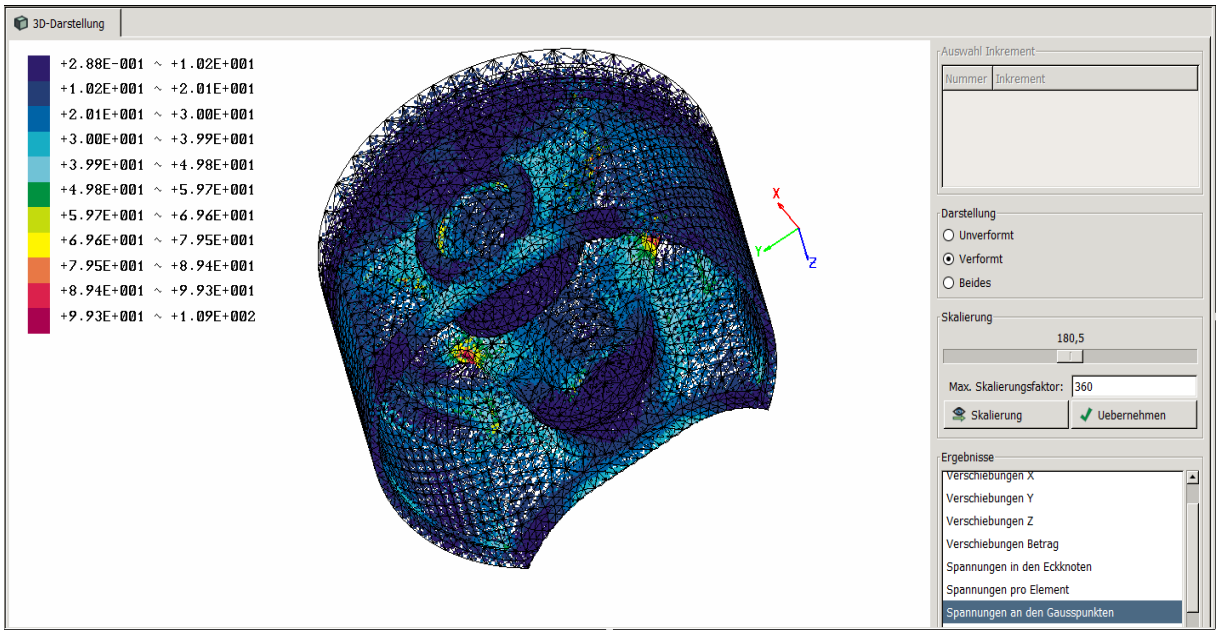
Z88 Aurora®

User manual



Version 5

An easily operated user interface for Z88® for Windows, Linux and macOS (64-bit).

This freeware version is the literary property of the Chair for Engineering Design and CAD, University of Bayreuth, Germany, composed and edited by Professor Dr.-Ing. Frank Rieg.

With the aid of:
Dr.-Ing. Bettina Alber-Laukant;
Dipl.-Ing. Daniel Billenstein; Maximilian Braun, M.Sc.;
Kevin Deese, M.Sc.; Christian Dinkel, M.Sc.; Pascal Diwisch, M.Sc.;
Dr.-Ing. Michael Frisch; Johannes Glamsch, M.Sc.;
Christian Glenk, M.Sc.; Dr.-Ing. Daniel Goller;
Dipl.-Wirtsch.-Ing. Reinhard Hackenschmidt;
Stefan Hautsch, M.Sc.; Florian Hüter, M.Sc.;
Dipl.-Ing. Claudia Kleinschrodt; Christopher Lange, M.Sc.;
Dr.-Ing. Martin Neidnicht; Dr.-Ing. Florian Nützel; Dr.-Ing. Bernd Roith;
Matthias Roppel, M.Sc.; Frank Rudolph, M.Sc.; Dr.-Ing. Alexander Troll;
Dr.-Ing. Felix Viebahn; Dr.-Ing. Christoph Wehmann; Tobias Weiß, M.Sc.;
Dipl.-Ing. Johannes Wittmann; Aljoscha Zahn, M.Sc.; Dr.-Ing. Jochen Zapf;
Dr.-Ing. Markus Zimmermann; Dr.-Ing. Martin Zimmermann

Version 5, April 2019

ABOUT Z88AURORA® AND WHAT'S NEW IN Z88AURORA V5

Z88 is a software package for solving structural mechanical, static problems with the aid of the Finite Element Analysis (FEA), which is available under the GNU-GPL as free software with source code. The software, originally created by Professor Frank Rieg in 1986, is currently being further developed by a team under the supervision of Professor Rieg at the University of Bayreuth.

In addition to the present compact Z88, which is currently available in the 14th version, an extended program Z88Aurora is on the market since 2009. Z88Aurora® is based on Z88 and is available for Windows 64-bit, Linux 64-bit and Mac OS X for free download (as an executable file). In addition to the efficient solvers contained in Z88, Z88Aurora® offers a graphical user interface (GUI), a completely new preprocessor and an extension of the approved postprocessor Z88O. During the development of Z88Aurora®, great effort was put into improving the usability.

Since version V2, Z88Aurora® offers, in addition to static strength analysis, a material database containing more than 50 established construction materials and modules such as nonlinear strength calculations, natural frequency analysis and thermal analyses. Since Z88Aurora® V3, the program includes an extended nonlinear equation solver, which now also allows to consider non-linear material behavior besides geometrically nonlinear analysis. Therefore, three plastic material laws are implemented. The surface has been enhanced accordingly in order to be able to enter the additionally required material data. Since version 4, Z88Aurora® includes a contact module, which provides frictionless and bonded contact formulations. The GUI offers different part operations for creating and aligning assemblies (e.g. importing parts, moving, rotating, scaling and copying).

The current version Z88Aurora® V5 offers an extended thermo-mechanical FE-solver for the simulation of convective heat transfer from a solid body to the surrounding. Furthermore, the new postprocessor of Z88Aurora® V5 provides the possibility to choose between different view options and includes a new feature for exporting images of the current viewport.

Other modules such as a module for transient analysis are in development.

The modules:

Module for linear static analysis

The basic module of Z88Aurora® is the linear static module, which is used for linear static analyses. This is the first solver included in Z88Aurora®. The only material parameters required are Young's modulus and Poisson's ratio. The user has the choice of a number of boundary conditions: displacements, forces, pressures or surface loads.

Module for thermal analysis

In this module of Z88Aurora® steady state thermal conduction and the thermal expansion are calculated. The simulated temperature profile is treated separately and is time-independent, i.e. the state of equilibrium is displayed. By linking the thermal and mechanical boundary conditions the user can calculate thermo-mechanical displacements or stress in addition to the thermal results, like temperature or heat flow. With this, statements can be made about the influence of temperature on the construction component.

Module for natural frequency f

Since Version V2, Z88Aurora® also gives the possibility to analyse a component regarding its natural frequency. If the material properties Young´s modulus, Poisson's ratio and densitiy are known, this module can calculate the natural frequency. Alternatively fixing constraints for sets of nodes in one or several spatial directions can be applied. As a result the user gains information about the smallest natural frequency as well as the distortion of the component.

Module Z88NL for non-linear calculation

Lastly there is the possibility to conduct non-linear calculations of the structural mechanics. Regarding the boundary conditions homogeneous and heterogeneous displacements, different kinds of applying forces as well as pressure loads can be observed. The equation solver Z88NL evaluates the Finite Element Analysis considering the geometrical nonlinearities or, since V3, also considering non-linear material behavior. With this module the displacements (Z88NLO2.TXT) as well as Chauchy's stress (Z88NLO3.TXT) can be calculated for the the element types 1 (Hexahedron with 8 nodes), 4 (truss), 10 (hexahedron

with 20 nodes), 7 (plain stress element with 20 nodes), 8 (torus with 8 nodes), 16 (tetrahedron with 10 nodes) and 17 (tetrahedron with 4 nodes).
Since the release of Z88Aurora® V3 three different plastic material laws can be calculated.

The contact module Z88KONTAKT
Since version 4, Z88Aurora® can handle multiple parts and can conduct contact analyses. To this end, node-surface or surface-surface contact with an automated contact search is implemented. The possible contact types are bonded or frictionless. The methods for including these restrictions are Lagrange, perturbed Lagrange or penalty. The meshed (linear or quadratic hexahedrons or tetrahedrons) parts can be imported, positioned and manipulated via the GUI. The rest of the preprocessing can be done analoguous to the linear static analysis. It's important to note that the contact module only works in conjunction with the linear static module.

The Z88 philosophy is also valid for Z88Aurora®!

- *f*ast and compact: Developed for PC, no ported mainframe system
- full 64-bit support for Windows, Linux and Mac
- native Windows and Mac OS X programs, no emulations
- Windows and Mac OS X versions use the same computing kernels
- full data exchange from and to CAD systems (AutoCAD *.DXF, *.STP, *.STL)
- FE structure import (*.COS, *.NAS, *.ANS, *.INP)
- context sensitive online-help and video tutorials
- simple installation with Microsoft® Installer (MSI)
- Z88Aurora® V5 is completely compatible with Z88 V14/15 OS and Z88Aurora® V2/V3/V4. Already existing Z88 V13 files or Z88Aurora® V1 files can be easily imported with the conversion tool "Mitoo"!

Note:

Always compare FE calculations with analytical rough calculations, results of experiments, plausibility considerations and other tests without exception!

Keep in mind that sign definitions of Z88Aurora® (and also other FEA programs) differ from the usual definitions of the analytical technical mechanics from time to time.

⚠ **Unit conventions are independently managed by the user. The material database integrated in Z88Aurora® uses the units mm/t/N.**

Z88Aurora® is a powerful, complex computer program, which is still under development. If you have suggestions concerning future functionality, feel free to write an email to z88aurora@uni-bayreuth.de or contact us via the forum http://forum.z88.de.

The compatibility of Z88Aurora® with other programs is not fully tested. Especially data exchange is highly dependent on developments of the exporting third party software. Changes on this end are often not directly visible and are prone to cause problems. In these cases please contact Z88 support. FAQs can be found on the forum or in the handbooks.

System Requirements

- Operating systems: Microsoft® Windows® 7/8/8.1/10, Linux, MacOS® (64 bit)
- Graphics requirements: OpenGL driver
- Main memory: 1 GB minimum, recommended: 8 GB
- Documentation and videos require PDF-reader, video player, browser

Installation

For more information see the installation guide which comes with the installation of the Z88Aurora® package. For starting either use the desktop icon "Z88Aurora V5" or open the program in the start menu: "Z88Aurora V5" → "Z88Aurora V5". If you haven't installed the desktop icon nor the start menu entry you can start the program via the Windows-Explorer:
"C:\Z88AuroraV5\win\bin\z88aurora.exe".

Documentation

The Z88Aurora® documentation consists of:

- User manual containing a detailed overview of GUI
- Theory manual with an elaborate description of the embedded modules
- Examples for the most common applications in mechanical analyses
- Element library displaying the integrated element types in Z88Aurora®
- Video manual containing some topics of special interest
- SPIDER-Workflow: Process support by a workflow tool

TABLE OF CONTENTS

1. AN OVERVIEW OF THE USER INTERFACE

Z88Aurora® is characterized by an intuitive operation of the pre- and postprocessor. The project data management is carried out by means of a project folder management. A status display provides better ease of use.

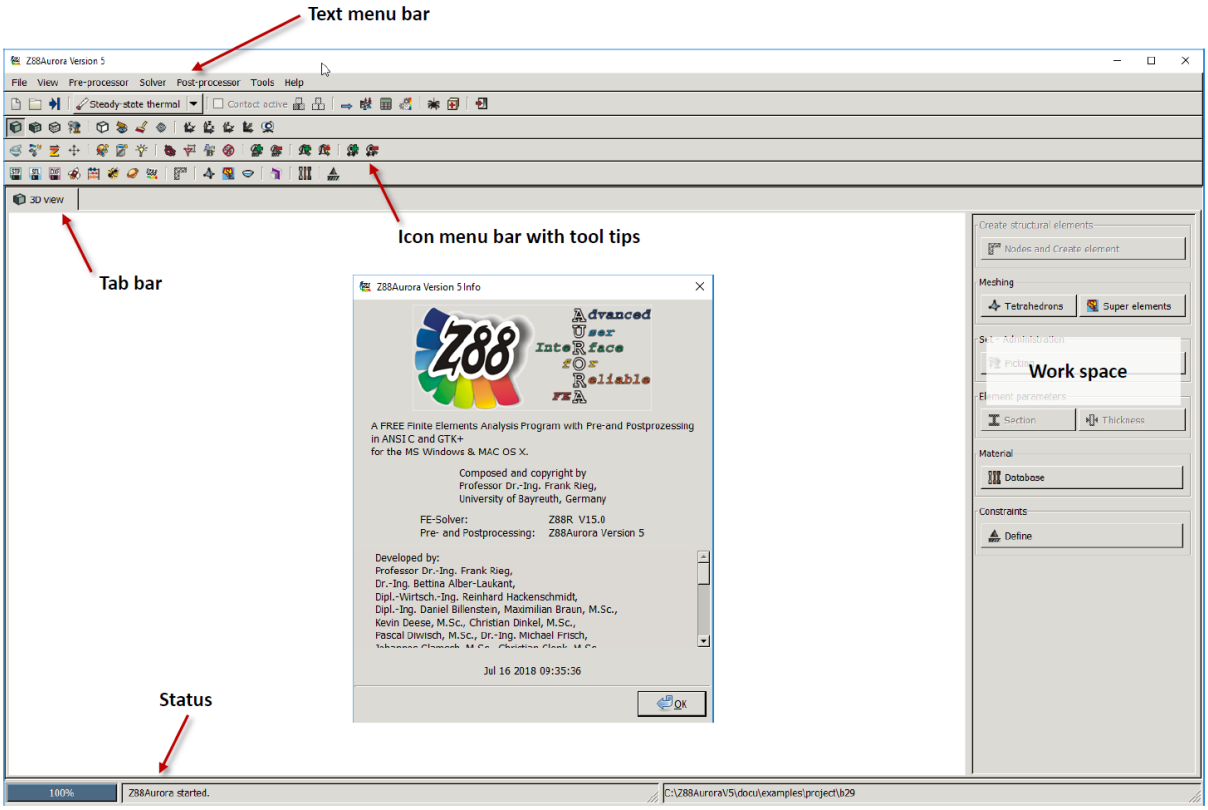


Figure 1: User interface of Z88Aurora® V5

2. MENU BARS

Several menu bars are of importance for operation. The four icon menu bars provide quick access to all functions of Z88Aurora®. The main functions of the first icon menu bar, such as preprocessor  , open additional side menus. The other three icon menu bars contain view, color and import options and the functionalities for the preprocessor.

The text menu bar contains all functionalities of the icon menu bar and the side menus, the correspondent icons precede the text commands. Depending on the current procedure, there are several tabs on the tab bar, such as the material cards in the material menu, between which you can switch. Each tab can be closed by clicking the corresponding “X”.

The icon menu bar is separated into different areas: the project folder management, the type of analysis and the pushbuttons, which access the context sensitive side menus and the

support. Depending on the state of the analysis several icons are displayed in grey, because their functions aren't available at that time.

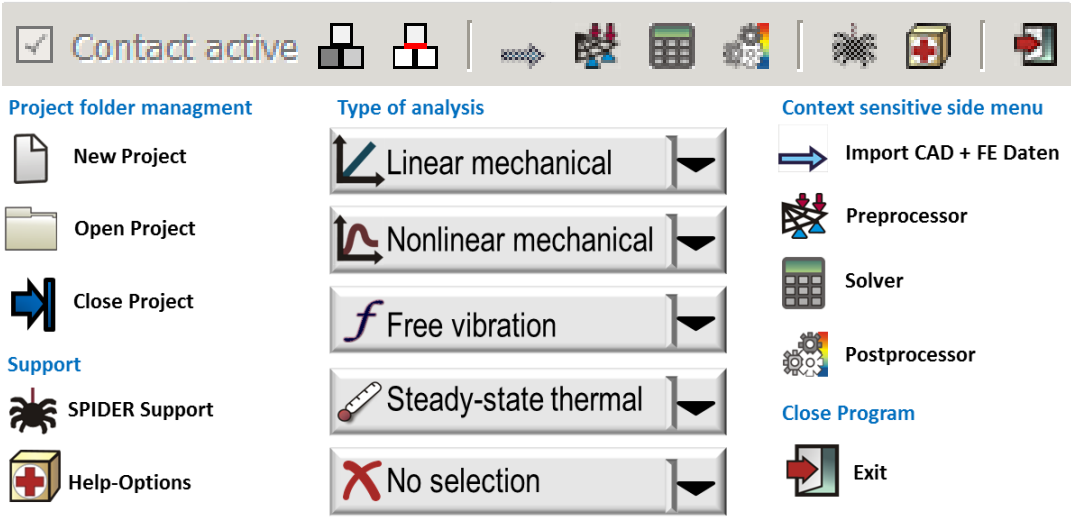


Figure 2: Pushbuttons of the icon menu bar

Please always note the status display at the lower left edge of the user interface. Here you can find references to the next steps and information about operation!

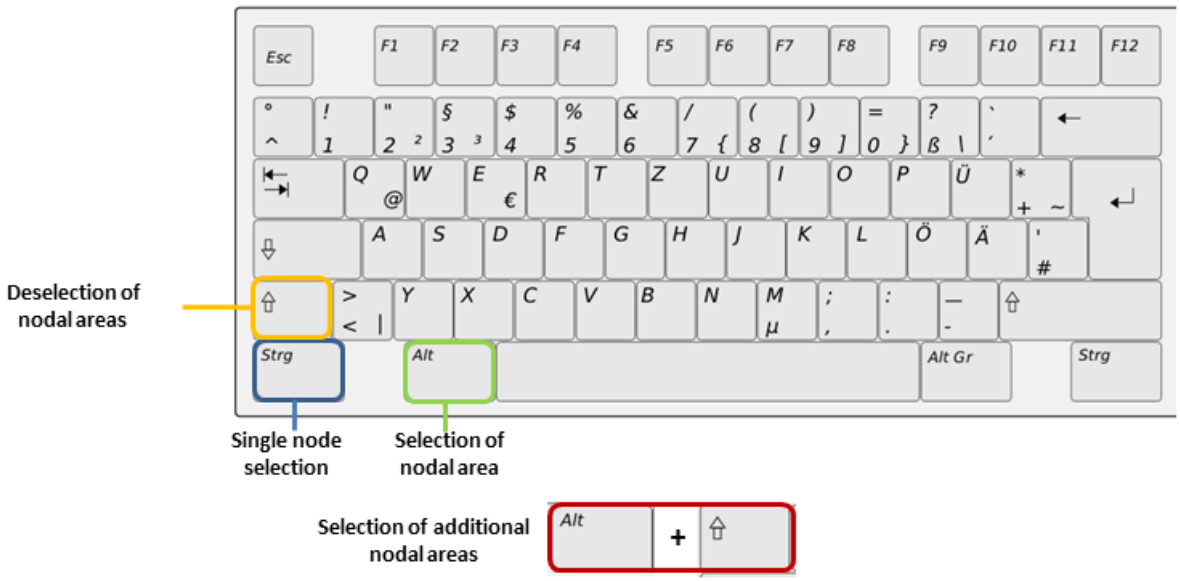


Figure 3: Keyboard layout

2.1 Project folder management

Depending on the status of the project it is possible to launch a new project folder or to open an existing project. Options which aren't accessible at the time are displayed in grey.

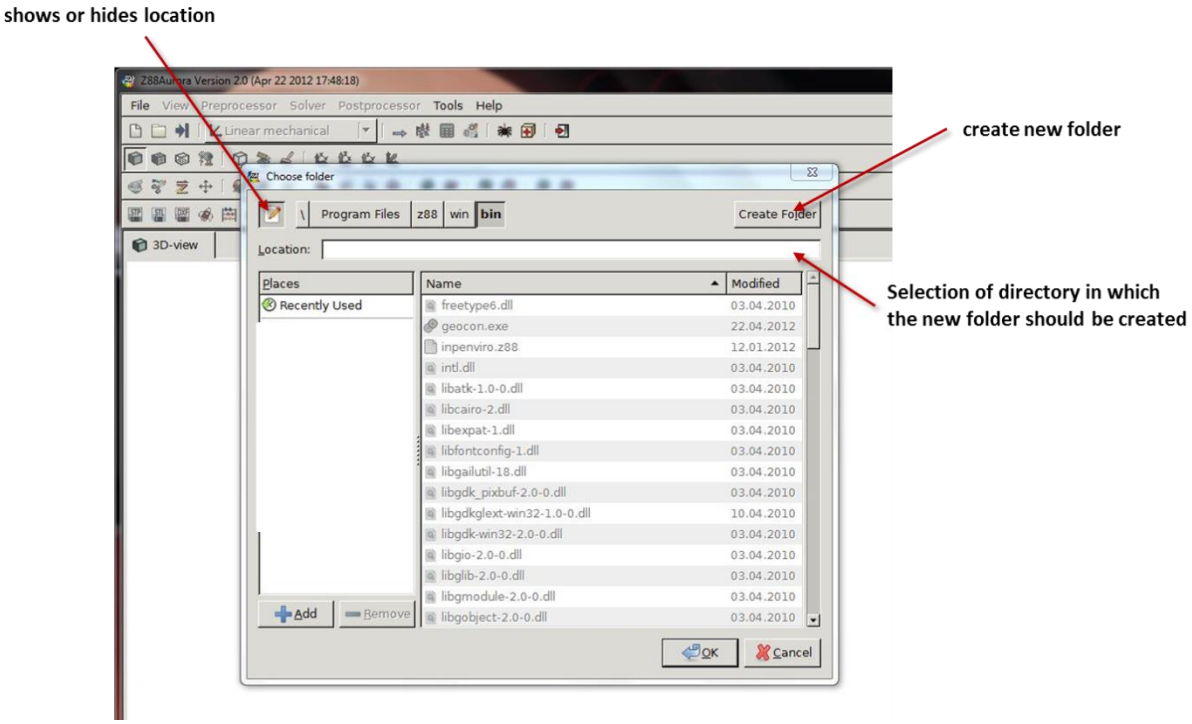


Figure 4: Project folder management of Z88Aurora®

2.2 Creating a New Project Folder

⇨ Create a new folder

⇨ Enter folder name „Name“

⇨ Confirm (Return)

⇨ Click OK to confirm

The input mask disappears and you can start the compilation of the computation model.

For further use, the project folder can be put into the quick access! (⇨ Add)

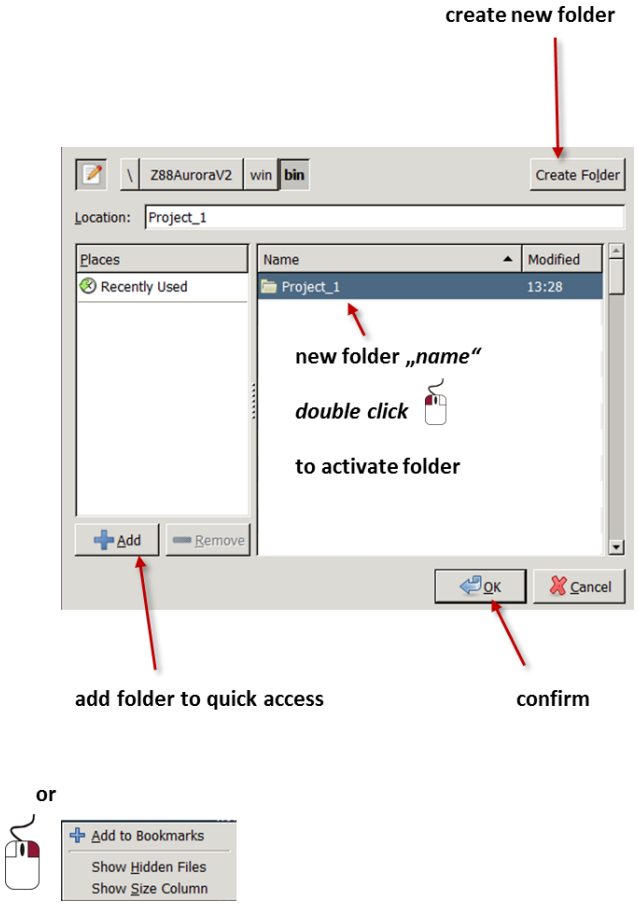


Figure 5: Launching a new project folder

2.3 Opening a Project Folder

⇨ Select a project folder to open

⇨ Click "OK" to confirm. The project is displayed in the work area.

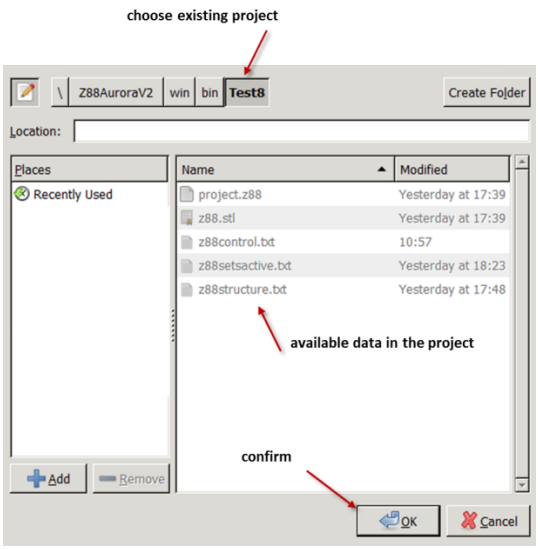


Figure 6: Opening an existing project folder

⇨ Generally it is possible to open the current project folder directly at the GUI with a double click, see Figure 7.

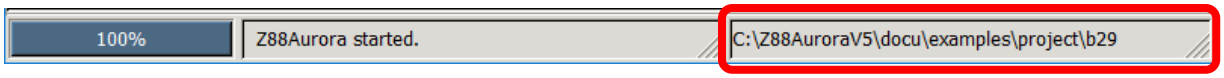


Figure 7: Direct file picker

2.4 Closing a Project Folder

With this button the presently open project folder is closed.

You must always close the current project folder before creating a new one or opening another project!

2.5 Project Folder Management in the Text Menu Bar

In addition to the icon menu bar, Z88Aurora® possesses a text menu bar above the icon menu bar. This either contains further functionalities or you can access the same functions as in the icon menu bar. The text menu bar with its respective functions is described in the corresponding chapters.

Functionalities which aren't available are displayed in grey.

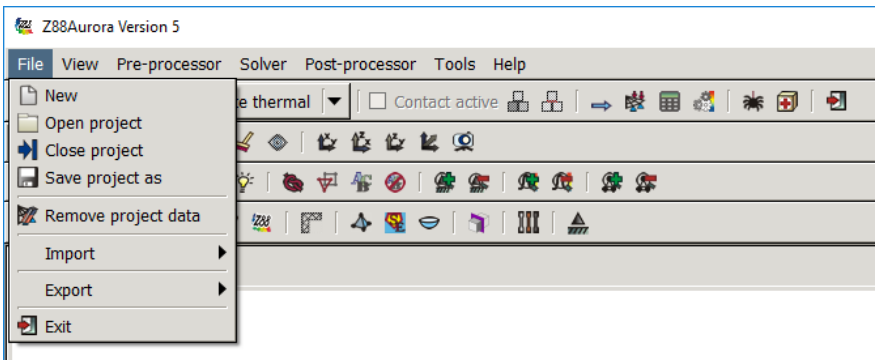


Figure 8: Project folder management in the text menu bar

2.6 Deleting Project Files

In the text menu bar there is also the option to delete the complete content of the project folder. The folder itself is preserved.

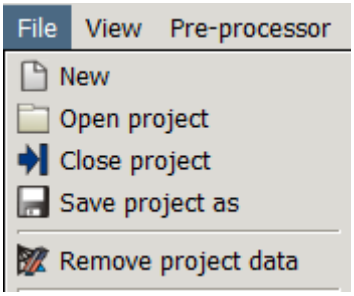


Figure 9: Deleting project data in the text menu bar

3. VIEW

The view options can be edited in many ways in Z88Aurora®. It is possible to display often required tool bars and to change their arrangement, to change the light, material and legend color or to switch miscellaneous additional view options on and off. With “labels” is possible to switch the numeration of nodes and elements on an off.

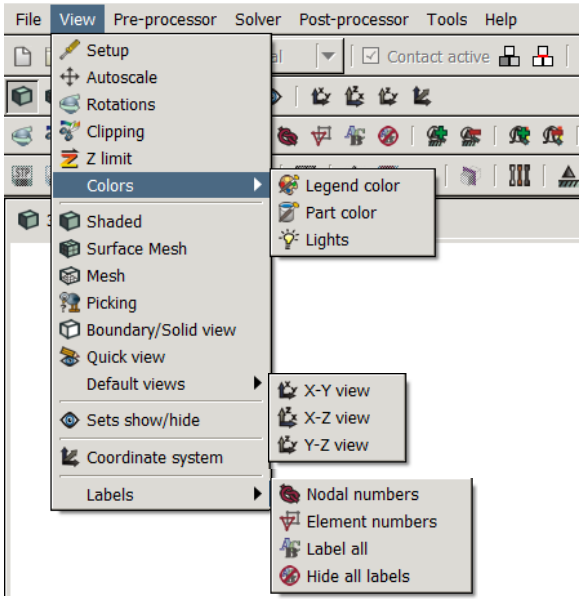


Figure 10: View options

3.1 Toolbars

For import/export, view and preprocessor it is possible to show additional toolbars. This can be done permanently via the settings in the file z88enviro.dyn or session-oriented via the menu "View">"Setup".

