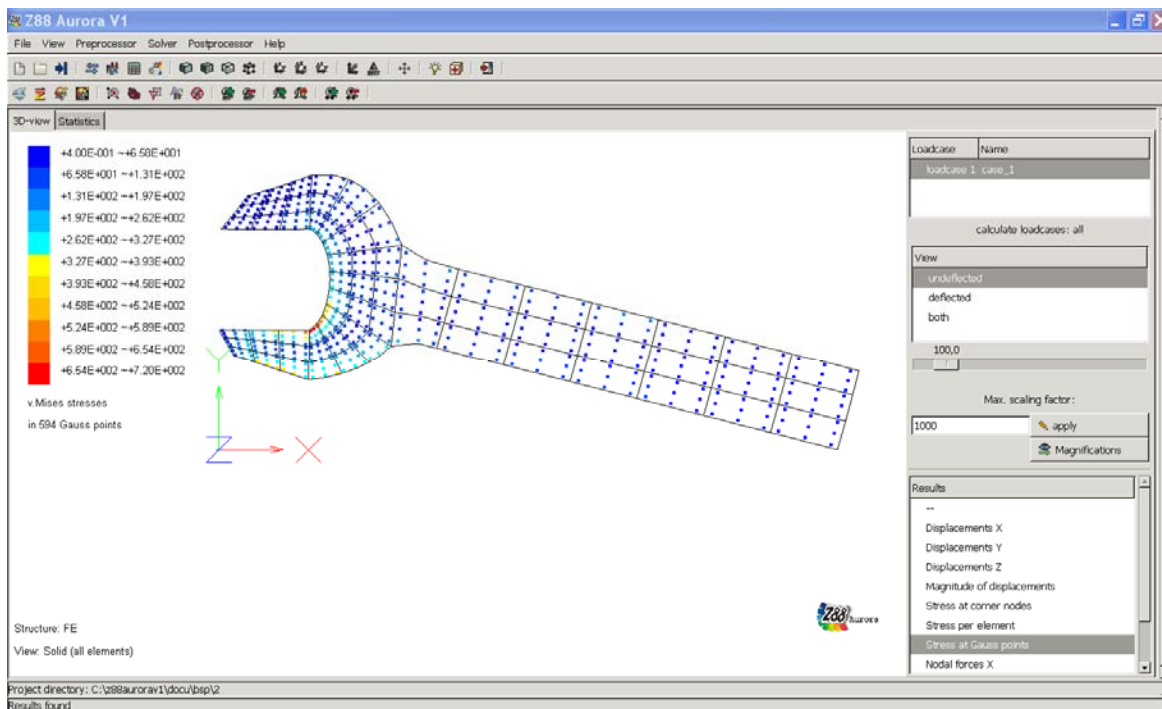




User Manual



Version V1a



*An easily operated user interface for Z88[®]
for all Windows- and Mac OS X-Computers
for 32- and 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. Wirtsch.-Ing. Reinhard Hackenschmidt,
Dipl.-Math. Martin Neidnicht, Dipl.-Ing. Florian Nützel,
Dr.-Ing. Bernd Roith, Dipl.-Ing. Alexander Troll, Dipl.-Ing. Christoph Weh-
mann, Dipl.-Ing. Jochen Zapf, Dipl.-Ing. Markus Zimmermann,
Dr.-Ing. Martin Zimmermann*

All rights reserved by the editor

Version 1 December 2010



is a registered trademark (No. 30 2009 064 238) of Professor Dr.-Ing. Frank Rieg

INSTALLATION AND OPERATION OF Z88 AURORA

Z88 is a software package for solving structural mechanical, static problems with the aid of the Finite Element Method (FEM), which is available under 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 of ten under the supervision of Professor Rieg at the University of Bayreuth.

In addition to the present compact Z88, which is currently available in the 13th version, an extended program Z88 Aurora is on the market since 2009. Z88 Aurora is based on Z88 and is available for Windows 32-BIT and 64-BIT for free download (as executable file). In addition to the efficient solvers contained in Z88, Z88 Aurora offers a graphic user interface. A completely new preprocessor and an extension of the approved postprocessor Z88O. Developing Z88 Aurora great importance was attached to intuitive operation.

The present version Z88 Aurora 1 offers, in addition to static strength analyses, a material database containing more than 50 established construction materials. Further modules such as non-linear strength calculations, natural frequency analysis, contact and thermal analyses are under development.

THE Z88 PHILOSOPHY IS ALSO VALID FOR Z88 AURORA!

- Fast and compact: Developed for PC, no ported mainframe system
- full 64-BIT support for Windows 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
- FE structure import (*.COS, *.NAS and new: *.BDF, *.ANS, *.INP)
- context sensitive online-help and video tutorials
- simplest installation with *Microsoft® Installer (MSI)*
- Z88 Aurora ist voll kompatibel zu Z88 V13. Bestehende Z88 V13 Dateien können einfach importiert werden!

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 Z88 (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 Z88 Aurora uses the units mm/t/N.

Z88 Aurora is a powerful, complex computer program, which is still in the development phase. Please note that currently not all the functions are implemented, therefore certain functions cannot be selected and the modification of parameters in the user interface to some extent show no effect.

How Z88 deals with other programs and utilities etc. is not predictable! It is the aim of this research version to give you an understanding of the fundamental operating concept. The developers of Z88 Aurora are interested in constantly improving this software. Proposals, suggestions, and remarks can be sent to z88aurora@uni-bayreuth.de. In addition, FAQs are available on the homepage www.z88.de .

LICENSE

Software Products: Z88 Aurora - Software as delivered, ("Software")

Licensor: Chair for Engineering Design and CAD ("LCAD")

This is a legal agreement between you, the end user, and
Chair for Engineering Design and CAD, Universitaetsstr. 30, 95447 Bayreuth, Germany.

By installing, by downloading or by agreeing to the integrated conditions of this End-User License Agreement, you are agreeing to be bound by the terms of this agreement. If you do not agree to the terms of this agreement, promptly return the Software and the accompanying items (including written materials and binders or other containers) to the place you obtained them for a full refund.

1. Grant of license

This LCAD license agreement (license) permits you to use a copy of the Software acquired with this license on any computer in multiple number of installations. The Software is in use on a computer when it is loaded into the temporary memory or installed into the permanent memory (e.g. hard disk, CD ROM, or other storage device) of that computer.

2. Copyright

The Software is owned by LCAD and is protected by copyright laws, international treaty provisions, and other national laws. Therefore, you must treat the Software like any other copyrighted material (e.g. a book). There is no right to use trademarks, pictures, documentation, e.g. without naming LCAD.

3. Other restrictions

You may not rent or lease the Software, but you may transfer your rights under this LCAD license agreement on a permanent basis provided you transfer all copies of the Software and all written materials, and the recipient agrees to the terms of this agreement. You may not reverse engineer, decompile or disassemble the Software. Any transfer must include the most recent update and all prior versions. The Software is for calculation Finite-Element-Structures; there is no warranty for accuracy of the given results.

4. Warranties

LCAD gives no warrants; the Software will perform substantially in accordance with the accompanying documentation. Any implied warranties on the Software are not given.

5. No liability for consequential damages

In no event shall LCAD be liable for any other damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or other pecuniary loss, personal damage) arising out of the use of or inability to use this Software product, even if LCAD has been advised of the possibility of such damages.

7. Governing Law

This Agreement shall be governed exclusively by and be construed in accordance with the laws of Germany, without giving effect to conflict of laws.

SYSTEM REQUIREMENTS

- Operating systems: Microsoft® Windows® XP, Windows Vista™ or Windows 7®, 32- and 64-BIT respectively, MAC OS-X Snow Leopard.
- Graphics requirements: Open GL driver
- Main memory: for 32-BIT 512 MB minimum, for 64-BIT 1 GB minimum
- Documentary and videos require PDF-Reader, Videoplayer, Browser

INSTALLATION


For more information see the installation guide which comes with the installation of the Z88 Aurora package.
















DOCUMENTATION











The Z88 Aurora documentation consists of:

- User manual containing a detailed overview of GUI (Graphical User Interface)
- Theory manual with an elaborate description of the embedded modules
- Examples for the most common applications in mechanical analyses
- Element library of the integrated element types in Z88 Aurora
- Video manual containing some topics of special interest

TABLE OF CONTENTS

1.	AN OVERVIEW OF THE USER INTERFACE	10
2.	MENU BARS	10
3.	KEYBOARD LAYOUT	12
4.	 PROJECT FOLDER MANAGEMENT	13
4.1	 Launching a New Project Folder	13
4.2	 Opening a Project Folder	14
4.3	 Closing a Project Folder	15
4.4	Project Folder Management in the Text Menu Bar	15
5.	VIEW	16
5.1	Toolbars	16
5.2	 Camera Settings	17
5.3	 Colours	17
5.4	 Displays	17
	 Picking	18
5.5	 Views and View Options	27
5.6	 Labels	29
	 Labelling: Picking nodes	29
	 Labelling: Nodes	29

	 Labelling: Elements	29
	 Labelling: Nodes and Elements	29
	 Labelling Nothing: Nodes and Elements	29
5.7	 Size of Boundary Conditions / Gauss Points/ Pick-Points	30
	 Size of boundary conditions	30
	 Size of Gauss Points	30
	 Size of Pick-Points	30
	Toolbar Views	30
6.	 CONTEXT SENSITIVE SIDE MENUS	31
6.1	 Import and Export of CAD and FE Data	31
	Importing	31
	Exporting	34
	Import/Export in the Text Menu Bar	35
	Toolbar Import/Export	35
	 Exporting an Image	35
6.2	 Preprocessor	36
	Preprocessor in the text menu bar	37
	Toolbar preprocessor	37
	 Creating FE Structures: Trusses/Beams	37
	 Meshing	40
	 Prove Mesh	42
	 Element parameters	43

	 Material	44
	 Applying Boundary Conditions	48
6.3	 Solver	54
	Option "Cuthill-McKee Algorithm" in "Extended Options" in the Solver Menu	57
	The Solver in the Text menu bar	58
6.4	 Postprocessor	59
6.5	 Help, Support and Option settings	62
	 Help	62
	 Examples	63
	 Information	67
	 Support	68
	 Option Settings	68

1. AN OVERVIEW OF THE USER INTERFACE

Z88 Aurora is characterised 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.

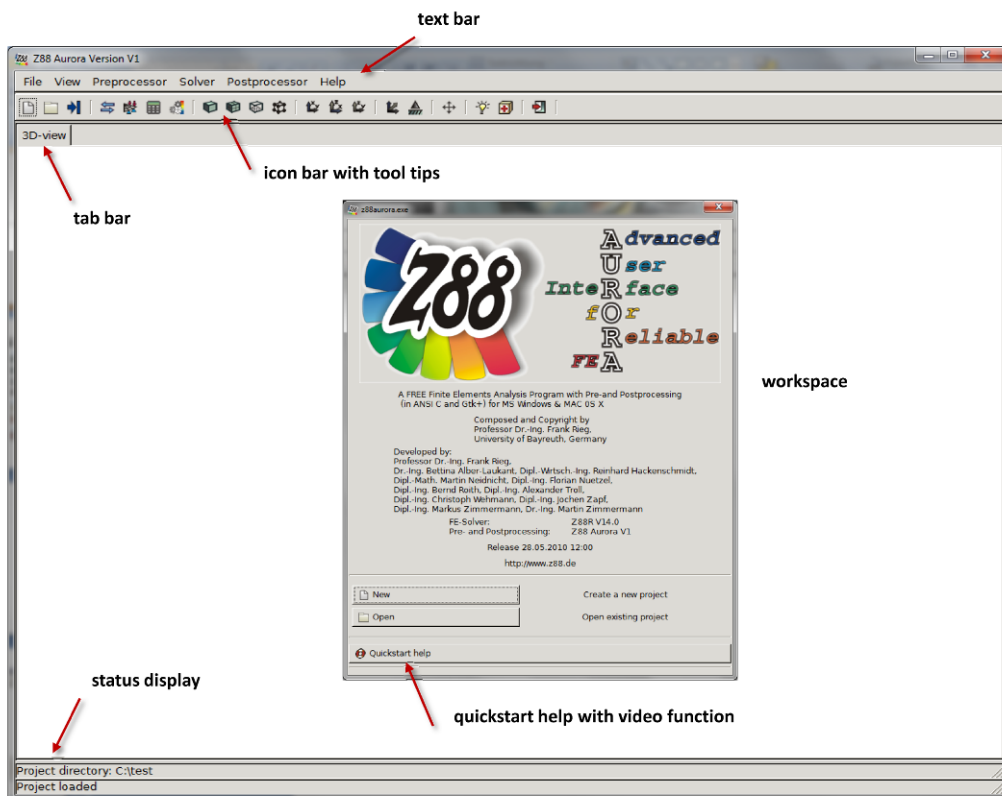



Figure 1: User interface of Z88 Aurora

2. MENU BARS

Several menu bars are of importance for operation. The icon menu bar provides quick access to all functions of Z88 Aurora. The main functions of the icon menu bar, such as preprocessor , open additional side menus. 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 statistics function in the postprocessor or the material cards in the material menu, between which you can switch.

The icon menu bar is separated into different areas: the project folder management, the pushbuttons, which access the context sensitive side menus, the display options, the views and the view options, the scaling, and the support.

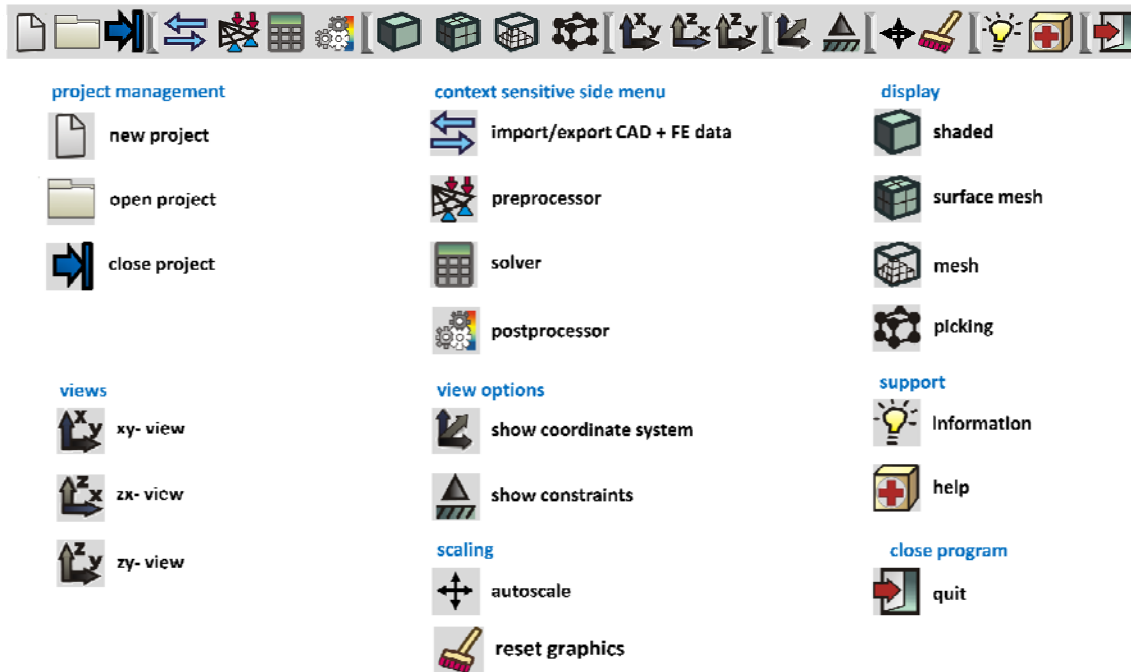



Figure 2: Pushbuttons of the icon menu bar

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

3. KEYBOARD LAYOUT

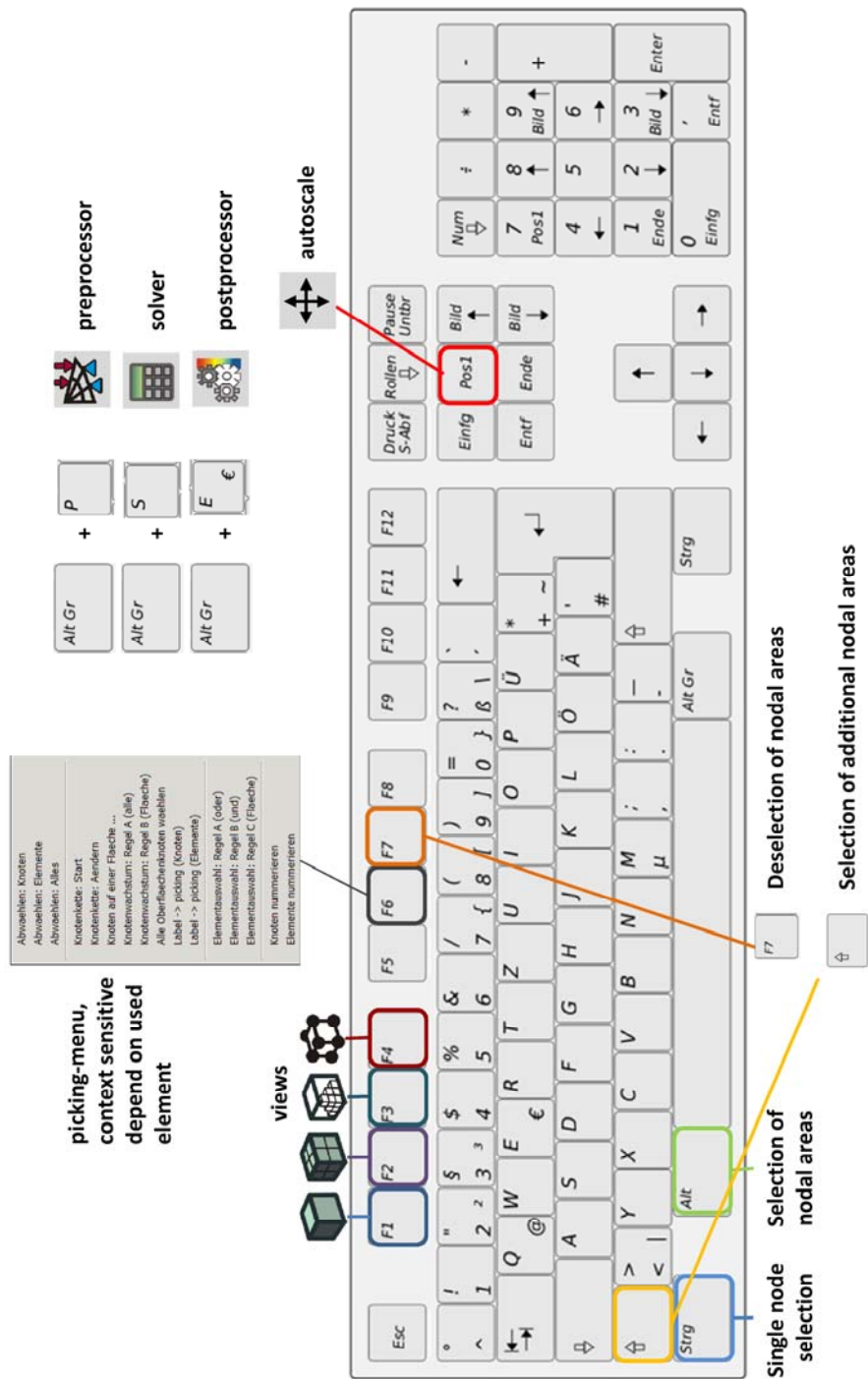


Figure 3: Keyboard layout

4. 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. In the first version of Z88 Aurora, one kind of simulation is possible, the linear strength analysis.

