

Short manual to the EXCEL-tool: TRUSS

1. Announcements / Preparations

Announcements

This short manual to the EXCEL-based tool TRUSS which accompanies the course in Non-linear FEM shall help to give a first introduction and some hints concerning the possibilities and the handling of the software.

Where to download:

EXCEL-based tool TRUSS can be downloaded at the Moodle page of the course Non-linear FEM or the homepage of the chair of structural analysis under Home/Software/Education/Truss

Content:

The manual is subdivided in three main parts:

- Sections 2-7 explain the basic steps in a simple example (two-bar truss)
- Section 8 presents the different solution algorithms (i.e. path following methods) and section 9 illustrates some special features
- Section 10 and 11 explain how to export & import models.
- Section 12 answers some questions and shall give hints to work around some bugs

If you would like to enhance the tutorial with more hints you are cordially invited.

Preparations

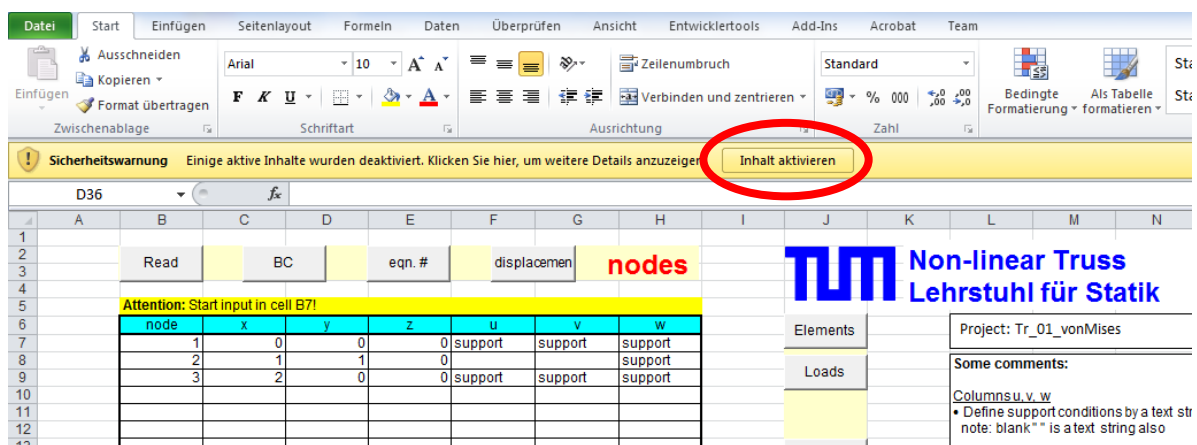
Generally: This tutorial implies the use of Microsoft EXCEL® 2007, 2010 or 2013.

General preparations in Excel:

TRUSS consists of one main file *Truss-16.xlsm*, which is an EXCEL® file with macros. For this reason you have to enable the use of macros in EXCEL®. There are two ways to do so:

1. Standard settings:

By default macros are blocked and EXCEL® and when opening the main file *Truss-16.xlsm* EXCEL® asks if you would like to activate them. You have to confirm this.



The screenshot shows the Microsoft Excel 2010 interface. A yellow warning bar at the top states: "Sicherheitswarnung Einige aktive Inhalte wurden deaktiviert. Klicken Sie hier, um weitere Details anzuzeigen" with a button "Inhalt aktivieren" circled in red. The spreadsheet data is as follows:

node	x	y	z	u	v	w
1	0	0	0	support	support	support
2	1	1	0	support	support	support
3	2	0	0	support	support	support

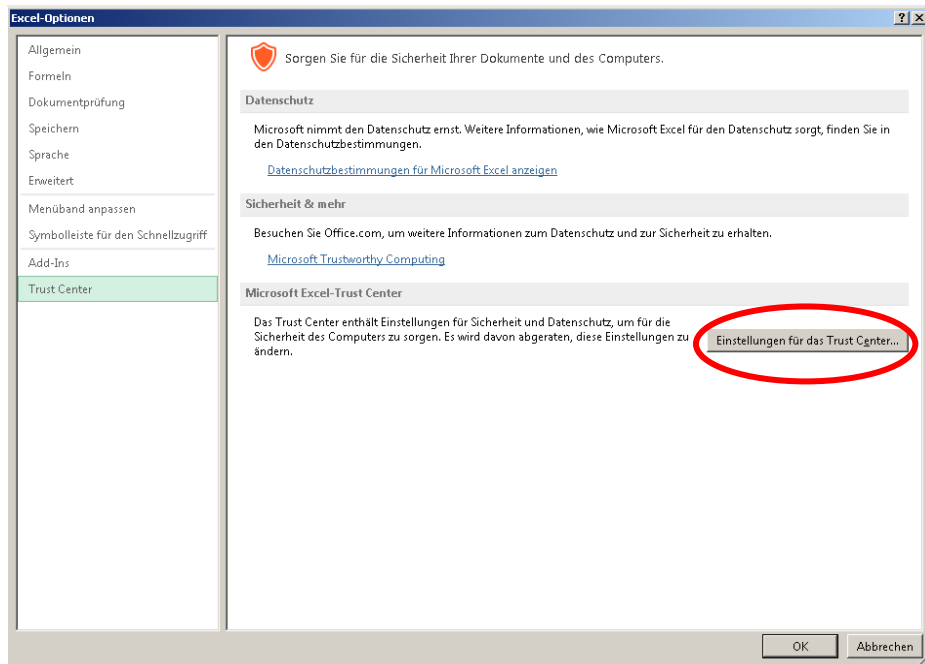
Other visible elements include the ribbon (Datei, Start, Einfügen, etc.), a sidebar with "Elements" and "Loads" buttons, and a "Comments" section with the text: "Columns u, v, w • Define support conditions by a text string note: blank "" is a text string also".

