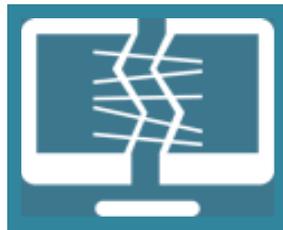


Virtual Lab for Fibre Reinforced Concrete Design by Simulation Prototyping

FibreLAB

Project funded by the European Community under the
Eurostars project E!10316



D5.2 User's Manual for FibreLAB Software Tools

Responsible author: Jan Cervenka

Co-authors: Zdenek Janda,

Tomas Altman,

Status: final

Type: software/pilot

Access: public

Version: 3.0

Date: 17.7.2019

Project website:

<http://www.fibrelab.eu>



Executive summary:

This project document is developed during the project FibreLab. This document is an user's manual for the software tools developed during the FibreLAB project.

The project develops software tools to support the design of advanced structures or products from fiber reinforced concrete (FRC) using simulation prototyping. The software supports engineers during the design process, which is based on the simulation of the structural performance during the foreseen design scenarios for the individual design limit states: serviceability and ultimate limit states as well as the new design states such as: robustness, durability and service life verification.

The software is developed based on the existing product ATENA developed and will be distributed by CER. The project develops a new module of this system specifically targeted for fibre reinforced concrete industry.

The product supports parametric modelling and embedded scripting language to enable the fast development of even more specialized design tools for the development and design of specific construction products for pre-cast industry or other mass production.

This documents provides the basic information how to apply the software tools developed in the research and development project FibreLAB for the analysis, design and performance evaluation of structures or structural components made of fibre-reinforced concrete. The document is divided into three main chapters which corresponds to the main three products developed during FibreLAB research project:

Chapter 2 - Rapid Prototyping for Parametric Modelling of FRC Products/Structures – ATENA Studio Pre

This chapter describes the main application of the prototyping and modelling software ATENA Pre. This software allows the parametric description and modelling of FRC products and structures.

Chapter 3 - FRC Model Parameters Identification

This chapter provides step by step description of the FRC model parameter identification based on experimental data using the procedures and software tools developed during FibreLAB project.

Chapter 4 - FRC Design using Simulation Prototyping

This chapter provides a guidance how to perform the global structural assessment and design using the new ATENA software developed during FibreLAB project for fibre-reinforced structures and products.

Revision history:

date	author	status	changes
16. 6. 2019	J. Červenka	1.0	draft
16. 7. 2019	Z. Janda	2.0	Draft 2
17. 7. 2019	J. Červenka	3.0	First complete version for official release

Table of contents:

REVISION HISTORY:	3
1. GENERAL	6
2. RAPID PROTOTYPING FOR PARAMETRIC MODELLING OF FRC PRODUCTS/STRUCTURES – ATENA STUDIO PRE	7
2.1 GEOMETRICAL AND NUMERICAL MODELS	8
2.2 MAIN WINDOW	9
2.3 MAIN DATA TREE	11
2.4 SCRIPT HISTORY	12
2.5 GLOBAL PARAMETERS	13
2.6 CONSTRUCTION COMMANDS	18
2.8 DELETE COMMAND	19
2.9 TRANSFORMATION COMMANDS	20
2.10 MATERIAL DEFINITION.....	22
2.10.1 Concrete FRC – Fiber Reinforced Concrete	24
2.10.2 Concrete User.....	27
2.10.3 Concrete SHCC – Strain Hardening Cementitious Composite for Fibre Reinforced Concrete	30
2.11 MODELLING OF STRUCTURAL ELEMENTS USING BIM MODEL.....	34
2.12 ATENA IN RELATION TO BIM	36
2.13 IFC FILE IMPORT TO THE ATENA STUDIO PRE	37
2.14 IFC DATA COMPATIBILITY IN ATENA STUDIO PRE.....	40
2.14.1 Materials	40
2.14.2 Building elements	40
2.14.3 Reinforcing elements	41
2.14.4 Profiles	42
2.14.5 Geometry.....	44
3. FRC MODEL PARAMETERS IDENTIFICATION	46
3.1 SIMPLIFIED APPROACH FOR FRC PARAMETER IDENTIFICATION	48
3.2 MANUAL INVERSE ANALYSIS FOR FRC PARAMETER IDENTIFICATION	50
3.3 RULE-BASED INVERSE ANALYSIS FOR FRC PARAMETER IDENTIFICATION	61
4. FRC DESIGN USING SIMULATION PROTOTYPING	67
4.1 ASSESSMENT OF STRUCTURAL SAFETY AND RELIABILITY	67
4.1.1 Full Probabilistic Analysis	68
4.1.2 ECOV Method – Estimate of Coefficient of Variation	72
4.1.3 Method based on EN 1992-2	72

4.1.4 Partial Safety Factors	73
4.2 VALIDATION OF SAFETY FORMATS FOR ORDINARY REINFORCED CONCRETE STRUCTURES	73
4.3 PERFORMANCE OF FRC STRUCTURES	74
5. SUMMARY.....	77
6. REFERENCES.....	78

1. General

The project develops a software tool to support the design of advanced structures or products from fiber reinforced concrete (FRC) using simulation prototyping. The software supports engineers during the design process, which will be based on the simulation of the structural performance during the foreseen design scenarios for the individual design limit states: serviceability and ultimate limit states as well as the new design states such as: robustness, durability and service life verification.

The software is developed based on the existing product ATENA developed and distributed by CER. The project develops a new version of this system specifically targeted for fibre reinforced concrete industry.

The product supports parametric modelling and embedded scripting language to enable the fast development of even more specialized design tools for the development and design of specific construction products for pre-cast industry or other mass production.

The software prototype was developed as part of work package W2, which develops a tool for rapid prototyping of parametric modelling of fiber reinforced concrete structures and products. The tool will be based on the current input capabilities of the simulation system ATENA, which contains several methods for the problem definition:

- (a) GiD solid modeller (www.gidhome.com) with embedded fully automated mesh generator and scripting language based on TCL, GiD modeller is a third party product and CER is the member of its partner network and an official re seller.
- (b) ATENA system includes its own definition language for geometric modelling and Python scripting language enabling the parametric definition of the geometric model. It also includes a powerful fully automated 2D/3D mesh generator for both simplicial and non-simplicial meshes.
- (c) ATENA Engineering version contains its own pre-processing interactive module, which is however technologically obsolete now, and will not be used in this project for further development.

The development will build on the current scripting capabilities using either Python or TCL to develop a tool, which will allow rapid development of specialized pre-processing and data preparation tools for specific structures and/or products. This tool will be extremely useful to increase the attractiveness and applicability of the proposed project to the precast industry or for the mass manufactures of products for building and construction industry.

Typical examples are producers of precast elements, but it is attractive also for manufactures of various anchoring or connecting systems.

2. Rapid Prototyping for Parametric Modelling of FRC Products/Structures – ATENA Studio Pre

ATENA Studio PRE user interface consists of three main windows, through which a user can control three phases of execution: Pre-processing, Run (calculation) and Post-processing. ATENA has a modular structure. Calculation phases are processed by these program modules separately by opening a corresponding window. This can be done by run the analysis in Tasks option in the tree. Main window of ATENA Studio PRE you can see in the subsequent figure:

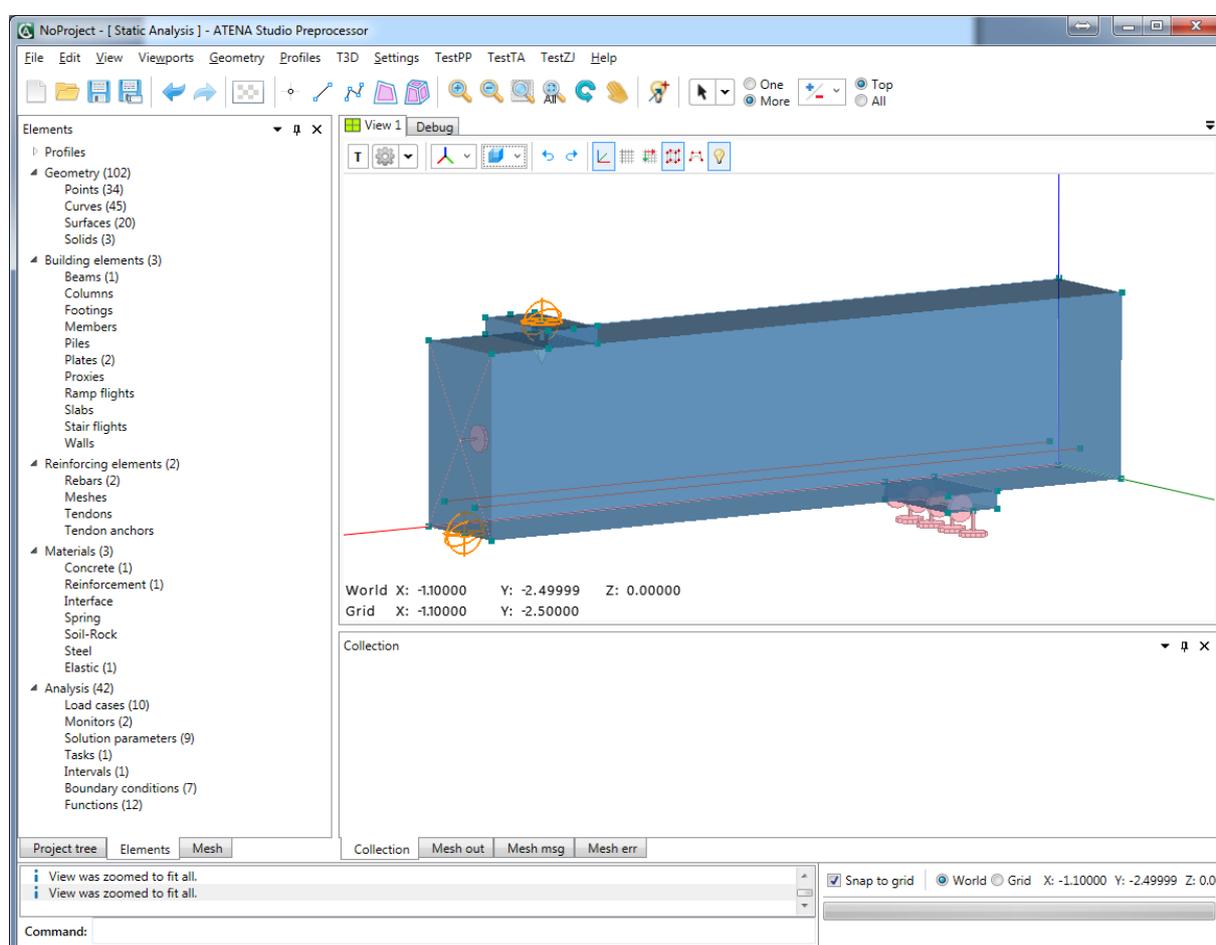


Fig. 2-1: Main window of ATENA Studio PRE

2.1 Geometrical and Numerical Models

ATENA recognizes two models, geometrical and numerical. Data of these models are treated strictly separately:

Geometrical model represents dimensions, properties and loading. It consists of an assembly of building elements (created from solids, surfaces, lines and points). Reinforcement can be modeled by two ways. First way is a discrete bar, which is defined as a geometrical multi-linear object. It is embedded in solid objects. Its geometry is defined independently of building elements. Thus one reinforcing bar can be embedded in any number of building elements. Second way is a smeared reinforcement, defined as a composite material, which has no effect on geometrical model. Geometrical model is completed by defining loading (load cases and load history) and construction cases.

Numerical model is generated based on geometrical model and represents a numerical approximation of the structural analysis problem. Numerical model is a result of discretization made by the finite element method. The mesh generator in ATENA makes possible to generate automatically meshes for solid and reinforcing objects.

Terminology

Material – Constitutive model of material used in ATENA to represent a certain material type.

Geometrical model – Solid structure described as an assembly of basic geometrical objects (macroelements, their contacts, reinforcing, springs, etc.)

Solid – basic geometrical 3D object defined by surfaces.

Surface – planar face of object circumvented by lines.

Curve – curve (line) connecting two joints.

Point – basic geometrical entity.

Reinforcement bar – multi-linear object embedded in macroelements

Load case - load action defined on assembly of geometry.

Load step – increment of load action.

Task – analysis of some type.

Intervals (Analysis steps) – loading history defined as a sequence of load increments.

Analysis steps – loading history defined as a sequence of load increments.

Finite element – basic element used for approximation of deformation and stress state of a solid.

Node – point in finite element mesh.

DOF – degree of freedom used as independent variable in finite element analysis.

2.2 Main window

In this chapter we shall describe the functionality of the pre-processing window in the sequence as they appear on the screen. Tool-bars available in the pre-processing window are described in Fig. 2. Detail description follows. Most functions can be accessed in two ways:

(1) Using graphical tools form the tool bars (see Sections 3.3 to 3.6.5 and 3.7.1 to 3.7.6). This is the recommended way.

(2) Using the main menu items, see Fig. 2-3 and Sections 3.2.1 to 3.2.7. Some graphical functions cannot be accessed from the main menu.