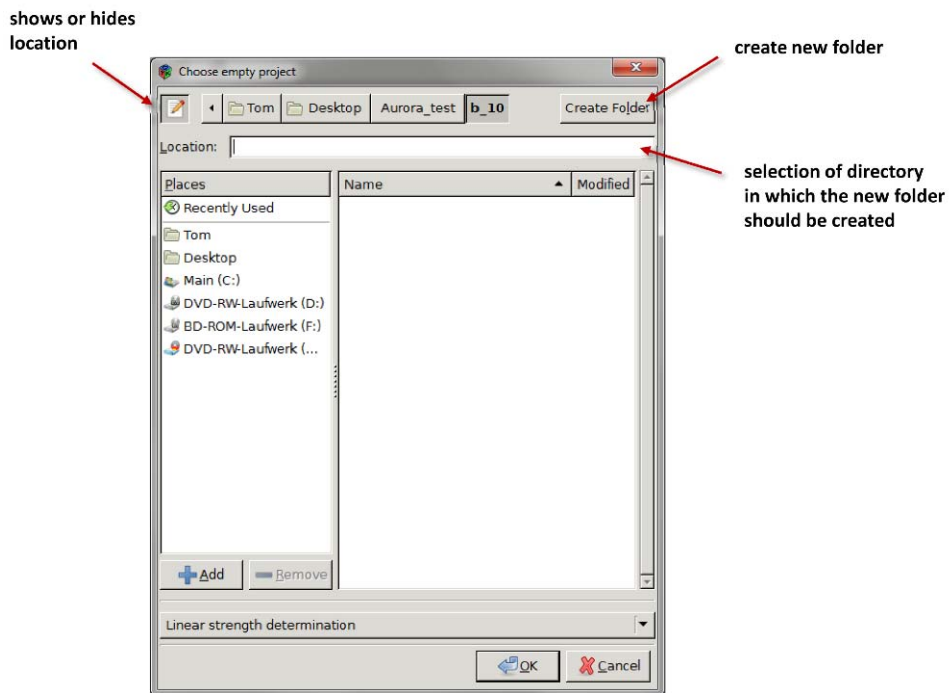


Figure 4: Project folder management of Z88 Aurora

4.1 Launching a New Project Folder

⇒ Create a new folder

⇒ Enter folder name „Name“

⇒ Confirm  (Return) and *double click*  (left mouse button) to activate folder

⇒ Click OK to confirm

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

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

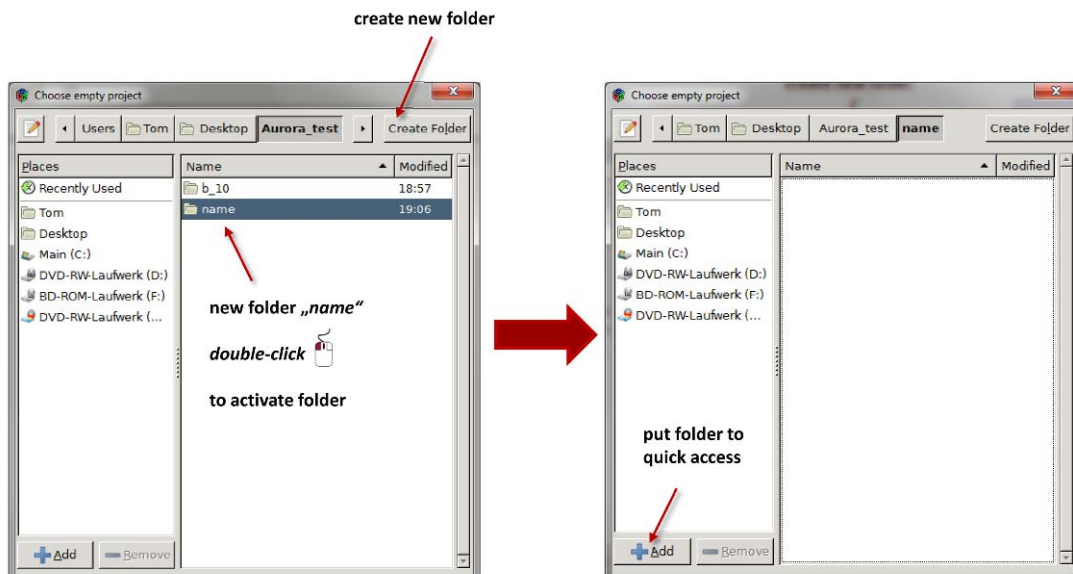


Figure 5: Launching a new project folder

4.2 Opening a Project Folder

- ⇒ Select a project folder to open
- ⇒ Select Project information, all relevant data about the project folder are shown
- ⇒ Click "OK" to confirm. The project is displayed in the work area.

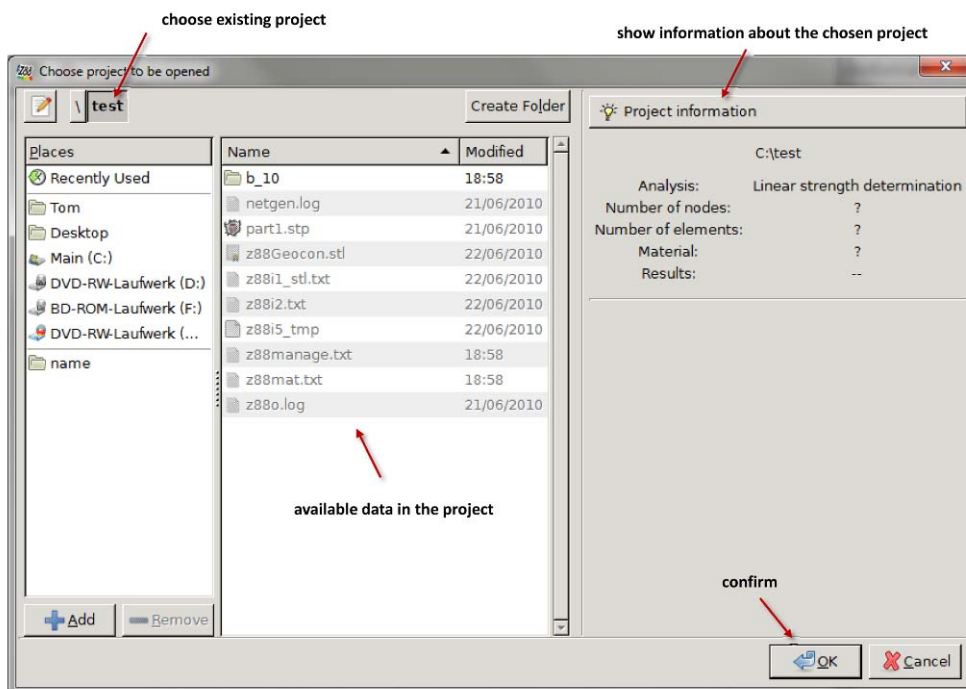



Figure 6: Opening an existing project folder

4.3 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!**

4.4 Project Folder Management in the Text Menu Bar

In addition to the icon menu bar, Z88 Aurora 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.

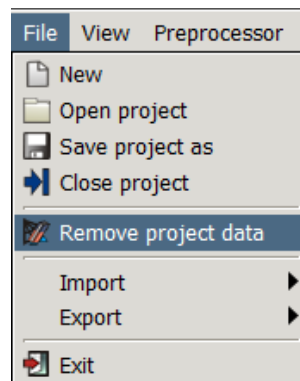


Figure 7: Project folder management in the text menu bar

5. VIEW

The view display can be edited in many ways in Z88 Aurora. In addition to the functionalities of Z88 V13 it is possible to display often required tool bars, to change the background and legend colour or to switch miscellaneous additional view options on and off.

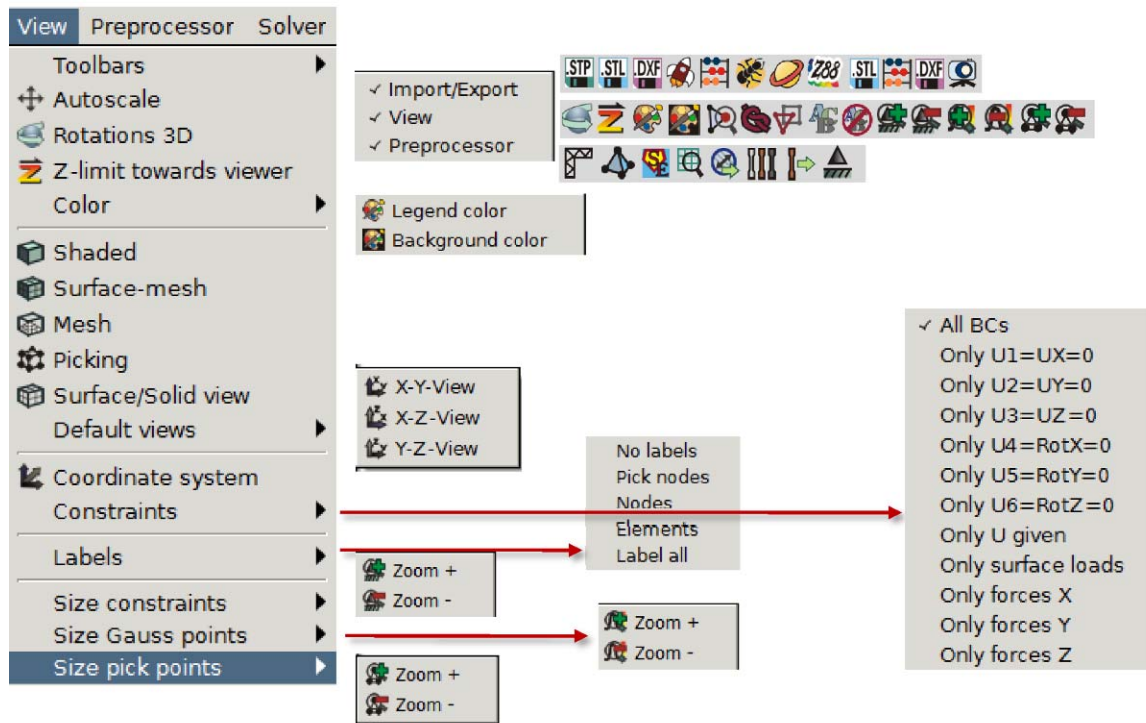


Figure 8: View options

5.1 Toolbars

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

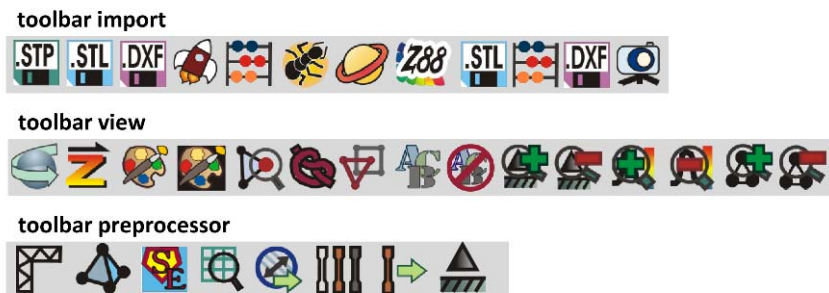





Figure 9: Toolbars

5.2 Camera Settings

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

5.3 Colours

The legend colour as well as the background colour 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 colour.

5.4 Displays

There are four possibilities of view display. These can be accessed via the icons in the icon menu bar or with the keys F1-F4.

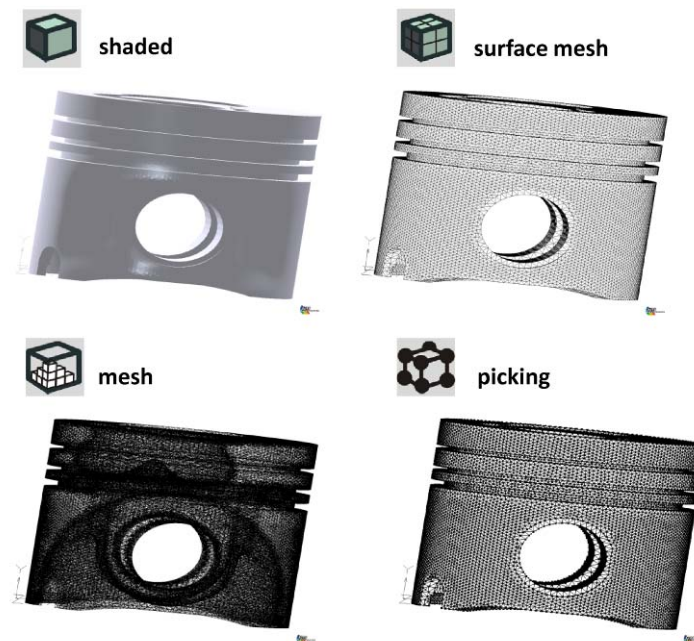



Figure 10: Display options in Z88 Aurora

The display modes shaded, surface mesh and mesh can be applied by the user according to his needs; the Picking display is used for the selection of nodes and elements. It is used in the boundary conditions menu; in addition, it can execute further functions via the button



(see next chapter).

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

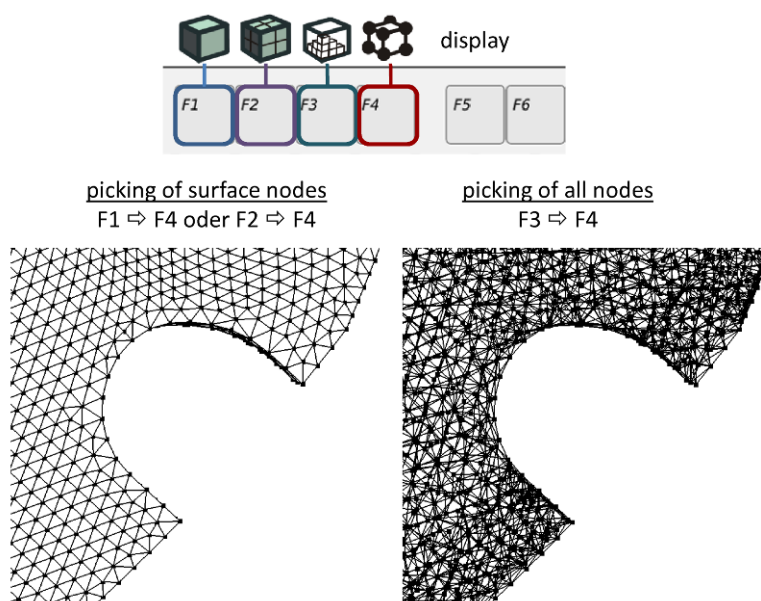












Figure 11: Switching to the display option "Picking"

Picking


One of the main innovations of Z88 Aurora is the possibility to apply boundary conditions, such as forces, pressures and fixations on the new, graphic user interface with one mouse click. Hereafter, this functionality will be referred to as "Picking". For the Picking there is a separate view, which you can display in the main window by clicking the button  or by pressing the key .

Short Cuts

By means of the mouse and a few shortcuts it is possible to "pick" single or several nodes, in order to define the designated boundary conditions:

-  +  (click) Selection of single nodes
-  +  (hold) Selection of several nodes in a rectangular window while maintaining the previous selection
-  +  (hold) New Selection of several nodes in an area while discarding the previous selection
-  +  (hold) Opening a rectangular window to deselect several nodes in an area

Picking Menu

Pushing the button  opens the "Picking menu". Depending on the current element type, different possibilities are offered to select single or several nodes (Figure 12).

The respective picking menus are divided into five parts:

- **Deselecting**
- **Node selection**
- **Element selection**
- **Hiding**
- **Numbering**

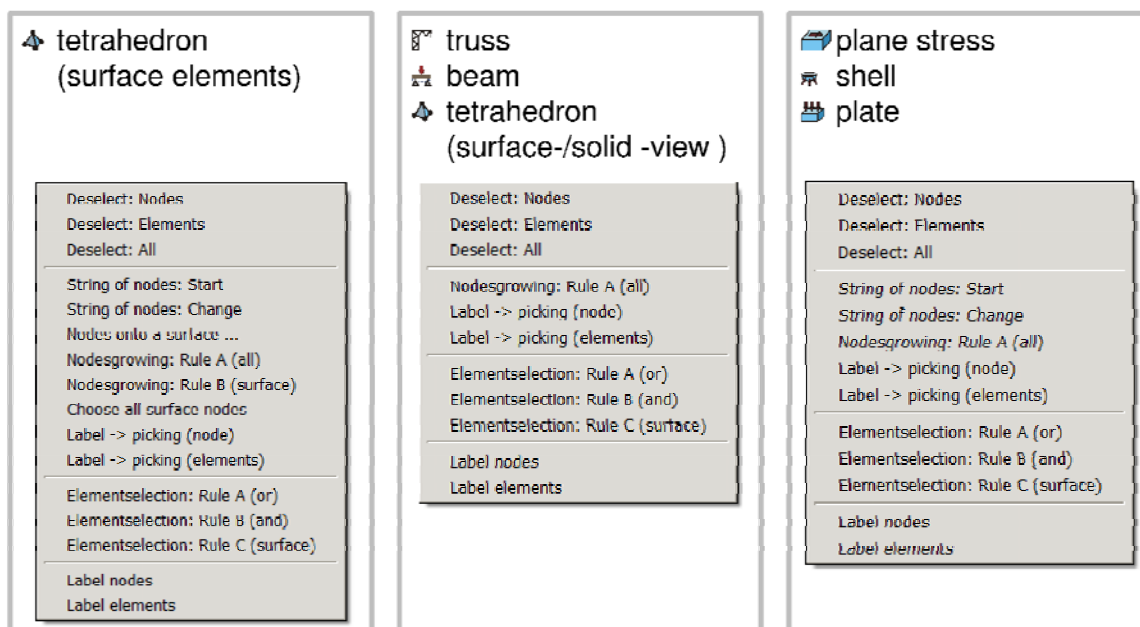


Figure 12: Picking menu

Deselecting

The "Deselecting" area includes the following sub items:

- *Deselect: Nodes* selected nodes are deselected
- *Deselect: Elements* selected elements are deselected
- *Deselect: All* selected nodes and elements are deselected

Node Selection

The "Node Selection" area contains the following functions:

- *Node chain*

A "node chain" is a group of adjacent nodes running along the edge of an FE model. By means of this picking option it is possible to select edges of drill holes or circumferential edges of a profile. The following options are possible:

- *Node chain: Start*
Starts a node chain along the edge circumferentially or between two selected points.
- *Node chain: Alter*
Switches between the different alternatives of the edge connection between two selected points successively.

The single possibilities are illustrated in Figure 13.

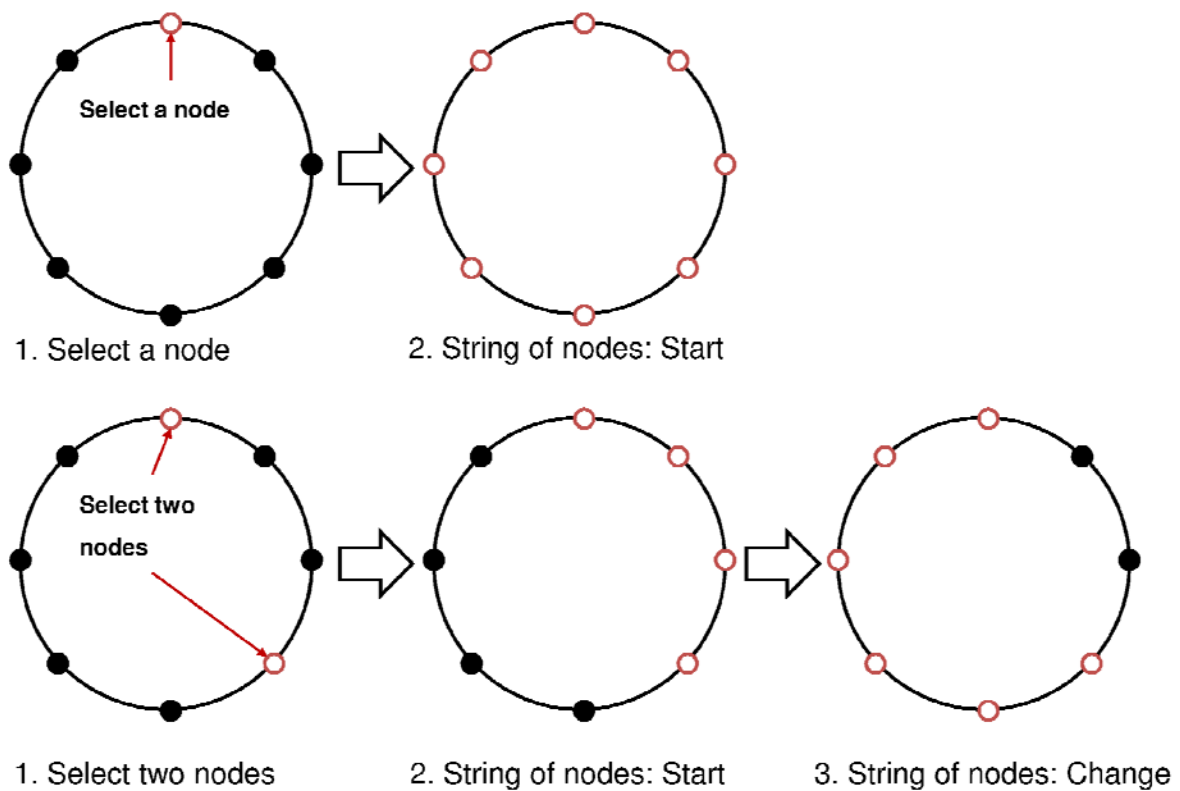


Figure 13: Node chain

- *Nodes on an area*

If you want to select, for example, the inside surface of a drill hole for the application of boundary conditions, you can use the function "Nodes on an area" (Figure 14).

