The tiny Finite Element System for Android Smartphones

Composed and edited by
Professor Dr. Frank Rieg
Chair for Engineering Design and CAD
University of Bayreuth, Germany

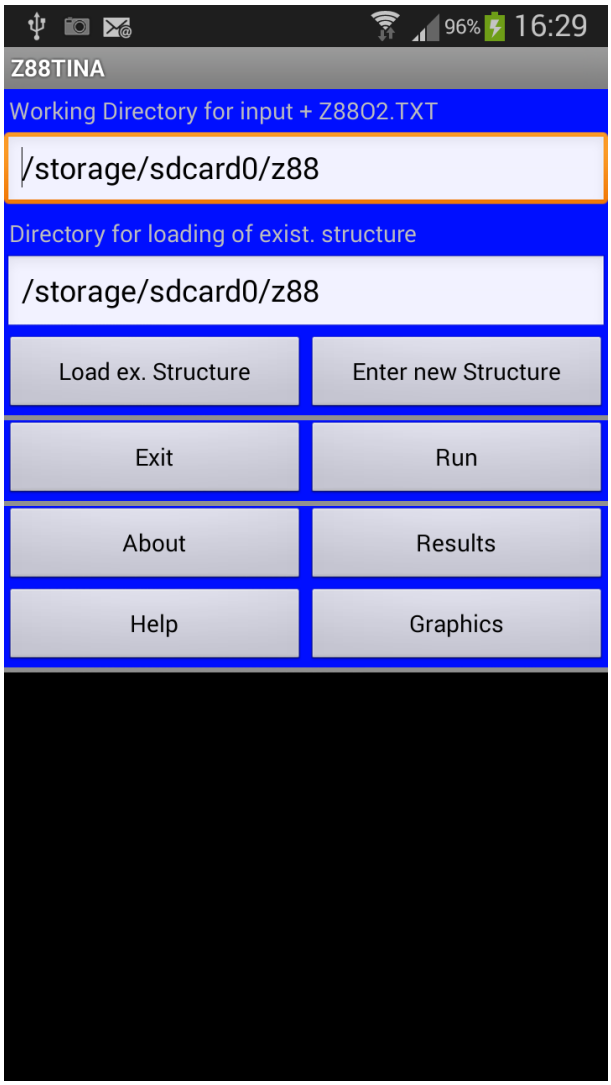
All rights reserved by the editor

Version 1.0

TABLE OF CONTENTS

1 THE FINITE ELEMENTS PROGRAM Z88TINA 3
2 A QUICK TOUR THROUGH Z88TINA 8
2.1 ENTER A NEW STRUCTURE – 1ST EXAMPLE 8
2.2 ENTER A NEW STRUCTURE – 2ND EXAMPLE 30
2.3 READ IN AN EXISTING STRUCTURE – 3RD EXAMPLE 53
2.4 READ IN AN EXISTING STRUCTURE – 4th EXAMPLE 59
3 EDITING INPUT FILES 66
3.1 GENERAL INFORMATION 66
3.2 GENERAL STRUCTURE DATA Z88I1.TXT 67
3.3 BOUNDARY CONDITIONS Z88I2.TXT 70
4 DESCRIPTION OF THE FINITE ELEMENTS 72
4.1 BEAM NO.2 WITH 2 NODES IN SPACE 72
4.2 TRUSS NO.4 IN SPACE 73
4.3 CAM ELEMENT NO.5 WITH 2 NODES 73
4.4 TRUSS NO.9 IN PLANE 74
4.5 BEAM NO.13 WITH 2 NODES IN PLANE 75
4.6 PLANE STRESS ELEMENT NO.14 WITH 6 NODES 76
4.7 TORUS NO.15 WITH 6 NODES 77
4.8 PLATE NO.18 WITH 6 NODES 77
5 FURTHER EXAMPLES 79
5.1 CRANE TRUSS WITH TRUSSES NO.4 79
5.2 TRANSMISSION CAM WITH CAM ELEMENTS NO.5 82
5.3 BEAM IN PLANE WITH BEAMS NO.13 86
5.4 PIPE UNDER INTERNAL PRESSURE, TORUS NO.15 89

1 THE FINITE ELEMENTS PROGRAM Z88TINA



Z88Tina (www.z88tina.de) is a very, very small cousin of our full-scale FEA program Z88Aurora (www.z88.de) for Windows, LINUX and Mac OS-X and is designed for Android Smartphones and Android Tablets. Z88Tina was thoroughly tested on Samsung smartphones Galaxy S2 and S3, and on a Samsung Tablet Tab3 8".

Z88Tina can operate in two modes:

1. Enter a new structure interactively
2. Read in an existing structure

Before we begin please keep in mind:

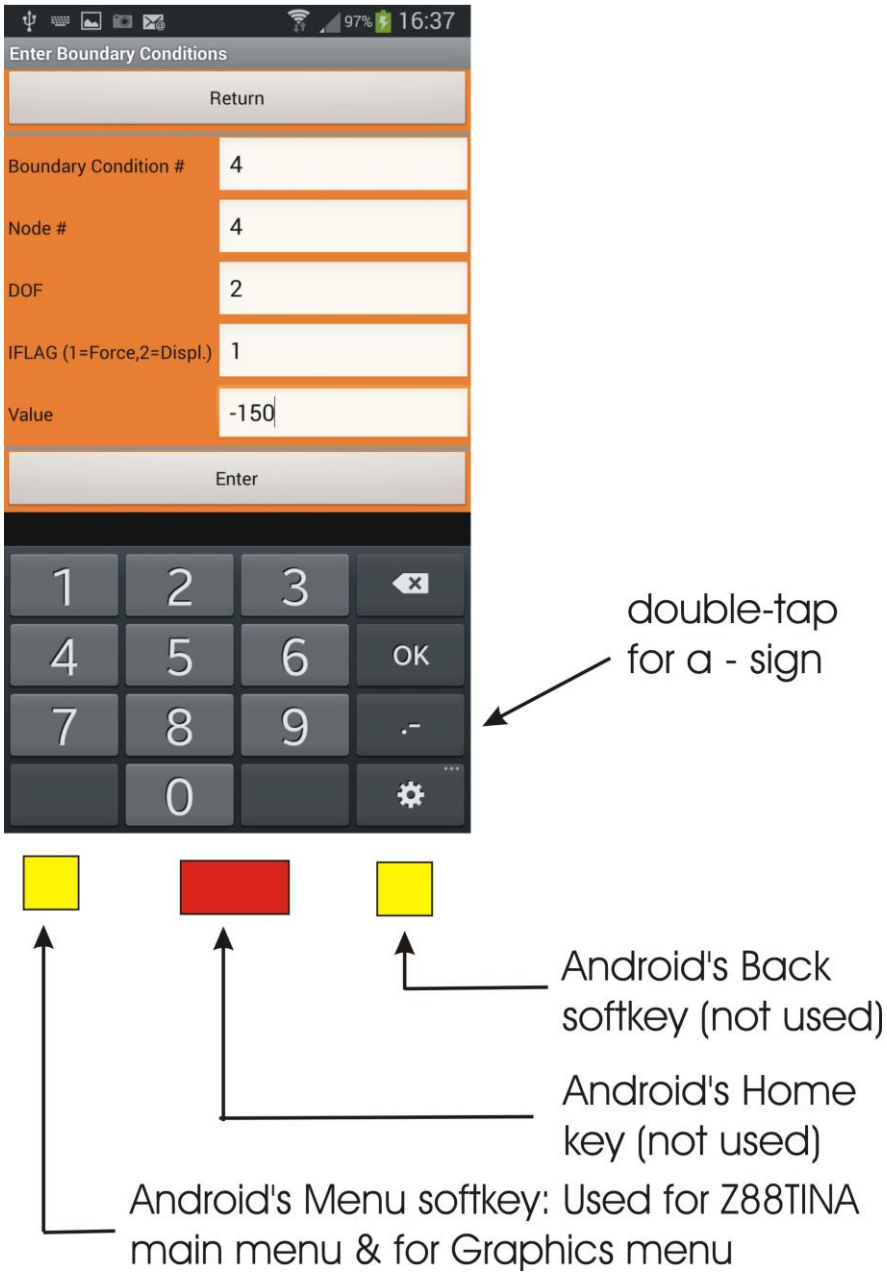
Always compare FE calculations with analytical rough calculations, results of experiments, plausibility considerations and other tests without exception!

Sign definitions of Z88Tina (and also other FEA programs) may differ from the usual definitions of the analytical technical mechanics from time to time .

If you already have FEA experiences, you can start at once. If you are a beginner in this area, i would recommend secondary literature. Here are a few choices – especially the first two ones are excellent:

- *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 Elemente Procedures. Prentice Hall, 1995*
- *Rieg,F.; Hackenschmidt, R., Alber-Laukant, B.: Finite Elemente Analyse für Ingenieure. Carl Hanser Verlag, München Wien 2012, 4th edition (in German language – an English edition is under way)*

Some hints for operating T88Tina:



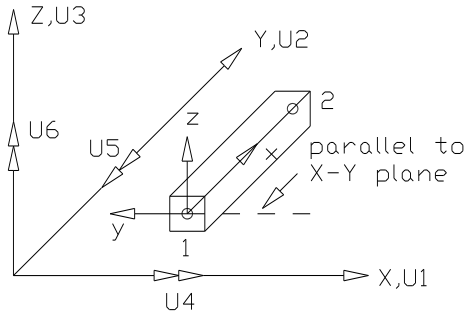
Return from OpenGL graphics screen: Tap on graph screen.

Summary of the Z88 element library:

(You will find the exact description of the element library in chapter 4.)

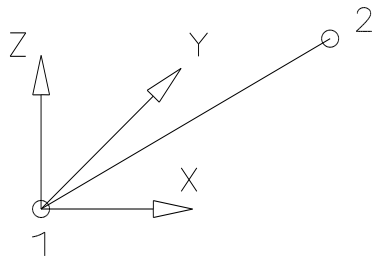
Beam No. 2

- 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



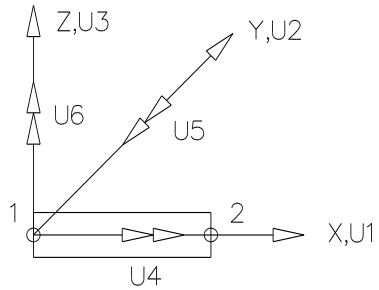
Truss No. 4

- 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



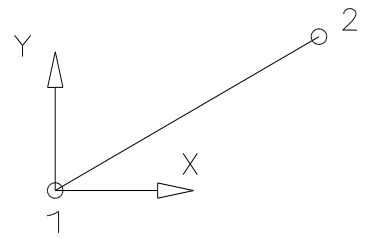
Cam No. 5

- 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



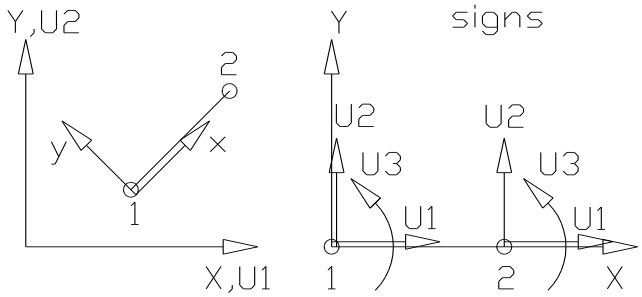
Truss No. 9

- 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



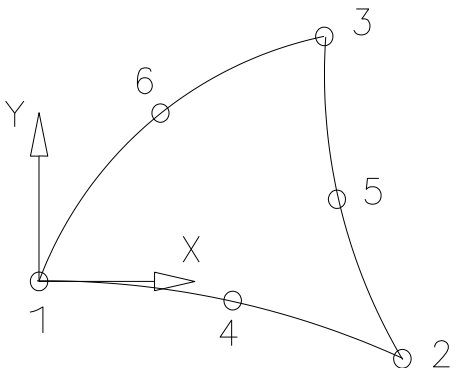
Beam No. 13

- 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



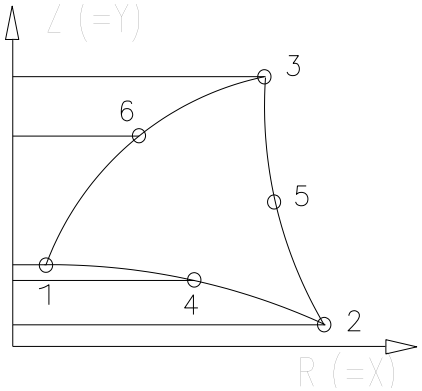
Plane Stress Isoparametric Element No. 14

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



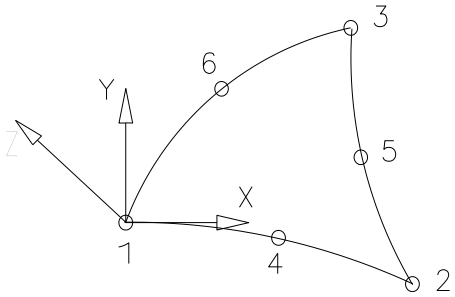
Torus No. 15

- Quadratic Isoparametric Serendipity element
- Quality of displacements 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

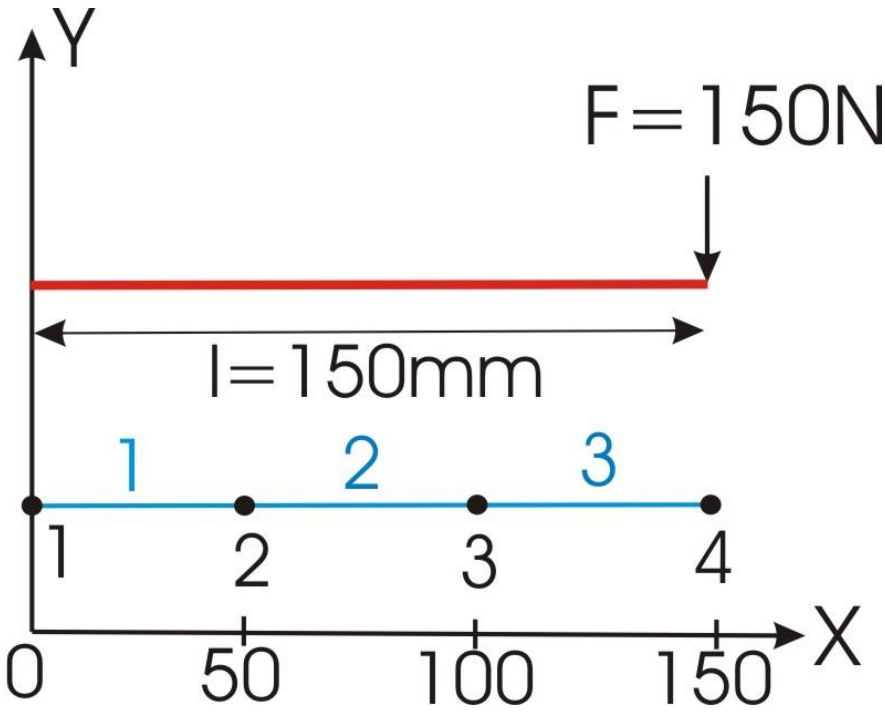
- Quadratic Isoparametric Serendipity element following Reissner- Mindlin's theory
- Quality of displacements very good
- Quality of stresses in the corner nodes acceptable
- Computing effort: medium
- Size of element stiffness matrix: 18 × 18



2 A QUICK TOUR THROUGH Z88TINA

2.1 ENTER A NEW STRUCTURE – 1ST EXAMPLE

Let us assume the following very simple structure: A beam fixed on the left side and with a load on the right side:



The parameters for cross-section area, second moment of inertia zz (bending around zz axis), max. distance from neutral axis zz , Young's modulus and Poisson's ratio are:

$$A = 100\,mm^2\ ,\ I_{zz} = 3000\,mm^4\ ,\ e_{zz} = 10\,mm\ ,\ E = 206000\,\frac{N}{mm^2}\ ,\ \nu = 0.3$$

We want to see results not only for the right side but in addition for x = 50 mm and x = 100 mm. Thus, we need to have 4 nodes, resulting in 3 beam elements. Remember: *You'll get results only where nodes are.*

One computes analytically:

$$w = \frac{F \cdot \ell^3}{3 \cdot EI} = \frac{-150 \cdot 150^3}{3 \cdot 206000 \cdot 3000} = -0.27\,mm$$

$$\psi = \frac{F \cdot \ell^2}{2 \cdot EI} = \frac{-150 \cdot 150^2}{2 \cdot 206000 \cdot 3000} = -0.0027\,rad$$

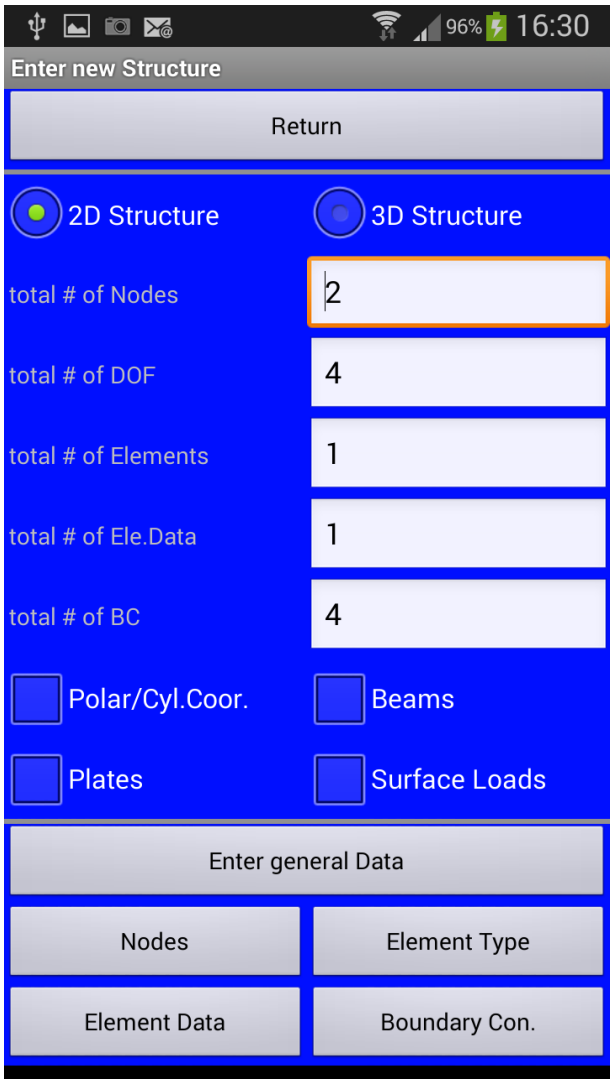
$$M_{bleft} = F \cdot \ell = -22500\,Nmm$$

$$\sigma_{bleft} = \frac{M_b}{I_{zz}} e_{zz} = -75\,\frac{N}{mm^2}$$

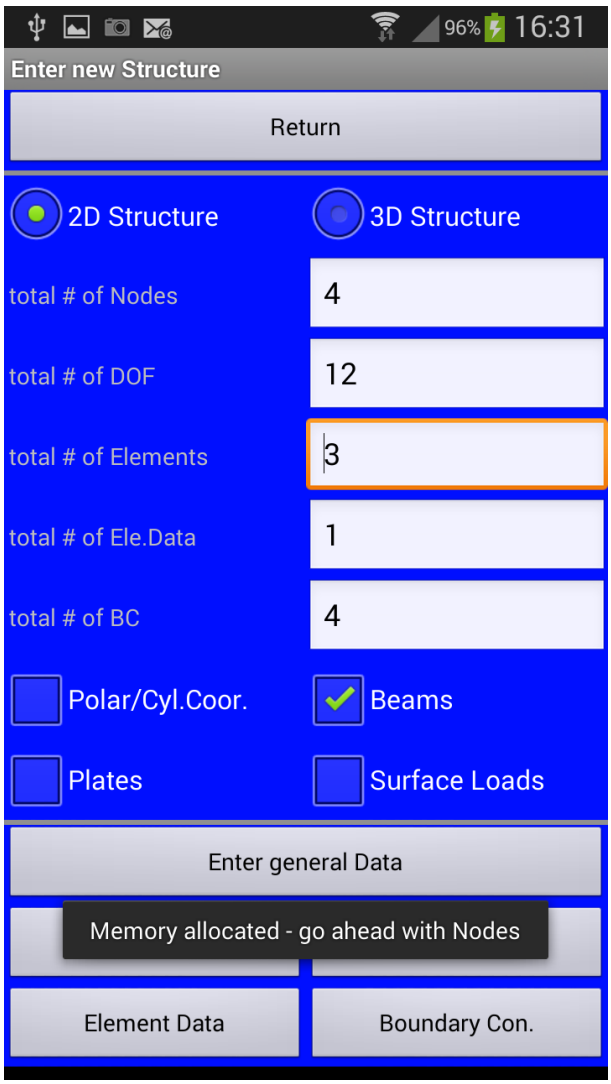
Now we'll solve this problem with Z88Tina:

In the main menu *Z88TINA* press *Enter new Structure* and enter now the general data, i.e.
- the total number of nodes = 4 for our example
- the total number of DOF (degrees of freedom) = 12 for our example because we want to use Beams No.13 (Beam in plane) with 3 DOF each times 4 nodes
- the total number of elements = 3 for our example
- the total number of Element data = 1 for our example because we'll use 3 beams with the same parameters
- the total number of BC (boundary conditions) = 4 for our example because we'll fix the first node in all the 3 DOF and enter a load onto node 4.
- set the appropriate flags, here *Beams* (internally IBFLAG). The flags *Polar/Cyl.Coor.* (internally KFLAG), *Plates* (internally IPFLAG) and *Surface Loads* (internally IQFLAG) don't apply here for this example. See chapters 3 & 4 for a detailled explanation of these flags.

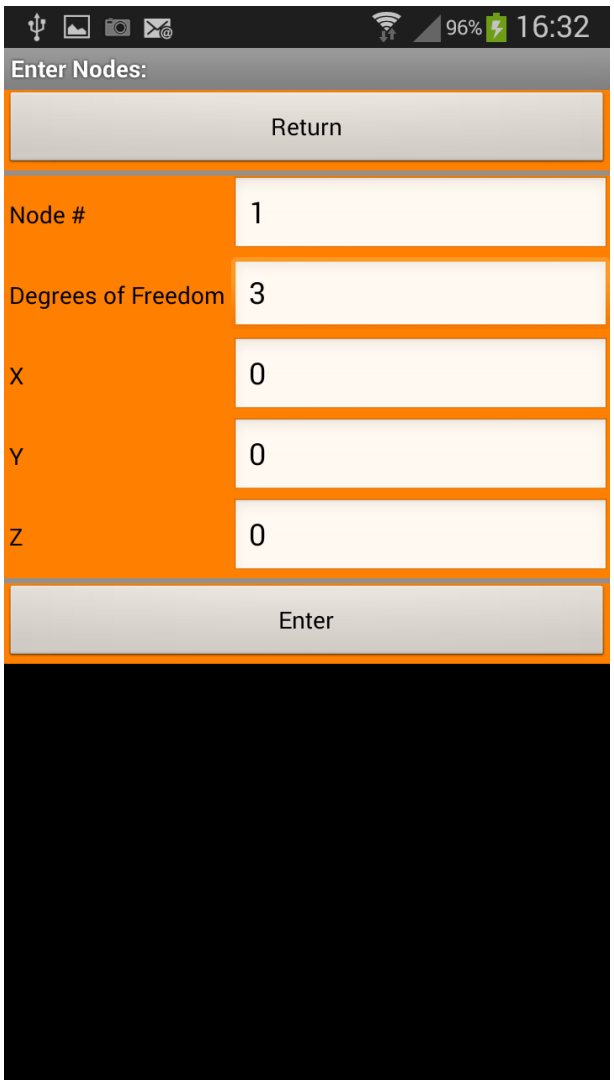
Press *Enter general Data* then.



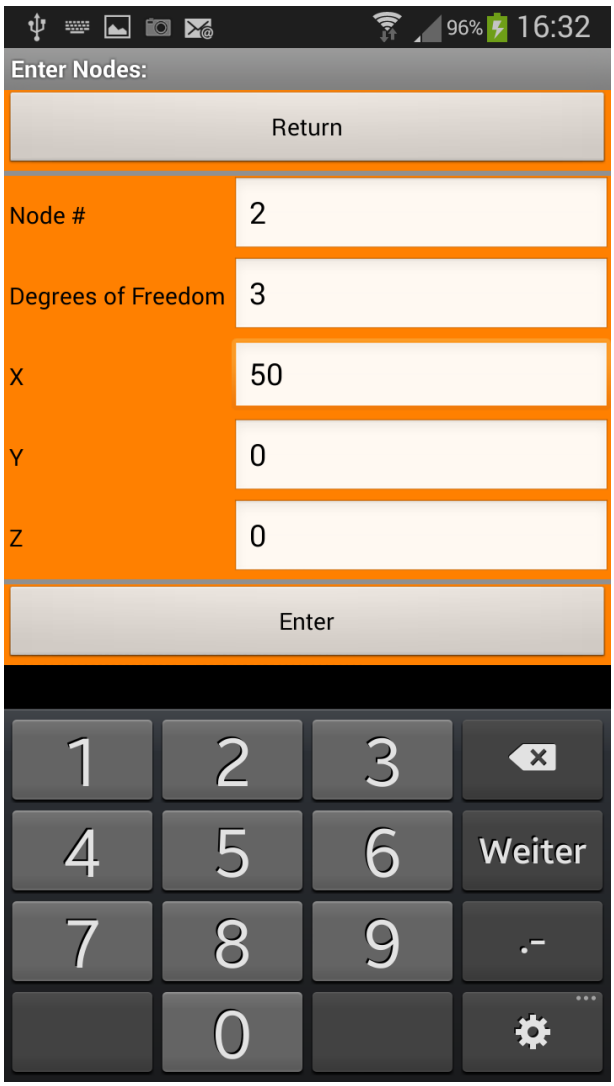
The default menu.



