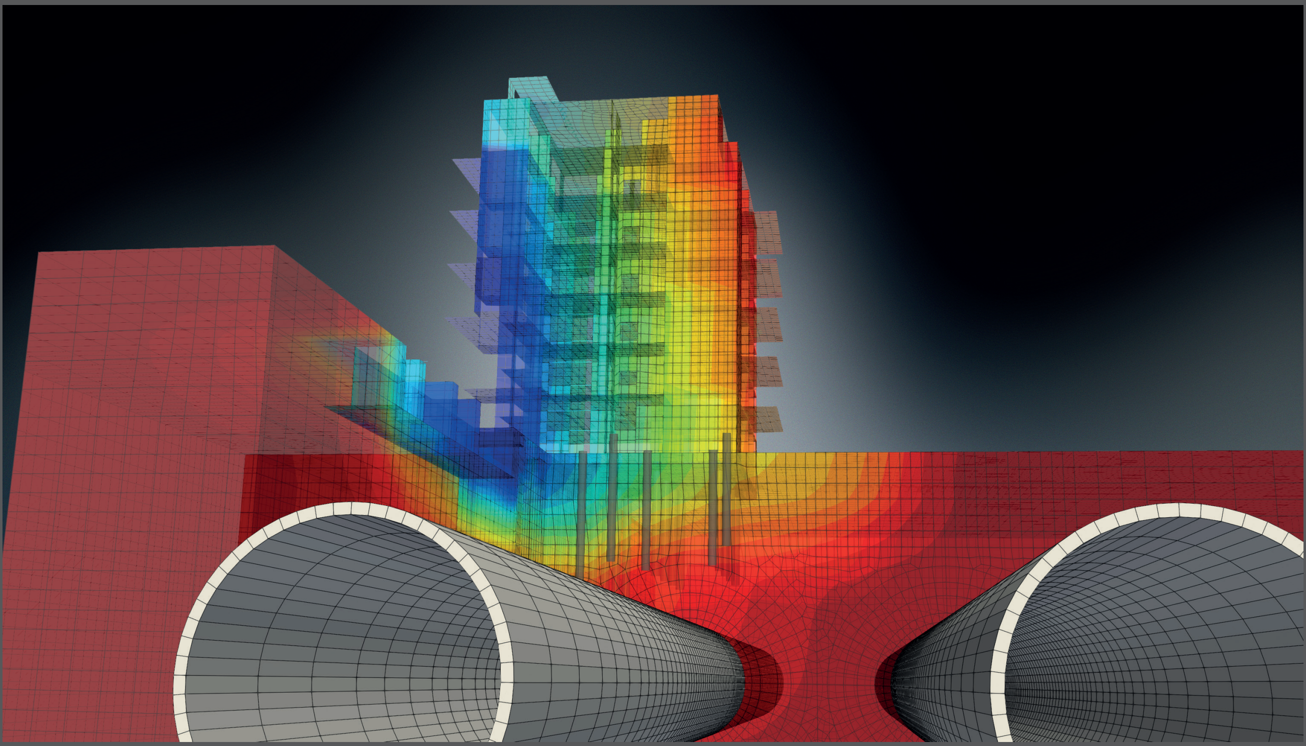
ZSoil

for geotechnics & structures

User manual
QUICK HELP



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

zsoil.com

ZSoil

for geotechnics & structures

QUICK HELP

User manual

ZSoil®2026

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

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

WARNING

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

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

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

ISBN 2-940009-08-2

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

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

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

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

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

1. GRANT OF LICENSE

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

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

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

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

3. OTHER RIGHTS AND LIMITATIONS

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

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

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

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

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

WARRANTIES & LIMITATIONS TO WARRANTIES

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

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

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

NO OTHER WARRANTIES.

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

NO LIABILITY FOR CONSEQUENTIAL DAMAGES.

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

OTHER PROVISIONS.

SUPPORT: If included in license price, assistance will be provided by **GeoDev Sarl**, by e-mail exclusively, during the first year following purchase. This service excludes all forms of consulting on actual projects. Installation support is limited to initially supported OS and a four year duration.

PROFESSIONAL VERSIONS of **ZSOIL** are meant to be used in practice & in research centers.

ACADEMIC VERSIONS of **ZSOIL** are meant to be used exclusively for teaching and research in academic institutions.

ACADEMIC WITH CONSULTING VERSIONS of **ZSOIL** are meant to be used exclusively for teaching, research and consulting in academic institutions.

The terms of this agreement may be amended in the future, by **GeoDev Sarl**, when necessary. In such cases the revised agreement will be resubmitted for user approval on the software's front screen.

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

LAUSANNE 15.11.2025

Chapter 1

MENU CONTENTS DESCRIPTION

The ZSoil® 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*" where *i* is a subsequent number of the job during the session. Calls the dialog to select **Analysis Type** 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 indiced job, closing (and saving) the recent one.

Related Topics

- *data preparation: Batch Processing*

1.2 CONTROL

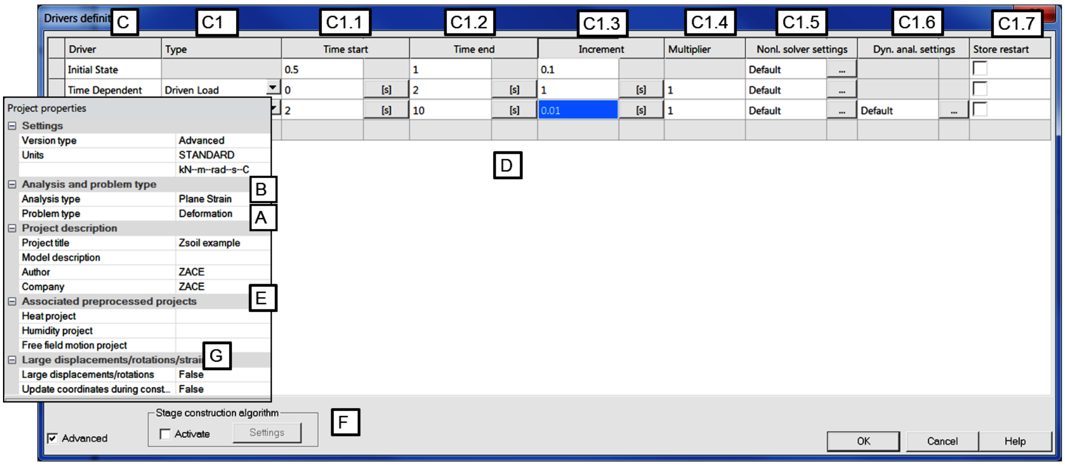
Following options allow to control computational analysis

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



	Driver	Type	Time start		Time end		Increment		Multiplier	Nonl. solver settings		Dyn. anal. settings		Store restart
	Initial State		0.5		1		0.1			Default	...			☐
	Time Dependent	Driven Load	0	[s]	2	[s]	1	[s]	1	Default	...			☐
			2	[s]	10	[s]	0.01	[s]	1	Default	...	Default	...	☐

Project properties

⊟ Settings	
Version type	Advanced
Units	STANDARD
	kN--m--rad--s--C
⊟ Analysis and problem type	
Analysis type	Plane Strain
Problem type	Deformation
⊟ Project description	
Project title	Zsoil example
Model description	
Author	ZACE
Company	ZACE
⊟ Associated preprocessed projects	
Heat project	
Humidity project	
Free field motion project	
⊟ Large displacements/rotations/strai	
Large displacements/rotations	False
Update coordinates during const...	False

☑ Advanced

Stage construction algorithm

☐ Activate Settings

	Key	Options	Comment
A	Problem Type	Deformation	for single phase analysis
		Deformation+flow	for 2 phase analysis
		Flow	for single phase flow analysis
		Heat	for thermal diffusion analysis
		Humidity	for humidity diffusion analysis
B	Analysis	3D	
		Plane strain	
		Axisymmetry	
C	Driver		Depends on **A**
C1	Driver Type		Depends on **C**
C1.1	Start		Context depends on **C1**
C1.2	End		Context depends on **C1**
C1.3	Increment		Context depends on **C1**
C1.4	Multiplier		Context depends on **C1**
C1.5	Nonlinear solver settings		Set algorithm , convergence tolerances, storage frequency
C1.6	Dynamic anal.settings		Set parameters for dynamic driver (algorithm, damping coeff.)
C1.6	Pushover anal.settings		Set parameters for pushover driver (force pattern, direction)
D	Drivers list		Displays the sequence of drivers
E	Associated projects		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
C	`Driver`	`Initial State`	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)
C1	`Driver Type`	none	
C1.1	`Start`		Initial value of the load factor (default 1.0, must be greater than 0.0)
C1.2	`End`		Final value of the load factor (default 1.0)
C1.3	`Increment`		Load factor increment
C1.4	`Multiplier`		Multiplier of load factor increment
C	`Driver`	`Time dependent`	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
C1	`Driver Type`	`Driven load`	
C1.1	`Start`		Time at which driver is switched on
C1.2	`End`		Time at which driver is switched off
C1.3	`Increment`		Initial time step increment
C1.4	`Multiplier`		Multiplier of time step increment (meaningful mainly for creep analysis)

