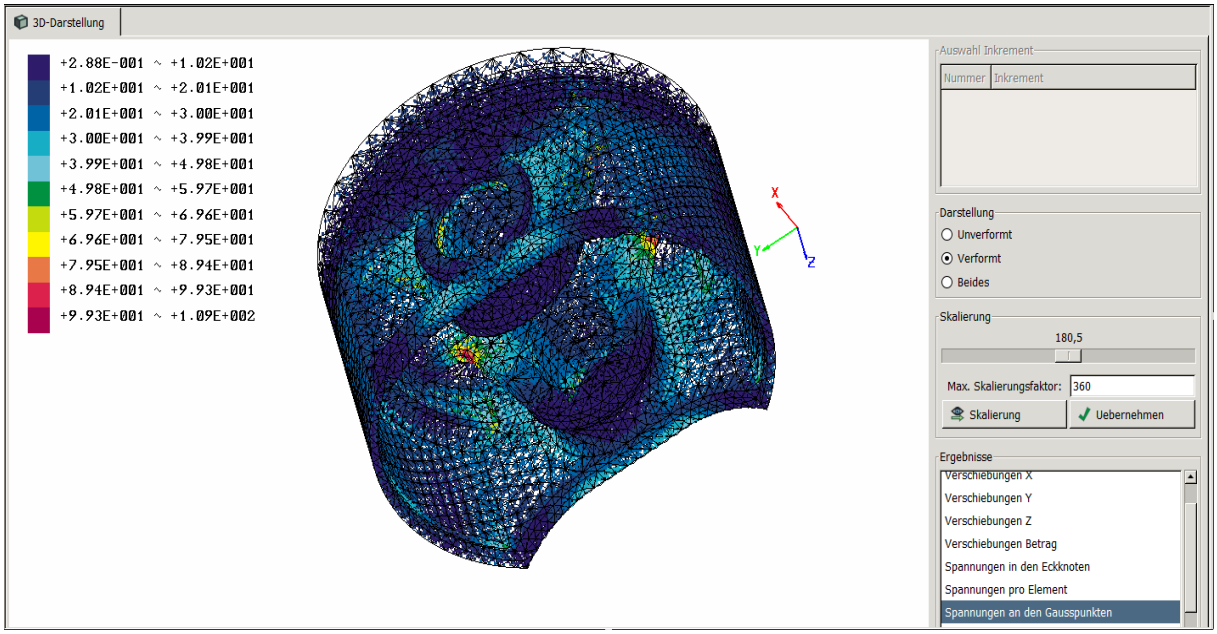
Theory manual



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

All rights reserved by the editor
Version 5, April 2019

Z88 Aurora® is a registered trademark (No. 30 2009 064 238) of Prof. Dr.-Ing. Frank Rieg

WELCOME TO Z88AURORA®!

Z88Aurora® is a software package for solving structural mechanical, linear and nonlinear static problems, eigen problems, and thermal problems by the Finite Element Analysis (FEA) and is developed by a team of ten under the supervision of Professor Frank Rieg since 2009. Z88Aurora® is based on Z88 OpenSource V15 and is available for Windows 64-bit, LINUX 64-bit and Mac OS-X for free download (as executable file). In addition to Z88 OpenSource V15, Z88Aurora® offers a graphical user interface, a completely new pre-processor and an extension of the approved post-processor Z88O along with multi-core solvers for static linear problems (Z88R), vibrations (Z88EI), thermal problems (Z88TH), nonlinear static problems (Z88NL) and contact simulations (Z88KONTAKT). Z88Aurora® was developed with great care for easy and intuitive operation.

This Z88Aurora® V5 is an extended development of the successful version V4 and its forerunners V1/V2/V3 of Z88. Since version 4 Z88Aurora® allows the simulation of interacting parts for both “frictionless” and “fixed” contact as well as different contact types (node-surface, surface-surface). In version 5, the software module for thermomechanical simulations has been extended. It supports multiple material settings for different parts of a solid body and allows to define 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 (shading, mesh) and to hide/show the legend as well as the coordinate system triad. Additionally, an image export function has been integrated into Z88Aurora®.

Z88Aurora® does not want to compete with commercial FEA programs which exceed Z88Aurora® ’s functionality by far, but are hardly affordable and complicated to operate. While you are still puzzling about installation and start of some programs of this genre, you will already have calculated the first examples with Z88Aurora®. And the online-help is always only one keystroke or mouse click away. The Z88 system may operate with English or German language depending on your setting (ENGLISH or GERMAN) in the options menu.
In addition to this Theory Manual, there are a User Manual, an Example Manual, an Installation Manual and video sequences available.

If you already have FEA experiences, you may start at once. If you are a beginner in this area, I would recommend secondary literature. Here are a few choices:

- *Zienkiewicz, O.C.; Taylor, R.L.: The Finite Element Method, Volumes 1-3, 5th edition, Butterworth- Heinemann and John Wiley & Sons, 2000*
- *Bathe, K.J.: Finite Element Procedures. Prentice Hall, 1995*
- *Rieg, F.; Hackenschmidt, R., Alber-Laukant, B.: Finite Element Analyse for Engineers. Carl Hanser Verlag, Munich, Vienna: 2014, 1st edition*

The Z88 website: www.z88.de. Give us your feedback!

Professor Dr. Frank Rieg
Lehrstuhl Konstruktionslehre und CAD
(Chair for Engineering Design and CAD)
Faculty of Engineering Science
University of Bayreuth, Germany
frank.rieg@uni-bayreuth.de

Bayreuth, April 2019

http://www.konstruktionslehre.uni-bayreuth.de

License

Software Products: Z88Aurora - 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.

TABLE OF CONTENTS

WELCOME TO Z88AURORA®! 2

1. THE FINITE ELEMENT PROGRAM Z88AURORA® 8

1.1 GENERAL OVERVIEW ON Z88AURORA® 8
1.2 SUMMARY OF THE Z88 ELEMENT LIBRARY: 9
2D PROBLEMS: PLANE STRESS, PLATES, BEAMS, TRUSSES 9
AXISYMMETRIC PROBLEMS: 12
SHELL PROBLEMS: 14
SPATIAL PROBLEMS: 16

2. The Z88 Computing Units: 20

2.1 OVERVIEW: 20
2.2 A SHORT DESCRIPTION OF THE MODULES: 20
I. THE PRE- AND POST-PROCESSOR 20
II. THE SOLVERS 20
2.2.1 THE LINEAR SOLVER Z88R 20
2.2.2 THE VIBRATION SOLVER Z88EI 22
2.2.3 THE THERMO SOLVER Z88TH 23
2.2.4 THE NONLINEAR SOLVER Z88NL 24
III. THE INTERFACES TO CAD & FEA SYSTEMS 25
IV. THE MAPPED MESHERS 25
2.3 WHICH Z88 ELEMENTS CAN BE PRODUCED AUTOMATICALLY? 26

3. The Input and Output of Z88Aurora® 27

3.1 COMPARISON OF Z88 FILE FORMATS 29
3.2 FILE LAYOUT IN Z88AURORA® 30
3.2.1 GENERAL STRUCTURE DATA Z88STRUCTURE.TXT 30
3.2.2 GROUP DATA Z88MARKS.TXT & Z88SETS.TXT 32
3.2.3 CHARACTERISATION FILE Z88SETSACTIVE.TXT 34
3.2.4 MATERIAL DATA FILE Z88MAT.TXT 34
3.2.5 MATERIAL DATA *.TXT 35
3.2.6 MESH GENERATOR INPUT FILE Z88NI.TXT 36
3.2.7 SOLVER CONTROL FILE Z88CONTROL.TXT 40
3.2.8 SOLVER CONTROL FILE Z88.DYN 46
3.2.9 DEFINITION FILE Z88ENVIRO.DYN 48
3.2.10 FILE FOR HISTORY VARIABLES Z88NLI7.TXT 52
3.2.11 OUTPUT FILES Z88O.TXT 55

4. THE Z88 MODULES 56

4.1 CAD & FE INTERFACES 56
4.1.1 IMPORTING Z88 FILES 58
4.1.2 MANUALLY CREATING Z88 FILES 59
4.1.3 THE STEP CONVERTER Z88GEOCON 69
4.1.4 THE STL CONVERTER Z88GEOCON 70
4.1.5 THE STL-TOOLS 72
4.1.6 THE DXF CONVERTER Z88X 74

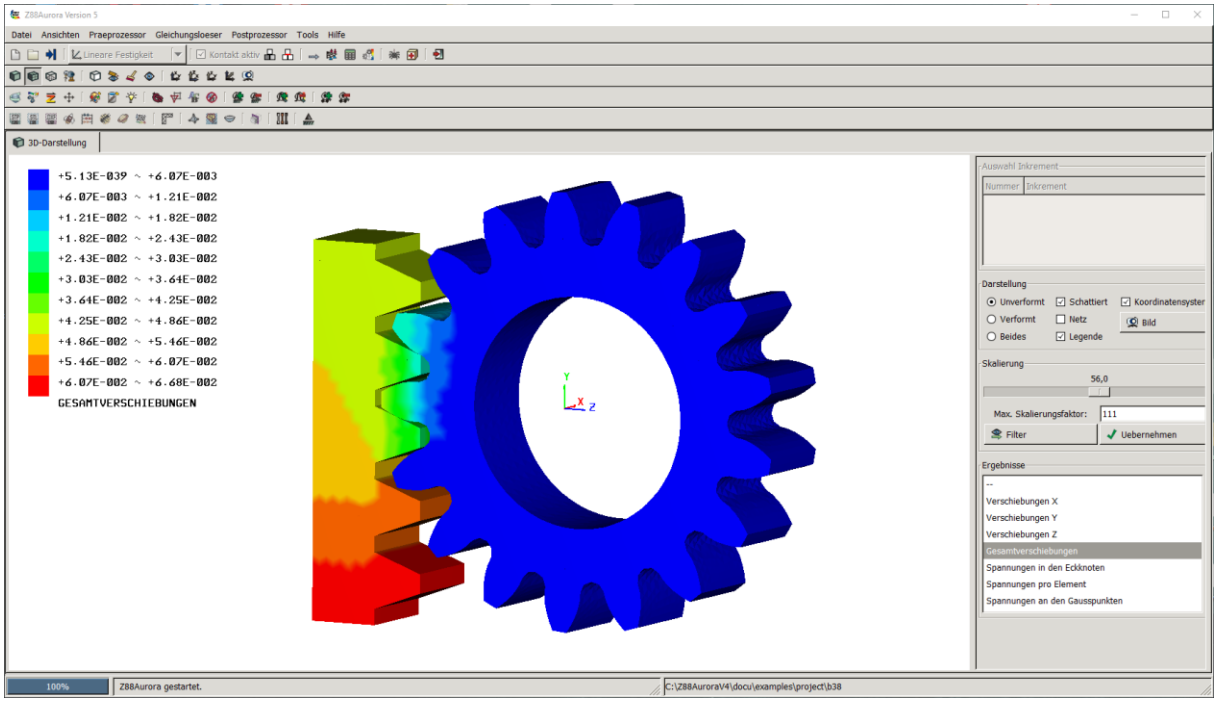
Z88X IN DETAIL 75
EXAMPLE 1 FOR Z88X: FINITE ELEMENTS STRUCTURE 83
EXAMPLE 2 FOR Z88X: SUPER ELEMENTS STRUCTURE 93
4.1.7 THE NASTRAN & COSMOS CONVERTER Z88G 100
4.1.8 THE ANSYS CONVERTER Z88ASY 102
4.1.9 THE ABAQUS CONVERTER Z88AINP 104
4.2 PICKING & SET MANAGEMENT 106
4.2.1 SURFACE LOADS 107
4.3 THE LINEAR SOLVER Z88R 109
4.3.1 CHOOSING THE RIGHT SOLVER 113
4.3.2 SOME NOTES ON STRESS CALCULATION 114
4.3.3 SOME NOTES ON NODAL FORCE CALCULATION 114
4.4 THE VIBRATION SOLVER Z88EI 115
4.5 THE THERMAL SOLVER Z88TH 118
4.6 THE NONLINEAR SOLVER Z88NL 122
4.7 THE CONTACT MODULE Z88KONTAKT 125
4.8 THE MAPPED MESHERS 128
4.8.2 Z88N FOR 2D AND 3D ELEMENTS 128
4.8.3 THE TETRAHEDRON REFINER 130
4.8.4 THE SHELL THICKENER 132
4.9 THE POST-PROCESSOR 133

5. DESCRIPTION OF THE FINITE ELEMENTS 137

5.1 HEXAHEDRON NO. 1 WITH 8 NODES 137
5.2 BEAM NO. 2 WITH 2 NODES IN SPACE 139
5.3 PLANE STRESS TRIANGLE NO. 3 WITH 6 NODES 141
5.4 TRUSS NO. 4 IN SPACE 143
5.5 SHAFT ELEMENT NO. 5 WITH 2 NODES 144
5.6 TORUS NO. 6 WITH 3 NODES 145
5.7 PLANE STRESS ELEMENT NO. 7 WITH 8 NODES 146
5.8 TORUS NO. 8 WITH 8 NODES 148
5.9 TRUSS NO. 9 IN PLANE 150
5.10 HEXAHEDRON NO. 10 WITH 20 NODES 151
5.11 PLANE STRESS ELEMENT NO. 11 WITH 12 NODES 153
5.12 TORUS NO. 12 WITH 12 NODES 155
5.13 BEAM NO. 13 IN PLANE 157
5.14 PLANE STRESS ELEMENT NO. 14 WITH 6 NODES 159
5.15 TORUS NO. 15 WITH 6 NODES 161
5.16 TETRAHEDRON NO. 16 WITH 10 NODES 163
5.17 TETRAHEDRON NO. 17 WITH 4 NODES 165
5.18 PLATE NO. 18 WITH 6 NODES 167
5.19 PLATE NO. 19 WITH 16 NODES 169

5.20 PLATE NO. 20 WITH 8 NODES 172
5.21 SHELL NO. 21 WITH 16 NODES 174
5.22 SHELL NO. 22 WITH 12 NODES 176
5.23 SHELL NO. 23 WITH 8 NODES 178
5.24 SHELL NO. 24 WITH 6 NODES 180
5.25 BEAM NO. 25 WITH 2 NODES IN SPACE 182

1. THE FINITE ELEMENT PROGRAM Z88AURORA®



1.1 GENERAL OVERVIEW ON Z88AURORA®

THE Z88 PHILOSOPHY IS ALSO TRUE FOR Z88AURORA®!

- Fast and compact: Developed for PC, no ported mainframe system
- full 64-bit support for Windows, LINUX and Mac
- *native* Windows, LINUX and Mac OS-X programs, no emulations
- Windows, LINUX and Mac OS-X versions use the same computing kernels
- full data exchange from and to CAD systems (*.DXF, *.STP, *.STL)
- FE structure import (*.COS, *.NAS, *.BDF, *.ANS, *.INP) and FE export (*.INP)
- context sensitive online-help and video tutorials
- simplest installation with *Microsoft® Installer (MSI)*
- Z88Aurora® can load input files from the OpenSource Z88 V15 directly
- Z88Aurora® V5 is fully compatible with Z88 V14/15 OS and Z88Aurora® V2/3/4. Existing files and projects from previous versions can be migrated easily!

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) may differ from the usual definitions of the analytical technical mechanics from time to time.

The file formats of the Z88 versions: Z88Aurora® V5, Z88Aurora® V4, Z88Aurora® V3, Z88Aurora® V2, Z88Aurora® V1, Z88 V15 OS and Z88 V14 OS are quite similar but especially Z88Aurora® V5/V4/V3/V2 use more and different input files than former versions for better operation of the GUI. Proper migration tools are included in the software package.

The compatibility of Z88Aurora® with other programs and utilities is not fully tested! It is the aim of this research version to give you an understanding of the fundamental operating concept. The developers of Z88Aurora® are interested in constantly improving this software.

Proposals, suggestions, and remarks can be sent to aurorasupport@z88.de. In addition, FAQs are available on the homepage www.z88.de and users can exchange experiences in a forum.

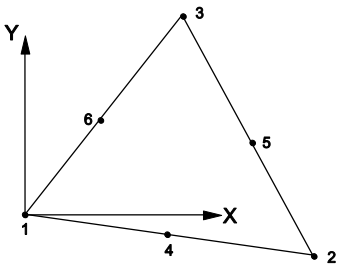
The present version Z88Aurora® V5 was tested on Windows 10 64 bit, Windows 8.1 64 bit, Windows 7 64 bit, Ubuntu 16.04 and openSuSE 12.2 LINUX, and Mac OS-X Sierra.

1.2 SUMMARY OF THE Z88 ELEMENT LIBRARY:

2D PROBLEMS: PLANE STRESS, PLATES, BEAMS, TRUSSES

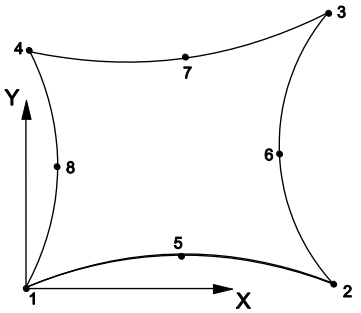
Plane Stress Triangle Element No. 3

- Shape functions quadratic, but linear
- Quality of displacements very good
- Quality of stresses in the centre of gravity good
- Computing effort: average
- Size of element stiffness matrix: 12 × 12



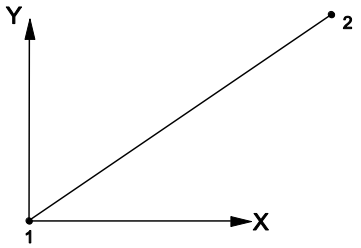
Plane Stress isoparametric Element No. 7

- Quadratic isoparametric Serendipity element
- Quality of displacements very good
- Quality of stresses in the Gauss points very well
- Quality of stresses in the corner nodes good
- Computing effort: High
- Size of element stiffness matrix: 16 × 16



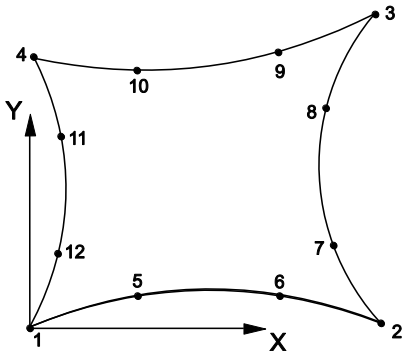
Truss No. 9

- Linear function
- Quality of displacements exact (Hooke's law)
- Quality of stresses exact (Hooke's law)
- Computing effort: Minimal
- Size of element stiffness matrix: 4 × 4



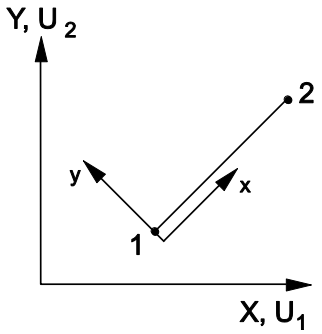
Plane Stress Isoparametric Element No. 11

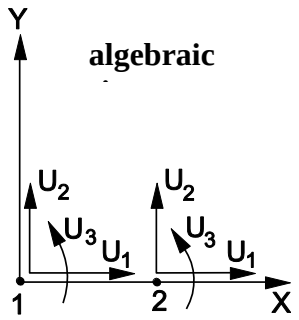
- Cubic isoparametric Serendipity element
- Quality of displacements excellent
- Quality of stresses in the Gauss points excellent
- Quality of stresses in the corner nodes good
- Computing effort: Very high
- Size of element stiffness matrix: 24 × 24



Beam No. 13

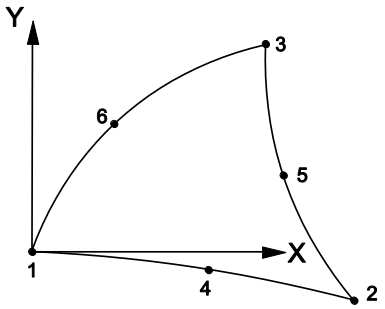
- Linear function for tensile stress, cubic function for bending stress
- Quality of displacements exact (Hooke's law)
- Quality of stresses exact (Hooke's law)
- Computing effort: Low
- Size of element stiffness matrix: 8 × 8





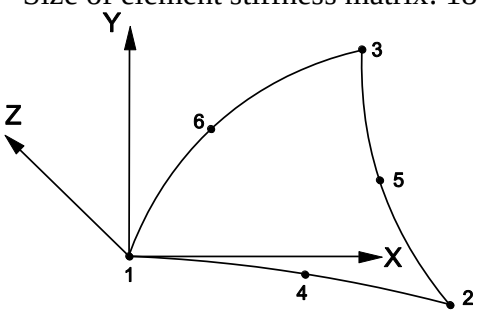
Plane Stress Isoparametric Element No. 14

- Quadratic Isoparametric Serendipity element
- Quality of displacements very good
- Quality of stresses in the Gauss points very good
- Quality of stresses in the corner nodes good
- Computing effort: medium
- Size of element stiffness matrix: 12 × 12



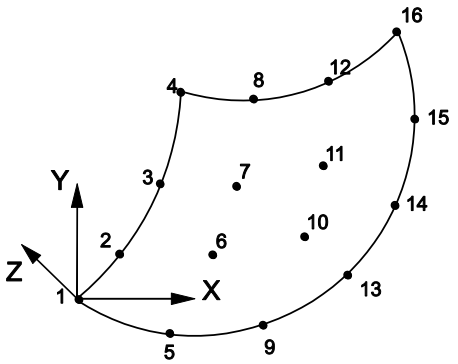
Isoparametric Plate Element No. 18

- Quadratic Isoparametric Serendipity element following Reissner-Mindlin's theory
- Quality of displacements very good
- Quality of stresses in the Gauss points good
- Quality of stresses in the corner nodes acceptable
- Computing effort: medium
- Size of element stiffness matrix: 18 × 18



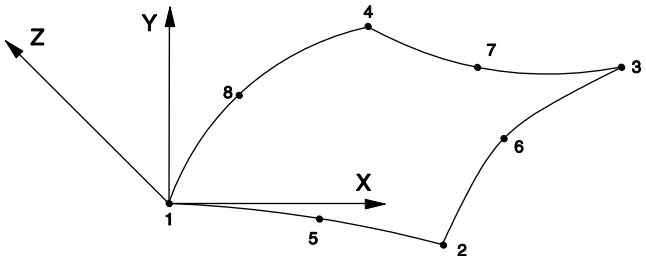
Isoparametric Plate Element No. 19

- Cubic Isoparametric Lagrange element following Reissner-Mindlin's theory
- Quality of displacements very good
- Quality of stresses in the Gauss points very good
- Quality of stresses in the corner nodes good
- Computing effort: High
- Size of element stiffness matrix: 48 × 48



Isoparametric Plate Element No. 20

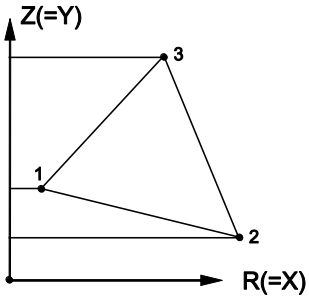
- Quadratic Isoparametric Serendipity element following Reissner-Mindlin's theory
- Quality of displacements very good
- Quality of stresses in the Gauss points good
- Quality of stresses in the corner nodes quite good
- Computing effort: medium
- Size of element stiffness matrix: 24 × 24



AXISYMMETRIC PROBLEMS:

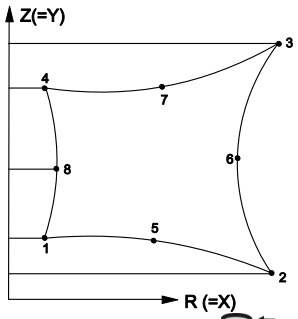
Torus No. 6

- Linear function
- Quality of displacements average
- Quality of stresses in the corner nodes inaccurate
- Computing effort: Low
- Size of element stiffness matrix: 6 × 6



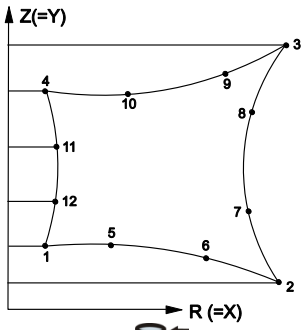
Torus No. 8

- Quadratic Isoparametric Serendipity element
- Quality of displacements very good
- Quality of stresses in the Gauss points very good
- Quality of stresses in the corner nodes good
- Computing effort: High
- Size of element stiffness matrix: 16 × 16



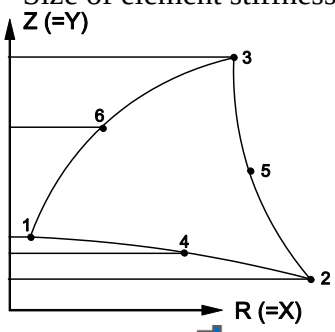
Torus No. 12

- Cubic Isoparametric Serendipity element
- Quality of displacements excellent
- Quality of stresses in the Gauss points excellent
- Quality of stresses in the corner nodes good
- Computing effort: Very high
- Size of element stiffness matrix: 24 × 24



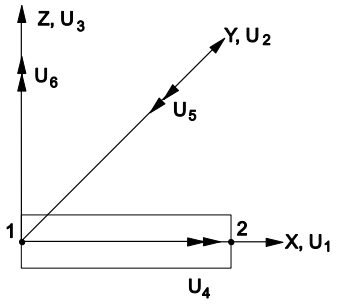
Torus No. 15

- Quadratic Isoparametric Serendipity element
- Quality of displacements very good
- Quality of stresses in the Gauss points very good
- Quality of stresses in the corner nodes good
- Computing effort: High
- Size of element stiffness matrix: 12 × 12



Cam No. 5

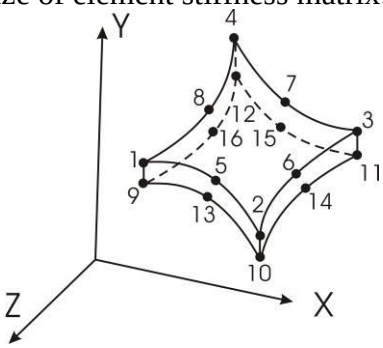
- Linear function for torsion and tensile stress, cubic function for bending stress
- Quality of displacements exact (Hooke's law)
- Quality of stresses exact (Hooke's law)
- Computing effort: Low
- Size of element stiffness matrix: 12 × 12



SHELL PROBLEMS:

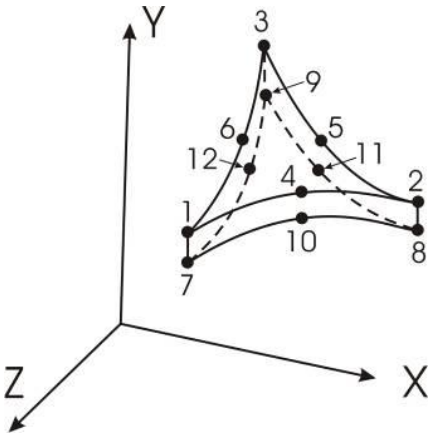
Shell No. 21

- curvilinear, isoparametric Serendipity volume shell element
- isoparametric transformation
- arbitrary curvature of element possible
- good calculation of both displacements and stresses
- Stresses in the corner nodes (good for an overview) or in the Gauss points (substantially more exact)
- Computing effort: high
- Size of element stiffness matrix: 48×48



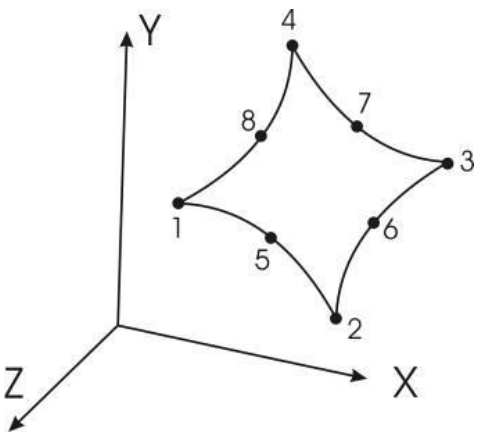
Shell No. 22

- curvilinear, isoparametric Serendipity volume shell element
- isoparametric transformation
- arbitrary curvature of element possible
- good calculation of both displacements and stresses
- Stresses in the corner nodes (good for an overview) or in the Gauss points (substantially more exact)
- Computing effort: average
- Size of element stiffness matrix: 36×36



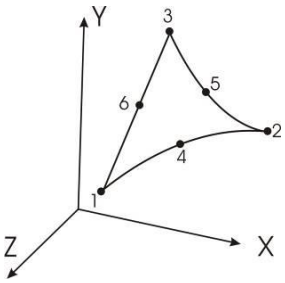
Shell No. 23

- curvilinear, isoparametric Serendipity shell element
- Shape functions quadratic
- isoparametric transformation
- all nodes in one plane
- good calculation of both displacements and stresses
- Stresses in the corner nodes (good for an overview) or in the Gauss points (substantially more exact)
- Computing effort: high
- Size of element stiffness matrix: 48×48



Shell No. 24

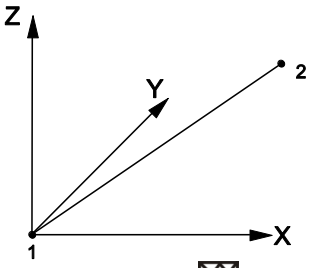
- curvilinear, isoparametric Serendipity shell element
- Shape functions quadratic
- isoparametric transformation
- all nodes in one plane
- good calculation of both displacements and stresses
- Stresses in the corner nodes (good for an overview) or in the Gauss points (substantially more exact)
- Computing effort: average
- Size of element stiffness matrix: 36×36



SPATIAL PROBLEMS:

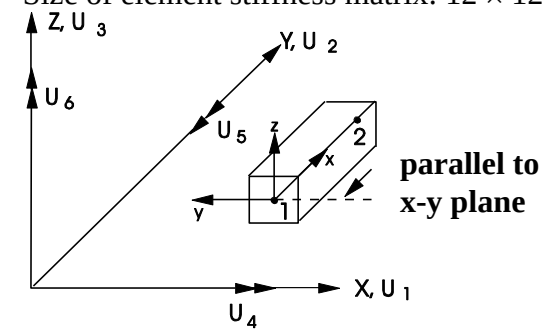
Truss No. 4

- Linear function
- Quality of displacements exact (Hooke's law)
- Quality of stresses exact (Hooke's law)
- Computing effort: Minimal
- Size of element stiffness matrix: 6 × 6



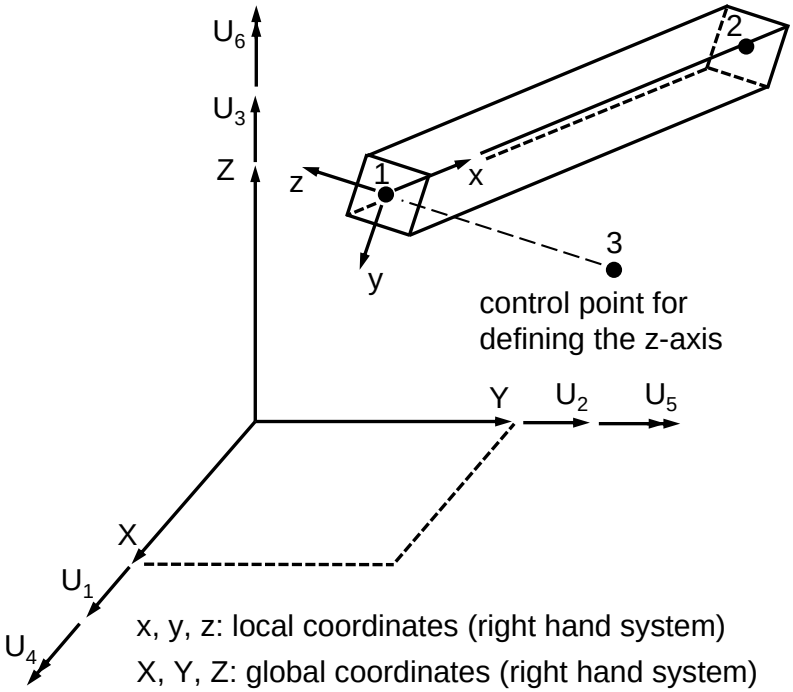
Beam No. 2

- Linear function for tensile stress, cubic function for bending stress
- Quality of displacements exact (Hooke's law)
- Quality of stresses exact (Hooke's law)
- Computing effort: Low
- Size of element stiffness matrix: 12 × 12



Beam No. 25

- Linear function for tensile stress, cubic function for bending stress
- Quality of displacements exact (Hooke's law)
- Quality of stresses exact (Hooke's law)
- Computing effort: Low
- Size of element stiffness matrix: 12 × 12
- Arbitrary definition of the orientation of the cross section in 3D possible
- Control point for defining the orientation
- Theory of Bernoulli and theory of Timoshenko can be used



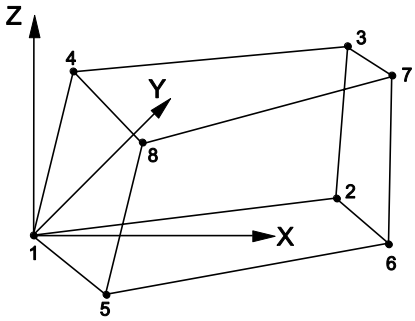
x, y, z: local coordinates (right hand system)

X, Y, Z: global coordinates (right hand system)

U_1, U_2, U_3, U_4, U_5, U_6: degrees of freedom

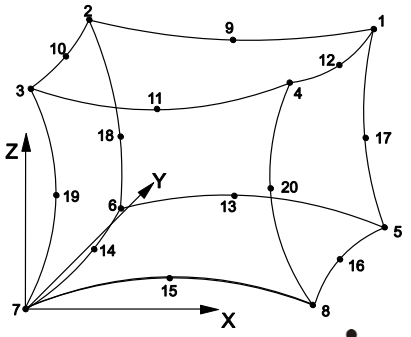
Hexahedron No. 1

- Linear shape functions
- Quality of displacements average
- Stresses in the Gauss points useable
- Stresses in corner nodes inaccurate
- Computing effort: very high
- Size of element stiffness matrix: 24 × 24



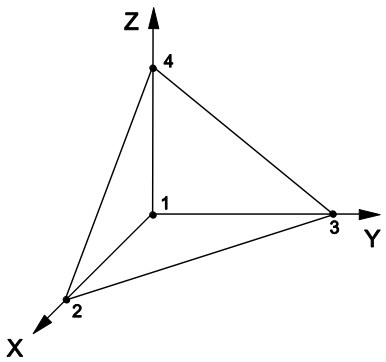
Hexahedron No. 10

- Quadratic Isoparametric Serendipity element
- Quality of displacements very good
- Stresses in the Gauss points very good
- Stresses in corner nodes good
- Computing effort: extremely high
- Size of element stiffness matrix: 60 × 60



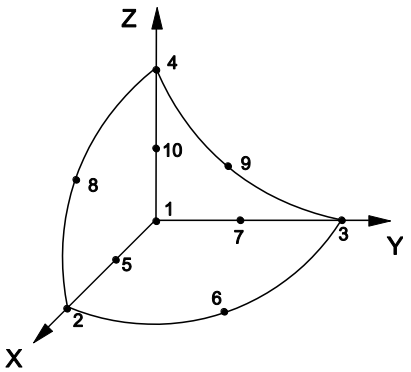
Tetrahedron No. 17

- Linear shape functions
- Quality of displacements bad
- Stresses in the Gauss points inaccurate
- Stresses in corner nodes very inaccurate
- Computing effort: medium
- Size of element stiffness matrix: 12 × 12



Tetrahedron No. 16

- Quadratic Isoparametric Serendipity element
- Quality of displacements very good
- Stresses in the Gauss points very good
- Stresses in corner nodes good
- Computing effort: very high
- Size of element stiffness matrix: 30 × 30



2. THE Z88 COMPUTING UNITS:

2.1 OVERVIEW:

Z88Aurora® always exclusively works on the tasks required at the moment. Under the new user interface the established Z88 programs are launched. Z88 is no gigantic, monolithic program, but consists of several separate running modules according to the UNIX philosophy "Small Is Beautiful". They are loaded into the main memory according to your requirements, execute their tasks and release the main memory again. In this way Z88 achieves its enormous speed and faultlessness beating many other FE programs! The Z88 modules communicate by files, cf. Chapter 3.

2.2 A SHORT DESCRIPTION OF THE MODULES:

I. THE PRE- AND POST-PROCESSOR

In addition to the established Z88 modules, Z88Aurora® possesses a graphic user interface. All input which in Z88 V15 OS was made via the input files Z88I1-Z88I5.TXT is now directly made in Z88Aurora®. But of course, input files from Z88 V15 OS may be loaded directly into Z88Aurora® V5, while existing input files from Z88 V15 OS and Z88Aurora® V1 may be migrated to Z88Aurora® V5 easily (by MITOO). For the plotting of results the approved Z88O was extended and adjusted. Furthermore, for the further use of the results the files Z88O0-Z88O4.TXT can be displayed and printed.

II. THE SOLVERS

2.2.1 THE LINEAR SOLVER Z88R

The linear solver Z88R is the heart of any FEA system. It reads the general structure data, the data for boundary conditions and surface and pressure loads along with the integration order, the elements parameters and the material definitions. Basically, the Z88 input files can be created by CAD converter Z88X, by 3D-converter Z88G, by mesh generator Z88N, by editor or word processor system or by a mixed procedure, e.g. by CAD and editor. The solver generates prepared structure data Z88O0.TXT and processed boundary conditions Z88O1.TXT, 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 in Z88O2.TXT. Thus, the main task of every FEA system, the calculation of displacements, is solved. Thereupon the stresses are calculated and stored in Z88O3.TXT; afterwards the nodal forces are calculated and stored in Z88O4.TXT. Furthermore, the solver generates two files Z88O5.TXT and Z88O8.TXT, which are used for the communication with Z88Aurora®.

Z88R features three different solvers:

- A so-called *Cholesky solver without fill-in.* It is easy to handle and very fast for small and medium structures. It is your choice for small and medium structures, up to 20,000

... 30,000 degrees of freedom. **In Z88Aurora® the Cholesky solver can only be used for truss or beam structures!**

- A so-called *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 a good choice for medium structures, up to 150,000 degrees of freedom on ordinary PCs. However, we've computed structures with ~ 1 million of DoF (degrees of freedom) very fast using a computer featuring 32 (!) Gbyte of memory, 4 CPUs, 64 bit Windows version of Z88.
- A so-called *sparse matrix iteration solver*. It solves the system of equations by the method of conjugate gradients featuring SOR-preconditioning (SORCG) or pre-conditioning by an incomplete Cholesky decomposition (SICCG) depending on your choice. As our tests have shown 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. 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 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 any time, thus, you may use it as your standard solver.*

In Z88Aurora® the solver types are selected via the solver menu:

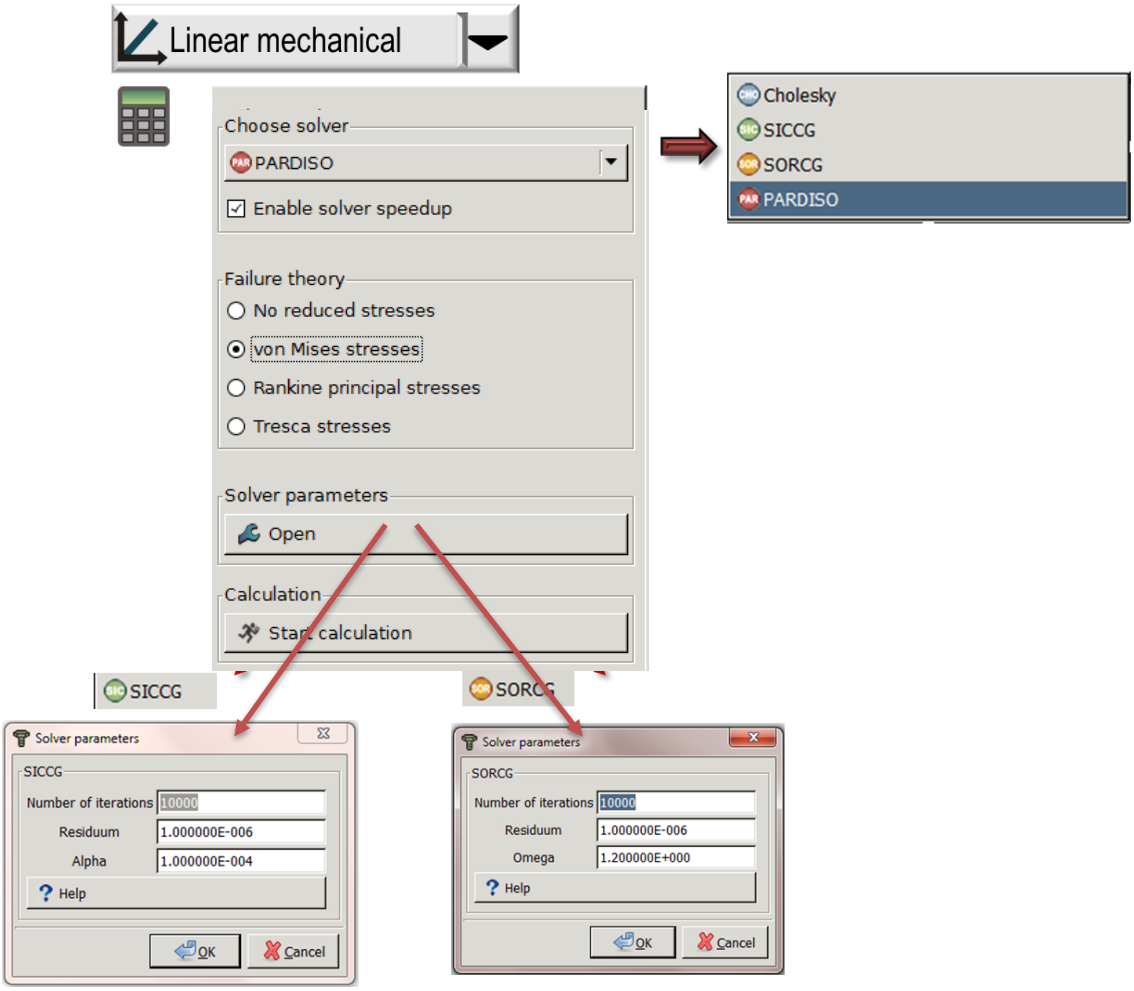


Figure 1: Solver menu

As new features since version Z88Aurora® V2b, there are accelerated versions of the linear solvers. 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 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 accelerated, too. All changes will be known as Z88RS and are used when a check mark is set for "Solver Speedup".

2.2.2 THE VIBRATION SOLVER Z88EI

This eigen solver for natural frequency uses a numeric method that is especially approved in FEA and was introduced already in 1950 by Cornelius Lanczos. Although no one could have thought about numeric model analysis then, the algorithm features 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.

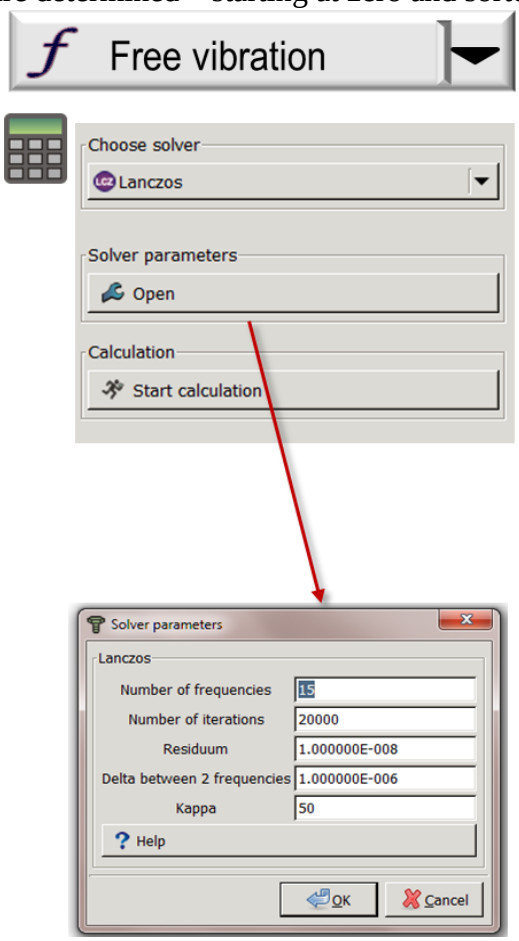


Figure 2: Solver menu of the solver for natural frequencies

2.2.3 THE THERMO SOLVER Z88TH

For the calculation the module for thermo-mechanical 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 necessary 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 runs 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.

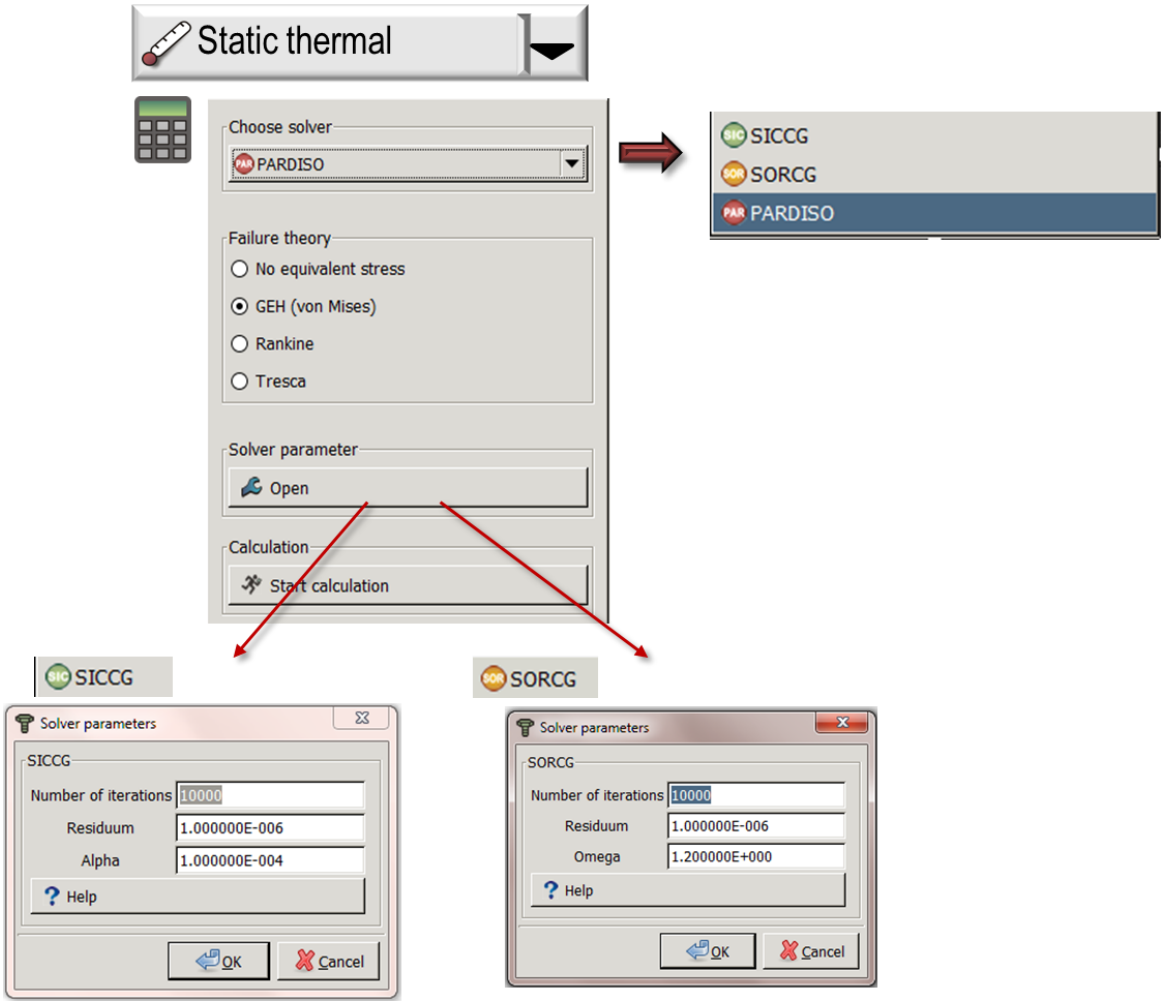


Figure 3: Solver menu of the thermo solver Z88TH

2.2.4 THE NONLINEAR SOLVER Z88NL

The module Z88NL represents an equation solver which is designed for nonlinear calculation. Z88NL can handle large displacement effects (geometrical non-linearity) or material non-linearity (plasticity).

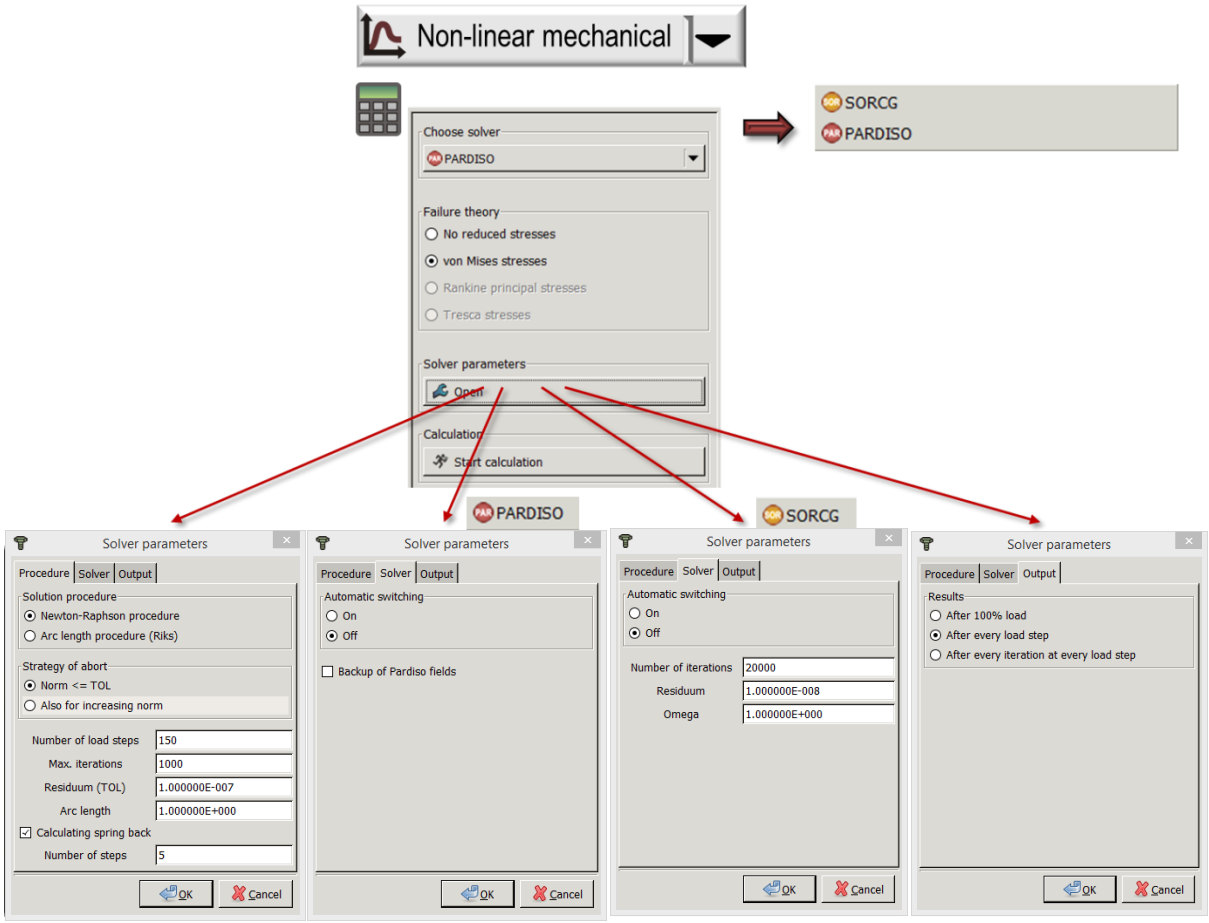


Figure 4: Control of the non-linear solver Z88NL

III. THE INTERFACES TO CAD & FEA SYSTEMS

Z88Aurora® offers the possibility to import a multitude of established file formats from commercial simulation programs, pure geometry data or super structures, as well as to migrate existing files from Z88 OpenSource V15 or Z88Aurora® V1 by the use of our migration tools (MITOO). Each of these converters offers an individual range of functions and its own setting options, if necessary. In chapter 3.2 the functions of the auxiliary programs as well as the procedure when using them is explained in detail. You have the following possibilities:

Import of Z88-files to Z88Aurora®. Import of data decks from previous Z88 releases i.e. Z88V13 and Aurora V1 files may be converted first by MITOO into Z88V14 OS data decks.

STEP import: You may import 3D geometry data in the STEP data format according to DIN ISO 10303 AP 203 and AP 214. This format is supported by most 3D CAD systems.

AUTOCAD DXF import and export: You have the possibility to import 2D and 3D FE structures which were generated AutoCAD and to process. For this a bit of groundwork is needed, c.f. Chapter 4.1.5

STL import: Z88 processes stereo lithography data which contain a triangulated 3D structure. This format is also typically used as input data for CAM programs. This is why most CAD programs can generate this file type.

NASTRAN import: The CAD system Pro/ENGINEER and other commercial programs can write FE data (continuum elements and boundary conditions) as *.nas file. These can be directly imported into Z88Aurora®.

ABAQUS import: Similar to the NASTRAN case, the input files (*.inp) of the program ABAQUS can be loaded.

ANSYS import: Direct transformation of ANSYS-PREP7 data into data for Z88Aurora®.

COSMOS import: The import of COSMOS files known from previous versions is still supported.

IV. THE MAPPED MESHERS

There are three possibilities in Z88Aurora® to improve the mesh:

- super element generator for hexahedrons, axisymmetric elements, plain stress elements, plates and volume shells
- tetrahedron improver
- shell thickener for 2D shells
- STL-refiner for imported STL-meshes

Depending on which element is used different adjustments have to be done.

