

USER'S GUIDE

---

# **Bending Test**

**Data Processing and Evaluation Software**

---

*Author:*  
Václav NEŽERKA

March 26, 2012

## Part I

# About The Program

The program was developed for fast and user-friendly evaluation of the data obtained from a three-point bending test on a notched specimens and can be used only in the MATLAB environment (version 2008 and newer). If you encounter any problems or bugs when using the program, please don't hesitate and contact the author at [nezerka.v@seznam.cz](mailto:nezerka.v@seznam.cz). The author would like to thank his supervisor, Jan Zeman, PhD., who provided him with fruitful consultations during the software development. This work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS12/027/OHK1/1T/11.

## 1 Program Structure

The program is started using the file `main.m` in the main folder. The program structure is following (see Fig.1):

- All the files needed for the program to run (including graphics) are in the folder "files" and the user needn't open this folder.
- The functions needed for a calculation of the Young's modulus and fracture energy can be found in the folder "functions".
- Folder "input" is of the biggest importance for the user, since all the input data should be stored there. The program starts searching for the input files right in this folder.
- The "output" folder is used for storing the data for postprocessing (see Section 13).

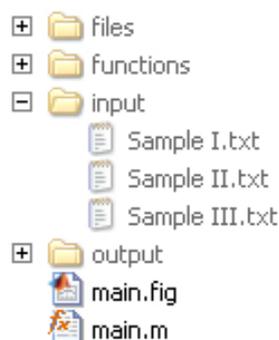


Figure 1: Program data structure

## 2 Input File Format

The input file must be in an ASCII format, it can have a header and the data must be in columns - first column represents loading [N] and third column is supposed to be an array with displace-

ments [m] (see Fig.2). Experienced MATLAB user can easily change this order in the script located at: .../files/input\_processing.m.

```

"Sample I. D.", "Malty-sada1.mss"
"Specimen Number", "10"

"Load (N)", "Time (s)", "Crosshead (mm)", "Stress (MPa)", "_strain1 (mm/mm)"
0.2616, 0.136, 0.0000, 0.021, 0.00000
0.1931, 0.160, 0.0000, 0.016, 0.00000
0.0000, 0.168, 0.0000, 0.000, 0.00000
0.0187, 0.200, 0.0000, 0.002, 0.00000
0.0311, 1.100, -0.0001, 0.003, 0.00000
0.0436, 1.900, 0.0003, 0.004, 0.00000
0.0498, 2.700, 0.0007, 0.004, 0.00000
0.0934, 3.500, 0.0012, 0.008, 0.00000
0.1246, 4.300, 0.0017, 0.010, 0.00000
0.0007, 5.100, 0.0022, 0.008, 0.00000

```

1st column = load  
2nd column = displacement

Figure 2: Format of an input file

### 3 Calculation of Young's Modulus

The Young's modulus is determined from the elastic part of the load-displacement diagram, considering reduced beam stiffness. The derivation of formulas needed for a determination of the notched beam stiffness can be found in the Appendix, A. Knowing the Young's modulus, the stress-strain curve can be calculated. **However, it is valid only for the part of the diagram representing the elastic response - and even this is only approximate, since there can arise stress singularities near the notch tip. Therefore the stress-strain diagram does not represent the material behavior and the curves serve only as a comparison among individual specimens.**

### 4 Calculation of Fracture Energy

The fracture energy is calculated using the standard RILEM approach and therefore it is specimen size dependent characteristic. In order to eliminate the size effect, use the specimens as large as possible.

### 5 Statistical Evaluation of Data

The data from multiple output files (see Section 13) are evaluated. The program calculates the mean values from the maximum forces reached during the experiment, corresponding displacements, maximum stresses in the edge fibres (based on linear elastic distribution of stresses in the cross-section), corresponding strains, Young's moduli and fracture energies. For the same quantities, standard deviation and coefficient of variation (equal to standard deviation divided by mean value) are also calculated.

## Part II

# Main Menu

After opening the program, a user should follow the order of buttons on the right-hand side in the main menu from the very top to the bottom. First step is to choose the option "input" to load a data into the program.

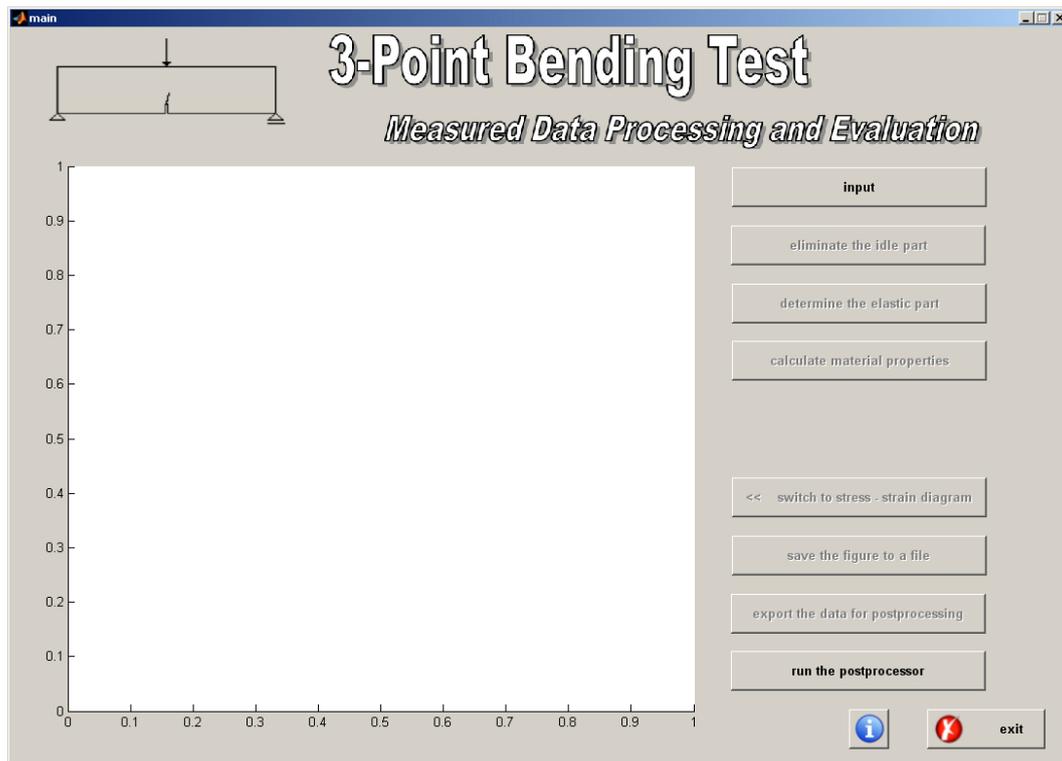


Figure 3: Main menu of the program

## 6 Input of Data

After pushing the button "input" in the main menu a user is asked about the geometry of the tested specimen and an input file with the data from a measurement. After pushing "choose a file" in the input menu, the user is directed to the folder "input" (see Fig.5). When the file is chosen and geometry of the specimen is set, the green check button must be pushed in order to load the data into the program (see Fig.4).