2. Allow macros by default:



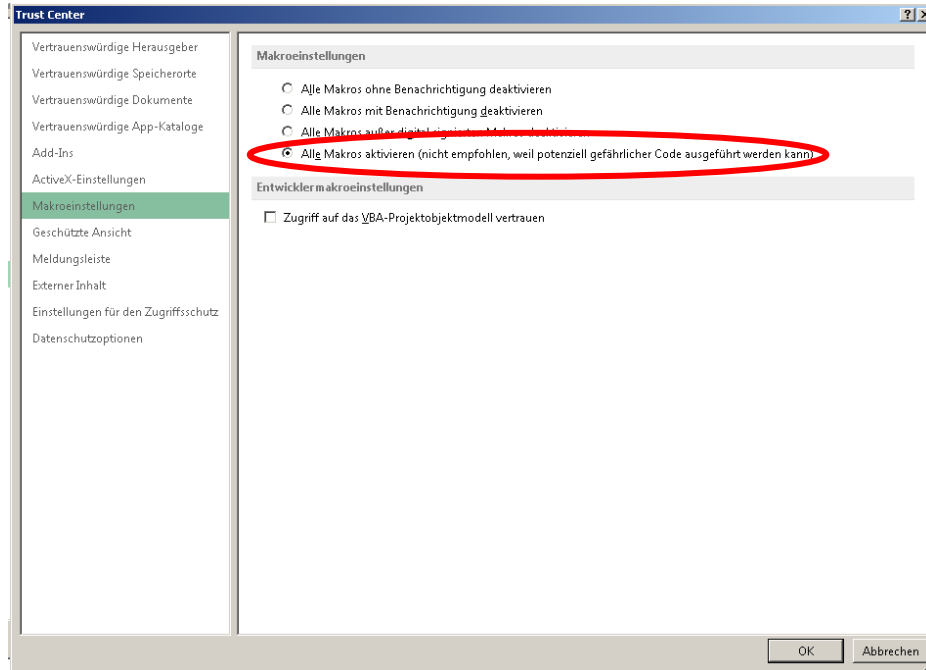
Main Menu in EXCEL 2013

Then go to *Trust Center* and click on *Trust Center Settings*.



Trust Center in EXCEL 2013

Go to *Macro Settings* and activate the check box *Enable all macros*.



Trust Center in EXCEL 2013

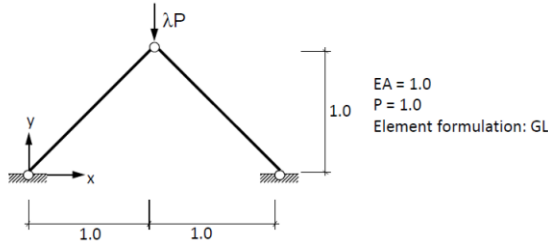
Remark: This option allows Excel in general the usage of Macros!

Start a calculation

- Download the program from the website of the course NFEM
- Unzip the file to a local directory (e.g. Desktop)
- Open the file *Truss-16.xlsm* (above of Cell A1, a security warning dialog appears; click on “Options” and mark the buttons “enable this content” in order to activate the macros).
- The file is already filled with the input of the following 2-bar-truss example.

In the following, the principles of how to work with the program will be explained in a short example. If you encounter an error message, have a look at Section 10.

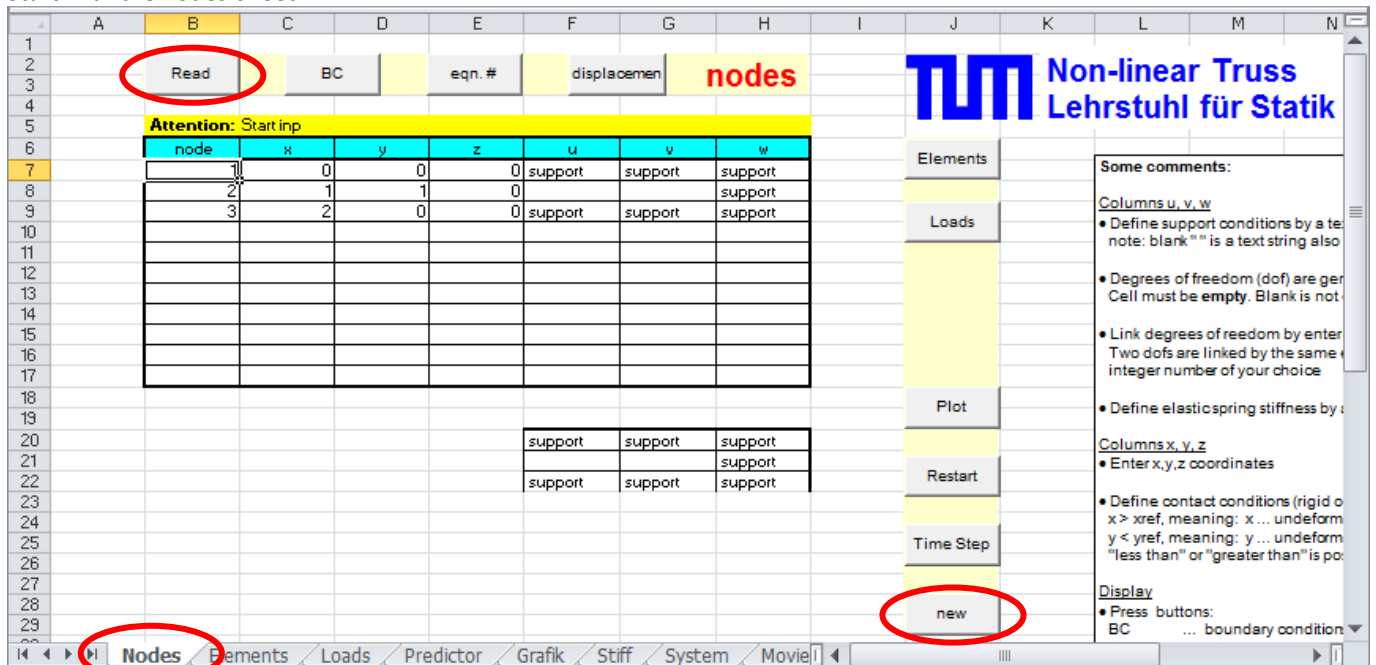
Example:



Task: Analyse the deformation behaviour of the system given above. Follow the primary path using proper path-following methods and determine critical points.

2. Nodes & Boundary Conditions

Start with the nodes-sheet:



node	x	y	z	u	v	w
1	0	0	0	support	support	support
2	1	1	0			
3	2	0	0	support	support	support

- Initialize a new problem by clicking on **new**
- Enter the node numbers and the respective coordinates
- Enter the support conditions for every node:
Every node has 3 degrees of freedom (DOF) u,v,w in the global directions x,y,z which can be either fixed free. To ensure that a DOF is **fixed**, this means the deformation in this direction is blocked, enter any string in the appropriate cells (e.g. "support"). In case the DOF should be **free** leave the cell empty.
If the support consists of a **spring**, enter the spring's stiffness in these cells.
- Read in the entered data by clicking **Read** (you automatically jump to the Elements-sheet)
- By clicking on **BC**, **eqn.#** or **displacement**, you can switch between the following views in the Nodes-sheet:

BC	Boundary conditions (initial view)
eqn.#	Equation number of the specific DOF
displacement	displacement of the specific DOF

3. Elements

Element	Node i	Node k	E	A	F ₀	Formulation
1	1	2	1	1	0	GL
2	2	3	1	1	0	GL

- Enter the element number, its beginning (i) and ending node (k)
- Enter the Young's modulus **E** and the cross-section **A**
- **F₀** would be the pre-stress, which we don't need for the moment. Just leave it empty (i.e. = 0)
- Choose the element formulation. Use the acronyms explained in the blue box on the right.
- Read in the entered data by clicking **Read** (you jump automatically to the Loads-sheet)

4. Loads

Node i	F _x	F _y	F _z	Load factor
2	0	-1	0	1,00E-02

Step	Factor
1	0,01
2	0,02
3	0,03
4	0,05
5	0,1
6	0,0136
7	0,015
8	0,3
9	0,5
10	0,7
11	1

- Define the position of the loads by entering the node-number (column B)
- The components of the load (**F_x, F_y, F_z**) have to be given in global directions (negative sign → opposite direction to the global axis); per node only one set of loads (**F_x, F_y, F_z**) can be used.
- Enter the load factor in the red marked cell e.g. **0,01**. All loads will be scaled with the **one and only** load factor λ .
- Read in the entered data by clicking **Read** (you automatically jump to the Grafik-sheet)

5. Graphic

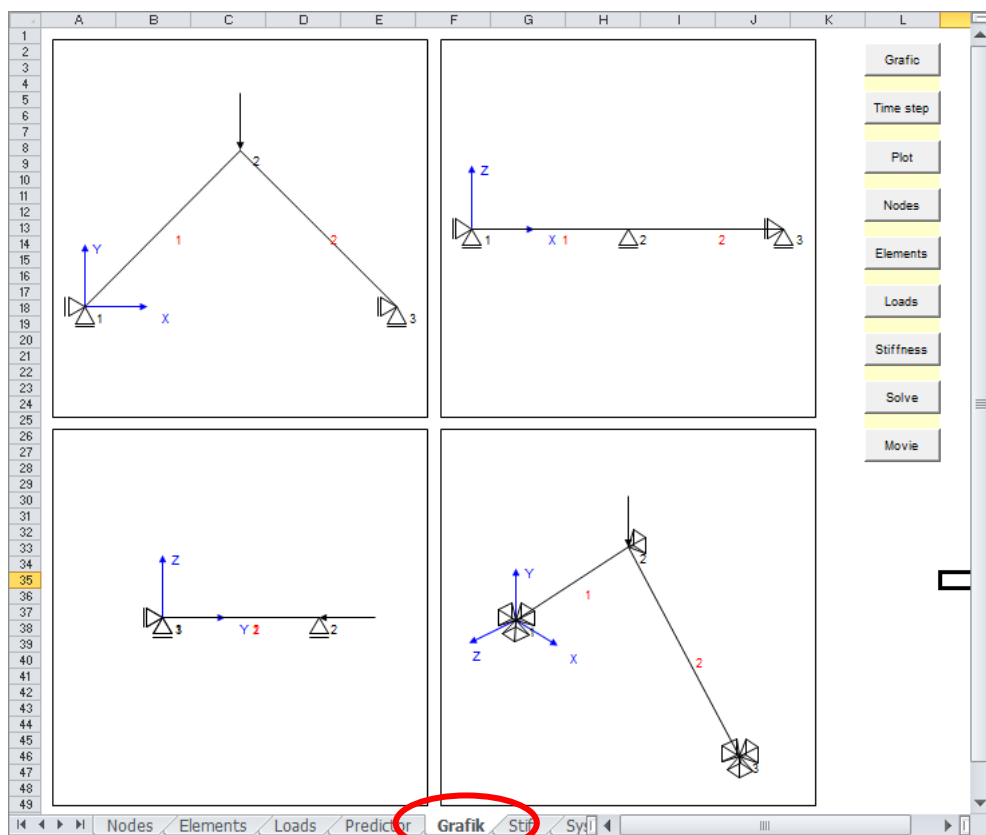
A pop-up menu appears, giving you different options. By confirming with **OK** you accept the options and come to the Grafik-sheet.