	Key	Options	Comment
C	`Driver`	`Stability`	Use to compute safety factor (SF) and failure pattern at present state (time) through successive reduction of the yield surface
C1	`Driver Type`	$C - tan(\phi)$	Define SF as divisor of both cohesion and friction angle
		`Stress level`	Define SF as the maximum possible multiplier of deviatoric stress
		C	Define SF as divisor of cohesion exclusively
C1.1	`Start`		Initial value of safety factor
C1.2	`End`		Final value of safety factor
C1.3	`Increment`		Safety factor increment
C1.4	`Multiplier`		not meaningful (kept equal to 1.0)
C	`Driver`	`Dynamics`	Use to carry out time history analysis of structures (continuum elements cannot be used !)
C1	`Driver Type`	`Driven load`	
C1.1	`Start`		Time at which driver is switched on
C1.2	`End`		Time at which driver is switched off
C1.3	`Increment`		Initial time step increment
C1.4	`Multiplier`		Multiplier of time step increment
C	`Driver`	`Pushover`	Use to estimate bearing capacities of structures (continuum elements cannot be used !) subject to seismic risk
C1	`Ctrl node label`	`user def.node`	Use to indicate a node (obligatory) to check target displacement
C1.1	`Start`		Initial displacement increment
C1.2	`End`		Target displacement
C1.3	`Increment`		Displ. increment reduction factor
C1.4	`Multiplier`		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
C	`Driver`	`Initial State`	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
C1	`Driver Type`	none	
C1.1	`Start`		Initial value of the load factor (default 1.0, must be greater than 0.0)
C1.2	`End`		Final value of the load factor (default 1.0)
C1.3	`Increment`		Load factor increment
C1.4	`Multiplier`		Multiplier of load factor increment
C	`Driver`	`Time dependent`	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
C1	`Driver Type`	`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`	Undrained total stress analysis
		`Consolidation`	Fuly coupled total stress analysis
C1.1	`Start`		Time at which driver is switched on
C1.2	`End`		Time at which driver is switched off
C1.3	`Increment`		Initial time step increment
C1.4	`Multiplier`		Multiplier of time step increment (meaningful mainly for consolidation or creep analysis)

	Key	Options	Comment
C	`Driver`	`Stability`	Use to compute safety factor (SF) and failure pattern at present state (time) through successive reduction of the yield surface
C1	`Driver Type`	$C - tan(\phi)$	Define SF as divisor of both cohesion and friction angle
		`Stress level`	Define SF as the maximum possible multiplier of deviatoric stress
		C	Define SF as divisor of cohesion exclusively
C1.1	`Start`		Initial value of safety factor
C1.2	`End`		Final value of safety factor
C1.3	`Increment`		Safety factor increment
C1.4	`Multiplier`		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
C	`Driver`	`Initial State`	Use to compute initial pore pressures assuming steady state conditions at time t=0
C1	`Driver Type`	none	
C1.1			**C1.1**÷**C1.4** are all not meaningful
C1.2			
C1.3			
C1.4			
C	`Driver`	`Time dependent`	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
C1	`Driver Type`	`Steady state`	Sequence of steady state solutions of Darcy flow
		`Transient`	Evolution of the fluid pressure and velocity fields in time due to transient boundary conditions
C1.1	`Start`		Time at which driver is switched on
C1.2	`End`		Time at which driver is switched off
C1.3	`Increment`		Initial time step increment
C1.4	`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
C	`Driver`	`Initial State`	Use to compute temperature field assuming steady state conditions at time t=0
C1	`Driver Type`	none	
C1.1			**C1.1**÷**C1.4** are all not meaningful
C1.2			
C1.3			
C1.4			
C	`Driver`	`Time dependent`	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
C1	`Driver Type`	`Steady state`	Sequence of steady state solutions of heat transfer
		`Transient`	Evolution of the temperature field due to transient boundary conditions
C1.1	`Start`		Time at which driver is switched on
C1.2	`End`		Time at which driver is switched off
C1.3	`Increment`		Initial time step increment
C1.4	`Multiplier`		Multiplier of time step increment (meaningful mainly for transient case)

DRIVERS DATA for `Problem Type Humidity`

Use for humidity transfer analysis[1].

See also Data Preparation: HUMIDITY TRANSFER

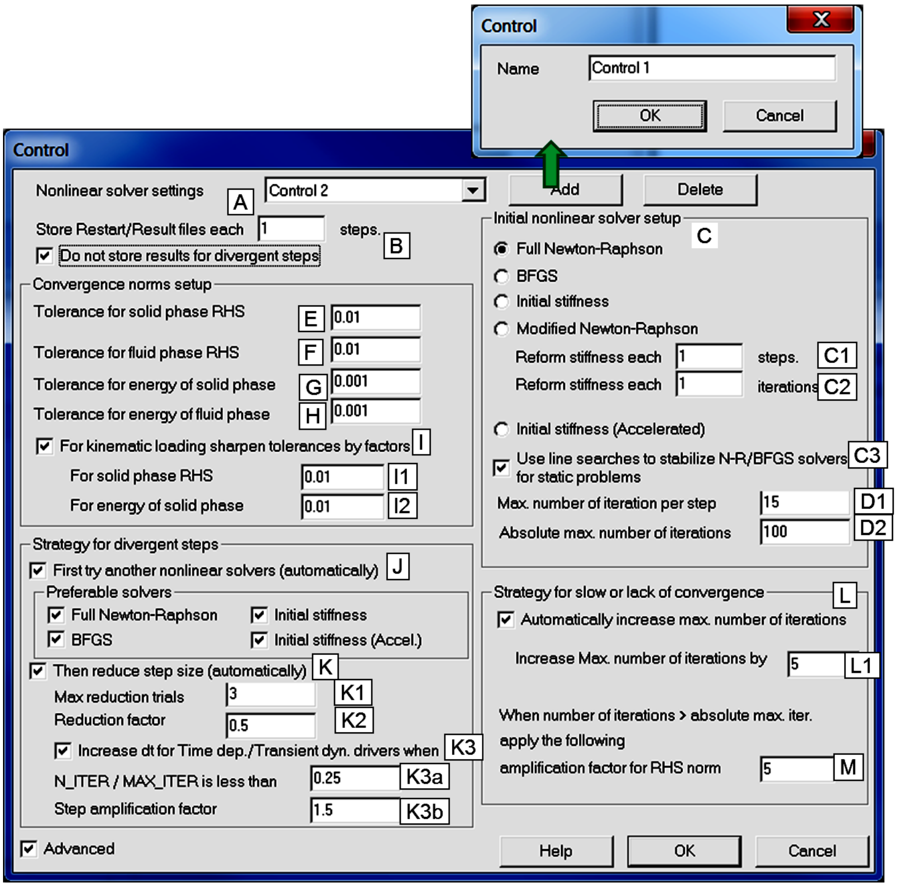
	Key	Options	Comment
C	`Driver`	`Initial State`	Use to compute relative humidity field assuming steady state conditions at time t=0
C1	`Driver Type`	none	
C1.1			**C1.1**÷**C1.4** are all not meaningful
C1.2			
C1.3			
C1.4			
C	`Driver`	`Time dependent`	Use to compute problems with variable in time geometries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

---

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

1.2.2 CONTROL

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



Control

Name: Control 1

Control

Nonlinear solver settings: Control 2

Store Restart/Result files each 1 steps.

☑ Do not store results for divergent steps

Convergence norms setup

Tolerance for solid phase RHS: 0.01

Tolerance for fluid phase RHS: 0.01

Tolerance for energy of solid phase: 0.001

Tolerance for energy of fluid phase: 0.001

☑ For kinematic loading sharpen tolerances by factors

For solid phase RHS: 0.01

For energy of solid phase: 0.01

Strategy for divergent steps

☑ First try another nonlinear solvers (automatically)

Preferable solvers

☑ Full Newton-Raphson ☑ Initial stiffness

☑ BFGS ☑ Initial stiffness (Accel.)

☑ Then reduce step size (automatically)

Max reduction trials: 3

Reduction factor: 0.5

☑ Increase dt for Time dep./Transient dyn. drivers when

N_ITER / MAX_ITER is less than: 0.25

Step amplification factor: 1.5

☑ Advanced

Initial nonlinear solver setup

● Full Newton-Raphson

○ BFGS

○ Initial stiffness

○ Modified Newton-Raphson

Reform stiffness each 1 steps.