After the input is confirmed the user can continue with the other options in the main menu. The loaded load-displacement diagram should appear in the axes in the main menu (see Fig.6).

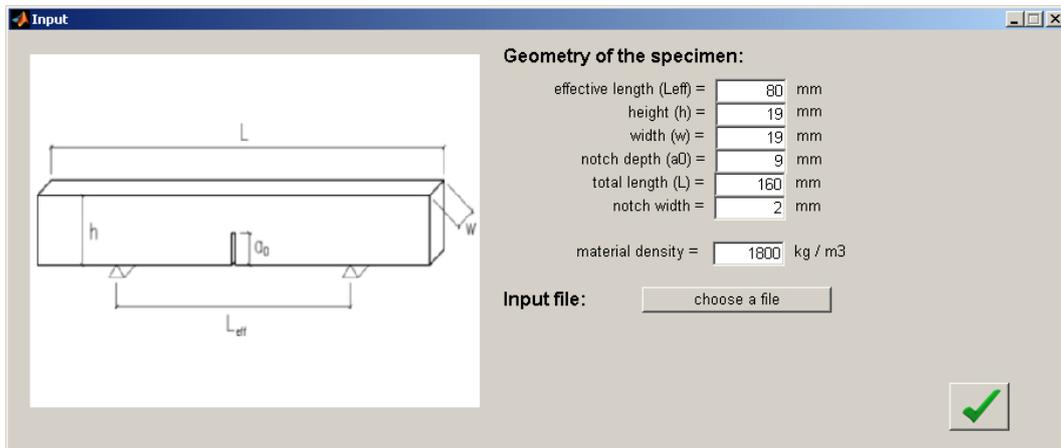


Figure 4: Input menu

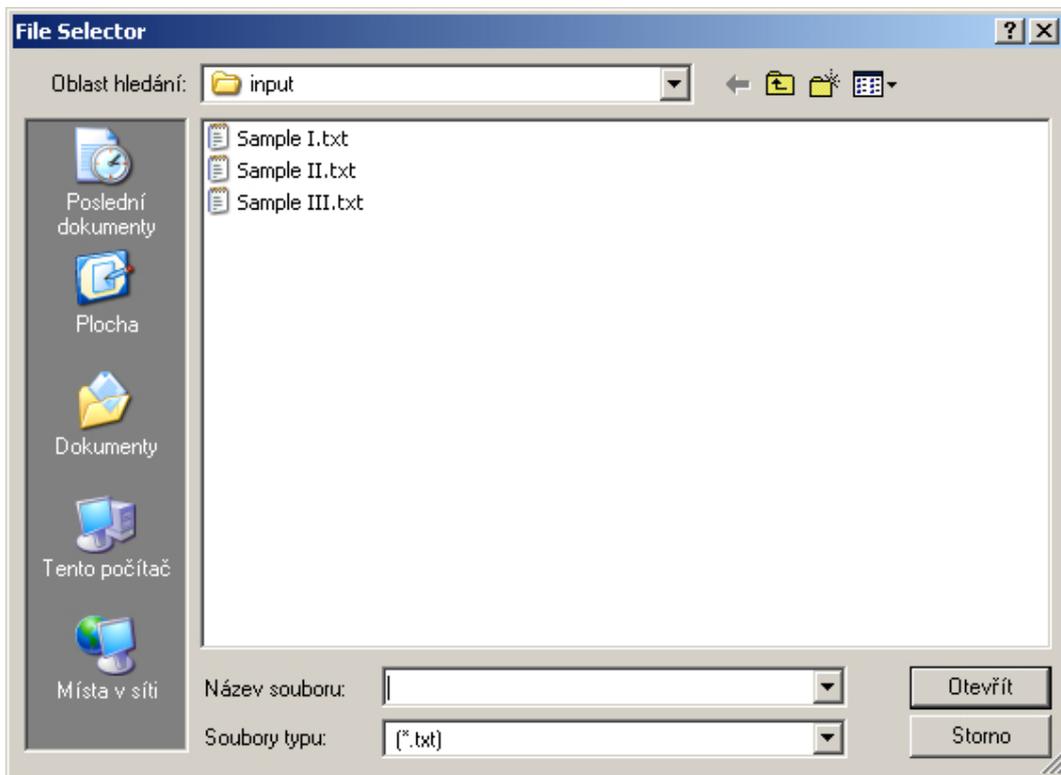


Figure 5: Choosing the input file

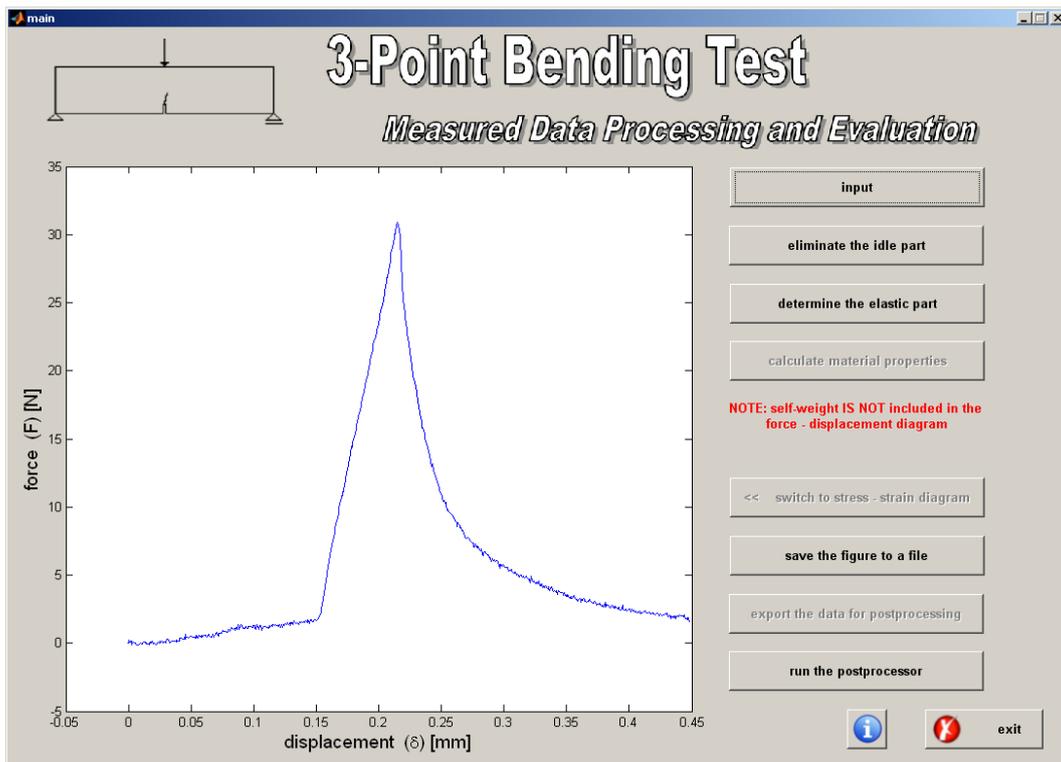


Figure 6: Main menu after loading the input data

## 7 Elimination of Idle Part in Diagram

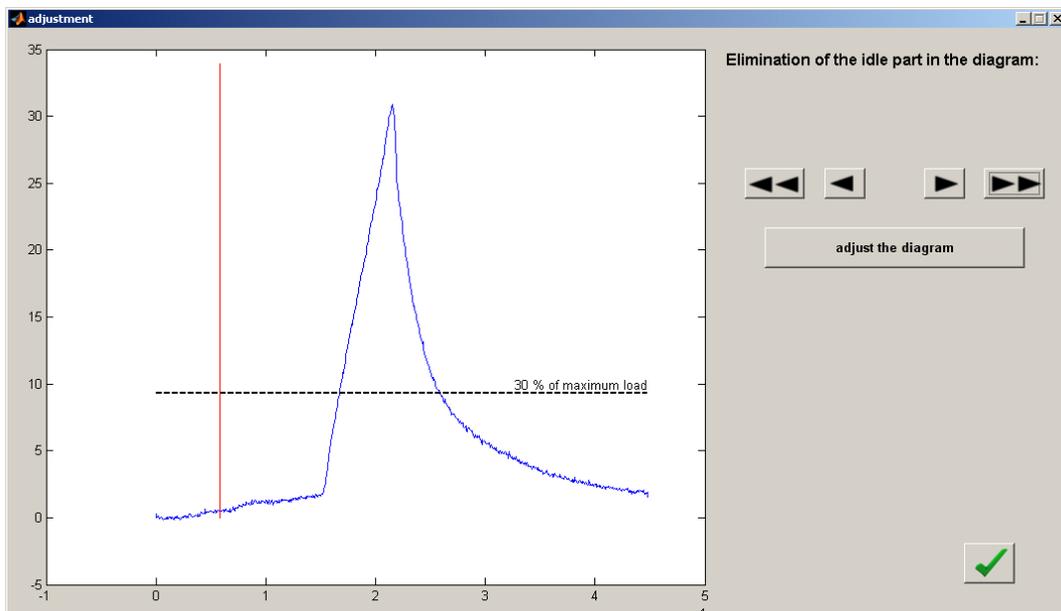


Figure 7: Setting of the elimination by arrows