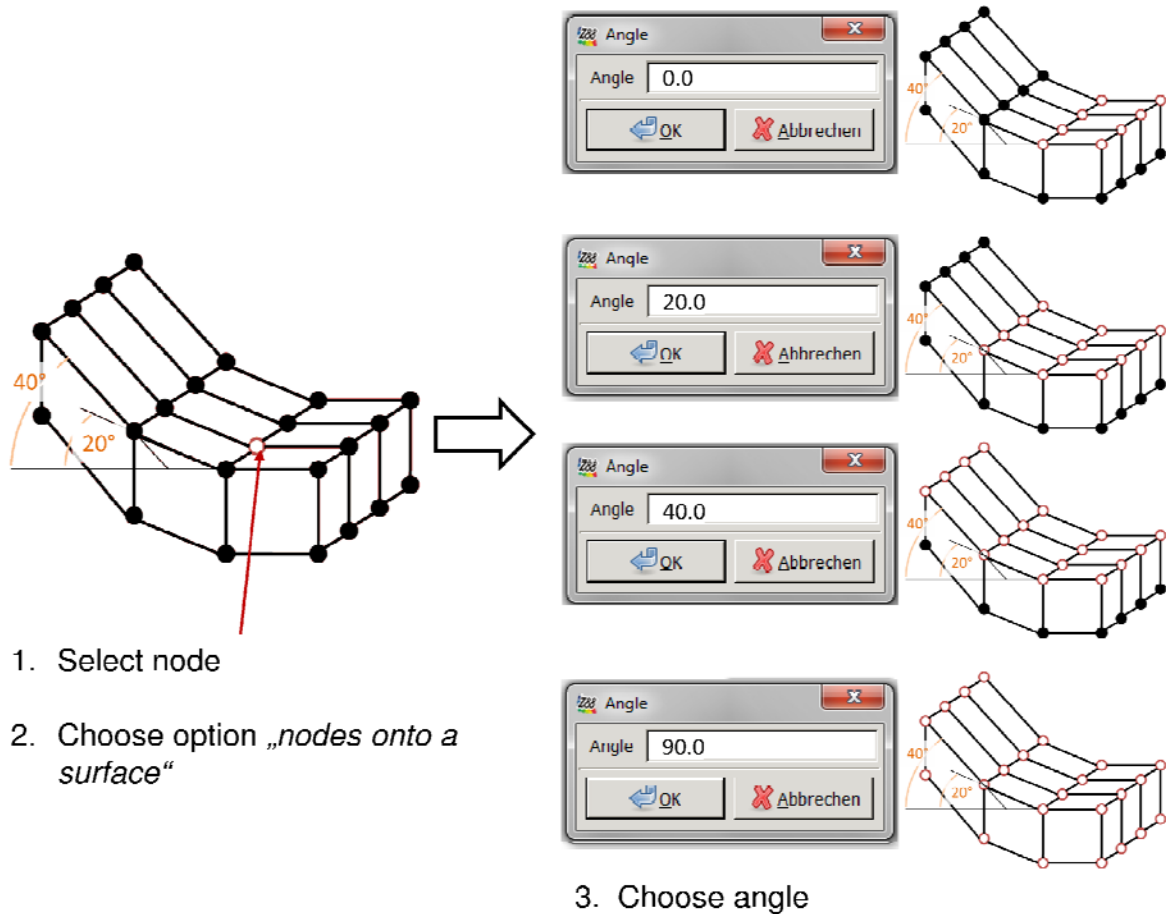




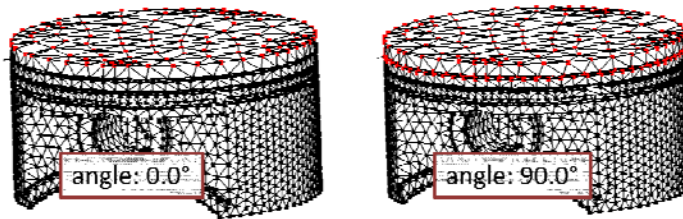
Figure 14: Nodes on an area

Pick a node with  +  (click). Now select the function mentioned above; an input mask appears, where you have to enter an angle and confirm the dialog with "OK". This value describes the angle between the element containing the selected node and the adjacent elements. If the value is less than or equal to the one you entered, the nodes of the respected elements are selected.

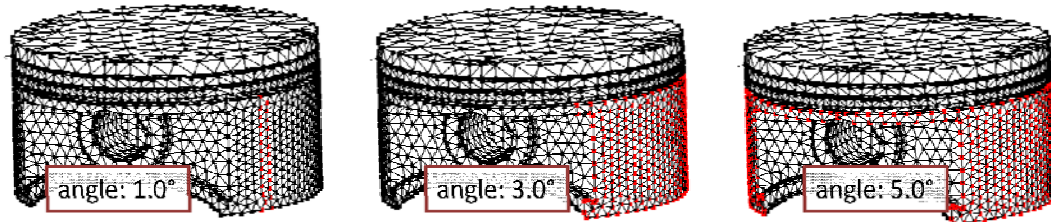
To find the appropriate settings for your designated area, you may have to try several values before you achieve the desired result. The following settings serve as reference values (Figure 15):

- Plane area: 0.0°
- A double row of nodes on a large curvature radius: $1^\circ - 2^\circ$
- Lateral surface (partial or complete) of a large curvature radius: ca. $5^\circ - 10^\circ$
- Inside surface of drill hole: ca. $10^\circ - 20^\circ$

Picked node on a plain surface



Picked node on a curved surface



Picked node in the drill hole

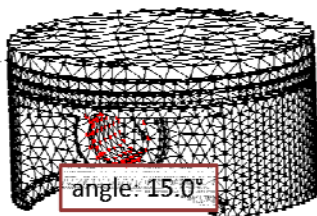



Figure 15: Angle settings

- *Node growth*

The "node growth"-algorithm calculates the nodes which are adjacent to selected elements (see "Element selection").

• *Node growth: rule A (all)*

With "Node growth: rule A (all)", for example, contact surfaces of bolts adjacent to the drill hole can be comfortably selected by means of a suitable combination of "Node chain", "Node growth" and "Element selection" (Figure 16).

 If you do not achieve the desired result, please note the information in the status display below the work area

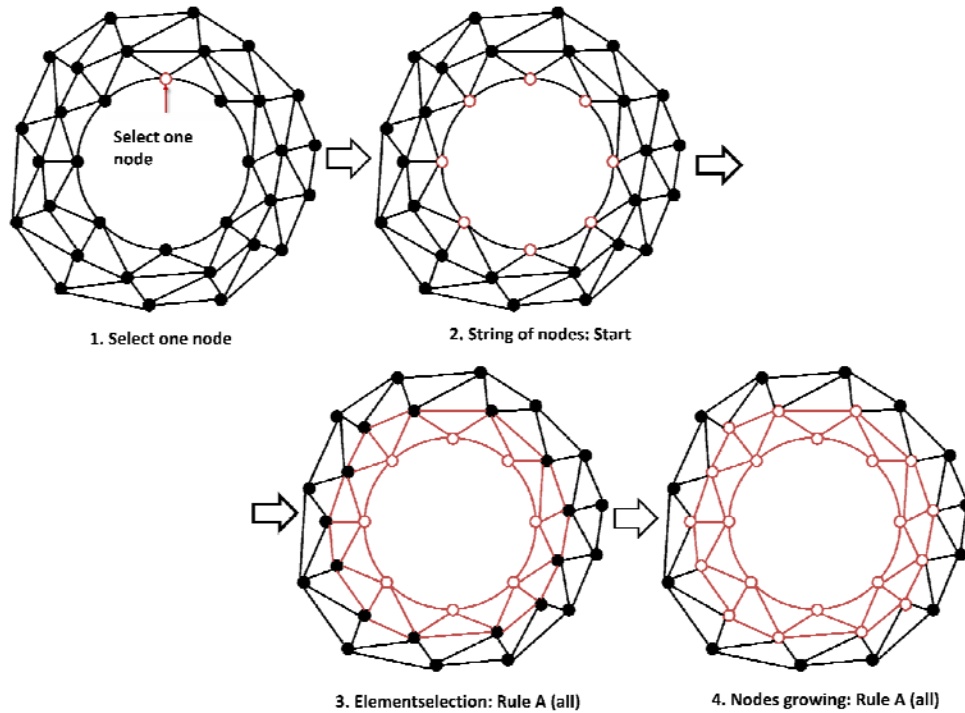


Figure 16: Node selection: Rule A (all)

- *Node growth: Rule B (area)*

With this function a range of nodes on an area can be selected. This option, too can be combined with others, thus selecting the respective surface elements (Figure 17).

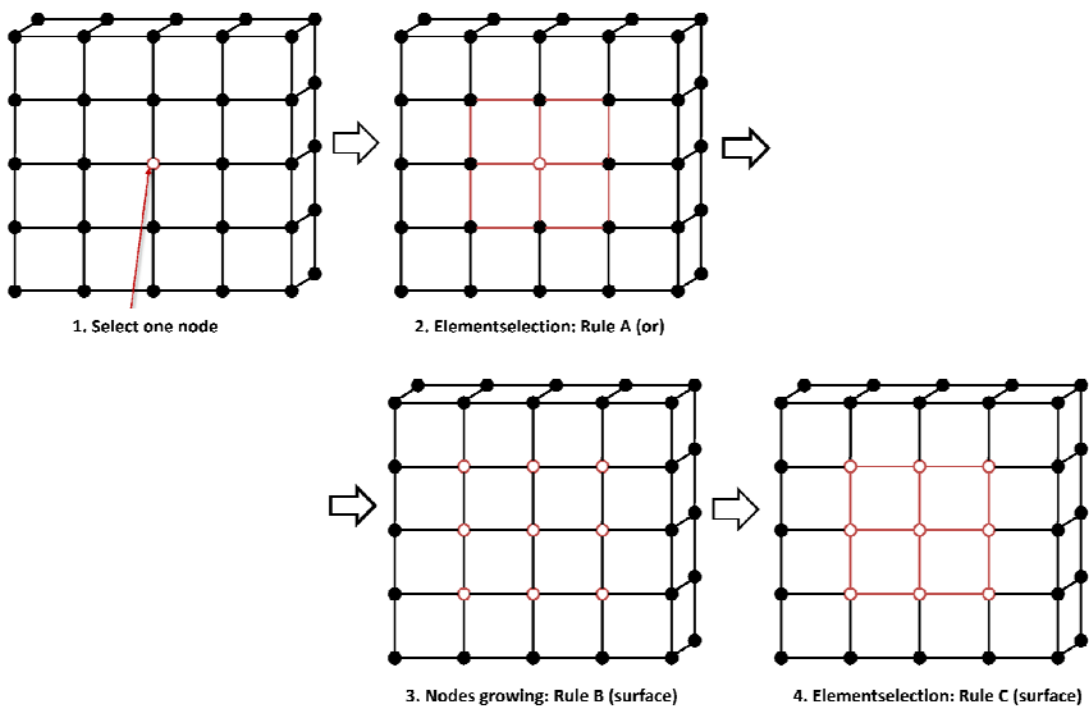


Figure 17: Node growth rule B (surface)

- *Selecting all surface nodes*

The option „*Selecting all surface nodes*“ selects all nodes on the surface; this way, hydrostatic pressures, for example, can be applied to a component (Figure 18).

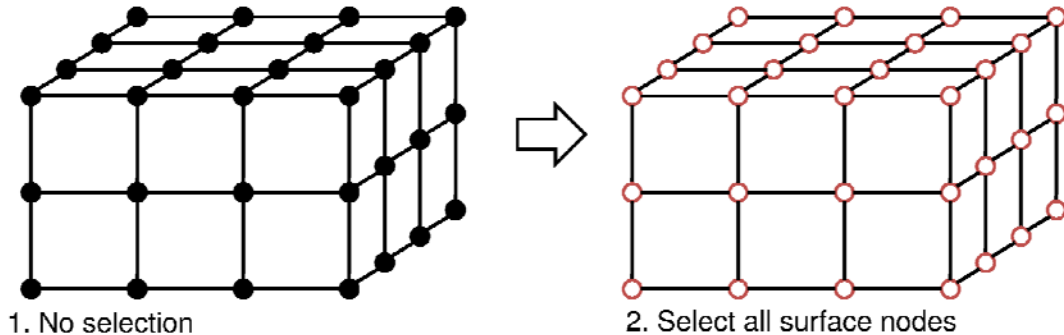


Figure 18: Selecting all surface nodes

- *Labels*

The functions "*Label -> picking (nodes)*" and "*Label -> picking (elements)*" effect the corresponding node/element numbers of the selected objects to be displayed.

- *Label -> picking (Nodes)*

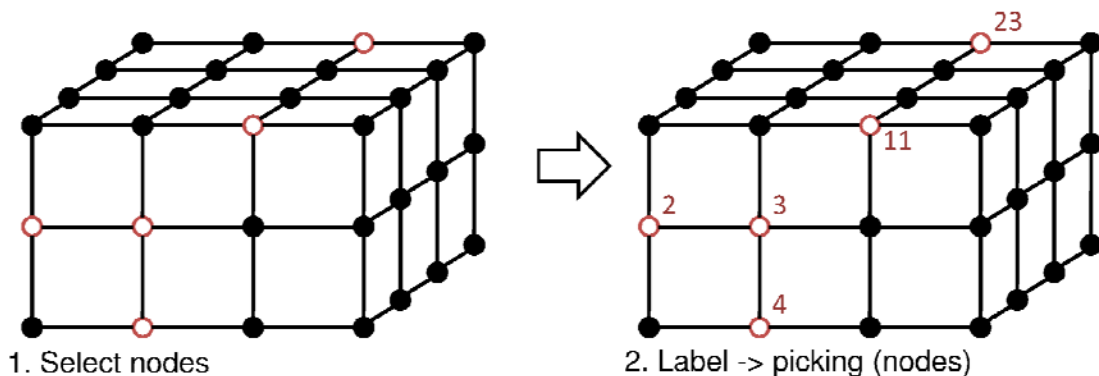


Figure 19: Labelling of nodes

- *Label -> picking (Elements)*

Element Selection

For the selection of elements adjacent to the selected nodes, the following options are available in the "*Element Selection*"-area:

- *Element selection: rule A (or)*

With rule A (or) all elements directly adjoining the picked node(s) are selected. The remaining nodes belonging to the elements are not selected. With "*Node growth: rule A (all)*" these can be added later (see "*Node growth*").

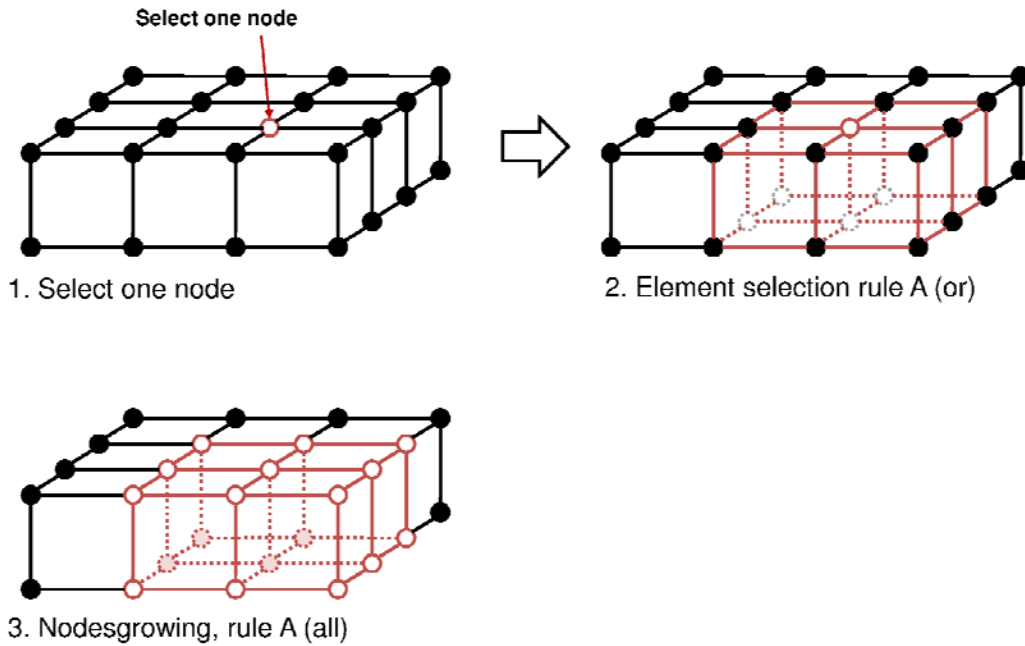


Figure 20: Element selection: rule A (or)

- *Element selection: rule B (and)*

In contrast to "rule A (or)", when applying "rule B (and)" all nodes included in the element must be selected, in order to mark the element completely (Figure 21).

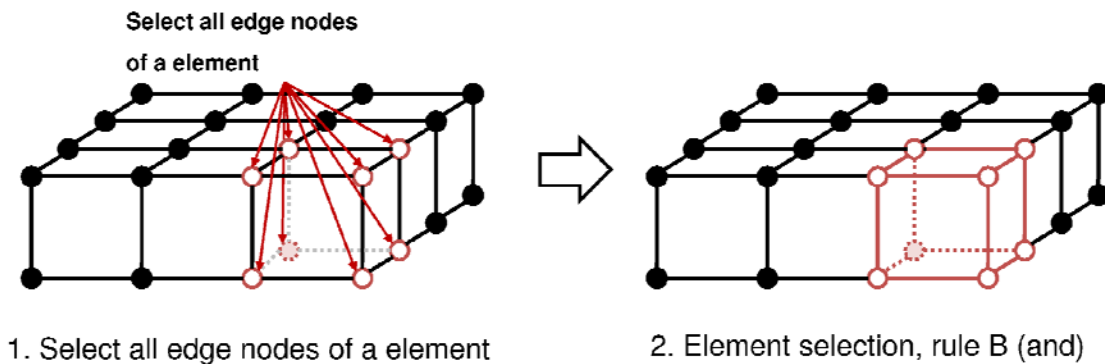


Figure 21: Element selection: Rule B (and)

- *Element selection: rule C (area)*

Using this function, the nodes adjoining an area must be selected in order to select the area completely (Figure 22).