This sheet provides a view of the entered system. Different options and feature for the visualisation can be entered in the menu behind the **Grafik** -button.

You can visualize your system at any time. By clicking on **Time step** you can specify a certain time step. However, after choosing the time step the graphic still needs to be updated.

Note: The graphic is not updated automatically. For an updated graphic click on **Grafik** (you will come to the same pop-up menu as before) and then on **OK**

Click **Solve** (you automatically jump to the System-sheet)



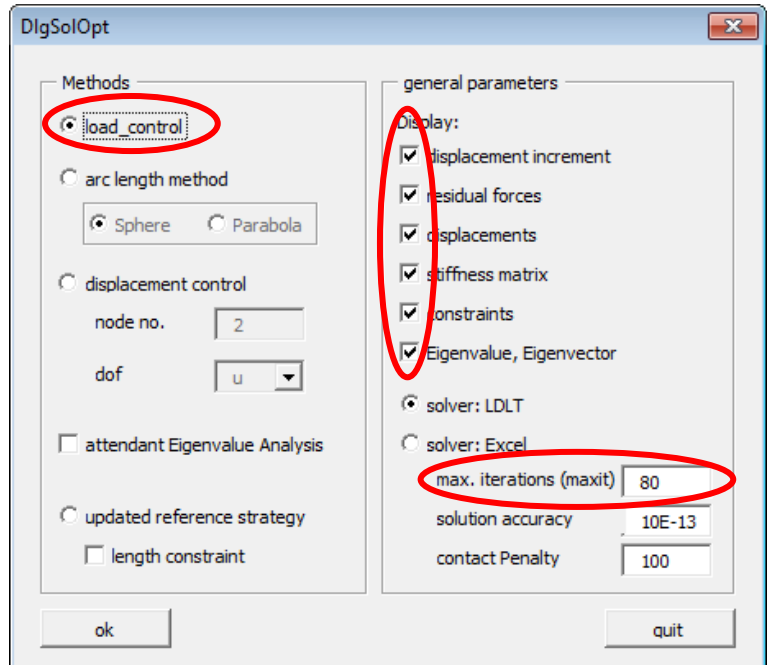
6. Solving the problem

A pop-up menu “solution options” automatically appears, giving three solution algorithms:

- load control
- arc length method
- displacement control

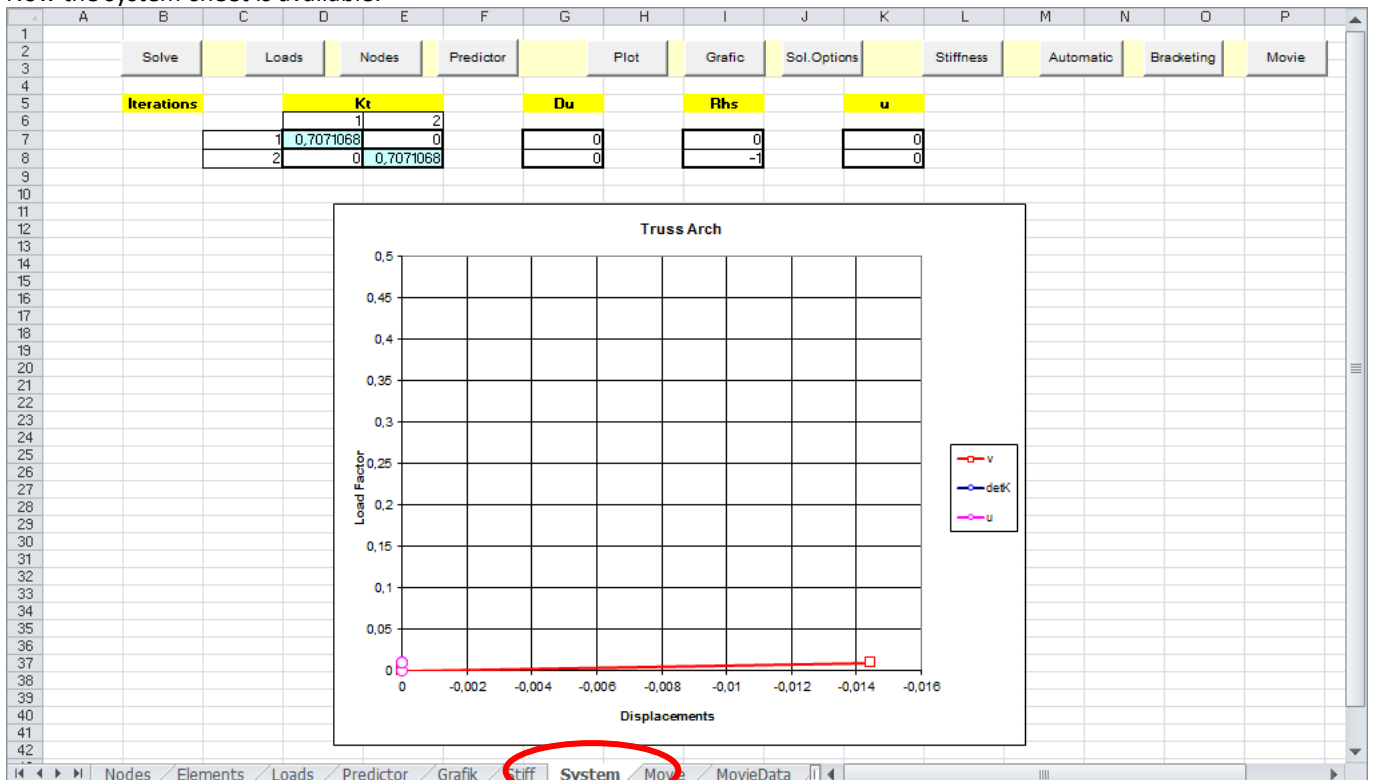
Note: these algorithms and their handling will be explained more in detail in section 7.

