ČERVENKA CONSULTING

**Červenka Consulting s.r.o.**
Na Hrebenkach 55
150 00 Prague
CzechRepublic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

# ATENA Program Documentation
# Part 4-7

# ATENA Science – GiD FRC Tutorial

Step by step guide for nonlinear analysis of fiber reinforced concrete structures with ATENA and GiD

Written by
**Tereza Sajdlová**

**Prague, August 22, 2016**

*Trademarks:*
*ATENA is registered trademark of Vladimir Cervenka.*
*GiD is registered trademark of CIMNE of Barcelona, Spain.*
*Microsoft and Microsoft Windows are registered trademarks of Microsoft Corporation.*
*Other names may be trademarks of their respective owners.*

*Copyright © 2015-2021 Červenka Consulting s.r.o.*

# CONTENTS


1. INTRODUCTION ..... 1

2. BRIEF OVERVIEW ..... 3

3. THE STEPS BEFORE MATERIAL MODEL DETERMINATION ..... 4

3.1 Starting Program ..... 4

3.2 Loading the Model ..... 4

4. FRC MATERIAL MODEL ..... 12

4.1 Necessary Information for Creating FRC Material Model ..... 12

4.2 Determining the FRC Material Model Parameters ..... 13

4.2.1 Manual Inverse Analysis ..... 14

4.2.2 Inverse Analysis by Consoft ..... 41

5. APPLICATION OF FRC MATERIAL MODEL ..... 42

5.1 Vault Bending Tests ..... 42

5.2 Model of Segment (Keystone) in Compression ..... 43

6. CONCLUSION ..... 44

7. PROGRAM DISTRIBUTORS AND DEVELOPERS ..... 45

8. LITERATURE ..... 46

# 1. INTRODUCTION

Fiber reinforced concrete (FRC) composites are more and more often applied in structures and therefore the need of numerical models describing these materials increases. Material behavior is more difficult to describe in comparison with ordinary concrete. Cementitious composites are combined with randomly oriented fibers made from different materials (steel, polymers, glass, etc.) and applied in different fractions.

The issue is to find the appropriate input material parameters to successfully model FRC.In particular, the tensile parameters that are important for FRC must be determined properly. The measured response of direct tensile test can serve for direct input of the parameters into the material model. Unfortunately, preparation of test sample is complicated and the test is not performed very often. The three or four point bending tests are more common. Results can be also used for the material model however they cannot be directly put into the model. Inverse analysis of the results must be performed to identify model parameters correctly.

This tutorial provides instructions regarding modelling of fiber reinforced concrete (FRC) materials using the programs **ATENA** and **GiD**. **ATENA-GiD** is a finite element based software system specifically developed for the nonlinear analysis of reinforced concrete structures. Besides the material models for ordinary reinforced concrete, **ATENA** offers special numerical models accounting for the FRC specifics such as shape of tensile softening branch, high toughness and ductility.

This tutorial contains a step by step explanation on how to obtain the input parameters of the FRC material model and its application for the analysis of the segmental tunnel lining. The four point bending test (simply supported beam loaded by two loads) is used for the determination of the material parameters, see Figure 1. Usually, results of this test are available for FRC together with those from a compressive test. It is also possible to use the three point bending test results, modify the geometry of the model, and follow the instructions it the tutorial. More details about the problem or experiment can be obtained from the program developer or distributor.

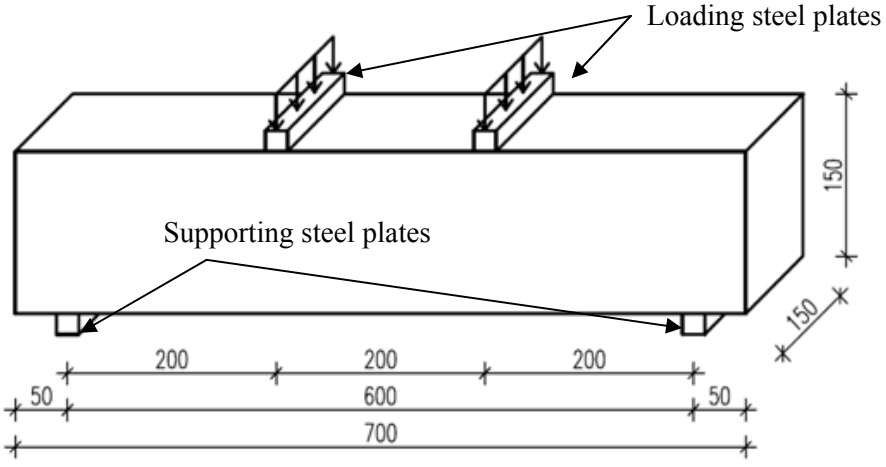



**Figure 1: Geometry of the structure for inverse analysis of material model parameters**

Tutorial is targeted for intermediate **ATENA-GiD** users who have already finished the basic **ATENA** and **GiD** tutorial. The preparation of the model is not described in detail, as it can be done according to the basic ATENA-GiD tutorial. This document is focused just on the material model determination and its application.

## 2. BRIEF OVERVIEW

To summarize this tutorial, basic steps of the procedure are highlighted now. The method how to obtain the material model for fiber reinforced concrete requires the knowledge of material model for ordinary concrete that is subsequently modified to FRC. To have an opportunity to define model more in the detail, **CC3DNonLinCementitious2User** is utilized and user can define his own laws in this fracture-plastic material model. Most important parameters for modeling FRC include:

- tensile stress – strain law (tensile function) – describes ductility in tension, it is the most important parameter for FRC modeling,
- tensile strength – tensile strength of FRC is usually higher than for ordinary concrete, thus it should be modified, this value also governs the y-axis of tensile function,
- compressive stress – strain law (compressive function) – describes ductility in compression,
- reduction of compressive strength due to cracks (Fc Reduction-COMPRED) – function describing how cracks reduce material compressive strength, default function drops sharply, for FRC it is recommended to keep the function constant with value 1 (it means no reduction of compressive strength due to cracks).

Procedure how to determine material model for FRC consists of following steps:

- initial setup of the tensile parameters (tensile strength, tensile function),
- running the analysis,
- export the load-displacement diagram from **ATENA** to excel file or similar,
- comparison of load-displacement diagram from calculation and laboratory test,
- if the difference between diagrams is within the user accuracy limits, the determination of FRC model is completed,
- if the difference between diagrams is not within the user accuracy limits, the input parameters must be modified in order to achieve better results, and again compare with the laboratory results. This step must be repeated until the results are satisfactory.

# 3. THE STEPS BEFORE MATERIAL MODEL DETERMINATION

## *3.1 Starting Program*

After the installation of the programs **GiD** and **ATENA** during the basic tutorial [1], you can start a nonlinear analysis using the **ATENA-GiD** system. Firstly, the program **GiD** is started, e.g. from the start menu of your computer using the following menu path: **Start | All Programs | CervenkaConsulting | ATENA Science | GiD**. This opens the program **GiD** which is used for the preparation of the numerical model of the analyzed structure.

If you do not have installed the above mentioned programs yet, you can install them following the instructions from the ATENA-GiD manual [2].

## *3.2 Loading the Model*

To start the model determination described in the following chapters, it is necessary to load the prepared model by selecting **Files | Open** in **GiD** and choosing the file: **%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.FRC_FRC_user_material_calibration\FRC_4PBT_2D_initial model.gid**.

The prepared model contains beam loaded by two described displacement at the top plates and supported by two bottom plates, see Figure 2. The geometry corresponds to the beam in Figure 1. The top and bottom plates are connected to the beam through fixed contacts (Master-Slave). The reactions are monitored at the supports, both on the left (Reaction L) and the right (Reaction R) bottom plate. The displacement of the beam is monitored in the middle at the point under the beam by boundary condition **Monitor for Point Nearest Node IP**, see Figure 3.

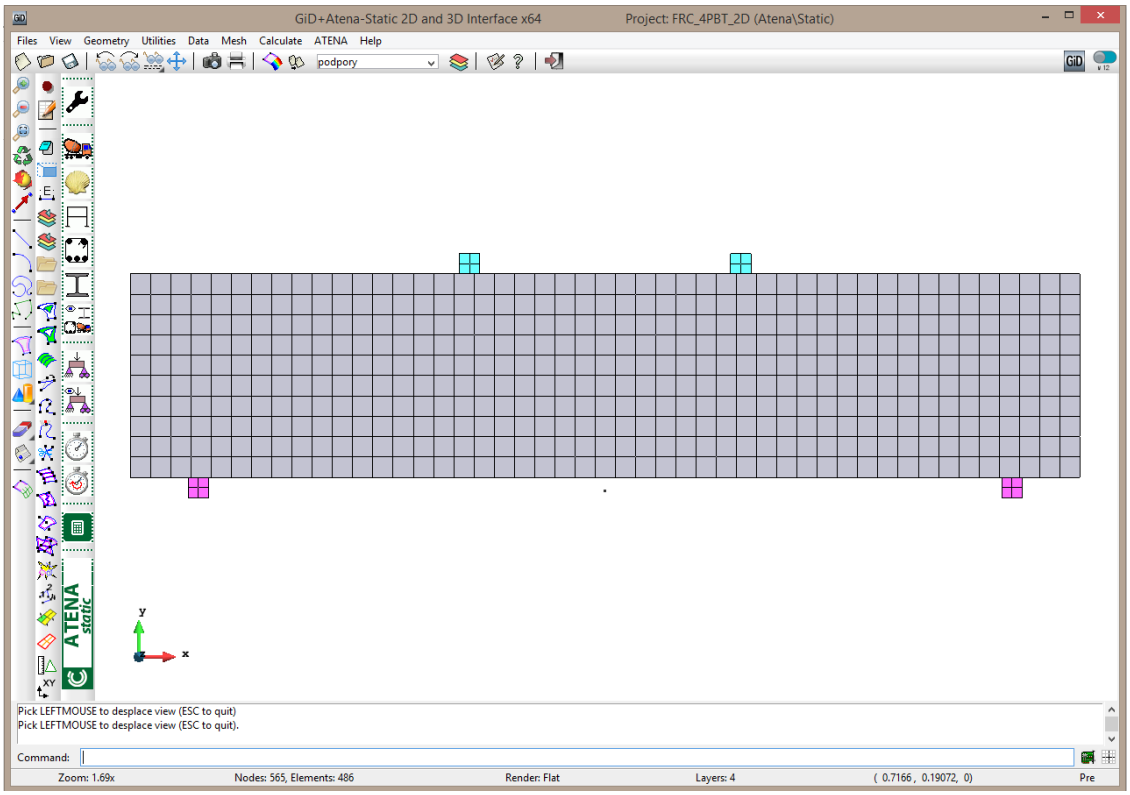



**Figure 2: Prepared model of the four point bending test in GiD**

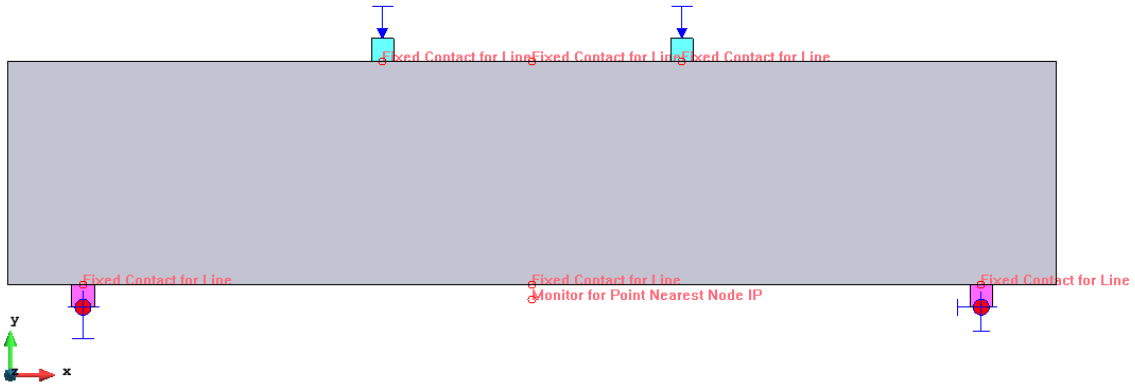



**Figure 3: Boundary conditions**

The size of the finite element mesh is 0.015 m (10 finite elements per height and 47 elements per length). The model consists of 486 finite elements (470 elements in beam and 16 elements in plates), see Figure 4. Two different materials are used in the model. The material named **Plates** is **SOLID Elastic** and it represents steel, the parameters are shown in the Figure 7 to Figure 9. The second material **FRC beam** is **SOLID Concrete** with the material prototype **CC3DNonLinCementitious2User** (see Figure 10 to Figure 16). At this stage, the material model has default values of its parameters. The process how to modify these parameters in order to model FRC successfully is described in the following sections.

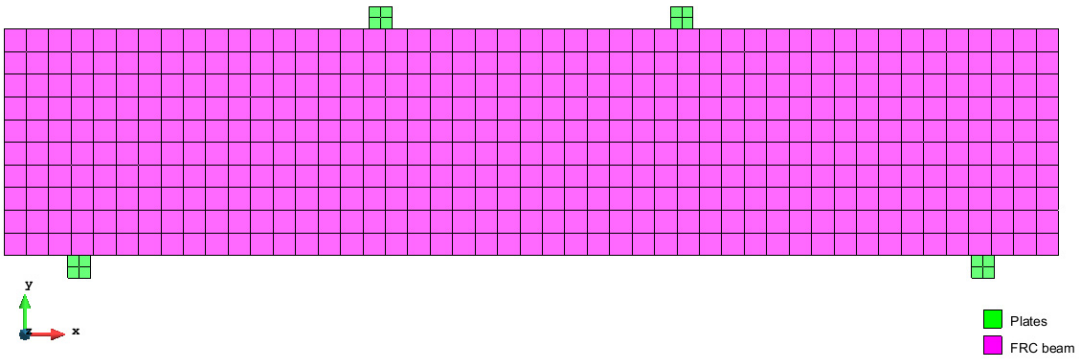



**Figure 4: Finite element mesh and applied material models**

Material model **CC3DNonLinCementitious2User** is the most sophisticated and most general model of FRC available in **ATENA** and it combines constitutive models for tensile (fracturing) and compressive (plastic) behavior. It describes the tensile behavior according to the material response measured in tests point-wise in terms of the stress-strain relationship (Figure 5). The first part of the diagram is the usual stress-strain constitutive law. After exceeding the localization strain $\varepsilon_{loc}$ the material law assumed for the characteristic crack band width $L_{ch}$ is adjusted to the actual crack band width $L_t$. Usually, $\varepsilon_{loc}$ is the strain after which the diagram is entering into the softening regime.

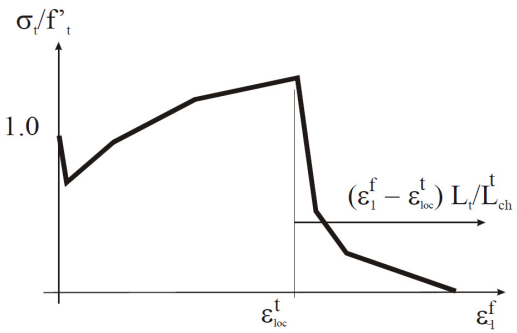



**Figure 5: User defined tensile behavior [4]**

The compressive stress-strain law of the mentioned material model is described in Figure 6. The softening law in compression is linearly descending and the end point of the softening curve is defined by plastic strain $\varepsilon_d$. By increasing material parameter $\varepsilon_d$ the contribution of the fibers to the compressive behavior of concrete is considered. Another important compressive parameter for FRC modelling is reduction of compressive strength due to cracks (labelled as Fc Reduction-COMPRED, see Figure 14) which says how the strength is reduced while the material is subjected to lateral tension.

See also the ATENA Troubleshooting Manual [5], section 2.2.4 I want to use the user-defined stress-strain law of concrete to replace that used in ATENA program. How can I do it? for more information about the **CC3DNonLinCementitious2User** material model.

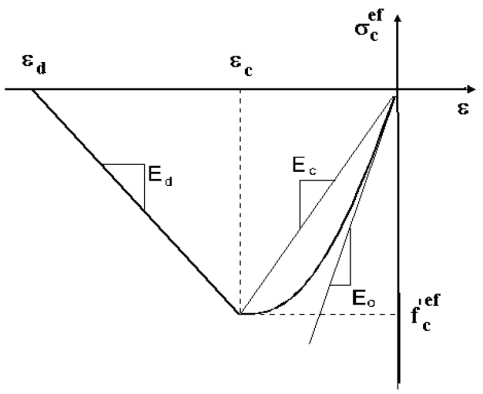



**Figure 6: Compressive stress-strain law [4]**

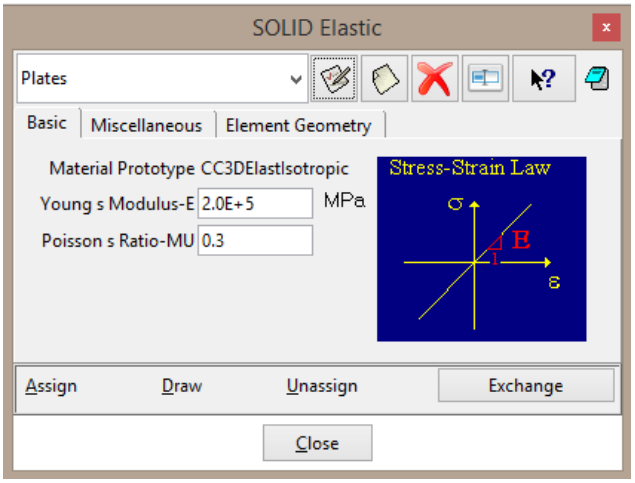



**Figure 7: Material model of plates – Basic parameters**

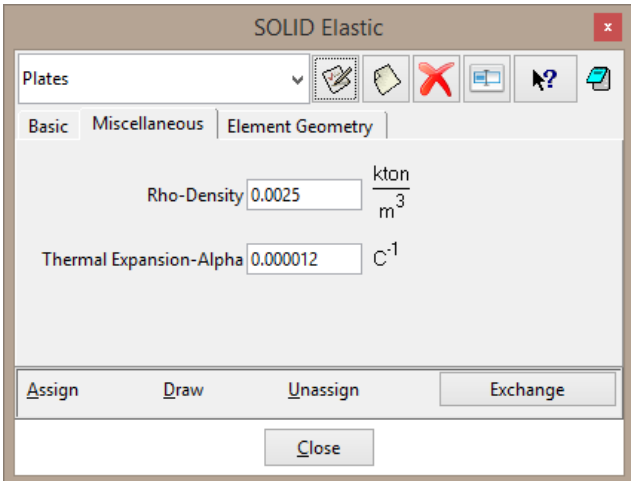



**Figure 8: Material model of plates – Miscellaneous parameters**

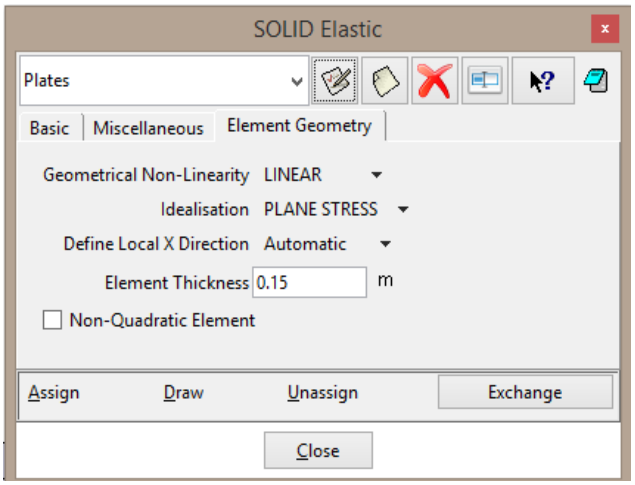



**Figure 9: Material model of plates – Element geometry – plane stress idealization, element thickness 0.15 m according the geometry in Figure 1**