Reform stiffness each 1 iterations

○ Initial stiffness (Accelerated)

☑ Use line searches to stabilize N-R/BFGS solvers for static problems

Max. number of iteration per step: 15

Absolute max. number of iterations: 100

Strategy for slow or lack of convergence

☑ Automatically increase max. number of iterations

Increase Max. number of iterations by: 5

When number of iterations > absolute max. iter. apply the following amplification factor for RHS norm: 5

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 (Δt, SF or increment of gravity load for the `Initial state`)

K3. activates adaptive time step (for `Time dependent` or `Transient dynamics` drivers) amplification (if previously the step was reduced)

K3a. time step size can be amplified when ratio of number of iterations needed to achieve the convergence state to the declared max. number of iterations per step (D1) is less than given value

K3b. time step size, if possible, will be amplified by a given factor (larger than 1.0 but less than 1.0 divided by the reduction factor K2)

L. flag that allows the calculation module to increase maximum number of iterations once the iterative scheme is converging slowly

L1. once the flag (L) is set on maximum number of iterations will be increased by a given value

M. once the absolute maximum number of iterations (D2) is exceeded the RHS tollerance is amplified by a given factor

Remarks:

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

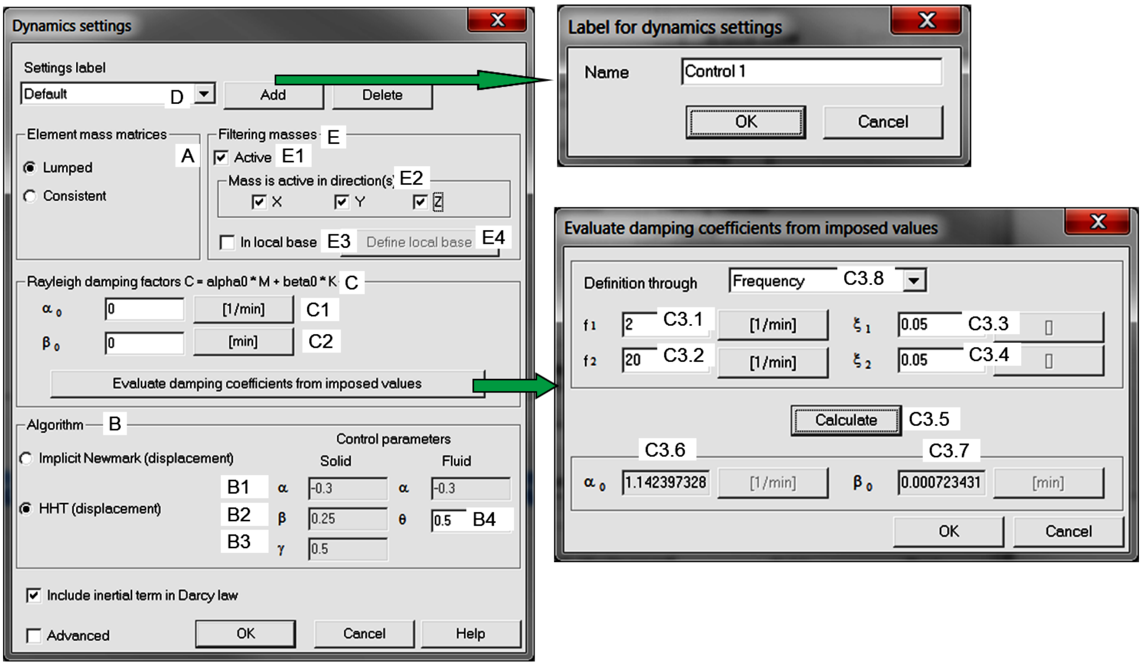
Related Topics

Theory

- ALGORITHMS

1.2.3 DYNAMICS

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



Dynamics settings

Settings label: Default

Element mass matrices: ● Lumped ○ Consistent

Filtering masses: ☑ Active

Mass is active in direction(s): ☑ X ☑ Y ☑ Z

☐ In local base

Rayleigh damping factors C = alpha0 * M + beta0 * K

α₀: 0 [1/min]

β₀: 0 [min]

Algorithm: ○ Implicit Newmark (displacement) ● HHT (displacement)

Control parameters, Solid: α -0.3, β 0.25, γ 0.5

Control parameters, Fluid: α -0.3, θ 0.5

☑ Include inertial term in Darcy law

☐ Advanced

Label for dynamics settings

Name: Control 1

Evaluate damping coefficients from imposed values

Definition through: Frequency

f₁: 2 [1/min]; ξ₁: 0.05

f₂: 20 [1/min]; ξ₂: 0.05

α₀: 1.142397328 [1/min]

β₀: 0.000723431 [min]

- **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 α parameter for HHT algorithm (default -0.3), $\alpha = 0$ corresponds to Newmark alg.
- **B2.** sets β parameter for HHT (depends on α) and Newmark (default 0.25)
- **B3.** sets γ parameter for HHT (depends on α) and Newmark (default 0.50)
- **B4.** sets θ parameter for integration scheme for pore pressure (default 0.50)
- **C.** sets Rayleigh damping factors α_o (applies to the mass) and β_o (applies to the stiffness)
- **C1.** sets directly Rayleigh damping factor α_o
- **C1.** sets directly Rayleigh damping factor β_o
- **C3.** runs Rayleigh damping factors calculator and puts results in **C1.** and **C2.** edits
- **C3.1** sets 1-st angular frequency ω_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 ω_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 ω_1, f_1 or T_1
- **C3.4** sets percentage of the critical damping for ω_2, f_2 or T_2
- **C3.5** runs α_o and β_o calculation
- **C3.6** shows calculated α_o
- **C3.7** shows calculated β_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`

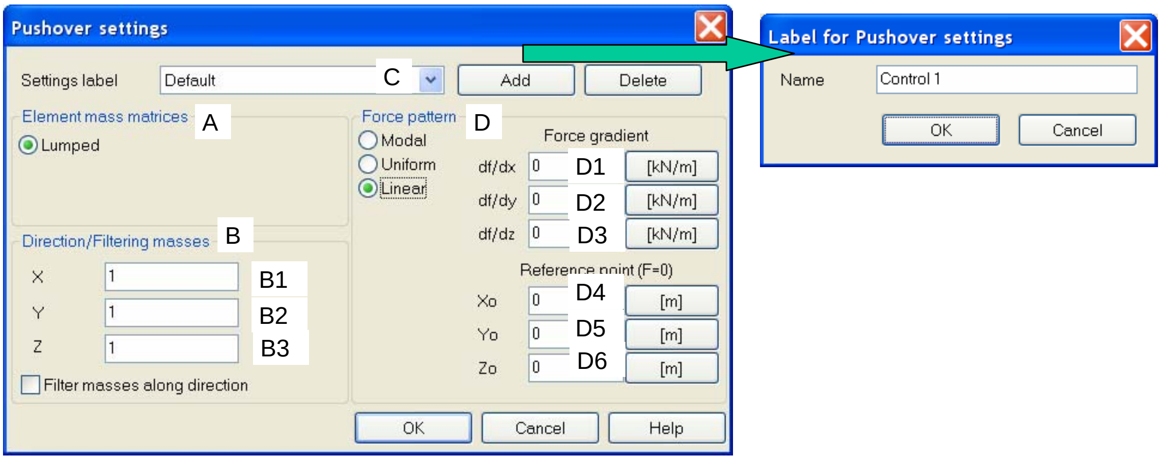
Related Topics

Theory

- ALGORITHMS

1.2.4 PUSHOVER

Sets parameters for drivers: `Pushover`



Settings label: Default

● Lumped

X: 1
Y: 1
Z: 1

☐ Filter masses along direction

○ Modal
○ Uniform
● Linear

df/dx: 0 [kN/m]
df/dy: 0 [kN/m]
df/dz: 0 [kN/m]
Xo: 0 [m]
Yo: 0 [m]
Zo: 0 [m]

Name: Control 1

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

Related Topics

Theory:

- ALGORITHMS

1.3 ASSEMBLY

Following options allow to build the computational model

Preprocessing▶	Runs unified 2D/3D graphical preprocessor to set model geometry, generate mesh, loads, initial and boundary conditions. • *INDEX* • *USER INTERFACE*
Materials▶	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). Model Data Groups ▶, Material Data Base ▶, Model Selection ▶, Material parameters ▶
Existence Function▶	**Existence Function** is a Heaviside type function ($0 \rightarrow$ non–existence, $1 \rightarrow$ existence) specified at the series of time ranges $t_1 \div t_2 \ldots t_N \div t_{N+1}$ (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.
Load Function▶	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 **Load Time Function** as a list of pairs $(t_k, f(t_k)$, assuming linear interpolation for intermediate points. Values $f(t_k)$ can be scaled by a given factor and time instances t_k may be shifted by a given time value.
Evolution function▶	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 **Evolution function** as a list of pairs $(a_k, f(a_k)$, assuming linear interpolation for intermediate points.
Gravity▶	Specify gravity multipliers in $x, y, (z)$ directions (minus sign downwards) and (optionally) set global value of the K_o^{insitu} coefficient of the lateral earth pressure (at the initial state) for all the materials related to continuum. Specifying K_o^{insitu} at the material level will overwrite the global definition.
Seismic input▶	Specify global (applied to all elements) acceleration time history, including direction vector, to be used during execution of dynamic drivers.