(3) Using the command line

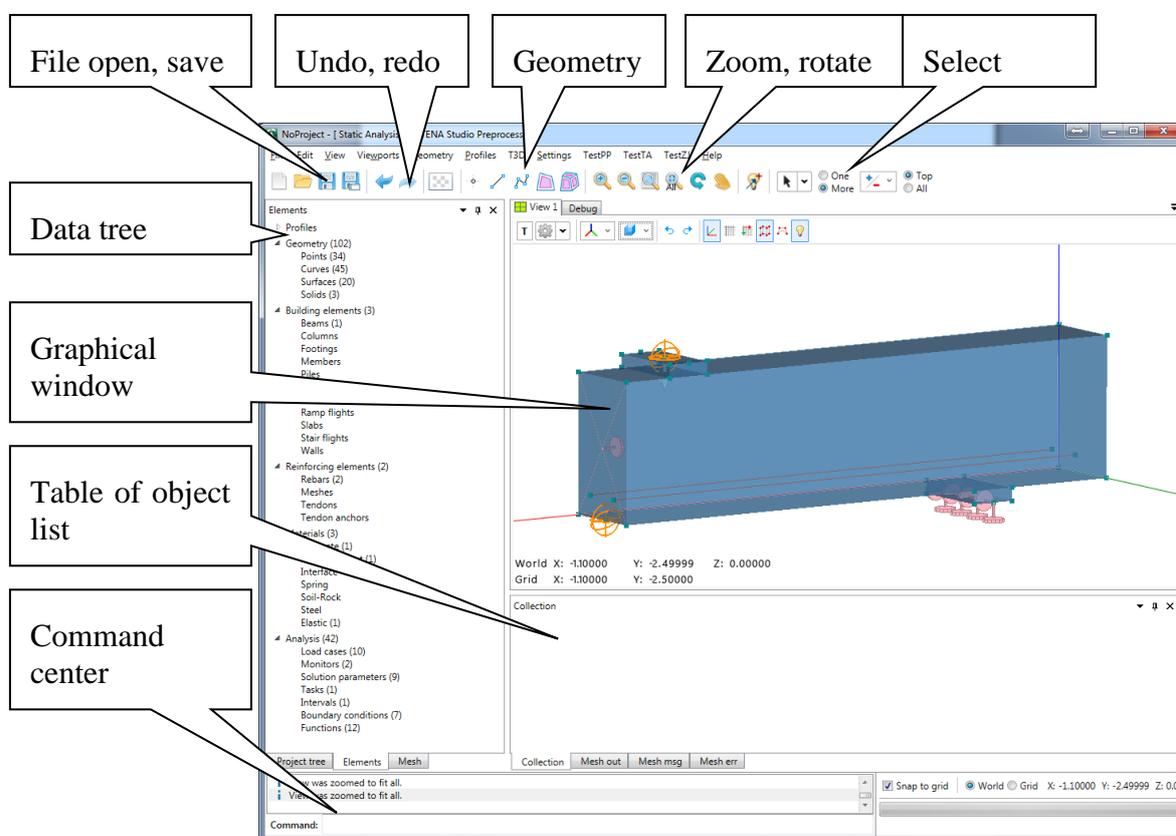


Fig. 2-2: Main window of ATENA Studio PRE – description

Main Menu

Items of the menu you can see in Fig. 2-3. In the item **Open other** following ATENA files can be open:

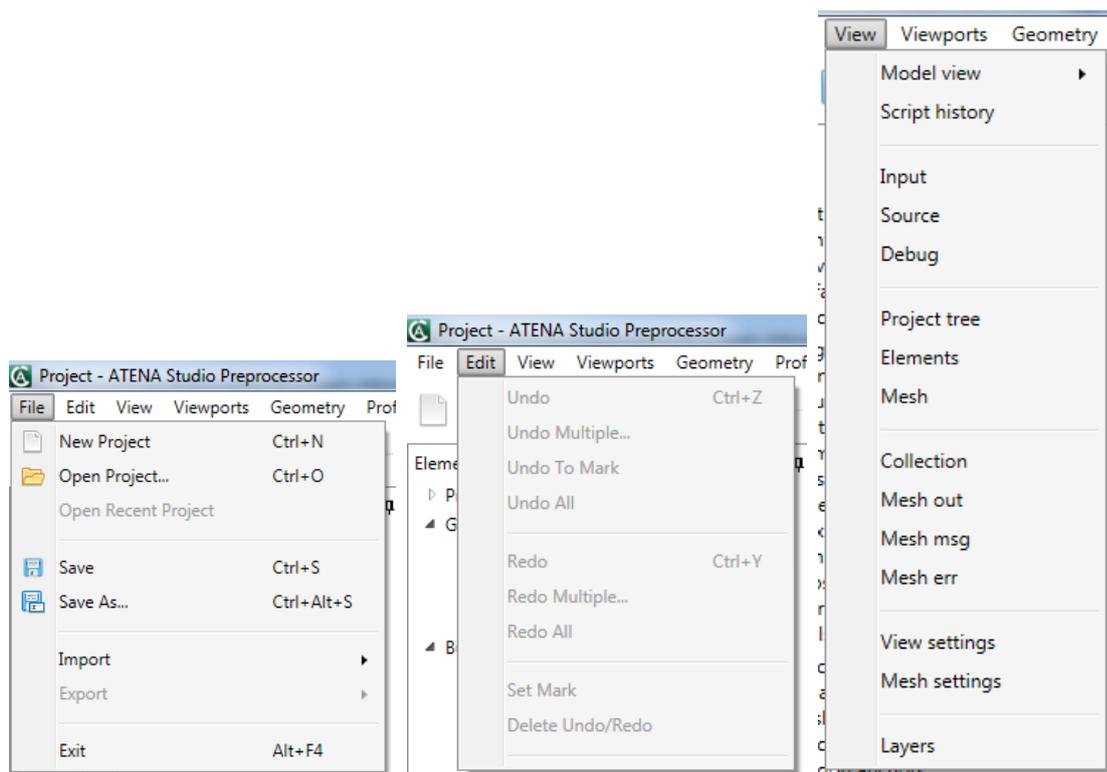


Fig. 2-3 Main menu a) File, b) Edit, c) View

Import feature supports import from multiple file types (IFC, IGES). The Industry Foundation Classes (IFC) data model is intended to describe building and construction industry data. It is a commonly used collaboration format in Building information modeling (BIM) based projects. The Initial Graphics Exchange Specification (IGES) (pronounced eye-jess) is a vendor-neutral file format that allows the digital exchange of information among computer-aided design (CAD) systems.

2.3 Main Data tree

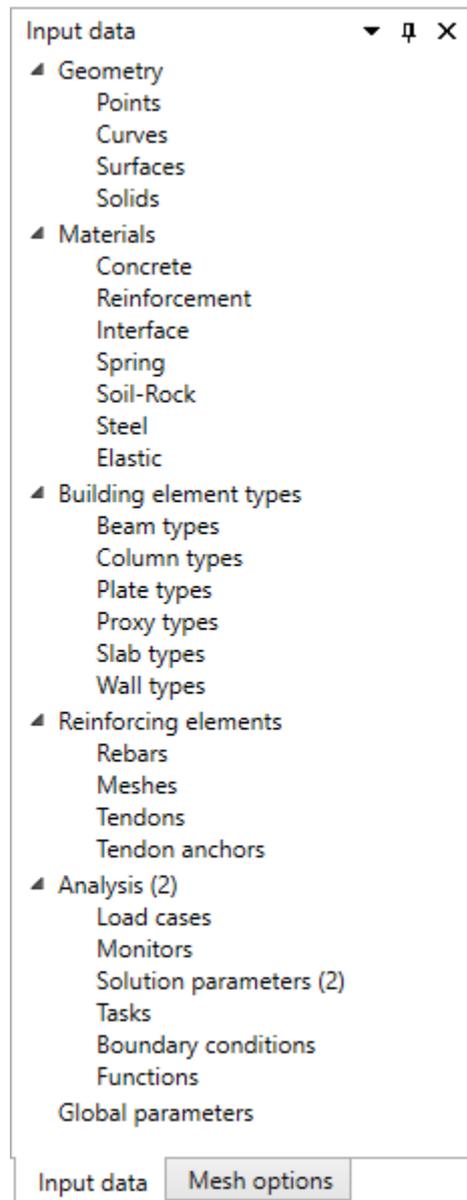


Fig. 2-4: main data tree

Main data tree shows structure of model. You can go from top to down, and step by step enter all items. In Profiles you can set many types of cross-sections, which can be used for 1D elements. All geometry items are composite from 4 types of geometric entities: Point, Curve, Surface and Solid.

Building elements define entities which include some geometric entities, and some special settings. By building elements, you can define for example beams, piles, slabs, or walls. The next item is

Reinforcing elements, it is the same like the building elements, but only for the reinforcements. There you can use couple of types of reinforcements like classic rebars, reinforcing meshes or tendons. Next group is used for definition of materials. There are several types of material, for different types of materials. More about the definition of materials will be in the next section.

The last part is the Analysis group... Load cases are the groups of condition (Body forces, support, forces, shrinkage, prestressing, temperature, ...) Next you can set some monitors, it is special point (or other geometry entities) who measure value of the quantity (displacement, reaction, stress, strain, ...) Solution parameters, are groups of parameters which is necessary for control the calculation. Task is simulating one experiment. Each task has several intervals, and each interval have combination of load cases. So, you can set different boundary conditions in different part of experiment.

2.4 Script history

In preprocessor all commands are handled by the script center. This center works in Python. Every executed script is saved in history and can be reprocessed at any time, see Fig. 2-5

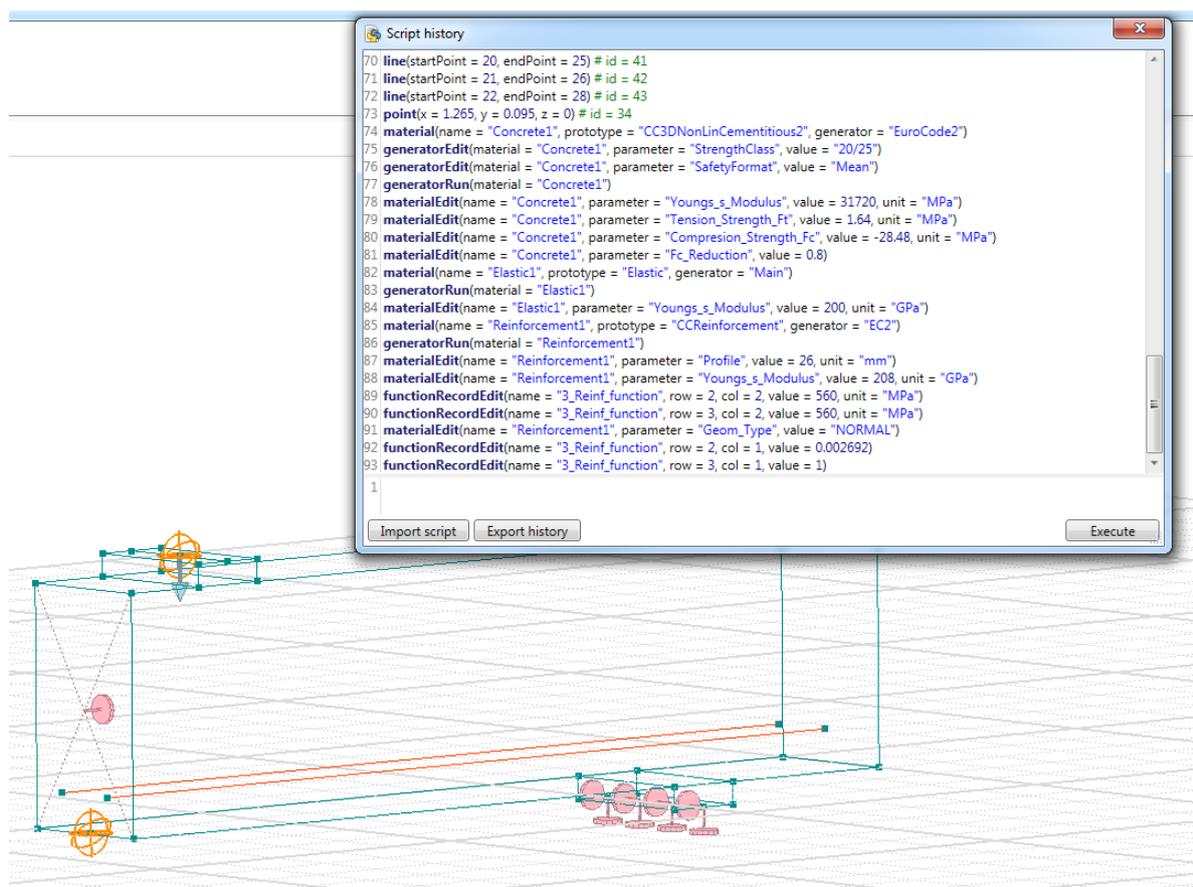


Fig. 2-5: Example of script history for model Shear beam

2.5 Global parameters

If we know relationships of geometry, we can create models with variable parameters.

In the preprocessor each point coordinate can be set parametrically. The parameter must be created before its usage. It can be done in command line or directly in settings textboxes when the command (e.g. points creating) is in process. An equation of creating of the parameter must be in a format e.g. “ $a = 4$ ”. When the parameter is created, it can be displayed in global parameters table and is offered during writing in coordinate textbox.

If we have done the model with parameters, we have complete python script saved in script history which can be used again with changed parameters.

See examples of models created parametrically below.