This dialog is also available by clicking **Sol.Options** in the System-sheet.



- Choose “load_control” as solution algorithm (for this example).
- Check all display-options in the “general parameters” column.
- Choose a solver and set the number of “max. iteration” to e.g. 80.
- Confirm with **OK**

Now the system-sheet is available:

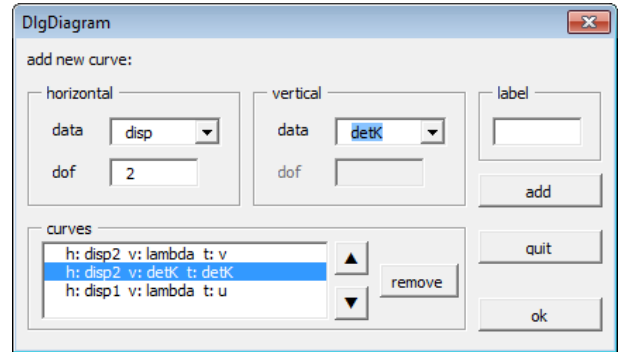


The diagram depicts the “Load Factor – Displacement” Curve.

The data to be shown in the diagram can be chosen by clicking on

Plot

In this dialog you can enter and add (or remove) specific curves of the diagram.



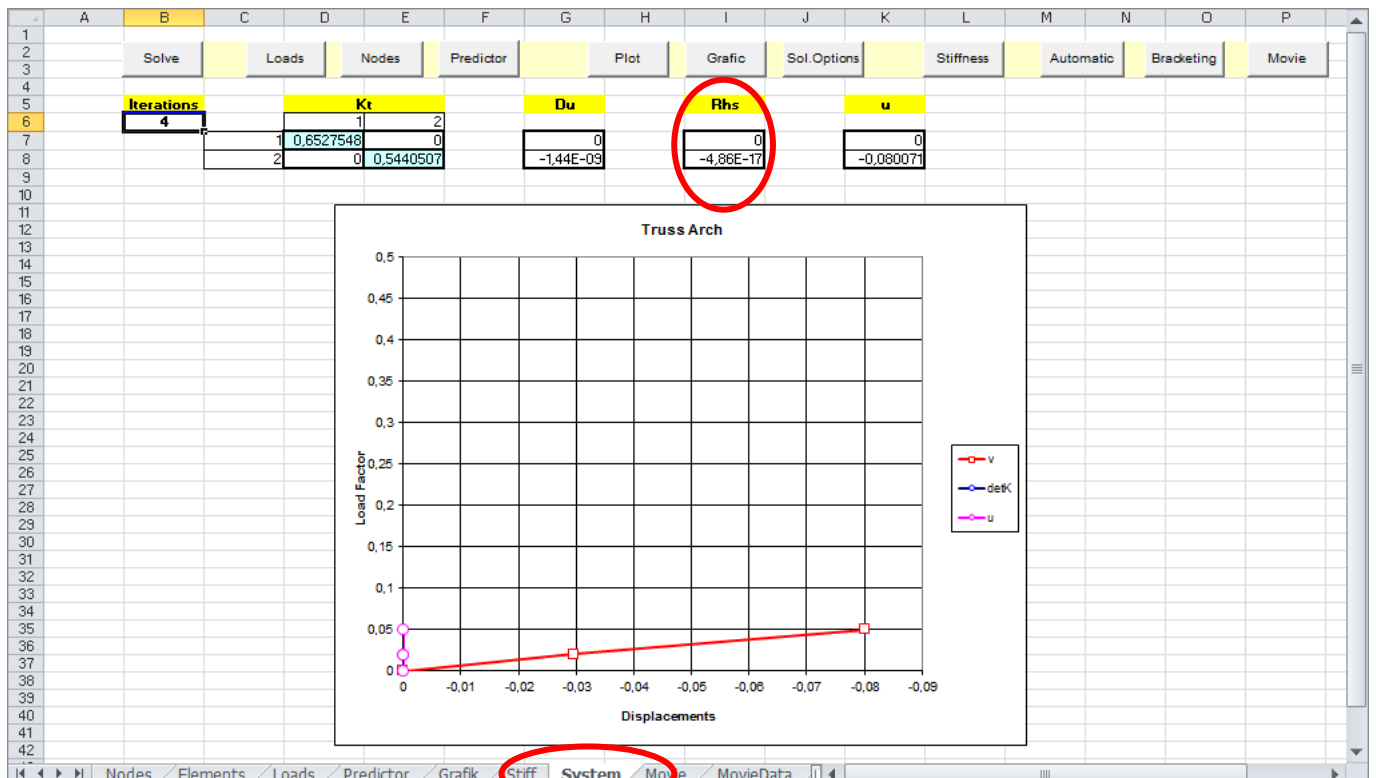
- To remove a line, select it and the click on **remove**
- To add a line, specify the data plotted in the horizontal and the vertical axis. For displacements (“disp”), select the DOF which should be used. You can find the DOF for a certain displacement of a node in the Nodes-sheet by clicking on **eqn.#**

- Do not forget to label the new data curve with a meaningful label.
- To add the new data curve click on **add**
- Confirm with **OK**

Above the diagram, you can see the equation system with the current stiffness matrix, the load, the residuum (RHS) and displacement for the specific DOFs.

