# BE-FRAMES

## REANALYSIS AND INFLUENCE FUNCTIONS

FEBRUARY 2021

PROF. DR.-ING. FRIEDEL HARTMANN
HAGEBUTTENWEG 12
D-34225 BAUNATAL

TEL. +49-561-949-1510
FAX. +49-561-949-1511

hartmann@be-statik.de

www.be-statik.de

## CONTENTS

# 1. INTRODUCTION

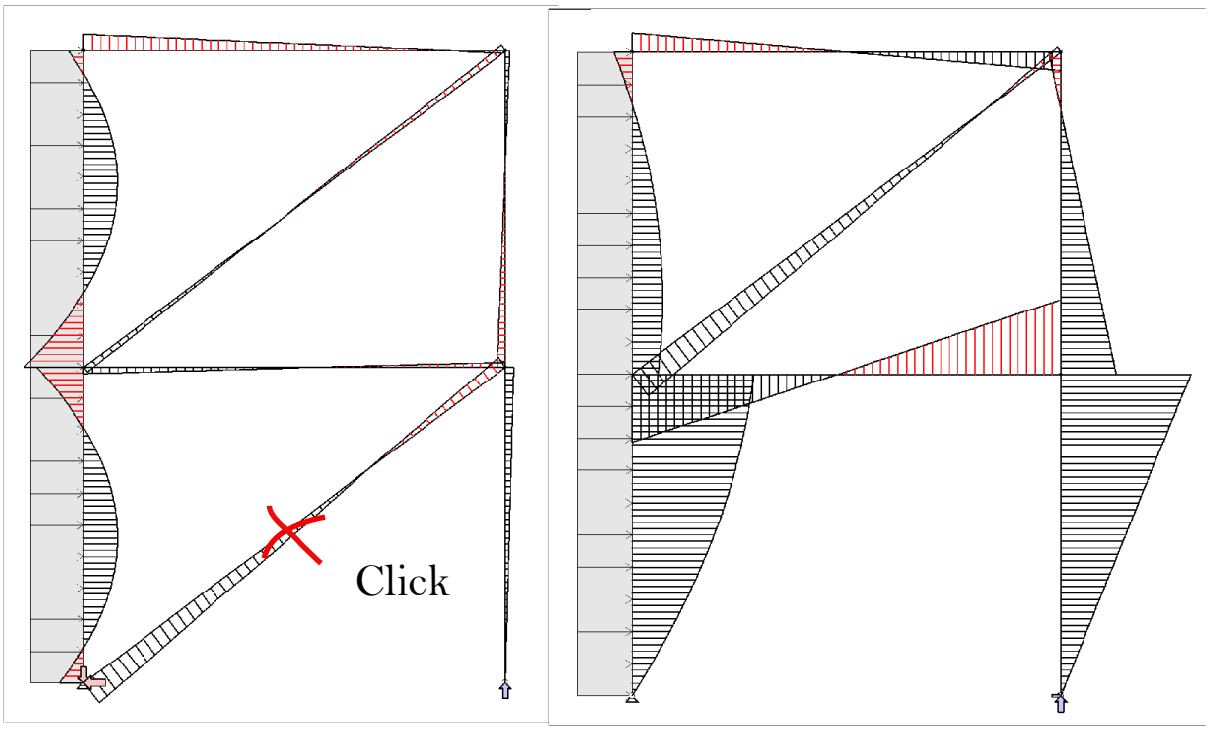
This program is a learning tool for students who want to understand:

- How the design influences the load carrying capacity of a structure.
- How changes in the stiffness of single frame elements influence the distribution of the forces in a frame
- The shapes of the different influence functions for internal forces, displacements, and support reactions in a frame.

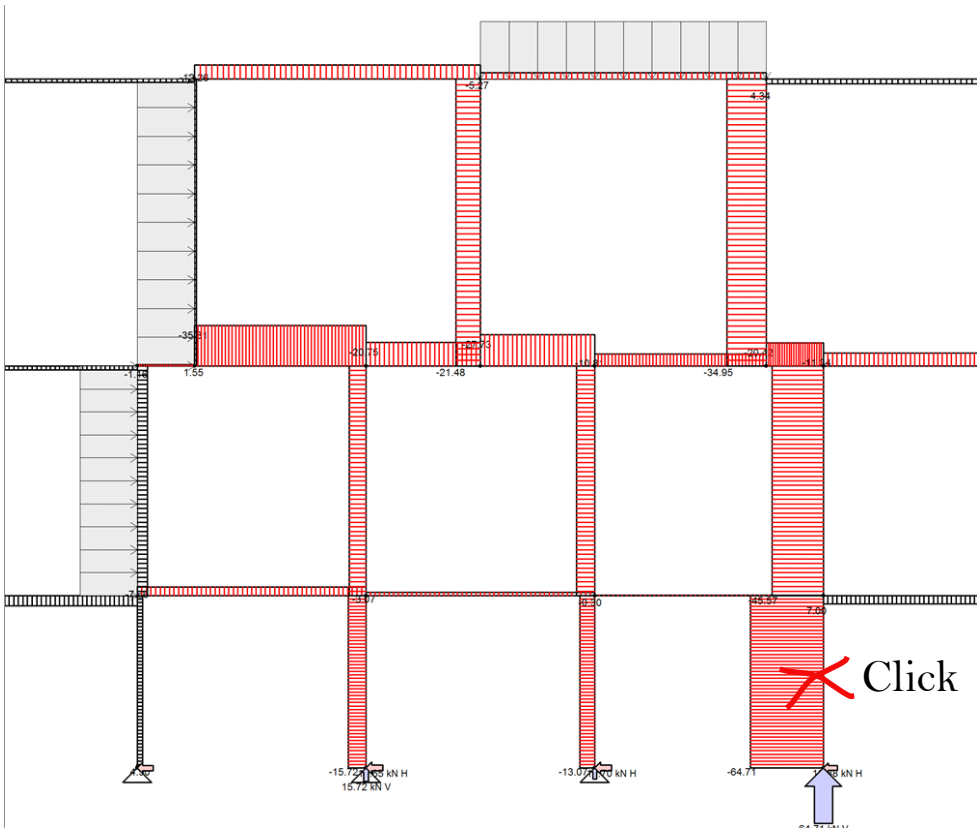
## 1.1. EXAMPLES

Two features characterize the program: **One-Click Reanalysis** and **Influence Functions**.

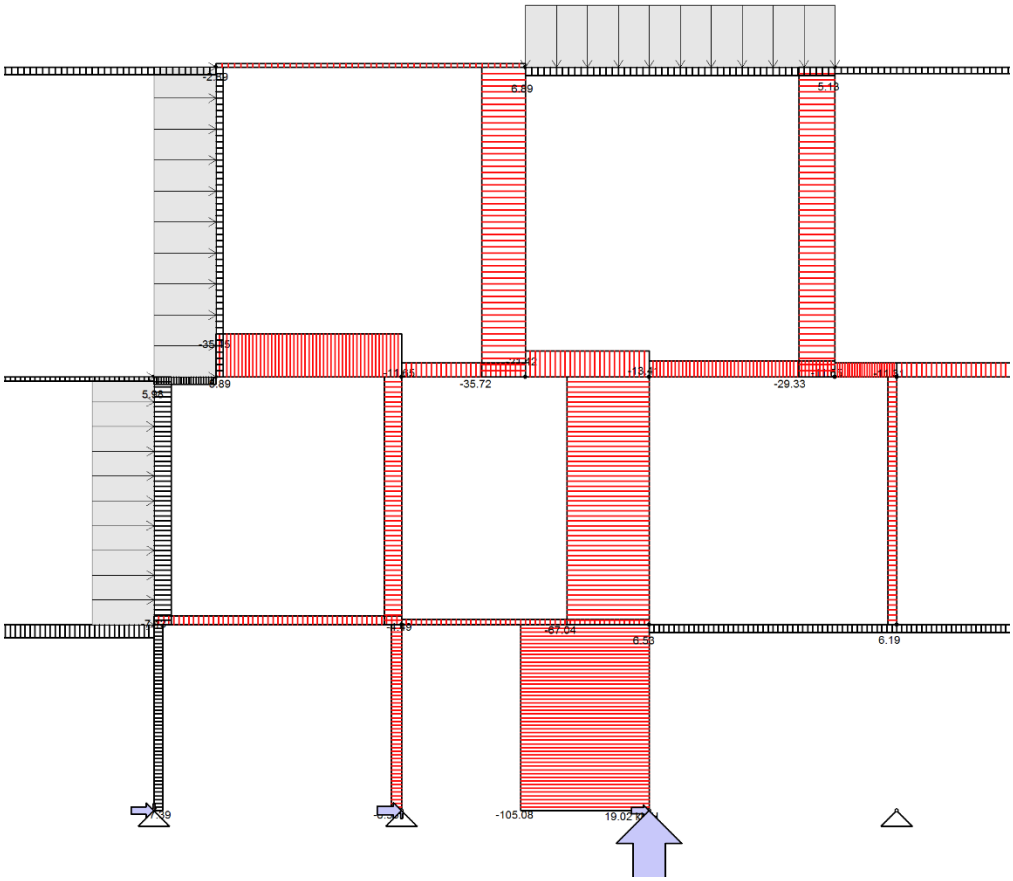
One-click reanalysis means that you can click on any element and the effects of a stiffness modification in that element immediately become visible, as in the following figure where a click on the lower diagonal removes the diagonal and the new bending moment distribution is displayed