2.3 WHICH Z88 ELEMENTS CAN BE PRODUCED AUTOMATICALLY?

Table 1: Automatically producible element types

element type	*function*	**.COS*	**.NAS*	**.INP*	**.ANS*	**.DXF*
hexahedron						
hexahedron No. 1	linear	✗	✗	✓	✓	✓
hexahedron No. 10	quadratic	✗	✗	✓	✓	✓
tetrahedron						
tetrahedron No. 16	quadratic	✓	✓	✓	✓	✗
tetrahedron No. 17	linear	✓	✓	✓	✓	✗
plane stress						
plane stress No. 3	quadratic	✗	✗	✗	✗	✓
plane stress No. 7	quadratic	✓	✓	✗	✗	✓
plane stress No. 11	cubic	✗	✗	✗	✗	✓
plane stress No. 14	quadratic	✓	✓	✗	✗	✓
torus						
torus No. 6	linear	✗	✗	✗	✗	✓
torus No. 8	quadratic	✓	✓	✗	✗	✓
torus No. 12	cubic	✗	✗	✗	✗	✓
torus No. 15	quadratic	✓	✓	✗	✗	✓
plate						
plate No. 18	quadratic	✓	✓	✗	✗	✓
plate No. 19	cubic	✗	✗	✗	✗	✓
plate No. 20	quadratic	✓	✓	✗	✗	✓
shell						
shell No. 21	quadratic	✗	✗	✗	✗	✓
shell No. 22	quadratic	✗	✗	✗	✗	✓
shell No. 23	quadratic	✗	✓	✗	✗	✓
shell No. 24	quadratic	✗	✓	✗	✗	✓
truss and beam structures (with special case cam)						
truss No. 4	exact	✗	✗	✗	✗	✓
truss No. 9	exact	✗	✗	✗	✗	✓
beam No. 2	exact	✗	✗	✗	✗	✓
beam No. 13	exact	✗	✗	✗	✗	✓
cam No. 5	exact	✗	✗	✗	✗	✓
beam No. 25	exact	✗	✗	✗	✗	✓

3. THE INPUT AND OUTPUT OF Z88AURORA®

Generally, the input and output files in Z88Aurora®, unlike in Z88 V15 OS, are created while operating the user interface. Of course, it is possible to load existing Z88 V15 files directly and to migrate Z88Aurora® V1 input files by use of our external migration tools to Z88Aurora®. Additionally, all boundary conditions from existing files can be edited and altered directly on the Z88Aurora® user interface. The following table offers an overview of the input and output files:

Table 2: Input and output of Z88Aurora®

Name	**Purpose**
Control files	
Z88.DYN	memory and language header file
Z88.FCD	fonts, colors, dimensions header file
Z88ENVIRO.DYN	setting variables Aurora
Z88MAN.TXT	header file Aurora
1.TXT - *.TXT	material data for data base in Aurora user interface
Z88MARKS.TXT	node and element marks
Z88SETS.TXT	node and element sets for applying boundary conditions and materials
Z88SETSACTIVE.TXT	currently used sets of project
Z88STRUCTURE.TXT	structure data, similar to Z88I1.TXT in Z88 V15
Output files	
Z88O0.TXT	processed structure data
Z88O1.TXT	processed constraints
Z88O2.TXT	computed displacements
Z88O3.TXT	computed stresses
Z88O4.TXT	computed nodal forces
Z88O5.TXT	for internal use in Z88Aurora®
Z88O8.TXT	for internal use in Z88Aurora®
Z88O14.TXT	master and slave contact nodes (for internal use)
Z88TO0.TXT	computed temperature
Z88TO1.TXT	computed heat flow
Z88TO2.TXT	computed thermal expansions
Z88TO3.TXT	computed thermal forces
Z88TO4.TXT	computed displacements
Z88TO6.TXT	computed forces (thermo mechanical)
Z88TO6.TXT	computed stresses (thermo mechanical)
Z88NLO2.TXT	computed displacements, nonlinear calculation with Z88NL
Z88NLO3.TXT	computed Cauchy stresses, nonlinear calculation with Z88NL
Z88NLOH.TXT	computed process variables, nonlinear calculation with Z88NL
Z88AURORA.LOG	protocol file in Z88Aurora®
Z88TRAIL.TXT	protocol file in Z88Aurora®
Import files	
*.COS	COSMOS-FE-file for converter Z88G
*.NAS	NASTRAN-FE-file for converter Z88G
*.INP	ABAQUS-FE-file for converter Z88AINP
*.ASY	ANSYS-PREP7-FE-file for converter Z88ASY
*.DXF	AUTOCAD-CAD-file for converter Z88X
*.STP	import of geometry for internal mesh generator
*.STL	import of geometry for internal mesh generator
Z88NI.TXT	Input file for mesh generator Z88N

Dimensions, i.e. measurement units, are not used explicitly. You may work in optional measurement systems, e. g. in the Metric or Imperial measurement system. Use inches, Newton's, pounds, tons, millimetres, meters, and yards - whatever you prefer. But make sure to keep the one chosen measurement unit throughout all computations of this structure. Example: You want to work with mm and N, thus, Young's modulus must be used in N/mm^2.

Why working with files? Is that not old-fashioned and isn't working "interactively" better? In Z88Aurora® you have both possibilities!
Any kind of pre-processing and post-processing is possible without restrictions. You can generate the input files by small, self-written pre-programs (such a pre-program is the mesh generator Z88N) or leave the job of processing the output data to other programs: You can quite easily load Z88 output files – because it's pure ASCII - into EXCEL and analyse them there. Or you use Z88Aurora® and manually adjust the input in the text editor later, if necessary, because only few boundary conditions have changed or you want to use a different material for calculation. Very often, input files are produced **much faster** than by any interactive queries: Many input lines are similar to prior lines: Use the block operations of your editor for copying and pasting!

Every FEA program can, and so does Z88Aurora®, produce a huge amount of data junk from time to time. You are very often interested only in very specific results, e.g. of special nodes. The output files are simple ASCII files. You may edit and shorten them as you like and print only the **really interesting results**.

Downward compatibility:
Z88 V13 and Z88Aurora® V1 files may be migrated by external migration tools (MITOO). Z88 V14/15 OS and Z88Aurora® V2/V3/V4 files can be opened directly.

This is why we do not want to refrain from explaining the input and output of the program. Z88Aurora® V5 is supposed to be as transparent to experienced users as Z88 V15!

Rules for entering values within the files:
There is no need for special rules or field divisions, only the usual C rules apply:

- *All values are to be separated by at least one blank*
- *Integer numbers may not contain any points or exponents*
- *For floating point numbers, no points need to be provided*
- *Numerical values which are 0 (zero) have to be entered explicitly.*

Integer numbers

Right	1	345	55555	0
Wrong	1.	345.	55555E+0	no entry

Floating point numbers (Z88 uses internally double precision floating point numbers *[Double]*)

Right	1.	345	5.5555E+10	0
Wrong	1,	345,	O (letter O)	no entry

Z88 input files may have comments in every line if all corresponding data has been entered before. Separate the last data and the comment at least by one blank. Lines in Z88 input files

can include 250 bytes (really needed are noticeably less than 80). Blank lines and pure comment lines are not permitted.

Problems which often occur when editing text files:

- Are the files really pure text files in ASCII format?
- Have hidden control characters been added by your text processor without being noticed?
- Is the last line of an input file terminated by at least one *RETURN* ?
- Is the coincidence list defined properly? Especially Hexahedrons No. 10 are very sensitive to wrong numbering.

Z88Aurora® input files for UNIX and Windows have the same structure. You may load without restriction Z88 UNIX files (i.e. LINUX and Mac files) into Windows and vice versa.

As a matter of principle, the user can generate the desired calculation model completely in Z88Aurora®. Users who already know Z88, however, are supposed to get the possibility to edit the input files with an editor or word processor as usual. These edited files may be loaded directly for subsequent use in Z88Aurora®. In case of word processor systems you have to pay attention to create pure ASCII texts, which means without concealed control characters. Every word processor program includes such an option. The solver input files are:

3.1 COMPARISON OF Z88 FILE FORMATS

The file formats of Z88 versions Z88Aurora® V5/V4/V3/V2, Z88 V15/14 OS and Z88Aurora® V1 are very similar, but especially in Z88Aurora® V5/V4/V3/V2 the input is distributed to more and different input files compared to older versions, so that the GUI can be used more efficiently and expansion can be taken into account. The same file structure is used in Z88 V15 OS. Adequate converters for older formats are available.

Table 3: Data formats of the six Z88 versions
Z88Aurora V4, Z88Aurora V3, Z88Aurora® V2, Z88V14 OS, Z88V13 and Z88Aurora® V1

	Z88 V15/14 OS	***Z88Aurora® V5/4/3/2***	***Z88Aurora® V1***	***Z88 V13 OS***
Structure data	*Z88I1.TXT*	*Z88STRUCTURE.TXT*	*Z88I1.TXT*	*Z88I1.TXT*
Material definitions	*Z88MAT.TXT* **.TXT*	*Z88SETSACTIVE.TXT* **.TXT*	*Z88MAT.TXT* **.TXT*	*Z88I1.TXT*
Element parameters	*Z88ELP.TXT*	*Z88SETSACTIVE.TXT*	*Z88ELP.TXT*	*Z88I1.TXT*
Integration orders for displacement calc.	*Z88INT.TXT*	*Z88ENVIRO.DYN*	*Z88MAT.TXT*	*Z88I1.TXT*
Integration orders for stress calc.	*Z88INT.TXT*	*Z88ENVIRO.DYN*	*Z88MANAGE.TXT*	*Z88I3.TXT*
Solver parameters	*Z88MAN.TXT*	*Z88CONTROL.TXT*	*Z88MANAGE.TXT*	*Z88I4.TXT*
Boundary conditions	*Z88I2.TXT*	*Z88SETSACTIVE.TXT*	*Z88I2.TXT*	*Z88I2.TXT*
Surface loads	*Z88I5.TXT*	*Z88SETSACTIVE.TXT*	*Z88I5.TXT*	*Z88I5.TXT*
Number of mat. rules NEG	*Z88MAT.TXT*	*Z88SETSACTIVE.TXT*	*Z88MANAGE.TXT*	*Z88I1.TXT*
Coordinate flag KFLAG	*Z88I1.TXT*	*Z88STRUCTURE.TXT*	*Z88I1.TXT*	*Z88I1.TXT*

Beam -Flag IBFLAG	*Z88MAN.TXT*	*Not applicable*	*Z88I1.TXT*	*Z88I1.TXT*
Plate-Flag IPFLAG	*Z88MAN.TXT*	*Not applicable*	*Z88I1.TXT*	*Z88I1.TXT*
Surface load - Flag IQFLAG	Not applicable	*Not applicable*	*Z88MANAGE.TXT*	*Z88I1.TXT*
Shell-Flag IHFLAG	*Z88MAN.TXT*	*Not applicable*	*Z88I1.TXT*	*Z88I1.TXT*
Radial/tangential-Flag: KDFLAG	*Z88MAN.TXT*	*Z88CONTROL.TXT*	*Z88MANAGE.TXT (here: KSFLAG)*	*Z88I3.TXT (here: KFLAG)*
Equivalent stress-Flag: ISFLAG	*Z88MAN.TXT*	*Z88CONTROL.TXT*	*Z88MANAGE.TXT*	*Z88I3.TXT*

3.2 FILE LAYOUT IN Z88AURORA®

Basically, Z88Aurora® V5 reflects the file structure of Z88 V15 OS and Z88Aurora® V4/V3/V2/V1, but the input is divided into more structure files to guarantee an optimal operation and expansion possibilities. The input files of Z88Aurora® V5 are:

- *Z88STRUCTURE.TXT (general structure data, coordinates, coincidence)*
- *Z88MARKS.TXT (data sets of nodes or elements, which can be used to create SETs)*
- *Z88SETS.TXT (node and element allocation for parts, boundary conditions and material definitions)*
- *Z88SETSACTIVE.TXT (properties of the SETS valid for the current calculation)*
- *1.TXT ~ ?.TXT (material data files)*
- *Z88NI.TXT (input file of mesh generator Z88N)*
- *Z88CONTROL.TXT (control parameters for solver)*
- *Z88DYN.TXT (control file Z88)*
- *Z88ENVIRO.TXT (control file for Z88Aurora®)*

3.2.1 GENERAL STRUCTURE DATA Z88STRUCTURE.TXT

In Z88STRUCTURE.TXT the geometry data of the structure is entered.

<u>1st input group:</u>
General data in the first line contain general structure data. Write all numbers into a line, separate them at least by one blank respectively. All numbers here are of the type [Long].

1st number: Dimension of the structure (2 or 3)
2nd number: Number of nodes of the FEA structure
3rd number: Number of elements
4th number: Number of degrees of freedom
5th number: Coordinate flag KFLAG (0 or 1) Attention: This position was in former Z88 versions reserved for the number of materials NEG.
and the identifier #AURORA_Vx

Explanations:

KFLAG:
On input of 0 the coordinates are expected Cartesian coordinates while on input of 1 polar or cylindrical coordinates are expected. The latter are then converted into Cartesian coordinates and thereupon stored in this form in Z88O0.TXT. Caution: The axisymmetric elements No. 6, 8, 12 and 15 positively expect cylindrical coordinates, set KFLAG to 0 here!

2nd input group:
Starting with line 2, containing coordinates of nodes, one line per node, node numbers strictly ascending.

1st number: node number [Long]
2nd number: Number of the degrees of freedom for this node [Long]
3rd number: X-coordinate or, if KFLAG is 1, R- coordinate [Double]
4th number: Y-coordinate or, if KFLAG is 1, PHI-coordinate [Double]
5th number: Z-coordinate or, if KFLAG is 1, Z-coordinate [Double]

The Z coordinate can be dropped at 2-dimensional structures. Enter angles PHI in radian.

Example 1: *The node No. 156 has 2 degrees of freedom and the coordinates X = 45.3 and Y = 89.7 . > Thus: 156 2 45.3 89.7*

Example 2: *The node No. 68 is supposed to have 6 degrees of freedom (a Beam No. 2 is attached) and cylindrical coordinates R = 100., PHI = 0.7854 (corresponds to 45 °), Z = 56.87. > Thus: 68 6 100. 0.7854 56.87*

3rd input group:
Starting after last node, containing coincidence, i.e. the allocation of the element type and the corresponding nodes of every element. Enter two lines for every finite element. The element numbers, like the node numbers, must be entered strictly ascending.

1st line:
1st number: Element number [Long].
2nd number: Element type (1 to 25) [Long].

2nd line: Depending on element type
1st number: 1st node number for coincidence [Long]
2nd number: 2nd node number for coincidence [Long]
...
20th number: 20th node number for coincidence [Long]

Write all numbers into a line, separate at least by one blank respectively. All numbers here are of type [Long].

Example: *An Isoparametric Serendipity Plane Stress Element No. 7 has element number 23. The coincidence has the global nodes 14, 8, 17, 20, 38, 51, 55, 34 (locally these are the nodes 1-2-3-4-5-6-7-8) > Thus resulting in two lines:*
23 7
14 8 17 20 38 51 55 34

3.2.2 GROUP DATA Z88MARKS.TXT & Z88SETS.TXT

In Z88Aurora® it is possible to create node and element groups using Z88MARKS.TXT and Z88SETS.TXT and afterwards applying part definitions, material properties or boundary conditions in the file Z88SETSACTIVE.TXT. Z88MARKS.TXT contains nodes and elements, which have been selected in the user interface, as a group. You can select surfaces, edges, labeled nodes and elements. For more information please refer to the user manual.
Z88MARKS.TXT works like the marker function in a word processing program.

Z88MARKS.TXT is composed as follows:

1st value: number of markers
1st input group:

1. identifier: #NODES, #ELEMENTS or #SURFACE
2. value: consecutive number of the marker
3. value: number of nodes/elements in the marker
4. "name of the marker"

2st input group:
List of nodes/elements of the respective marker

Explanations:
The markers are written in the files consecutively. How many markers can be imported is controlled by the first value in the file.

Example: There are two markers. The first, "left", contains the nodes No. 2174, 2175, 2176, 2177, 2179. The second, "right", contains the nodes No. 1929, 1931, 1932, 1933, 1934, 1935, 1936.

```
>>
2
#NODES 1 5 "left"
      2174       2175       2176       2177       2179
#NODES 2 7 "right"
     1929       1931      1932       1933       1934       1935       1936
```

With Boolean operations these markers can be used to create sets. Sets defined the allocation of nodes and elements to their application as boundary conditions, material, …

Z88SETS.TXT is composed as follows:

1st value: number of sets
1st input group:

1. identifier: #NODES, #ELEMENTS or #SURFACE
2. keyword (e.g. CONSTRAINTS, CONTACT, MATERIAL, MESH, ELEMENTGEO, PART, UNKNOWN)
3. consecutive number of the set
4. number of nodes/elements in the set
5. "name of set"

2st input group:
List of nodes/elements of the respective set

Explanations:

The sets are written in the files consecutively. How many sets can be imported is controlled by the first value in the file.

Example: There is a set “Material” which contains 22948 elements.

```
>>
1
#ELEMENTS MATERIAL 1 22948 “Material”
         1          2          3          4          5          6          7…
```

Example:

```
3
#NODES CONSTRAINT 1 58 "Constraints"
        86          87          88          89          90          91          92
93         94         95
        96         125         126         127         132         133         134
135        136        137
       138         139         140         141         142         143         144
172        173        466
       467         468         469         470         471         472         473
474        475        476
       477         478         479         480         481         482         483
484        485        486
       487         488         489         490         491         492         493
494
#NODES CONSTRAINT 2 54 "Pressure"
       174         175         176         177         178         179         180
181        182        183
       184         185         186         187         188         189         190
191        321        322
       323         324         325         326         327         328         329
330        331        332
       333         334         335         336         337         338         339
340        341        342
       343         344         345         346         347         348         349
350        351        352
       353        354        355        356
#ELEMENTS MATERIAL 3 2764 "Material"
         1           2           3           4           5           6           7
8          9         10
        11          12          13          14          15          16          17
18         19         20
        21          22          23          24          25          26          27
28         29         30
        31          32          33          34          35          36          37
38         39         40
        41          42          43          44          45          46          47
48         49         50
        51          52          53          54          55          56          57
58         59         60
        61          62          63          64          65          66          67
68         69         70
        71          72          73          74          75          76          77
78         79         80
        81          82          83          84          85          86          87
88         89         90
        91          92          93          94          95          96          97
98         99        100
       101         102         103         104         105         106         107
108        109        110
       111         112         113         114         115         116         117
118        119        120
       121         122         123         124         125         126         127
128
```

3.2.3 CHARACTERISATION FILE Z88SETSACTIVE.TXT

Here is the specific information and the application status determined. Characteristics of the mesh, of the mesher, number of elements, size of elements, size of load, material name or beam properties (cross section, moment of inertia,…) etc. are saved in this file.

1st value: number of lines
1st input group:

1. identifier: #NODES, #ELEMENTS, #MESH, #CONTACT_DEF, #PART
2. keyword (e.g. CONSTRAINTS, MATERIAL, FREE_MESH, MAPPED_MESH, ELEMENTGEO), omitted when using #CONTACT_DEF or #PART
3. value 1 (SET active) or 2 (SET inactive)
4. consecutive number
5. x: value: Property, depending on application

x+1: “name of set”

Example:

```
4
#MESH FREE_MESH 1 1 1 14 17 1.000000E+001 "mesh rule1"
#ELEMENTS MATERIAL 1 2 3 1 1 "steel"
#NODES CONSTRAINT 1 3 1 11 123 1 0.000000E+000 "fixed"
#NODES CONSTRAINT 1 4 2 11 123456 2 5.000000E+000 "pressure"
```

3.2.4 MATERIAL DATA FILE Z88MAT.TXT

The File Z88MAT.TXT features the material data. It is created when the solver has been started. It is defined as follows:

<u>1st input group:</u>
It includes one value that declares the number of following lines. This number corresponds the number of material data.

1st value: number of material data

<u>2nd input group:</u>
Material data. One line for each material. Structure of line:

1st value: from element
2nd value: to element
3rd value: filename of file that contains the material (description in the following chapter)
4th value: material law (only relevant for Z88NL)

*3.2.5 MATERIAL DATA *.TXT*

Aurora features a material database with 50 standard materials. These materials cannot be edited, but you may create and edit copies or create a new material. (For more information refer to the user manual).

Example for material data file:

```
#AURORA_Vx_MATERIAL

#COMMON START
 ID                  1
 NAME1               "Maschinenbau Stahl"
 NAME2               "Engineering steel"
 DESCRIPTION         "E295"
 NUMBER              "1.0050"
 ANNOTATION          "N/mm/t"
#COMMON END

#LINEAR START
 YOUNG_MODULUS       210000.00
 POISSON_RATIO       0.30
 DENSITY             7.85E-009
#LINEAR END

#THERMAL START
 THERMAL_CONDUCTIVITY  0.054
 THERMAL_EXPANSION     1.11E-005
#THERMAL END

#NONLINEAR START
  #MATERIAL_MODELS START
    #VON_MISES 6
      +0.0000000E+000   +6.0000000E+002
      +5.0000000E-002   +8.0000000E+002
      +1.0000000E-001   +9.0000000E+002
      +3.0000000E-001   +1.2000000E+003
      +5.0000000E-001   +1.5000000E+003
      +1.0000000E+000   +3.0000000E+003
    #PDF_MODEL 6
      FLOW_DIRECTION +7.0000000E-001
      +0.0000000E+000   +6.0000000E+002
      +5.0000000E-002   +8.0000000E+002
      +1.0000000E-001   +9.0000000E+002
      +3.0000000E-001   +1.2000000E+003
      +5.0000000E-001   +1.5000000E+003
      +1.0000000E+000   +3.0000000E+003
    #PDFV_MODEL 6 6
      +0.0000000E+000   +6.0000000E+002
      +5.0000000E-002   +8.0000000E+002
      +1.0000000E-001   +9.0000000E+002
      +3.0000000E-001   +1.2000000E+003
      +5.0000000E-001   +1.5000000E+003
      +1.0000000E+000   +3.0000000E+003
      +0.0000000E+000   +4.6000000E-001
      +5.0000000E-003   +4.7000000E-001
      +5.0000000E-002   +8.0000000E-001
      +1.0000000E-001   +8.0000000E-001
      +5.0000000E-001   +6.0000000E-001
      +1.0000000E+000   +5.0000000E-001
  #MATERIAL_MODELS END
#NONLINEAR END
```

3.2.6 MESH GENERATOR INPUT FILE Z88NI.TXT

The layout of Z88NI.TXT is very similar to the layout of Z88I1.TXT or Z88STRUCTURE.TXT, i.e. the input files for the linear FE solver: Only the &-labelled data is required in addition.

Mind the following formats:
[Long] = 4 bytes or 8 bytes integer number
[Double] = 8 bytes floating point number, alternatively with or without point
[Character] = A letter

1st input group:

1st number: Dimension of the structure (2 or 3)
2nd number: Number of nodes of the super structure
3rd number: Number of super-elements
4th number: Number of degrees of freedom
5th number: Coordinate flag KFLAGSS for the super-elements (0 or 1)
& 6th number: Trap radius flag NIFLAG (0 or 1)
& 7th number: Coordinate flag KFLAG for the finite elements (0 or 1)

Write all numbers into a line, separate at least by one blank respectively. All numbers here are of the type [Long].

Explanations:

KFLAGSS:
On input of 0 the coordinates are expected Cartesian while on input of 1 polar or cylindrical coordinates are expected. The latter are then converted into Cartesian coordinates and thereupon stored in this form in Z88I1.TXT if KFLAG (see below) is set to 0. Caution: The axisymmetric elements No. 8 and 12 positively expect cylindrical coordinates, set KFLAGSS to 0 here!

Example: *Super-structure 2-dimensional with 37 nodes, 7 super elements, 74 degrees of freedom. Cylindrical coordinates (= 1), trap radius default value (= 0), Coordinate flag KFLAG for the finite elements 1= output into Z88I1.TXT in cylindrical coordinates. Thus:*
2 37 7 74 1 0 1

NIFLAG:
In order to identify already defined nodes the mesh generator needs a trap radius. The defaults are 0.01 for EPSX, EPSY and EPSZ if NIFLAG is 0. These values can be modified at extremely small or large structures. To initiate this change, set NIFLAG to 1. The new trap radiuses of EPSX, EPSY and EPSZ are then defined in Z88NI.TXT as the 5th input group.

Example: *Super-structure 2-dimensional with 37 nodes, 7 super elements, 74 degrees of freedom. cartesian coordinates, trap radius default value (= 0), output into Z88I1.TXT in cartesian coordinates. Thus:*
2 37 7 74 1 0 0 0

KFLAG:
Internally, Z88N works with natural or Cartesian coordinates. Sometimes, though, you might want to store the output of Z88N as polar or cylindrical coordinates. With this flag (=1) the output takes place in polar or cylindrical coordinates. This is independent from the flag KFLAGSS for the input file Z88NI.TXT.

Example: *Super-structure 2-dimensional with 37 nodes, 7 super elements, 74 degrees of freedom. Cylindrical coordinates (= 1), trap radius default value (= 0), Coordinate flag KFLAG for the finite elements 1= output into Z88I1.TXT in cylindrical coordinates.*
Thus:
2 37 7 74 1 0 1

2nd input group:
Starting in line 2, contains coordinates of nodes, one line per node, node numbers strictly ascending.

1st number: Node number [Long]
2nd number: Number of the degrees of freedom for this node [Long]
3rd number: X-coordinate or, if KFLAG is 1, R- coordinate [Double]
4th number: Y-coordinate or, if KFLAG is 1, PHI-coordinate [Double]
5th number: Z-coordinate or, if KFLAG is 1, Z-coordinate [Double]

The Z coordinate may be skipped for 2-dimensional structures.

Example: *The node No. 8 has 3 degrees of freedom and the coordinates X = 112.45, Y = 0. , Z = 56.75. Thus: 8 3 112.45 0. 56.75*

3rd input group:
Starting after the last node, contains coincidence, i.e. the allocation of the element type and the corresponding nodes of every element. Edit two lines for every super element. The element numbers, like the node numbers, must be entered strictly ascending.

1st line:
1st number: Element number [Long].
2nd number: Super-element type (1, 7, 8, 10, 11, 12, 20, 21) [Long].

2nd line: Depending on element type
1st number: 1st node number for coincidence [Long]
2nd number: 2nd node number for coincidence [Long]
.....
20th number: 20th node number for coincidence [Long]

Write all numbers into a line, separate at least by one blank respectively. All numbers here are of the type [Long].

These are the mesh generator-suitable elements:

Element No. 1: Isoparametric Hexahedrons 8 nodes
Element No. 7: Isoparametric Serendipity Plane Stress Element 8 nodes
Element No. 8: Isoparametric Serendipity Torus 8 nodes
Element No. 10: Isoparametric Serendipity Hexahedron 20 nodes
Element No. 11: Isoparametric Serendipity Plane Stress Element 12 nodes
Element No. 12: Isoparametric Serendipity Torus 12 nodes
Element No. 20: Isoparametric Serendipity Plate 8 nodes
Element No. 21: Isoparametric Serendipity Shell 16 nodes

Example: *An Isoparametric Serendipity Plane Stress Element No. 7 has element number 23. The coincidence has the global nodes 14, 8, 17, 20, 38, 51, 55, 34 (locally these are the nodes 1-2-3-4-5-6-7-8, see chapter 4.7). Thus resulting in two lines:*
23 7
14 8 17 20 38 51 55 34

<u>& 4th input group:</u>
Starting after last coincidence line contains the descriptive details for the mesh generation process. Two lines for every super element.
1st line:
1st number: Super element No. [Long]
2nd number: Finite element type (types 1, 7, 8, 10, 19, 20, 21) to be generated [Long]

2nd line:
1st number: Number of finite elements in local x direction [Long]
2nd number: Type of subdivision of CMODE x [Character]
3rd number: Number of finite elements in local y direction [Long]
4th number: Type of the subdivision CMODE y [Character]
5th number: Number of finite elements in local z direction [Long]
6th number: Type of the subdivision of CMODE z [Character]

The two values for Z are skipped at 2-dimensional structures.

Explanations: **CMODE** can accept the following values:
- "E": Subdivision equidistant, "e" is also permitted
- "L": Subdivision increasing geometrically in local coordinate direction
- "l": Subdivision decreasing geometrically in local coordinate direction

The local x-, y and z axes are defined as follows:
- Local x axis points in direction of local nodes 1 and 2
- Local y axis points in direction of local nodes 1 and 4
- Local z axis points in direction of local nodes 1 and 5

See following sketch below.

Example: Subdivide an Isoparametric Serendipity Plane Stress Element with 12 nodes (Element No. 11) into finite elements of type Isoparametric Serendipity Plane Stress Element with 8 nodes (Element No. 7). Subdivide in local x direction three times equidistantly and subdivide 5 times increasing geometrically in local y direction. The super element is supposed

to have the number 31. Thus resulting in two lines:
31 11
7 3 E 5 L (e or E for equidistant are equivalent)

<u>& 5th input group, optionally after the end of input group 4:</u>
Input group 6 is required if NIFLAG was set to 1, i.e. the trap radius is supposed to be modified. 1 line:

1st number: Trap radius in global X direction EPSX [Double]
2nd number: Trap radius in global Y direction EPSY [Double]
3rd number: Trap radius in global Z direction EPSZ [Double]

Skip the Z detail for 2-dimensionalen structures.

Example: The trap radiuses shall be set to 0.0000003 for X, Y and Z respectively.
> Thus: *0.0000003 0.0000003 0.0000003*
This is effective only if NIFLAG was set to 1 in the first input group!

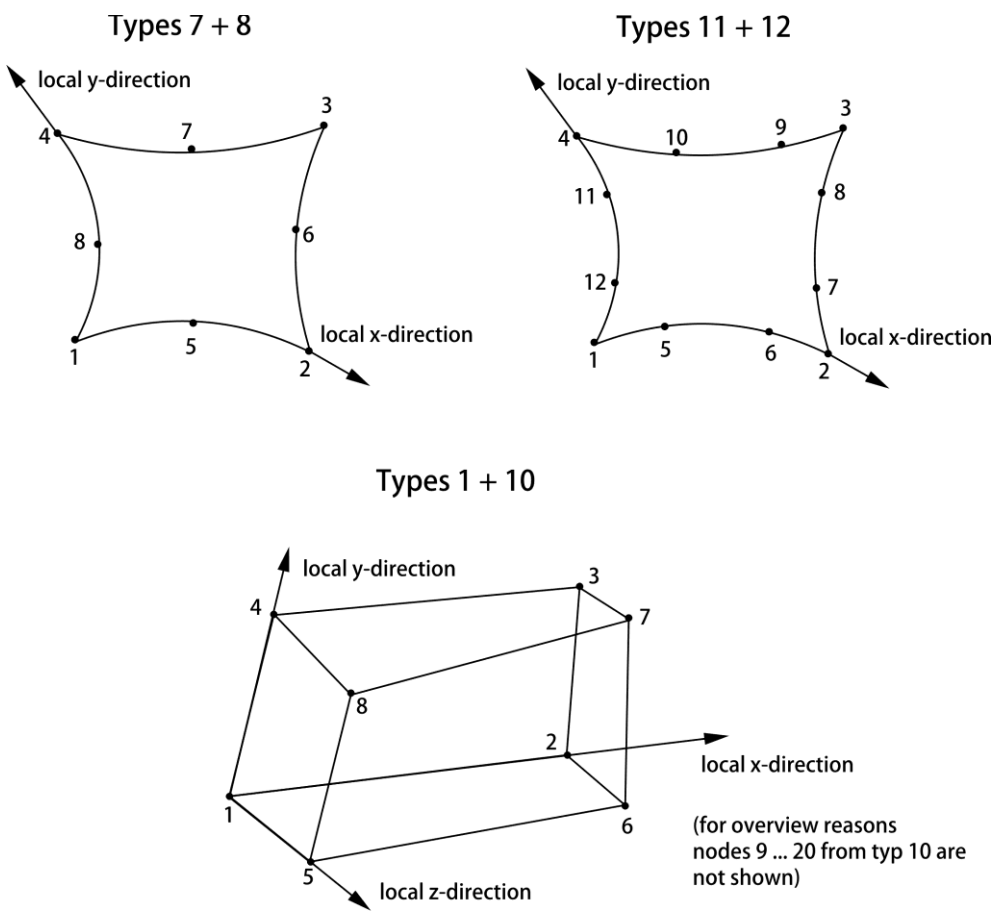


Figure 5:Definition of local x, y and z direction using the example of different element types

3.2.7 SOLVER CONTROL FILE Z88CONTROL.TXT

The solver control file Z88CONTROL.TXT is divided into three parts: the GLOBAL, the solver parts, and the STRESS part. The next figure shows a typical Z88CONTROL.TXT:

```
DYNAMIC START
*-------------------------------------------------------------------------------
Z88Aurora Version 4
--------------------------------------------------------------------------------*

*-------------------------------------------------------------------------------
GLOBAL
--------------------------------------------------------------------------------*

GLOBAL START
   SIMCASE         11
   ICORE           4
GLOBAL END

*-------------------------------------------------------------------------------
CONTACT SETTINGS
--------------------------------------------------------------------------------*

CONTACT START
   CONTACT        1
   FO10           0
   FO11           0
   FO12           0
   FO13           0
   FO14           1
   FO15           0
   FO16           0
   FO17           0
   FO18           0
   FO19           0
CONTACT END

*-------------------------------------------------------------------------------
LINEAR SOLVER
--------------------------------------------------------------------------------*

LMSOLVER START
   ICFLAG          4
   SOLVER_SPEEDUP  -8
   MAXIT           10000
   EPS             1.00E-006
   ALPHA           1.00E-004
   OMEGA           1.20
LMSOLVER END

*-------------------------------------------------------------------------------
NONLINEAR SOLVER
--------------------------------------------------------------------------------*

NLSOLVER START
   ICFLAG          4
   MAXIT           20000
   EPS             1.00E-008
   ALPHA           1.00E-004
   OMEGA           1.20E+000
   NLFLAG          1
   NLAERH          25
   MAXNLIT         1000
   EXIT            1
   TOL             1.00E-007
   AUTOGAUSS       0
   OUTPUTFLAG      1
   OUT_CAUCHY      1
   OUT_INT90FFS    1
   PARSP           0
   ELEFORM         1
   BGLAENG         1.00E+000
   LASTST          0
   SPRING_BACK     0
NLSOLVER END

*-------------------------------------------------------------------------------
VIBRATION SOLVER
--------------------------------------------------------------------------------*

ESSOLVER START
   ICFLAG          5
   MAXIT           20000
   EPS             1.00E-008
   EIGDIFF         1.00E-006
   EIGNUM          15
   EIGSTEP         50
ESSOLVER END

*-------------------------------------------------------------------------------
THERMAL SOLVER
--------------------------------------------------------------------------------*
```

```
TMSOLVER START
   ICFLAG          4
   MAXIT           10000
   EPS             1.00E-006
   ALPHA           1.00E-004
   OMEGA           1.20
   THERMOMODE      1
   THERMOMECHANIC  0
TMSOLVER END

*-------------------------------------------------------------------------------
STRESS
-------------------------------------------------------------------------------*

STRESS START
   KDFLAG          0
   ISFLAG          1
STRESS END

DYNAMIC END
```

Explanation:

GLOBAL

SIMCASE

11 linear static mechanical analysis
19 nonlinear static mechanical analysis
37 natural frequency analysis
66 steady state thermal analysis

ICORE is a control parameter for the calculation of constraints and for the PARDISO solver. It determines the number of CPUs on multi-core computers.

CONTACT SETTINGS:

CONTACT 0 activates the Lagrange method
CONTACT 1 activates the perturbed Lagrange method
CONTACT 2 activates the penalty method

FO10 Write the surface information (0 = inactive; 1 = active)
FO11 Write all element facets (0 = inactive; 1 = active)
FO12 Write master and slave information (0 = inactive; 1 = active)
FO13 Write contact data (0 = inactive; 1 = active)
FO14 Write maser and slave contact nodes (0 = inactive; 1 = active)
FO15 Write contact stiffness (0 = inactive; 1 = active)
FO16 Write global stiffness (0 = inactive; 1 = active)
FO17 Not used
FO18 Calculate contact pressure (0 = inactive; 1 = active)
FO19 Calculate contact shear stress (0 = inactive; 1 = active)

LINEAR SOLVER

ICFLAG 1 Cholesky solver
ICFLAG 2 SIC solver
ICFLAG 3 SOR solver
ICFLAG 4 PARDISO solver

SOLVER_SPEEDUP stating whether the accelerated linear solver to use (yes = -7) or not (no = -8)

MAXIT is the first termination criterion. When reaching this number of iterations the iteration solvers SICCG and analogously SORCG are terminated in any case. The values of the solution vector reached up to this point are printed, however.

EPS This value is compared to a norm of the residual vector. When this value is reached for the iteration solvers SICCG and SORCG, the solution reached should have a good precision. This is the second termination criterion. Enter a relatively small value, e.g. 0.00001 or 0.0000001. Note that there is no absolute truth in this field! No matter which norm of the residual vector is compared against this limit - you can never be sure that all elements of the solution vector are precise. The choice of EPS influences the iteration count and, thus, the computing speed enormously. Remember this when comparing Z88Aurora® to the big, commercial solvers (you do not know which termination criterions are internally used anyway): The limit you can adjust there may have absolutely nothing to do with EPS of Z88. However, extensive tests proved that the deflections of different nodes compared quite well to those from the commercial solvers if EPS was between 0.00001 and 0.0000001 with similar computing time. Please note: When computing large FEA structures with different solvers, you will never know which solver delivers the best result anyway!

ALPHA is *the convergence acceleration parameter* α. With this parameter for the SIC pre-conditioner you choose the shift factor α for the iteration solver SICCG (from 0 to 1, good values may vary from 0.0001 to 0.1). 0.0001 is a good initial value.

OMEGA is *the convergence acceleration parameter* ω. With this parameter for the SOR pre-conditioner you choose the relaxation factor ω for the iteration solver SORCG (from 0 to 2, good values may vary from 0.8 to 1.2). Which value to choose for ω? Good question! Try ω with 1, which will never lead to totally bad results, and then try other values for further runs with this structure.

Example 1:

You want to use the Iterations Sparse matrix Solver and stop after 5000 iterations, the limit is 0.0000001 and the convergence acceleration parameter ω for SOR is 0.9, since you want to use the SORCG-Solver ("SORCG: sparse, iterative"). Thus:

MAXIT	*5000*
EPS	*0.0000001*
ALPHA	*Standard value without significance*
OMEGA	*0.9*

Example 2:

You want to use the Iteration Sparse Matrix Solver and you want to stop positively after 10000 iterations, the limit shall be 10^{-9} and the convergence acceleration factor α for SIC shall be 0.001 because you want to use the SICCG-Solver ("SICCG: sparse, iterative"). Thus:

MAXIT	*10000*
EPS	*1e-9*
ALPHA	*0.001*
OMEGA	*Standard value without significance*

Example 3:

You want to use the direct Sparse Matrix Solver with fill-in ("Pardiso: sparse, direct") and you have two double core CPUs installed in your computer. Thus:

MAXIT	*Standard value without significance*
EPS	*Standard value without significance*
ALPHA	*Standard value without significance*
OMEGA	*Standard value without significance*
ICORE	*4*

Example 4:

You want to use the Cholesky solver. Thus the control values MAXIT, EPS, ALPHA, OMEGA and ICORE can be arbitrary and are without significance.

VIBRATION SOLVER

ICFLAG 5 is the Lanczos solver Z88EI

MAXIT is the first termination criterion. When reaching this number of iterations the Lanczos solver is terminated in any case. The values of the solution vector reached up to this point are printed, however.

EPS is a termination criterion. It's a measurement to determine, that the calculated eigenvalues won't change significantly anymore.

EIGDIFF is the difference between two eigenvalues that determines that the two eigenvalues are handled as two different values.

EIGNUM is the number of frequencies that are determined. Only the lowest frequencies are calculated.

EIGSTEP determines the number of iterations, after which is checked if EPS is reached.

THERMAL SOLVER

ICFLAG 2 → see Linear Solver
ICFLAG 3 → see Linear Solver
ICFLAG 4 → see Linear Solver

MAXIT → see Linear Solver
EPS → see Linear Solver.
ALPHA → see Linear Solver
OMEGA → see Linear Solver

THERMOMODE is an internal control flag. For steady state thermal analysis, and thermo mechanical analysis it has to be set to 1

THERMOMECHANIC is required for thermo-mechanical analysis. It has to be set to 1 (0: steady state thermal analysis)

NONLINEAR SOLVER

ICFLAG 2 → see Linear Solver
ICFLAG 3 → see Linear Solver
ICFLAG 4 → see Linear Solver

MAXIT → see Linear Solver
EPS → see Linear Solver
ALPHA → see Linear Solver
OMEGA → see Linear Solver

NLFLAG determines the method: *Newton-Raphson* (1) or the arc length method by *Riks* (2)

NLAERH determines how many steps are used to apply the total load.

MAXNLIT determines the maximum number of iterations of the nonlinear solver

EXIT quits the solver if the norm is smaller than TOL (1) or if the norm is still rising (2)

TOL termination bound: value of the norm, must be smaller than TOL so that the solution is found.

AUTOGAUSS controls the automatic change of the solver: If the flag is activated (1), the change is done automatically if there are extremely small structures, if it is deactivated (0) there is no change.

OUTPUTFLAG controls the solution output: output only for the total load at the end of the algorithm (0), output after each partial load (1) or output after each iteration (2).

OUT_CAUCHY controls the stress calculation: (1) stress is calculated or (0): no stress calculation

OUT_INT9OFFS activates the provision of a special storage field that is required e. g. for stress calculation (if stress calculation is active, this flag must be active too): 1=active, 0=inactive

PARSP organizes the storage management: if the flag is active (1), additional storage is provided to save the sparse pointer IP and IEZ to accelerate the calculation; if it is inactive (0) an iterative recalculation is carried out.

ELEFORM determines the used element formulation. Default is 1 for the Lagrangian element formulation, that detects geometrical nonlinearities. The Euler element formulation, also suitable for geometric nonlinearities is used with 2. For material nonlinearities, ELEFORM must be set to 3. Then the multi-field formulation of Simo and Hughes is applied. Usually this flag does not have to be set manually, because Z88Aurora® or the converter AG2NL automatically sets the flag in dependence of the constitutive model.

LASTST determines either to use equidistant load steps (without spring back) (set LASTST to 0) or to activate the load control over the file Z88NLI8.TXT (set LASTST to 1). The load control is automatically activated if the spring back should be calculated.

SPRING_BACK activates the spring back calculation and specifies the number of load steps for the calculation. A value of 5 means for example, that five steps are used for the spring back. The spring back calculation is disabled at a value of 0.

BGLAENG determines the arc length for the method by *Riks*

STRESS:

KDFLAG: *[Long]*
0 = standard stress calculation
1 = additional calculation of the radial and tangential stresses for the element No. 3, 7, 11, 14

ISFLAG: *Choice of the reduced stress hypothesis [Long]*
0 = no calculation of the reduced stresses
1 = *von Mises* stresses
2 = principal or *Rankine* stresses
3 = *Tresca* stresses

Example 1: *A structure featuring plane stress elements No. 7 may calculate in addition radial and tangential stresses, thus KDFLAG = 1. The reduced stresses calculation may use the v.Mises criterion: ISFLAG = 1.*
Thus:
KDFLAG 1
ISFLAG 1

Example2: *A structure featuring plane stress elements No. 7 may calculate only default stresses, thus KDFLAG = 0. No reduced stresses calculation: ISFLAG =0.*
Thus:
KDFLAG 0
ISFLAG 0

In the menu *"Solver"* under *"Solver options"* the parameters of the different solvers may be edited. For further information on the use of the solver menu see User Manual.

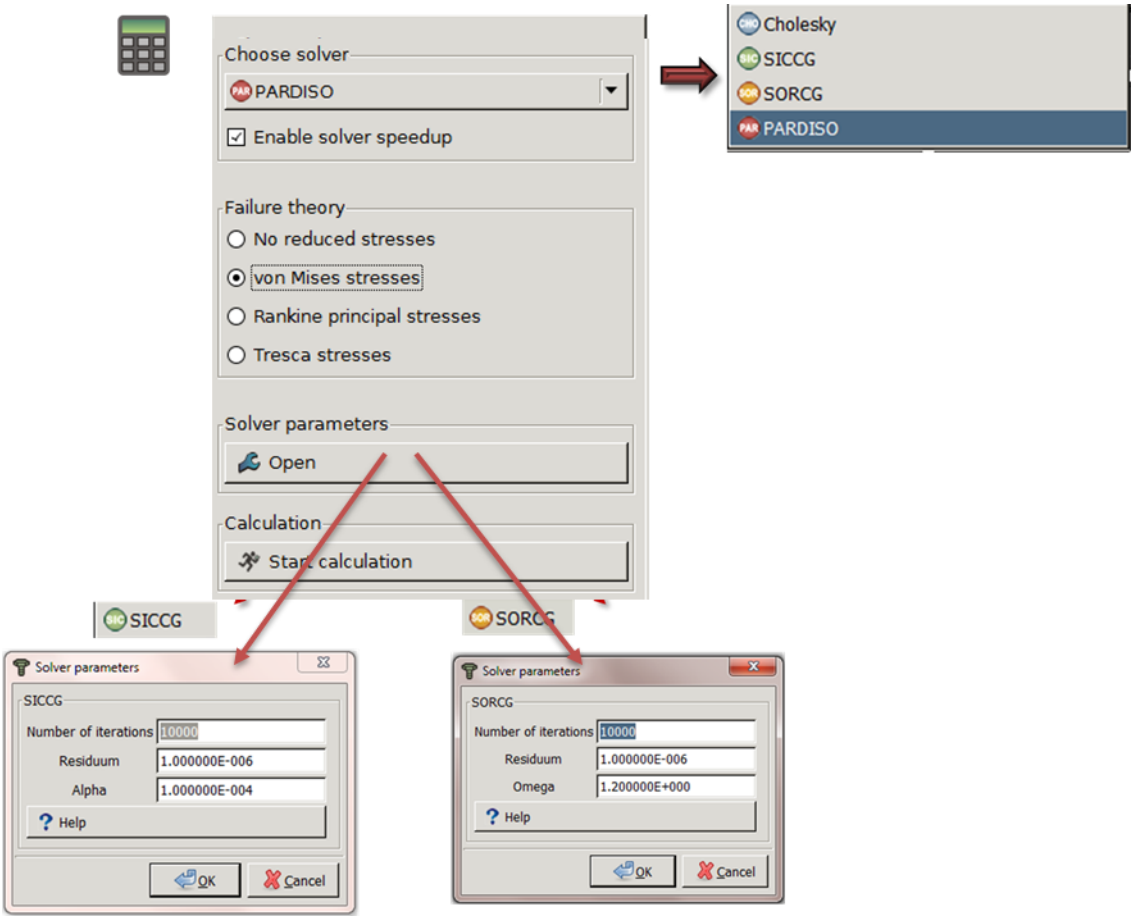


Figure 6: Solver options menu for the control of the solver parameters of the four integrated solver types

3.2.8 SOLVER CONTROL FILE Z88.DYN

General settings, such as memory needs or the appearance of Z88Aurora® are defined in the two definition files Z88.DYN and Z88ENVIRO.DYN. The user can influence their control via the option menu (*under "Help" > "Options"*). For further information about the settings in the option menu see the Z88Aurora® User Manual.

The files are located in the working directory of Z88Aurora®, which is, depending on the platform, in c:/`z88auroravxb/*`(*Selection* operating system/`bin`. The working directory must not be confused with the project directory, which is selected or defined independently by the user when starting the program.

Purpose and structure of the definition file Z88.DYN

At the start of the program Z88Aurora® requests a certain amount of memory, which can be controlled via the file Z88.DYN. Apart from this, Z88.DYN defines the language for Z88Aurora® and any accessed Z88 modules. For the allocation of memory, the file features different parameters which define the maximum possible size of structures to be computed. MAXE, for example, determines the maximum number of elements for the finite element calculation. If it becomes apparent during the use of Z88Aurora® that the memory does not suffice, you will get a respective error message, see figure 7.

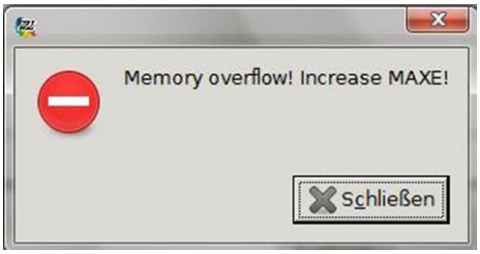


Figure 7: Memory overflow because of too many nodes

After that, the dialog box *"Options"* opens, where the respective parameter can be increased under the tab *"Memory"*, see figure 8. The memory parameters always have an offset of about five for safety and stability reasons. Thus, for the calculation of a model with 1000 elements, the memory parameter MAXE should be set to 1005.
After closing the dialog box, Z88Aurora® is quitted. In the background, the definition file was changed according to the adjustments. When running Z88Aurora® the next time, these changes are taken into account. There will be no data loss! The memory parameters can also be edited without previous memory overflow alert. For this purpose, select the function *"Options*“ in the menu „*Help"*. The tab “Memory” contains all memory parameters MAXK for the maximum node numbers and MAXE for the maximum number of elements. After closing the dialog box Aurora has to be rebooted.
The file Z88.DYN can also be edited manually by experienced users. The important thing is that certain keywords remain in any case. Blank lines or comments are optional, only the uppercased keywords are recognized. After the keyword follows an integer value, separated by at least one blank. The order of the keywords is optional.

There are no limits for the size of the structures for Z88. The maximum size is limited only by virtual memory of your computer and your imagination! However, for very large structures you may use Z88 with 64 bit integers and pointers (i.e. the 64 bit versions for Windows and Mac OS X) to avoid overflows of internal loop counters.

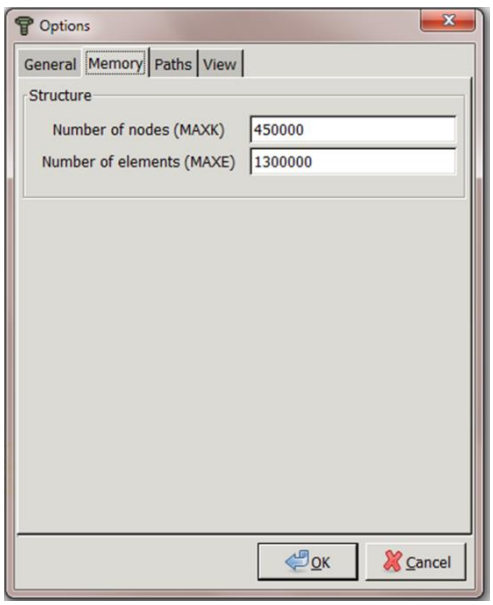


Figure 8: Memory settings in the options menu

In the 64 bit version Z88Aurora® uses

- Floating point numbers with doubles = 8 bytes and
- Integers and pointers with longs = 8 bytes.

Figure 9 shows an example for a file Z88.DYN with the mentioned keywords.

```
DYNAMIC START
*-----------------------------------------------------------------------
Z88Aurora
-----------------------------------------------------------------------*

*-----------------------------------------------------------------------
LANGUAGE
-----------------------------------------------------------------------*
GERMAN

*-----------------------------------------------------------------------
Common entries for all modules
-----------------------------------------------------------------------*

  COMMON START
    MAXE            500000
    MAXK            500000
  COMMON END
DYNAMIC END
```

Figure 9: Example of control file Z88.DYN

The file must start with the keyword DYNAMIC START and end with the keyword DYNAMIC END. By entering GERMAN, German is selected as language, for English select ENGLISH. Between the lines COMMON START and COMMON END there are the memory parameters.

MAXK Maximum number of nodes in the structure.

MAXE Maximum number of elements in the structure.

3.2.9 DEFINITION FILE Z88ENVIRO.DYN

Z88Aurora® includes a project folder management. While working with Z88Aurora®, a project directory must be selected. All input and output files are stored here. The main directory contains the protocol data. Apart from that, several other paths are stored here as well. They permit, for example, the automated access to text viewers like Adobe Reader. Furthermore, some control flags for the configuration of the user interface are stored here, e.g. a flag for the definition of the number of processors set by default (CPU_NUM).

Table 4: List of flags of file Z88ENVIRO.DYN

Flag	Purpose	Possible values
SHOW_SURFACE	Controls if, after starting the program, all elements are shown (-8) or only surface elements (-7)	-8 -7
SCROLLER	Speed of scroll wheel for view display	1 to 299
ROTATOR	Speed of rotation for view display	0,1 to 2,0
TRANSLATOR	Speed of displacement for view display	0,1 to 2,0
RESOLUTION	Size of Aurora input window	Possible sizes, see option menu
MPC_RIGID	No influence in Z88Aurora	-
MPC_USER	No influence in Z88Aurora	-
MPC_TYP	No influence in Z88Aurora	-
CULLING	Backface-Culling. Culling of not viewed surfaces to achieve a faster display	-8: no culling -7: with culling
SPIDER_START	Work flow support SPIDER	-8: no support -7: with support
SHOW_START_INFO	Show the start screen	-8: no -7: yes
CPU_NUM	Number of computing kernels used in the calculation	1 to …
CONFIRM_DIALOG_ TRANSFORM_BC	Determines whether each part manipulation (translation, rotation, scaling) triggers a message box.	1: show 0: do **not** show
CONFIRM_SOLVER_START	Determines whether or when you click the button "Start calculation", the solver is started immediately or only passes through the converter.	1: only converter 0: run immediately
PICKING_USE_CTRL_KEY	Determines whether the picking can be don with the CTRL key (1) or without (0).	1: picking with CTRL 0: picking without CTRL
SHOW_LEGEND_BC	Determines whether the legend of the boundary conditions is displayed.	1: show 0: do **not** show
SHOW_LEGEND_POST	Determines whether the legend of the results is displayed.	1: show 0: do **not** show
INTORD_TYP_1…24	Integration order in compilation	value: number of Gauss points
INTOS_TYP_1…24	Integration order for stress display	value: number of Gauss points
TOOLBAR 1	Shows toolbar; values relate to respective icon, see user manual	0: hidden 1: shown
TOOLBAR 2	Shows additional toolbar; values relate to respective icon, see user manual	0: hidden 1: shown

TOOLBAR 3	Shows additional toolbar; values relate to respective icon, see user manual	0: hidden 1: shown
TOOLBAR 4	Shows additional toolbar; values relate to respective icon, see user manual	0: hidden 1: shown
EXPORT_PIC_WIDTH	Defines the width of the image	integer > 0
EXPORT_PIC_HEIGHT	Defines the height of the image	integer > 0
EXPORT_PIC_REMOVE_BG	Flag to remove the background	0: colored 1: transparent
EXPORT_PIC_FORMAT	Controls the file format of the image	0: Bitmap (.bmp) 1: Portable Network Graphics (.png)
EXPORT_PIC_NAME	Specifies the name of the image to export	ASCII letters (no whitespace)

INTOS_TYP_1…24 defines the number of Gauss points which are used for the stress calculation.