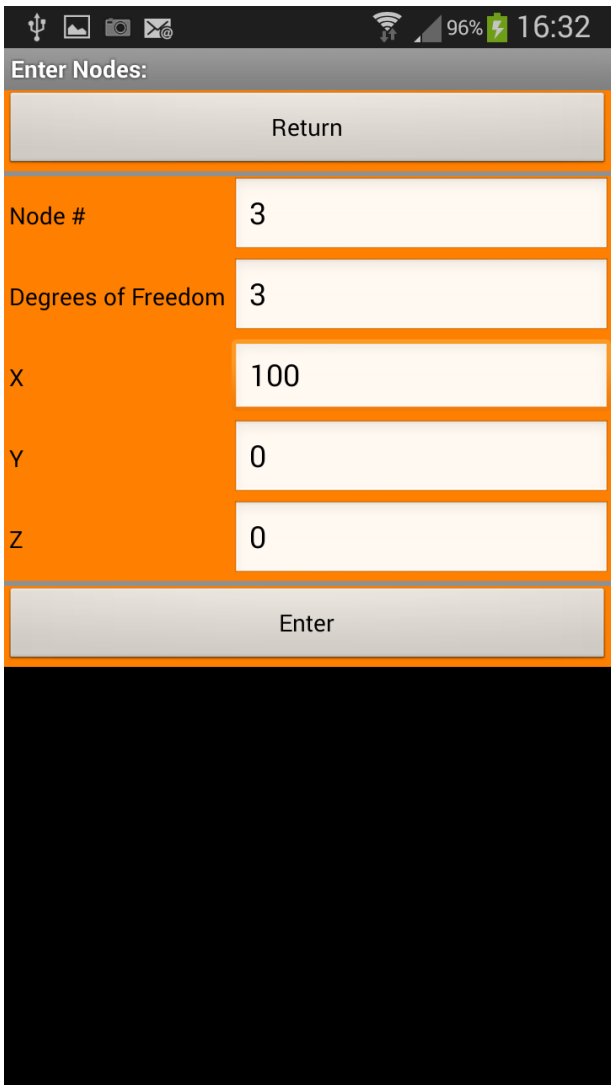
After entering the appropriate data press *Enter general Data* to allocate memory. Proceed with *Nodes*:



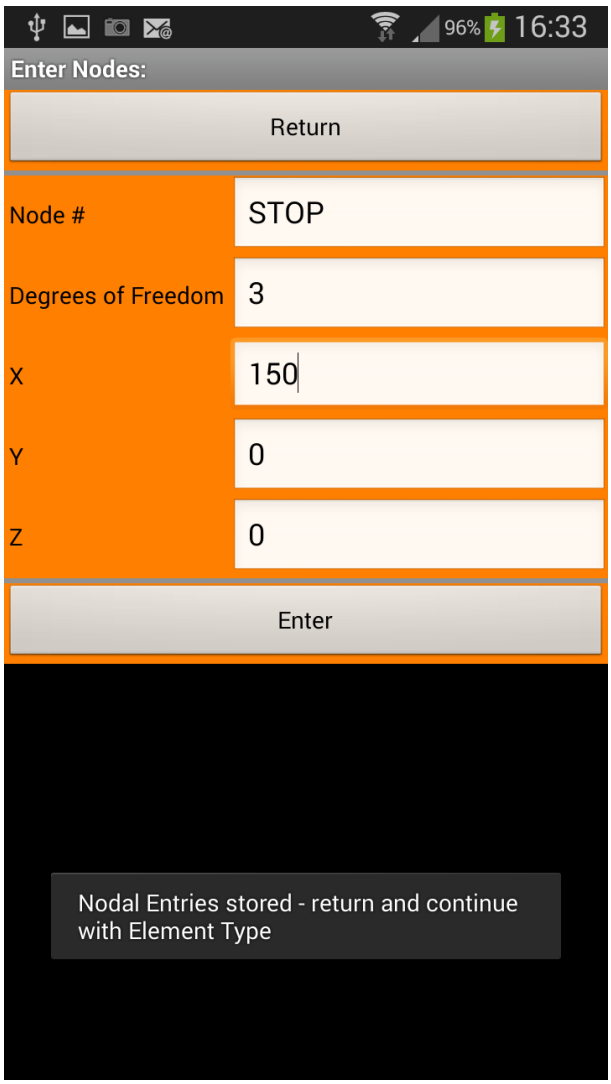
The first node. Press *Enter* after each node. The node counter is incremented (though you may enter your own node number if you made a mistake) up to the highest counter you entered one screen before for *total # of nodes*.



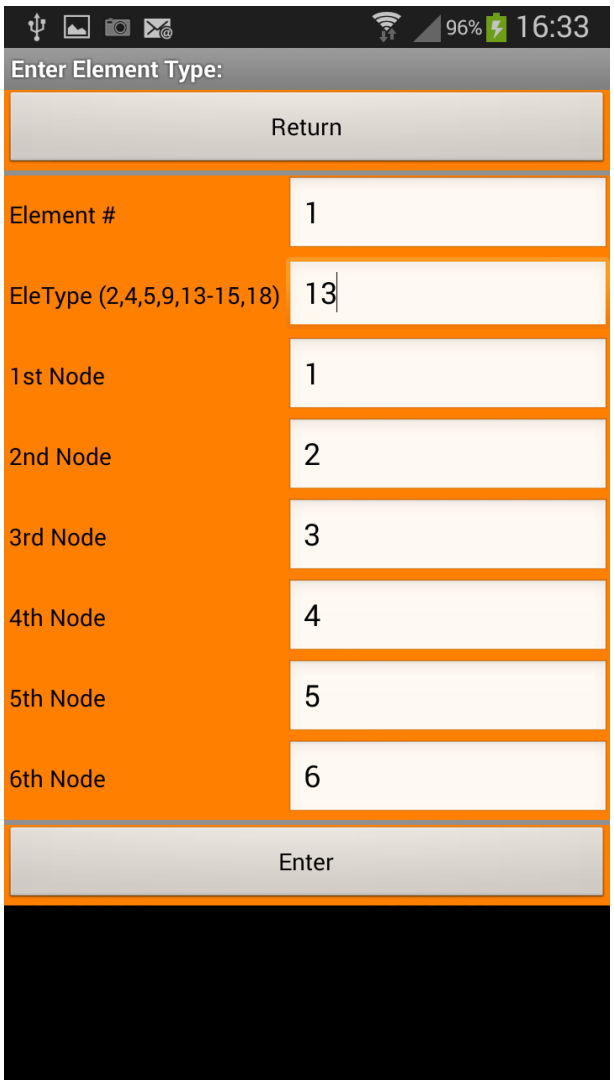
The second node.



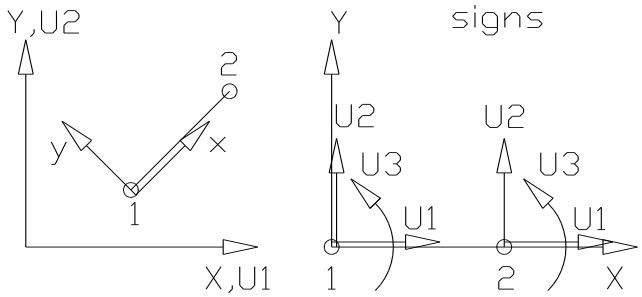
The third node.



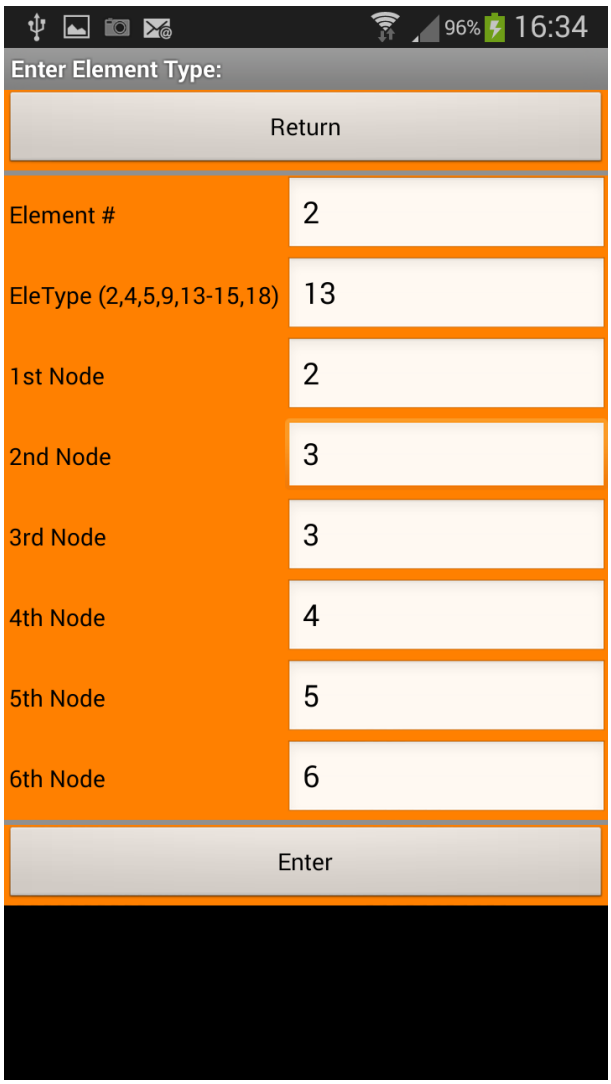
The forth node. Then *Return* to the *Enter new Structure* screen. Proceed with *Element Type*:



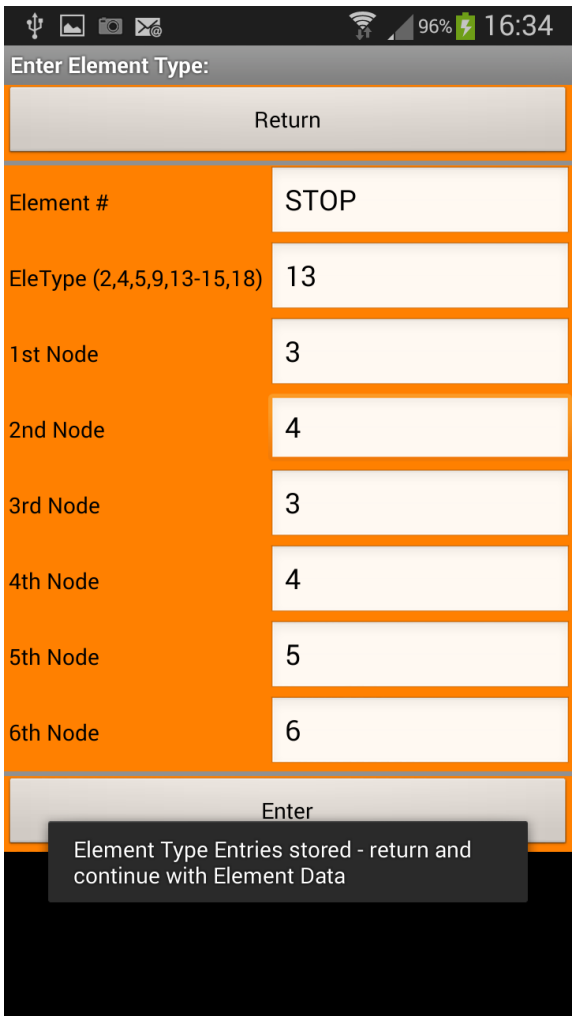
We'll use Beam No.13 in plane:



You may enter only the first two node, i.e. *1st Node* and *2nd Node* because a Beam No. 13 is defined by 2 nodes. Thus, don't care about the nodes 3 to 6.

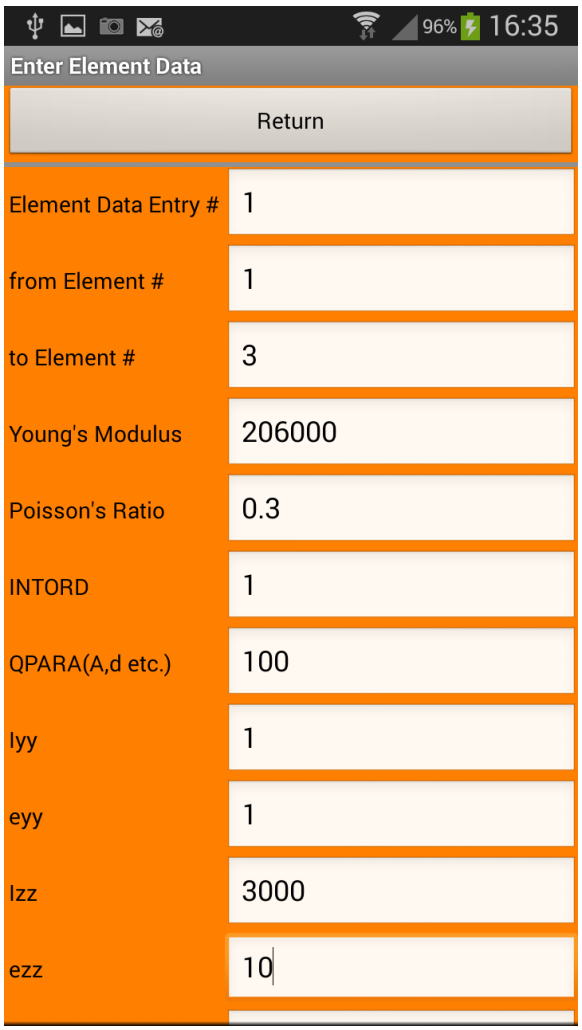


The second beam.



As you may see on the screen you'll get a hint after entering element 3 by pressing *Enter* for each element's coincidence that you may proceed with the next step. Thus, Return to the *Enter new Structure* screen and proceed with *Element Data*:

Please note that this menu is quite large, thus, you'll need to scroll.



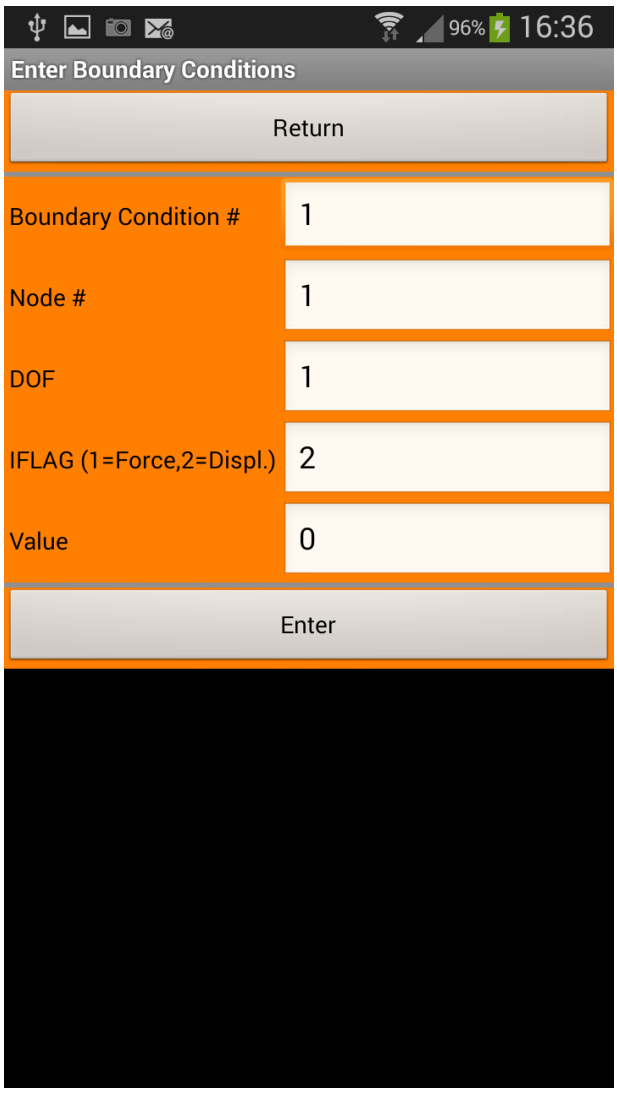
Here's why:

from element #	1	*this data set is valid from element 1*
to element #	3	*to element 3*
Young's Modulus	206000	
Poisson's Ratio	0.3	
INTORD	1 (*arbitrary)*	*arbitrary for beams*
QPARA	100	*for beams & trusses: cross-section area*
Iyy	1 (*arbitrary)*	*Second moment of inertia yy (bending around yy axis), applies only for Beam No.2 in space*
ezz	1 (*arbitrary)*	*max. distance from neutral axis zz, applies only for Beam No.2 in space*
Izz	3000	*Second moment of inertia zz (bending around zz axis)*
ezz	10	*max. distance from neutral axis zz*
It	1 (*arbitrary)*	*Second moment of area (torsion), only for Beam No.2*
Wt	1 (*arbitrary)*	*Second modulus (torsion), only for Beam No.2*

Remark: The Beams No.2, No.13 and Cam No.5 are so-called Bernoulli beams. Our full-scale FEA program Z88Aurora features in addition so-called Timoshenko beams.

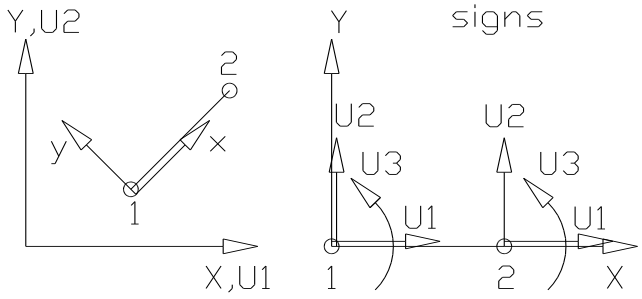
Very important: Scroll the menu down and press *Enter* to store your just entered data internally.

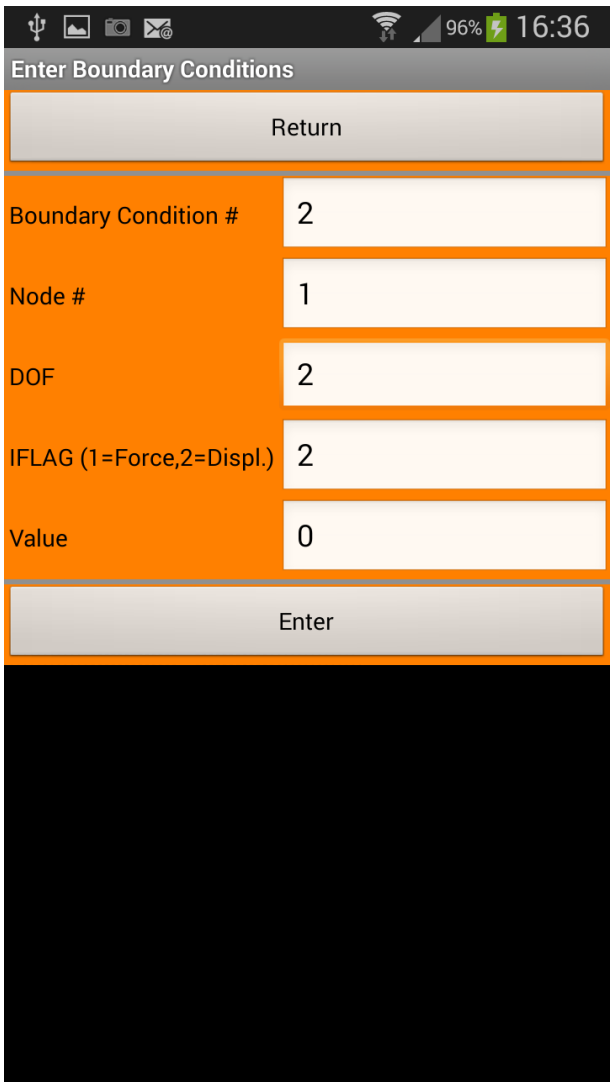
As we have here only one data set we may now *Return* to the *Enter new Structure* screen and proceed with *Boundary Con.*:



Here's why:

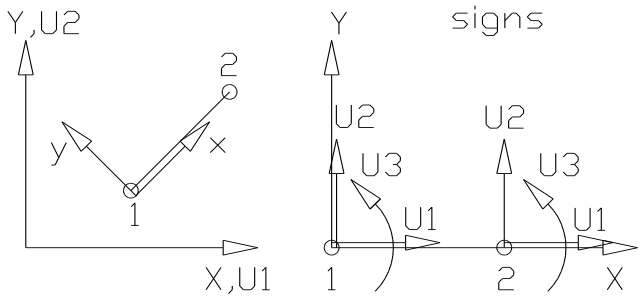
Boundary Condition #	1	*every BC needs an unique #, strictly ascending*
Node #	1	*the node with the BC*
DOF	1	*the degree of freedom of the node, here: DOF U1 of node 1*
ISFLAG	2	*2 means Displacement applied, 1 means Force applied*
Value	0	*a Displacement of 0 = fixed, i.e. DOF U1 of node 1 is fixed*

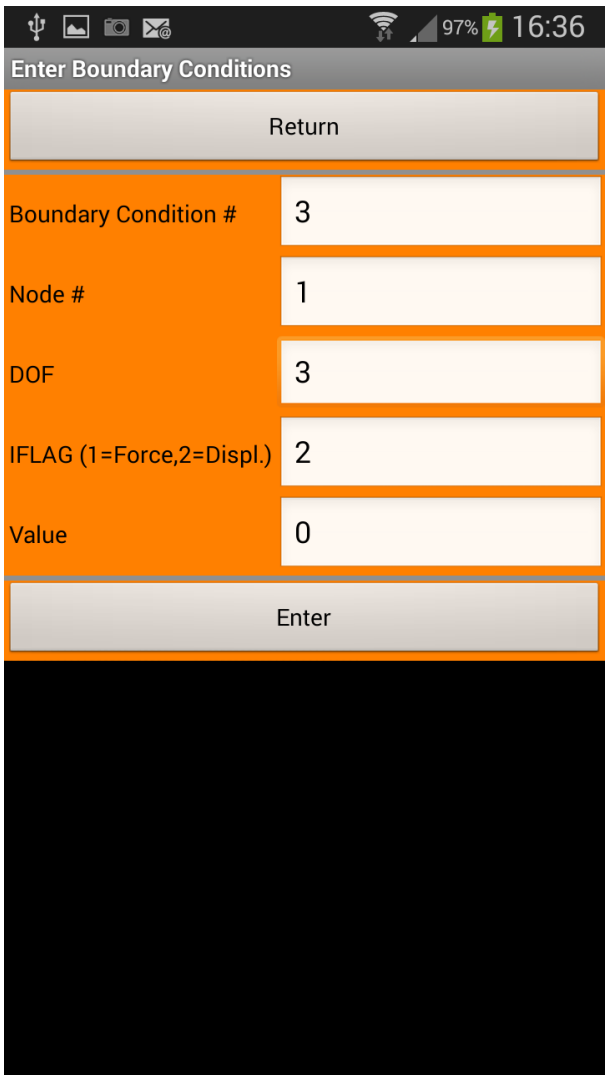




Here's why:

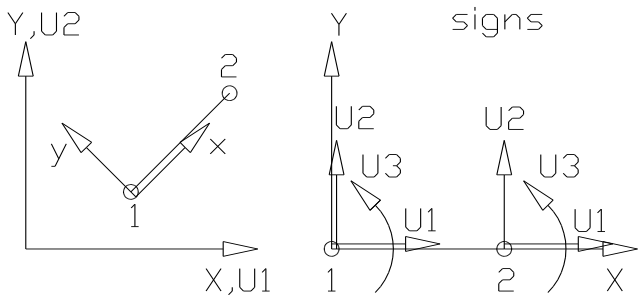
Boundary Condition #	2	*every BC needs an unique #, strictly ascending*
Node #	1	*the node with the BC*
DOF	2	*the degree of freedom of the node, here: DOF U2 of node 1*
ISFLAG	2	*2 means Displacement applied, 1 means Force applied*
Value	0	*a Displacement of 0 = fixed, i.e. DOF U2 of node 1 is fixed*

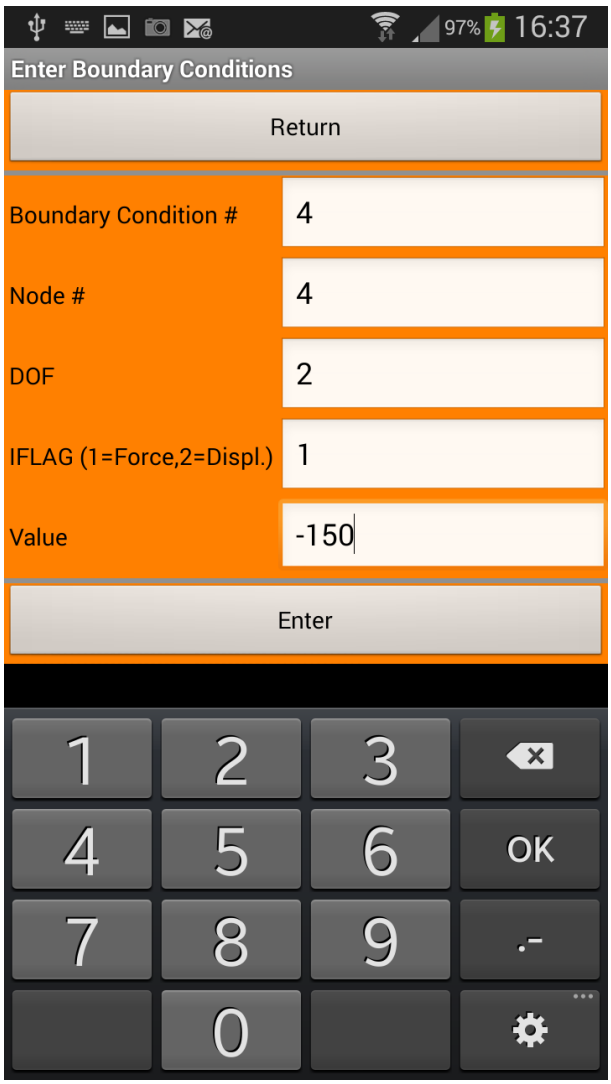




Here's why:

Boundary Condition #	3	*every BC needs an unique #, strictly ascending*
Node #	1	*the node with the BC*
DOF	3	*degree of freedom of the node, here: DOF U3 of node 1*
ISFLAG	2	*2 means Displacement applied, 1 means Force applied*
Value	0	*a Displacement of 0 = fixed, i.e. DOF U3 of node 1 is fixed*

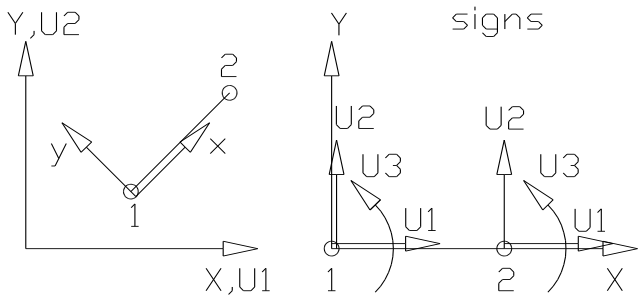




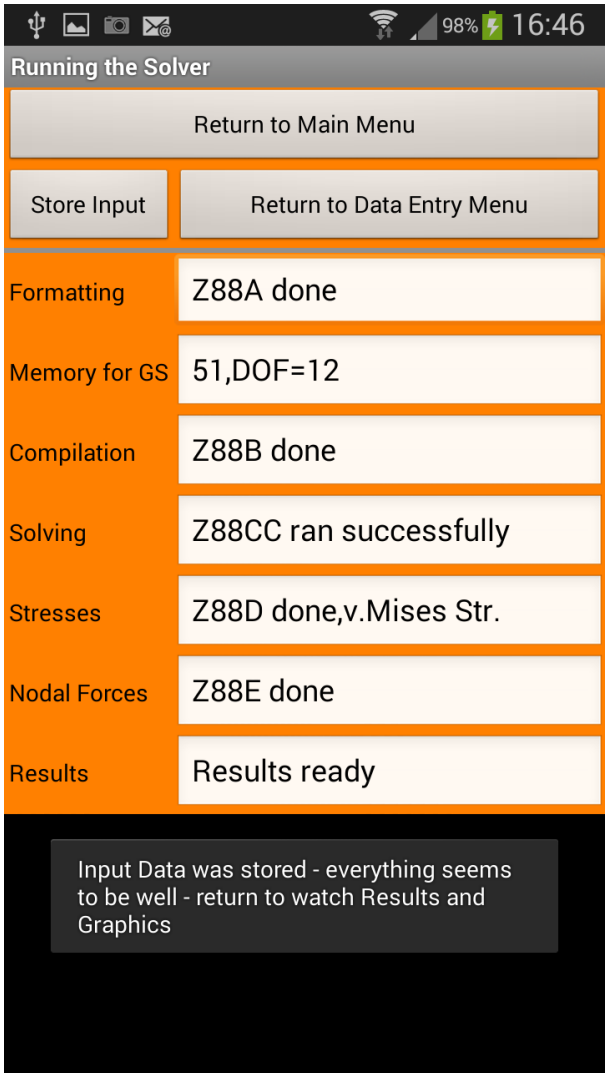
How to enter a negative sign? Double-tap the key .-

Here's why:

Boundary Condition #	4	*every BC needs an unique #, strictly ascending*
Node #	4	*the node with the BC*
DOF	2	*degree of freedom of the node, here: DOF U2 of node 1*
ISFLAG	1	*1 means Force applied, 2 means Displacement applied*
Value	-150	*a Force of 150 applied onto DOF U2 of node 4*



After entering the last BC, *Return* to *Enter new Structure* menu and *Return* to *Z88TINA* main menu and launch the *Solver*:



Looks good, thus, press *Store Input*: Your input data will be stored in the two files

- Z88I1.TXT : the structure data
- Z88I2.TXT : the boundary conditions

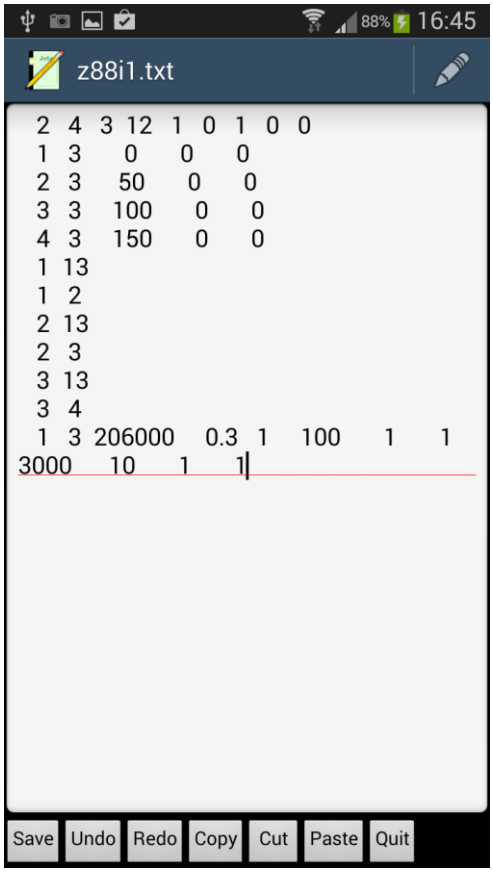
in the *Working Directory* (usually /storage/sdcard0/z88) you selected in the *Z88TINA* main menu.