Press **Solve** to solve the equation system and to go one step further on the path of equilibrium. The load-displacement curve will be updated accordingly.

For control: the residuum vector (right hand side vector) **Rhs** should be approximately equal to zero.



To compute the next time step on the path of equilibrium, click on **Loads** and enter a new load-factor in the Loads-sheet.

Confirm with **Read**

In the System-sheet press **Solve** to solve the new equation system. The load-displacement curve will be updated accordingly.

7. Movie

By clicking on **Movie** you come to the Movie-sheet where you can watch the results for each time step. The view of the system (Grafik-sheet) can be changed by rotating and moving.

Basic functions are:

- Scroll
- Rotation (all axes)
- Scaling factor
- Time steps

8. Solution algorithms

Clicking on **Sol. options** in the System-sheet gives options for the solution algorithm:

The goal of all algorithms is to find the next point of equilibrium on the unknown load-displacement curve.

- By using the "load control method" displacements are calculated to satisfy equilibrium to the given load factor.
- The "displacement control method" determines the load factor to satisfy equilibrium for prescribed displacements.
- The "arc length method" creates a scaled tangent vector (predictor) at the end of the existing equilibrium path. The initial point of this vector is the midpoint of a circle, the length is the radius. For the so-defined circle the intersection with the equilibrium path will be evaluated. This intersection is the next point of equilibrium, which is displayed in the plot. In general both, the load factor and the displacements, are changed at the same time by prescribing the predictor.

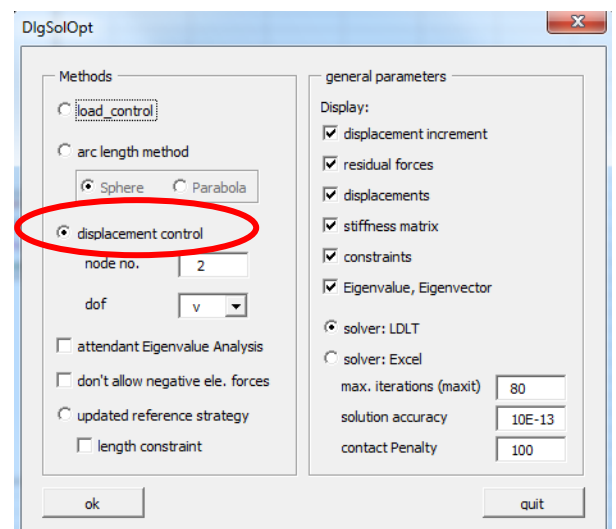
8.1 Load-control

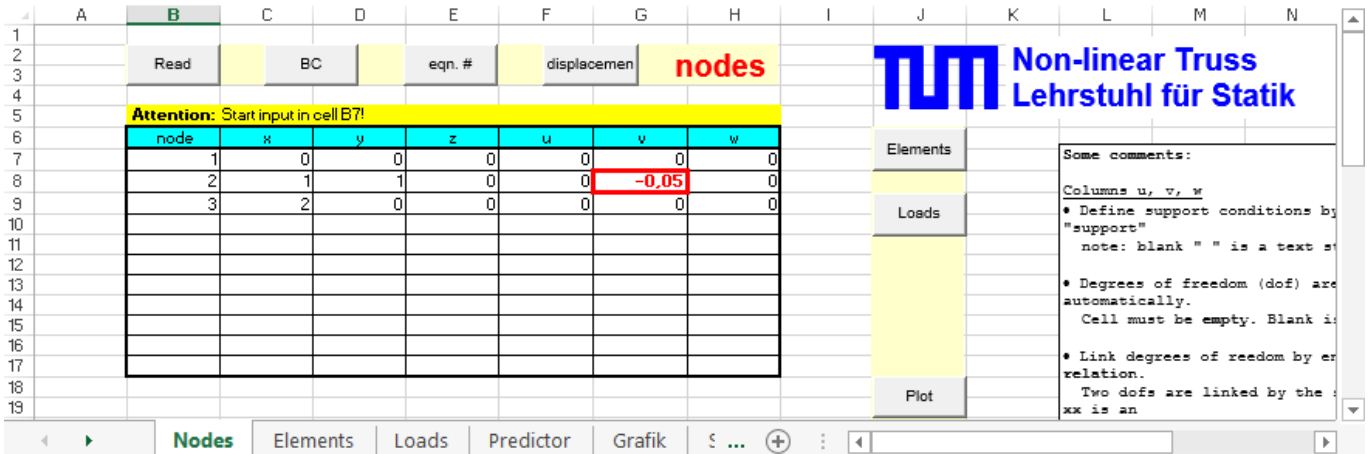
→ Already described in the example above.

8.2 Displacement Control

Click on **Solution options** in the System-sheet in order to get to the following dialog:

- Activate "displacement control" .
- This method requires the prescribed displacement for one DOF.
For our example this could be the vertical displacement of the top node: displacement v of node 2
- Confirm your input with **OK**
- Click on **Nodes** to get to the nodes-sheet. Here you can enter the prescribed displacement for the specified node in the now the *red marked cell*.





node	x	y	z	u	v	w
1	0	0	0	0	0	0
2	1	1	0	0	-0.05	0
3	2	0	0	0	0	0

- Read in the new displacement by clicking on **Read**.
- Back in the System-sheet, solve the equation system by clicking on **Solve**. The load-displacement curve will be updated accordingly.
- Repeat the last 3 steps for more incremental steps on the path of equilibrium.