1.3.1 MODEL DATA GROUPS

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

`Elastic`	For most models it includes 2 parameters: • Young modulus E $\{[\text{kPa}];\ [>0];\ 100000\}$ • Poisons ratio ν $\{[-];\ [0, 0.49999];\ 0.3\}$ For some nonlinear continuum models it can be different.
`Unit Weight`	For all models for solid set: • unit weight for the solid γ $\{[\text{kN/m}^3];\ [\geq 0];\ 0\}$ For continuum solid models (dry or saturated) set: • unit weight (bouyant, saturated or dry) to be used for single phase problems • dry unit weight to be used for for two phase problems • unit weight for the fluid γ_F $\{[\text{kN/m}^3];\ [\geq 0];\ 10\}$ • initial void ratio e_0 $\{[-];\ [\geq 0];\ 0\}$ For continuum models unit weights for both single phase and two-phase problems should be specified to avoid data inconsistency when switching from **`Deformation`** to **`Deformation+Flow`** analysis. A special application is available to compute both values from specimen data. The specimen data is memorized.
`Geom`	Used by models attached to structural elements. Defines different geometric characteristics, see detailed description for each model separately
`Creep`	Set up creep data independently from other attributes and type of constitutive model attached to the continuum or shells (except **`Aging Concrete`** 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.
`Non--Linear`	Set up data to represent nonlinear mechanical behavior of the material; these parameters are distinct for each model

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 $k_{()}$ $\{[\text{m/s}];\ [>0];\ 1.0\}$ • Angle of local x' to global x coordinate system for principal permeability direction definition β • Residual saturation ratio S_r $\{[-];\ [0,1];\ 0\}$ • Parameter related to the air entry suction α $\{[\text{m}^{-1}];\ [>0];\ 2.0\}$ • Measure of the pore-size distribution n $\{[-];\ [>1];\ 2.0\}$. • Permeability function for unsaturated medium {Irmay or Mualem; Irmay} • Pore pressure weighting term $\{S\vert S_e^{1/(n\cdot m)};\ S\}$ • 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 α $\{[-];\ (e_0,1];\ 1.0\}$ • 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 Δp 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. α 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	
`Initial State`	Sets up data concerning coefficient(s) of the lateral earth pressure K_o in the *in situ* state. • for 2D analysis: ⋆ K_{ox} in x' direction ⋆ K_{oz} in z' direction ⋆ inclination angle of x' axis with respect to horizontal axis • for 3D analysis: ⋆ K_{ox} in x_1 direction ⋆ K_{oz} in x_3 direction ⋆ direction vectors for x_1 and x_3 **Remarks**: 1. This data applies exclusively to models for continuum 2. This data specified at the material level overwrites (for this material only) global setting made under `Gravity`
`Heat`	Set up: • Problem type: `Heat`(models for continuum only !) ⋆ thermal conductivity λ_T (it can be set as dependent on temperature via evolution function) ⋆ heat capacity c^* (it can be set as dependent on temperature via evolution function) ⋆ hydration heat source characteristics • Problem type `Deformation`, `Deformation+Flow`, (all `Solid Phase Models`) ⋆ thermal dilatancy coefficient α (it can be set as dependent on temperature via evolution function)
`Humidity`[2]	Set up: • Problem type `Humidity`(models for continuum only !) ⋆ moisture conductivity parameters a, D_1, w_1, χ. • Problem type `Deformation`, `Deformation+Flow`, (all `Solid Phase Models`) ⋆ hygral dilatancy coefficient β
`Stability`	Set up control data for stability driver, which for given material overwrites global settings made under `Control`→`Analysis & Drivers`.
`Damping`	Set up damping parameters α_o (applies to the mass) and β_o (applies the stiffness) which for this material will overwrite α_o and β_o globally defined in `Control`/`Dynamics` menu.

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

---

[2] 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:

Models for Continuum

`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.) ▶

Truss (anchor) model

`Uni--Axial Model` (simp.) ▶

Beams and Axisymmetric Shells

`Beam` (simp.) ▶
`Axisymmetric Shell` (simp.) ▶

Models for Shells

`Shell Section` (simp.) ▶
`Shell Component` (simp.) ▶

Models for Membrane

`Membrane--Fiber` (simp.) ▶
`Membrane--Plane Stress` (simp.) ▶
`Membrane--Fabric` (simp.) ▶

Models for Infinite m.

`Infinite Media` (simp.) ▶

Models for Interface

`Heat Convection` ▶
`Humidity Convection` [a] ▶
`Contact` (simp.) ▶
`Bond interface` (simp.) ▶
`Nodal interface` (simp.) ▶
`Seepage` (simp.) ▶

---

[a] 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 [3]

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
`Elastic`	obligatory • nP - number of maturity points for interpolation in time (default 10) • nM - number of Maxwell elements (max=6) • Young moduli E_k [$E_k > 0$ [kPa] and their distribution factors W_{ki} $(\sum_{k=1}^{nM} W_{ik} = 1)$ over nM Maxwell elements, defined at maturity points $(M_i\ i = 1..nP)$ • ν - Poisson ratio same for all Maxwell elements $\nu[[-],[<0.49999],0.2]$ • τ_k - retardation times $[h]$ for each Maxwell element $(k = 1..nM)$
`Unit Weight`	optional; (see more information)
`Flow`	optional; (see more information)
`Non--Linear`	needed for output purposes only $f_t(M_i)$ – tensile strengths [kPa] digitized at nP maturity points
`Heat`	optional; (see more information) • for `Heat` analysis use standard (see more information) • for `Deformation` and `Deformation+Flow` analysis it serves parameters needed to integrate maturity M based on nonstationary temperature field given in associated `Heat Transfer` project; if group ☒ **Heat** is set OFF then maturity is equivalent to the time of existence of a given material: ⋆ α – thermal dilatancy; α $[[1/°\text{C}], \geq 0, 10^{-5}]$. ⋆ Q/R – hydration heat source constant $Q/R = 4000[°\text{K}]$ use same value as for corresponding material in associated `Heat Transfer` project ⋆ T_F – reference temp. for hydration heat source $T_F = 20[°\text{C}]$ use same value as for corresponding material in associated `Heat Transfer` project
`Damping`	optional (see more information)

---

[3] 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 K0 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: $\nu(\bigstar)$,$\{[-],[0<\nu<0.49999],0.3\}$
Unit Weight	optional; (see more information)
Flow	optional; (see more information)
Non--Linear	obligatory; • Slope of critical state line in p-q plane M_{c}, $\{[-],[>0],[<3],1.0\}$ • Slope of primary consolidation in e-ln(p) λ, $\{[-],[>0],0.1\}$ • Slope of secondary consolidation in e-ln(p) κ, $\{[-],[>0],\kappa<\lambda,0.01\}$ • Minimum preconsolidation pressure $\underline{p}_{\text{co}}$ $(\bigstar)$, $\{[\text{kPa}],[>0],10\}$ • Overconsolidation ratio OCR, $\{[-],[\geq 1],1\}$ • Strength anisotropy factor (ratio between tension and compresion meridian slopes $k = M_{\text{e}}/M_{\text{c}}$) can be: ⋆ **disabled** $(k=1)$ ⋆ **default** $(k=\dfrac{3}{3+M_c})$ ⋆ **prescribed** by the user k, $\{[-],0.5<k<1.0,0.75\}$
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 StateK_0	optional for **Deformation**, **Deformation**+**Flow**; (see more information)
Non--Linear	obligatory; • Drucker–Prager data • Cap parameters: ⋆ λ - slope of primary consolidation line λ $\{[-],[>0],0.1\}$ in $(e-\ln\ p)$ ⋆ OCR - overconsolidation ratio; OCR $\{[-],[\geq 1.0],1.0\}$ ⋆ $\underline{p}_{\text{co}}$ and R - initial preconsolidation pressure and initial shape cap surface parameters 1. for **Direct Definition** enter: ♦ $\underline{p}_{\text{co}}$ (★) $\{\text{kPa},[>0],10^{38}\}$ ♦ R $\{[-],[1.1<R<2.7],1.5\}$ 1. for indirect definition through an **Oedometric Test** enter: ♦ vertical preconsolidation stress $\underline{\sigma}_{\text{VM}}$, $\{\text{ kPa},[>0],0\}$ ♦ coefficient of lateral pressure at normal consolidation state K_0^{NC}, $\{[-],[>0],0.5\}$(Jaky's formula $K_0^{\text{NC}}=1-\sin(\phi)$ can be applied)
Humidity	optional for **Humidity**, **Deformation**, **Deformation**+**Flow**; (see 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 K0 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 ; • Cohesion C (★), {kPa, $[>0]$, 20} • Friction angle φ (★), $\{[\mathrm{deg}],\ [0,89], 30\}$ • Dilatancy angle ψ (★), $\{[\mathrm{deg}],\ [0,89],\ 0\}$ Set $\psi=0$ for incompressible behavior • Tensile cut–off(★); optional, ignore if not konwlegable of effects • Size adjustment with respect to the Mohr-Coulomb criterion: : **External Edges**, **Internal Edges**, **Elastic**, **Plane Strain** (default for plane strain), **Intermediate** (default for axisymmetry)
`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 $K0$ 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 K0 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; see more information

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 StateK_0	optional for **Deformation**, **Deformation**+**Flow**; (see more information)
Non--Linear	obligatory; • Compressive uni-axial strength f_{c}, $\{[\mathrm{kPa}], [> 0, > f_{\mathrm{t}}], 15000\}$ • Tensile uni-axial strength f_{t}, $\{[\mathrm{kPa}], [> 0, < f_t], 1000\}$ • Biaxial compressive/uniaxial compressive strength ratio f_b/f_c, $\{[-], [> 1], 1.16\}$ • **Tensile Cut--Off** flag ON/OFF first stress invariant $I_{1\mathrm{t}}$ (if **Cut--Off** is ON), $\{[\mathrm{kPa}], [> 0], 0\}$ • Plastic flow type **Deviatoric**, **Drucker--Prager**, **Tensile Meridian**, **Hoek--Brown (M--W)**, • Dilatancy angle at uniaxial compressive strength ψ_{c} (if **Drucker--Prager** or **Hoek--Brown (M--W)** flow is ON), $\{[\mathrm{deg}], [\arctan\left(\frac{f_{\mathrm{t}}}{\sqrt{2}f_{\mathrm{c}}}\right) \leq 35.3°], 0\}$ • Softening flag ON/OFF[4]. This option must be used with caution. If activated, introduce: ⋆ Crack opening at failure $w_r, \{[m], [> 0], 0.0001]\}$ ⋆ Steepness parameter $a, \{[-], [> 0], 5.0]\}$ ⋆ Fictitious number of cracks for compression $b, \{[-], [> 1], 10.0]\}$
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 K0 state*
- *data preparation: Stability*
- *data preparation: Heat*
- *data preparation: Humid*