Contrary to Z88Aurora® V1 the arrangement of the icons can be completely adjusted to the user’s requirements. For this the default settings of the toolbars are stored in the z88enviro.dyn file with their respective icon number:

```
**Buttons of the toolbars:
TOOLBAR 1 1
2 3 4 0 1 0 5 6 7 8 0 9 10 0 11 -1
TOOLBAR 2 1
50 51 52 53 0 54 55 27 0 22 23 24 25 -1
TOOLBAR 3 1
12 13 14 26 0 15 17 16 0 18 19 20 21 0 28 29 0 30 31 0 32 33 -1
TOOLBAR 4 1
```

```
41 42 43 44 45 46 47 48 0 35 0 36 37 64 0 65 0 39 0 40 -1
```

Figure 11: Extract of z88enviro.dyn

In the following figures the default arrangement of the toolbars and their meanings are displayed:

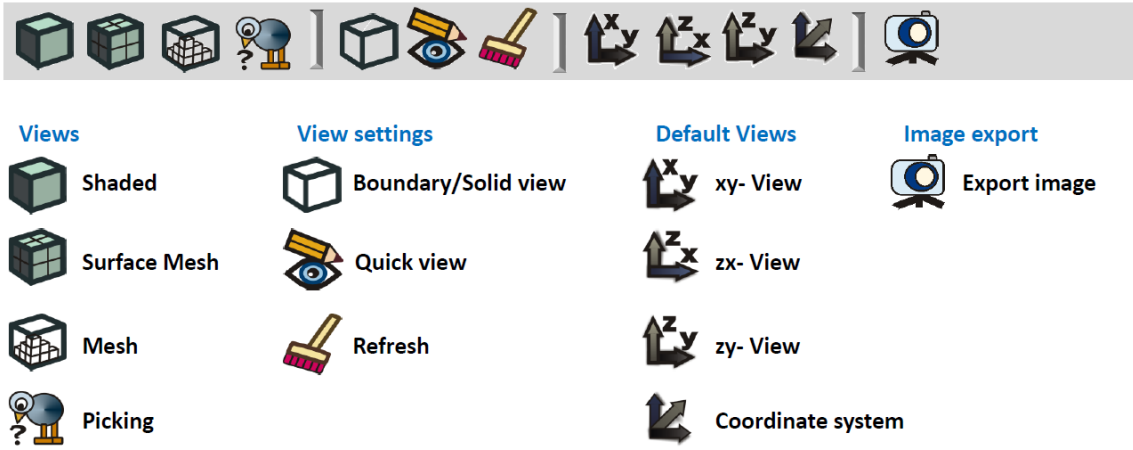


Figure 12: push buttons of the second toolbar

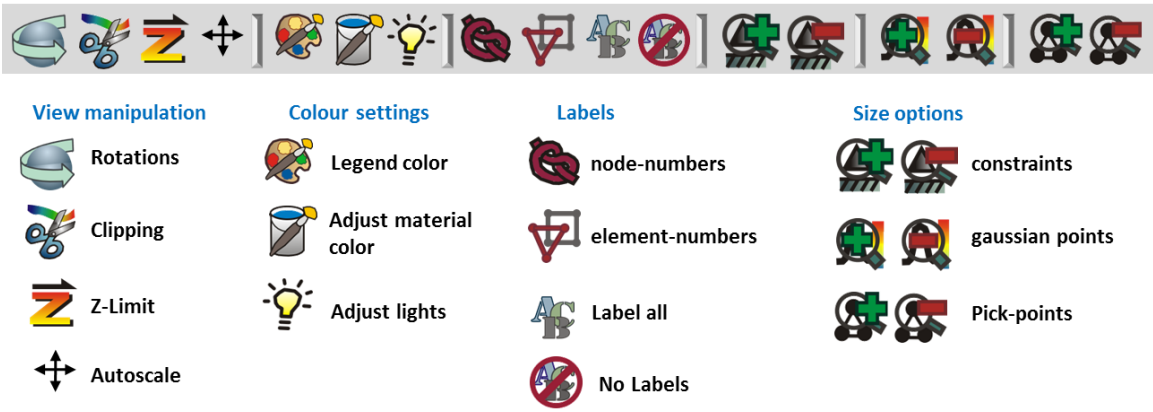


Figure 13: push buttons of the third toolbar

Figure 14: push buttons of the forth toolbar

To adjust the toolbar simply arrange the respective numbers.

3.2 Camera Settings

Auto scale offers the possibility to fit the model into the Open GL window. With Rotations 3D a precise rotation can be applied. Z limit towards the user is a clipping option. By setting a defined Z plane the component can be viewed from inside.

With all three planes can be used for clipping via the scroll bar.

3.3 Colors

The legend color as well as the background color of the Open GL window can be changed arbitrarily. For this, you can resort to defined standards (black/white, white/black, default) or manually set a certain color. The component is adequately displayed via the component color and the light settings.

3.4 Displays

There are four possible view options. These can be accessed via the icons in the icon menu bar.

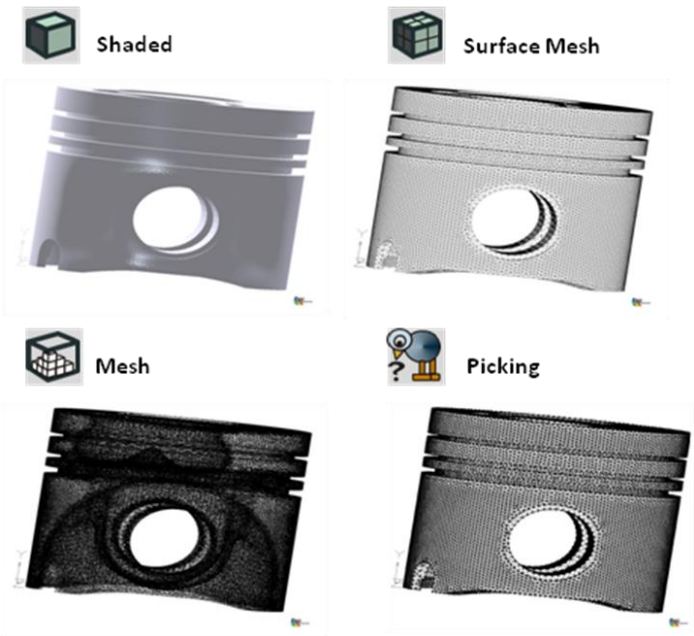


Figure 15: Display options in Z88Aurora®

The view options shaded, surface mesh and mesh can be applied by the user according to his needs; the Picking view is used for the selection of nodes, surfaces and elements. Activating quick view in combination with the display mode “shaded” enables large components to be moved quickly. To improve the display rate of the picking mode in combination with large

components, the Boundary/Solid View can be activated. In this mode only the surface of the component can be selected.

The Picking display depends on the previously selected display mode. Thus, you can either select all nodes or only surface nodes!

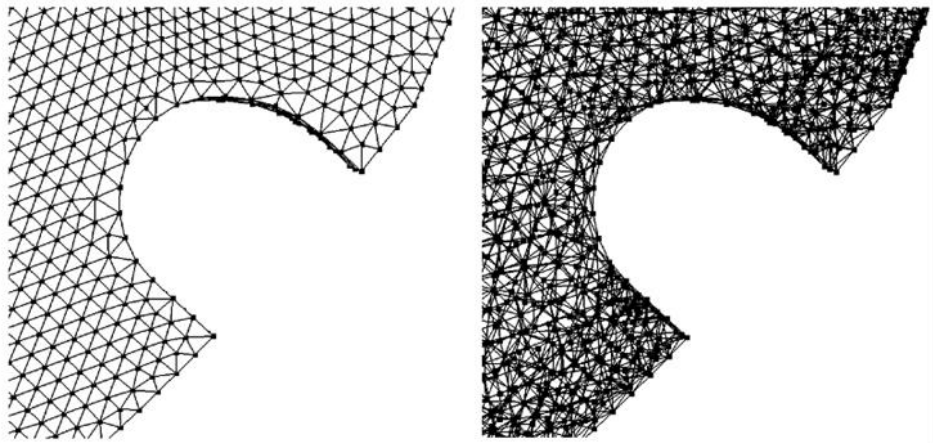


Figure 16: Switching to the display option "Picking", left: surface, right: all nodes

By clicking the “eye” existing sets and parts can be shown or hidden via the following menu, which is especially interesting for contact simulations.

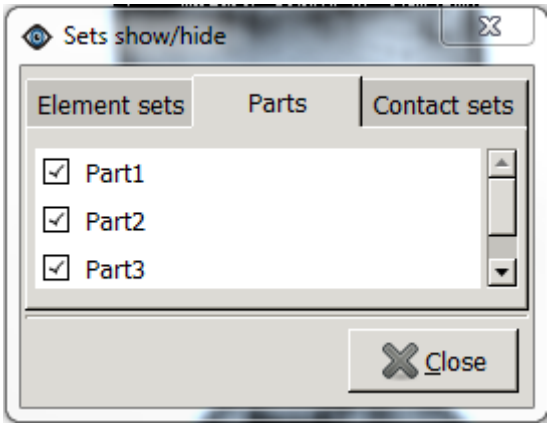


Figure 17: Show/Hide parts

This menu also shows the contact sets which signify the master and slave nodes which are considered during the contact analysis.

3.5 Views and View Options

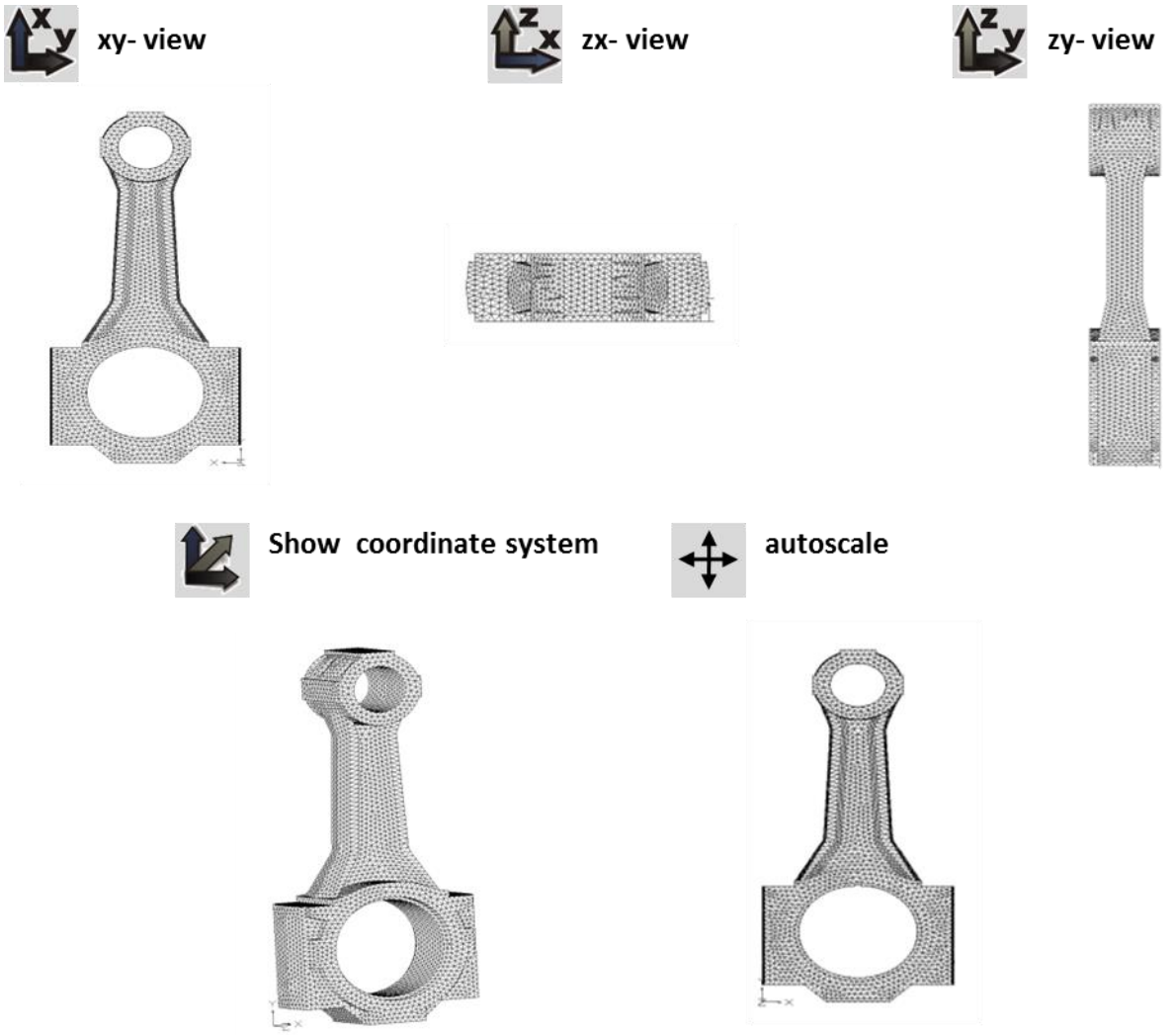


Figure 18: View options in Z88Aurora®

Double clicking on the respective icon or further clicking after the first orientation will rotate the view by 180°.

3.6 Image export

Since version 5, Z88Aurora® provides a feature for exporting images of the current viewport. Clicking on the camera symbol opens a dialog box for defining the export settings. The user can choose between different file formats (.bmp or .png), specify the name, and image size (in pixels) individually for each picture. The background of the exported image can be made transparent via the “Remove background” check box. Z88Aurora® saves the image file to the current project folder.

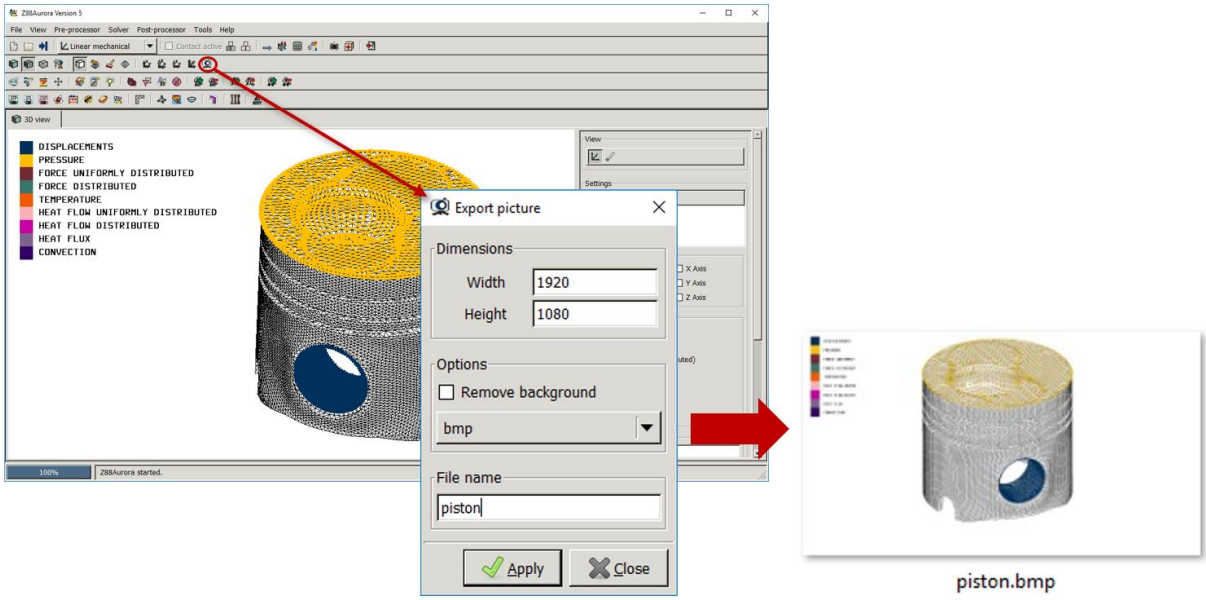


Figure 19: Image export in Z88Aurora®

3.7 Labels

The menu item "Labels“ is used to indicate the respective nodes and element numbers of selected objects and contains the following sub items:

Labels: Nodes

A window appears in which the numbers of the desired nodes must be entered. The dialog is ended with "OK“.

Labels: Elements

Similar to "Labels → nodes“, the desired element numbers must be entered, in order to display them.

Labels: Nodes and Elements

This function displays the labels of all nodes and elements.

Please keep in mind that this function might make the display of big structures with many elements and nodes confusing and, apart from that, may influence the speed of the program negatively, depending on the hardware used.

Hide all labels: Nodes and Elements

This function hides the labels of all nodes and elements.

3.8 Size of Boundary Conditions / Gauss Points/ Pick-Points

Size of boundary conditions

The function "Size of boundary conditions“ causes the shown boundary conditions to be displayed enlarged or reduced in the preprocessor menu.

Size of Gauss Points

With the menu item "Size of Gaussian points“ the size of the calculated Gaussian points, depicted here in the Z88Aurora® postprocessor, is defined (Figure 20).

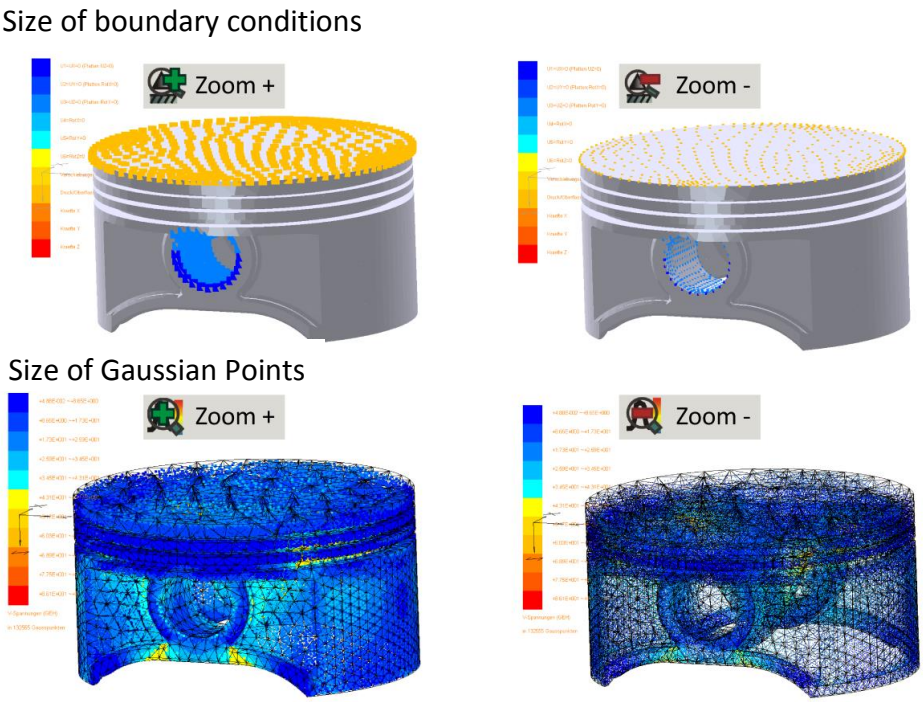


Figure 20: Display of boundary conditions and Gaussian points

Size of Pick-Points

The function "Size of Pick-Points“ causes the shown pick-points to be displayed enlarged or reduced in the menu "boundary conditions”.

4. CONTEXT SENSITIVE SIDE MENUS

When you have started a project, you can perform different actions. You can display and alter an existing project, but you can also import a structure from a CAD program as well as from an FE program.

4.1 Import of CAD and FE Data

After creating a new project folder it is possible to import geometry data as well as FE structures and to continue using them in Z88Aurora®. You will find an overview of the available formats in Figure 18.

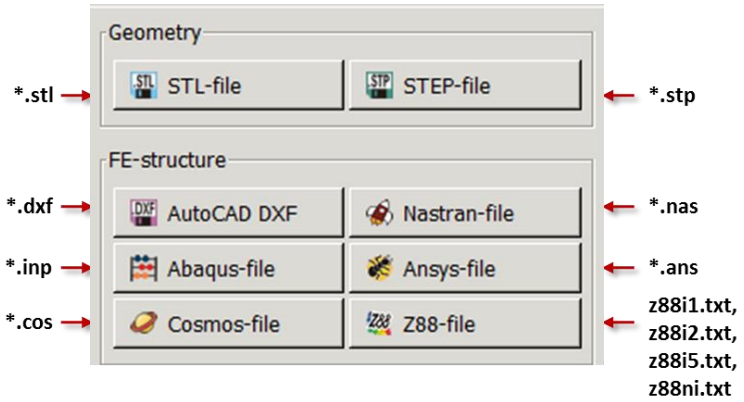


Figure 21: Import and export options in Z88Aurora®

Note: In case of a contact simulation including several parts, no import is possible. For carrying out a contact simulation, please refer to chapter Assembly management / Contact analysis.

<u>Compatibility with other versions of Z88</u>

For users who have already worked with Z88V15 OS there is the possibility to import existing Z88 input files into Aurora. In the process, the definition files required by Z88Aurora® are created automatically. A more profound insight into the file structure of Z88Aurora® is offered by chapter 3 of the Theory Manual. The input files Z88I1.TXT, Z88I2.TXT, Z88I5.TXT and the mesh generator file Z88NI.TXT can be imported. The files Z88I3.TXT and Z88I4.TXT are not required any longer in Z88Aurora®.

Older projects from Z88Aurora® V1/V2 and Z88V13 files can be imported by the migration tool “Mitoo.exe” , which is provided since Aurora V2. It can be found in the “bin” directory. Double clicking opens the migration dialog. The files are converted after choosing the respective directory and clicking on “Start”.

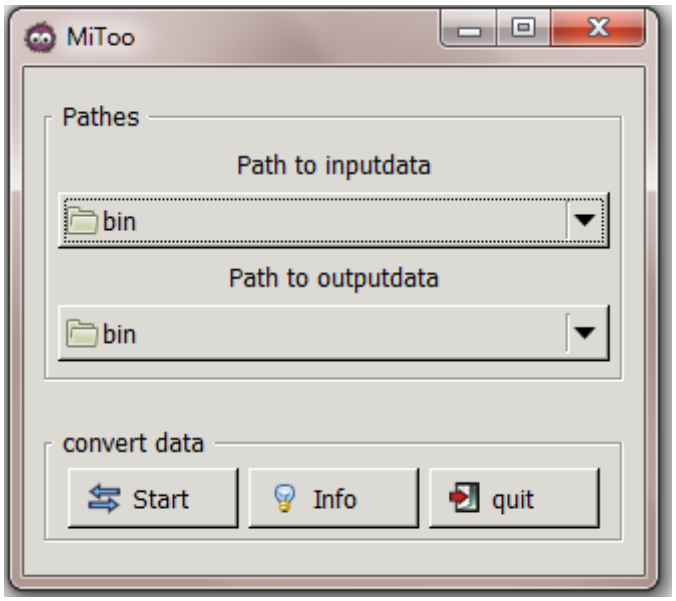


Figure 22: Migration tool Mitoo

For further processing files in Z88 V14/15 OS you can insert “enable write_only” in the file Z88.fcd. This creates a complete data set for Z88V14OS in the folder “bin”>“Z88V14OS”. The previous folder is replaced. If you still want to use these files you should save them in another directory.

Data Import

As an example, the import procedure of a STEP file is demonstrated (Figure 23):

⇨ Select Import/Export

⇨ Click " STEP file“, a selection window is opened

⇨ Select file

⇨ Click OK to confirm

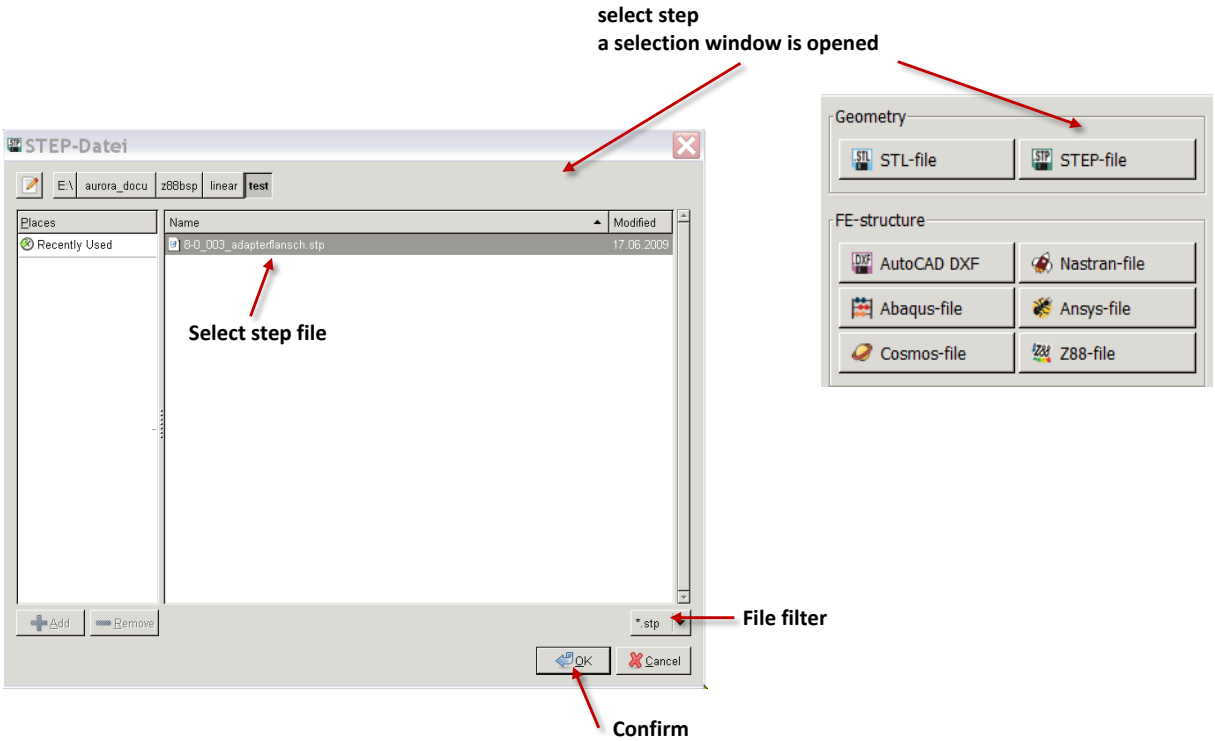
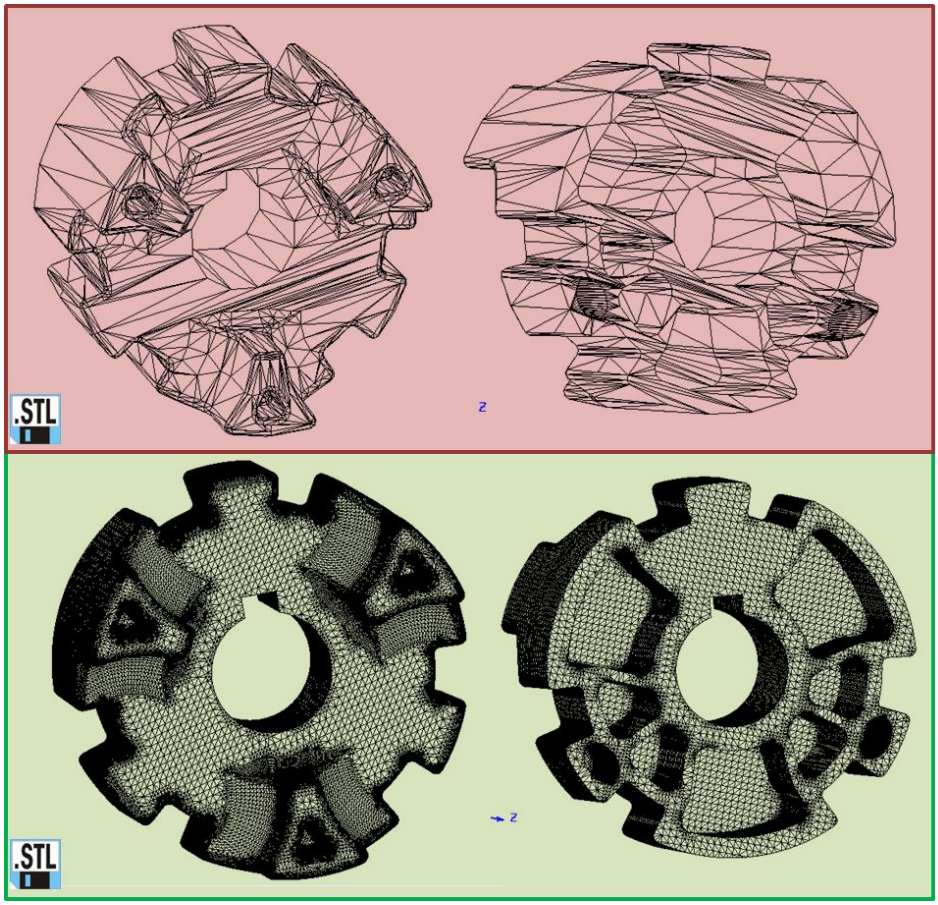


Figure 23: Import of a STEP file

- **The default setting determining which input file is supposed to be imported, is defined by the user.**
- **The import procedure depends on the quality of the given data. Incomplete or damaged STEP or STL data lead to incorrect displays and faulty meshing in Z88Aurora®.**

In this case it is necessary to adjust the export settings. Depending on the CAD program the lateral length, the interior angle or the width-to-height ratio can be changed.



Default setting
STL-Export Wildfire 5.0

☑ **Import**

☒ **Meshing**

Adjusted setting
STL-Export Wildfire 5.0

☑ **Import**

☑ **Meshing**

Figure 24: Import of a STL-file

All import features are described in detail in chapter 4.1 of the Theory Manual. Table 1 offers an overview of the model data, which can be transferred from FE structure data.

Table 1: model data which can be transferred from FE-structure data

	Z88V15.OS	DXF Autocad	ABAQUS	ANSYS	COSMOS	NASTRAN
FE structure	✓	✓	✓	✓	✓	✓
FE superstructure	✓	×	×	×	×	×
Nodal loads	✓	×	✓	✓	✓	✓
No displacements	✓	×	✓	✓	✓	✓
Surface load	✓	×	×	✓	×	✓

AUTOCAD-DXF files can be imported as four different file types (Figure 25). More information about creating an AutoCAD files and their preparation can be found in the Theory Manual, Chapter 4.1.5.

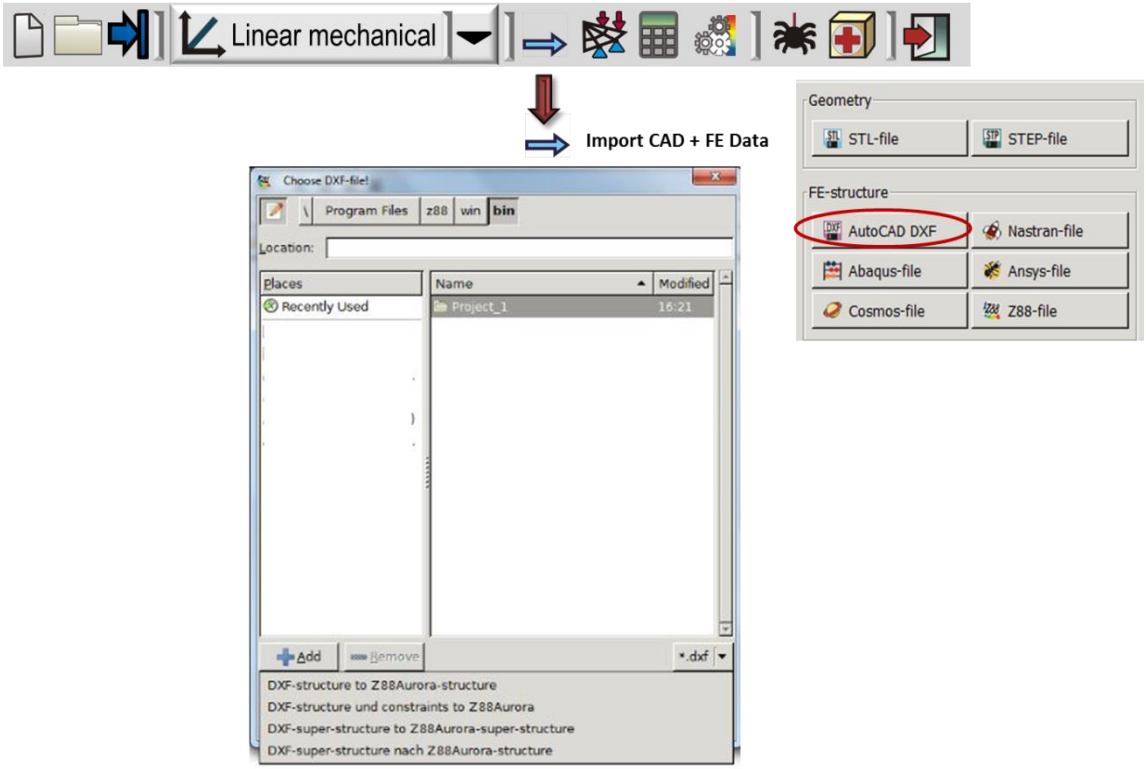


Figure 25: Possibilities to import a DXF-structure

The DXF import is designed for files created by the CAD system AutoCAD. If you are using a different program, the import might fail.