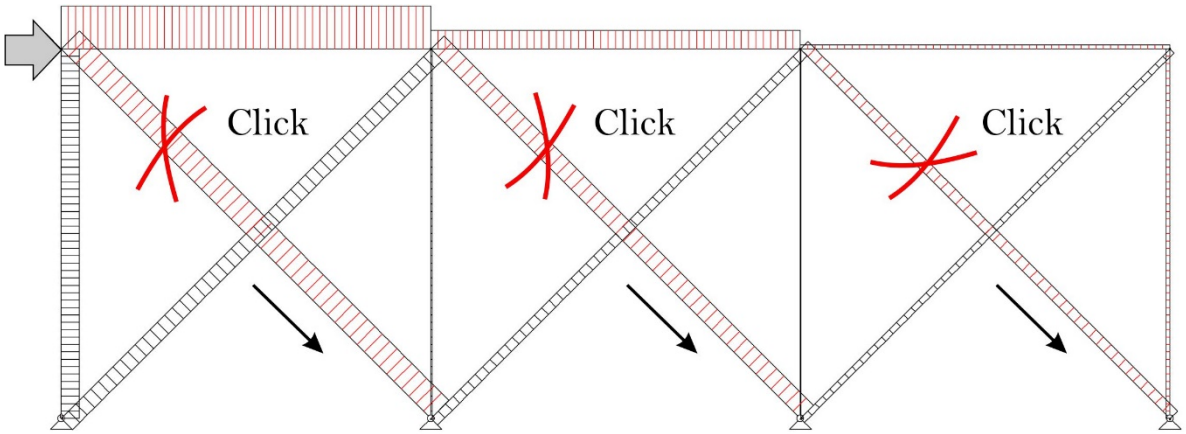
A click on a column removes the column

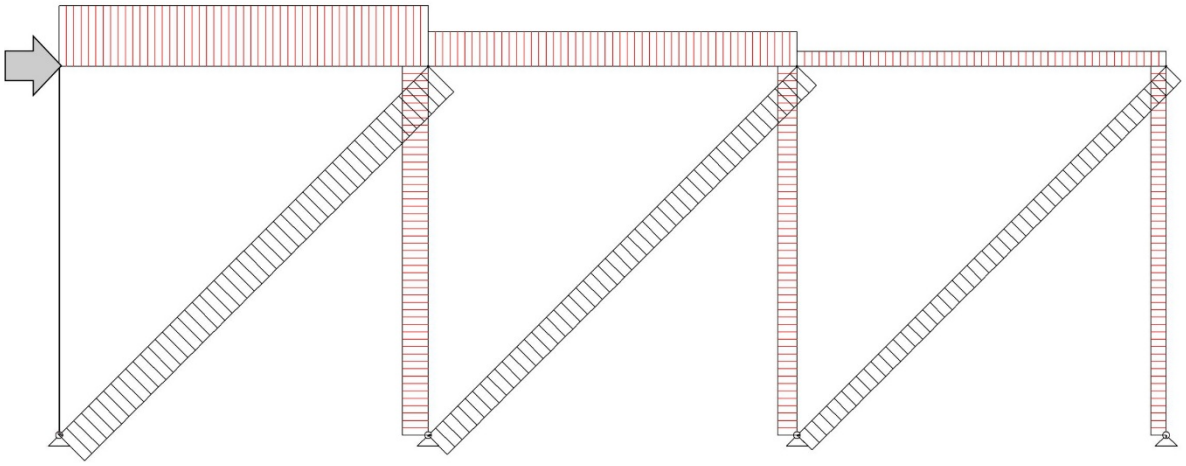



and the effect on the normal forces is seen immediately

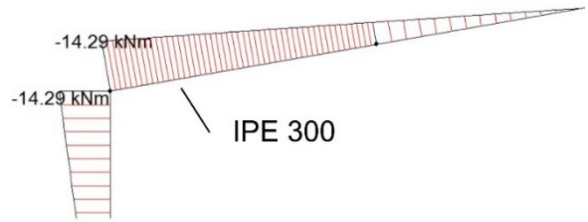


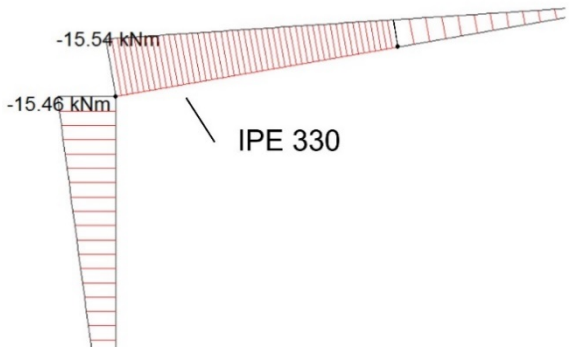
Or with three simple clicks you can eliminate the frame elements which buckle under compression



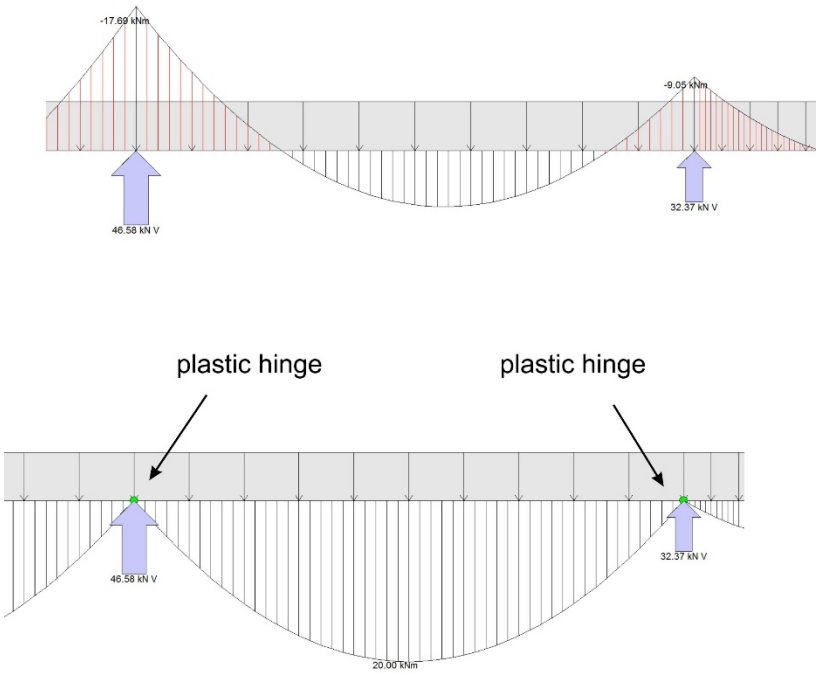



and see how the forces distribute in that situation (this can also be done automatically). Or you change the cross section of a girder to study how this effects the bending moments in the portal frame

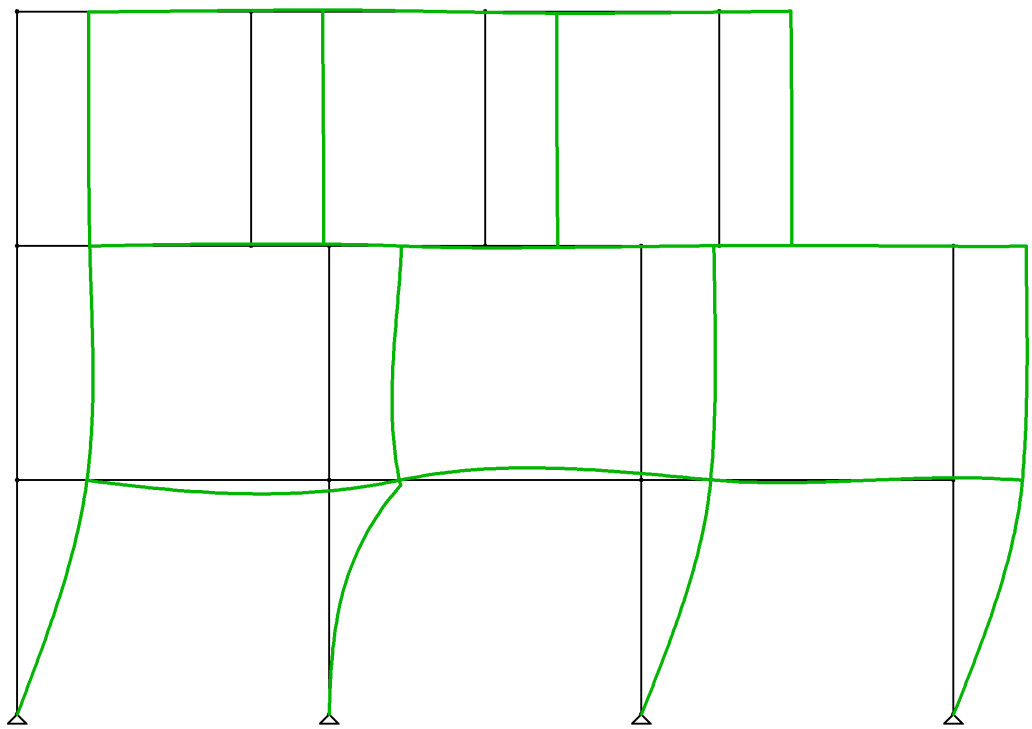


You can even introduce plastic hinges in a frame as in the following drawing




In addition, the program generates the influence functions for all the standard values of a frame



Influence function for a bending moment

## 1.2. HOW REANALYSIS IS DONE

While traditionally any variation in the design requires a new build of the global stiffness matrix

$$\boldsymbol{K}\,\boldsymbol{u} = \boldsymbol{f} \qquad \rightarrow \qquad (\boldsymbol{K} + \boldsymbol{\Delta K})\,\boldsymbol{u}_c = \boldsymbol{f}$$

to determine the new equilibrium position $\boldsymbol{u_c}$ of the frame, the $\boldsymbol{f}^+$ technique

$$\boldsymbol{K}\,\boldsymbol{u}_c = \boldsymbol{f} - \boldsymbol{K}^{-1}\boldsymbol{\Delta}\,\boldsymbol{u}_c = \boldsymbol{f} + \boldsymbol{f}^+ \qquad (\boldsymbol{f}^+ = -\boldsymbol{K}^{-1}\boldsymbol{\Delta K}\,\boldsymbol{u}_c)$$

provides the same answer $\boldsymbol{u_c}$ but keeps the original stiffness matrix intact. The new position vector $\boldsymbol{u_c}$ is either found by iteration

$$\boldsymbol{u}_c^{(i+1)} = -\boldsymbol{K}^{(-1)}\,\boldsymbol{\Delta K}\,\boldsymbol{u}_c^{(i)} + \boldsymbol{u} \qquad i = 1, 2, \ldots \;\; \boldsymbol{u}_c^{(0)} = \boldsymbol{u}$$

or by direct solution of a small 6 x 6 system, see Section 13.

## 1.3. PROS AND CON

Pros: No need for a new build of the global stiffness matrix and no need to update the data structure. The consequences of the modifications are immediately seen on the screen.
Cons: The inverse stiffness matrix $\boldsymbol{K}^{(-1)}$ of the frame must be calculated and stored. This has at least the one advantage that with $\boldsymbol{K}^{(-1)}$ you also have the influence functions of the nodes immediately at hand.

## 1.4. CAVEAT

This program is not intended as a professional tool. Its main purpose is to provide students with an instrument to probe different design concepts with one-click reanalysis. We cannot guarantee that the results are always correct. The $\boldsymbol{f}^+$ approach itself is an approximate method. Not every decimal place will be correct.

# 2. INSTALLATION

## 2.1. INSTALLATION

1. Unpack the file BE-FRAMES.ZIP and click on the file setup.exe

2. The program will be installed in the folder

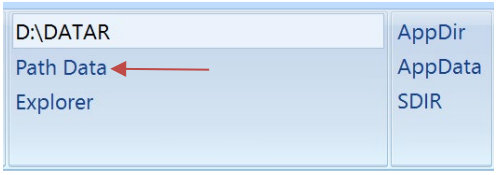
   C:\Programs (x86)\BE-STATIK\BE-FRAMES

3. The program is written in C++ and built with Visual Studio 2010 and the MFC extensions. In case the standard dll libraries of Visual Studio 2010 are not installed on the computer (which is rare) you must install the dll's manually. Start the Windows-Explorer and open the program folder (see above) and do a double click on the file vcredist_x86.exe. This will install the missing dll's.
   Simple test: if the program starts the dll's are installed on the computer.

4. The program uses a Data Path. The Data Path is the name of a folder on your computer which you specify. It serves as the 'root folder' for the subdirectories SDIR… which contain the datafiles of the single positions. After the installation, the Data Path points to the directory

   C:\Users\...\AppData\Roaming\BE-FRAMES

   which is an inconveniently long path for editing chores. In addition, Windows does not allow to store data files in this directory. So, specify a new legitimate Data Path. For this click on the button Path Data




   and specify a more convenient folder as your root folder. Either you pick an existing folder, or you enter the name of a new folder which the program will generate for you. End the input of a new folder with the Enter-key. In the example above the Data Path is D:\DATAR.
   Be careful not to choose a folder to which the program has no access, where it cannot store files as e.g. installation folders (prohibited by Windows)

## 2.2. POSITIONS

Each frame is assigned a position number

**123 200 3400 7777**

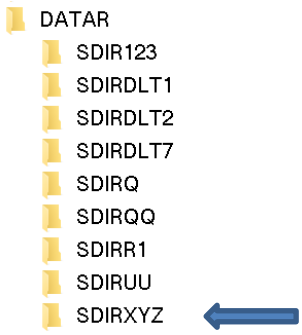
which can also be a string

**TESTABC**

if the string does not contain special characters as in **231.2** or **A+1**.
The data of a position such as XYZ are stored in a folder

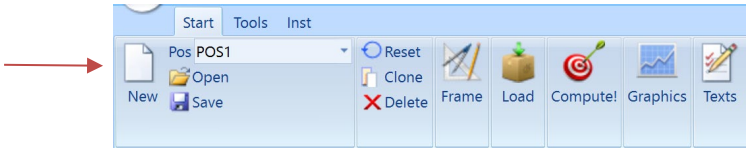
**SDIRXYZ**

which branches off from the Path Data (here D:\DATAR).