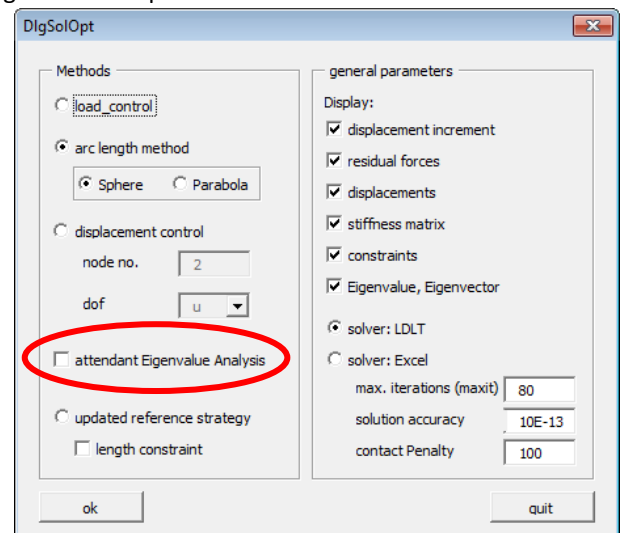
7.3 Arc Length Method

Hint: In the Nonlinear Truss Program, a secant is used instead of a tangent for the specification of the direction of the predictor. Therefore, it is necessary to do a first iteration-step with the "load control method" in order to obtain 2 points from the equilibrium path to determine the direction of the secant.

Procedure:

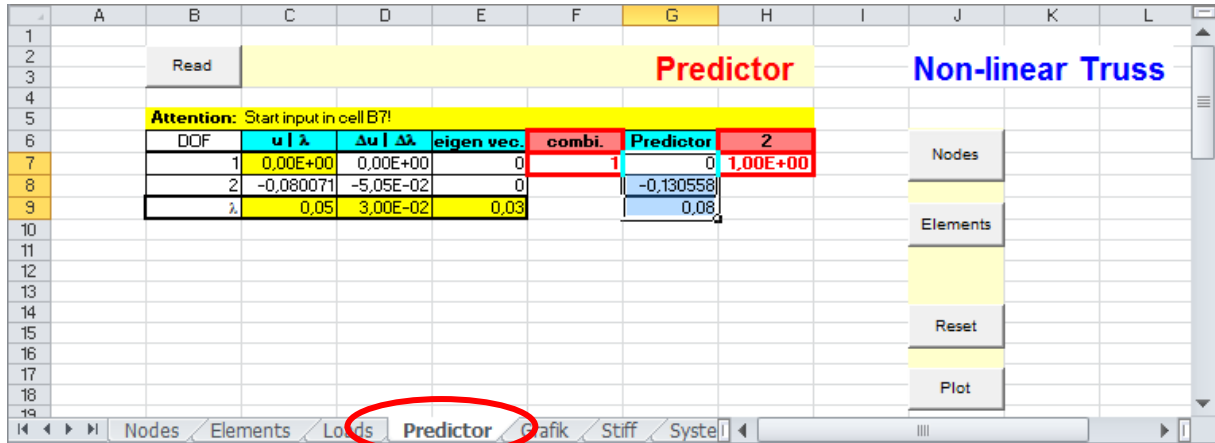
In the System-sheet:

- Check "arc length method" in the **Sol. options**.
- Check also „constraints“ in the column of „Display“.
- Confirm your choices with **OK**.
- Click **Predictor** to get to the Predictor-sheet.



- An equation system appears in matrix notation.

In the column C the current displacements and the current load factor are displayed. Column D shows the distance of the next point of equilibrium in the direction of the displacements and the load vector respectively, which will be reached with the next predictor.



DOF	u Δ	Δu ΔΔ	eigen vec.	combi.	Predictor	
1	0.00E+00	0.00E+00	0	1	0	1.00E+00
2	-0.080071	-5.05E-02	0		-0.130558	
Δ	0.05	3.00E-02	0.03		0.08	

- Change the length of the predictor (column H) e.g. **1.0**
Note that the length of the new predictor is always calculated relatively to the length of the last increment (the length of the last secant).
- The new value has to be read in. **Read**
- Back in the System-sheet, solve the equation system by clicking on **Solve**.
The load-displacement curve will be updated accordingly.
- Repeat the last 3 steps to change the length of the predictor again and calculate more steps on the equilibrium path.

9. Special features

- 1) If the „arc length method“ is active the equilibrium path can be generated automatically:
Click on **Automatic** and enter the number of incremental steps which should be done.

Note that the length of the predictor will be constant for these incremental steps.

- 2) You can go back in your calculation to any previous step using the **Reset** -button:
 - for the „load control method“ in the Loads-sheet
 - for the "displacement control method" in the Nodes-sheet
 - and for the "arc length method" in the Predictor-sheet sheet

- 3) When the "arc length method" is selected you can find critical points ($\det K = 0$) by clicking on **Bracketing** and entering a giving amount of steps, if you are next to such critical point.

Types of critical points:

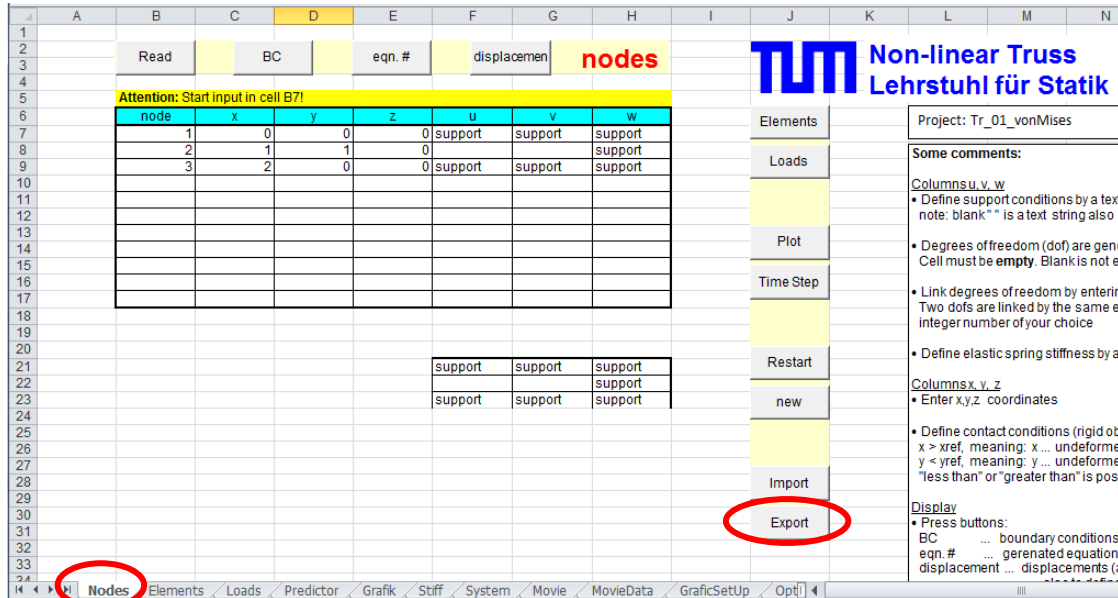
- bifurcation point ($\det K = 0$)
- limit point ($\det K = 0$ and horizontal tangent of the equilibrium paths)
- turning point ($\det K = 0$ und vertikal tangent) → not in this example

- 4) How to get secondary path in this diagram?

- a. Set the “Combi” factor in the Predictor-sheet to **0**.The search direction of the predictor will be changed from the primary path to the secondary path.
- b. The new input again has to be read in by clicking **Read**
- c. Continue solving the problem as described in section 7.3.

10. Export

You can export your calculation example into an *.xlsx* file (EXCEL® file without macros) which contains all your input like nodal coordinates, elements and loads – and also solution information if you already solved your problem. The exported *.xlsx* file is only a data storage. In order to run an exported file, look at chapter 11. Import.



The screenshot shows the Excel interface with the 'Nodes' sheet selected. The 'Export' button in the right-hand sidebar is circled in red. The spreadsheet contains the following data:

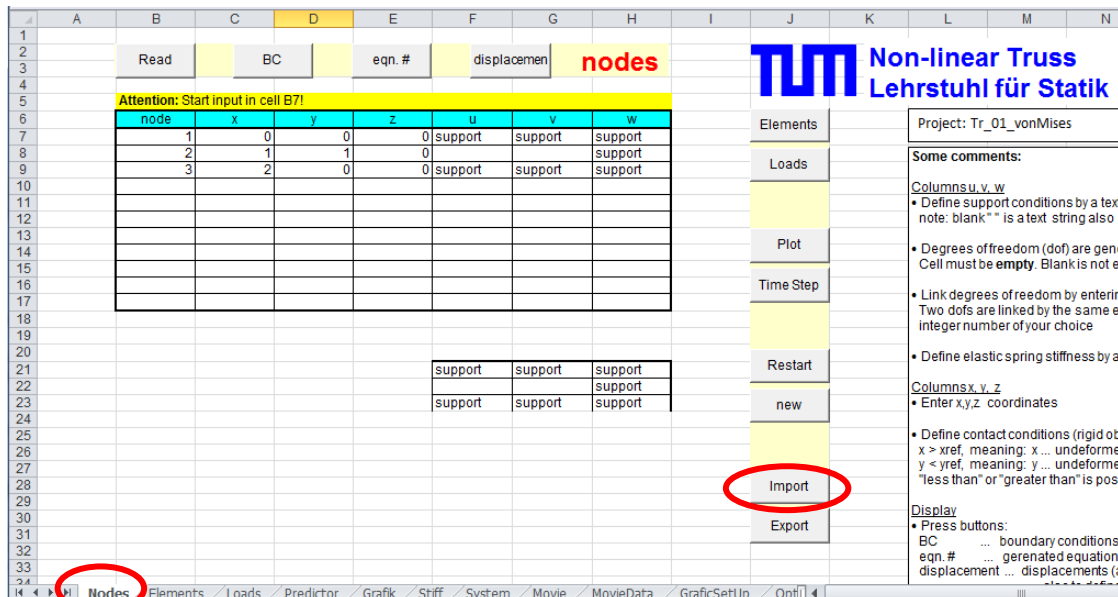
node	x	y	z	u	v	w
1	0	0	0	support	support	support
2	1	1	0	support	support	support
3	2	0	0	support	support	support

Support conditions are listed in columns F, G, and H:

eqn. #	displacement	nodes
	support	support
	support	support
	support	support

11. Import

You can import the data base of an exported example by clicking on Import in the nodes sheet and selecting the respective *.xlsx* file.



The screenshot shows the Excel interface with the 'Nodes' sheet selected. The 'Import' button in the right-hand sidebar is circled in red. The spreadsheet contains the same data as in the previous screenshot.

If you just want to use the imported data base as a base for your new calculation, click on **new** on the nodes sheet and proceed with steps 1-9.

If you want to restart from the results of your exported file, or just want to visualize the results, click **Restart** and *select take results of previous run.*

Then you have to confirm the node, elements and loads by clicking **Read** at the respective sheets. Now you are able to visualize the results of the previous run in the movie sheet, or to continue your non-linear calculation.

12. Remarks

If you encounter problems with the movie sheet, try the **Refresh** button on the movie sheet.

Sometimes also clicking **new** on the nodes sheet can help.

Please let us know if you encounter any errors / bugs, so that we can fix them as soon as possible.