Import Text Menu Bar

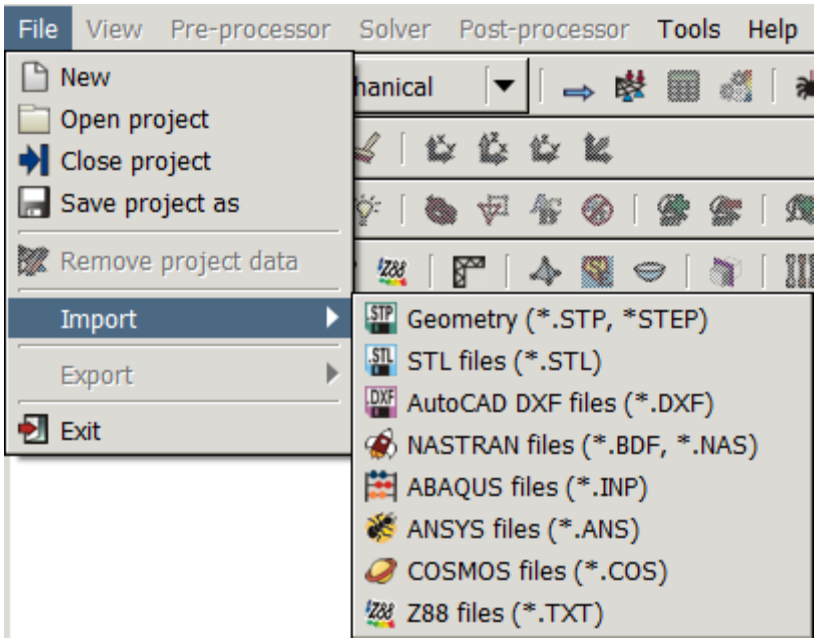


Figure 26: Import in the text menu bar

Import tool bar

The tool bar “Import” is shown by default. In the menu “View” and “Setup” you can deselect the view option “Import”.

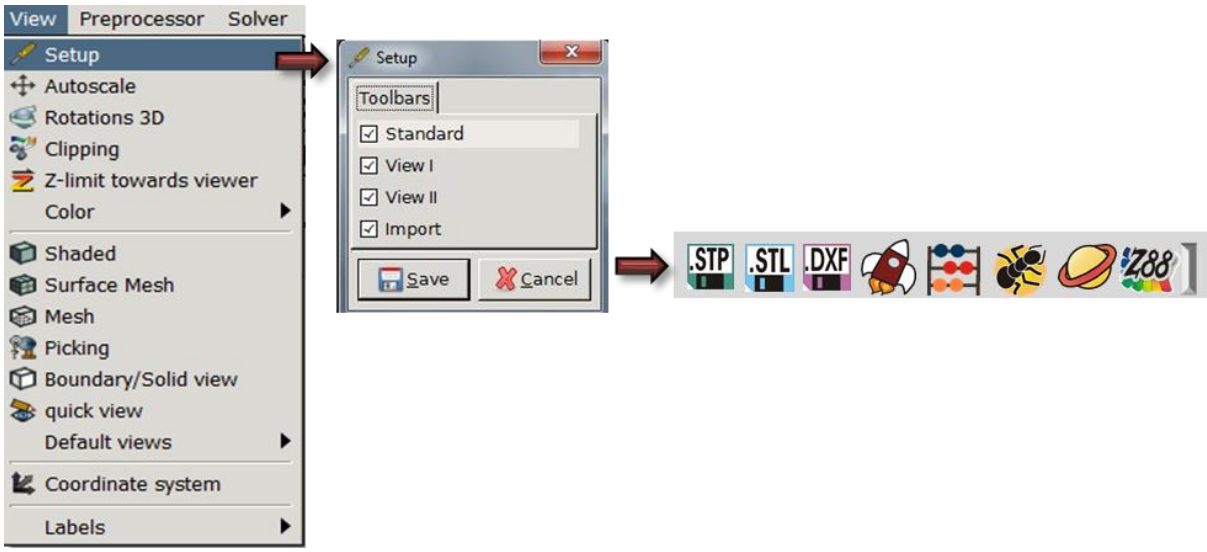


Figure 27: Tool bar Import

Import Text Menu Bar

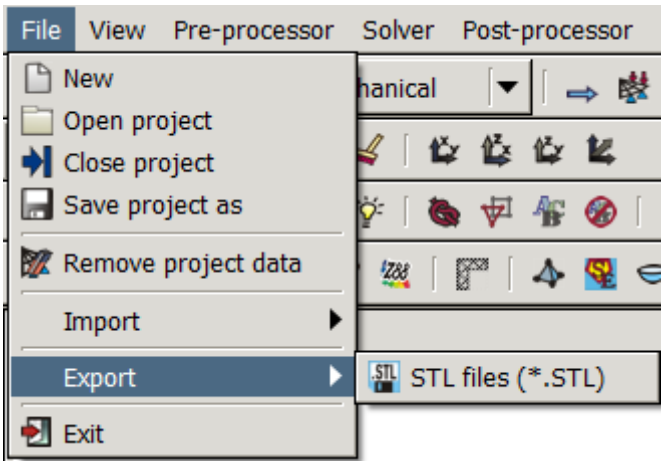


Figure 28: Export menu

A currently loaded FE structure can be written out again using the export function as an STL file.

4.2 Preprocessor

Clicking the preprocessor icon opens the context sensitive side menu "Preprocessor" (Figure 29) you can either create a FE structure or mesh an imported geometry. Afterwards it is possible to select a material from the database or to edit your own material. In addition, all mechanical boundary conditions can be applied.

All possibilities of the preprocessor are introduced separately below.

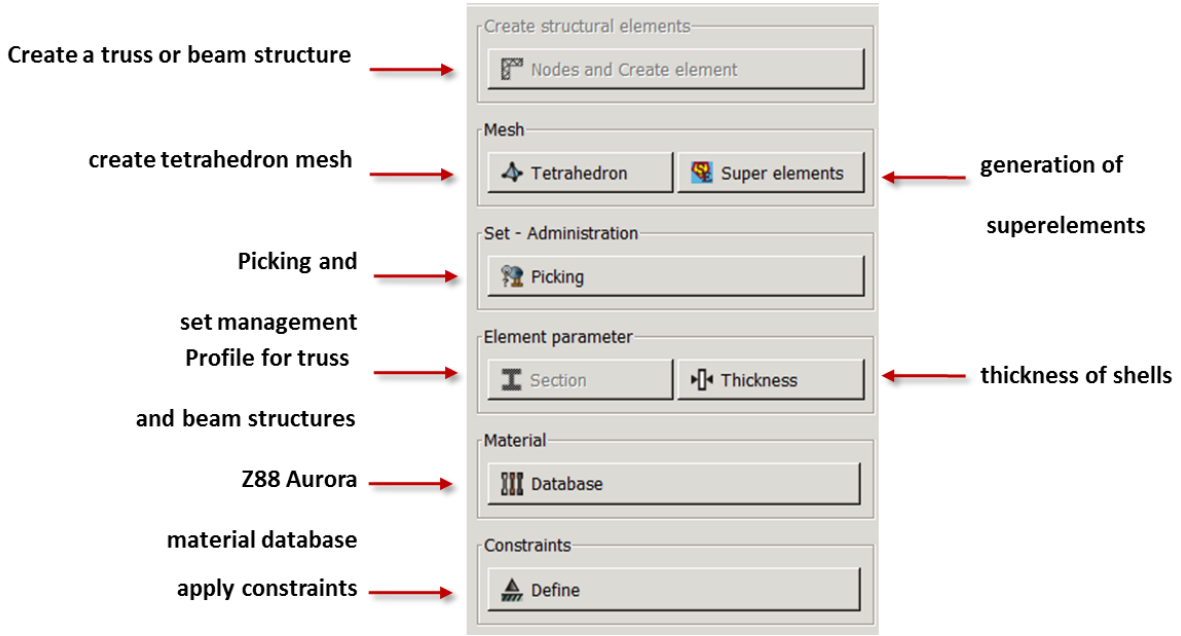


Figure 29: Side menu „Preprocessor“

Preprocessor in the text menu bar

All functions of the preprocessor can be accessed via the text menu bar.

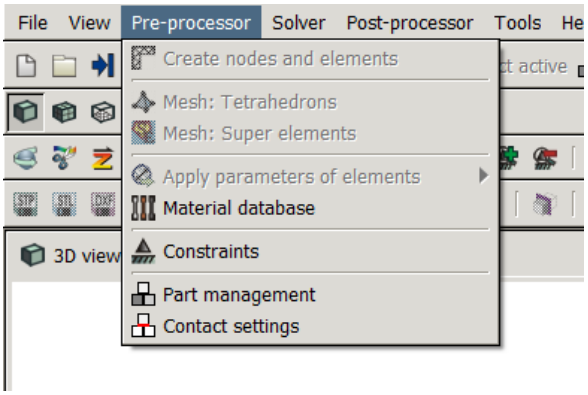


Figure 30: Text menu bar „Preprocessor“

Tool bar Preprocessor

In the menu “View” and “Setup” → “Toolbars” you can select “Import” to show the preprocessor buttons:

Picking

A main innovation in Z88Aurora® is the possibility to apply boundary conditions such as forces, pressure and restraints with only one mouse click in the graphic user interface.

This will be called “Picking” in the following chapters.

For Picking there is a separate view, which can be displayed by clicking the button in the main window.

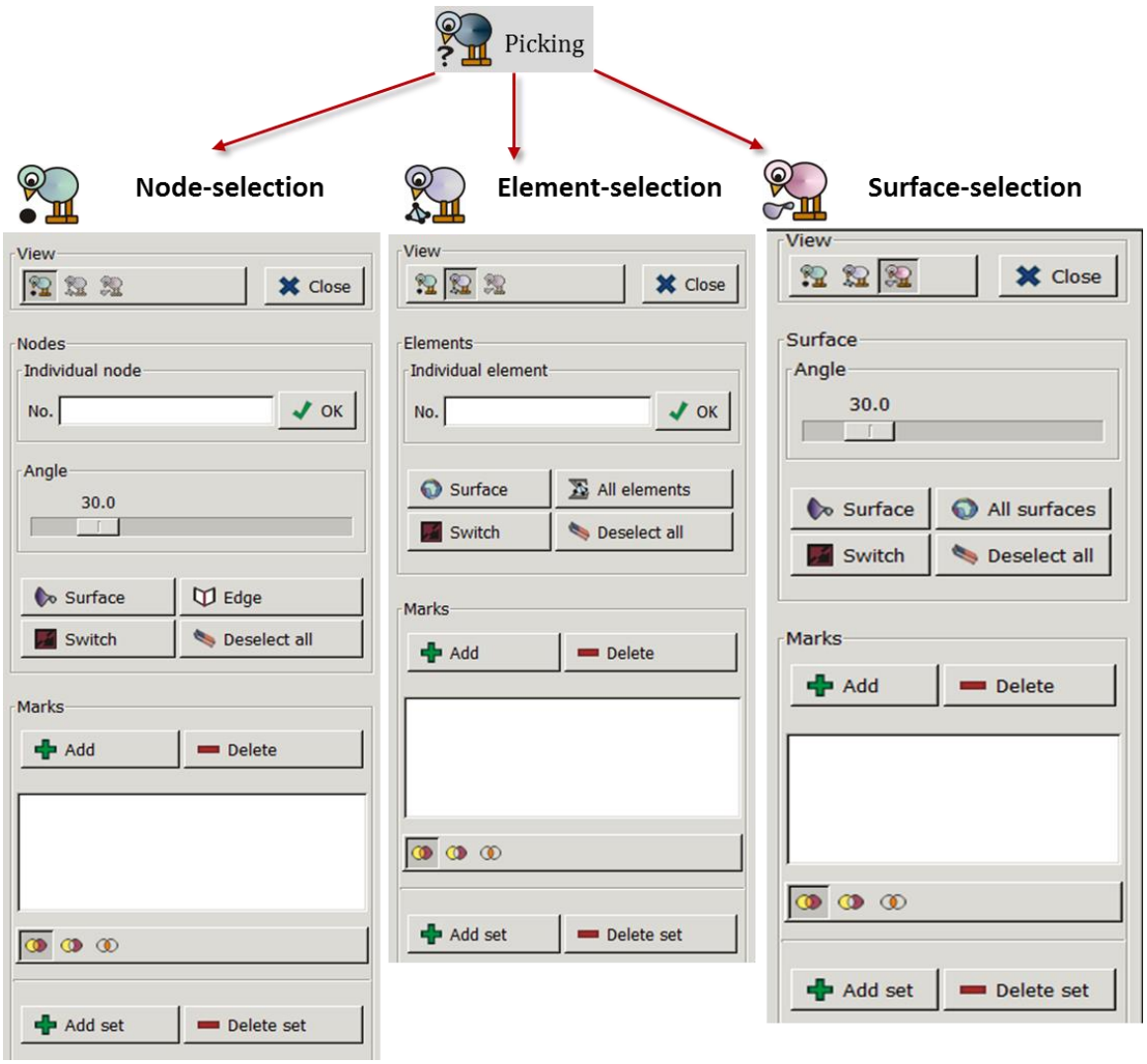


Figure 31: Picking options in Z88Aurora® V5

Hot keys

With the mouse and a view shortcuts it is possible, to “pick” single or several nodes, elements or surfaces, in order to define the boundary conditions you need:

Strg + (click) Selecting single nodes

Alt + (hold) New selection of several nodes in a rectangular area while discarding the previous selection.

Alt + ⇧ + (hold) Additional selection of several nodes in a rectangular area while maintaining the previous selection.

⇧ + (hold) Selecting a rectangular area for deselecting several nodes.

The respective nodes, elements and surfaces are marked as small black rectangles.

Selecting is carried out by clicking on the rectangles and using the shortcuts.

Picking of nodes

The Picking of nodes includes the following options:

- single nodes
- surface (doesnot work with shell elements)
- edge
- switch
- deselect all

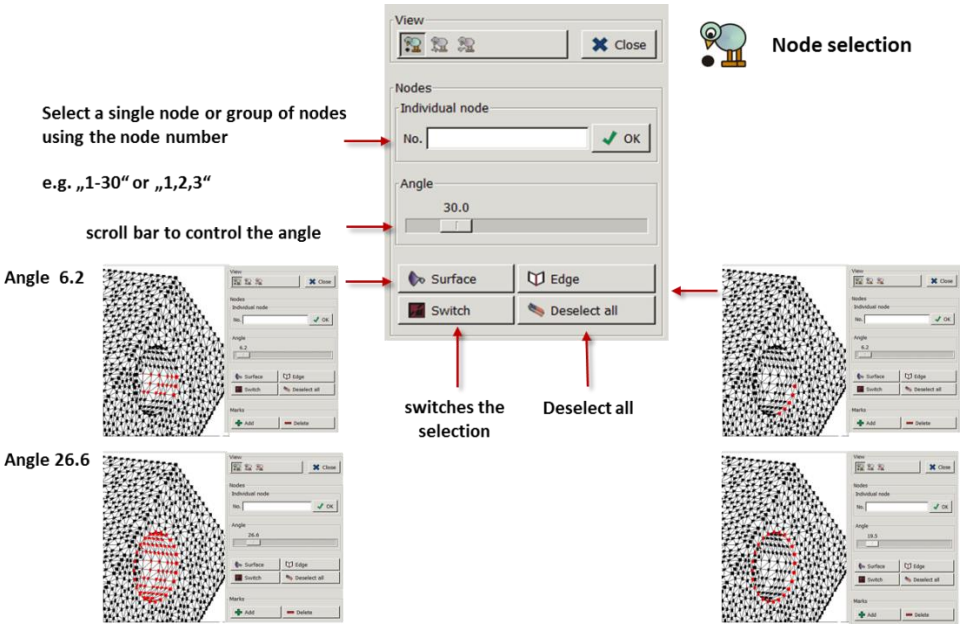


Figure 32: Picking of nodes

Single nodes: You can select single nodes via the node number as well as adjacent areas.

Surface: If you would like to select the interior surface of a drill hole to apply boundary conditions you can use the option “surface”. Pick a node with Strg+(click). With the scroll bar you can choose an angle, which influences the selection. This value describes the angle between the element that contains the selected node and the adjacent element. If the value is smaller or equal the value chosen via the scroll bar, the nodes on these elements will be selected.

To find the proper selection for the surface you want to choose, you might have to try some different values in order to achieve the desired result.

These settings can be used as a starting point to select the right surfaces (Figure 33):

- Planar surface: 0.0°
- A double line of nodes with a large curvature radius: 1° - 2°
- Lateral surface (complete or partial) with a large curvature radius: 5° - 10°
- Interior surface of a drill hole: 10° - 20°

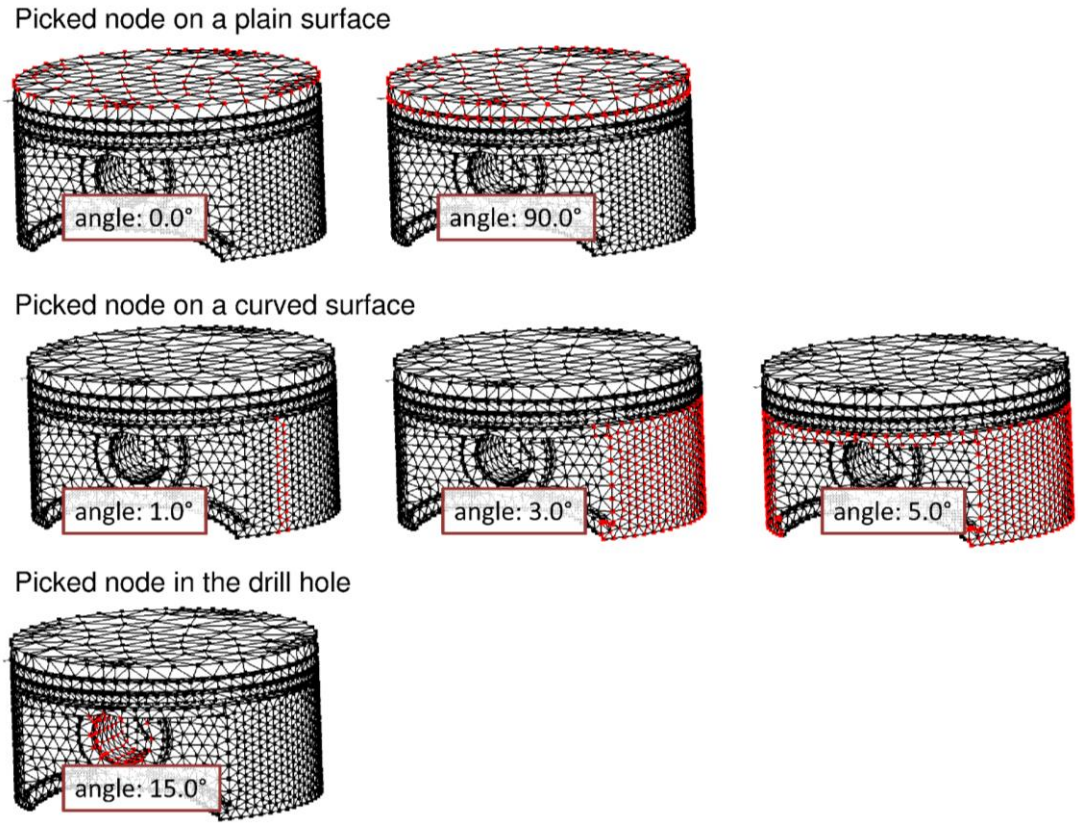


Figure 33: Setting of angles

⚠ **Only the selection of a corner node is possible (no nodes in the middle of the element)!**

Edge: A group of adjacent nodes, which are located on an edge of a FE-model, can be selected via the option “Edge”. With this option it is possible to select nodes on an edge of a drill hole or on a circumferential edge of a profile. Only one node on the edge must be selected. With the scroll bar also only a part of the edge can be selected.

Switch: With “switch” the selection is inverted.

Deselect: Deselecting of selected areas.

Picking of elements

There are following options for picking elements:

- surface
- all elements
- switch
- deselect

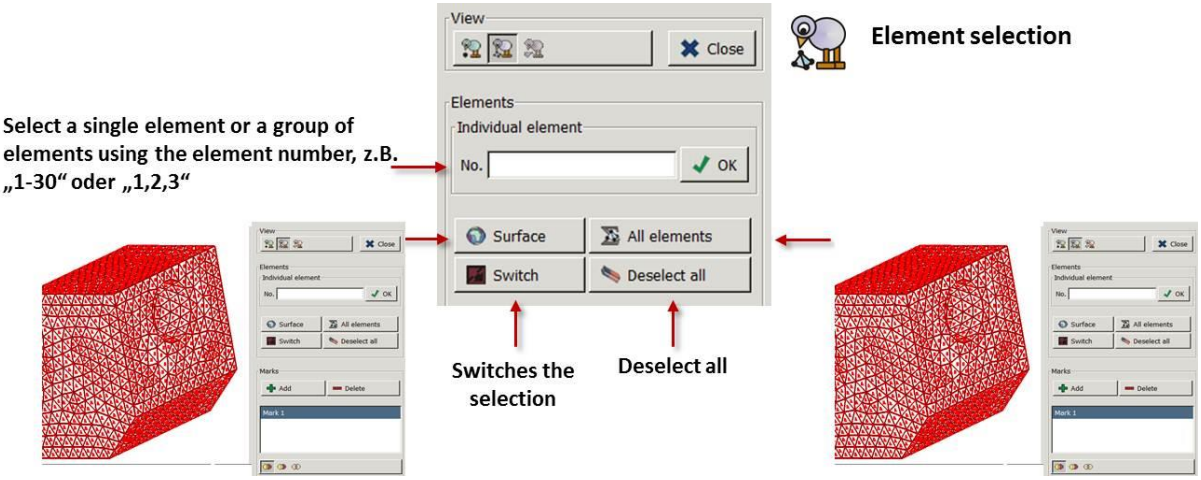


Figure 34: Picking of elements

To assign a material it can be advantageous to assign different material properties to different elements. This can be done with “picking of elements” with the normal picking option or with “picking of surfaces”. For rod- and beam-elements the selection of the elements can only be done by using the element number. The elements cannot be selected by the mouse-control.

Picking of surfaces

There are following options for picking surfaces:

- surface
- edge
- switch
- deselect

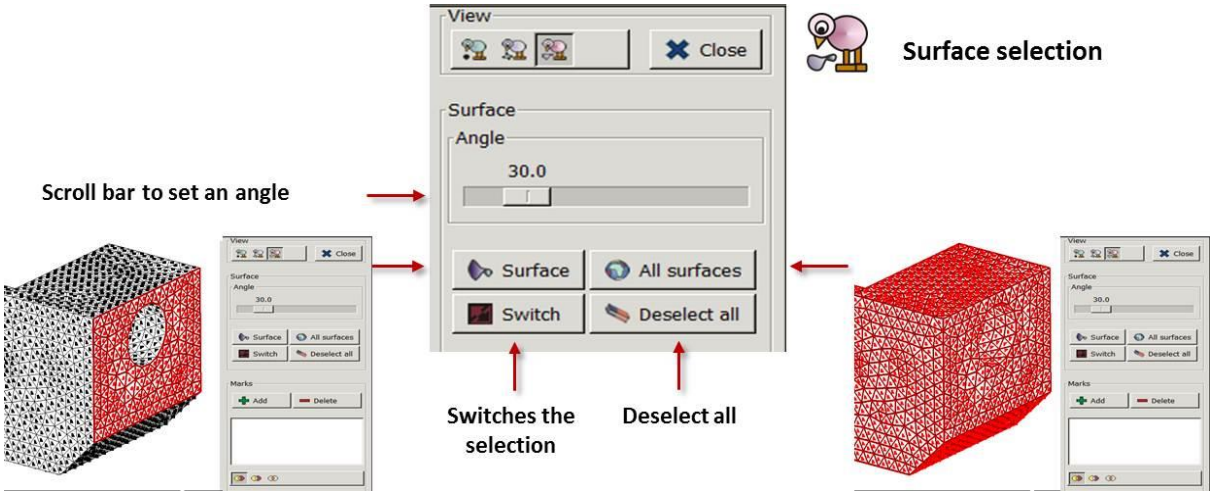


Figure 35: Picking of elements

The option “picking surface” works like the option “picking of nodes”. With both options you can select the surface of a building element. Pick with Strg+(click) a surface an select „surface“. „All surfaces“ selects the entire surface.

How to select the best picking option

Node selection	Element selection	Surface selection
Constraints	Improving tetrahedron mesh	Pressure (tetrahedron, Hexahedron) volume shells
	Assign a material	
	Locale meshing, Hexahedron mesh	

Set management

Every selection done by the picking menu can be saved as a marker. Simply click on “Add” at the marker you would like to save. These markers serve as starting basis for defining sets and later on applying boundary conditions, material assignments, refinements of tetrahedrons or displaying options. With Boolean operators several markers can be added or trimmed. When you achieve the desired result click on “Add Set”.

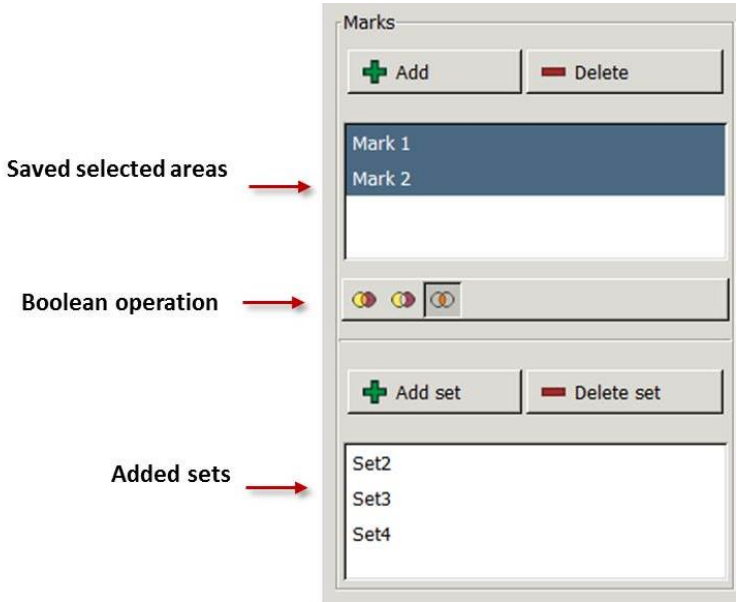


Figure 36: Markers and sets in the picking menu

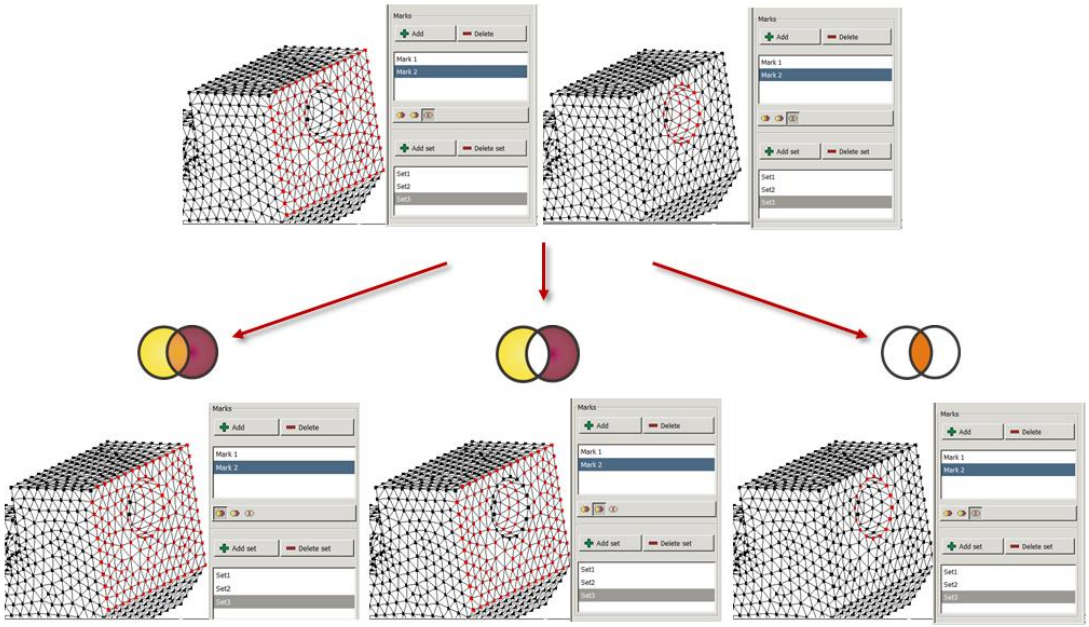


Figure 37: Boolean operation to create a set from a combination of different markers

Creating FE Structures: Trusses/Beams

Like in Z88 V15 OS, it is possible in Z88Aurora® to create and calculate truss and beam structures.

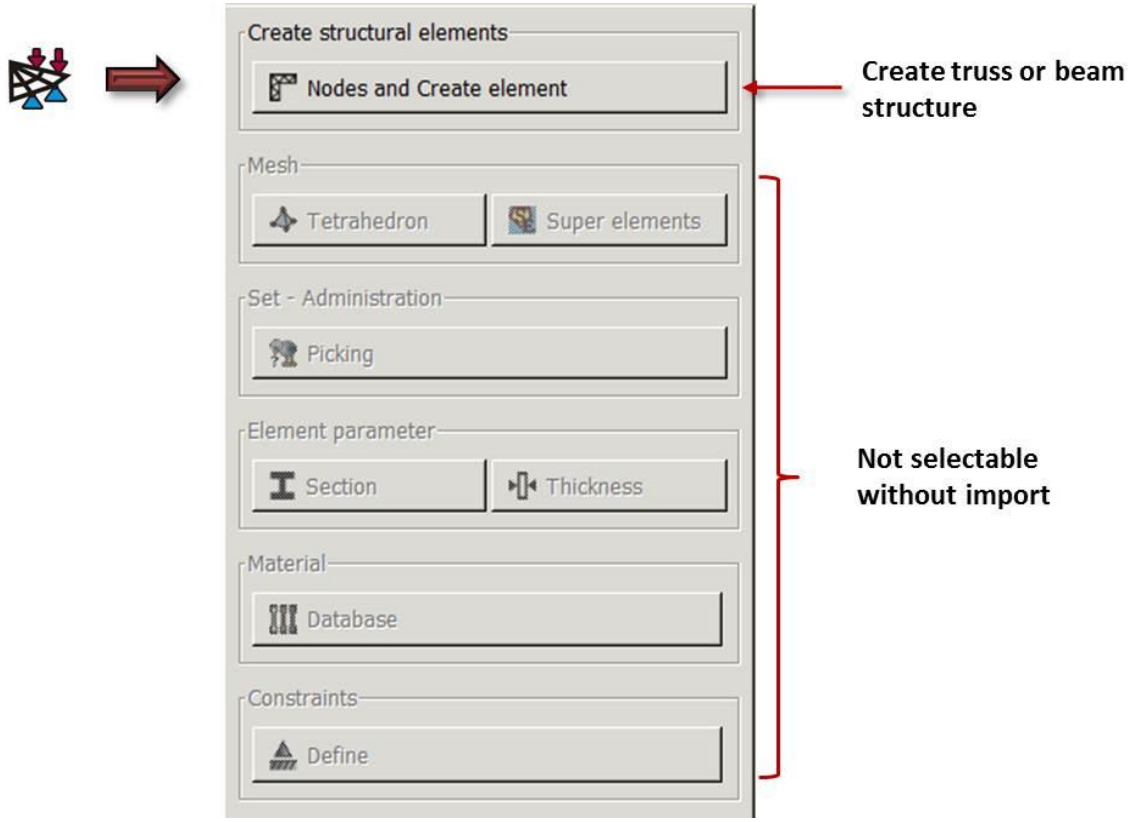


Figure 38: Creating structural elements

In the submenu "Creating nodes and elements" nodes are created by entering the coordinates, and then the element type is selected and created.