Fig. 2-6: Example of a tram-line FRC product to be analysed by ATENA system

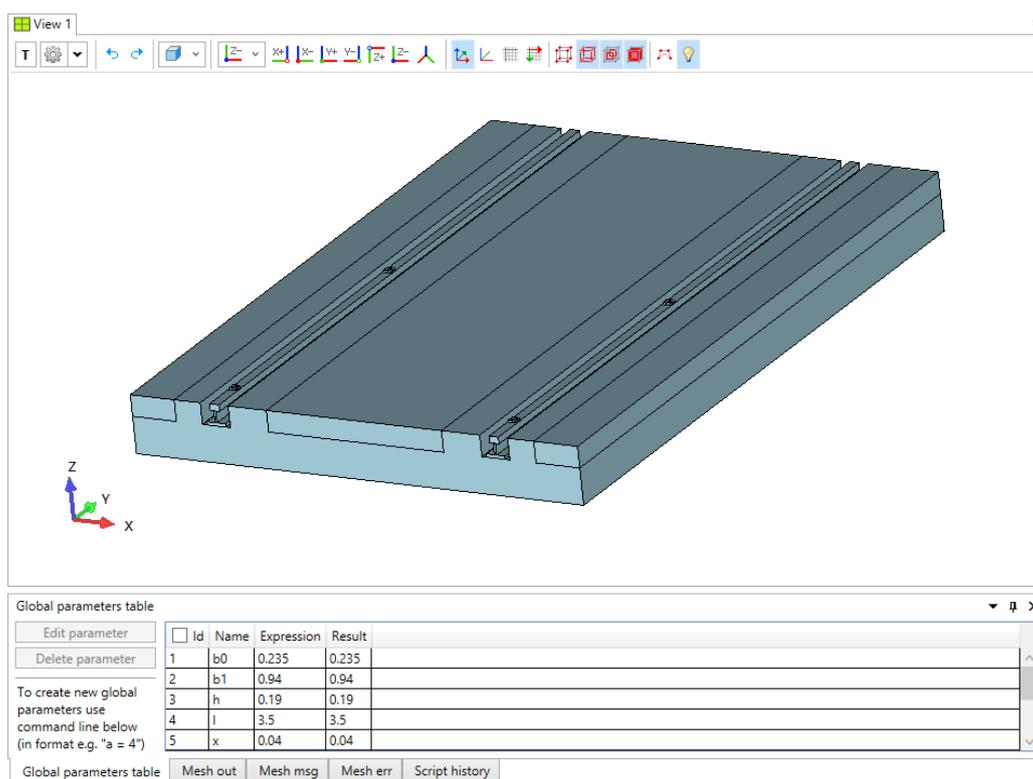


Fig. 2-7: Example of parametrically created tram lines model. View and global parameters table

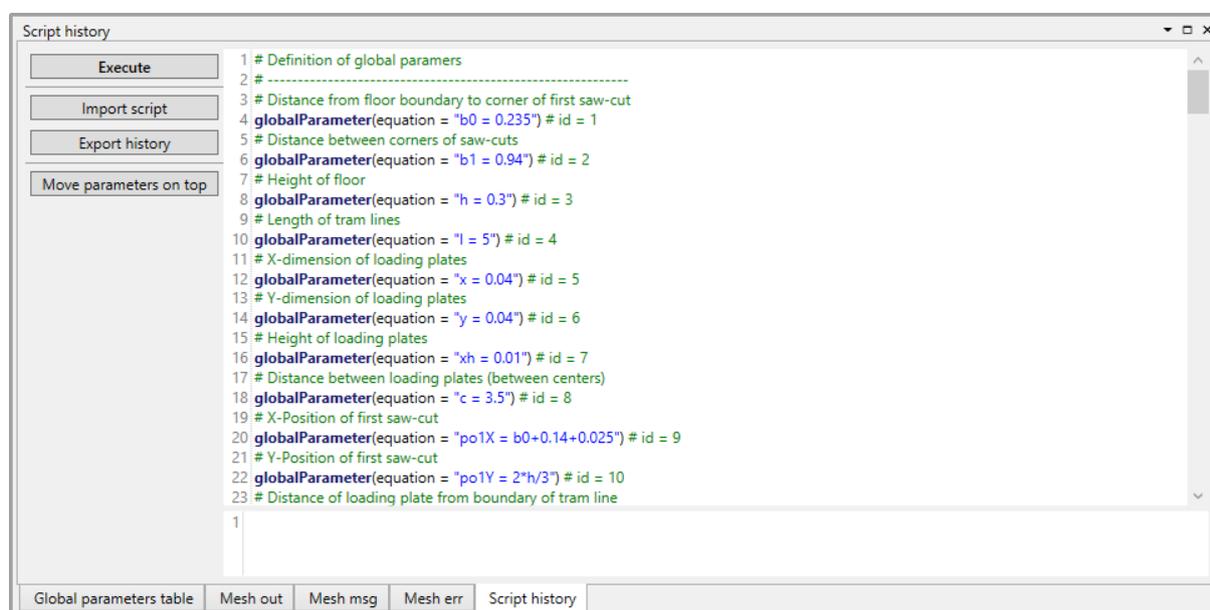


Fig. 2-8: Example of python script of parametrically created tram lines model.

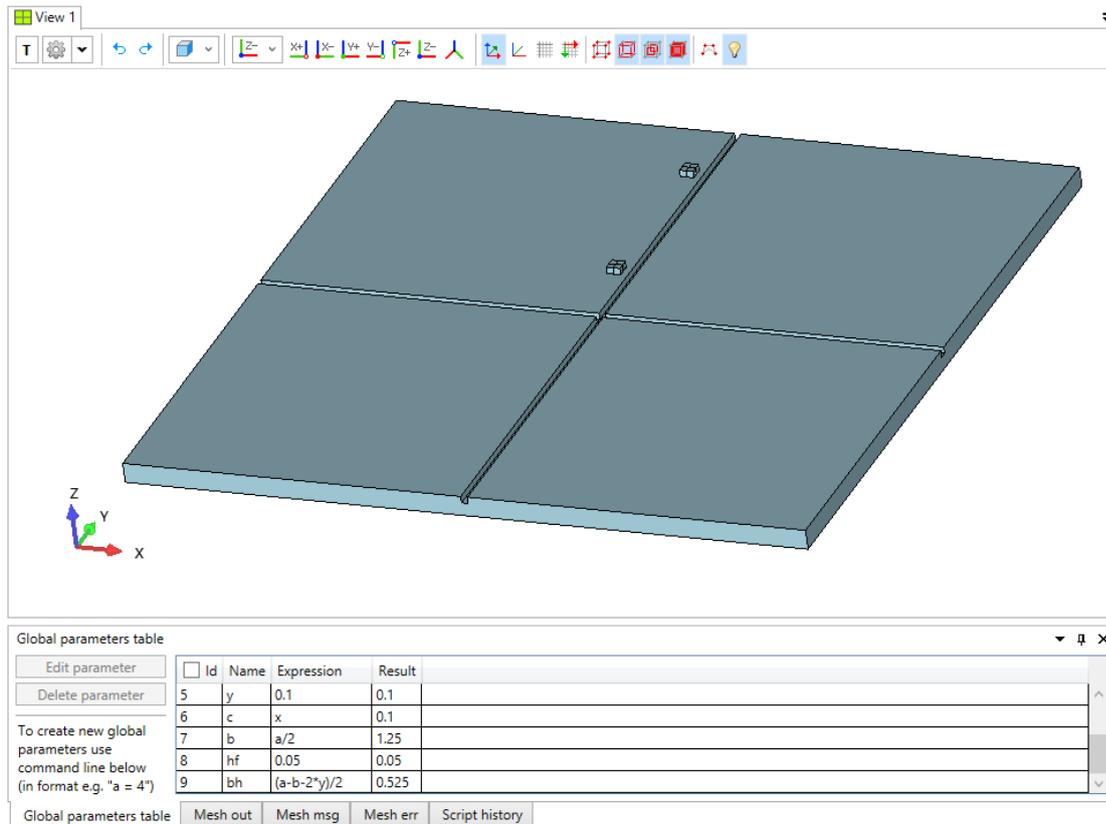


Fig. 2-9: Example of parametrically created industrial floor model. View and global parameters table

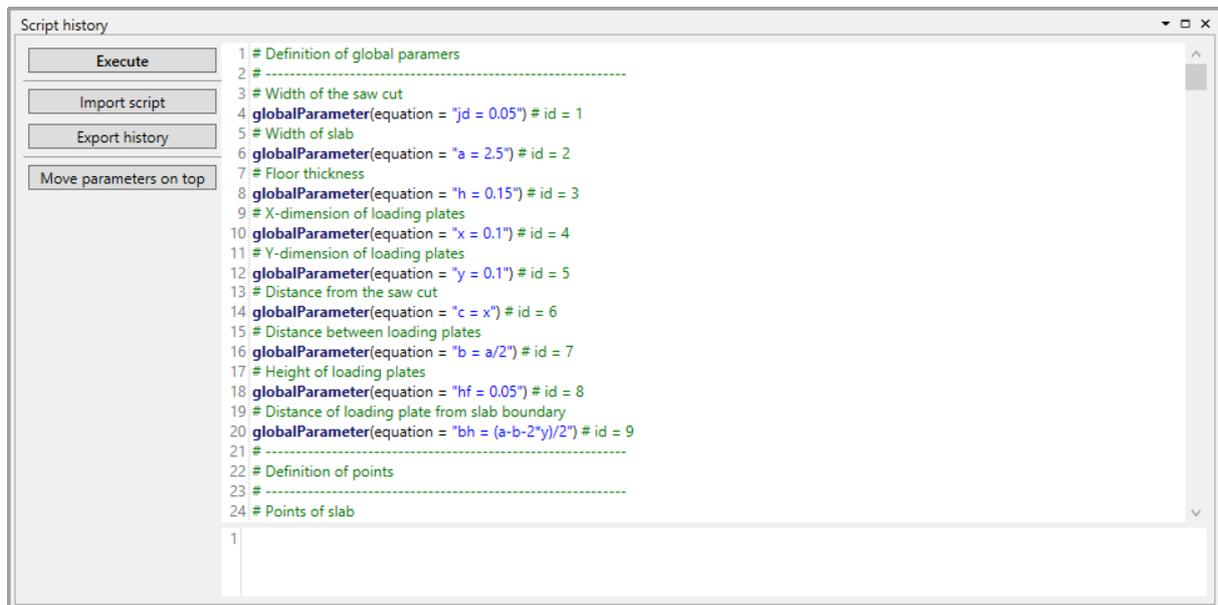


Fig. 2-10: Example of python script of parametrically created industrial floor.

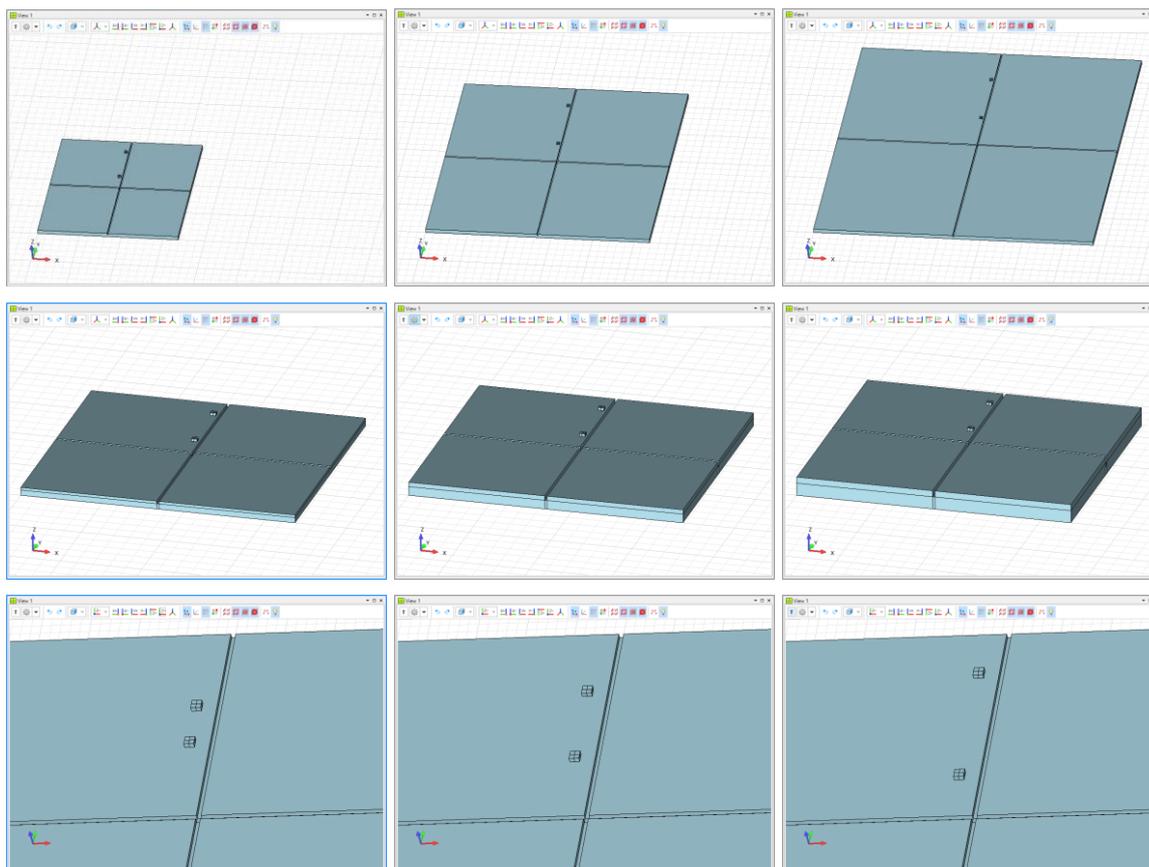


Fig. 2-11: Examples of changing parameters in industrial floor model (changing length, height and distance between loading plates).