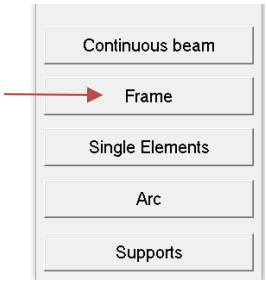



## 2.3. FIRST FRAME

Click on the button New and choose a name, say POS1, for the first frame.




Click on the button Frame in the main menu and next on the button Frame in the program that opens

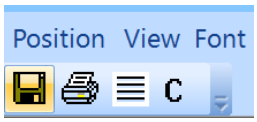



Enter the following numbers to generate a three-story frame

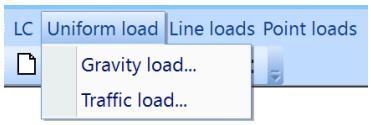
Frame

| Floor | Floor height [m] | Number of columns | Spacing [m] |
|---|---|---|---|
| 1 | 3 | 4 | 4 |
| 2 | 3 | 4 | 4 |
| 3 | 3 | 4 | 4 |
| 4 | | | |

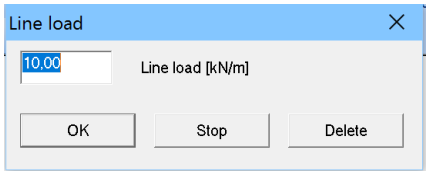
Click on the OK-button (in this dialog) and click on the disk-symbol in the tool bar to save the input




and return to the main menu of the program. Back in the main menu click on the Load button and next on Uniform load -> Traffic load




Accept the default value




and save. By a click on save (disk icon) the position is immediately analyzed and the results are ready for your inspection (no need to click the Compute! Button).

Next, start the Graphics program by clicking on the button Graphics in the main menu and in the Graphics program press the N, S or M-key to display the normal forces, the shear forces, or the bending moments of position POS1. Use the mouse wheel to scale the figures and click the right mouse button to iterate through the scales

- Default: Displacements, normal forces, etc. are scaled by a turn of the mouse wheel
- 1st click: The load is scaled
- 2nd click: The support reactions are scaled
- 3rd click: Loop repeats…

## REANALYSIS

**C-key = remove or alter an element**

Next try reanalysis. Have the bending moments or the normal forces of the frame displayed on the screen. *Press the C-key*. The cursor becomes a small square. *Click on any element*. The element is removed, and you see at once the effects. (Actually, the effects depend on the parameter k chosen. For this do a right click with the mouse). To repeat, *press again the C-key* and choose another element. To navigate forward or backward in the chain of modifications, use the *arrow-keys*, left and right, or click on the space bar. The *space bar* toggles directly between the original system and the most modified version of the system.

Press the ESC-key to reset everything to the original state. Next try the F-key.

**F-key = plastic hinge M_plastic = k * M where k = chosen parameter (right click)**

The cursor turns again into a small square. *Click on any element*. Depending on where you click the element, a plastic hinge is installed at three possible locations: left, center, right.

**G-key = shear-hinge V_plastic = k * V where k = chosen parameter (right click)**

When you do the same with the G-key a shear-hinge is installed.

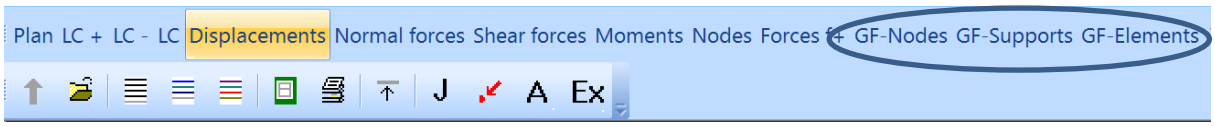
**H-key = V = M = 0**

When you hit the H-key and you click on a frame element, a shear hinge (V = 0) and a plastic hinge (M = 0) are installed.

In the C-mode (C-key is hit) you can also remove supports. To do this click slightly below the support. In the dialog you can specify the support stiffness in different directions. To remove the support simply click on the button

Click here to set all values k to zero = no support

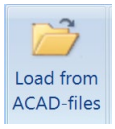
When you are done with the modifications click on the ESC-key to set everything back to the original state.



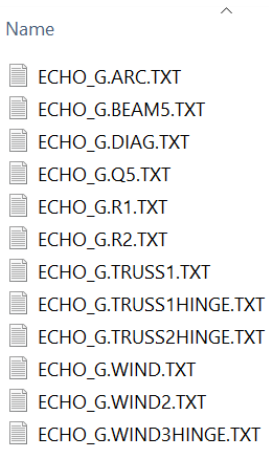

Other buttons activate the display of the Green's functions of the nodal displacements, the support reactions and of the displacements and internal actions at the mid-points of the elements. In this mode, the Up- and Down-keys let you switch between the single degrees of freedom. You can also press the right mouse button to advance to the next degree of freedom.

## 2.4. ADDITIONAL EXAMPLES

The program comes with a list of ready-to-install positions. To open this list, click on the button Tools and next on the button




If all is well, you see a folder EXAMPLES which contains files called ECHO_G.*.TXT



By double-clicking on any one of these files the corresponding position is automatically loaded, and you can e start the analysis by clicking on the button Calculate!

In case the folder EXAMPLES is not visible, do the following: Navigate to the folder that contains the program

C:\Program Files (x86)\BE-STATIK\BE-FRAMES

and look for the folder EXAMPLES, open the folder and pick an ECHO_G.*.TXT file. The program remembers the path from then on.

## 2.5. STORE AND RELOAD

By clicking on the button Save


your input is stored on the hard drive in the form of two text-files

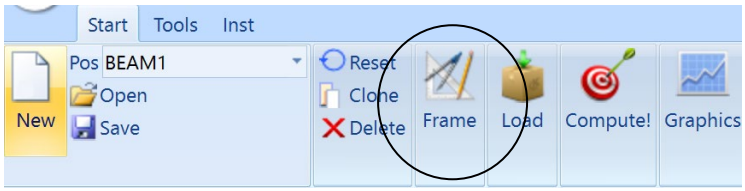
ECHO_G.POS.TXT (geometry)
ECHO_L.POS.TXT (load)

where POS is the name of the actual position.

To reload the position, open the Explorer, click on the file ECHO_G.POS.TXT and drag it onto the open program.

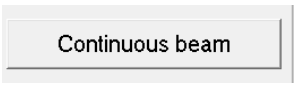
# 3. CONTINUOUS BEAM

Start in the main menu, and enter a new position, choose for example the name BEAM1,




and click on the button Frame and then on the buttons Material and Cross Sections to specify these values

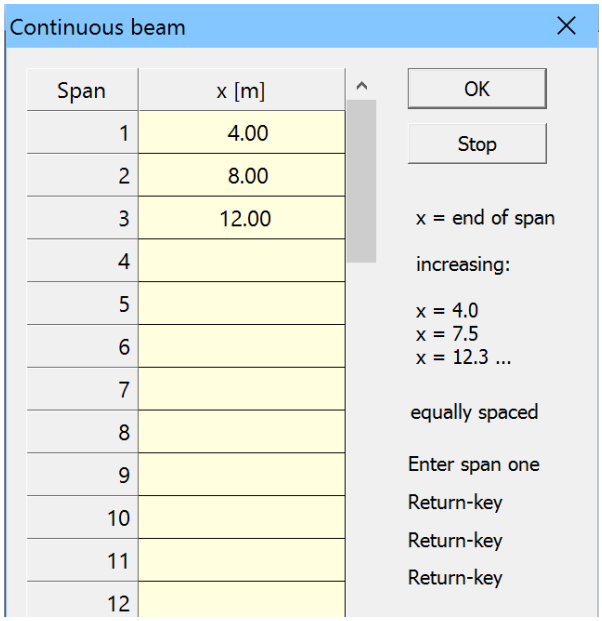

Next click on the button Continuous Beam

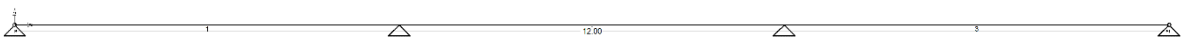



A continuous beam with equally spaced supports is best entered in the following way:

Enter the x-coordinate of the endpoint of the first span and then press the Enter-key 3 times to generate two more spans, each with the same length.

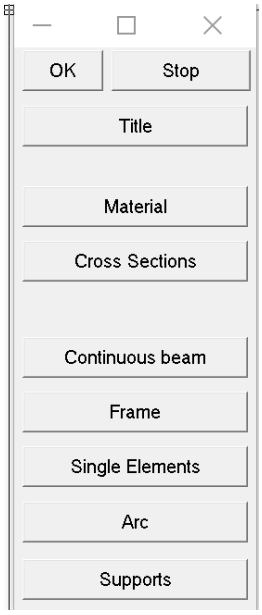
The supports are automatically generated.



# 4. FRAME

Click on the button Frame

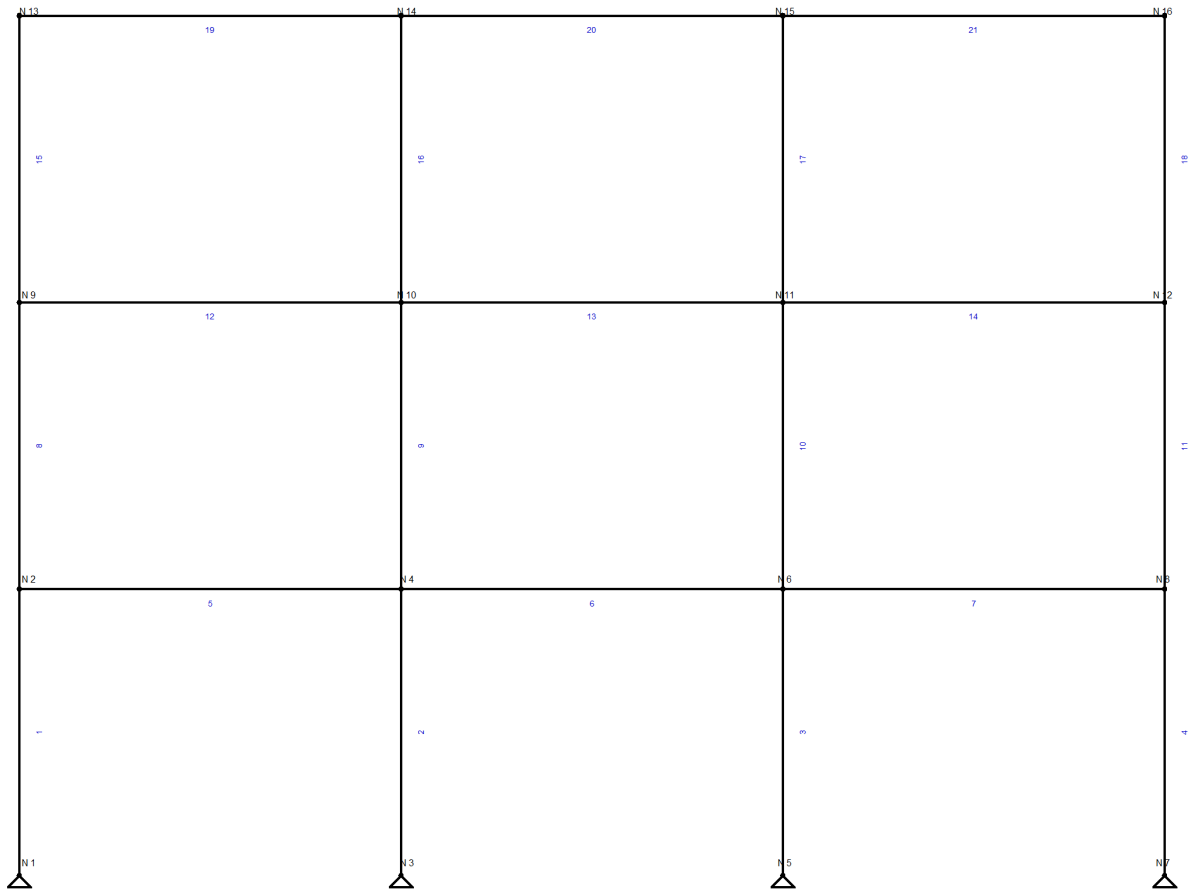



and specify the floor height, the number of columns and the spacing in between

Frame

| Floor | **Floor height [m]** | Number of columns | Spacing [m] |
|---|---|---|---|
| 1 | 3 | 4 | 4 |
| 2 | 3 | 4 | 4 |
| 3 | 3 | 4 | 4 |
| **4** | | | |
| 5 | | | |

If the second floor is a copy of the first, then simply hit the Enter-key to copy the values from the row above.