It is not possible to create mixed structures with different element types!

To do this, the use of Z88V15 OS is recommended.

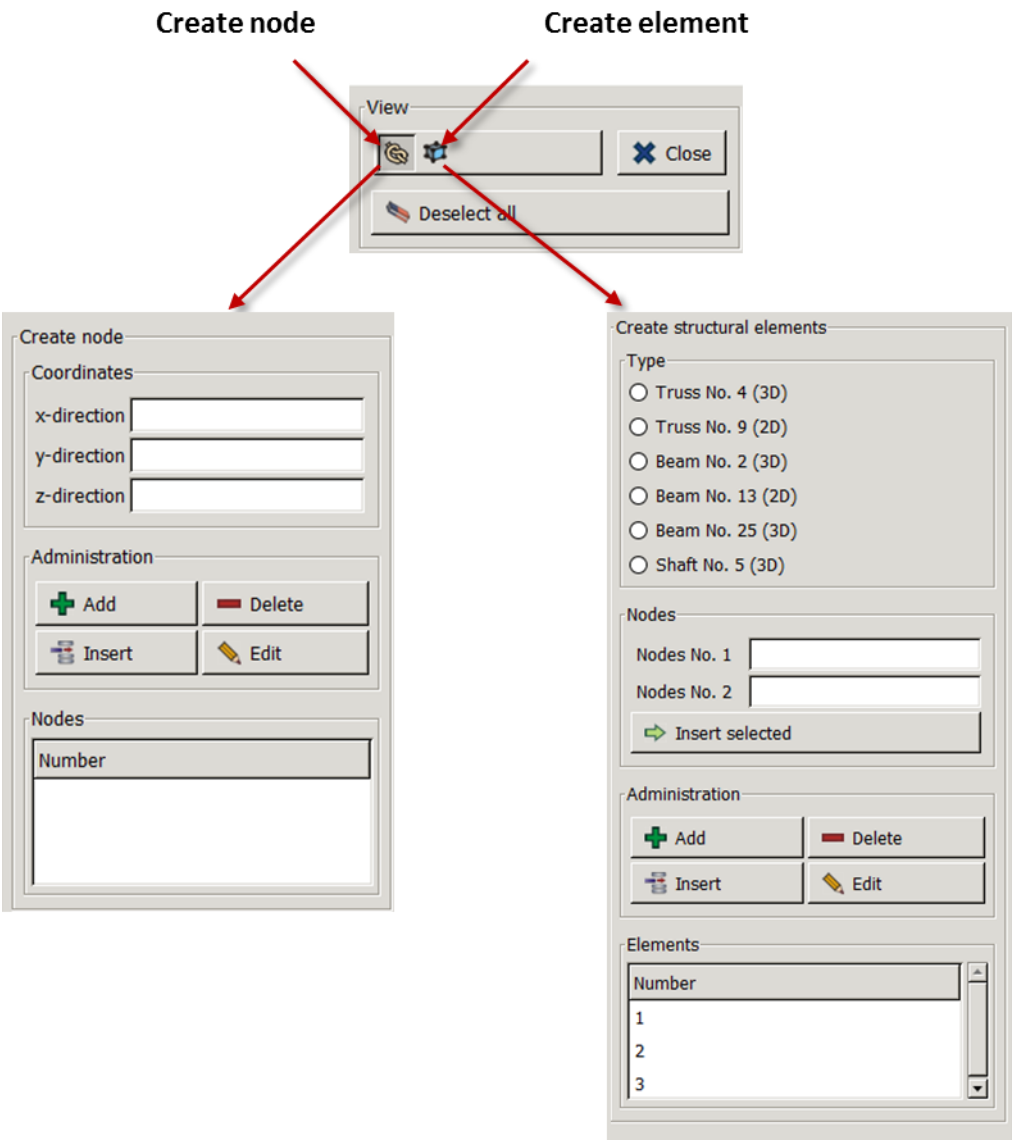
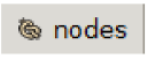


Figure 39: Creating structural elements



⇨ create new node coordinates

- enter "x“
- enter "y“
- enter "z“

⇨ click 

After entering the data, the nodes can be edited or deleted:

The selection of the nodes to be changed can be done via the selection from the list.

⇨with 

the node is selected for futher processing in the pop up menu “Edit”.

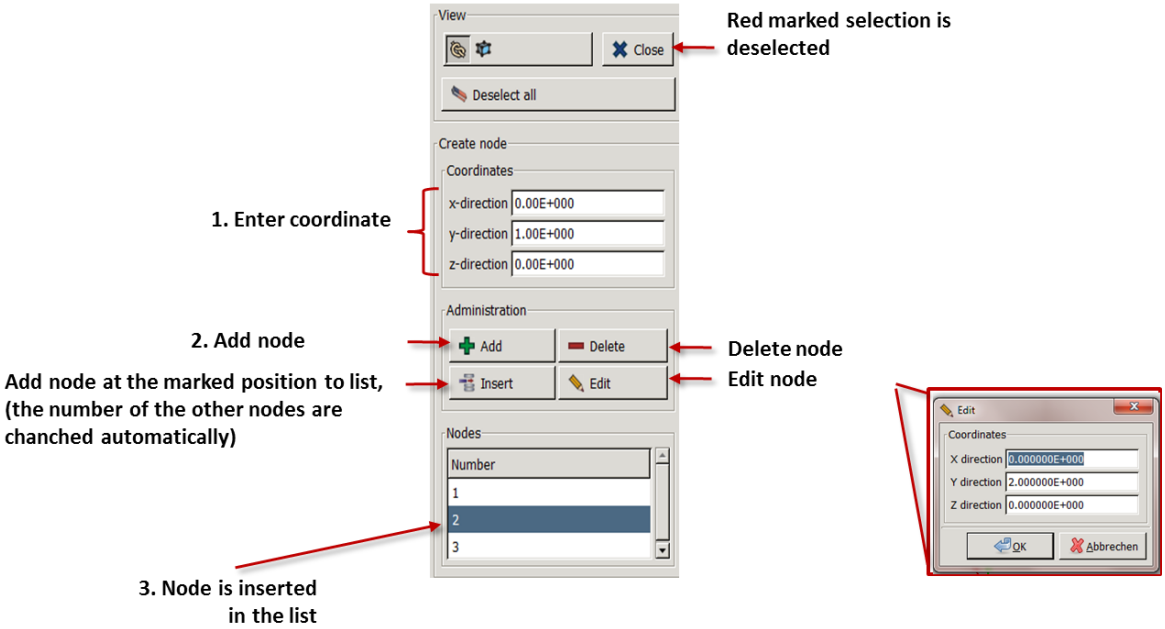


Figure 40: Node menu

For further information about the selection of nodes, see chapter "Picking"

Selecting from the list:

⇨ + select the node to be edited from the list → the node turns red

Afterwards, the selected node can be edited or deleted.

When all nodes have been created, the elements can be defined. For this, you have to switch to the menu. Elements

With “Add” a node can be inserted in the node list afterwards, the other nodes are renumbered automatically.

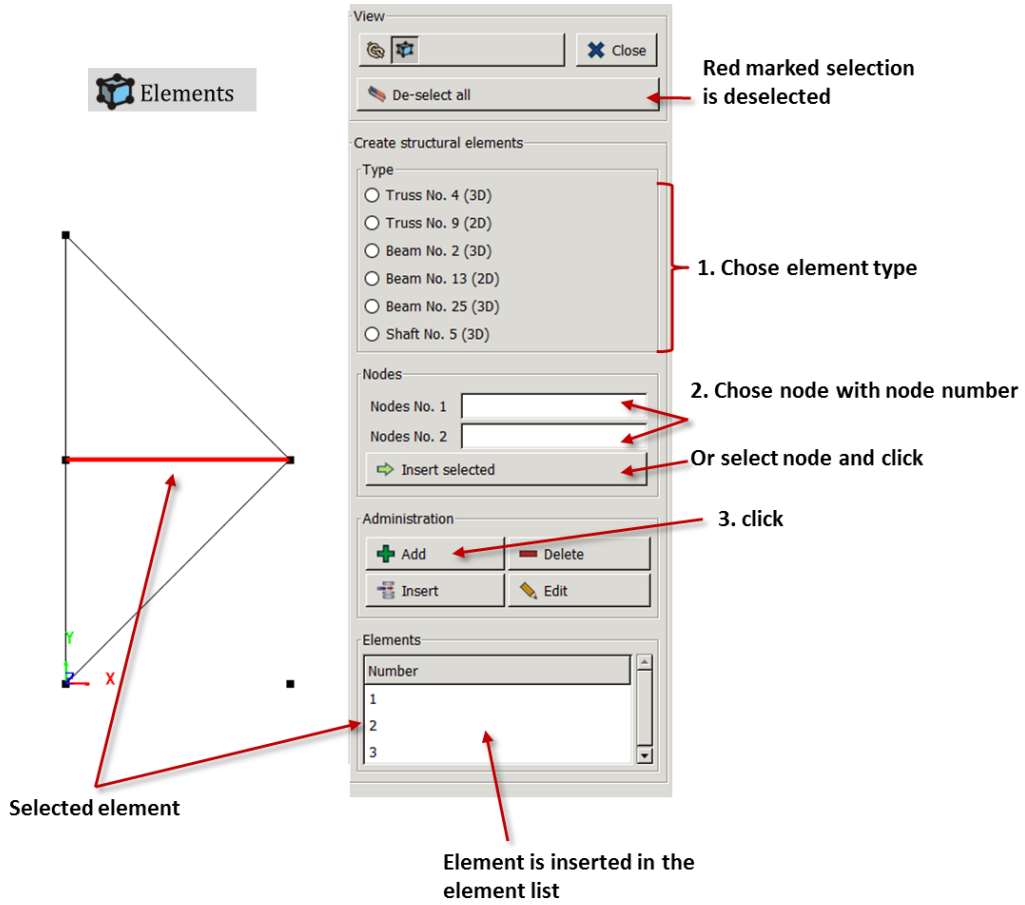


Figure 41: menu "Elements"

⇨ create new element

⇨ determine element type (Truss No. 4 / No. 9, Beam No. 2 / No. 13 / No. 25, Cam No. 5)

for further information please see the Theory Guide

- enter node 1 (by direct selection of the node via mouse+ Add selected)
- enter node 2 (or by entering the node number)

⇨ click add

After the elements have been entered, they can still be edited or deleted. The selection is done via the element table.

The compilation of the entry file is now completed. You can save the data and close the submenu.

In the next steps element parameters (geometry, cross section etc.), material and boundary conditions must be allocated. For this, please consult the help for “Element parameters”, “Material“ or “Constraints“.

Meshing

You have three possibilities of meshing parts in Z88Aurora®. On the one hand, a continuum can be meshed to miscellaneous FE structures with the mesh generator Z88N via the intermediate step of super element creation. On the other hand, two Open Source meshers, TetGen and NETGEN, for the creation of tetrahedron meshes, are integrated in Z88Aurora®. It is also possible to convert STL files directly into shell elements.

Creating a tetrahedron mesh

After import of geometry via *.STEP or *.STL, the part can be meshed by tetrahedrons. Two Open Source meshers are available:

- TetGen was developed by Dr. Hang Si of the research group "Numerical Mathematics and Scientific Computing" at the Weierstrass Institute for Applied Analysis and Stochastics in Berlin. In Z88Aurora® this mesher can be used for tetrahedrons with 4 or 10 nodes. In Z88Aurora® this mesher can be used for tetrahedrons with 4 or 10 nodes.
- NETGEN was mainly developed by Professor Joachim Schöberl (Institute of Analysis and Scientific Computing at the Vienna University of Technology, research group Computational Mathematics in Engineering) within the framework of the projects "Numerical and Symbolic Scientific Computing" and the Start Project "hp-FEM". In Z88Aurora® this mesher can be used for tetrahedrons with 4 nodes. In Z88Aurora® this mesher can also be used for tetrahedrons with 4 or 10 nodes.

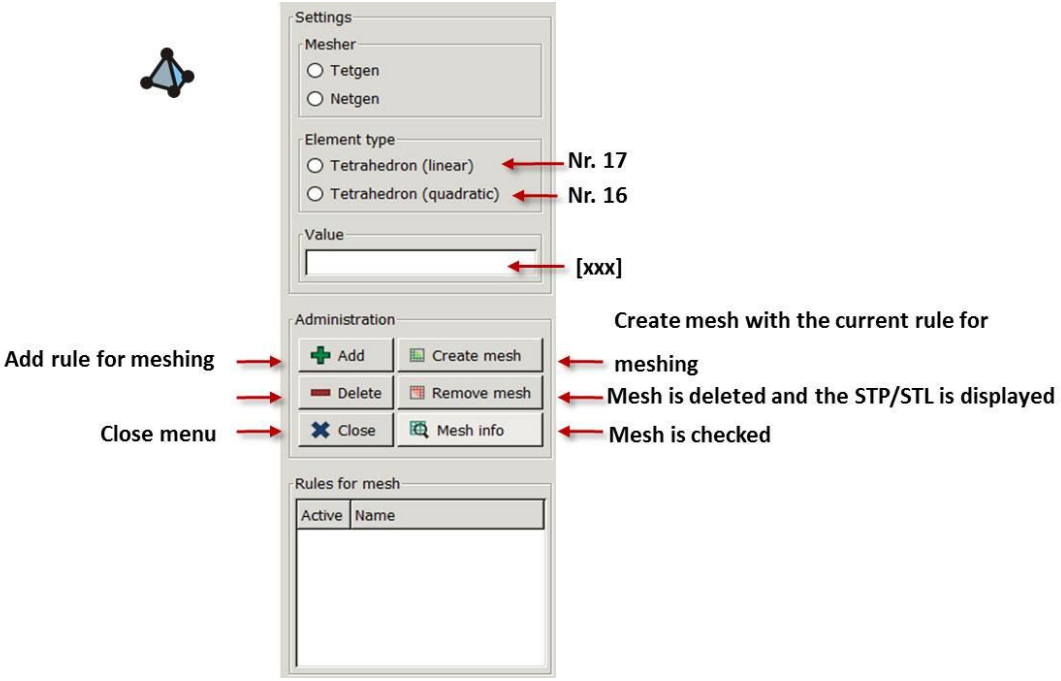


Figure 42: Creating of tetrahedron meshes and the options Tetgen/Netgen

⇨ Select TetGen or NETGEN

⇨ determine mesh parameter (accuracy of the mesh) and element type (this value correlates to the edge length in the respective unit of length)

⇨ click Add (the rule for meshing is created)

The data for the rule can be viewed any time.

⇨ afterwards either Create mesh or add a new rule

⇨ with Close leave the tetrahedron menu

⚠ **Depending on the selected mesher, the mesh creation may take some time. Please note the information window "meshing" and the status display! Chose the accuracy of the mesh adequately to the size of your component.**

Proof Mesh

As an additional function there is the feature "proof mesh" for the quality check of imported or self-created meshes. Please keep in mind that the results of the FE calculation are only plausible when you have a sufficiently good mesh. Therefore always conduct a quality check at the end of the meshing. If the mesh is faulty the flawed element is displayed in red (view option: “mesh”). Additionally the file z88det.txt is created in the project folder.

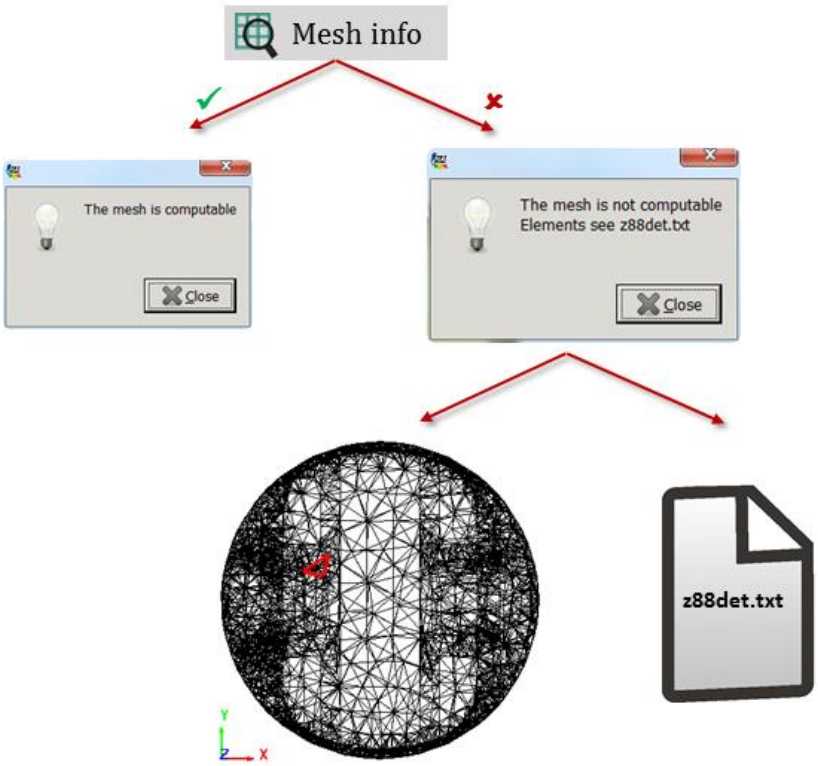


Figure 43: Prove mesh

Generating Super elements / mesh generator Z88N

The mesh generator Z88N from Z88 is integrated in Z88Aurora® with additional features:

- Z88N for hexahedrons, tori, plain stress elements, plate and volume shells
- Tetrahedron refiner
- Shell thickener → Volume shell
- STL refiner

These features are openend with the icon Super elements in the preprocessor menu

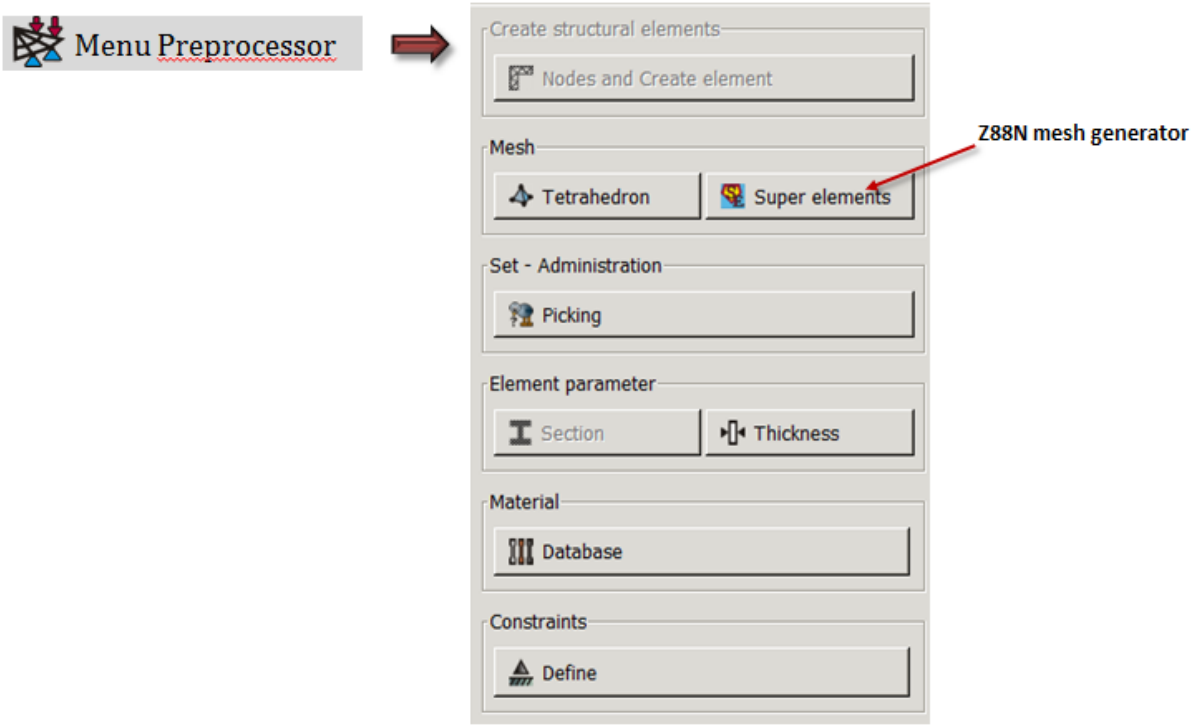


Figure 44: menu “Preprocessor” with the start icon “Super elements” of the mesh generator Z88N

Usage of Z88N in Z88Aurora®

The mesh generator can create 2 or 3 dimensional FE structures from super structures. Generating a mesh is only possible with continuum elements.

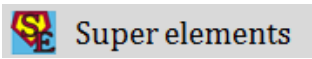
Table 2 offers an overview of the possible FE structures.

Table 2: Overview of possible super structures in Z88Aurora®

Super structure	Finite Element Structure
Plane Stress Element No. 7	Plane Stress Element No. 7
Torus No. 8	Torus No. 8
Plane Stress Element No. 11	Plane Stress Element No. 7
Torus No. 12	Torus No. 8
Hexahedron No. 10	Hexahedron No. 10
Hexahedron No. 10	Hexahedron No. 1
Hexahedron No. 1	Hexahedron No. 1
Plate No. 20	Plate No. 20
Plate No. 20	Plate No. 19
Shell No. 21	Shell No. 21

In all spatial directions a super structure can be refined uniformly, in ascending or descending order. For this element sets must be created, the rules for meshing must be defined and afterwards meshed.

For example:

⇨Define 3 element sets, change to 

⇨Define element type

⇨local x-direction: uniform, ascending or descending fragmentation

⇨local y-direction: uniform, ascending or descending fragmentation

⇨local z-direction: uniform, ascending or descending fragmentation

⇨ click on Add (the rule for meshing is created)

The data for the rule can be displayed any time.

⇨ either Create mesh or create another rule; only one rule can be defined per set

⇨ leave the menu with Close

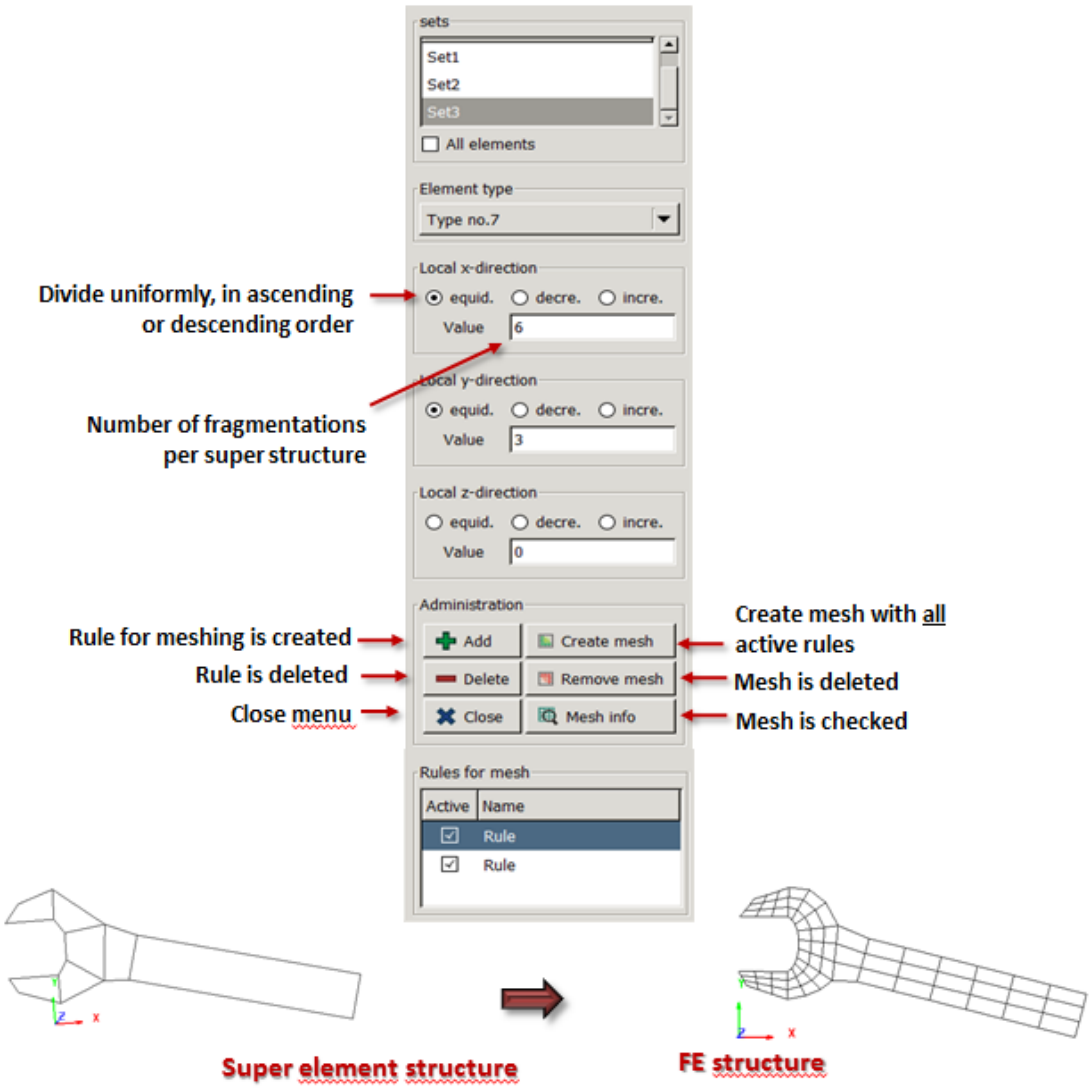


Figure 45: menu “Super elements” of the mesh generator Z88N

⚠ **After creating the mesh, the rules for meshing are deleted!**

Refining a tetrahedron mesh

With this option it is possible to improve already existing tetrahedron meshes. With “picking” a set can be created with the tetrahedron that should be refined. Each element is divided into 8 tetrahedrons.

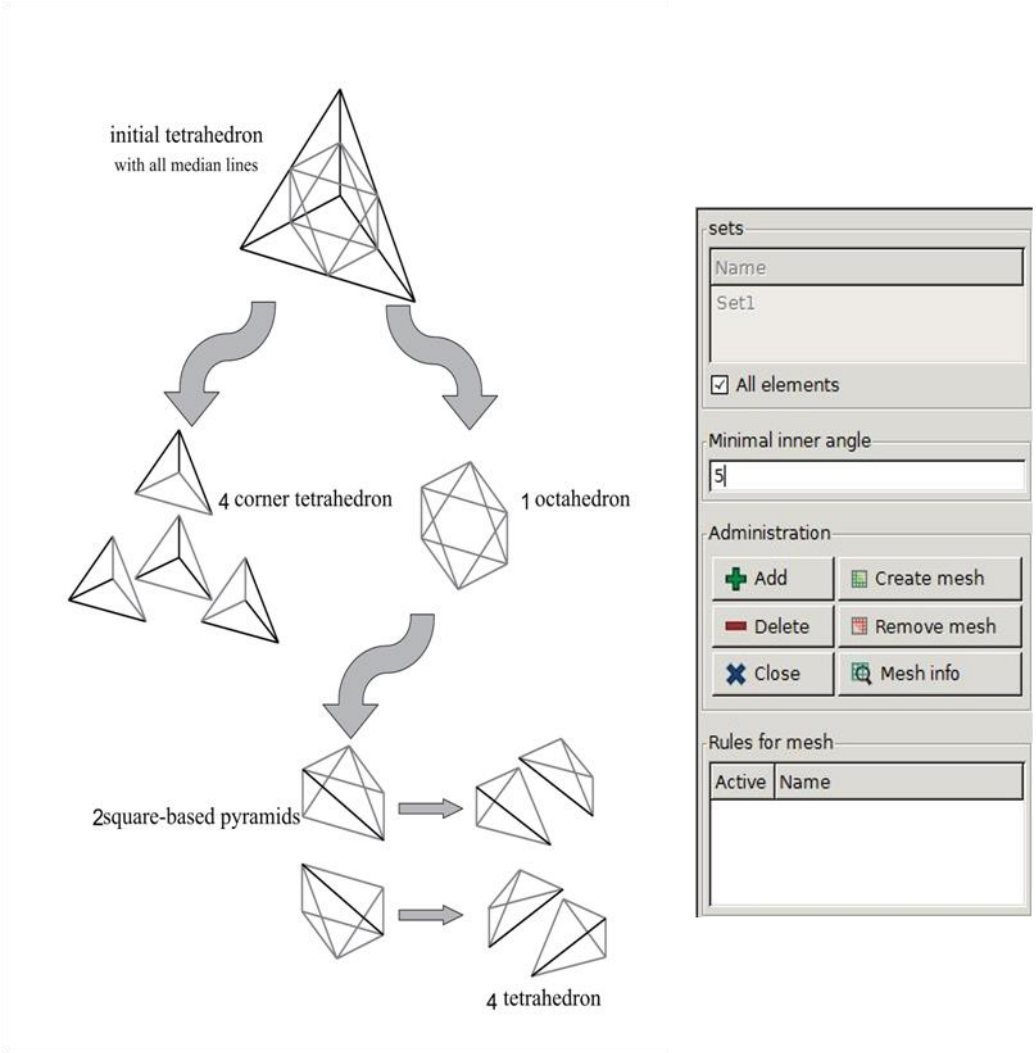


Figure 46: refining of a tetrahedron: input mask (right), process (left)

The adjacent elements are adjusted to the changed node numbers and are divided as well. For this a minimum element angle must be specified to prevent a large deformation. Instead of the ideal interior angle of 60°, an angle of 3°-10° is realistic.