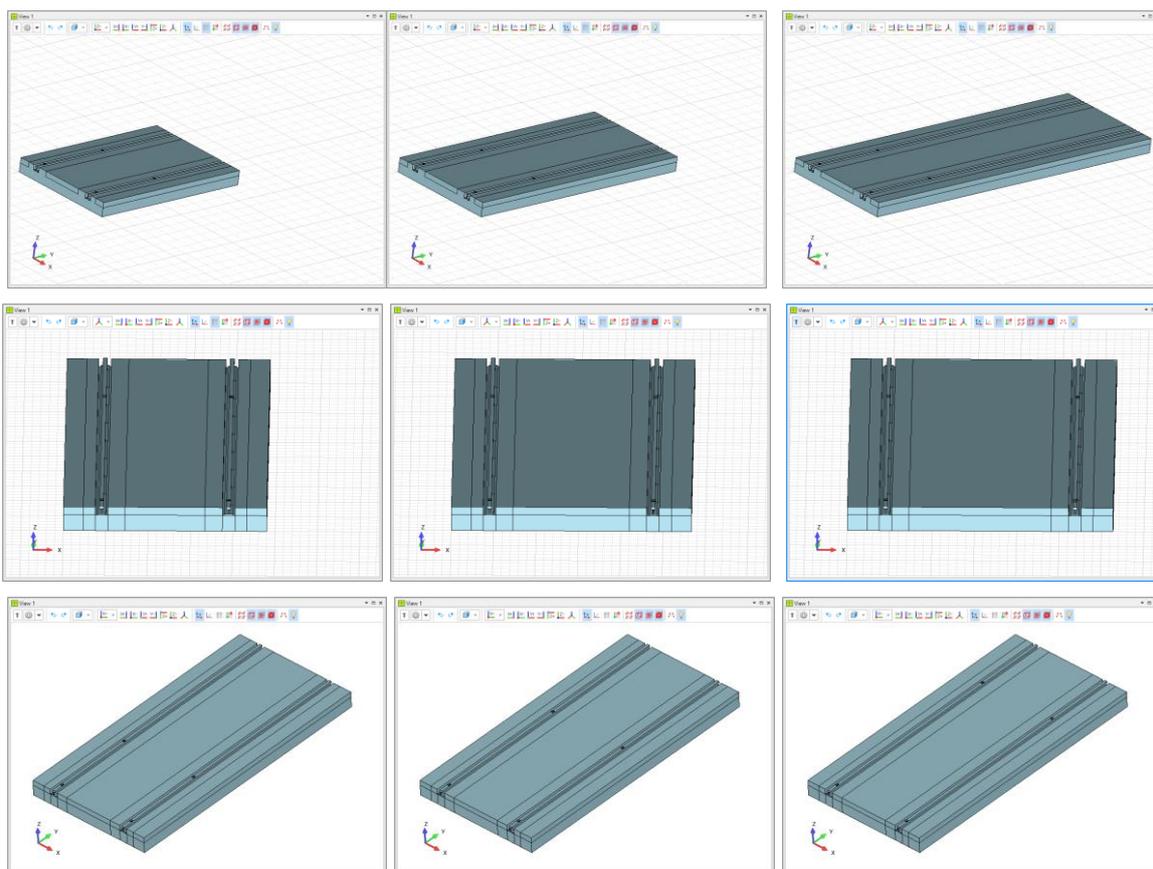


Fig. 2-12: Examples of changing parameters in tram lines model (changing length, height and distance between loading plates).

2.6 Construction Commands

The geometry of Preprocessor can be imported from IGES or IFC, can be created by python script or created by user self. If the user decides to create a new model from the beginning or just some part, there are four basic buttons in main toolbar which serve to create points, line segments, polygonal surfaces and boundary represented (B-rep) solids dynamically. Each of these commands has appropriate settings window placed in layout, where it can be seen which entities are selected or which point coordinates are currently used (see Fig. 2-13)

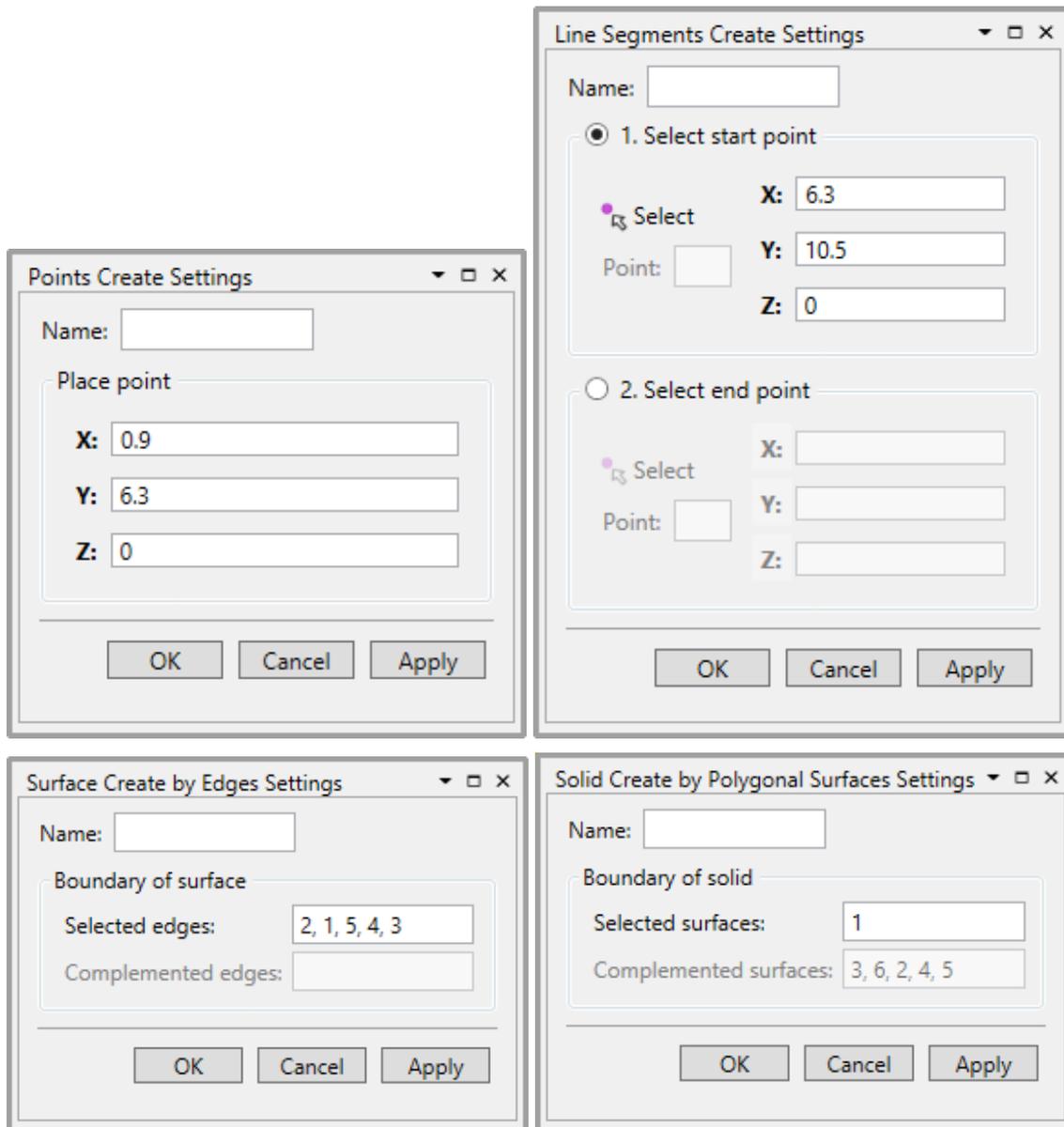


Fig. 2-13: Points, lines, polygonal surfaces, BREP solids creating settings

2.8 Delete Command

Each entity, which can be seen in data table, can be deleted by a button near the table, by the icon on the toolbar or using the Delete key, if the entity is selected. Always it is asked to confirm deletion and shown entity ids with total count which will be deleted. If the icon on the toolbar is used, a delete settings window is shown where can be chosen what really delete and see what will be deleted and what will remain before confirmation of deletion, see Fig. 2-14.

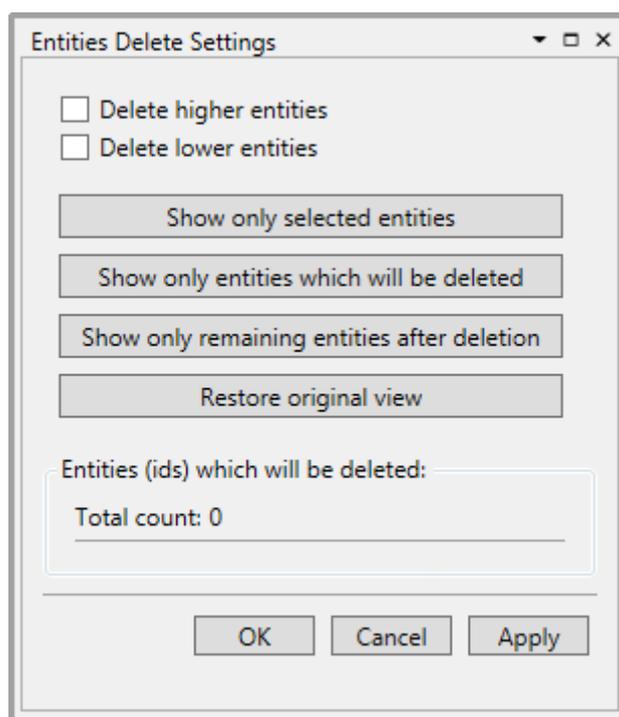


Fig. 2-14: Entity deleting settings

2.9 Transformation Commands

In Preprocessor are supported three basic geometry transformations: translation, scaling and rotation. They are accessible on the toolbar and each of these transformations has own settings window (see Fig. 2-15). Transformed geometry can be copied and collapsed with close geometry. If the geometry is not copied and is part of geometry not selected to transform it occurs disconnecting transformed from remaining geometry and creating geometry where would be created holes.