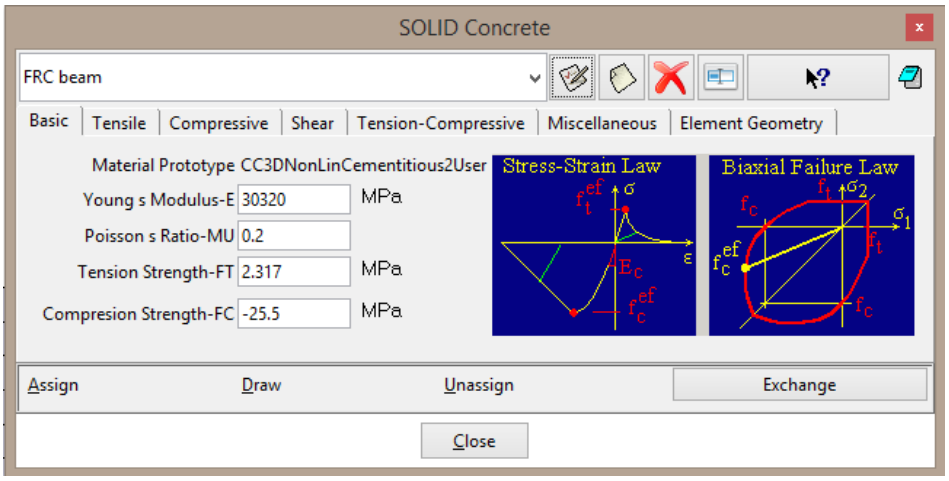



**Figure 10: Material model of FRC – Default basic parameters**

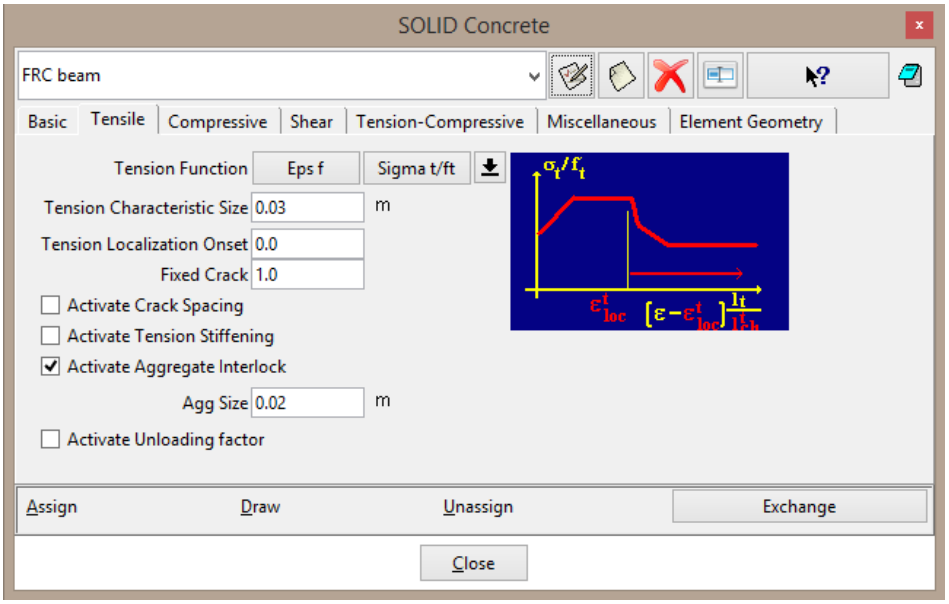



**Figure 11: Material model of FRC – Default tensile parameters**

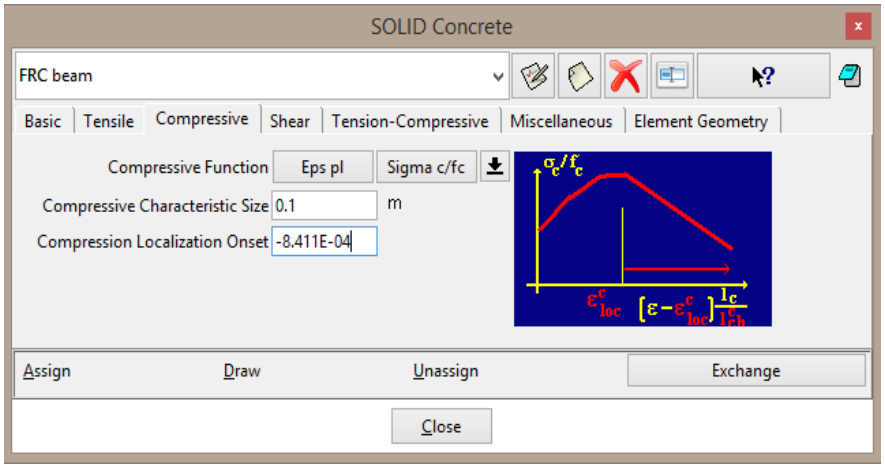



**Figure 12: Material model of FRC – Default compressive parameters**

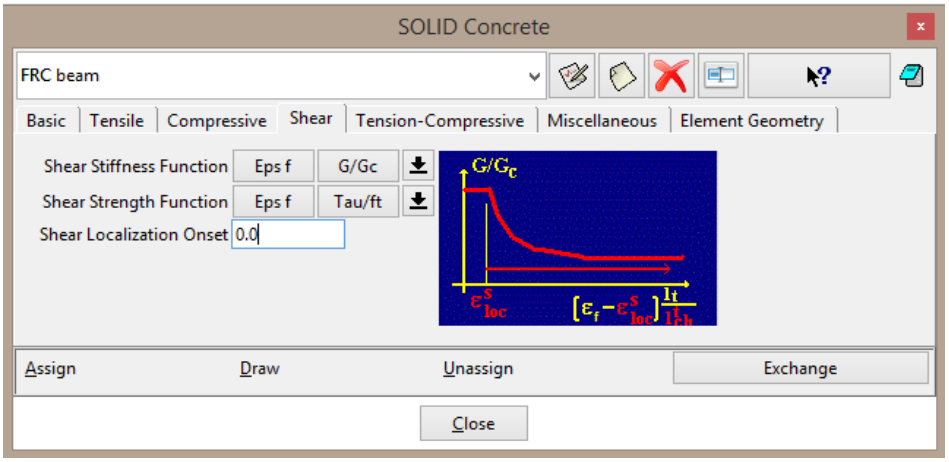



**Figure 13: Material model of FRC – Default shear parameters**

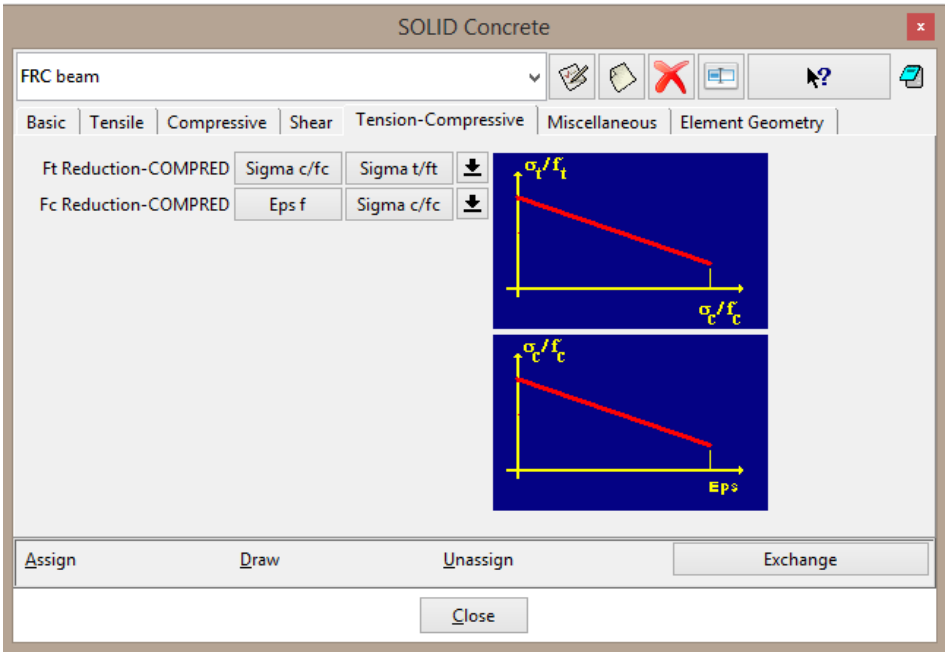



**Figure 14: Material model of FRC – Default tension-compressive parameters**

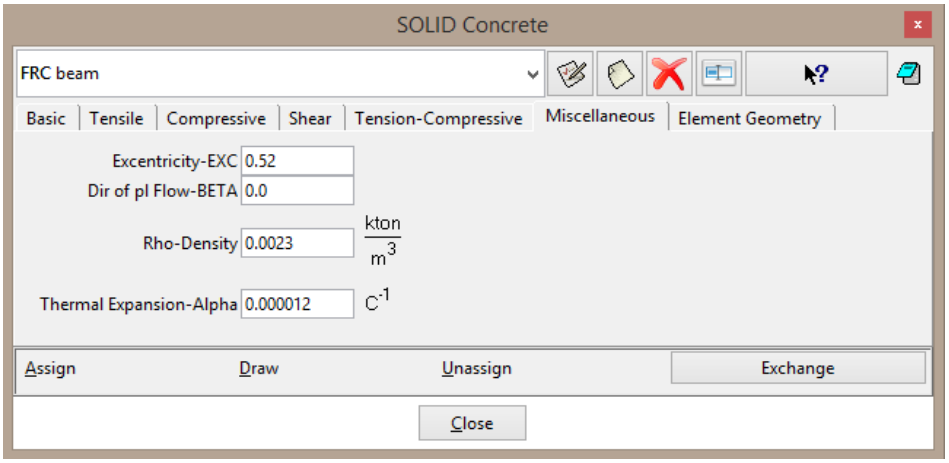



**Figure 15: Material model of FRC – Default miscellaneous parameters**

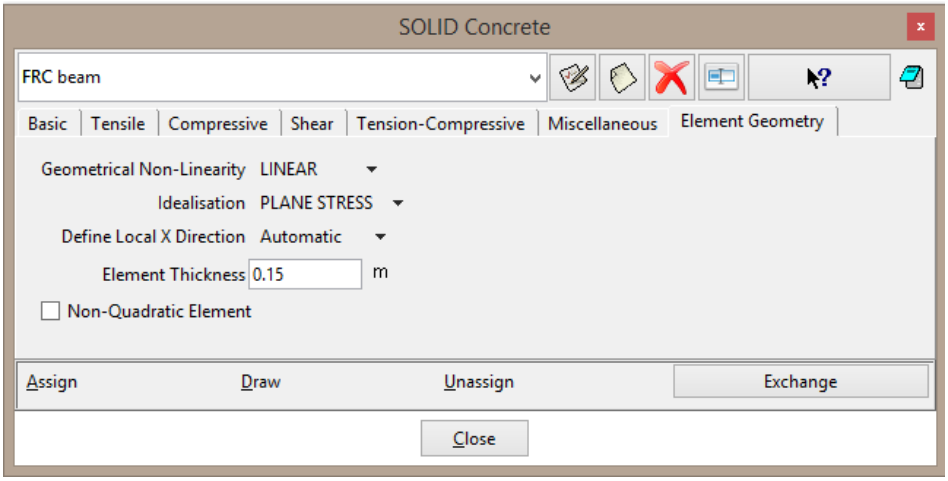



**Figure 16: Material model of FRC – Element geometry – plane stress idealization, element thickness 0.15 m according the geometry in Figure 1**

The last part of the prepared model are Interval data (Figure 17). The load is applied in the 70 steps with interval multiplier 70. It means that each step contains the assigned load multiplied by 1. The number of steps is based on the specific calculation which the user wants to provide. In the presented case, it is known from the experimental results that the beam should fail at the deflection of about 7 mm, see Figure 18. The prescribed displacement applied at the top plates is 0.1 mm. This means that the predefined displacement should be multiplied approximately 70 times to reach the failure. Based on this assumption, the load interval is multiplied by 70. Naturally, such a load should not be applied to the structure in one moment. Therefore, it is necessary to subdivide the interval in several load steps. In this case the interval will be divided in 70 load steps.

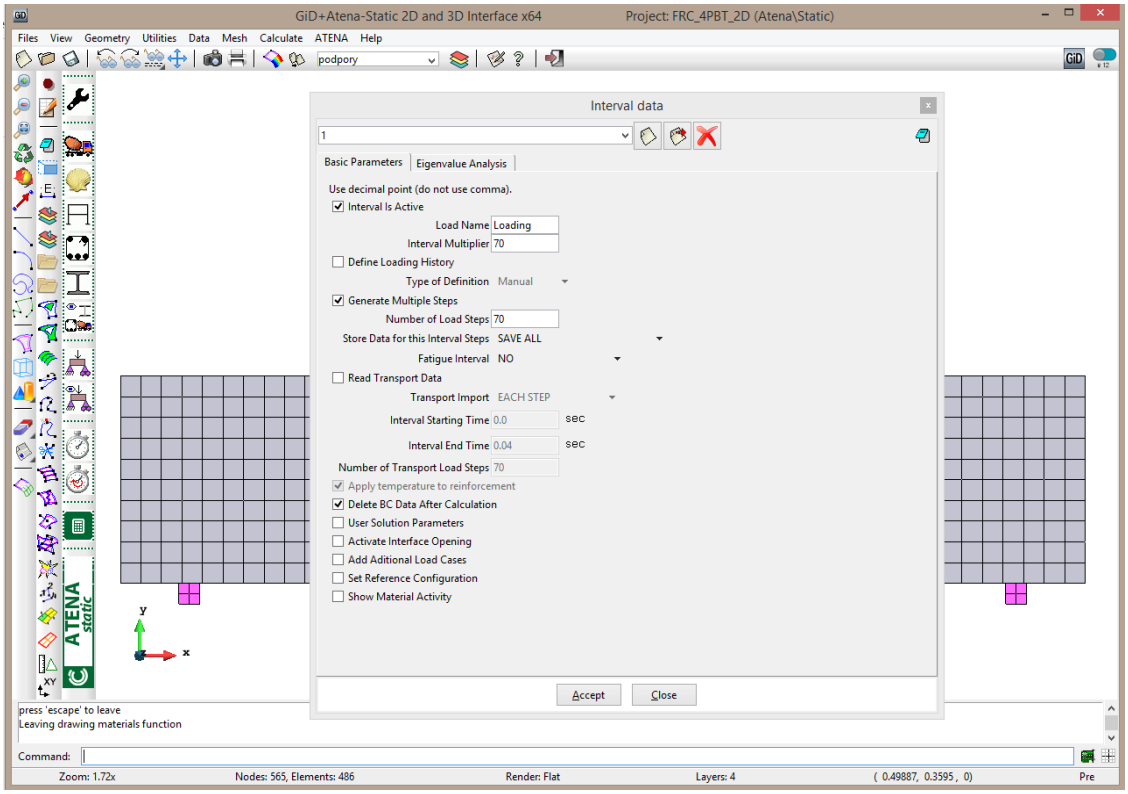



**Figure 17: Interval data**

# 4. FRC MATERIAL MODEL

## *4.1 Necessary Information for Creating FRC Material Model*

For determining FRC parameters that serve as an input for material model in **ATENA**, it is necessary to use results from laboratory tests, e.g. three (3PBT) or (better) four point bending (4PBT) tests, compression tests on cubes or cylinders and test for the elastic (Young´s) modulus.

The data from four point bending test (load-displacement diagrams, see Figure 18), compression test (compressive strength) and Young´s modulus are available for the case presented in this tutorial and are summarized in Table 1. It is also known that concrete belongs to class C110/130 and it is reinforced by steel fibers BASF Masterfiber 482 in volume fraction 1.5 %. The fibers length is 13 mm and the diameter is 0.2 mm.

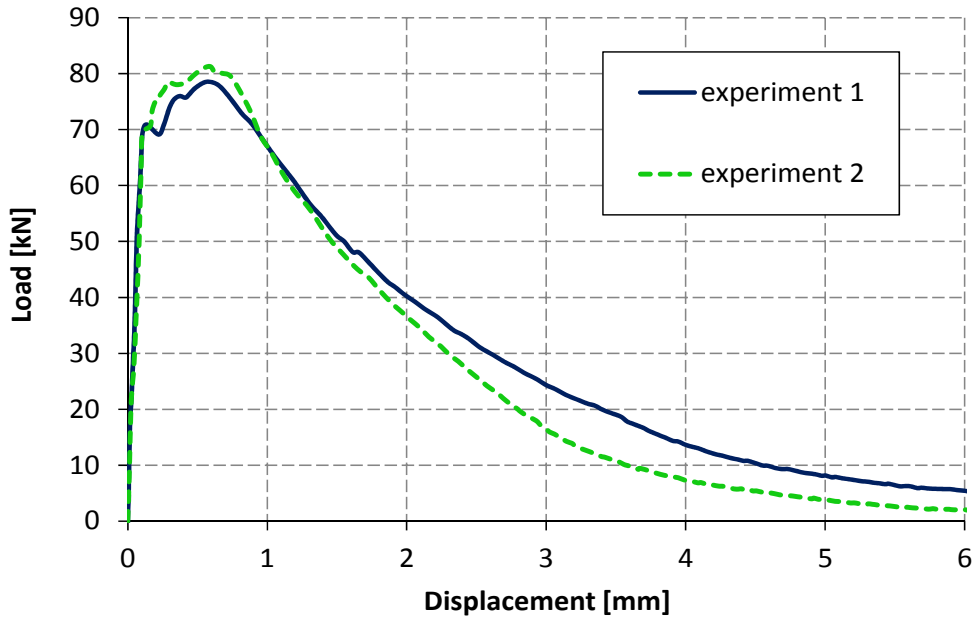



**Figure 18: Measured load-displacement diagrams of laboratory tests**

**Table 1: Available material parameters**

| Material parameter [unit] | Value |
|---|---|
| Cylindrical compressive strength [MPa] | 125 |
| Young´ s modulus [MPa] | 45000 |

## 4.2 Determining the FRC Material Model Parameters

Based on the available results from chapter 4.1 the material model for FRC can be determined. Clear parameters as compressive strength and Young´s modulus are directly put into the model.

The issue is to obtain tensile parameters as tensile strength and tension function (Figure 11), i.e., the key FRC parameters. The measured response of direct tensile test can serve for direct input of the parameters into the material model. Unfortunately, preparation of test sample is complicated and the test is not performed very often. Therefore, the results of four point bending test are utilized for parameters determination but they cannot be directly put into the model. Inverse analysis of the load-displacement diagrams from Figure 18 needs to be performed to identify parameters correctly. There are two possible methods, the first one is manual inverse analysis which consists of following steps:

- initial setup of the tensile parameters (tensile strength, tensile function),
- running the analysis,
- export the load-displacement diagram from **ATENA** to excel file or similar,
- comparison of load-displacement diagram from calculation and laboratory test,
- if the difference between diagrams is within the user accuracy limits, the determination of FRC model is completed,
- if the difference between diagrams is not within the user accuracy limits, the input parameters must be modified in order to achieve better results, and again compare with the laboratory results. This step must be repeated until the results are satisfactory.

The step by step procedure of this method is described in chapter 4.2.1.

The second method utilizes a sophisticated program for inverse analysis instead of the manual approach. The program is called **Consoft** and it was developed by Prof. Dr.-Ing. Volker Slowik and his colleagues at the University of Applied Sciences in Leipzig, Germany (HTWK). Automatic analysis based on the evolutionary algorithms is used for determining the tensile function [6]. Before the Consoft analysis, it is necessary to specify the initial function parameters and their limits. The experiment is then numerically simulated and the initial parameters are modified during an iterative process to achieve the best possible agreement between the experimental results and the numerical model. As a result, **Consoft** gives the tensile strength and tensile function to the user and afterwards it can be used for **ATENA** simulations. The process is described in detail in chapter 4.2.2. The chapter is now under construction and will be completed as soon as possible.

## 4.2.1 Manual Inverse Analysis

### 4.2.1.1 Modification of the Default Material Model

The material modification starts by selecting the **Define SOLID Concrete material** icon or with the command **Data | Materials | SOLID Concrete** in main menu. After selecting this command, the window for the definition of the **SOLID Concrete** appears (see Figure 19). The prepared material called FRC beam should be selected (see Figure 20) and then its parameters can be changed.