To edit the supports, click on the button Supports

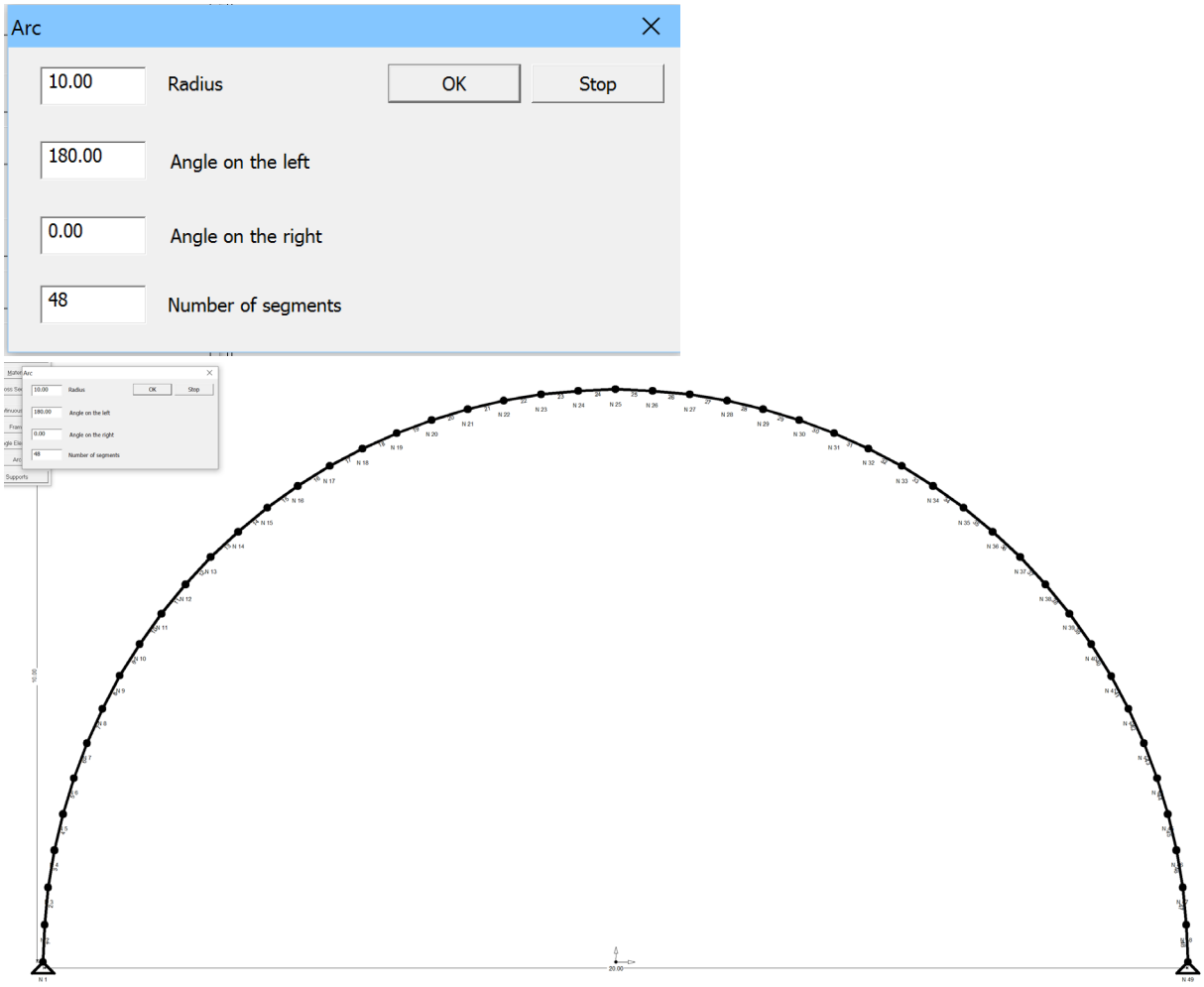
Nodes

| Nodes | x | y | u | v | phi |
|---|---|---|---|---|---|
| 1 | 0.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 2 | 0.00 | 3.00 | 0.0000 | 0.0000 | 0.0000 |
| 3 | 4.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 4 | 4.00 | 3.00 | 0.0000 | 0.0000 | 0.0000 |
| 5 | 8.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 6 | 8.00 | 3.00 | 0.0000 | 0.0000 | 0.0000 |
| 7 | 12.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 8 | 12.00 | 3.00 | 0.0000 | 0.0000 | 0.0000 |
| 9 | 0.00 | 6.00 | 0.0000 | 0.0000 | 0.0000 |
| 10 | 4.00 | 6.00 | 0.0000 | 0.0000 | 0.0000 |
| 11 | 8.00 | 6.00 | 0.0000 | 0.0000 | 0.0000 |
| 12 | 12.00 | 6.00 | 0.0000 | 0.0000 | 0.0000 |
| 13 | 0.00 | 9.00 | 0.0000 | 0.0000 | 0.0000 |
| 14 | 4.00 | 9.00 | 0.0000 | 0.0000 | 0.0000 |
| 15 | 8.00 | 9.00 | 0.0000 | 0.0000 | 0.0000 |
| 16 | 12.00 | 9.00 | 0.0000 | 0.0000 | 0.0000 |
| 17 | | | | | |

The entries in the columns u, v and phi, the values 0 to 1, represent the stiffness of the supports.
0 = free, 1 = fixed, and any value in between is allowed. The actual support stiffness of a node are these weights times the base stiffness, which (in numbers) is set to max E * max A (= max cross-section).

# 5. ARC

Arcs are approximated by a sequence of straight frame elements.




# 6. SINGLE INPUT

You can specify the elements of a frame as well single handedly, element for element.

| xa [m] | ya [m] | xb [m] | yb [m] | Mat. | Sect. | C | T | H left | H right |
|---|---|---|---|---|---|---|---|---|---|
| 0.00 | 0.00 | 0.00 | 3.00 | 1 | 1 | 1 | 1 | 0 | 0 |
| 4.00 | 0.00 | 4.00 | 3.00 | 1 | 1 | 1 | 1 | 0 | 0 |
| 0.00 | 3.00 | 4.00 | 3.00 | 1 | 1 | 1 | 1 | 0 | 0 |
| 0.00 | 0.00 | 4.00 | 3.00 | 1 | 1 | 0 | 1 | 1 | 1 |
| 4.00 | 0.00 | 0.00 | 3.00 | 1 | 1 | 0 | 1 | 1 | 1 |
| | | | | | | | | | |

OK

Hinges: H (left) H (right)

0 = No
1 = Yes

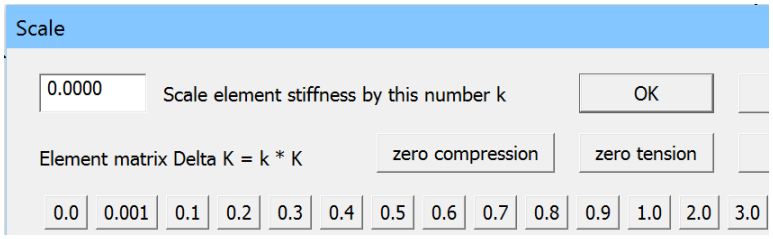
C = Compression, 0 = No, 1 = Yes

T = Tension, 0 = No, 1 = Yes

A zero in column C says that the element will buckle under compression and a zero in column T says that tensile forces are not carried by the element.

In the ground state these attributes C and T are ignored, that is all elements carry compressive as well as tensile forces.

To disable all members of type C = 0, start the program Graphics, click on the menu bar entry Parameter, and click on the button zero compression



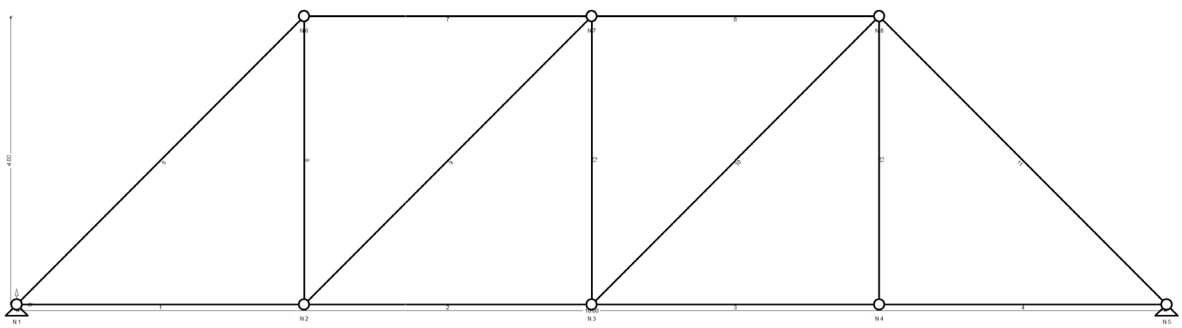

The program will then try to find iteratively an equilibrium position of the frame where all frame elements of type C = 0 are consecutively deactivated, are removed, if N is less than zero.

# 7. TRUSS

In trusses each frame element has a hinge (H) at both ends, H left = 1, H right = 1

| Element | xa [m] | ya [m] | xb [m] | yb [m] | Mat. | Sect. | C | T | H left | H right |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | 0.00 | 0.00 | 4.00 | 0.00 | 1 | 1 | 1 | 1 | 1 | 1 |
| 2 | 4.00 | 0.00 | 8.00 | 0.00 | 1 | 1 | 1 | 1 | 1 | 1 |
| 3 | 8.00 | 0.00 | 12.00 | 0.00 | 1 | 1 | 1 | 1 | 1 | 1 |
| 4 | 12.00 | 0.00 | 16.00 | 0.00 | 1 | 1 | 1 | 1 | 1 | 1 |

as in the following figure



The hinges are displayed in this way when you click on the menu entry Elevation in the Graphics program. Internally, of course, each such cluster of hinges concentrated a node is one single hinge.

# 8. SUPPORT STIFFNESS

The program calculates with a base stiffness

base stiffness = (max E) x (max A)  A = max cross section

All fixed supports are modeled as very stiff springs with k equal to the base stiffness.

This approach ends the need to distinguish between a reduced and a non-reduced global stiffness matrix and makes it easy to model stiffness changes in supports.

The base stiffness times the entries in the following table is the actual support stiffness of the single fixed nodes.