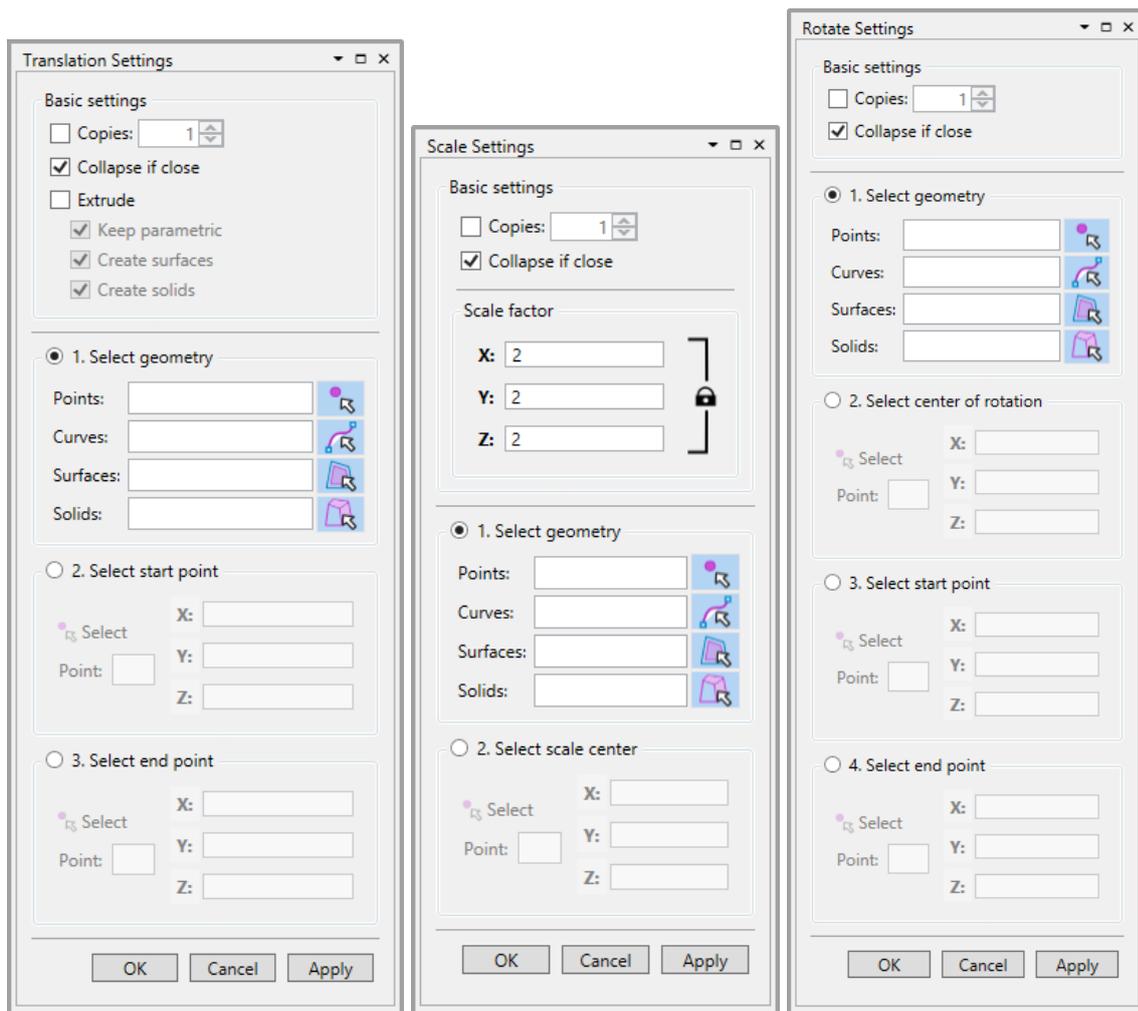


Fig. 2-15: Geometry transformation settings

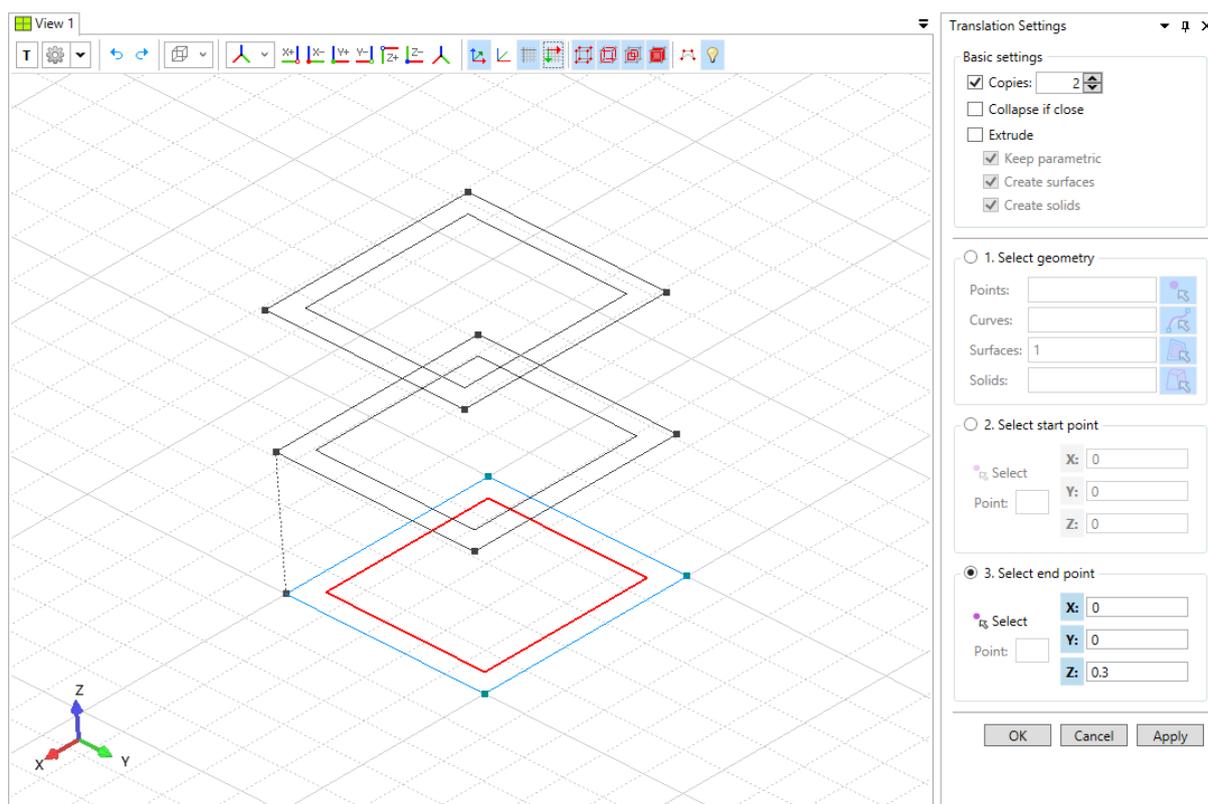


Fig. 2-16: Example of copying a surface during the translation.

2.10 Material Definition

The input of material types and their parameters is done by choosing the menu item in the access tree. ATENA support several types of material types: Concrete, Reinforcement, Interface, Spring, Soil-Rock, Steel and Elastic material. By double click or click to button create material will start dialog, in which you can select from couple of automatic generators, which you like to use. The generators are predominantly based on standards used for nonlinear computing in the world (Euro Code, Model Code, Fib, Model Code + FRC generator, ...) Each material prototype has specific material parameters, which is generated by this automatic tool.

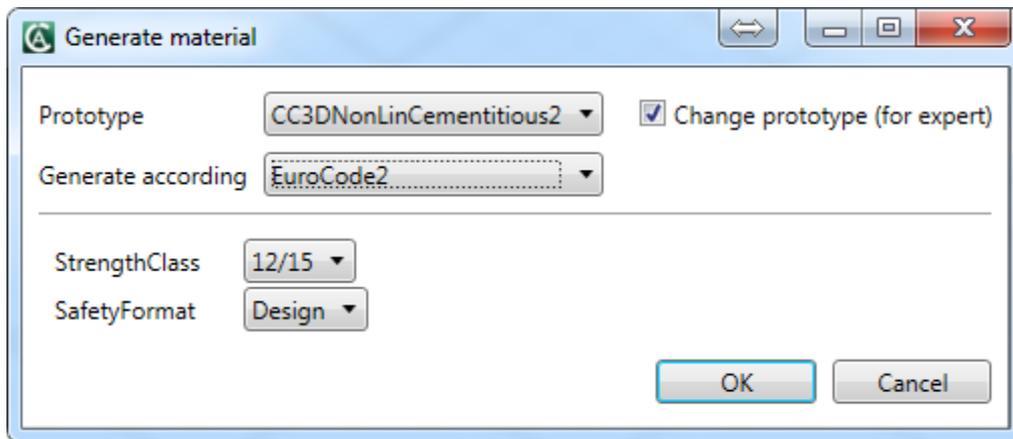


Fig. 2-17: Generator dialog for concrete

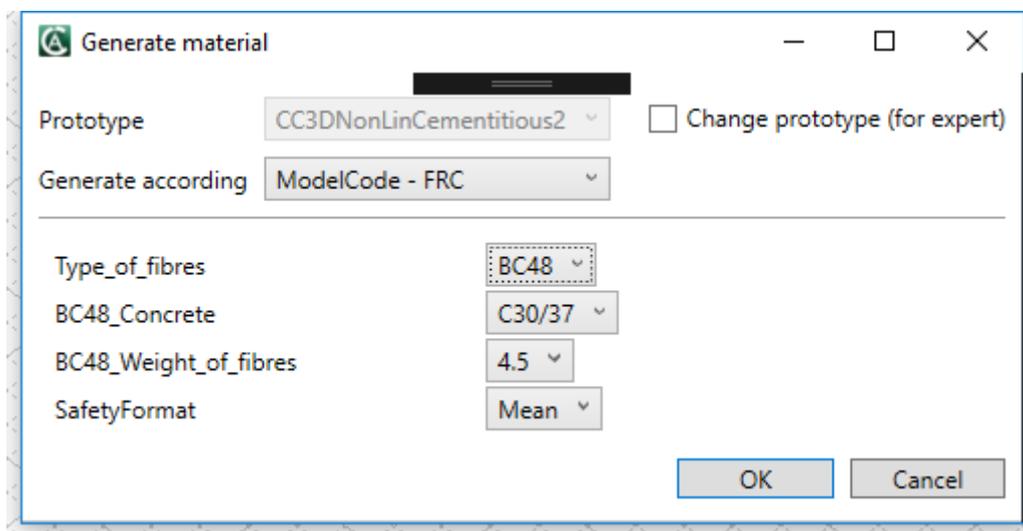


Fig. 2-18: Generator dialog for concrete FRC

The input of material properties is accompanied by explaining comments and pictures. All material models offer a set of default parameters. These default parameters of models for concrete-like materials are derived from cube compressive strength. The relations for material parameters (Elastic modulus, tensile strength, etc.) are taken mainly from the FIB Model Code 90, Euro Code and other sources. In cases where user does not know all parameters, the default values can be used. If precise values of material properties are known the default parameters can be edited and changed to the desired ones.

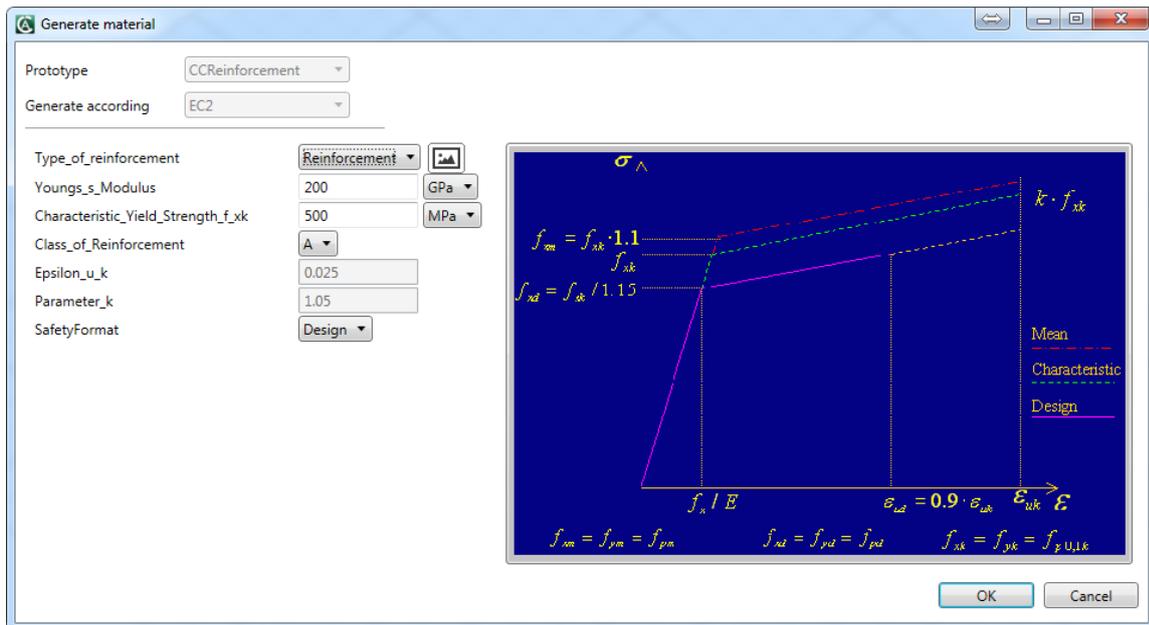


Fig. 2-19: Generator dialog for reinforcement

Python scripting

All generators, and material parameters export are made using by scripting Python. These scripts are saved in text forms, so the user can modify it. Or make a new one (for example when new model code for some country is written).

```

0
1
2 <Generators>
3 <Generator Name="EuroCode2" Prototype="CC3DNonLinCementitious2">
4   <EnumMaterialParameter TabSection="Basic" Name="StrengthClass" Value="12/15" ToolTip="" ValueRange="12/15,16/20,20/25,25/30,30/37,35/45,40/50,45/50">
5   <EnumMaterialParameter TabSection="Basic" Name="SafetyFormat" ToolTip="" Value="Design" ValueRange="Characteristic, Design, Mean, EN1992-2" />
6
7   <GeneratePython>
8   materialProxy.InitMaterial()
9
10  StrengthClass = materialProxy.GeneratorParameters("StrengthClass").GetValue()
11  SafetyFormat = materialProxy.GeneratorParameters("SafetyFormat").GetValue()
12
13  #print StrengthClass
14  #print SafetyFormat
15
16  gamma = 1.0
17  if SafetyFormat == "Design" :
18    gamma = 1.5
19  if SafetyFormat == "Characteristic" :
20    gamma = 1.0
21  if SafetyFormat == "Mean" :
22    gamma = 1.0
23  if SafetyFormat == "EN1992-2" :
24    gamma = 1.18577
25
26  if StrengthClass == "12/15" :
27    fck = 12
28    beton = 15
29    fcm = 20
30    fctm = 1.6
31    fctk = 1.1
32    moduLE = 27000
33
34  if StrengthClass == "16/20" :
35    fck = 16
36    beton = 20
37    fcm = 24
38    fctm = 1.9
39    fctk = 1.3
40    moduLE = 29000
41
42  if StrengthClass == "20/25" :
43    fck = 20
44    beton = 25
45    fcm = 28
46    fctm = 2.2
47    fctk = 1.5
48    moduLE = 30000
49

```

Fig. 2-20 : Example of python scripting of EuroCode2 generator for concrete

For simulating fiber reinforced concrete, you can use three different material in ATENA (Concrete FRC, Concrete User and Concrete SHCC).

2.10.1 Concrete FRC – Fiber Reinforced Concrete

The *CC3DNONLINCEMENTITIOUS2FRC* is one of suitable materials for fiber reinforced concrete. It has special generation script, for generation the proper material parameters for FRC.