In order to eliminate the part of the diagram, where there is no response from the beam (loading device is only approaching the specimen), the user has to select the "eliminate the idle part" option (see Fig.7). Then the user is asked to define the limit for the curve elimination and substitution by a new part (extrapolated from the rest of the load displacement diagram) - see Fig.7 and Fig.8.

If the user is satisfied with the adjustment the green check button in the input window can

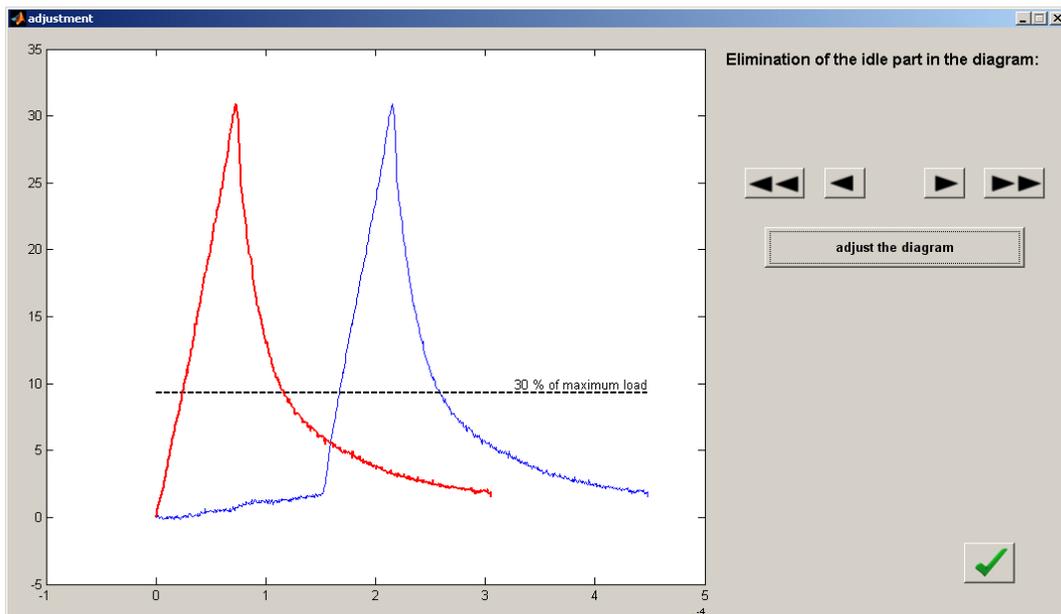


Figure 8: Eliminated and extrapolated load-displacement diagram

be pushed to confirm the change. The adjusted diagram should appear in the axes in the main manu - see Fig.9.