Nodes

| Nodes | x | y | u | v | phi |
|---|---|---|---|---|---|
| 1 | 0.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 2 | 0.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 3 | 4.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 4 | 4.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 5 | 8.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 6 | 8.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 7 | 12.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 8 | 12.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 9 | 16.00 | 0.00 | 1.0000 | 1.0000 | 0.0000 |
| 10 | 16.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 11 | 3.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 12 | 6.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 13 | 9.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 14 | 15.00 | 4.00 | 0.0000 | 0.0000 | 0.0000 |
| 15 | 0.00 | 8.00 | 0.0000 | 0.0000 | 0.0000 |
| 16 | 3.00 | 8.00 | 0.0000 | 0.0000 | 0.0000 |
| 17 | 6.00 | 8.00 | 0.0000 | 0.0000 | 0.0000 |
| 18 | 9.00 | 8.00 | 0.0000 | 0.0000 | 0.0000 |
| 19 | 12.00 | 8.00 | 0.0000 | 0.0000 | 0.0000 |
| 20 | 15.00 | 8.00 | 0.0000 | 0.0000 | 0.0000 |

OK  Stop

0 = free, 1 = fixed

or any value in between

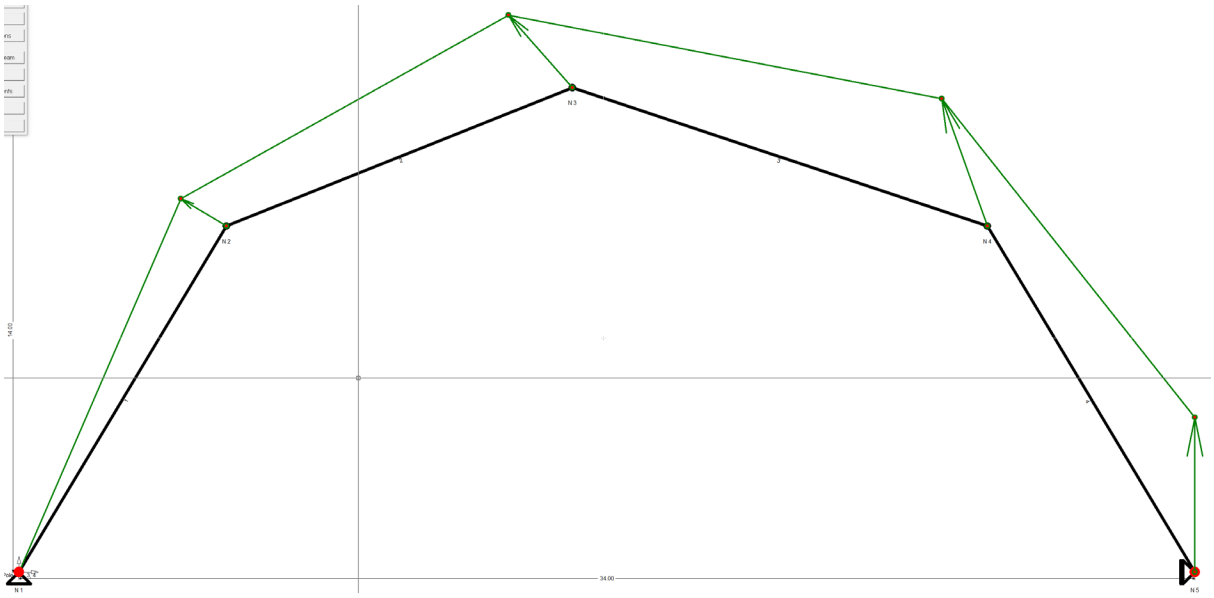
Table entry * base stiffness

is support stiffness

Base stiffness

2.8489E+006  kN/m (kNm)

An entry of 1.0000 means the node is fixed and an entry 0.0000 means that the node is unrestrained. Any other values in between are also possible.

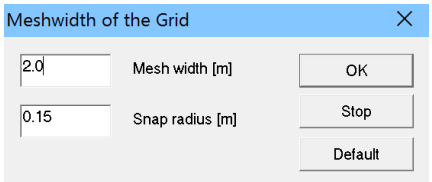


If the system is kinematic because a support is missing or because it contains one hinge too many the program flags this as an error and displays the possible rigid-body movement as in the figure above where the missing vertical support at the right end allows the frame to rotate about the left support.

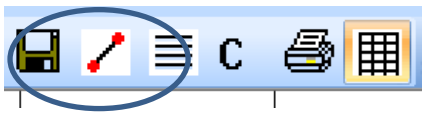
# 9. GRAPHICAL INPUT

You can also draw a frame with the mouse. First click on the Grid-button

and choose a suitable mesh width




To activate the drawing mode, click on the line-icon



and then click on the nodes of the grid to outline the shape of the frame. (You must not use the grid).
To pause, press the spacebar and to resume press the spacebar again. To stop the input mode, click on ESC.

To finalize the drawing click on the button Single Elements and if everything is correct specify the supports by clicking on the button Supports.

Note that at the end of the drawing mode you **must** click on the Single Elements dialog. This way the program has a chance to understand the layout of the frame.

# 10. FINITE ELEMENTS 'PURE'

A click on the icon


displays the equivalent nodal forces which belong to the actual load case. The reaction of the structure to these nodal forces is the FE-solution 'pure'. The nodal values of this solution are the entries of the vector $\boldsymbol{u}$ which solves the system $\boldsymbol{K}\,\boldsymbol{u} = \boldsymbol{f}$.

The complete solution is the FE-solution 'pure' plus the local solutions in the single frame elements

- Complete solution = FE-solution 'pure' + local solutions (not displayed when key is pressed)

The local solutions are the element solutions when the frame elements are clamped at both ends.

To display the complete solution simply turn the FE-key off.

# 11. WHEN ARE RESULTS EXACT?

Point values are only exact if

- the Green's function of that point value lies in $V_h$ or
- the error in the Green's function is orthogonal to the load.

The second condition is met with if the exact solution lies in $V_h$.

Consider a simple hinged beam with a uniform load p and subdivided into four elements. The FE-solution is exact also in between the nodes - but why?

- The exact solution is a $4^{th}$ order polynomial which does not lie in $V_h$ because all shape functions are $3^{rd}$ order polynomials.

- The Green's functions for point values, say w(x), in between nodes do not lie in $V_h$.

So theoretically point values in between the nodes should not be exact. The trick is that the program adds to the pure FE-solution (only equivalent nodal forces) the local solutions in the elements.

The point values of the pure FE-solution are not exact, but the values of the complete solution are

- Complete solution = FE-solution 'pure' + local solution = exact solution

Note: In all elements which are load-free the local solution is zero (no need for a clamped element to deflect when there is no load) and in these elements the pure FE-solution is the complete, the exact solution.

# 12. EXAMPLE

An illustrative example of the difference between the pure FE-solution and the complete solution supplies the next picture. In Figure a the exact influence function for the deflection at the mid-point of the first element is displayed

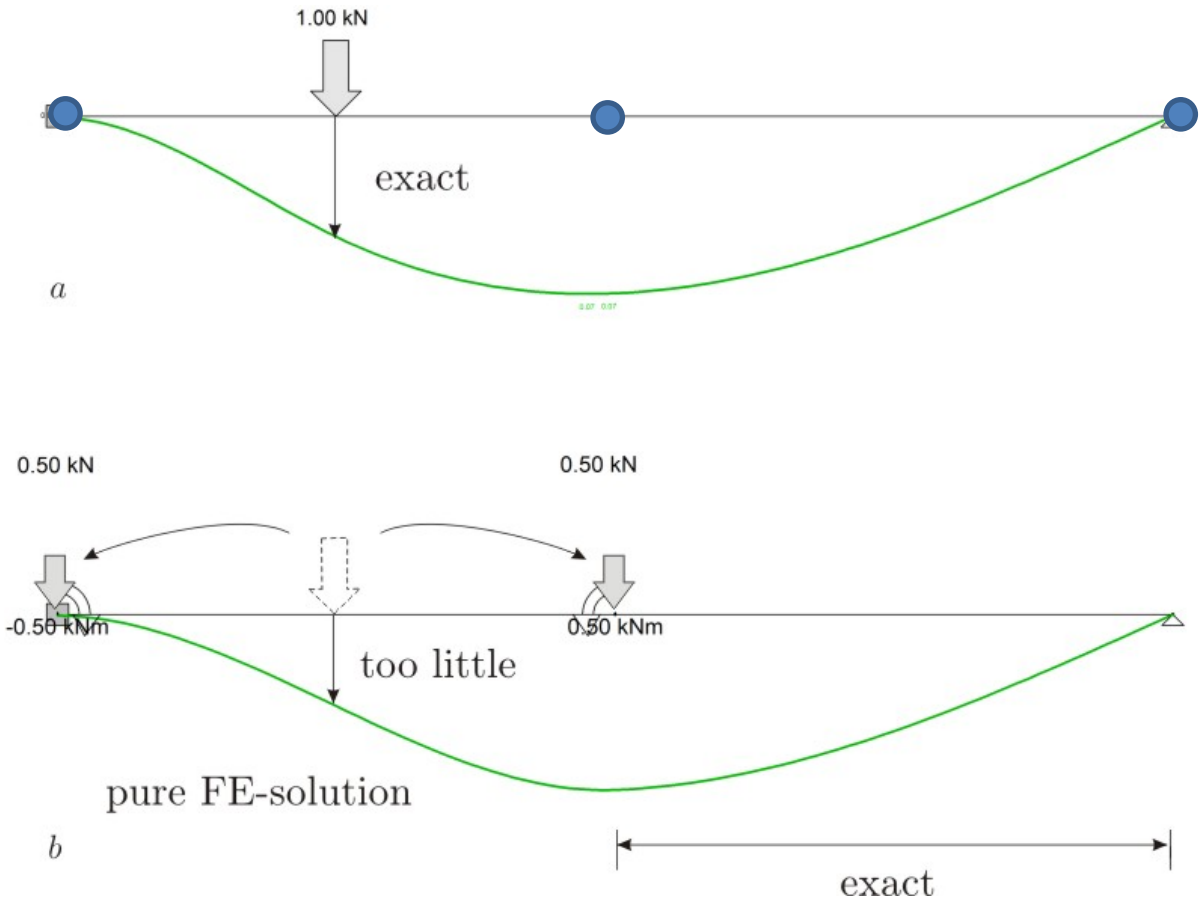
The FE-program cannot solve this load case and so it replaces the point load by equivalent nodal forces, see Figure b. The result is an influence function which – in the first element - is only an approximation.

Outside the element the FE-solution is exact because the exact influence function G is outside of the element a homogeneous solution, $EI\, G^{IV} = 0$, a cubic, and cubics lie in $V_h$.

To turn this 'defect' on you must click on the FE-button. Otherwise, the program displays the complete solution which is of course exact because the program adds the local solution to the pure FE-solution.

The local solution is the response of a clamped one-element beam loaded with a unit point load at its center.

In frame analysis, it is possible to add the local solutions to the pure FE-solution but in 2D and 3D problems this is not possible because the local solutions are unknown and because the shape functions are not homogeneous solutions of the governing equation. In 2D and 3D problems the FE-solution is the pure FE-solution.



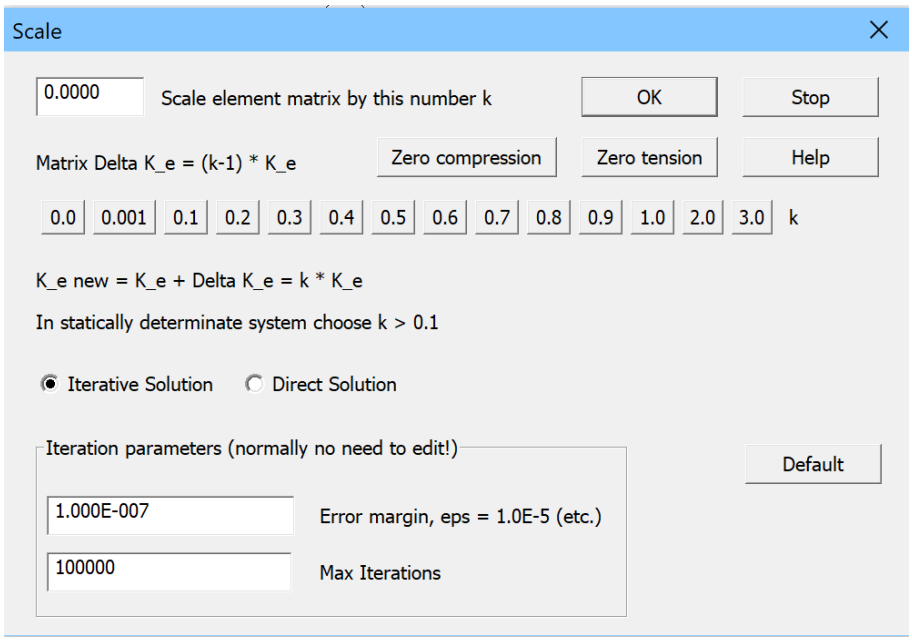

# 13. REANALYSIS

Reanalysis is done in one click.