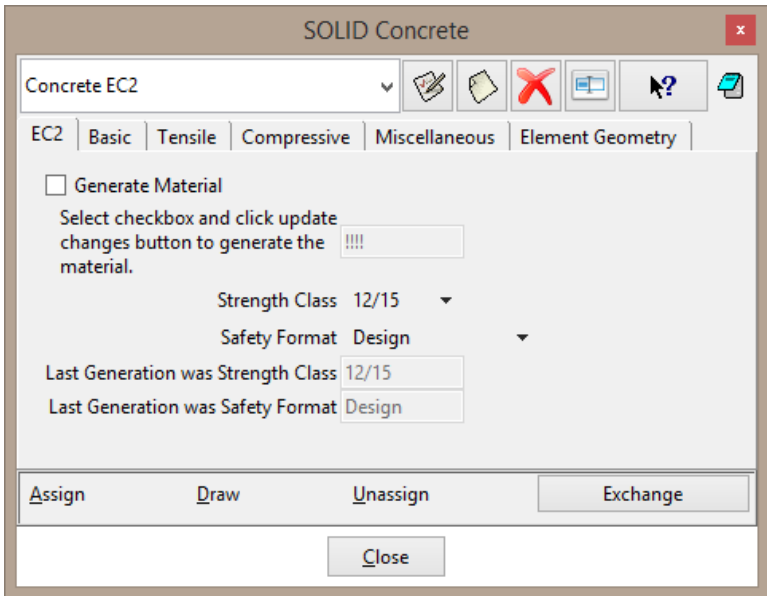



**Figure 19: The window for the definition of the SOLID Concrete**

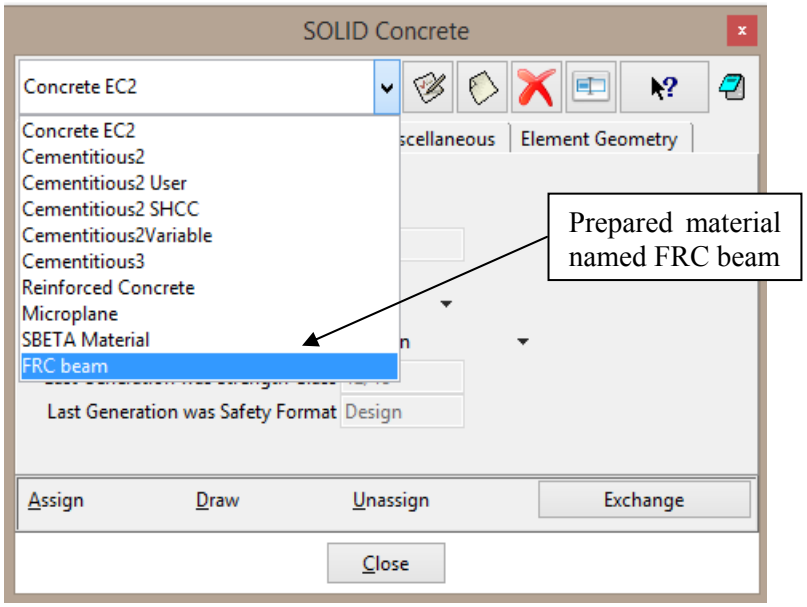
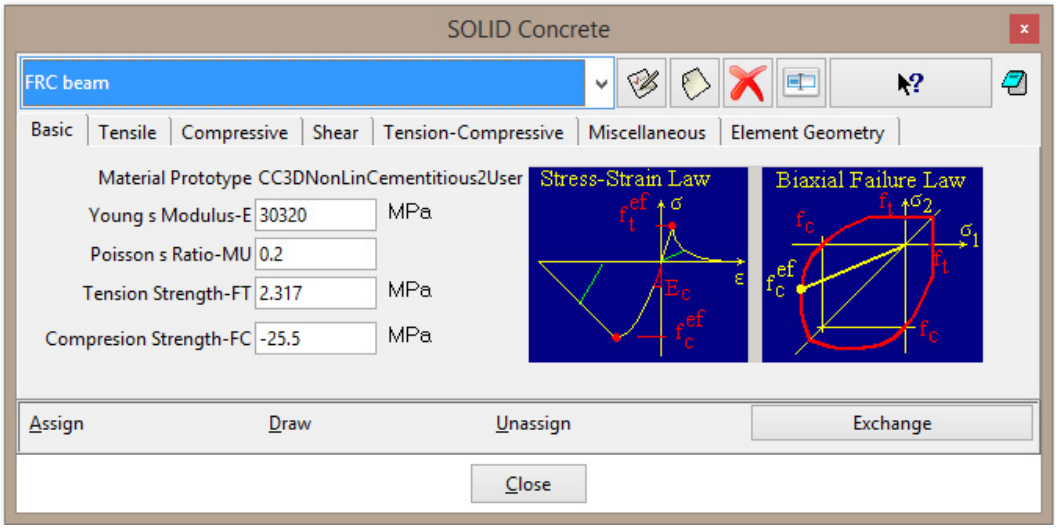
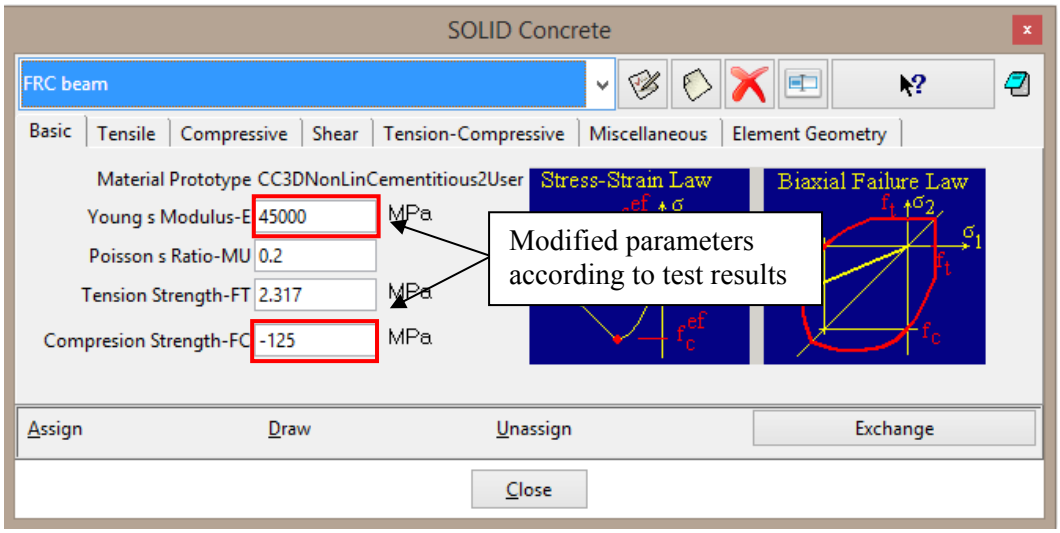



**Figure 20: The selection of the FRC beam material**

Firstly, parameters directly obtained by test results are modified in the material model. In this case, the compressive strength and Young´s modulus are known and can be edited in the model according to the values in Table 1. The data before and after the modification are depicted in Figure 21. The modified parameters are saved by clicking on the **Update Changes** icon .




(a)




(b)

**Figure 21: The Basic parameters of the material FRC beam before (a) and after (b) modification**

Secondly, other parameters must be estimated. Attention should be paid to the tensile parameters which are of crucial importance for FRC materials. The first peak of the Load - Displacement (L-D) diagram represents the formation of tensile crack and it is governed by the tensile strength. According to the assumptions of linear elastic behavior, the tensile strength $F_t$ can be determined by formula for flexural stress $\sigma$:

$$\sigma = M/W \quad (1)$$

where $M$ is bending moment at the central part of the beam loaded by four point bending test and it can be calculated as:

$$M = Fl/6 \quad (2)$$

where $F$ is total load acting on the beam and $l$ is the span of the beam.

The denominator of formula (1) is section modulus of the beam:

$$W = bh^2/6 \quad (3)$$

where $b$ is the width of the beam cross section and $h$ is the height of the beam cross section. By utilizing formulas (2) and (3) in the formula (1), the tensile strength $F_t$ can be calculated as:

$$F_t = F_{max}\, l/bh^2 \quad (4)$$

where $F_{max}$ is the load corresponding to the tensile strength (first peak in the L-D diagram).

*Fib* Model Code for Concrete Structures 2010[7]describes a procedure how to find the tensile stress – strain law (tensile function) based on the results of three point bending test on the notched beam according to EN 14651 [8]. The diagram of the applied force versus the crack mouth opening displacement is a required result of the test. Afterwards, residual flexural tensile strength can be calculated for individual crack openings by the formula (1) where the bending moment is adapted to the three point bending test. Thus obtained tensile function can serve as a first estimation of the function in the model.

As there is no information about crack opening from the four point bending test, the procedure described in *Fib* Model Code [7] can be used at least for determining the tensile strength. Load $F_{max}$ corresponding to the tensile strength is approximately 70 kN, see the first peak in the Figure 18. The cross section dimensions are 150 x 150 mm and span of the beam is 600 mm, see Figure 1. The tensile strength calculated by formula (4) is 12.4 MPa and this value is used in the model, see Figure 22. The tensile strength is quite high but considering high compressive strength 125 MPa it is possible.

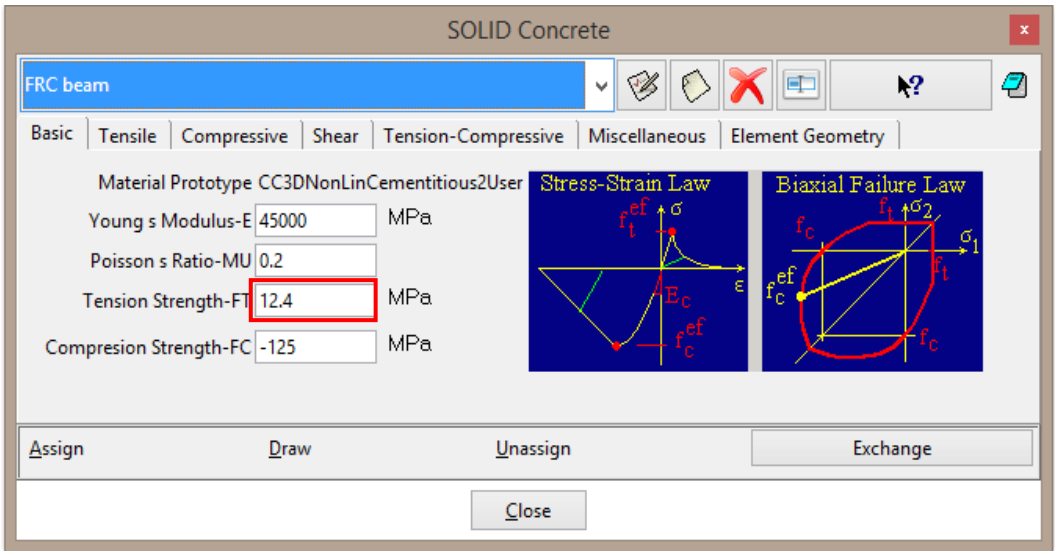



**Figure 22: The modification of the tensile strength**

Another important tensile parameter is the tensile function. In the fracture model, each crack is represented on the finite element level and its response is characterized by the traction-separation relationship (tensile function). A crack is perceived as a displacement discontinuity, which, however, is capable of transferring stress between its faces. This cohesive stress is related to the crack opening displacement through a traction-separation relationship. To this end, the crack band model is employed and the band width (characteristic length) is related to the element size to remain objective results with respect to the size of the finite elements.

The aforementioned tension function is specified in the model as relationship between fracture strain on the horizontal axis and tensile stress divided by tensile strength on the vertical axis. The fracture strain can be calculated by the following formula:

$$\varepsilon_f = w_c / L_t \qquad (5)$$

where $\varepsilon_f$ is fracture strain, $w_c$ is crack opening and $L_t$ is characteristic length. More information can be found at ATENA Theory Manual [4]. The characteristic length is a material parameter and it should be equal to the size of the element, in this case 0.015 m (Figure 24). Then, the tensile function describes the evolution of tensile stresses in one finite element after crack formation.

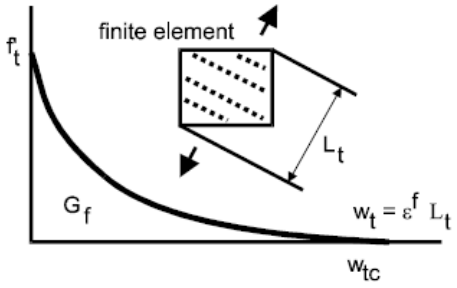



**Figure 23: Tensile softening and characteristic length [4]**

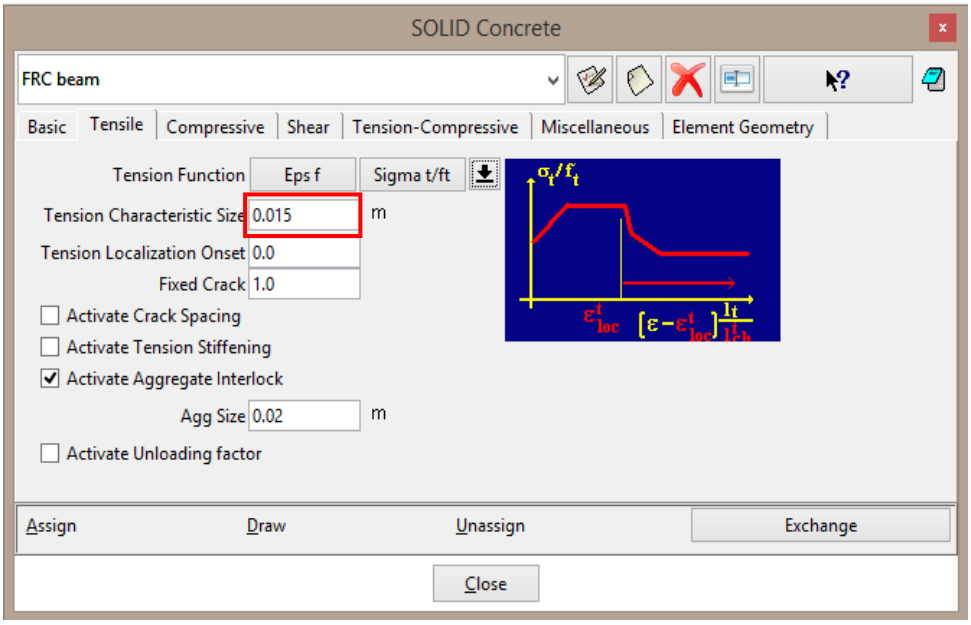



**Figure 24: Modification of tension characteristic length**

In other words, the post-peak behavior and the shape of diagram is governed by tensile function. In the **GiD**, the user can display function by opening the dialog for diagram input and clicking on the button . The results of four point bending tests do not provide information about crack propagation, crack widths and stresses that can be used for the first estimation of the function. Information that can help at this stage is that crack width in FRC during similar tests is usually in millimeters, and based on this fact the order of fracture strain can be at least determined according to the formula (5). Fracture strain corresponding to the crack width 1 mm is 0.07, therefore, the last point of the function, describing the maximal crack opening, is shifted to position [0.1, 0]. It means that initial value of strain is multiplied approximately 10 times. The second point is shifted with the same multiplier, see Figure 26. The horizontal values are maintained because it is not clear now how to modify them.

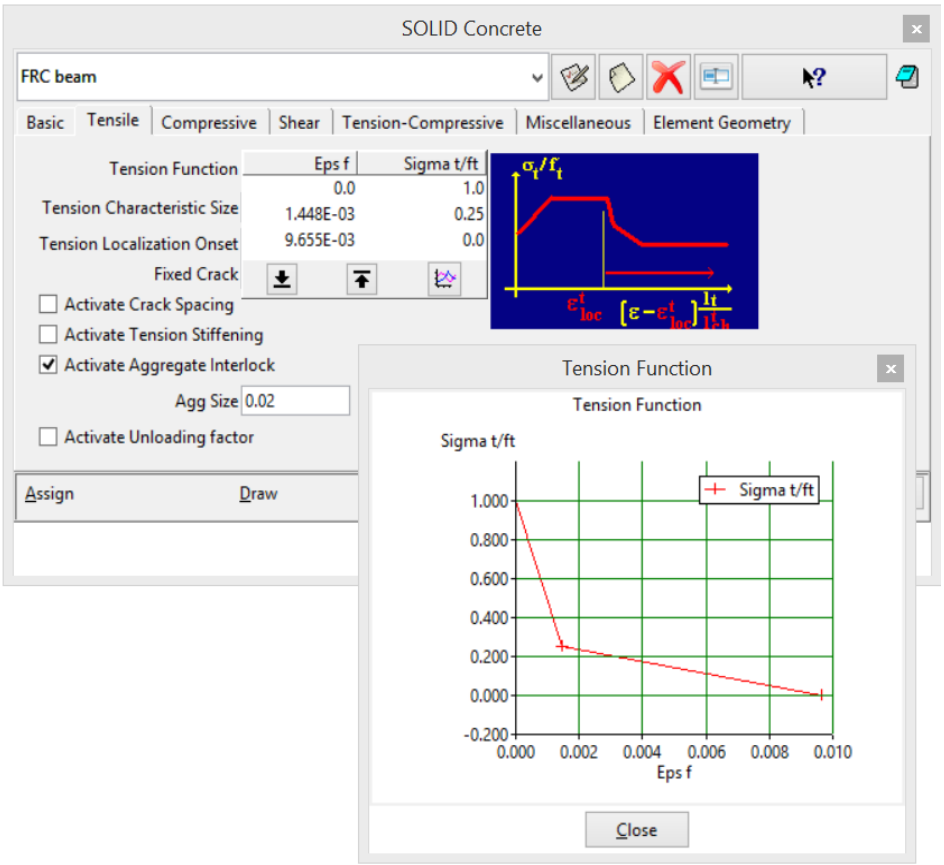



**Figure 25: Default tensile function**

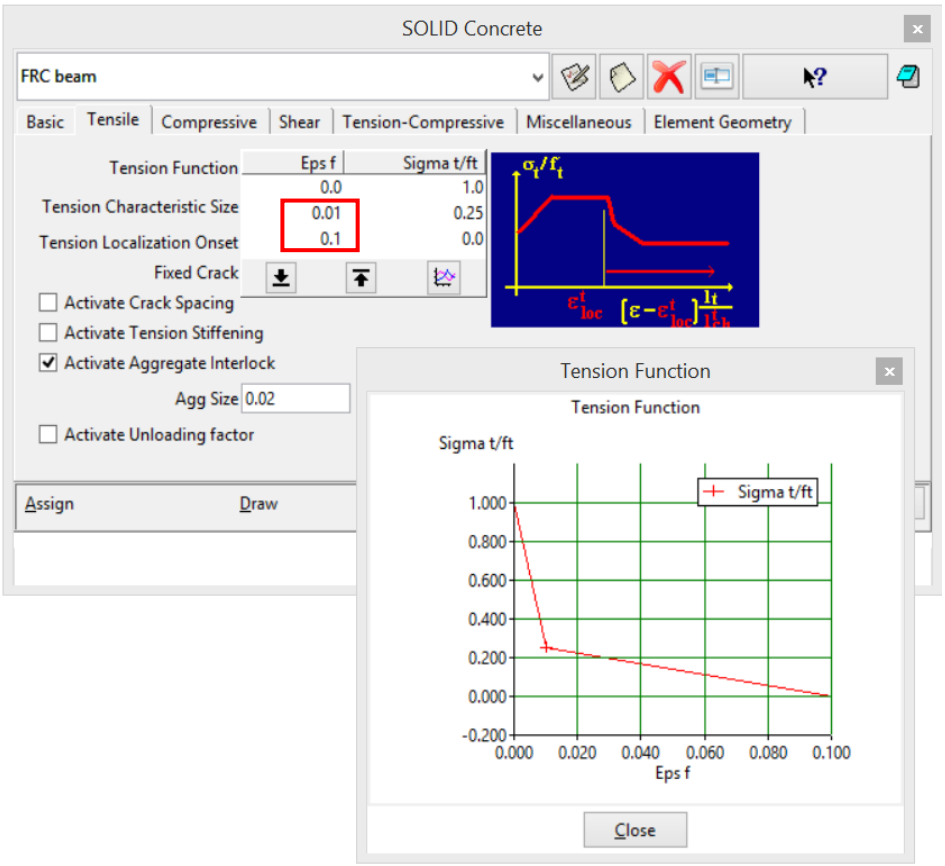



**Figure 26: Modified tensile function**

The last part of the material model that should be adapted to FRC behavior is compression. As it is known that FRC has higher ductility in tension in comparison with ordinary concrete, the compression ductility also increases. In the model **CC3DNonLinCementitious2User**, the ductility in compression is described by compressive function, the default function is depicted in Figure 27. In contrast to the tension, there is no result of compressive test that can help to determine the shape of compression function. Due to this fact, the first point of the diagram is just moved 100 times (the multiplier is based on the developers experience) to increase the ductility, see Figure 28.