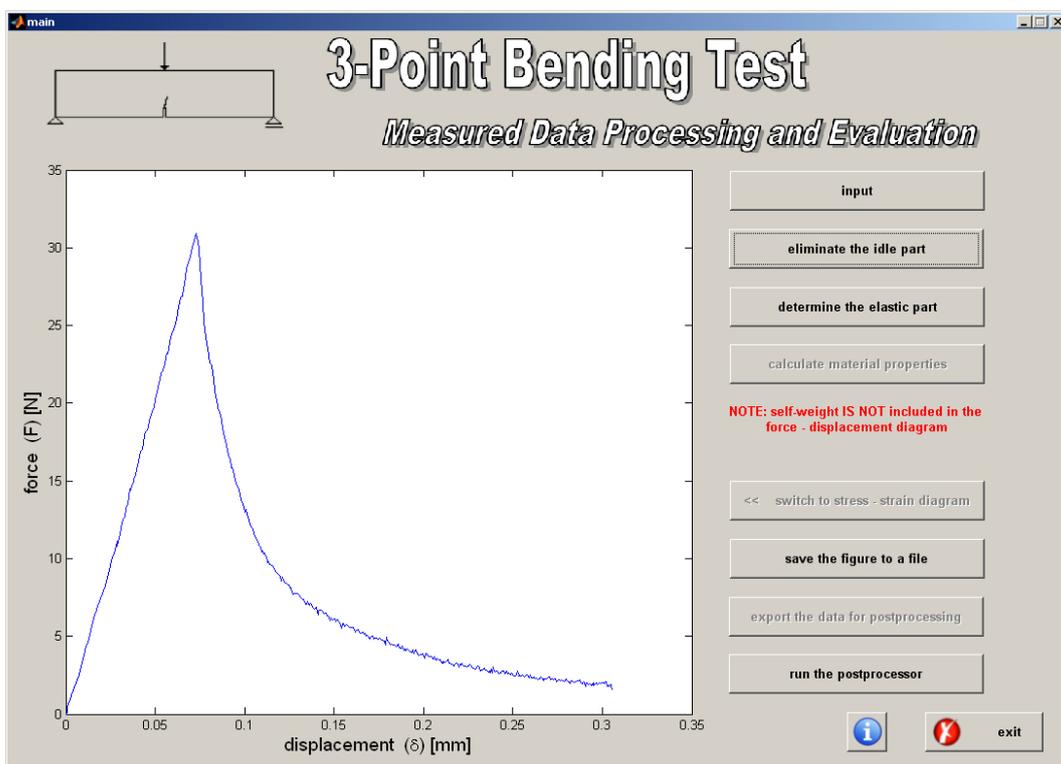


Figure 9: A force-displacement diagram after the adjustment

## 8 Elastic Part Determination

The next step is to determine the elastic response in the diagram. After the user chooses the option "determine the elastic part" in the main menu, a window as in Fig.10 appears. There the

user can move with the red circle using black arrows until the peak of the elastic part is reached. Then, again, the setting is confirmed by pushing the green check button in the right-bottom corner.

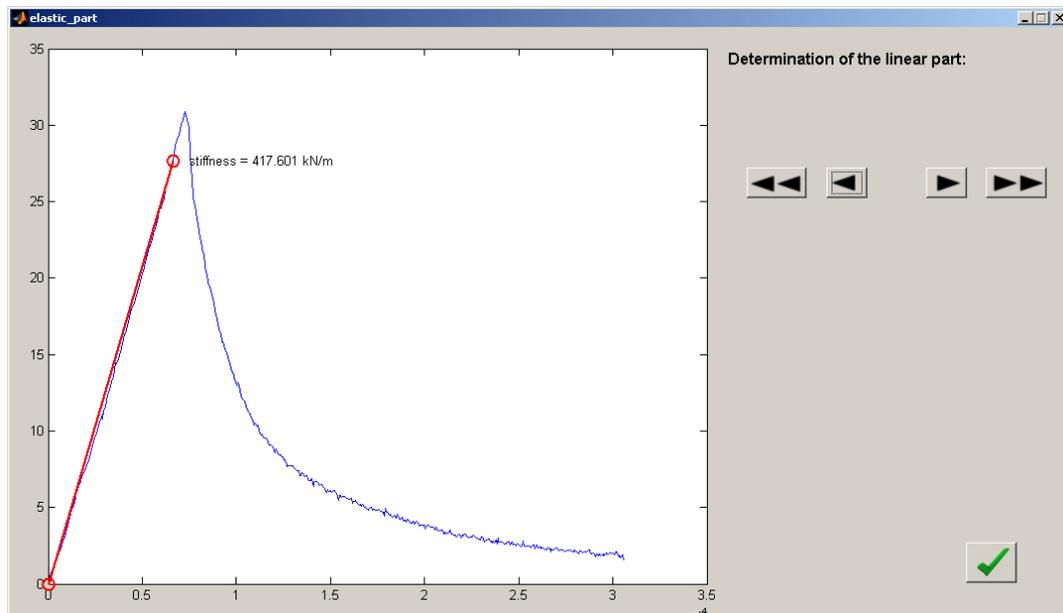


Figure 10: Determination of the elastic part in the diagram

## 9 Displaying Results

When all the adjustments and stiffness determination is done, the user can see the calculated results by pushing the "calculate material properties" button in the main menu. The window with summarized results shows up, as in Fig. 11.

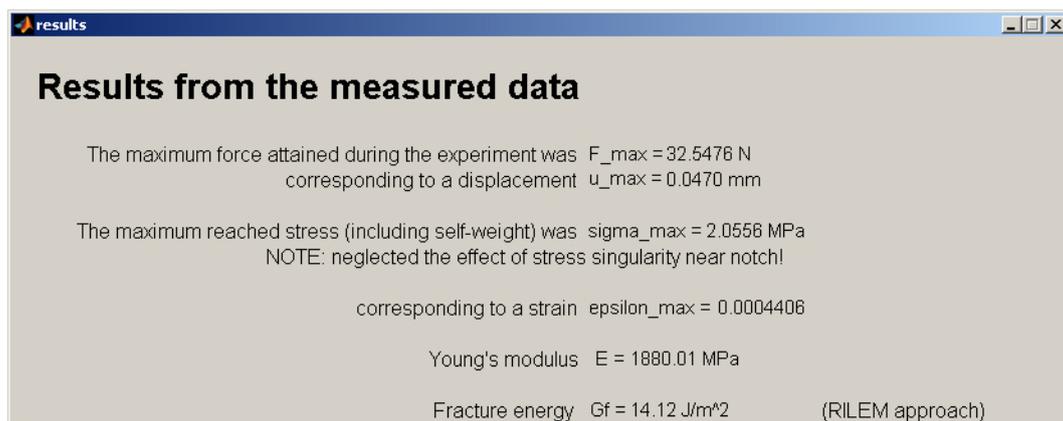


Figure 11: Displaying the results after pushing "calculate material properties" button

## 10 Switching to Stress-Strain / Load-Displacement Curve

A pseudo stress-strain diagram (where only the elastic part is true, for explanation read the Section 3) can be displayed on the axes in the main menu. This switch is done using the "switch to stress - strain diagram" button in the main menu.