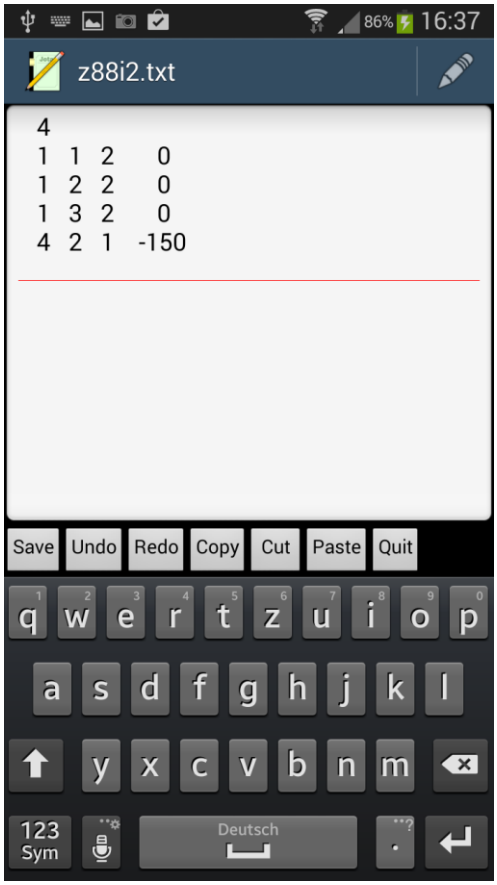
If the Solver ran successfully, the file Z88O2.TXT containing the displacements is stored, too, in the *Working Directory*. You may now *Return to Main Menu* and watch the *Results* and the *Graphics*.

What if the Solver collapsed? You may now press *Return to Data Entry Menu* (note: this works only if you did not press *Store Input* because then the files are closed!) and enter better data: If a FEA Solver collapses means in many cases wrong or too few boundary conditions especially for beams.

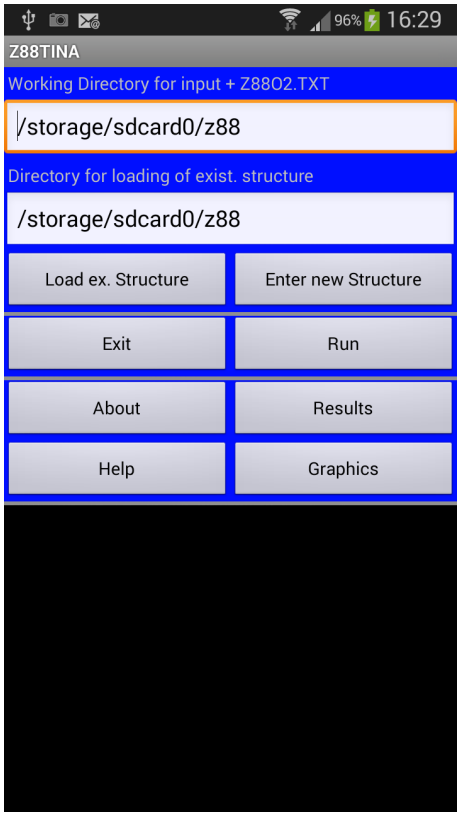
And quite freaky: Or you may *Store Input* and check and edit the input files Z88I1.TXT and Z88I2.TXT *directly*. How? Download any txt-editor from the Google PlayStore (for example: JOTA, DroidEdit etc.). The format of these two files is descriped precisely in chapter 3. Be sure to operate with the correct coding standard when editing: As Android is an UNIX system the

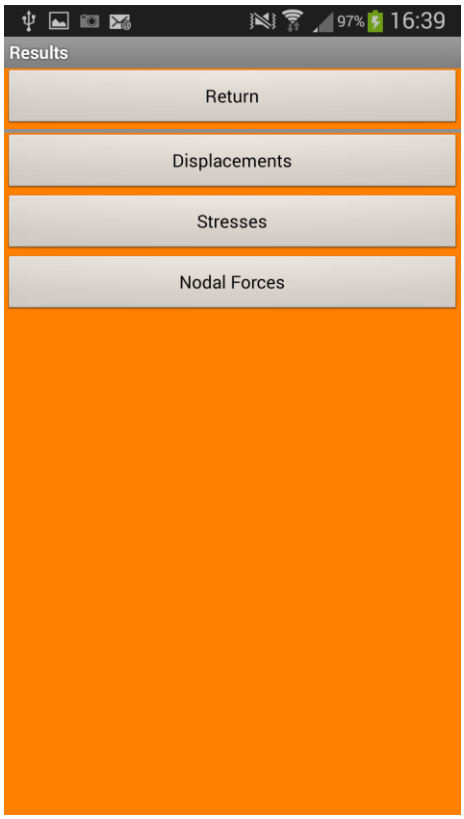
coding standards for such files are ASCII or ISO 8859-1 or Windows 1252. Here, the two files look as follows:

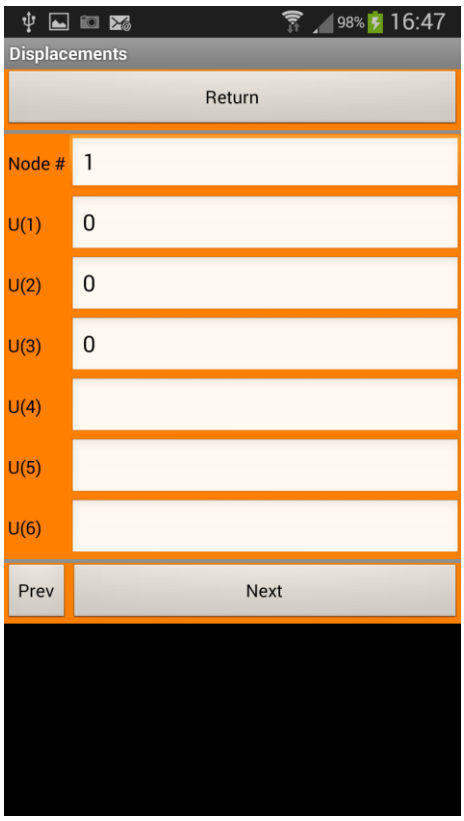




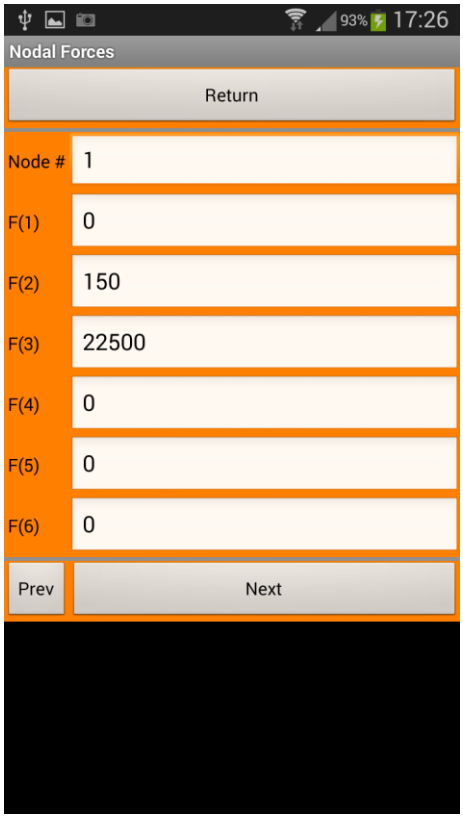
Let us continue with some *Results*:



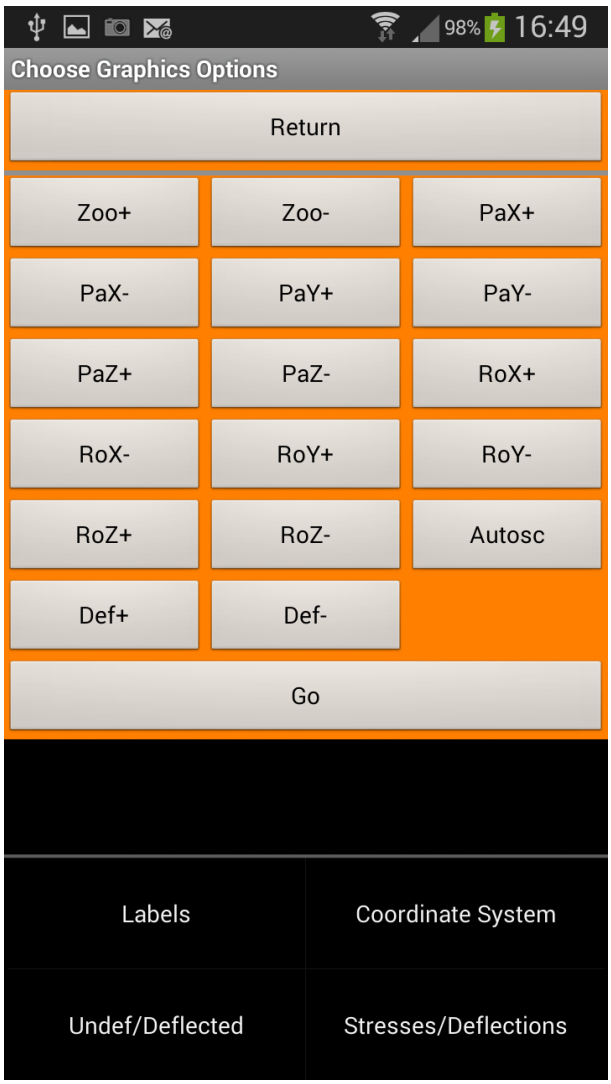



98% 16:47
Displacements
Return
Node # 1
U(1) 0
U(2) 0
U(3) 0
U(4)
U(5)
U(6)
Prev
Next

93% 17:25
Displacements
Return
Node # 4
U(1) 0
U(2) -0.27305825
U(3) -0.0027305825
U(4)
U(5)
U(6)
Prev
Next



And now the *Graphics*:



You should know:
1. You'll return from the OpenGL screen by just tapping it.
2. The graphics of Z88Tina are optimized for *vertical* position, thus, hold your smartphone or tablet in a vertical position.
3. Tapping on Android's menu "button" left below let you see this menu directing you to 4 submenus which are quite self-explaining.
4. You first "build" an OpenGL scene with this menu and then you *Go*. If in doubt just press *Autosc* and *Go* and watch the scene. Then apply improvements. The buttons are:

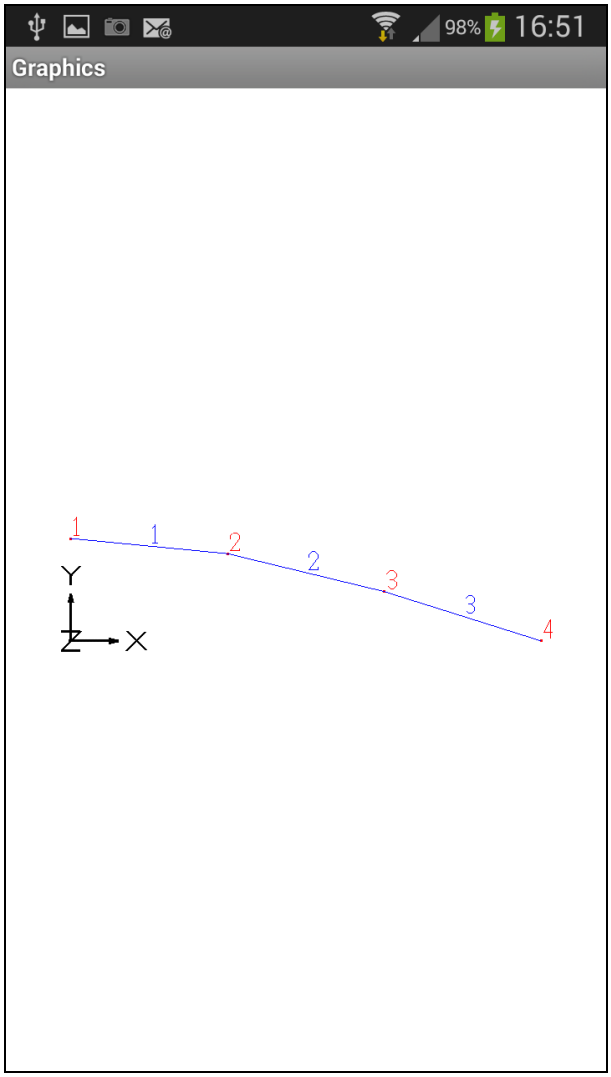
Zoo+: increase zoom
Zoo-: decrease zoom

PaX+: increase panning in X direction
PaX-: decrease panning in X direction
… just the same for Y and Z

RoX+: increase rotation around the X axis by 30°
RoX- : decrease rotation around the X axis by -30°
… just the same for Y and Z axis

Def+ : increase the enlargement of the deflections (default is 100)
Def- : decrease the enlargement of the deflections

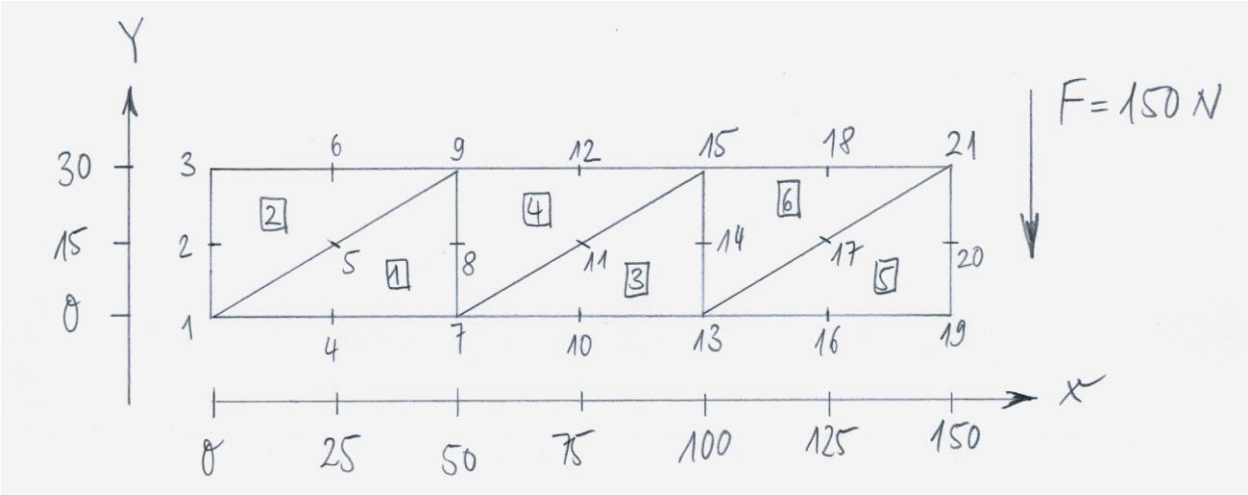
Autosc(ale) : fit the structure in the screen, set all graphics parameters to default values
5. The OpenGL screen does not react on gestures with the one exception of tapping because there are too few gestures defined in Android for stearing a real, full-scale 3D graphics program.



Here's a scene with the 3 beams deflected (note: Z88Tina connects the nodes always with straight lines for speed reasons – thus, you see here no real cubic bending curve) and labeled.

2.2 ENTER A NEW STRUCTURE – 2ND EXAMPLE

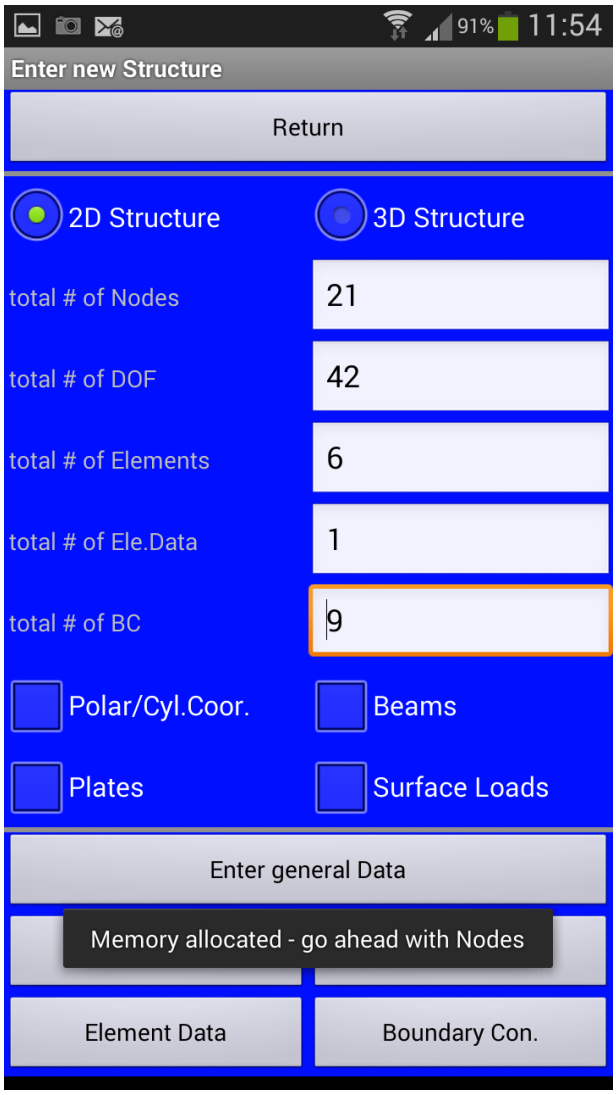
Let us assume the following structure of Plane Stress Elements No.14 – the sketch below was made old-fashioned by hand with paper and pencil; took 10 minutes:

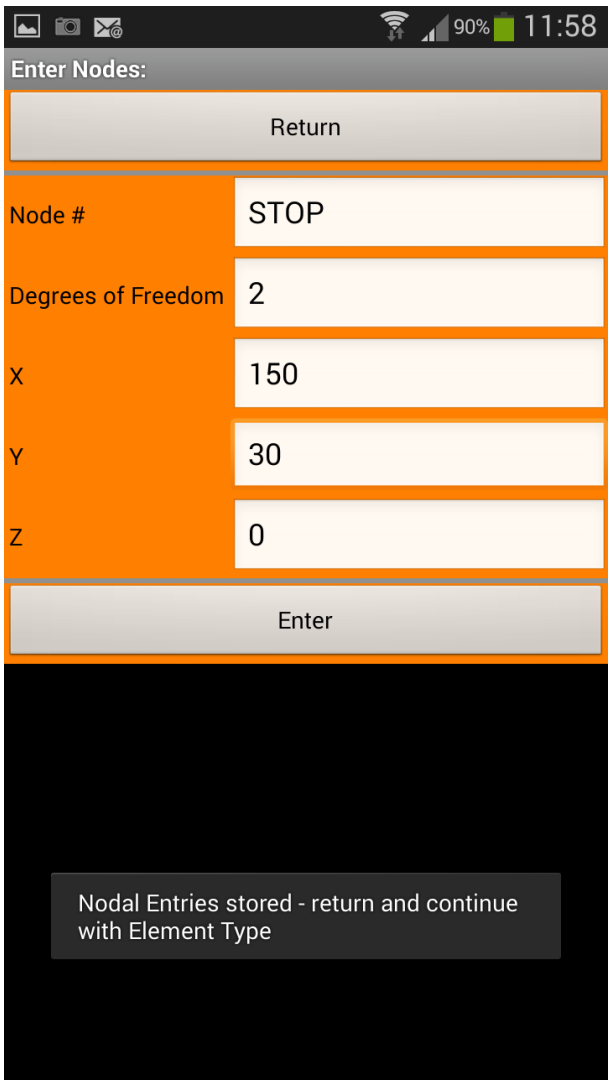


with these parameters:

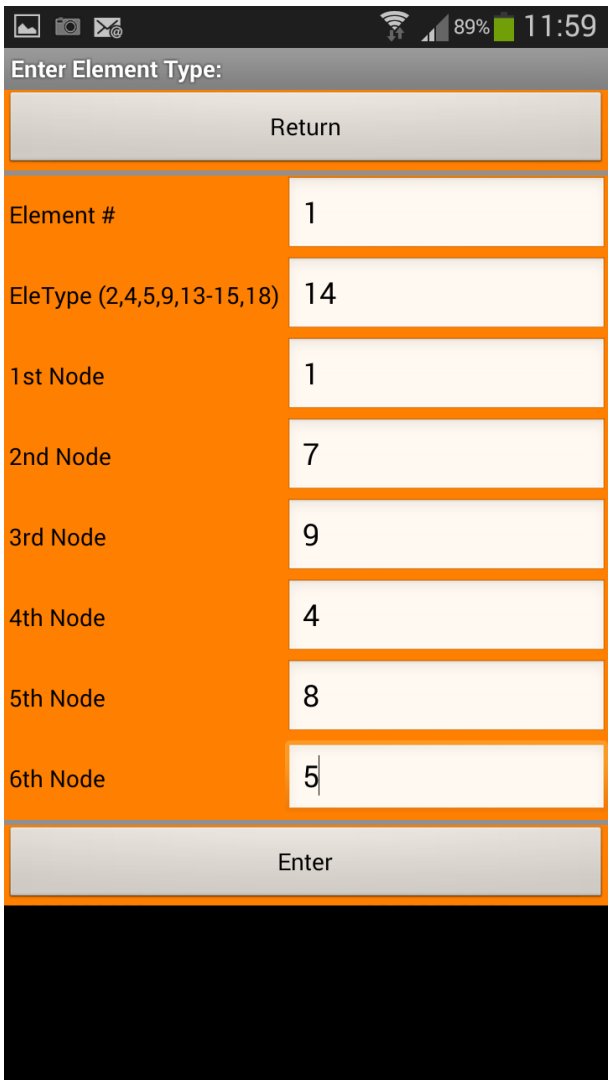
$$E = 206000 \frac{N}{mm^2}, \nu = 0.3, t = 1.3333mm$$

This structure implements a “beam” similar to chp. 2.1, thus, it will be of interest if there will be the same results. A serie of screenshots should give you some advise:

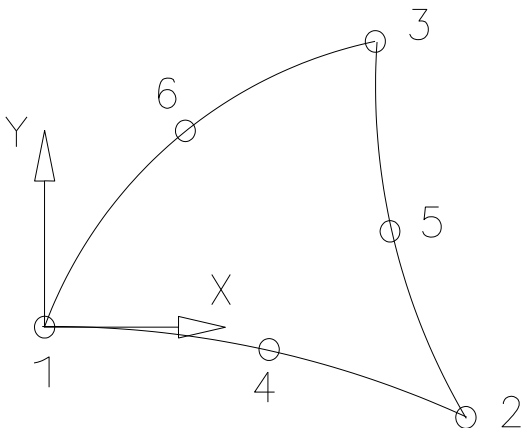


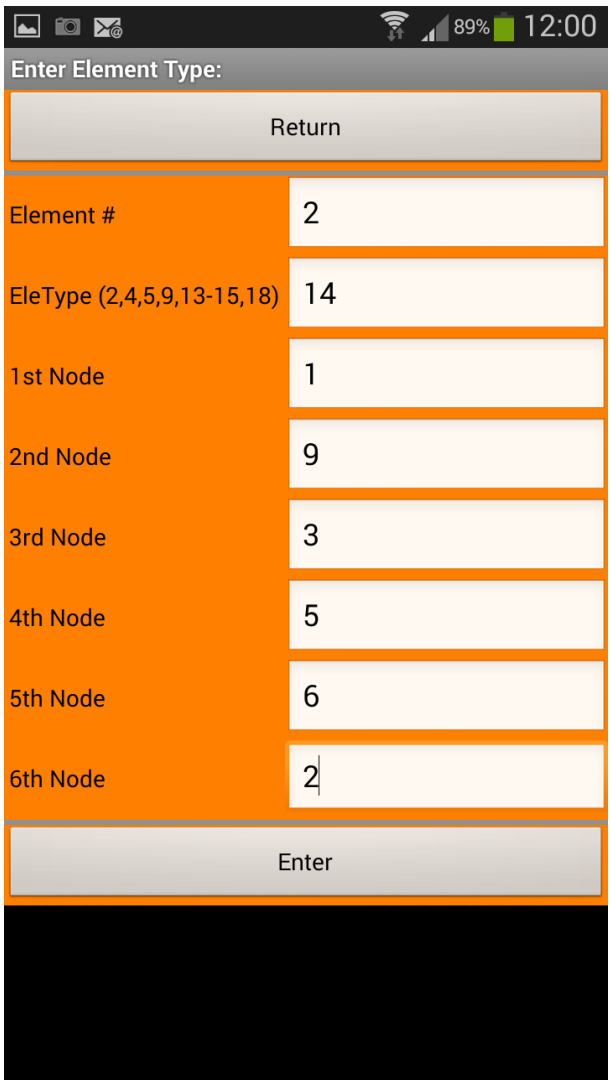


To enter the 21 nodes is fairly easy.

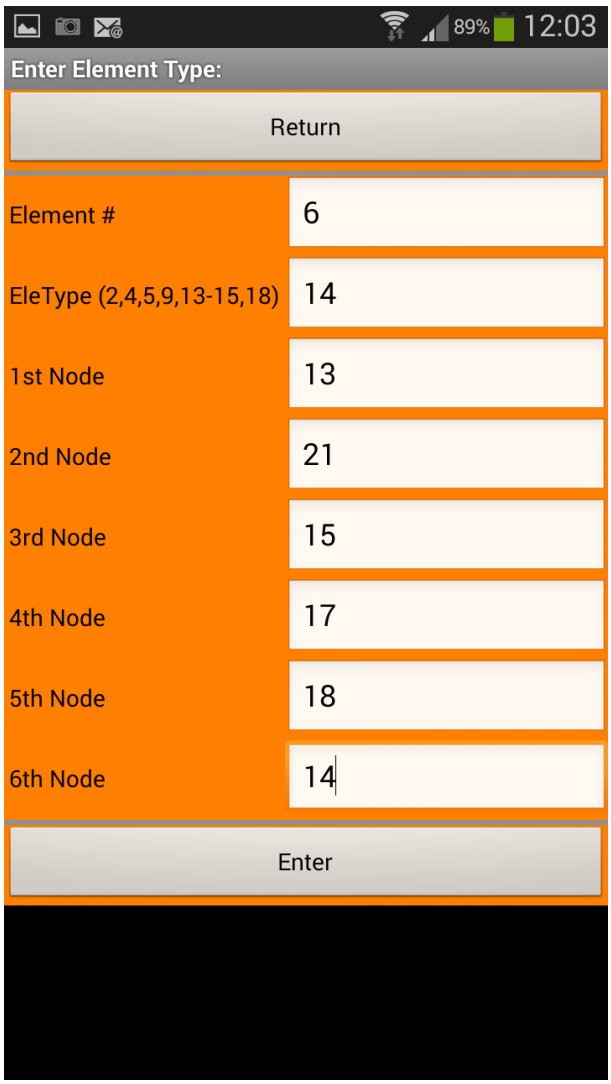


The first element. Please keep in mind to enter at first the 3 corner nodes and then the 3 middle nodes. You may declare any of the six nodes to 1st node, then enter the next corner node and so on – **counterclockwise**! Thus, you could have entered the coincidence of element 1 also as 3 – 1 – 2 – 6 – 4 – 5 .