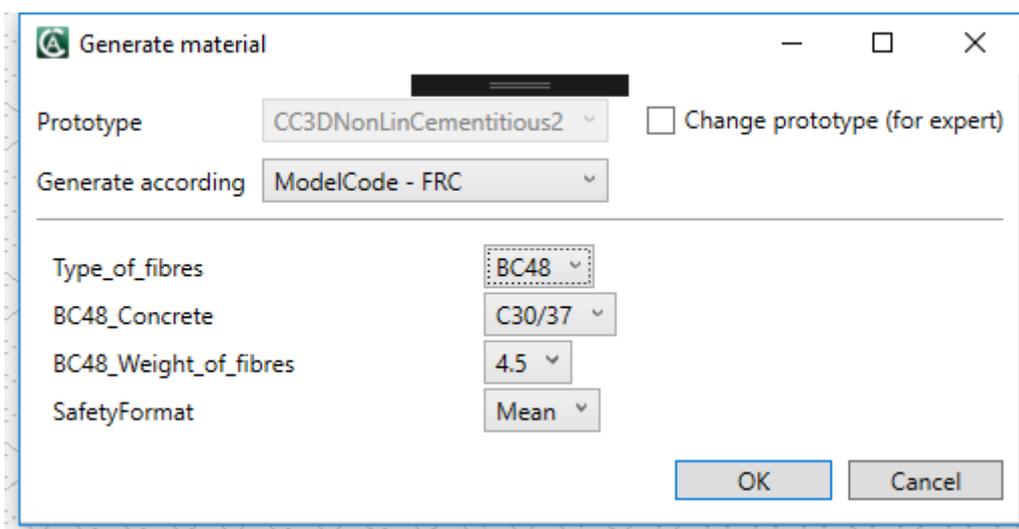


Fig. 2-21 : Generator for FRC material

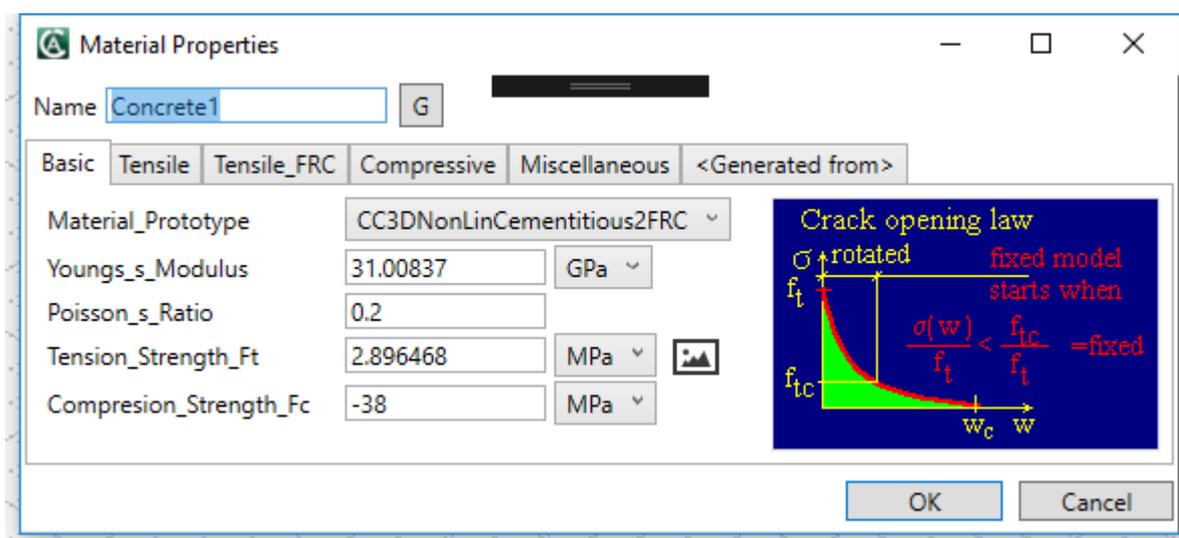


Fig. 2-22: Dialog Concrete FRC – Basic parameters

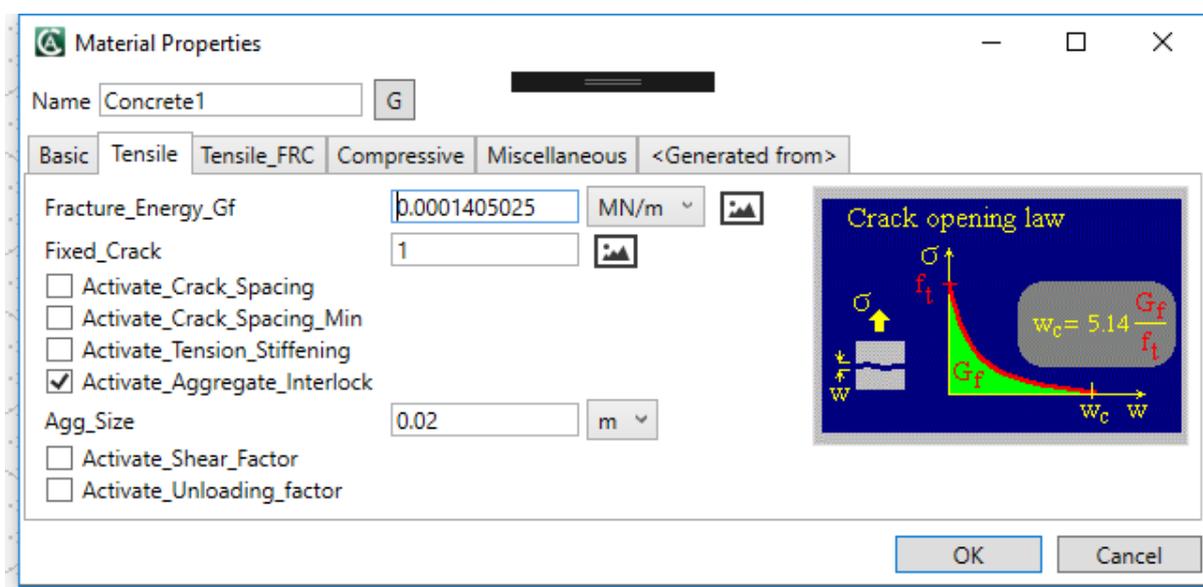


Fig. 2-23: Dialog Concrete FRC – Tensile parameters

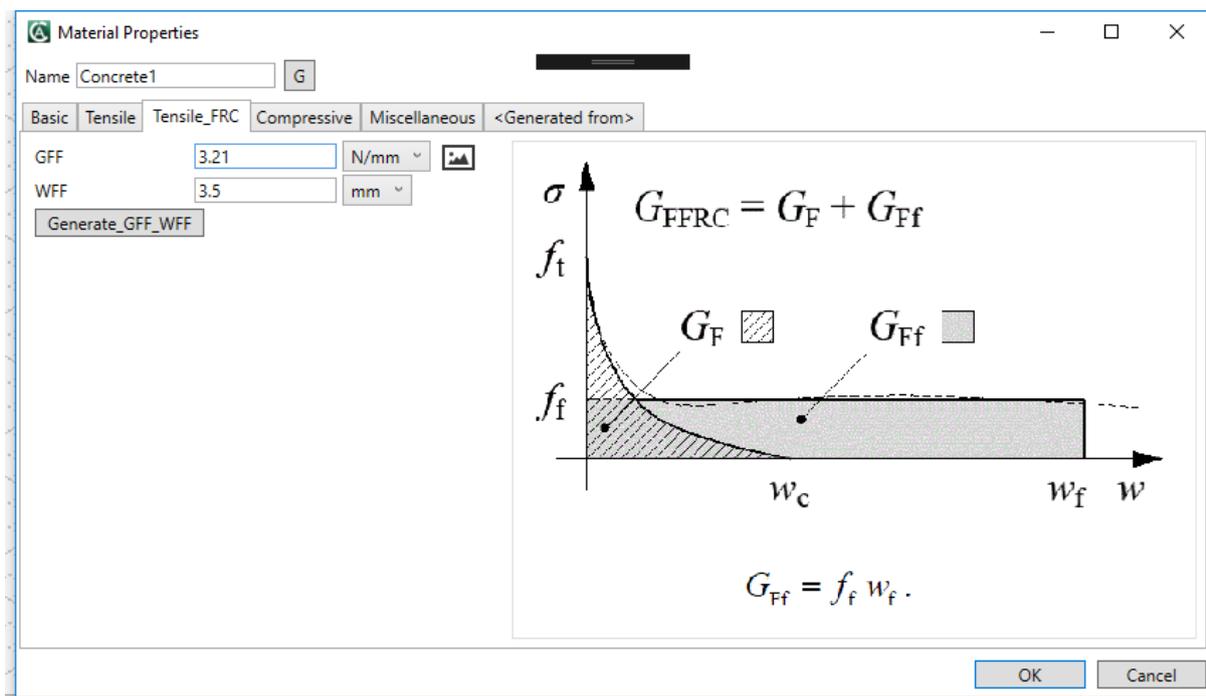


Fig. 2-24: Dialog Concrete FRC – Tensile FRC parameters

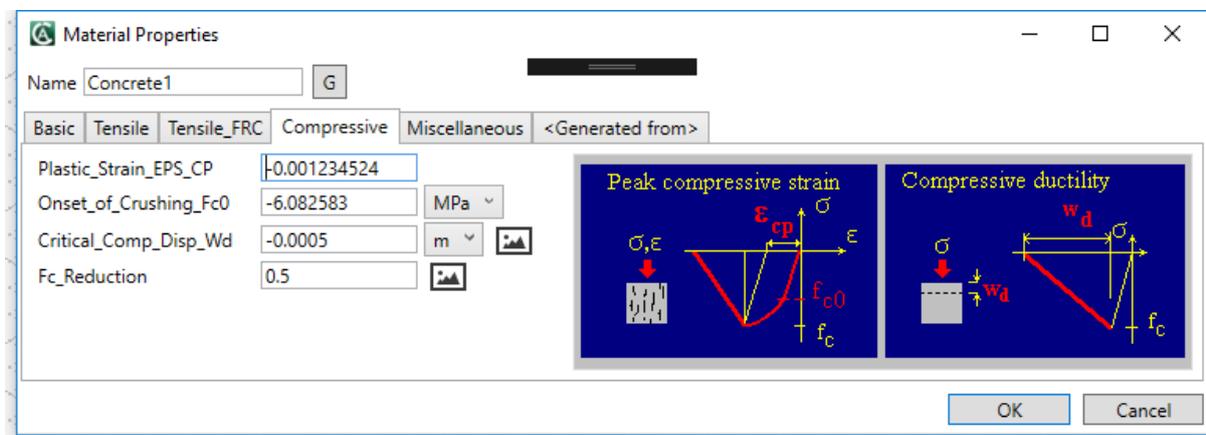


Fig. 2-25: Dialog Concrete FRC – Compressive parameters

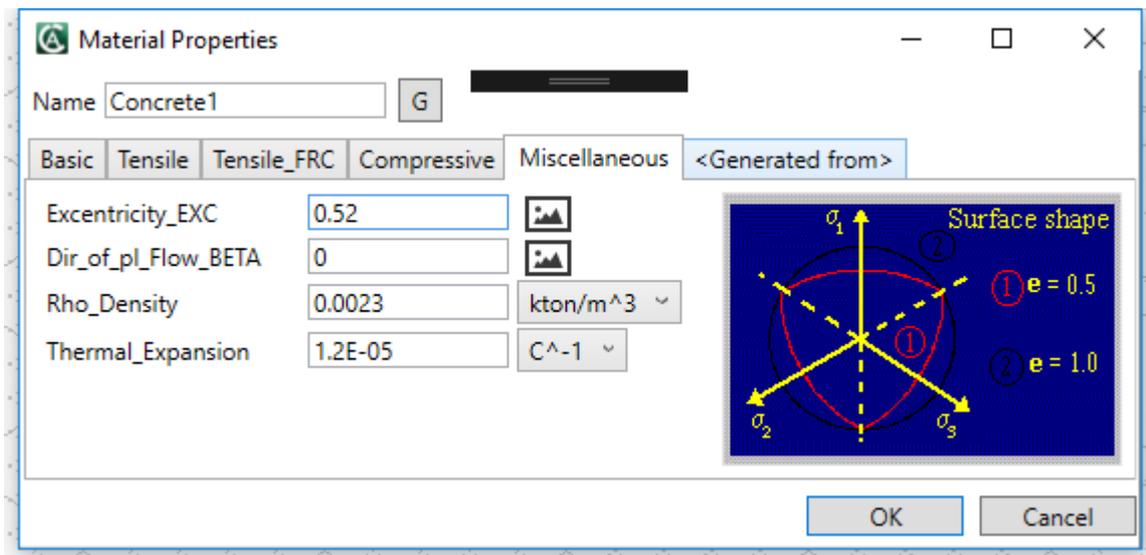


Fig. 2-26: Dialog Concrete FRC – Miscellaneous parameters

2.10.2 Concrete User

The Fiber reinforced concrete is suitable as well the *CC3DNONLINCEMENTITIOUS2USER* material. Special setting for FRC is on tab Tensile. We can use the special sub generator.