## 13.1. THE C-KEY

It is best to have the normal forces or the bending moments on display when you start the reanalysis. You begin by specifying the scale, by how much the stiffness of a clicked element should change. For dramatic effects, it is best to set the scale k to zero (if the frame is not statically determinate).

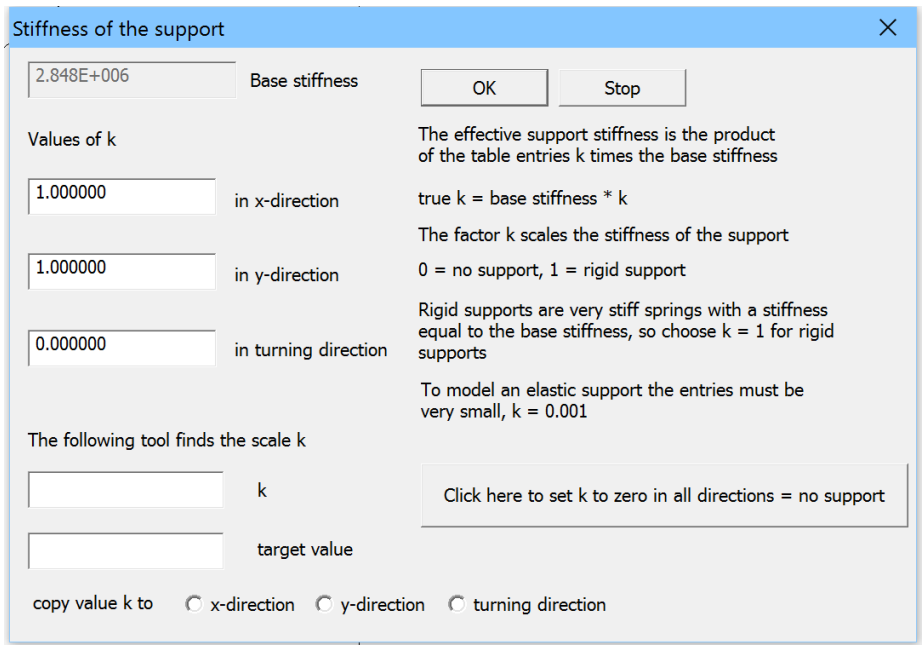
In the menu bar click on Parameter or simply press the right mouse button. This opens the following dialog



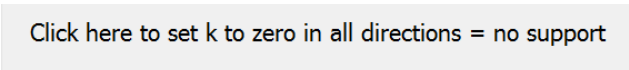

where you can specify the scale k. Set k equal to zero. Close the dialog and then hit the C-key (you are then in 'C-mode') and next click on the frame element you want to remove. To repeat, hit again the C-key and click on the next element.

With the cursor keys, ← and →, you can switch between the different modifications. The space bar toggles between the original frame and the most modified version of the frame.

To remove a support, click on the support (a little below center) and in the dialog




click on



Otherwise, you can choose any value between 0 and 1 to model a support stiffness.

The program finds the new equilibrium position of the frame either

- by an iterative analysis. The entry in the fields Error margin and Max Iterations (see above) serve to steer the iteration, or
- by direct solution

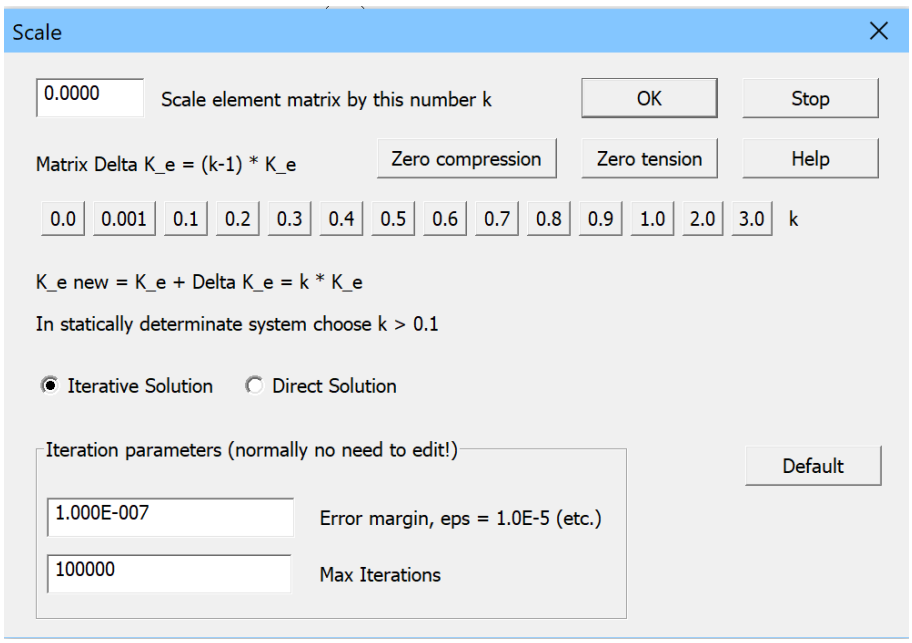
In the direct solution approach, the program solves an auxiliary 6 x 6 system and the solution of this small system suffices to determine the new equilibrium position of the frame, regardless of how many degrees of freedom the frame has. Each additional element which gets modified makes the size of the auxiliary system increase from 6 to 12 to 18 … columns and rows.

## 13.2. THE F-KEY

When you press the F-key the program is ready to place a plastic hinge (M = 0) in any frame element at three possible spots, left, center or right, depending on where you click on the element.

To repeat, press the F-key anew and click on the next element. In this mode, it is best to have the bending moments on display or the displacements to see the kinks at the plastic hinges.

The 'degree' of the plastic hinge can be scaled by choosing a suitable value of k in the following dialog




An entry k = 0 corresponds to a true plastic hinge (M = 0) while an entry k = 0.8 means a reduction of the bending moment by 20 %, that is $M_c = M * k$. What you see on the screen are the bending moments in the frame given that the load carrying capacity of the plastic hinge is limited to M * k. In this situation, the factor k only affects the capacity of the plastic hinge. Element matrices are not changed.

### 13.3. THE G-KEY

The G-key works the same way it is only that a shear hinge (V = 0) is installed in the frame element.

### 13.4. THE H-KEY

The H-key finally is a combination of an M-hinge and a V-hinge, that is the frame element is (not exactly) split into two parts. Not exactly because normal forces can pass through. Only one such combo-hinge can be placed on the frame at any time, and it is always placed in the middle of the frame element (no mathematical reason for this). The scale k is automatically set to zero, M = V = 0.

## 14. INFLUENCE FUNCTIONS (OR GREEN'S FUNCTIONS)

The program can display the Green's functions for

- The nodal displacements;
- The support reactions
- The displacements and internal actions at the mid-points of the elements or anywhere else

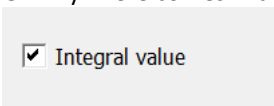

- The Green's function of a node is the displacement of the frame if a force P = 1 or a moment M = 1 pushes the node in the direction of one of the three degrees of freedom $u_1$, $u_2$ or $u_3$.

- At supports the Green's function is the influence function for the chosen support reaction.

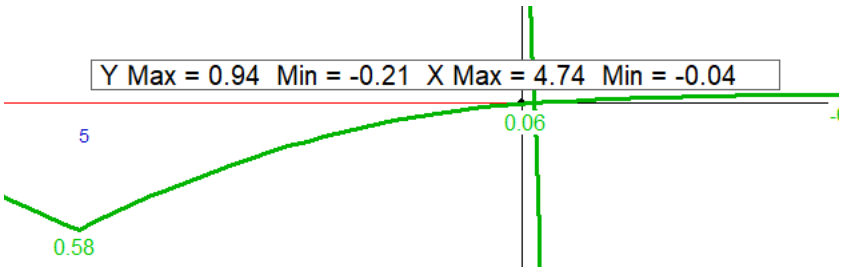
- The element Green's functions are the influence functions for u, w, w', N, V, M at the mid-point of the element.

With the Up- and Down-keys you can advance to the next Green's function. The same effect has a press on the right mouse button.

GF-Anywhere comes with the option to display the integral of the influence function




that is if all elements would carry a horizontal or vertical unit load p = 1 [kN/m]




Y Max = max value under vert. load, X Max = max value under horiz. load, Min are the min values under Y and X load.

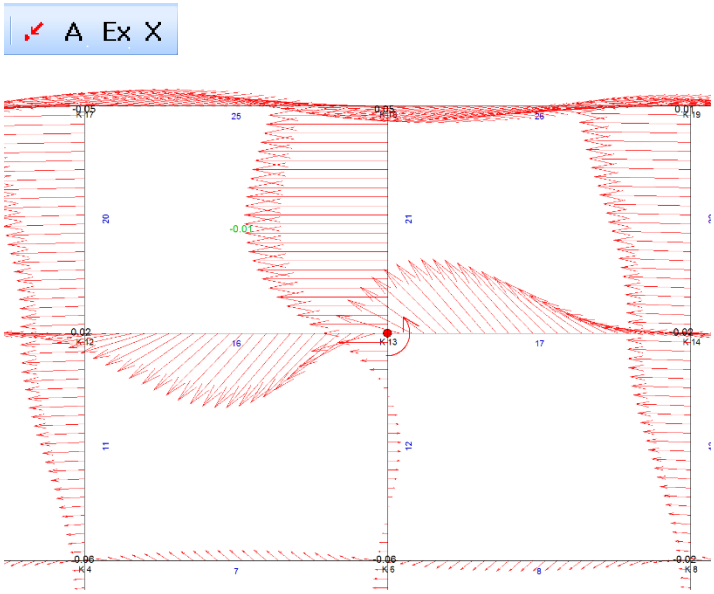
When an element Green's function is displayed, here of a normal force N at the center of the element, a text as

GF N [kN], H = 0.691 V = 0.795 R = 0.004 (H is result if all horizontal f_i = 1, V = result if all vertical f_i = 1, etc.)

is displayed. It is H = 0.691 the normal force N in the element when all nodes carry a unit nodal force $f_j = 1$ pointing in horizontal direction, V = 0.795 is the normal force N when all nodes carry unit nodal forces $f_j = 1$ pointing in vertical direction and R = 0.004 is the value of N if the nodes carry moments $f_j = 1$.

## 14.1. ARROWS

A click on the button A adds arrows to the display of the Green's functions to better visualize the character of the function




A single point load has maximum effect if it points in these directions. The length of the arrow is proportional to the influence the point load has, in the picture above on the rotation of the center node.