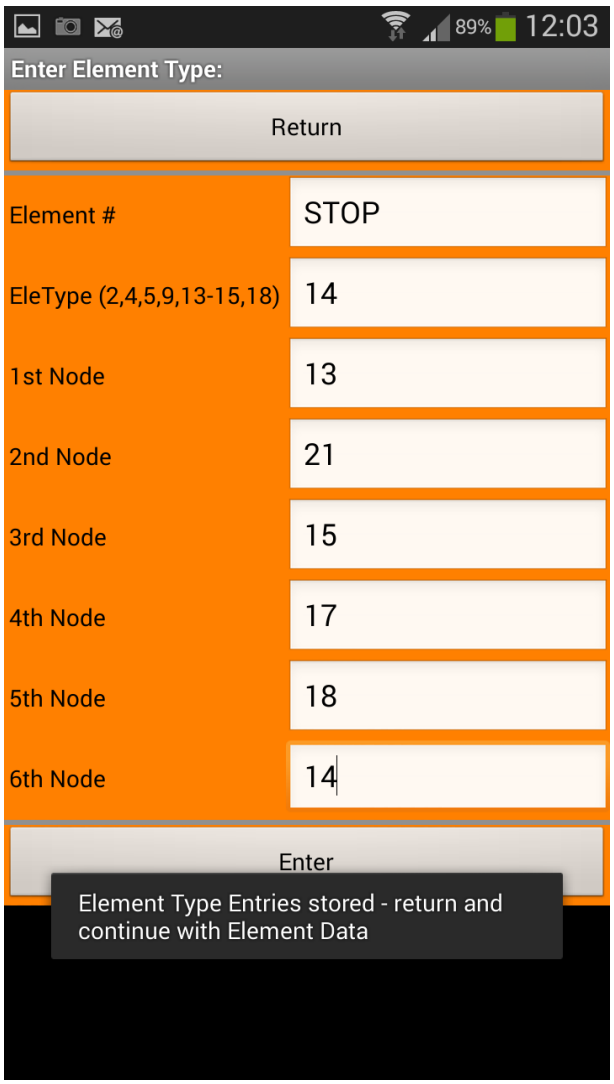




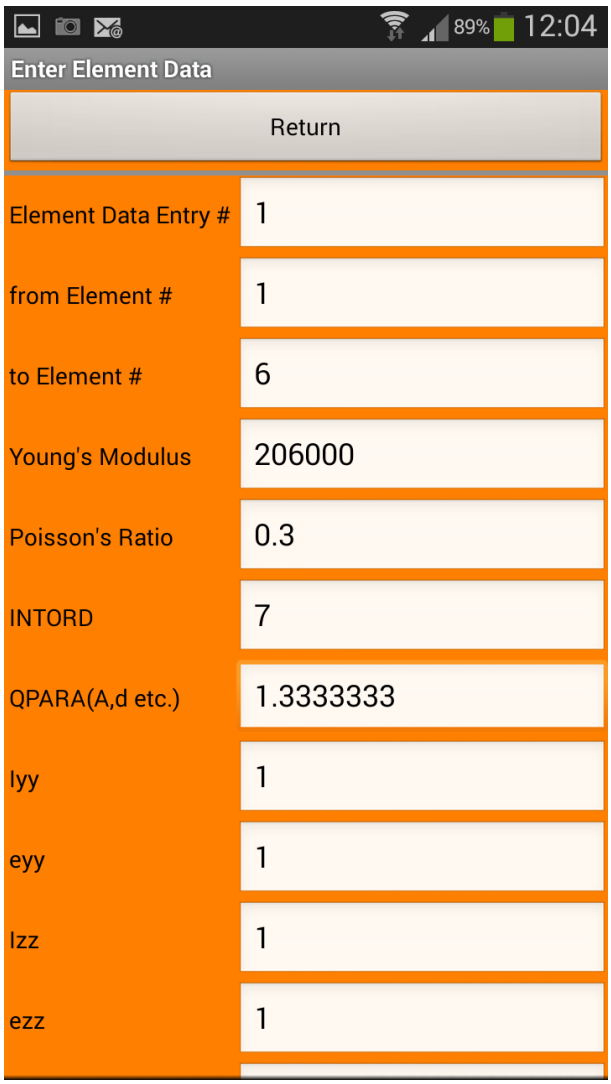
Coincidence for element 2. I think you now know how to enter elements 3 ~ 6 and I'll present the screenshot for element 6:



… just before *Enter*

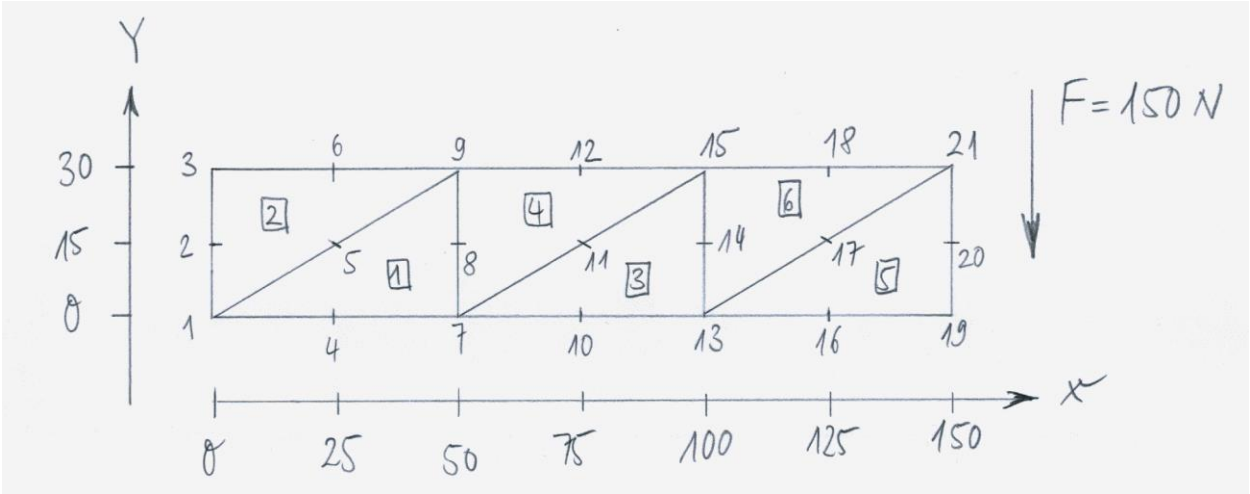


We may now enter the element parameters. As all the 6 elements feature a thickness of 1.3333 mm (Why? Crazy thickness! Just because the height of 30 mm and the thickness of 1.3333 mm makes an Izz of 3000 mm^4)

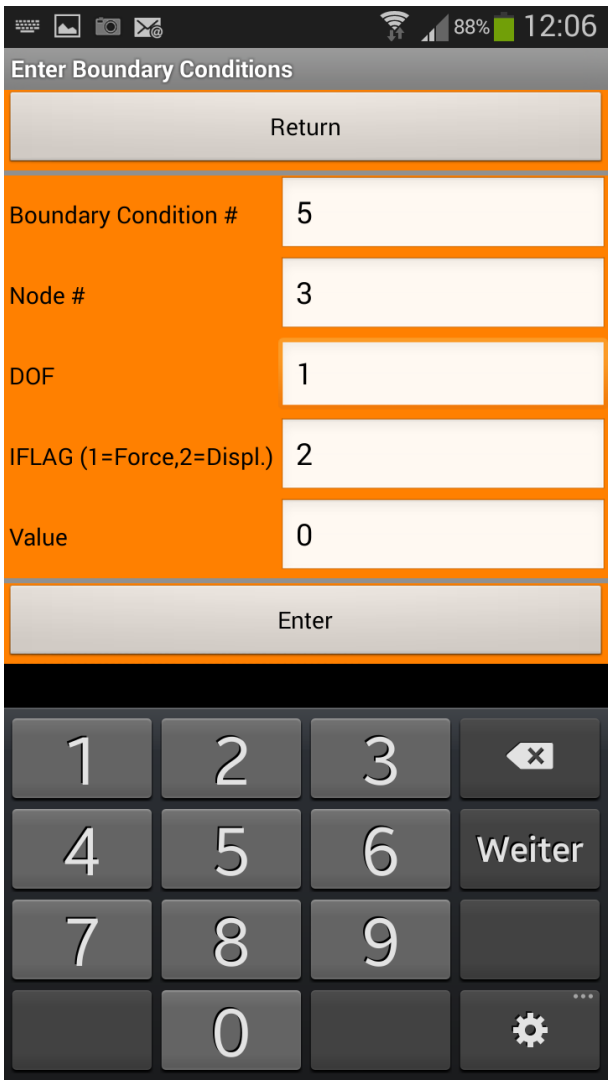


Don’t forget to scroll down to press *Enter* :

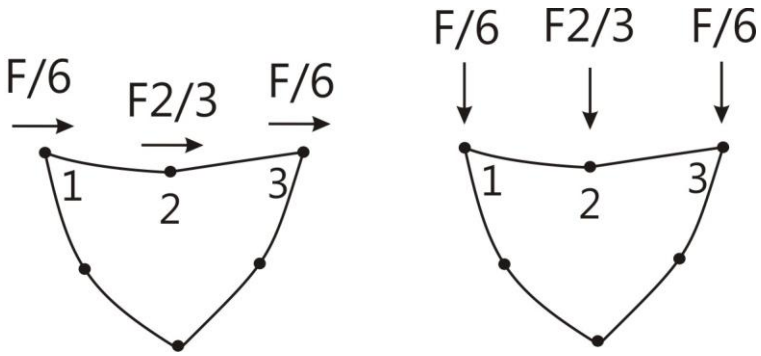
Let us continue with the *Boundary Conditions*:



However, we'll need to fix the DOF 1 & 2 of the nodes 1,2 and 3 each. The load will be distributed onto the nodes 19, 20 and 21. Here's a screenshot of DOF 1 of node 3:

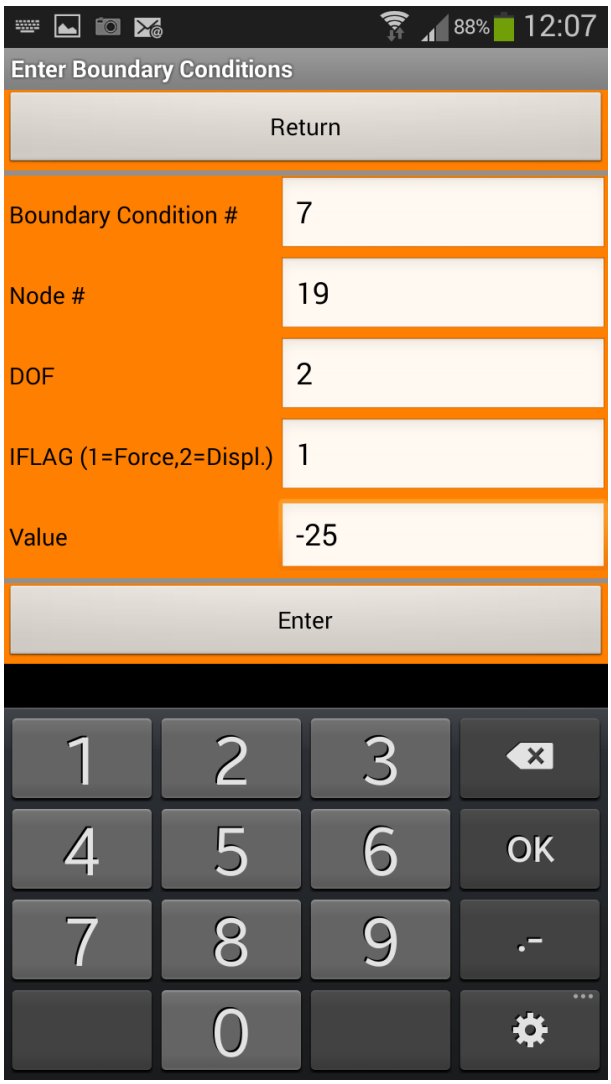


The distribution of the load is tricky: You'll need to distribute a force put on a side of an element in this way (see chp. 3.3 for more information):

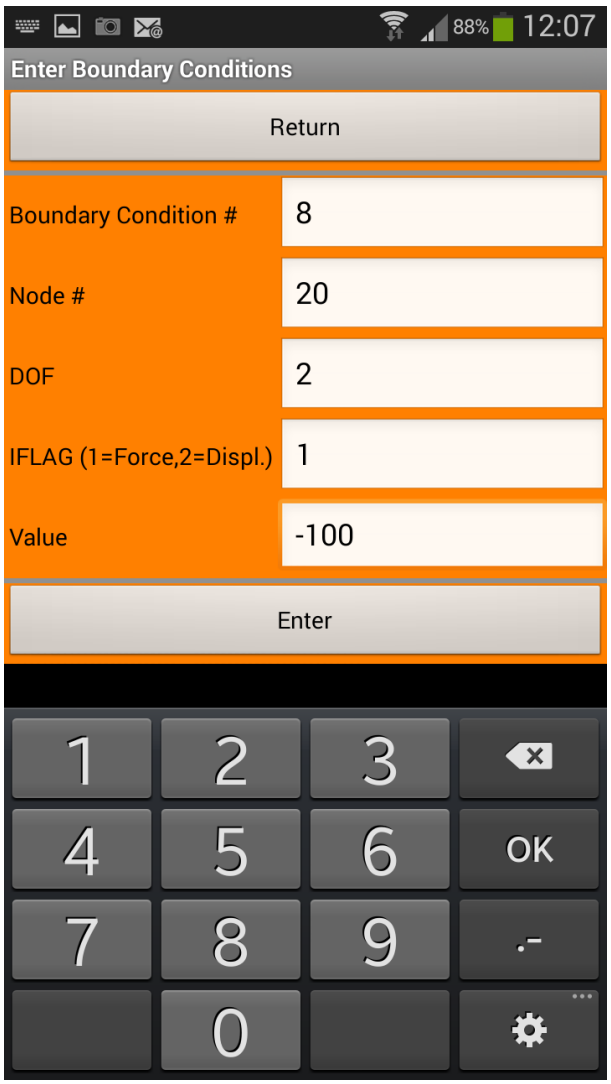


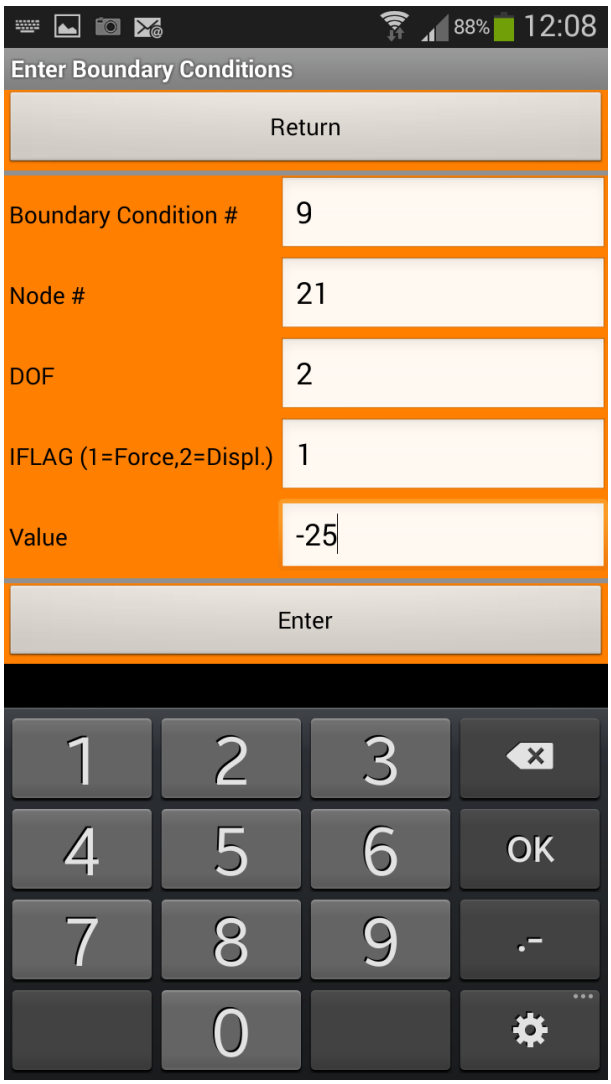
Of course, it would be possible to put the whole load on only one single node either node 19 or node 20 or node 21 but this would lead to a stress concentation in this single point – not recommended!

Thus:

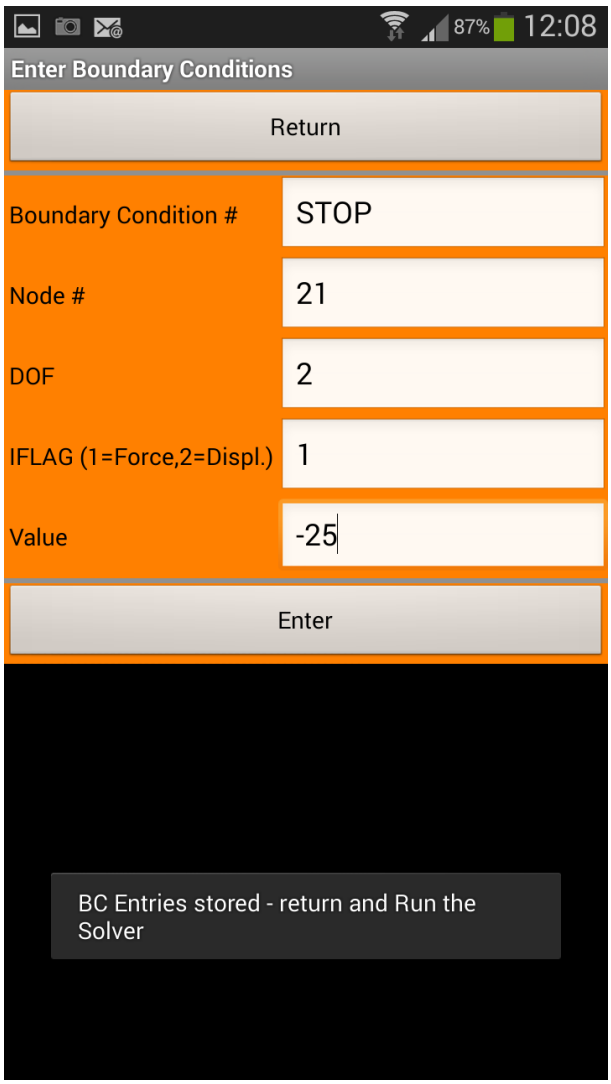


How to enter the – sign? Double-tap onto the key .-

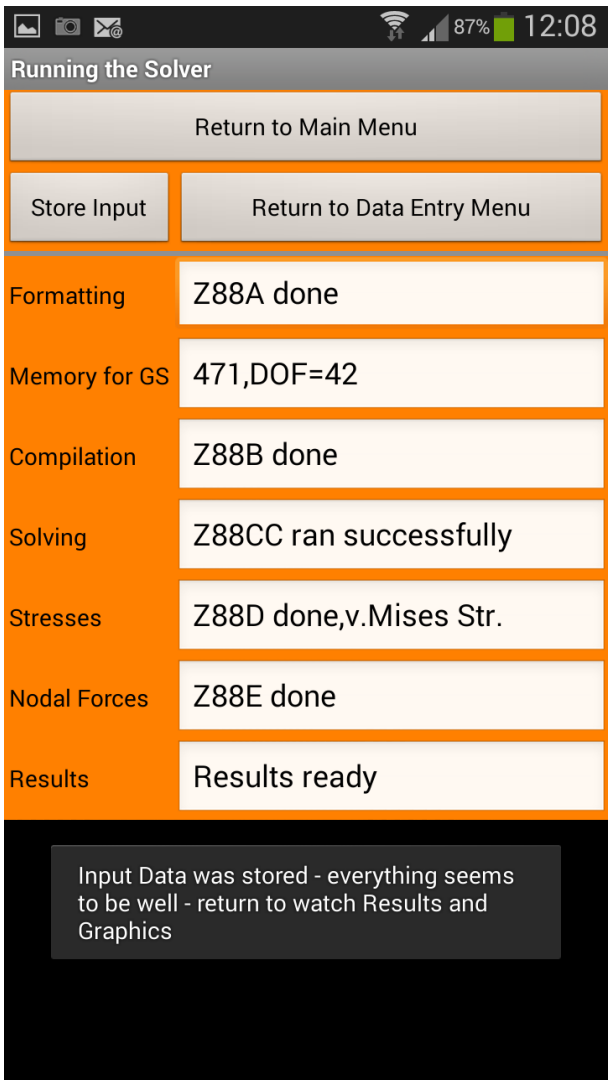

88% 12:07
Enter Boundary Conditions
Return
Boundary Condition # 8
Node # 20
DOF 2
IFLAG (1=Force,2=Displ.) 1
Value -100
Enter
1 2 3
4 5 6 OK
7 8 9 .-
0



Just before *Enter*



We may now run the *Solver*:



As the Solver ran successfully we may now *Store* the *Input*. If you want to take a look to these two files Z88I1.TXT and Z88I2.TXT you may do this by any editor (see chp. 3 for a details of these files):

Z88I1.TXT:

```
2   21   6   42   1   0   0   0   0
 1   2    0    0
 2   2    0   15
 3   2    0   30
 4   2   25    0
 5   2   25   15
 6   2   25   30
 7   2   50    0
 8   2   50   15
 9   2   50   30
10   2   75    0
11   2   75   15
12   2   75   30
13   2  100    0
14   2  100   15
15   2  100   30
16   2  125    0
```

```
17  2 125  15
18  2 125  30
19  2 150   0
20  2 150  15
21  2 150  30
1  14
1  7  9  4  8  5
2  14
1  9  3  5  6  2
3  14
7  13  15  10  14  11
4  14
7  15  9  11  12  8
5  14
13  19  21  16  20  17
6  14
13  21  15  17  18  14
1  6  206000  0.3  7  1.333333  0  0  0  0  0  0
```

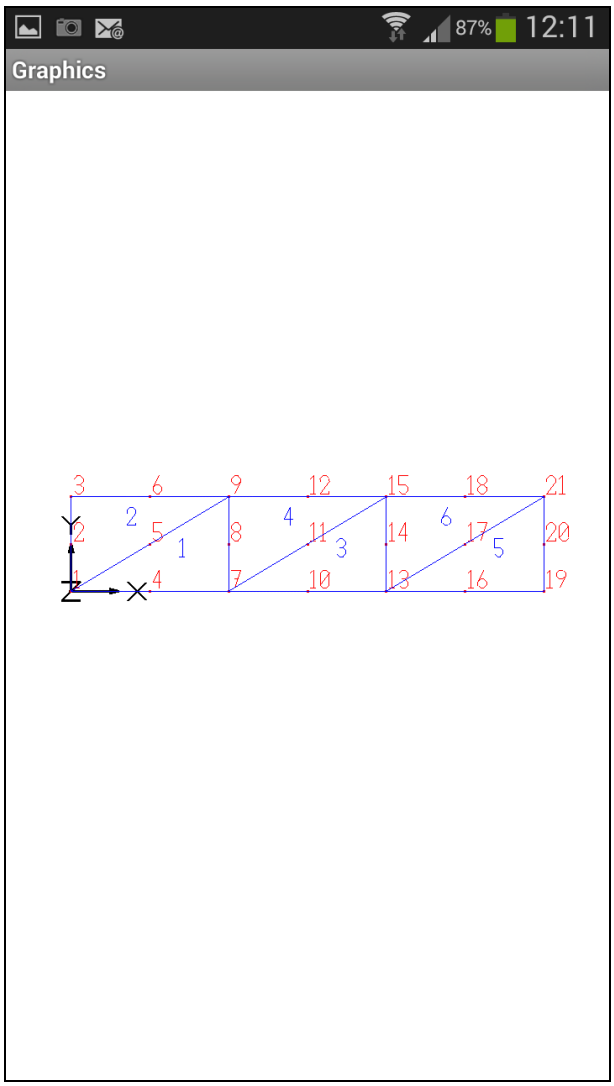
Z88I2.TXT

```
9
 1  1  2  0
 1  2  2  0
 2  1  2  0
 2  2  2  0
 3  1  2  0
 3  2  2  0
19  2  1  -25
20  2  1  -100
21  2  1  -25
```

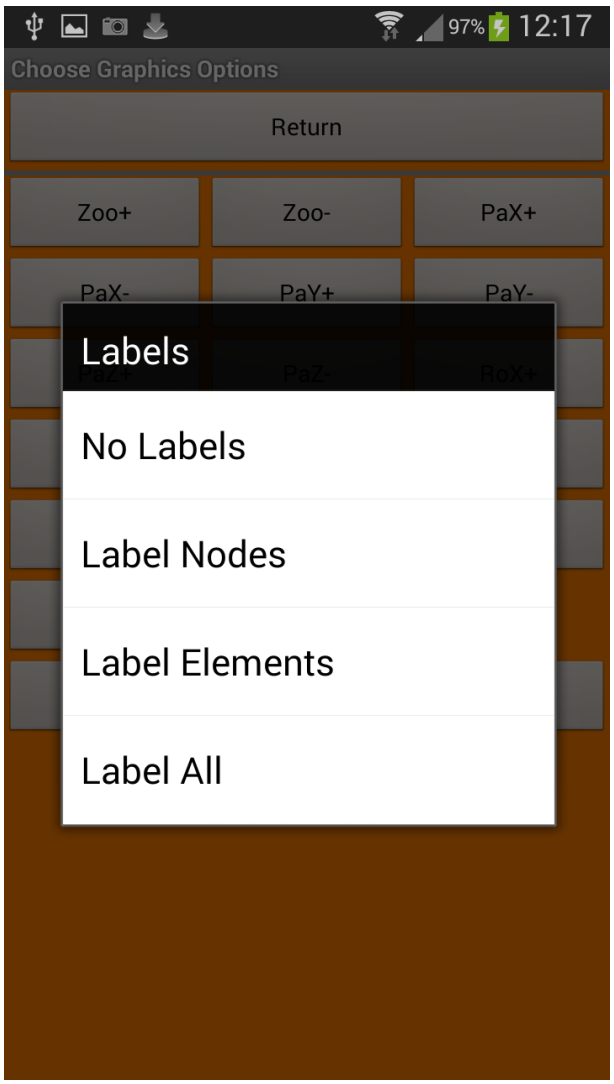
If you want to keep these files for later use you should move or copy them to a (new) directory other than the working directory. For control reasons, the file Z88O2.TXT – the displacements – are stored in your working directory.

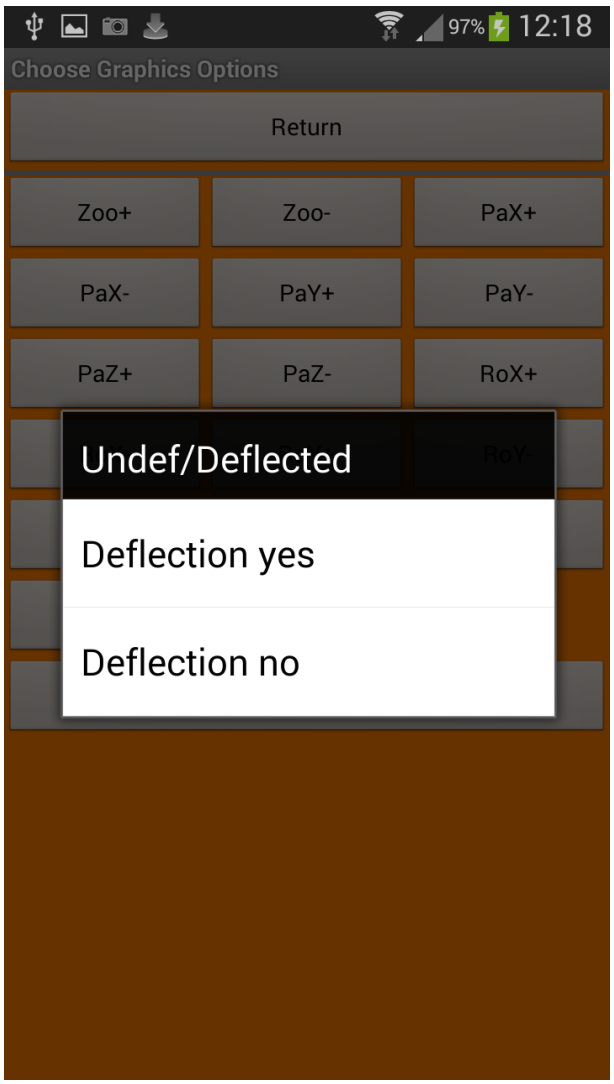
Now some results:

The displacement of -0.27 mm fits quite good to the result of the beam problem chp. 2.1.
If you'll add the Nodal Forces of the nodes 1, 2 and 3 in vertikal direction you'll notice that 337 – 596 + 409 makes 150 – the raction force on our load of 150N.