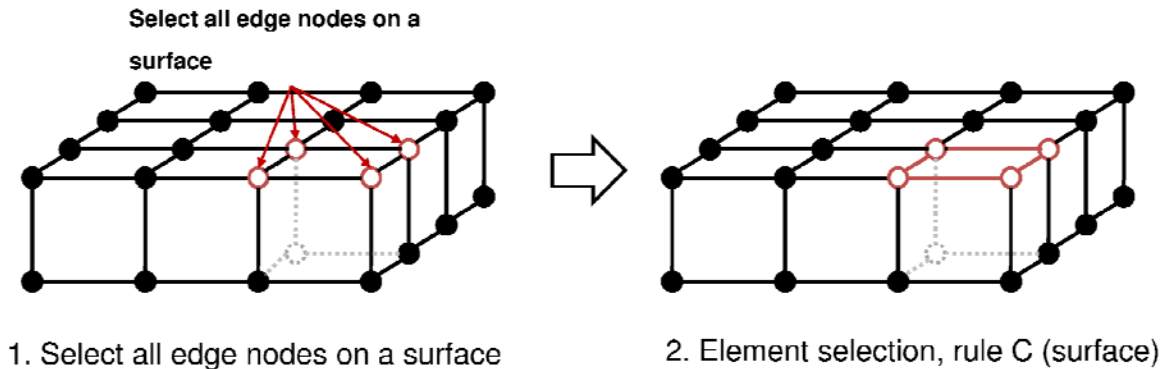





Figure 22: Element selection: Rule C (area)

Hiding

- *Hide: Selected elements*
- *Hide: Not selected elements*

With this functionality you can hide and show elements of the FE mesh in pre- as well as with postprocessor. This can be especially helpful when applying boundary conditions to hidden edges and recesses; furthermore, it can facilitate the evaluation of the displacements, stresses etc. in the postprocessor.

For example, you can create a half section as follows (Figure 23):

- Switch to the picking view with  by pressing the key F4. There you open a window with  +  (*hold*) in order to select the desired node area.
- With F6 you access the picking menu and select the item "Element selection: rule A (or)". Thus, all elements adjoining the selected points are selected.
- Now access the Picking menu again with F6 to find the "Hide menu" activated there. Use "Hide: Selected elements", to suppress the selected elements; "Hide: Not selected elements" hides all other elements.
- If you want to show the elements again, you have to remove the check mark in front of the previously selected option in the picking menu (F6). To access the selection again, you can mark the option once more.

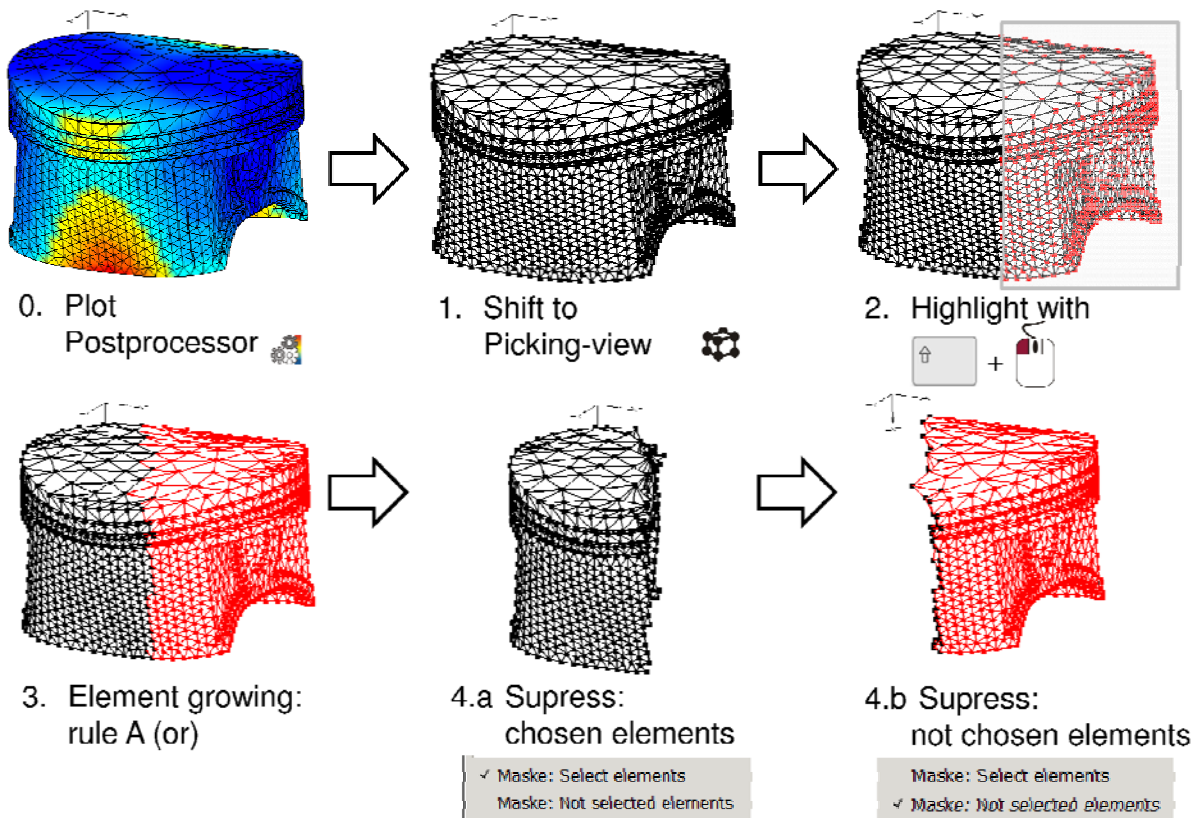


Figure 23: Hiding elements

The menu "Hide" is only activated, if you have already selected some elements.

Numbering

The "Numbering" menu includes the following sub items:

- *Number nodes* Label nodes
Node numbers of selected nodes are shown / hidden.
- *Number elements* Label elements
Element numbers of selected nodes are shown / hidden.

5.5 Views and View Options

Several predefined views with standard orientations are available in Z88 Aurora (see Figure 24). The boundary conditions as well as the coordinate system can be shown or hidden at your choice. For further information about showing the boundary conditions see chapter " Applying Boundary Conditions".

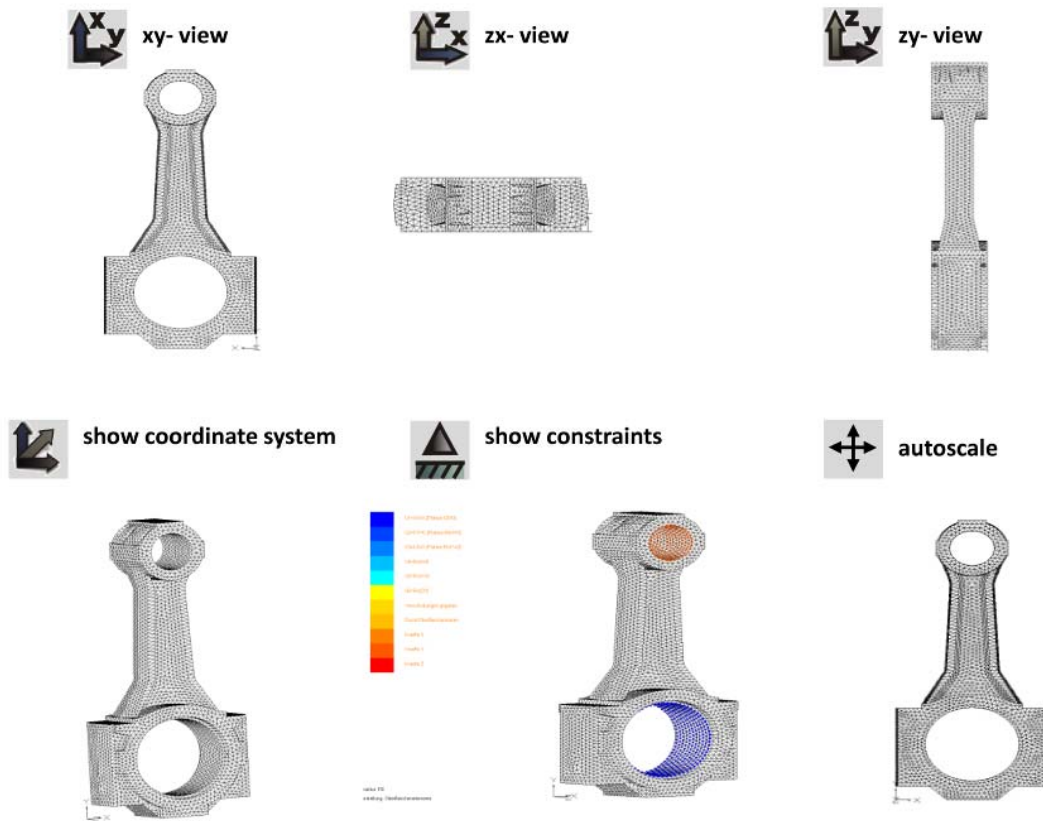


Figure 24: View options in Z88 Aurora

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

5.6 Labels

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

Labelling: Picking nodes

Nodes selected by picking are provided with their corresponding node numbers (see "Picking → Label -> picking (nodes)").

Labelling: Nodes


A window appears in which the numbers of the desired nodes must be entered, in order to display them. The dialog is ended with "OK".

Labelling: Elements

Analogously to "Labels → nodes", here, too, the desired element numbers must be entered, in order to display them.

Labelling: 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.**

Labelling Nothing: Nodes and Elements

This function hides the labels of all nodes and elements.

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

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.

Size of Gauss Points

With the menu item "Size of Gauss points" the size of the calculated Gauss points, depicted here in the Z88 Aurora postprocessor, is defined (Figure 25).

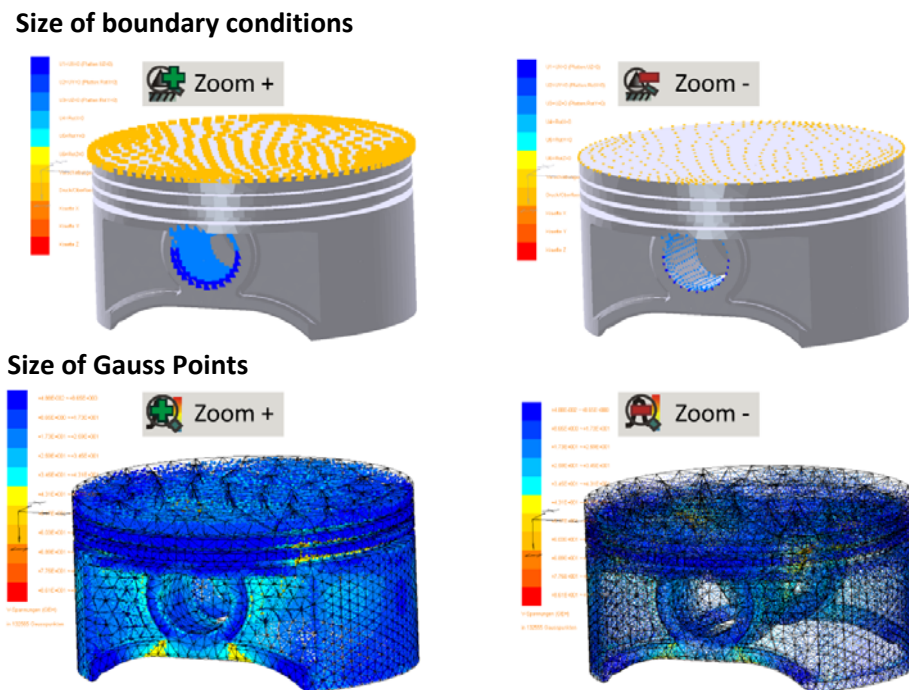


Figure 25: Display of boundary conditions and Gauss points

Size of Pick-Points

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

Toolbar Views

In the menu "Help" under "Options" you find the tab "View" where the possibility to display

the toolbar "Views"  is offered.

6. CONTEXT SENSITIVE SIDE MENUS

When you have started a project, you can perform different actions. On the one hand, you can display and alter an existing project, on the other hand, you can import a structure from a CAD program as well as from an FE program.

6.1 Import and Export 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 26.

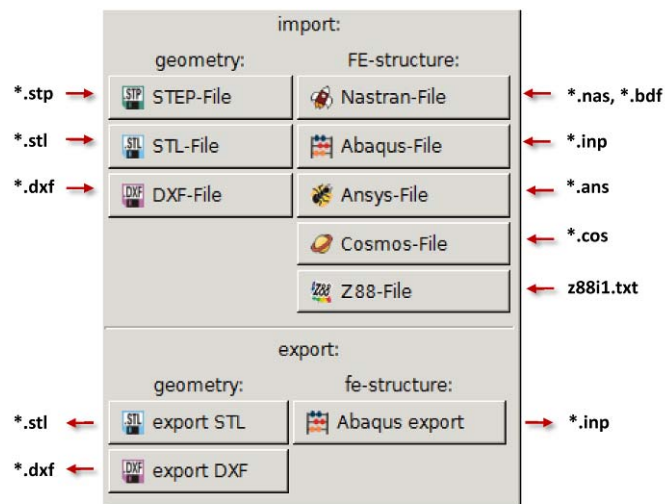


Figure 26: Import and export options in Z88 Aurora


Importing

As an example, the import procedure of a STEP file is demonstrated (Figure 27) :

⇒ Select Import/Export

⇒ Click " STEP file", a selection window is opened

⇒ Select file

⇒ Click OK to confirm 

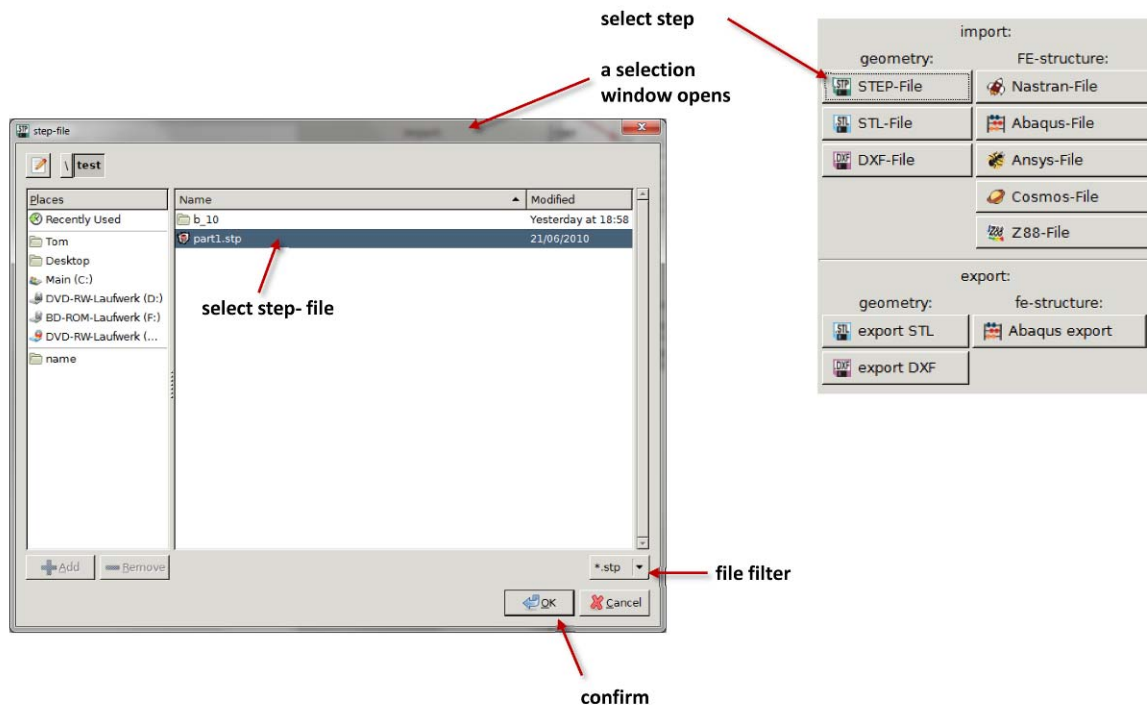


Figure 27: Import of a STEP file

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

- ⚠ **The default setting, which input file is supposed to be imported, is defined by the user (Figure 28).**
- ⚠ **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 Z88 Aurora.**

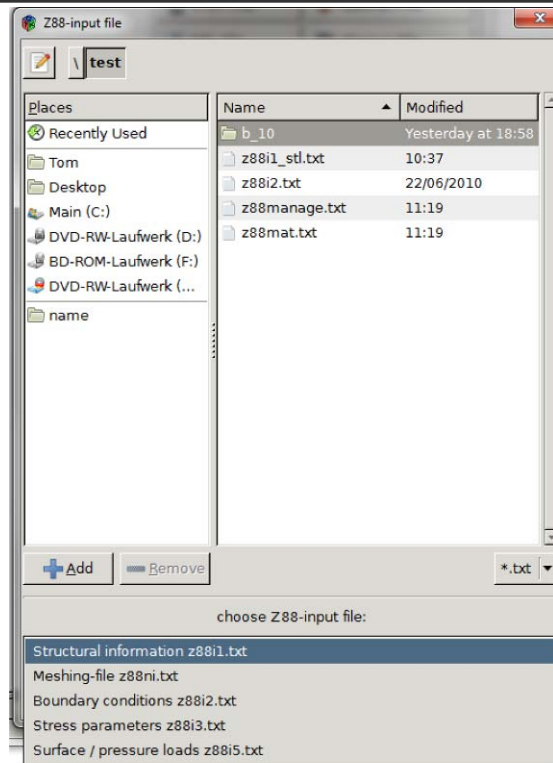








Figure 28: Import function for existing Z88 files

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

	Z88V13 	DXF 	ABAQUS 	ANSYS 	COSMOS 	NASTRAN 
FE structure	✓	✓	✓	✓	✓	✓
FE super structure	✓	✓	✗	✗	✗	✗
Material laws	✓	✓	✓ (1 material)	✓	✓	✓
Point loads	✓	✓	✓	✓	✓	✓
Supports	✓	✓	✓	✓	✓	✓
Surface loads	✓	✓	✓ only import	✓	✓	✓
Solver options	✓	✗	✗	✗	✗	✗

DXF-files can be imported as four different file types (Figure 29).

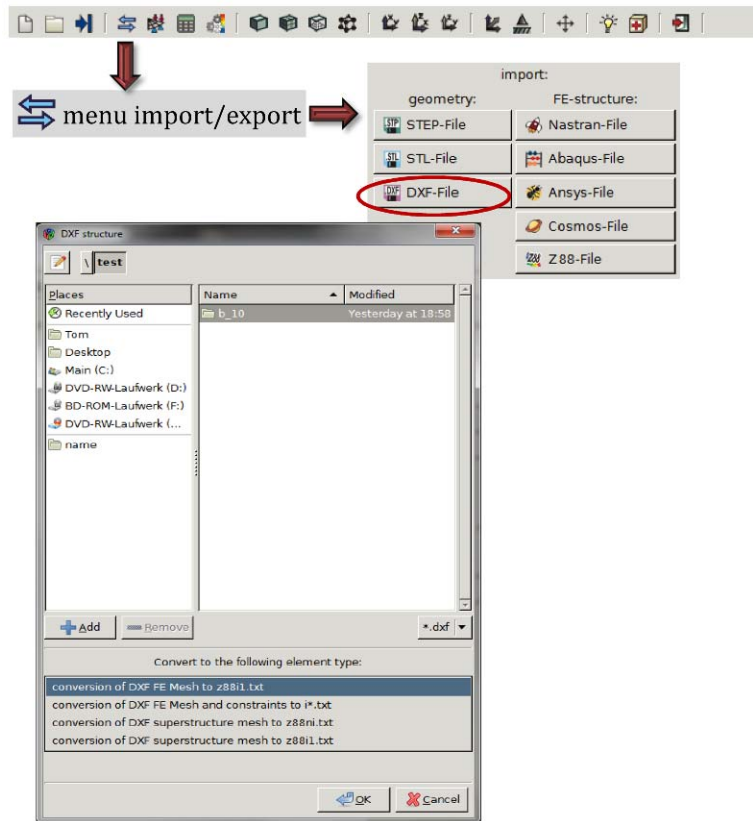


Figure 29: Import alternatives DXF-structure

Exporting

The export is done by selecting the desired export option (Figure 30).