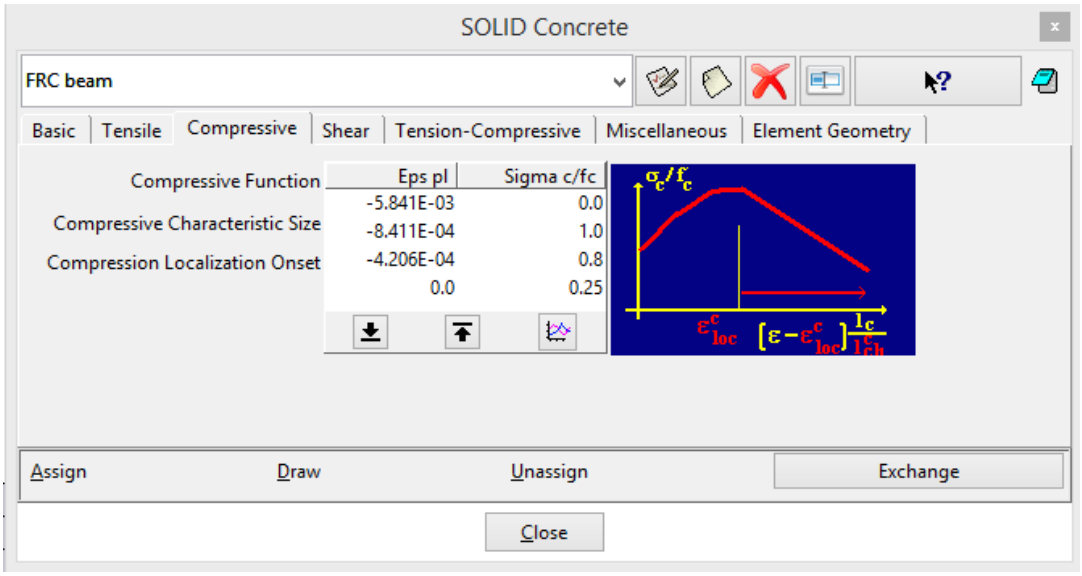



**Figure 27: Default compressive function**

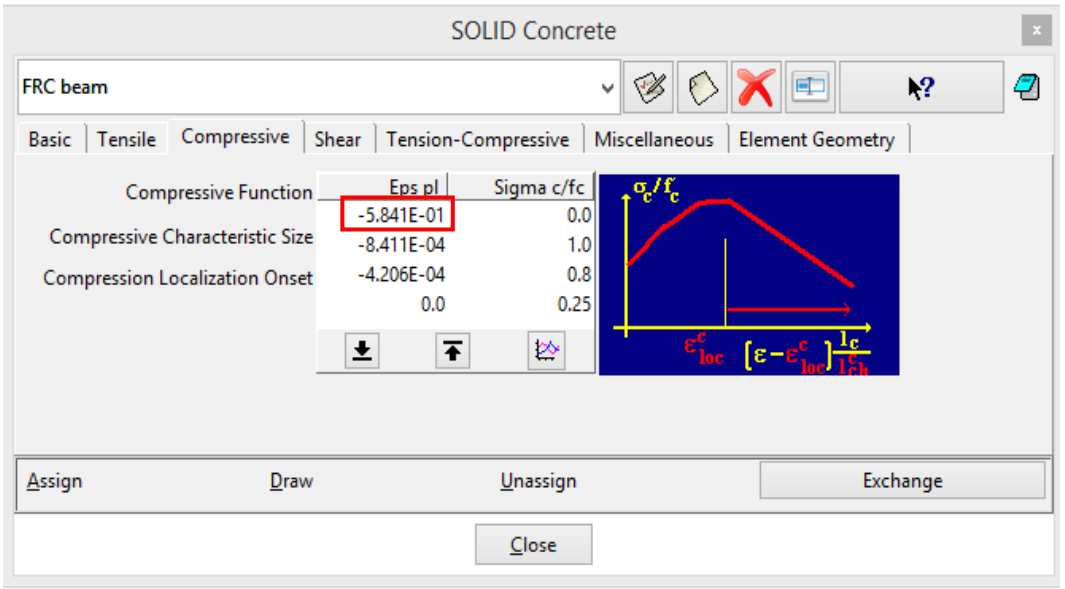



**Figure 28: Modified compressive function**

Second parameter connected with compression is Fc Reduction-COMPRED (reduction of compressive strength due to cracks) which says how the strength is reduced while the material is subjected to lateral tension. In the model **CC3DNonLinCementitious2User**,

the parameter is described by function shown in Figure 29. Based on the developers experience, this reduction should be deactivated by adjusting the function to the constant value 1, see Figure 30.

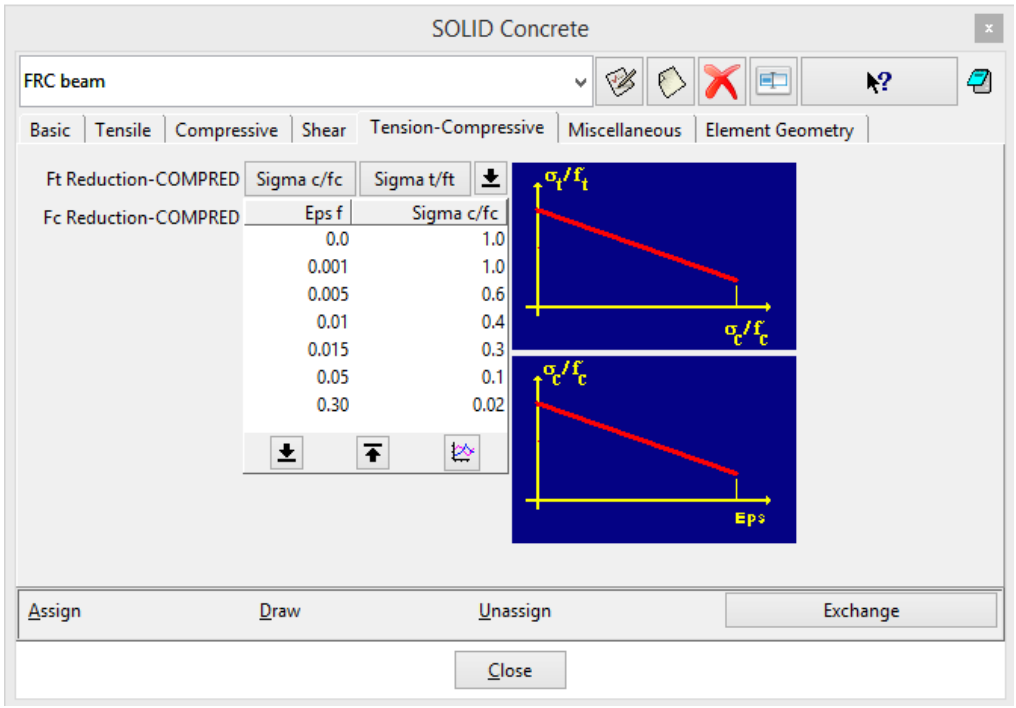



**Figure 29: Default parameter Fc Reduction-COMPRED**

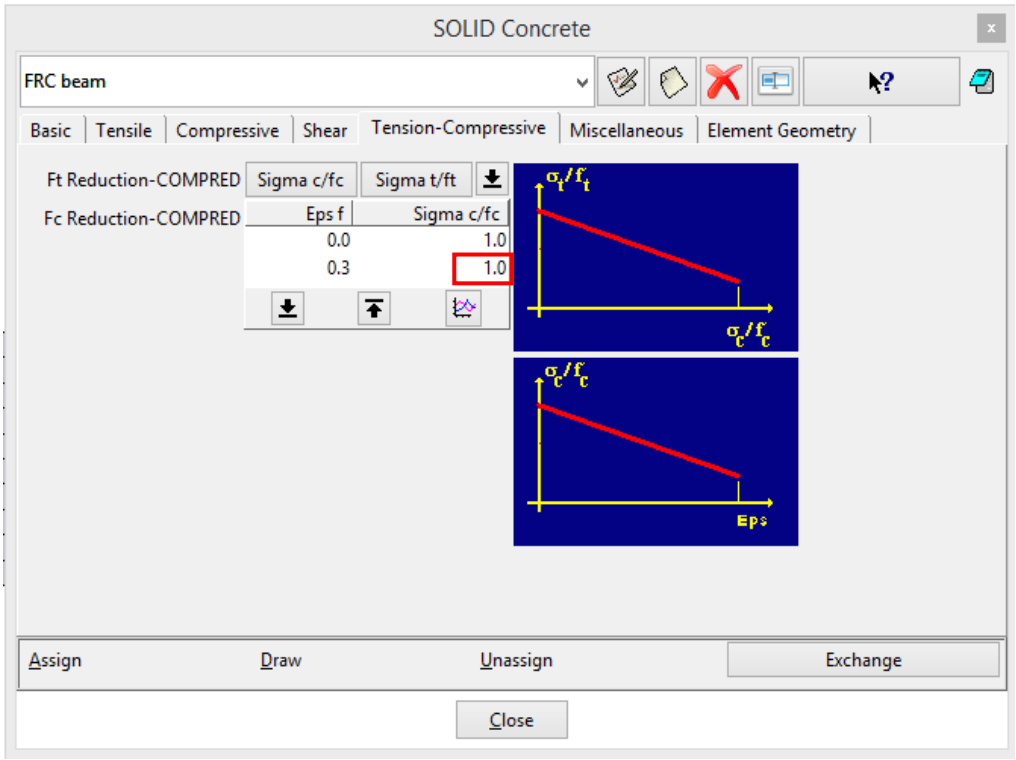



**Figure 30: Modified parameter Fc Reduction-COMPRED**

This is the initial model for the inverse analysis. Based on its results, material parameters will be modified to find the suitable model in accordance with experimental results.

### 4.2.1.2 Running the Analysis

After the material model modifications described in the previous chapter, the finite element analysis is started by clicking on the **Start ATENA static analysis**icon or by the using of command **Calculate | Calculate** or **ATENA | ATENA Analysis**. After selecting this command, the program will start to generate the input file. This process is indicated by the dialog box (see Figure 31). Then, the **ATENA Studio** window appears and analysis is in progress (see Figure 32).

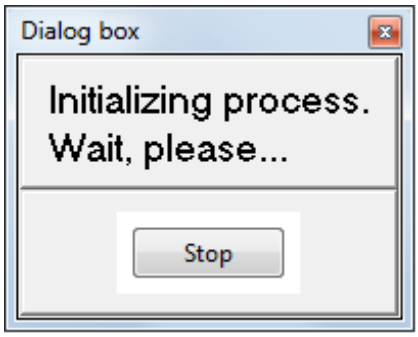



**Figure 31: The initializing of the analysis**

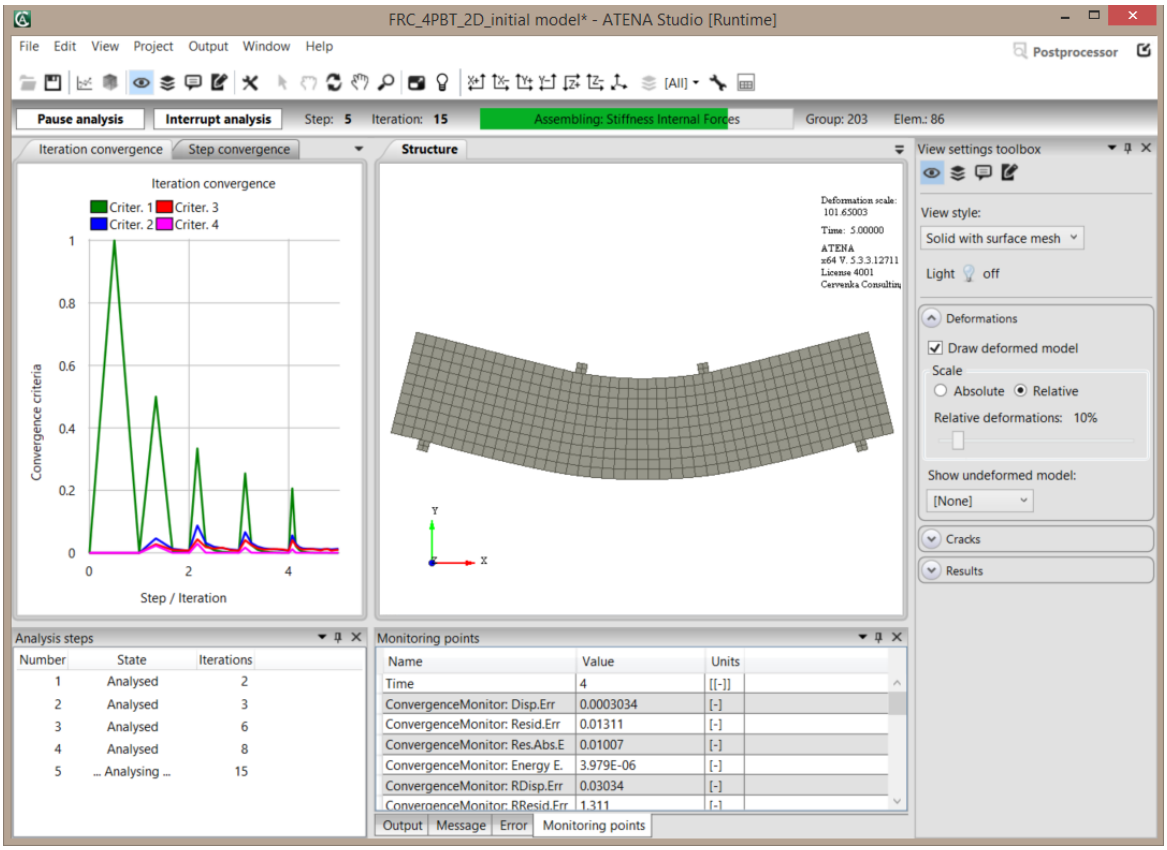



**Figure 32: The ATENA Studio interface window**

### 4.2.1.3 Load-Displacement Diagram and its export from ATENA to Excel File or Similar

During the analysis, it is very useful to see the evolution of the applied load and beam deflections. The progress of the load and deflection is available in the monitors that were defined in the model, see chapter 3.2 and Figure 3.

To visualize these monitors during the nonlinear analysis, new diagram window must be opened by the clicking on the icon . The empty window for the diagram and the diagram settings appears (see Figure 33). The new diagram is defined by diagram settings dialog (see Figure 34 to Figure 36). You may also wish to see the ATENA Studio User's Manual [3] for more options.

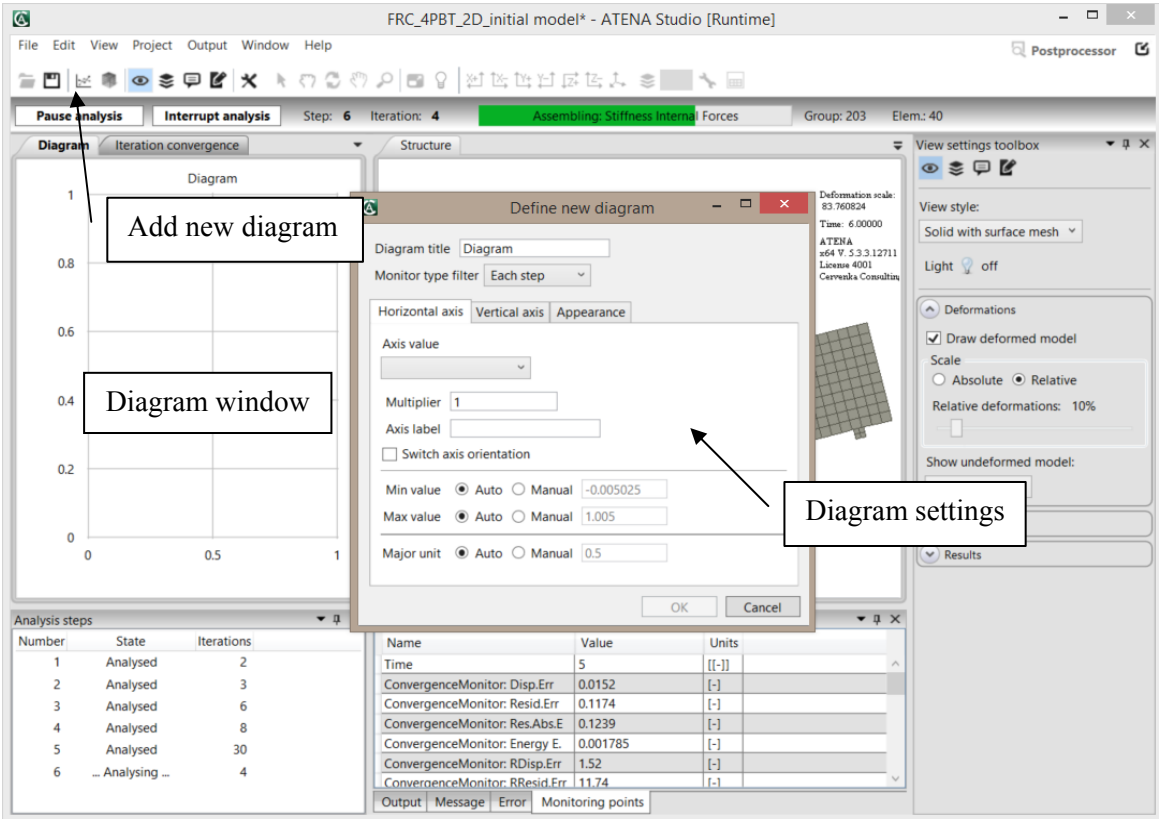



**Figure 33: The execution of the graph**

The diagram title can be **L-D** and the monitor type filter should be **Each step**. For the horizontal value the monitor labelled **displacement_DISPLACEMENTS** should be selected. The name of axis should be **Displacement [mm]** and the values are multiplied by -1000 to achieve switched orientation of the axis and millimeters instead of default meters, see Figure 34.

The vertical axis can display more series. Add new series, choose the **Reaction R_REACTIONS** for the right reaction and **Reaction L_REACTIONS** for the left reaction. Use multiplier 1000 to achieve reactions in the kilonewtons instead of meganewtons. The axis label can be **Reactions [kN]**. The series definition must be applied by the **OK** button above Axis label, see Figure 35 and Figure 36. The definition of the diagram parameters is finished by clicking on the **OK** button. After this, the L-D diagram is shown on the left side of the **ATENA Studio** interface, see Figure 37. This diagram is

showing actual stage of the running analysis and it changes as the analysis progresses based on the current loads and deflections.