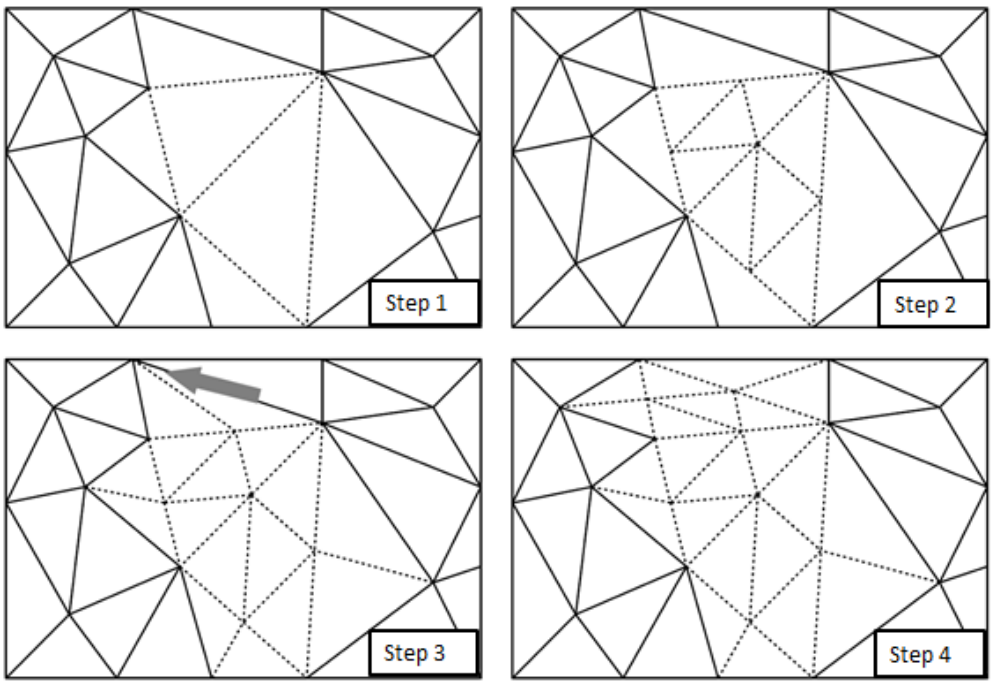


Figure 47: Process of the algorithm for refinement

Shell thickener

With this feature it is possible to thicken already existing shells, which have nodes in only one plane (e.g. from Nastran or DXF import), and thus creating volume shells (element No. 21 and No. 22)

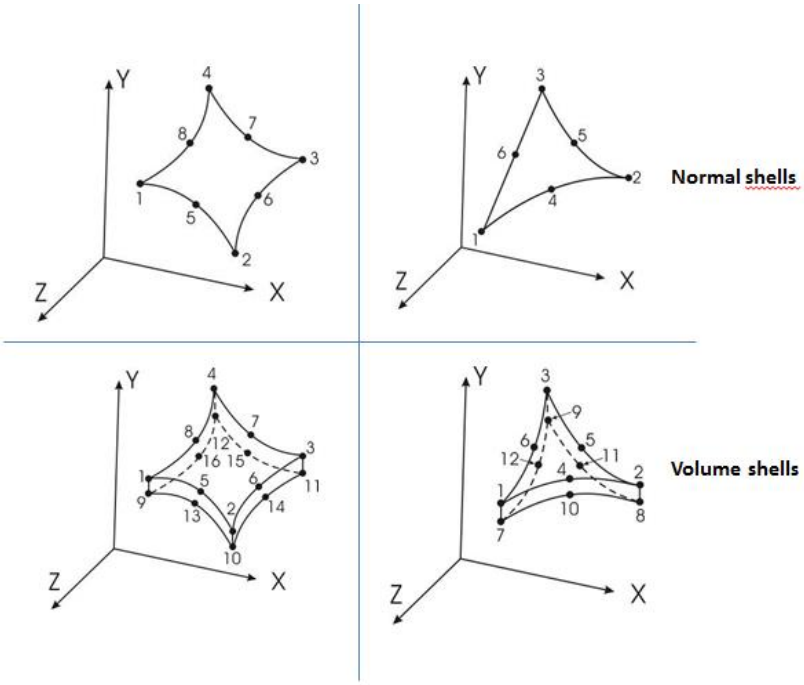


Figure 48: normal shells (above) and volume shells (below)

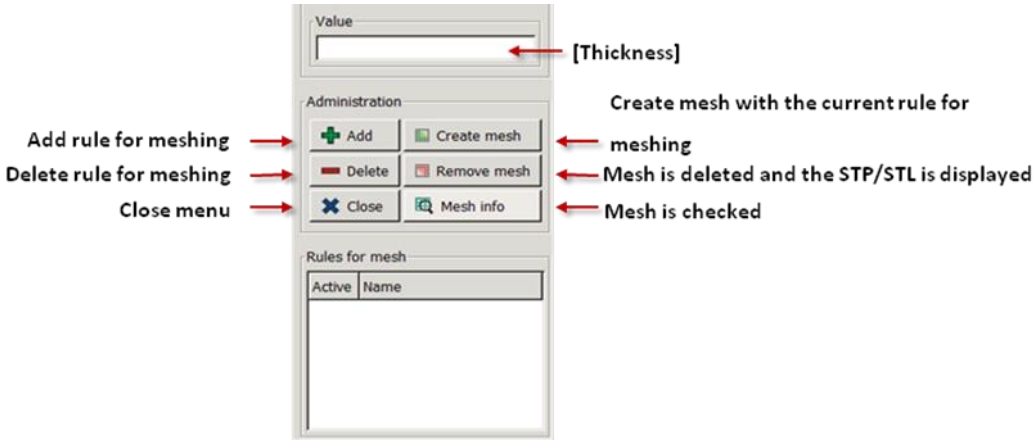


Figure 49: Shell thickener in Z88Aurora®

More information see "Creating a tetrahedron mesh"

Element parameters

You can allocate element parameters for the element types plate, shell, truss, beam and cam element and plain stress element.

According to the element types following icons are selectable:

: beam, truss, cam

: plate, plain stress element, shell

beam, truss, cam

If you have created these structures beforehand, they can be edited here.

Element parameters of imported Z88 files (Z88V15, Z88V14, Z88V13, Z88Aurora® V1) must be created anew, because they cannot be imported with the structure!

Depending on what element type you have chosen, you can assign the respective geometry date. You can assign one geometry to all elements All elements **or you can define sections (from/to element) and assign one geometry per section.**

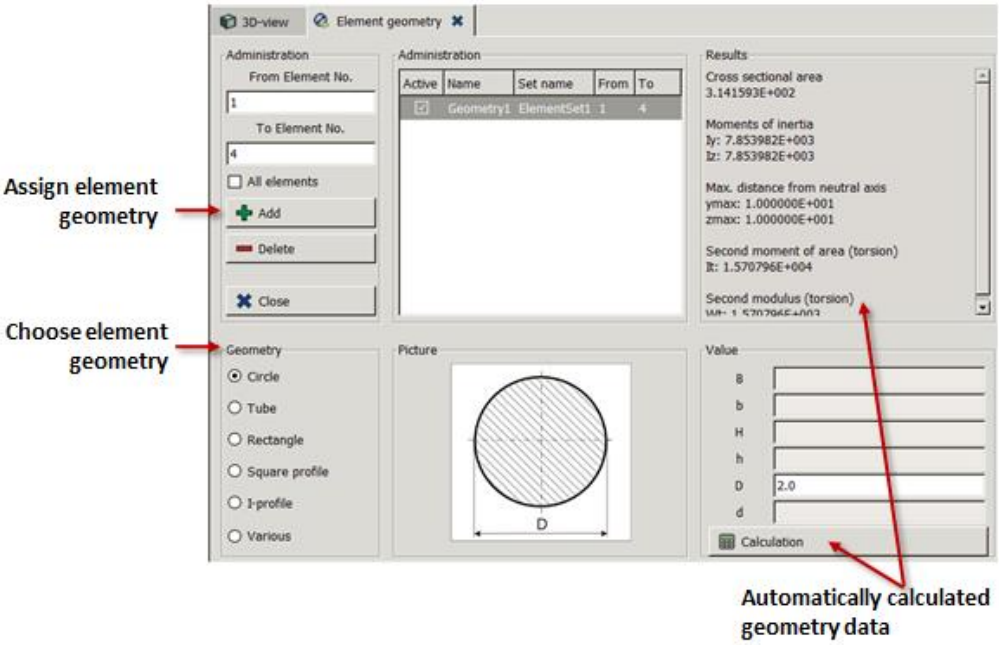
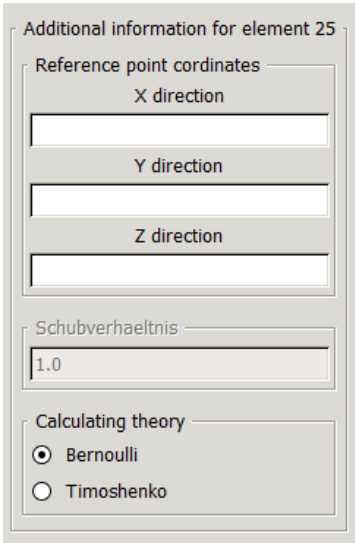


Figure 50: Assigning cross section in the menu “element parameter”

When using the element type 25, another menu is available, which need special settings for the control node can be made. For details, see theory manual.



The element parameters can be inserted by hand. Additionally Z88Aurora® V5 can calculate the element geometry of a circle, tube, rectangle, square profile or I-profile.

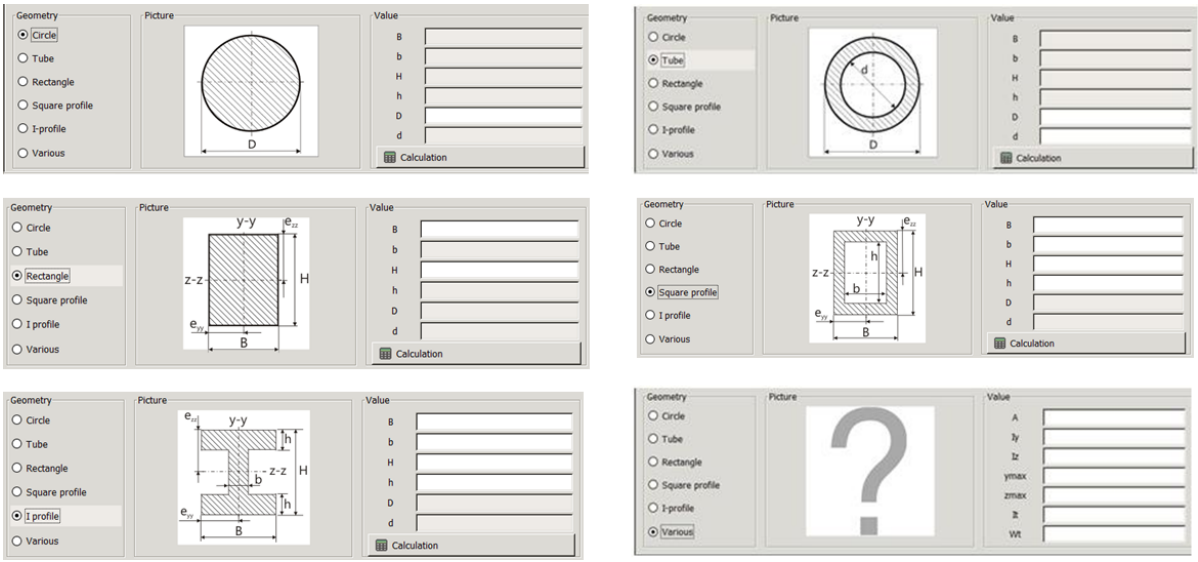


Figure 51: Cross sections that can be automatically calculated

To do this:

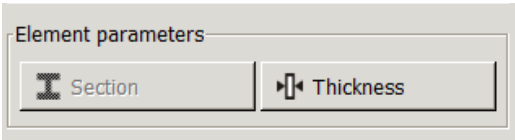
⇨ select element geometry

⇨ insert entry parameters (depending on the selected element type only the required data is used for the calculation)

⇨ Calculation

With Add the element parameters are assigned to the structural elements.

Plate, plain stress element, shell



You can assign one geometry to all elements All elements or you can define sections (from/to element) and assign one geometry per section.

⇨ Add

⇨edit thickness with double clicking

⇨ Close

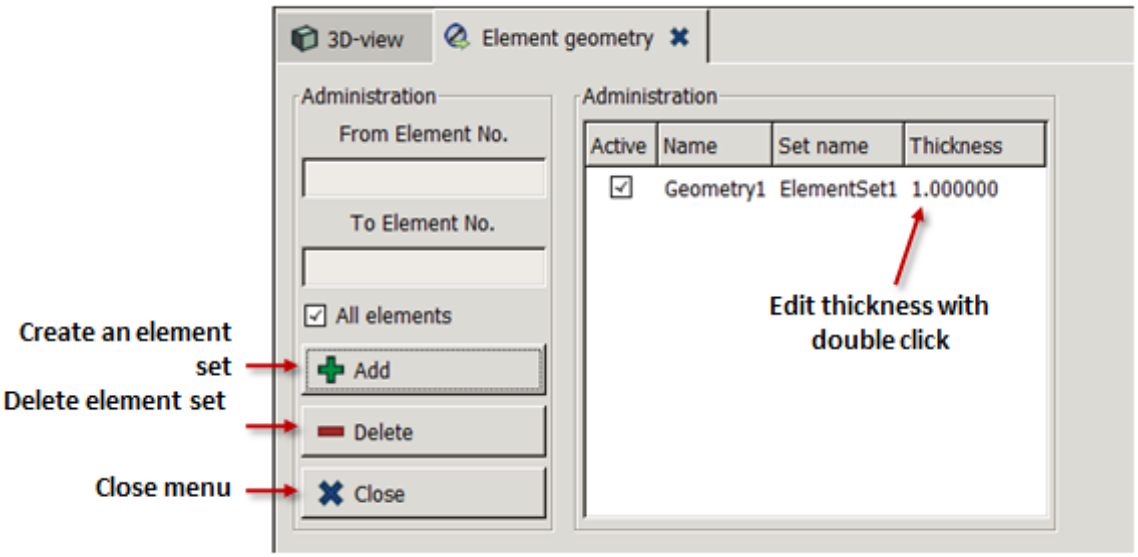


Figure 52: Assign thickness to plain stress elements, plates and shells

Assembly management / Contact analysis

A detailed workflow for carrying out a contact analysis is shown in Figure 53. The import of assemblies directly from a CAD program is not possible.

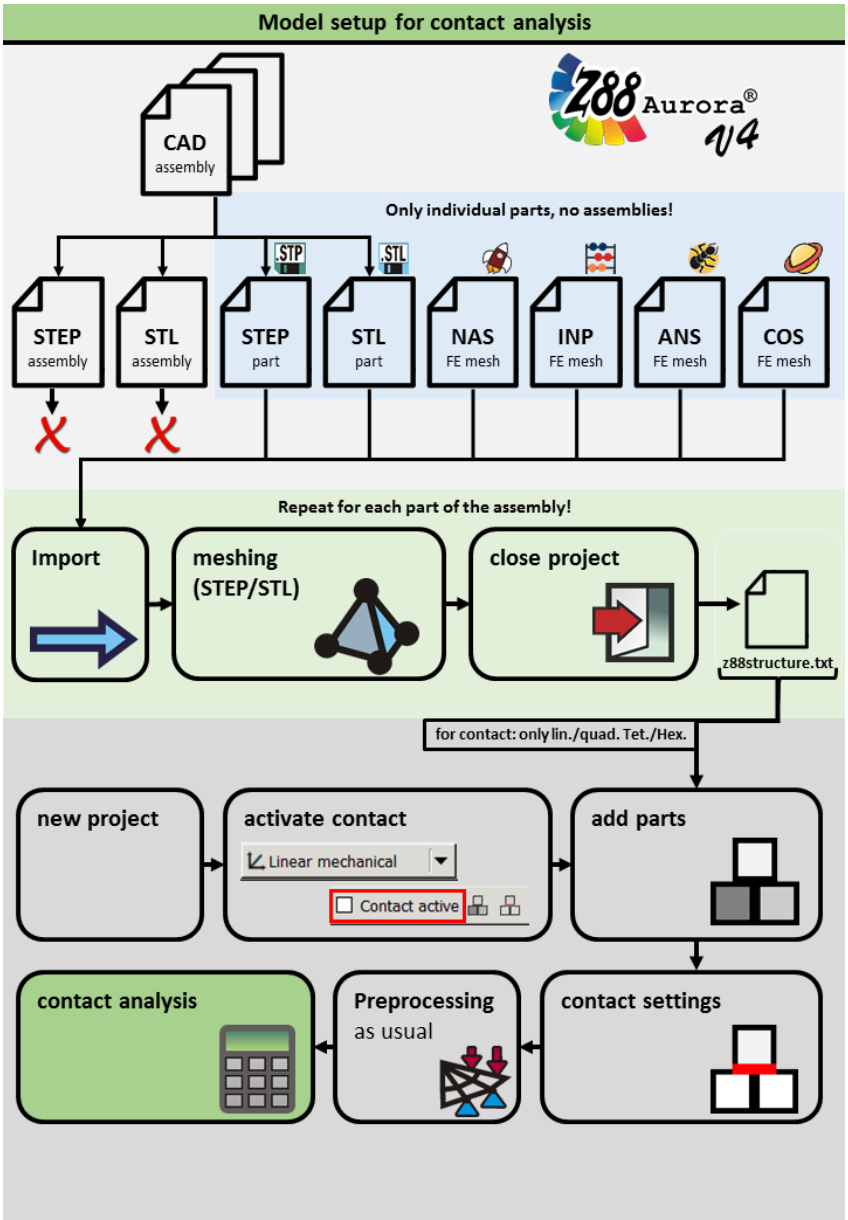


Figure 53: Setup of a contact analysis

To carry out a contact analysis in Z88Aurora® a new project has to be created. In this project the contact module has to be activated (checkbox “Contact active”) and multiple parts have to be imported. These parts have to be in the Z88 structure format (z88i1.txt or z88structure.txt). If the parts have not already been meshed and are only available as either a *.STEP or *.STL, they have to be imported into individual Z88Aurora® projects and meshed. The resulting structure file can then be found in the project folder. The element type used

for all parts has to be the same. The element types available for a contact analysis are linear tetrahedron (no. 17), quadratic tetrahedron (no. 16), linear hexahedron (no. 1) and quadratic hexahedron (no. 10). Note that the checkbox "Contact active" 

is only available for a linear mechanical analysis. Once the contact module has been activated, the part manager can be activated via the corresponding button next to the contact checkbox. The part management menu is shown in Figure 54.

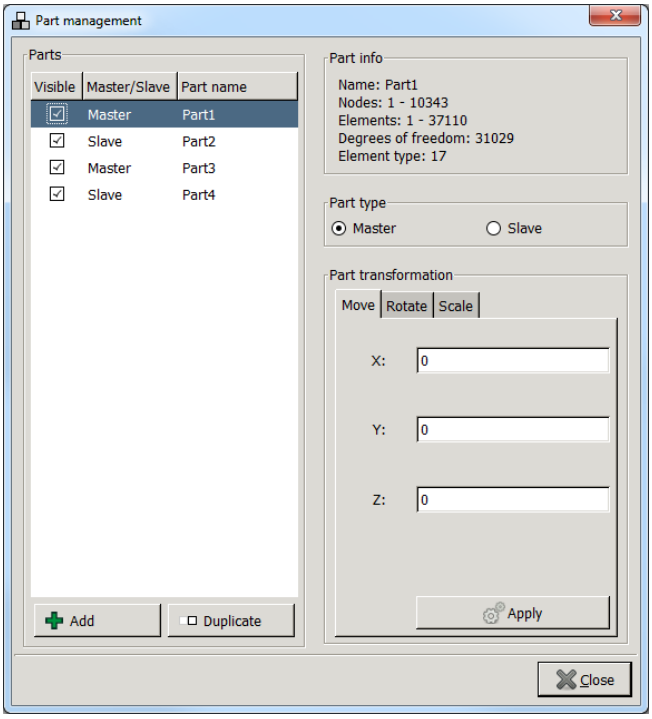


Figure 54: Part management

An arbitrary number of parts can be added to the assembly via the "Add" button as long as they are available as z88i1.txt or z88structure.txt. Additionally, a number of part transformations are available:

- Move: Part can be moved in x, y and z direction
- Rotate: Part can be rotated around the x, y or z-axis or a user-defined axis
- Scale: Part can be scaled by a specific factor in x, y or z direction

The defined transformation can be applied to the selected part via the "Apply" button.

Every part can be transformed in a different way, making the creation of an assembly consisting of different parts possible. Using the part management menu, imported parts can be duplicated.

Via the part management menu each part has to be assigned a role, i.e. master or slave. A contact can only exist between a master and a slave part. The contact search fails between two master or two slave parts. Special care has to be taken when assigning the roles.

The following procedure is recommended for simulating assemblies that have been created in a CAD system:

- Design the assembly in CAD.
- Export the assembly as individual parts as STL or STEP files.
- Import each part as a new project and mesh it and close the project.
- Create a new project, activate the “Contact active” checkbox and import all meshed parts.
- Now the assembly should already be scaled and positioned correctly, only the contact roles have to be assigned.

After the part management, the contact settings have to be assigned, see Figure 55.

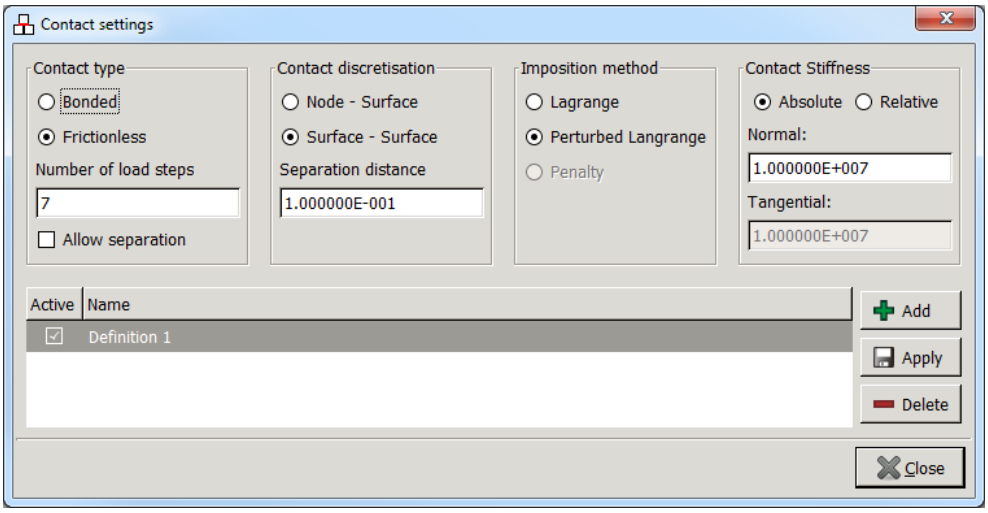


Figure 55: Contact settings

Various settings regarding the contact type (bonded or frictionless), contact discretization (node-surface, surface-surface), integration method (Lagrange, perturbed Lagrange, penalty) and the contact stiffness can be made here. The theoretical background and the effects of the different parameters are described in the theory handbook.

The chosen settings can be saved as a contact definition via the “Add” button. To change values in an existing definition, change them and save them via the “Apply” button. A contact definition can be activated via the “Active” checkbox. Only one contact definition can be active at a time, which then is applied to the whole assembly.

Warning! For each calculation the contact search is only carried out once within the defined contact distance. If no contact is found at the start of the calculation, it will not be updated during the course of the simulation - a so-called linear contact without coordinate update. Defining a wrong contact distance or role may result in a statically indeterminate structure or parts penetrating themselves.

Material

In order to carry out static strength analyses, natural frequency analyses and thermic calculations the present version of Z88Aurora® offers a material database containing more than 50 established construction materials.

Z88Aurora® Material Database

The Z88Aurora® material database is selected in the preprocessor menu ( ) via the button (or via "Preprocessor" → “Material Database”). To facilitate your work with Z88Aurora®, several materials, such as miscellaneous types of steel and aluminum, have already been predefined.

If you select a material from the list on the left, its allocated properties can be viewed in “Details” (Figure 56). If a material should be edited, a copy of the material is created so that the internal data base remains consistent. The flow curves are **not** copied by non-linear material. Already existing materials from other projects can also be imported.

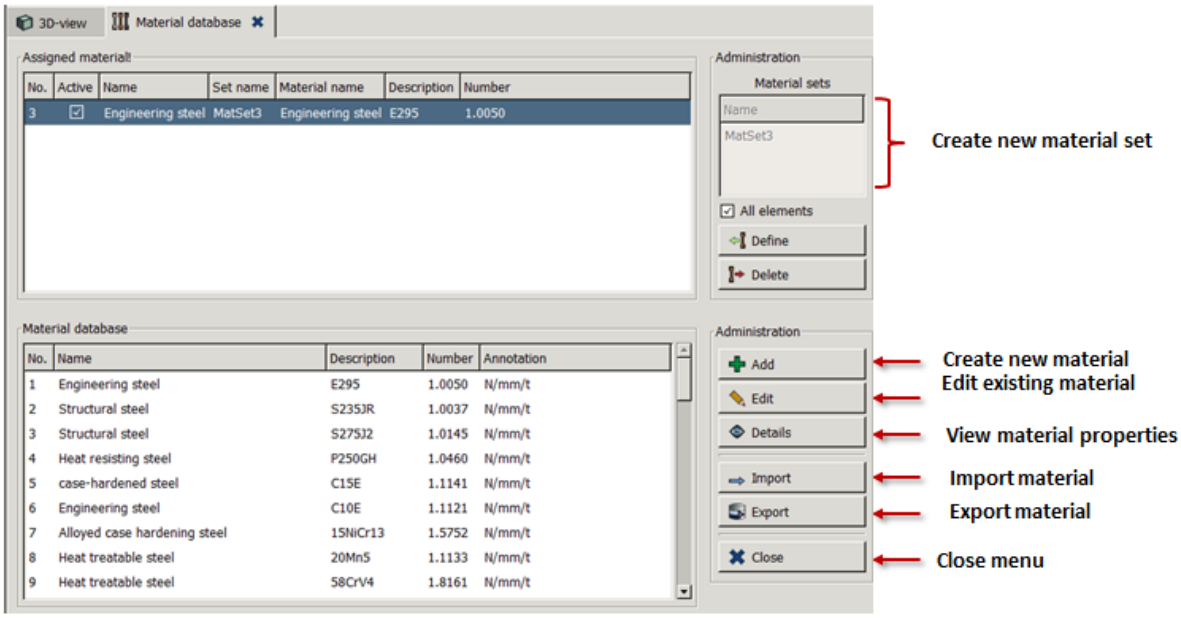


Figure 56: Z88Aurora® Material database

If the required material is not included, you have the possibility to define new materials in the database. For this, click add in the right menu and the context menu "Material Parameters" is opened (Figure 57). In the first input array you can define the material type by means of "Material Name", "Identifier" and "Material Number". In the second input array

the material properties, such as Young's Modulus, Poisson's ratio and density (⚠ **Unit density: t/mm³**) are entered for the linear analysis.

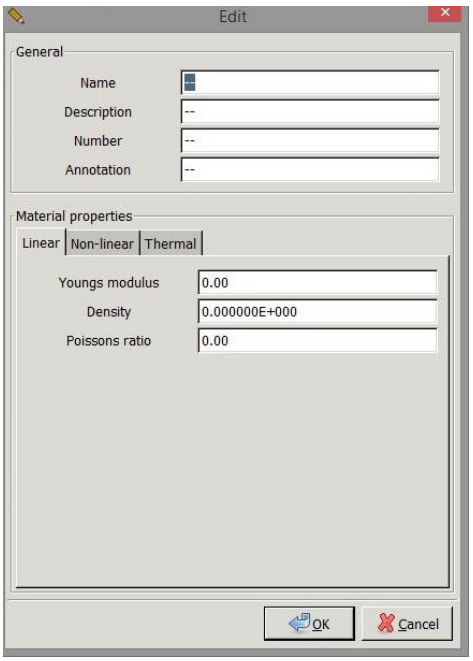


Figure 57: Context menu material parameters

In the case of unalloyed construction steel (according to DIN EN 10025-2) this would look as follows:

- Name: construction steel (common name)
- Description: S235JR
- Number: 1.0038
- Annotation: own annotations e.g. delivery
- Young's Modulus: 210000 N/mm²
- Density: 7.85 E-9 t/mm³
- Poisson's Ratio: 0.29

For thermal analysis you have to assign heat conductivity and heat expansion (tab “Thermal”, cp. figure below).

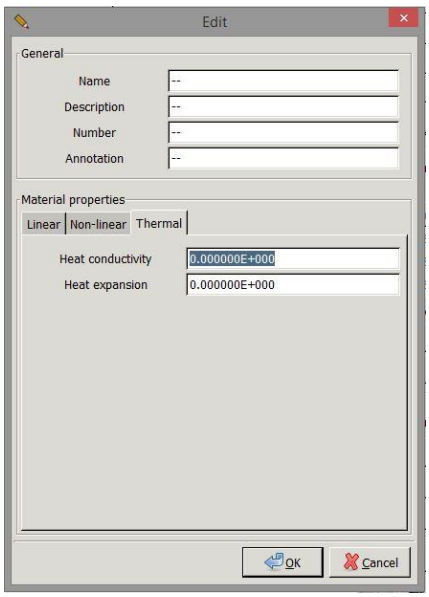


Figure 58: Input for thermal analyses

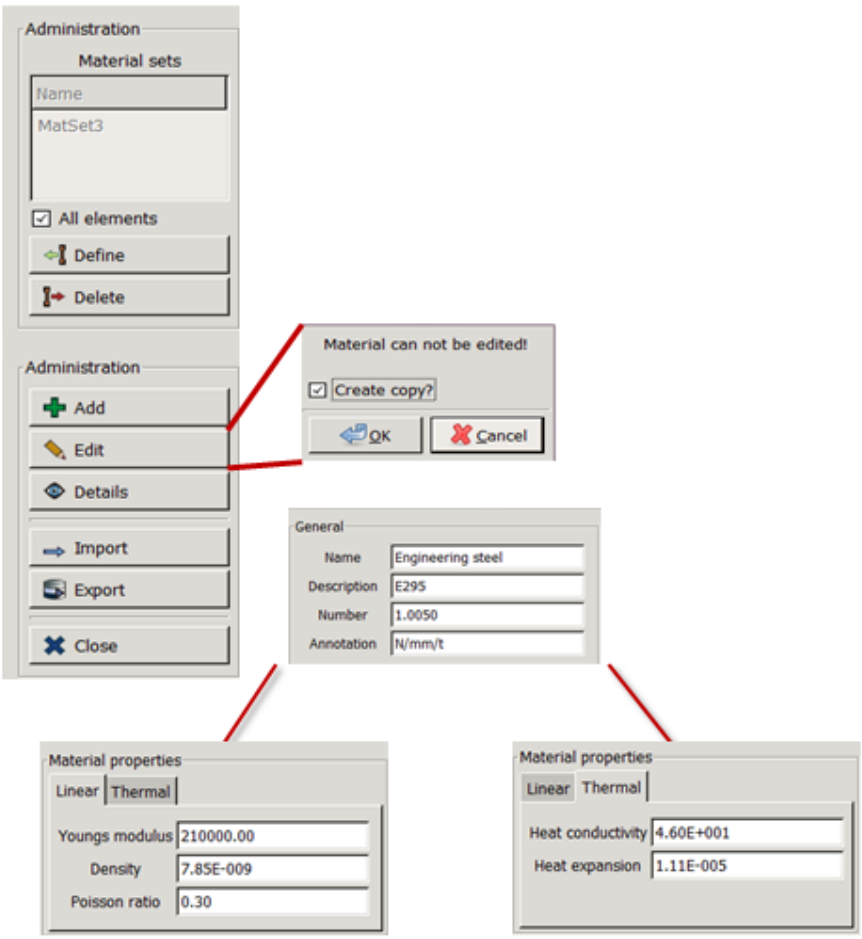


Figure 59: Context menu material parameters II

Only the material properties required by the respective type of analysis have to be entered.

Please note that you have to enter a dot as decimal point and that the material name must be unique (e.g. "construction steel1“, "construction steel2“, etc.).

With the pushbutton edit you can edit already entered materials. A copy of the material is created so that the data base remains consistent.

With Define the material is added to the structure, with Delete it is deleted.

If you want to apply one material to the whole structure, set a check mark at “All elements”. Otherwise you can apply different materials to different elements sets, e.g. to create a bimetal. With Close the data base is saved and the tab is closed.

Materialmodels

In nonlinear analyses different material laws can be chosen to consider elastic or plastic material behavior.