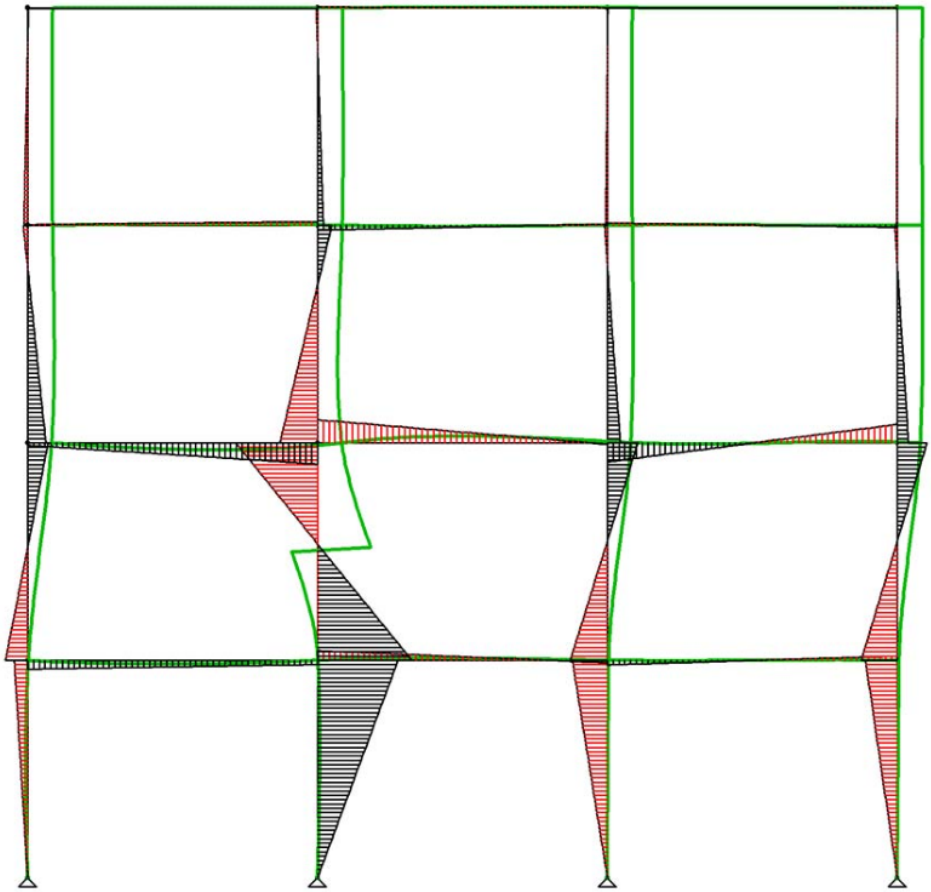
# 15. SENSITIVITY

How sensitive is a point value, M(x), V(x), etc. to a change of the stiffness in the neighboring elements? According to the formula, [1] p. 321,

$$M_c - M = -d(w_c, G) = -\frac{\Delta EI}{EI}\int_0^{l_e} \frac{M_c\, M_G}{EI_c}\, dy$$

the difference between the new value, $M_c(x)$, and the old value M(x), is the scalar product of the new bending moment ($M_c$) in the beam element and the bending moment $M_G$ of the influence function for M(x).
This would also be true if we study a change in N(x) or V(x). $M_G$ is always the bending moment of the influence function for the terms on the left. By plotting $M_G$ we see which elements will affect the most the value on the left. To see the weighting function $M_G$ for a given value, say, V(x), do the following:

1. Click on the button GF-Anywhere
2. Click on the point x to display the influence function for V(x)
3. Click on the button Moments to display the bending moment $M_G$ of the influence function



In this figure: Influence function for the shear force V(x) in a frame and the bending moment $M_G$ of the influence function. It indicates which elements have the most influence on V(x) in the sense that a change of EI in these elements has the most effect.

For the above formula to provide the value $M_c$ – M the bending moment distribution $M_c$ would have to be known. For a first estimate it will suffice to substitute for $M_c$ the previous bending moment distribution M.

**Correction**: Because the above equation is based on the Green's function G of the bending moment M = - EI w'', it is

$$-\underset{\uparrow}{EI}\, w_c''(x) - M(x) = -d(w_c, G)$$

or

$$M_c(x) = (M(x) - d(w_c, G)) \cdot \frac{EI_c}{EI}$$

so that

$$M_c(x) \cdot \frac{EI}{EI_c} - M(x) = -d(w_c, G)$$

is a weighted difference of the two moments. In the case of displacements terms, say $w_c$ – w = -d($w_c$,G) the result is correct.

# 16. EIGENVECTORS AND EIGENVALUES

With MATLAB you can calculate the eigenvectors and eigenvalues of the frame.

This is how it is done: Copy the file **frameeigen.m** from the program directory of BE-FRAMES to your MATLAB folder (where you current MATLAB projects reside).

Open MATLAB and open the file

frameeigen.m

Edit the first two lines, that is specify the so called DataPath and the name of the position.

Run the file

Start the program and in graphics click on the menu entry Eigen. With the Up- and Down-keys you can switch between the eigenvectors.

(The file frameeigenEx.mlx in the program directory is a lengthier version with more comments interspersed.)

**The file frameeigen.m**

```
DataPath = 'd:\dat';    % <---------- specify the PathData = name of the folder from
which the folder SDIR… branches off
NamePosition = 'cbeam';    % <---------- specify the name of the position

path = strcat(DataPath,'\sdir',NamePosition,'\matrix.',NamePosition);

% With the above input the path points to the file
% d:\dat\sdircbeam\matrix.cbeam
%
% If the name of the position is abc and the folder sdirabc lies in the
% folder
% E:\FrameData then DataPath = 'E:\FrameData' and NamePosition = abc'
%
% path = 'E:\FrameData\sdirabc\matrix.abc'

fid = fopen(path);
VectorM = fread(fid,'double');  % column vector
fclose(fid);

[row,col] = size(VectorM);    % col = 1

n = 0.5 + sqrt(0.25 + row) - 1; n = cast(n,'int32');  % n = number of columns

K = zeros(n+1,n);    % the stiffness matrix with an extra row at the top

for i = 1:n
  K(:,i) = VectorM(1 + (i-1) * (n+1) : 1 + (i-1) * (n+1) + n);
end

K(1,:) = [];  % this matrix has now size n x n and is the original stiffness matrix

[V,P] = eig(K);  % all eigenvalues must be positive

path1 = strcat(DataPath,'\sdir',NamePosition,'\eigv.',NamePosition);
fid = fopen(path1,'wb');  % eigenvalues
fwrite(fid,diag(P),'double');
fclose(fid);

path2 = strcat(DataPath,'\sdir',NamePosition,'\ev.',NamePosition);
fid = fopen(path2,'wb');    % eigenvectors
```

```
for i = 1:n
  fwrite(fid,V(:,i),'double');
end
fclose(fid);
```

# 17. KEYS AND MOUSE

## 17.1. DISPLAY RESULTS

- Press N, M, S, D-key to display Normal forces, Moments etc.
- Strg + click on an element displays single values
- To display the values in one single element, press the P-key and then click on the element
- X-key, only horizontal displacements
- Y-key, only vertical displacements
- <-key scales the plots of M, N and V so that they fit on the page
- The mouse wheel acts on
  1. N, M, S, D
  2. The load
  3. The support reaction, in this order
- Click right mouse button to iterate through this list

## 17.2. TOOLBAR


- A click on one of the first three icons loads text files (dimensions, load cases, results)
- Print preview (green icon)
- Print
- Reset displaced texts (after you have moved the texts with the mouse)
- J = jpeg
- Red arrow resets scales
- A is for arrows on the influence functions
- Ex = exponentials, scientific notation
- X = reminder to reset the frame after the C-mode or any other mode
- FE = 'Pure FE-Results', before the local solutions are not added

## 17.3. WHEEL

- Wheel = scale drawing,
- Shift + Wheel = scale size of the frame,
- Shift + Ctrl + Wheel = font size
- Alt + Shift + Wheel = line width
- A + Wheel = scale support reactions
- Ctrl + Wheel = scale loads

To zoom, first press the Z-key, and then draw a rectangle with the mouse, ESC-key = zoom off

## 17.4. MULTI-CLICK AND MOUSE WHEEL

At the start the mouse wheel acts as a scale on the displacements, the normal forces, etc.
1st click: The load is scaled
2nd click: The support reactions are scaled
3rd click: Loop repeats…

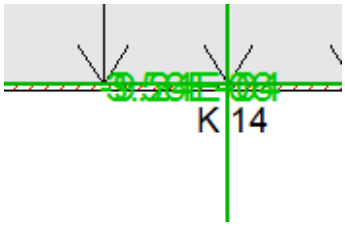
## 17.5. MOUSE CLICKS

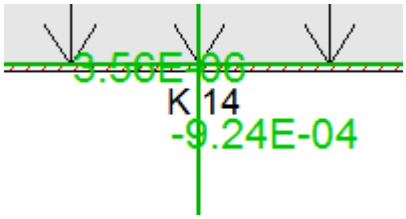
When influence functions are displayed: A Right-click advances to the next influence function. The Up- and Down-keys have the same effect.

To scale the support reactions, click near any support and rotate the mouse wheel. Same as A + Wheel. Or use the multi-click technique.

To scale the load, click on any element and turn the mouse wheel. Same as Ctrl + Wheel. Or use multi-click.

Most numbers on the screen can be displaced. Click on the number and pull it sideways. This helps to untangle messy heaps of numbers.




## 17.6. KEYS

- D = Displacements
- E = Element numbers on/off
- F = F-mode, plastic hinge (M = 0)
- G = G-mode, shear hinge (V = 0)
- H = H-mode, element is cut in half, M = V = 0
- K = Node numbers on/off
- N = Normal forces
- M = Moments,
- P = Activates ‘click-mode’ to display N, M, S, D of single elements by clicking on the elements
- S = Shear forces
- R = rotate horizontal displacements, display in vertical direction
- Strg-key + click on an element displays single value of N, M, S, D at that point
- W = energy ½ $\boldsymbol{u}^T \boldsymbol{K}\, \boldsymbol{u}$ * 10 000 of single elements, button toggles
- X = only results in horizontal direction, on/off
- Y = only results in vertical direction, on/off
- Z = Numbers, on/off
- Space key = Switch between original structure and modified structure

Cursor-keys (Up and Down) = Change load case
Cursor-keys (Left and Right) = Move forward or backward in a series of modifications

## 17.7. HORIZONTAL DISPLACEMENT

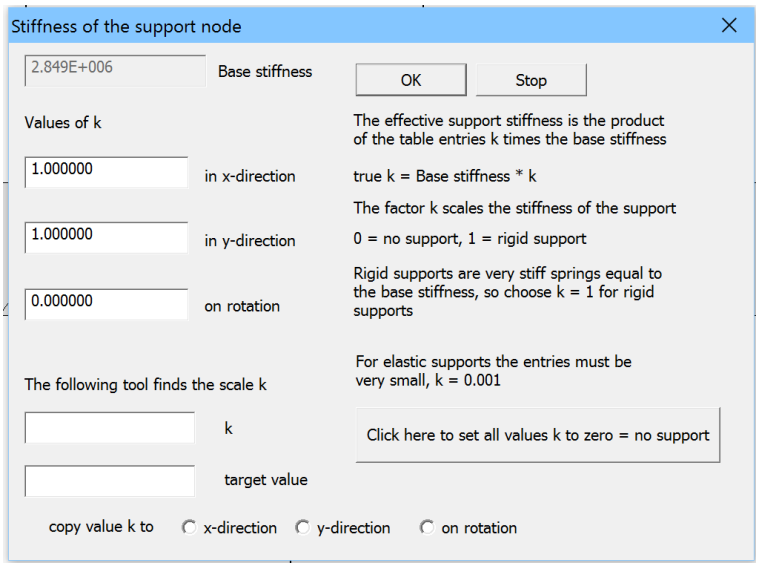
To display the horizontal displacement of a horizontal frame element, proceed as follows:
Press the X-key, press the P-key and click on the frame element.
Alternative method: Press the R-key, if it is a continuous beam

## 17.8. MODIFYING ELEMENTS (C-MODE)

- Press the C-key and then click on the element which is to be removed or modified
- Repeat!
- Traverse the sequence of modifications with the Cursor-keys (Left and Right)
- Space key = Switch between original structure and the modified structure (deepest level)
- ESC-key leaves the C-Mode

## 17.9. CHANGING SUPPORTS (C-MODE)

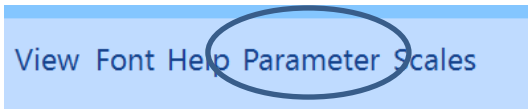
- Press the C-key to activate the C-Mode and click on the support (a little below the support)
- Edit the stiffness of the support by choosing a suitable value for k