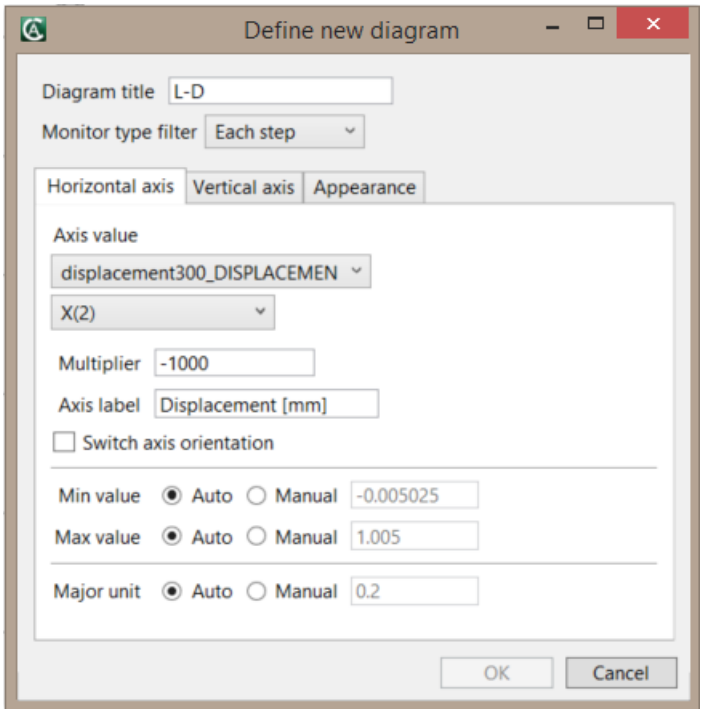



**Figure 34: The diagram definition – properties of the horizontal axis**

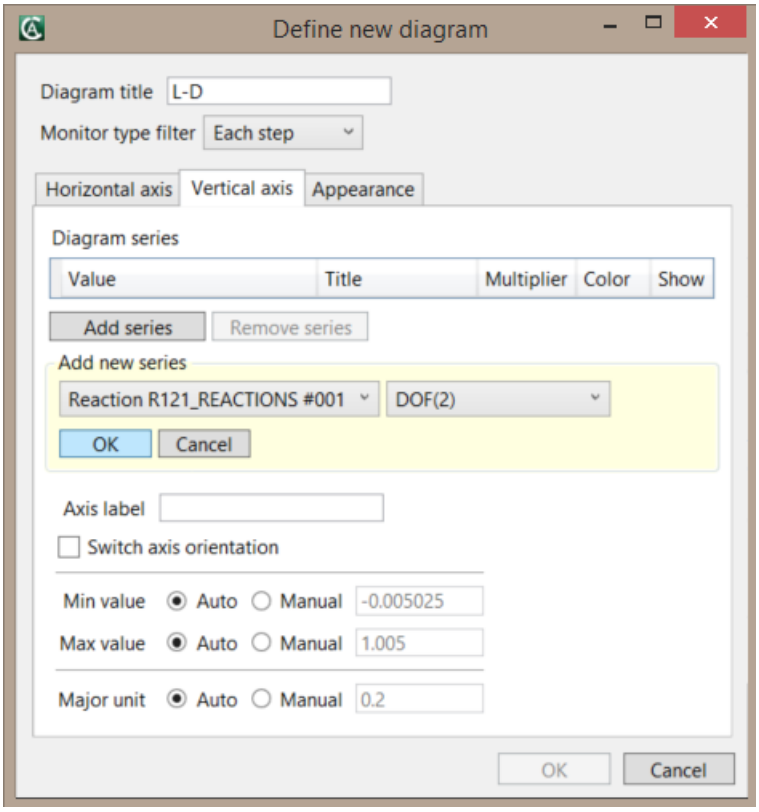



**Figure 35: The diagram definition – selection of the series for vertical axis**

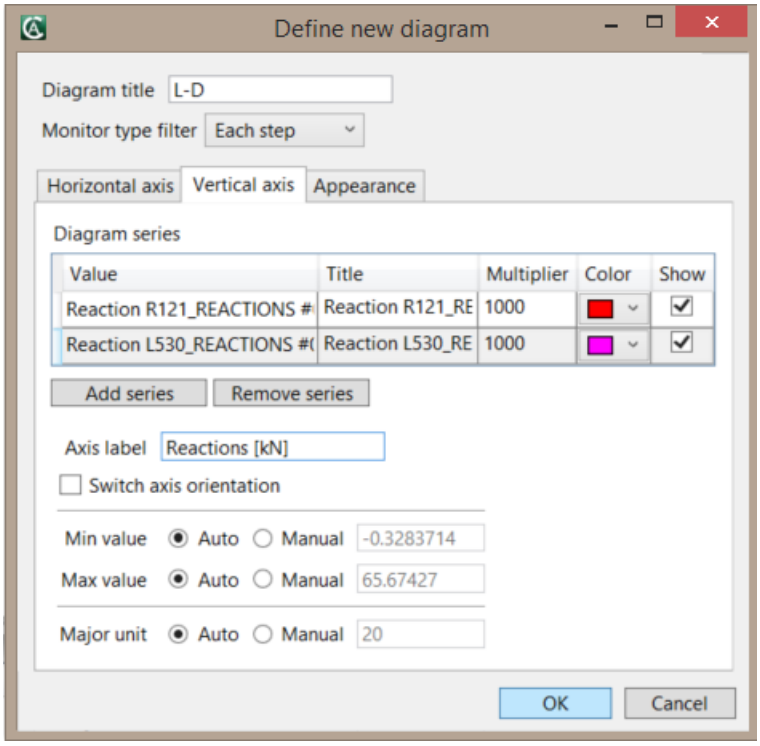



**Figure 36: The diagram definition – properties of the vertical axis**

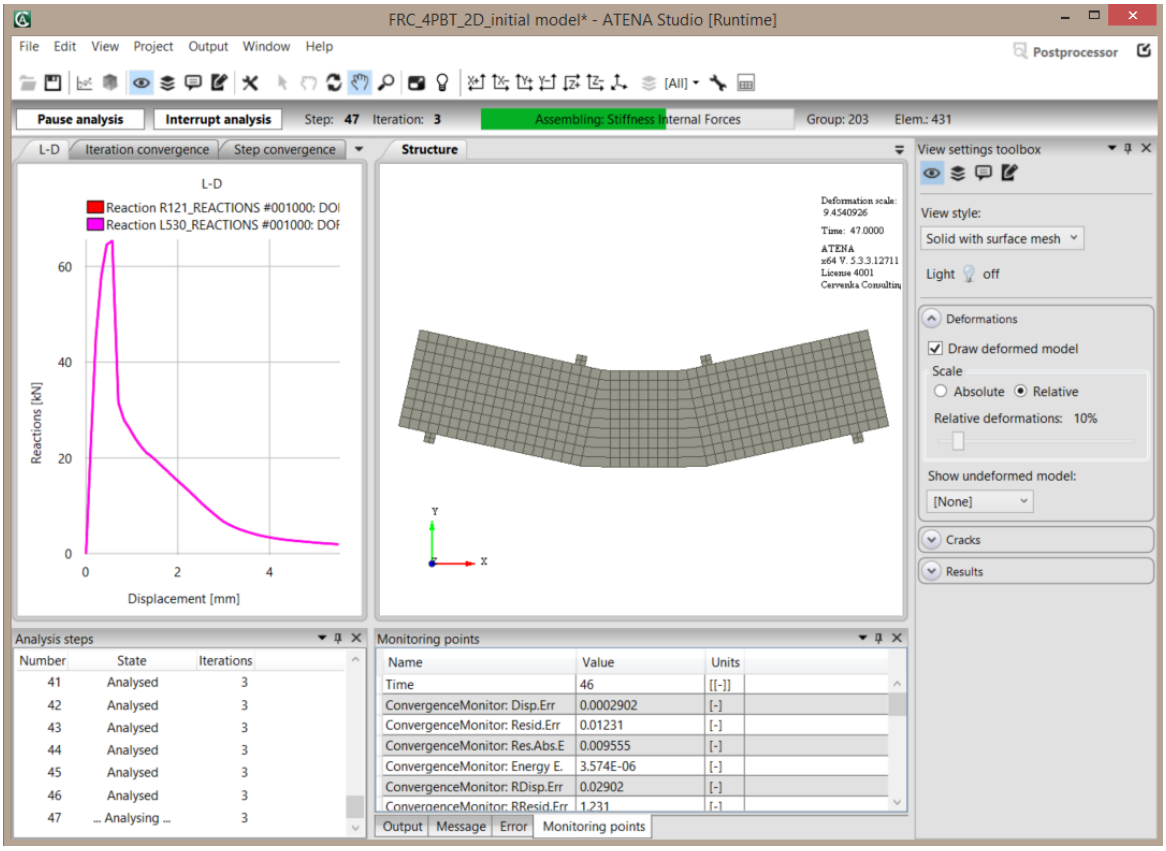



**Figure 37: The L-D diagram showing stage of the running analysis**

At the stage shown in the Figure 37, the calculation can be stopped. It is obvious from the L-D diagram that applied load decreases almost to the zero and the beam fails. The calculation is interrupted by clicking on the button **Interrupt analysis** above the diagrams. The dialog appears and asks the user if the analysis should be stopped after the step, iteration or as soon as possible, see Figure 38. The third option is chosen in this case.

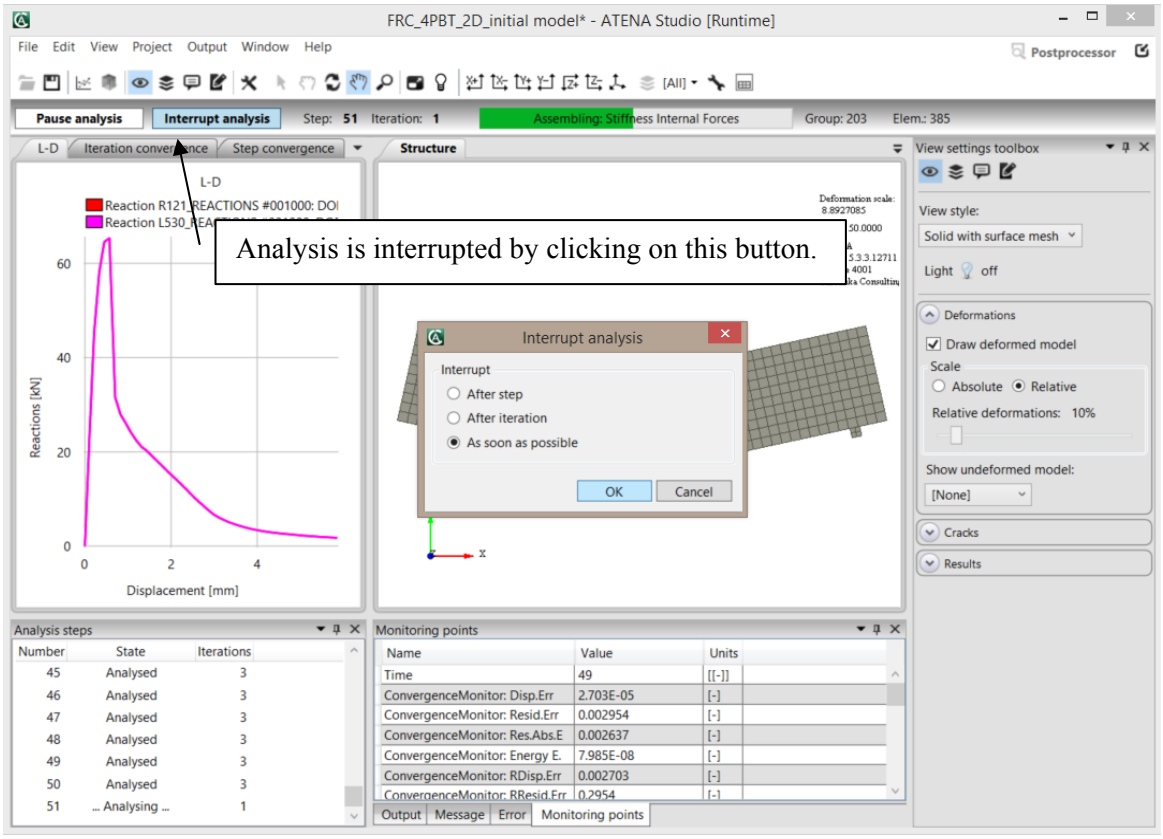



**Figure 38: Dialog for interrupting the analysis**

When the calculation is interrupted, the data from L-D diagram must be copied into the excel file or similar and compared with experimental data. It can be done at the runtime mode, it is not necessary to go to the postprocessor. Click on the window with L-D diagram and use Ctrl-C to copy the data. The dialog shown in Figure 39 appears, copy data by clicking on the OK button.

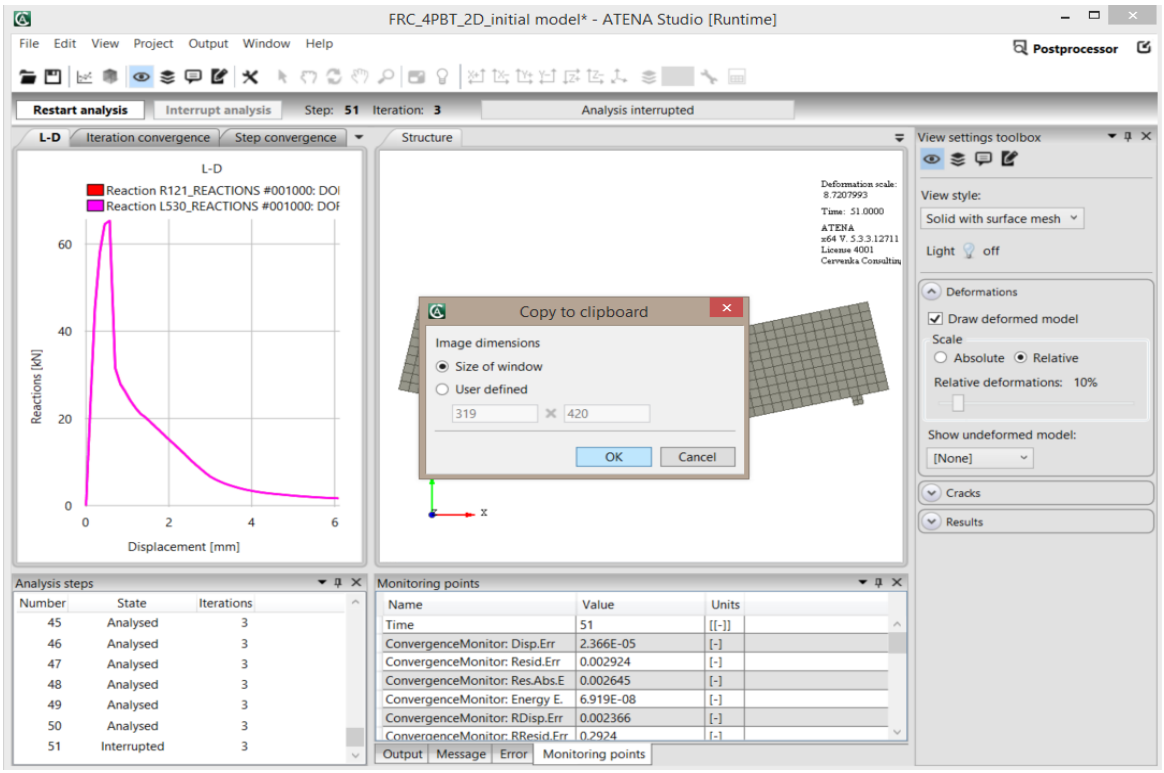



**Figure 39: Dialog for the copy of data in the L-D diagram**

For processing the copied data, **Microsoft Excel** or some similar program needs to be utilized. The Excel file named **LD_diagrams_empty.xlsx** is used in this tutorial and it can be found on following address: **%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.FRC_FRC_user_material_calibration\LD_diagrams_empty.xlsx**. There are three lists, the first one contains data from laboratory tests and relevant L-D diagram (see Figure 40), the second list is prepared for the **ATENA** data and is empty at this moment, the third list serves to compare the results.

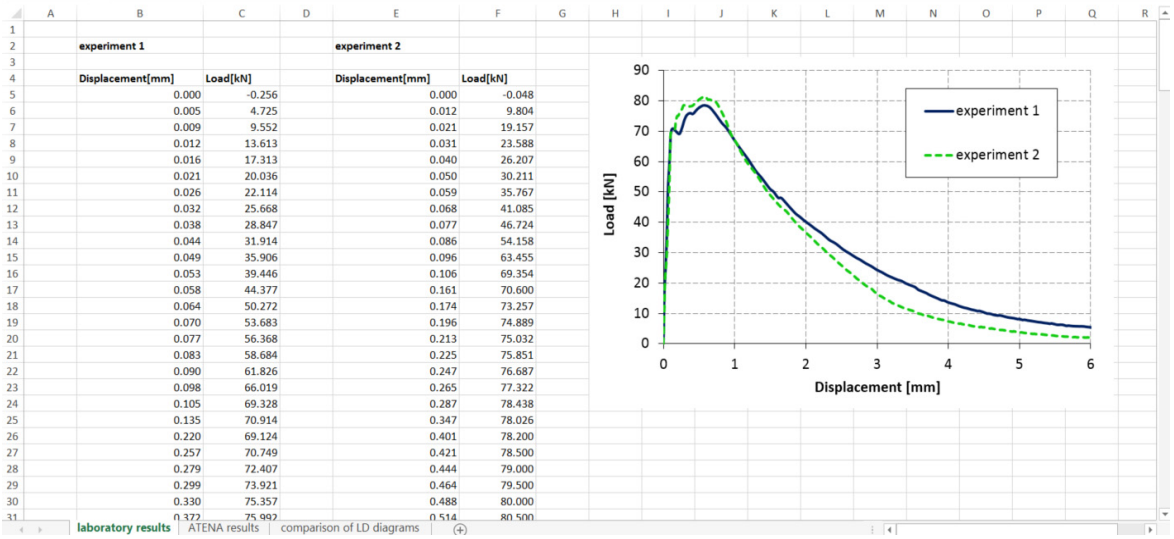
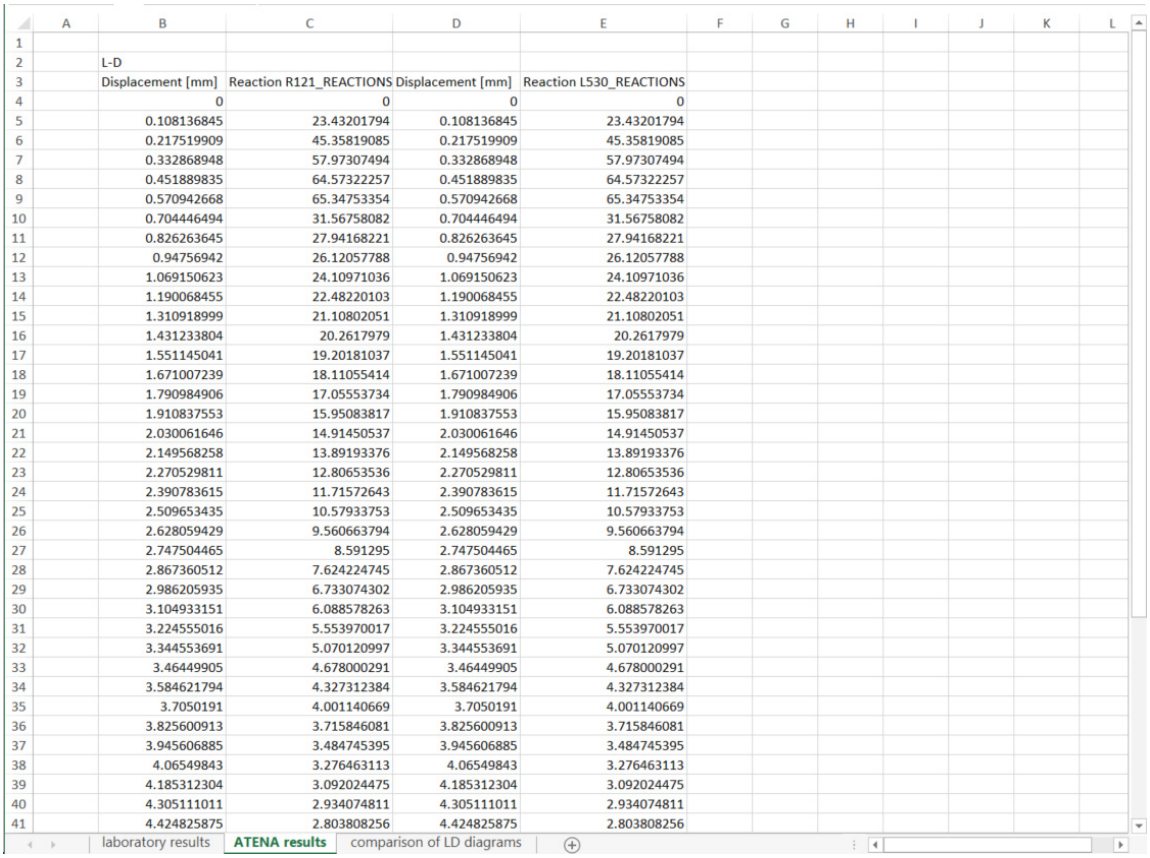