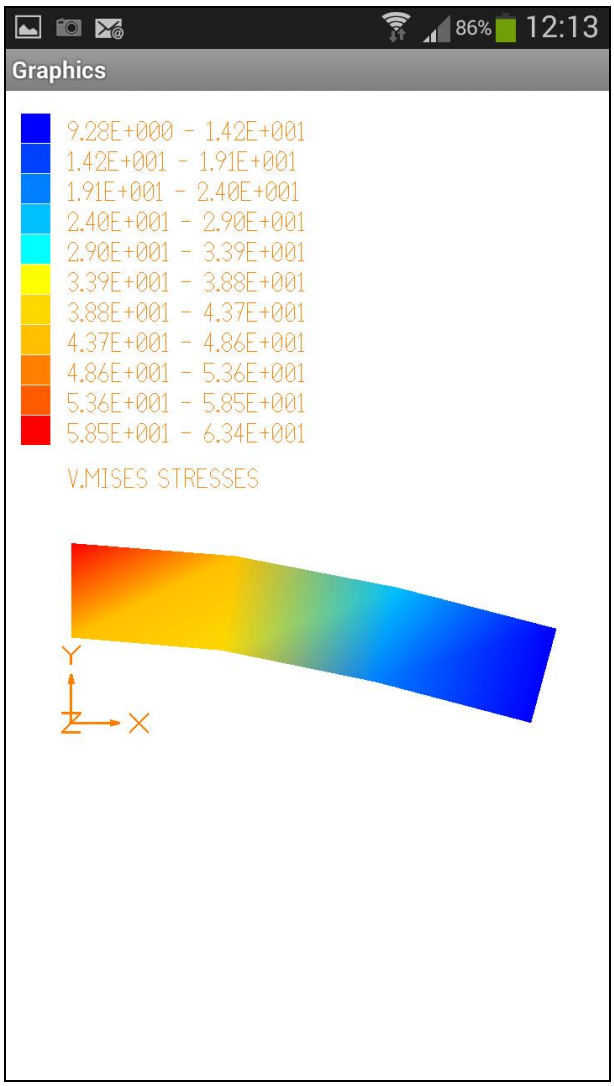


A plot of the structure with nodes and elements labeled – use the menu left-side of the Home button.

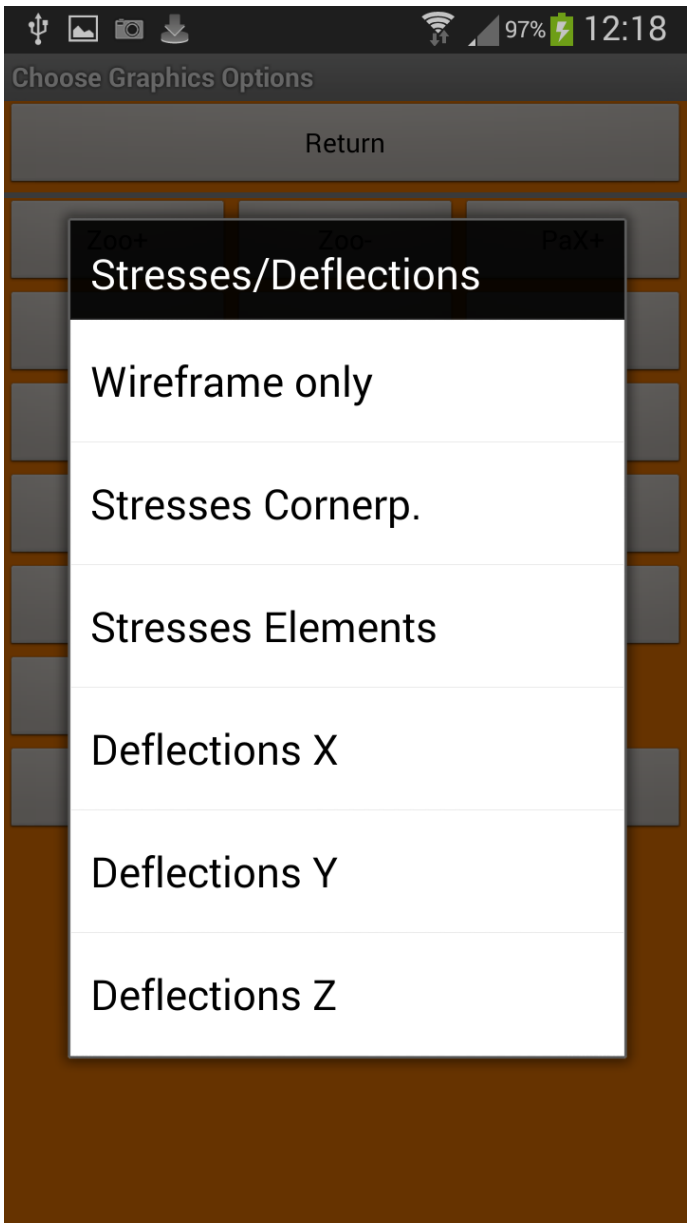

97% 12:17
Choose Graphics Options
Return
Zoo+
Zoo-
PaX+
PaX-
PaY+
PaY-
Labels
No Labels
Label Nodes
Label Elements
Label All



Choose *Deflection* from the menu left-side of the Home button.



Von Mises stresses. If you want to see *Pricipal* or *Tresca* stresses adjust this in the menu of the Main page before running the Solver.



Where is the difference betwenn *Stresses Cornerp*(oints) and *Stresses 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:

- *von Mises/principla/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 but in Gauss points laying in the near of 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.
- *von Mises/principla/Tresca stresses as a mean value for each element.* The stresses are

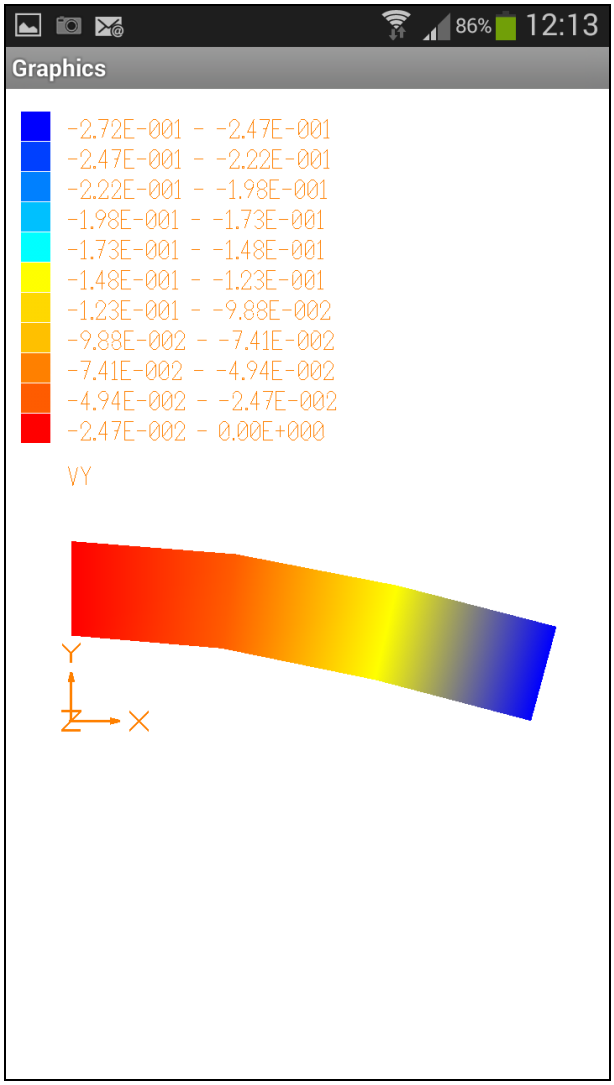
computed in the Gauss points of the current element, added and then divided by the current number of Gauss points. This results in a mean value for the *von Mises/principla/Tresca stress* per element.

- *von Mises/principla/Tresca stresses in Gauss points.* This is most accurate but delivers not always very pretty pictures. Not implemented in Z88Tina, only in Z88Aurora.

Z88Tina may show these stresses after a run of the *Solver* – but only one type of stress:

- *von Mises stresses*
- *Rankine or principal stresses*
- *Tresca stresses*

Thus, if you have computed the *von Mises* stresses with a *Solver* run Z88Tina's *Graphics* will show the *von Mises* stresses. If you want to see now the *Tresca* stresses you must choose *Tresca* stresses and re-run the *Solver* This looks awkward but don't you know before starting the FE computations which type of stresses is suitable and correct for your task?



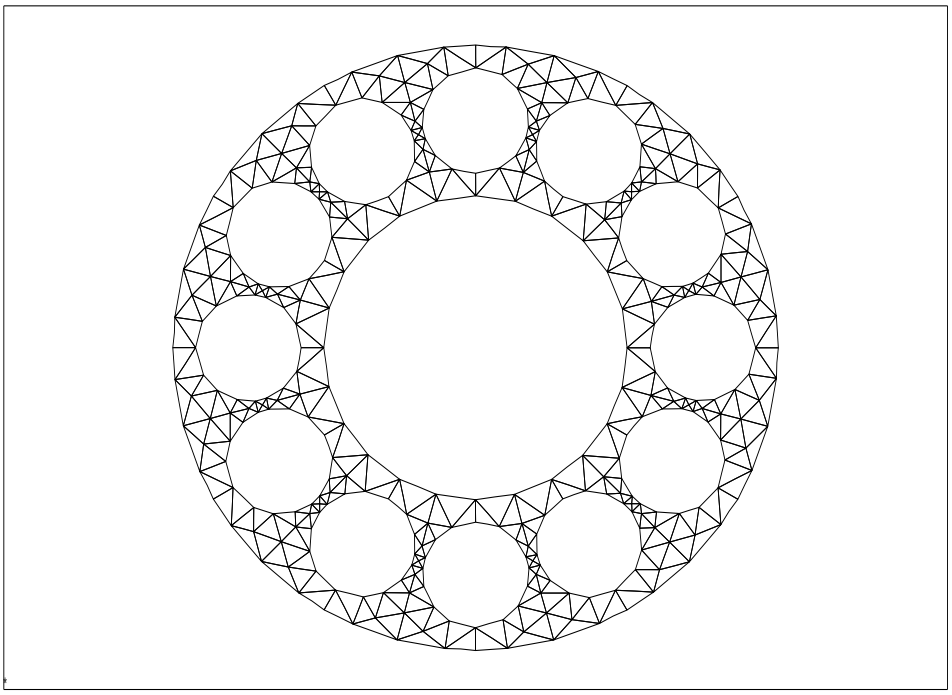
The deflections V in Y direction.

2.3 READ IN AN EXISTING STRUCTURE – 3RD EXAMPLE

The following example is taken from the book

Rieg,F.; Hackenschmidt, R.; Alber-Laukant, B.: Finite Elemente Analyse für Ingenieure. Carl Hanser Verlag, München Wien 2012, 4th edition (in German language – an English edition of this book is under way)

for our full-scaled FEA program Z88Aurora, see www.z88.de . Thus, because this structure is too heavy for manual input we'll read it into Z88Tina.



The is the torsion element reacting as a torsion spring for a torque sensor. The inner edge is fixed while the node of the outer edge with a diamter of 80 mm are moved 1°. This means a tangetial displacement of 0,7 mm:

$$s = \varphi \cdot r = 1° \cdot \frac{\pi}{180} \cdot 40\ \text{mm} = 0,7\ \text{mm}$$

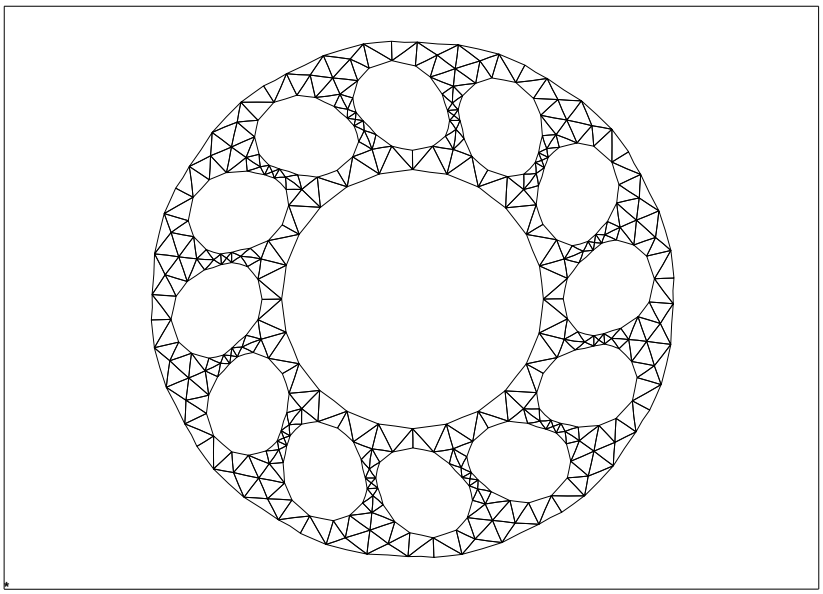
Nodes which are not on 90°, 180°, 270° or 360°/0° need an angular transformation. Let us assume the node 201 is located on 45° or 1:30 o'clock:

$$c^2 = a^2 + b^2 = 2 \cdot a^2$$

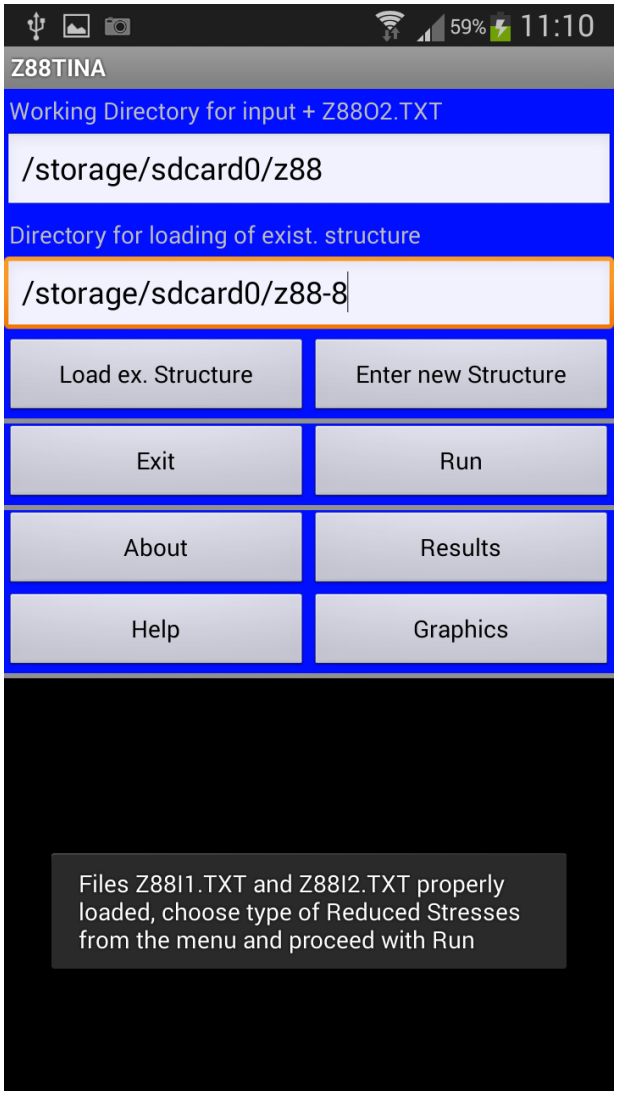
$$c = \sqrt{2a^2}$$

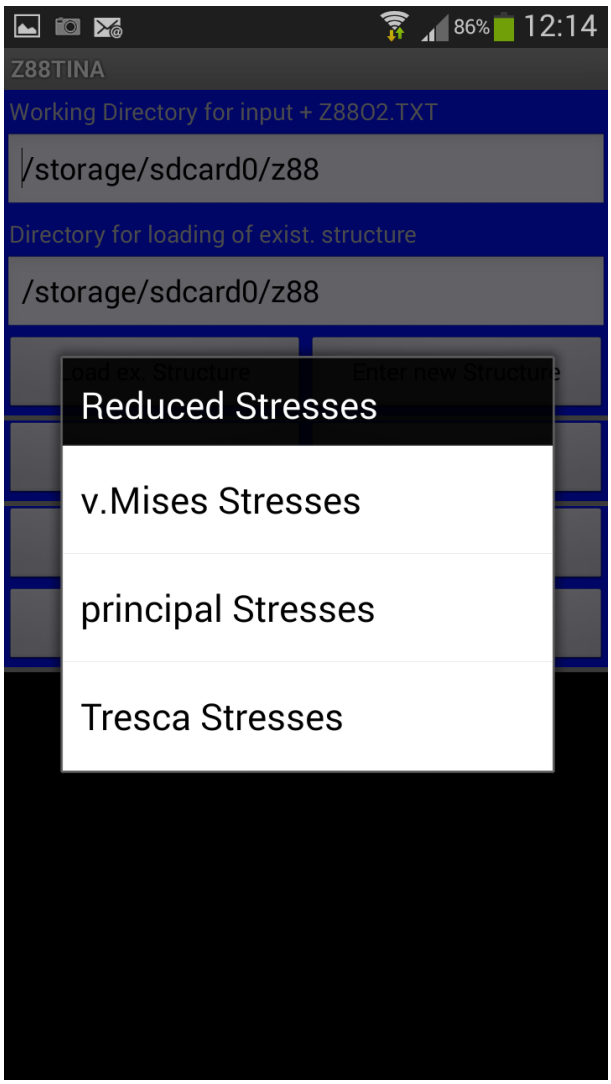
$$a = \frac{c}{\sqrt{2}} = 0,4949$$

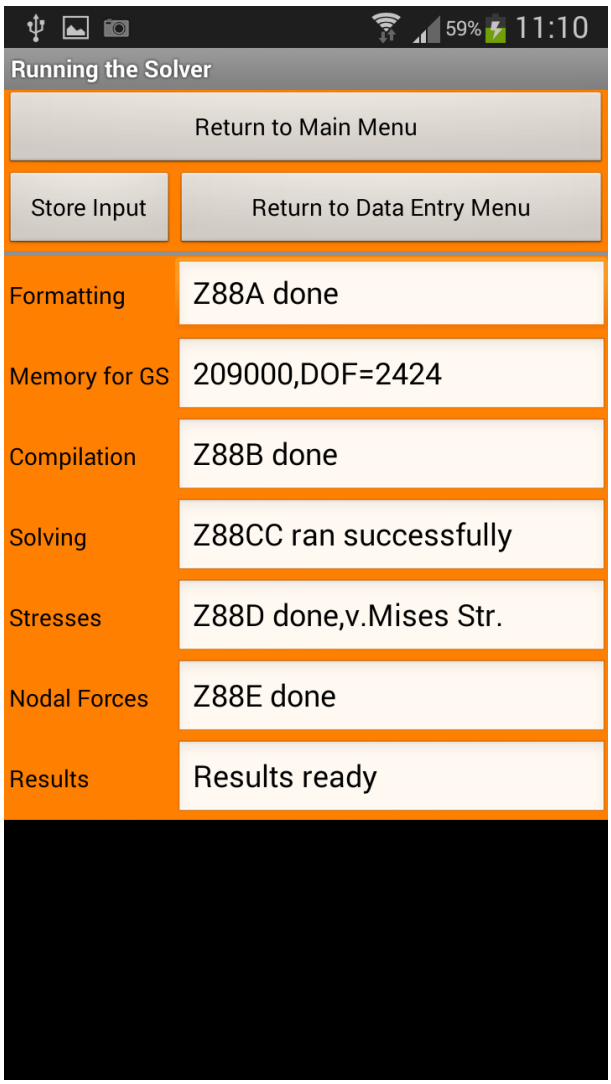
Thus, node 201 features a displacement of –0,494 in X direction and +0,494 in Y direction.

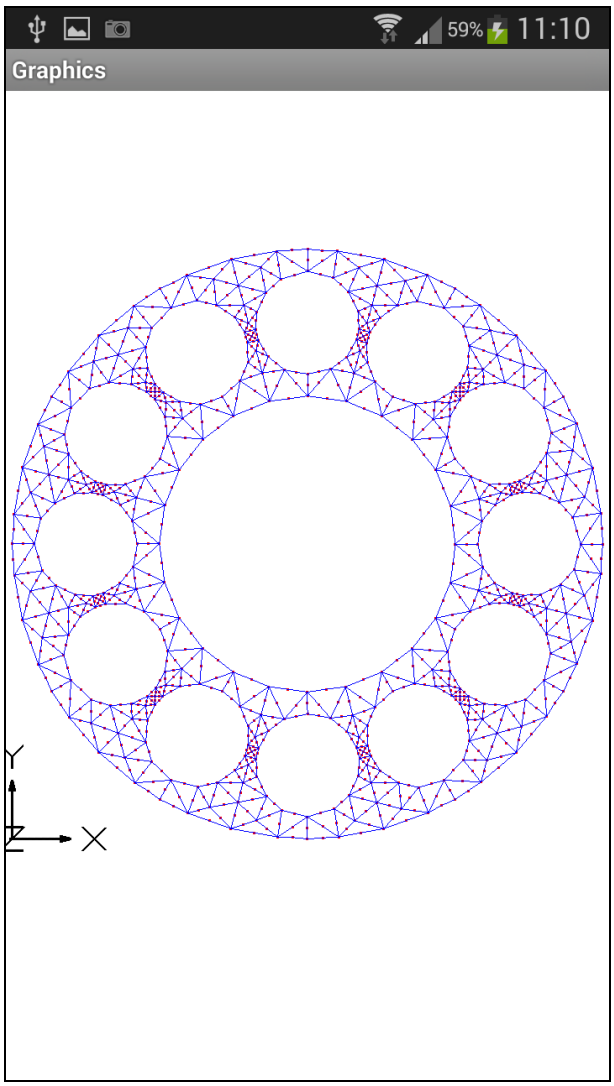


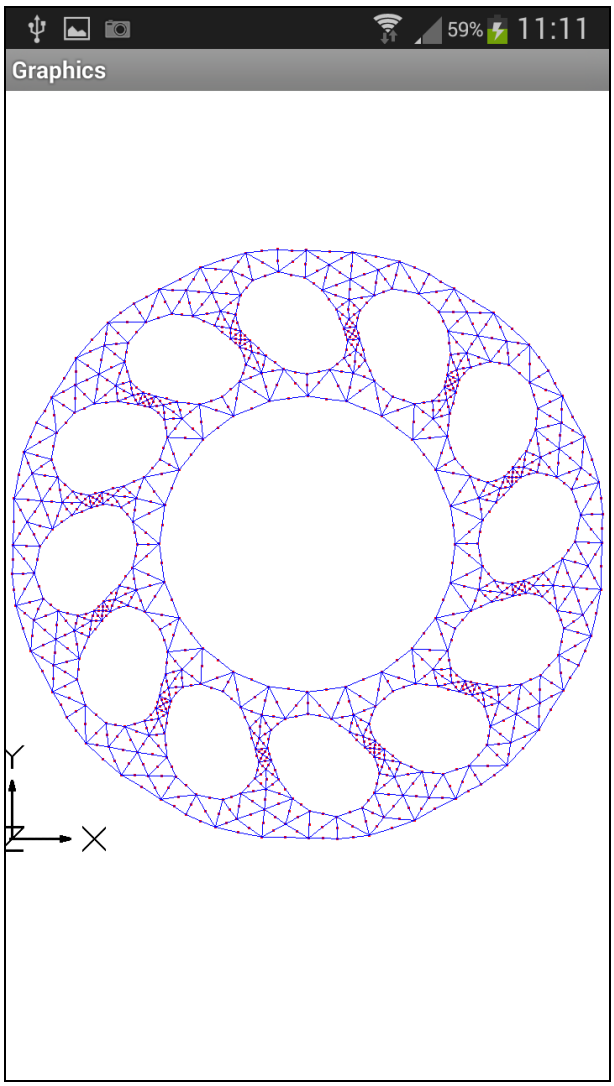
To proceed, please load the files from example z88-8, see www.z88tina.de.