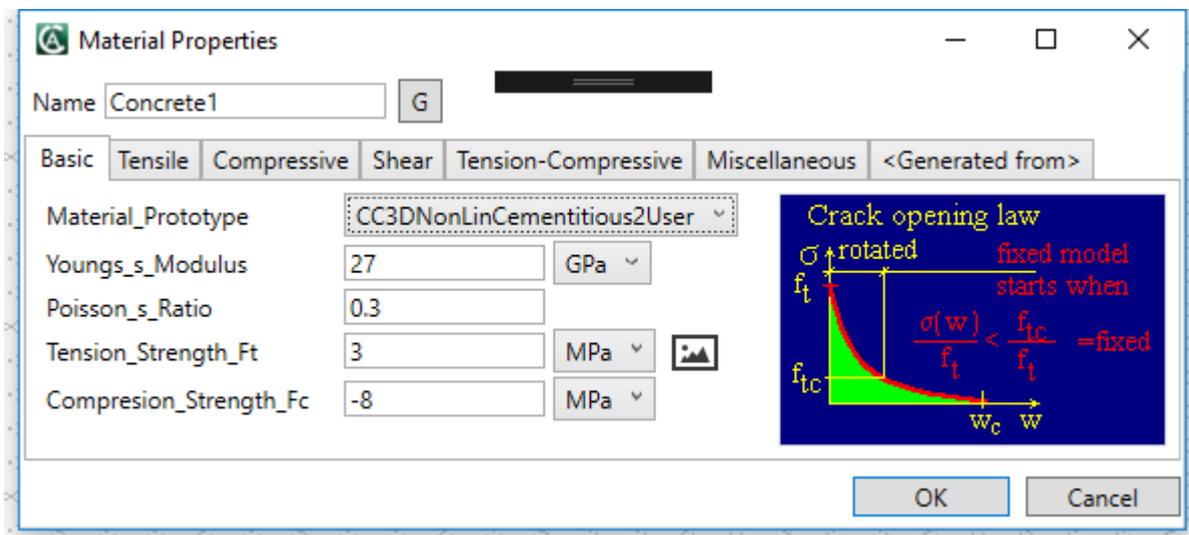


Fig. 2-27: Dialog Concrete User – Basic parameters

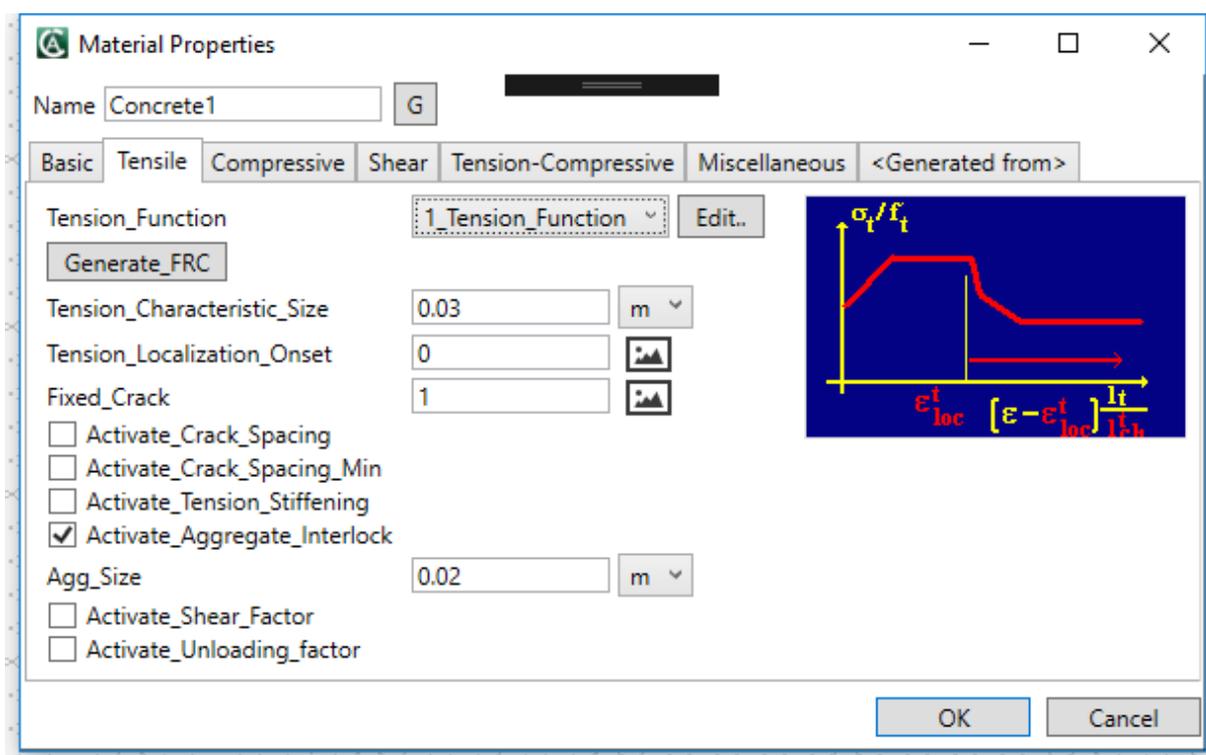


Fig. 2-28: Dialog Concrete User – Tensile parameters



Fig. 2-29: Dialog Concrete User – Compressive parameters

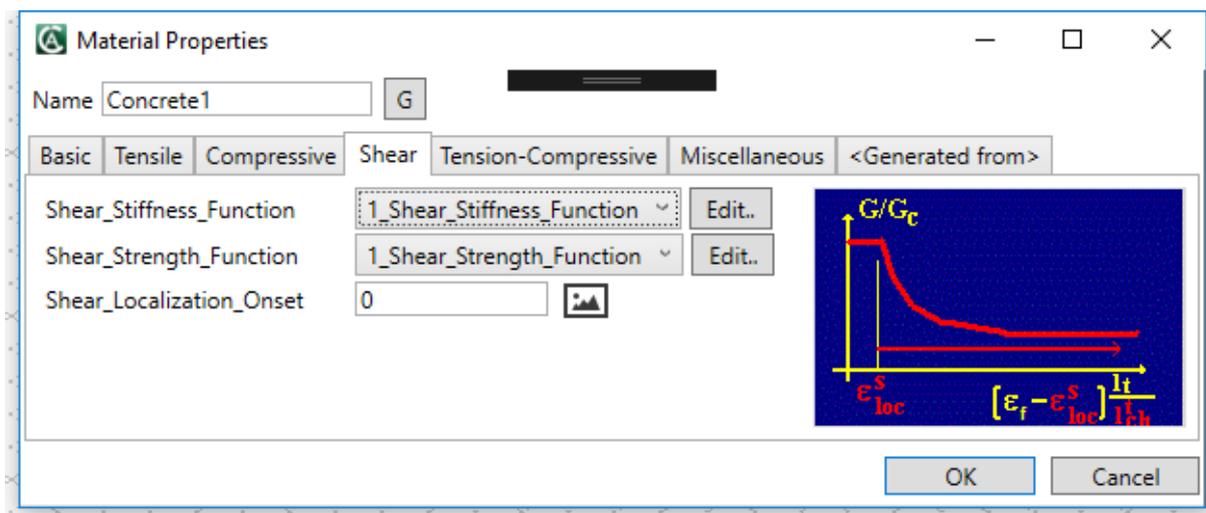


Fig. 2-30: Dialog Concrete User – Shear parameters

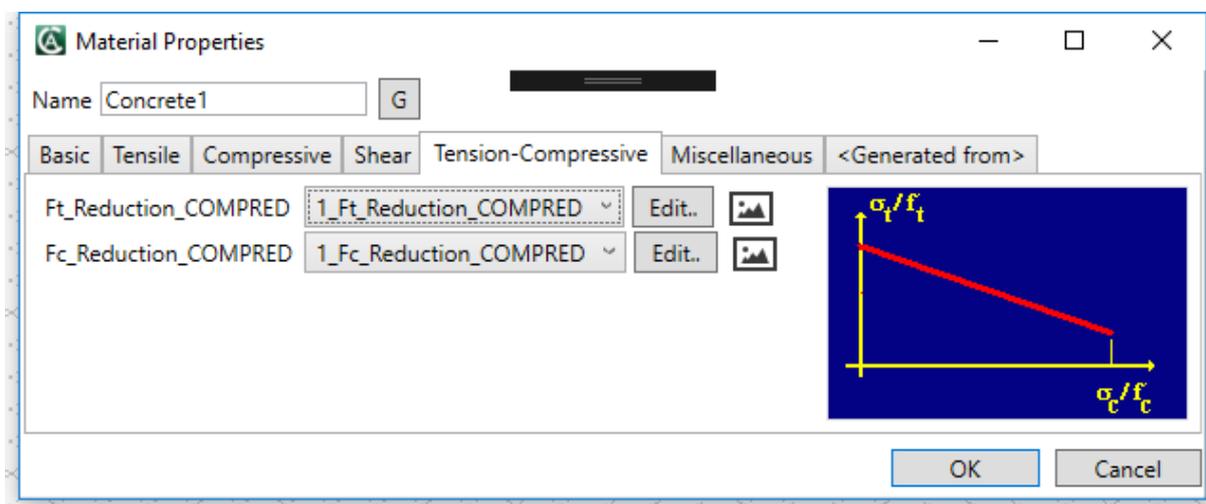


Fig. 2-31: Dialog Concrete User – Tension-Compressive parameters

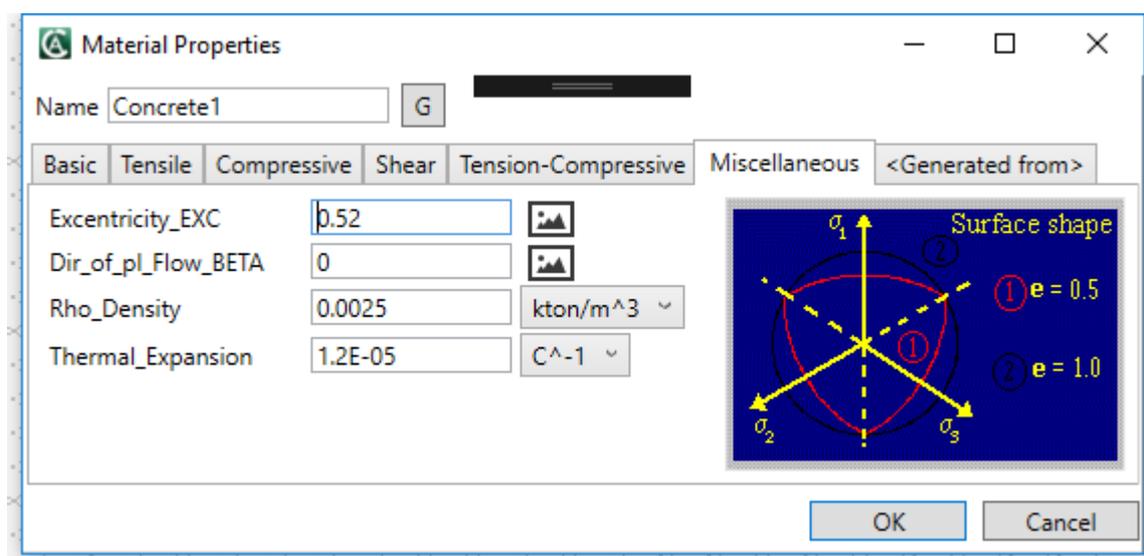


Fig. 2-32: Dialog Concrete User – Miscellaneous parameters

2.10.3 Concrete SHCC – Strain Hardening Cementitious Composite for Fibre Reinforced Concrete

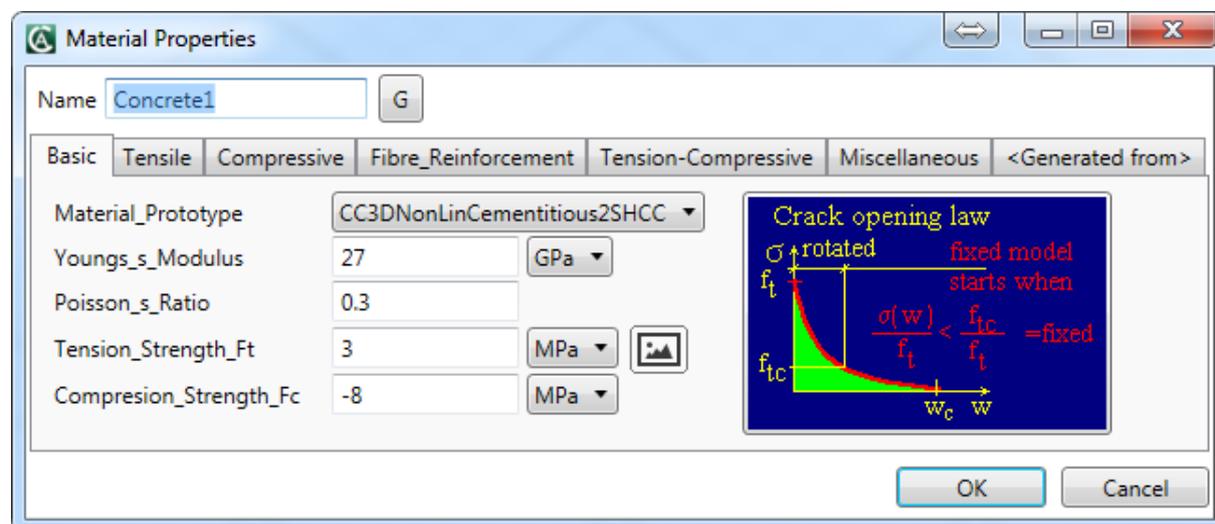


Fig. 2-33: Dialog Concrete SHCC – Basic parameters

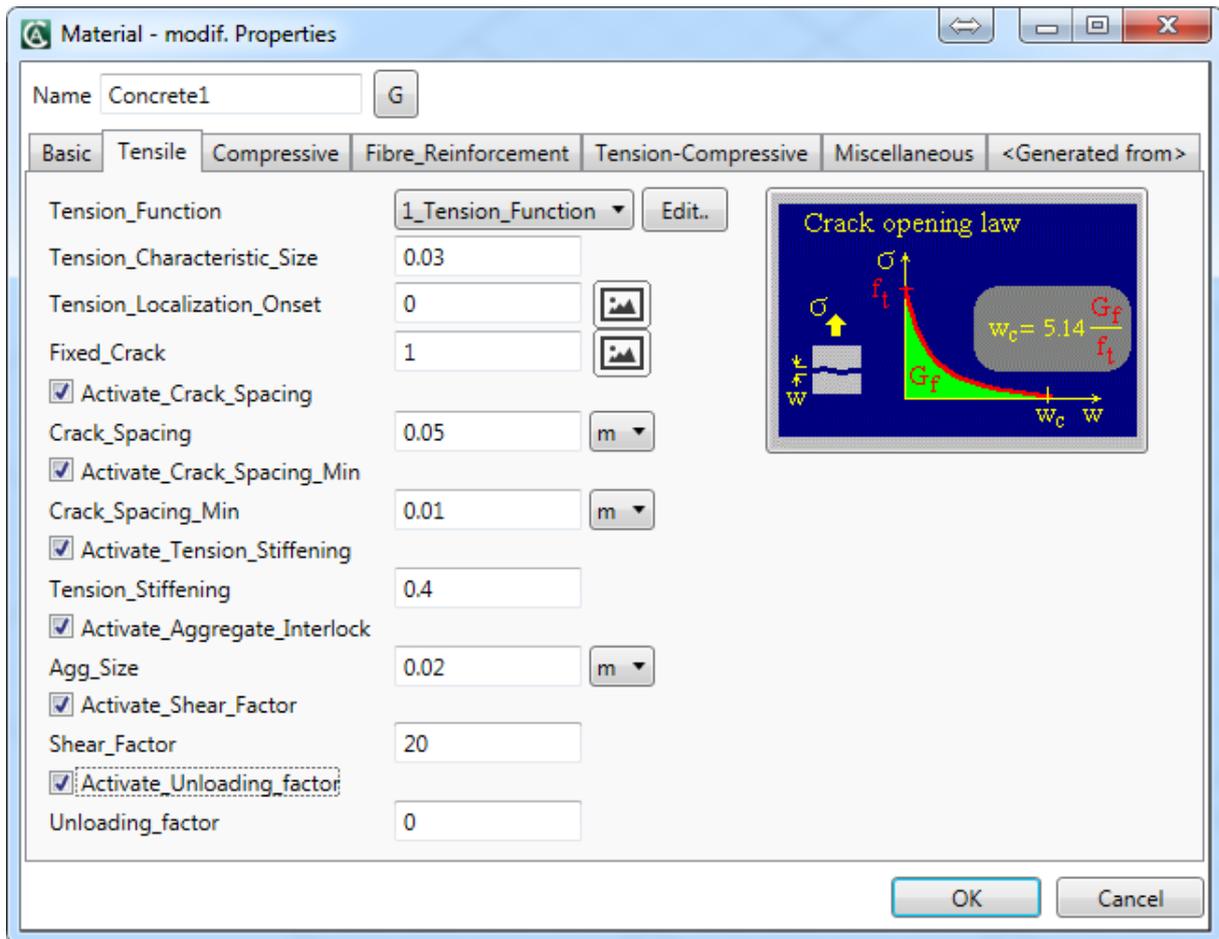


Fig. 2-34: Dialog Concrete SHCC – Tensile parameters

The *CC3DNONLINECEMENTITIOUS2SHCC* is suitable for fibre reinforced concrete, such as SHCC (Strain Hardening Cementitious Composites) and HPCFRCC or UHPFRCC (high and ultrahigh performance fiber reinforced concrete) materials.