1.3.3.8 MOHR–COULOMB (M-W)(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; • Cohesion c, $\{[\text{kPa}], [>0], 20\}$ • Friction angle ϕ, $\{[\text{deg}], [0, 89], 30\}$ • `Cut--Off` flag ON/OffFlg • Tensile strength f_t $\{\{[\text{kPa}], [0, c/\tan(\phi)], 0\}$ (if `Cut--Off` flag is ON), • Plastic flow type `Deviatoric`, `Drucker--Prager` • Dilatancy angle ψ, $\{[\text{deg}], [0, 89], 0\}$ (if `Drucker--Prager` flow is ON, set $\psi = 0$ for incompressible behavior • Size adjustment flag ON/OFF
`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*
- *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`	obligatory; set • E_{ur}^{ref} is the unloading/reloading Young modulus given at the reference stress σ^{ref} $\{[\text{kPa}], [>0],\ 80000\})$ • $E_{ur,v}^{ref}$ is the vertical unloading/reloading Young modulus given at the reference stress σ^{ref} $\{[\text{kPa}], [>0],\ 80000\})$, (if ☒ **Anisotropy** is set ON) • Reference stress σ^{ref}, $\{[\text{kPa}], [>0],\ 100\}$ is the stress value at which the reference stiffness moduli E_{ur}^{ref} is defined • ν_{ur} is the unloading/reloading Poisson coefficient; it varies from 0.15 to 0.3, hence for sands it is recommended to assume $\nu_{ur} = 0.2..0.25$ and for clays $\nu_{ur} = 0.25..0.3$, $\{[-], [0, 0.49999],\ 0.2\}$ • $\nu_{ur,hh}$ is the unloading/reloading Poisson's ratio for horizontal strain due to complementary horizontal strain. $\{[-], [-1.0 < \nu_{ur,hh} < 0.5], 0.2\}$, (if ☒ **Anisotropy** is set ON) • m is the exponent in stress dependency power law; it varies from $m = 0.4$ to $m = 0..9$; it is smaller for dense sands and larger for clays • σ_L is the minimum allowed stress (σ_3 or p) value used for evaluation of stiffness moduli $\{[\text{kPa}], [>0],\ 10\}$ • stiffness stress dependency type can be one of three following functions 1. $\dfrac{\sigma_3 + c\ \cot\phi}{\sigma^{ref} + c\ \cot\phi}$ 2. $\dfrac{\sigma_3}{\sigma^{ref}}$ 3. $\dfrac{p}{\sigma^{ref}}$ the last two functions can be used in simplified undrained analyses when $\phi = 0$ is assumed; the third function is recommended in general but then reference stiffness moduli and power exponent m must be recalculated once they were calibrated assuming σ_3 type stress dependencies • small strain formulation can be selected from the set of the three possible options 1. **`Disabled`** 2. **`HSs formulation`** - classical 3. **`Brick formulation`** The **`Brick formulation`** is superior with respect to the classical HSs.
	continued on next page

continued from previous page

`Elastic`	obligatory; set • E_o^{ref} is the Young modulus at very small strains at the reference stress σ^{ref}; In case of lack of information on E_o^{ref} one may try to estimate E_o^{ref} based on Alpan's diagram (see external report on Hardening Soil model) assuming $E_s = E_{ur}$ • $E_{o,v}^{ref}$ is the vertical Young modulus at very small strains at the reference stress σ^{ref}, (if ☒ **Anisotropy** is set ON) • $\gamma_{0.7}$ is threshold shear strain at which secant shear moduls G reaches 70 % of its initial value G_o (note that $G_o = \dfrac{E_o}{2(1+\nu_{ur})}$); in the current implementation $\gamma_{0.7}$ is assumed to be constant; in case of lack of information on $\gamma_{0.7}$ one may use the diagram by Vucetic and Dobry for cohesive soils and diagram by Wichtmann and Triantafyllidis for cohesionless ones included in the report on Hardening Soil model; $\gamma_{0.7} = 0.0001..0.0002$, $\{[-], [0.0001, 0.0005], 0.0002\}$ • α_G $\{[-], [0.7 \leq \alpha_G \leq 3.0], 2.0\}$, (if ☒ **Anisotropy** is set ON) • β $\{[-], [1.0 \leq \beta \leq 2.0], 1.6\}$, (if ☒ **Anisotropy** is set ON) • Tilt angle θ $\{[\text{deg}], [-90 \leq \theta \leq 90], 0\}$, (if ☒ **Anisotropy** is set ON) • Azimuthal angle φ $\{[\text{deg}], [0 \leq \varphi \leq 180], 90\}$, (if ☒ **Anisotropy** is set ON)
`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`	obligatory; • E_{50}^{ref} is the secant Young modulus at 50 % of failure deviatoric stress q_f derived from the $q - \varepsilon_1$ curve in drained triaxial test (see report on Hardening Soil model for more details) • ϕ is the friction angle $\{[\text{deg}], [> 0],\ 30\}$ • ψ is the dilatancy angle $\{[\text{deg}], [> 0],\ 0\}$ • c' is the effective cohesion $\{[\text{kPa}], [\geq 0],\ 20\}$ • R_f is the failure ratio $\{[-], [0.8 < R_f < 1],\ 0.9\}$ • f_t is the tensile strength $\{[\text{kPa}], [\geq 0],\ 0\}$ • e_{max} is the maximum allowed void ratio; if current void ratio exceeds the e_{max} dilatancy angle is switched to $\psi = 0$, $\{[-], [> 0],\ 1\}$ • D is the Rowe's dilatancy multiplier in the contractant domain, for standard HS model $\{[-], [0 \leq D \leq 1],\ 0.0\}$ while for HS-small $\{[-], [0 \leq D \leq 1],\ 0.25\}$ • Cap surface parameter M and hardening parameter H are derived by using a simple calculator which simulates an oedometric test; for given tangent oedometric modulus E_{oed} at a given reference vertical stress σ_{oed}^{ref} and for assumed K_o^{NC} parameter (here Jaky's formula can be used) values of H and M are evaluated (press button **Evaluate M,H**); one may assume $E_{oed} = E_{50}^{ref}\left(\dfrac{\sigma_{oed}^{ref} + c\ \cot\phi}{\sigma^{ref} + c\ \cot\phi}\right)^m$ as a default • Setting the initial state variables γ_o^{PS} and p_{co} can be carried out by means of assumed OCR or preoverburden pressure q^{POP}; pairs K_o^{SR} and OCR ($OCR \geq 1.0$) or K_o^{SR} and q^{POP} are needed to setup the initial position of the cap surface and the initial value of the hardening parameter γ^{PS} (please refer to the report on Hardening Soil model for the detailed explanation) • p_{co}^{min} is the minimum allowed value for the initial preconsolidation stress $\{[kPa], [> 0],\ 10\}$
`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`	obligatory; set • E_{ur}^{ref} is the unloading/reloading Young modulus given at the reference stress σ^{ref} $\{[\text{kPa}], [>0],\ 80000\}$); note that the elastic stiffness moduli depend on p rather on σ_3 value in this model • Reference stress σ^{ref}, $\{[\text{kPa}], [>0],\ 100\}$ is the stress value at which the reference stiffness moduli E_{ur}^{ref} is defined • ν_{ur} is the unloading/reloading Poisson coefficient; it varies from 0.15 to 0.3, hence for sands it is recommended to assume $\nu_{ur} = 0.2..0.25$ and for clays $\nu_{ur} = 0.25..0.3$, $\{[-], [0, 0.49999],\ 0.2\}$ • m is the exponent in stress dependency power law; for loose sands $m \approx 0.5$ • σ_L is the minimum allowed reference stress value used for evaluation of stiffness moduli $\{[\text{kPa}], [>0],\ 10\}$ • stiffness stress dependency type can be one of three following functions 1. $\frac{\sigma_3 + c\ \cot\phi}{\sigma^{ref} + c\ \cot\phi}$ 2. $\frac{\sigma_3}{\sigma^{ref}}$ 3. $\frac{p}{\sigma^{ref}}$ the last two functions can be used in simplified undrained analyses when $\phi = 0$ is assumed; the third function is recommended in general but then reference stiffness moduli and power exponent m must be recalculated once they were calibrated assuming σ_3 type stress dependencies • small strain formulation can be selected from the set of the three possible options 1. **`Disabled`** 2. **`HSs formulation`** - classical 3. **`Brick formulation`** • E_o^{ref} is the Young modulus at very small strains at the reference stress σ^{ref}; note that the current very small strain stiffness modulus E_o depends on p rather than σ_3 (as in the HS model)