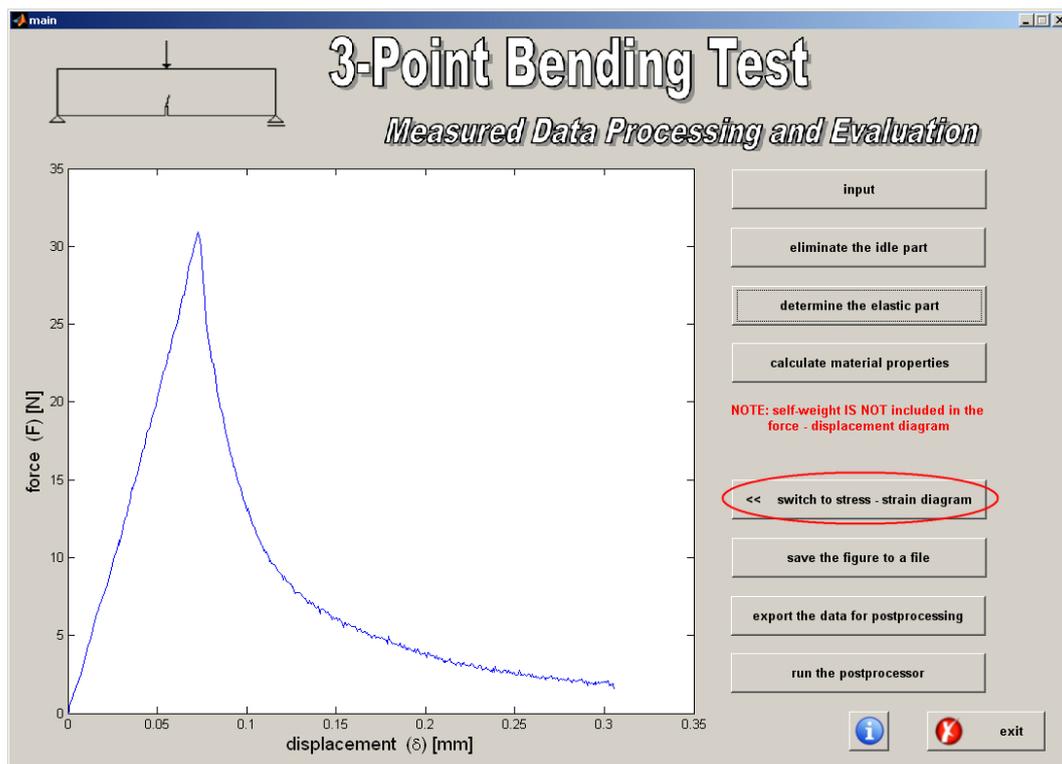


Figure 12: Switching to stress - strain diagram

When the stress-strain diagram is displayed, the self-weight is considered in the diagram (unlike in the case of force-displacement diagram).

## 11 Saving Diagram

The figure (axes from the main menu) can be easily saved. After pushing the "save the figure to a file" button in the main menu, the window as in Fig.13 shows up. Using the menu buttons at the top, the user can easily find the options for a figure adjustments and export the figure in a desired format and quality.

## 12 Export of Data for Postprocessing

The last-but-one option from the main menu is to export the data (i.e. the adjusted diagram, stiffness and other calculated results) for postprocessing. This is simply done by selecting the "export the data for postprocessing" option. This may take a second or two until a dialog as in Fig.14 shows up.

The data are actually saved into two separate files - the first containing the arrays with displacements and forces from the adjusted diagram and the second one (having the same name) that contains the additional data goes into the folder `aux_data` placed in the output folder (see Fig.15). The output file names are identical with the names of a chosen input file in the beginning.

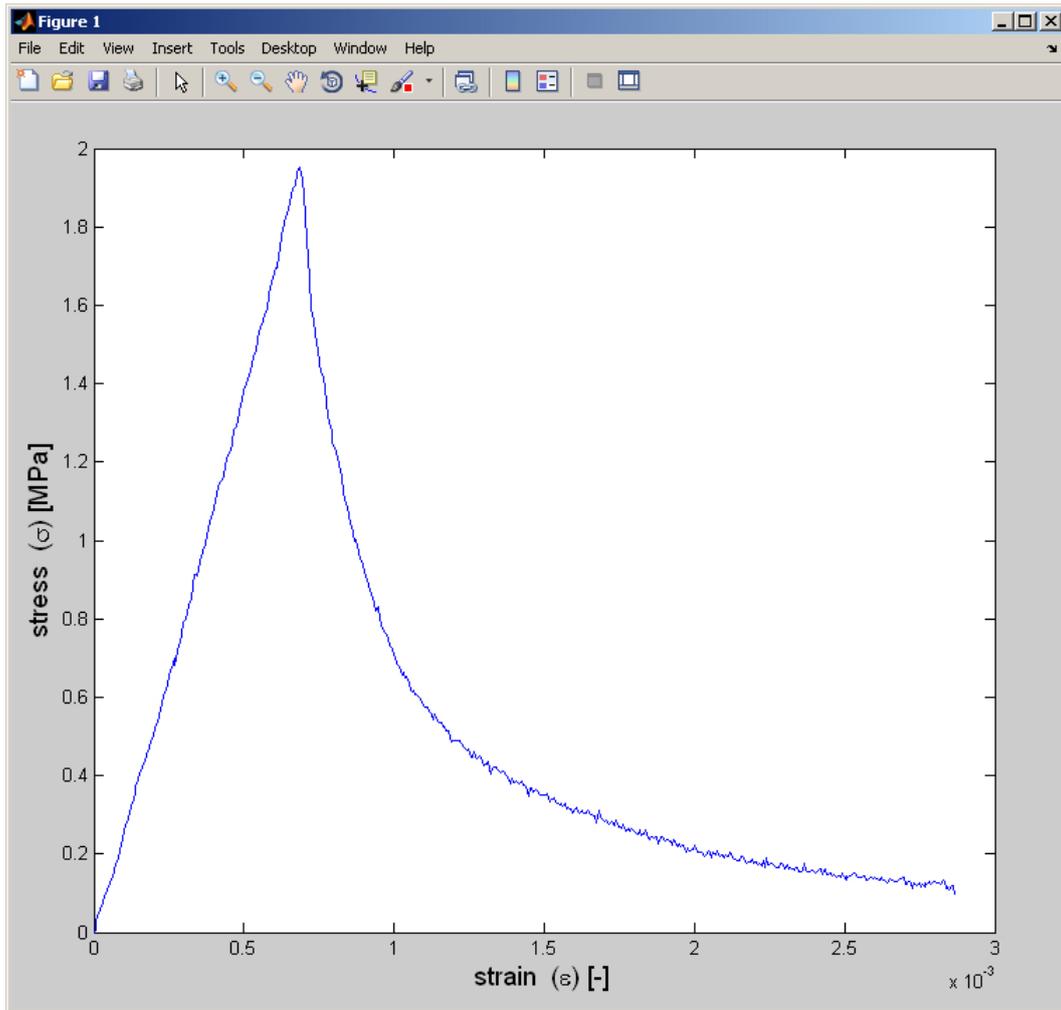


Figure 13: A figure prepared for printing or export



Figure 14: A dialog after saving the data for postprocessing

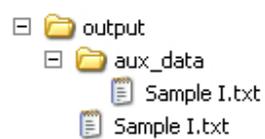


Figure 15: Structure of output files

## Part III

# Postprocessing - Evaluation of Data

In the postprocessing menu the data from multiple measurements can be displayed and compared. After choosing the "run the postprocessor" option in the main menu, the user is asked to choose multiple files (see Fig.16).