0 = Calculation of stress in the corner nodes, equivalent stress calculation is not possible.

For isoparametric elements Nr.1, 7, 8, 10, 11, 12, 19, 20, 21 and 23:
1, 2, 3 or 4 (i.e. N×N) = Calculation of stress in the Gauss points, equivalent stress calculation is possible. A good value is 3 (= 3×3 gauss points). For type 1 and type 20 2 (=2x2 gauss points) may suffice, for type 19 4 (=4x4 gauss points) is the right value.

For isoparametric elements Nr.14, 15, 18, 22 and 24:
3, 7 or 13 (i.e. N) = Calculation of stress in the Gauss points, equivalent stress calculation is possible. A good value is 7 (=7 gauss points). For type 18 3 gauss points may suffice

For isoparametric elements Nr.16 and 17:
1, 4 or 5 (i.e. N) = Calculation of stress in the Gauss points, equivalent stress calculation is possible. A good value is 7 for type 16.
For type 17 1 gauss point may suffice
.

This value has no meaning for elements Nr.2, 3, 4, 5, 6, 9, 13 and 25. It's best to enter a 1.

In figure 10 you'll find an example of the definition file Z88ENVIRO.DYN. Z88ENVIRO.DYN is also located in the subdirectory c:/`z88auroraVx/*`(*Selection* operating system)`/bin`.

*----------

```
Z88Aurora Version 5
-----------------------------------------------------------------------------*


**FLAGS zum Ansteuern der verschiedenen Optionen
FLAG  SHOW_SURFACE          -8
FLAG  SCROLLER              150
FLAG  ROTATOR               1.000000
FLAG  TRANSLATOR            1.000000
FLAG  RESOLUTION            800 x 600
FLAG  CULLING               -8
FLAG  SPIDER_START          -7
FLAG  SCOUNTER              257
FLAG  SHOW_START_INFO       -8
FLAG  LOAD_INFO_OFF         -8
FLAG  CPU_NUM               8
FLAG  CONFIRM_SOLVER_START 0
FLAG  CONFIRM_DIALOG_TRANSFORM_BC 0
FLAG  PICKING_USE_CTRL_KEY 1
FLAG  SHOW_LEGEND_BC        1
FLAG  SHOW_LEGEND_POST      1
FLAG  GOD                   0
FLAG  STYLE_POST            1
FLAG  POSITION_CSYS         2
FLAG  EXPORT_PIC_WIDTH      1920
FLAG  EXPORT_PIC_HEIGHT     1080
FLAG  EXPORT_PIC_REMOVE_BG 0
FLAG  EXPORT_PIC_FORMAT     0
FLAG  EXPORT_PIC_NAME       DefaultPic
FLAG  MODEL_ROTATION        0.000000,0.000000,0.000000
FLAG  PROJECT_PIC_PREVIEW   1
FLAG  INTORD_TYP_1          3
FLAG  INTORD_TYP_7          3
FLAG  INTORD_TYP_8          3
FLAG  INTORD_TYP_10         3
FLAG  INTORD_TYP_11         3
FLAG  INTORD_TYP_12         3
FLAG  INTORD_TYP_14         7
FLAG  INTORD_TYP_15         7
FLAG  INTORD_TYP_16         4
FLAG  INTORD_TYP_17         4
FLAG  INTORD_TYP_18         3
FLAG  INTORD_TYP_19         4
FLAG  INTORD_TYP_20         2
FLAG  INTORD_TYP_21         3
FLAG  INTORD_TYP_22         7
FLAG  INTORD_TYP_23         3
FLAG  INTORD_TYP_24         7
FLAG  INTOS_TYP_1           2
FLAG  INTOS_TYP_7           3
FLAG  INTOS_TYP_8           3
FLAG  INTOS_TYP_10          3
FLAG  INTOS_TYP_11          3
FLAG  INTOS_TYP_12          3
FLAG  INTOS_TYP_14          7
FLAG  INTOS_TYP_15          7
FLAG  INTOS_TYP_16          4
FLAG  INTOS_TYP_17          4
FLAG  INTOS_TYP_18          3
FLAG  INTOS_TYP_19          4
FLAG  INTOS_TYP_20          2
FLAG  INTOS_TYP_21          3
FLAG  INTOS_TYP_22          7
FLAG  INTOS_TYP_23          3
FLAG  INTOS_TYP_24          7


**PFADE zu externen Programmen
PATH  P01_ACROBAT      "C:\Program Files\Internet Explorer\iexplore.exe"
```

```
PATH   P02_PLAYER        "C:\Program Files\Windows Media Player\wmplayer.exe"
PATH   P03_BROWSER       "C:\Program Files\Internet Explorer\iexplore.exe"
PATH   P04_PROJECT       "C:\Z88AuroraV5\docu\examples\project\b25"


**Buttons der Toolleisten
TOOLBAR 1 1
2 3 4 0 1 0 71 73 74 0 5 6 7 8 0 9 10 0 11 -1
TOOLBAR 2 1
50 51 52 53 0 54 55 27 72 0 22 23 24 25 49 -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 10: Example Z88ENVIRO.DYN

Lines which are preceded by * are ignored by Z88Aurora®. They are comment lines. Behind PATH there is the directory of extern programs. The paths have to be written in uppercase letters, otherwise blinds cannot be imported correctly. The flags listed in Table 4 are marked by the preceding keyword FLAG. They can be changed via *"Help"* > *"Options"* in the tab *"View"*, see figure 11.

Behind "PATH" you find the paths of external programs which can be automatically accessed. If no path is present, C:\ is used as standard. These paths as well as all other settings of Z88ENVIRO.DYN can be modified under "*Options*" in the menu "*Help*", see figure 11. Experienced users can edit them directly in the file. The path “P04” changes dynamically, because it contains the last project directory, which is used as basis to open the project folder.

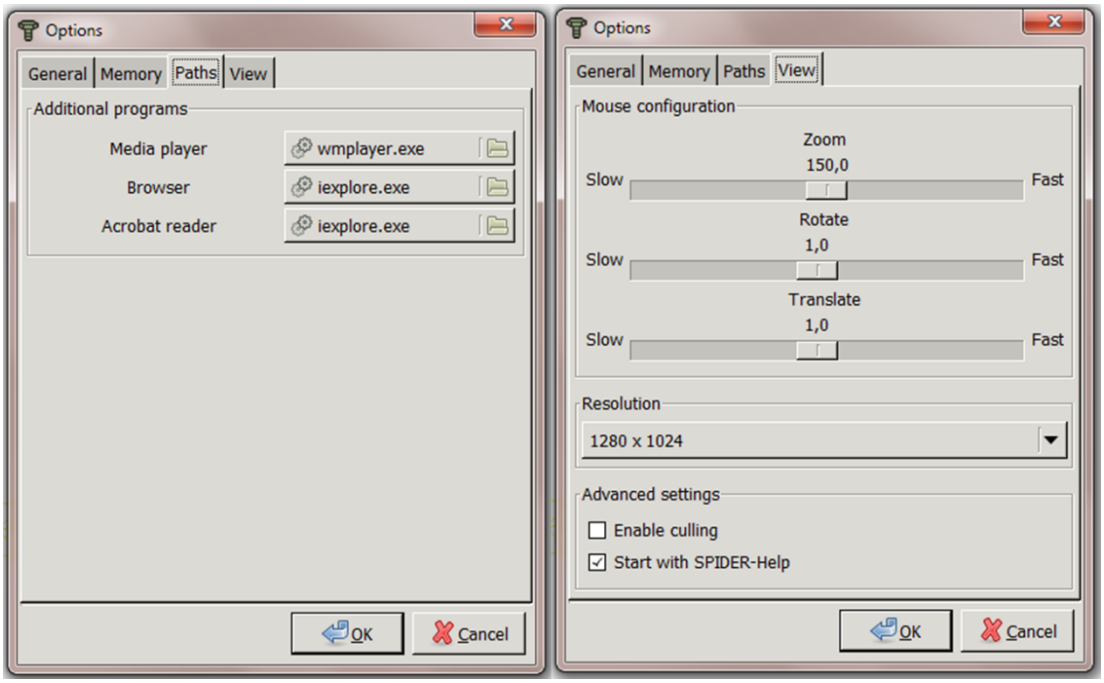


Figure 11: Flags for the standard view (left) and the path settings (right)

3.2.10 FILE FOR HISTORY VARIABLES Z88NLI7.TXT

This file only is relevant to the non-linear solver Z88NL. It allows e.g. recording the displacement of a node over the individual load steps. In this way, force-displacement characteristics in discrete locations can be created quickly. This file is not created automatically, you

must create it manually.
Hereafter, the structure of the file Z88NLI7.TXT is shown:

```
nhv
var[1] loc[1] iot[1]
var[2] loc[2] iot[2]
...
var[nhv] loc[nhv] iot[nhv]
```

The number nhv represents the number of different history variables. Such a variable can be e.g. the strain λ, whereat in this case all entries of the strain vector are listed. For each variable, the file Z88NLI7.TXT contains a line with the three key figures var, loc and iot. The number var determines the physical dimension, e.g. elongation, the number loc defines the location, for example, node or element number, and the number iot the integration point, if it is an element. If it is a node, it is recommended to set this number identical -1. Below all relevant indicators are listed:

key figure	**meaning**	**example**
nhv	number of history variables	3
var	identifier of the variables	2
loc	location of the variables	2589
iot	integration point number	4

The physical dimensions that can be recorded have a unique variable type. These variable types are listed below:

variable type	physical dimension	list of outputs
1	load step i	i
2	displacements $\boldsymbol{u}$	u, v, w
3	stresses $\boldsymbol{P}$ (nominal stresses)	$P_{11}, P_{12}, P_{13}, P_{21}, P_{22}, P_{23}, P_{31}, P_{32}, P_{33}$
4	strain $\boldsymbol{\lambda}$	$\lambda_{11}, \lambda_{22}, \lambda_{33}, \lambda_{12}, \lambda_{23}, \lambda_{31}$
5	material matrix $\boldsymbol{C}$	$C_{11}, C_{12}, C_{13}, C_{14}, C_{15}, C_{16,}$ $C_{21}, C_{22}, C_{23}, C_{24}, C_{25}, C_{26,}$ … $C_{61}, C_{62}, C_{63}, C_{64}, C_{65}, C_{66}$
6	Cauchy-stesses $\boldsymbol{\sigma}$ (true stresses)	$\sigma_{11}, \sigma_{22}, \sigma_{33}, \sigma_{12}, \sigma_{23}, \sigma_{31}$

7	equivalent plastic strain	ε_{vgl}^{pl}
8	comparison stress	σ_v
9	inner forces	I_k
10	sum of inner forces	$\sum_k I_k$

The last type is a special feature: here all history variables of type 9 (inner nodal forces) are summed up. Therefore, type 10 may occur only once and also requires at least one variable of type 9. Type 10 is very useful if you e.g. prescribe displacements and would like to know the corresponding force.

Finally, an example of an input file Z88NLI7.TXT is shown.

```
3
2 27 -1
5 389 4
3 391 1
```

In this example, three history variables are created. The first variable stores the displacements of node 27 after each load step. The second is the material matrix of the 4th integration point of the 389th element. Finally, the developing of the stress of the first integration point of element no. 391 is recorded. If a file Z88NLI7.TXT exists, the solver Z88NL creates an additional output file Z88NLOH.TXT. This has the following structure:

```
nhv
var[1] loc[1] iot[1]
var[2] loc[2] iot[2]
...
var[nhv] loc[nhv] iot[nhv]
var[1] loc[1] iot[1]
1 Wert_x Wert_y Wert_z
2 Wert_x Wert_y Wert_z
...
n Wert_x Wert_y Wert_z
var[2] loc[2] iot[2]
1 Wert_x Wert_y Wert_z
2 Wert_x Wert_y Wert_z
...
n Wert_x Wert_y Wert_z
...
var[nhv] loc[nhv] iot[nhv]
1 Wert_x Wert_y Wert_z
2 Wert_x Wert_y Wert_z
...
n Wert_x Wert_y Wert_z
```

Here n is the number of load steps. If it is about stresses, respectively there are values Wert_xx, Wert_yy, Wert_zz, Wert_xy, etc.

3.2.11 OUTPUT FILES Z88O.TXT

The following list is an overview of the Z88Aurora® 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 thermal flow)*
- *Z88TO2.TXT – (calculated thermal expansion)*
- *Z88TO3.TXT – (calculated thermal forces)*
- *Z88TO4.TXT – (calculated displacements)*
- *Z88TO5.TXT – (calculated nodal forces)*
- *Z88TO6.TXT – (calculated forces (thermo-mechanical))*
- *Z88TO7.TXT – (calculated stress (thermo-mechanical))*
- *Z88NLO2.TXT – (calculated displacements, nonlinear calculation with Z88NL)*
- *Z88NLO3.TXT – (calculated Cauchy stress, nonlinear calculation with Z88NL)*
- *Z88NLOH.TXT – (calculated history variables, nonlinear calculation with Z88NL)*

The files Z88O5.TXT, Z88O8.TXT, Z88TO5.TXT, Z88TO8.TXT and Z88O14.TXT are no regular Z88 output files. They are internally used for the postprocessor and stored as ASCII files, so that experienced users can use them for their own routines, if necessary

4. THE Z88 MODULES

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

4.1 CAD & FE INTERFACES

Z88Aurora® offers the possibility to import a multitude of established file formats from commercial simulation programs, pure geometry data or super structures, as well as to load existing FE data from Version 15/14 of the Open Source program Z88. Each of these converters offers an individual range of functions and its own setting options, if necessary. Z88V13 and Z88Aurora® V1 files can be migrated with the external migration tool MITOO into the Z88V15 file format.
But since especially the actual proprietary data formats of simulation programs do not meet any national or international standards, the respective producers can conduct changes in the files when issuing a new version, which may influence the converters. When using neutral formats for geometry or product data (STL or STEP), some appropriate adjustments might have to be made in the CAD programs, in order to generate a functioning FE model in the desired accuracy.

CAUTION: In the following paragraphs the range of functions of the converters are listed as well as the programs with which they were tested. In spite of intensive tests we cannot guarantee the compatibility of files from other programs or newer versions. Please note the respective support in the explanations.

There are two possibilities to access the import and export functions of files:
1. Via the text menu (figure 12)
2. Via the toolbar (figure 13)

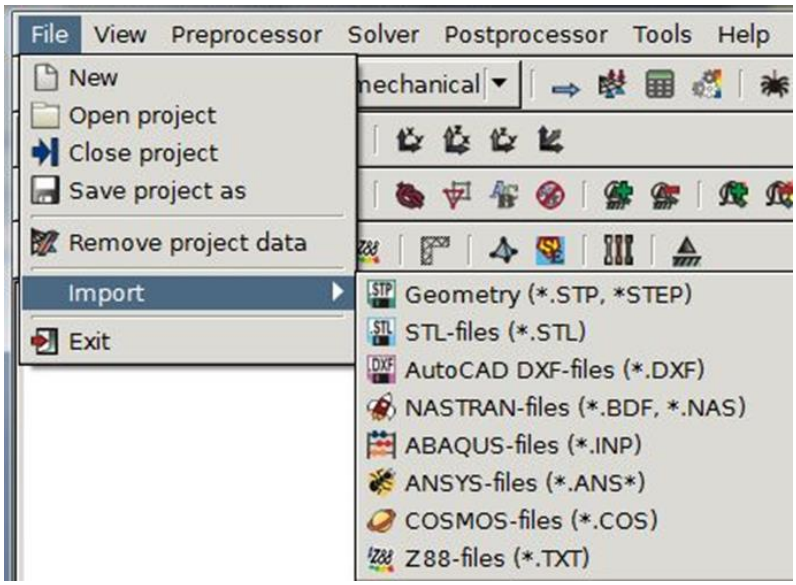


Figure 12: Import in the text menu

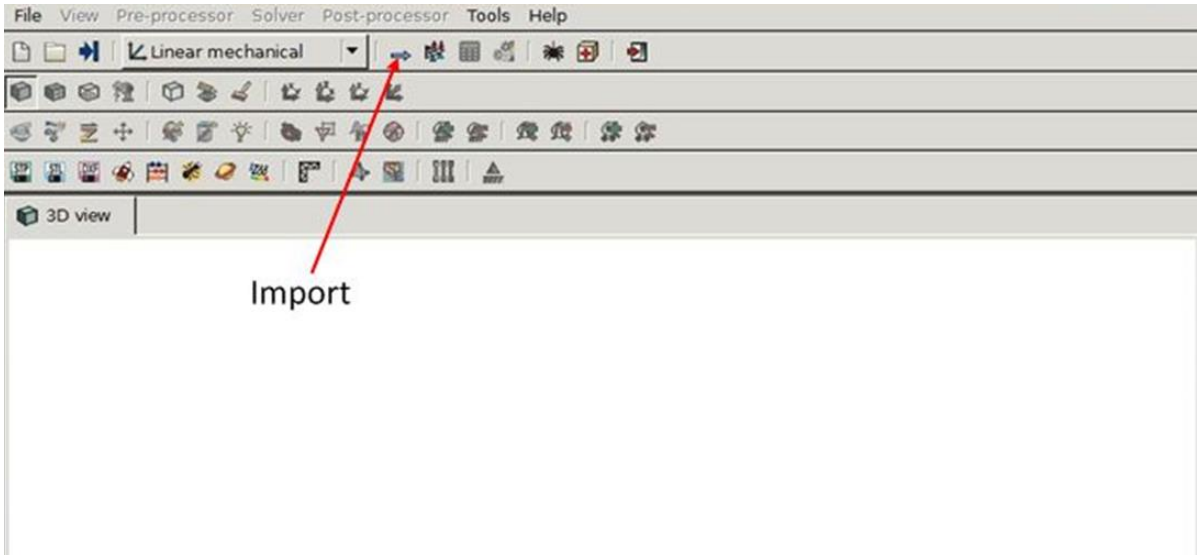


Figure 13: Import and export via the toolbar

Depending on the range of functions of the converter, a multitude of FE model data can be imported or exported. You have the possibility to generate a complete FE structure or a super structure which can be meshed further by means of the integrated mapped mesher Z88N. The finite elements of the source program are properly transformed into the corresponding type in Z88Aurora®, and material data can be adopted.

As expected, the pure geometry interfaces only contain the function to import a 3-dimensional image without any FE information.

Table 5: Model data which can be transferred from FE structure data

	Z88V15OS *	DXF AutoCAD	ABAQUS	ANSYS	COSMOS	NASTRAN
FE structure	✓	✓	✓	✓	✓	✓
FE super structure	✓	×	×	×	×	×
Point loads	✓	×	✓	✓	✓	✓
Boundary Conditions	✓	×	✓	✓	✓	✓
Surface and pressure loads	✓	×	✓	✓	×	✓

* (via MITOO)

Please note:
The import of AutoCAD DXF files only imports geometry data or FE mesh data with optionally boundary conditions. The material data, elements parameters and integration orders can be defined easily in Z88Aurora® afterwards. Only then data consistency in Z88Aurora® projects can be guaranteed.

NASTRAN, ANSYS, ABAQUS, and COSMOS files can be imported with boundary conditions. The material properties cannot be transferred intentionally; they are added in Z88Aurora®. An overview over material data that can be imported can be found in table 5, information about the possible element types can be found in table 1 in chapter 2.3.

4.1.1 IMPORTING Z88 FILES

What is the basic idea and which are the features?
Downward compatibility is one of the basic prerequisites for the effective application of a simulation system. Who wants to revise their models every six months? Therefore all input files from previous versions can be used in Z88Aurora® V5. If necessary, only a few small adjustments must be made.

Which Z88 versions can cooperate with Z88Aurora® V4?
Basically every input file from every previous version can be imported. The function is mainly intended for the import of files from Z88 V14/V15 OS or Z88Aurora®V2/V3/V4 and – via the migration tool MITOO – Z88 V13 and Z88Aurora® V1. Therefore, older files must be updated to these versions. For this purpose it is often sufficient to add some flags or values.

The Migration Tool MITOO
Files from Z88V14/15 OS and Z88Aurora® V2/V3/V4 can be imported directly to Z88Aurora® V5 without MITOO.
MITOO may be used for an easy migration of Z88 V13 and Z88Aurora® V1 files. MITOO generates a Z88V14/15 OS data set (Z88I1.TXT, Z88I2.TXT, Z88I5.TXT, Z88MAN.TXT, Z88MAT.TXT, *.TXT, Z88INT.TXT, Z88ELP.TXT).

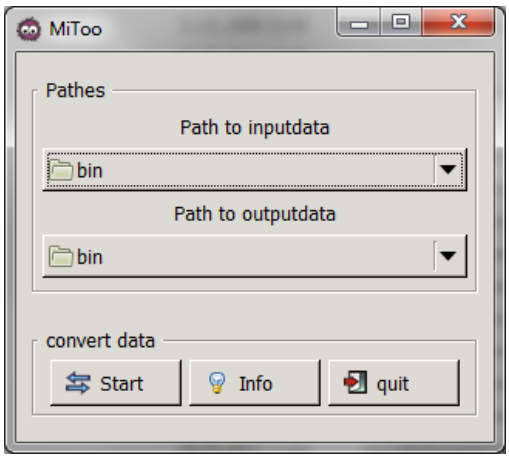


Figure 14: Migration tool MITOO for older Z88 files

The executable MITOO can be found in the `bin` folder. Double clicking opens the migration dialogue. Selecting the respective folder and “Start” converts the data.

Afterwards, the import menu and the import can be done as usual in Z88Aurora®, see fig. 15. For complete projects the following import option can be chosen:

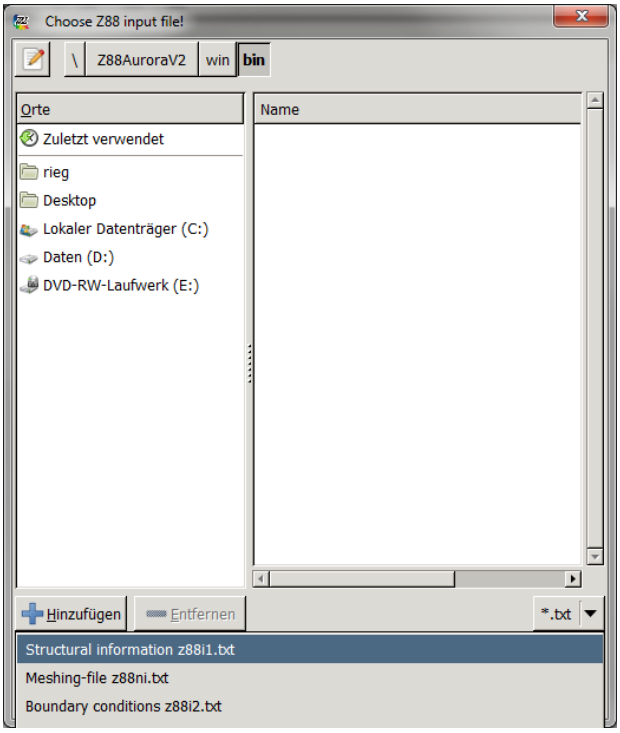


Figure 15: Import of Z88 files

Application of Z88Aurora® files

For further processing of files in Z88V14/15 OS, you have to insert "enable write_only" in `Z88.FCD`. With this the files are created as a complete data set for Z88V14/15 OS in the folder `Z88V14OSdata` in the `bin` directory. This folder is overwritten. If you want to use these files later, you should move them to another folder.

4.1.2 MANUALLY CREATING Z88 FILES

Z88V14 files can be created manually and imported into Z88Aurora®. For this the files are assembled by hand.

GENERAL STRUCTURE DATA Z88I1.TXT

In Z88I1.TXT the geometry data of the structure is entered.

<u>1st input group:</u>

General data in the first line contain general structure data. Write all numbers into a line, separate them at least by one blank respectively. All numbers here are of the type [Long].

1st number: Dimension of the structure (2 or 3)
2nd number: Number of nodes of the FEA structure
3rd number: Number of elements
4th number: Number of degrees of freedom
5th number: Coordinate flag KFLAG (0 or 1) Attention: This position was in former Z88 versions reserved for the number of materials NEG.

<u>Explanations:</u>

KFLAG:

On input of 0 the coordinates are expected Cartesian coordinates while on input of 1 polar or

cylindrical coordinates are expected. The latter are then converted into Cartesian coordinates and thereupon stored in this form in Z88O0.TXT. Caution: The axisymmetric elements No. 6, 8, 12 and 15 positively expect cylindrical coordinates, set KFLAG to 0 here!

2nd input group:
Starting with line 2, containing coordinates of nodes, one line per node, node numbers strictly ascending.

1st number: node number [Long]
2nd number: Number of the degrees of freedom for this node [Long]
3rd number: X-coordinate or, if KFLAG is 1, R- coordinate [Double]
4th number: Y-coordinate or, if KFLAG is 1, PHI-coordinate [Double]
5th number: Z-coordinate or, if KFLAG is 1, Z-coordinate [Double]

The Z coordinate can be dropped at 2-dimensional structures. Enter angles PHI in radian.

Example 1: *The node No. 156 has 2 degrees of freedom and the coordinates X = 45.3 and Y = 89.7 .*
> Thus: 156 2 45.3 89.7

Example 2: *The node No. 68 is supposed to have 6 degrees of freedom (a Beam No. 2 is attached) and cylindrical coordinates R = 100. , PHI = 0.7854 (corresponds to 45 °), Z = 56.87.*
> Thus: 68 6 100. 0.7854 56.87

3rd input group:
Starting after last node, containing coincidence, i.e. the allocation of the element type and the corresponding nodes of every element. Enter two lines for every finite element. The element numbers, like the node numbers, must be entered strictly ascending.

1st line:
1st number: Element number [Long].
2nd number: Element type (1 to 25) [Long].

2nd line: Depending on element type
1st number: 1st node number for coincidence [Long]
2nd number: 2nd node number for coincidence [Long]
...
20th number: 20th node number for coincidence [Long]

Write all numbers into a line, separate at least by one blank respectively. All numbers here are of the type [Long].

Example: *An Isoparametric Serendipity Plane Stress Element No. 7 has element number 23. The coincidence has the global nodes 14, 8, 17, 20, 38, 51, 55, 34 (locally these are the nodes 1-2-3-4-5-6-7-8) > Thus resulting in two lines:*
23 7
14 8 17 20 38 51 55 34

BOUNDARY CONDITIONS FILE Z88I2.TXT

In the file Z88I2.TXT the boundary conditions, displacements and forces affecting the model are entered. Surface loads are put into the file Z88I5.TXT.

Mind the following formats:
[Long] = 4 bytes or 8 bytes integer number
[Double] = 8 bytes floating point number, alternatively with or without point

1st input group:
Number of boundary conditions/loads.

1st number: Number of boundary conditions/loads [Long]

2nd input group:
The boundary conditions and loads are defined. For every boundary condition and for every load one line, respectively.

1st number: node number with boundary condition: load or constraint [Long]
2nd number: Respective degree of freedom (1,2,3,4,5,6) [Long]
3rd number: Condition flag: 1 = force [Long] or 2 = displacement [Long]
4th number: Value of the load or displacement [Double]

Example: *The node 1 shall be fixed at his 3 degrees of freedom, respectively. Node 3 features a load of -1,648 N in Y direction (i.e. DoF 2), the degrees of freedom 2 and 3 are supposed to be fixed for the node 5. This will result in 6 boundary conditions.*
> Thus:

```
6
1  1  2  0
1  2  2  0
1  3  2  0
3  2  1  -1648
5  2  2  0
5  3  2  0
```

It is a good idea to define surface and pressure loads in the file Z88I5.TXT.

Only forces and constraints should be entered here into Z88I2.TXT.

Of course, it is possible, too, to convert surface loads into concentrated forces "manually" and to enter these forces into Z88I2.TXT (which is the classical way but somewhat cumbersome).

For the elements with linear shape function, e.g. Hexahedrons No. 1 and Torus No. 6, edge loads and surface loads are distributed to the elements simply and straight-forward onto the respective nodes.

However, for elements with higher shape functions, i.e. square (Plane Stress No. 3, No. 7, Torus No. 8, Hexahedron No. 10 etc.) or cubic (Plane Stress No. 11 and Torus No. 12) edge and surface loads have to be put onto the elements according to certain rules which are not always physically obvious but mathematically absolutely correct. Amazingly, some load components can have negative values. Though these facts are not obvious, nevertheless they

lead to correct results, which is not the case for intuitive distribution of loads to the respective nodes.

An example may clarify the facts, see figure 16:

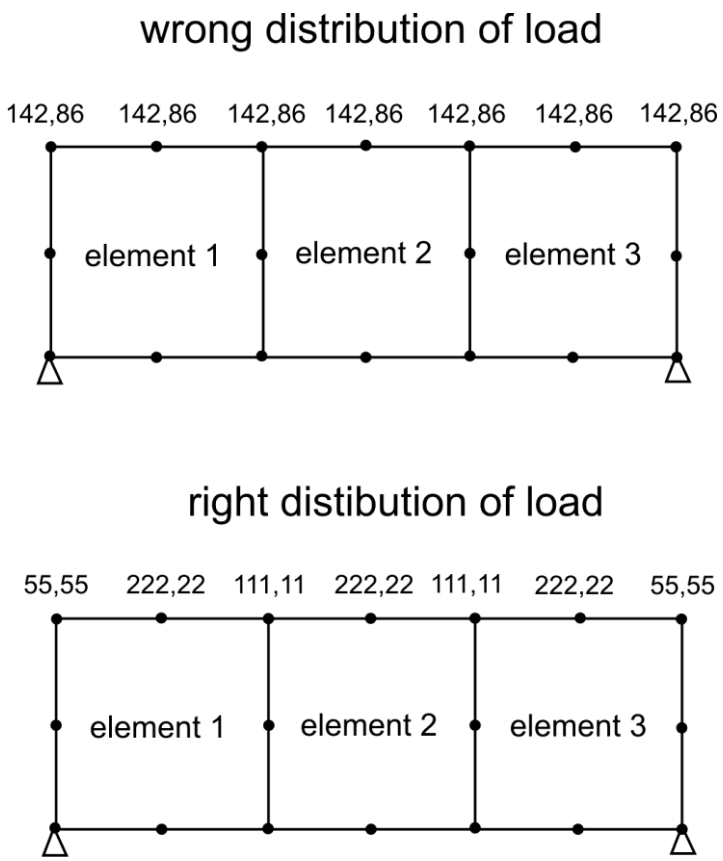


Figure 16: Load distribution on the nodes

A FE structure consists of three plane stress elements No. 7 with the load of 1,000 N distributed on the upper edge in Y direction, see figure 16:

Incorrect: 1,000 N / 7=142.86 N per node. Not correct for elements with square shape function.

Correct: 2 × 1/6 + 2 × (1/6+1/6) + 3 × 2/3 = 18/6 = 3, corresponds to 1,000 N
"1/6 points" = 1,000/18×1 = 55.55
"2/6 points" = 1,000/18×2 = 111.11
"2/3 points" = 1,000/18×4 = 222.22
Control: 2×55.55 + 2×111.11 +3×222.22 = 1,000 N, o.k.

Here's why:

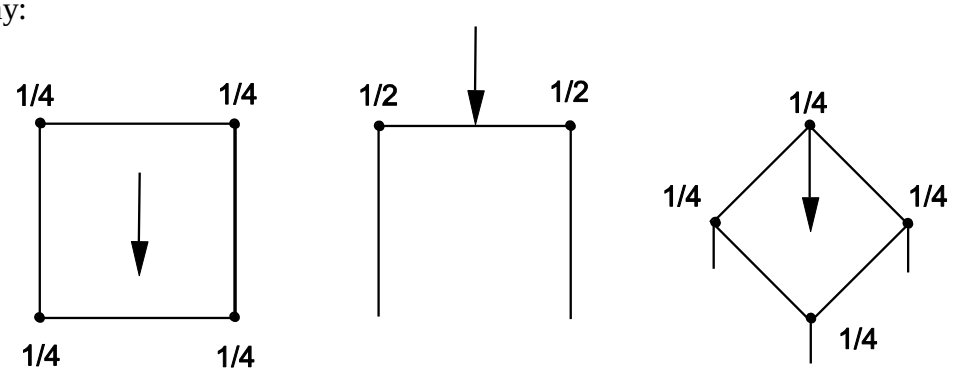


Figure 17: Elements with linear shape functions, e.g. Hexahedron No. 1

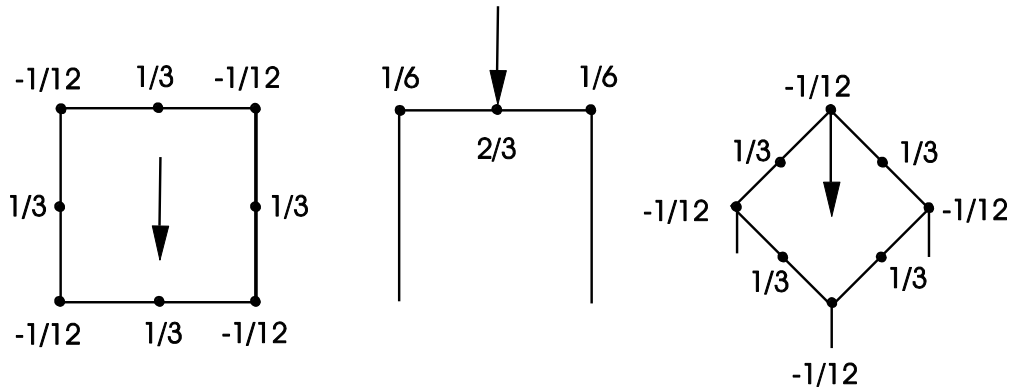


Figure 18: Elements with quadratic shape functions,
e.g. plane stress element No. 3 and 7, Torus No.8, Hexahedron No. 10

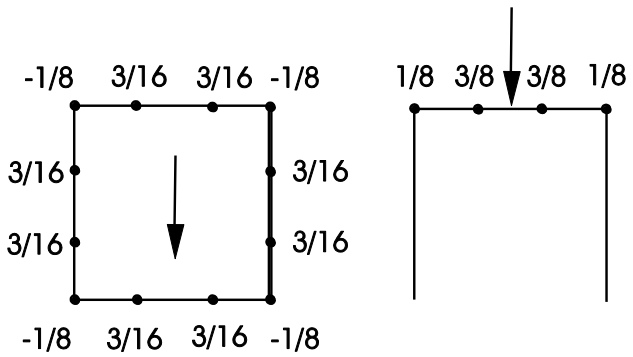


Figure 19: Elements with cubic shape functions,
e.g. plane stress element No. 11, Torus No. 12

SURFACE AND PRESSURE LOADS FILE Z88I5.TXT

This file must always exist. If you do not have any surface and pressure loads, enter a 0 (zero) into the first line, add a *RETURN* and skip the second input group.

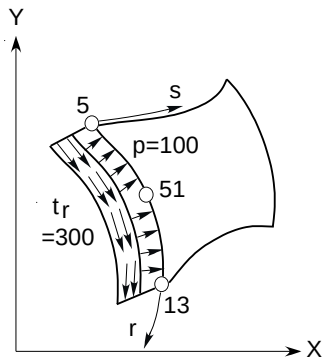
Mind the following formats:
[Long] = 4 bytes or 8 bytes integer number
[Double] = 8 bytes floating point number, alternatively with or without point

1st input group:
1st number: Number of surface and pressure loads [Long]

2nd input group:
Surface and pressure loads – one line per load. Of course, an element may have more than one load applied. The following entries depend from the element type with surface and pressure load to avoid unnecessary data entries.
As for the local directions: Define the local r and s directions by the nodes and their sequence. These local directions for the surface loads may differ from the local r and s coordinate system of the finite element. The numbering has to conform to the element numbering, see chap. 5.

Plain stress element No. 7 and 14 and Torus elements No. 8 and 15:
Element number with surface load [Long]
Pressure, positive if pointing towards the edge [Double]
Tangential shear, positive in local r-direction [Double]
3 nodes of the loaded edge [3 × Double]

Example: The plain stress element 97 features surface load. The load should be applied onto the edge defined by the corner nodes 5 and 13 and by the mid node 51. One surface load is applied normally to the edge with 100 N/mm and the other surface load is applied tangentially and positive in local r direction with 300 N/mm (defined by the two corner nodes).
Thus:
> 97 100. 300. 5 13 51



Plane stress element with surface loads

Hexahedron No. 1:
Element number with surface and pressure load [Long]
Pressure, positive if pointing towards the surface [Double]
Tangential shear, positive in local r direction [Double]
Tangential shear, positive in local s direction [Double]
4 nodes of the loaded surface [4 × Long]

Example: The hexahedron 356 features surface loads. The load should be applied onto the surface defined by the corner nodes 51, 34, 99 and 12 .The first surface load is pressure with 100 N/mm. The second surface load is applied tangentially and positive in local r direction with 200 N/mm. The third surface load is applied tangentially and positive in local s direction with 300 N/mm. Thus
> 356 100. 200. 300. 51 34 99 12

Hexahedron No. 10:
Element number with surface and pressure load [Long]
Pressure, positive if pointing towards the surface [Double]
Tangential shear, positive in local r direction [Double]
Tangential shear, positive in local s direction [Double]
8 nodes of the loaded surface [8 × Double]

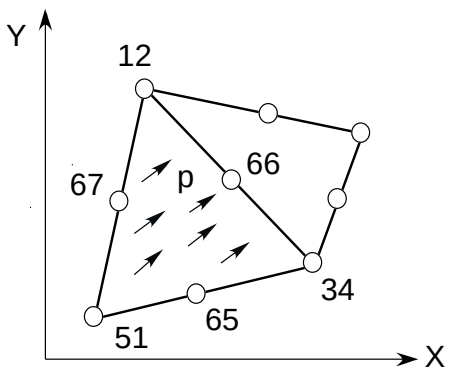
Example: The hexahedron 456 features surface loads. The load should be applied onto the surface defined by the corner nodes 51, 34, 99 and 12 and the mid nodes 102, 151, 166 and 191. The first surface load is pressure with 100 N/mm. The second surface load is applied tangentially and positive in local r direction with 200 N/mm. The third surface load is applied tangentially and positive in local s direction with 300 N/mm. Thus
> 456 100. 200. 300. 51 34 99 12 102 151 166 191

Tetrahedron No. 17:
Element number with pressure load [Long]
Pressure, positive if pointing towards the surface [Double]
3 nodes of the loaded surface [3 × Double]

Example: The tetrahedron 356 features surface loads. The load should be applied onto the surface defined by the corner nodes 51, 34 and 12.The surface load is pressure with 100 N/mm pointing towards the surface, i.e. positive. Thus:
> 356 100. 51 34 12

Tetrahedron No. 16:
Element number with pressure load [Long]
Pressure, positive if pointing towards the surface [Double]
6 nodes of the loaded surface [6 × Double]

Example: The tetrahedron 888 features surface loads. The load should be applied onto the surface defined by the corner nodes 51, 34 and 12 and the mid nodes 65, 66 and 67.The surface load is pressure with 100 N/mm pointing towards the surface, i.e. positive. Thus:
> 888 100. 51 34 12 65 66 67

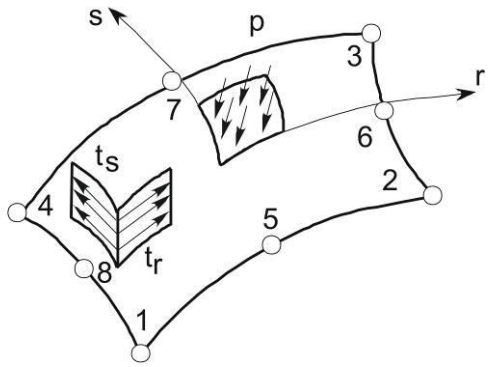


Tetraeder with pressure load on one element side

Plate elements No. 18, 19 and 20:
Element number with pressure load [Long]
Pressure, positive if pointing towards the surface [Double]

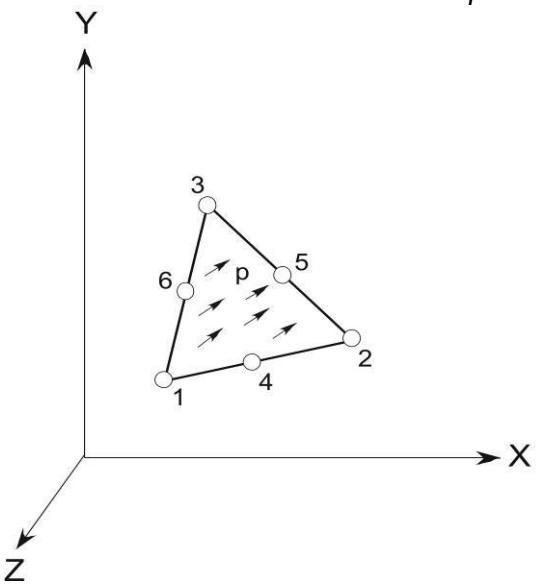
Shell No. 21:
> Element number
> Pressure, positive if pointing towards the surface
> Tangential shear in local r direction
> Tangential shear in local s direction
> 4 corner nodes and 4 mid nodes of the loaded surface. Mathematically positive in top view.

The local r direction is defined by the nodes 1 - 2; the local s direction is defined by the nodes 1-4. The local nodes 1 to 8 for the surface load may differ from the local nodes 1 to 8 used for the coincidence.



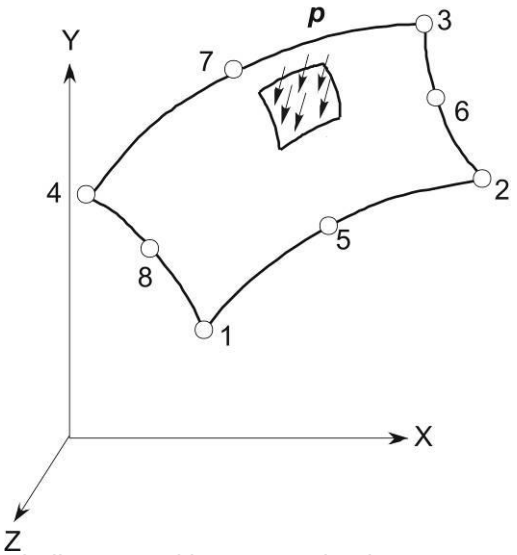
Shell No. 21 with pressure load

Shell No. 22:
> Element number
> Pressure, positive if pointing towards the surface
> 3 corner nodes and 3 mid nodes of the loaded surface. Mathematically positive in top view.



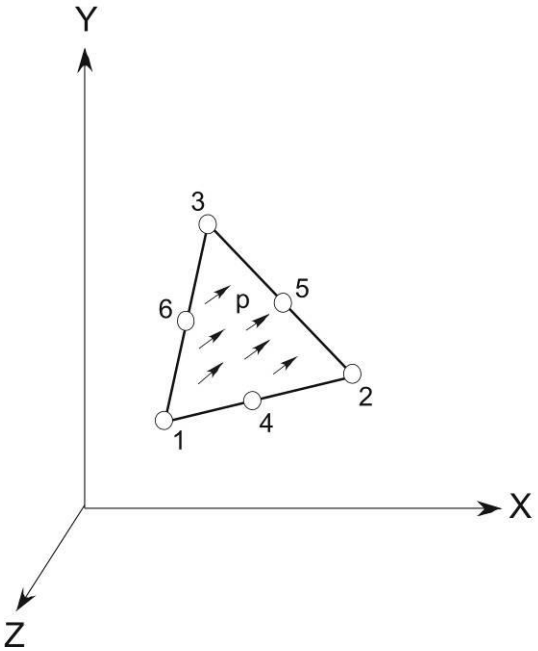
Shell No. 22 with pressure load

Shell No. 23:
> Element number
> Pressure, positive if pointing towards the surface
> 4 corner nodes and 4 mid nodes of the loaded surface. Mathematically positive in top view.



Shell No. 23 with pressure load

Shell No. 24:
> Element number
> Pressure, positive if pointing towards the surface
> 3 corner nodes and 3 mid nodes of the loaded surface. Mathematically positive in top view.



Shell No. 24 with pressure load

Checking of Z88V15 input data files by Z88VRY

The file checker Z88VRY investigates input data sets for Z88 V15 and for Z88Aurora®. Z88VRY was written in Perl to have one universal program for all operating systems, see *docu*. Perl is always installed under LINUX and Snow Leopard. For Windows you may load Perl from www.perl.org. You may install either Strawberry-Perl or ActiveState-Perl. This is a one-click-installation without any problems and does no harm to your system. Then launch the file checker in a Windows-„command prompt“ or an UNIX terminal as follows:

```
perl z88vry.pl -english -aurora
```

This will check the Z88 files Z88I1.TXT, Z88I2.TXT, and Z88I5.TXT.

Although Z88VRY recognizes many conceivable faulty possibilities and is internally quite tricky, situations like with compilers may occur where faults are not detected or seem to be recognized on other passages. Z88VRY stops when detecting the first error because otherwise resulting sequence errors are usually generated from this. Therefore, a recognized error must be fixed right now. An error-free input file recognized from Z88VRY can nevertheless lead to subtle faults at the later program run. However, the probability is low to some extent. This statement refers to formal errors: Z88VRY neither recognizes inconsistent structures, nor wrong or too few boundary conditions!

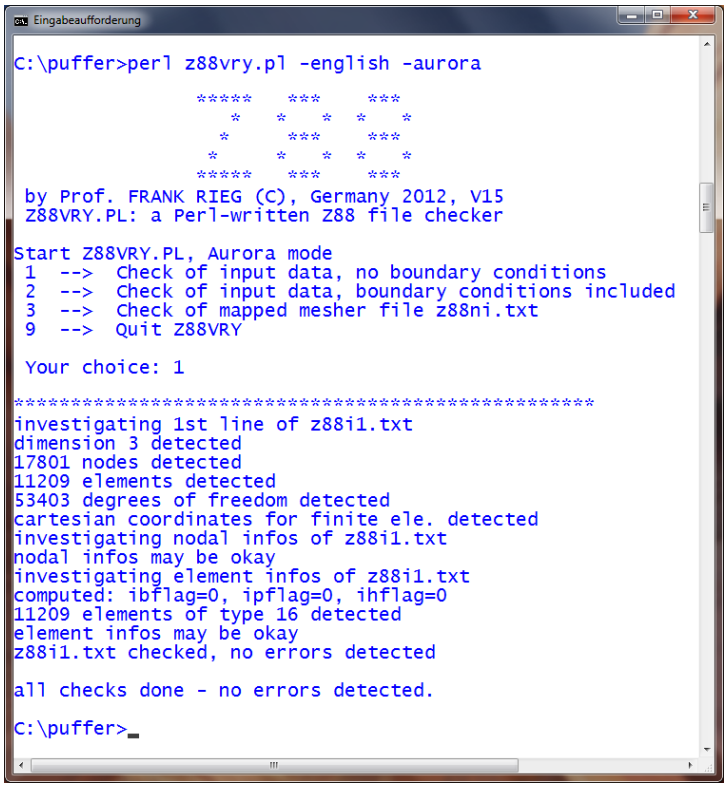


Figure 20: Z88VRY console application

4.1.3 THE STEP CONVERTER Z88GEOCON

What is the basic idea and which are the features?

The present STEP converter is based on the parsing and output routines of the Open Source 3D-Suite OpenCASCADE. Therefore, the relevant sources (stepread.cpp and geocon.cpp) as well as a copy of the GPL license accompany Z88Aurora®.

Most 3D CAD systems feature the possibility to store models in files according to the international standard DIN ISO 10303 (STEP: STandard for the Exchange of Product model data). In most of these cases the application logs AP203 and AP214 are used. These store the 3D geometry, described in highly accurate form, in text files. At the moment, only few CAD producers accommodate the fact that STEP could transfer a lot more (notes, parameters, materials and a lot more), according to the definition. The geometry, however, can mostly be used well also in FE programs, if a few points are taken into account:

- Any STEP converter of a CAD program is only as good as the applied graphic kernel. Thus, if the CAD display contains mistakes, these mistakes are exported along with the data and might impede the processing. These mistakes are partially caused by the kernel itself, partially by the export of cluttered models.
- In the CAD system, apply a modelling tolerance as high as possible (geometrical tolerance e.g. < 0.01), if you have the possibility to repeatedly specify a tolerance during export, select a higher one than the modelling tolerance (e.g. = 0.01).
- Make sure that you use AP203 or AP214 during export.
- Should problems arise during the import, consider simplifying your model before exporting it. Often, small fillets or chamfers are the cause for very small edges and surfaces, which impede the processing. If they are not mandatory for the FE simulation, they can be eliminated and therefore not exported.

Which CAD systems can cooperate with the Step converter?
Any CAD systems which can export, i.e. write STEP files. However, we cannot guarantee any success.

Which elements are supported by the converter?
Z88Aurora® first generates an STL file from the imported STEP files for visualization. This can be transferred into structures from elements No. 16 or No. 17 (linear or quadratic Tetrahedrons) as well as elements Nr. 23 or Nr. 24 (8-node- or 6-node-shell) by means of the existing meshers.

Which functions does the converter offer?
*Z88Geocon > Conversion > from *.step or *.stp to visualised super structure Z88I1.txt*

How to proceed?

1. Construct the 3D geometry to be calculated in your CAD system. In the process, please keep in mind the above mentioned particularities and that only single volumes (assemblies have to be combined) can be imported. Export the geometry as STEP AP203 or AP214 file. Please take care to export a volume model, not 2D or wireframe

model! It is recommended to check the original model and the interchange file with an integrated geometry check for defective and very small surfaces.

2. In Z88Aurora® select *File* > *Import* > *STEP data*. In the subsequent selection box you can only select STEP files. Therefore select the desired file (Figure 21).
3. From your file, Z88GEOCON generates an STL, which is required for visualisation in Z88Aurora®. This data type can now be meshed and processed.

The same functionalities are available when you access the STEP import via the toolbar.

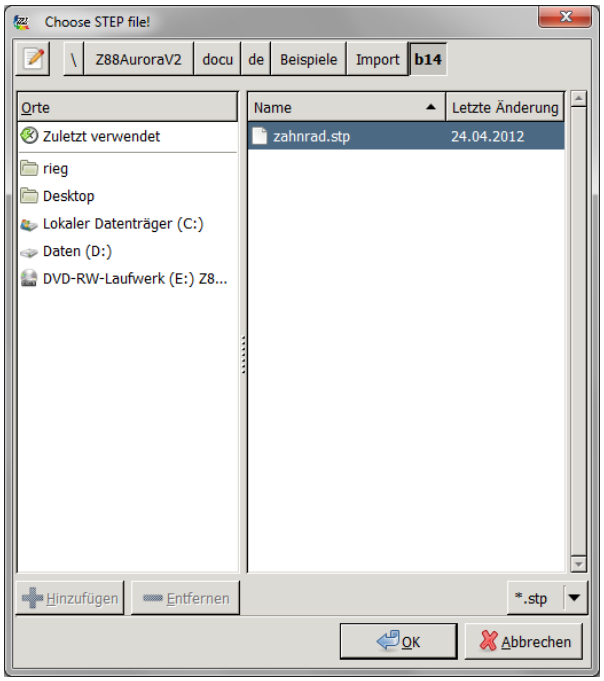


Figure 21: Importing STEP files

4.1.4 THE STL CONVERTER Z88GEOCON

What is the basic idea and which are the features?

Like STEP, STL (stereo lithography) is an established and standardised interchange format which can be generated from many CAD and CAM systems and is often used for Rapid Prototyping and mold flow simulations. In addition, there are also systems in the area of Reverse Engineering, which can generate STL data from a 3D capture. This means that components can also be simulated without using a CAD model.

In contrast to STEP – which can describe the surface of the component very accurately by means of Bézier curves or splines – STL is always a discretisation of the component, i.e. all surfaces are divided into straight edged triangles. Therefore, a loss of accuracy occurs especially at fillets, roundings or holes. However, this occurs in the FEA anyway, however, after the meshing at the latest. But you should take into account that a poorly generated STL leads to an even more extensive loss of quality in the meshing process. Therefore, you should check the following settings when generating STL data, if your source system offers this possibility:

1. Angle control: If you can define the minimum angle in a surface triangle in your CAD system, it is recommended to permit angles with at least 30°. Very acute angles, depending on the mesher, in most cases lead to acute angles in the elements of the FE

mesh, which inevitably either generates elements which cannot be calculated (too small or negative Jacobian determinant) or create bad results.

2. Chord length: A very small chord length also leads to triangles as equilateral as possible, and to especially small triangles for the surface plot. You should select the shortest straight edge of your model and conduct the meshing with, for example, half this value.
3. Edge ratio: The quotient of the longest and the shortest edge of a triangle is also a measure for its regularity. Here, you should select a value close to 1.

Which CAD systems can cooperate with Z88GEOCON?
Any CAD systems which can export, i.e. write STL files in ASCII format. However, we cannot guarantee any success.

Which elements are supported by the converter?
Z88Aurora® first generates a visualisation from the imported STL files. This can be transferred into structures from elements No. 16 or No. 17 (linear or quadratic Tetrahedrons) as well as elements Nr. 23 or Nr. 24 (8-node- or 6-node-shell) by means of the existing meshers.

How to proceed?

1. Construct the 3D geometry to be calculated in your CAD system. In the process, please keep in mind the above mentioned particularities and that only single volumes (assemblies have to be combined) can be imported. Export the geometry as STL file. It is recommended to check the original model and the interchange file with an integrated geometry check for defective and very small surfaces. Have a closer look at the STL and search for triangles with very acute angles. If they are located in a part of the component which is important for the calculation, it is recommended to export the interchange file one more time with modified settings.
2. *In Z88Aurora® select File > Import > STL data.* In the subsequent selection box you can only select STL files. Therefore select the desired file (figure).
3. The geometry is visualised in Z88Aurora® and can be processed.

The same functionalities are available when you access the STEP import via the toolbar.

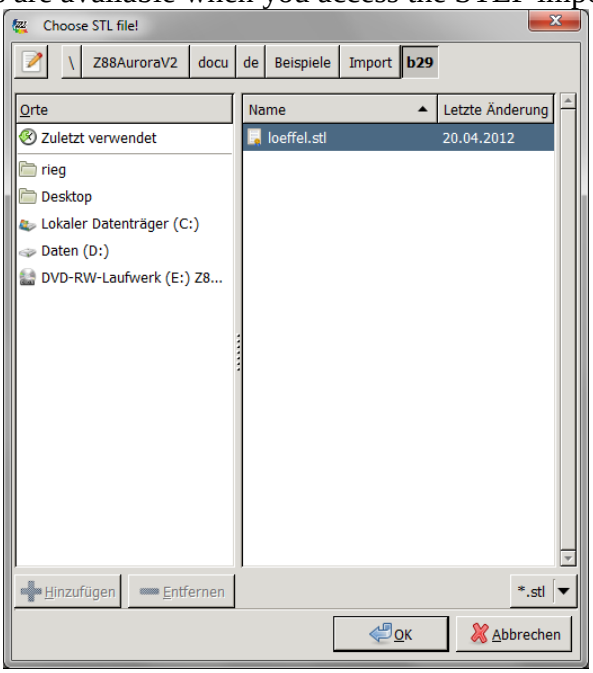
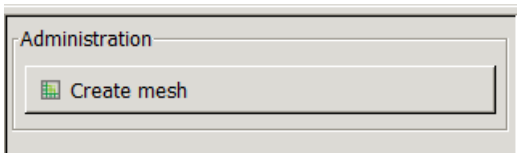


Figure 22: Importing STL files

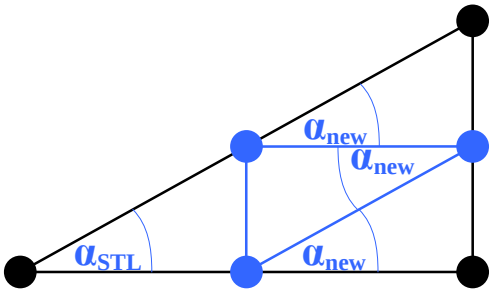
4.1.5 THE STL-TOOLS

STL refiner

STL files can be refined before the real mesher will be started. The advantage is the perpetuation of a STL-structure. This way all possibilities of the previously mentioned mesh generators are available. As submenu the button "create mesh" is implemented.



Parameter entries are not provided. A consisting STL-triangle is automatically divided into four new triangles as can be seen in Figure 23. The smallest existing **angle α** in the new, refined mesh remains **constant**.



STL-element

$ne_{new} = 4 * ne_{STL}$

Figure 23: STL refinement

Convert STL to shell element

With the above displayed symbol the STL-mesh is directly converted into a mesh consisting of shells. As explained in Figure shell elements no. 23 or no. 24 are creatable. When shell element no. 23 is chosen an automatized refinement is implemented. The original STL-element is divided into tree shells no. 23. Based on the centre of the inscribed circle new corner nodes are created.

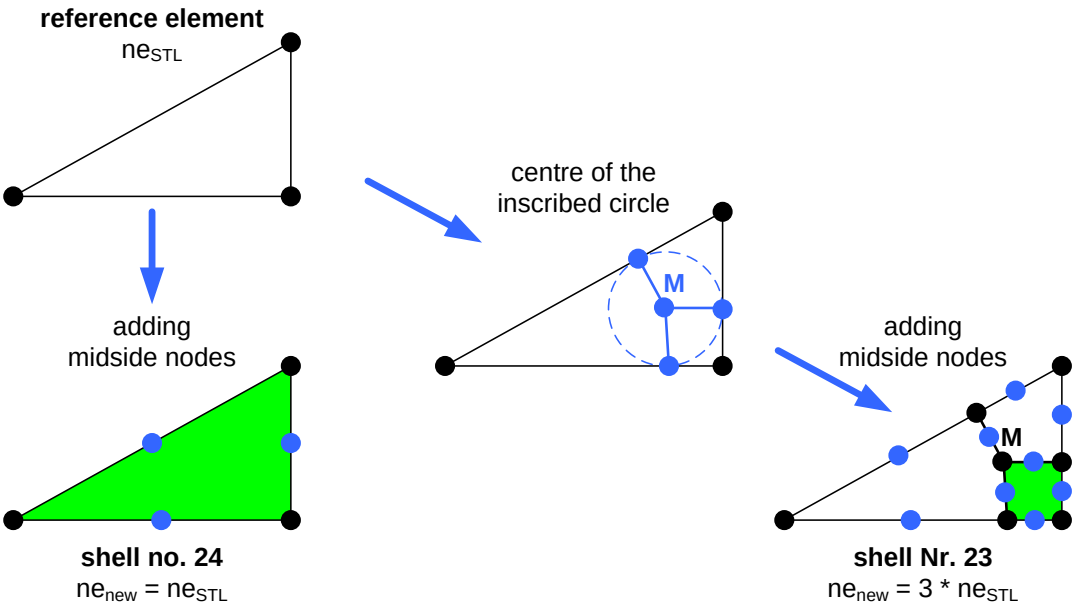
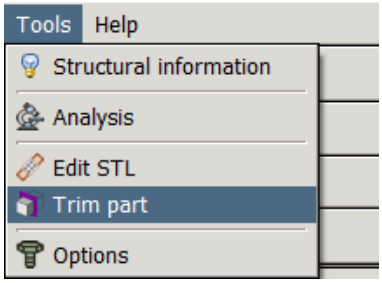


Figure 24: Converting STL-mesh into shell mesh

Trim part



Z88Aurora® offers the user a lot of different elements. A selection of 2D-elements for the conversion of STL-structures is listed in the submenu. The user enters a value for the z-coordinate in the field *cutting plane*. With the *accuracy* ε an area surrounding the z-coordinate is defined. In the blue marked area, see Figure , all gravity points G_x are selected which are within the blue marked area. With the button *preview* the selected STL-triangles can be highlighted. The button create starts the conversion into the previously picked element type. But beware that only works with STL files.

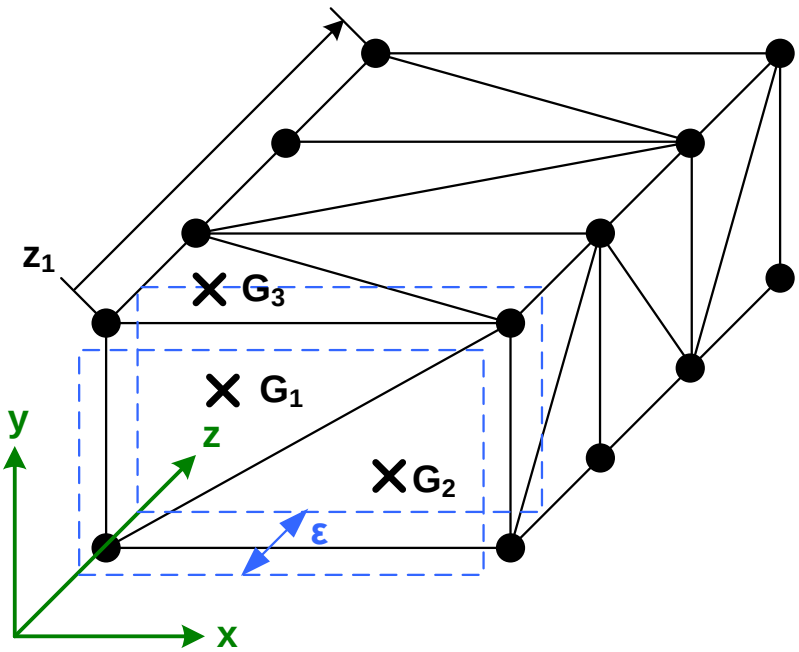
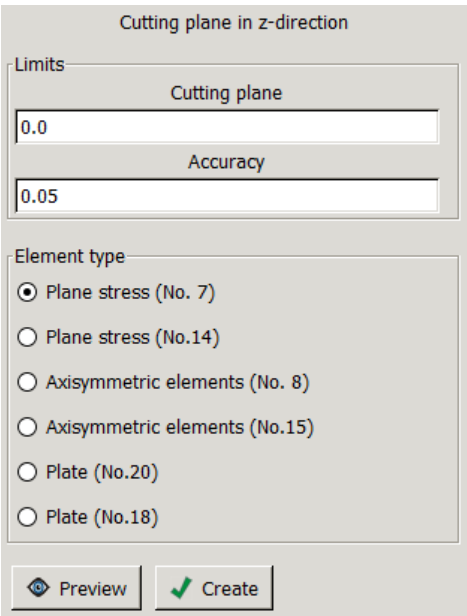


Figure 25: Creating a new STL-structure

The menu looks as following:



4.1.6 THE DXF CONVERTER Z88X

What is the basic idea and which are the features?
2D CAD systems like AutoCAD offer a simple possibility to transfer complex 2D or 2½D structures into Z88Aurora® without an expensive 3D system. For this purpose, the layer-based structure of the DXF files is perfectly suited.

Which CAD systems can cooperate with Z88X?
Any CAD systems which can import (read) and export (write) DXF files. However, we cannot guarantee any success. Z88Aurora®has been tested together with different AutoDesk AutoCAD and AutoCAD LT versions, and AutoDesk's DXF guidelines have been regarded as the inventor of the DXF interface, i.e. according from AC1009 to AC1024. Choose AutoCAD R12 DXF format, if in doubt, but AutoCAD 2011 DXF works, too.

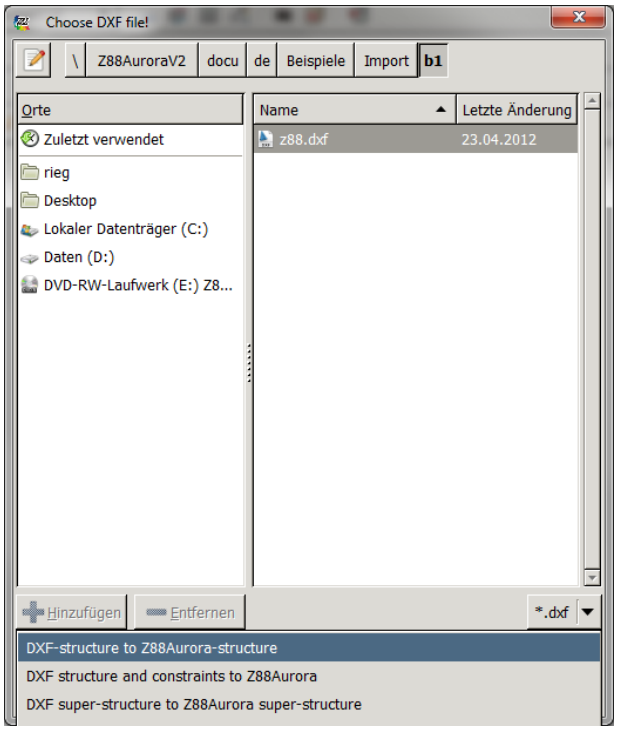
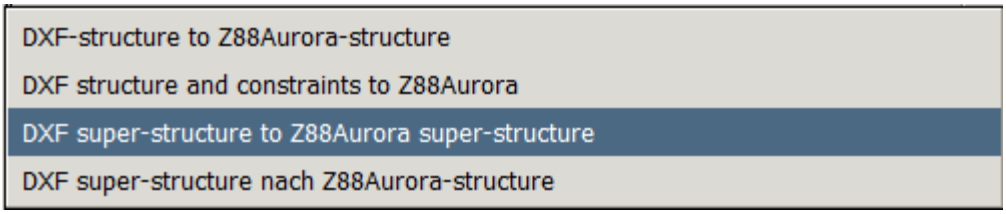


Figure 26: Accessing the DXF converter Z88X and import options in Z88Aurora®

Which elements are supported by the converter? All element types from 1 to 25.

Which functions does the converter offer?



How to proceed?

<u>In the CAD system:</u>

1) Design your component. Order and layers as you like.
2) Define the FEA structure or the super structure by lines and points. Any order and layers,

therefore unproblematic and fast.
3) Number the nodes with the TEXT function on the layer Z88KNR. Any order, therefore unproblematic and fast.
4) Write the element information with the TEXT function on the layer Z88EIO. Any order, therefore unproblematic and fast.
5) Outline each element with the LINE function on the layer Z88NET. The only section with firm work rules and orders (because of the topological information).
6) Write general information, material information and control information for the stress processor Z88D on the Layer Z88GEN.
7) Define the boundary conditions on the layer Z88RBD.
8) Define the surface and pressure loads (if needed) on the layer Z88FLA.
9) Export or store your 3-D model or 2-D drawing under the name **Z88X.DXF**.

In Z88Aurora®: Launch the CAD converter Z88X

Select in the menu *File > Import > DXF files.*
In the selection menu you can choose which of the following files are supposed to be created:

- *a file of general structure data Z88I1.TXT or*
- *a complete Z88 data record with Z88I1.TXT, Z88I2.TXT, Z88I3.TXT, and Z88I5.TXT, if applicable*
- *a Z88I1.TXT from a super structure which you can mesh manually in Aurora*
- *a Z88I1.TXT from a super structure which is meshed with the information deposited in the DXF file (for this purpose, the mapped mesher Z88N is launched directly afterwards)*

The same functionalities are available when you access the Z88 import via the toolbar. Then, define material data, element parameters and integration orders in Z88Aurora®.

Z88X IN DETAIL

Proceed in the following steps and reserve the following layers

Z88GEN: Layer for *general information* (1st input group in the mesh generator input file Z88NI.TXT and general structure data file Z88STRUCTURE.TXT).
Z88KNR: Layer including the node numbers.
Z88EIO: Layer including the *element information* like element type and in the case of mesh generator input file Z88NI.TXT control information for the mesh generator.
Z88NET: Layer containing the *mesh* which was drawn or outlined in defined order.

A further layer, **Z88PKT**, is produced by Z88X if you convert from Z88 to CAD. It shows all nodes with a *point marker* in order to better recognize the nodes. For the reverse step, from CAD to Z88, it is completely insignificant.

1st step: Design your component in the CAD system as usual. You do not need to maintain a definite order and you can use any layers. It is highly recommended to put symbols on one layer, edges on another layer, dimensions on a third layer, invisible lines and centre lines on a fourth layer and so on. This enables you to remove all unnecessary information in the next step.

2nd step: Plan your mesh subdivision that means suitable finite element types and their distribution. Subdivide the FE structure or the super structure into elements by lines, insert **all**

points which are not yet existing (for example intersection points or end-points of lines are usable). Any order and layer. However, it is recommended not to use the Z88-layers like Z88NET, Z88GEN, Z88PKT, Z88KNR, Z88EIO, Z88FLA, and Z88RBD. Better define any new layer for this or use already available layers from step 1.

3rd step: Define the Z88-Layer Z88KNR and make it the active layer. Catch or trap every FE node, which were already defined in the 1st step by your construction or have been completed in the 2nd step, and number them. Write to every node **P blank node-number** e.g. *P 33*, with the TEXT function of the CAD program. Be very careful to snap exactly the node and attach the number exactly to the node's location. Take your time! With the snap modes of AutoCAD (intersection point, end-point, point etc.) this works well. Choose any order of the work sequence as you like, you can well number the node 1 (*P 1*), then the node 99 (*P 99*) and then node 21 (*P21*). However, the numbering of the nodes must make sense and must be meaningful for an FE analysis. *You* define which node in node 99 and which other node reads 21.

4th step: Define the Layer **Z88EIO** and make it the active layer. Write the element information with the TEXT function anywhere (of course, it looks nicer with the element info's placed in middle of the respective finite element or super element). The order of the work sequence is up to you. You can describe element 1 first, step to the attaching element 17 and then proceed with element 8. However, your element choice and description must make sense for an FE analysis. The following information has to be written:

For all finite element types from 1 to 25:

FE Element number Element type
Write into one line, separate each item by at least one blank.

Example: An Isoparametric Serendipity Plane Stress Element No. 7 is supposed to get the element No. 23. Write e.g. into the middle of the element with the TEXT function *FE 23 7*

For super-elements 2-dimensional No. 7, 8, 11, 12 and 20:

SE
Element number
Super-element type
Type of the finite elements to be produced by meshing
Subdivision in local x direction
Type of subdivision in local x direction
Subdivision in local y direction
Type of subdivision in local y direction
Write into one line, separate each item by at least one blank.

Example: Subdivide an isoparametric Serendipity Plane Stress Element with 12 nodes (Element type 11) used as super-element into finite elements of type 7, i.e. isoparametric Serendipity Plane Stress Elements with 8 nodes (Element type 7). Subdivide in local x direction three times equidistantly and subdivide in local y direction 5 times ascending geometrically. The super element is supposed to have the number 31. Write e.g. into the middle of the element with the TEXT function: *SE 31 11 7 3 E 5 L* (e or E for equidistant is equivalent)

For super-elements 3-dimensional Hexahedrons No. 10 and Shells No. 21:

SE
Element number
Super-element type
Type of the finite elements to be produced by meshing
Subdivision in local x direction
Type of subdivision in local x direction
Subdivision in local y direction
Type of subdivision in local y direction
Subdivision in local z direction
Type of subdivision in local z direction
Write into one line, separate each item by at least one blank.

Example: Subdivide an Isoparametric Serendipity Hexahedron with 20 nodes (Element type 10) as super element into finite elements of the type Isoparametric Hexahedrons with 8 nodes (Element type 1). Subdivide equidistantly three times in local x direction, 5 times ascending geometrically in local y direction and subdivide equidistantly 4 times in local z direction. The super element is supposed to have the number 19. Write e.g. into the middle of the element with the TEXT function:
SE 19 10 1 3 E 5 L 4 E (e or E for equidistant is equivalent)

5th step: Define the Layer **Z88NET** and make it the active layer. You need concentration for this step, because a firm and rigid work sequence must now be kept because of the topological information. One of the most important information, the coincidence, is defined in this step that means which elements are defined or outlined by which nodes. Choose a proper colour which differs well from the colours used till now and remove all superfluous information by switching off unused layers. Select the **LINE command** and select the proper snap options e.g. points, **intersection points** and, if necessary, end-points.

Start at the first element. For Z88 the first element is the element with which you start now, that means the one which you have chosen for your first element (*SE 1 or FE 1*). Select the node you want to be the first node of this element (this can be e.g. globally the node 150) and draw a line to the node which shall be the second node of this element (this can be e.g. globally the node 67). From there, draw a line to the third node of this element (this can be e.g. globally the node 45). Connect all required nodes with lines and draw at last a line to the starting point, the first node, and then quit the LINE function.

Then you do the same with the second element. Remember: **You determine with this order which of the elements will be the real second element now**. In the previous 4th step you have only defined what kind of element the second element is. You determine here **how** the element is defined topologically.

The third element follows and so on. If you should make a mistake at the outlining of an element then delete all previous lines of this element (e.g. with an UNDO function) and start again at the first point of the element in question. But if you notice now just outlining element 17 that you have made a mistake at element 9, then you must delete all lines of the elements 9 to 17 and restart with element 9.

For your comfort, you must keep the following outline orders which partly differ from the orders shown at the element descriptions when entering the coincidence by hand. Z88X then sorts internally correctly.

Example: The coincidence for the element type 7 is as follows in the element description: First the corner nodes, then the mid nodes, read 1-2-3-4-5-6-7-8. The coincidence list must look like this in the Z88 input files. However, for Z88X' use for comfortably outlining the elements the order is 1-5-2-6-3-7-4-8-1 (left picture), respectively A-B-C-D-E-F-G-H-A (right picture):

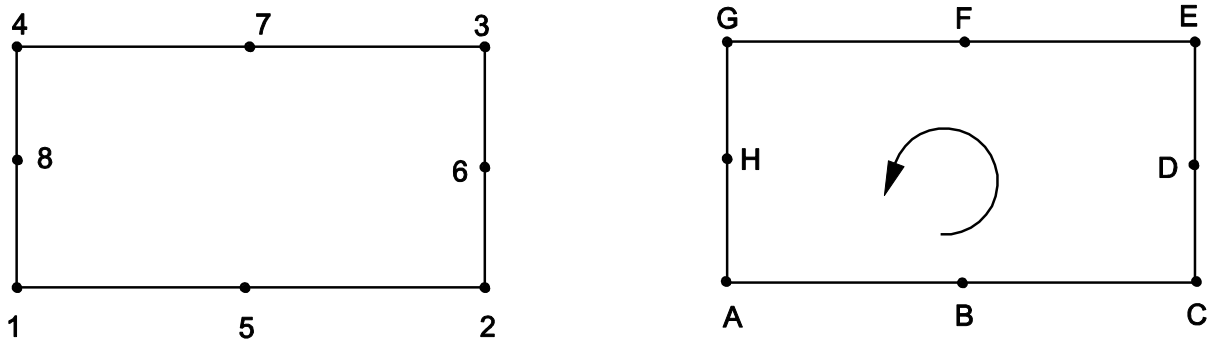
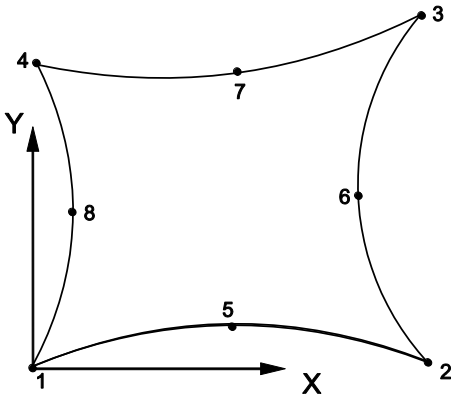


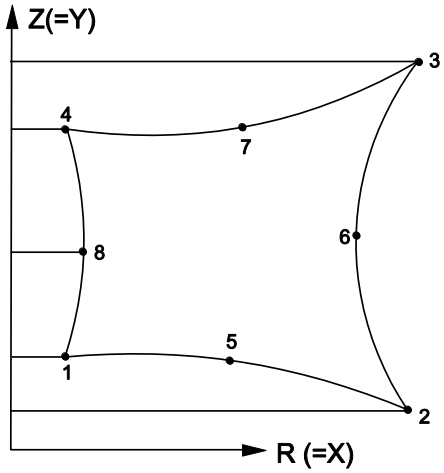
Figure 27: Example for correct outline orders

Following the CAD outline orders for all elements:

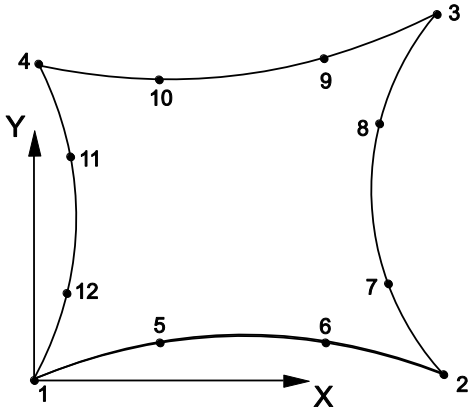
<u>Element No. 7, No. 20 and No. 23:</u> 1 - 5 - 2 - 6 - 3 - 7 - 4 - 8 - 1



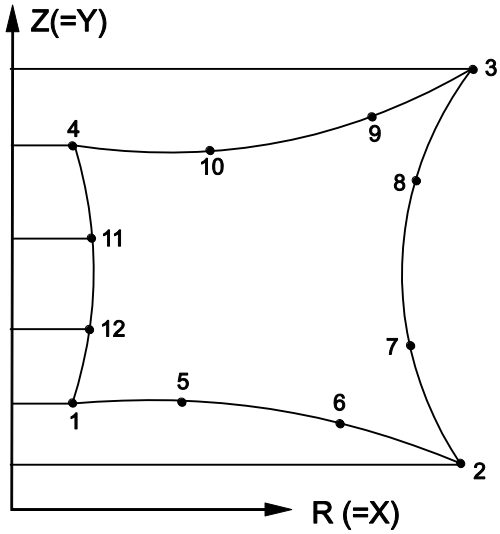
Element No. 8: 1 - 5 - 2 - 6 - 3 - 7 - 4 - 8 - 1



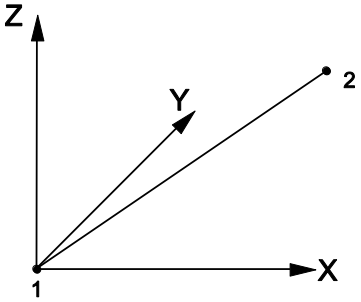
Element No. 11: 1 - 5 - 6 - 2 - 7 - 8 - 3 - 9 - 10 - 4 - 11 - 12 - 1



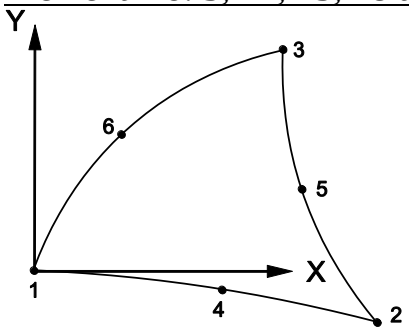
Element No. 12: 1 - 5 - 6 - 2 - 7 - 8 - 3 - 9 - 10 - 4 - 11 - 12 - 1



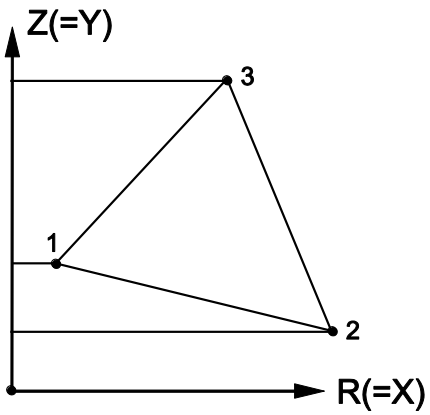
Element No. 2, 4, 5, 9, 13: Line from node 1 to node 2



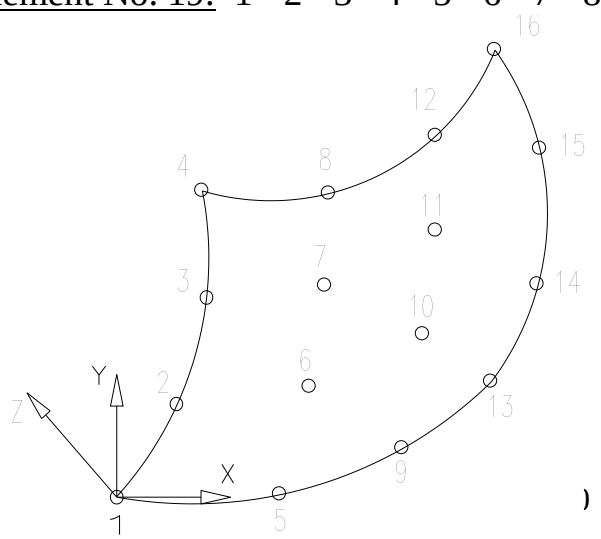
Element No. 3, 14, 15, 18 and 24: 1 - 4 - 2 - 5 - 3 - 6 - 1

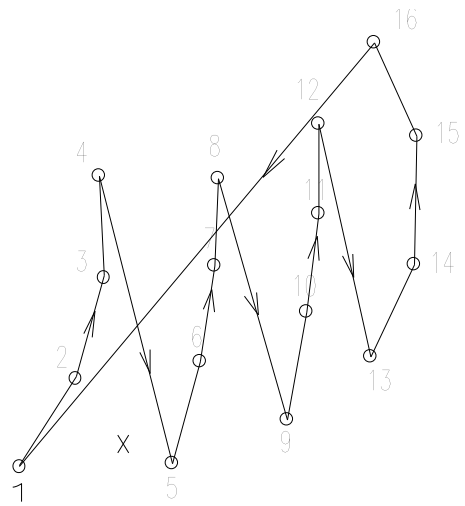


Element No. 6: 1 - 2 - 3 – 1



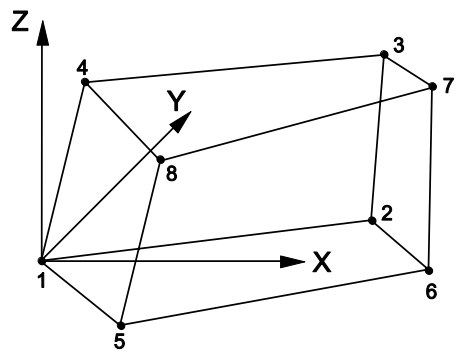
Element No. 19: 1 - 2 - 3 - 4 - 5 - 6 - 7 - 8 - 9 -10 - 11 - 12 - 13 - 14 - 15 - 16 – 1





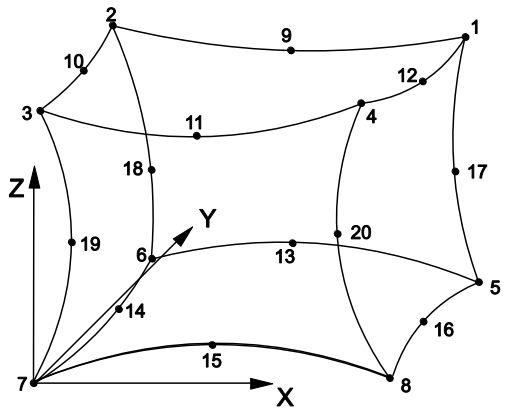
Element No. 1:

Upper plane: 1 - 2 - 3 - 4 - 1, quit LINE function
Lower plane: 5 - 6 - 7 - 8 - 5, quit LINE function
1 - 5, quit LINE function
2 - 6, quit LINE function
3 - 7, quit LINE function
4 - 8, quit LINE function



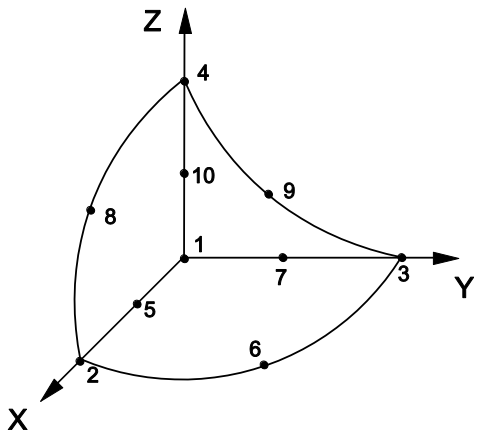
Element No. 10:

Upper plane: 1 - 9 - 2 - 10 - 3 - 11 - 4 - 12 - 1, quit LINE function
Lower plane: 5 - 13 - 6 - 14 - 7 - 15 - 8 - 16 - 5, quit LINE function
1 - 17 - 5, quit LINE function
2 - 18 - 6, quit LINE function
3 - 19 - 7, quit LINE function
4 - 20 - 8 , quit LINE function



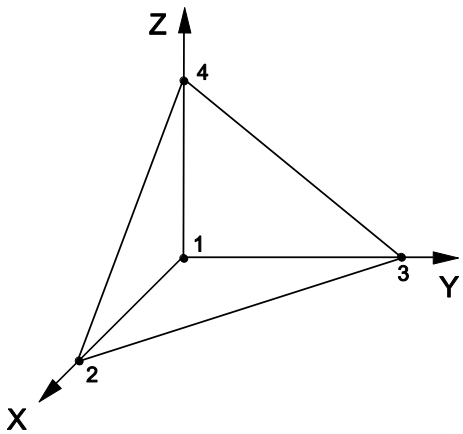
Element No. 16:

XY-Plane: 1 - 5 - 2 - 6 - 3 - 7 - 1, quit LINE function
2 - 8 - 4 quit LINE function
3 - 9 - 4, quit LINE function
1 - 10 - 4, quit LINE function



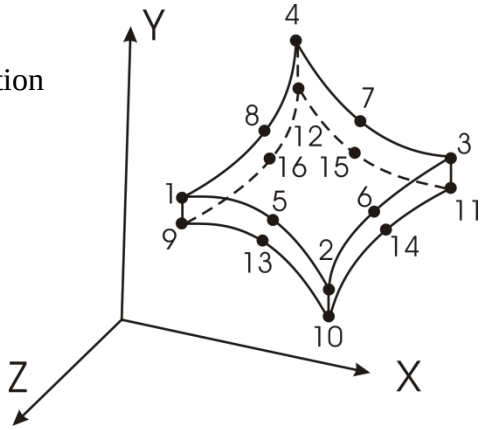
Element No. 17:

XY-Plane: 1 - 2 - 3 - 1, quit LINE function
2 - 4, quit LINE function
3 - 4, quit LINE function
1 - 4, quit LINE function



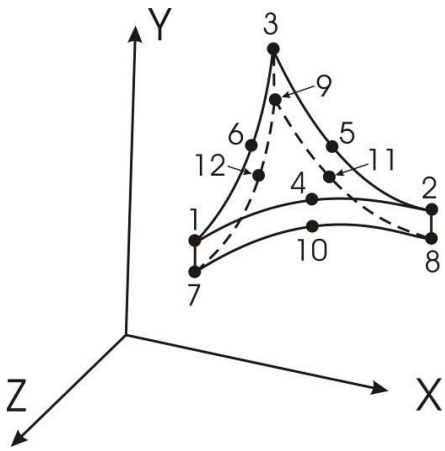
Element No. 21:

Upper plane: 1 - 5 - 2 - 6 - 3 - 7 - 4 - 8 - 1, quit LINE function
Lower plane: 9 - 13 - 10 - 14 - 11 - 15 - 12 - 16 - 9,
quit LINE function
1 - 9, quit LINE function
2 - 10, quit LINE function
3 - 11, quit LINE function
4 - 12, quit LINE function



Element No. 22:

Upper plane: 1 - 4 - 2 - 5 - 3 - 6 - 1,
quit LINE function
Lower plane: 7 - 10 - 8 - 11 - 9 - 12 - 7,
quit LINE function
1 - 7, quit LINE function
2 - 8, quit LINE function
3 - 9, quit LINE function



6th step: Define the layer **Z88GEN** and switch it active. Write with the TEXT function into any place of your drawing: the **general information,** i.e. the first input group of the general structure data Z88I1.TXT or the mesh generator file Z88NI.TXT.

In case of Z88I1.TXT (i.e. FE mesh):

Z88I1.TXT
Dimension of the structure
Number of nodes
Number of finite elements
Number of degrees of freedom DoF
Coordinate flag (0 or 1)

Write into one line, separate each item by at least one blank. **Definitely write in the layer Z88GEN.**

Example: 3-dimensional FE structure with 150 nodes, 89 finite elements, 450 degrees of freedom. Input with cylindrical coordinates. Thus:
Z88I1.TXT 3 150 89 450 1

In case of Z88NI.TXT (i.e. super structure):

Z88NI.TXT
Dimension of the structure
Number of nodes
Number of super element
Number of degrees of freedom DoF
Coordinate flag for super elements (0 or 1)
Trap radius header flag (mostly 0)
Coordinate flag for finite elements (0 or 1)
Write into one line, separate each item by at least one blank.

Example: 2-dimensional super-structure with 37 nodes, 7 super elements, 74 degrees of freedom. Polar coordinates for the super elements, use default for trap radius, and use default (= cartesian coordinates) for coordinate flag for the finite elements. Thus:
Z88NI.TXT 2 37 7 74 1 0 0

Write into one line, separate each item by at least one blank. **Make sure to write in the layer Z88GEN.**

9th step: Store your model or drawing in the DXF file format. Choose AutoCAD R12 DXF format, if in doubt, but AutoCAD 2011 DXF works, too. For precision of decimal positions take the default value which the CAD program suggests

Caution: Use the Z88X keywords **"P number, FE values, SE values, FLA, RBD, Z88NI.TXT, Z88I1.TXT, Z88I2.TXT and Z88I5.TXT"** only where they are really needed. Take care that they do not appear in other drawing captions!
Please note: The import of DXF, ABAQUS, ANSYS, NASTRAN, and COSMOS input decks is positively limited for this version Z88Aurora® to the FE geometry, boundary conditions/forces and surface and pressure loads - in contrary to Z88 V13 and Z88Aurora® V1. And here's why: After the import of commercial input decks you may define the material data, the element parameters and the integration orders very comfortable in Z88Aurora® (even if some of these information are included in the input decks, however), because only in this way a proper use of Aurora's own material data base is possible. This will give you a clean data base for Z88Aurora® projects

EXAMPLE 1 FOR Z88X: FINITE ELEMENTS STRUCTURE

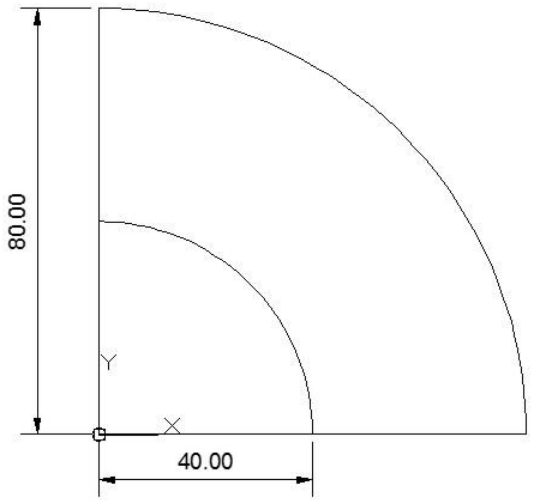
Consider a pipe under internal pressure of 1,000 bar (=100 N/mm^2). Inside diameter of the pipe is 80 mm, outside diameter of the pipe is 160 mm. The length is 40 mm. If one chooses the supports cleverly, a quarter of the pipe is enough to reflect the problem.

Such structures are best suited for polar coordinates. The internal pressure of 1,000 bar corresponds to a force of 251,327 N while the edge load is:

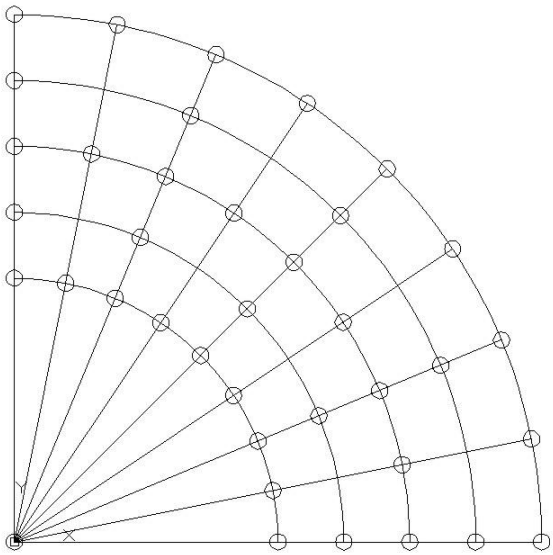
$$q = \frac{F}{\ell} = \frac{F}{r\,\varphi} = \frac{251327}{40\,\pi/2} = 4000\,N/mm$$

1st step: Design your component in the CAD system as usual. You do not need to maintain a definite order and you can use any layers. It is highly recommended to put symbols on one layer, edges on another layer, dimensions on a third layer, invisible lines and centre lines on a fourth layer and so on. This enables you to remove all unnecessary information in the next step. For this example you may enter the main data by command line. Recall AutoCAD's numeric data formats:

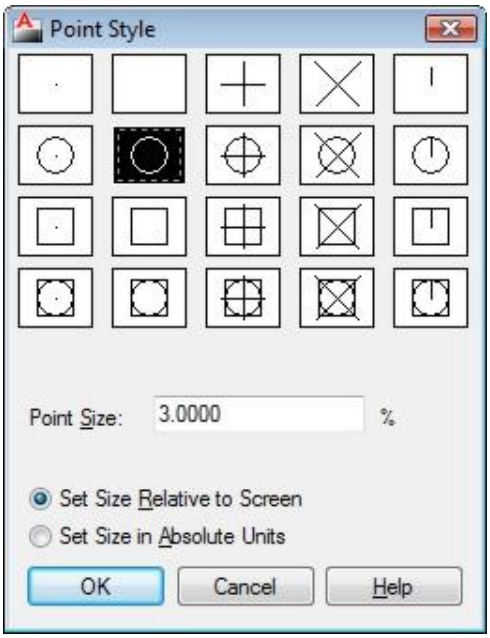
- absolute cartesian coordinates: X,Y
- relative cartesian coordinates: @DeltaX,DeltaY
- absolute polar coordinates: Radius<Angle
- relative polar coordinates: @Radius<Angle



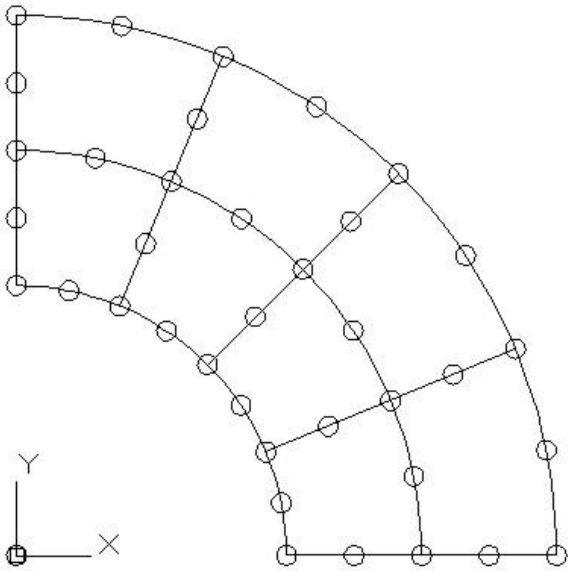
2nd step: Plan your mesh subdivision that means suitable finite element types and their distribution. Subdivide the FE structure or the super structure into elements by lines, insert **all** points which are not yet existing (for example intersection points or end-points of lines are usable). Any order and layer. However, it is recommended not to use the Z88-layers like Z88NET, Z88GEN, Z88PKT, Z88KNR, Z88EIO, Z88FLA, and Z88RBD. Better define any new layer for this or use already available layers from step 1.



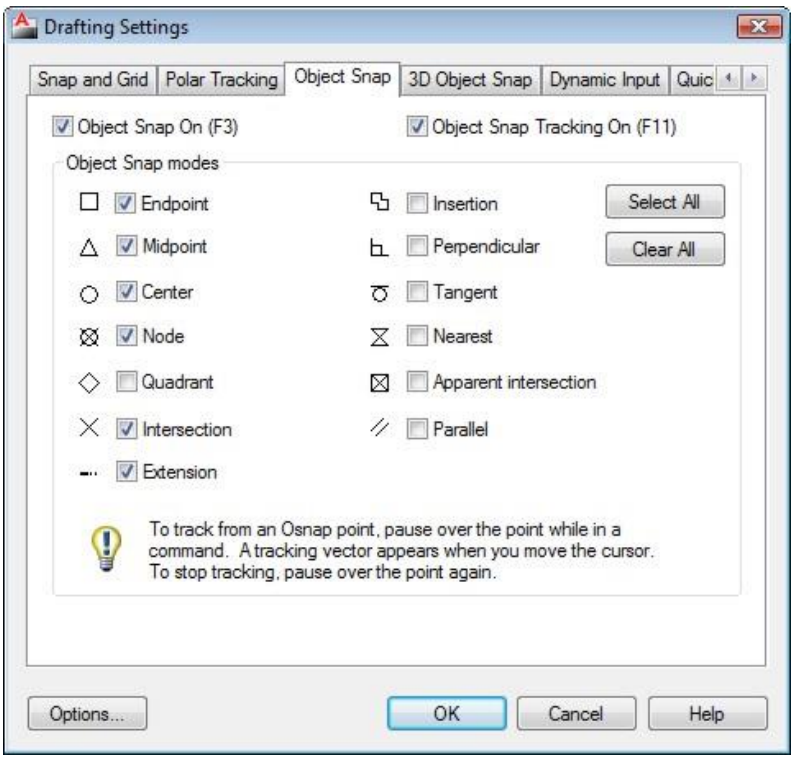
Take care to set nice looking points. Use the AutoCAD command DDPTYPE:

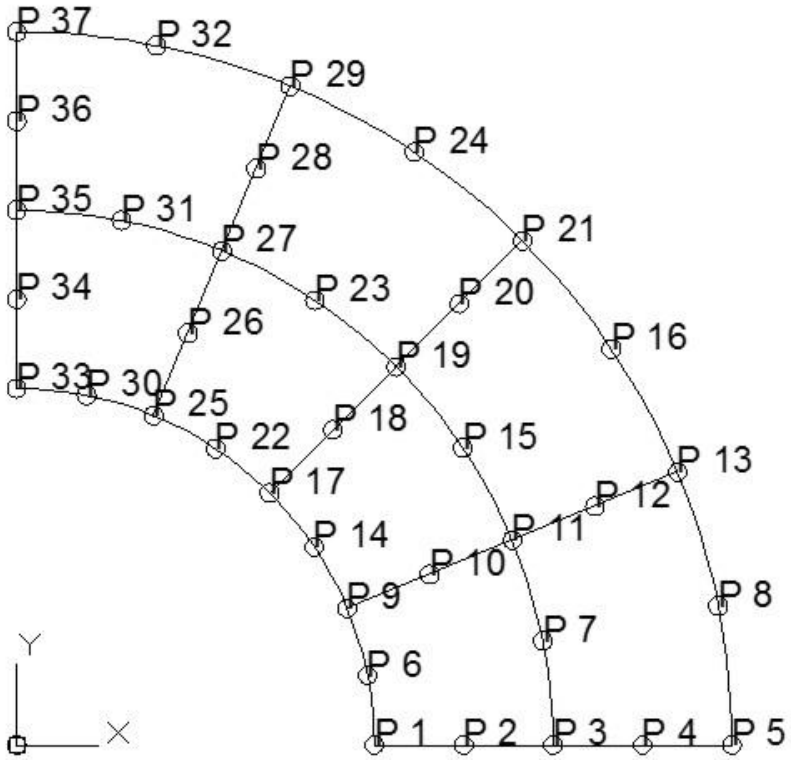


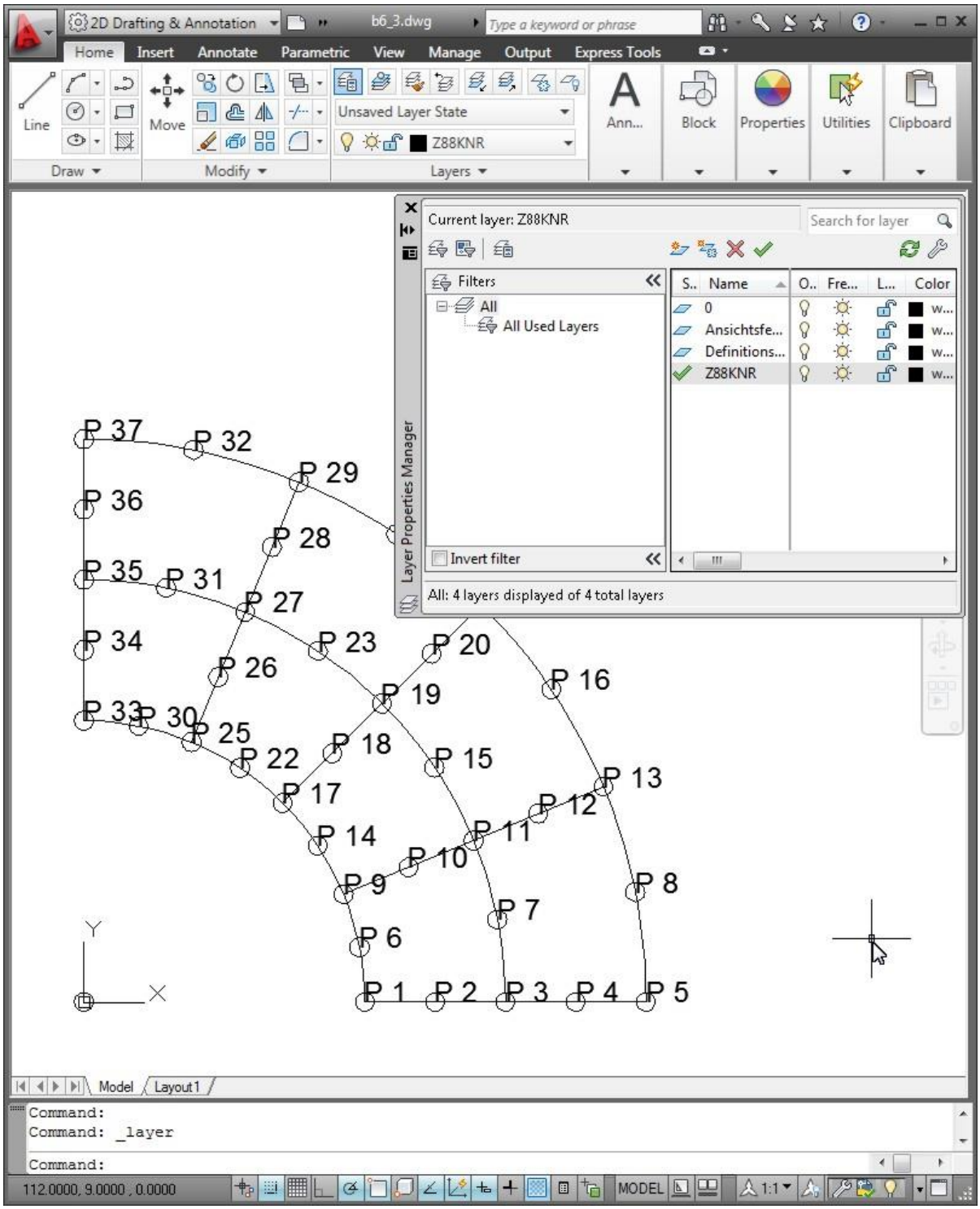
Now you may delete any auxiliary lines, arcs etc. to see the true FE structure:



3rd step: Define the Z88-Layer Z88KNR and make it the active layer. Catch or trap every FE node, which were already defined in the 1st step by your construction or have been completed in the 2nd step, and number them. Write to every node **P blank node-number** e.g. *P 33*, with the TEXT function of the CAD program. Be very careful to snap exactly the node and attach the number exactly to the node's location. Take your time! With the snap modes of AutoCAD (intersection point, end-point, point etc.) this works very well. Choose any order of the work sequence as you like, you can well number the node 1 (*P 1*), then the node 99 (*P 99*) and then node 21 (*P21*). However, the numbering of the nodes must make sense and must be meaningful for an FE analysis. *You* define which node is node 99 and which other node is 21.



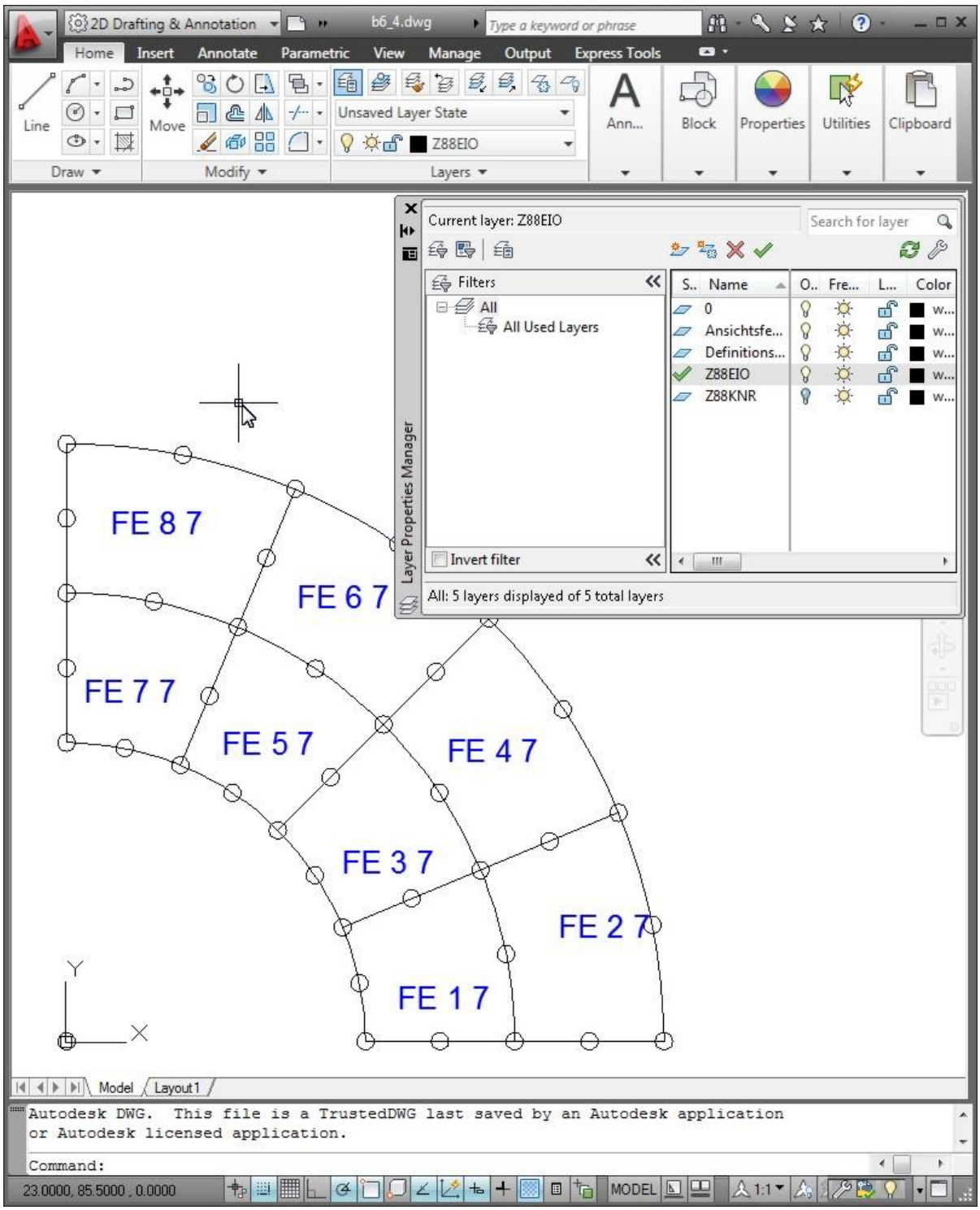

P 37
P 32
P 29
P 36
P 24
P 28
P 35
P 31
P 27
P 21
P 34
P 23
P 20
P 26
P 16
P 19
P 33
P 30
P 25
P 18
P 22
P 15
P 13
P 17
P 12
P 14
P 11
P 10
P 9
P 8
P 7
P 6
Y
X
P 1
P 2
P 3
P 4
P 5



4th step: Define the Layer **Z88EIO** and make it the active layer. Write the element information with the TEXT function anywhere (of course, it looks nicer with the element info's placed in middle of the respective finite element or super element). The order of the work sequence is up to you. You can describe element 1 first, step to the attaching element 17 and then proceed with element 8. However, your element choice and description must make sense for an FE analysis. The following information has to be written:

FE Element number Element type

It might be a good idea to use another colour for the objects of layer Z88EIO – here, blue. However, you don't need to. For better information, the former layer Z88KNR is switched off.

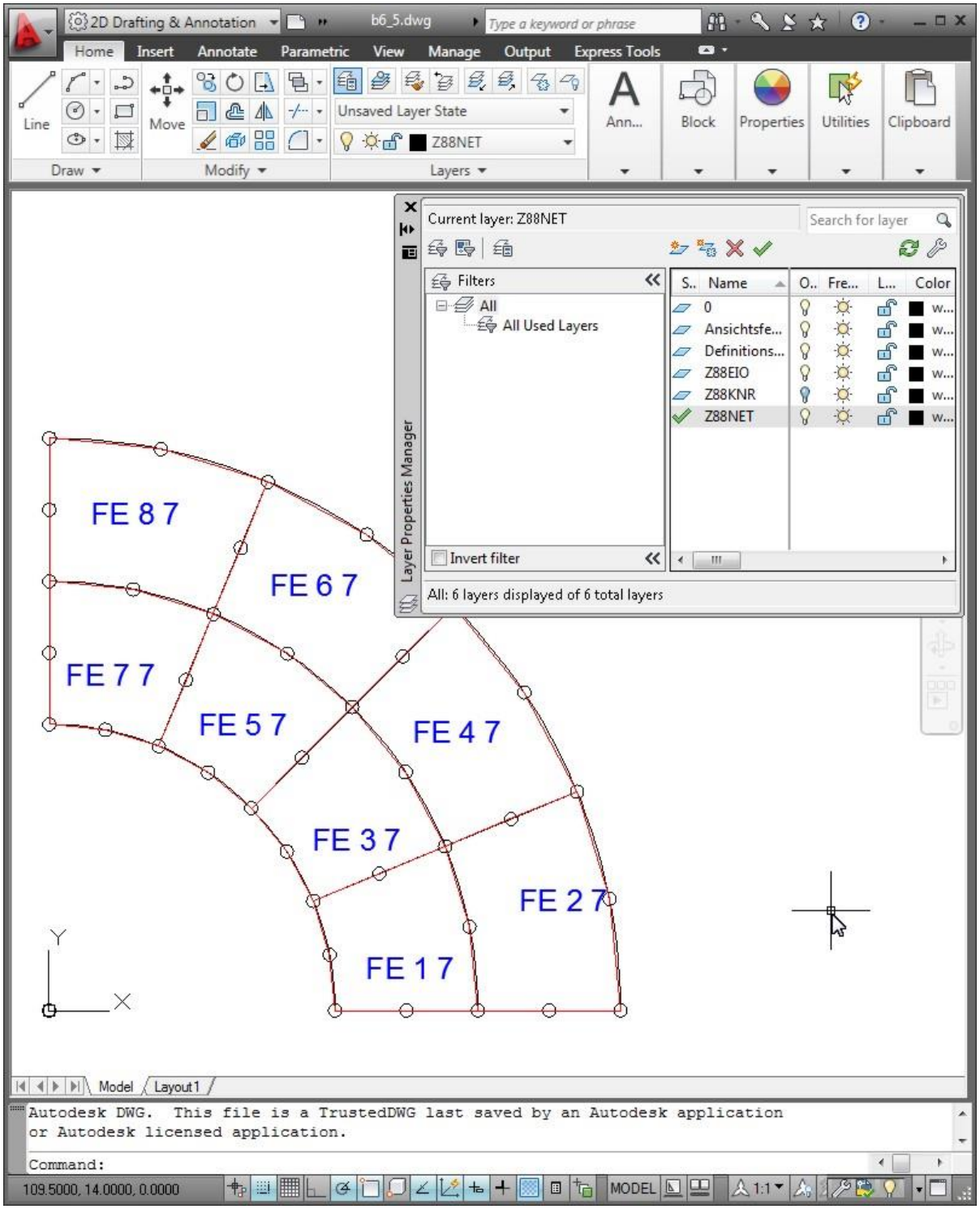


5th step: Define the Layer **Z88NET** and make it the active layer. You need concentration for this step, because a firm and rigid work sequence must now be kept because of the topological information. One of the most important information, the coincidence, is defined in this step that means which elements are defined or outlined by which nodes. Choose a proper colour which differs well from the colours used till now and remove all superfluous information by switching off unused layers. Select the **LINE** command and select the proper snap options e.g. points, intersection points and, if necessary, end-points.

Start at the first element. For Z88 the first element is the element with which you start now, that means the one which you have chosen for your first element (*FE 1*). Select the node you want to be the first node of this element and draw a line to the node which shall be the second node of this element. From there, draw a line to the third node of this element. Connect all required nodes with lines and draw at last a line to the starting point, the first node, and then quit the LINE function. Thus, we might draw this line: P1-P2-P3-P7-P11-P10-P9-P6-P1, quit LINE. However, these lines would do fine, too: P9-P6-P1-P2-P3-P7-P11-P10-P9 or P3-P7-P11-P10-P9-P6-P1-P2-P3 or P11-P10-P9-P6-P1-P2-P3-P7-P11.

Then do the same with the second element. Remember: **You determine with this order which of the elements will be the real second element now**. In the previous 4th step you have only defined what kind of element the second element is. You determine here **how** the element is defined topologically. Thus, we might draw this line: P3-P4-P5-P8-P13-P12-P11-P7-P3, quit LINE. Let the other elements follow.

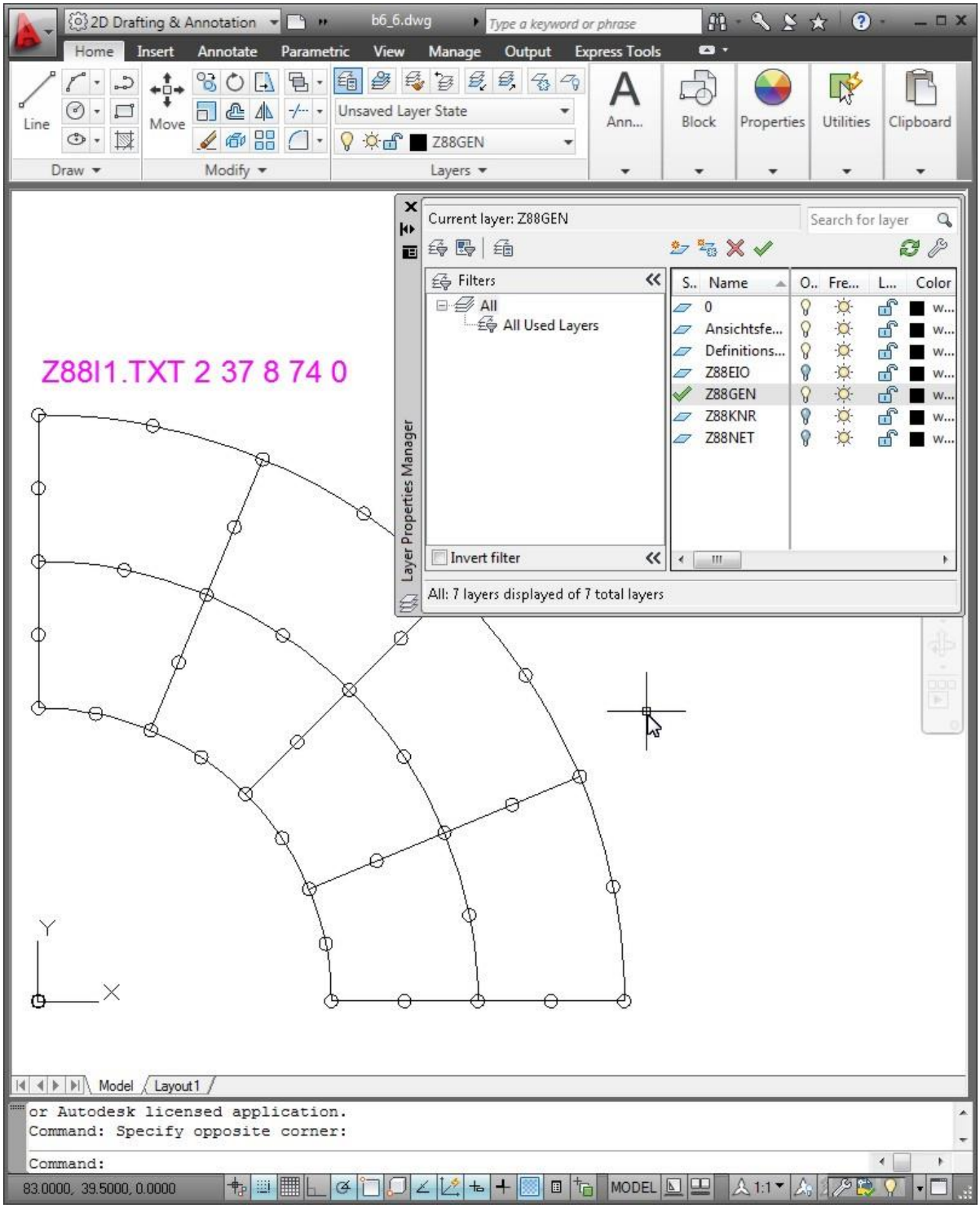
This procedure sounds strange and complicated but be assured that it will work much more easily and quickly than one can describe it: For these 8 elements you will finish work in less than two minutes.



6th step: Define the layer **Z88GEN** and switch it active. Write with the TEXT function into any place of your drawing the **general information,** i.e. the first input group of the general structure data Z88I1.TXT:

Z88I1.TXT
Dimension of the structure
Number of nodes
Number of finite elements
Number of degrees of freedom DoF
Coordinate flag (0 or 1)

Thus, here: Z88I1.TXT 2 37 8 74 0



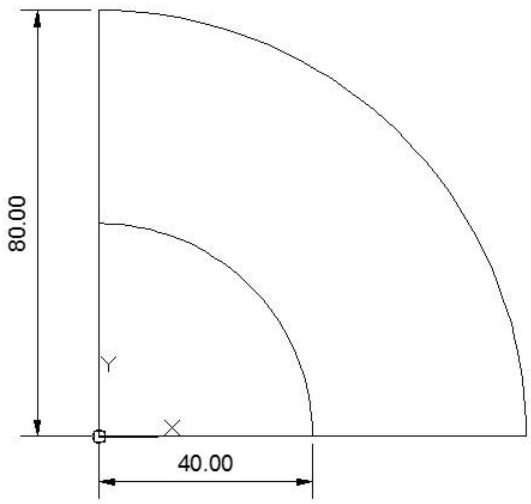
7th step: Store your model or drawing in the DXF file format. Choose AutoCAD R12 DXF format, if in doubt, but AutoCAD 2011 DXF works, too. For precision of decimal positions take the default value which the CAD program suggests. You may import this DXF file into Aurora by DXF import *"DXF structure to Z88Aurora® structure"*. Later in Aurora, you may enter the boundary conditions and surface loads.

EXAMPLE 2 FOR Z88X: SUPER ELEMENTS STRUCTURE

This example is very similar to the first one but now we will generate a super structure. This super structure will be loaded into Z88Aurora® by DXF import and then automatically meshed by the mapped mesher Z88N resulting in a finite elements structure.

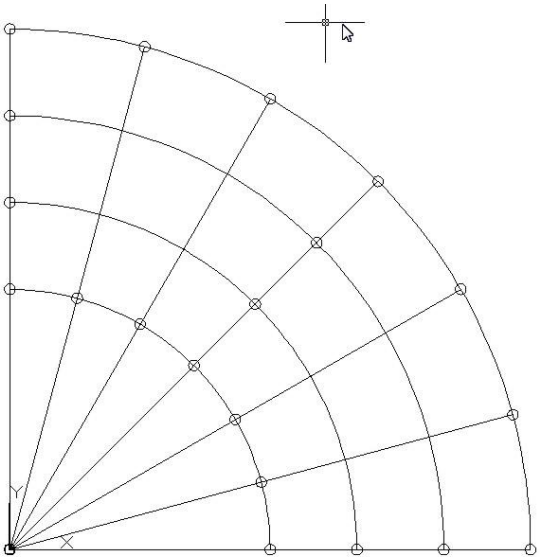
Consider a pipe under internal pressure of 1,000 bar (=100 N/mm^2). Inside diameter of the pipe is 80 mm; outside diameter of the pipe is 160 mm. The length is 40 mm. If one chooses the supports cleverly, a quarter of the pipe is enough to reflect the problem.

1st step: Design your component in the CAD system as usual.

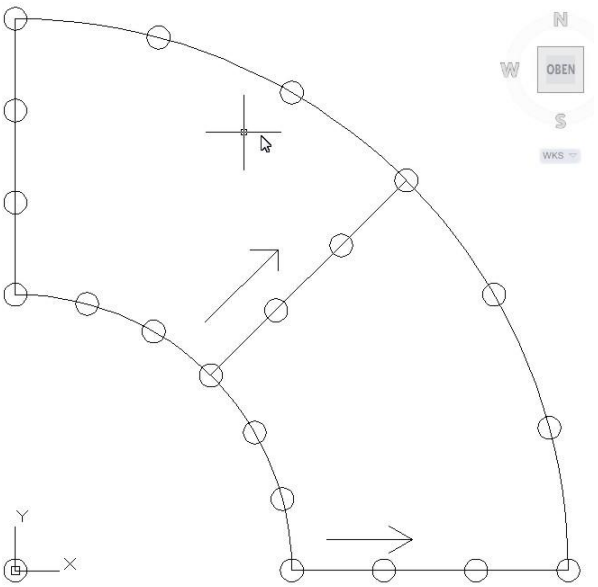


2nd step: We will only use 2 super elements No. 11 with 12 nodes each – and this will do for a nice 90° arc because of the cubic interpolation functions of element No. 11. Subdivide the super structure into elements by lines, insert **all** points which are not yet existing (for example intersection points or end-points of lines are usable). Any order and layer. However, it is recommended not to use the Z88-layers like Z88NET, Z88GEN, Z88PKT, Z88KNR, Z88EIO, Z88FLA, and Z88RBD. Better define any new layer for this or use already available layers from step 1.

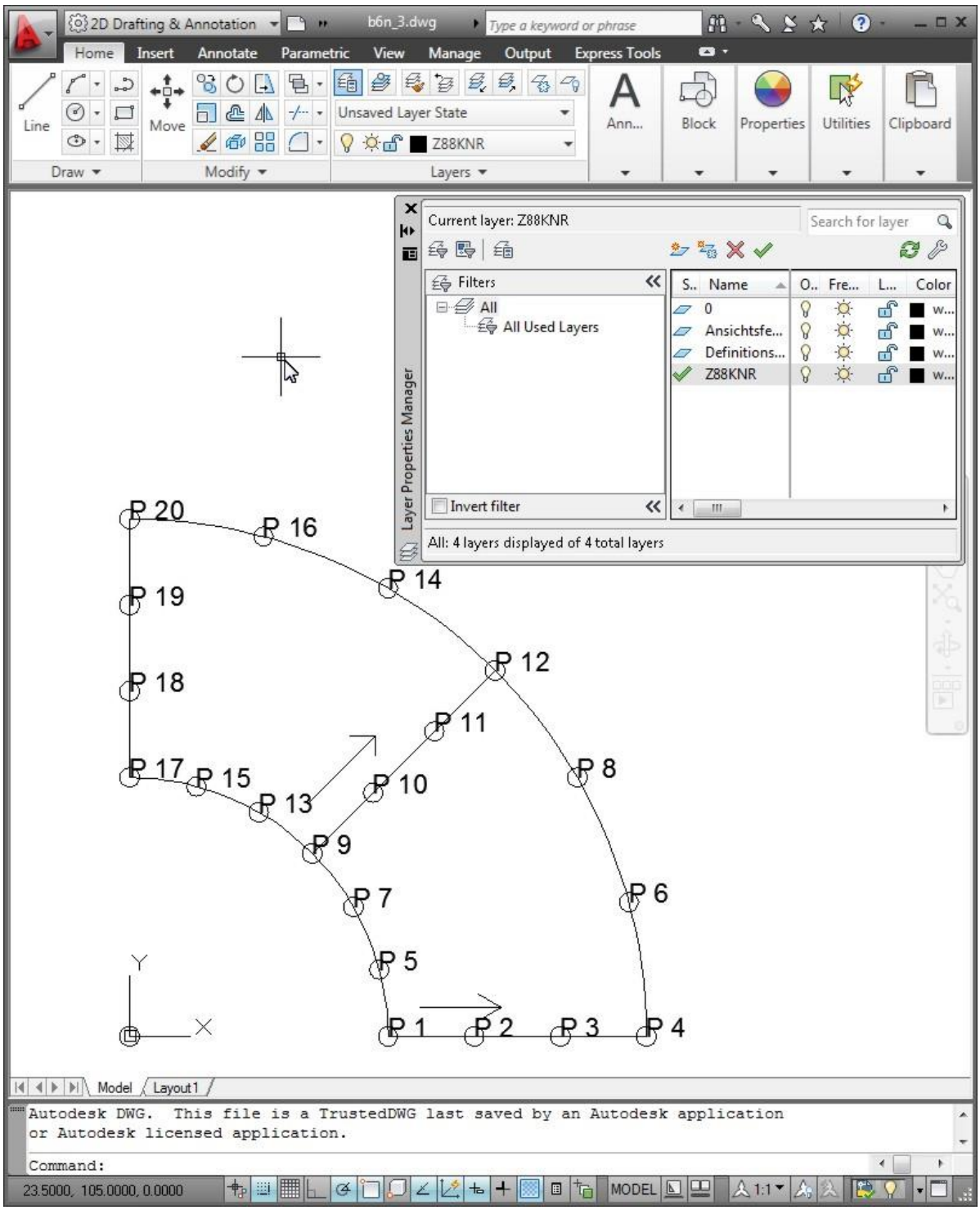
Take care to set nice looking points. Use the AutoCAD command DDPTYPE:



Now you may delete any auxiliary lines, arcs etc. to see the true super elements structure. When working with super elements it is always a good idea to insert arrows to mark the local x-axis for more easy operation later. Thus, you will have the starting point for the multi-line, too.



3rd step: Define the Z88-Layer Z88KNR and make it the active layer. Catch or trap every super node, which were already defined in the 1st step by your construction or have been completed in the 2nd step, and number them. Write to every node **P blank node-number** e.g. *P 33*, with the TEXT function of the CAD program. Be very careful to snap exactly the node and attach the number exactly to the node's location. Take your time! With the snap modes of AutoCAD (intersection point, end-point, point etc.) this works very well. Choose any order of the work sequence as you like.

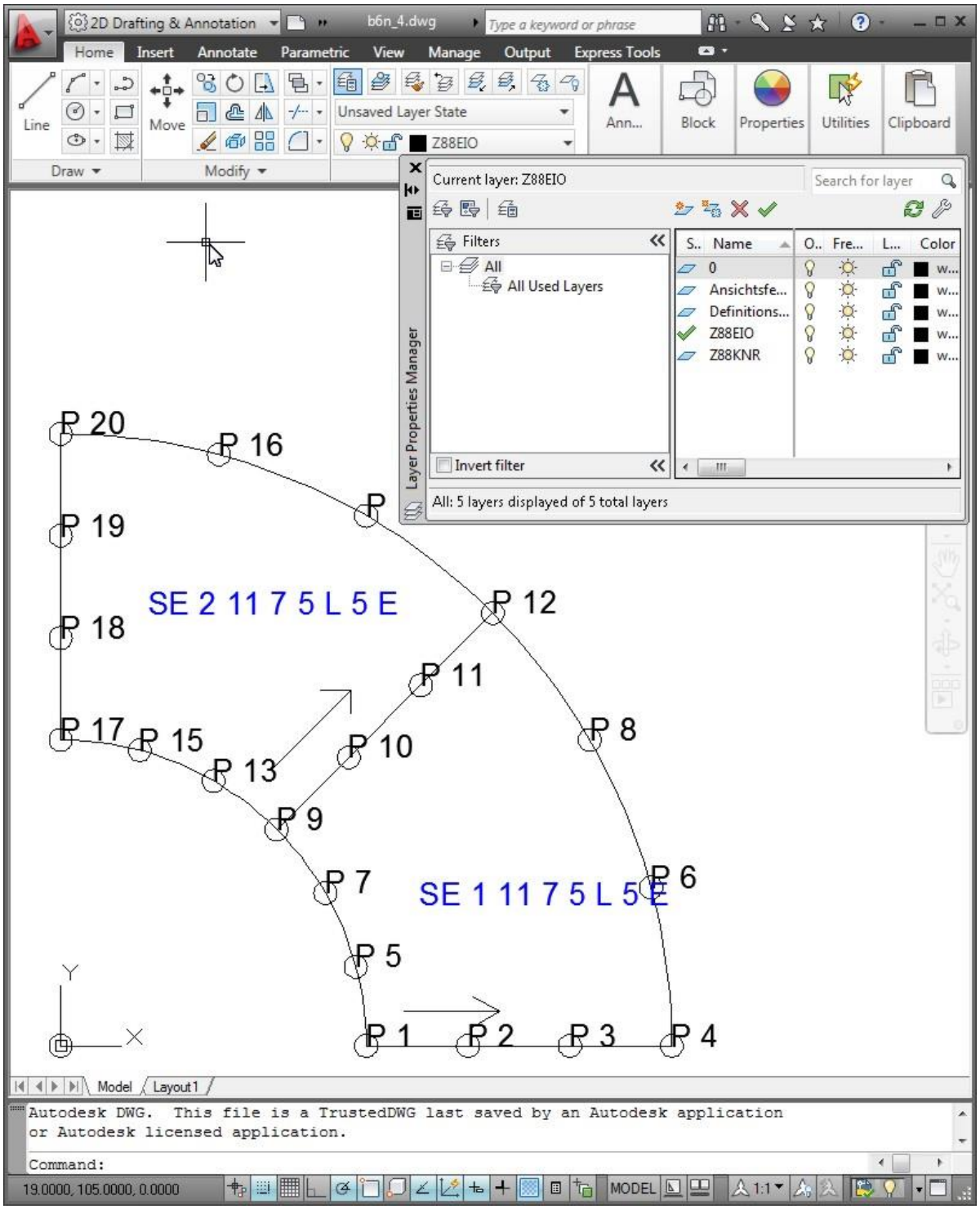


4th step: Define the Layer **Z88EIO** and make it the active layer. Write the super element information with the TEXT function anywhere (of course, it looks nicer with the element info's placed in middle of the respective finite element or super element). The order of the work sequence is up to you. The following information has to be written:

SE
Element number
Super-element type
Type of the finite elements to be produced by meshing
Subdivision in local x direction

Type of subdivision in local x direction
Subdivision in local y direction
Type of subdivision in local y direction

Suppose to mesh the first super element of type No. 11 into finite elements of type No. 7. Subdivide in local x direction 5 times ascending geometrically and subdivide in local y direction 5 times equidistantly. Thus: SE 1 11 7 5 L 5 E
It might be a good idea to use another colour for the objects of layer Z88EIO – here, blue. However, you don't need to.



5th step: Define the Layer **Z88NET** and make it the active layer. You need concentration for this step, because a firm and rigid work sequence must now be kept because of the topological information. One of the most important information, the coincidence, is defined in this step that means which super elements are defined or outlined by which nodes. Choose a proper colour which differs well from the colours used till now and remove all superfluous information by switching off unused layers. Select the **LINE** command and select the proper snap options e.g. points, intersection points and, if necessary, end-points.

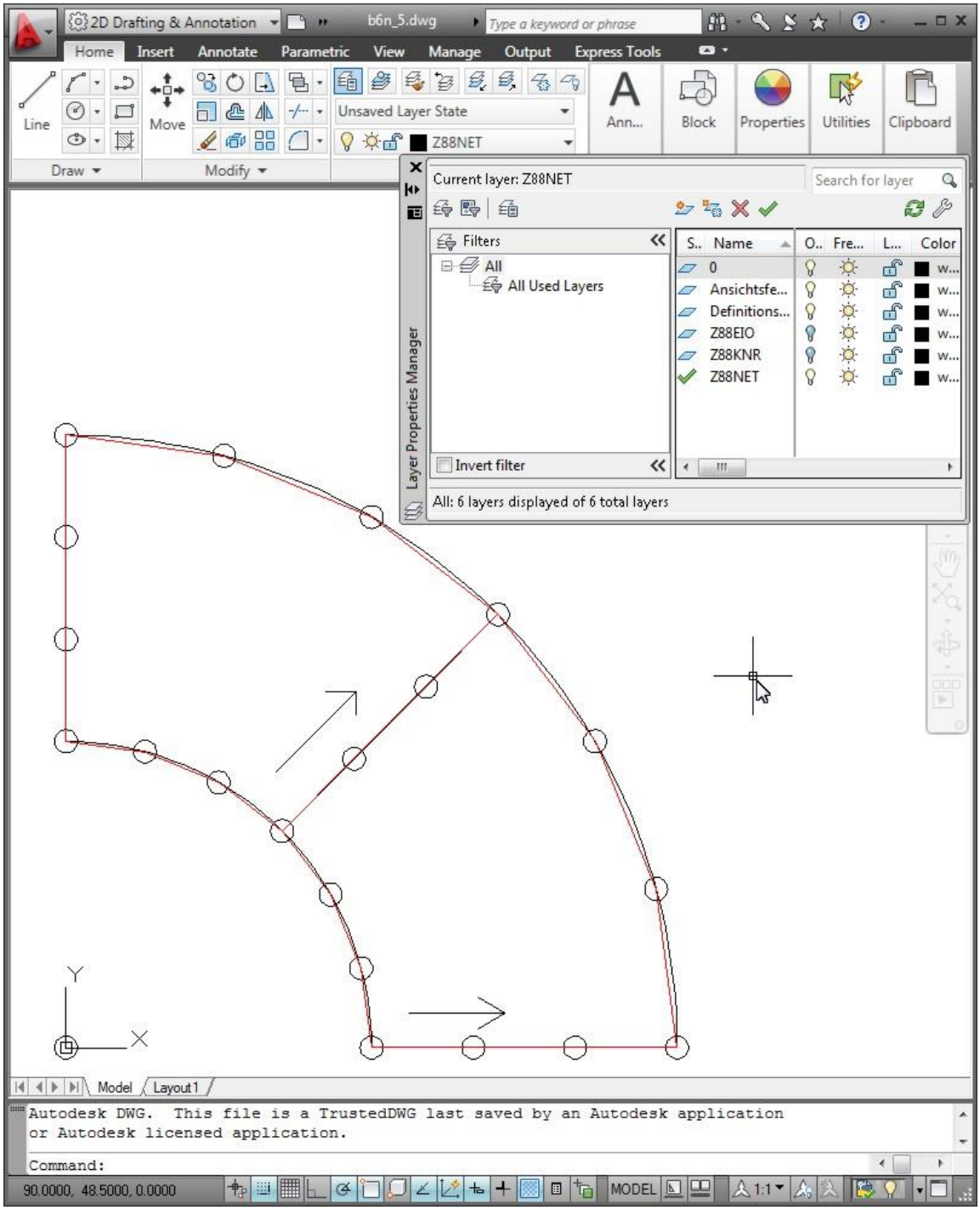
Start at the first element. For Z88 the first element is the element with which you start now, that means the one which you have chosen for your first super element (*SE 1*). Select the node you want to be the first node of this element and draw a line to the node which shall be the second node of this element. From there, draw a line to the third node of this element. Connect all required nodes with lines and draw at last a line to the starting point, the first node, and then quit the LINE function.

Thus, we might draw this line: P1-P2-P3-P4-P6-P8-P12-P11-P10-P9-P7-P5-P1. By the two corner nodes P1 and P4 the local x-axis is defined and this we have already marked by an arrow. This auxiliary arrow has no meaning for Z88X – it was only a hint for us. This fits fine with our definition SE 1 11 7 5 L 5 E.

What would have happened if we would have drawn this line: P4-P6-P8-P12-P11-P10-P9-P7-P5-P1-P2-P3-P4? Basically, nothing. You are only to change the super element's definition to SE 1 11 7 5 E 5 l (← small letter L). And here is why: By drawing the line P4-P6-P8… you did define the local x-axis by the nodes P4 and P12 and, thus, the local y-axis from node P4 to node P1.