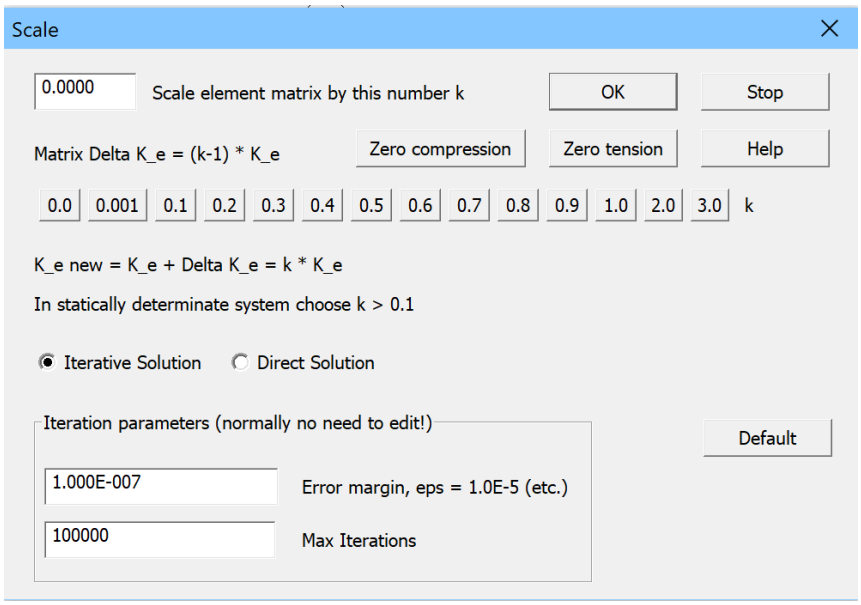

- ESC-key leaves the C-Mode

## 17.10. FINETUNING OF THE C-MODE

A click on the entry Parameter in the menu bar, or simply a click of the right mouse button,




opens the following dialog




where you can specify the element stiffness in relation to the standard stiffness of the element.

A change in the stiffness of an element means that the element stiffness matrix is scaled with the factor k. It is not possible to scale EA or EI separately.

## 17.11. ZERO COMPRESSION OR ZERO TENSION

Frame members which cannot carry any compression or tension have the attribute

C = 0 or T = 0

In the ground state the program ignores these specifics and analyzes the frame as if all elements can carry compressive as well as tensile forces.

When you click in the above dialog on the button

Zero compression

the program will try to find iteratively (or directly, button direct solution) an equilibrium position of the frame where all frame elements of type C = 0 are deactivated if N is less than zero. If k = 0 the rule is enforced strictly, otherwise it can be attenuated by choosing a small positive factor k > 0. The technique is based on the forces **$f^+$.**

## 17.12. FORCES $F^+$

In the C-mode you can display the forces **$f^+$.** These are the forces which when they act on the original frame produce the same effects as the modifications in the C-mode. To scale the forces, turn the mouse wheel. To scale the moments, also press the CTRL-key.

# 18. HOW IT IS DONE

When an element is removed, or altered, the program finds the new equilibrium position $\mathbf{u}_c$ of the frame by iteration

$$\boldsymbol{u}_c^{(i+1)} = -\boldsymbol{K}^{(-1)}\boldsymbol{\Delta K}\,\boldsymbol{u}_c^{(i)} + \boldsymbol{u} \qquad i = 1, 2, \ldots \qquad \boldsymbol{u}_c^0 = \boldsymbol{u}$$

where $\Delta\boldsymbol{K}$ is the modified element stiffness matrix and $\boldsymbol{K}^{(-1)}$ is the inverse of the global stiffness matrix, see [1] p. 282. The matrix $\Delta\boldsymbol{K}$ has as many columns and rows as $\boldsymbol{K}$ but it contains only the entries of the modified 6 x 6 element matrix $\Delta\boldsymbol{K}_e$ of the element that is removed or altered.

Alternatively, the program calculates the new vector $\mathbf{u}_c$ directly, by solving the system

$$(\boldsymbol{I} + \boldsymbol{K}^{-1}\boldsymbol{\Delta K})\,\boldsymbol{u}_c = \boldsymbol{u}$$

Because $\Delta\boldsymbol{K}$ is sparse the program must only solve a (6 x 6)-system when one single element is modified or removed. No need to solve the full system. For two elements, it would be a (12 x 12) system, etc.

When you would apply the forces

$$\boldsymbol{f}^+ = \boldsymbol{\Delta K}\,\boldsymbol{u}_c$$

to the original structure then the equilibrium position would exactly come out as the vector $\mathbf{u}_c$.

$$\boldsymbol{K}\boldsymbol{u}_c = \boldsymbol{f} + \boldsymbol{f}^+$$

The nodal influence functions are identical with the columns of the inverse stiffness matrix, or to be precise, the nodal values of these influence functions are the columns of $\boldsymbol{K}^{(-1)}$.

To calculate the element influence functions the program applies the corresponding Dirac deltas at the mid-points of the elements, that is a point force P = 1 in x- and y-direction, a moment M = 1, etc.
A plastic hinge is generated by applying a Dirac delta $\delta_2$ (= influence function for M), that is an artificial kink and by then sizing the Dirac delta in such a way that the sum of the bending moment of the load and of the Dirac delta is zero.

The same technique is applied to generate a shear hinge with a Dirac delta $\delta_3$ (= influence function for V) and to cut a frame element into two halves, two Dirac deltas $\delta_2$ and $\delta_3$ are applied simultaneously.

## 18.1. LIMITS

Theoretically, the iteration does not converge if the modified frame is kinematic, that is if its stiffness matrix

$$\boldsymbol{K}_c = \boldsymbol{K} + \boldsymbol{\Delta K}$$

is singular**.** This can happen when a structure is statically determinate and an element is removed, in which case $\Delta\boldsymbol{K} = -\boldsymbol{K}_e$ where $\boldsymbol{K}_e$ is the element stiffness matrix, so that $\boldsymbol{K}_c = \boldsymbol{K} - \boldsymbol{K}_e$. To see this, we argue as follows: Because the new $\boldsymbol{K}_c$ is singular there exists a non-trivial vector $\boldsymbol{u}_0$ so that

$$(\boldsymbol{K} + \boldsymbol{\Delta K})\,\boldsymbol{u}_0 = \boldsymbol{0}$$

Multiplying this equation from the left with the inverse $\boldsymbol{K}^{-1}$ gives

$$\boldsymbol{K}^{-1}(\boldsymbol{K}+\boldsymbol{\Delta K})\,\boldsymbol{u}_0 = (\boldsymbol{I}-\boldsymbol{K}^{-1}\boldsymbol{\Delta K})\,\boldsymbol{u}_0 = \boldsymbol{0}$$

which means that $\boldsymbol{u_0}$ is an eigenvector of $\boldsymbol{K}^{-1}\,\Delta\boldsymbol{K}$ with eigenvalue 1 and the consequence is that the error $\boldsymbol{e}_{i+1} = \boldsymbol{u}_{i+1} - \boldsymbol{u}_i$ cannot decrease

$$\boldsymbol{e}_{i+1} = -\boldsymbol{K}^{-1}\boldsymbol{\Delta K}\boldsymbol{e}_i \qquad i = 1, 2, \ldots$$

because such a fixed-point iteration only succeeds if the absolute values of the eigenvalues of the iteration matrix are less than one, [2] p. 564.

Note

$$\boldsymbol{e}_{i+1} = \boldsymbol{u}_{i+1} - \boldsymbol{u}_i = -\boldsymbol{K}^{-1}\boldsymbol{\Delta K}\boldsymbol{u}_i + \boldsymbol{u} + \boldsymbol{K}^{-1}\boldsymbol{\Delta K}\,\boldsymbol{u}_{i-1} - \boldsymbol{u} = -\boldsymbol{K}^{-1}\boldsymbol{\Delta K}(\boldsymbol{u}_i - \boldsymbol{u}_{i-1}) = -\boldsymbol{K}^{-1}\boldsymbol{\Delta K}\boldsymbol{e}_i$$

A curious point is that under mild circumstances the iteration will converge even if the removal of an element(s) has made the structure unstable because the program iterates and does not try to calculate the non-existing inverse of the singular matrix $\boldsymbol{K} + \Delta\boldsymbol{K}$. The results are seemingly the same as if you would substitute for the inverse $(\boldsymbol{K} + \Delta\boldsymbol{K})^{-1}$ the pseudoinverse.

## 18.2. NOT IMPLEMENTED

Reanalysis is not implemented for load cases where supports settle. Technically such load cases pose no additional problems. The reanalysis must focus on the change in the support stiffness of the node. To this end a unit force $f_i = 1$ is applied at the support and a reanalysis is done. The inverse of the support displacement caused by the unit force is the new support stiffness and with this value the necessary force can be calculated to displace the support.
Repeating this at each node one could theoretically calculate the inverse $\boldsymbol{K_c}^{-1}$ of the modified structure step by step.

# 19. MATH

The program solves $\boldsymbol{K}\,\boldsymbol{u} = \boldsymbol{f}$ two times.

First it reduces the matrix $\boldsymbol{K}$ to reduced-row-echelon-form (= unit matrix + possible singular columns) to see whether the frame is stable.

If not then the last columns of rref($\boldsymbol{K}$) (row-reduced echelon form, MATLAB) are columns which lie in the kernel of $\boldsymbol{K}$, in the null-space of $\boldsymbol{K}$, [3] p. 267. These rigid-body-displacements generate no forces. If these exist, then the system is unstable, is kinematic. The program displays these modes on the screen and notifies the user.

If everything is okay the program next calculates the inverse of $\boldsymbol{K}$ by applying Gauss' method.

Files in readable form in the folder SDIR… :

| | |
|---|---|
| MATH_KMATRIX.TXT | The original matrix $\boldsymbol{K}$, in text-form |
| MATH_RREF.TXT | The reduced-row-echelon form (to detect possible rigid-body-movements) |
| MATH_TRIANGULAR.TXT | The upper triangular matrix |
| MATH_UKINEMATIC.TXT | A possible rigid-body-displacement |
| MATH_RHS0X.TXT | The right-hand side of load case X, (read 0 as number null) |
| MATH_SOL0X.TXT | The solution of $\boldsymbol{K}\,\boldsymbol{u} = \boldsymbol{f}$ in load case X |

If there are more than one null-solutions only the last one is stored and displayed.

# 20. LITERATURE

[1] Hartmann F, Jahn P (2021) Statics and Influence Functions – from a Modern Perspective, Springer Verlag, 2nd Ed.

[2] Strang G (2007) Computational Science and Engineering, Wellesley-Cambridge Press

[3] Strang G (2014) Differential Equations and Linear Algebra, Wellesley-Cambridge Press

Carl O (2011) Statische und dynamische Sensitivitätsanalysen von geschädigten Tragwerken mit Greenschen Funktionen, PhD-Thesis, University of Siegen

Franke W, Kunow T (2007) Kleines Einmaleins der Baustatik, Kassel University Press, Kassel

Carl O, Hartman F, Zhang C (2017) Schnelle Berechnung von Änderungen und Varianten bei komplexen Tragsystemen (3D-Modellen) – Neue Ansätze in der Baustatik unter Verwendung von Einflussfunktionen, to appear Stahlbau (March 2017)

Hartmann F, Jahn P (2016) Statik und Einflussfunktionen - vom modernen Standpunkt aus. Kassel University Press, ISBN 978-3-7376-0100-9 (print), ISBN 978-3-7376-0101-6 (e-book)

Hartmann F, Katz C (2007) Structural Analysis with Finite Elements, (2nd Ed.), Springer-Verlag

Hartmann F, Jahn P (2014) Steifigkeitsänderungen bei finiten Elementen, Bauingenieur 89, 209-215

Kunow T (2010) Modellfehler und Greensche Funktionen, PhD-Thesis, University of Kassel