⚠ For the export of an FE structure a complete FE model must be present!

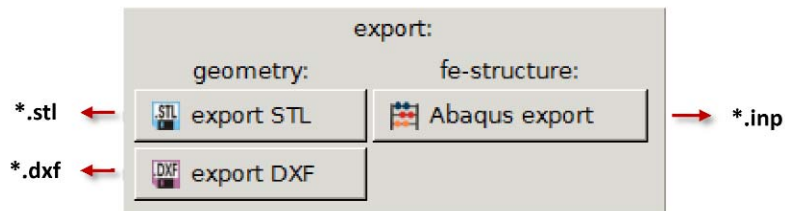


Figure 30: Export options

Import/Export in the Text Menu Bar

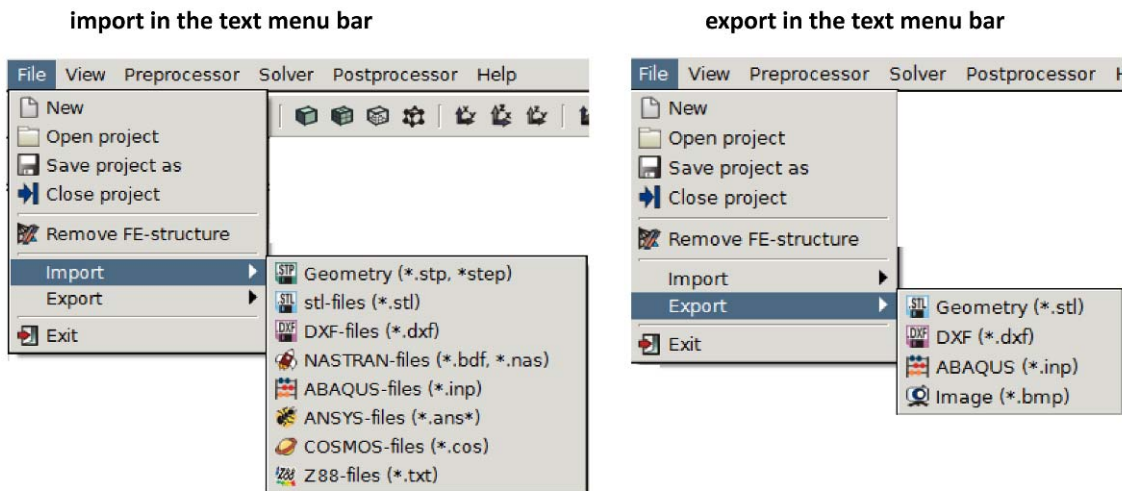


Figure 31: Import/Export in the text menu bar

Toolbar Import/Export

In the menu "Help" under "Options" you find the tab "View" where the possibility to display

the toolbar "Import"  is offered.

Exporting an Image

A plot within Z88 Aurora can be exported as an image (*.bmp) at any time. For this, either click the icon in the Import/Export toolbar or in the file menu (see Figure 32).

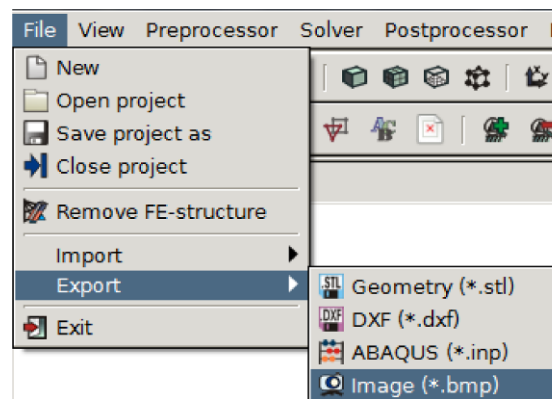



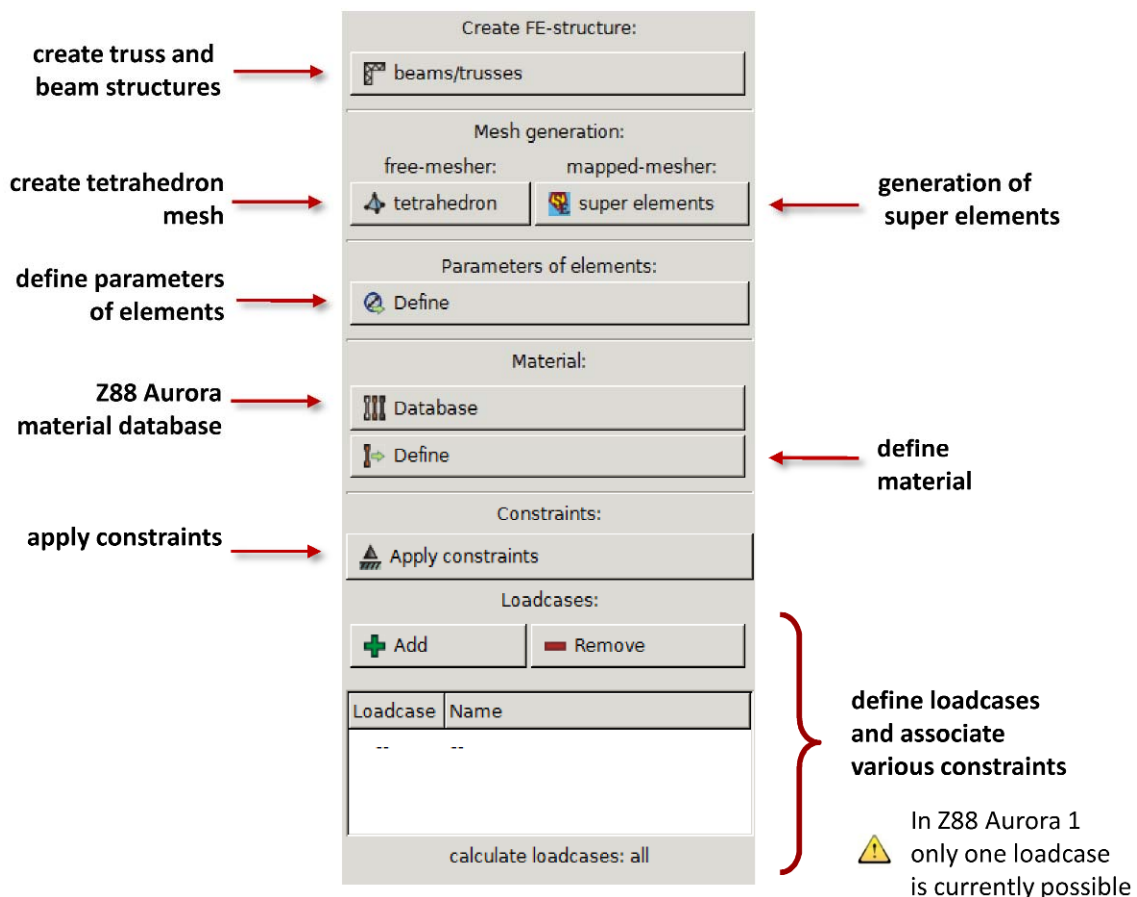
Figure 32: Exporting a picture

 **For the export of a picture, the side menu must always be closed. This is done by clicking the respective symbol of the icon menu bar again!**

6.2 Preprocessor

Clicking the preprocessor icon opens the context sensitive side menu "Preprocessor" (Figure 33). You can either create an 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. In later versions of Z88 Aurora, it will be possible to allocate different boundary conditions to certain load cases and to display them separately in the postprocessor. At the moment, Z88 Aurora is limited to one load case

All possibilities of the preprocessor are introduced separately below.



create truss and beam structures →

create tetrahedron mesh →

define parameters of elements →

Z88 Aurora material database →

apply constraints →

generation of super elements ←

define material ←

define loadcases and associate various constraints ←

⚠ In Z88 Aurora 1 only one loadcase is currently possible

Figure 33: Side menu „Preprocessor“

Preprocessor in the text menu bar

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

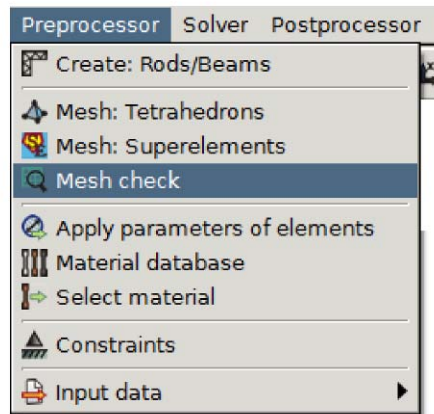



Figure 34: Text menu bar „Preprocessor“

As an additional function, the text menu bar offers the function "prove mesh" for quality checks of imported or created meshes, and the function "input data", where the Z8811.TXT-Z8815.TXT entries can be viewed.

Toolbar preprocessor

In the menu "Help" under "Options" you find the tab "View" where the possibility to display the toolbar "Preprocessor"  is offered.

Creating FE Structures: Trusses/Beams

Like in Z88 V13, it is possible in Z88 Aurora, to create and calculate truss and beam structures.

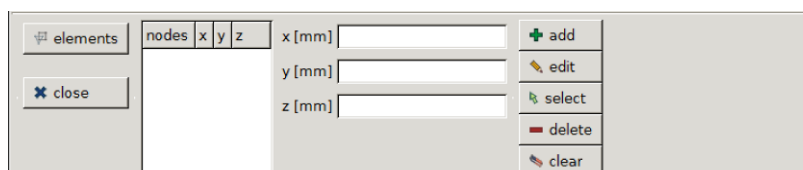
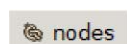


Figure 35: Creation truss/beam; menu "Nodes"

⇒ creating 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 by means of the mouse or via the selection from the list.


Selection via mouse:

⇒  + **Strg** + the node to be edited => the node turns red

⇒ with  the node is selected in the table for further editing

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

Selection from the list:

⇒  + select the node to be edited from the list => the node turns red

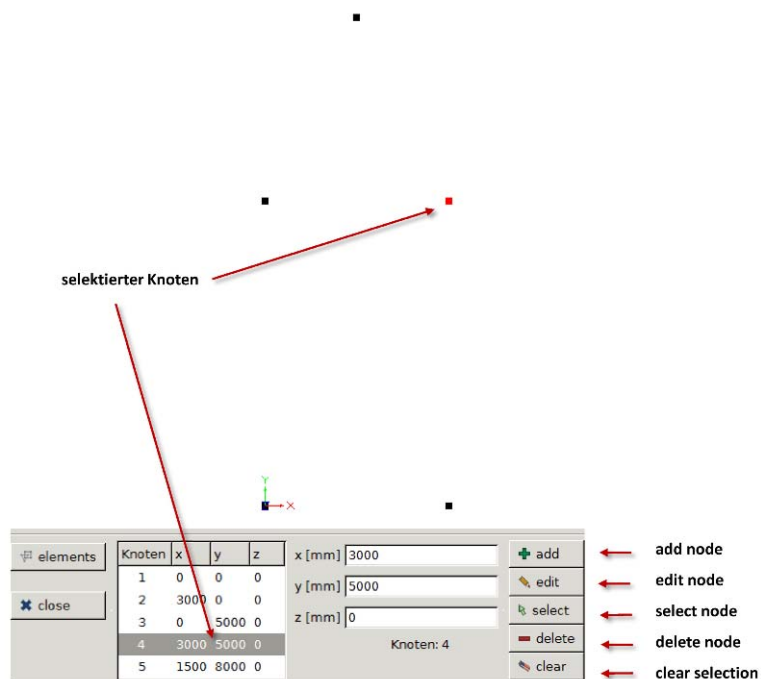


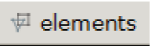


Figure 36: Selected node in the truss/beam menu

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  .

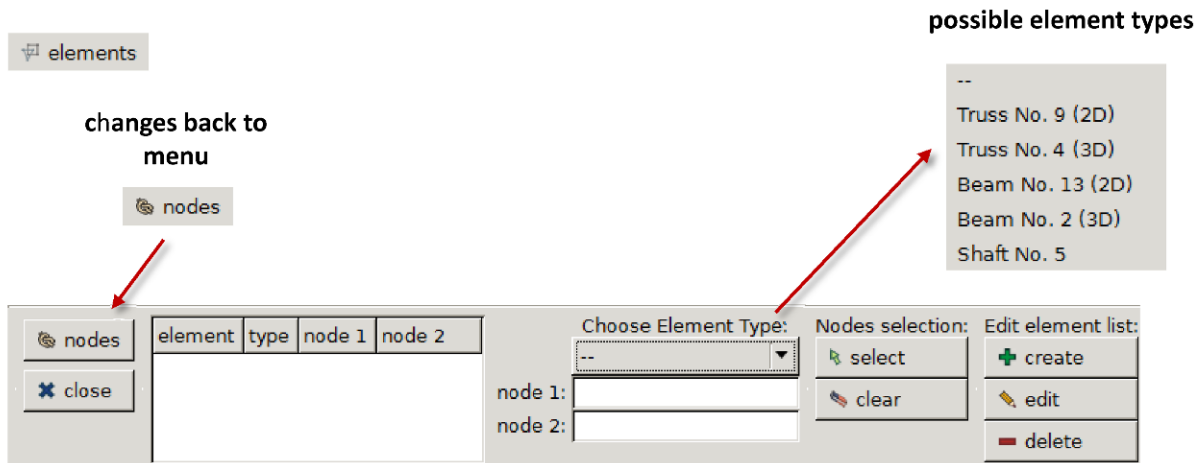


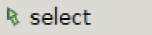
Figure 37: Truss/beam creation; menu "Elements"

⇒ create new element

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

for further information please see Theory Manual chapter 5

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

⇒ click 

⇒ click 

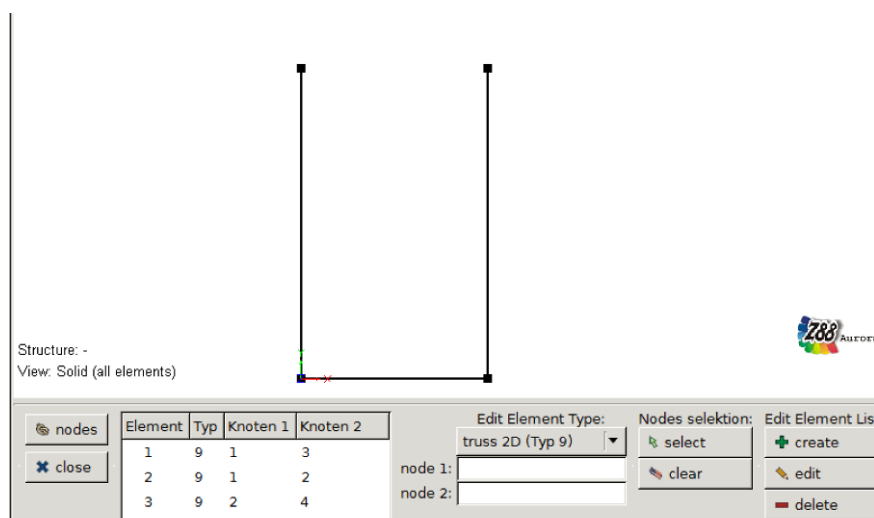


Figure 38: Truss/Beam creation; creating an element list

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.

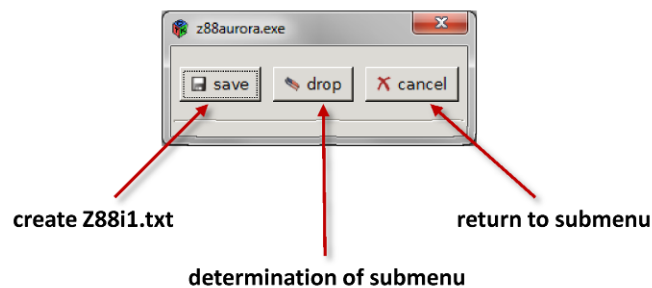






Figure 39: Exiting the submenu

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  Z88 Aurora Material Database.

Meshing

You have two possibilities of meshing parts in Z88 Aurora. 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 Z88 Aurora.

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 Z88 Aurora this mesher can be used for tetrahedrons with 4 or 10 nodes. Possibility of influence on the meshing within Z88 Aurora: Input of the maximum edge length

- 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 Z88 Aurora this mesher can be used for tetrahedrons with 4 nodes. Possibility of influence on the meshing within Z88 Aurora: Input of the maximum element size and the refinement factor.

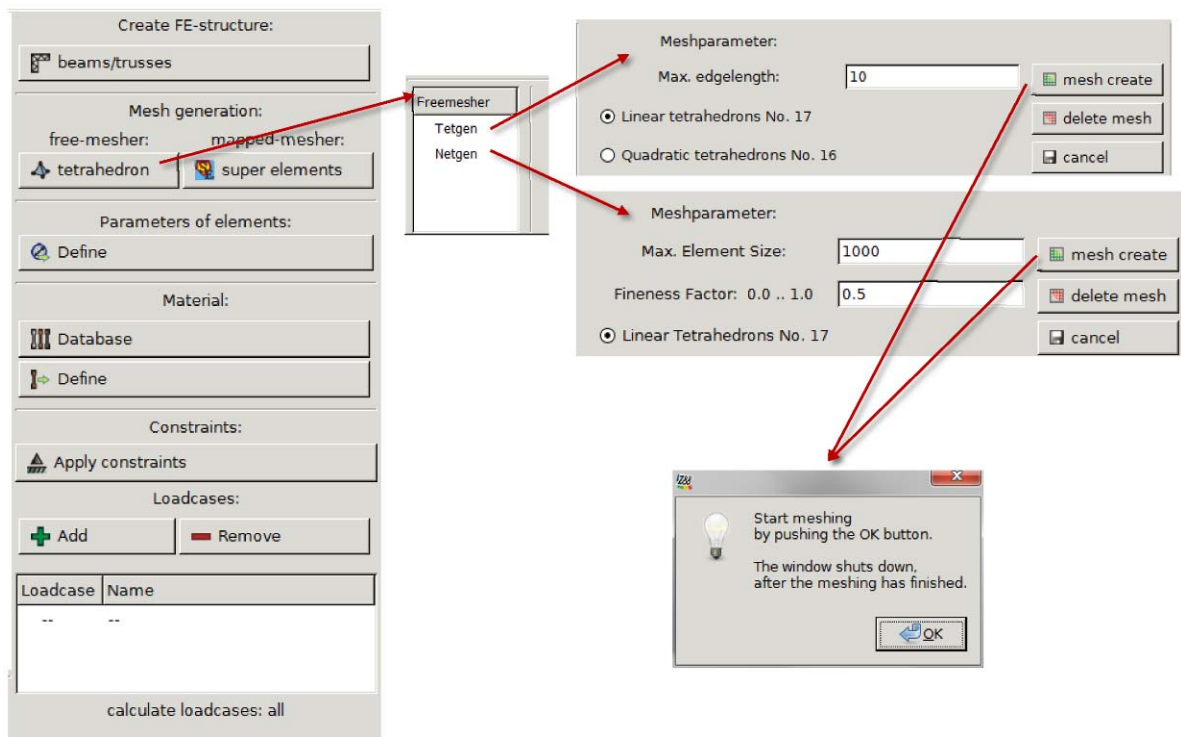




Figure 40: Creating tetrahedron meshes and options TetGen/NETGEN


⇒ Select TetGen or NETGEN

⇒ determine mesh parameter and element type

⇒ click  mesh create

⇒ afterwards either  delete mesh and change parameter

or

⇒  Save and 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!

 **Generating Super Elements / Mesh Generator Z88N**

The mesh generator can produce 2-dimensional and 3-dimensional Finite Element meshes from super structures. A mesh generation is sensible and permitted only for continuum elements. Table 2 Offers an overview of the possible Finite Element structures.

Table 2: Overview of possible super structures in Z88 Aurora

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

 **Prove Mesh**

As an additional function, the text menu bar offers the feature "prove 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 please always conduct a quality check at the end of the meshing.