Table 3: Overview of material models

	material behavior	Young´s-Modulus	Poisson Ratio	Flow curve	Further parameter
Hooke	linear-elastic	✓	✓	×	×
Von Mises	elastic-plastic	✓	✓	✓	×
Wehmann	elastic-plastic, different poisson ratio in plastic sector	✓	✓	✓	• flow direction parameter *w* <u>or</u> • course of the flow direction parameter f(*w*)

Therefore, the table with the assigned materials contains an additional column in the module nonlinear strength. *Figure 60* shows this column, in which the favored material law can be selected. This selection has to be confirmed with the "ENTER" button so that it is adopted. In case of Hooke's law, linear-elastic material behavior is present, geometric nonlinearity alone will be taken into account. Regarding the material parameters, only Young´s modulus

and Poisson's ratio are necessary in this case. These two parameters are specified under the “Linear” tab (see. Figure 57).

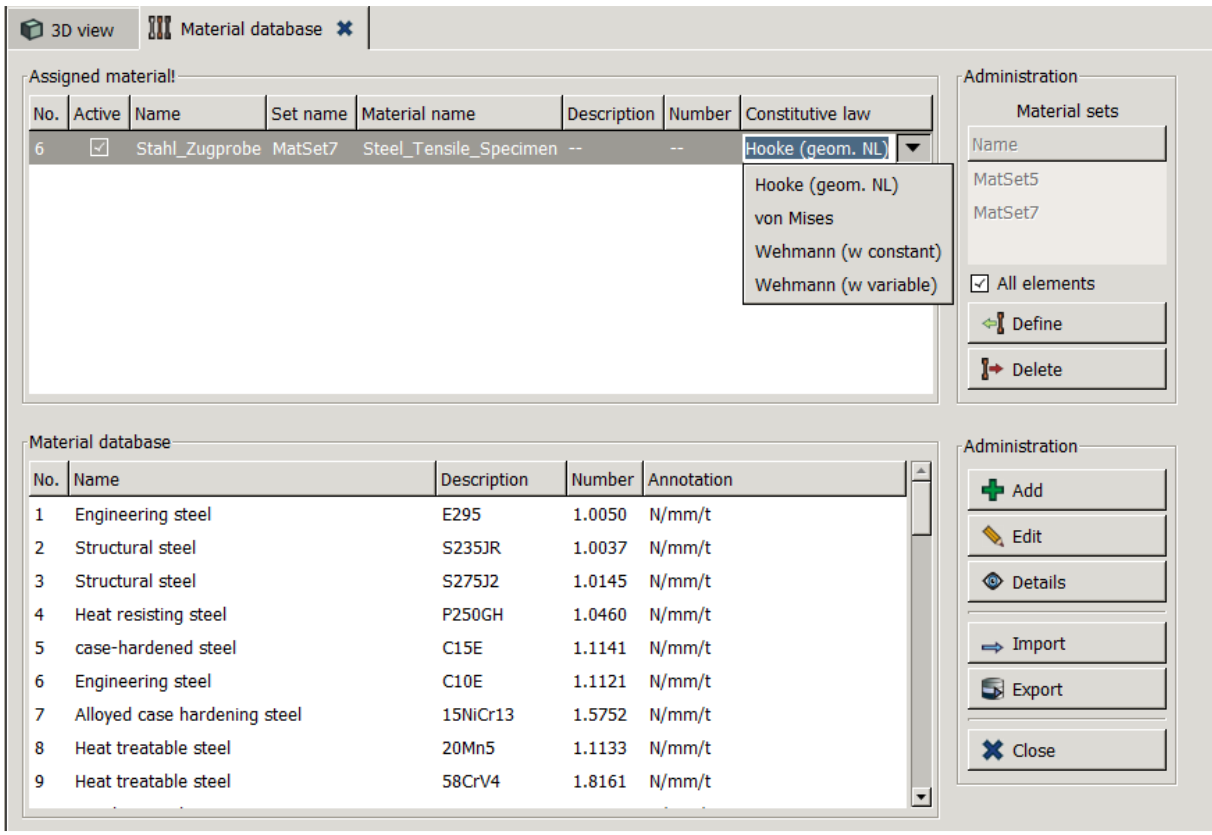


Figure 60: Choice of the material law for nonlinear analyses

By selecting one of the other laws, plastic material behavior is present and additional material data must be specified. In these cases, no geometrical non-linearities are taken into account. Necessary additional parameters for the plastic material models must be specified under the “Non-linear” tab.

Figure 61 shows which parameters must be entered for the "von Mises" law. You have to enter the flow curve, which is defined by pairs of values of the plastic elongation and yield stress. By clicking the button “Add” you can define a new pair of values, the values can always be selected and edited by double-clicking. With the “Delete” button the selected value pair is deleted. Also, importing a text file with all pairs of values is possible.

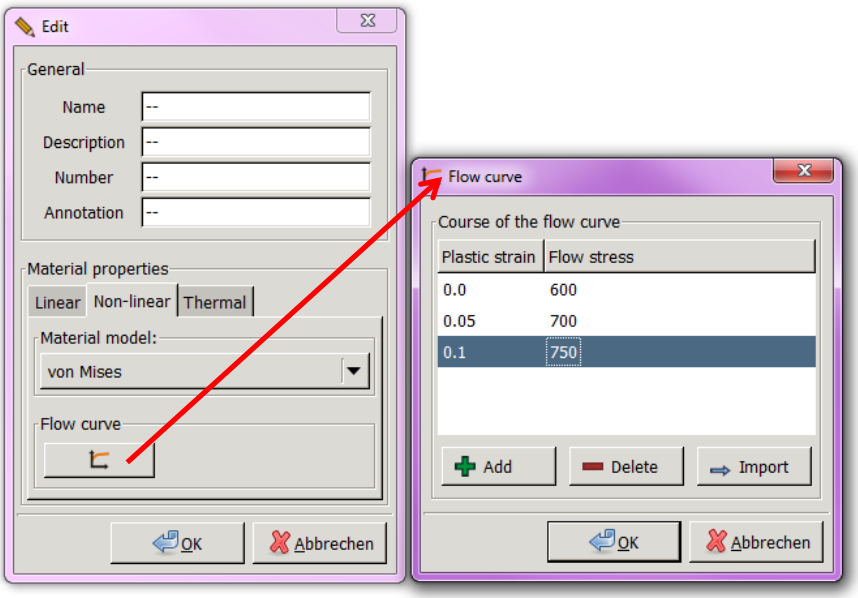


Figure 61: Input of the material data for the material law "von Mises"

Figure 62 shows the structure of a sample file. After its import, the values result in the way shown in *figure 61*. It is important that the first line of the file contains the number of pairs of values.

```
3
0.0     600.0
0.05    700.0
0.10    750.0
```

Figure 62: Structure of a txt-file for the import as yield curve

The unit for the plastic strain is always 1 (dimensionless), i.e. 0.05 corresponds to 5% plastic strain. The flow curve can be determined by the following two equations from the Stress-strain curve (σ-ε diagram).

$$k_f = \sigma$$

$$\varepsilon_{pl} = \varepsilon - \varepsilon_{el} = \varepsilon - \frac{\sigma}{E}$$

Thereby, k_f is the yield stress and ε the total strain. Figure 64 shows an exemplary stress-strain diagram and the corresponding flow curve.

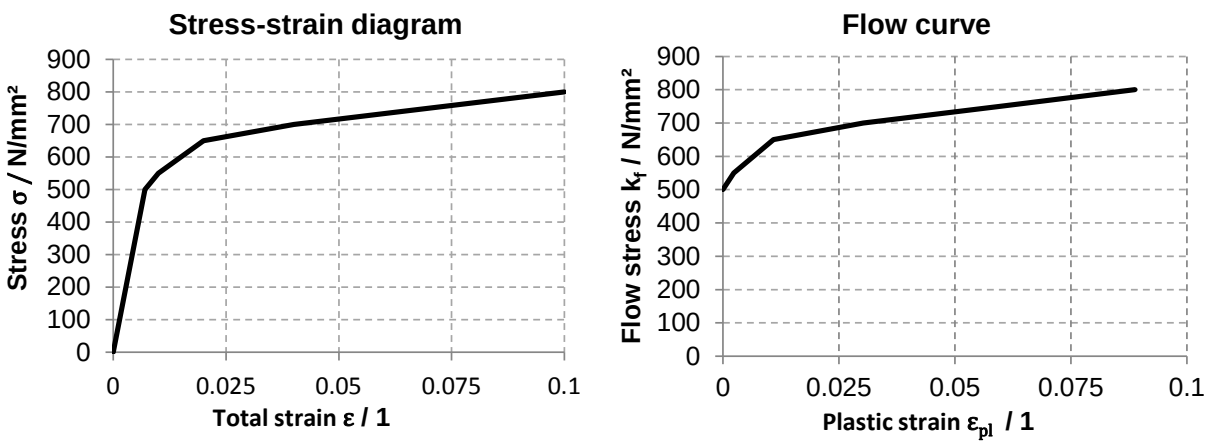


Figure 63: Stress-strain diagram and the corresponding flow curve

In case of the Wehmann model, the selection must be switched to " Wehmann (w constant)" as shown in figure 64 on the left. This model allows to adjust the transverse contraction in the plastic range decoupled, with the help of an additional material parameter. If you, for example, notice that the von Mises-law does not describe the transverse contraction correctly, you can achieve an improved description with the help of the Wehmann model. For Wehmann model, as for von Mises, a yield curve must be specified.

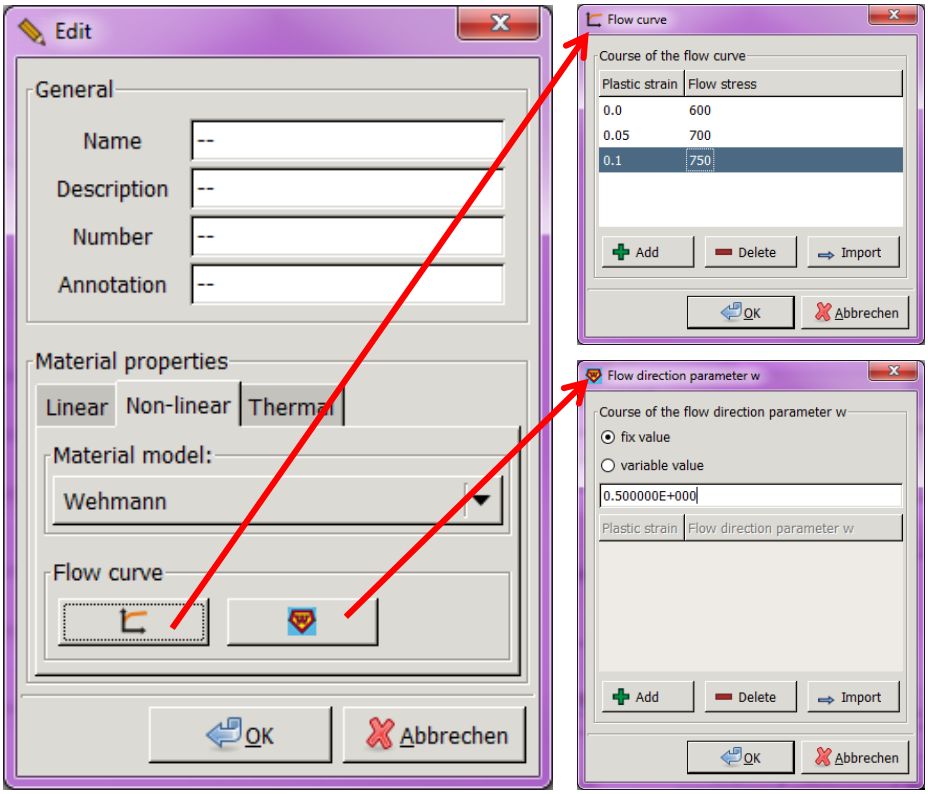


Figure 64: Input of the material data for the material law „Wehmann model"

In addition, the flow direction parameter w has to be specified, which can assume values between 0 and 1 (see. figure 64 right). The parameter can be determined in a tensile test, by including the measurement of the transverse contraction. Using a circular sample having a diameter D_0 the cross-sectional area A_0 and length L results in the following determinative equation.

$$w = -2\,\frac{\frac{\Delta D}{D_0} + \frac{\nu\,F}{E\,A_0}}{\frac{\Delta l}{L} - \frac{F}{E\,A_0}}$$

ΔD and Δl are the change of diameter and the change of length. Note that $\Delta D = D - D_0 < 0$ and $\Delta l = l - L > 0$ are applied. With E and ν Young's modulus (elastic) Poisson's ratio are identified. F is the force in the tensile test. Further information on Wehmann model can be found in [Wehm14].

In case that the parameter w according to the equation above for tensile test data is not constant, the modified Wehmann model exists. Once the selection is switched to " Wehmann (w variable)", the additional material parameters can be entered (see figure 65).

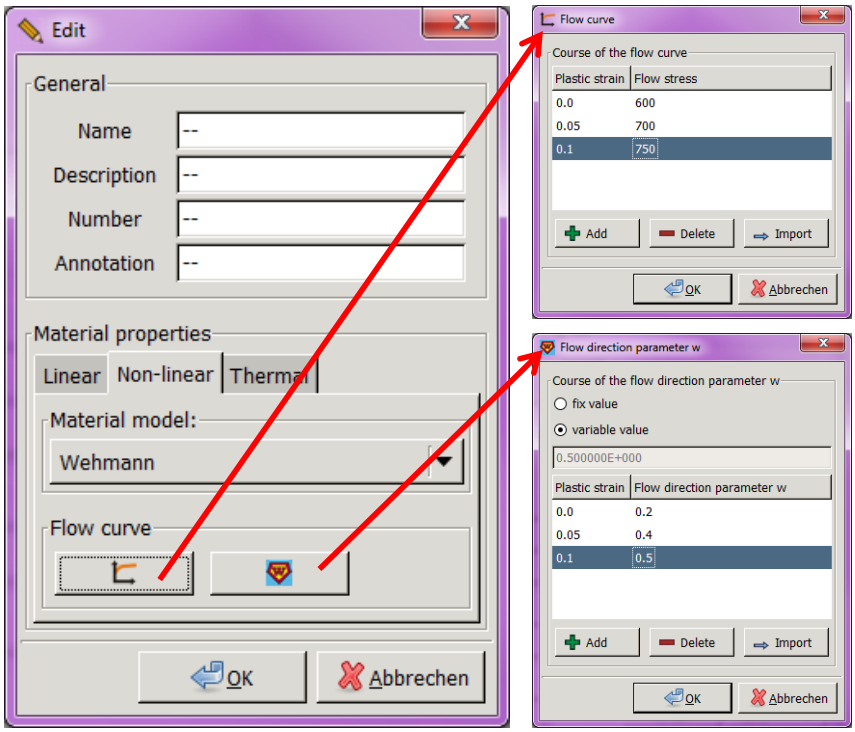


Figure 65: Input of the material data for the material law „modified Wehmann“

Entering the course of the w parameter is similar to the input of the flow curve (see figure 65 right). Again, a file can be imported with the value pairs. The file has the same structure as the one for the flow curve (see *figure 62*), only the parameter w must be entered instead of the yield stress, respectively. Thereby the parameter is determined according to the equation below from tensile test data.

$$w = -2\,\frac{\partial \varepsilon_{pl}^{q}}{\partial \varepsilon_{pl}}$$

With ε_{pl}^{q} the plastic transverse strain is denoted. The following applies:

$$\varepsilon_{pl}^{q} = \varepsilon^{q} - \varepsilon_{el}^{q} = \frac{\Delta D}{D_0} + \nu\,\varepsilon_{el} = \frac{\Delta D}{D_0} + \nu\,\frac{\sigma}{E}$$

So the course of the plastic transverse strain plotted over the plastic elongation is to be determined from the tensile test. The gradient of this curve corresponds to the w-parameter. Also applies $0 \leq w \leq 1$ here. For further information on the theoretical background of modified Wehmann model we refer to [Wehm14].

Applying Boundary Conditions

Z88Aurora® offers the possibility to define all boundary conditions within the preprocessor. Boundary conditions can only be applied to sets, so first of all these have to be defined with Picking (for more information see chapter „Picking“)

Picking of nodes	**Picking of elements**
All mechanical and thermal boundary conditions	**Pressure for tetrahedrons, hexahedrons and volume shells**

Figure 66: Picking options for boundary conditions

Imported structures can either be calculated with the existing boundary conditions in Z88Aurora® or new entries can be applied. For imported boundary conditions sets are created automatically, that can be viewed in boundary conditions menu.

For both types of analysis Linear mechanical and Static thermal two different views of the menu are available, that can be selected in the upper left corner of the menu.

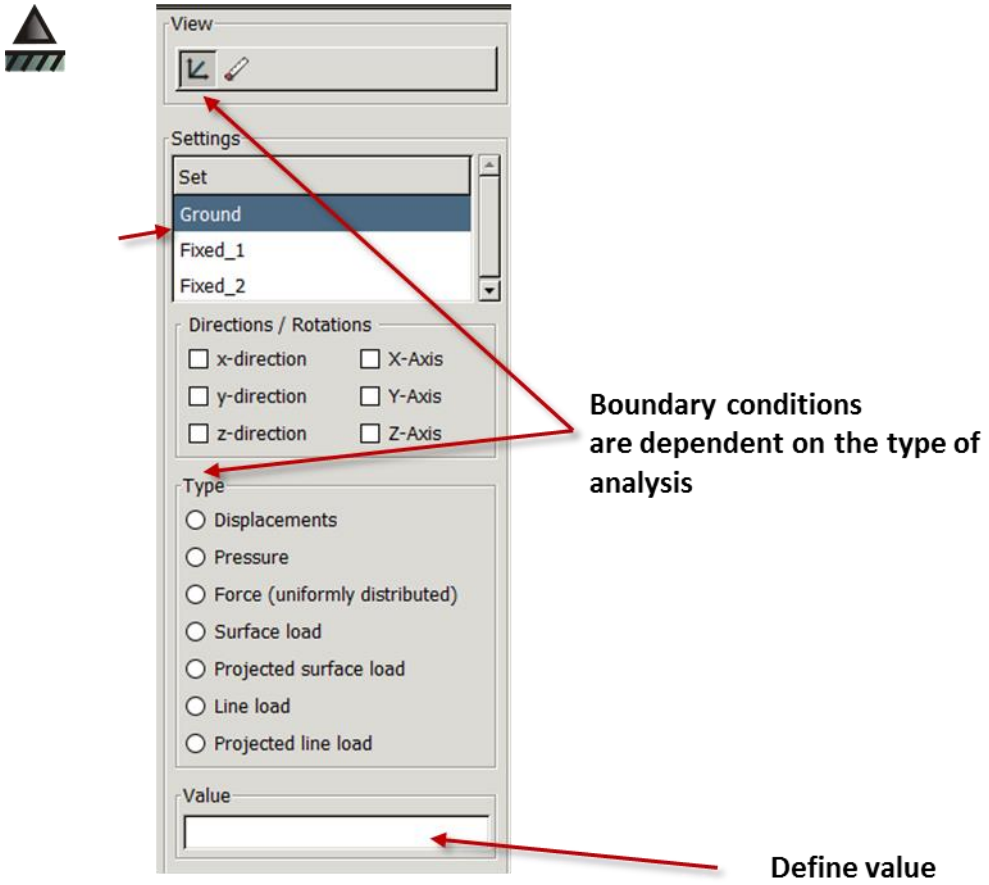


Figure 67: Creating boundary conditions I

In order to apply a boundary condition, proceed as follows:

⇨ Select analysis type

⇨ Select set

⇨ Select Directions/Rotations, e.g. x-direction

⇨ Select type, e.g. “Displacements”

⇨ Enter value, e.g. “0”

⇨ Enter name, e.g. “fixed”

⇨

In the OpenGL window the boundary condition appears with the respective color.

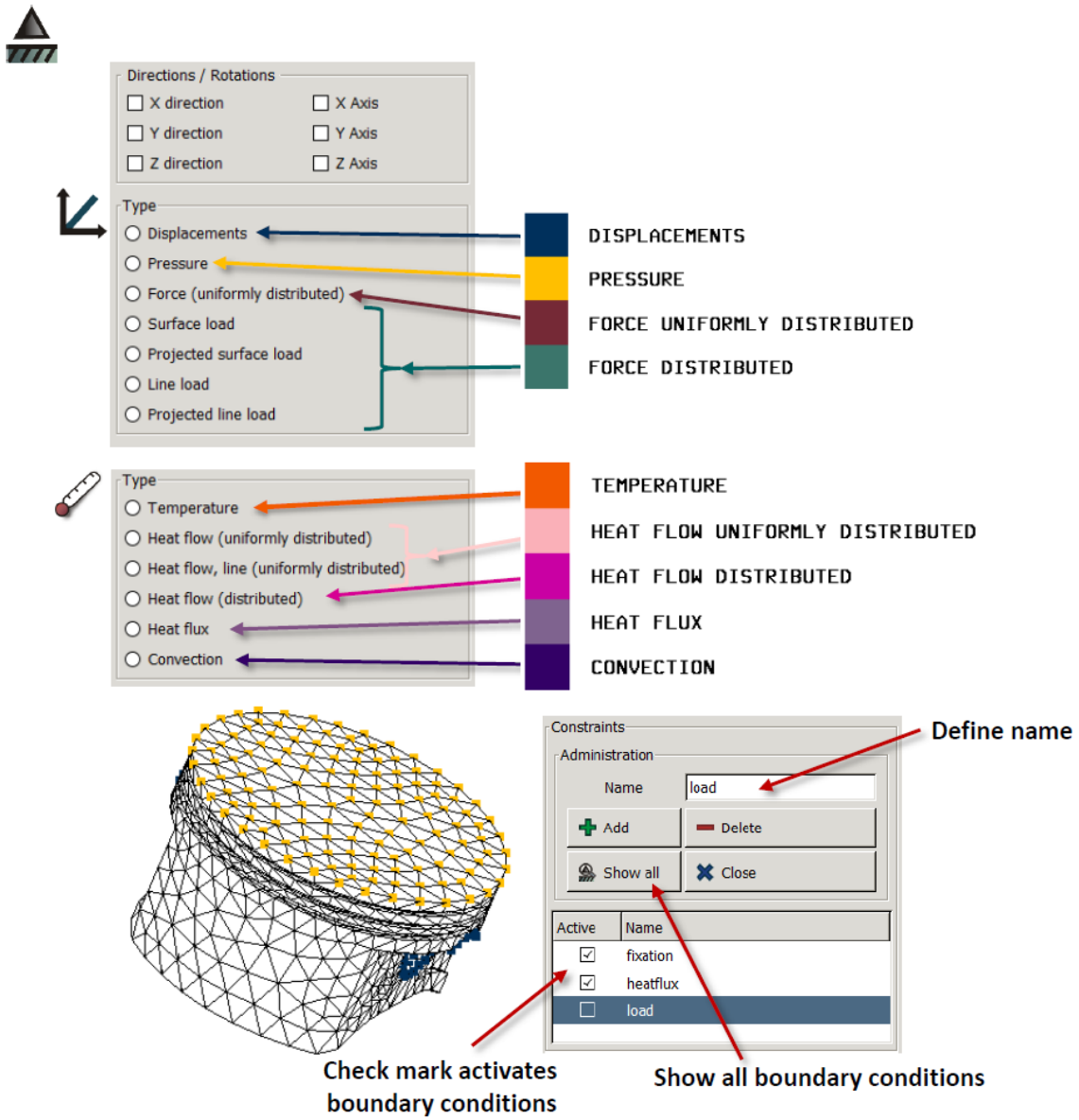


Figure 68: Creating boundary conditions II

Figure 68 shows the possibilities to apply boundary conditions. You can apply displacements, pressure and forces. You can choose between uniformly distributed force, surface load, projected surface load, line load and projected line load.

Force (uniformly distributed) applies always the same force to the nodes while Surface load and Line load distributes the load according to FEA rules (for further information see theory manual)

- ⚠ **Thermal boundary conditions are not dependent on direction!**
- ⚠ **Pressure always requires a surface-set.**

deletes existing boundary conditions. “Show all” shows the boundary conditions, which are activated via the check mark. The different boundary conditions are shown in their respective color:

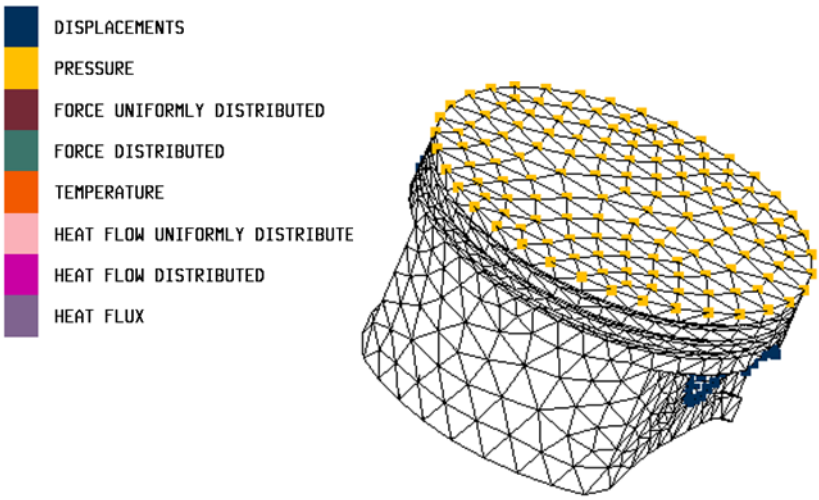


Figure 69: view options “boundary conditions“

To view single boundary conditions separately, the respective constraint can be selected via "Administration".

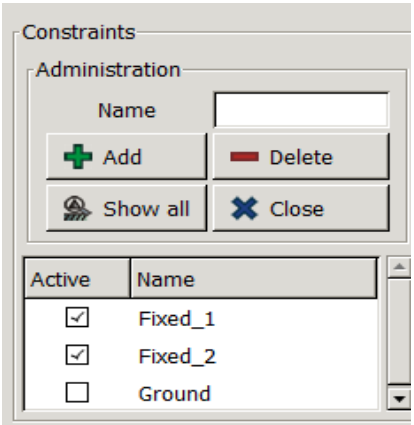


Figure 70: viewing boundary conditions separately

With 

the menu is closed.

Size of boundary conditions

The function "Size of boundary conditions" effects that the shown boundary conditions are displayed at a larger or smaller scale in the preprocessor menu.

Figure 71: Changing the size of boundary conditions

The labeling of the boundary conditions is not scaled by the size of the component. If you do not see applied boundary conditions, please change the size via the tool bar "View" or the sub item "Size of boundary conditions" in the "View" menu.

4.3  Solver

The **solver** is the heart of the program system. It calculates the element stiffness matrices, compiles the total stiffness matrix, scales the system of equations, solves the (huge) system of equations and stores the displacements, the nodal forces and stresses.

The linear Solvers Z88R and Z88RS

Z88 features three different solvers:

- A Cholesky solver without fill-in. It is easy to handle and very fast for small and medium structures. However, like any direct solver Z88F reacts badly on ill-numbered nodes but you may improve the situation with the Cuthill-McKee program Z88H. Z88F is your choice for small and medium structures, up to 20,000 ... 30,000 degrees of freedom.
- A direct sparse matrix solver with fill-in. It uses the so-called PARDISO solver. This solver is very fast but uses very much dynamic memory. It is your choice for medium structures, up to 150,000 degrees of freedom. The PARDISO solver is more sensitive towards static indetermination and therefore identifies problems in the preprocessing at the start of the calculation.
- A sparse matrix iterative solver. It solves the system of equations by the method of conjugate gradients featuring SOR- preconditioning or preconditioning by an incomplete Cholesky decomposition depending on your choice. This solver deals with structures with more than 100,000 DOF (degrees of freedom) at nearly the same speed as the solvers of the large and expensive commercial FEA programs as our tests showed. At the same time the memory requirement is minimal. It's the right choice for structure from 100,000 DOF upwards but can also deal with structures with millions of DOF. This proven and stable solver often works even when the structure is statically indeterminate, a problem in the preprocessing which is only identifiable when the calculation has already finished.

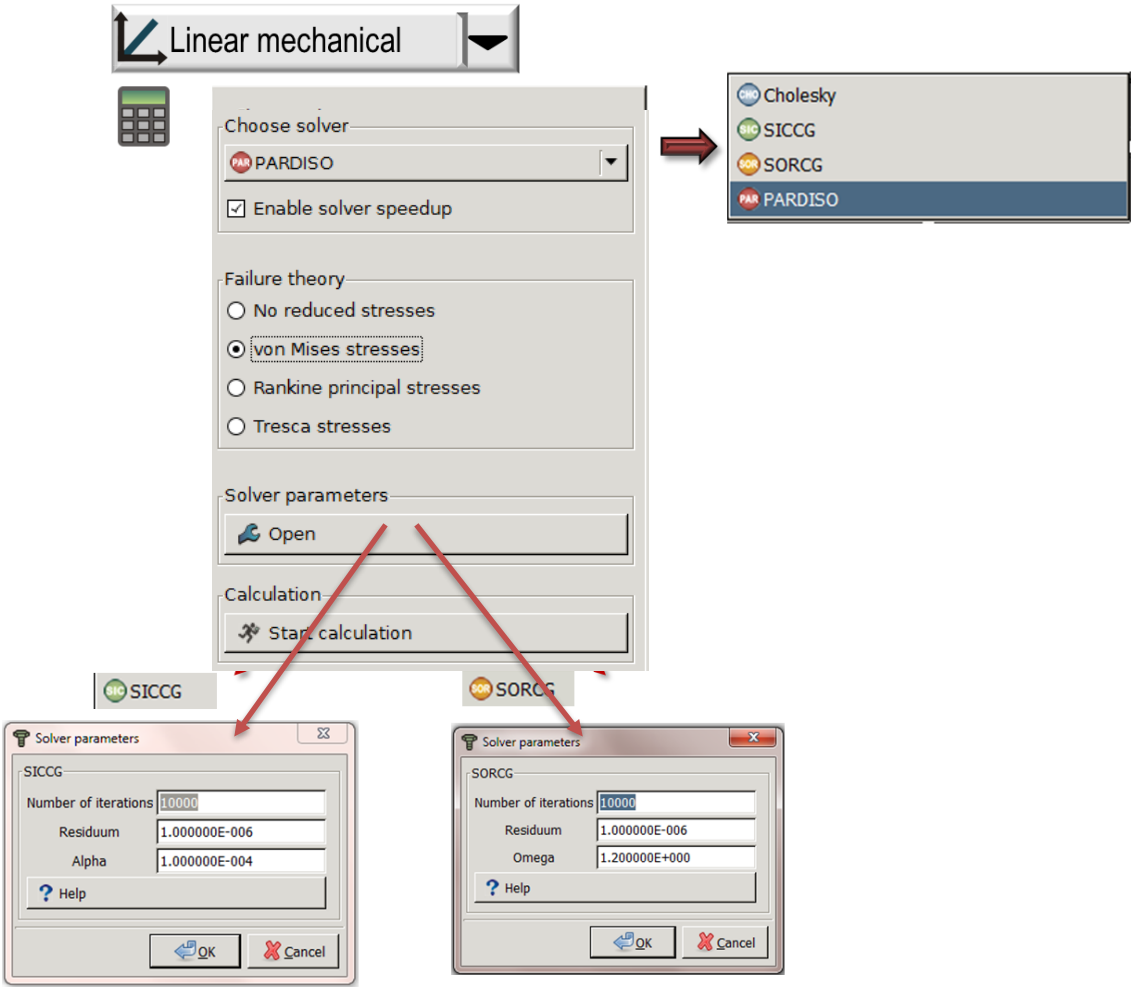


Figure 72: solver menu linear mechanical analysis

Accelerated versions of the linear solvers were already added to Z88Aurora® V2b. Some steps in the solution process could be reduced, without compromising the quality of results. In particular, the two variants of the CG iterative solver were parallelized, thereby not only one CPU but any number can be used simultaneously. Therefor it requires twice as much memory as the CG-solvers from Z88R. Further, the termination criterion for the mathematical residual is now used, which includes the Euclidean norm of the error vector. This threshold corresponds to the root of the termination criterion of the Z88R. Even with the boundary conditions and other task-phase modifications were made, so that the already parallelized PARDISO solver could be made even faster. All changes are marked as Z88RS and are used when a check mark is set for "Solver Speedup". The following Table 4 should serve for orientation when each solver can be used.