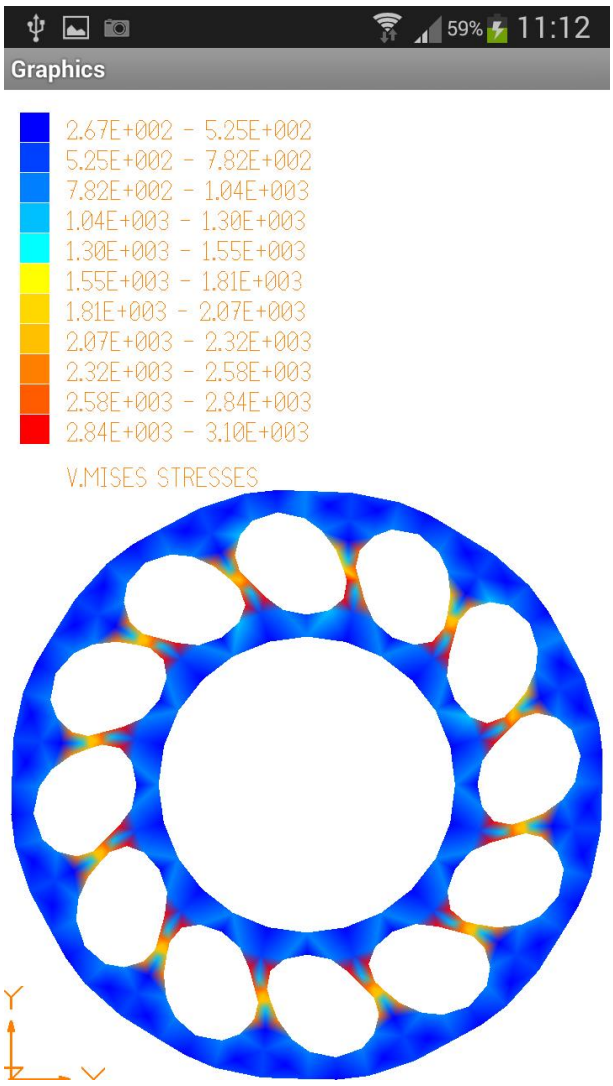



86% 12:14
Z88TINA
Working Directory for input + Z88O2.TXT
/storage/sdcard0/z88
Directory for loading of exist. structure
/storage/sdcard0/z88
Reduced Stresses
v.Mises Stresses
principal Stresses
Tresca Stresses


59% 11:10
Running the Solver
Return to Main Menu
Store Input
Return to Data Entry Menu
Formatting
Z88A done
Memory for GS
209000,DOF=2424
Compilation
Z88B done
Solving
Z88CC ran successfully
Stresses
Z88D done,v.Mises Str.
Nodal Forces
Z88E done
Results
Results ready


59%
11:10
Graphics
Y
Z
X


59%
11:11
Graphics
Y

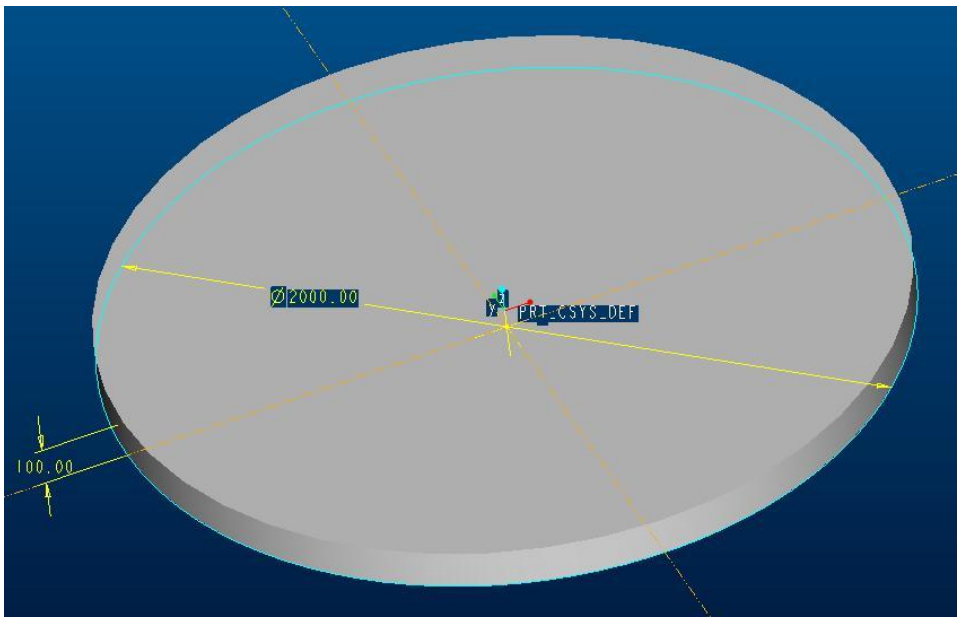


2.4 READ IN AN EXISTING STRUCTURE – 4th EXAMPLE

The following example is taken from the book

Rieg,F.; Hackenschmidt, R.; Alber-Laukant, B.: Finite Elemente Analyse für Ingenieure. Carl Hanser Verlag, München Wien 2012, 4th edition (in German language – an English edition of this book is under way)

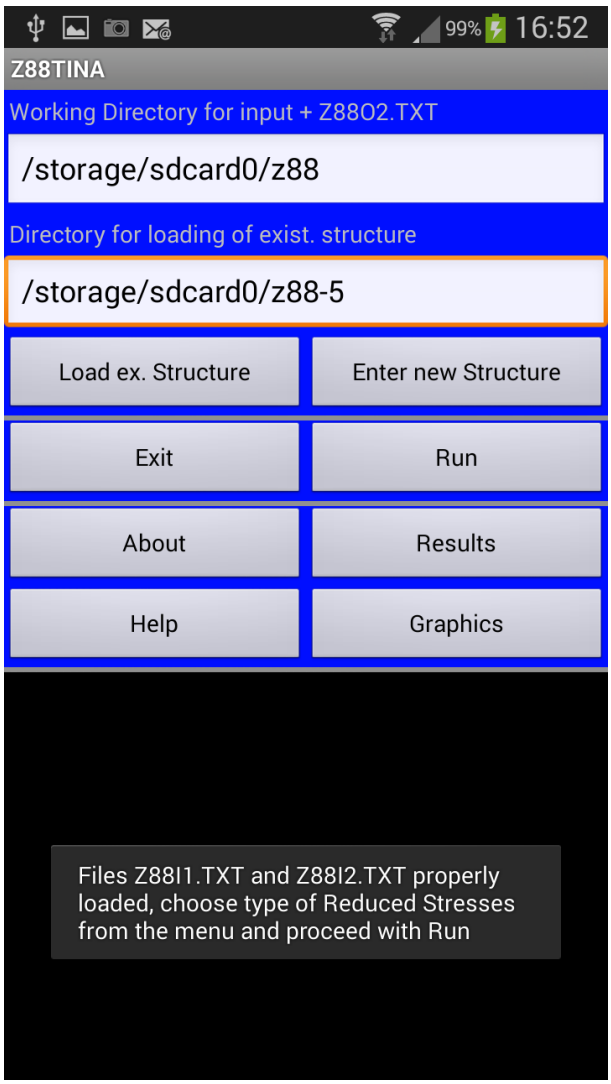
for our full-scaled FEA program Z88Aurora, see www.z88.de . Thus, because this structure is too heavy for manual input we'll read it into Z88Tina.

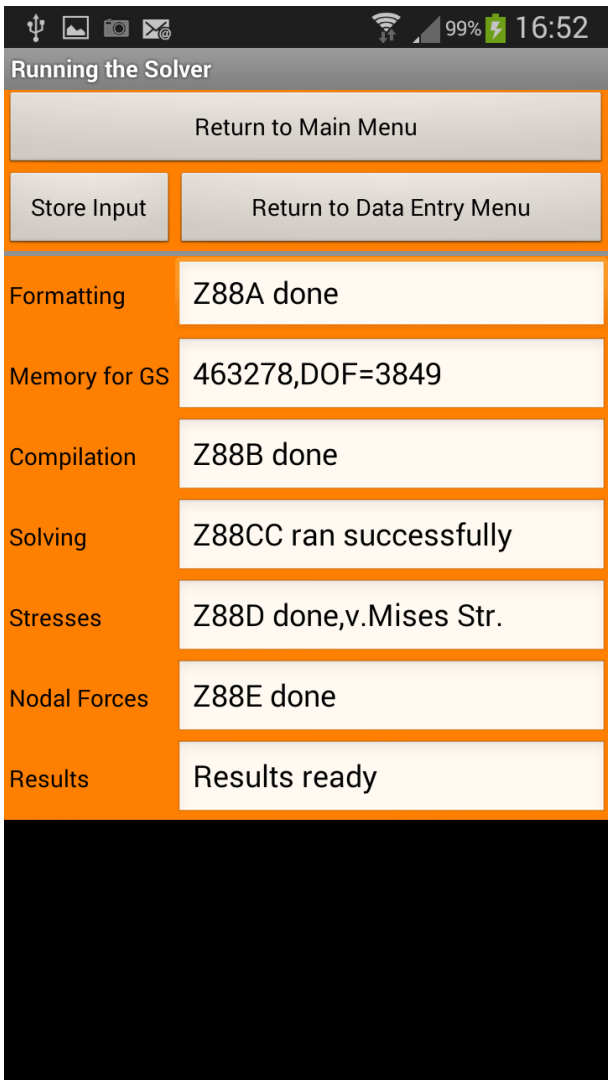


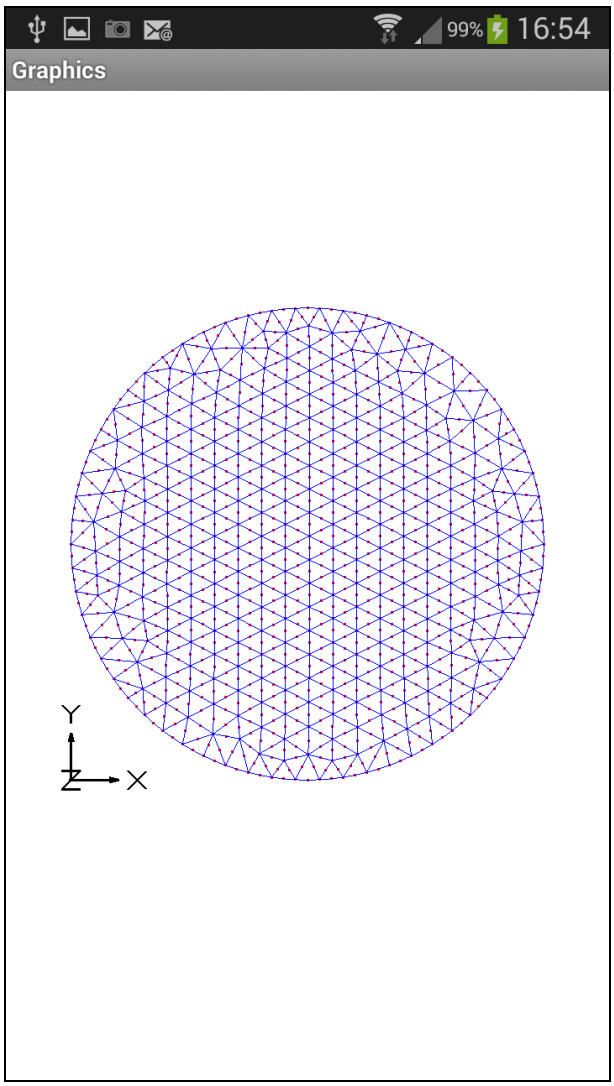
The circular plate was modelled in PTC Pro/ENGINEER.

It features 2.000 mm diameter and 100 mm thickness, material steel with E = 206.000 N/mm^2 and $\nu = 0{,}3$, a surface load of 2 N/mm^2. The outer edge is fixed in *Z* direction.

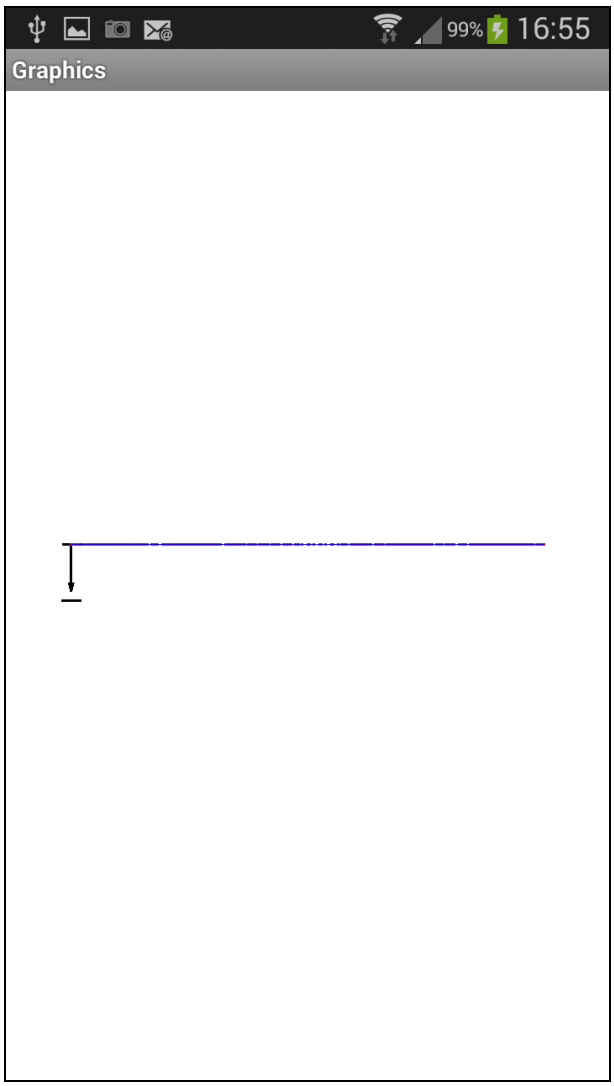
To proceed, please load the files from example z88-8, see www.z88tina.de.


99% 16:52
Z88TINA
Working Directory for input + Z88O2.TXT
/storage/sdcard0/z88
Directory for loading of exist. structure
/storage/sdcard0/z88-5
Load ex. Structure
Enter new Structure
Exit
Run
About
Results
Help
Graphics
Files Z88I1.TXT and Z88I2.TXT properly loaded, choose type of Reduced Stresses from the menu and proceed with Run

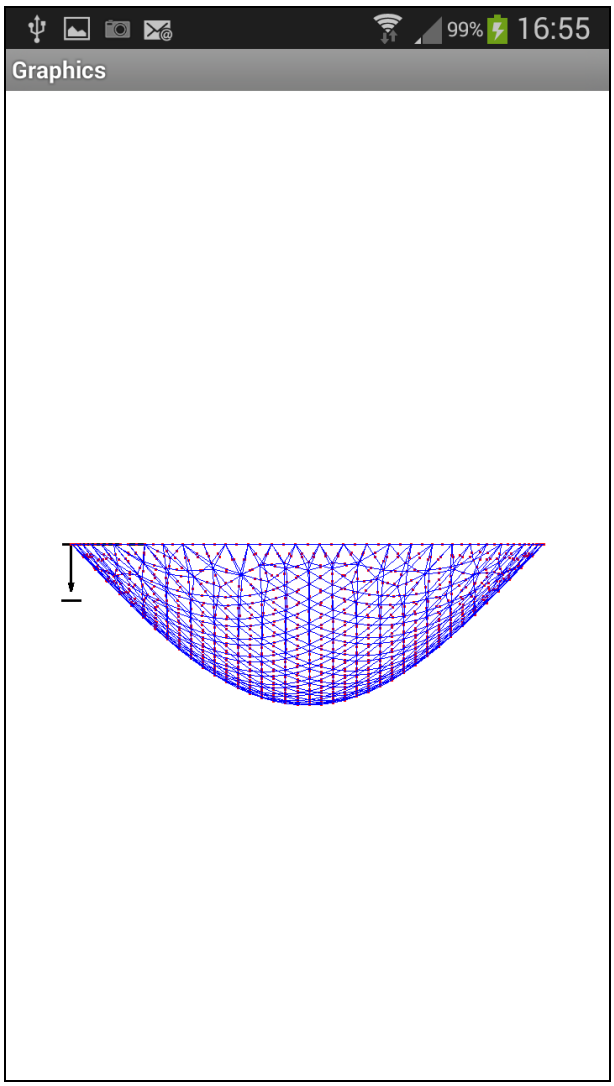

99% 16:52
Running the Solver
Return to Main Menu
Store Input
Return to Data Entry Menu
Formatting
Z88A done
Memory for GS
463278,DOF=3849
Compilation
Z88B done
Solving
Z88CC ran successfully
Stresses
Z88D done,v.Mises Str.
Nodal Forces
Z88E done
Results
Results ready



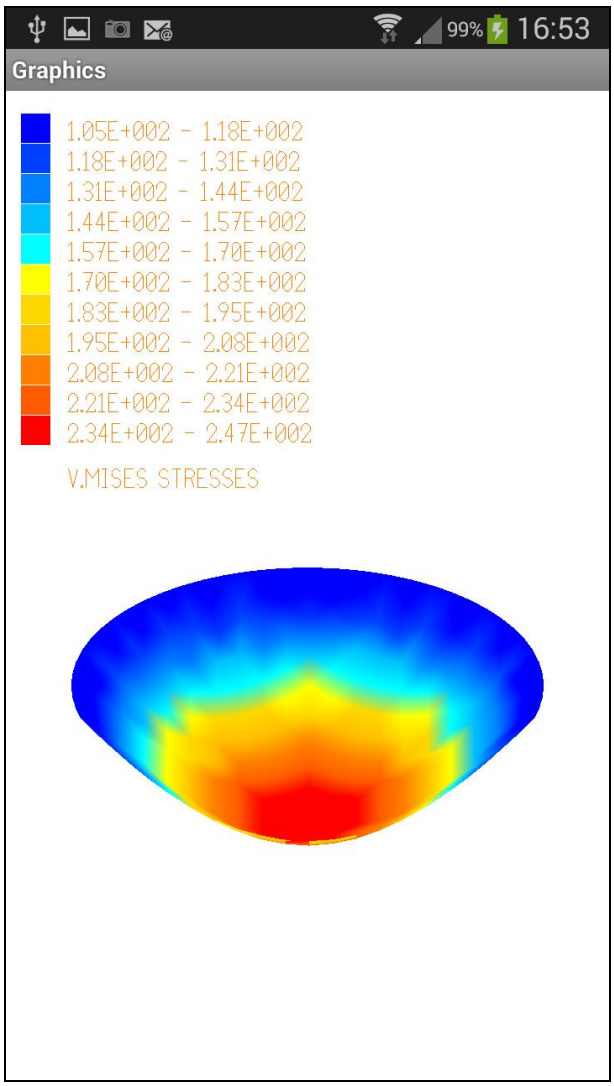
The undeflected structure in default view. Now rotate 90° around the X axis by pressing *Rot+* three times:



Now activate *Deflected yes*:



Now the v. Mises stresses, deflected:



3 EDITING INPUT FILES

3.1 GENERAL INFORMATION

Z88Tina works in reading mode with the following files and writes them in editing mode, thus, you may edit these files on a PC and store them via USB on the smartphone:

(1) Input Files:

- *Z88I1.TXT (general structure data, coordinates, coincidence, material informations)*
- *Z88I2.TXT (boundary conditions, loads, constraints)*

Details to the input files see sections 3.2 pp.

(2) Output Files:

- *Z88O2.TXT (calculated displacements)*

Rules for entering values:
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 contain any point 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 filled out before. Separate the last data and the comment at least by one blank. Blank lines and pure comment lines are not permitted.

Examine when error messages or abnormal program stops of Z88:

- Are the files really pure text files, well in the ASCII format? Or have they been added unnoticed hidden control characters by your text processor?
- Is the last line of an input file terminated by at least one *RETURN* ?
- Is your structure statically determined or in any way statically overdefined (allowed!)? Or is it statically indetermined, i. e. boundary conditions are missing which may cause serious trouble. Statically indetermined structures can appear easily for Beams No.2, Cams No.5 and Beams No.13 (take care of the rotation degrees of freedom).
- Is the coincidence list defined properly?
- Plot the initial structure. If you won't see some pretty good stuff, then the rest can hardly be better!
- Always do a rough calculation! Are the calculated deflections extremely high? Then check the boundary conditions quite carefully!

3.2 GENERAL STRUCTURE DATA Z88I1.TXT

Use the data types [Int] and [Double].

1st input group, i. e. first line, contains:

Dimension of the structure (2 or 3) [Int]
Number of nodes of the FEA structure [Int]
Number of elements [Int]
Number of degrees of freedom [Int]
Number of material information lines [Int]
Coordinate flag KFLAG (0 or 1) [Int]
Beam flag IBFLAG (0 or 1) [Int]
Plate flag IPFLAG (0 or 1) [Int]
Surface and pressure loads flag IQFLAG (0 or 1) [Int] – not used for Z88Tina V1.0

Write all numbers into a line, separate at least by one blank respectively.

Explanation KFLAG:
At input of 0 the coordinates are expected cartesian while at input of 1 polar or cylindrical coordinates are expected. The latter are then converted into cartesian coordinates. Caution: The

axisymmetric elements No. 15 positively expect cylindrical coordinates, set KFLAG to 0 here!

Explanation IBFLAG:
If Beams No.2 or Beams No.13 appear in the structure, then set beam flag IBFLAG to 1, otherwise it must be 0.

Explanation IPFLAG:
If Plates No.18 appear in the structure, then set plate flag IPFLAG to 1, otherwise it must be 0.

Caution: This Z88 release allows only beams or plates in a structure, not both in the same structure, because the DOF of the beams and the plates are not compatible!

Explanation IQFLAG:
- Set to 0 for this release of Z88Tina.

Example: Your structure features 4 nodes with 3 beams No.13 (=3 DOF per node, 2-dimensional), the same parameters for all 3 beams, cartesian coordinates.
Thus: *2 4 3 12 1 0 1 0 0*

2nd input group, starting with line 2, contains:
Coordinates, one line per node.

Node number, strictly ascending [Int]
Number of the degrees of freedom for this node [Int]
X-coordinate or, if KFLAG is 1, R- coordinate [Double]
Y-coordinate or, if KFLAG is 1, PHI-coordinate [Double]
Z-coordinate or, if KFLAG is 1, Z-coordinate [Double]
The Z coordinate can be dropped at 2-dimensionalen structures. Enter angles PHI in radian.

Write all numbers into a line, separate at least by one blank respectively.

Example 1: The node no.15 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, contains:
Coincidence, two lines for every finite element

1st line:
Element number, strictly ascending
Element type (2, 4, 5, 9, 13, 14, 15, 18), i.e.
2 = Beam No.2 in space, 2 nodes, 6 DOF
4= Truss No.4 in space, 2 nodes, 3 DOF
5= Cam No.5 (along the X-axis), 2 nodes, 6 DOF
9= Truss No.9 in plane, 2 nodes, 2 DOF
13= Beam No.13 in plane, 2 nodes, 3 DOF
14= Plane Stress Element No.14 in Plane, 6 nodes, 2 DOF
15= Axisymmetric Element No.15, 6 nodes, 2 DOF

18= Plate Element No.18, 6 nodes, 3 DOF

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

2nd line: Depending on element type
1st node number for coincidence
2nd node number for coincidence
.....
6th node number for coincidence

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

Example: An Isoparametric Serendipity Plane Stress Element No.14 has element number 23. The coincidence has the global nodes 16, 8, 17, 20, 38, 51 (locally these are the nodes 1-2-3-4-5-6, see chapters 4.6 ~ 4.8) . Thus resulting in two lines:
23 14
16 8 17 20 38 51

4th input group, starting after last element, contains:
Material information, one line for each material information.

This material information line starts with element no. inclusively [Int]
This material information line ends with element no. inclusively [Int]
Youngs's Modulus [Double]
Poisson's Ratio [Double]
*Integration order (0, 1, 3, **7** or 13)* [Int]
Cross section value QPARA [Double]

... And if beams (but not plates!) are defined in addition:
Second moment of inertia yy (bending around yy axis) [Double]
Max. distance from neutral axis yy [Double]
Second moment of inertia zz (bending around zz axis) [Double]
Max. distance from neutral axis zz [Double]
Second moment of area (torsion) [Double]
Second modulus (torsion) [Double]

... And if plates (but not beams !) are defined and IQFLAG=0, in addition:
area load

Write all numbers into a line, separate at least by one blank respectively.

Explanation cross section value QPARA:
QPARA is element type-dependent, e.g. for trusses QPARA is the cross-sectional area and for plane stress elements QPARA is the thickness. See chapter 4.

Example: The structure has 6 finite elements No.14. The thicknesses is supposed to vary: Elements 1 to 2 thickness 10 mm, element 3 15 mm and elements 4 to 6 now 18 mm. Material steel. Integration order is supposed to be 7. Thus three material information lines:

1 2 206000 0.3 7 10.
3 3 206000 0.3 7 15.
4 6 206000 0.3 7 18.

3.3 BOUNDARY CONDITIONS Z88I2.TXT

Use the data types [Int] and [Double].

1st input group, i. e. first line, contains:

Number of the boundary conditions: loads and constraints [Int]

2nd input group, starting in line 2, contains:

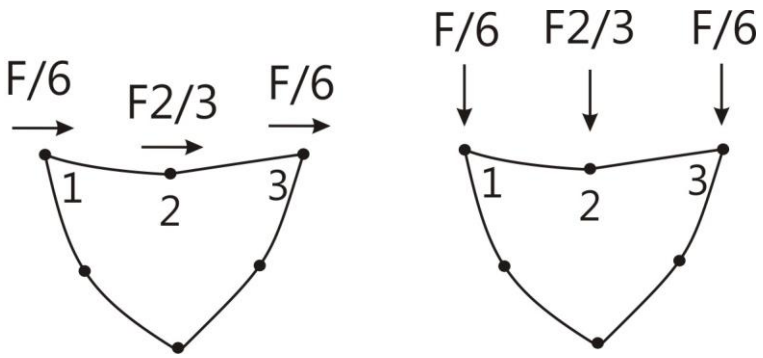
Boundary conditions and loads. For every boundary condition and for every load respectively one line.

node number with boundary condition: load or constraint [Int]
Respective degree of freedom (1,2,3,4,5,6) [Int]
Header flag: 1 = force [Int] *or 2 = displacement* [Int]
Value of the load or displacement [Double]

Example: The node 1 shall be fixed respectively at his 3 degrees of freedom: support. Node 3 gets a load of -1648 N in Y direction (i.e. DOF 2), the degrees of freedom 2 and 3 is supposed to be fixed for the node 5. Resulting in 6 boundary conditions. Thus :

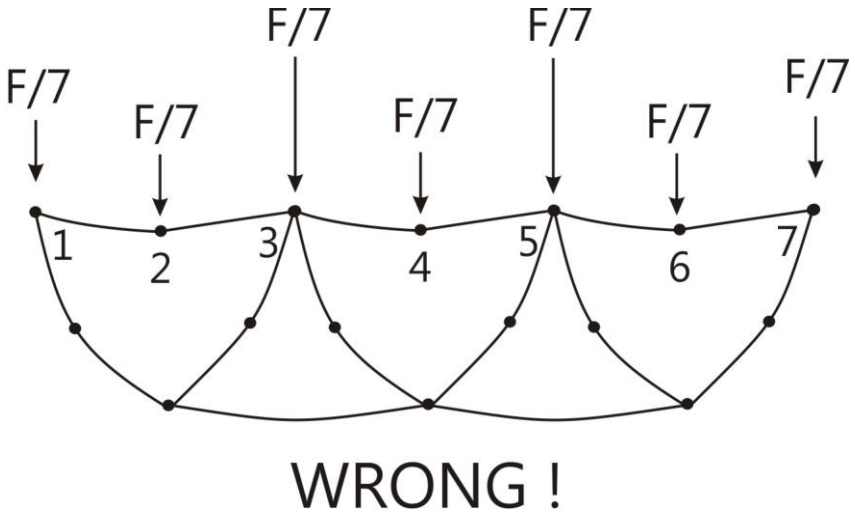
6
1 1 2 0
1 2 2 0
1 3 2 0
3 2 1 -1648
5 2 2 0
5 3 2 0

However, for elements with higher shape functions, i. e. Plane Stress No.14, Torus No.15, and Plates No.18 edge and surface loads have to be put onto the elements according to fixed rules which are not always physically obvious. Really funny, some load components can have negative values (for cubic interpolation function). Though these facts are not obvious, nethertheless they lead to correct results which is not the case for intuitive distribution of loads to the respective nodes - please see any good book on finite elements:



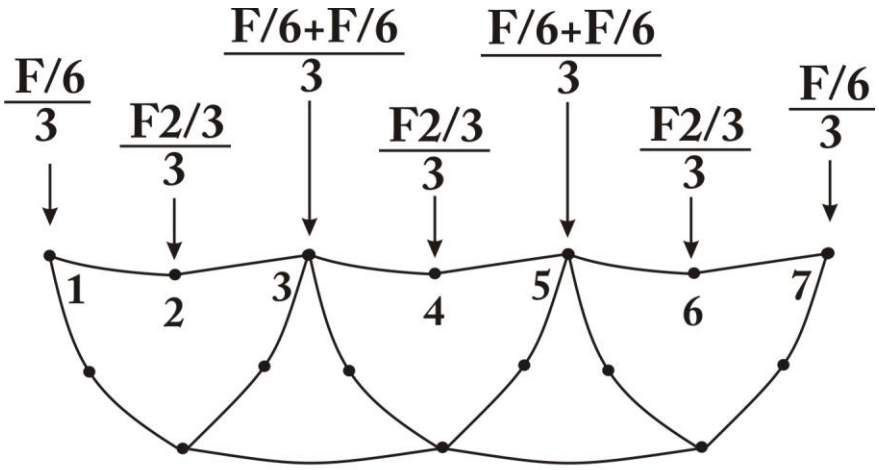
An example shall clarify the facts:

We want to distribute a force of 5400 N onto three Plane Stress Eements No.14 - the same is true for Torus Elements No.15 and Plate Elements No.18 because they feature 3 nodes per edge i.e. a quadratic interpolation function:



WRONG !

An equal distribution for all nodes i.e. 771 N per node is only correct for elements with linear interploation functions (not true for Z88Tina's continuum elements). Here's how to do it correctly:



CORRECT !

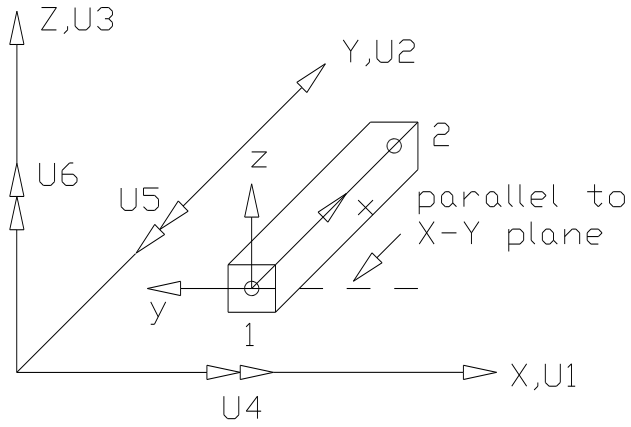
Thus, you'll put on node 1 a force of 5400N/6/3 elements= 300N, on node 2 a force of 5400N x

2/3/3 elements= 1200N, on node 3 a force of 5400N x 2 /6/3 elements= 600N and so on. This results in: 300 + 1200 + 600 + 1200 + 600 + 1200 + 300 = 5400N = correct!