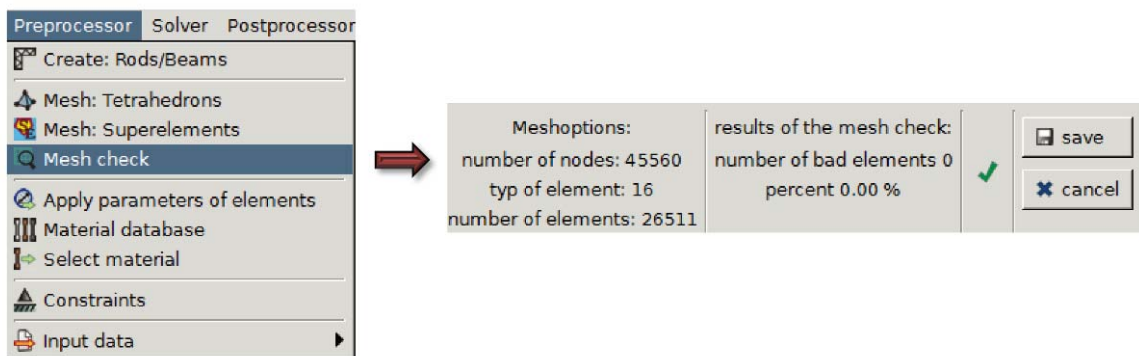


Figure 41: Prove mesh in the text menu bar

Element parameters

You can allocate element parameters for the element types plate, shell, truss, beam and cam element. If these have previously been compiled via the truss/beam menu elements, they can be edited here. Data from existing Z88 V13 files, imported into Z88 Aurora, can also be edited here.

Depending on the selected element type, the respective geometry data can be entered now.

In the process, you can allocate one geometry for all elements (all elements) or step-by-step different geometry ⇨ "from/to" for single elements.

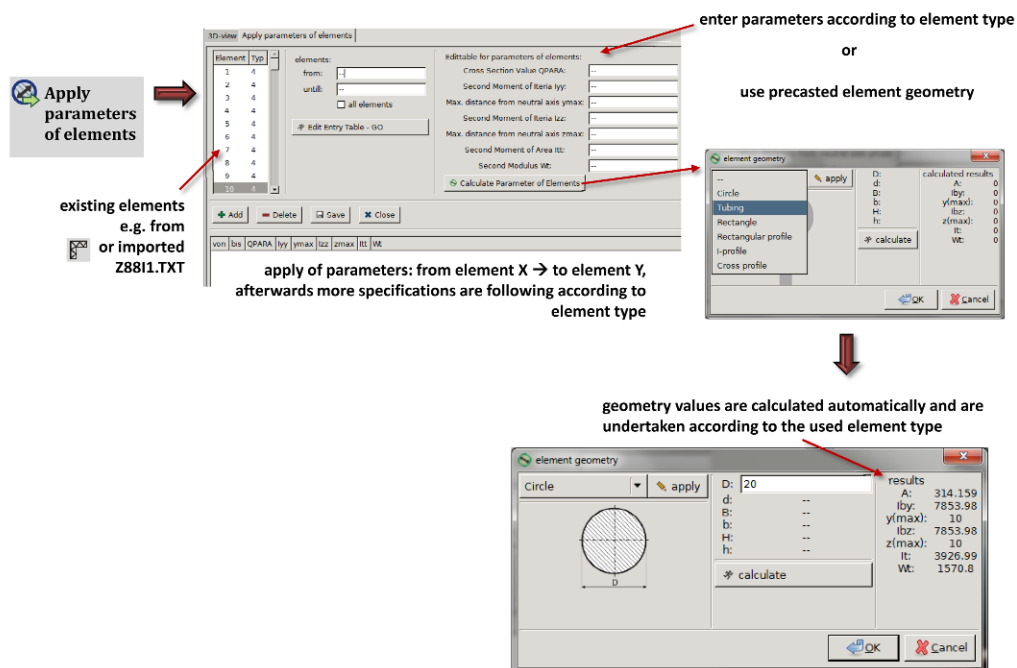


Figure 42: Element parameters-menu

The element parameters can be entered manually. Additionally, it is possible to have element shapes, such as circle, pipe, rectangle, square profile, I-profile or cross profile computed automatically. This is done via:

⇒ Calculate Parameter of Elements

⇒ Select element shape and apply (shape and input parameters appear)

⇒ Enter input parameters

⇒ calculate

⇒ "OK"

⇒ add

Depending on the selected element type only the required data are context sensitively chosen and used for calculation.

Material

In order to carry out static strength analyses, the present version of Z88 Aurora offers a material database containing more than 50 established construction materials.

Z88 Aurora Material Database

The Z88 Aurora material database is selected in the preprocessor menu () via the button (or via "Preprocessor" → Material Database). To facilitate your work with Z88 Aurora, several materials, such as miscellaneous types of steel and aluminium, have already been predefined. When you select a material from the list on the left, its allocated properties will be displayed on the right side (Figure 43).

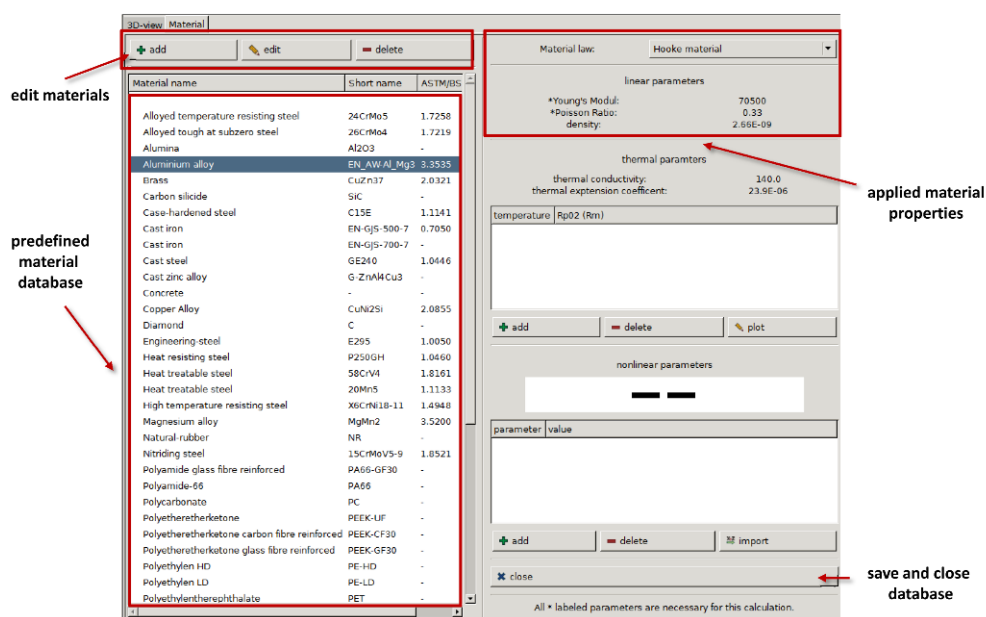
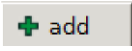



Figure 43: Z88 Aurora Material database

If the required material is not contained, you have the possibility to define new materials in the database. For this, click  in the left menu and the context menu "Material Parameters" is opened (Figure 44). 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.

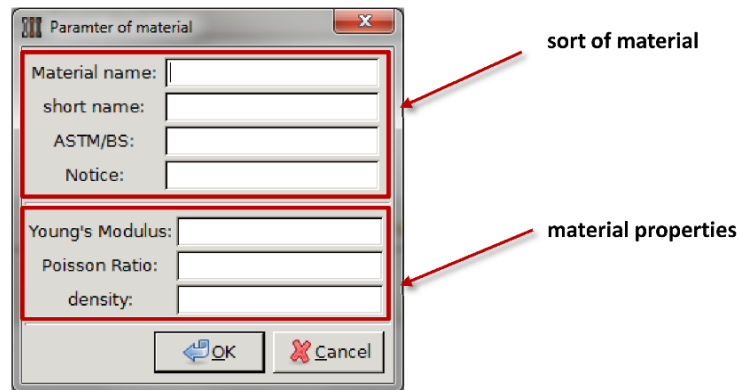

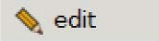
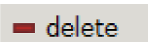
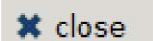


Figure 44: Context menu material parameters

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



- Material name: construction steel (common name)
- Identifier: S235JR
- Material number: 1.0038
- Young's Modulus: 210.000 N/mm²
- Poisson's Ratio: 0,29
- Density: 7,85 E-9 t/mm³

 Please note that you have to enter a dot as decimal point and that the material name must be clear (e.g. "construction steel1", "construction steel2", etc.). With „OK“ the material is transferred into the database.

With the pushbutton  you can edit already entered materials and remove them again, if necessary, with . Via  the database is saved and the tab closes.

⚠ For modules which are still in development, such as non-linear strength calculations, natural frequency analysis, contact and thermal analyses input masks ("thermal" and "non-linear properties") already exist; they will not be accessible, however, before the above-mentioned modules are implemented in a later version of Z88 Aurora.

 **Allocating Material**

The materials defined in the material database can be allocated to the imported or created FE models. Via  you access the preprocessor menu; there you click the pushbutton  **Select material** to open the "Allocate Material" tab (alternatively: "Preprocessor" → "Allocate Material"). The allocation menu is divided into several parts (Figure 45). In the left window, materials saved in the material database are displayed, which can be allocated to the model via the middle pushbutton (right sector).

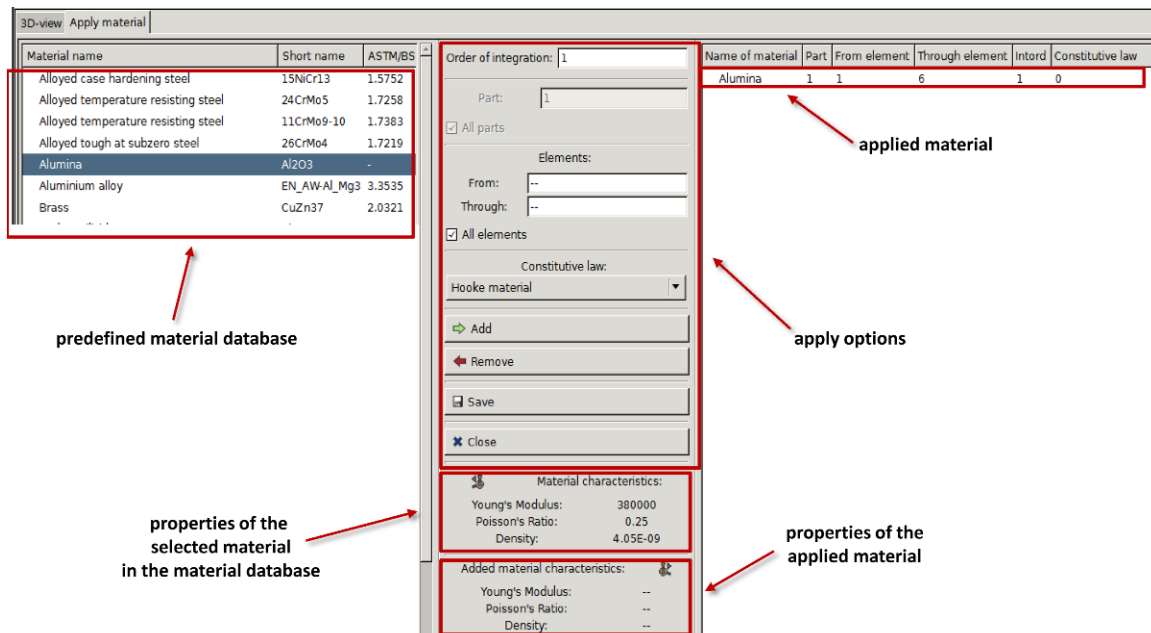


Figure 45: Allocate Material-tab

When you select a material from the list on the left, its properties are displayed in the middle area under "Material properties". There you will find the allocation options, where you can exactly specify the material allocation to the computation model (Figure 46).

The parameter "Integration order" determines the accuracy of display of the used meshing elements; for tetrahedron and hexahedron meshes, a value of 3 to 4 has proved practical; in

case of heavily distorted elements the order must be increased (for further information see Theory Manual).

The allocation parameters determine, to which areas of the current computation model the selected material properties are selected. If you want to apply the material to the whole part, keep the check marks at "all Elements". If not, you can assign different materials to the single elements ("from" element "to" element), for example, in order to model a bimetal.

⚠ In a future version of Z88 Aurora it will be possible to simulate different components in a model; the respective pushbutton will remain inactive until then.

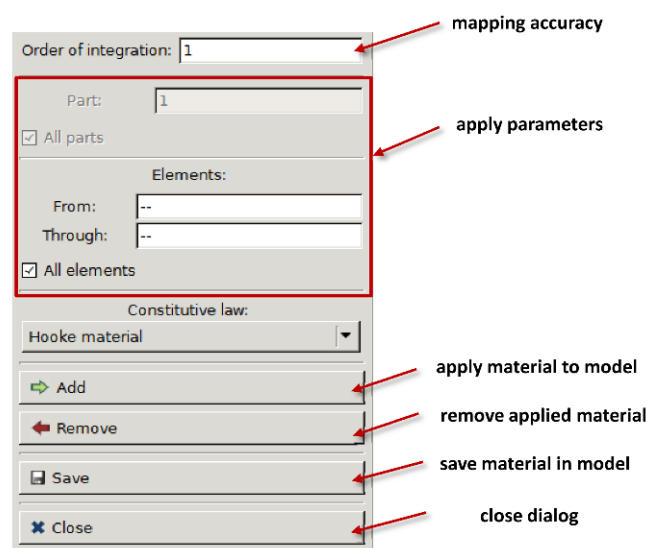
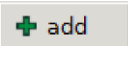
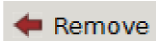




Figure 46: Allocation options

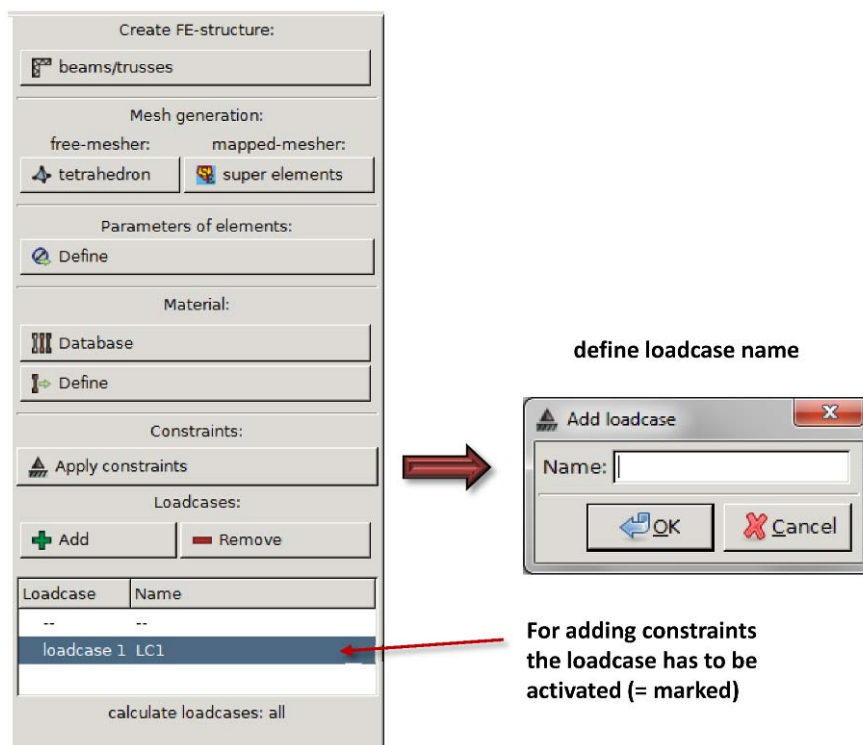
To allocate the selected material to the component click , to remove it click . The properties of the respective material are displayed in the middle area under "allocated material properties", when the corresponding material is selected on the right. With  the material will be transferred into the model and  closes the dialog window.

⚠ If you close the dialog without saving the material, it will not be transferred for the computation model.

Applying Boundary Conditions


Z88 Aurora offers the possibility to define all boundary conditions within the preprocessor; there do not have to be any conditions available in advance. Imported structures can either be calculated with the existing boundary conditions in Z88 Aurora or new entries can be applied. A set of boundary conditions is assigned to a load case. At the moment, only one loadcase is available in Version 1 of Z88 Aurora.

Clicking the "Apply constraints" icon opens the dialog window, where you can edit the name of the loadcase. For the application of the boundary conditions the loadcase must be active.



For adding constraints the loadcase has to be activated (= marked)

Figure 47: Creating boundary conditions

 Next time a loadcase is to be created, an alert window opens, because this function will be available in future Versions of Z88 Aurora.

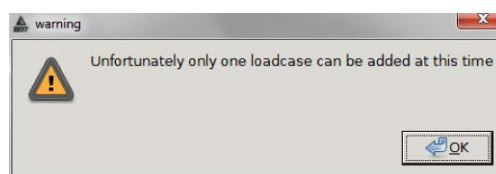
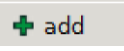
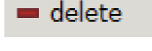
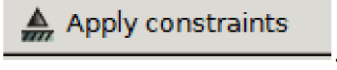


Figure 48: Warning: Number of load cases exceeded

With  the load case can also be created,  deletes all existing load cases.

After the creation of the load case and its activation, the dialog window "Constraints" can be

opened with .

Adding boundary conditions

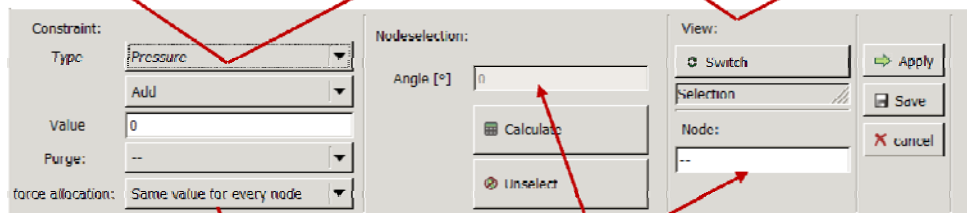
Types of constraints varying with element type

Pressure
 Force FG 1 (x-Richtung)
 Force FG 2 (y-Richtung)
 Force FG 3 (z-Richtung)
 Constraint (all directions)
 Constraint FG 1 (x-direction)
 Constraint FG 2 (y-direction)
 Constraint FG 3 (z-direction)

Force DOF 1 (x direction)
 Force DOF 2 (y-direction)
 Constraint (all directions)
 Constraint DOF 1 (x direction)
 Constraint DOF 2 (y-direction)
 Pressure

Constraints-view

Picking-view



Set current constraint

Save current constraint set

Leave dialog „constraints“

Calculation with FE-rules
 Same value for every node

Selection of nodes

(entire surface or individual)

Figure 49: Dialog box "Constraints"

Figure 49 shows the possibilities of the application of boundary conditions. Displacements, pressures and forces can be applied. With respect to forces you have to choose between a distribution of the force according to FE rules (for further information see Theory Manual chapter 3.1.3 "boundary conditions file Z88I2.TXT") and an even distribution (each node gets the given force).


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


⇒ Select boundary condition type, e.g. "Displacement FG 1 (x-direction)"

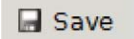
⇒ Add

⇒ Enter value, e.g. "50"

⇒ Select distribution of force, e.g. even distribution (distribution according to FE-rules only active in case of forces)

⇒ Select nodes, either via angle selection or by entering the number of the respective node (for further information please consult chapter  Picking).

⇒  applies the current boundary condition.

If the constraint is correct, you can either save the loadcase by clicking  , or you can add further boundary conditions.

- **In order to apply additional boundary conditions you must always first remove the presently selected nodes from the active selection!**

⇒  .

After that you can proceed as usual.

The different boundary conditions are displayed in a colour scale.