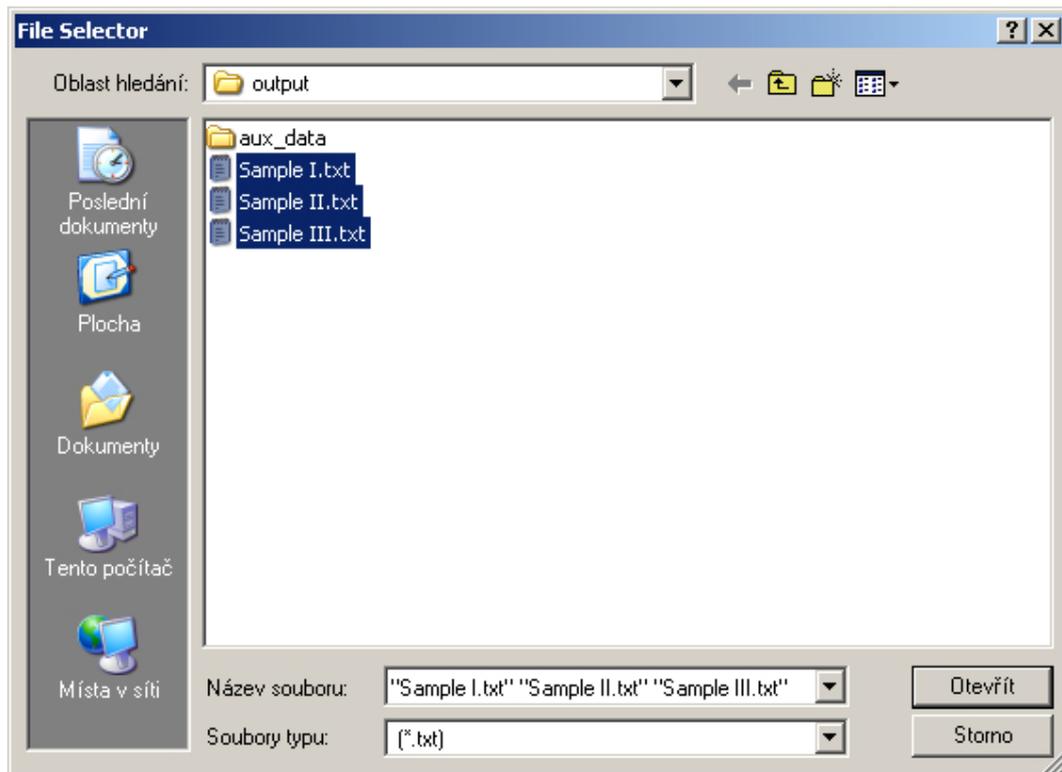


Figure 16: Choosing of files for postprocessing

Then the postprocessing window shows up, displaying the multiple stress strain curves (see Fig.17).

## 13 Displaying Results of Postprocessing

The "switch to stress - strain diagram" button and "save the figure to a file" button have the same functions as the buttons having the same name in the main menu. The only unknown option is "calculated values". After pushing this button a summary of results, together with histograms as in Fig.18, shows up.