4 DESCRIPTION OF THE FINITE ELEMENTS

4.1 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 Z88I1.TXT. Thus, you can use any symmetric profile in contrast to other FEA programs which sometimes 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.



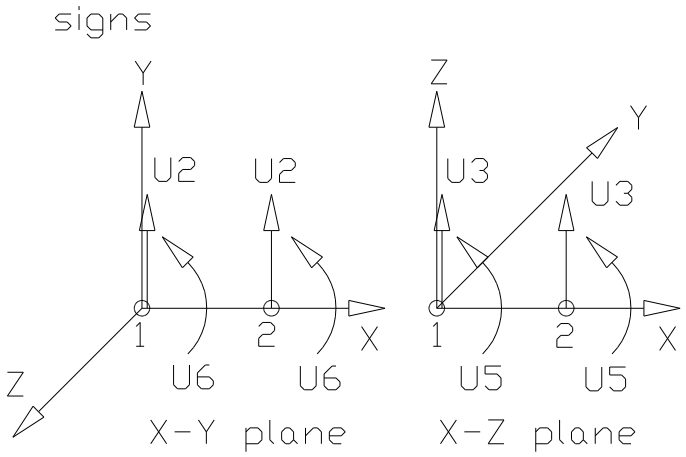
Input:

> KFLAG for cartesian (0) or cylindrical coordinates (1)
> Set beam flag IBFLAG to 1
> 6 degrees of freedom in a node (Attention: DOF5 (not right hand rule), see below)
> Element type is 2
> 2 nodes per element

At the material information lines:
> Integration order INTORD is arbitrary, has no meaning
> Cross-sectional area QPARA
> Second moment of inertia RIYY (bending around y-y axis)
> Max. distance EYY from neutral axis y-y
> Second moment of inertia RIZZ (bending around z-z axis)
> Max. distance EZZ from neutral axis z-z
> Second moment of area (torsion) RIT
> Second modulus (torsion) WT

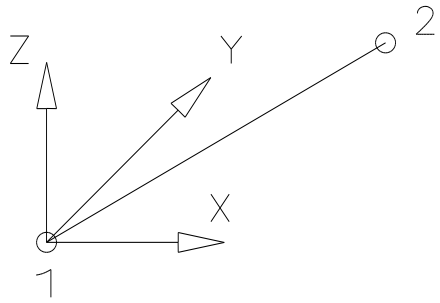
Results:
Deflections in X, Y and Z and rotations around X, Y and Z. Attention DOF5 (not right hand rule), see below

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 (= outer forces) for each node.



4.2 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 matches Hooke's law exactly .



Input:

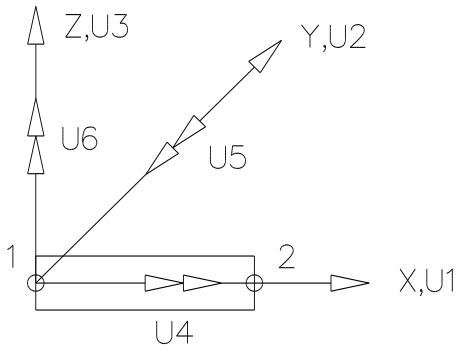
> KFLAG for cartesian (0) or cylindrical coordinates (1)
> 3 degrees of freedom for each node
> Element type is 4
> 2 nodes per element
> Cross-section parameter QPARA is the cross-sectional area of the truss

Results:
Displacements in X, Y and Z
Stresses: Normal stresses
Nodal forces (= outer forces) for each node.

4.3 CAM ELEMENT NO.5 WITH 2 NODES

The cam 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. Inputs and calculations are simplified strongly through this. Like with the beam element the results are exact according to Bernoulli's bend theory and Hooke's law, and not approximate solutions like with the continuum elements.

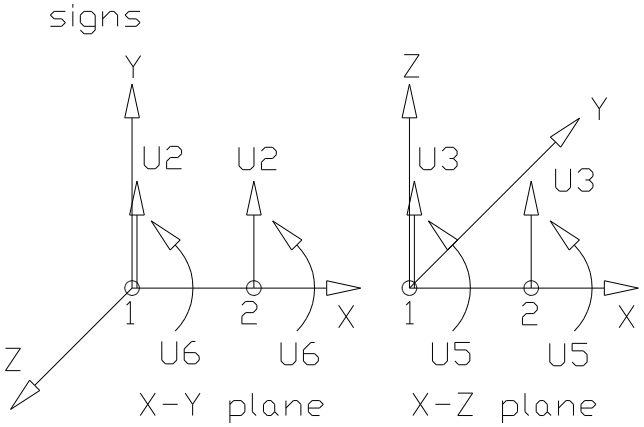


Input:

> Set KFLAG on 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
> Cross-section parameter QPARA is the diameter of the cam

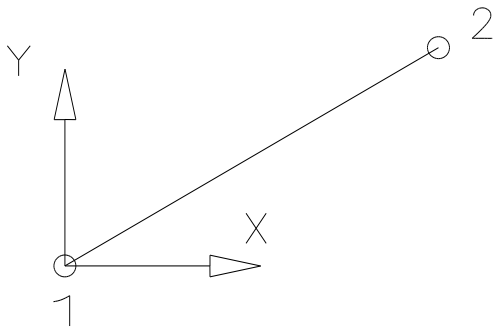
Results:

Deflections 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, 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 (= outer forces & moments) for each node.



4.4 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 matches Hooke's law exactly.



Input:

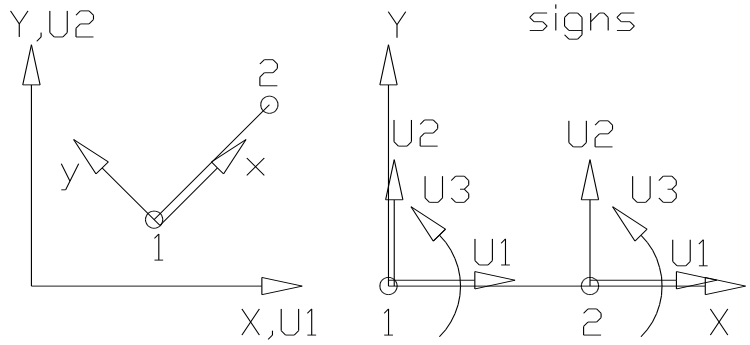
> KFLAG for cartesian (0) or polar coordinates (1)
> 2 degrees of freedom for each node
> Element type is 9
> 2 nodes per element
> Cross-section parameter QPARA is the cross-sectional area of the truss

Results:

Displacements in X and Y
Stresses: Normal stresses
Nodal forces (= outer forces) for each node.

4.5 BEAM NO.13 WITH 2 NODES IN PLANE

Beam element with any symmetric profile. The profile values are provided in Z88I1.TXT. Thus, you can use any symmetric profile in contrast to other FEA programs which sometimes 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:

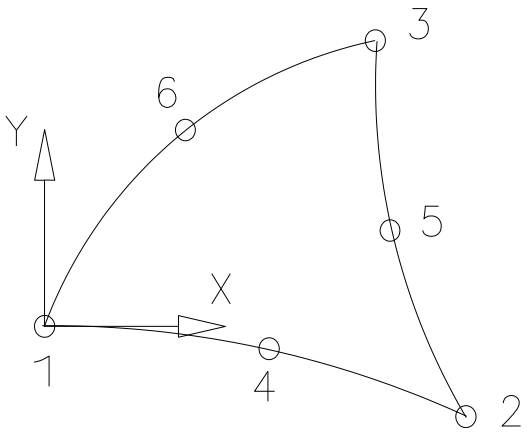
> KFLAG for cartesian (0) or polar coordinates (1)
> Set beam flag IBFLAG to 1
> 3 degrees of freedom in a node
> Element type is 13
> 2 nodes per element

At the material information lines:
> *Any integration order INTORD (1...4), has no influence*
> *Cross-sectional area QPARA*
> *Insert 0 for second moment of inertia RIYY (bending around y-y axis)*
> *Insert 0 for max. distance EYY from neutral axis y-y*
> *Second moment of inertia RIZZ (bending around z-z axis)*
> *Max. distance EZZ from neutral axis z-z*
> *Insert 0 for second moment of area (torsion) RIT*
> *Insert 0 for second modulus (torsion) WT*

Deflections in X and Y and **rotations** around Z.
Stresses: SIGXX, TAUXX: Direct 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 (= outer forces & moments) for each node.

4.6 PLANE STRESS ELEMENT NO.14 WITH 6 NODES

This is a curvilinear Serendipity plane stress element with square shape functions. The transformation is isoparametric. The integration is carried out numerically according to Gauss-Legendre. Consequently, the integration order can be selected in Z88I1.TXT in the material information lines. The order 7 (=7 Gauss points) is mostly sufficient. This element calculates both displacements and stresses very exactly.



Input:

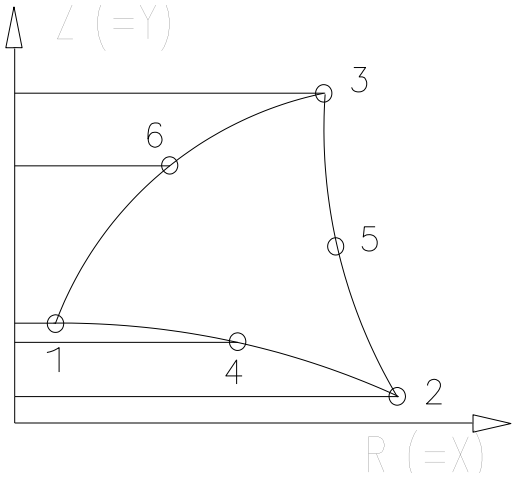
> *KFLAG for cartesian (0) or polar coordinates (1)*
> *IQFLAG=1 if edge loads for this element are filed in Z88I5.TXT*
> *2 degrees of freedom for each node*
> *Element type is 14*
> *6 nodes per element*
> *Cross-section parameter QPARA is the element thickness*
> *Integration order INTORD per each mat info line. 7 is usually good. Possible is: 3 for 3 Gauss points, 7 for 7 Gauss points and 13 for 13 Gauss points.*

Results:

Displacements in X and Y.
Stresses: The reduced stresses SIGV are calculated in the corner nodes and per element, each. You may choose this when plotting. Choose *von Mises* or *principal* or *Tresca* stresses in the menu before running the solver.
Nodal forces(= outer forces) for each node.

4.7 TORUS NO.15 WITH 6 NODES

This is a curvilinear Serendipity torus element with square shape functions. The transformation is isoparametric. The integration is carried out numerically according to Gauss- Legendre. Thus, the integration order can be selected in Z88I1.TXT in the material information lines. The order 7 is mostly sufficient. This element calculates both displacements and stresses very exactly.



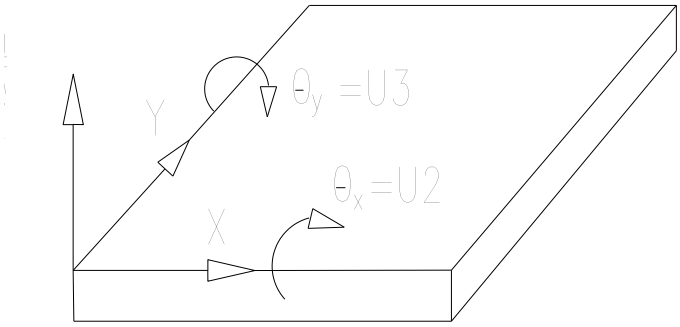
Input:

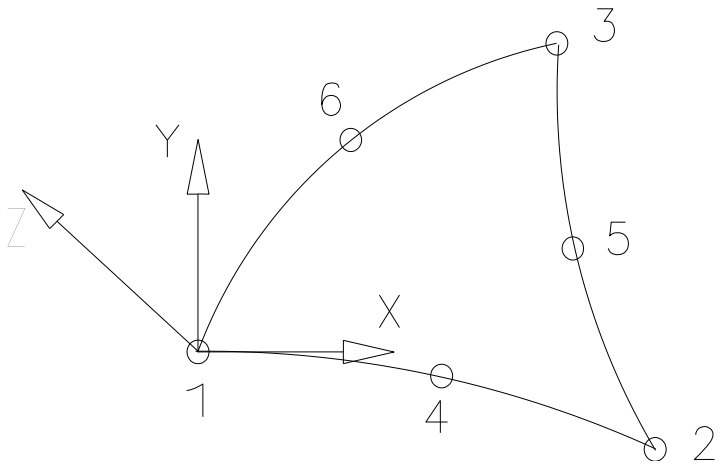
> In principle cylindrical coordinates are expected: KFLAG must be 0 !
R coordinate (= X), always positive
Z coordinate (= Y), always positive
> IQFLAG=1 if edge loads for this element are filed in Z88I5.TXT
> 2 degrees of freedom for each node, DOF R and Z (= X and Y).
> Element type is 15
> 6 nodes per element
> Cross-section parameter QPARA is 0 or any value, no influence
> Integration order INTORD per each mat info line. 7 is usually good. Possible is: 3 for 3 Gauss points, 7 for 7 Gauss points and 13 for 13 Gauss points.

Results:
Displacements in R and Z (= X and Y).
Stresses: The reduced stresses SIGV are calculated in the corner nodes and per element, each. You may choose this when plotting. Choose *von Mises* or *principal* or *Tresca* stresses in the menu before running the solver.
Nodal forces(= outer forces) for each node.

4.8 PLATE NO.18 WITH 6 NODES

This is a curvilinear Serendipity *Reissner-Mindlin* plate element with square 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 Z88I1.TXT in the material information lines. The order 3 (= 3 points) is mostly sufficient (reduced integration). This element calculates both displacements and stresses quite good. Area loads are defined in the appropriate material lines, file Z88I1.TXT, instead of Second moment of inertia RIYY. 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.





Input:

> KFLAG for cartesian (0) or cylindrical coordinates (1)
> set plate flag IPFLAG to 1 (or 2, if you want to reduce the shear influence)
> set surface and pressure loads flag IQFLAG to 0 for your convenience. Then the entry of the pressure is done via the "Second moment of inertia RIYY", see below.
> 3 degrees of freedom for each node (w, θ_x,θ_y)
> Element type is 18
> 6 nodes per element
> Cross-section parameter QPARA is the element thickness
> "Second moment of inertia RIYY" is the pressure load
> Integration order INTORD per each mat info line. 3 is usually good. Possible is: 3 for 3 Gauss points, 7 for 7 Gauss points and 13 for 13 Gauss points.

Results:

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

Stresses: T **Stresses:** The reduced stresses SIGV are calculated in the corner nodes and per element, each. You may choose this when plotting. Choose *von Mises* or *principal* or *Tresca* stresses in the menu before running the solver.
Nodal forces(= outer forces) for each node.

5 FURTHER EXAMPLES

Example 1: Crane truss. Modelled with Trusses No.4.

Example 2: Transmission cam. Cam with different diameters, forces and moments in different planes with cam elements No.5, statically over-defined.

Example 3: Beam in plane, repeatedly statically over-defined. On both sides firmly fixed Beam No.13.

Example 4: Press fit. Axially symmetric problem with Tori No.15.

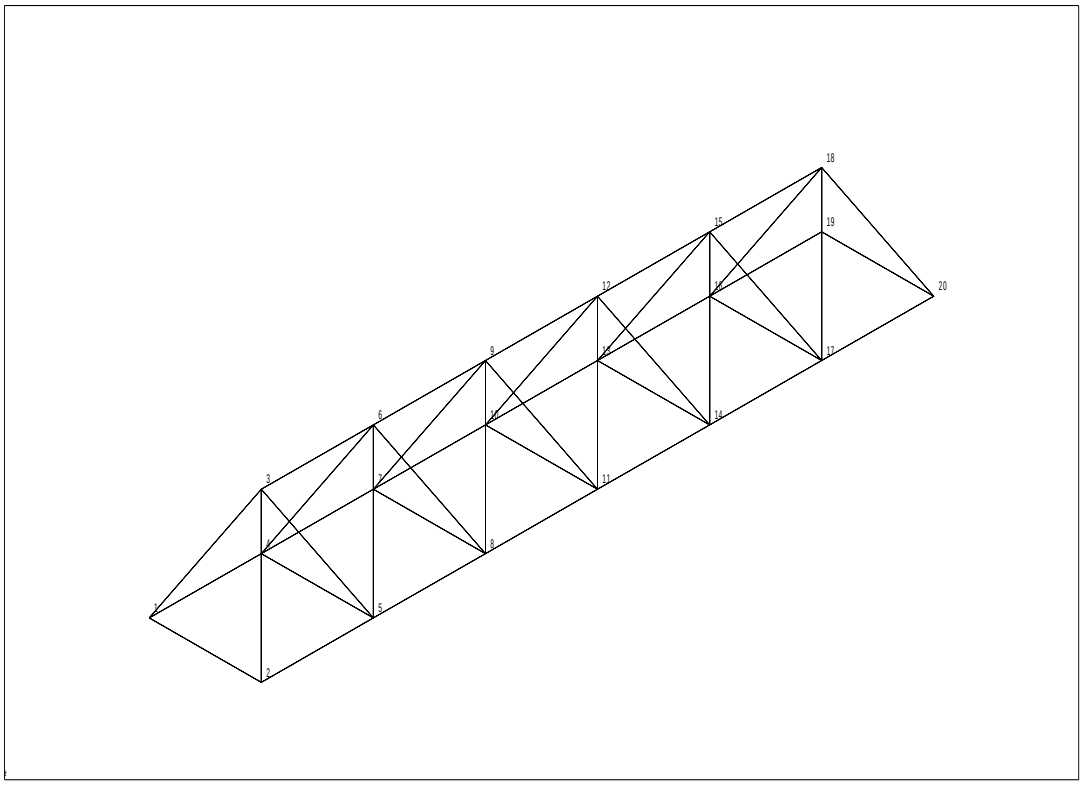
5.1 CRANE TRUSS WITH TRUSSES NO.4

Copy the example files Z88I1.TXT and Z88I2.TXT from www.z88tina.de , directory **z88-4**.

The example is simple and straight. Experiment with the 3D features of the *Graphics* section.

A crane truss consists of 54 trusses, 20 nodes and forms a spatial framework. The nodes 1, 2 and 19, 20 are fixed, the nodes 7 and 8 are loaded per -30,000 N. The total length is 12 m. The inputs in the sample file are in mm but inputs in meters are just as possible if the other entries like Young's modulus and cross-sectional area also refer to meters (or yards or inches). The Young's modulus is 200,000 N/mm^2, Poisson's ratio 0.3, the cross-sectional area 500 mm^2 each.

This example is taken from the (very good) book SCHWARZ, H.R.: FORTRAN Programme zur Methode der Finiten Elemente. Teubner Verlag, Stuttgart, Germany 1984.



Z88I1.TXT:

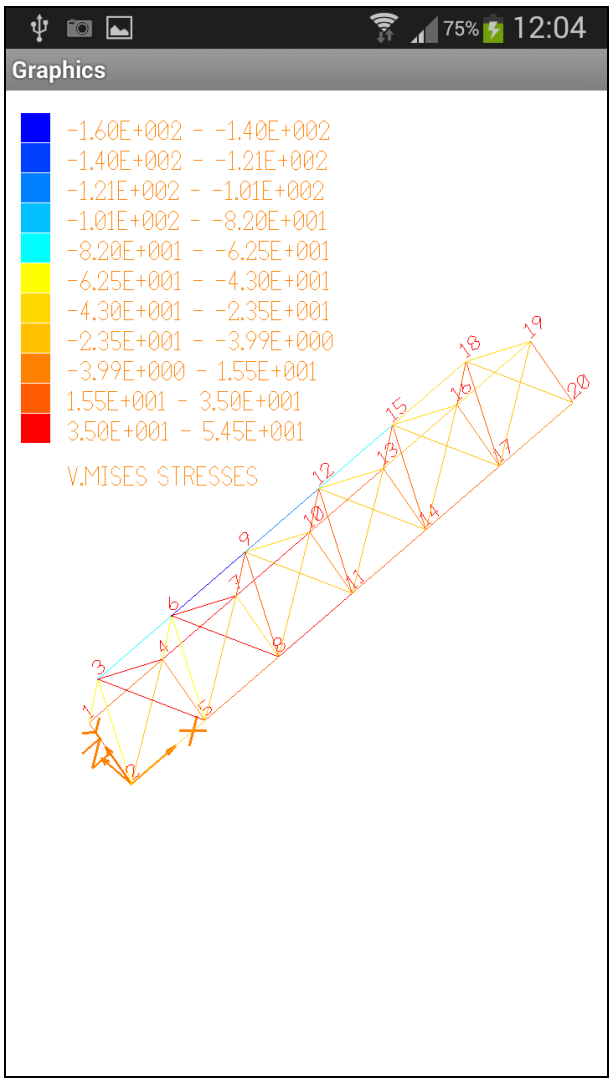
```
3  20  54  60  1  0  0  0  0  (3-dim, 20 nodes, 54 elements, 60 DOF, 1 mat info
line, flags 0)
 1  3    0    2000     0     (1st node, 3 DOF, X, Y and Z coordinate)
 2  3    0       0     0     (2nd node, 3 DOF, X, Y und Z coordinate)
 3  3   1000  1000  2000
 4  3   2000  2000     0
 5  3   2000     0     0
.....                         (nodes 6 ..18 dropped here)
19  3  12000  2000     0
20  3  12000     0     0
 1    4              (1st element, type Truss No.4)
 1    2              (coincidence 1st element)
 2    4              (2nd element, type Truss No.4)
 4    5              (coincidence 2nd element)
 3    4
 7    8
.....               (elements 4 ..53 dropped here)
54    4
17   19
1  54  200000  0.3  1  500  (mat info from ele 1 to 54,Young's modulus,Poisson's
ratio, INTORD (any), QPARA is cross-section area of the trusses)
```

The structure should be fixed to the node 1, 2 and 19, 20. A load of 30,000 N each is applied to the nodes 7 and 8. The load should be applied downward, therefore -30,000 N:

Z88I2.TXT:

```
10                              (10 boundary conditions)
 1    2    2    0              (Node 1, DOF 2, Displacement 0 (=fixed in Y direction)
 1    3    2    0              (Node 1, DOF 3, Displacement 0 (=fixed in Z direction)
 2    1    2    0              (Node 2, DOF 1, Displacement 0 (=fixed in X direction)
 2    3    2    0              (Node 2, DOF 3, Displacement 0 (=fixed in Z direction)
 7    3    1    -30000         (Node 7, DOF 3, load -30,000)
 8    3    1    -30000
19    1    2    0
19    3    2    0
20    2    2    0
20    3    2    0
```

Reduced stresses are not provided in Z88Tina for Trusses No.4. But Z88Tina does it because tensile stresses of trusses are equivalent to *von Mises* stresses. Why don't you try it?

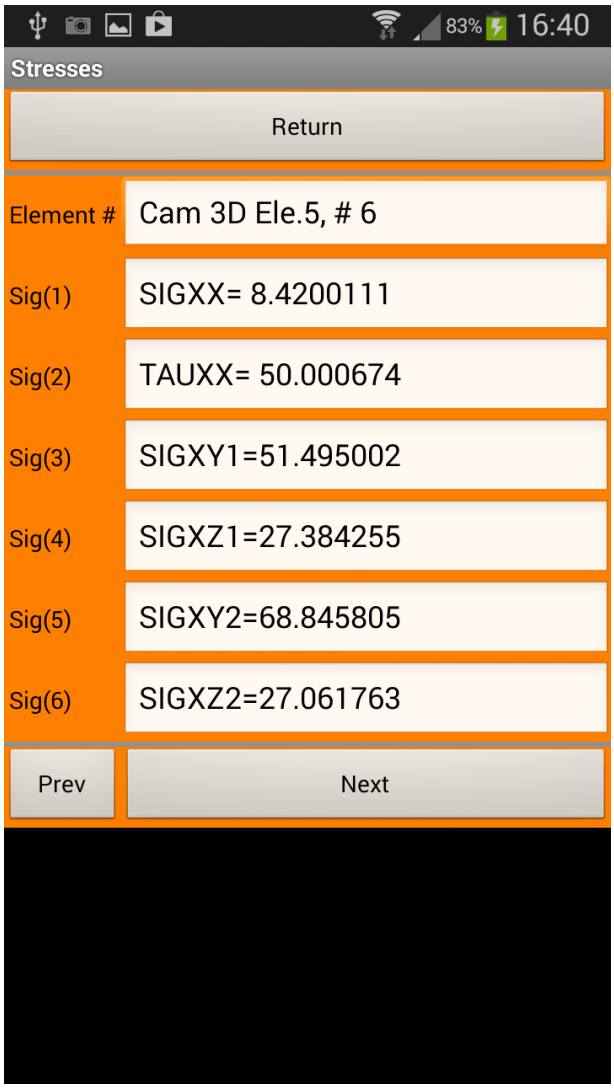


And indeed, there is an element with ~ -160 N/mm^2 (*Results* → *Stresses* → Ele # 45) and one with ~ 54.5 N/mm^2 (*Results* → *Stresses* → Ele # 38).

5.2 TRANSMISSION CAM WITH CAM ELEMENTS NO.5

Copy the example files Z88I1.TXT and Z88I2.TXT from www.z88tina.de , directory **z88-3**. However, this example is so short und simple that you should consider to enter the data interactively.

Basically, the calculation and displaying of *reduced stresses* is not provided in Z88Tina for cams No.5, because newer literal sources state correctly that reduced stresses for cams and other machinery parts under dynamic loads do not only depend on the normal and direct stresses (which are computed by Z88Tina), but also on stress concentration factors (impossible to calculate in Z88Tina, Z88Aurora and other FEA systems with beam or cam elements; however, plane stress elements like No.14 would do this job, of course) and other factors. Of course, you'll get the several stresses for each cam element, and it's up to you to compute reduced stresses on your own:



Task: A transmission cam is designed as follows:

- Cam section, D= 30 mm, L = 30 mm , fixed bearing at the left end
- Gear wheel 1, reference circle D = 45 mm, L = 20 mm

- Cam section, D = 35 mm, L = 60 mm, moveable bearing in the middle
- Gear wheel 2, reference circle D = 60 mm, L = 15 mm
- Cam section, D = 40 mm, L = 60 mm, moveable bearing at the right end

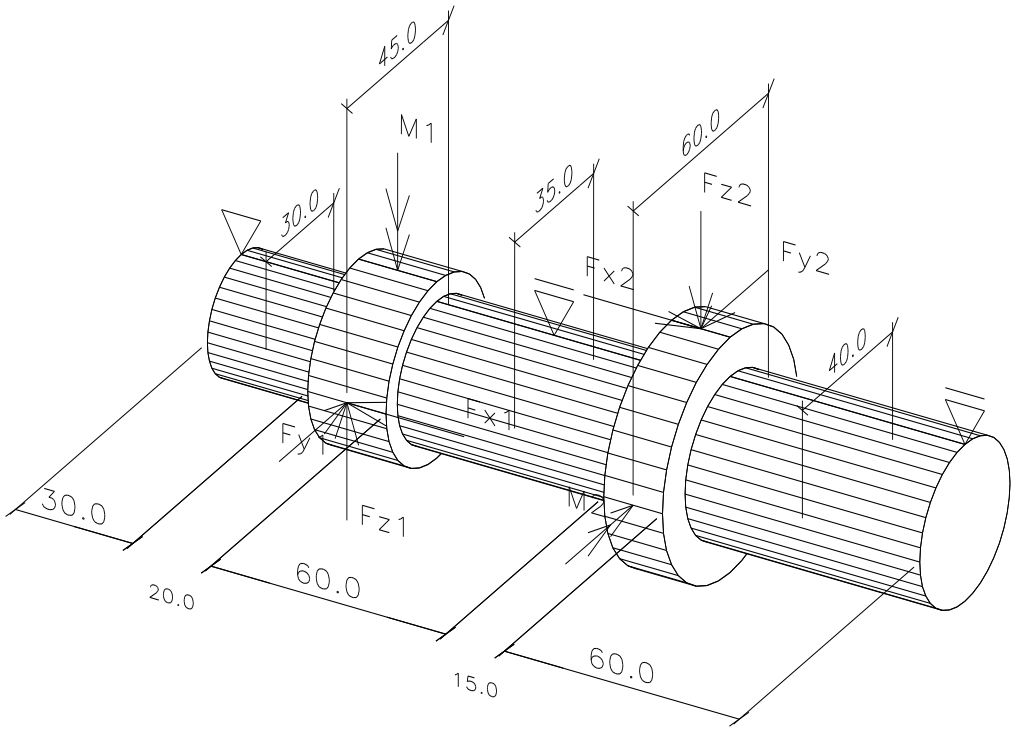
For the loads we picture the cam with the following coordinate system: If we look onto the cam as the main view, then the origin should be at the left end in the middle of the cam. X runs along the cam, Z runs to the upper direction, Y runs in the rear.