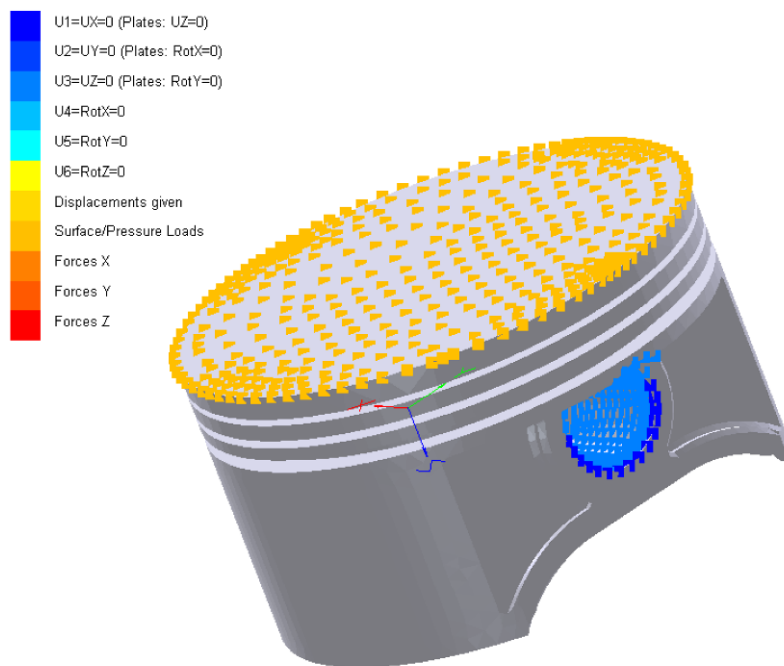


Figure 50: Display type "Boundary conditions"

To view single boundary conditions separately, the respective constraint can be selected via "View" > "Constraint" > "Only ...".

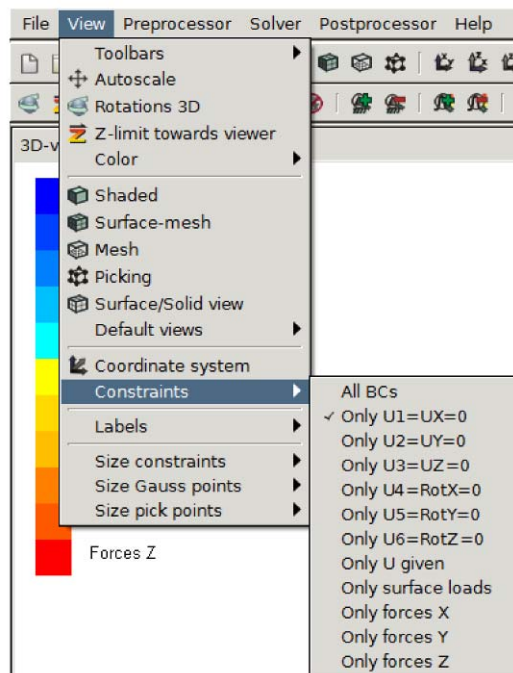



Figure 51: View display separate boundary condition

Defined boundary conditions can be displayed or hidden via  in the icon menu bar at any time.

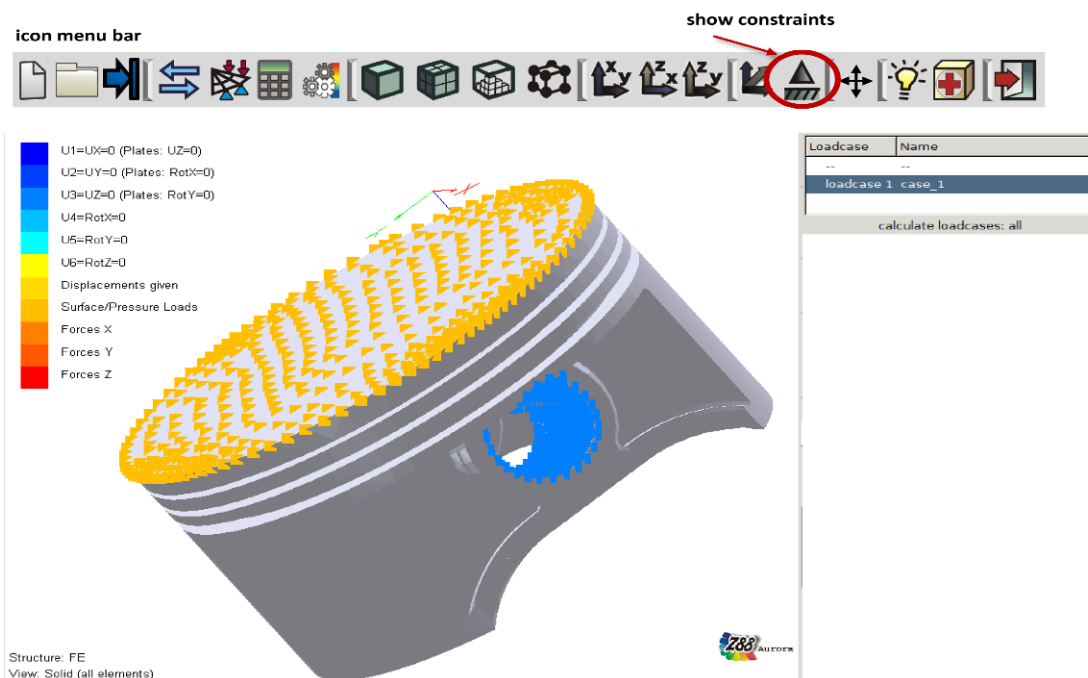


Figure 52: Accessing the boundary condition display via the boundary conditions icon in the icon menu bar

Removing boundary conditions

There are two possibilities to remove applied boundary conditions. On the one hand, the number of nodes where the boundary condition is applied, can be modified (command "Remove"), on the other hand all displacements, forces or pressures can be removed (command "Clean"). The selection of the nodes is done via "Picking" again.

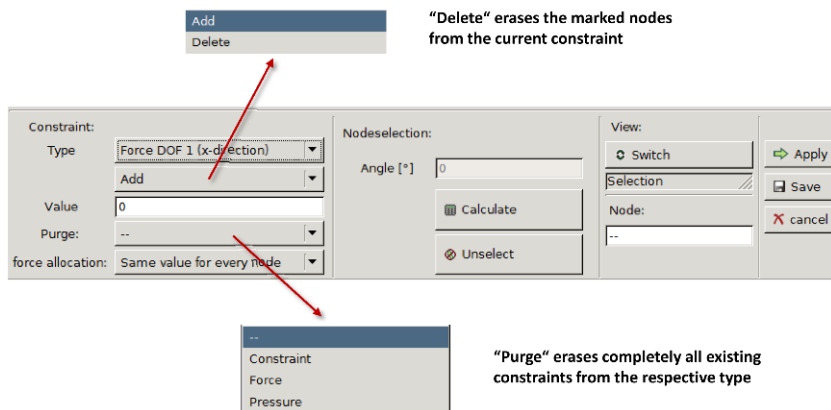


Figure 53: Removing boundary conditions

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.

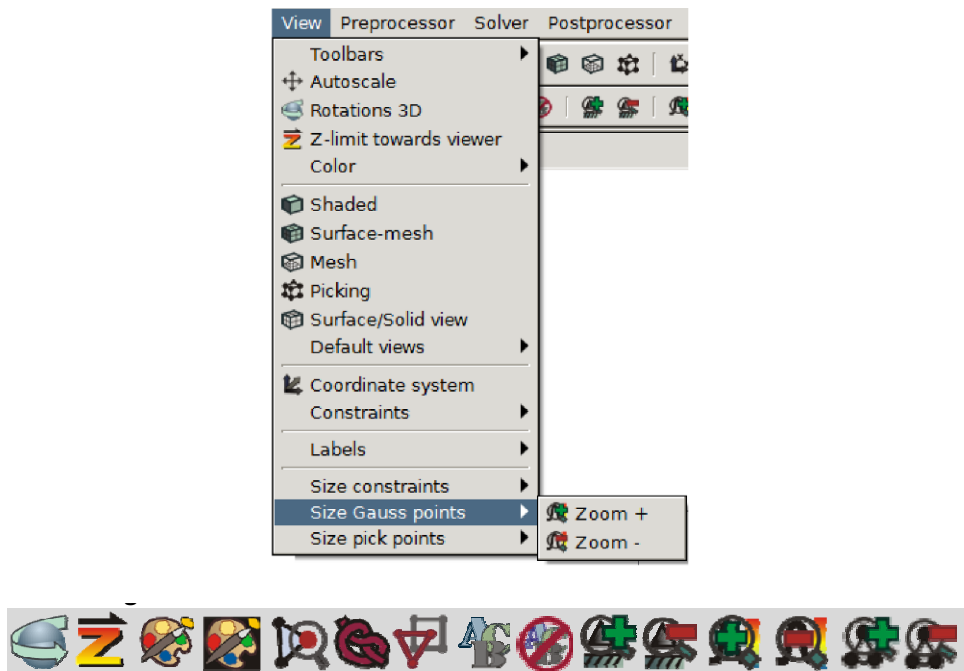


Figure 54: Changing the size of boundary conditions

- ⚠ **The labelling 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.

6.3 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.

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.
- A *sparse matrix iteration 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 at nearly the same speed as the solvers of the large and expensive commercial FEA programs as our tests showed. In addition, a minimum of storage is needed. This solver is your choice for large structures with more than 150,000 ... 200,000 DOF. FE-structures with ~ 5 million DOF (degrees of freedom) are no problem for it if you use a 64-BIT operation system (Windows or LINUX or Mac OS X) along with the 64-BIT version of Z88 and about 6 GByte of memory. *This very stable and approved solver works always, thus, you may use it as your standard solver.*

Further information and theoretical backgrounds about the integrated solvers can be found in chapter 4.2 of the Theory Manual.

In Z88 Aurora the solver types are selected via the solver menu:

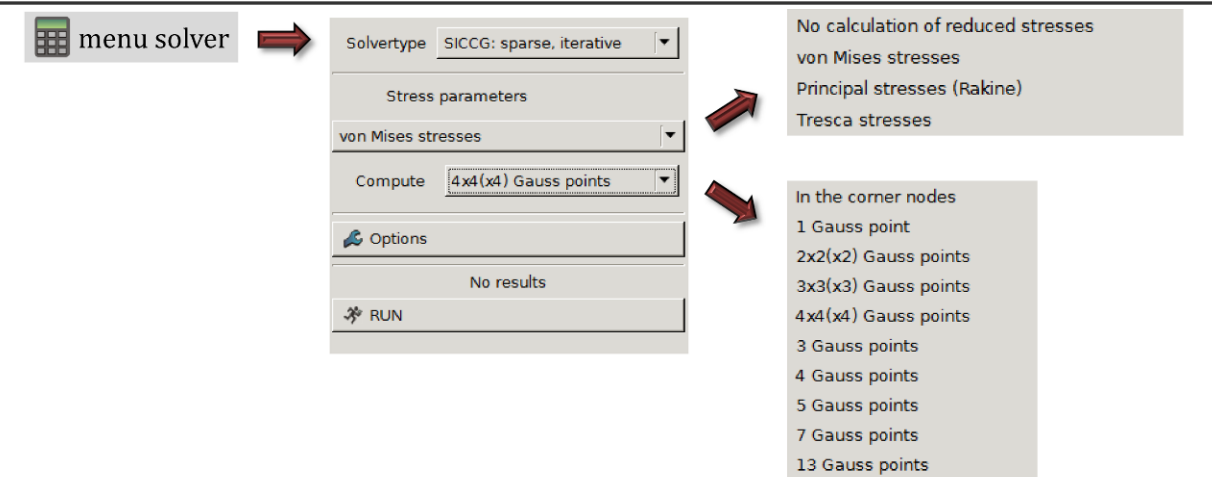


Figure 55: Solver menu

In the sector "Stress Parameters", the following equivalent stresses can be selected – but only one at a time – depending on the previous computation run:

- Distortion Energy Theory *DET*, i.e. *von Mises*
- Principal stress hypothesis *PSH*, i.e. *Rankine*
- Shear stress hypothesis *SH*, i.e. *Tresca*

"Number" determines the location of the stress calculation, in the corner nodes or Gauss points. Depending on the element type, the number of Gauss points differs. Further information in the Theory Manual, chapter 5.

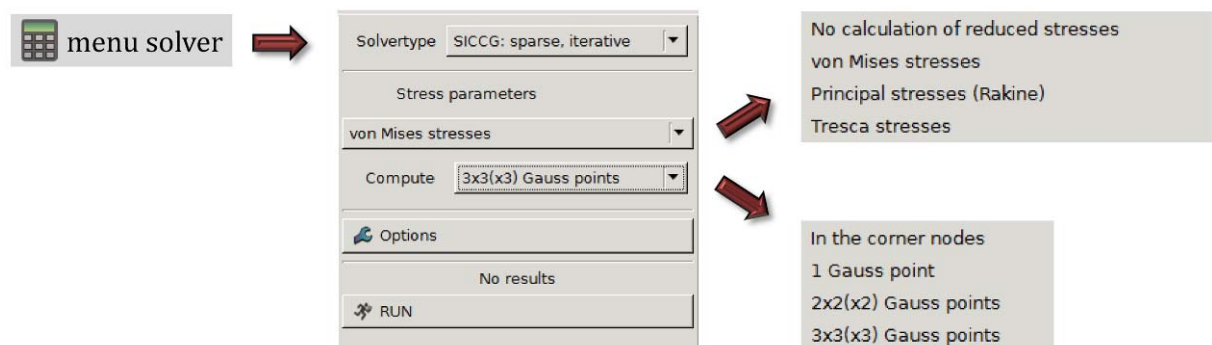


Figure 56: Stress parameters in the solver menu

In addition, you must supply several parameters. This is done via "extended options" in the menu "Solver":

- termination criterion: maximum count of iterations (for example 10000) reached
- termination criterion: residual vector < limit *Epsilon* (for example 1e-7)

- parameter for the SIC convergence acceleration. Shift factor *Alpha* (from 0 to 1, good values may vary from 0.0001 to 0.1). For further information consult the literature).
- parameter for the SOR convergence acceleration. Relaxation factor *Omega* (from 0 to 2, good values may vary from 0.8 to 1.2).
- Number of CPUs.
- COC-Incore-memory in Megabyte

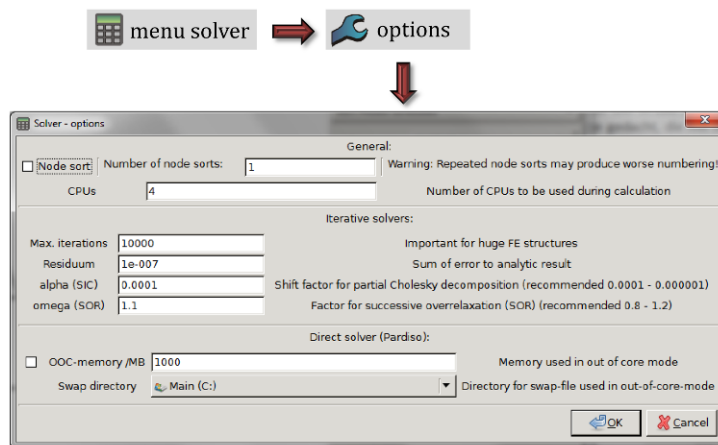


Figure 57: Extended options of the solver menu

- **The local settings for CPU and memory selected here are independent from the global settings in the options menu, which are used at every start.**

Table 3: Overview of the integrated solvers and their efficiency

Solver	Type	Number of DOF	Memory needs	Speed	Multi-CPU	Notes
Z88R -t/c -choly	Cholesky Solver without Fill-In	up to ~ 30.000	medium	medium	no	running Z88H first is recommended
Z88R -t/c -parao	direct Solver with Fill-In	up to ~ 150.000 for 32-BIT PCs	very high	very high	yes	useful with several CPUs and <i>very much</i> memory
Z88R -t/c -siccg or -sorcg	Conjugated gradients solver with pre-conditioning	no limits (up to 12 mio. DOF were run on an ordinary PC)	an absolute minimum	medium	no	a very stable and reliable solver for very large structures

Option "Cuthill-McKee Algorithm" in "Extended Options" in the Solver Menu

The choice of the nodal numbers is extremely important for the compilation of the stiffness matrix. Bad nodal numbering may result in huge memory needs which are not really necessary. However, Z88H may reduce the memory needs for the direct Cholesky Solver Z88F greatly; the sparse matrix iteration solver Z88I1/Z88I2 may also gain some advantages from a Z88H run, but the iteration solver is a-priori very stable regarding node numbering because of storing the non-zero elements only.

The Cuthill-McKee program Z88H, integrated in Z88 Aurora was originally designed for finite element meshes generated by 3D converter Z88G. However, Z88H can deal with all Z88 meshes. The Cuthill-McKee program Z88H may sometimes compute counterproductive results, i.e. a worse nodal numbering than the original mesh. You should have some experiments because the *Cuthill-McKee algorithm* may not always improve a given mesh (further information in chapter 4.2.4 of the Theory Manual).

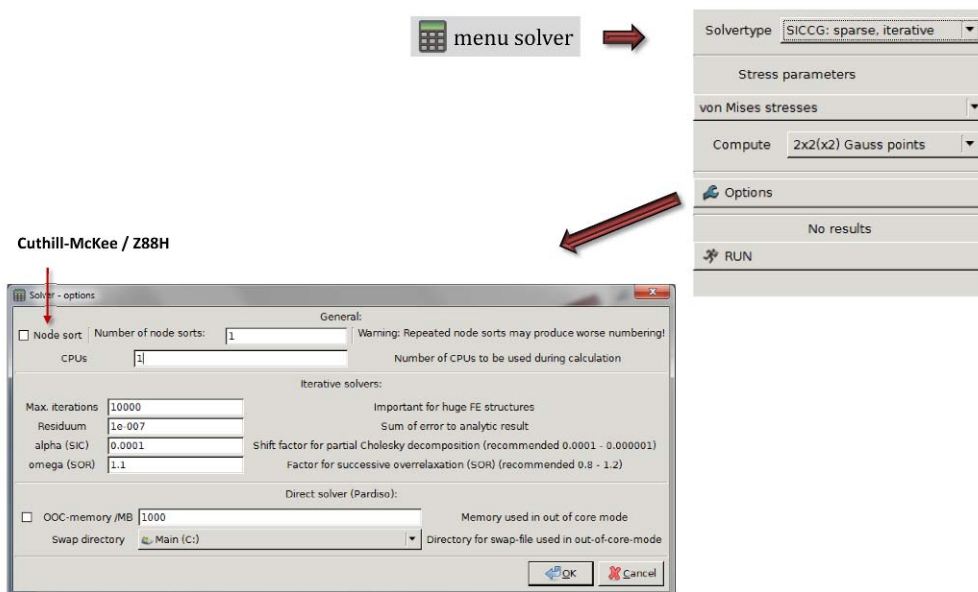



Figure 58: Solver menu, or extended options with "Cuthill-McKee algorithm"

After setting all required parameters, the calculation is started by pushing the button  . An information window opens, which displays the duration of the calculation. Start the calculation by confirming the message.

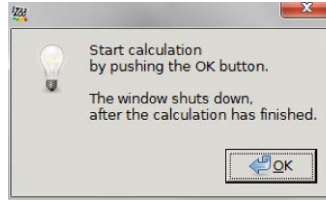


Figure 59: Information window: calculation

The Solver in the Text menu bar

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

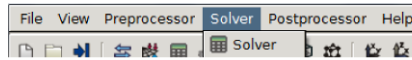



Figure 60: Solver access the text menu bar

6.4 Postprocessor

After the calculation has been carried out, the results can be displayed in the Z88 Aurora postprocessor by clicking the button  (Figure 61).