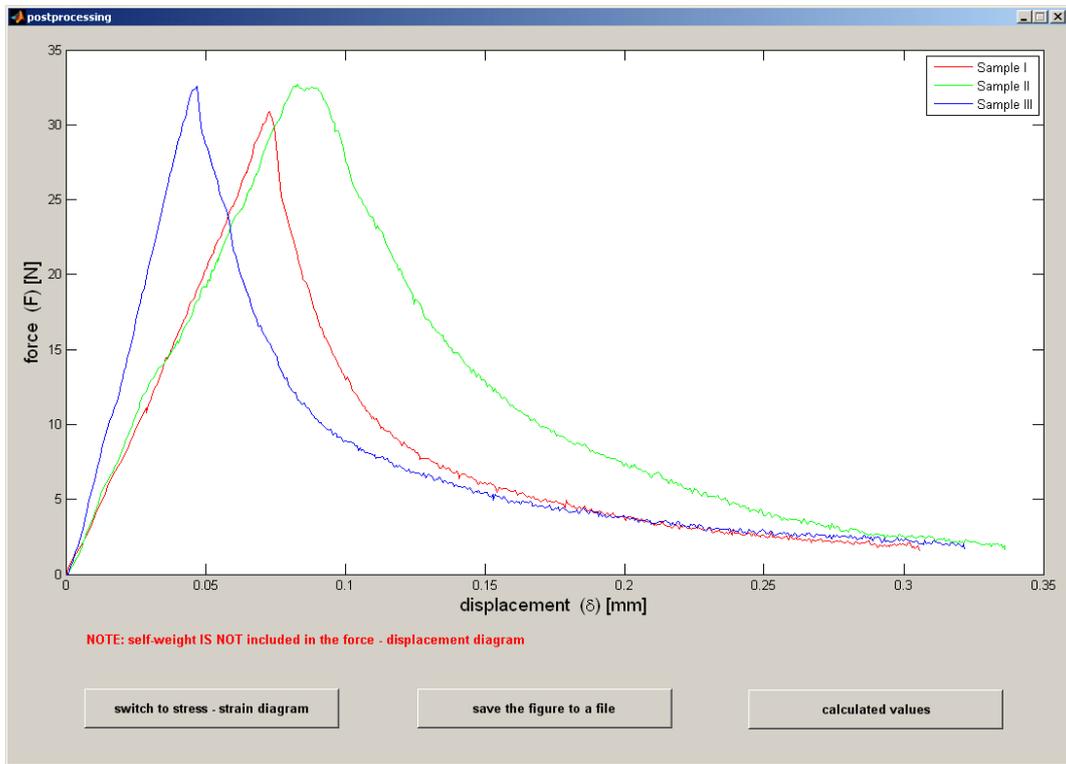


Figure 17: Postprocessor menu

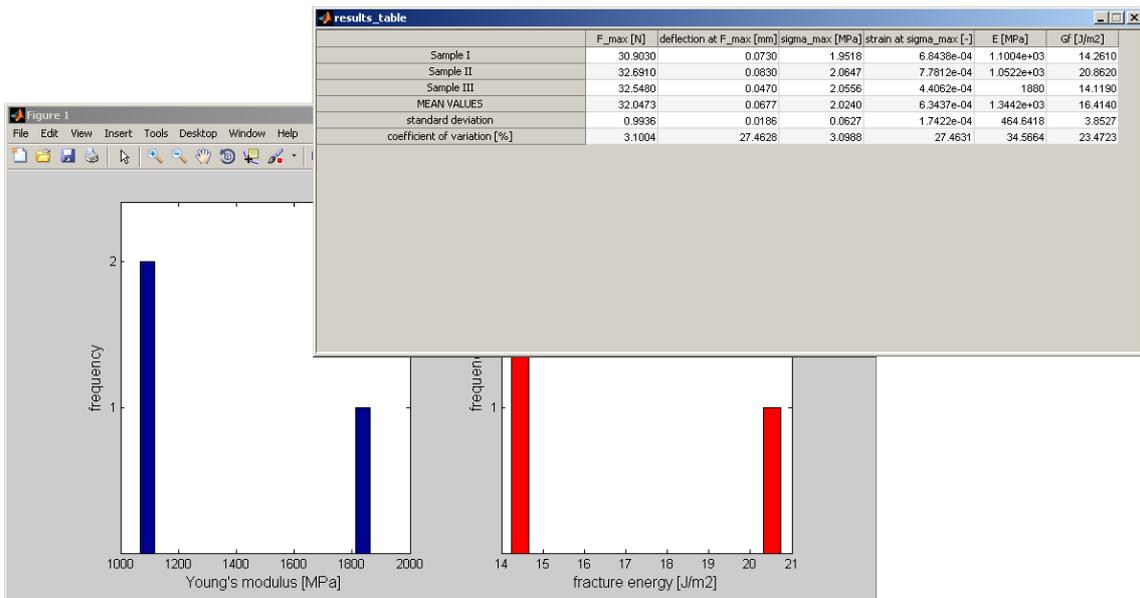


Figure 18: Summarized results

# Appendix

## A Determination of Notched Beam Stiffness

Bending stiffness of a beam can be determined from a force-displacement diagram obtained during a three-point bending test. If the beam has a uniform cross-section and elastic properties along, the bending stiffness to a concentrated force applied in the midspan can be calculated as

$$k = \frac{48EI}{L_{\text{eff}}^3} \quad (1)$$

where  $L_{\text{eff}}$  is the effective span, i.e. distance between supports,  $E$  is the Young's modulus and  $I$  is the second moment of area for the cross-section. Since the beam stiffness,  $k$ , is known from the force-displacement diagram, we can obtain the elastic material stiffness (Young's modulus) as

$$E = \frac{kL_{\text{eff}}^3}{48I} \quad (2)$$

### A.1 Investigated Specimen

The investigated specimen had a configuration as depicted in Fig.19, with material properties characterized by  $E = 2$  GPa and  $\nu = 0.2$ . The specimen was loaded right above the notch.

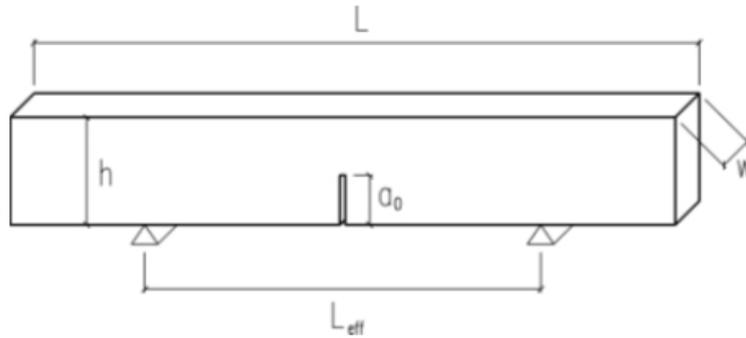


Figure 19: Geometry of the tested beam

It was found, that the beam stiffness is non-linearly dependent on the ratio of a notch length,  $a_0$ , to a beam depth,  $h$ , and on ratio of a notch width,  $w_a$ , to a beam span,  $L_{\text{eff}}$ . The results of a finite element analysis were compared to results given by the direct stiffness method. It turned out, that the direct stiffness method is not capable of predicting accurate values, since it neglects the transition zone near the beam notch and therefore exaggerates the beam stiffness by far.

### A.2 FE Analysis

The numerical calculations, investigating several geometry configurations, were conducted to investigate the influence of a notch on the beam stiffness. As a reference, a beam without notch was also investigated by the finite element analysis.

The analysis was done using the ADINA software. The material used was linear elastic and isotropic. The problem was solved by 2D plane stress model, using quadrilateral elements with 9 nodes.

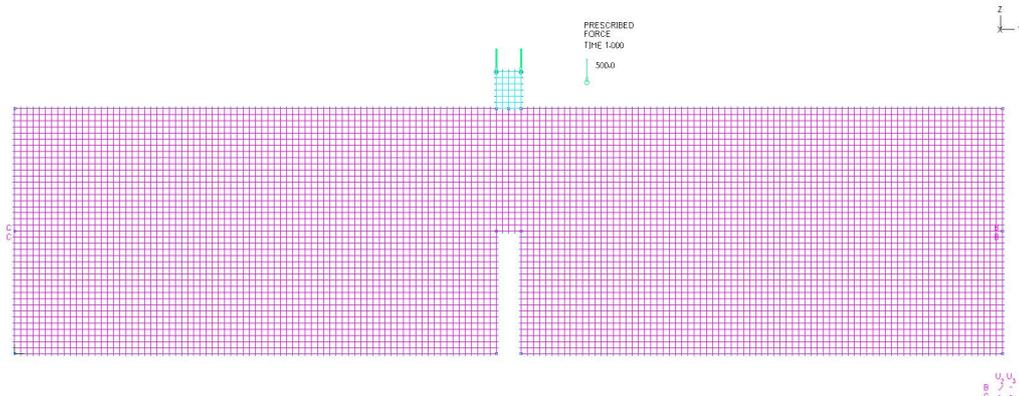


Figure 20: Mesh of beam for numerical analysis

The load (unit force) was not applied directly on the structure, but rather through the stiff distributing element to avoid a load concentration. The same problem could have been encountered near supports and therefore the supports were shifted from the bottom to a stiffer part near the center of the cross-section.

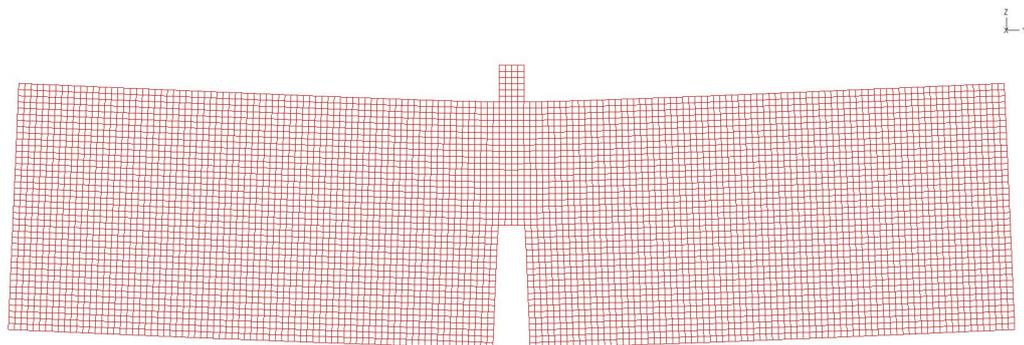


Figure 21: Deformed beam (exaggerated deformation)

The tested ratios of a notch length,  $a_0$ , to a beam depth,  $h$ , were in a reasonable range. The obtained results were used for curve fitting and regression to find a relation between the bending stiffness of beam without notch and a notched beam, based on a relative notch size.