**Figure 40: Excel file for comparison or results – List with the results of laboratory tests**

The **ATENA** data can be inserted on the second list named **ATENA** results by Ctrl-V or by clicking right mouse button and choosing **Insert**. The data are organized in four columns. The first two columns belong to the diagram with the displacement on the horizontal axis and right reaction on the vertical axis. Other two columns belong to the diagram with the same horizontal axis but there is left reaction on the vertical axis, see Figure 41.



| | A | B | C | D | E | F | G | H | I | J | K | L |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | | | | | | | | | | | | |
| 2 | | L-D | | | | | | | | | | |
| 3 | | Displacement [mm] | Reaction R121_REACTIONS | Displacement [mm] | Reaction L530_REACTIONS | | | | | | | |
| 4 | | 0 | 0 | 0 | 0 | | | | | | | |
| 5 | | 0.108136845 | 23.43201794 | 0.108136845 | 23.43201794 | | | | | | | |
| 6 | | 0.217519909 | 45.35819085 | 0.217519909 | 45.35819085 | | | | | | | |
| 7 | | 0.332868948 | 57.97307494 | 0.332868948 | 57.97307494 | | | | | | | |
| 8 | | 0.451889835 | 64.57322257 | 0.451889835 | 64.57322257 | | | | | | | |
| 9 | | 0.570942668 | 65.34753354 | 0.570942668 | 65.34753354 | | | | | | | |
| 10 | | 0.704446494 | 31.56758082 | 0.704446494 | 31.56758082 | | | | | | | |
| 11 | | 0.826263645 | 27.94168221 | 0.826263645 | 27.94168221 | | | | | | | |
| 12 | | 0.94756942 | 26.12057788 | 0.94756942 | 26.12057788 | | | | | | | |
| 13 | | 1.069150623 | 24.10971036 | 1.069150623 | 24.10971036 | | | | | | | |
| 14 | | 1.190068455 | 22.48220103 | 1.190068455 | 22.48220103 | | | | | | | |
| 15 | | 1.310918999 | 21.10802051 | 1.310918999 | 21.10802051 | | | | | | | |
| 16 | | 1.431233804 | 20.2617979 | 1.431233804 | 20.2617979 | | | | | | | |
| 17 | | 1.551145041 | 19.20181037 | 1.551145041 | 19.20181037 | | | | | | | |
| 18 | | 1.671007239 | 18.11055414 | 1.671007239 | 18.11055414 | | | | | | | |
| 19 | | 1.790984906 | 17.05553734 | 1.790984906 | 17.05553734 | | | | | | | |
| 20 | | 1.910837553 | 15.95083817 | 1.910837553 | 15.95083817 | | | | | | | |
| 21 | | 2.030061646 | 14.91450537 | 2.030061646 | 14.91450537 | | | | | | | |
| 22 | | 2.149568258 | 13.89193376 | 2.149568258 | 13.89193376 | | | | | | | |
| 23 | | 2.270529811 | 12.80653536 | 2.270529811 | 12.80653536 | | | | | | | |
| 24 | | 2.390783615 | 11.71572643 | 2.390783615 | 11.71572643 | | | | | | | |
| 25 | | 2.509653435 | 10.57933753 | 2.509653435 | 10.57933753 | | | | | | | |
| 26 | | 2.628059429 | 9.560663794 | 2.628059429 | 9.560663794 | | | | | | | |
| 27 | | 2.747504465 | 8.591295 | 2.747504465 | 8.591295 | | | | | | | |
| 28 | | 2.867360512 | 7.624224745 | 2.867360512 | 7.624224745 | | | | | | | |
| 29 | | 2.986205935 | 6.733074302 | 2.986205935 | 6.733074302 | | | | | | | |
| 30 | | 3.104933151 | 6.088578263 | 3.104933151 | 6.088578263 | | | | | | | |
| 31 | | 3.224555016 | 5.553970017 | 3.224555016 | 5.553970017 | | | | | | | |
| 32 | | 3.344553691 | 5.070120997 | 3.344553691 | 5.070120997 | | | | | | | |
| 33 | | 3.46449905 | 4.678000291 | 3.46449905 | 4.678000291 | | | | | | | |
| 34 | | 3.584621794 | 4.327312384 | 3.584621794 | 4.327312384 | | | | | | | |
| 35 | | 3.7050191 | 4.001140669 | 3.7050191 | 4.001140669 | | | | | | | |
| 36 | | 3.825600913 | 3.715846081 | 3.825600913 | 3.715846081 | | | | | | | |
| 37 | | 3.945606885 | 3.484745395 | 3.945606885 | 3.484745395 | | | | | | | |
| 38 | | 4.06549843 | 3.276463113 | 4.06549843 | 3.276463113 | | | | | | | |
| 39 | | 4.185312304 | 3.092024475 | 4.185312304 | 3.092024475 | | | | | | | |
| 40 | | 4.305111011 | 2.934074811 | 4.305111011 | 2.934074811 | | | | | | | |
| 41 | | 4.424825875 | 2.803808256 | 4.424825875 | 2.803808256 | | | | | | | |

laboratory results | ATENA results | comparison of LD diagrams

**Figure 41: Excel file for comparison or results – List with the data from ATENA**

The diagram comparable with the experimental results must be created from **ATENA** data. Both reactions must be summed up because the load in the experimental results expresses the overall force acting on the beam. Then the data are ready for making the diagram (see Figure 42) that can be put into one graph together with laboratory results and compared (see Figure 43).

| | A | B | C | D | E | F | G | H | I | J |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | | | | | | | | | | |
| 2 | | L-D | | | | | | | | |
| 3 | | Displacement [mm] | Reaction R121_REACTIONS | Displacement [mm] | Reaction L530_REACTIONS | | ummation of reactions [kN] | | | |
| 4 | | 0 | 0 | 0 | 0 | | 0 | | | |
| 5 | | 0.108136845 | 23.43201794 | 0.108136845 | 23.43201794 | | 46.86403588 | | | |
| 6 | | 0.217519909 | 45.35819085 | 0.217519909 | 45.35819085 | | 90.71638169 | | | |
| 7 | | 0.332868948 | 57.97307494 | 0.332868948 | 57.97307494 | | 115.9461499 | | | |
| 8 | | 0.451889835 | 64.57322257 | 0.451889835 | 64.57322257 | | 129.1464451 | | | |
| 9 | | 0.570942668 | 65.34753354 | 0.570942668 | 65.34753354 | | 130.6950671 | | | |
| 10 | | 0.704446494 | 31.56758082 | 0.704446494 | 31.56758082 | | 63.13516165 | | | |
| 11 | | 0.826263645 | 27.94168221 | 0.826263645 | 27.94168221 | | 55.88336442 | | | |
| 12 | | 0.94756942 | 26.12057788 | 0.94756942 | 26.12057788 | | 52.24115576 | | | |
| 13 | | 1.069150623 | 24.10971036 | 1.069150623 | 24.10971036 | | 48.21942072 | | | |
| 14 | | 1.190068455 | 22.48220103 | 1.190068455 | 22.48220103 | | 44.96440206 | | | |
| 15 | | 1.310918999 | 21.10802051 | 1.310918999 | 21.10802051 | | 42.21604102 | | | |
| 16 | | 1.431233804 | 20.2617979 | 1.431233804 | 20.2617979 | | 40.5235958 | | | |
| 17 | | 1.551145041 | 19.20181037 | 1.551145041 | 19.20181037 | | 38.40362074 | | | |
| 18 | | 1.671007239 | 18.11055414 | 1.671007239 | 18.11055414 | | 36.22110828 | | | |
| 19 | | 1.790984906 | 17.05553734 | 1.790984906 | 17.05553734 | | 34.11107468 | | | |
| 20 | | 1.910837553 | 15.95083817 | 1.910837553 | 15.95083817 | | 31.90167634 | | | |
| 21 | | 2.030061646 | 14.91450537 | 2.030061646 | 14.91450537 | | 29.82901074 | | | |
| 22 | | 2.149568258 | 13.89193376 | 2.149568258 | 13.89193376 | | 27.78386752 | | | |
| 23 | | 2.270529811 | 12.80653536 | 2.270529811 | 12.80653536 | | 25.61307072 | | | |
| 24 | | 2.390783615 | 11.71572643 | 2.390783615 | 11.71572643 | | 23.43145287 | | | |
| 25 | | 2.509653435 | 10.57933753 | 2.509653435 | 10.57933753 | | 21.15867506 | | | |
| 26 | | 2.628059429 | 9.560663794 | 2.628059429 | 9.560663794 | | 19.12132759 | | | |
| 27 | | 2.747504465 | 8.591295 | 2.747504465 | 8.591295 | | 17.18259 | | | |
| 28 | | 2.867360512 | 7.624224745 | 2.867360512 | 7.624224745 | | 15.24844949 | | | |
| 29 | | 2.986205935 | 6.733074302 | 2.986205935 | 6.733074302 | | 13.4661486 | | | |
| 30 | | 3.104933151 | 6.088578263 | 3.104933151 | 6.088578263 | | 12.17715653 | | | |
| 31 | | 3.224555016 | 5.553970017 | 3.224555016 | 5.553970017 | | 11.10794003 | | | |
| 32 | | 3.344553691 | 5.070120997 | 3.344553691 | 5.070120997 | | 10.14024199 | | | |
| 33 | | 3.46449905 | 4.678000291 | 3.46449905 | 4.678000291 | | 9.356000582 | | | |
| 34 | | 3.584621794 | 4.327312384 | 3.584621794 | 4.327312384 | | 8.654624768 | | | |
| 35 | | 3.7050191 | 4.001140669 | 3.7050191 | 4.001140669 | | 8.002281338 | | | |
| 36 | | 3.825600913 | 3.715846081 | 3.825600913 | 3.715846081 | | 7.431692161 | | | |
| 37 | | 3.945606885 | 3.484745395 | 3.945606885 | 3.484745395 | | 6.969490789 | | | |
| 38 | | 4.06549843 | 3.276463113 | 4.06549843 | 3.276463113 | | 6.552926227 | | | |
| 39 | | 4.185312304 | 3.092024475 | 4.185312304 | 3.092024475 | | 6.18404895 | | | |
| 40 | | 4.305111011 | 2.934074811 | 4.305111011 | 2.934074811 | | 5.868149622 | | | |
| 41 | | 4.424825875 | 2.803808256 | 4.424825875 | 2.803808256 | | 5.607616512 | | | |

laboratory results | ATENA results | comparison of LD diagrams

**Figure 42: Excel file for comparison or results – Modification of the data from ATENA**

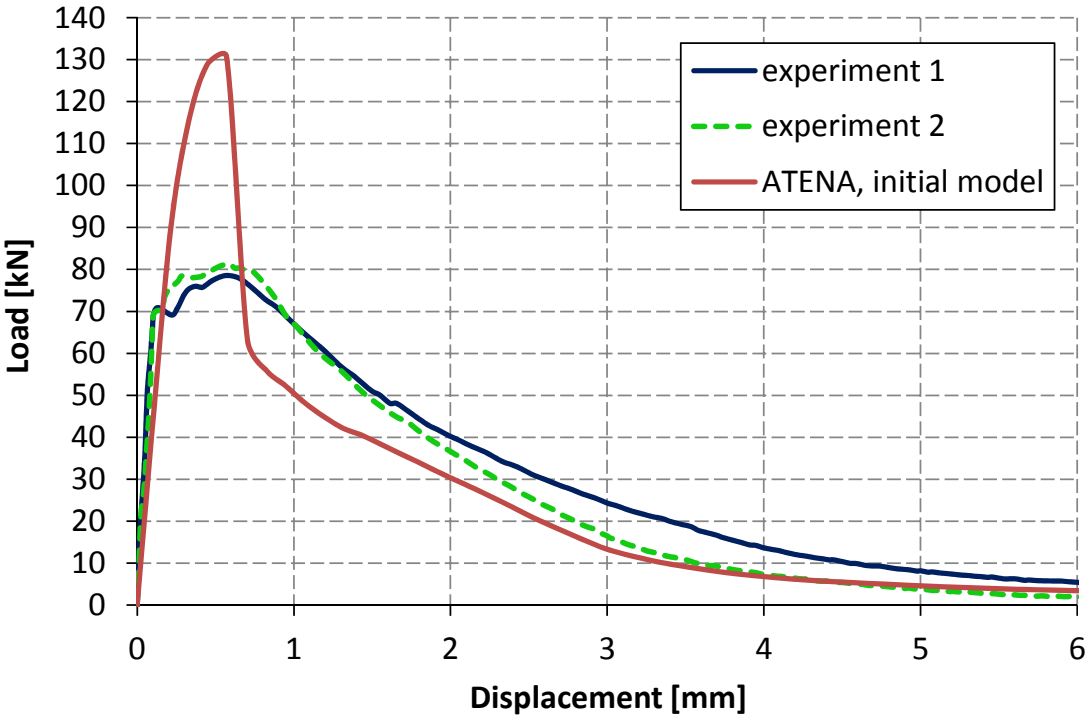



**Figure 43: Comparison of L-D diagrams**

### 4.2.1.4 Analysis of the results and subsequent material modification

It is obvious from the comparison of results in Figure 43 that material model does not describe behavior of FRC properly and needs modifications. As the beam is subjected to bending, the main attention should be paid to the tensile strength and tensile function.

By modification of the tensile strength, the whole L-D diagram moves up and down. The shape of the diagram can be adjusted through the tensile function. The position of the diagram peak is influenced both by the tensile strength and the tensile function, specifically by the slope of the first part of the diagram. The user can choose which parameter to modify to obtain the right position of the peak. Since the value of tensile strength is based on the formulas from previous chapter, the tensile function is adjusted.

It is useful to find fracture strains corresponding to point where the load should be changed. For example, at the peak of the L-D diagram, the displacement is 0.6 mm and the load is 131 kN while it should be 80 kN. The fracture strain at this step can be calculated by formula (5) using the crack width found in the postprocessor, see Figure 44. The fracture strain is 0.003, see Table 2. At this point, the relative stress should be decreased in the same proportion as is between the loads. The original position of the point is [0.003, 0.775] (the vertical coordinate is calculated from the first part of the tensile function, i.e. line beginning at the point [0, 1] and ending at [0.01, 0.25]). The new position of the point is [0.003, 0.473], the relative stress is decreased to 61 % based on the proportion between calculated and required load.

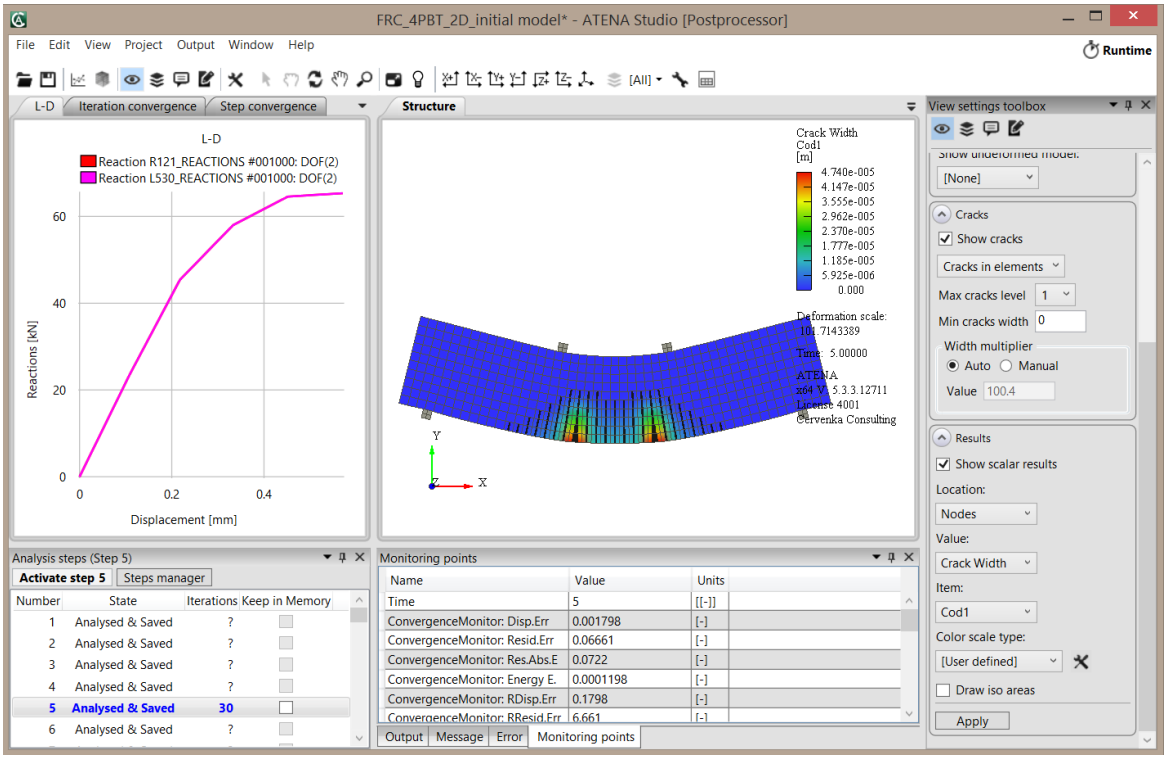



**Figure 44: Crack width in postprocessor**

Contrary to the peak of the diagram, the load corresponding to the displacement 1, 2, 3 and 4 mm should be increased by approximately 30 %. In the same proportion, the relative stresses in the tensile function are increased. The corresponding fracture strains are summarized in the Table 2. According to the fracture strain at the displacement 6 mm, the maximal fracture strain in the tensile function is shifted from position [0.1, 0] to [0.15, 0]. Modified tensile function is depicted in Figure 45. Model with described modifications can be found at the location: **%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.FRC_FRC_user_material_calibration\FRC_4PBT_2D_modification_1.gid**.