Then do the same with the second element. Remember: **You determine with this order which of the elements will be the real second element now**. In the previous 4th step you have only defined what kind of element the second element is. You determine here **how** the element is defined topologically. Thus, we might draw this line: P9-P10-P11-P12-P14-P16-P20-P19-P18-P17-P15-P13-P9, quit LINE.

This procedure takes less than half a minute.

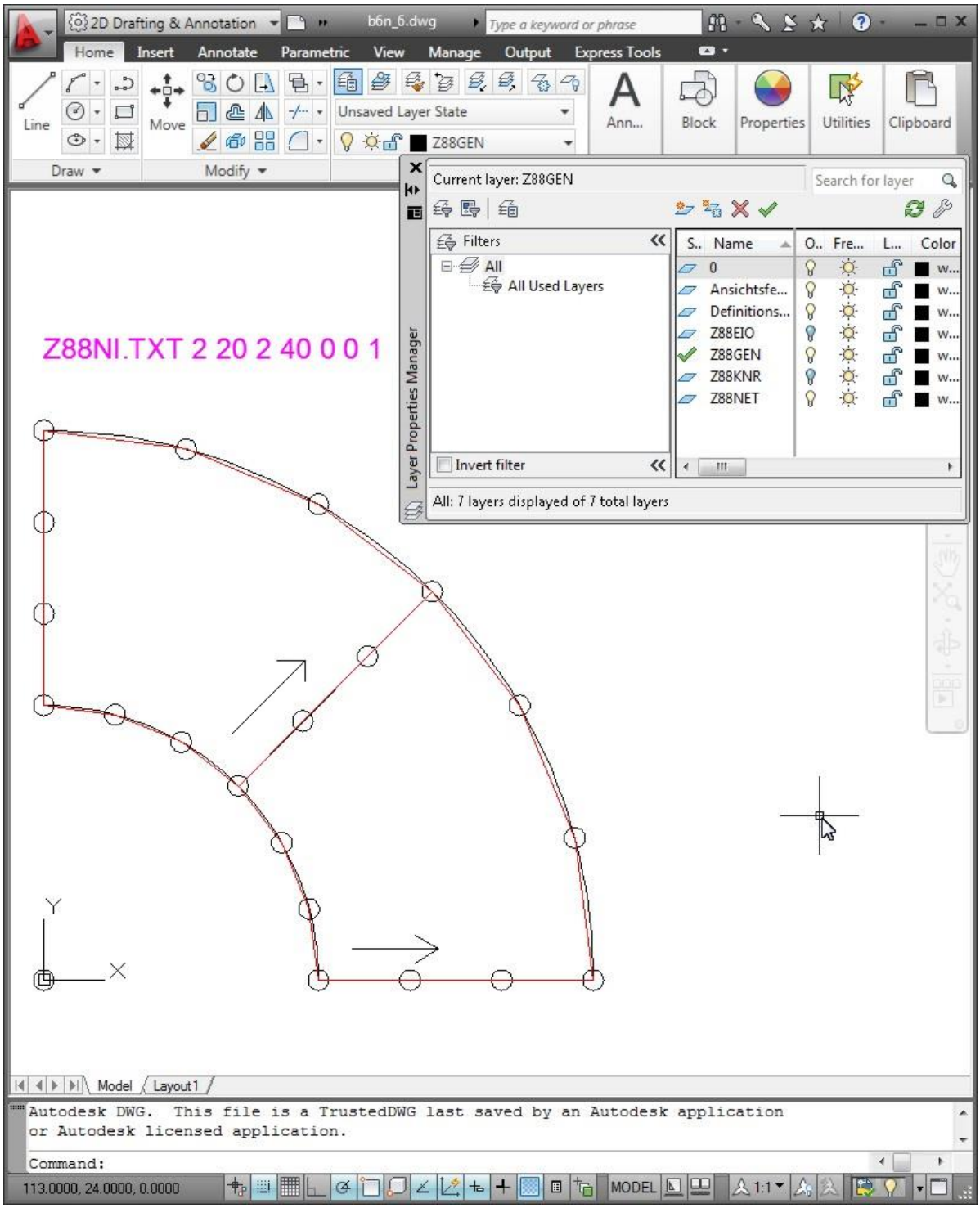


6th step: Define the layer **Z88GEN** and switch it active. Write with the TEXT function into any place of your drawing the **general information,** i.e. the first input group of the general structure data Z88NI.TXT:

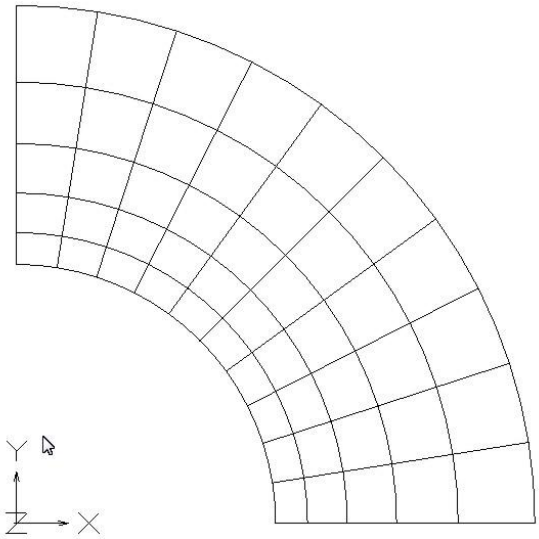
Z88NI.TXT
Dimension of the super structure
Number of nodes
Number of super elements
Number of degrees of freedom DoF
Coordinate flag for super elements (0 or 1)

Trap radius header flag (mostly 0)
Coordinate flag for finite elements (0 or 1)

However, here: Z88NI.TXT 2 20 2 40 0 0 1. Thus, the resulting FE structure will be written by the mapped mesher Z88N in polar coordinates.



7th step: Store this drawing as a DXF file. In Z88Aurora®, we import it by DXF Import with the option *"DXF super structure to Z88Aurora® structure"*. Thus, Z88Aurora® interprets the DXF file as a super structure and launches immediately the mapped mesher Z88N resulting in this FE structure:



Now you may add in Aurora comfortably boundary conditions, materials, integration orders and element thickness.

4.1.7 THE NASTRAN & COSMOS CONVERTER Z88G

Some 3D CAD programs include so-called automeshers which divide a CAD model into finite elements. This generated mesh can be stored in some output format to fit the needs of the various FEA programs. Typical output formats are the COSMOS and the NASTRAN format for the *COSMOS* or the *NASTRAN* FEA program.

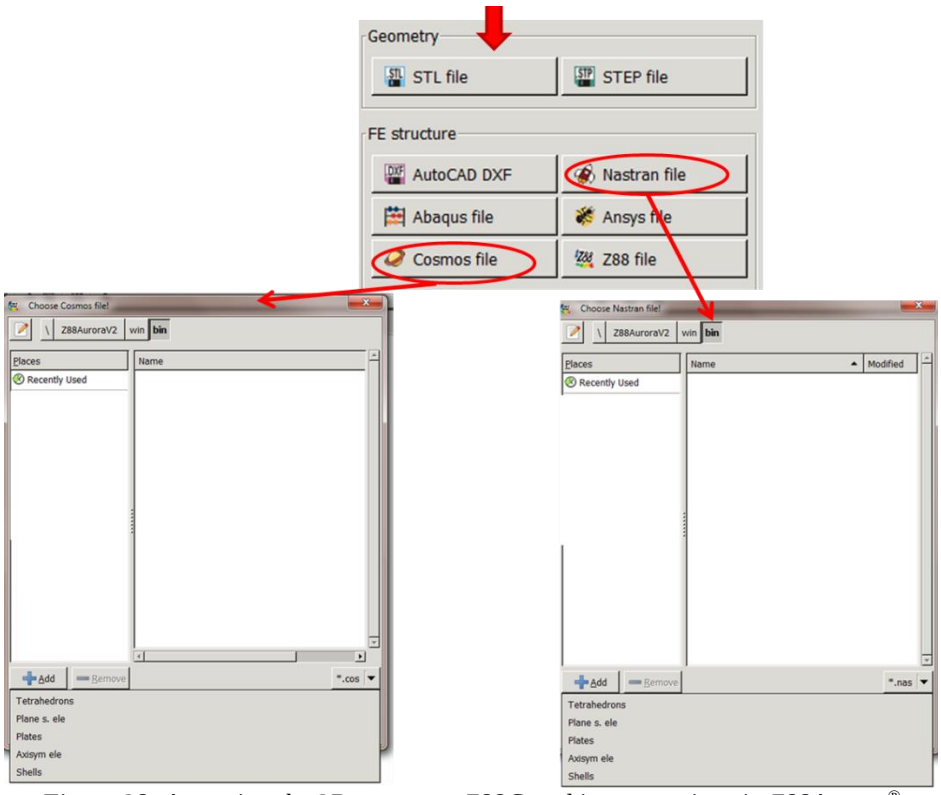


Figure 28: Accessing the 3D converter Z88G and import options in Z88Aurora®

Z88G is developed and tested for *Pro/ENGINEER* by Parametric Technology, USA. Pro/ENGINEER must include the option (the additional module) *Pro/MECHANICA*. Then you may activate *FEA* in the Pro/ENGINEER program after designing your 3D model, define a coordinate system (which must be in harmony with Z88!) and add forces and boundary conditions to single points. Create these single points with *Feature* > *Datum* > *Point*. For plates the direct entry of the pressure load is allowed. Do not forget to define an analysis. Otherwise, no boundary conditions are filed! Modify the mesh control values, if necessary. Create the mesh with *Make Model* and choose the element type e.g. *Tet Mesh*. Store the mesh with *Output Model*, choose *NASTRAN* or *COSMOS/M* and *linear* or *parabolic* (the option *toggle "fix elements"* is not bad for this purpose). Enter *filename.nas* for NASTRAN files or *filename.cos* for COSMOS files for the output file name. Then the converter **Z88G** is launched automatically if you load either a NASTRAN file or a COSMOS file, see figure 28.
Specify the element type to be generated. Of course, both must correlate with what you have previously designed in Pro/ENGINEER. The background, especially of the selection of the element type, is that the output of Pro/ENGINEER is the type shell, even if we deal with plane stress elements, axisymmetric elements or plates. The converter produces the Z88 input files Z88I1.TXT, Z88I2.TXT and Z88I5.TXT automatically. Then, you may enter the material data, element parameters and integration orders directly in Z88Aurora®.

Please note: The import of DXF, ABAQUS, ANSYS, NASTRAN, and COSMOS input decks is positively limited for this version Z88Aurora® to the FE geometry, boundary conditions/forces and surface and pressure loads - in contrary to Z88 V13 and Z88Aurora® V1. And here's why: After the import of commercial input decks you may define the material data, the element parameters and the integration orders very comfortable in Z88Aurora® (even if some of these information are included in the input decks, however), because only in this way a proper use of Aurora's own material data base is possible. This will give you a clean data base for Z88Aurora® projects.

If you find a 3D model totally flat: You have defined a coordinate system CS0 in Pro/ENGINEER which does not fit Z88's needs. Simply define a new correct coordinate system in Pro/ENGINEER and define it as datum when outputting the model. Keep in mind that those exchange file formats and their Pro/ENGINEER output are subject to change every some months.

You may create the following Z88 element types with Z88G:

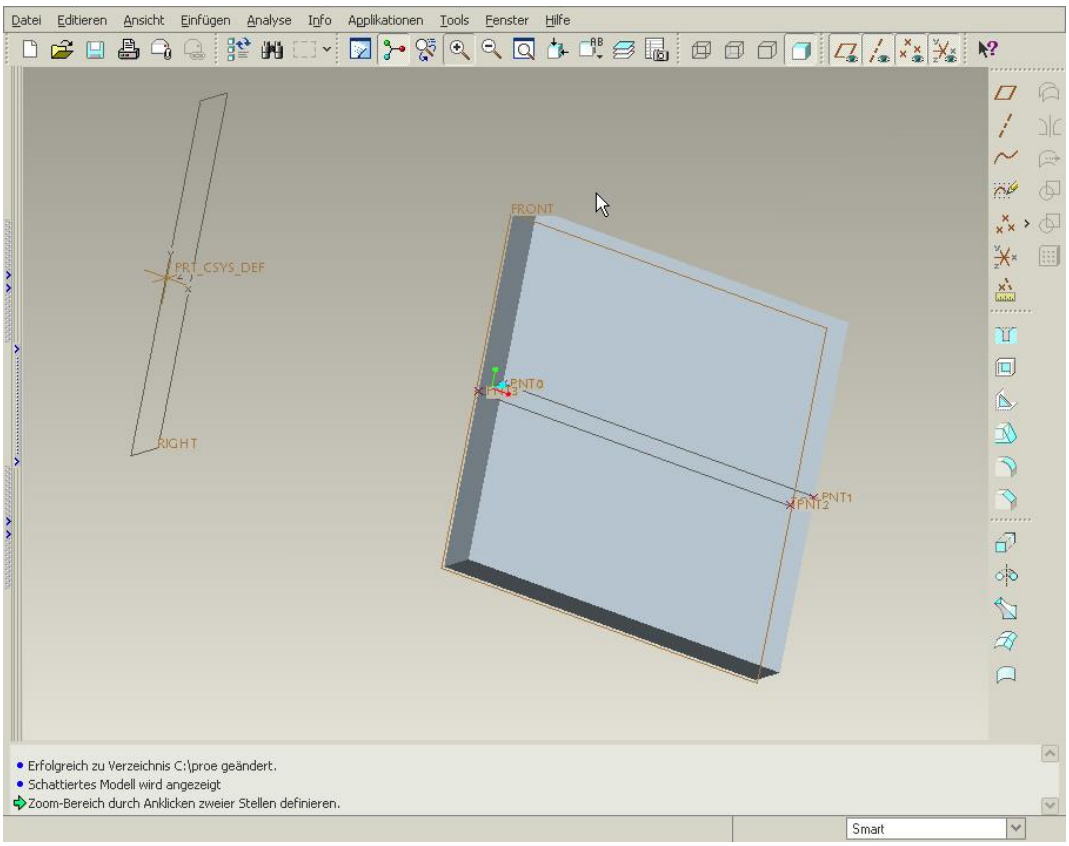
- Tetrahedron No. 16 (*Tetrahedron, parabolic* in Pro/ENGINEER)
- Tetrahedron No. 17 (*Tetrahedron, linear* in Pro/ENGINEER)
- Plane stress No. 14 (*Shell, triangle, parabolic* in Pro/ENGINEER)
- Plane stress No. 7 (*Shell, quadrangle, parabolic* in Pro/ENGINEER)
- Plate No. 18 (*Shell, triangle, parabolic* in Pro/ENGINEER)
- Plate No. 20 (*Shell, quadrangle, parabolic* in Pro/ENGINEER)
- Torus No. 15 (*Shell, triangle, parabolic* in Pro/ENGINEER)
- Torus No. 8 (*Shell, quadrangle, parabolic* in Pro/ENGINEER)
- Shell No. 23 (*Shell, quadrangle, parabolic* in Pro/ENGINEER)
- Shell No. 24 (*Shell, triangle, parabolic* in Pro/ENGINEER)

Please keep in mind that Z88G is capable to deal directly with pressure loads from Pro/ENGINEER only with NASTRAN files. In this case, the file for surface and pressure loads Z88I5.TXT is generated. This is not possible for COSMOS files: Here you are to enter pressure loads via nodal forces.

Choose element type before start.
Pro/ENGINEER makes no distinction between volume elements, plane stress elements, shells, axisymmetric elements, and plates, therefore it is up to you to feed Z88G the right information. The reason for this is that Pro/ENGINEER only recognises the FE type shell or volume. Here, too, you must enter the appropriate data, i.e. what you have already designed in Pro/ENGINEER.

Before running the conversion choose the right type of elements.
The generation of volumes is easy, but the generation of plane stress elements, plates and torus elements (axisymmetric elements) is tricky: First, build a volume with (small) thickness in Pro/E. Set *reference points*, especially for axis-symmetric elements. Launch Pro/MECHANICA and *idealize* the volume into shells: *Model* > *Idealizations* > *Shells* > *Midsurfaces*. This eliminates the depth. When working with axisymmetric elements keep in mind that you are working in cylinder coordinates: Your coordinate system coincidates with the axis of rotation and the "volume" lies on the corresponding radiuses (see figure):

Please keep in mind: These FEA output data formats, especially the NASTRAN format, are modified almost on a daily basis. Thus, consult our homepage www.z88.org for updates.
Anyway: Z88G looks quite harmless, but properly operated is Z88G a mighty tool which allows you to file very large FEA structures to Z88.



This is how torus elements are generated in Pro/ENGINEER. In case of plates and shells, proceed correspondingly.

4.1.8 THE ANSYS CONVERTER Z88ASY

What is the basic idea and which are the features?

Apart from NASTRAN and COSMOS, Pro/ENGINEER also supports the output of simulation data as ANSYS file (*.*ans*). These data can subsequently be transmitted to ANSYS as well as to Z88Aurora®. Please keep in mind, however, that this data format can also be arbitrarily altered by the producer, which might lead to compatibility problems.

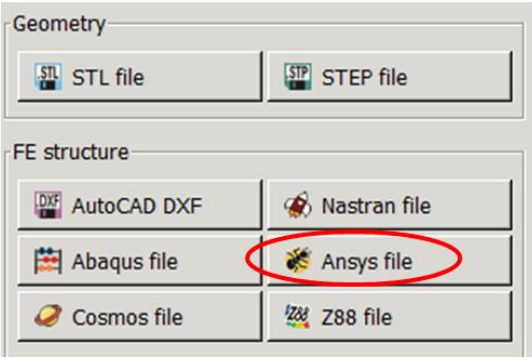


Figure 29: Accessing the ANSYS converter Z88ANS

A solid with any number of materials (linear elastic) can be converted. The solid must consist of one element type. Optional Cartesian boundary conditions, concentrated forces and forces on areas as well as pressures can be applied. You do not need to define any material in ANSYS because:

Please note: The import of DXF, ABAQUS, ANSYS, NASTRAN, and COSMOS input decks is positively limited for this version Z88Aurora® to the FE geometry, boundary conditions/forces and surface and pressure loads - in contrary to Z88 V13 and Z88Aurora® V1. And here's why: After the import of commercial input decks you may define the material data, the element parameters and the integration orders very comfortable in Z88Aurora® (even if some of these information are included in the input decks, however), because only in this way a proper use of Aurora's own material data base is possible. This will give you a clean data base for Z88Aurora® projects.

Which ANSYS files systems can be imported by Z88Aurora®?
ANSYS PREP7 data in ASCII format can feature very different structures and contents, depending on their origin. That is why accurate statements about compatibility cannot be made. Especially integrated scripts can cause problems. This converter was developed and tested for Pro/ENGINEER Wildfire 4. Data generated by ANSYS Workbench cannot be imported.

Which elements are supported by the converter?
You can use tetrahedrons, as linear or as quadratic type:

Conversion > from TET 4 to element type 17 and vice versa
Conversion > from TET 10 (E 92) to element type 16 and vice versa
Which functions does the converter offer?

Import functions of the converter Z88ASY:
Generation > of Z88I1.TXT from an ANSYS file
Generation > of Z88I2.TXT from an ANSYS file

Generation > of Z88I5.TXT from an ANSYS file
Generation > of MAT.TXT from an ANSYS file*

How to proceed?

1. Construct your model according to the instructions for the NASTRAN converter Z88G.
2. Take care to select the ANSYS format when outputting the simulation data.
3. Import the simulation model into Z88Aurora® as described in figure 29. For this purpose select *File > Import > ANSYS file*.

4.1.9 THE ABAQUS CONVERTER Z88AINP

What is the basic idea and which are the features?
Today, ABAQUS has become a wide-spread simulation tool in the industrial field, offering a large range of performance as well as simple operation. Due to the extensive range of functions, the following restrictions were made with regard to the functions of the converter:

One solid (1 instance) with one material (linear elastic) can be converted. The solid must consist of one element type. Optional Cartesian boundary conditions, forces (Concentrated Force) and pressure can be applied. You do not need to define any material in ANSYS because:

Please note: The import of DXF, ABAQUS, ANSYS, NASTRAN, and COSMOS input decks is positively limited for this version Z88Aurora® to the FE geometry, boundary conditions/forces and surface and pressure loads - in contrary to Z88 V13 and Z88Aurora® V1. And here's why: After the import of commercial input decks you may define the material data, the element parameters and the integration orders very comfortable in Z88Aurora® (even if some of these information are included in the input decks, however), because only in this way a proper use of Aurora's own material data base is possible. This will give you a clean data base for Z88Aurora® projects.

Which ABAQUS versions can cooperate with Z88Aurora®?
The converter at hand was tested with ABAQUS 6.12.3; therefore the full range of functions is available in this case. Since the ABAQUS format is proprietary, modifications may occur at any time, resulting in malfunction of the converter. Older versions of ABAQUS, e.g. 6.6 or 6.7 also do not write any version information into the files. Therefore a version-dependent conversion is difficult as well.

Which elements are supported by the converter?
You can use any Tetrahedrons and Hexahedrons from ABAQUS, but since normally no acoustic or thermal simulation data are exchanged between ABAQUS and Z88Aurora®, the following element transformations will occur:
Conversion > from C3D4 to element type 17 and vice versa
Conversion > from C3D10 to element type 16 and vice versa
Conversion > from C3D8 to element type 1
Conversion > from C3D20 to element type 10

Which functions does the converter offer?

Import > of Z88STRUCTURE.TXT, Z88SETSACTIVE.TXT and Z88SETS.TXT from an ABAQUS input file.

How to proceed?
You can use files from ABAQUS CAE as well as your own input decks. Please look for the corresponding keywords in the ABAQUS documentation, which you may enter into the file INPENVIRO.Z88 and pay attention to upper and lower case characters. For ABAQUS 6.12.3 is enclosed the suitable INPENVIRO.Z88 in the BIN directory of Z88Aurora®. ABAQUS scripts cannot be processed.

For the import of an ABAQUS file proceed as follows:

Importing and meshing the component in ABAQUS:
The ABAQUS converter only processes components which you can arbitrarily import into ABAQUS-CAE and fit into an *Assembly*. It is up to you, whether you select *Mesh on Part* or *Mesh on Instance*. Allocate either *Hex* or *Tet* as meshing properties (see figure 30) and generate a mesh which meets your requirements.

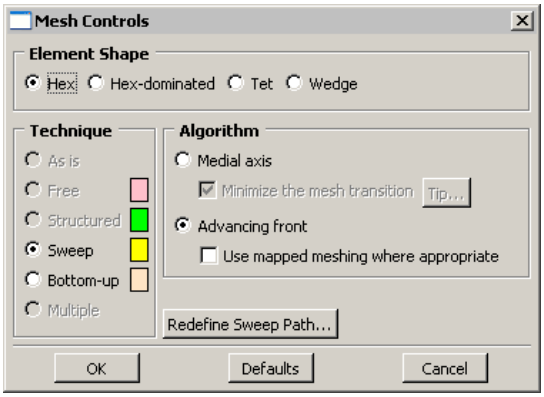


Figure 30: Allocation of the appropriate element type in ABAQUS 6.12.3

Note: Extended settings of the mesh control and element selection are not adopted, since there are no corresponding equivalents in Z88Aurora®. Thus, in case you have chosen hybrid formulation or an element for acoustic analysis, this will be transformed into a pure Z88 type when being imported into Z88.

Remarks: Z88AINP processes all loads of a *step* which are given in the ABAQUS file. Should you have defined several simulation steps in your CAE model, note that you should provide a new model by copying in which all steps are deleted except the desired simulation step.

1. Write out the input deck as a *.inp file.
2. In Z88Aurora® you select from the menu *File> import> ABAQUS data*. In the following choice dialog you can select automatically only .inp files. Select the desired file.
3. The converted structure is plotted and you may show constraints and loads in Z88Aurora.

4.2 PICKING & SET MANAGEMENT

Boundary conditions, material definitions, surface loads - all these qualities of a FE analysis which are controlled in the Open Source version by the input files Z88I1.TXT to Z88I5.TXT are tied together in Z88Aurora® with the *set management*. In principle, the action is always the same. In the Picking menu nodes, element, or surfaces groups can be marked.

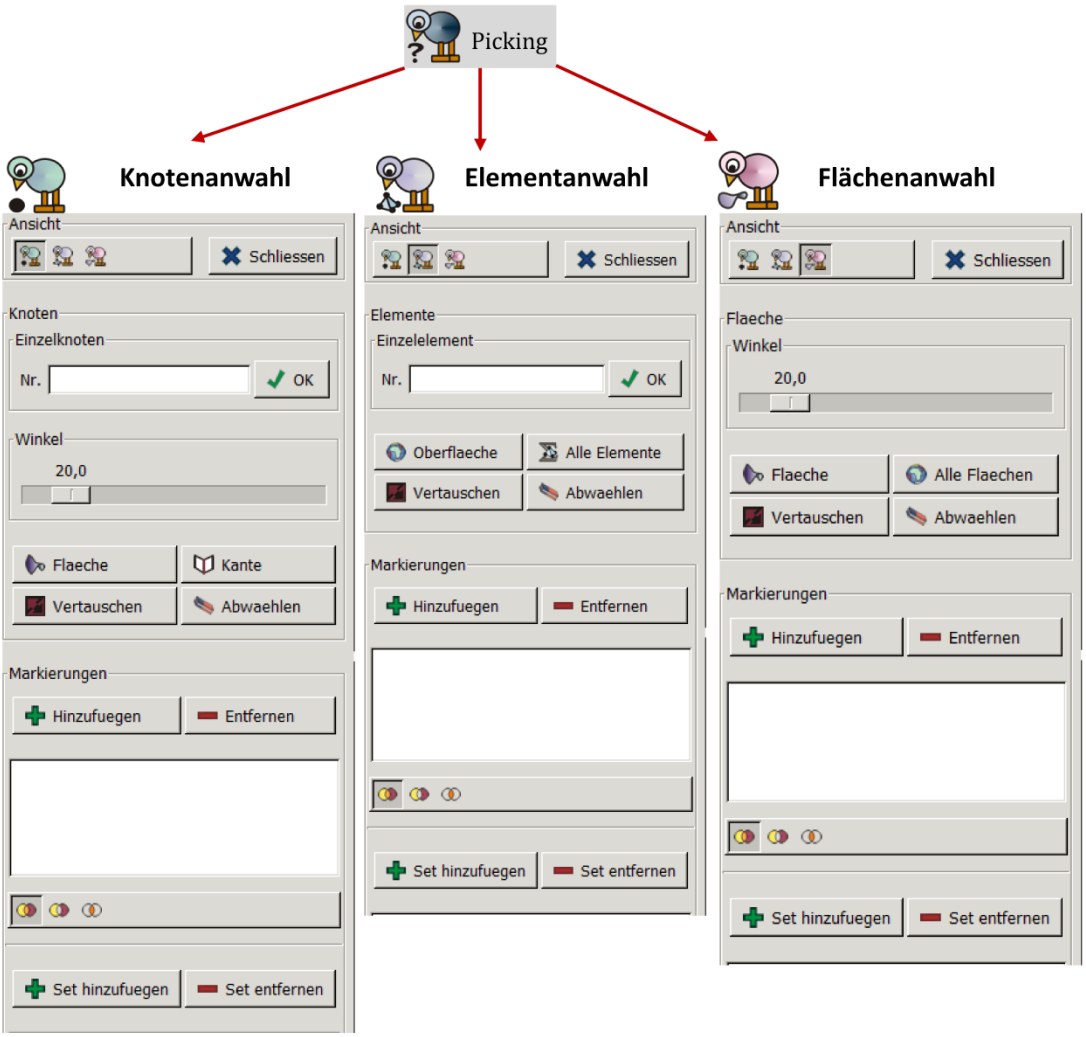


Figure 31: Picking options in Z88Aurora®

During the pre-processing, properties can be assigned to this set, like materials, loads, boundary conditions, meshing rules, etc. The last entries in the respective menus, stored in the file SETSACTIVE.TXT, control afterwards the calculation. Z88Aurora® is planned in the shape that the pre-processing is carried out interactively by the user in a graphical surface interface i.e. a GUI. Though the files are clear and built up logically, nevertheless, the production of hand is difficult. This is not the key feature of Aurora, but from Z88 V15 OS. The use of the Pickings and the set management is found in the user's handbook.

4.2.1 SURFACE LOADS

Although the entering of the boundary conditions is not required by hand in Aurora, you need some knowledge on the peculiarities regarding surface loads. With distributed loads and surface loads like pressure loads and tangential shear some specific features are to be followed.

Seldom is a load in a few points on a body, mostly loads reflecting surface loads. For the distribution of these loads on the component there are several possibilities:

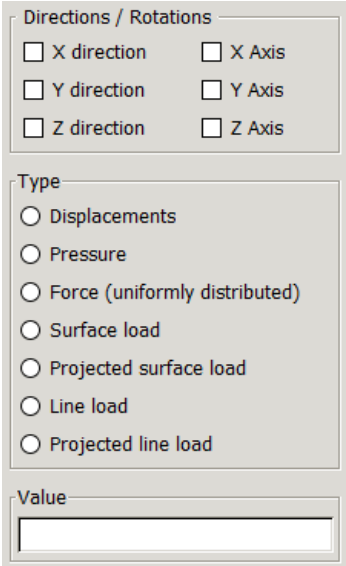


Figure 32: Possibilities of the boundary conditions in the boundary condition menu

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 the elements with linear shape function, e.g. Hexahedrons No. 1 and Torus No. 6, edge loads and surface loads are distributed to the elements simply and straight-forward onto the respective nodes.

However, for elements with higher shape functions, i.e. square (Plane Stress No. 3, No. 7, Torus No. 8, Hexahedron No. 10 etc.) or cubic (Plane Stress No. 11 and Torus No. 12) edge and surface loads have to be put onto the elements according to certain rules which are not always physically obvious but mathematically absolutely correct. Amazingly, some load components can have negative values. Though these facts are not obvious, nevertheless they lead to correct results, which is not the case for intuitive distribution of loads to the respective nodes.

An example may clarify the facts:

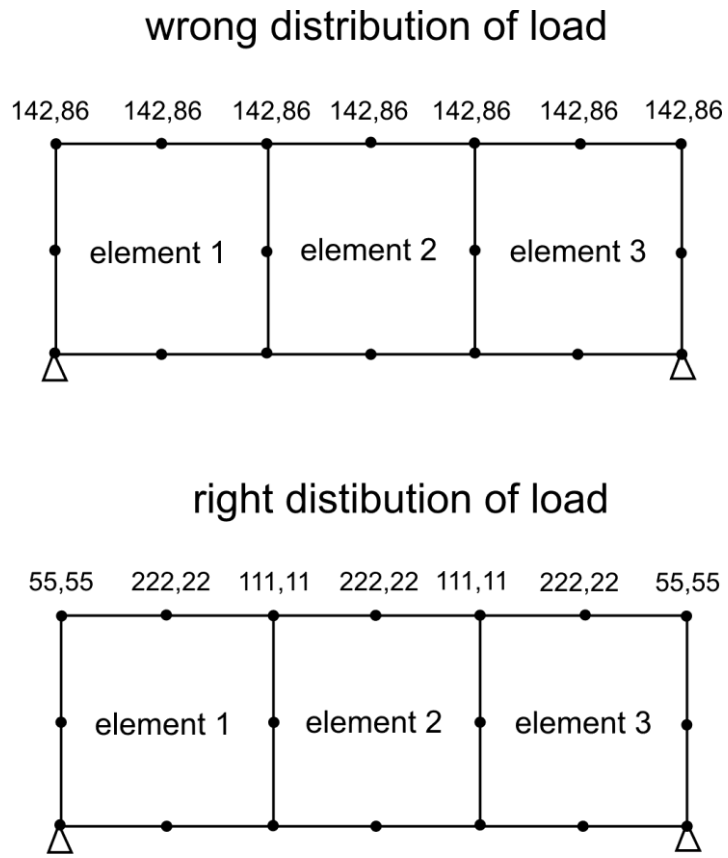


Figure 33: The distribution of loads

An FE structure consists of three plane stress elements No. 7 with the load of 1,000 N distributed on the upper edge in Y direction, see figure 35:

Incorrect: 1,000N/7=142.86 N per node. Not correct for elements with square shape function.

Correct: 2 × 1/6 + 2 × (1/6+1/6) + 3 × 2/3 = 18/6 = 3, corresponds to 1,000 N
"1/6 points" = 1,000/18×1 = 55.55
"2/6 points" = 1,000/18×2 = 111.11
"2/3 points" = 1,000/18×4 = 222.22
Control: 2×55.55 + 2×111.11 +3×222.22 = 1,000 N, o.k.

Here's why:

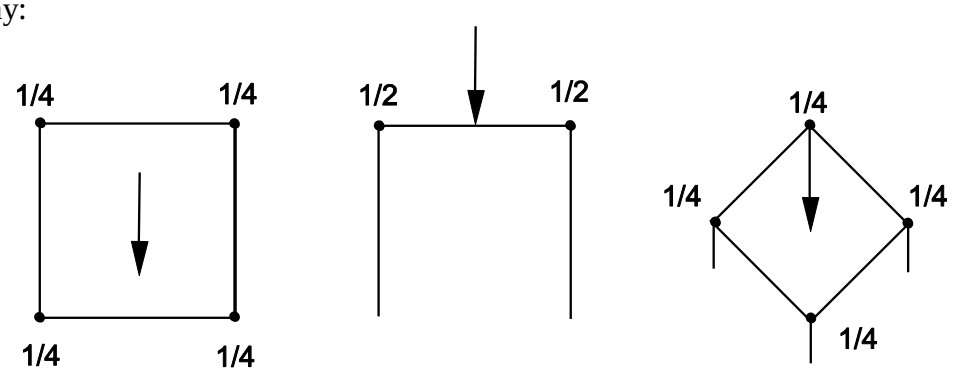


Figure 34: Elements with linear shape functions, e.g. Hexahedron No. 1

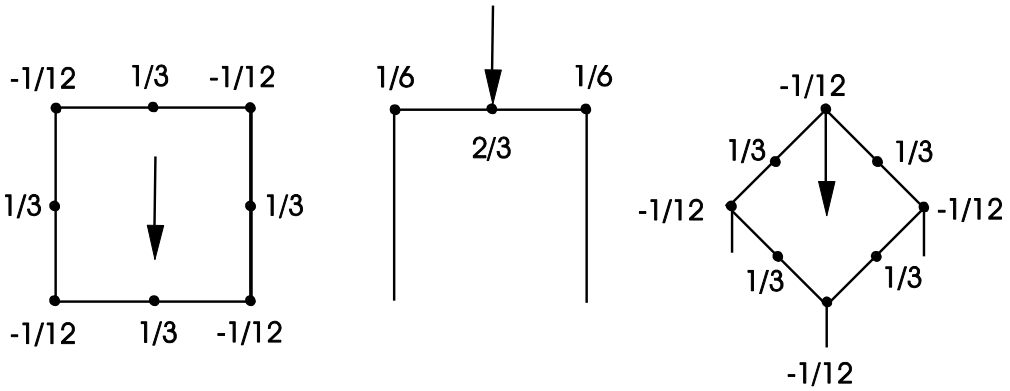


Figure 35: Elements with quadratic shape functions,
e.g. plane stress element No. 3 and 7, Torus No. 8, Hexahedron No. 10

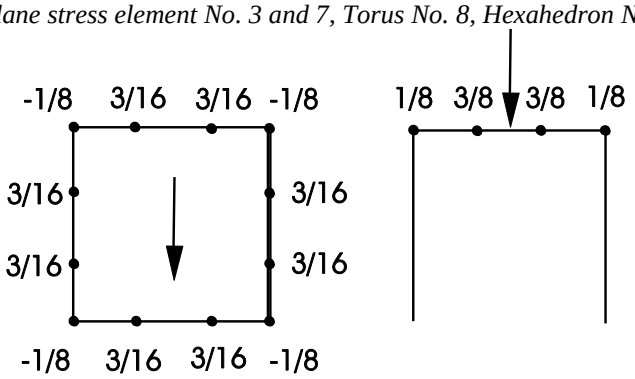


Figure 36: Elements with cubic shape functions,
e.g. plane stress element No. 11, torus No. 12, plate No. 19

The choice surfaces *„projected surface load“* and *„projected line load“* take the fact into account that meshing with Free-Meshern very often means irregular node distributions, particularly with curved surfaces. If a surface contains more nodes on her right half than in the left, the load application occurs one-sided. This effect is weakened by the projection.

Further details to the application of boundary conditions see user's handbook.

4.3 THE LINEAR SOLVER Z88R

The linear solver Z88R includes internally three different solvers:

- The so-called *Cholesky solver* without *fill-in*, with so-called Jennings storage. It is easy to handle and fast for small and medium structures. `Z88R -choly` is your choice for small and medium structures, up to 20,000 ... 30,000 degrees of freedom. **In Z88Aurora® can be calculated only truss and beam frameworks with this solver.**
- A so-called *direct sparse matrix solver with fill-in.* It uses the so-called PARDISO solver `Z88R -parao`. This solver is very fast, since it is multi-CPU compliant, but it uses very much dynamic memory, therefore the program is likely to quit with an error message, if the main memory is exhausted. This solver is your choice for medium structures, up to 150,000 degrees of freedom on ordinary PCs. However, we have computed structures with ~ 1 million of DoF very fast using a computer featuring 32 (!) Gbyte of memory, 4 CPUs, 64 bit Windows version of Z88.
- The so-called *sparse matrix iteration solver*. It solves the system of equations by the method of conjugate gradients featuring SOR preconditioning `Z88R -sorcg` or

preconditioning by an incomplete Cholesky decomposition `Z88R -siccg` depending on your choice. This solver needs a minimum of storage. This is your choice for large structures with more than 150,000 ~ 200,000 DoF. FE-structures with ~ 5 million DoF are no problem for it if you use a 64 bit operation system (Windows, LINUX or Mac OS X) with about 6 Gbyte of memory. The largest processed structure up to now in Z88R featured 12 million DoF – using an ordinary PC! *This very stable and approved solver works always, thus, you may use it as your standard solver.*

Note: The following explanations for the "manual" launch of the solver are only meant for a deeper understanding, if necessary. Z88Aurora® takes care of everything for you!

The solver Z88R runs in console mode and requires two control flags:

z88r -mode -solver

mode means:
`-t`: Test mode. Z88R determines the required memory and enters these settings into the memory definition file Z88R.DYN.
`-c`: Computing mode. Z88R.DYN is imported.

Run the solver in test mode first, and then a second time in computing mode, using the *same setting* of the second parameter *solver*:

solver means:
`-choly`: Launch the simple Cholesky solver without fill-in with Jennings storage
`-parao`: Launch the direct sparse matrix solver with fill-in and solver PARDISO
`-siccg`: Launch the iteration solver conjugated gradients with SIC preconditioning
`-sorcg`: Launch the iteration solver conjugated gradients with SOR preconditioning

Explanations to the sparse matrix iteration solvers SICCG and SORCG

An iteration solver uses only the so-called non-zero elements – which results in an absolute minimum of storage requirements. It builds the following pointers for the lower part of the total stiffness matrix GS:

- Pointer vector IP points to the diagonal elements GS(i, i)
- Pointer vector IEZ points to the column index GS(x, j)

Example (ref. Schwarz, H.R: Methode der finiten Elemente): Let the lower part of GS be:

GS(1,1)					
GS(2,1)	GS(2,2)				
	GS(3,2)	GS(3,3)			
GS(4,1)			GS(4,4)		
GS(5,1)		GS(5,3)		GS(5,5)	
	GS(6,2)		GS(6,4)		GS(6,6)

GS results in the following vector of non-zero elements:

GS(1,1)	GS(2,1)	GS(2,2)	GS(3,2)	GS(3,3)	GS(4,1)	GS(4,4)
GS(5,1)	GS(5,3)	GS(5,5)	GS(6,2)	GS(6,4)	GS(6,6)	

IEZ will result in:

1	1	2	2	3	1	4	1	3	5	2	4	6

and IP:

1	3	5	7	10	13

The pointer IEZ holds MAXIEZ elements, the vector GS holds MAXGS elements. These limits are determines in the test mode of the solver.
In the second run, the actual computation run, the solver computes the element stiffness matrices, compiles the total stiffness matrix, incorporates the boundary conditions, scales the system of equations and solves the (huge) system of equations by the conjugate gradient algorithm. Preconditioning is done for better convergence. You can choose whether to use a *SOR* step or a so-called incomplete Cholesky decomposition for precondition. Default is the incomplete Cholesky decomposition (*shifted incomplete Cholesky decomposition SIC*), because the main parameter, the so-called shift factor α, is easy to handle. The *SOR* preconditioning needs less memory, but the control parameter, the relaxation parameter ω, cannot be determined a-priori.

In addition, you must enter several parameters into the file Z88CONTROL.TXT. This is done via "Solver parameters" in the menu "Solver" for every solver:

- termination criterion: maximum count of iterations (e.g. 10000)
- termination criterion: residual vector < limit *Epsilon* (e.g. 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; start with 0.0001). 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).

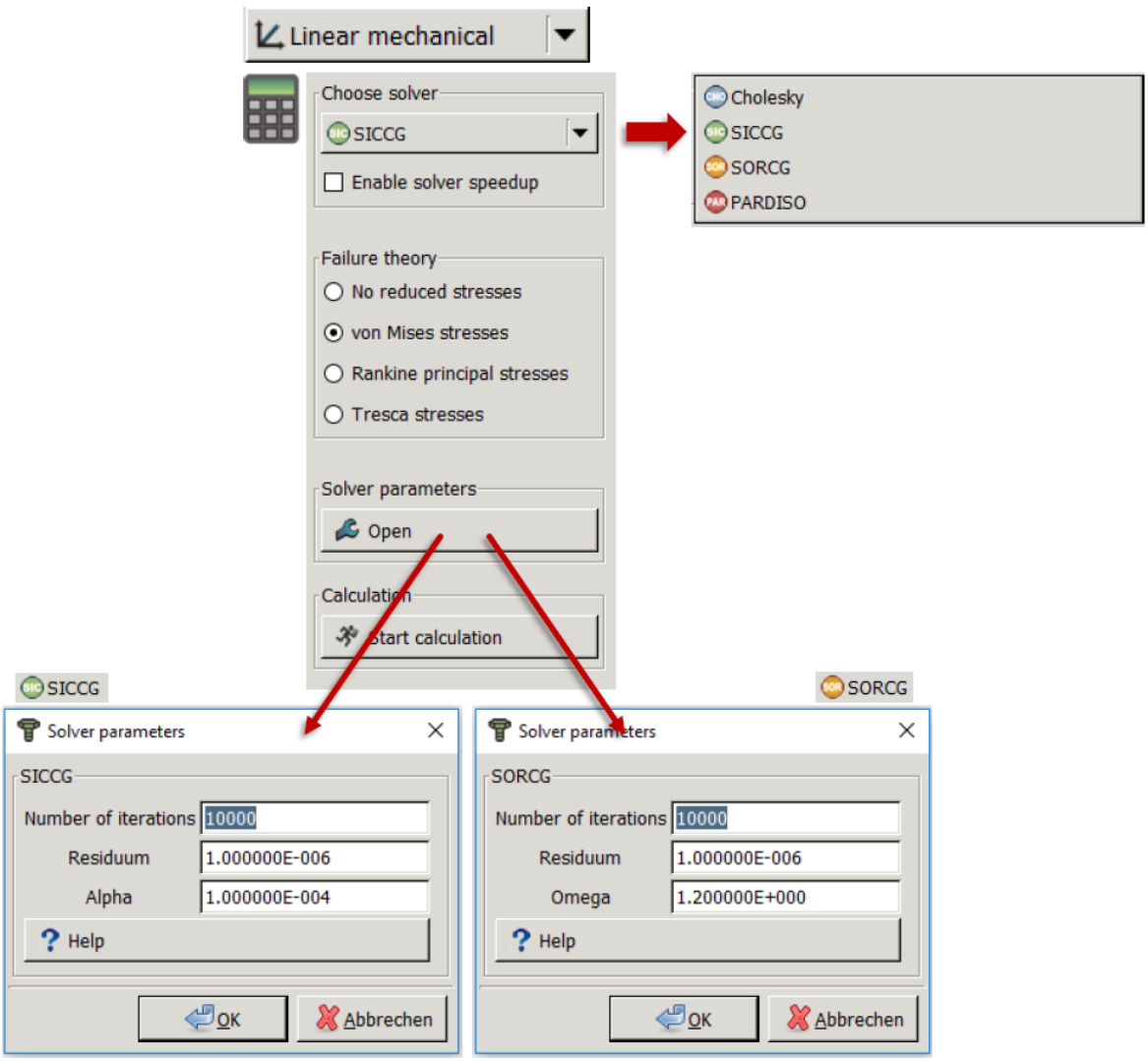


Figure 37: Options of the solver menu according to the chosen solver

Explanations to the direct sparse matrix solver with fill-in

This solver does direct matrix decomposition, but in contrary to the simple Cholesky solver, this solver operates with *fill-in*. Fill-in means allocating dynamic memory for the new matrix elements created by the decomposition process. Thus, the memory needs cannot be calculated before running the solver. If the memory is exhausted during the calculation, the solver will inevitably quit with an error message. This solver works at very high speed for medium structures (100.000 ~ 1.000.000 DoF) because it is multi-processor compliant, but needs more memory than the iteration solver by several orders of magnitude. Therefore, this solver is only really useful on machines with very much memory and 64 bit pointers and integers. We recommend the 64 bit version of Z88Aurora®, a 64 bit Windows operating system and a minimum of 4 Gbyte (8 or 16 bByte are even better) of memory for this solver. The actual solver core used is PARDISO by O. Schenk, University of Basel, Switzerland. Define the number of CPUs in Z88MAN.TXT. The values preceding have no significance; they must be there, however. Please take care that in the Windows settings System *Properties* > *Advanced* > *Environment Variable* you do not have this kind of variable: NUM_THREADS, OMP_SET_NUM_THREADS. This might clash with the settings in Z88CONTROL.TXT.

Input files for both modes:

- Z88I1.TXT general structure data
- Z88I2.TXT boundary conditions
- Z88I5.TXT surface and pressure loads
- Z88MAT.TXT material defini. and one or more material files in .TXT format
- Z88ELP.TXT element parameters
- Z88INT.TXT integration orders
- Z88MAN.TXT solver parameters

Output files in computing mode:

- Z88O0.TXT prepared structure data for documentation
- Z88O1.TXT prepared boundary conditions for documentation
- Z88O2.TXT displacements
- Z88O3.TXT stresses
- Z88O4.TXT nodal forces

4.3.1 CHOOSING THE RIGHT SOLVER

Roughly spoken: Use the simple and reliable Cholesky solver Z88R –choly for small truss and beam structures. The sparse matrix iteration solver Z88R –siccg or –sorcg *always* works. For medium sized structures the direct sparse matrix solver with fill-in Z88R –parao is very suitable because of its tremendous speed.

Since all modifications for the advanced versions of SICCG, SORCG and PARDISO not concern the numeric algorithms, the advantages and disadvantages of the solver selection remain in principle. However, especially in the iterative CG method even with two CPUs significant time savings are expected, so the speed drops to PARDISO solver. Even for medium-sized structures is now an alternative given, which is comparable in speed. For the calculation of large and very large structures remain the CG-solver the tool of choice. Although twice memory in the parallelized-based version is needed, this requirement is also smaller PCs easily achievable.

Table 6: 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	only for trusses and beams in Z88Aurora®
Z88R -t/c -parao	direct Solver with Fill-In	up to ~ 150.000 with 32-bit PCs	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 (tested with more than 12 Mio. DoF on a normal PC)	an absolute minimum	medium	no	a very stable and reliable solver for very large structures

Z88RS -t/c -parao	direct Solver with Fill-In	up to ~ 150.000 with 32-bit PCs	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 (tested with more than 12 Mio. DoF on a normal PC)	very low	high	yes	requires twice as much memory as the CG-solvers from Z88R and enables several CPUs

Note that the termination criteria have changed in the iterative solvers in Z88RS. Enter the mathematical residual. This is the root of the criterion in Z88R!

4.3.2 SOME NOTES ON STRESS CALCULATION

The results are presented in the file Z88O3.TXT. The stress calculation is controlled via the file Z88MAN.TXT, see chapter 3. It defines, among other things:

- Calculation of the stresses at the Gauss points or at the corner nodes
- Additional calculation of radial and tangential stresses for elements No. 3, 7, 8, 11, 12, 14 and 15.
- Calculation of *von Mises* stresses for continuum elements No. 1, 3, 6, 7, 10, 11, 12, 14, 15 ~ 24.

4.3.3 SOME NOTES ON NODAL FORCE CALCULATION

The results are presented in Z88O4.TXT. The nodal forces are calculated separately for each element. If several elements meet a node, one gets the complete nodal force for this node by adding the nodal forces of all accessing elements. These results are presented further down in the nodal force file Z88O4.TXT.

4.4 THE VIBRATION SOLVER Z88EI

This eigen solver offers the possibility to examine components concerning her natural frequencies as well as the oscillation forms caused from it. The accelerating and damping nodal forces which are caused by dimension inertia or reset forces from stiffness qualities keep just the scales with this frequency. As with the linear solver Z88R information to the modulus of elasticity and the across contraction becomes necessary. In addition, still the material density is required for the mass calculation. All material data are deposited in the material data base. Figure 36 shows the choice and control of the natural oscillation module. The analysis is for tetrahedron and hexahedron with linear and square approach (element model: 1, 10, 16, 17) possibly.

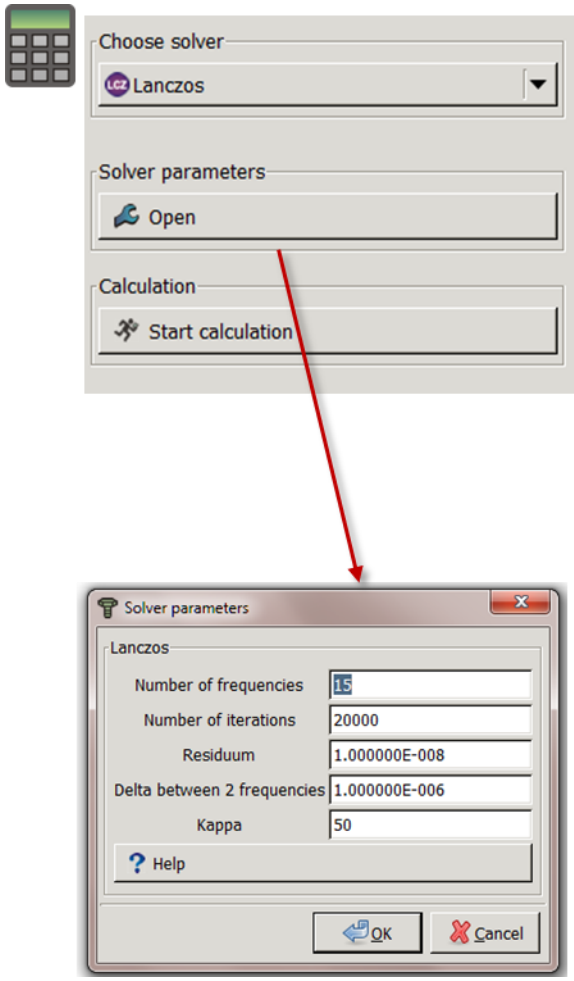


Figure 38: Parameters of the vibration solver

Taking care is necessary concerning the boundary conditions. While fixing boundary conditions (movement is zero) can be raised as usual on any nodal sets - also in single degrees of freedom - forces, pressures as well as inhomogeneous movements are ignored. The aim of the natural frequency simulation is the calculation of free natural frequencies; any dynamic suggestions or external loads do not fall under it. On the other hand, a component must be statically defined for the vibration analysis by no means. It can be also carried out totally without boundary conditions a calculation without any problems.

This time mathematically no equation system is to be solved but to carry out an eigenvalue calculation of a system matrix which contains information about the stiffness distributions as well as about mass distributions with regard to the FE mesh. The eigenvalue calculation is considerably numerically costlier than the solution of the equation system and demands in practice according to more arithmetic time. The procedure can be split up basically in two phases: First a so-called tridiagonal matrix is calculated iterative which is able to approximate eigenvalues of the system matrix just the smallest - and technically most interesting-. Nevertheless, the regulation of these approximated eigenvalues occurs not in every step, but is carried out in the interest of enhanced speed only in agreed step sizes. As soon as the eigenvalues do not change (almost) any more, the iteration is finished. The second phase contains the search for the eigenvectors and their transformation to the so-called form vectors. To every eigenvalue (defines the frequency) an accompanying vector is calculated by a single solution of the equation system.

For the control of the computing kernel the following five values are usable:

Number of frequency: Here it is fixed which number of the smallest natural frequencies should be determined. However, although one is interested often only in one or some frequency, the number should not be selected too small. Often so-called rigid body modes are for numerical reasons under the oscillations of the slightest frequency. The calculation time rises with bigger number even a little, because the iteration is done furthermore only once. Then merely the part of the eigenvalue approximation demands more time. A good default value is 15.

Number of iterations: Alike to the parameter MAXIT with iterative equation solvers a limit after which the iterative phase I of the Lanczos solver is stopped can be entered here. This serves primarily to be able to check with test calculations already after a short time whether the model is computable as desired. As usual with iterative processes, the results which were generated after the maximum iteration number should be taken with care. In these cases cannot be spoken yet of a convergence against the correct solution. The statement helps that at the latest after that iteration number which corresponds to the degree of freedom number of the model the maximum accuracy is reached.

Difference b. 2 frequency: Particularly with symmetrical components it often seems that two successive natural frequencies differ only very little. Then the oscillation forms are often turned around the symmetric axis and are absolutely the same otherwise. Besides, mathematically it concerns so-called multiples of the accompanying eigenvalues. To exclude this in phase I, a least difference (in Hz) can be stopped, from two frequencies should be also considered really as different. The value must be really greater than zero; quite small values like 1.0E-6 cause the desired results.

Kappa: This value determines after which fixed number of Lanczos iterations an eigenvalue approximation should be done. If for example, the default 50 is used, an approximation occurs only after 50, 100, 150, … iterations. All 49 intermediate steps are run in the interest of shorter arithmetic time without a complex check of the termination criterion, even if the convergence is already reached. An enlargement of the value leads at the same time to a stricter termination criterion, vice versa a small value weakens his influence.

The start of the calculation causes the call of the Solvers Z88EI which works with the following input and output files.

Input files:

- Z88I1.TXT general structure data
- Z88I2.TXT boundary conditions
- Z88I2EI.TXT the Lanczos solver parameters
- Z88MAT.TXT material defini. and one or more material files in .TXT format
- *.TXT material file
- Z88INT.TXT integration orders
- Z88MAN.TXT solver parameters

Output files:

- Z88O2.TXT Displacements for all frequencies

4.5 THE THERMAL SOLVER Z88TH

In the module Z88TH of Z88Aurora® a purely thermal conduction simulation as well as a thermo-mechanical simulation can be carried out. The first step to select the module and therefore the solver to the temperature calculation is to change the rider in the menu fillet from *linear mechanical* to *steady-state thermal*, see figure 39.

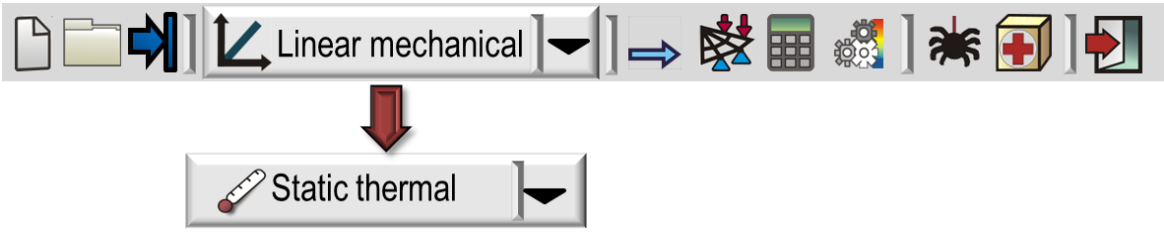


Figure 39: Choosing the steady-state thermal calculation mode

The first small difference to the linear mechanical calculation is the import of components. In general, it is possible to load a pure geometry structure by the known interfaces. However, with FE structures only the element types tetrahedron and hexahedron (linearly or squarely, either) can be imported without thermal boundary conditions. In the material data base must be added, for the thermo-analysis, the thermal conductivity and for the thermo-mechanical simulation, in further addition, the thermal expansion. Figure 40 shows exemplarily a material just available in the material data base where these values are already predefined.

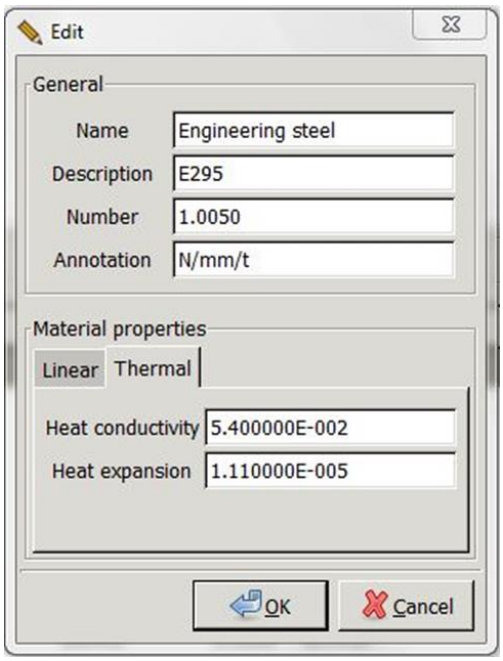


Figure 40: Material definitions for thermo-mechanical calculations

In the real boundary condition menu there is an addition (figure 41) for the thermal boundary conditions (as temperature, heat flux and heat flux density) which are marked. After defining the appropriate sets the boundary conditions may be assigned, and information of the direction is not necessary, because the thermal boundary conditions feature only one degree of freedom in the space.

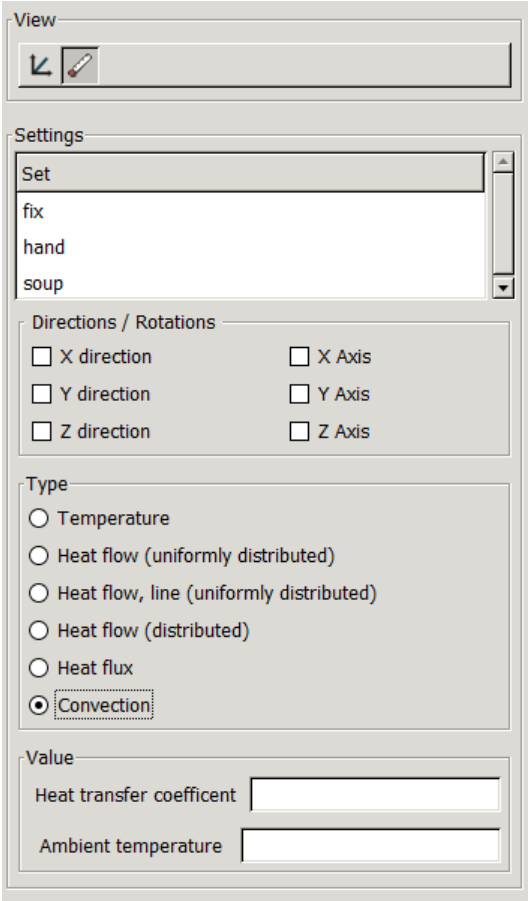


Figure 41: thermal boundary conditions

The user can enter the temperature either in Kelvin (K) or in ° Celsius ($°C$). This makes no difference for the calculation results. The heat flux density is a surface-related load, i.e. it defines the heat flux per surface (unit: W/mm^2). The heat flux (unit: W) as a nodal load can be applied either *uniformly distributed* or *distributed*. *Uniformly distributed* means here that the same entered value is assigned to every node. *Distributed* applies the loads according to FE rules. A new feature of Z88Aurora® V5 is the thermal boundary condition for modeling convective heat transfer. The convective heat transfer from a solid body to surrounding air can be modeled by defining the value of the convective heat transfer coefficient (unit: W/mm^2K). If one wants to carry out a thermo-mechanical simulation, one must define only new sets with additional mechanical boundary conditions (e.g., of a constraint). Then the Solver automatically recognises that the user would like to carry out such a simulation. As a calculation core can be selected between three equations solvers: PARDISO (directly and multi-core-running), SICCG and SORCG (iterative) (figure 42). The Cholesky-Solver cannot be chosen here.

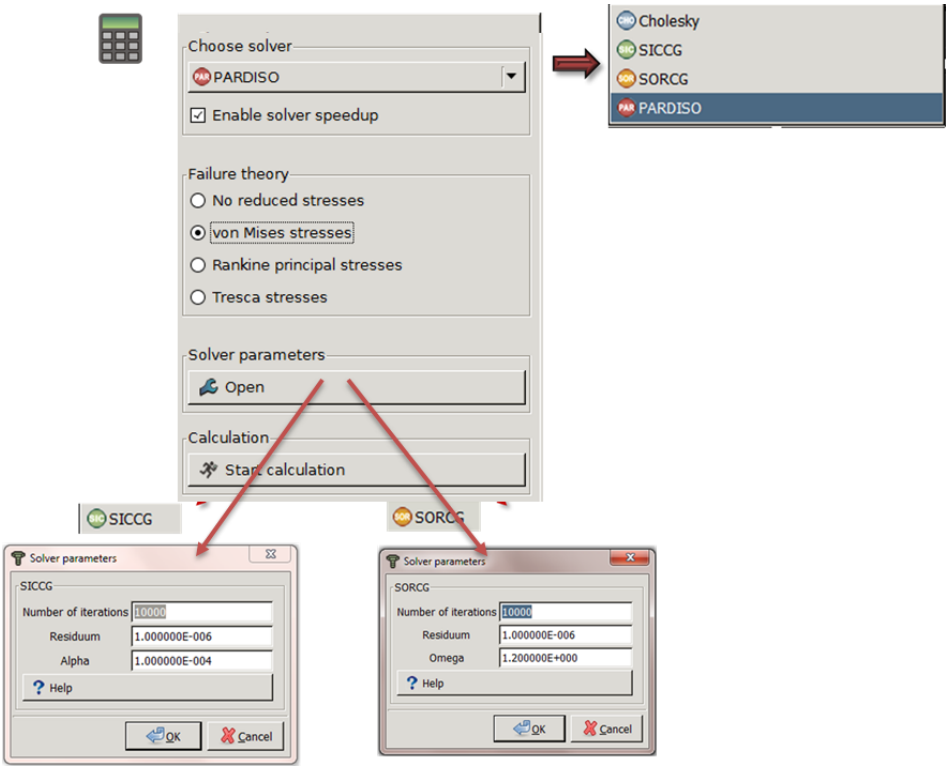


Figure 42: Choose the appropriate equation solver

In the post-processor the plots are extended by the thermal results temperature and heat flux as well as by the thermo-mechanical results thermal strains and thermal force.
For a thermal or thermo-mechanical calculation there are as usual also furthermore the input and issue data in the form of .txt files possible.

Input files:

- Z88I1.TXT general structure data
- Z88I2.TXT mechanical boundary conditions
- Z88I5.TXT mechanical surface loads or 0 in the first line
- Z88TI2.TXT thermal boundary conditions (temperature, heat flux)
- Z88TI5.TXT thermal surface loads (heat flux density) or 0 in the first line
- Z88TI5C.TXT convective boundary condition
- Z88MAT.TXT material defini. and one or more material files in .TXT format
- *.TXT material file
- Z88INT.TXT integration orders
- Z88MAN_TH.TXT solver parameters for Z88TH

Output files:

- Z88TH.LOG hints, errors, and warnings of Z88TH
- Z88AG2THI.LOG hints, errors, and warnings of the Z88TH converter
- Z88TH.DYN memory parameters (automatically computed)
- Z88TO0.TXT temperature results
- Z88TO1.TXT heat flux results
- Z88TO2.TXT thermal strains results
- Z88TO3.TXT thermal forces results
- Z88TO4.TXT displacements results
- Z88TO6.TXT forces results
- Z88TO7.TXT stresses results

As you may see, the files of the structure and the mechanical boundary conditions are identically to those of the linear mechanical analysis. In the thermal solver file Z88MAN_TH.TXT there are on the other hand beside the settings for the iterative solver two new Flags. The first Flag `THERMOMODE` must always be `1`. The second Flag `THERMOMECHANIC` must be `0` for a purely thermal simulation, and for a thermo-mechanical simulation `1`.