continued on next page

continued from previous page	
`Elastic`	obligatory; set • $\gamma_{0.7}$ is the threshold shear strain at which secant shear modulus G reaches 70 % of its initial value G_o (note that $G_o = \dfrac{E_o}{2(1+\nu_{ur})}$); in the current implementation $\gamma_{0.7}$ is assumed to be constant; in case of lack of information on $\gamma_{0.7}$ one may use the diagram by Wichtmann and Triantafyllidis included in the report on Hardening Soil model; $\gamma_{0.7} = 0.0001..0.0002$, $\{[-], [0.0001, 0.0005], 0.0002\}$
`Unit Weight`	optional; (see 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`	obligatory; • ϕ is the friction angle $\{[\deg], [> 0],\ 30\}$ • c' is the effective cohesion $\{[\text{kPa}], [\geq 0],\ 1\}$ • f_t is the tensile strength $\{[\text{kPa}], [\geq 0],\ 0\}$ • To activate internal shear plastic mechanism the flag must be set ON • R_f is the failure ratio $\{[-], [0.8 < R_f < 1],\ 0.9\}$ • E_{50}^{ref} is the secant Young modulus at 50 % of failure deviatoric stress q_f derived from the $q - \varepsilon_1$ curve in drained triaxial test (same as in the HS model so please refer to the report on Hardening Soil model for more details) • To activate densification mechanism user must switch ON the flag ☒ **Densification mechanism** and then choose one of the two densification laws, the one proposed by Zienkiewicz or by Sawicki • A, B, γ - parameters for densification law by Zienkiewicz • C_1, C_2 - parameters for densification law by Sawicki (typical values for C_1 are of order $0.005 \div 0.009$ and for C_2 of order 2*10^5) (please refer to the report on Dynamics for more information) • Strain mapping for the accumulated volumetric strain can be selected as *Isotropic* (here the increment of the accumulated volumetric strain is projected equally on all normal directions) or *Extended* (here the increment of the accumulated volumetric strain is projected on those normal directions that are not restrained by the boundary conditions); using the extended mapping is recommended; in the shear layer problem the *Extended* mapping will put the full accumulated volumetric strain only on the vertical direction • Setting the initial state variable γ_o^{PS} can be carried out by means of assumed OCR or preoverburden pressure q^{POP}; pairs K_o^{SR} and OCR ($OCR \geq 1.0$) or K_o^{SR} and q^{POP} are needed to setup the initial value of the hardening parameter γ^{PS} (please refer to the report on Hardening Soil model for the detailed explanation as the same procedure is used in this model)
`Humidity`	optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see 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; (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;for each active weakness plane set: • for 2D models: ⋆ Inclination angle α measured in anti-clockwise direction, $\{[\text{deg}], [-90 \leq \alpha \leq 90], 0\}$ • for 3D models: ⋆ Inclination angles α, β $\{[\text{deg}], [-360 \leq \alpha/\beta \leq 360], 0\}$, (if ☒ **Advanced mode** is set ON) ⋆ Vector $\mathbf{n}$ normal to the weakness plane (if ☒ **Advanced mode** is set OFF) • Cohesion c, $\{[\text{kPa}], [> 0], 0\}$ • Friction angle ϕ, $\{[\text{deg}], [0, 89], 0\}$ • Dilatancy angle ψ, $\{[\text{deg}], [0, 89], 0\}$; set $\psi = 0$ for incompressible behavior. • `Cut--Off` flag ON/OFF • Tensile strength f_t, $\{[\text{kPa}], [0, c/\tan(\phi)], 0\}$ (if `Cut--Off` flag is ON) • Matrix material model (elastic if ☒ **Advanced** is OFF) or `Mohr-Coulomb (M-W)`/ `Rankine (M-W)`/`Hoek-Brown`
`Humidity`	optional for `Humidity`, `Deformation`, `Deformation+Flow`; (see 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: 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; (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; set: • Tensile strength (uni-axial) f_t, $\{[\text{kPa}], [> 0], 0\}$ • Softening flag; if ☒ **Softening** is set ON define ⋆ Crack opening at failure $w_r, \{[m], [> 0], 0.0001]\}$ ⋆ Steepness parameter $a, \{[-], [> 0], 5.0]\}$ ⋆ Fictitious number of cracks for compression $b, \{[-], [> 1], 10.0]\}$
`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: 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 StateK_0	optional for **Deformation**, **Deformation**+**Flow**; (see more information)
Non--Linear	obligatory; • Cohesion C ($\bigstar$), $\{[\text{kPa}],\ [>0],\ 0\}$ • Friction angle φ ($\bigstar$), $\{[\deg],\ [0,89],\ 0\}$ • Dilatancy angle ψ ($\bigstar$), $\{[\deg],\ [0,89],\ 0\}$ Set $\psi = 0$ for incompressible behavior • Tensile cut–off(Rankine); (optional) f_t ($\bigstar$), $\{[\text{kPa}],[\geq 0],0\}$ • Dilatancy cut-off; (optional) e_{MAX} , $\{[-],[e_{\max} > e_o]\}$
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 $K0$ 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 StateK_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) • $\underline{\tilde{\sigma}}_c/f_c$ - stress level for damage calibration in compression (on post-peak branch)(≤ 1.0) • $\tilde{D}_c$ - damage parameter in uniaxial compression at the assumed reference stress level (≥ 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 (≥ 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 ($0.1 \div 0.3$) • Dilatancy type - $\mathrm{Constant/Variable}$ • α_p - Dilatancy parameter value for $\mathrm{Constant}$ option or dilatancy parameter in uniaxial compression when option $\mathrm{Variable}$ is used ($0.1 \div 0.5$) • α_{po} - dilatancy parameter in uniaxial tension option $\mathrm{Variable}$ is used ($0.1 \div 0.4$) • $\underline{\sigma}_{c,dil}/\underline{f}_c$ - activation of dilatancy in compression when $\mathrm{Variable}$ option is used; $\underline{\sigma}_{c,dil}/\underline{f}_c \geq \underline{f}_{co}/\underline{f}_c$; ($0.4 \div 1.0$) • α_d - smoothing factor for plastic potential; value 1.0 is recommended ($0.2 \div 2.0$)

continued on next page

continued from previous page	
`Nonlinear`	obligatory; • Char. length (RC) flag - enforced characteristic length flag for reinforced concrete (ON/OFF) • l_c^{RC} - characteristic length to be used for reinforced concrete when Char. length (RC) flag is ON (> 0) • Secant stiffness flag - use secant elastic stiffness ($\mathbf{D}^e(1-D)$) when ON or consistent tangent when OFF (secant stiffness is recommended for cases with poor convergence)
`Humidity`	optional for **`Humidity`**, **`Deformation`**, **`Deformation+Flow`**; (see more information)
`Heat`	optional for **`Heat`**, **`Deformation`**, **`Deformation+Flow`**; (see more information) parameters λ, c^* and α 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
Elastic	obligatory; set: Young's elastic modulus, E $\{[\text{kPa}]; [> 0]; 100000\}$,
Unit Weight	optional; set: volumic weight of the material γ, $\{[\text{kN/m}^3]; [> 0]; 0\}$.
Non--Linear	optional; set: uniaxial tensile strength $f_t\{[\text{kPa}], > 0, 1\}$ uniaxial compressive strength $f_c\{[\text{kPa}], > 0, 1\}$
Geometry	obligatory; set: cross–section area A, $\{[\text{m}^2]; [> 0]; 1\}$ distance (2D Plane Strain only) a, $\{[\text{m}]; [> 0]; 1\}$
Heat	optional; set: α, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient. In case of active **Data Group Heat** and active associated **Heat Transfer** project (if $\alpha > 0$), thermal strains will be evaluated.
Damping	optional; set Rayleigh damping factors α_o (applies to the mass) and β_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
Elastic	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 $G=\dfrac{E}{2(1+\nu)}$
Cross section	obligatory; cross–section characteristics, setting dependent on analysis type: • $a\{[\text{m}];[>0];1\}$ – the distance between members with given section, for **Plane Strain** • $A\ \{[\text{m}^2];[>0];1\}$ – cross–section area, for all **Analysis Type** • $I_x\{[\text{m}^4];[>0];1\}$ – 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 **Analysis Type** • $A_y=\kappa_y A\{[\text{m}^2];[>0];1\}$ – shear section area in Y_L direction (κ_y is a shear correction factor), for all **Analysis Type** • $A_z=\kappa_z A\{[\text{m}^2];[>0];1\}$ – shear section area in Z_L direction (κ_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	optional If **Automatic evaluation of shear modulus** is set ✓ (default), Kirchoff modulus G will be evaluated automatically basing on E, ν of `Core` material. Otherwise set: • Young's elastic modulus $E\{[\text{kPa}];[>0];100000\}$, • Poisons ratio $\nu\{[-];[0,0.49999];0.3\}$. Both will be used only to create shear Kirchhoff modulus $G = \dfrac{E}{2(1+\nu)}$, independently of `Core` material data. In any case, elasticity modulus E_X in axial direction has to be entered via **Cross section→ Core material → Edit material**
Cross section	obligatory Selection between section `Types` has to be done. Use alternatively: • `Profiles` - to set a typical manufactured (steel) section choosen form available data base. • `User` - to enter geometrical data of predefined types. In both cases cross section area will automatically be divided into layers with uni-axial elastic/elasto-plastic or user defined model marked as `Core` at each. In addition extra layers (possibly prestressed) can be added to the core material by activating ☒ **Additional layers** option. The latter option can be used to model reinforced concrete, although setting beam reinforcement can be made much easier during preprocessing (under `Beam` use option **Reinforcement sets**). Reinforcement sets defined at the preprocessing stage and extra layers defined at the material level will be added. To edit data for core material use ☒ **Edit material** and for extra layers use ☒ **Edit layers**. Moreover, cross sectional characteristics, related to shear or torsion, will be automatically evaluated: • $I_x\{[\text{m}^4];[>0];1\}$ – torsional momentum of inertia, for **3D** only • $A_y = \kappa_y A\{[\text{m}^2];[>0];1\}$ – shear section area in Y_L direction (κ_y is a shear correction factor), for all **Analysis Type** • $A_z = \kappa_z A\{[\text{m}^2];[>0];1\}$ – shear section area in Z_L direction (κ_z is a shear correction factor) for **3D** For **Plane Strain** set: • $a\{[\text{m}];[>0];1\}$ – the interval between beams with given section (leave $a = 1$ in case of a continues lining).