Gear wheel 1 gets the following loads in the (physical) point X1 = 40, Y1 = -22.5, Z1 = 0: Fx1 = -10,801 N, Fy1 = 6,809 N, Fz1 = 18,708 N. Fx1 results in a bending moment M1 around the Z axis of -243,023 Nmm.

Gear wheel 2 gets the following loads in the (physical) point X2 = 117.5, Y2 = 0, Z2 = 30: Fx2 = 8,101 N, Fy2 = -14,031 N, Fz2 = -5,107 N. Fx2 results in a bending moment M2 around the Y axis of -243,030 Nmm.

This results in loads in XY and XZ plane. The "physical" points do not exist in the FE calculation, of course, because a cam element is formed analytically only of two points along an axis. The Y and Z coordinates are always 0.

The cam is subdivided into eight cam elements No.5 = 9 nodes. The bearings are assumed in the nodes 1, 5 and 9. Very important: Node 1 is fixed in addition in the degree of freedom 4 (the torsion degree of freedom) in order to compute the torsion angle between the two gears. Otherwise, the structure is statically under-defined !



Z88I1.TXT:

```
3   9   8 54   3   0   0   0   0   (3D, 9 Node, 8 Ele, 54 DOF, 3 E-Gesetze, Flags 0)
1   6     0     0     0             (Node 1, 6 DOF, X-, Y- und Z-Koordinate)
```

```
2    6    30     0     0          (Node 2, 6 DOF, X-, Y- und Z-Koordinate)
3    6    40     0     0
4    6    50     0     0
5    6    80     0     0
6    6   110     0     0
7    6   117.5   0     0
8    6   125     0     0
9    6   185     0     0
1    5                          (Element 1, cam No.5)
1    2                          (Coincidence Ele 1)
2    5                          (Element 2, type 5)
2    3                          (coincidence Ele 2)
..... (Elemente 3 to 7 dropped here)
8    5
8    9
1   3   206000 0.3 1 30        (mat info from Ele 1 to 3,Young's,Poisson's, QPARA= 30)
4   6   206000 0.3 1 35        (mat info from Ele 4 to 6,Young's,Poisson's, QPARA= 35)
7   7   206000 0.3 1 40        (mat info from Ele 7 to 7,Young's,Poisson's, QPARA= 40)
                               INTORD is set here to 1, has no influence.
```

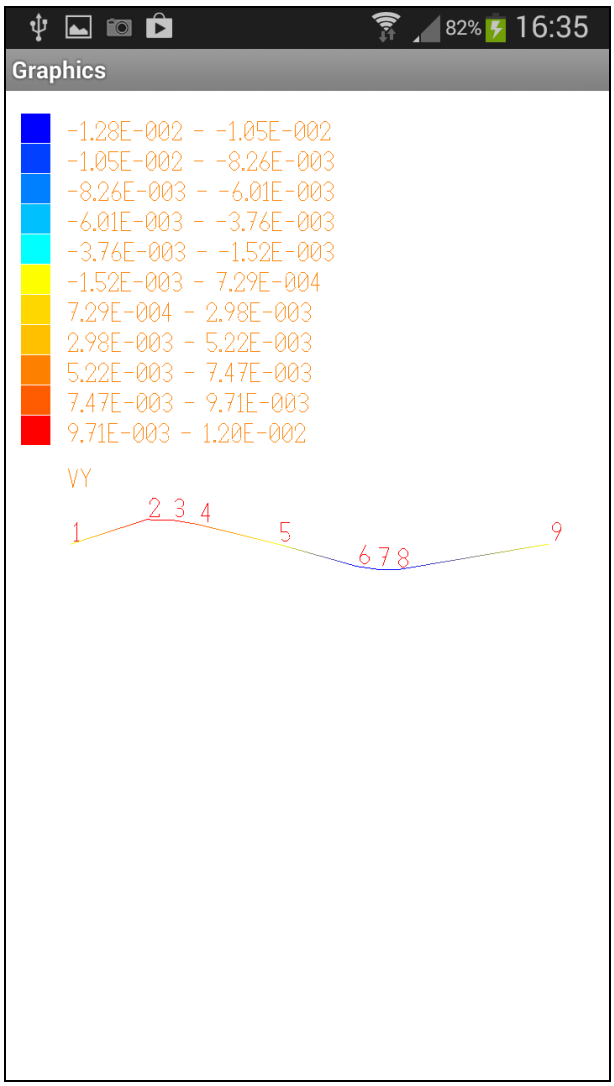
The boundary conditions Z88I2.TXT:

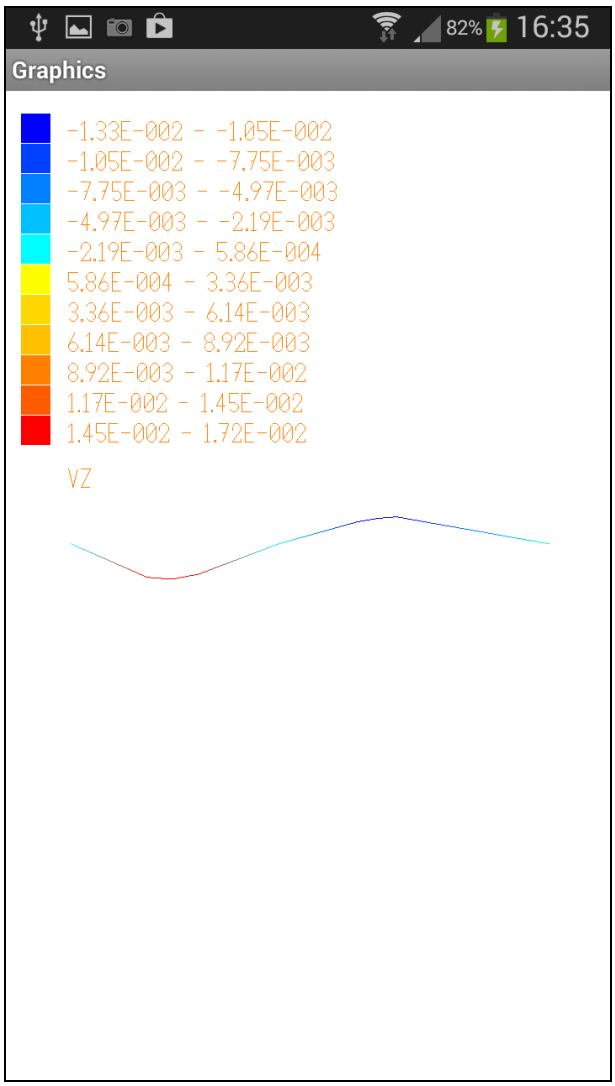
```
18                             (18 Boundary conditions)
1   1   2   0                  (Node 1, DOF 1 (=X) fixed)

1   2   2   0                  (Node 1, DOF 2 (=Y) fixed)
1   3   2   0                  (Node 1, DOF 3 (=Z) fixed)
1   4   2   0                  (Node 1, DOF 4 (=torsion) fixed)
3   1   1   -10801            (Node 3, DOF 1 (=X), load -10,801 N)
3   2   1   +6809             (Node 3, DOF 2 (=Y), load 6,809 N)
3   3   1   +18708            (Node 3, DOF 3 (=Z), load 18,708 N)
3   4   1   -420930           (Node 3, DOF 4 (torsion) -420,930 Nmm)
3   6   1   -243023           (Node 3, DOF 6 (bend. moment around Z),-243,023Nmm)
5   2   2   0
5   3   2   0
7   1   1   +8101
7   2   1   -14031
7   3   1   -5107
7   4   1   +420930
7   5   1   -243030
9   2   2   0
9   3   2   0
```

Results:
Keep in mind, that the "forces" of the DOF 4, 5 and 6 are really moments, because the DOF 4, 5 and 6 are rotations.

The following pictures of the plot program show the deflected structures in the XY and the XZ plane:


82% 16:35
Graphics
-1.28E-002 - -1.05E-002
-1.05E-002 - -8.26E-003
-8.26E-003 - -6.01E-003
-6.01E-003 - -3.76E-003
-3.76E-003 - -1.52E-003
-1.52E-003 - 7.29E-004
7.29E-004 - 2.98E-003
2.98E-003 - 5.22E-003
5.22E-003 - 7.47E-003
7.47E-003 - 9.71E-003
9.71E-003 - 1.20E-002
VY
1
2 3 4
5
6 7 8
9



5.3 BEAM IN PLANE WITH BEAMS NO.13

Copy the example files Z88I1.TXT and Z88I2.TXT from www.z88tina.de , directory **z88-7**.

This example deals with a beam, fixed on both sides, and loaded with 1,648 N in the middle in downward direction. This mechanical problem is covered in every mechanical and civil engineering handbook. Geometry: Length ℓ= 1,000 mm, cross-cut 50 x 10 mm. Thus: A = 500 $mm^{2,}$ I_{zz} = 4,167 mm^4, e_{zz} = 5 mm.

The deflection curve has inflection points, we therefore take 4 beams No.13. Nodes 1 and 5 will be fixed and node 3 is loaded.

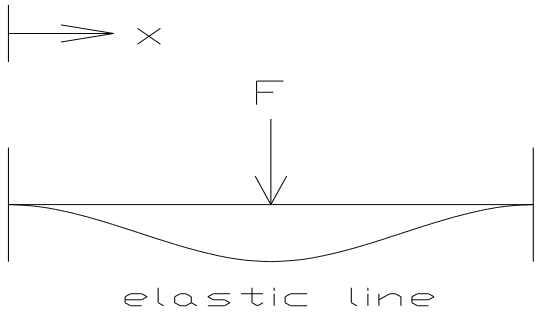
You would calculate analytically:

f in the middle: $f = \frac{F\ell^3}{192\,EI} = 10\,\text{mm}$

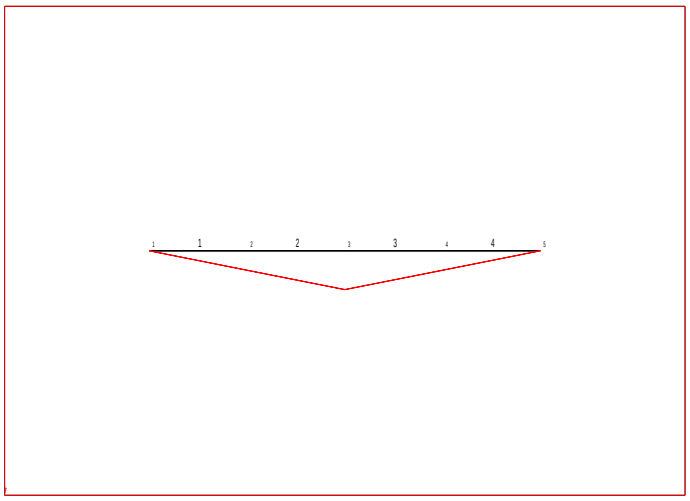
f in the inflection points: $f_w = \frac{f}{2} = 5\,\text{mm}$

The bending moments on the left, middle, on the right: $M_b = \frac{F\ell}{8} = 206.000\,\text{Nmm}$

The slope angle in the inflection points: $\psi = \text{atan}\left(\frac{3f}{\ell}\right) = 0{,}029991 \text{ rad}$



This example shows *that a FEA basically needs nodes in all locations where you want to get results.* As the beam is fixed left and right, the maximum of displacements appears in the middle for x = ℓ/2, but the bending curve features two inflection points for x = ℓ/4 and x = 3ℓ/4. To calculate results for this locations, the structure must be subdivided with nodes in x = 0, x = ℓ/4, x = ℓ/2 and x = 3ℓ/4. The undeflected and the deflected structure would like this because Z88Tina connects the nodes with straight lines, although the deflection curve represents a cubic parable in the case of a Beam No.13 or No.2 or Cam No.5. This means: Z88Tina shows the deformations correctly for the node, but straight lines are between the nodes. Therefore, no deflection curve is shown. If you want to plot a real nice deflection curve with Z88Tina, then use basically more nodes, e.g. 15 to 20 nodes for this example (the cubic bending curve is then featured by a couple of straight lines).



Thus, we'll subdivide the length ℓ into *20* Beam Elements No.13:

Z88I1.TXT so becomes:

```
 2    21    20   63    1    0    1    0    0      (2-D, 21 nodes, 20 ele, 63 DOF, 1
mat info, KFLAG 0, IBFLAG 1, IPFLAG 0, IQFLAG 0)
 1    3      0    0                  (1.node, 3 DOF, X and Y coordinate)
 2    3    250    0
...
21    3   1000    0
 1   13                 (1. element, type beam in plane No.13)
 1    2                 (coincidence for 1. element)
 2   13
```

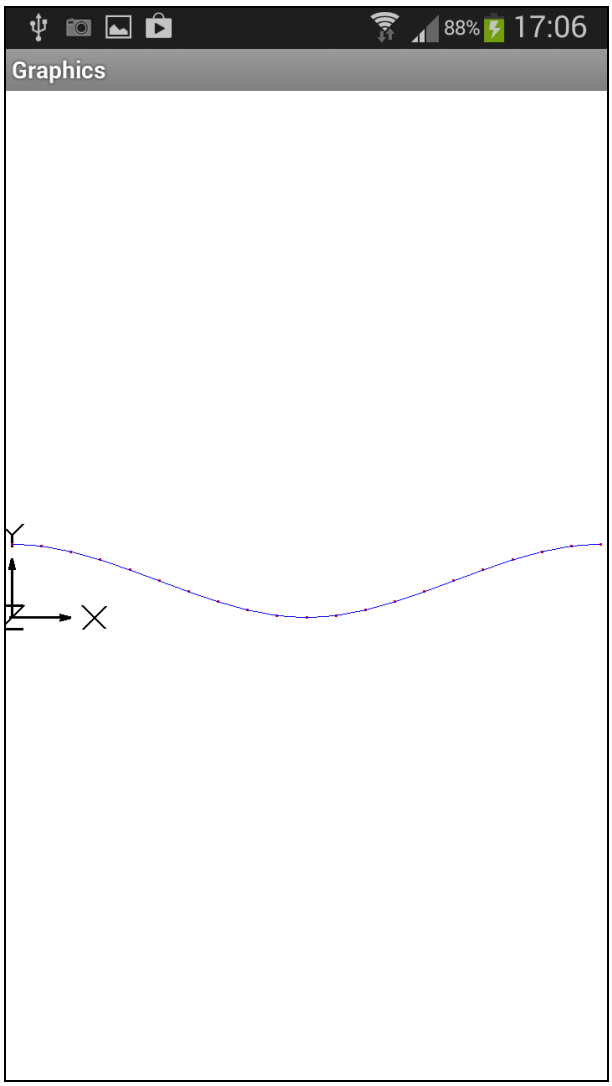
```
 2    3
...
20   13
20   21
1    20 206000 0.3 1 500 0 0 4167 5 0 0
```
(mat info for ele 1 to 4, Young's, Poisson's, INTORD (any), QPARA = area, I_{xx}=0, e_{xx}=0, I_{zz}, e_{zz}, I_t=0, W_t=0)

The node 1 is fixed in all degrees of freedom at the boundary conditions. It is important to fix especially the DOF 1 = displacement in X direction so that the structure cannot move. Node 5 is fixed in DOF 2 = displacement in Y direction and DOF 3 = rotation around Z axis. You could also fix DOF 1 for node 5, if you wish. But in reality one of the bearings or supports will allow for thermal expansion. This was taken into account in Z88I2.TXT.

Here Z88I2.TXT:

```
6                    (6 Boundary conditions)
 1    1    2        0        (Node 1, DOF 1 = X gets a displacment of 0 = DOF 1 fixed)
 1    2    2        0        (Node 1, DOF 2 = Y fixed)
 1    3    2        0        (Node 1, DOF 3= Mb  fixed (restraining moment)
11    2    1    -1648        (Node 3, DOF 2 gets load of -1,648 N)
21    2    2        0        (Node 21, DOF 2 = Y fixed)
21    3    2        0        (Node 21, DOF 3= Mb  fixed (restraining moment)
```

Such simple examples are well suitable to become aware of the sign definitions. Experiment with this example and calculate other bend cases from good handbooks. Frameworks with Beams No.2 are calculated accordingly. However, a real spatial structure then must be available: At least one Z coordinate must not equal 0.

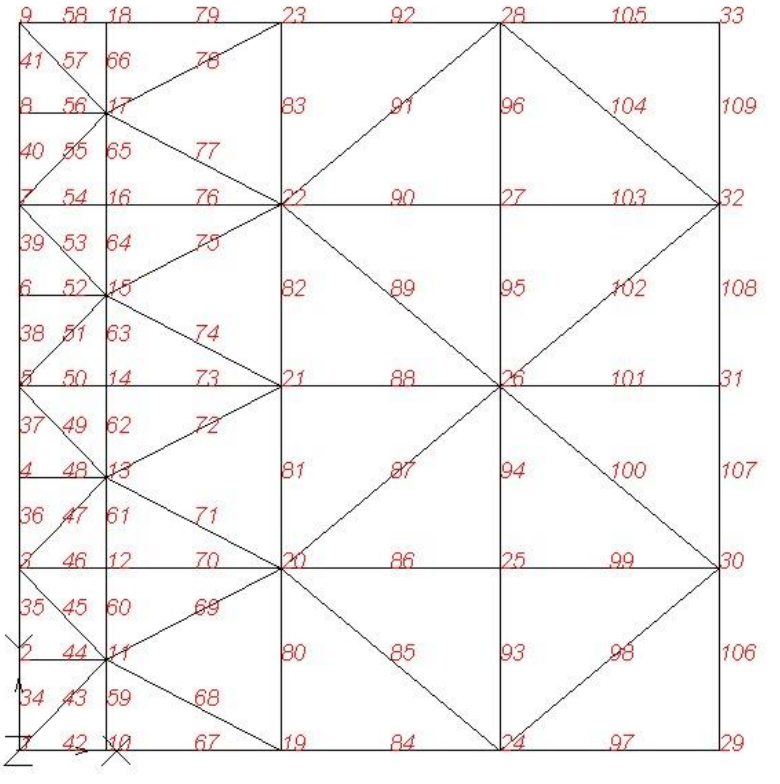


As you see we've got a quite nice line of deflection.
Keep in mind: Place nodes into your structure where you want to get results!

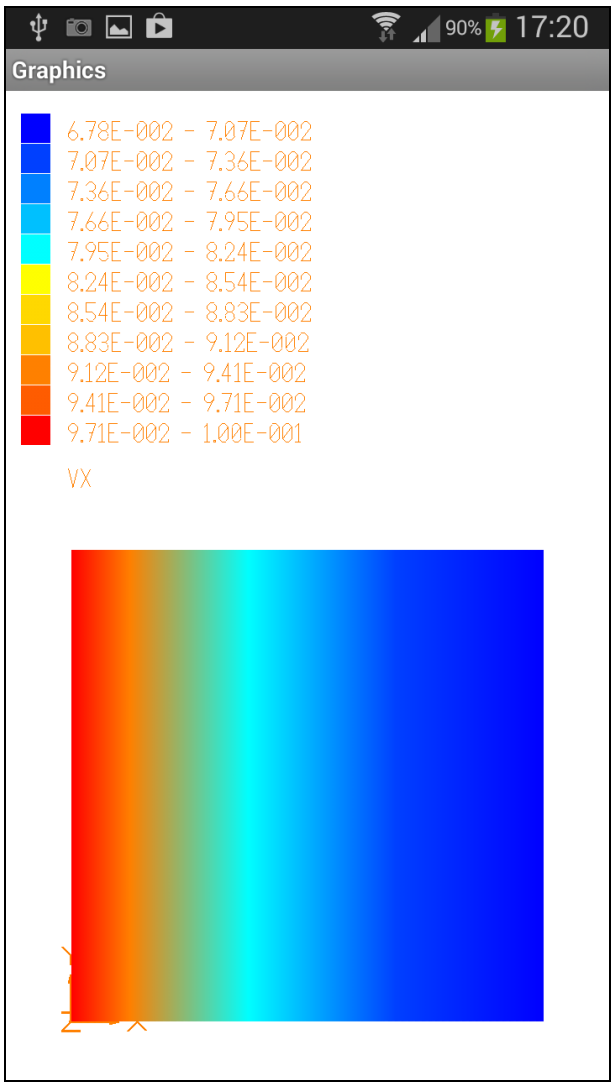
5.4 PIPE UNDER INTERNAL PRESSURE, TORUS NO.15

Copy the example files Z88I1.TXT and Z88I2.TXT from www.z88tina.de , directory **z88-9**.

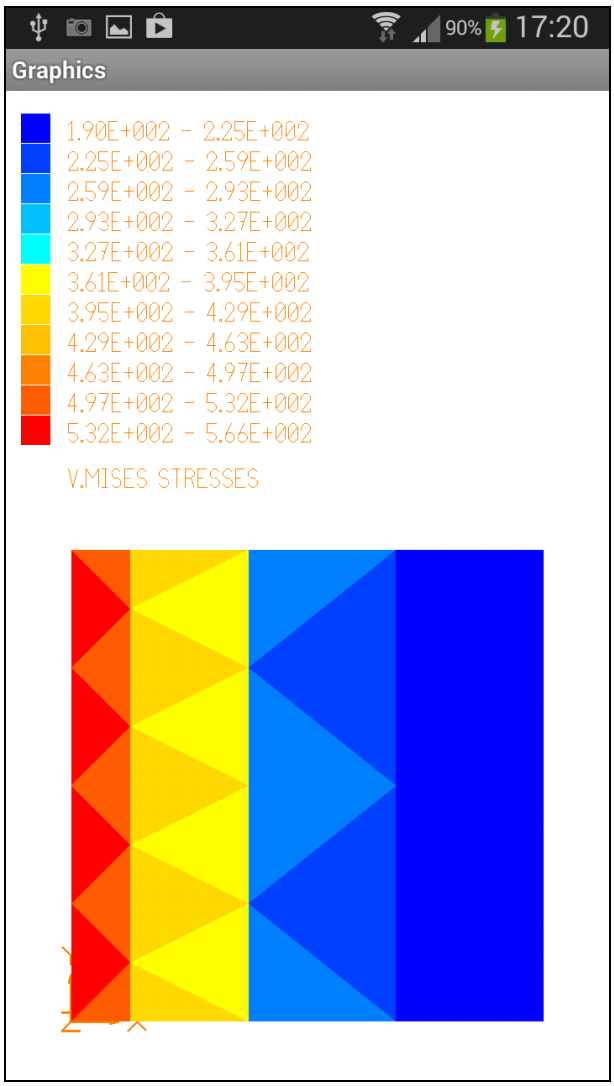
We look at a pipe under internal pressure. Pipe inside diameter 80 mm, pipe outside diameter 160 mm, length 40 mm. For Torus elements the cross-section of the pipe is important.
The inside radius shall be expanded by 0.1 mm = r_d (press fit). Attach this displacements to the nodes from 1 to 9 and from 34 to 41. To fix the structure in space, e.g. fix node 31 in Z direction.



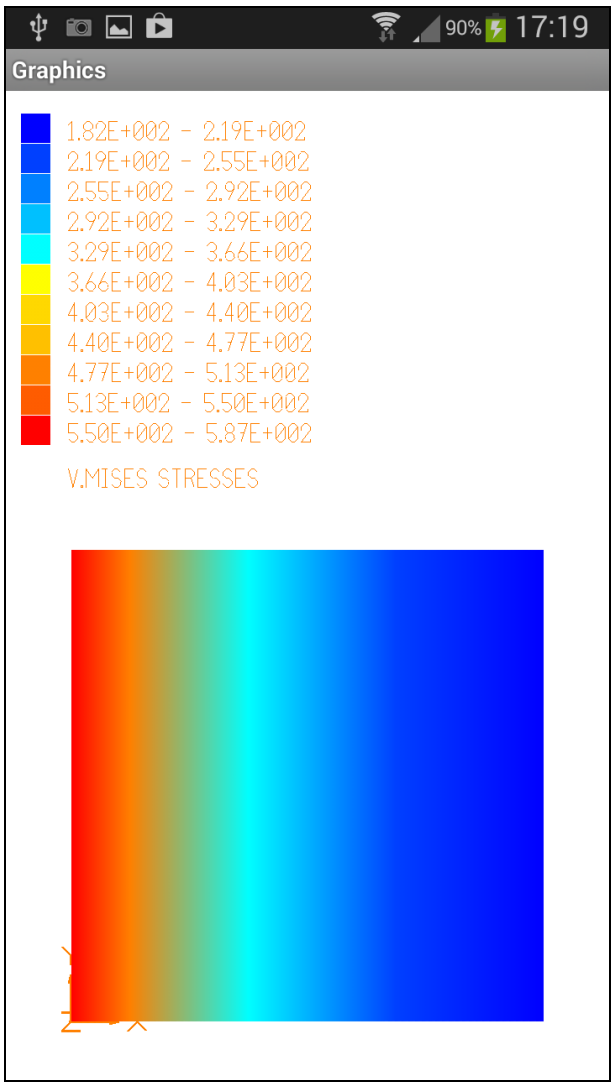
The results are:



The deflections in X (= R) direction.



The *v.Mises* stresses per element.



The *v.Mises* stresses at the corner nodes.

The force is: $F = p\,A = p\,2\,\pi\,r_i\,\ell = 2.633.911\,\mathrm{N}$

One calculates analytically (assuming Young's modulus E= 206000 N/mm² and Poisson's ratio υ= 0,3):

pressure: $$p = \frac{r_d E}{r_i} \cdot \frac{1}{\frac{1+q_a}{1-q_a} + \nu} = 262\ \mathrm{N/mm^2} = 2.620\ \mathrm{bar} \quad with \quad q_a = \frac{r_i^2}{r_a^2} = 0,25$$

radial stresses: $$\sigma_{r_i} = -p = -262\ \mathrm{N/mm^2}$$
$$\sigma_{r_a} = 0$$

tangential stresses: $$\sigma_{t_i} = p \cdot \frac{1+q_a}{1-q_a} = 437\ \mathrm{N/mm^2}$$
$$\sigma_{t_a} = 2p \cdot \frac{q_a}{1-q_a} = 175\ \mathrm{N/mm^2}$$

von Mises stresses: $\sigma_{Vi} = \sqrt{\sigma_r^2 + \sigma_\theta^2 - \sigma_r \cdot \sigma_\theta} =$

$$\sqrt{(-262)^2 + 437^2 - (-262) \cdot 437} = 612 \text{ N/mm}^2$$

So what? The highest *v.Mises* stresses are 587 N/mm^2, which is considerably lower than the analytical results of 612 N/mm^2.

In fact, it is not possible to calculate the stresses exactly in the corner points with good results – this is true for all *FEA programs*, neither Z88Tina nor Z88Aurora nor any of those expensive commercial FEA programs can do this! As it was described Z88Tina looks for the nearest Gaussian points to the respevtive corner and calculates then an average value. To get the exact results we would have to now the stresses in the Gaussian points (this is only possible with our full-scale FEA program Z88Aurora or the open source version Z88V14OS, see www.z88.de). Even then you are not ready to go: You'll have to extrapolate the stresses in the Gauss points which lare located inside an element to the according edges. Then you will see results which are very precise.

Please understand the stress calculations and stress plots of Z88Tina to give you a first impression.