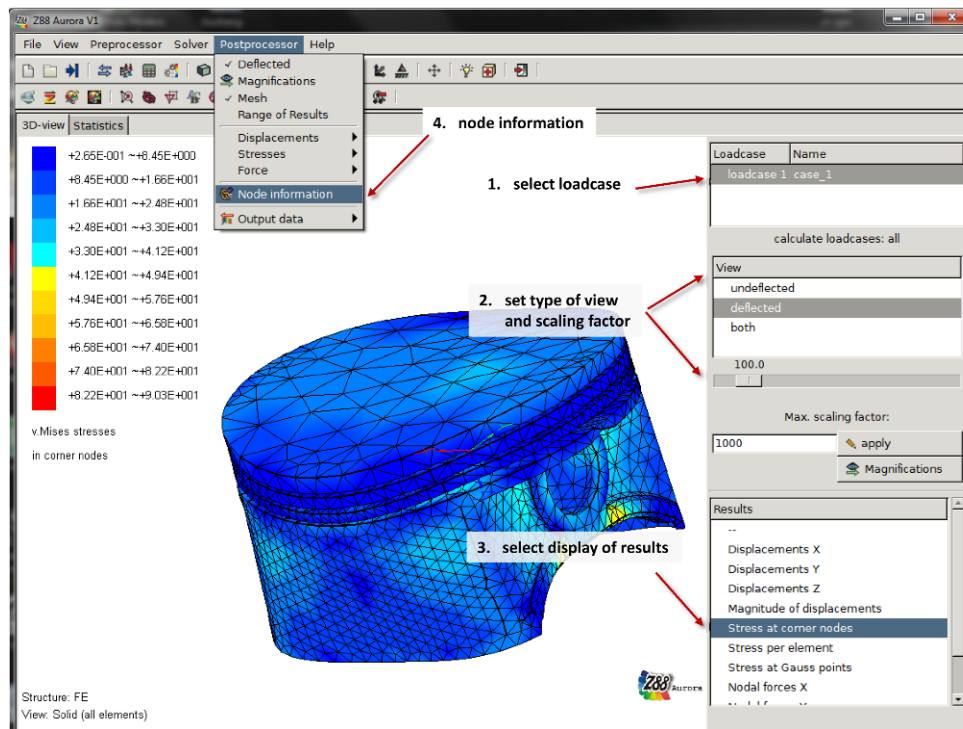


Figure 61: Z88 Aurora Postprocessor

On the right side of the screen a context menu appears. Here you must first select the load case; furthermore it is possible, to have the component displayed in the results window – deflected, undeflected or both conditions at the same time.

Below this is the results menu: here, 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 undeformed condition

These options can also be accessed via the menu bar "Postprocessor" (Figure 62). Apart from the "Displaced" view you can also change the "single factors" of the displacement display here, by adjusting the factors „FUX“, „FUY“, and „FUZ“ accordingly. Thus, displacements can be displayed at a larger or smaller scale as you wish.

With "Mesh", the surface mesh of the calculated structure is shown and hidden.

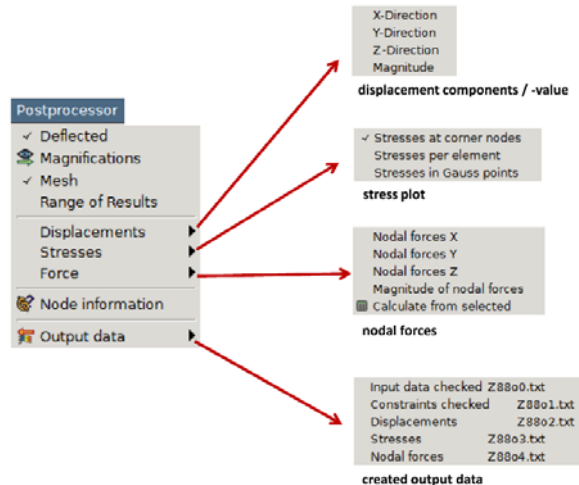


Figure 62: Postprocessor menu bar

„Scaling“ opens a context menu where you can enter the limits of the displayed stresses, deformations, and nodal forces. Depending on the given interval, only the values between the "MIN" and the "MAX" value are displayed in different colours (Figure 63). This way excessive stresses can be hidden or only critical stresses shown.

The options "Displacements", "Stresses" and "Forces" with corresponding submenus permit the display of components or values of the respective selection.

You can have the nodal force of a single or several nodes displayed. For this, switch to picking display via the key F4 and select the respective nodes with Strg + . Switch to the "Postprocessor → Nodal forces"-display. When you click "Forces → Calculate from selection" now, the nodal forces of the selected nodes are added up component-by-component and displayed as value in the main window.

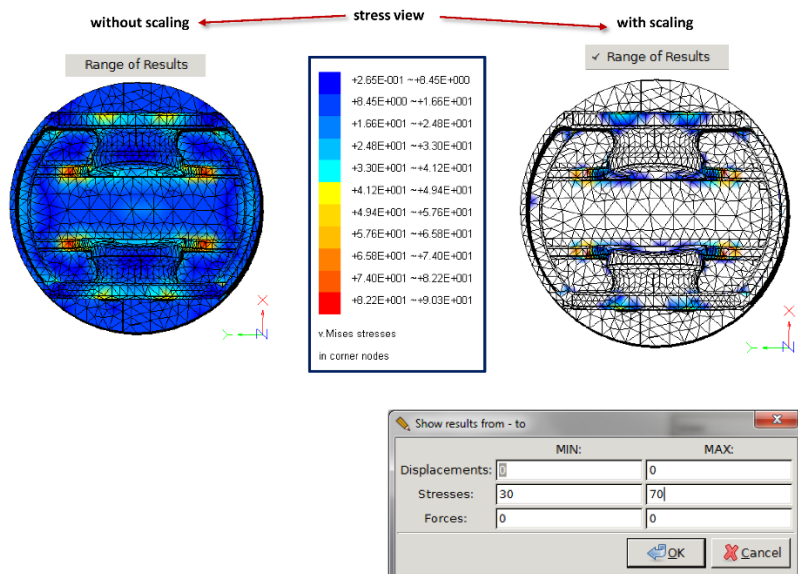



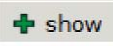


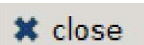





Figure 63: Scaling the colour plot

Furthermore, it is also possible to select single or several nodes directly in the 3D model, in order to have their displacements and stresses displayed. Via the menu "Postprocessor → "Node information" a context menu opens and the view display switches to "Picking". With  +  you can select single nodes; if you want to select several nodes at the same time, you can create an enveloping box by means of *Shift key* pressed and *left mouse button* (for further information see chapter  Picking). If you click , a list of selected nodes is displayed, which you can also export as .CSV-file via  export->csv . The node information contains node number, current coordinates and the displacement (component-by-component and value);  deletes the selection.  ends the dialog (Figure 64).

	Node number	x	y	z	Displ. x	Displ. y	Displ. z	Displ.	
	546	23.5051	44.1306	35.0688	0.0181927	-0.0155904	0.0344173	0.0419355	
	547	20.7219	45.5039	30.3791	0.0136213	-0.0159075	0.0387924	0.0440845	
	548	20.7158	45.5067	32.402	0.0141040	0.020025	0.0287201	0.0450522	

Node:


 export->csv

Figure 64: Node information

Under "Postprocessor → Output files" you can access the single output files of the calculation, in order to get the exact numerical values there (for further information see Z88 Aurora Theory Manual):

- Z88O0.TXT – prepared input data
- Z88O1.TXT – prepared boundary conditions
- Z88O2.TXT – calculated displacements
- Z88O3.TXT – calculated stresses
- Z88O4.TXT – calculated nodal forces

With the integrated statistics function (tab menu) the relative as well as the absolute statistic frequency of the calculated stress values (here according to v.Mises) can be displayed (Figure 65). In order to use this function, a stress output must be selected in the 3D-view.

If you leave the partition of the bar diagram at exactly 11 (standard value), the same colour screen as in the 3D display is used for visualisation. You can also output this statistic as a file.

To do so, you click  and enter a file name.

! We suggest you open the saved .CSV-file with a normal text editor, since some spreadsheet programs interpret this data format for example as date type and do not read them out as numerical values.

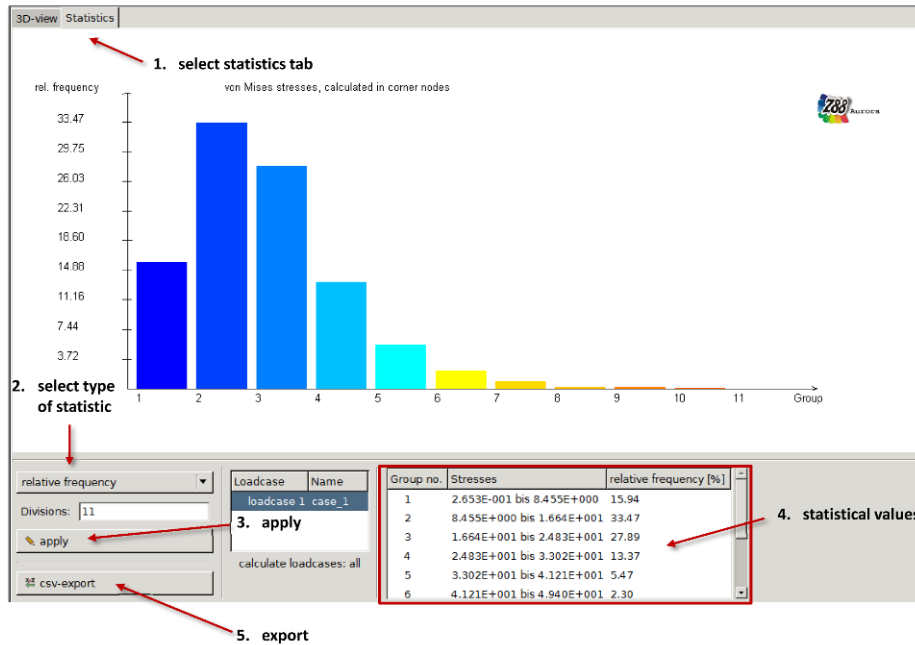


Figure 65: Statistics function

6.5 Help, Support and Option settings

Help

Z88 Aurora offers you several different help functions, which can be used separately. The following is an overview of the single help components.

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

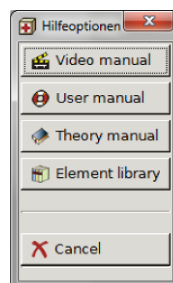



Figure 66: Help options


Video Tutorial

To increase clarity, video sequences dealing with some special topics are available. The single videos are accessed via the menu "Video help".

These are:

- Picking
- Views
- Node information

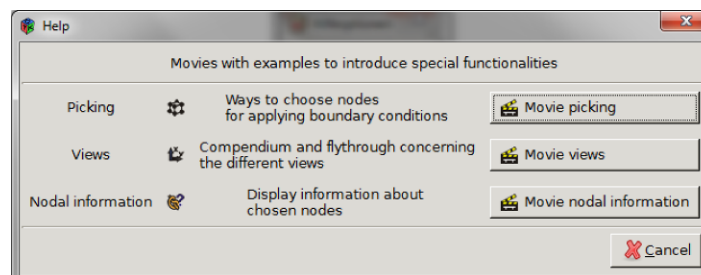



Figure 67: Video help in Z88 Aurora


User Manual

In the User Manual all functions available in Z88 Aurora are explained.


Theory Manual

The Theory Manual addresses the issue of the computation bases of Z88 Aurora. For experienced Z88 V13 users the differences between Z88 V13 and Z88 Aurora 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

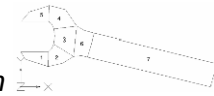
A short description of the element types integrated in Z88 Aurora.


Examples

By means of five different examples the basic functions are explained.

*Analytic elements:**Example: electrical tower*

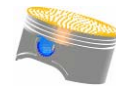
On the basis of this example, the data import from Z88 V13 files and the calculation of trusses are explained.

*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 Z88 Aurora 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.

*Importing geometry:**Example: piston rod*

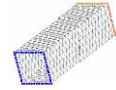
On the basis of this example it is explained how you can import geometry data from an STL-file into Z88 Aurora. The object in question is a piston rod, which was constructed in Pro/ENGINEER WF4.0 and exported as STL-file (surface mesh). STL-files can be created with practically every CAD system. These files only contain geometry data, but no ready-to-calculate FE mesh. With the integrated meshing algorithms of Z88 Aurora, these meshes can be created.

*Volume elements:**Example: engine piston*

As already described in previous chapters, you can import data from 2D- and 3D-CAD- and FE-systems in Z88 Aurora. 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 Z88 Aurora are demonstrated.

Shell elements:

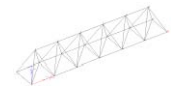
Example: square pipe



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 Z88 Aurora are demonstrated.

Truss elements:

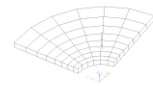
Example: crane girder



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.

Hexahedron elements:

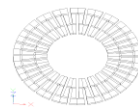
Example: plate segment



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.

Hexahedron elements:

Example: RINGSPANN pad



This example contains a so called RINGSPANN pad, used as force transmitter, similar to a sliced disc spring. Springs, operating according to Hooke's law, can be mapped as FEA-structures.

Tetrahedron elements:

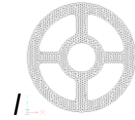
Example: motorcycle crankshaft



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:

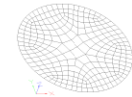
Example: gearwheel



A gearwheel, whose centre 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 centre's expansion to the gear teeth (which are left out for model simplification).

Plate element:

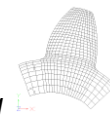
Example: circular plate



This sample is intended as 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.

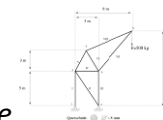
Plane stress elements:

Example: dedendum loading




The bearing capacity calculation of spur gears is very high sophisticated. This example shows a qualitatively way for estimate calculation. A complete calculation of the bearing capacity using DIN 3990 is very complex. In this example there are static forces and ideal geometries (no misalignments, no crowning and so on). The stiffness of dedendum of a geometric correct tooth is observed.

Creating a FE-structure and element parameters *Example: crane*



In Z88 Aurora an editor for creating beam and truss elements is included. The necessary nodes for creating the structure can be done by using coordinates. For the element parameters Z88Aurora is used.


Information

 Project information

Project information can be viewed in two different places.

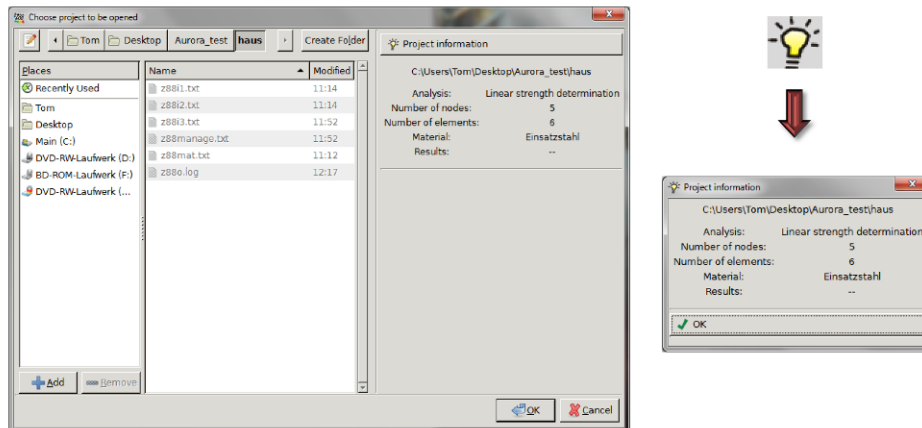


Figure 68: Project information at opening a project folder (left) and via the icon in the icon menu bar (right)

The analysis type, the number of nodes, the number of elements, the material, and possibly existing results are listed.

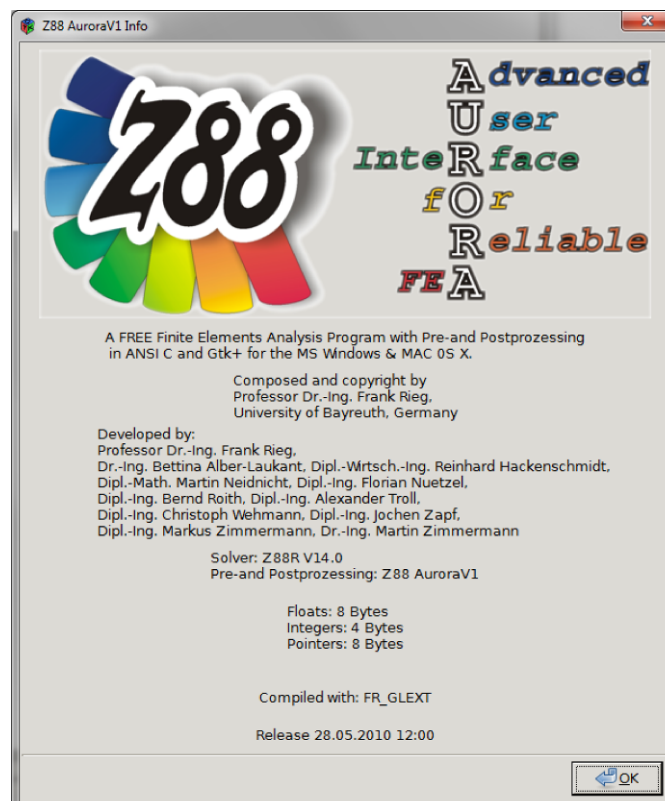

 About Z88 Aurora


Figure 69: Version information Z88 Aurora

Support

Homepage

See www.z88.de for further information!

Email

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

Forum

The homepage owns a forum with needful discussion about Z88 Aurora. Let's discuss...

Option Settings

Changes to the user interface can be made in the options menu. Here, the language, the single file paths, the memory settings and the view settings are modified.

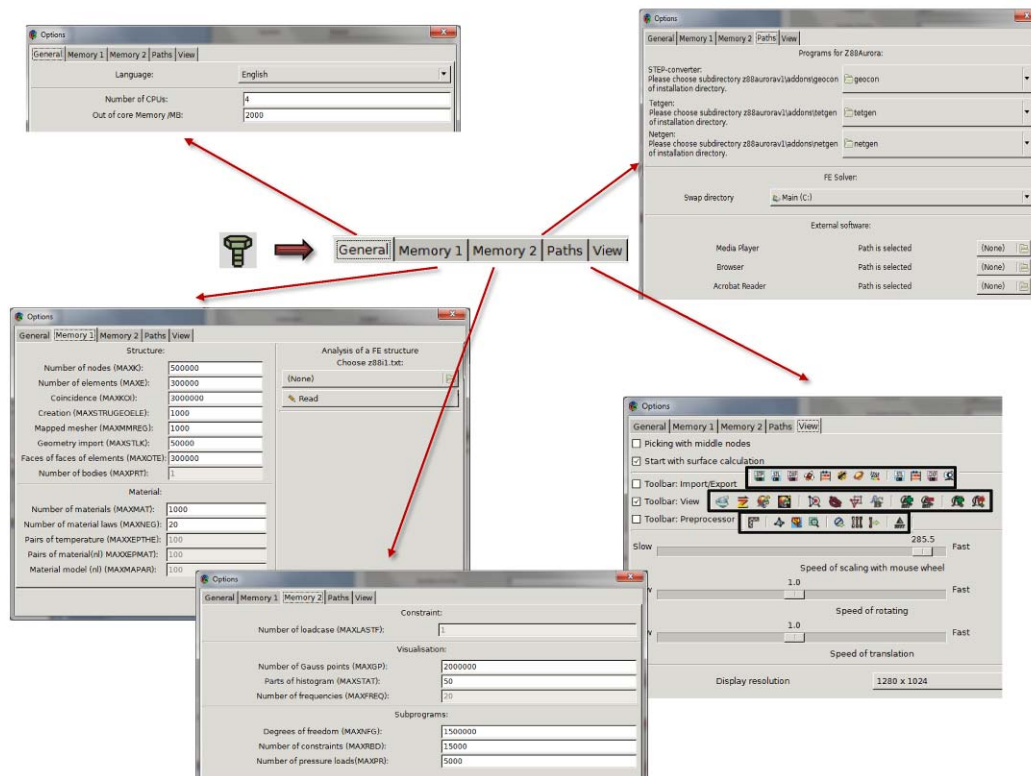


Figure 70: Option settings

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 Z88 Aurora!