continued on next page

continued from previous page

Heat	optional; set: • α, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - 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
`Unit Weight`	optional; • γ, $\{[\mathrm{kNm}^{-3}];[>0];0\}$ - unit weight of the shell (averaged over the whole section); • Density $\rho=\dfrac{\gamma}{g}$ is automatically computed from γ (and vice versa) • Multiplier for γ $\{[-];[\geq 0];1.0\}$ • Multiplier for ρ $\{[-];[\geq 0];1.0\}$ Setting γ 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 ρ multiplier to 0.0 will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if **`Dynamics`** or **`Pushover`** is set ON in the project preselection).
`Heat`	optional α, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient. In case of active **`Data Group Heat`** and active associated **`Heat Transfer`** project (if $\alpha>0$) thermal strains will be computed.
`Stability`	optional Handles material strength parameters reduction, see: *LOCAL STABILITY* for the details.
`Damping`	Rayleigh damping factors α_o (applies to the mass) $\{[s^{-1}];[\geq 0];0.0\}$ and β_o (applies to the stiffness) $\{[s];[\geq 0];0.0\}$; 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
`Elastic`	obligatory • Young's elastic modulus, E $\{[\text{kPa}]; [> 0]; 100000\}$, • Poisson ratio $\nu\{[-]; [0, 0.49999]; 0.3\}$ E and ν constant for whole section, used for all stiffness components
`Cross section`	obligatory • shell thickness $h\{[m]; [> 0]; 1\}$ • shear correction factor $\kappa\{[-]; [> 0]; 0.83333\}$ The shear stiffness of the section will be then evaluated as: $GA_y = \kappa\frac{Eh}{2(1+\nu)}$ 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
`Elastic`	optional If `Automatic evaluation of shear modulus` is set ✓ (default), Kirchoff modulus G will automatically be evaluated basing on E, ν of `Core` material. Otherwise set: • Young's elastic modulus $E\{[\text{kPa}]; [> 0]; 100000\}$, • Poisons ratio $\nu\{[-]; [0, 0.49999]; 0.3\}$. Both will be used only to compute shear Kirchhoff modulus $G = \dfrac{E}{2(1+\nu)}$ independently of `Core` material data. **Remark**: In any case, elastic constants for each component separately, are set in the material data under group ⊠ **Cross section** by activating ⊠ **Layered cross section** option. *(to be continued on the next page)*

Data Group	Use & Set up
`Cross section`	obligatory 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: • Set shell thickness $h\{[m];[>0];1\}$ • The section will automatically be split into core material layers with bi-axial elastic or elasto-plastic model • To enter material model data for core use **Edit material** button. In the grid type dialog box edit elastic and plastic properties of the shell core material. • To activate reinforcing radial/circumferential fibers set ON the option ☒ **Additional layers**; to edit properties of extra layers enter to the dialog box by pressing the button **Edit layers**; in the dialog define additional layers by setting ⋆ `Orientation` of the reinforcement in given layer switching between `Radial` and `Circumferential` ⋆ `Type` of the reinforcement area definition by selecting `Density` (standard setup in the practice where reinforcement is set with the unit $[m^2/m]$ to preserve certain percentage) (valid for both radial and circumferential direction) or `Total` (kept mostly for back compatibility with previous versions) (here unit is m^2)(valid only for the radial direction) ⋆ `Y-Position` of the given layer refered to the top, bottom or the middle of the cross section ⋆ `Y-distance` that is an offset between selected core fibers defined in column `Y-Position` and current reinforcement layer ⋆ `Area` of the reinforcement defined with unit that may depend on the reinforcement `Type`

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

`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
`Elastic`	obligatory • Young's elastic modulus, E $\{[\text{kPa}];[>0];100000\}$, • Poisson ratio $\nu\{[-];[0,0.49999];0.3\}$

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

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
`Elastic`	obligatory; • Young's elastic modulus $E\{[\text{kPa}];[>0];100000\}$ • Poisons ratio $\nu\{[-];[0,0.49999];0.3\}$,
`Creep`	optional (same as for continuum) ;
`Heat`	optional; α, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared thermal strains will be computed (if $\beta > 0$).
`Humidity`	optional; β, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient; In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared hygral strains will be computed (if $\beta > 0$).
`Non--Linear`	• `Elastic`, `Mohr--Coulomb (M--W)`, `Hoek--Brown (M--W)`, `Aging Concrete` shells, `Plastic damage (concrete)` shells – obligatory set data as for continuum models • `Huber--Mises` shell – obligatory; set: ⋆ uniaxial tensile strength f_t, $\{[\text{kPa}];[>0];1\}$
`Stability`	optional for `Deformation`, `Deformation+Flow`;

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

Data groups	Parameters
Elastic	obligatory; Young's elastic modulus, E $\{[\text{kPa}]; [> 0]; 100000\}$ 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)
Creep	not supported
Heat	optional; α, $\{[C^{-1}]; [> 0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group ☒ **Heat** is active and preprocessed **Heat Transfer** project is declared (if $\alpha > 0$) thermal strains will be computed; these parameter can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see dedicated report on reinforced concrete structures subject to elevated temperatures)
Humidity	optional; β, $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient; In case when group ☒ **Humidity** is active and preprocessed **Humidity Transfer** project is declared (if $\beta > 0$) hygral strains will be computed.
Non--Linear	optional; • Uniaxial tensile strength f_t, $\{[\text{kPa}]; [f_t > 0]; 1\}$ • Uniaxial compressive strength f_c, $\{[\text{kPa}]; [f_c \geq 0]; 1\}$ these parameters can be modified by the predefined set of thermal evolution functions to carry out fire analyses (see dedicated report on reinforced concrete structures subject to elevated temperatures)
Geometry	obligatory; • Set X, Y, Z being the coordinates of an arbitrary direction vector in 3D space indicating an orientation of fibers in 3D space;
Stability	optional for **Deformation**, **Deformation+Flow**

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
Elastic	obligatory; Young's elastic modulus, E $\{[\text{kPa}]; [>0]; 100000\}$
Unit Weight	optional; • γ, $\{[\text{kNm}^{-2}]; [>0]; 0\}$ - unit weight of fiber material • Density $\rho = \frac{\gamma}{g}$ is automatically computed from γ (and vice versa) • Multiplier for γ $\{[-]; [\geq 0]; 1.0\}$ • Multiplier for ρ $\{[-]; [\geq 0]; 1.0\}$ Setting γ 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 ρ multiplier to 0.0 will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if **Dynamics** or **Pushover** is set ON in the project preselection).
Creep	not supported
Heat	optional; α, $\{[C^{-1}]; [>0]; 1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group ☒ **Heat** is active and preprocessed **Heat Transfer** project is declared (if $\alpha > 0$) thermal strains will be computed.
Humidity	optional; β, $\{[-]; [\geq 0]; 0.01\}$ - hygral dilatancy coefficient; In case when group ☒ **Humidity** is active and preprocessed **Humidity Transfer** project is declared (if $\beta > 0$) hygral strains will be computed.
Non--Linear	optional • Uniaxial tensile strength f_t, $\{[\text{kPa}]; [f_t > 0]; 1\}$ • Uniaxial compressive strength f_c, $\{[\text{kPa}]; [f_c \geq 0]; 1\}$
Geometry	obligatory • For **Plane Strain**: ⋆ Area per unit length A, $\{[\text{m}^2/\text{m=m}], [>0], [1]\}$ • For **Axisymmetry**: ⋆ Direction: one of: ◎ **Circumferential** or ◎ **Longitudinal** ⋆ Area per unit length A, $\{[\text{m}^2/\text{m=m}], [>0], [1]\}$ • For 3D: ⋆ Set X, Y, Z being the coordinates of an arbitrary direction vector in 3D space indicating an orientation of fibers in 3D space; ⋆ Area per unit length A, $\{[\text{m}^2/\text{m=m}], [>0], [1]\}$
Damping	optional (same definition as in the global *Control/Dynamics dialog)*

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
`Elastic`	obligatory (same as for continuum);
`Unit Weight`	optional; • γ, $\{[\text{kNm}^{-3}];[>0];0\}$ - unit weight • Density $\rho = \frac{\gamma}{g}$ is automatically computed from γ (and vice versa) • Multiplier for γ $\{[-];[\geq 0];1.0\}$ • Multiplier for ρ $\{[-];[\geq 0];1.0\}$ Setting γ 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 ρ multiplier to 0.0 will cancel contribution of all elements with that material to the mass matrix; both multipliers are active if `Dynamics` or `Pushover` is set ON in the project preselection).
`Geometry`	obligatory; Thickness t, $\{[\text{m}];[>0];0.1\}$
`Creep`	not supported
`Heat`	optional; α, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group ☒ **Heat** is active and preprocessed `Heat Transfer` project is declared thermal strains will be computed (if $\beta > 0$).
`Humidity`	optional; β, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient; In case when group ☒ **Humidity** is active and preprocessed `Humidity Transfer` project is declared hygral strains will be computed (if $\beta > 0$).