Table 4: Overview solver

Solver	Type	Number of DOF	Required Memory	Speed	Multi -CPU	Notes
Z88R -t/c -choly	Cholesky Solver without Fill-In	up to ~ 30,000	medium	medium	no	only for trusses and beams
Z88R -t/c -parao	direct Solver with Fill-In	up to ~ 150,000	very high	very high	yes	useful with several CPUs and *very much* memory
Z88R -t/c -siccg or -sorcg	conjugated gradients solver with pre-conditioning	no limits (millions of DOF)	an absolute minimum	medium	no	a very stable and reliable solver for very large structures
Z88RS -t/c -parao	direct Solver with Fill-In	depending on RAM up to millions of DOF	very high	maximum	yes	once more accelerated version of the direct Pardiso Solver from Z88R
Z88RS -t/c -siccg or -sorcg	conjugated gradients solver with pre-conditioning	no limits (millions of DOF)	very low	high	yes	requires twice as much memory as the CG-solvers from Z88R and enables several CPUs

The nonlinear solver Z88NL

The module Z88NL represents a solver for nonlinear calculation. Nonlinearities can either be geometrical origin or material nonlinearities, for which the stress-strain relationships are nonlinear. Regarding the material properties Young's modulus and Poisson's ratio are required for pure geometrical nonlinearity, just like for the linear solver Z88R. For material nonlinearities additional data such as the flow curve have to be specified (see section 5.2). The same boundary conditions as in the linear mechanical calculation can be applied. Following elements can be handled with geometrical nonlinearity: Type 1 (hexahedron with 8 nodes), type 4 (3dimensional truss), type 7 (plain stress element with 8 nodes), type 8 (torus with 8 nodes), type 10 (hexahedron with 20 nodes), type 16 (tetrahedron with 10 nodes) and type 17 (tetrahedron with 4 nodes). Type 1 and type 16 can be used for material nonlinearities.

The main difference to linear calculation is the parameterization of the solvers, which are described in detail in the theory manual. Also the postprocessor is different: Now results for every load step are available. This affects both the node based results of displacements and the integration point based stress results. For stress results the comparison stress is calculated after the von Mises yield criterion, which is based on Cauchy stress.

The thermal solver Z88TH

For the calculation the module for steady-state thermal simulation resorts to the solver types PARDISO, SORCG and SICCG. The number of values used in the system of equations is decreased by using the finite elements for pure thermal analysis (hexahedrons, tetrahedrons) because of the reduction of the DOF to one (instead of three), so the system of equations itself is reduced. In contrast there are no changes in the thermo-mechanical calculation, the usual three DOF have to be regarded. The thermal conductivity is the only material property that is required for determining the steady-state thermal conduction. If a thermo-mechanical simulation is to be conducted the coefficient of thermal expansion is also needed (in addition to the material properties used in elastostatic problems: Young's modulus, Poisson's ratio). No additional adjustments have to be done for a thermo-mechanical calculation. The solver Z88TH automatically conducts the desired simulation, if thermal and mechanic boundary conditions have been applied. If only thermal boundary conditions are applied a steady-state thermal calculation is conducted.

The vibration solver Z88EI

The module for natural frequency uses a numeric method, which is especially approved in FEA and was introduced already in 1950 by Cornelius Lanczos. Although no one could have thought about numeric modal analysis then, the algorithm possesses many advantages for FE programing. The basic idea, to reduce the matrix to a tridiagonal matrix (non-zero elements only in the main diagonal and the first diagonal below and above) by iteration is very effective regarding memory management. Additionally it is guaranteed mathematically, that the eigenvalues of this tridiagonal matrix are approximately equal to the eigenvalues of the original matrix. Each iteration of the solver can be divided into two stages.

Initially an additional row or column of the tridiagonal matrix is calculated – basically only three matrix values, because all previously calculated entries preserved.

In the second stage the eigenvalues of the matrix are determined – starting at zero and sorted in ascending order.

The contact module's solver

The contact module is based on the same solvers that are already used for the linear mechanical module. Both preconditioned iterative solvers SICCG and SORCG as well as the direct solver PARDISO are available. All solver types in the contact module are available in their accelerated variants so that using multiple cores will accelerate the contact analysis. The solver options are the same as in the linear mechanical module.

Further information and theoretical background on the solvers can be found in the theory manual. The solver types are selected via the solver menu, which offers different options depending on the type of analysis.

Under "failure theory" the following comparison stresses can be selected, depending on the preceding calculation:

- von Mises theory
- Rankine's theory / principal stresses
- Tresca's theory

Additionally a view control values for the respective solver have to be defined in the menu solver" → "solver parameters"

SICCG

- Termination criterion: maximum number of iterations (e.g. 10000)
- Termination criterion: residual vector < Epsilon (e.g. 1e-7)
- Parameter for convergence acceleration: shift factor Alpha (between 0 and 1, good values may vary from 0.0001 to 1; start with 0.0001).

SORCG

- Termination criterion: maximum number of iterations (e.g. 10000)
- Termination criterion: residual vector < Epsilon (e.g. 1e-7)
- Parameter for convergence acceleration: relaxation factor Alpha (between 0 and 2, good values may vary from 0.8 to 1.2).

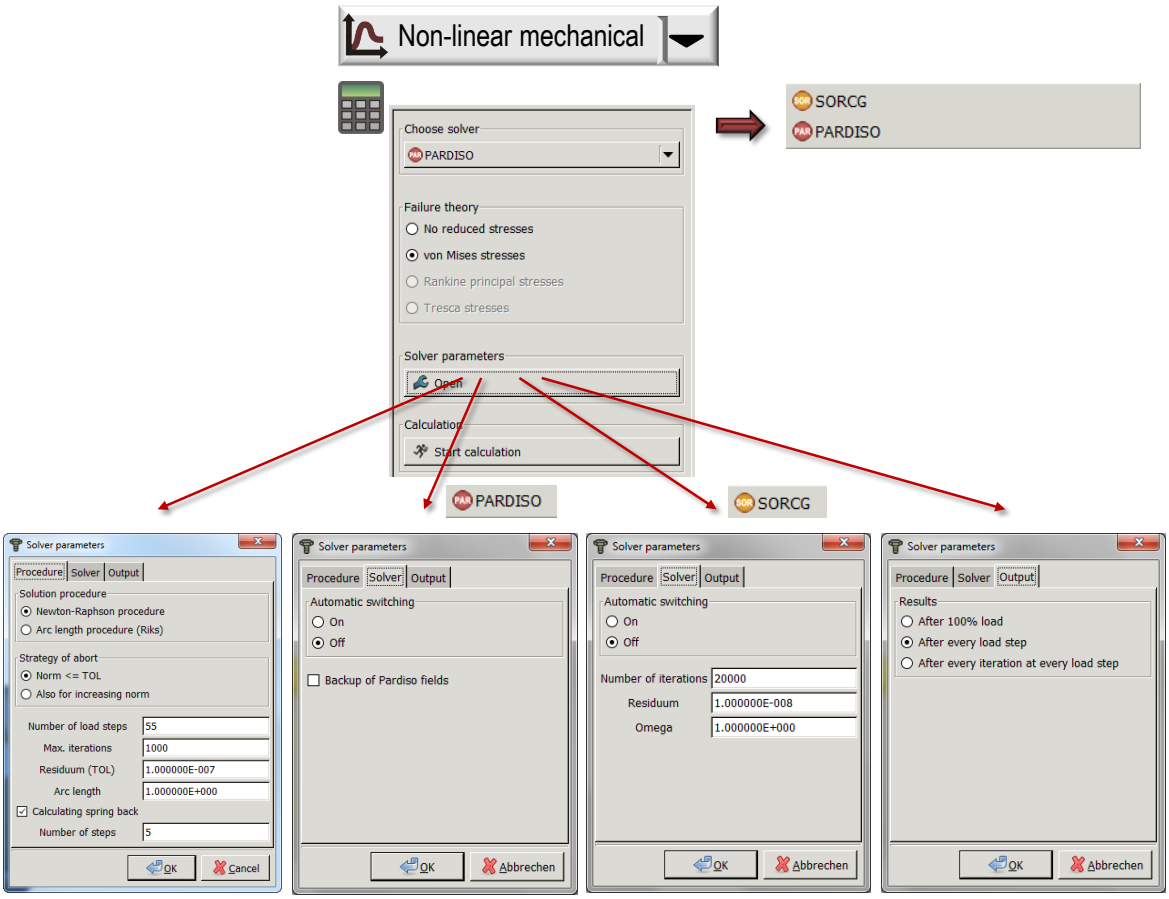


Figure 73: Solver menu non-linear calculation

⚠ ***Calculation of equivalent stress with Z88NL is only possible with von Mises theory.***

Since version V3 (see figure above, bottom left) there is the possibility of spring back calculation. This is provided for calculations with the plastic material laws. If the hook is set, the number of steps for the spring back calculation can be determined. With five steps chosen, for example, the load is reduced at the end in 20 % increments from 100 % to 0 %.

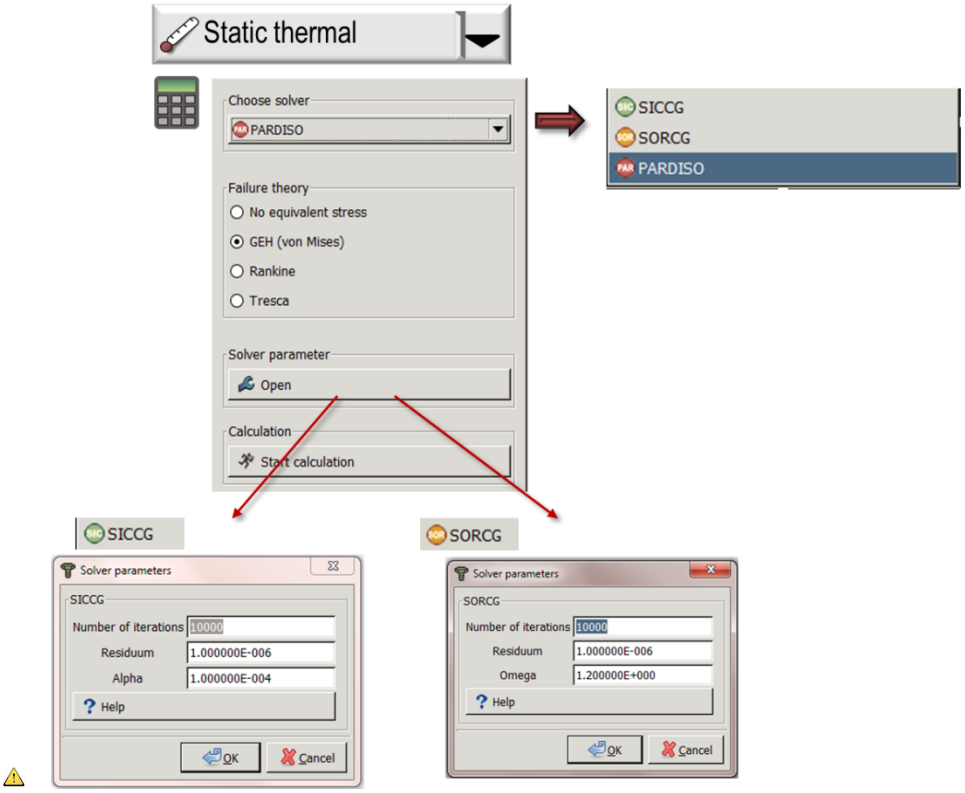


Figure 74: Solver menu steady-state thermal calculation

The settings correlate to those of the linear mechanical calculation.

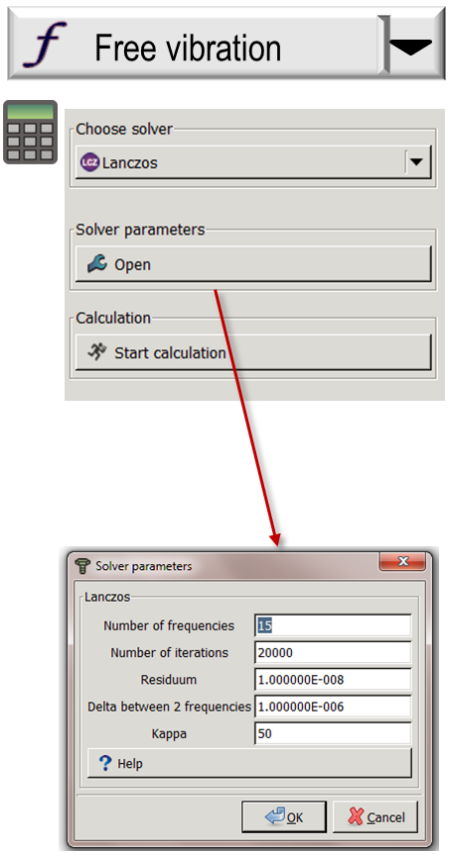
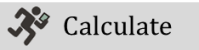


Figure 75: Solver menu natural frequency

- Number of frequencies that are calculated
- Termination criterion: maximum number of iterations (e.g. 10000)
- Termination criterion: residual vector, eigenvalue remains constant
- Difference between two frequencies: Difference between two eigenvalues (if the difference is smaller, the eigenvalues cannot be distinguished)
- Block length Kappa: critical value; after this number of iterations the residual vector is checked

After defining of all required parameters the calculation is started with. 

When the calculation is finished, an information window is opened automatically.

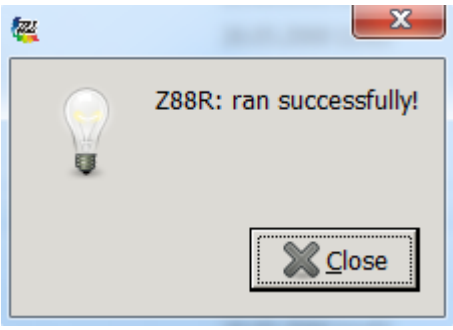


Figure 76: Information window

The solver in the text menu bar

The solver can also be accessed via the text menu bar:

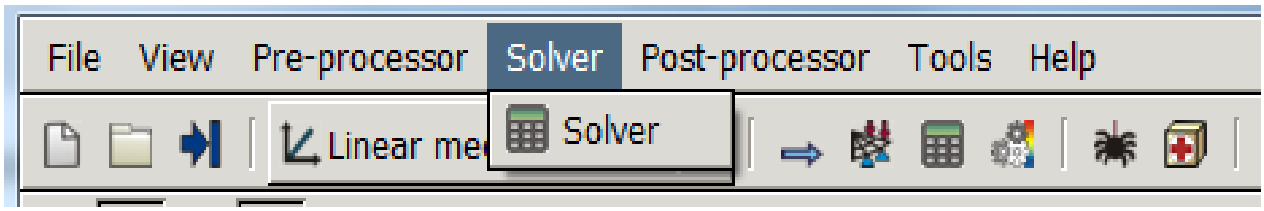


Figure 77: Selecting the solver via the text menu bar

Available solver types for the respective finite elements

elementtype	*approach*				*f*
Hexahedron					
Hexahedron Nr.1	linear	✓ SOR PAR SIC	✓ SOR PAR	✓ SOR PAR SIC	✓ LCZ
Hexahedron Nr.10	quadratic	✓ SOR PAR SIC	✓ SOR PAR	✓ SOR PAR SIC	✓ LCZ
Tetrahedron					
Tetrahedron Nr.16	quadratic	✓ SOR PAR SIC	✓ SOR PAR	✓ SOR PAR SIC	✓ LCZ
Tetrahedron Nr.17	linear	✓ SOR PAR SIC	✓ SOR PAR	✓ SOR PAR SIC	✓ LCZ
Plain stress element					
Plain stress element Nr.3	quadratic	✓ SOR PAR SIC	✗	✗	✗
Plain stress element Nr.7	quadratic	✓ SOR PAR SIC	✓ SOR PAR	✗	✗
Plain stress element Nr.11	cubic	✓ SOR PAR SIC	✗	✗	✗
Plain stress element Nr.14	quadratic	✓ SOR PAR SIC	✗	✗	✗
Torus					
Torus Nr.6	linear	✓ SOR PAR SIC	✗	✗	✗
Torus Nr.8	quadratic	✓ SOR PAR SIC	✓ SOR PAR	✗	✗
Torus Nr.12	cubic	✓ SOR PAR SIC	✗	✗	✗
Torus Nr.15	quadratic	✓ SOR PAR SIC	✗	✗	✗
Plate					
Plate Nr.18	quadratic	✓ SOR PAR SIC	✗	✗	✗
Plate Nr.19	cubic	✓ SOR PAR SIC	✗	✗	✗
Plate Nr.20	quadratic	✓ SOR PAR SIC	✗	✗	✗
Shell					
Shell Nr.21	3 D quadratic	✓ SOR PAR SIC	✗	✗	✗
Shell Nr. 22	3 D quadratic	✓ SOR PAR SIC	✗	✗	✗
Shell Nr. 23	2 D quadratic	✓ SOR PAR SIC	✗	✗	✗
Shell Nr. 24	2 D quadratic	✓ SOR PAR SIC	✗	✗	✗
Truss and beam structures					
Truss Nr.4	exact	✓ SOR PAR SIC CHO	✓ SOR PAR	✗	✗
Truss Nr.9	exact	✓ SOR PAR SIC CHO	✗	✗	✗
Beam Nr.2	exact	✓ SOR PAR SIC CHO	✗	✗	✗
Beam Nr.13	exact	✓ SOR PAR SIC CHO	✗	✗	✗
Cam Nr.5	exact	✓ SOR PAR SIC CHO	✗	✗	✗
Beam Nr.25	exact	✓ SOR PAR SIC CHO	✗	✗	✗

4.4 Postprocessor

After the calculation has run, the results can be displayed in the Z88Aurora® postprocessor by clicking the button .

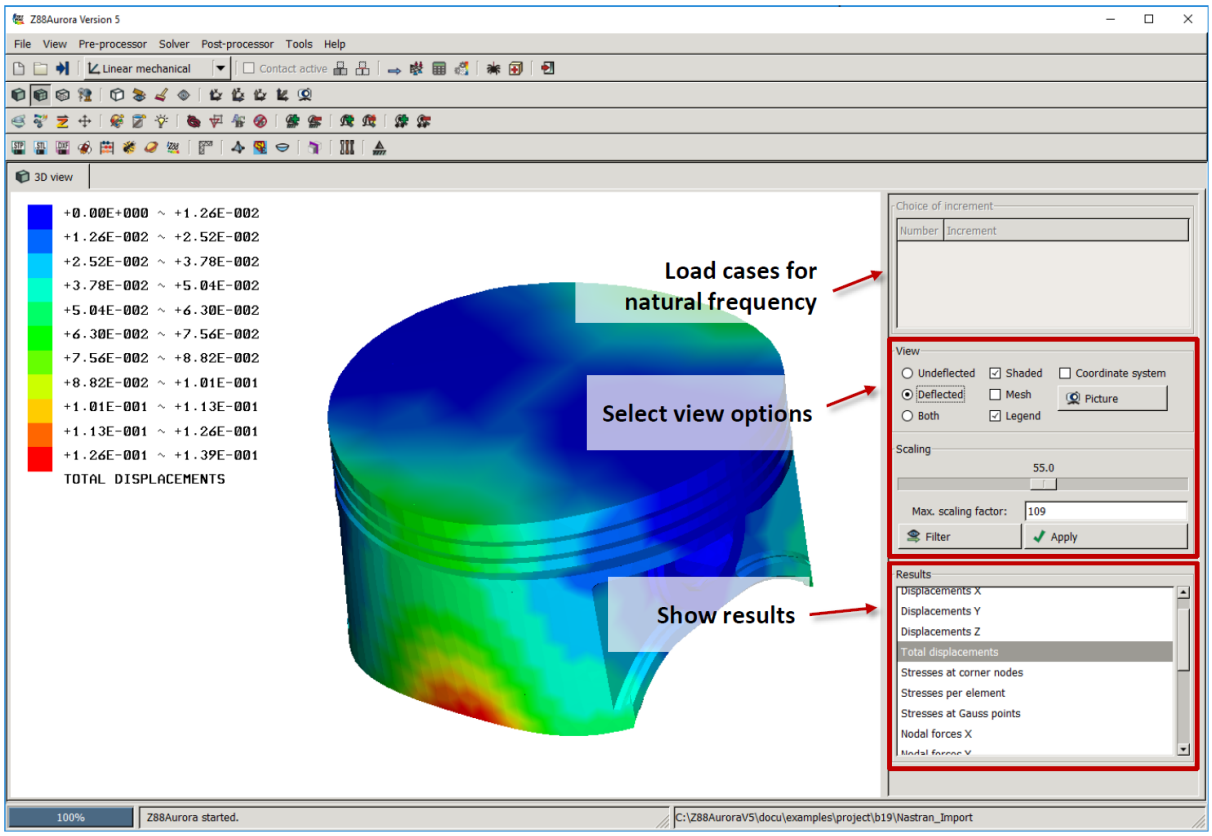


Figure 78: Z88Aurora® Postprocessor

On the right side of the screen a context menu appears. Here it is possible to have the component displayed in the results window in various ways: deflected, undeflected or both at the same time. Additionally, the user can choose between different view options: shading, hide/show mesh, legend or coordinate system triad.

A new feature of Z88Aurora® V5 is the OnClick-image export. The user can take a picture of the current viewport by pressing the button Picture. The image is automatically exported to the current project folder. Z88Aurora® uses the default image setting (width, height, file format) as specified in the file *z88enviro.dyn* (for further information see Z88Aurora® Theory Manual).

Below there is the results menu: the displacements (component-by-component and as value) as well as the stresses (at the corner nodes, averaged by elements and at the Gauss points) can be shown, the Gauss point display, however, is only shown in an undeflected condition.

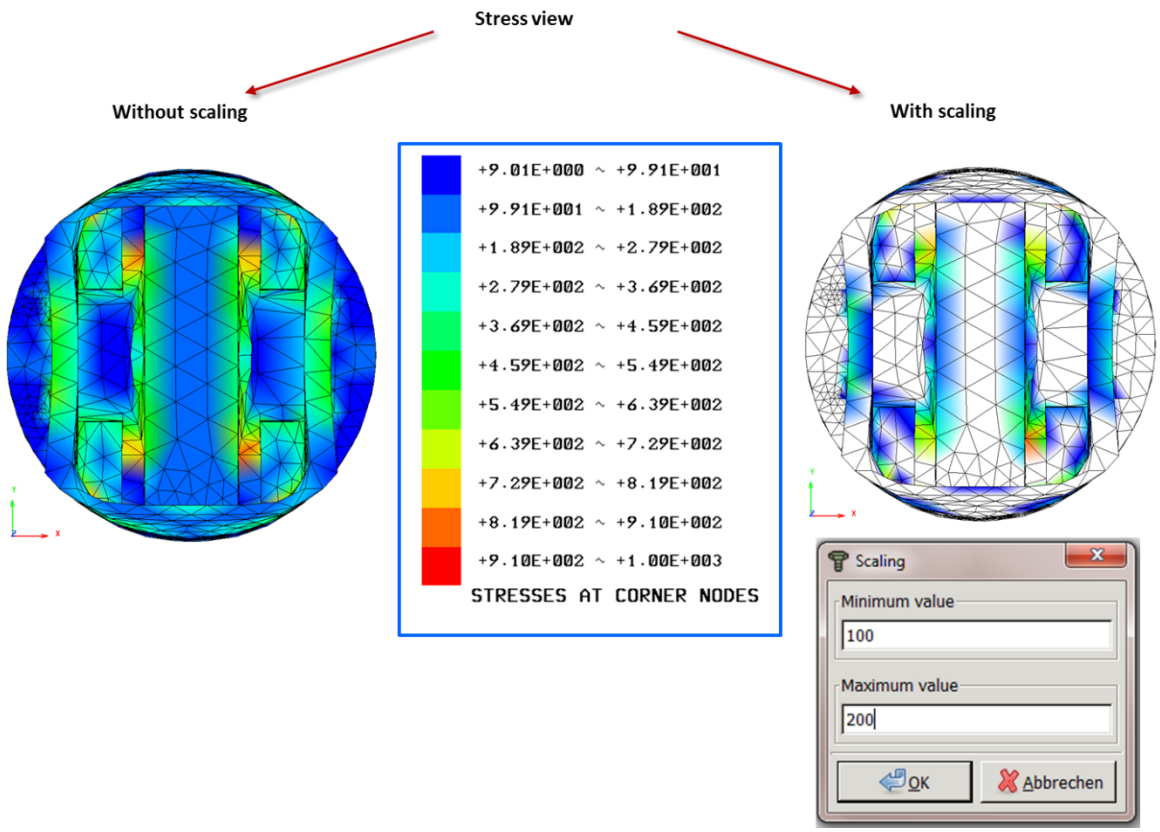


Figure 79: Color scale

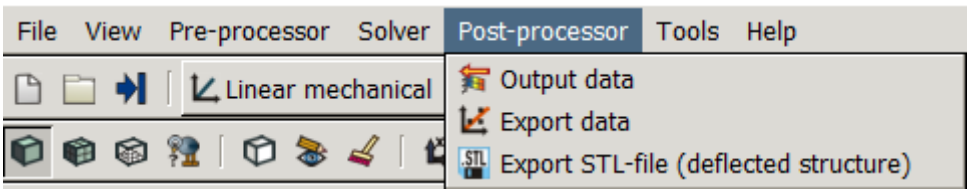


Figure 80: Menu bar Post-processor

Under "Postprocessor → Output data“ you can access the single output files of the calculation, in order to get the exact numerical values (for further information see Z88Aurora® Theory Manual):

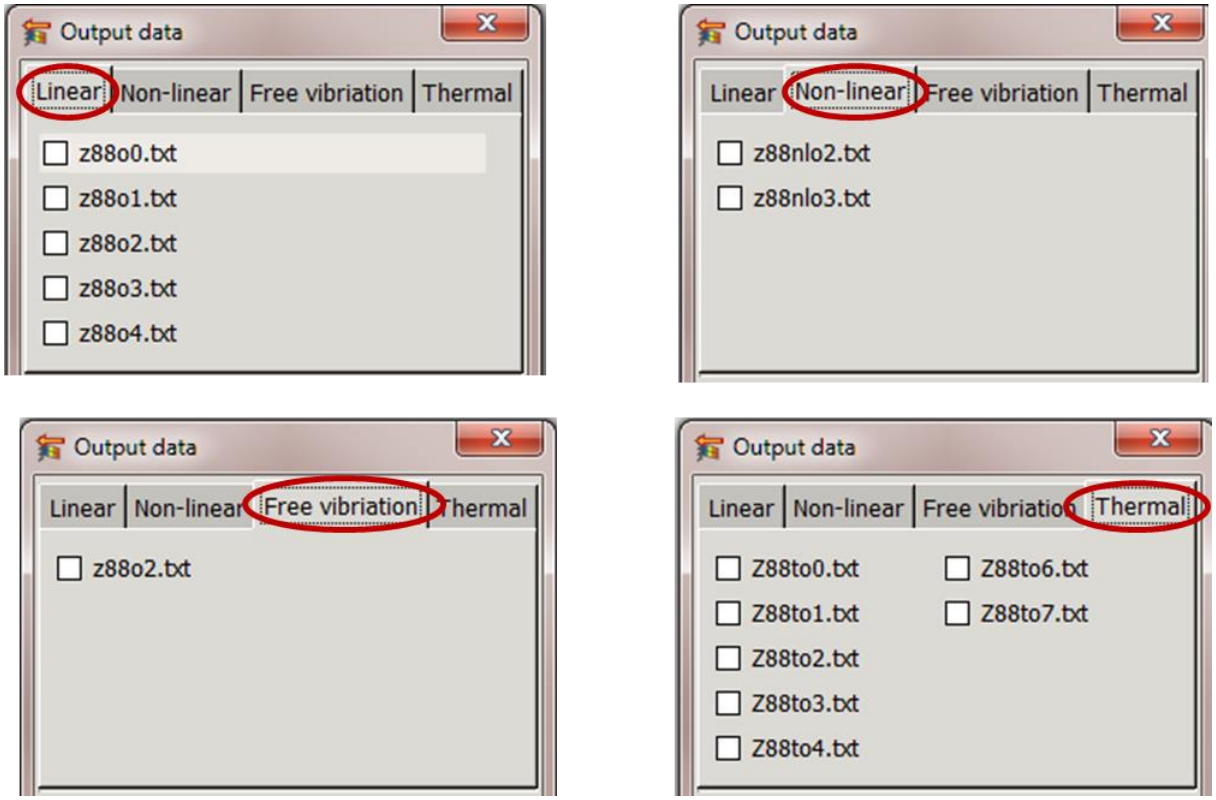


Figure 81: Output files

- Z88O0.TXT – prepared input data
- Z88O1.TXT – prepared boundary conditions
- Z88O2.TXT – calculated displacements
- Z88O3.TXT – calculated stresses
- Z88O4.TXT – calculated nodal forces
- Z88TO0.TXT – calculated temperature
- Z88TO1.TXT – calculated heat flow
- Z88TO2.TXT – calculated thermal expansion
- Z88TO3.TXT – calculated thermal forces
- Z88TO4.TXT – calculated displacements
- Z88TO6.TXT – calculated nodal forces (thermo-mechanic)
- Z88TO7.TXT – calculated stress (thermo-mechanic)
- Z88NLO2.TXT - calculated displacements, nonlinear calculation with Z88NL
- Z88NLO3.TXT – calculated Cauchy-stress, nonlinear calculation with Z88NL

In addition to the results data in a text file *.txt, where all nodes-, element- or Gauss point-information are stored, since version Z88Aurora® V2b it is possible by user-defined node- or element-sets ("Pre-processor → Picking") to write out the results of a specific component range. Therefore you have to choose "Export data" in the menu bar post-processor (figure 68). Afterwards a new menu pops up, where on the right hand all sets are listed. These include the sets that were created automatically and the sets that have been created individually.