**Table 2: Fracture strain corresponding to different displacements in the beam analysis (initial model)**

| Displacement [mm] | Load [kN] | Step [-] | Crack width [mm] | Fracture strain [-] |
|---|---|---|---|---|
| 0.6 | 131 | 5 | 0.047 | 0.0031 |
| 1 | 52 | 8 | 0.229 | 0.0153 |
| 2 | 30 | 17 | 0.610 | 0.0407 |
| 3 | 13 | 25 | 0.988 | 0.0659 |
| 4 | 7 | 33 | 1.400 | 0.0933 |
| 6 | 3 | 50 | 2.200 | 0.1467 |

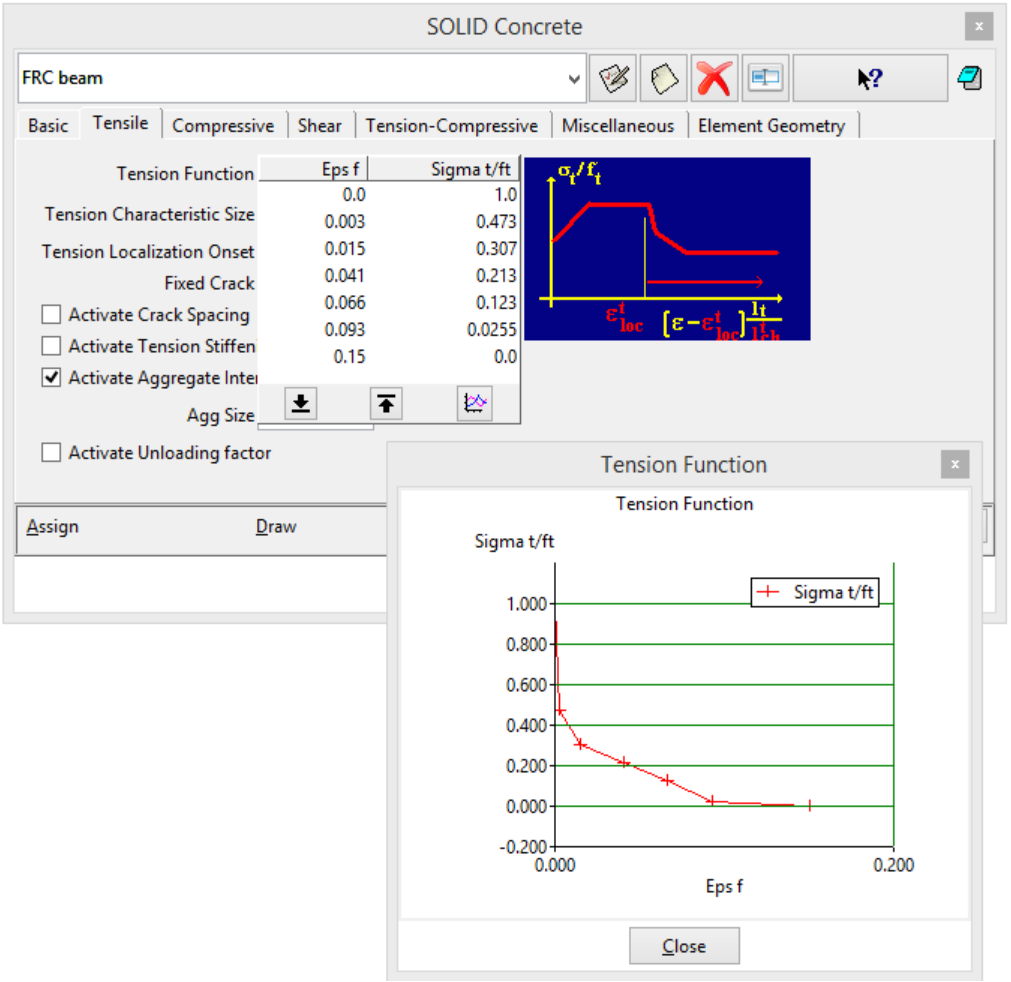



**Figure 45: Modification of the tensile function no. 1**

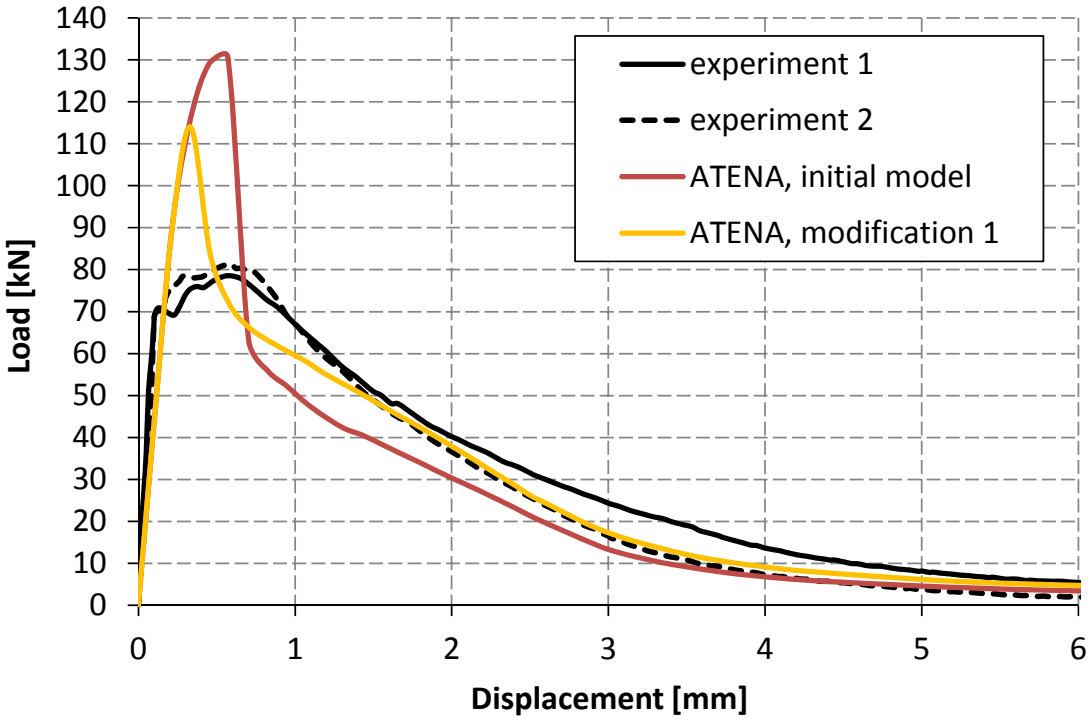



**Figure 46: Comparison of L-D diagrams with new result for modification no. 1**

Figure 46 shows that peak of the diagram decreases in comparison with the initial model but it still need further modifications. The number of subsequent modifications depends on the required accuracy of the material model. It is recommended to the user to do next modifications by himself to find the way how the model works and how the different adjustments affect the model behavior. The procedure of model determination done by a developer is presented in the following part of the tutorial to help users as far as they do not know how to find the appropriate parameters.

Description of the modifications to find the suitable model:

- **Modification no. 2**

  The result with modified curve no. 1 should be improved in the area around the maximum load (load is higher than in the experiment) and at the displacements 1, 3 and 4 mm (load is lower). The fracture strain corresponding to diagram peak is 0.0005, see Table 3, and at this point, the relative stress should be lower. Thus, the second point of the tensile diagram which original position was [0.003, 0.473] is shifted to [0.0005, 0.473], see Figure 47.

  To find appropriate modifications around the displacements of 1, 3 and 4 mm, the fracture strains from Table 3 are utilized again. At the point corresponding to the displacement of 1 mm, the relative stress should be increased by approximately 14 % (it is defined by the relation between loads: the load is 58 kN and it should be 66 kN). The original point [0.015, 0.307] is shifted to position [0.015, 0.349].

  The same procedure is done with tensile function in the area around displacement 3 and 4 mm, see Figure 47, the relative stresses at the corresponding fracture strains are increased.

Model with described modifications can be found at the location: **%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.FRC_FRC_user_material_calibration\FRC_4PBT_2D_modification_2.gid**.

**Table 3: Fracture strain corresponding to different displacements in the beam analysis (modification no. 1)**

| Displacement [mm] | Load [kN] | Step [-] | Crack width [mm] | Fracture strain [-] |
|---|---|---|---|---|
| 0.33 | 114 | 3 | 0.00748 | 0.0005 |
| 1 | 60 | 8 | 0.227 | 0.0151 |
| 3 | 17 | 25 | 0.981 | 0.0654 |
| 4 | 9 | 33 | 1.370 | 0.0913 |

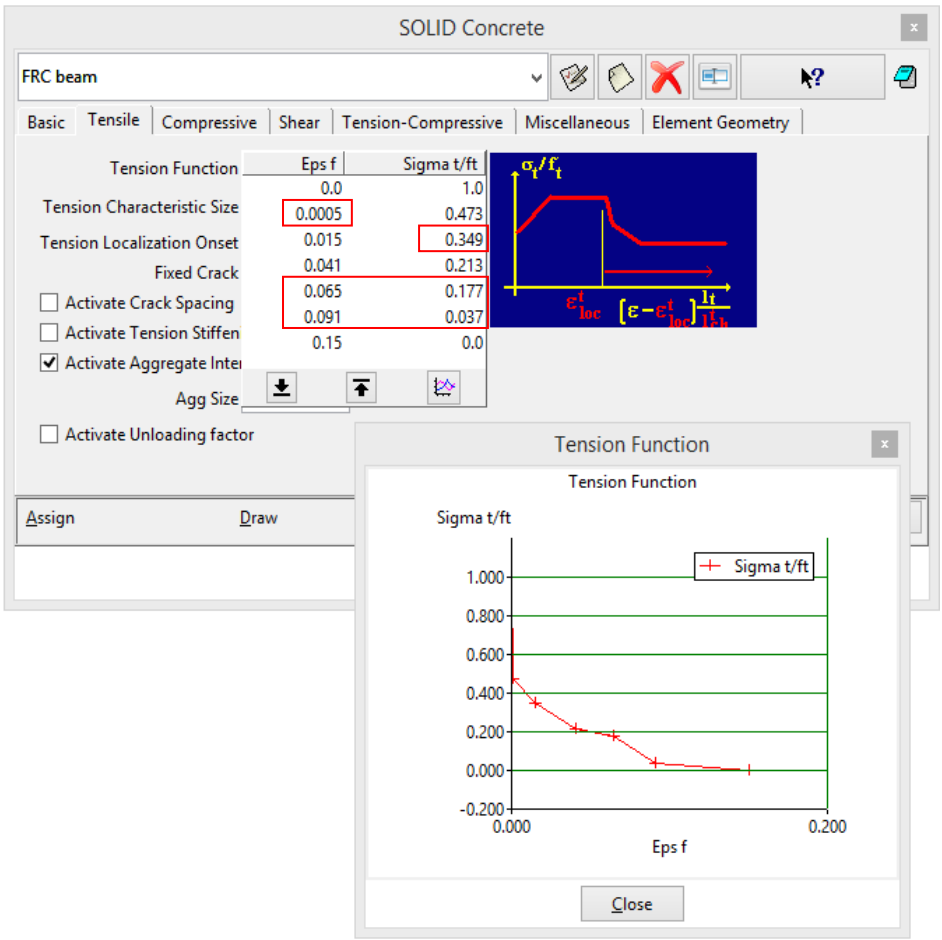



**Figure 47: Modification of the tensile function no. 2**

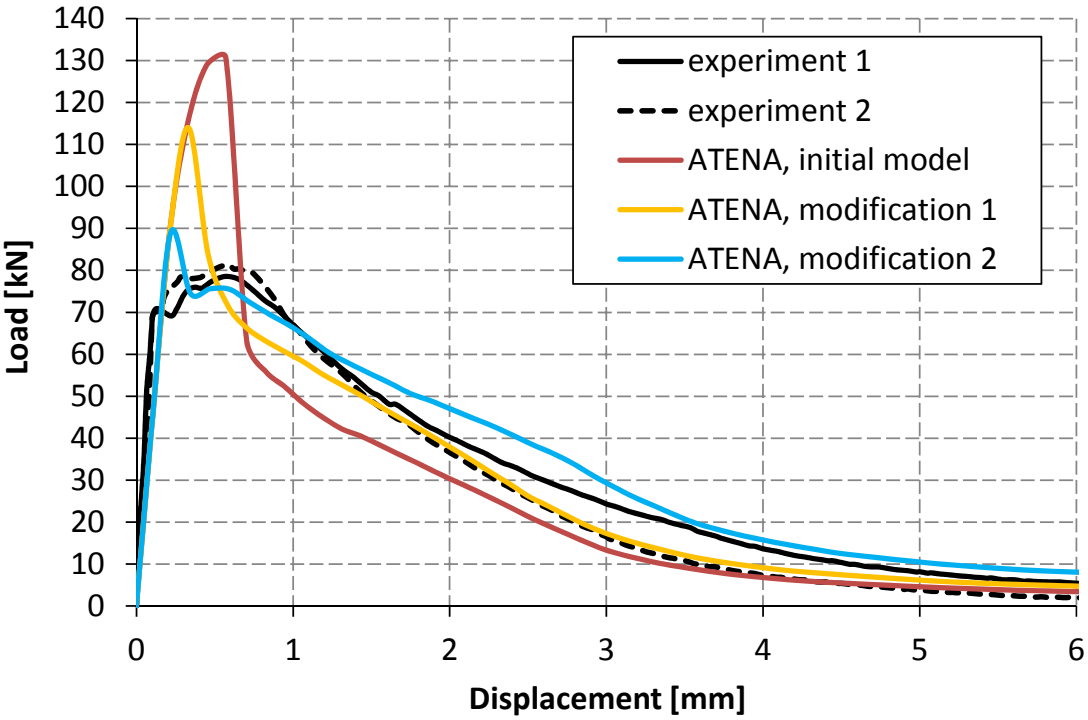



**Figure 48: Comparison of L-D diagrams with new result for modification no. 2**

- **Modification no. 3**

  The result with modified curve no. 2 is shown in the Figure 48. The diagram peak is still higher than it is required. The load at the displacement 2 and 3 mm should be also lower. The fracture strains corresponding to these displacements are summarized in

Table 4.

The fracture strain corresponding to diagram peak is 0.0001 and the relative stress should be lower at this point. Thus, the second point of the tensile diagram which original position was [0.0005, 0.473] is shifted to [0.0001, 0.473], see Figure 48. The diagram can be also improved around the displacement of 0.5 mm (fracture strain 0.005). The load should be similar to the peak load at this area. For that reason, the new point is added to the tensile function and its coordinates are [0.005, 0.473].

At the fracture strains corresponding to the displacement of 2 and 3 mm, the relative stresses are decreased. The same procedure as during the previous modification is done and relative stresses are adjusted in the same proportion as is between loads, see Figure 49.

Model with described modifications can be found at the location: **%Public%\Documents\ATENA Examples\Science\GiD\Tutorial.FRC_FRC_user_material_calibration\FRC_4PBT_2D_modification_3.gid**.

**Table 4: Fracture strain corresponding to different displacements in the beam analysis (modification no. 2)**

| Displacement [mm] | Load [kN] | Step [-] | Crack width [mm] | Fracture strain [-] |
|---|---|---|---|---|
| 0.22 | 90 | 2 | 0.00209 | 0.0001 |
| 0.5 | 76 | 4 | 0.078 | 0.0052 |
| 2 | 47 | 16 | 0.547 | 0.0365 |
| 3 | 28 | 24 | 0.912 | 0.0608 |

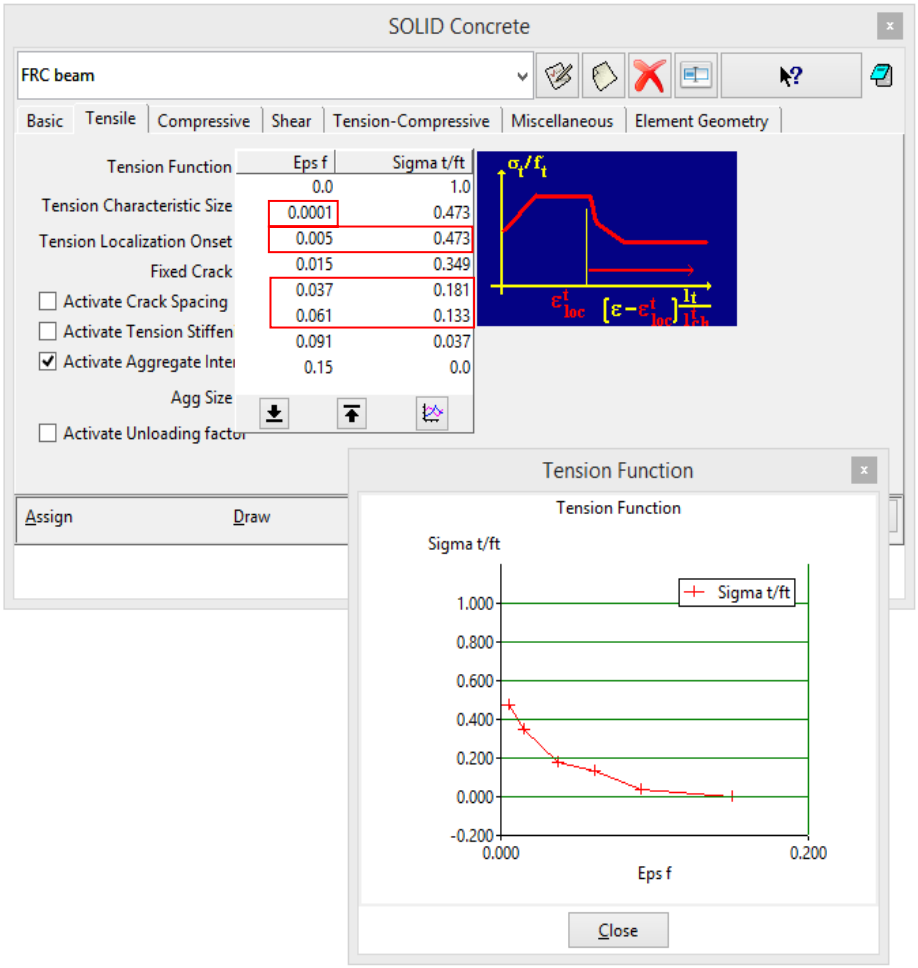



**Figure 49: Modification of the tensile function no. 3**

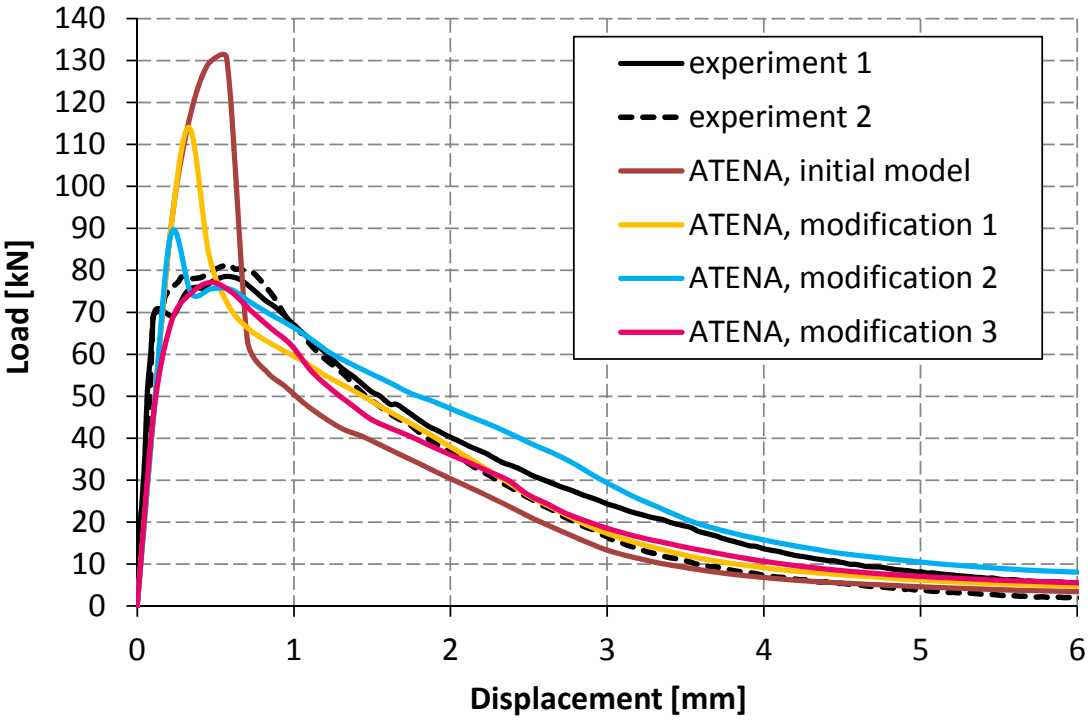



**Figure 50: Comparison of L-D diagrams with new result for modification no. 3**

- **Modification no. 4**

  As the diagram in the Figure 50 shows, the L-D diagram with modified tensile function no. 3 described the FRC behavior very well. It is possible to accept this result as satisfactory or to slightly increase the peak load and load around displacement 1 to 3 mm to obtain more accurate response.

  The relative stress at the peak points (fracture strains 0.0001 and 0.005) is increased by 4 %. Finally, the relative stress corresponding to displacement 1, 2 and 3 mm is also increased in the proportion that is between calculated and required load, see Figure 51.

  The result of modification no. 4 is shown in Figure 52. The behavior of the FRC beam in four point bending test is in accordance with the experiments and this model can be designated as a final model. For better illustration, the final model is shown in comparison with test results without the other models in Figure 53.

  Final model can be found at the location: **%Public%\ Documents\ATENA Examples\Science\GiD\Tutorial.FRC_FRC_user_material_calibration\FRC_4PBT_2D_modification_4.gid**.