The calculations were done on 40 different configurations of the beam geometry.

### A.3 Analysis of Results

It was found from the results of the numerical analysis that the ratios of a notch length to a beam depth and notch width to a beam span are mutually independent and therefore their influence can be expressed by means of independent coefficients. The dependence of the beam stiffness on the relative notch length can be seen in Fig.22.

The dependence of the relative notch width (related to a reference relative notch length) can be seen in Fig.23.

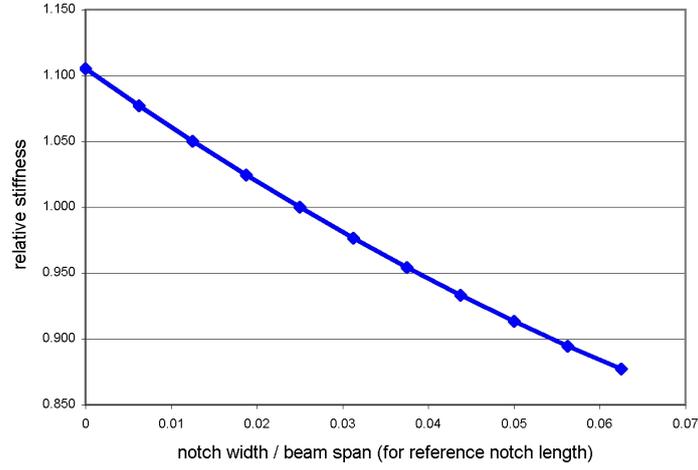


Figure 22: Dependence of beam stiffness on relative notch length

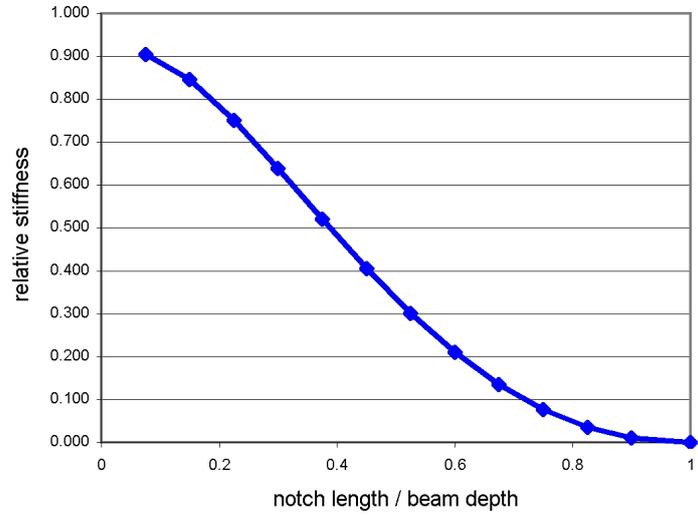


Figure 23: Dependence of beam stiffness on relative notch width

Using the formula for the beam stiffness given by equation 1 as a reference value, the solution was found in the form

$$k = \frac{48EI}{L_{\text{eff}}^3} \alpha \beta \quad (3)$$

where the coefficients,  $\alpha$  and  $\beta$ , represent the relative notch size, obtained from the regression analysis. The coefficient taking into account the relative notch length,  $a_0/h$ , was determined as

$$\alpha = -2.3617a^4 + 6.587a^3 - 5.2356a^2 + 0.0815a + 0.9277 \quad (4)$$

where  $a = a_0/h$ . The coefficient taking into account the relative notch width,  $w_a/L_{\text{eff}}$ , was found in the following form:

$$\beta = 14.981b^2 - 4.587b + 1.1053 \quad (5)$$

where  $b = w_a/L_{\text{eff}}$ . These coefficients should be valid for a relative notch length,  $a_0/h$ , ranging from 0.075 to 1.0 and for a relative notch width,  $w_a/L_{\text{eff}}$ , in the range of 0 and 0.0625, providing results with a negligible error.

## A.4 Verification

For the verification, a completely new configuration was considered. The tested beam was made of an isotropic material, having the elastic stiffness  $E = 20$  GPa and Poisson's ratio equal to 0.2.

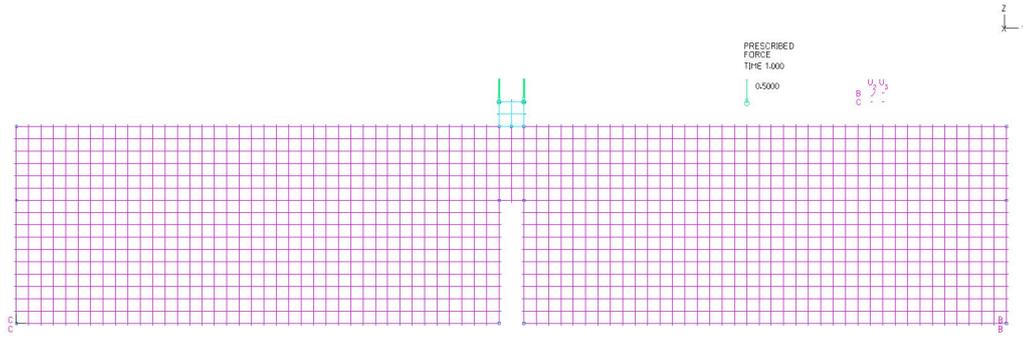


Figure 24: Tested beam for verification (FE model)

The effective beam span was set as  $L_{\text{eff}} = 0.4$  m, width  $w = 0.5$  m, depth  $h = 0.08$  m and the beam was loaded by a force of  $F = 1$  N in the midspan, right above the notch. The notch depth was set as  $a_0 = 0.05$  m and width as  $w_a = 0.005$  m.

First the midspan deflection of the beam loaded by a unit force was calculated. Then the notch was introduced into the model and the deflection was calculated again. The beam stiffness, determined as an inverse value of the calculated deflection, was calculated as introduced into the model  $k_1 = 27.79$  MN/m for the beam without notch and  $k_2 = 5.276$  MN/m for the notched beam.

Based on the beam geometry, the reduction coefficients from equations 4 and 5 were calculated as  $\alpha = 0.181$  and  $\beta = 1.05$ . The reduced stiffness of the beam, determined as  $k_{\text{cal}} = k_1 \alpha \beta$ , was then equal to  $k_{\text{cal}} = 5.291$  MN/m.

The difference between the stiffness of the notched beam determined by the finite element analysis,  $k_2$ , and stiffness calculated from the beam stiffness without notch,  $k_1$ , with use of the proposed coefficients,  $\alpha$  and  $\beta$ , was lower than 0.3 %.