`Non--Linear`	set:
	• For `Huber--Mises` and `Rankine (M--W)` (obligatory):
	  ⋆ Uniaxial tensile strength f_t, $\{[\text{kPa}];[\geq 0];1\}$
	• For `Hoek--Brown (M--W)` (obligatory):
	  ⋆ Uniaxial tensile strength f_t, $\{[\text{kPa}];[\geq 0];1\}$
	  ⋆ Uniaxial compressive strength f_c, $\{[\text{kPa}];[f_c > f_t \geq 0];1\}$

`Damping`	optional (same definition as in the global *`Control/Dynamics` dialog)*

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; α, $\{[C^{-1}];[>0];1.0e^{-5}\}$ - thermal dilatancy coefficient In case when group ☒ **Heat** is active and preprocessed **`Heat Transfer`** project is declared thermal strains will be computed (if $\beta > 0$).
`Humidity`	optional; β, $\{[-];[\geq 0];0.01\}$ - hygral dilatancy coefficient; In case when group ☒ **Humidity** is active and preprocessed **`Humidity Transfer`** project is declared hygral strains will be computed (if $\beta > 0$).
`Damping`	optional (same definition as in the global *`Control/Dynamics` dialog*)

`Isotropic` membrane: specific data.

Data	Use & Set up
`Elastic`	obligatory; • Elastic stiffness K (equivalent to $E\times$thickness) $\{[\text{kN/m}];[>0];100000\}$, • Poisson ratio ν, $\{[-];[0,0.49999];0.3\}$
`Non--Linear`	optional If not activated elastic membrane with unlimited strength will be considerd • Uniaxial tensile strength f_t, $\{[\text{kN/m}];[>0];1\}$ • Uniaxial compressive strength f_c, $\{[\text{kN/m}];[>0];1\}$

`Anisotropic` membrane: specific data.

Data	Use & Set up
`Elastic`	obligatory; • Elastic stiffness in anisotropy direction K_{XX}, K_{YY}, K_{XY} $\{[\text{kN/m}]; [> 0]; 100000\}$,
`Geometry`	obligatory X, Y, Z the coordinates of an arbitrary direction vector in 3D space. It determines anisotropy axis X
`Non--Linear`	optional If not activated elastic membrane with unlimited strength will be considered • Uniaxial tensile strength in anisotropy direction X,Y: f_tX, f_tY, $\{[\text{kN/m}]; [\geq 0]; 1\}$ • Uniaxial compressive strength in anisotropy direction X,Y: f_cX, f_cY, $\{[\text{kN/m}]; [\geq 0]; 1\}$

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 • Heat convection coefficient h_T $\{[\text{kN}/(\text{m}\cdot{}^\circ\text{K}\cdot\text{s})], >0, 10\}$ (see the report on reinforced concrete structures subject to elevated temperatures to set up proper value of this parameter in case of fire analyses)

Related Topics

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

1.3.3.24 HUMIDITY CONVECTION

The model is applicable exclusively to **`Humidity Problem Type`** [5].

It is used to describe convective properties of the boundary and surrounding media (with ambient relative humidity), in order to control humidity exchange through the boundary. It may be attached to **`Humidity Convection`** elements exclusively.

Data group	Parameters
`Humidity`	obligatory; • Humidity convection coefficient h_W $\{[1/(\mathrm{m}^2 \cdot \mathrm{d})], > 0, 10\}$

Related Topics

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

---

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

1.3.3.25 CONTACT (simple)

Data	Use & Set up
`Elastic`	optional • `Direct values of interface stiffness` is set OFF(default). Elastic stiffness of the contact surfaces is set automatically basing on properties of surrounding media. The user may, if necessary, change the default using: ⋆ normal stiffness K_n multiplier $\{[-],[>0],[0.01]\}$ i.e. $K_n = \texttt{multiplier}* K_n^{def}$ ⋆ tangential/normal stiffness ratio K_t/K_n $\{[-],[>0],[0.01]\}$ • `Direct values of interface stiffness` is set ON. User must set elastic properties of the interface explicitly specifying: ⋆ normal stiffness K_n, $\{[kN/m^3],[>0],[10000]\}$ K_n is equivalent to $K_n = E/h$ where h is the depth of the very thin weak (interface) layer and E is its elasticity modulus. ⋆ tangential stiffness K_t, $\{[kN/m^3],[>0],[10000]\}$ K_t is equivalent to $K_t = G/h$ where h is the depth of the weak layer and G is its shear modulus.
`Flow`	optional User may choose between `Impermeable Surface` of the interface ($k = 0$) and `Fully Permeable Surface` with permeability k estimated by the program as large enough to assure pressure compatibility, or as `Permeable Surface` with permeability given by the user . Following data have to be entered only if `Fully Permeable Surface` is set OFF. • permeability in the normal direction -k'_z $\{[1/\text{s}],[>0],[0.01]\}$ • in-plane permeability -k'_x, k'_y.$\{\text{m}^2/\text{s}, >0, 1\}$ If `Anisotropic Flow` in interface is set ON, then in–plane permeabilities k'_x, k'_y have to be set in 2 principal anisotropy directions. The anisotropy direction of $x\prime$ will be evaluated as the projection of `Orientation Vector` for anisotropy $\{v_x, v_y, v_z\}$. If set OFF in-plane flow properties are assumed to be isotropic characterized by permeability k'_x.

`Non--Linear`	obligatory If ◎ **Direct input** is set ON data for Coulomb type frictional law describing contact interface material model are: • cohesion c, $\{[\text{kPa}], [\geq 0], 0\}$ (valid only for small deformation contact formulation) • friction angle ϕ, $\{[\text{deg}], [0, 89], 0\}$ • dilatancy angle ψ, $\{[\text{deg}], [0, 89], 0\}$ (valid only for small deformation contact formulation) If ◎ **Set table of scaling factors to cohesion, friction...** is set ON then user may set up multipliers to $\tan(\phi)$, c and $\tan(\psi)$ at each material for **`Continuum`** or **`Continuum for structures`**; the current contact strength parameters are inherited from the adjacent continuum elements and modified (if multiplier is different from 1.0) internally by the calculation module; to edit multipliers click on button **Setup**. User may set several tables with scaling factors but only one can be applied to the given interface material. **Remark**: Tensile `Cut--Off` is implicitly activated in the model

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 $K_v^{ref} = \dfrac{k}{\gamma^F \; h^e}$ 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 ϵ is a large number (penalty method is used), but there are also two other possibilities (since version 2018) to set this value. User may set direct value of K_v parameter or may set his own multiplier to the K_v^{ref}.

Data	Use & Set up
`Flow`	obligatory; • **`Multiplier`** to $K_v^{ref} \; \epsilon$, $\{[-], [>0], 1\}$ • Direct value of K_v $\{[m^3/(kN \; s)], [>0], 1\}$ • **`Multiplier`** to K_v^{ref}, $\{[-], [>0], 1\}$ Use standard option with multiplier 1.0 unless computed pore pressure fields do not satisfy the assumed pressure boundary conditions with required order of accuracy; in such cases try to enlarge the **`Multiplier`**; this may however lead to convergence problems. In special cases when modeling drains with limited influx use second (direct value of K_v) or third setting (user given multiplier to the reference value K_v^{ref})

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 `Analysis & Driver`) 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... **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**.
`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

`Postprocessing`	Runs unified 2D/3D graphical postprocessor to visualize results of computation: input data, deformed mesh, plastic zones, stress resultants in form of diagrams or color contours, stress/strain contours, time histories of any selected nodal or element quantities, color contours and diagrams in cross sections, principal stresses in continuum elements, principal membrane forces and/or bending moments in membrane/shell elements, section forces and moments in beams/trusses.
`Pushover Results`	Performs automatically seismic demand assessment according to given design spectra (EC8 or custom) Each action will invoke MS-EXCEL spreadsheet for numerical results and graphics. These include: demand spectra in A-D or A-T format, multi-DOF(i.e. original system) capacity curve, SDOF (i.e. equivalent single DOF system)capacity curve and demand spectra reduced by ductility, pushover analysis summary.
`Results for piles`	Shows results for piles, modeled as beams embedded in the 3D continuum, in the form of Excel sheet.
`Results for fix.anch.zones`	Shows results for fixed anchor zones in the form of Excel sheet.
`Eigen vectors`	Exports eigenfrequencies, mass participation factors and eigenvectors to the Excel sheet.
`View log file *.log`	Browse the log file (created by calculation module) to trace warnings which may indicate source of potential mistakes in the model. In case of fatal error, computation is stopped and explanation is given at the end of the file including some hints on how to remedy the problem.
`View element results`	Browse the file (*.str) containing results stored at Gauss points at each element according to the specification set under **`Control/Results content`** menu.
`View nodal results`	Browse the file (*.dis) containing results stored at nodal points according to the specification set under **`Control/Results content`** menu.
`View echo of data`	Browse the file (*.eda) containing echo of the data (*.dat) file generated by the menu and sent to the calculation module.

1.6 EXTRAS

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

`Run Analysis`	Calls computational module without storing job file (*.inp) nor creating *.dat file. Use in case when *.dat file is modified manually.
`Write *.dat`	Creates data file (*.dat) for the computation module without calling it
`View *.dat`	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)
`View *.inp`	Edits standard input file (*.inp) allowing for its modification (risk of complete loss of data)
`Units`	Activates a tool to convert units
`Databases`	Gives an access to the steel profiles data base through submenu **`Database with steel profiles`** and stored materials database through submenu **`Database with material data`**
`Protection report`	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 **Extras**) 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