```
TMSOLVER START
  MAXIT                    10000
  EPS                      1.000000E-006
  RALPHA                   1.000000E-004
  ROMEGA                   1.200000E+000
  THERMOMODE               1
  THERMOMECHANIC           1
TMSOLVER END
```

4.6 THE NONLINEAR SOLVER Z88NL

The module Z88NL represents an equation solver which is designed for nonlinear calculation. Z88NL can handle large displacement effects (geometrical non-linearity) or material non-linearity (plasticity).

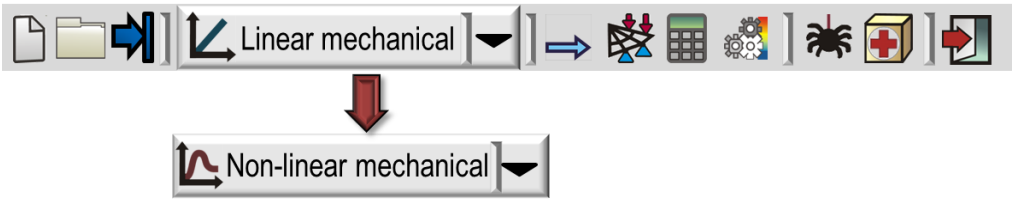


Figure 43: Choosing the non-linear solver

Figure 43 shows how Z88Aurora® is switched to non-linear calculations. For the import and the pre-processing, no differences arise in comparison to the linear mechanical calculation, except that merely the elements 1, 4, 7, 8, 10, 16, 17 can be used. In the menu, cf. figure 44, the settings can be adjusted in three different tabs for the non-linear solution process (tab Procedure), the linear sub-equation solver (tab Solver), and the result issue (tab Output). A reduced stress calculation by Z88NL is only for *von Mises* stresses possible.

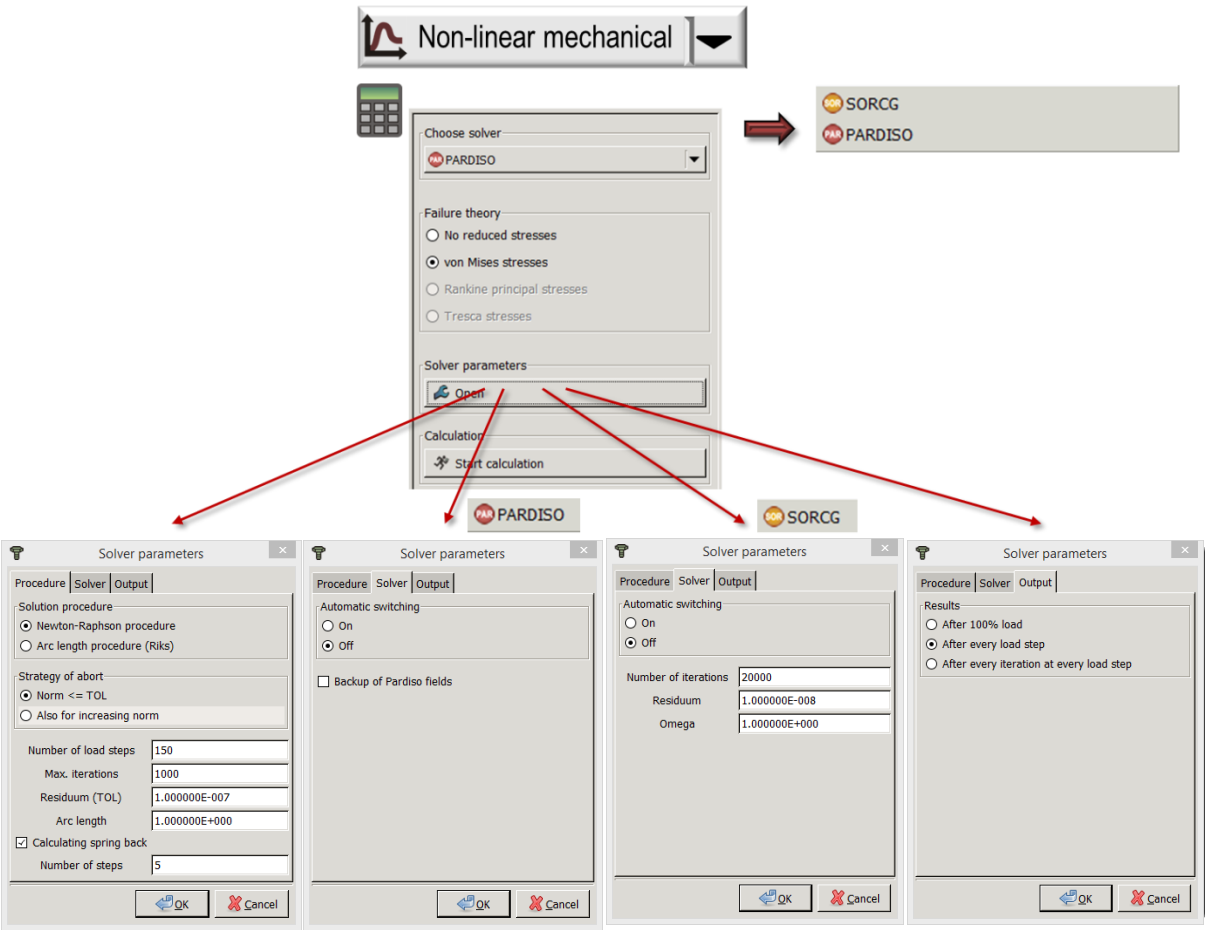


Figure 44: Adjusting the non-linear solver Z88NL

The start of the calculation causes the call of the equation solver Z88NL which works with the following input files and output files.

Input files:

- Z88I1.TXT general structure data
- Z88I2.TXT mechanical boundary conditions
- Z88I5.TXT mechanical surface loads or 0 in the first line
- Z88NLI7.TXT definition of history variables or 0 in the first line
- Z88MAT.TXT material definition and one or more material files in .TXT format
- Z88ELP.TXT element parameters
- Z88INT.TXT integration orders
- Z88MAN.TXT solver parameters for Z88NL

Output files:

- Z88NL.LOG hints, warnings, and errors of Z88NL
- Z88NLKV.LOG convergence of Z88NL
- Z88NL.DYN memory parameters (automatically computed)
- Z88NLO2.TXT displacements
- Z88NLO3.TXT stresses (Cauchy-stresses if chosen)
- Z88NLOH.TXT output of history variables

All input files except the files Z88NLI7.TXT and Z88MAN.TXT are named identically with the input files of Z88R. But the parameters file Z88MAN.TXT of the solver contains an additional section which would be briefly described in the following. The new section becomes limited by the key words NONLINEAR START and NONLINEAR END:

```
NONLINEAR START
  NLFLAG                  1
  NLAERH                 25
  MAXNLIT              1000
  EXIT                    1
  TOL                  1E-7
  AUTOGAUSS               0
  OUTPUTFLAG              1
  OUT_CAUCHY              1
  OUT_INT9OFFS            1
  PARSP                   1
  ELEFORM                 1
  BGLAENG               1.0
  LASTST                  0
NONLINEAR END
```

The parametres are adjusted by the settings in the solver menu. They have the following meanings.

- 1st value: Solution processes: NLFLAG: Newton-Raphson method (1) or arc length method according to Riks (2).
- 2nd value: Number of the load steps: NLAERH: says in how many steps the whole load is applied.
- 3rd value: maximum iteration number: MAXNLIT: says in how many steps the non-linear solution process is carried out at most.

- 4th value: Control of the termination criterion: EXIT: termination of the non-linear solution process only if the norm is smaller than the termination criterion TOL(1) or termination even if the norm increases (2).
- 5th value: Termination criterion or residuum TOL: The norm must be smaller than TOL so that the solution is found.
- 6th value: Automatic solver switching: AUTOGAUSS: If the Flag is activated (1), an automatic switching of the solver occurs with very small structures, if it is deactivated (0), no switching occurs.
- 7th value: Output control: OUTPUTFLAG: Controls the output of the results: Output only at the end of the solution process for the whole load (0), output after every load step (1) or output after every iteration step of every load step (2).
- 8th value: Stress calculation: OUT_CAUCHY: Stress calculation: is carried out (1) or is not carried out (0).
- 9th value: Memory management for integration point sizes with 9 values: OUT_INT9OFFS: activates the supply of a special memory field which is necessary, for example, for the stress calculation (with activated stress calculation this flag must be also activated), active (1) or inactive (0).
- 10th value: Memory management: PARSP: If the flag is activated (1), additional memory is allocated for the backup of the sparse pointers IP and IEZ to speed up the calculation process, if it is deactivated (0), an iterative new calculation occurs.
- 11th value: ELEFORM determines the used element formulation. Default is 1 for the Lagrangian element formulation that detects geometrical nonlinearities. The Euler element formulation, also suitable for geometric nonlinearities is used with 2. For material nonlinearities ELEFORM must be set to 3. Then the multi-field formulation of Simo and Hughes is applied. Usually, this flag does not have to be set manually because Z88Aurora® or the converter AG2NL automatically sets the flag in dependence of the constitutive model.
- 12th value: Arc length: BGLAENG: defines the arc length needed for the method of Riks.
- 13th value: LASTST determines either to use equidistant load steps (without spring back) (set LASTST to 0) or to activate the load control over the file Z88NLI8.TXT (set LASTST to 1). The load control is automatically activated if the spring back should be calculated.

4.7 THE CONTACT MODULE Z88KONTAKT

The contact module Z88KONTAKT can simulate the interaction between parts (small displacements and no rigid body movements). A conforming mesh in the contact area is not needed, so the only requirement is that both parts are of the same element type. Contact analysis is available for tetrahedrons and hexahedrons with linear and quadratic shape functions (element types 1, 10, 16, 17).

The contact module has to be activated at the start of a new project because the definition of an assembly is not possible at a later point.

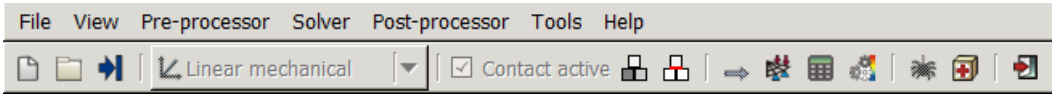


Figure 45: Control of the contact module

Instead of the standard import the part manager has to be used which allows the import and positioning of various parts. The imported parts already have to be meshed, since only Z88 structure files (z88structure.txt, z88i1.txt) can be added to an assembly. The parts each have to be meshed beforehand in a separate project folder.

Within the contact module each part has to be assigned a role within the assembly (master, slave). During calculation contact areas are located on the respective surfaces of the master and slave parts. Thus, an interaction is only possible between a master and a slave part. Therefore, contact between surfaces of a single part or contact between parts of the same role cannot be realized. The following profile fits a standard master part:

- Low degree of discretization
- Higher order of shape function
- More rigid body
- Larger contact area
- Lesser curvature of contact area

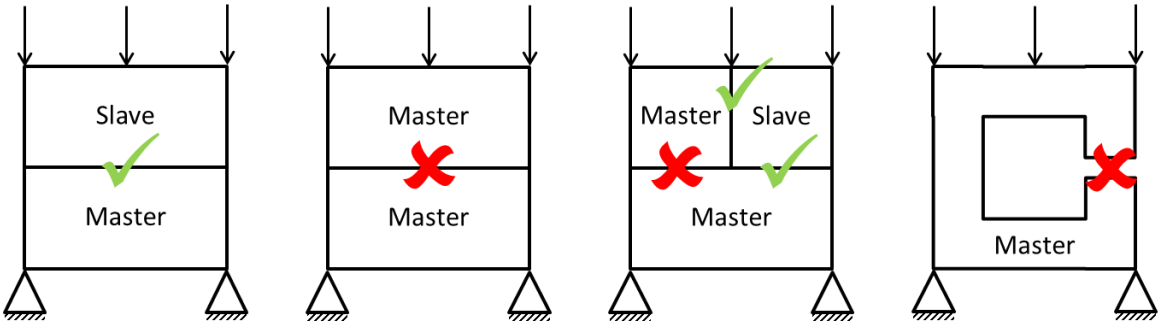


Figure 46: Possible contact types

In addition to the part manager a contact control menu was introduced. This menu included the following options:

- Contact type: Describes the type of interaction between the located contact areas
    - Bonded: All normal and tangential contact forces are propagated in the contact area. A separation of the parts is not possible.

- Frictionless: Only normal contact forces can be propagated in the contact area. The number of load steps defines the subdivision of the applied force and therefore defines the number of iterations for finding the equilibrium (typical values between 5 and 15). The additional option “Allow separation” defines whether the parts can separate under tensile stress. Compression stress can always be propagated in the contact area regardless of this choice.

  Note: The assemblie’s parts have to be fixed in all directions either by displacement boundary conditions or by neighboring contacts.

- Contact discretization: Describes the contact realization.

  - Node-Surface: Nodes of one part are in contact with the supporting points of the other part.

  - Surface-Surface: Supporting points on the surface of one part are in contact with supporting points on the surface of the other part. This leads to a detailled depiction of the contact zone.

  - Separation distance: Defines the tolerance for the contact search. Areas of two parts, whose euclidian distance is below this value are considered to be in contact.

- Imposition method: Describes the mathematical method used to incorporate the contact conditions into the equation system.

  - Perturbed Lagrange: The global stiffness matrix is augmented by the contact conditions, which is why a new compilation is not necessary. This enlarges the equation system. The reciprocal of the chosen penalty factor (contact stiffness) is put on the diagonal field of the added equations.

  - Lagrange: This corresponds to the perturbed Lagrange method with a penalty factor of 0.
    Note: This method is numerically unstable.

  - Penalty: Contact definitions are added to the global stiffness matrix so that the equation system is not enlarged. A too low penalty factor (contact stiffness) leads to a violation of the non-penetration condition. A too high value leads to a badly conditioned equation system.

- Contact stiffness: Describes the value of the penalty factor for the perturbed Lagrange method and penalty method.

  - Normal/Tangential: Define the value of the penalty factor in normal and tangential direction of the contact zone. Typically equal values are chosen for both directions when a fixed contact is used. In case of frictionsless contact the tangential direction is omitted since only forces in normal direction are propagated.

- Absolute: The input is used directly for the chosen imposition method. Typical values range from 1.0E+005 to 1.0E+012. This is extremely problem-specific.

- Relative: The input is multiplied by the highest value on the diagonal of the global stiffness matrix. The resulting value is then used for the chosen imposition method. Typical values range from 1.0E+003 to 1.0E+005. This is extremely problem-specific.

The remaining preprocessing (boundary conditions, material definitions etc.) along with the postprocessing is unchanged compared to a standard linear FE calculation without contact.

The following input and output files are used when starting a calculation within the contact module:

Input files:

- Z88I1.TXT general structure data
- Z88I1P.TXT assignment of nodes/elements to the different parts
- Z88I2.TXT boundary conditions without shear loads or pressure
- Z88I5.TXT shear loads, pressure or 0 in the first line
- Z88MAT.TXT material control data and one or more *.TXT material files
- *.TXT material file
- Z88ELP.TXT element parameters (0 in the first line for contact)
- Z88INT.TXT integration order
- Z88MAN.TXT solver parameters
- Z88KON.CTRL contact control of output files and imposition method
- Z88KON.PROP contact properties

Outpu file:

- Z88O0.TXT processed structure data
- Z88O1.TXT processed boundary conditions
- Z88O2.TXT calculated displacements
- Z88O3.TXT calculated stresses
- Z88O4.TXT calculated node forces
- Z88O5.TXT equivalent stresses at Gauss points
- Z88O8.TXT equivalent stresses in corner nodes
- Z88O14.TXT master and slave contact nodes

4.8 THE MAPPED MESHERS

The mesh generator Z88N from Z88 is integrated into Z88Aurora® with enhanced functionalities:

- Z88N for hexahedrons, axisymmetric elements, plane stress elements, plates and volume shells
- The Tetrahedron refiner for tetrahedrons
- The Shell thickener for flat shells
- STL-refiner for imported STL-meshes (4.1.5)

Access of the meshers in the pre-processor menu via the icon Super elements.

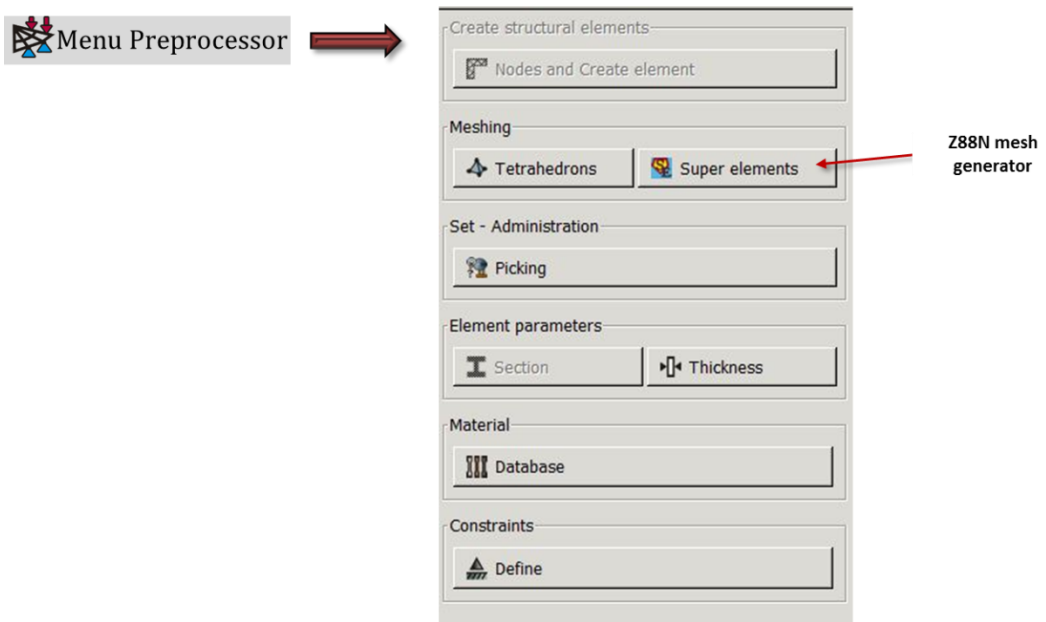


Figure 45: Menu "pre-processor" with start icon "Super Elements" of the mesh generator Z88N

4.8.2 Z88N FOR 2D AND 3D ELEMENTS

A mesh generation is only usefull and permitted for continuum elements. An overview of the possible finite element structures can be found in Table 7.

Table 7: Possible super structures in Z88Aurora®

Superstructure	**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

Mixed structures e.g. containing Plane Stress Elements No. 7 and Trusses No. 9, cannot be processed.

Mode of operation of the mesh generator:

For generating FE meshes proceed as follows: The continuum is described by so-called super elements (short SE), which practically corresponds to a quite rough FE structure.
The super structure is then subdivided. This is done super element wise, starting with SE 1, SE 2 up to the last SE. SE 1 produces the finite elements (short FE) 1 to j, SE 2 the FE j+1 to k, SE 3 the FE k+1 to m and so on. Within the SE, the direction of the local coordinates determines the nodal numbers and the element numbers of the FE structure. Definition:

- Local x axis points in direction of local nodes 1 and 2
- Local y axis points in direction of local nodes 1 and 4
- Local z axis points in direction of local nodes 1 and 5

Super structures in space are subdivided first in z, then in y and finally in x direction i. e. the FE element numbers start along the z direction. To plane and axially symmetric structures applies analogously: The numbering starts along the y axis or for axially symmetric elements along the z axis (cylinder coordinates!).
Along the local axes, a subdivision can be conducted as follows:

- equidistant
- increasing geometrically from node 1 to 4 or 5 : mesh becomes rougher
- decreasing geometrically from node 1 to 4 or 5 : mesh becomes finer

It is obvious, that for lines or areas, which are shared by two super elements, the super elements must be subdivided exactly the same! The mesh generator does not check this and then generates useless or totally meaningless FE meshes. Example:

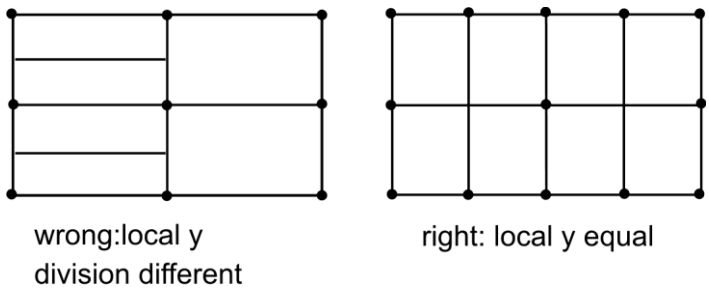


Figure 46: Subdivision of the super elements

Since the local axes x, y and z are defined by the location of the local nodes 1, 4 and 5, it is possible to generate almost arbitrary numberings for nodes and elements of the FE structure by corresponding construction of the coincidence list in the mesh generator input file Z88NI.TXT.
See figure 46 for an example for the generation of a FE structure with 8 FE Plane Stress Elements No. 7 from a super structure with 2 Plane Stress Elements No. 7 (looks the same with axisymmetric elements No. 8).

Specials:

The mesh generator checks which nodes are already known at the production of new FE nodes. For this check it needs a trap radius (a computer cannot meet a floating point number exactly). This trap radius is provided for all 3 axes per default 0.01. Modify the trap radiuses when processing very small or very large numerical values.

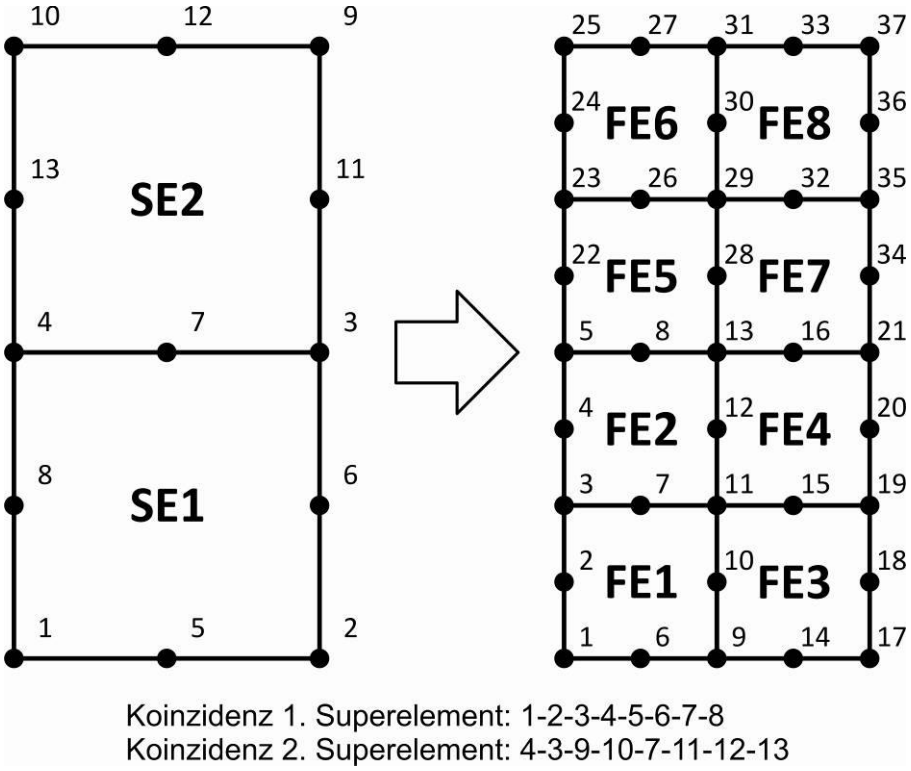


Figure 47: Transformation of super elements into finite elements

Attention mesh generator Z88N: The generator can generate input files easily which blast all limits of the FE solver. Generate therefore at first rougher FE structures, check the results then refine if necessary. A good starting point: Produce approx. 5 ~ 10 times more finite elements than super elements.

Note mesh generator Z88N: If the coordinate flag KFLAGSS is set in the mesh generator input files Z88NI.TXT, i.e. input values are polar or cylindrical coordinates, then the mesh generator output files Z88I1.TXT normally have Cartesian coordinates and KFLAG is set to 0. If you set the coordinate flag output KFLAG to 1, however, then the coordinates are polar or cylindrical in the output file Z88I1.TXT and KFLAG is set to 1 in Z88I1.TXT.

4.8.3 THE TETRAHEDRON REFINER

With this functionality it is possible to refine existing tetrahedron meshes. By means of picking a set with those tetrahedrons which should be refined. The division of every element occurs through 8 tetrahedrons.

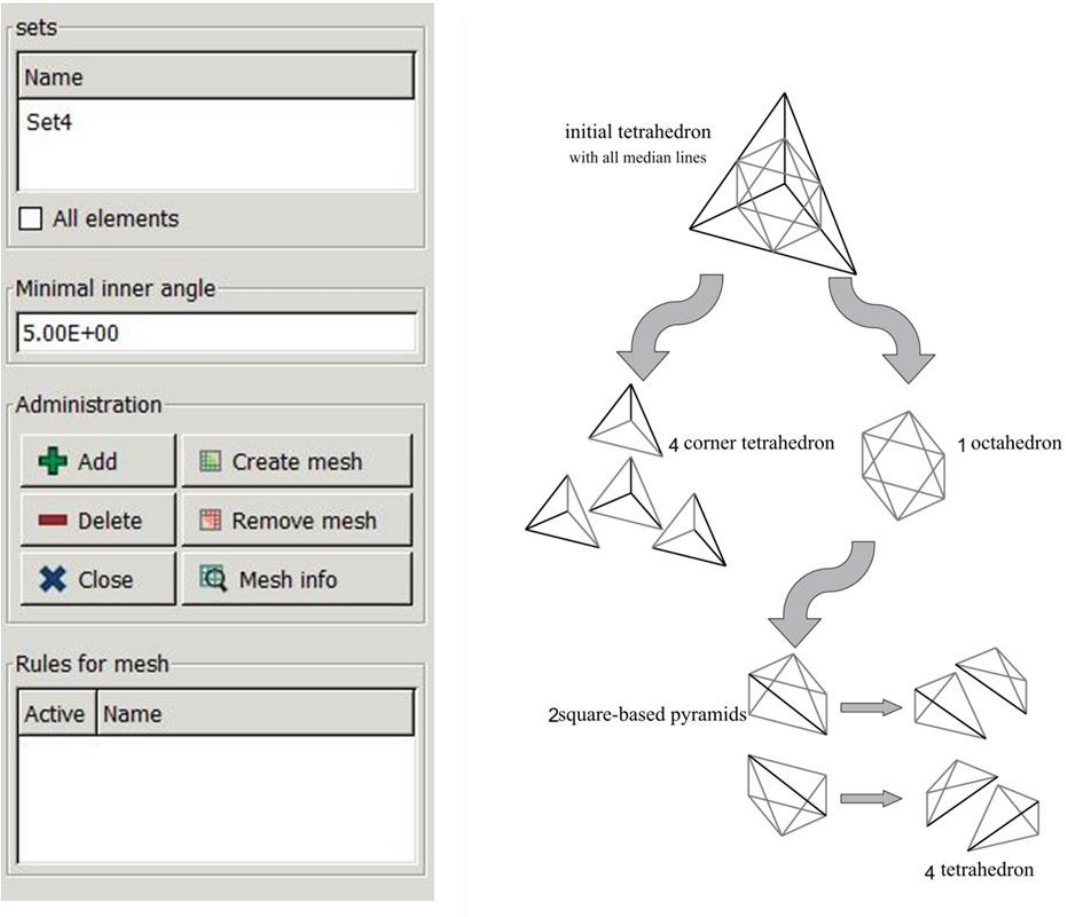


Figure 48: Entry field for tetrahedron refiner (left), method of tetrahedron refining (right)

Afterwards the adjoining elements are adapted to the changed nodal numbers and are also divided. On this occasion, a minimum element angle has to be entered to prevent of a too strong distortion.

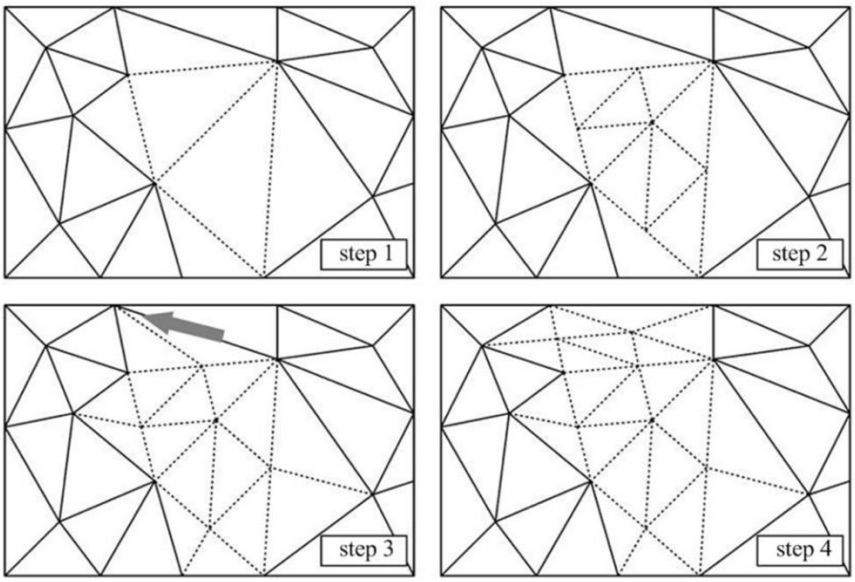


Figure 49: Course of the refinement algorithm with angle criterion

Instead of the ideal internal angle of 60° a default of the angle is realistic by a FE meshing between 3-10°. Further details to the application, see user's handbook.

4.8.4 THE SHELL THICKENER

With this functionality it is possible to thicken existing flat shells from Nastran or DXF import and, thus, to convert to volume shells (element No. 21 and element No. 22). Further details to the application see user's handbook.

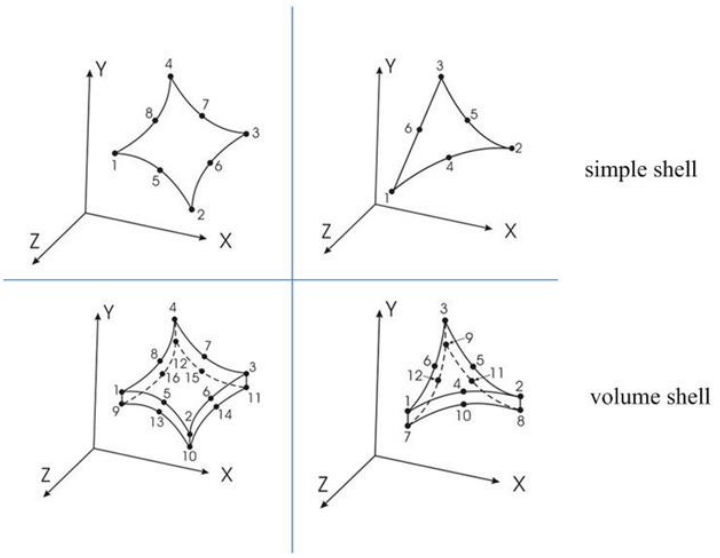


Figure 50: Flat shells (upper) and volume shells (lower)

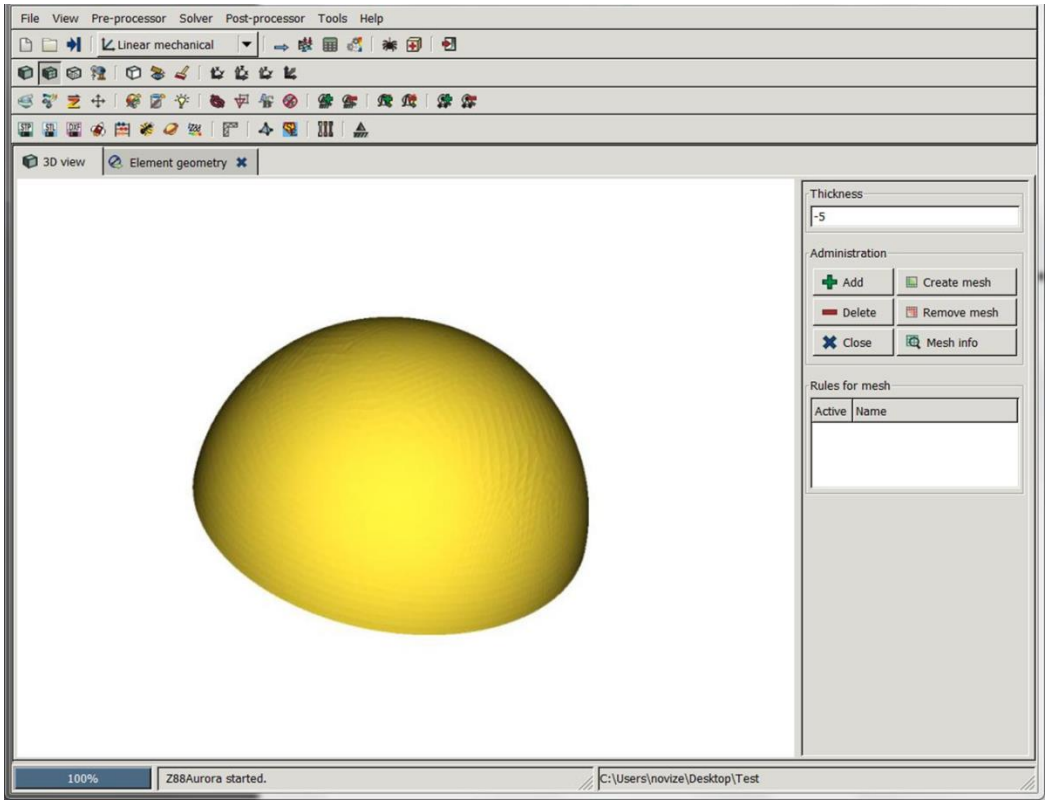


Figure 51: Shell thickener in Z88Aurora®

4.9 THE POST-PROCESSOR

Structures illuminated with three light sourced, wireframe- or hidden line-structures can be plotted undeflected, deflected or both of them overlaying. In the same way, a colour range for stresses and X, Y, and Z deflections can be displayed. In case of node and element numbers areas can be specified, which is very helpful in case of large structures. A plotter or printer output is not explicitly intended. Why should it be? Simply make a screenshot with *Shift-Print* in the clipboard and edit or print it with the Windows program *Paint* or a paint program such as, for example *CorelPaint* etc.

Z88Aurora® uses OpenGL. Therefore, your computer must be able to deal with OpenGL. In case of the more recent Windows versions this is activated by default, and usually a cheap graphic adapter will be sufficient. To be on the safe side, however, check the system settings – sometimes OpenGL hardware acceleration can be activated.

Your choice of colours, screen size, light features material properties, the polygon offset etc. can be defined in the file Z88.FCD. But be careful with changes in Z88.FCD: You must have some basic knowledge of how OpenGL works, if you want to change light effects etc. Otherwise, there will be long faces, because nothing seems to work properly anymore. Some hints are included in Z88.FCD in the form of remarks, but we cannot give an introduction to OpenGL in this context. Please consult, for example, *Shreiner, D: OpenGL Programming Guide. 7th edition. Addison-Wesley/Pearson: 2010.*

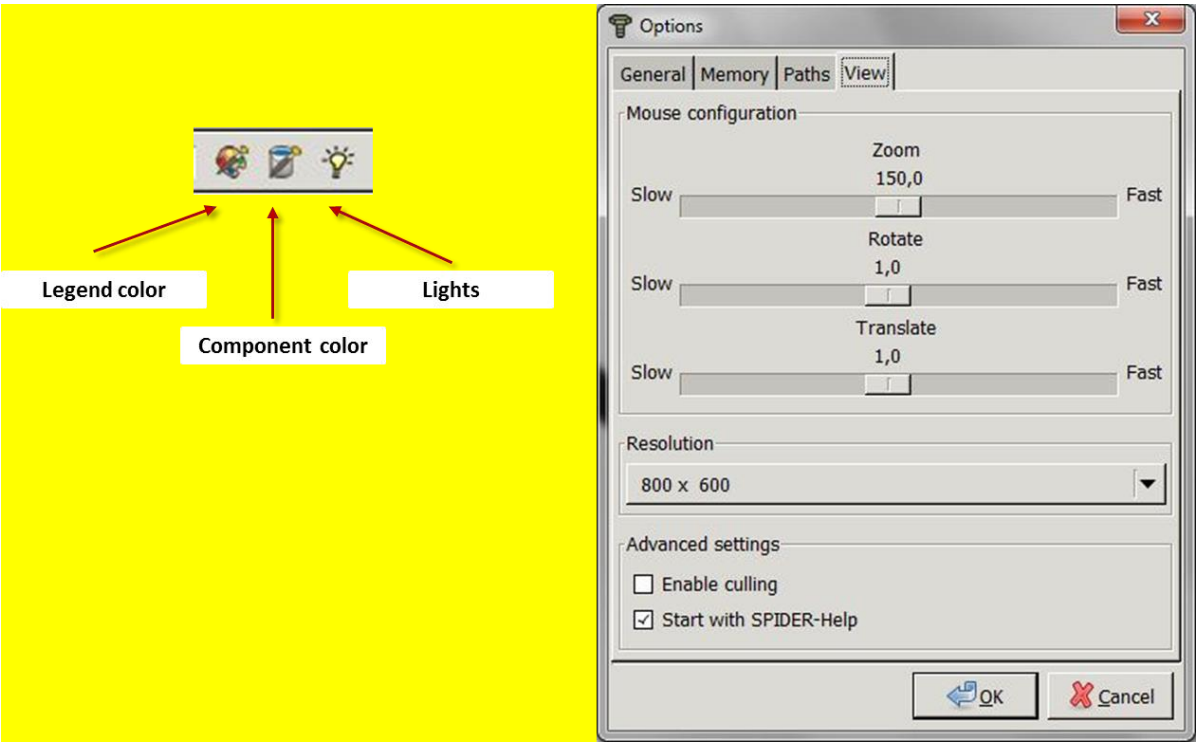


Figure 52: Option menu „View“ and the choice of lights, labels and colors by the appropriate icons

Features of rendering: For fastest operation Z88Aurora® connects the nodal points in case of scenes with lighting and in hidden line mode – and only the corner points – with straight lines, although for Serendipity elements the edges of the elements are square or cubic curves; in wireframe mode all nodes are connected with straight lines. Especially illuminated scenes need a huge amount of computational power. If a part renders pretty fast in your CAD system, Pro/ENGINEER for example, and the same part renders quite slowly in Z88O - this is normal business because CAD systems are „drawing“ only some outline curves. In contrast, FEA system have to render *every* finite element i.e. compute the normal vectors for any element surface, compute light effects for every tetrahedron etc. Hidden line scenes put very heavy load on the CPU. This can be solved by applying "*Surface/Solid View*", and *“Quick View”* which can be found in the view menu. Here, only the outer solid edges are calculated and completely displayed, but this procedure is not suitable for all functionalities.

What can I plot? Nearly everything if a solver was run which stored the deflection file Z88O2.TXT and the three stress files Z88O3.TXT (for you to check the stresses), Z88O5.TXT (for Z88O internally) and Z88O8.TXT (for Z88Aurora® internally). Even for trusses you may plot the "von Mises“ stresses (i.e. tensile stresses) with different colours; only beams No. 2 and No. 13 and cams No. 5 allow only the plotting of deflections and nothing more. Why? Because you must compute for beams and cams also the stress concentration factor which is impossible for a FEA system which deals with a *whole structure of beams*. Of course, you may compute the stresses in a chamfer by putting a FE mesh around it. But this needs either plane stress elements or 3D elements but neither beam elements nor cam elements.

Plot of stresses: The kind of plotting the stresses within FEA programs is truly of philosophical character. As a matter of fact, numerous experiments and computer studies at the *Institute of Engineering Design and CAD* of the University of Bayreuth, Germany, showed, that some very expensive and well-known professional FEA programs produced *incorrect stress plots* in some situations! The best way is the computation of stresses directly in the Gauss points. However, this is odd for OpenGL in some modes so I decided for the following way after a lot of experiments:

1. *von Mises/principle/Tresca stresses in corner nodes.* In fact, the stresses are computed not really in the corner nodes, which would lead to very wrong results especially for very tapered elements (sic!), but in Gauss points lying near the current corner nodes. Stresses are computed for just the same number of Gauss points like the number of corner points. Because often a node is linked to more than one element the stresses are computed to a mean value from the "corner node“ stresses of all linked elements. This results in pretty balanced stress shadings which are mostly somewhat lower than the maximum stresses in the Gauss points, however. The value of the order of integration INTOS in the header file Z88ENVIRO.DYN has no meaning but INTOS should be greater than 0.
2. *von Mises/principle/Tresca stresses as a mean value for each element.* The stresses are computed in the Gauss points of the current element, added and then divided by the current number of Gauss points. This results in a mean value for the *von Mises/principle/Tresca stress* per element. The value of the order of integration INTOS in the header file Z88MAN.TXT is important and INTOS must be greater than 0.
3. *von Mises/principle/Tresca stresses directly in Gauss points.* This is most accurate but does not deliver as pretty pictures as 1. and 2. INTOS must be greater than 0.

Z88Aurora® may show the following reduced stresses – but only one at a time - depending on the previous computation run:

- *von Mises* stresses
- Rankine i.e. *principal* stresses
- *Tresca* stresses

Thus, if you have computed the *von Mises* stresses previously, Z88Aurora® will show them. If you want to show the *Tresca* stresses now, you have to run the solver again, in this case with the setting *Tresca*, see *Figure* . This looks awkward but do not you know before starting the FE computations which type of stresses is suitable and correct for your task? The choice of the appropriate reduced stresses hypothesis is not a matter of trial-and-error.

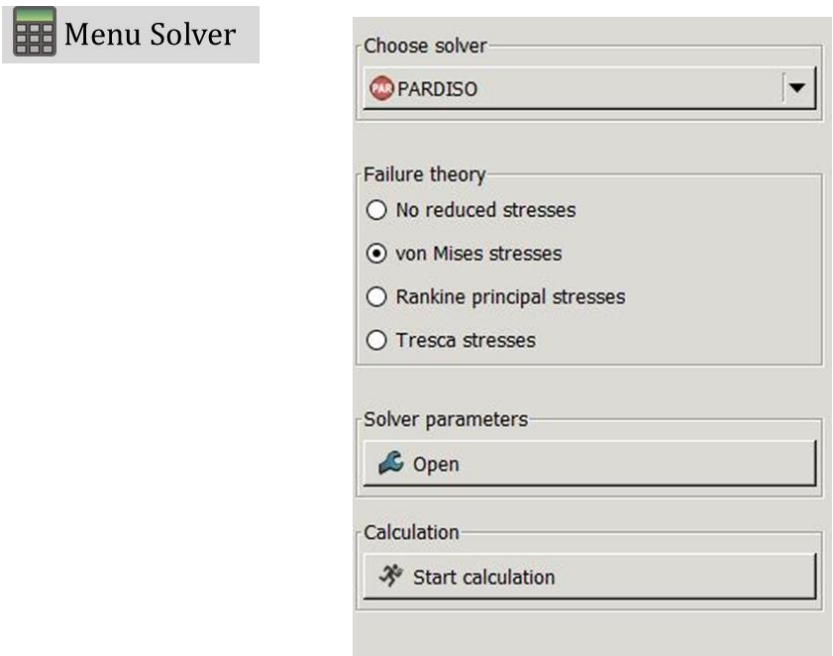


Figure 53: Setting options stress parameters in the menu "Solver"

Plot of deflections: You may plot the undeflected or the deflected structure or both of them overlaying. The enlargement factor is adjustable, normally the factor amounts to 10% of the biggest displacement amount. In addition, you may plot the deflections for X, for Y or for Z with colour shading. This is a pretty nice feature for large spatial structures. You may plot the shaded colours for stresses or for the deflections or the hidden line display or the wire frame display with the deflected structure. The background colours and legend display can be adjusted. For further information consult the Z88Aurora® User Manual. By means of a scrollbar the deflection can also be scaled continuously.

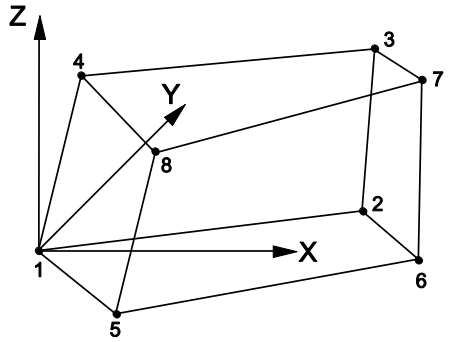
The „coordinate system“: OpenGL works with a *Clipping Volume*, i.e. with a kind of cube, defined by *Xmin* and *Xmax* in horizontal direction, by *Ymin* and *Ymax* in vertical direction and *Zmin* (points towards the user) and *Zmax* (points away from the user). If you use a too-large zoom factor or if you are panning the structure too near to you then the range of *Zmin* is exceeded and parts of the structure are lying outside the viewing volume. This offers a nice chance to look into a structure – also in order to see the internal stresses! Otherwise, change the value of Z*min* (default entry is –100) to lower values, e.g. -1000: use the menu "View" > *Z limit towards you*.

For further information regarding the application and options of post-processing, please consult the Z88Aurora® User Manual.

5. DESCRIPTION OF THE FINITE ELEMENTS

5.1 HEXAHEDRON NO. 1 WITH 8 NODES

The hexahedron element calculates deflections and stresses in space using linear shape functions. It is a transformed element; therefore it may be wedge-shaped or may have another oblique-angled form. The transformation is isoparametric. The integration is carried out numerically in all three axes according to Gauss-Legendre. Thus, the integration order can be selected in Z88ENVIRO.DYN. The order 2 is mostly sufficient. Hexahedron No. 1 is also well usable as a thick plate element, if the plate's thickness is not too small against the other dimensions. Hexahedrons No. 1 can be generated by the mesh generator Z88N from super elements Hexahedrons No. 10 and Hexahedrons No. 1.



Input:
CAD (see chapter 4.1.4**):**
Upper plane: 1 - 2 - 3 - 4 - 1, quit LINE function
Lower plane: 5 - 6 -7 - 8 - 5, quit LINE function
1 - 5, quit LINE function
2 - 6, quit LINE function
3 - 7, quit LINE function
4 - 8, quit LINE function

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or cylindrical coordinates (1)*
> *3 degrees of freedom for each node*
> *Element type is 1*
> *8 nodes per element*

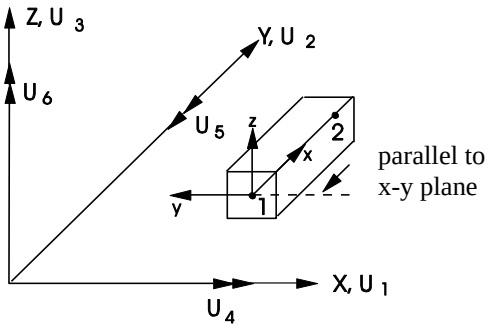
Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation: 2 is usually good.*
> *Integration order INTOS for stress calculation:*
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

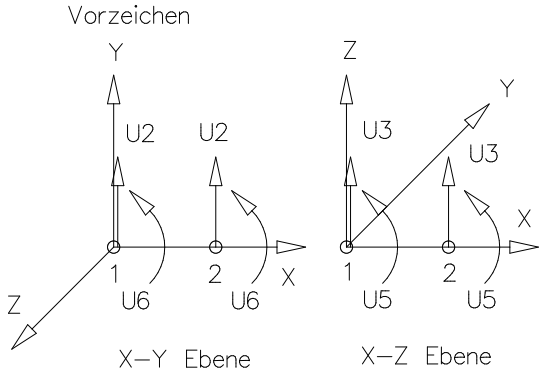
Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no influence
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)

3 = Tresca stresses in the Gauss points (INTOS not 0!)
Results:
Displacements in X, Y and Z
Stresses: SIGXX, SIGYY, SIGZZ, TAUXY, TAUYZ, TAUZX, respectively for corner nodes or Gauss points. Optional von Mises or principal or Tresca stresses.
Nodal forces in X, Y and Z for each element and each node.

5.2 BEAM NO. 2 WITH 2 NODES IN SPACE

Beam element with any symmetric profile (no slanting bend) with the restriction that the local y-y axis must be parallel to the global X-Y coordinate system. The profile values are provided in the GUI. Thus, you can use any symmetric profile in contrast to other FEA programs which incorporate a variety of different special beam and profile subroutines without matching all symmetric profiles as necessary. The element matches exactly Bernoulli's bend theory and Hooke's law. It uses no approximate solution as for the continuum elements.

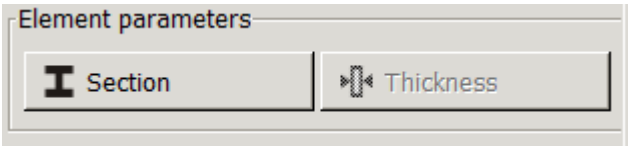




Input:

CAD (see chapter 4.1.4):
Line from node 1 to node 2

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or cylindrical coordinates (1)*
> *6 degrees of freedom in a node (Attention: DoF5 (not right hand rule), see below)*
> *Element type is 2*
> *2 nodes per element*



enter here the parameters for the elements

> *Cross-sectional area QPARA*
> *Second moment of inertia I_{yy} (bending around y-y axis)*
> *Max. distance e_{yy} from neutral axis y-y*
> *Second moment of inertia I_{zz} (bending around z-z axis)*
> *Max. distance e_{zz} from neutral axis z-z*
> *Second moment of area (torsion) I_T*
> *Second modulus (torsion) W_T*

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG* has no meaning

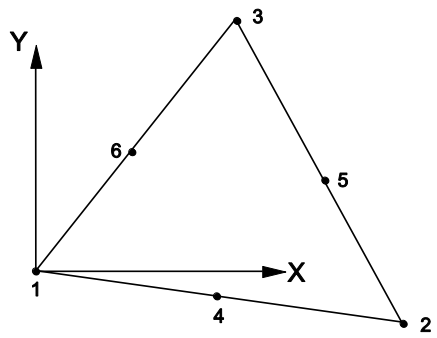
Results:

Displacements in X, Y and Z and rotations around X, Y and Z. Attention DoF5 (not right hand rule), see below
Stresses: SIGXX, TAUXX: Direct stress, shear stress, SIGZZ1, SIGZZ2: Bending stress around z-z for node 1 and node 2, SIGYY1 SIGYY2: Bending stress around y-y for node 1 and node 2
Nodal forces in X, Y and Z and nodal moments around X, Y and Z for each element and each node.

5.3 PLANE STRESS TRIANGLE NO. 3 WITH 6 NODES

This is a simple, triangular plane stress element with complete quadratic shape functions. This element is obsolete and kept in Z88 only for studies. Elements No. 7, 11 or 14 are much better. Pay attention to edge loads, cf. chapter 3.4. No entries into the surface and pressure loads file Z88I5.TXT!

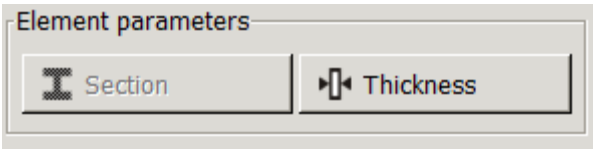
.



Input:

CAD (see chapter 4.1.4): *1-4-2-5-3-6-1*

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or polar coordinates (1)*
> *2 degrees of freedom for each node*
> *Element type is 3*
> *6 nodes per element*



enter here the thickness of the elements

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* = 0: Calculation of SIGXX, SIGYY and TAUXY
> *Radial/Tangential stress flag KDFLAG* = 1: Additional calculation of SIGRR, SIGTT and TAURT
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the center of gravity
2 = principal or Rankine stresses in the center of gravity
3 = Tresca stresses in the center of gravity

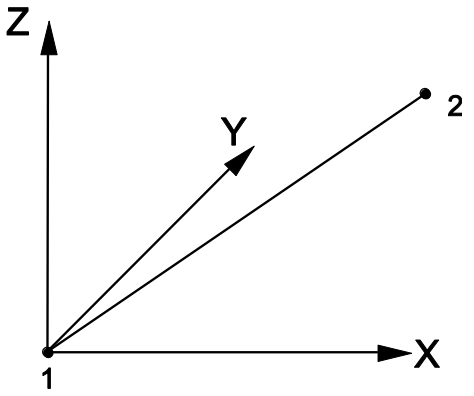
Results:

Displacements into X and Y
Stresses: The stresses are calculated in the element's centre of gravity. The coordinates of the

centre of gravity are thus printed. For KDFLAG = 1 the radial stresses SIGRR, the tangential stresses SIGTT and the accompanying shear stresses SIGRT are computed additionally (makes only sense if a rotational-symmetric structure is available). For easier orientation the respective radiuses and angles of the centre of gravity are printed. Optional von Mises stresses in the center of gravity.
Nodal forces in X and Y for each element and each node.

5.4 TRUSS NO. 4 IN SPACE

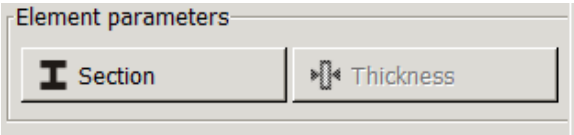
The truss element No. 4 can take any location in space. It is part of the simplest elements in Z88 and is calculated extremely fast. The truss elements match Hooke's law exactly. Hint: Trusses No. 4 are very suitable for modelling spring supports or oblique-angled supports.



Input:

CAD (see chapter 4.1.4): *Line from node 1 to node 2*

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or cylindrical coordinates (1)*
> *3 degrees of freedom for each node*
> *Element type is 4*
> *2 nodes per element*



enter here the cross-section area for the elements

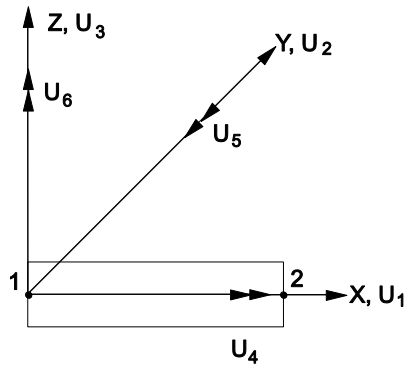
Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG* has no meaning

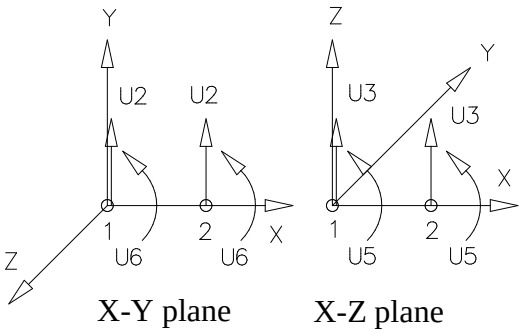
Results:
Displacements in X, Y and Z
Stresses: Normal stresses
Nodal forces in X, Y and Z for each element and each node.

5.5 SHAFT ELEMENT NO. 5 WITH 2 NODES

The shaft element is a simplification of the general beam element No. 2: It has always a circular cross-cut. The element lies concentrically to the X axis, consequently, local and global coordinates have the same direction. Thus, inputs and calculations are simplified strongly. Like the beam element, the results are exact according to Bernoulli's bend theory and Hooke's law, and not approximate solutions like the continuum elements.



Algebraic sign

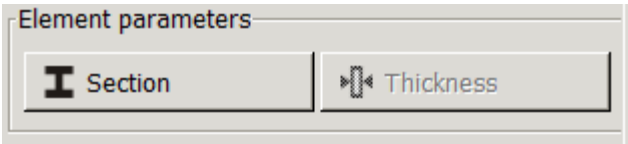


X-Y plane X-Z plane

Input:

CAD (see chapter 4.1.4): *Line from node 1 to node 2*

Z88STRUCTURE.TXT
> *Set KFLAG to 0 for Cartesian coordinates*
> *6 degrees of freedom in a node (Attention DoF5 (not right hand rule), see below)*
> *Element type is 5*
> *2 nodes per element*



enter here the diameter for the elements

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG* has no meaning

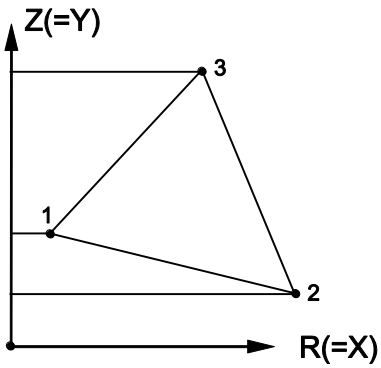
Results:

Displacements in X, Y and Z and rotations around X, Y and Z. Attention DoF5 (not right hand rule), see below

Stresses: SIGXX, TAUXX: Normal stress, shear stress, SIGXY1, SIGXY2: Bending stress in X-Y plane for node 1 and node 2, SIGXZ1 SIGXZ2: Bending stress in X-Z plane for node 1 and node 2

Nodal forces in X, Y and Z and **nodal moments** around X, Y and Z for each element and each node.

5.6 TORUS NO. 6 WITH 3 NODES



This element is implemented only for historical reasons and possible data exchange to other FEA systems. Much better: Torus No. 8 or Torus No. 12 or No. 15. No entries into the surface and pressure loads file Z88I5.TXT!

This is a simple, triangular torus element with linear shape functions for axisymmetric structures. The displacement results for this very simple element are quite useable, but the stress calculation results are inaccurate. The stresses are calculated in the corner nodes internally and then distributed as average value in the centre of gravity. However, the use of the torus elements No. 8 or No. 12 or No. 15 is highly recommended especially for accurate stress calculations.

Input:

CAD (see chapter 4.1.4): *1-2-3-1*

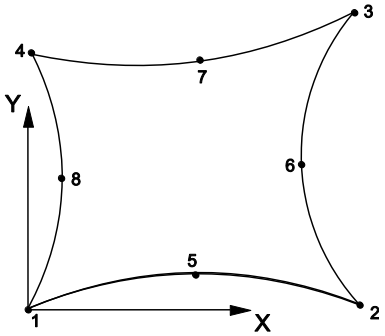
Z88STRUCTURE.TXT
> *In principle cylindrical coordinates are expected: KFLAG must be 0!*
R coordinate (= X), always positive
Z coordinate (= Y), always positive
> *2 degrees of freedom for each node, DoF R and Z (= X and Y)).*
> *Element type is 6*
> *3 nodes per element*

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no influence
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the center of gravity
2 = principal or Rankine stresses in the center of gravity
3 = Tresca stresses in the center of gravity

Results:
Displacements in R and Z (= X and Y)
Stresses: The stress are internally computed in the corner nodes, but plotted in the centre of gravity.
It is: SIGRR = stress in R direction = radial stress (= X direction), SIGZZ = stress in Z direction (= Y direction), TAURZ = shear stress in RZ plane (= XY plane), SIGTE = stress in peripherical direction = tangential stress. Optional von Mises stresses.
Nodal forces for each element and each node.

5.7 PLANE STRESS ELEMENT NO. 7 WITH 8 NODES

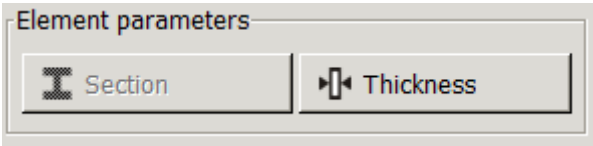


This is a curvilinear Serendipity plane stress element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss-Legendre. Consequently, the integration order can be selected in Z88ENVIRO.DYN. The order 3 is mostly sufficient. This element calculates both displacements and stresses very exactly. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). Pay attention to edge loads when using forces, cf. chapter 3.4. It is easier to enter edge loads via the surface and pressure loads file Z88I5.TXT. You may combine this element with elements No. 3 (not recommended) or elements No. 14 (good).
Plane Stress Elements No. 7 can be generated by the mesh generator Z88N from super elements Plane Stress Elements No. 7 or No. 11. Thus, the Plane Stress Element No. 7 is well suited as super element.

Input:

CAD (see chapter 4.1.4): *1-5-2-6-3-7-4-8-1*

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or polar coordinates (1)*
> *2 degrees of freedom for each node*
> *Element type is 7*
> *8 nodes per element*



enter here the thickness of the elements

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation: 3 is usually good.*
> *Integration order INTOS for stress calculation:*
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

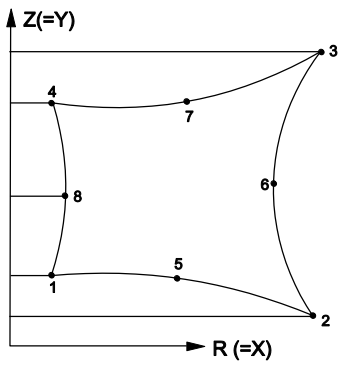
Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* = 0: Calculation of SIGXX, SIGYY and TAUXY
> *Radial/Tangential stress flag KDFLAG* = 1: Additional calculation of SIGRR, SIGTT and TAURT
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Results:

Displacements in X and Y.
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations**.** For KDFLAG = 1 the radial stresses SIGRR, the tangential stresses SIGTT and the accompanying shear stresses SIGRT are computed additionally (makes only sense if a rotational-symmetric structure is available). For easier orientation the respective radiuses and angles of the nodes/points are printed. Optional von Mises or principal or Tresca stresses
Nodal forces in X and Y for each element and each node.

5.8 TORUS NO. 8 WITH 8 NODES

This is a curvilinear Serendipity torus element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss-Legendre. Thus, the integration order can be selected in Z88ENVIRO.DYN. The order 3 is mostly sufficient. This element calculates both displacements and stresses very exactly. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). You may combine this element with elements No. 15. Torus elements No. 8 can be generated by the mesh generator Z88N from the super elements torus elements No. 8 or No. 12. Thus, Torus No. 8 is well suited as super element.



Input:
CAD (see chapter 4.1.4): *1-5-2-6-3-7-4-8-1*

Z88STRUCTURE.TXT
> *In principle cylindrical coordinates are expected: KFLAG must be 0!*
R coordinate (= X), always positive
Z coordinate (= Y), always positive
> *2 degrees of freedom for each node, DoF R and Z (= X and Y).*
> *Element type is 8*
> *8 nodes per element*

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation: 3 is usually good.*
> *Integration order INTOS for stress calculation:*
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

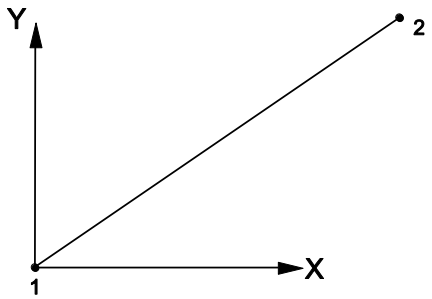
Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG*, any, has no meaning
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0 !)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0 !)
3 = Tresca stresses in the Gauss points (INTOS not 0 !)

Results:

Displacements in R and Z (= X and Y).
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. It is: SIGRR = stress in R direction = radial stress (= X direction), SIGZZ = stress in Z direction (= Y direction), TAURZ = shear stress in RZ plane (= XY plane), SIGTE = stress in peripherical direction = tangential stress. Optional von Mises, principal or Tresca stresses.
Nodal forces in R (= X) and Z (= Y) for each element and each node.

5.9 TRUSS NO. 9 IN PLANE

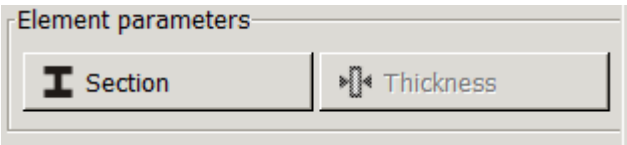
The truss element No. 9 can take any location in the X-Y plane. It is the simplest element in Z88 and is calculated extremely fast. The truss elements match Hooke's law exactly. Hint: Trusses No. 9 are very suitable for modelling spring supports or oblique-angled supports.



Input:

CAD (see chapter 4.1.4): *Line from node 1 to node 2*

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or polar coordinates (1)*
> *2 degrees of freedom for each node*
> *Element type is 9*
> *2 nodes per element*



enter here the cross-section area for the elements

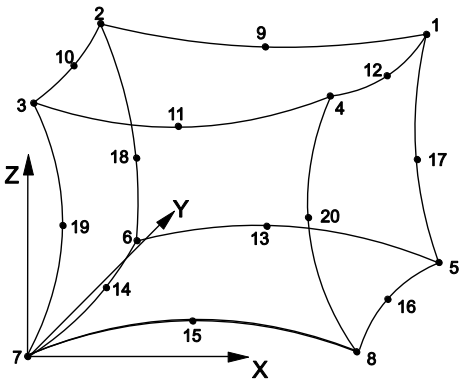
Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG* has no meaning

Results:

Displacements in X and Y
Stresses: Normal stresses
Nodal forces in X and Y for each element and each node.

5.10 HEXAHEDRON NO. 10 WITH 20 NODES



This is a curvilinear Serendipity volume element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in all axes according to Gauss- Legendre. Thus, the integration order can be selected in Z88ENVIRO.DYN. The order 3 is good. The quality of the displacement and stress calculations are far better than the results of the hexahedron element No. 1. Hexahedron No. 1 also applies well for thick plate elements, if the plate's thickness is not too small compared to the other dimensions. Or use shell elements No. 21 and No. 22.
The element causes heavy computing load and needs a large amount of memory because the element stiffness matrix has the order 60×60.

The nodal numbering of the element No. 10 must be done carefully and must exactly match the sketch below. Pay attention to the location of the axis system! The possible error message „Jacobi determinant zero or negative" is a hint for incorrect node numbering.
Hexahedron No. 10 can be generated by the mesh generator Z88N from super elements Hexahedron No. 10. Thus, the Hexahedron No. 10 is well suited as super element. Hexahedron No. 10 can also generate 8-node Hexahedrons No. 1.

Input:

CAD (see chapter 4.1.4):
Upper plane: 1 - 9 - 2 - 10 - 3 - 11 - 4 -12 - 1, quit LINE function
Lower plane: 5 - 13 - 6 - 14 - 7 - 15 - 8 - 16 - 5, quit LINE function
1 - 17 - 5, quit LINE function
2 - 18 - 6, quit LINE function
3 - 19 - 7, quit LINE function
4 - 20 - 8, quit LINE function

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or cylindrical coordinates (1)*
> *3 degrees of freedom for each node*
> *Element type is 10*
> *20 nodes per element*

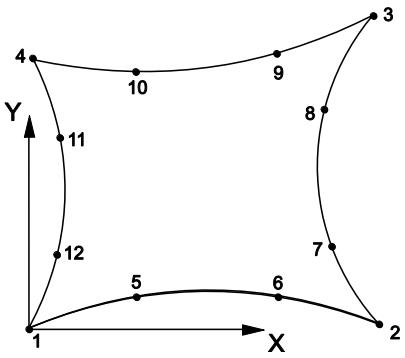
Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation: 3 is usually good.*
> *Integration order INTOS for stress calculation:*
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no influence
> *Reduced stress flag ISFLAG:*

0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Results:
Displacements in X, Y and Z
Stresses: SIGXX, SIGYY, SIGZZ, TAUXY, TAUYZ, TAUZX, respectively for corner nodes or Gauss points. Optional von Mises or principal or Tresca stresses.
Nodal forces in X, Y and Z for each element and each node.

5.11 PLANE STRESS ELEMENT NO. 11 WITH 12 NODES



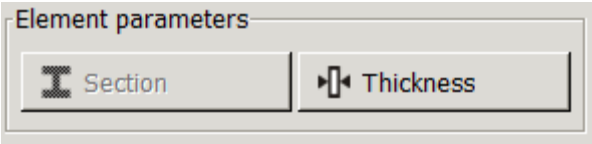
This is a curvilinear Serendipity plane stress element with cubic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss-Legendre. Thus, the integration order can be selected in Z88ENVIRO.DYN. The order 3 is mostly the best choice. This element calculates both displacements and stresses with outstanding precision. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly).

Plane Stress Elements No. 7 can be generated by the mesh generator Z88N from super elements Plane Stress Elements No. 11. Thus, the Plane Stress Element No. 11 is well suited as super element. However, Plane Stress Elements No. 11 cannot be generated by the mesh generator Z88N from super elements Plane Stress Elements No. 11.

Input:

CAD (see chapter 4.1.4): *1-5-6-2-7-8-3-9-10-4-11-12-1*

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or polar coordinates (1)*
> *2 degrees of freedom for each node*
> *Element type is 11*
> *12 nodes per element*



enter here the thickness of the elements

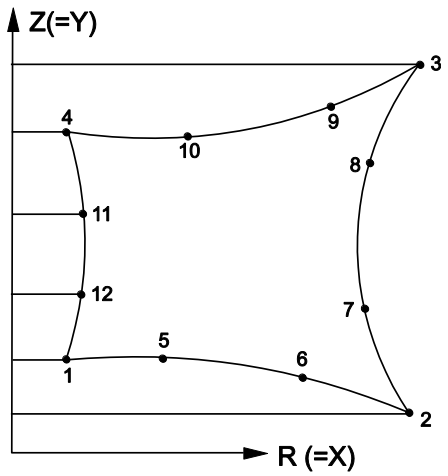
Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation: 3 is usually good.*
> *Integration order INTOS for stress calculation:*
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* = 0: Calculation of SIGXX, SIGYY and TAUXY
> *Radial/Tangential stress flag KDFLAG* = 1: Additional calculation of SIGRR, SIGTT and TAURT
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Results:

Displacements in X and Y.
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. For KDFLAG = 1 the radial stresses SIGRR, the tangential stresses SIGTT and the accompanying shear stresses SIGRT are computed additionally (makes only sense if a rotational-symmetric structure is available). For easier orientation the respective radiuses and angles of the nodes/points are printed. Optional von Mises or principal or Tresca stresses
Nodal forces in X and Y for each element and each node.

5.12 TORUS NO. 12 WITH 12 NODES



This is a curvilinear Serendipity torus element with cubic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss- Legendre. Thus, the integration order can be selected in Z88ENVIRO.DYN. The order 3 is mostly sufficient. This element calculates both displacements and stresses with outstanding precision. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly).
Torus elements No. 8 can be generated by the mesh generator Z88N from super elements torus elements No. 12. Thus, the torus element No. 12 is well suited as super element. However, torus elements No. 12 cannot be generated by the mesh generator Z88N from super elements torus elements No. 12.

Input:

CAD (see chapter 4.1.4): *1-5-6-2-7-8-3-9-10-4-11-12-1*

Z88STRUCTURE.TXT
> In principle cylindrical coordinates are expected: KFLAG must be 0!
R coordinate (= X), always positive
Z coordinate (= Y), always positive
> 2 degrees of freedom for each node, DoF R and Z (= X and Y).
> Element type is 12
> 12 nodes per element

Z88ENVIRO.DYN
> Integration order INTORD for displacement calculation: 3 is usually good.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

Z88CONTROL.TXT
> Radial/Tangential stress flag KDFLAG, any, has no meaning
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0 !)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0 !)
3 = Tresca stresses in the Gauss points (INTOS not 0 !)

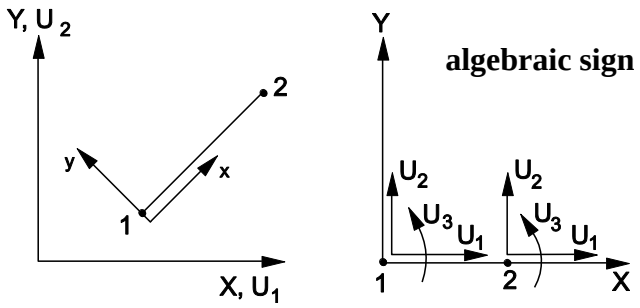
Results:

Displacements in R and Z (= X and Y).
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along

with their locations. It is: SIGRR = stress in R direction = radial stress (= X direction), SIGZZ = stress in Z direction (= Y direction), TAURZ = shear stress in RZ plane (= XY plane), SIGTE = stress in peripherical direction = tangential stress. Optional von Mises or principal or Tresca stresses.
Nodal forces in R (= X) and Z (= Y) for each element and each node.

5.13 BEAM NO. 13 IN PLANE

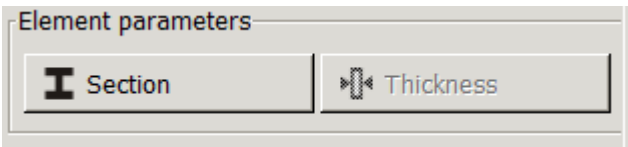
Beam element with any symmetric profile. Enter the element parameters into the GUI. Thus, you can use any symmetric profile in contrast to other FEA programs which incorporate a variety of different special beam and profile subroutines without matching all symmetric profiles as necessary. The element matches exactly Bernoulli's bend theory and Hooke's law. It uses no approximate solution compared to the continuum elements.



Input:

CAD (see chapter 4.1.4): *Line from node 1 to node 2*

Z88STRUCTURE.TXT
> *KFLAG for Cartesian (0) or polar coordinates (1)*
> *3 degrees of freedom in a node*
> *Element type is 13*
> *2 nodes per element*



enter here the parameters for the elements

> *Cross-sectional area QPARA*
> *insert 0 for second moment of inertia I_{yy} (bending around y-y axis)*
> *insert 0 for max. distance e_{yy} from neutral axis y-y*
> *Second moment of inertia I_{zz} (bending around z-z axis)*
> *Max. distance e_{zz} from neutral axis z-z*
> *insert 0 for second moment of area (torsion) I_T*
> *insert 0 for second modulus (torsion) W_T*

Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG* has no meaning

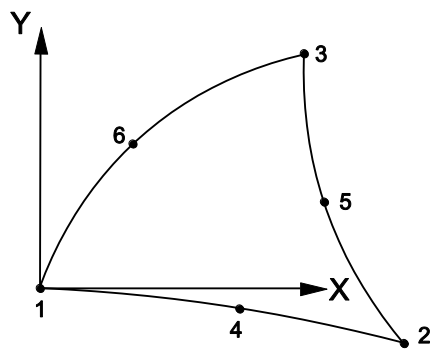
Results:

Displacements in X and Y and **rotations** around Z.
Stresses: SIGXX, TAUXX: Normal stress, shear stress, SIGZZ1, SIGZZ2: Bending stress around z-z for node 1 and node 2
Nodal forces in X and Y and **nodal moments** around Z for each element and each node.

5.14 PLANE STRESS ELEMENT NO. 14 WITH 6 NODES

This is a curvilinear Serendipity plane stress element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically according to Gauss- Legendre. Consequently, the integration order can be selected in Z88INT.TXT. The order 7 (= 7 Gauss points) is mostly sufficient. This element calculates both displacements and stresses very exactly. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). Pay attention to edge loads when using forces, cf. chapter 3.4. It is easier to enter edge loads via the surface and pressure loads file Z88I5.TXT.

This element type is implemented for use with automeshers. Thus, a mesh generation with Z88N is not possible. Use plane stress elements No. 7 for Z88N.
Use plane stress element No. 7 whenever possible. It is substantially more precise than this isoparametric triangle.



Input:

CAD (see chapter 2.7.2): *1-4-2-5-3-6-1*

Z88I1.TXT
> KFLAG for Cartesian (0) or polar coordinates (1)
> 2 degrees of freedom for each node
> Element type is 14
> 6 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is the element thickness

Z88INT.TXT
> Integration order INTORD for displacement calculation. 7 is usually good. Possible is: 3 for 3 Gauss points, 7 for 7 Gauss points and 13 for 13 Gauss points. For easy combination with plane stress elements No .7, function ISOD88 of Z88 uses internally these values:
integration order 1 or 2: 3 Gauss points
integration order 4: 7 Gauss points
Example: Z88INT.TXT uses an entry of 2 for INTORD: Thus, plane stress elements No. 7 use 2×2 = 4 Gauss points and plane stress elements No. 14 use 3 Gauss points for integration.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes

1,7,13 = Calculation of stresses in the Gauss points

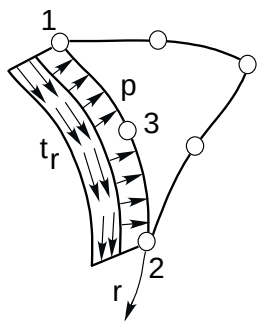
Z88MAT.TXT
> *Define materials, ref. chapter. 3.1.4 and 3.1.5.*

Z88MAN.TXT
> *Radial/Tangential stress flag KDFLAG* = 0: Calculation of SIGXX, SIGYY and TAUXY
> *Radial/Tangential stress flag KDFLAG* = 1: Additional calculation of SIGRR, SIGTT and TAURT
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces edge loads are applied onto element No. 14 – otherwise, enter a 0 into the first line:

> *Element number with surface and pressure load*
> *Pressure, positive if pointing towards the edge*
> *Tangential shear, positive in local r direction*
> *2 corner nodes and one mid node of the loaded surface. Mathematically positive in top view.*

The local r direction is defined by the nodes 1-2. The local nodes 1, 2, 3 may differ from the local nodes 1, 2, 3 used for the coincidence.



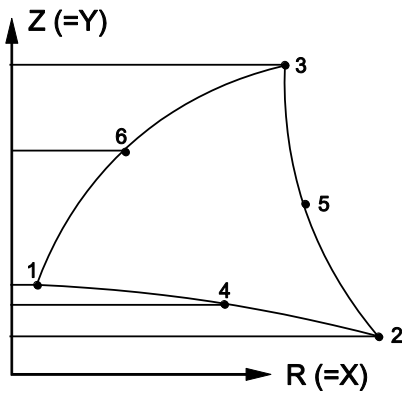
Results:

Displacements in X and Y.
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. For KFLAG = 1 the radial stresses SIGRR, the tangential stresses SIGTT and the accompanying shear stresses SIGRT are computed additionally (makes only sense if a rotational-symmetric structure is available). For easier orientation the respective radiuses and angles of the nodes/points are printed. Optional von Mises or principal or Tresca stresses.
Nodal forces in X and Y for each element and each node.

5.15 TORUS NO. 15 WITH 6 NODES

This is a curvilinear Serendipity torus element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically according to Gauss- Legendre. Thus, the integration order can be selected in Z88INT.TXT. The order 7 is mostly sufficient. This element calculates both displacements and stresses very exactly. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). Pay attention to edge loads when using forces, cf. chapter 3.4. It is easier to enter edge loads via the surface and pressure loads file Z88I5.TXT.

This element type is implemented for use with automeshers. Thus, a mesh generation with Z88N is not possible. Use torus elements No. 8 for Z88N.
Use torus element No .8 whenever possible. It is substantially more precise than this isoparametric triangle.



Input:

CAD (see chapter 2.7.2): *1-4-2-5-3-6-1*

Z88I1.TXT
> In principle cylindrical coordinates are expected: KFLAG must be 0!
R coordinate (= X), always positive
Z coordinate (= Y), always positive
> 2 degrees of freedom for each node, DoF R and Z (= X and Y).
> Element type is 15
> 6 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is 0 or any value, no influence

Z88INT.TXT
> Integration order INTORD for displacement calculation. 7 is usually good. Possible is: 3 for 3 Gauss points, 7 for 7 Gauss points and 13 for 13 Gauss points. For easy combination with torus elements No. 8, function ISOD88 of Z88 uses internally these values:
integration order 1 or 2: 3 Gauss points
integration order 4: 7 Gauss points
Example: Z88INT.TXT uses an entry of 2 for INTORD: Thus, torus elements No. 8 use 2×2 =

4 Gauss points and torus elements No. 15 use 3 Gauss points for integration.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,7,13 = Calculation of stresses in the Gauss points

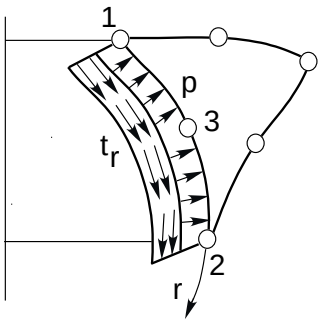
Z88MAT.TXT
> Define materials, ref. chapter. 3.1.4 and 3.1.5.

Z88MAN.TXT
> Radial/Tangential stress flag KDFLAG has no meaning
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces edge loads are applied onto element No. 15 – otherwise, enter a 0 into the first line:

> Element number with surface and pressure load
> Pressure, positive if pointing towards the edge
> Tangential shear, positive in local r direction
> 2 corner nodes and one mid node of the loaded surface. Mathematically positive in top view.

The local r direction is defined by the nodes 1-2. The local nodes 1, 2, 3 may differ from the local nodes 1, 2, 3 used for the coincidence.



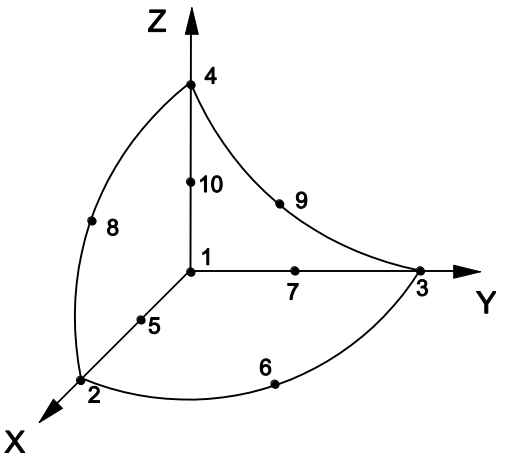
Results:
Displacements in R and Z (= X and Y).
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. It is: SIGRR = stress in R direction = radial stress (= X direction), SIGZZ = stress in Z direction (= Y direction), TAURZ = shear stress in RZ plane (= XY plane), SIGTE = stress in peripherical direction = tangential stress. Optional von Mises or principal or Tresca stresses.
Nodal forces in R (= X) and Z (= Y) for each element and each node.

5.16 TETRAHEDRON NO. 16 WITH 10 NODES

This is a curvilinear Serendipity volume element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically according to Gauss- Legendre. Thus, the integration order can be selected in Z88INT.TXT. The order 4 is good. The quality of the displacement and stress calculations are far better than the results of the tetrahedron element No. 17 but less precise than hexahedron No. 10.

This element type is implemented for use with automeshers. The converter functionality in Z88Aurora® offers the possibility to import and process files with this element type. For further information see chapter 4.1.8.

Tetrahedron No. 16 also applies well for thick plate elements, if the plate's thickness is not too small compared to the other dimensions. Pay attention to pressure loads when using forces, cf. chapter 3.4. It is easier to enter pressure loads via the surface and pressure loads file Z88I5.TXT.



The nodal numbering of the element No. 16 must be done carefully and must exactly match the sketch below. Pay attention to the location of the axis system! The possible error message *„Jacobi determinant zero or negative "* is a hint for incorrect node numbering.

Tetrahedron No. 16 cannot be generated by the mapped mesh generator Z88N. **Caution:** The automeshers of CAD systems very often produce very bad nodal numbering resulting in an useless large amount of memory needs of Z88R's Cholesky solver. Thus, you may renumber especially the nodes or use one of the sparse matrix solvers i.e. SICCG, SORCG, Pardiso.

Input:

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 3 degrees of freedom for each node
> Element type is 16
> 10 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is 0 or any other value, has no influence

Z88INT.TXT
> Integration order INTORD for displacement calculation. 4 is usually good. Allowed are 1 for 1 Gauss point, 4 for 4 Gauss points and 5 for 5 Gauss points
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,4,5 = Calculation of stresses in the Gauss points

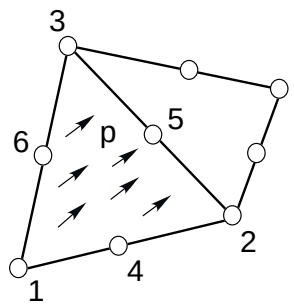
Z88MAT.TXT
> Define materials, ref. chapter. 3.1.4 and 3.1.5.

Z88MAN.TXT
> Radial/Tangential stress flag KDFLAG has no meaning
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces pressure loads are applied onto element No. 16 – otherwise, enter a 0 into the first line:

> Element number with pressure load
> Pressure, positive if pointing towards the edge
> 3 corner nodes and 3 mid nodes of the loaded surface. Mathematically positive in plain view.

The local nodes 1 to 6 may differ from the local nodes 1 to 6 used for the coincidence.



Results:
Displacements in X, Y and Z
Stresses: SIGXX, SIGYY, SIGZZ, TAUXY, TAUYZ, TAUZX, respectively for corner nodes or Gauss points. Optional von Mises or principal or Tresca stresses.
Nodal forces in X, Y and Z for each element and each node.

5.17 TETRAHEDRON NO. 17 WITH 4 NODES

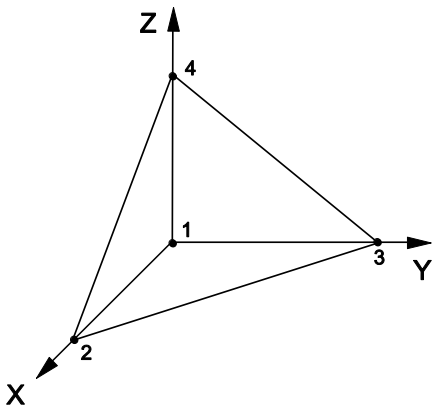
This is a volume element with linear shape functions. The transformation is isoparametric. The integration is carried out numerically according to Gauss- Legendre. Thus, the integration order can be selected in Z88INT.TXT. The order 1 is good.

This element type is implemented for use with automeshers. The converter functionality in Z88Aurora® offers the possibility to import and process files with this element type. For further information see chapter 4.1.8.

Tetrahedron No. 17 also applies well for thick plate elements, if the plate's thickness is not too small compared to the other dimensions.

Basically, this element calculates deflections and stresses very badly i.e. inaccurately. One needs very fine meshes to obtain useful results. Its one and only reason is the data exchange with 3D CAD systems. Use tetrahedrons No. 16, hexahedrons No. 1 and (best choice) hexahedrons No. 10.

Tetrahedron No. 17 cannot be generated by the mapped mesh generator Z88N.



Input:

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 3 degrees of freedom for each node
> Element type is 17
> 4 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is 0 or any other value, has no influence

Z88INT.TXT
> Integration order INTORD for displacement calculation. 1 is usually good. Allowed are 1 for 1 Gauss point, 4 for 4 Gauss points and 5 for 5 Gauss points
> Integration order INTOS for stress calculation:

0 = Calculation of stresses in the corner nodes
1,4,5 = Calculation of stresses in the Gauss points

Z88MAT.TXT
> Define materials, ref. chapter. 3.1.4 and 3.1.5.

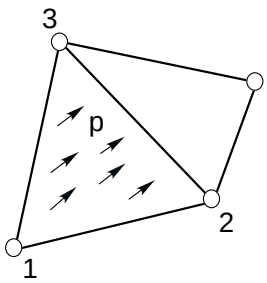
Z88MAN.TXT
> Radial/Tangential stress flag KDFLAG has no meaning
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces pressure loads are applied onto element No. 17 – otherwise, enter a 0 into the first line:

> Element number with pressure load
> Pressure, positive if pointing towards the edge
> 3 corner nodes of the loaded surface. Mathematically positive in plain view.

The local nodes 1 to 3 may differ from the local nodes 1 to 3 used for the coincidence.

Results:

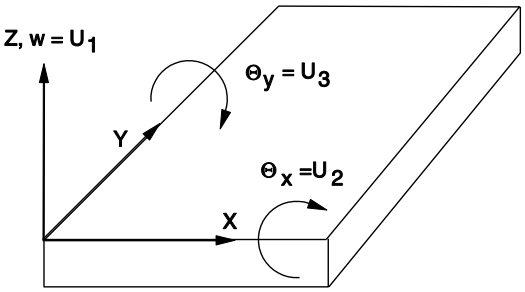


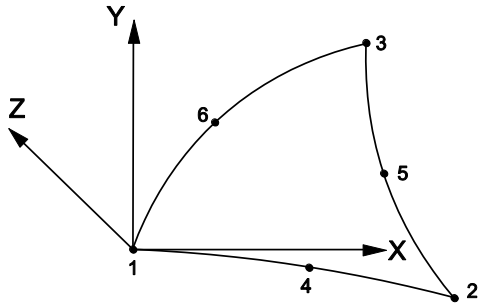
Displacements in X, Y and Z
Stresses: SIGXX, SIGYY, SIGZZ, TAUXY, TAUYZ, TAUZX, respectively for corner nodes or Gauss points. Optional von Mises or principal or Tresca stresses.
Nodal forces in X, Y and Z for each element and each node.

5.18 PLATE NO. 18 WITH 6 NODES

This is a curvilinear Serendipity *Reissner-Mindlin* plate element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss-Legendre. Consequently, the integration order can be selected in Z88INT.TXT. The order 3 (= 3 points) is mostly sufficient (reduced integration). This element calculates both displacements and stresses quite good. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). For this element you need to set the plate flag IPFLAG to 1. Attention: In contrary to the usual rules of the classic mechanics Z88 defines θ_x the rotation around the X-axis and θ_y the rotation around the Y-axis.

This element type is implemented for use with automeshers. Thus, a mesh generation with Z88N is not possible, because this will make no sense. Use plates No. 20 for the mapped mesher Z88N. Because plates No. 20 compute both the deflections and the stresses more exactly than the curvilinear triangle plates No. 18, you should prefer always plates No. 20.





Input:

CAD: *1-4-2-5-3-6-1*, ref. Chap. 2.7.2

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> set plate flag IPFLAG to 1 (or 2, if you want to reduce the shear influence)
> 3 degrees of freedom for each node (w, θ_x, θ_y)
> Element type is 18
> 6 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is the element thickness
Z88INT.TXT

> Integration order INTORD for displacement calculation. 3 is usually good. Possible is: 3 for 3 Gauss points, 7 for 7 Gauss points and 13 for 13 Gauss points. For easy combination with plate elements No. 20, function SPLA88 of Z88 uses internally these values:
integration order 1 or 2: 3 Gauss points
integration order 4: 7 Gauss points
Example: Z88INT.TXT uses an entry of 2 for INTORD: Thus, plate element No. 20 use 2×2 = 4 Gauss points and plate element No. 18 use 3 Gauss points for integration.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,7,13 = Calculation of stresses in the Gauss points

Z88MAT.TXT
> Define materials, ref. chapter. 3.1.4 and 3.1.5.

Z88MAN.TXT
> set plate flag IPFLAG to 1
> Radial/Tangential stress flag KDFLAG has no meaning
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces pressure loads are applied onto element No. 18 – otherwise, enter a 0 into the first line:

> Element number with pressure load
> Pressure, positive if pointing towards the surface

Results:

Displacements in Z (i.e. w) and rotations θ_x around X-axis and θ_y around the Y-axis.

Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. The following results will be presented:

- plate bending moments M_{xx} and M_{yy} (unit: force × length / length)
- plate torsion moments $M_{xy} = M_{yx}$ (unit: force × length / length)
- the shear forces Q_{yz} and Q_{zx} (unit: force / length)
- the true stresses resulting from plate bending moments and plate torsion moments

Optional *von Mises* or principal or Tresca stresses.
Nodal forces in X and Y for each element and each node.

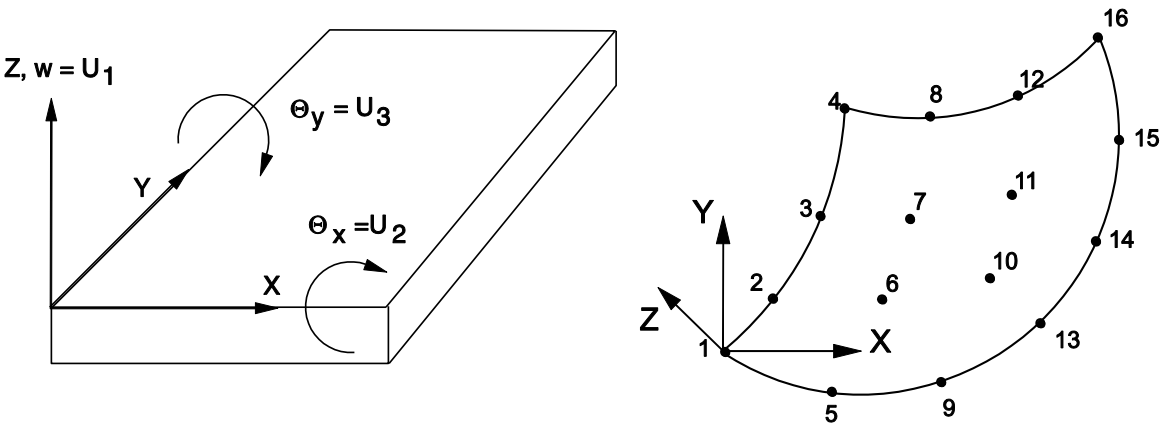
5.19 PLATE NO. 19 WITH 16 NODES

This is a curvilinear Lagrange-*Reissner-Mindlin* plate element with cubic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss-Legendre. Consequently, the integration order can be selected in Z88INT.TXT. The order 4 (= 4×4 points) is very good. This element calculates both displacements and stresses very precisely. The input amount is heavy; you should use the mesher Z88N.

The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). For this element you need to set the plate flag IPFLAG to 1. Attention: In contrary to the usual rules of the classic mechanics Z88 defines θ_x the rotation around the X-axis and θ_y the rotation around the Y-axis.

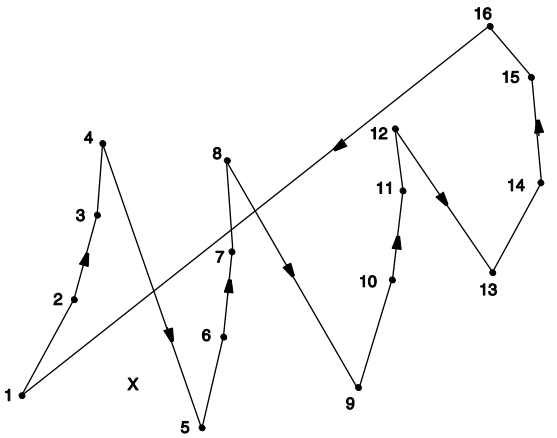
Mesh generation with Z88N: Use plates No. 20 for super elements, resulting in finite elements of type 19 (plates No. 20 may generated by AutoCAD or Pro/ENGINEER, ref. the chapters of Z88X and Z88G). A bit tricky, but works quite fine.
For example, some lines from a mesh generator input file Z88NI.TXT:

```
.....
5    20          super element 5 of type 20
20   25   27   22   24   26   28   21
.....
5    19          generate from super element 5 (which is of type 20 is, see above) finite elements of type 19
3E   3E          and subdivide them three times equidistant in X-direction and three times equidistant in Y- direction
```



Input:

CAD: *1-2-3-4-5-6-7-8-9-10-11-12-13-14-15-16-1*, ref. chap. 2.7.2. Usually, you will not work in this way. It is much easier to build within a CAD program a super element mesh with 8-node plates No. 20. Export this mesh as a DXF file and use Z88X to produce a mesh generator input file Z88NI.TXT. Run the mapped mesher Z88N and generate a finite elements mesh with plates No. 19. Then you may supply the boundary conditions.



Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> set plate flag IPFLAG to 1 (or 2, if you want to reduce the shear influence)
> 3 degrees of freedom for each node (w, θ_x,θ_y)
> Element type is 19
> 16 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is the element thickness

Z88INT.TXT
> Integration order INTORD for displacement calculation. 4 is usually good.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

Z88MAT.TXT
> Define materials, ref. chapter. 3.1.4 and 3.1.5.

Z88MAN.TXT
> set plate flag IPFLAG to 1
> Radial/Tangential stress flag KDFLAG has no meaning
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces pressure loads are applied onto element No. 19 – otherwise, enter a 0 into the first line:

> Element number with pressure load
> Pressure, positive if pointing towards the edge

Results:
Displacements in Z (i.e. w) and rotations θ_x around X-axis and θ_y around the Y-axis.

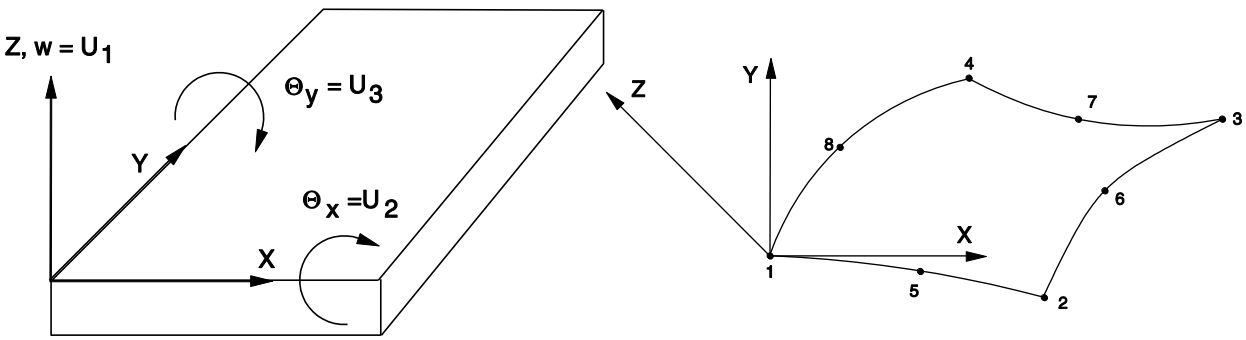
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. The following results will be presented:

- plate bending moments M_{xx} and M_{yy} (unit: force × length / length)
- plate torsion moments $M_{xy} = M_{yx}$ (unit: force × length / length)
- the shear forces Q_{yz} and Q_{zx} (unit: force / length)
- the true stresses resulting from plate bending moments and plate torsion moments

Optional *von Mises* or principal or Tresca stresses.
Nodal forces in X and Y for each element and each node.

5.20 PLATE NO. 20 WITH 8 NODES

This is a curvilinear Serendipity *Reissner-Mindlin* plate element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in both axes according to Gauss-Legendre. Consequently, the integration order can be selected in Z88INT.TXT. The order 2 (= 2×2 points) is mostly sufficient (reduced integration). This element calculates both displacements and stresses quite good. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). For this element you need to set the plate flag IPFLAG to 1. Attention: In contrary to the usual rules of the classic mechanics Z88 defines θ_x the rotation around the X-axis and θ_y the rotation around the Y-axis.
This element type is implemented for use with automeshers. In addition, a mesh generation with Z88N is possible. Super elements of type 20 can generate finite elements of type 20, and plates of type 19 with 16 nodes, too.



Input:

CAD: *1-5-2-6-3-7-4-8-1*, ref. chap. 2.7.2

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> set plate flag IPFLAG to 1 (or 2, if you want to reduce the shear influence)
> 3 degrees of freedom for each node (w, θ_x,θ_y)
> Element type is 20
> 8 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is the element thickness

Z88INT.TXT
> Integration order INTORD for displacement calculation. 2 is usually good.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

Z88MAT.TXT

> *Define materials, ref. chapter. 3.1.4 and 3.1.5.*

Z88MAN.TXT
> *set plate flag IPFLAG to 1*
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces pressure loads are applied onto element No. 19 – otherwise, enter a 0 into the first line:

> *Element number with pressure load*
> *Pressure, positive if pointing towards the edge*

Results:

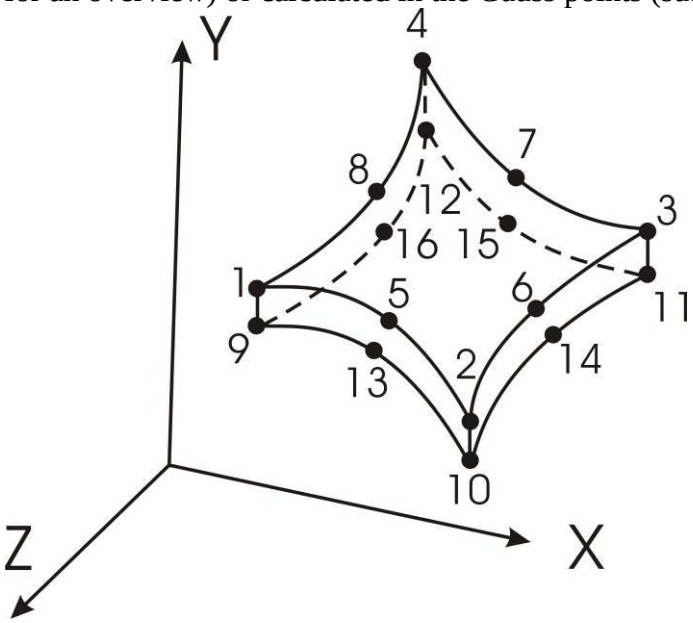
Displacements in Z (i.e. w) and rotations θ_x around X-axis and θ_y around the Y-axis.
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. The following results will be presented:

- plate bending moments M_{xx} and M_{yy} (unit: force × length / length)
- plate torsion moments $M_{xy} = M_{yx}$ (unit: force × length / length)
- the shear forces Q_{yz} and Q_{zx} (unit: force / length)
- the true stresses resulting from plate bending moments and plate torsion moments

Optional *von Mises* or principal or Tresca stresses.
Nodal forces in X and Y for each element and each node.

5.21 SHELL NO. 21 WITH 16 NODES

This is a curvilinear Serendipity volume shell element. The transformation is isoparametric. The integration is carried out numerically in all axes according to Gauss-Legendre. The element can be arbitrarily curved; it is actually a hexahedron with square shape functions on the surface and linear shape functions in the thickness direction. The integration order can be selected in Z88INT.TXT. The order 3 (i.e. 3×3 Gauss Points) is mostly sufficient. This element calculates both displacements and stresses very exactly. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly).



The three degrees of freedom are the global displacements in X, Y and Z. However, there are no rotational degrees of freedom, because type 21 is in fact a volume element. The element can be generated by the mapped mesher Z88N: type 21 → type 21.

Input:

CAD: upper plane: 1-5-2-6-3-7-4-8-1; lower plane: 9-13-10-14-11-15-12-16-9;
Lines: 1-9; 2-10; 3-11; 4-12, see chapter 4.1.7

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 3 degrees of freedom for each node
> Element type is 21
> 16 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is insignificant

Z88INT.TXT
> Integration order INTORD for displacement calculation: 3 is usually good.
> Integration order INTOS for stress calculation:
0 = Calculation of stresses in the corner nodes
1,2,3,4 = Calculation of stresses in the Gauss points

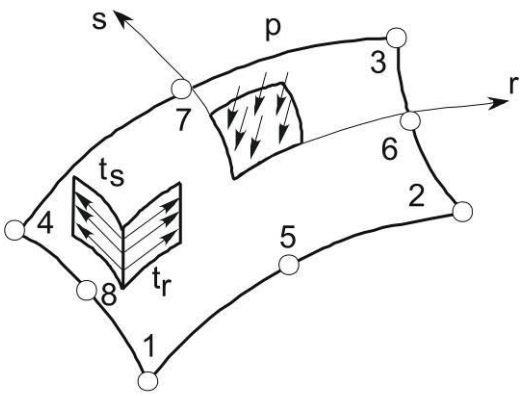
Z88MAT.TXT
> *Define materials, ref. chapter. 3.1.4 and 3.1.5.*

Z88MAN.TXT
> *Radial/Tangential stress flag KDFLAG* has no influence
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces surface and pressure loads are applied onto element No. 21 – otherwise, enter a 0 into the first line:

> *Element number*
> *Pressure, positive if pointing towards the surface*
> *Tangential shear, positive in local r direction*
> *Tangential shear, positive in local s direction*
> *4 corner nodes and 4 mid nodes of the loaded surface. Mathematically positive in top view.*

The local r direction is defined by the nodes 1-2, the local s direction is defined by the nodes 1-4. The local nodes 1 to 8 for the surface load may differ from the local nodes 1 to 8 used for the coincidence.

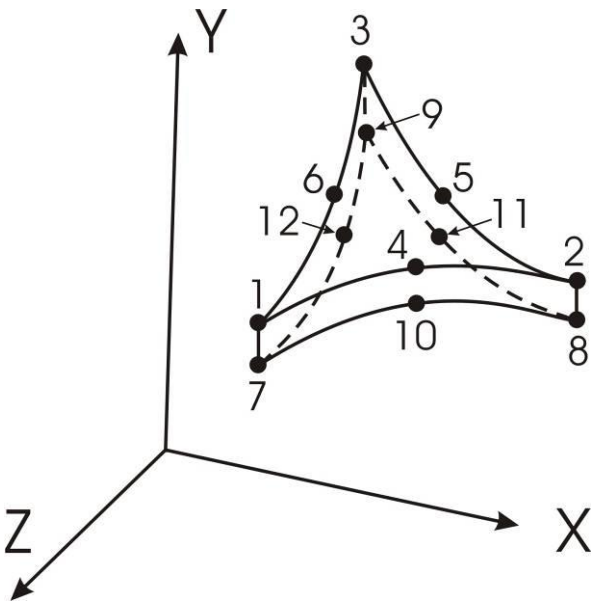


Results:

Displacements in X, Y and Z
Stresses: SIGXX, SIGYY, SIGZZ, TAUXY, TAUYZ, TAUZX, respectively for corner nodes or Gauss points. Optional *von Mises* or principal or Tresca stresses.
Nodal forces in X, Y and Z for each element and each node

5.22 SHELL NO. 22 WITH 12 NODES

This is a curvilinear Serendipity volume shell element. The transformation is isoparametric. The integration is carried out numerically in all axes according to Gauss-Legendre. The element can be arbitrarily curved; it is actually a kind of pie segment with square shape functions on the surface and linear shape functions in the thickness direction. The integration order can be selected in Z88INT.TXT. The order 3 (i.e. 3×3 Gauss Points) is mostly sufficient. This element calculates both displacements and stresses very exactly. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly).



The three degrees of freedom are the global displacements in X, Y and Z. However, there are no rotational degrees of freedom, because type 22 is in fact a volume element.

Input:

CAD: upper plane: 1-4-2-5-3-6-1; lower plane: 7-10-8-11-9-12-7;
Lines: 1-7; 2-8; 3-9; see chapter 4.1.7

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 3 degrees of freedom for each node
> Element type is 22
> 12 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is insignificant

Z88INT.TXT
> Integration order INTORD for displacement calculation: 3, 7, and 13 are possible. 7 is usually good.

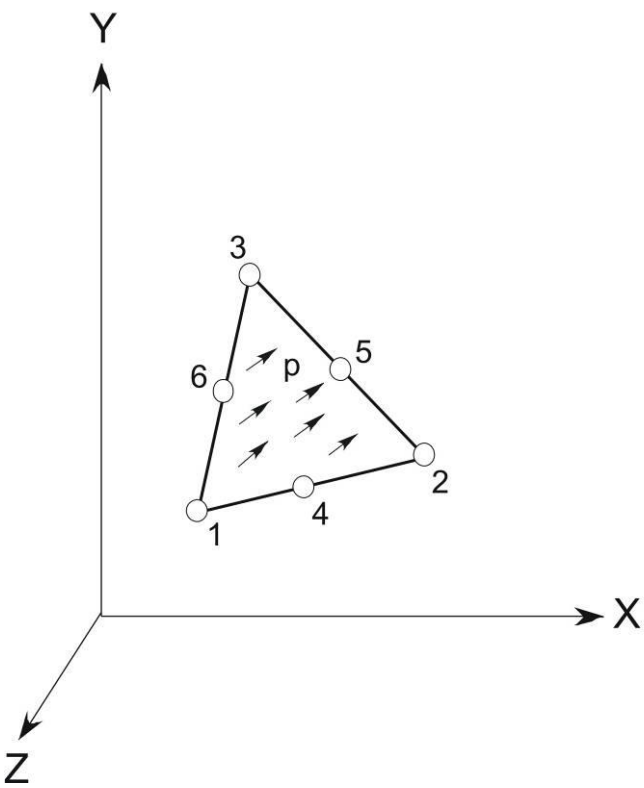
> *Integration order INTOS for stress calculation:*
0 = Calculation of stresses in the corner nodes
3,7,13 = Calculation of stresses in the Gauss points

Z88MAT.TXT
> *Define materials, ref. chapter. 3.1.4 and 3.1.5.*

Z88MAN.TXT
> *Radial/Tangential stress flag KDFLAG* has no influence
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces surface and pressure loads are applied onto element No. 22 – otherwise, enter a 0 into the first line:

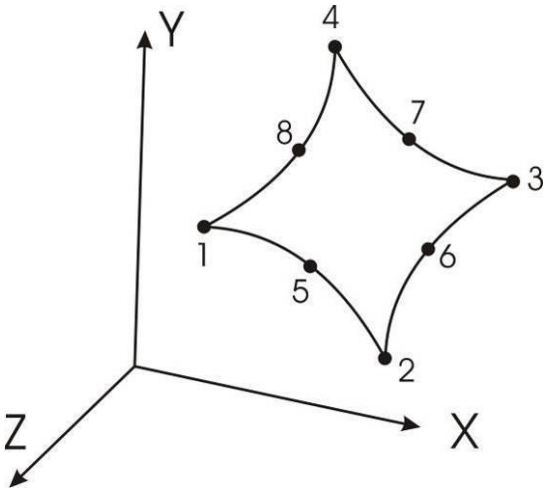
> *Element number*
> *Pressure, positive if pointing towards the surface*
> *3 corner nodes and 3 mid nodes of the loaded surface. Mathematically positive in top view.*



Results:
Displacements in X, Y and Z
Stresses: SIGXX, SIGYY, SIGZZ, TAUXY, TAUYZ, TAUZX, respectively for corner nodes or Gauss points. Optional *von Mises* or principal or Tresca stresses.
Nodal forces in X, Y and Z for each element and each node.

5.23 SHELL NO. 23 WITH 8 NODES

This is a curvilinear Serendipity shell element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in all axes according to Gauss-Legendre. All nodes have to be on a common surface which may be placed arbitrarily in a space – which is very useful for the data exchange with 3D CAD systems. The integration order can be selected in Z88INT.TXT. The order 3 (i.e. 3×3 Gauss Points) is mostly sufficient. This element calculates both displacements and stresses quite good. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). For this element, the shell flag IHFLAG should be set to 1 in Z88MAN.TXT. In case of thin shells, set IHFLAG to 2 or 3. In case of *very* thin shells set it to 4.



The first three degrees of freedom are the global displacements in X, Y and Z. The degrees of freedom 4 and 5 are the *global* torsions on the respective node (thus, quite useless); degree of freedom 6 is a pseudo-DoF without practical significance. Only the global displacements in X, Y and Z are practically useful and of interest for mechanical engineers.

Input:

CAD: *1-5-2-6-3-7-4-8-1* , see chapter 4.1.7

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 6 degrees of freedom for each node – but only DoF 1~3 are of interest
> Element type is 23
> 8 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is the element thickness

Z88INT.TXT
> Integration order INTORD for displacement calculation. 3 is usually good.
> Integration order INTOS for stress calculation:
0 = Calculation of the stresses in the corner nodes

1,2,3,4 = Calculation of the stresses in the Gauss points

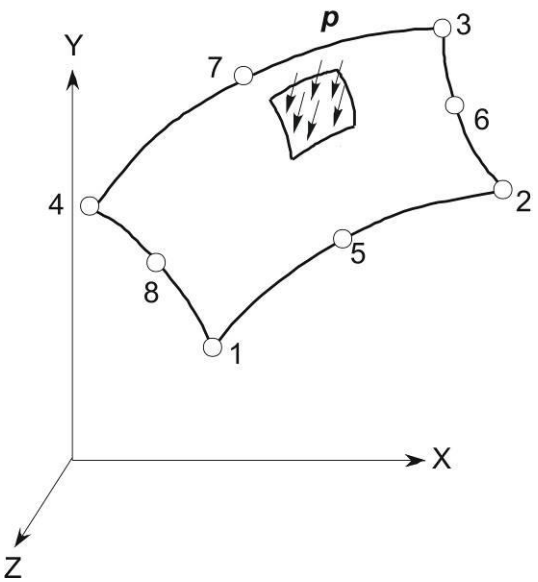
Z88MAT.TXT
> *Define materials, ref. chapter. 3.1.4 and 3.1.5.*

Z88MAN.TXT
> *Set shell flag IHFLAG to 1, or to 2 or 3 in case of thin shells, and to 4 in case of very thin shells*
> *Radial/Tangential stress flag KDFLAG* has no influence
> *Reduced stress flag ISFLAG:*
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces surface and pressure loads are applied onto element No. 23 – otherwise, enter a 0 into the first line:

> *Element number*
> *Pressure, positive if pointing towards the surface*
> *4 corner nodes and 4 mid nodes of the loaded surface. Mathematically positive in top view.*

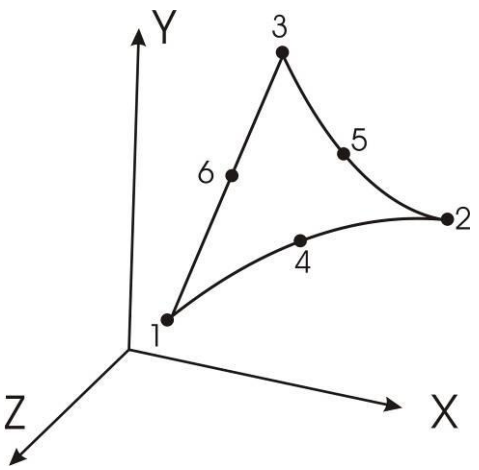
The local r direction is defined by the nodes 1-2, the local s direction is defined by the nodes 1-4. The local nodes 1 to 8 for the surface load may differ from the local nodes 1 to 8 used for the coincidence.



Results:
Displacements in X, Y and Z and global **Rotations** around X- and Y-axis (θ_x u. θ_y)
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. The stresses SIGXX, SIGYY and TAUXY as well as optionally *von Mises* or principal or Tresca stresses are output.
Nodal forces first for each element, then for each node.

5.24 SHELL NO. 24 WITH 6 NODES

This is a curvilinear Serendipity shell element with quadratic shape functions. The transformation is isoparametric. The integration is carried out numerically in all axes according to Gauss-Legendre. All nodes have to be on a common surface which may be placed arbitrarily in a space – which is very useful for the data exchange with 3D CAD systems. The integration order can be selected in Z88INT.TXT. The order 7 (i.e. 7 Gauss Points) is mostly sufficient. This element calculates both displacements and stresses quite good. The integration order can be chosen again for the stress calculation. The stresses are calculated in the corner nodes (good for an overview) or calculated in the Gauss points (substantially more exactly). For this element, the shell flag IHFLAG should be set to 1 in Z88MAN.TXT. In case of thin shells, set IHFLAG to 2 or 3. In case of *very* thin shells set it to 4.



The first 3 degrees of freedom are the global displacements in X, Y and Z. The degrees of freedom 4 and 5 are the *global* torsions on the respective node (thus, quite useless); degree of freedom 6 is a pseudo-DoF without practical significance. Only the global displacements in X, Y and Z are practically useful and of interest for mechanical engineers.

Input:

CAD: *1-4-2-5-3-6-1*, see chapter 4.1.7

Z88I1.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 6 degrees of freedom for each node – but only DoF 1~3 are of interest
> Element type is 24
> 6 nodes per element

Z88ELP.TXT
> Cross-section parameter QPARA is the element thickness

Z88INT.TXT
> Integration order INTORD for displacement calculation. 3, 7, and 13 are possible. 7 is usually good.
> Integration order INTOS for stress calculation:

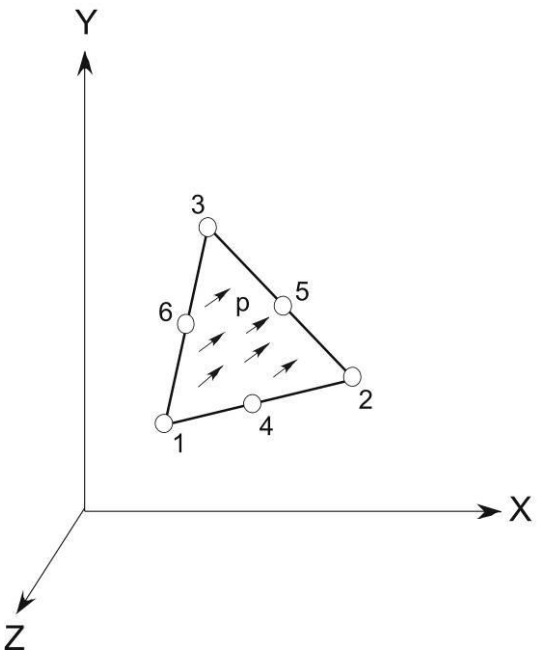
0 = Calculation of the stresses in the corner nodes
3,7,13 = Calculation of the stresses in the Gauss points

Z88MAT.TXT
> Define materials, ref. chapter. 3.1.4 and 3.1.5.

Z88MAN.TXT
> Set shell flag IHFLAG to 1, or to 2 or 3 in case of thin shells, and to 4 in case of very thin shells
> Radial/Tangential stress flag KDFLAG has no influence
> Reduced stress flag ISFLAG:
0 = no calculation of reduced stresses
1 = von Mises stresses in the Gauss points (INTOS not 0!)
2 = principal or Rankine stresses in the Gauss points (INTOS not 0!)
3 = Tresca stresses in the Gauss points (INTOS not 0!)

Z88I5.TXT
This file is only used (see 3.1.3) if in addition to nodal forces surface and pressure loads are applied onto element No. 24 – otherwise, enter a 0 into the first line:

> Element number
> Pressure, positive if pointing towards the surface
> 3 corner nodes and 3 mid nodes of the loaded surface. Mathematically positive in top view.

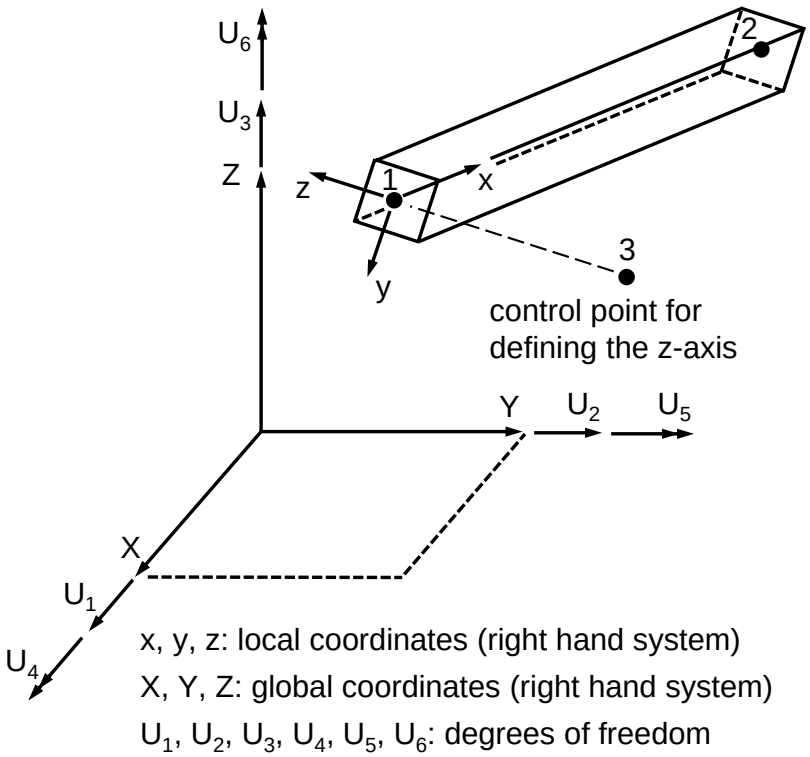


Results:

Displacements in X, Y and Z and global **Rotations** around X- and Y-axis (θ_x u. θ_y)
Stresses: The stresses are calculated in the corner nodes or Gauss points and printed along with their locations. The stresses SIGXX, SIGYY and TAUXY as well as optionally *von Mises* or principal or Tresca stresses are output.
Nodal forces first for each element, then for each node.

5.25 BEAM NO. 25 WITH 2 NODES IN SPACE

Beam element with any symmetric profile (no slanting bend). The cross section of this element is in contrary to element No.2 capable to take arbitrary orientations. The orientation is defined by an additional control point / control node (signed by “3” in the figure below). The control point do not has to be perpendicular to the axis 1-2 in the figure. But it is not allowed that the control point lies on the axis 1-2. The axis which lies in the plane 1-2-3 and perpendicular to 1-2 becomes the local z-axis. The local x-axis directs from 1 to 2 and the local y-axis is calculated by the cross product.
The profile values are provided in the GUI. They are related to the local coordinate system. The beam element No.25 can calculate on the basis of the theory of Bernoulli or the theory of Timoshenko. If the theory of Timoshenko is chosen, the shear ratio (shear correction factor) has to be defined by the user.
The element matches exactly Bernoulli's / Timoshenko’s bend theory and Hooke's law. It uses no approximate solution as for the continuum elements.



Input:

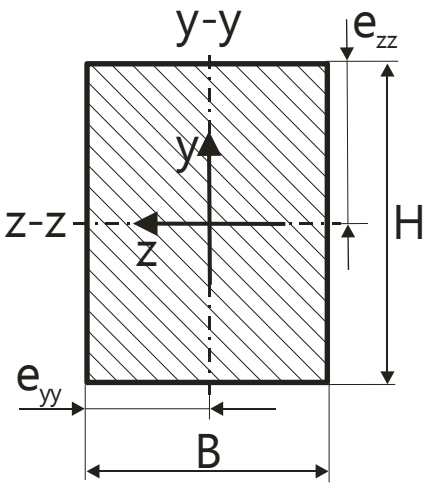
CAD (see chapter 4.1.4):
Line from node 1 to node 2

Z88STRUCTURE.TXT
> KFLAG for Cartesian (0) or cylindrical coordinates (1)
> 6 degrees of freedom in a node (Attention: DoF5 (not right hand rule), see below)
> Element type is 25
> 2 nodes per element

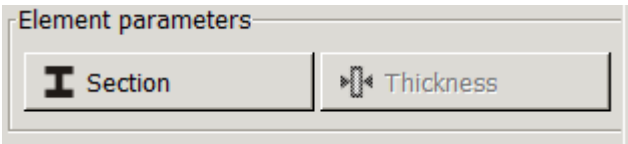
Z88ELP.TXT

> *Cross-sectional area QPARA*
> *Second moment of inertia I_{yy} (bending around y-y axis)*
> *Max. distance e_{yy} from neutral axis y-y*
> *Second moment of inertia I_{zz} (bending around z-z axis)*
> *Max. distance e_{zz} from neutral axis z-z*
> *Second moment of area (torsion) I_T*
> *Second modulus (torsion) W_T*
> *Flag for choosing the bend theory: Bernoulli (0) or Timoshenko (1)*
> *X-coordinate of the control point*
> *Y- coordinate of the control point*
> *Z- coordinate of the control point*
> *shear ratio (shear correction factor), for example in case of rectangular cross section* $= 5/6 \approx 0.8333$

The following figure is found in Z88Aurora® at the register card „element geometry“. It describes the definition of the max. distances from the neutral axes and the axes definition required for the second moments of inertia. Here, in addition to the original figure, the local axes are inserted. The second moment of inertia I_{yy} for example belongs to the case that rotation about the y-axis is considered.



The dialog for entering the element parameters listed above can be found via the button “Section”, as it is shown below.



Z88ENVIRO.DYN
> *Integration order INTORD for displacement calculation:* any order, has no influence
> *Integration order INTOS for stress calculation:* any order, has no influence

Z88CONTROL.TXT
> *Radial/Tangential stress flag KDFLAG* has no meaning
> *Reduced stress flag ISFLAG* has no meaning
Results:

Displacements in X, Y and Z and rotations around X, Y and Z.
Stresses: SIGXX, TAUXX: Direct stress, shear stress, SIGZZ1, SIGZZ2: Bending stress around z-z for node 1 and node 2, SIGYY1 SIGYY2: Bending stress around y-y for node 1 and node 2
Nodal forces in X, Y and Z and nodal moments around X, Y and Z for each element and each node.