**Table 5: Fracture strain corresponding to different displacements in the beam analysis (modification no. 3)**

| Displacement [mm] | Load [kN] | Step [-] | Crack width [mm] | Fracture strain [-] |
|---|---|---|---|---|
| 1 | 62 | 8 | 0.224 | 0.0149 |
| 2 | 36 | 16 | 0.559 | 0.0373 |
| 3 | 18 | 24 | 0.927 | 0.0618 |

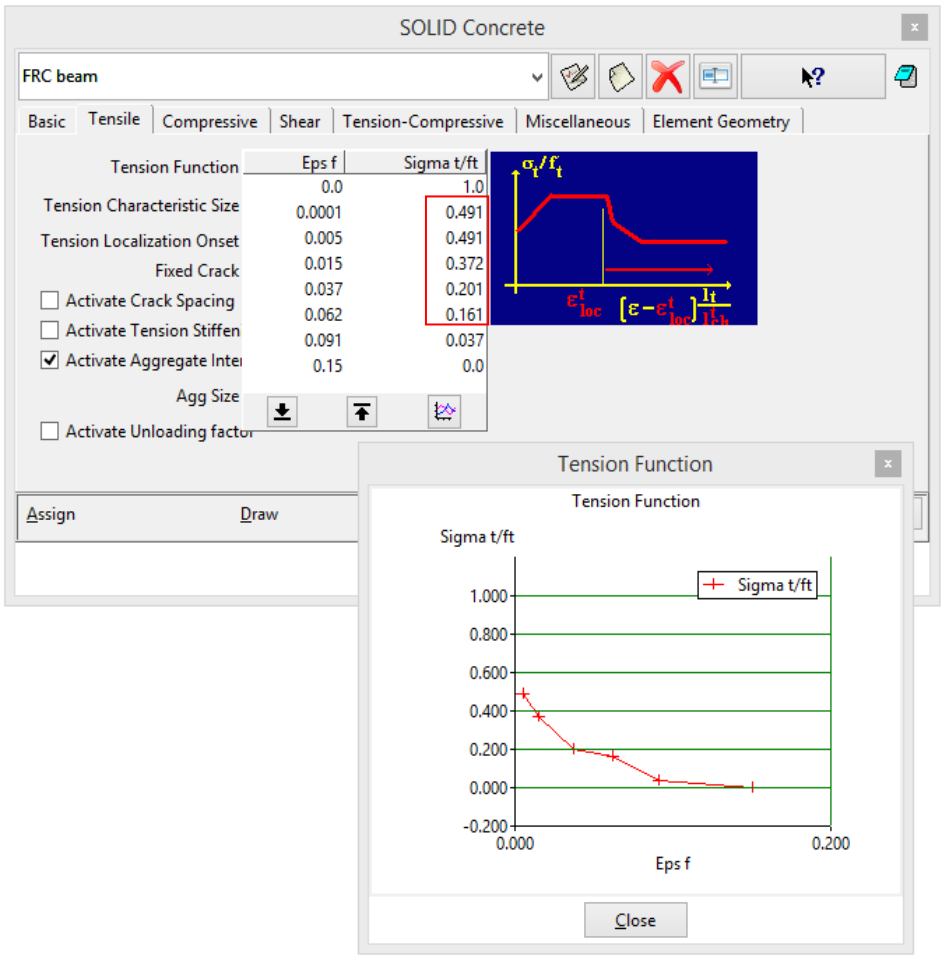



**Figure 51: Modification of the tensile function no. 4**

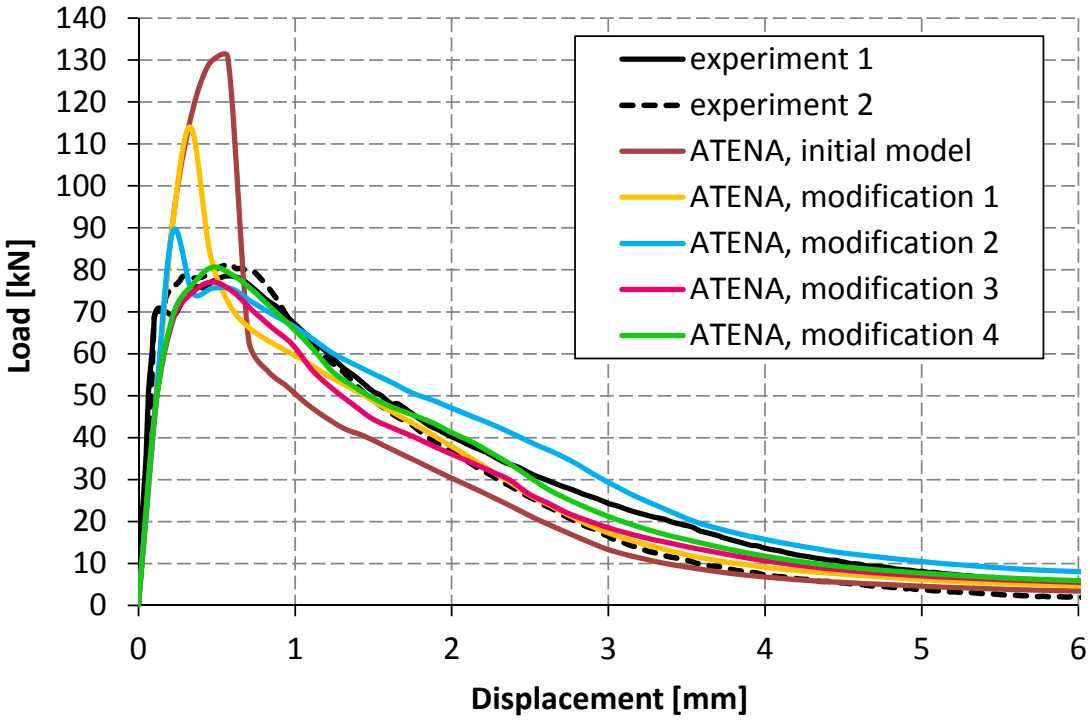



**Figure 52: Comparison of L-D diagrams with new result for modification no. 4**

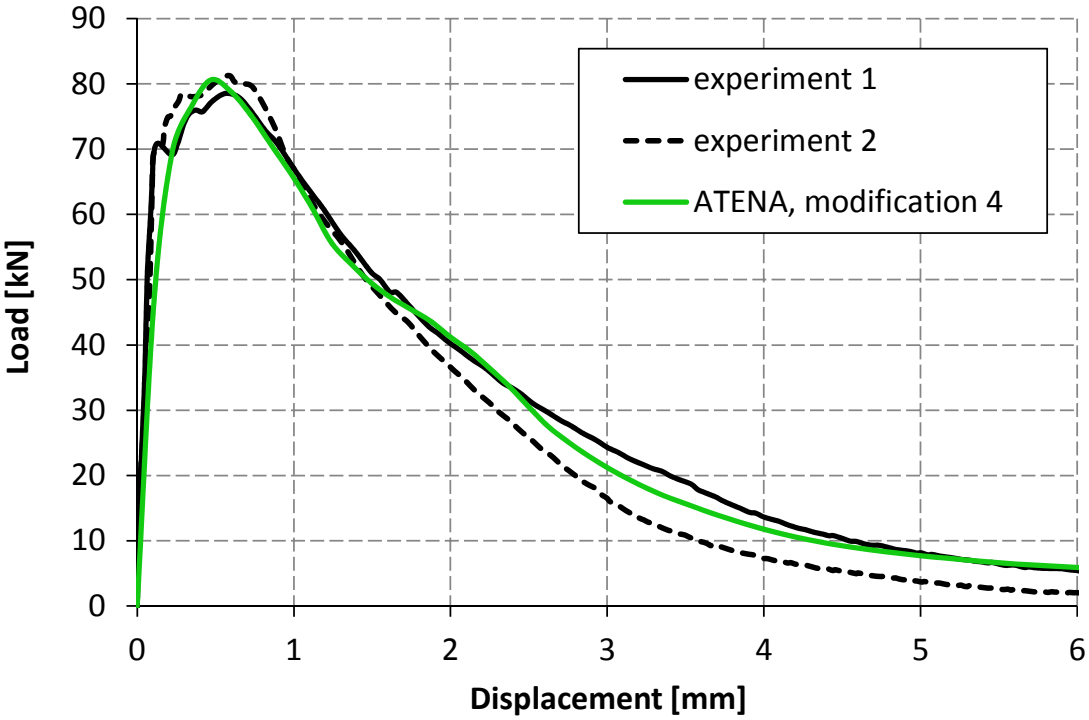



**Figure 53: Load-displacement diagram with the final result**

#### 4.2.1.5 General Instructions for Inverse Analysis

During the inverse analysis described in the previous chapter, the user should learn how to work with the material model and how modification of each parameter affects the results. To conclude this section of tutorial, it is possible to say that important parameters for determination of FRC model are (in case of the model **CC3DNonLinCementitious2User**):

- tensile strength,
- tension function,
- compressive function,
- Fc Reduction – COMPRED.

By adjusting tensile strength, the whole L-D diagram moves up and down. If the shape of the diagram needs to be modified, the tension function describing the ductility must be modified. For higher ductility in compression, the compressive function should be modified. The last parameter is Fc Reduction – COMPRED which says how the strength is reduced while the material is subjected to lateral tension. Based on developer's experience, it is useful to deactivate this parameter and suppose that for FRC the compressive strength do not decrease due to the lateral tension.

For more information about the material model and **ATENA** program, please read the documentation files ([1], [2], [4], [4] and[5]) or contact the support (following ATENA Troubleshooting [5], 2.1.1).

### 4.2.2 **Inverse Analysis by Consoft**

This section is under construction and will be completed as soon as possible.

# 5. APPLICATION OF FRC MATERIAL MODEL

Current significant application of FRC material in the Czech Republic is segmental tunnel lining installed by TBM (tunnel boring machine) during tunnel excavation. Utilization of FRC with steel fibers for segmental tunnel lining promises potential advantages in comparison to the traditionally reinforced concrete structures - efficient manufacturing, lower risk of corrosion, less damage during transport, etc. Červenka Consulting took a part in the project designing precast tunnel segments used in the underground and railway tunnels, see [9] and [10]. Some of the obtained results are presented in this section as a practical example of FRC material model application.

## *5.1 Vault Bending Tests*

Vault bending tests were performed on the whole segments or on a part of the segments. Subject of this experiment was to clarify behavior of the segment under local lateral load. If the laboratory results are known before the numerical modelling, they can be used for inverse analysis instead of three or four point bending tests on beams, described in the previous chapters, in case these tests are not available. The behavior of the segment is similar to the three point bending test.

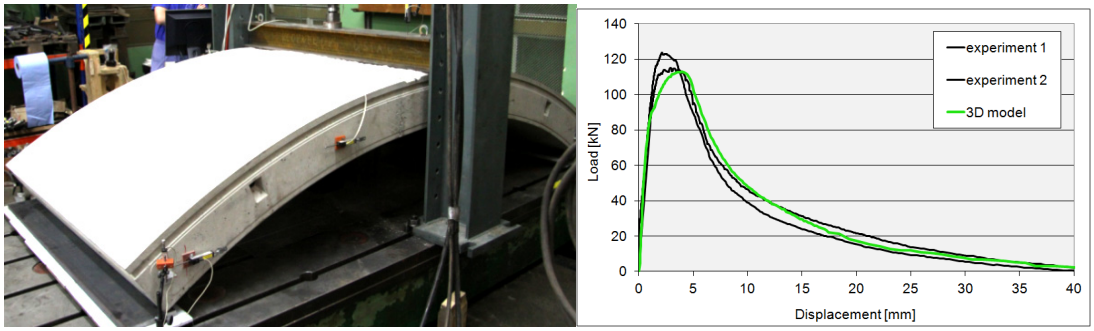



**Figure 54: Vault bending test of the whole segment – laboratory test (left), comparison of L-D diagrams of laboratory tests and 3D model**

Development of bending cracks in the model is very close to the structural behavior in experiment. Comparison of the final crack in the experiment and the model is shown in Figure 55; the significant cracking areas are marked. The cracks in both experiment and numerical analysis are here strongly localized even in the SFRC material.

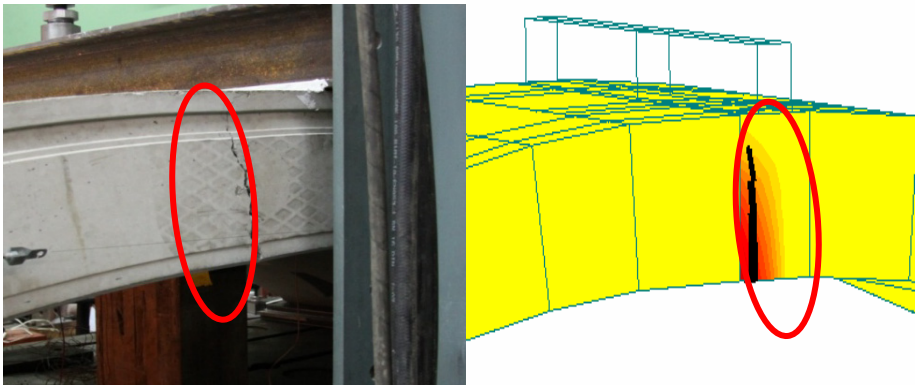


**Figure 55: Cracks in SFRC segment – vault bending: experiment (left), numerical simulation (right)**

Another vault bending test is presented in Figure 56 and it is performed just on part of the segment. In this case, the experimental results were used for inverse analysis to find appropriate parameters of FRC material model.

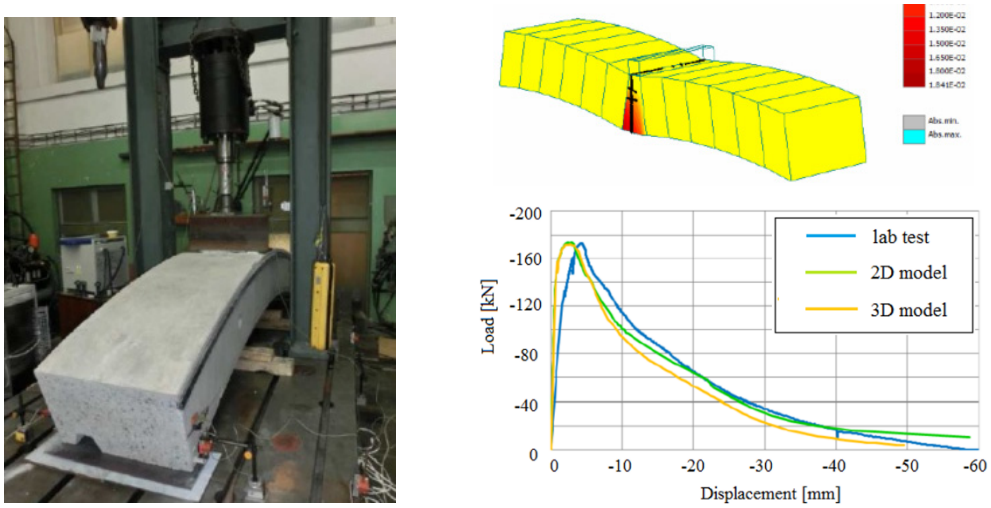



**Figure 56: Vault bending test in lab (left), failure of numerical model (top right), load-displacement diagram of test and models (bottom right)**

## *5.2 Model of Segment (Keystone) in Compression*

The next analysis contains keystone segment subjected to compression test. The keystone is loaded by concentrated pressure in the central part according to conditions during installation of segments by TBM. Comparison of results and failure mode between the laboratory test and the model (see Figure 57) confirmed that the FRC model works in accordance with the reality.

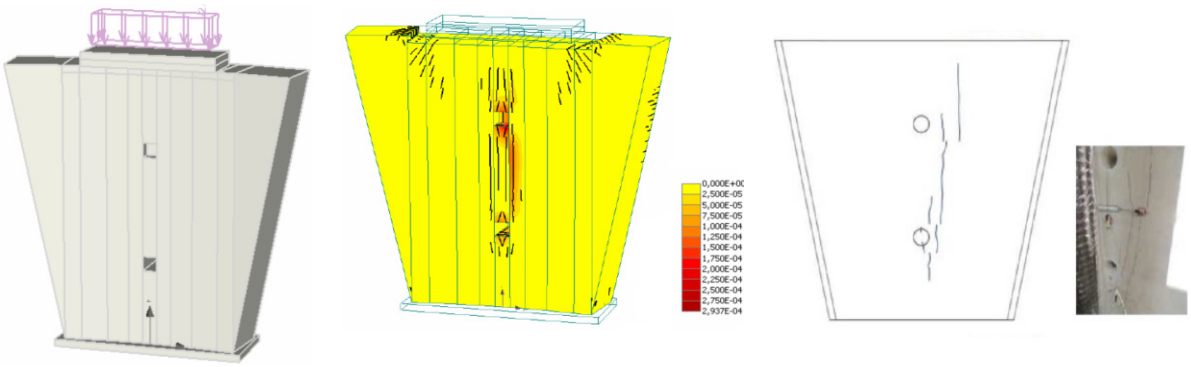


**Figure 57: Model of keystone, crack pattern in the model and results of laboratory test**

# 6. CONCLUSION

This tutorial provides a step by step instructions for determination of fiber reinforced concrete material model in **ATENA-GiD** interface. The issue is to find the appropriate tensile parameters to successfully model FRC. The measured response of direct tensile test could serve for direct input of the parameters into the material model. Unfortunately, preparation of test sample is complicated and the test is not performed very often. The three or four point bending tests are more common. Results can be also used for the material model but they cannot be directly put into the model. Inverse analysis of the results must be performed to identify model parameters correctly.

Two different methods of inverse analysis described in this tutorial can be utilized. The first method is manual analysis described in chapter 4.2.1. For the second method, the program **Consoft** based on the evolutionary algorithms is applied, the section is under construction yet. The developer team has experience with FRC simulation mainly of the segmental tunnel lining and some results are presented in the section 5.

The objective of this tutorial is to provide the user basic principles of FRC modeling and understanding of the material model behavior. For more information about the program the user should consult the ATENA documentation (e.g. [1], [2],[4] or [4]) or contact the program distributor or developers. Our team is ready to answer your questions and help you to resolve your problems.

The theoretical derivations and formulations that are used in the program are described in the Theory manual[4].

# 7. PROGRAM DISTRIBUTORS AND DEVELOPERS

Program developer: **Červenka Consulting s.r.o.**
Na Hrebenkach 55, 150 00 Prague 5, Czech Republic
phone: +420 220 610 018
fax: +420 220 612 227
website: www.cervenka.cz
email: cervenka@cervenka.cz

The current list of our distributors can be found on our websites:
http://www.cervenka.cz/company/distributors/

# 8. LITERATURE


[1] ATENA Program Documentation, Part 4-6, ATENA Science – GiD Tutorial, CERVENKA CONSULTING, 2016

[2] ATENA Program Documentation, Part 8, User's Manual for ATENA-GiD Interface, CERVENKA CONSULTING, 2015

[3] ATENA Program Documentation, Part 12, User's Manual for ATENA Studio, CERVENKA CONSULTING, 2016

[4] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2015

[5] ATENA Program Documentation, Part 11, ATENA Troubleshooting Manual, CERVENKA CONSULTING, 2016

[6] Consoft - software documentation. Available at: http://bauwesen.htwk-leipzig.de/nc/de/fakultaet-bauwesen/personen/name/slowik/

[7] *fib* Model Code 2010 - Final draft, Volume 1., Lausanne: International Federation for Structural Concrete (fib), 2012

[8] EN 14651, Test method for metallic fibre concrete - Measuring the flexural tensile strength (limit of proportionality (LOP), residual). European Committee for Standardization, B-1050 Brussels, September 2007

[9] Sajdlová, T. and Pukl, R., (2014): "Optimization of input parameters for material model of fibre reinforced concrete and application on the numerical simulation of tunnel lining", FRC 2014 - Fibre Reinforced Concrete: from Design to Structural Applications, Joint ACI-***fib*** International Workshop, Montreal, Canada, p. 13

[10] Marek, M.; Gramblička, M.; Baláž, D.; Pukl, R. and Sajdlová, T., (2016): "Steel fiber reinforced concrete precast tunnel segments: Design using nonlinear analysis", Podzemní stavby 2016, Prague, Czech Republic