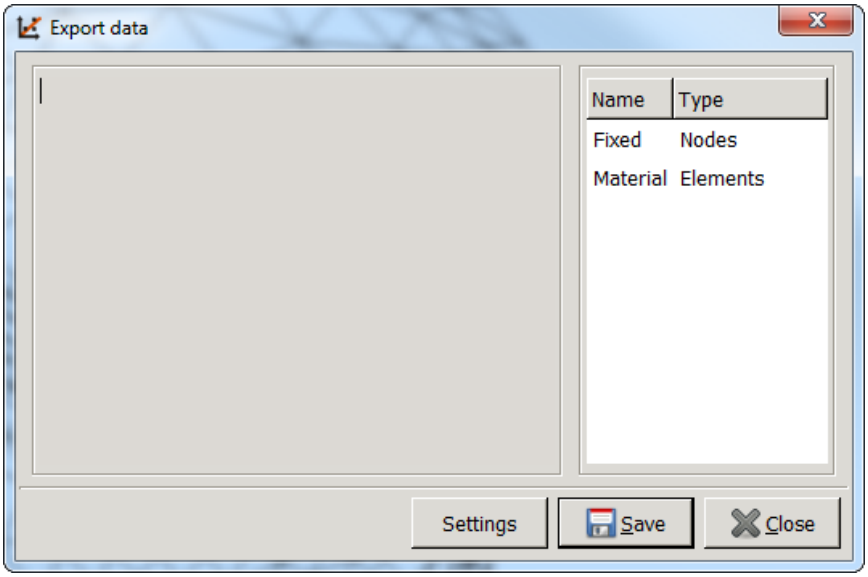


Figure 82: Export of results

If you choose one set, for example the node-set "Fixed" on the right hand, on the left hand the minimal/maximum displacements, the minimal/maximum/average stress and the minimal/maximal/average/sum of the forces will be shown.

In case of a contact analysis, the node sets containing the master and slave nodes respectively will also be shown here. This enables the user to review the nodes found during the contact search and therefore considered during the contact analysis.

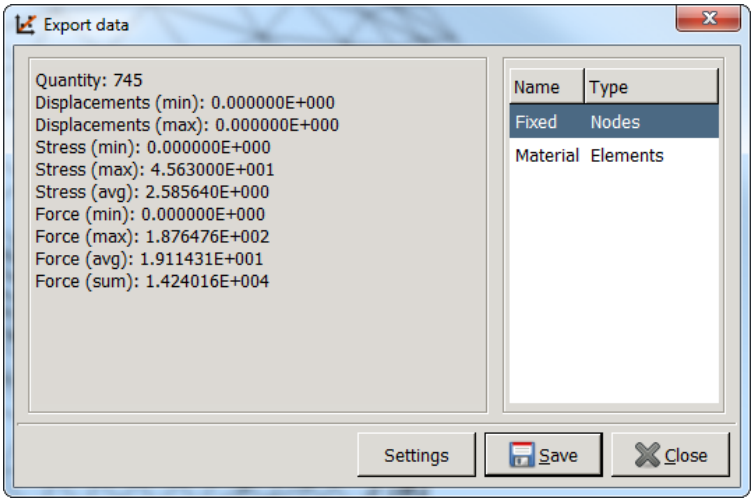


Figure 83: Results of the node-set “Fixed”

By selecting the button “Settings” (see figure 69) the user can decide individually which kind of export data of the chosen set should be written in a *.txt-file or a *.csv-file (figure 70).

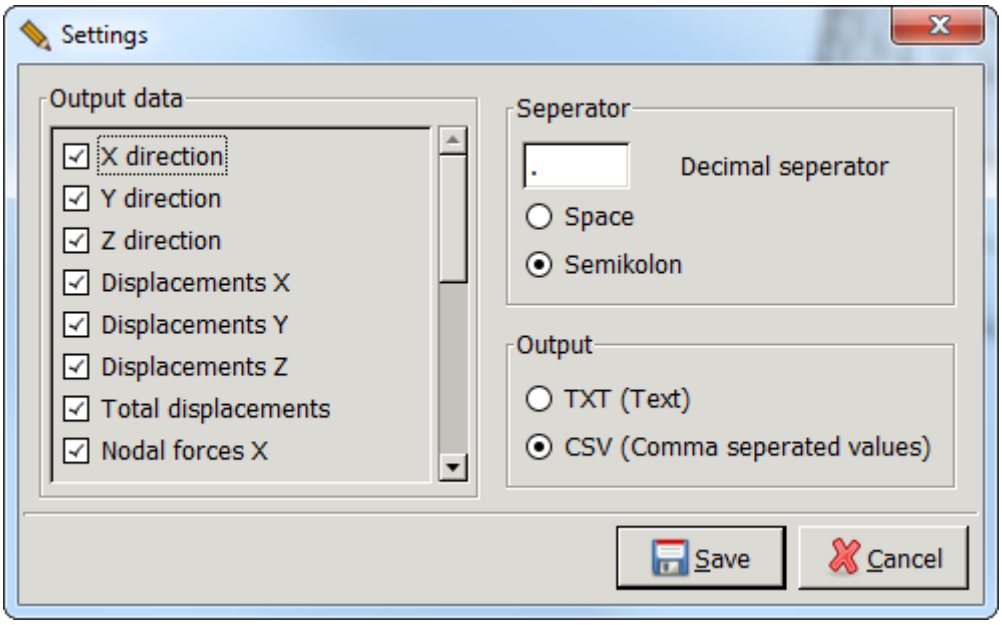


Figure 84: Individual outpout data

This is done by checking the corresponding output data set on the left side. Furthermore, the user can set the separator (space or semicolon) and the decimal separator for the output data (Please note which language setting the data processing software such as MS Excel, has!).

For the node-set “Fixed” with all selected output data the table looks like figure 73, for example:

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	name	Fixed											
2	count	745											
3	Kind	Nodes											
4	Nr	X	Y	Z	disX	disY	disZ	disMag	Fx	Fy	Fz	Fmag	stress
5	1001	4.03E+01	2.20E+01	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	5.10E+01	-3.46E+01	-3.96E+00	6.17E+01	4.25E+01
6	1002	3.94E+01	2.20E+01	3.98E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	3.31E+01	-2.43E+01	7.59E+00	4.17E+01	4.56E+01
7	1003	3.77E+01	2.20E+01	6.72E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	1.45E+01	-1.24E+01	6.32E+00	2.01E+01	4.27E+01
8	1004	3.57E+01	2.20E+01	8.49E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	1.90E+01	-8.27E+00	2.64E+00	2.09E+01	3.57E+01
9	1005	3.23E+01	2.20E+01	9.94E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	1.32E+01	-4.37E+00	2.81E+00	1.42E+01	2.65E+01
10	1006	2.87E+01	2.20E+01	1.01E+01	0.00E+00	0.00E+00	0.00E+00	0.00E+00	6.79E+00	-1.33E+00	3.95E+00	7.97E+00	1.82E+01
11	1007	2.50E+01	2.20E+01	8.90E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	8.41E-01	-7.92E-01	2.77E+00	3.00E+00	1.27E+01
12	1008	2.27E+01	2.20E+01	7.17E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	4.54E-01	-4.30E-02	1.65E+00	1.71E+00	9.12E+00

Figure 85: Example of a Postinfo file

The file itself will be saved automatically in the project file directory, it is called PostInfo_Fixed.csv (the name is a combination of PostInfo and the set-name).

It can be seen that the type of the output file *Fixed* is *Nodes* with a number (*count*) of 745 nodes. The following table 4 shows the shortcuts of the output data file.

Table 4: shortcuts of the Postinfo file

Nr	number
X	x-coordinate
Y	y- coordinate
Z	z- coordinate
disX	displacement in x-direction
disY	displacement in y-direction
disZ	displacement in z-direction
disMag	total displacements
Fx	forces in x-direction
Fy	forces in y-direction
Fz	forces in z-direction
Fmag	total forces
stress	stresses

It is also possible the currently displayed deformed structure to save as an STL file. This is available for all element types except the structural elements (type 2,4,5,9,13 and 25).

5. TOOLS

5.1 Analysis

For exact analysis of an FE-mesh or an imported model you can get information about nodes, elements and surfaces via “Analysis”. You select in the respective picking menu the object and go to “Tools”>”Analysis”. With “measuring” you can view the special orientation of two nodes.

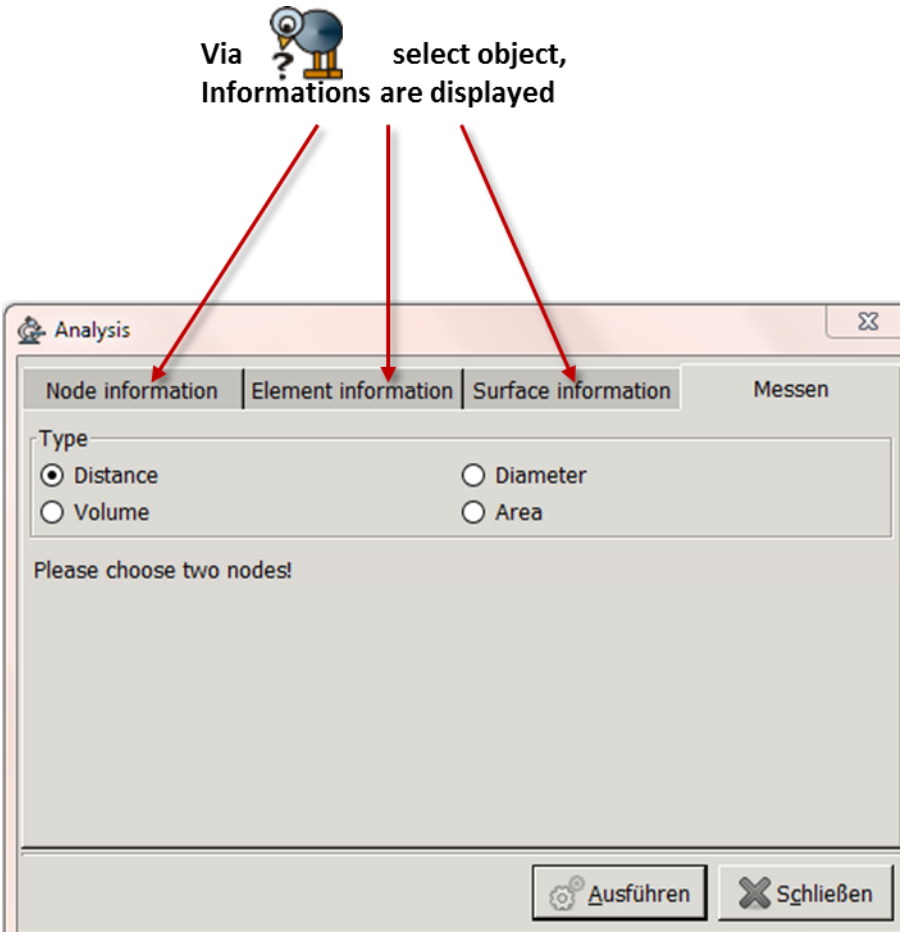


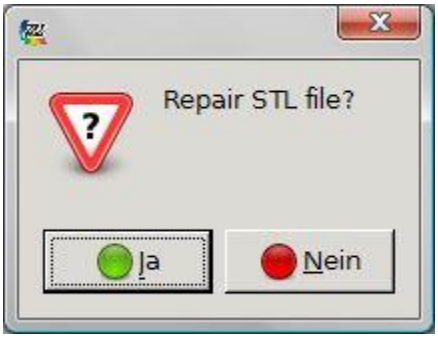
Figure 86: Tool for analysis

5.2 Edit STL

Switched surfaces in STL-import files, that cause a meshing abort, can be repaired via the tool “edit STL”. To do this select “Edit STL” in the “Tools” menu.

⇨ the switched surfaces are displayed in red

⇨click on one (!) green element



⇨ confirm with “Yes”

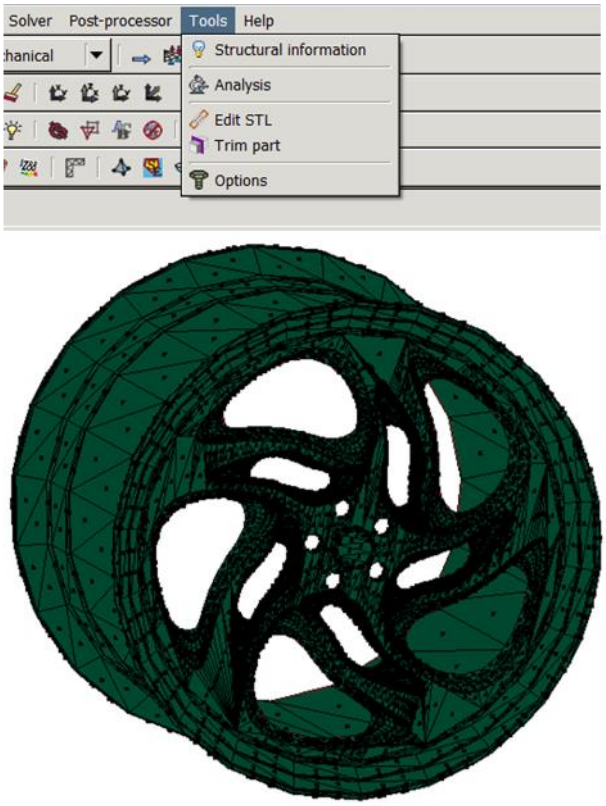


Figure 87: Edit STL

5.3 Options

Changes in the user interface can be done in the menu “Options”. Here you can select the language, the data folders, the memory settings and the view options.

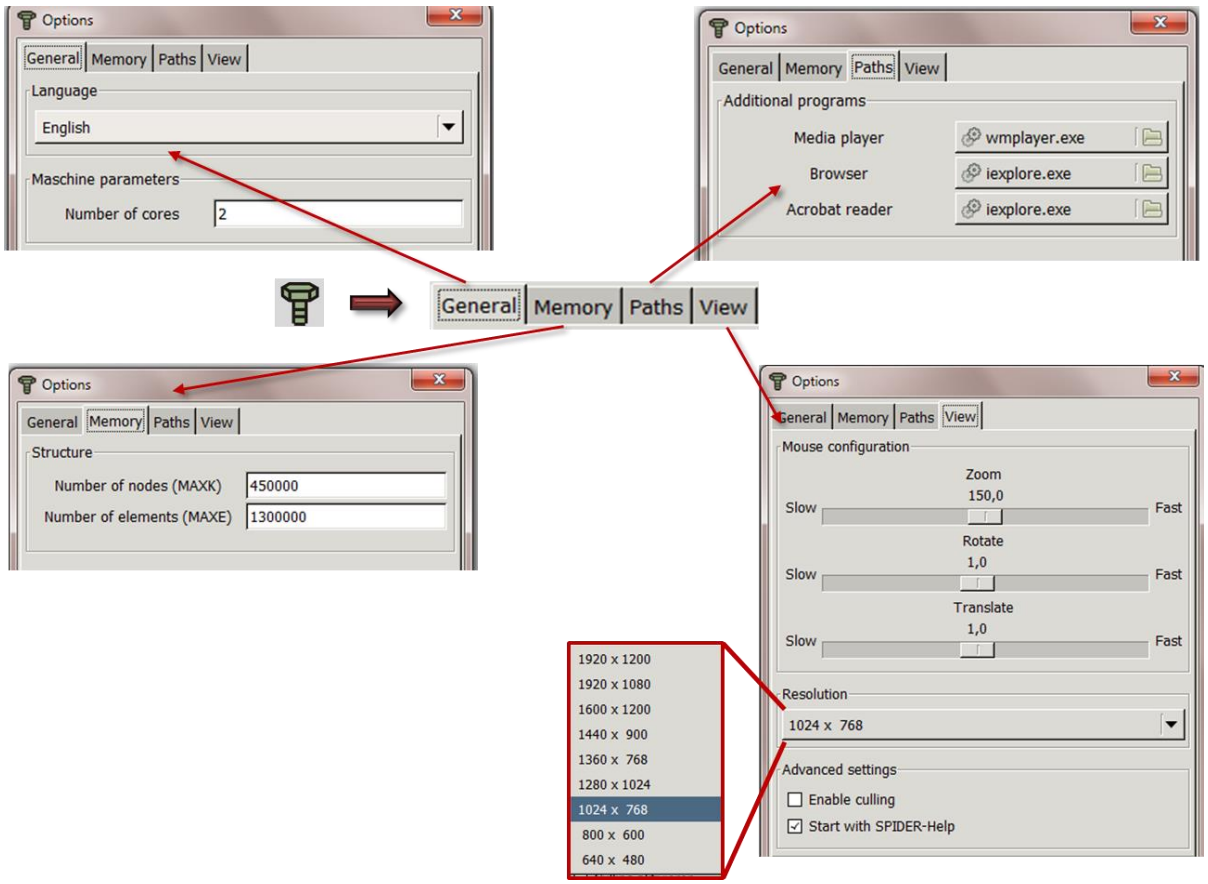


Figure 88: Options

⚠ The global settings for CPU and memory selected here are independent from the local settings in the solver options menu.

⚠ The changes only take effect after rebooting Z88Aurora®!

Media Player

Selecting the media player with which you can play the instruction videos.

e.g. VLC Media Player; Default path: „C:\Program Files\VideoLAN\VLC\vlc.exe “

Browser

Selecting the browser with which you can view the homepage and the user forum directly from Z88Aurora®

e.g. Mozilla Firefox; Default path: „C:\Program Files\Mozilla Firefox\firefox.exe“

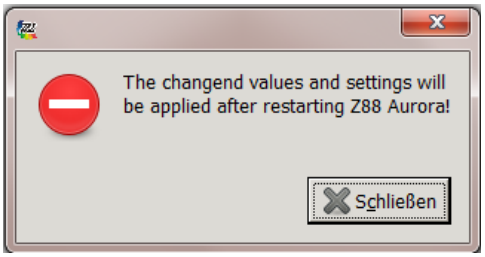
PDF-Reader

Selecting the PDF reader with which you can open the Z88 user manual and theory manual

e.g. Adobe Acrobat Reader; Default path:

„C:\Program Files\Adobe\Reader 11.0\Reader\AcroRd32.exe“

The dialogue and the information windows are closed with “OK”.



Afterwards you can restart Z88Aurora®.

6. HELP AND SUPPORT

Help

Z88Aurora® offers you several different help functions, which can be used separately. Following is an overview of the separate help components.

The icon in the icon menu bar opens the popup menu for the selection of the single help modules.

Figure 89: Help options

Video manual

To make learning easier, video sequences dealing with some special topics are available. The videos are accessed via the menu "Video manual".

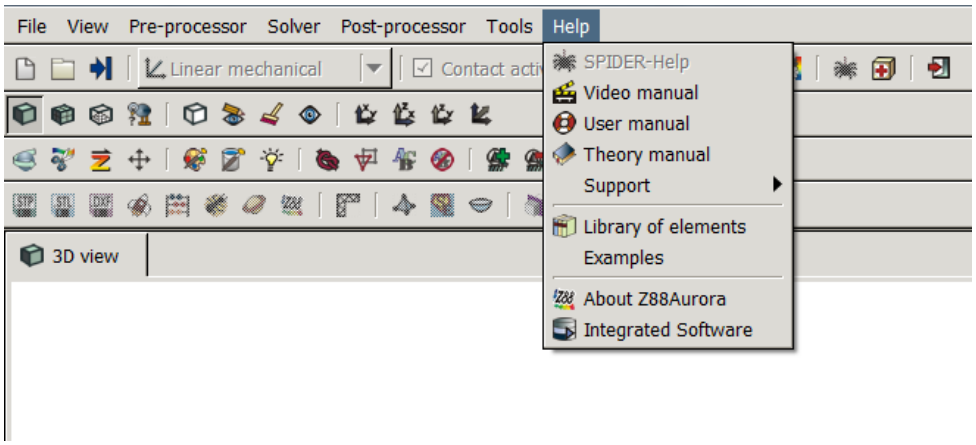


Figure 90: Video manual in Z88Aurora®

These are:

- Picking
- Views
- Node information

User Manual

In the User Manual all functions available in Z88Aurora® are explained.

Theory Manual

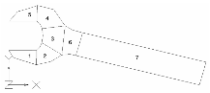
The Theory Manual addresses the issue of the computation bases of Z88Aurora®. For experienced Z88OS users the differences between Z88 V15 OS and Z88Aurora® are presented. Furthermore, all input and output files as well as the element types are illustrated in detail. The modules which are accessed from the user interface are explained here.

Element Library

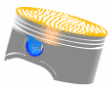
A short description of the element types integrated in Z88Aurora®.

Examples

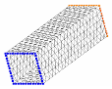
By means of different examples the basic functions are explained.

- *Plane elements:* *Example: fork wrench* 

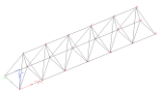
As an example, a DXF-file from AutoCAD was chosen– a fork wrench as plane stress element. By means of this component the export procedure of the structure from the CAD program as well as the import of DXF-files into Z88Aurora® is demonstrated. Furthermore, the creation and finer meshing of super structures is illustrated, as well as the execution and evaluation of a linear strength analysis.

- *Volume elements:* *Example: engine piston* 

As already described in previous chapters, you can import data from 2D- and 3D-CAD- and FE-systems in Z88Aurora®. The example cited here is an engine piston; it was designed in PTC Pro/MECHANICA and saved as a NASTRAN file. By means of this component, the import of the NASTRAN format and the calculation of tetrahedron meshes in Z88Aurora® are demonstrated.

- *Shell elements:* *Example: square pipe* 

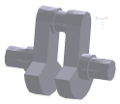
To display thin walled structures, such as bent sheet metal parts or profiles, shell models can be used. The component employed here is a square profile, which was designed as a shell model with an external FE program and saved as NASTRAN file together with the boundary conditions. By means of this component the import and the calculation of shell models in Z88Aurora® are demonstrated.

- *Truss elements:* *Example: crane girder* 

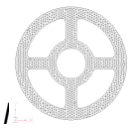
A simple example with 20 nodes and 54 trusses forming a spatial framework. These structures can easily be entered manually, CAD programs won't help much. Just try it for yourself.

- Volume element: Example: Two-stroke engine piston 

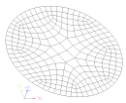
A piston for a two-stroke engine should be calculated. The load is the combustion pressure of 42.5 bar, the gudgeon pin is defined as bearing. The piston was designed in the 3D CAD program Pro/ENGINEER and meshed – selecting linear tetrahedrons - with Pro/MECHANICA. Then the structure and the mesh were exported as NASTRAN file. The piston contains 3211 nodes, thus 9633 degrees of freedom and 12489 elements (tetrahedron type 17 with 4 nodes).

- *Tetrahedron elements:* *Example: motorcycle crankshaft* 

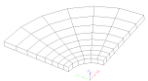
Applying a piston load of -5,000 N a single cylinder motorcycle crankshaft is to be calculated. In this case the constraints have to be considered in a special way, which is kind of tricky.

- *Plane stress element:* *Example: gearwheel* 

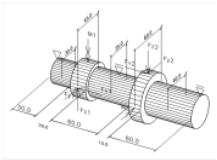
A gearwheel, whose center is pressed on a shaft, is examined. The interference fit assembly's groove pressure is 100 N/mm². Crucial point is the deformation transmitted by the center's expansion to the gear teeth (which are left out for model simplification).

- *Plate element:* *Example: circular plate* 

This sample is intended as an introduction for plate calculation. Z88 contains plate elements (Reissner-Mindlin approach) with 6-node Serendipity elements (type 18), 8-node Serendipity elements (type 20) and 16-node Lagrange elements (type 19). Nevertheless the plate is a 2D element. With plates thus being 2D elements the calculation requires some skills to map this paradox in FE software.

- *Hexahedron elements:* *Example: plate segment* 

A three-dimensional plate segment with curvilinear hexahedrons is calculated. Though seeming simple, this example can barely be solved analytically. It's a valuable sample for experiments with the mapped masher.

- Creating a structure and element parameters *Example: gear shaft* 

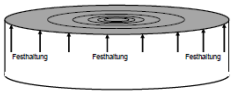
In Z88Aurora® an editor for creating beam and truss elements is included. The required nodes for creating the structure can be entered by using coordinates; the coincidence is created via the graphic user interface. This is explained using an example of a gear shaft.

- Shell thickener/Clipping *Example: submarine* 

A submarine, which was constructed in Pro/ENGINEER as shell structure is imported with the NASTRAN interface and thickened to create a volume shell in Z88Aurora®. The displacements and stresses in the submarine's hull should be calculated in the case of a dive depth of 50 m. The submarine is supposed to float in the water, thus it is fixed in Z88Aurora® via a virtual fixed-point.

- *ABAQUS-import/line load* Example: 3D gear 

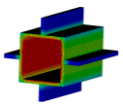
In this example a gear, which was imported as ABAQUS INP file, is calculated. This is done with a static load, so it cannot replace a strain calculation according to DIN 3990. The gear in this example is an involute spur gear without any deviations in the tooth flank. The gear body is constructed out of ribs to save weight.

- Natural frequency *Example: Drum* 

A popular example natural frequency analysis is the oscillating drumhead, because it can be calculated analytically as well. This is an example for pure natural frequency, because the drumhead achieves steady-state (final tone) only after one stroke because of the equilibrium between mass inertia and reset force. The fixing of the drumhead at the frame allows only certain modal form vectors, which can be described with mathematical methods. These form vectors should be simulated in FEA as well.

- Thermo mechanics *Example: Spoon* 

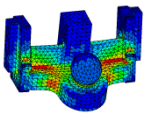
As example for heat flow and temperature analysis a spoon, which is hold with one hand and used for eating soup, should be simulated. The spoon was constructed in Pro/ENGINEER and imported as .stl-file into Z88Aurora® and was then meshed. At the handle the temperature is defined by the temperature of the hand, the heat flow through the handle is caused by the heat of the soup.

- Thermo mechanics *Example: Ripping* 

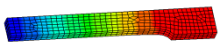
In this example, the influence of ripping on the heat transfer is analysed. The focus lies on the convective heat transfer from the solid body to the surrounding. The numerical results are compared to the analytical solution.

- Geometric nonlinearity *Example: Hinge* 

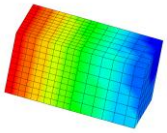
In this example a hinge to lock a cylindrical truss in position is examined. The clamping force is very high, so there are great deformations.

- Material non-linearity *Example: Ball coupling* 

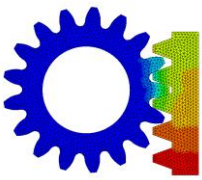
In this example, a heavy loaded ball coupling is considered. These couplings are used in agricultural engineering. In the notches, plastic deformations occur, so that taking plasticity into account leads to significantly improved stress results.

- Material non-linearity *Example: Tensile test* 

  This example deals with the simulation of a tensile test with the different material models which can be chosen. The models are compared to each other in order to explain the differences.

- Contact analysis *Example: Compression bar* 

  This example deals with a linear elastic contact simulation of a compression bar. The chosen example can easily be verified using analytic calculations.

- Contact analysis *Example: Rack* 

  This example visualizes the principle of frictionless contact between gear and rack.

SPIDER support

To achieve clearly arranged settings and functionalities for the user, the proceeding of FEA is displayed within the SPIDER work flow tool. Help can be accessed via the F1 key. You can choose between two different types of user support: the work flow scheme or additional videos, explanations or decision propositions.

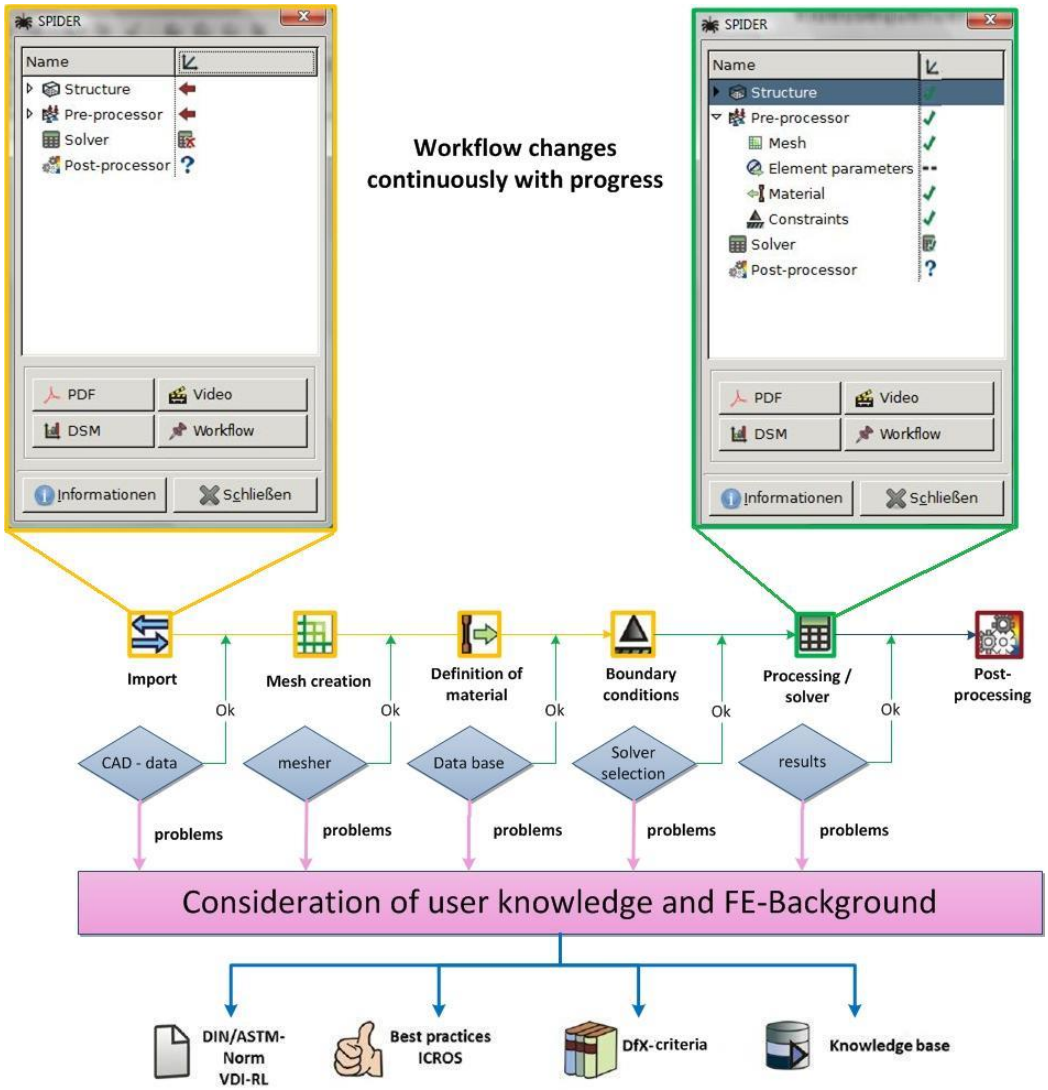


Figure 91: Work flow support SPIDER in Z88Aurora®

The SPIDER help is not available for the contact module.

About Z88Aurora®

Figure 92: Information about Z88Aurora®

Support

Homepage

See **http://z88.de** for further information!

Email

If you need support: z88aurora@uni-bayreuth.de!

Forum

On **http://forum.z88.de** you can find a forum where you can discuss Z88 topics and issues with the developers and other users.

Social Media

https://twitter.com/Z88Aurora

https://www.facebook.com/Z88Aurora/

For Development news on Z88Aurora®

7. LITERATURE

[RHA14] Rieg, F.; Hackenschmidt, R.; Alber-Laukant, B.: Finite Element Analysis for Engineers. Basics and Practical Applications with Z88Aurora®. 1st english edition, Carl Hanser, Munich, Vienna, 2014

[Wehm14] C. Wehmann: Nichtlineare Finite-Elemente-Analyse für Berechnungen im Maschinenbau. Geometrische Nichtlinearitäten und plastisches Material-verhalten ausgewählter Maschinenelemente. PhD Thesis, University of Bayreuth, Shaker, Aachen, ISBN 978-3-8440-3063-1, 2014