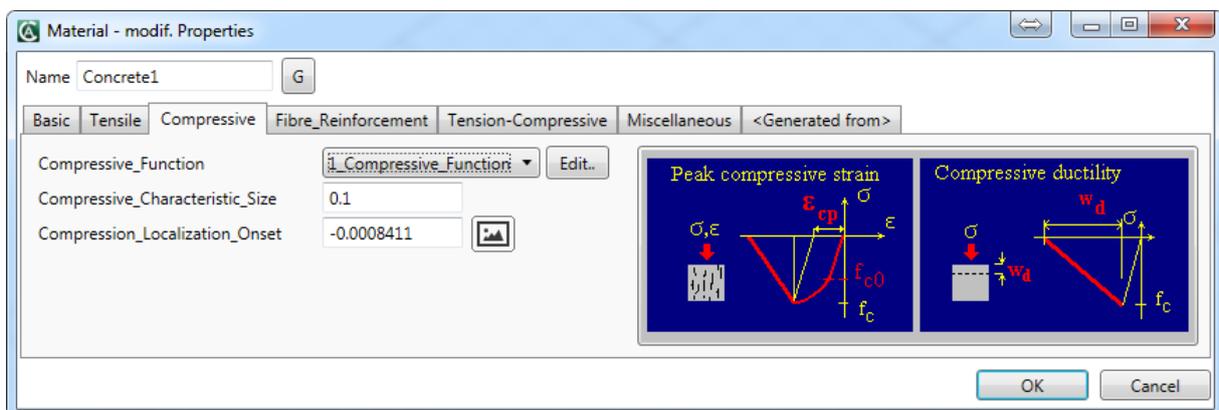


Fig. 2-35: Dialog Concrete SHCC – Compressive parameters

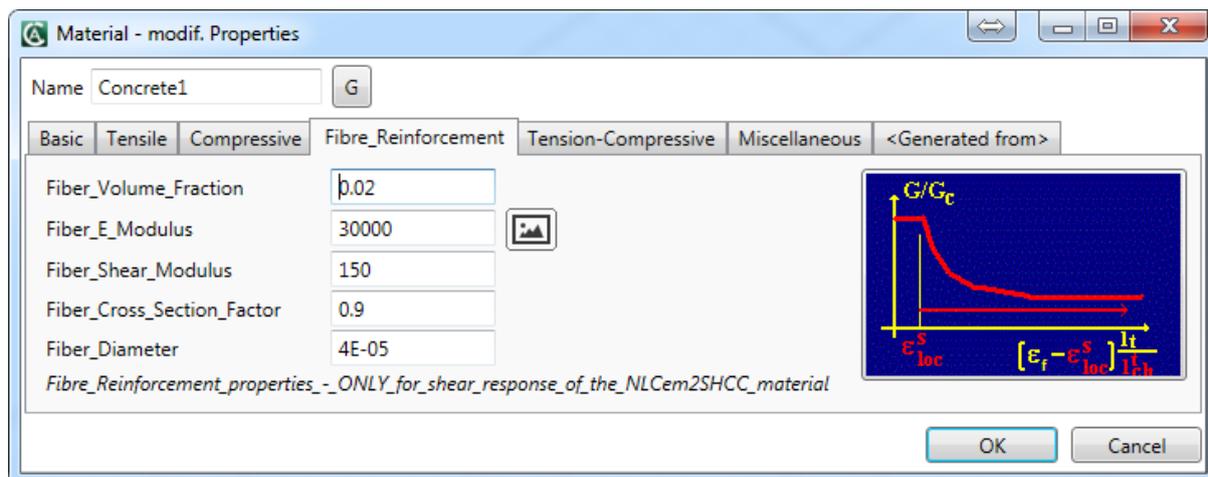


Fig. 2-36: Dialog Concrete SHCC – Fibre reinforcement parameters



Fig. 2-37: Dialog Concrete SHCC – Tension-Compressive parameters

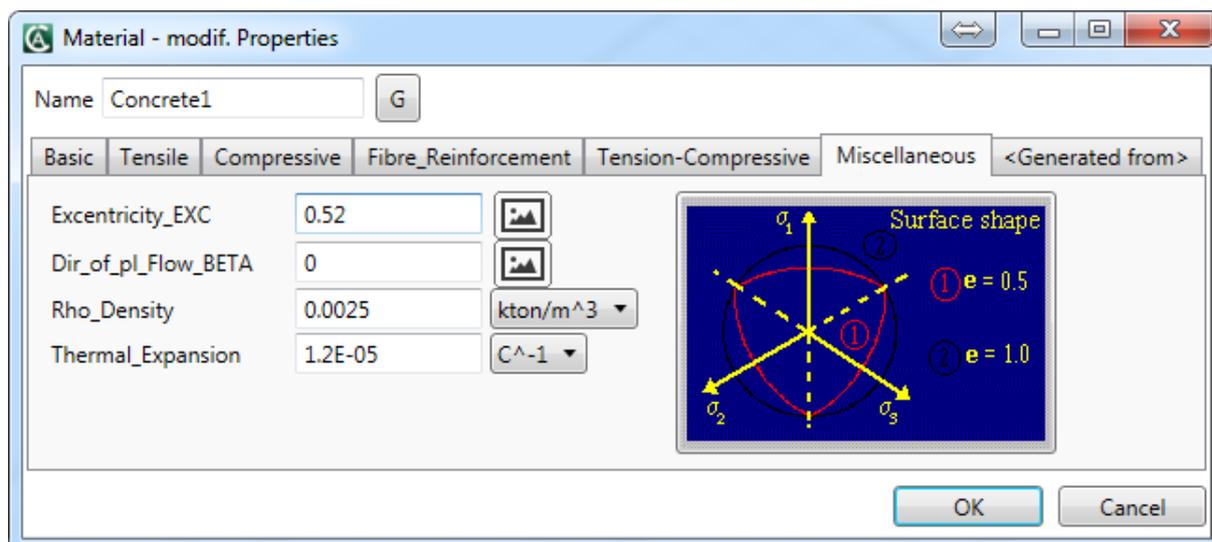


Fig. 2-38: Dialog Concrete SHCC – Miscellaneous parameters

2.1.1 Modelling of Structural Elements Using BIM Model

BIM (Building Information Modeling) often interpreted as **Building Information Management** is a term used for software tools, standards and data sets intended for creating, using and sharing information about buildings during their full lifetime from earliest conception to demolition. This include not merely projection and construction, but also building maintenance, facility management etc. There are many proprietary implementations of BIM idea and many proprietary data formats. Fortunately there is one BIM standard called **openBIM**.

The **openBIM** is an universal approach to the collaborative design, realization and operation of buildings based on open standards and workflows. This include open data formats and other standards for data exchange. openBIM is an initiative of **buildingSMART International Ltd.** and several leading software vendors using the open buildingSMART Data Model.

The **BIM** term used in following text is always meant as **openBIM** implementation if BIM abstract idea.

There are three most important standardized data formats that can be utilized in ATENA Studio PRE for import and export: IGES and STEP for CAD data exchange a and IFC for BIM data exchange.

IGES (Initial Graphics Exchange Specification) is the oldest and simplest neutral format for the exchange of CAD models. This standard is still used, despite the fact that it's last update was published in 1996.

STEP (Standard for the Exchange of Product model data) is more complex successor of IGES. It is standardized as ISO 10303-21 and constantly updated, with the last major update in 2016.

IFC (Industry Foundation Classes) is the core part of **openBIM** based on STEP standard. It is a platform neutral, open file format specification registered as an official international standard **ISO 16739:2013**. The latest version of IFC is **IFC4** but previous version **IFC2x3** is still used.

The **IFC** data standard is very complex so that subsets of its data are defined as **Model View Definitions (MVDs)**. These subsets must be supported by software tools to enable correct data exchange for desired purpose. For ATENA Studio PRE the most important are **IFC2x3 Coordination View MVD** and **IFC4 Design Transfer View MVD**. These MVDs define IFC subsets for exchanging data of construction structures.

The **IFC2x3 Coordination View** targets the coordination between the architectural, mechanical and structural engineering tasks during the design phase. The last version of this MVD is 2.0.

The **IFC4 Design Transfer View** is the successor of the IFC2x3 Coordination View and targets all work flows based on models that are handed over to perform in next work flows, allowing modifications of its content. Current version of this MVD is 1.0.

For **IFC2x3** there also exists the **Structural Analysis View** MVD for exchanging data of structural analysis. Applicability of this standard for ATENA was verified in CER by developing the **Ifc2AtenaConverter** software application. Unfortunately, the usability of Structural Analysis View MVD is very problematic and there are some serious disadvantages:

- Structural Analysis View doesn't exist for IFC4 and seems to be no longer developed
- data structures for structural analysis included in IFC are limited and enable only static analysis
- conversion of 1D or 2D structural elements included in IFC to their 3D geometric representation is very difficult and enable to create only very simple models
- there are many imperfections in IFC exported by third-party structural analysis software

For these reasons the Structural Analysis View is not supported by current version of ATENA Studio PRE, but may be implemented in the future, if some changes will occur.

2.12 ATENA in relation to BIM

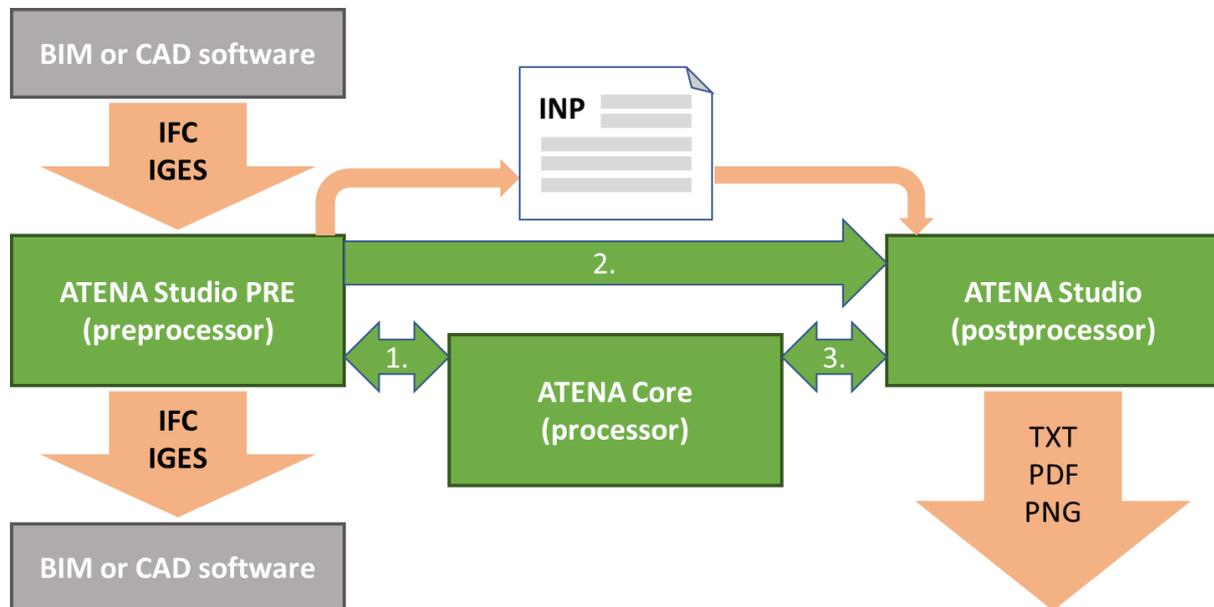


Fig. 2-39: Main ATENA modules and dataflow

ATENA Studio PRE workflow:

1. CAD or BIM model is imported to the ATENA Studio PRE, or
2. CAD model is created directly in the ATENA Studio PRE using its CAD functions.
3. CAD model is converted to BIM model by creating building and reinforcing elements (non-graphical properties only) and assigning them to graphical objects.
4. FE mesh parameters are set globally or can be refined for individual elements of geometric model.
5. Geometric model is sent through data channel 1 to the ATENA Core module to generate FE mesh.
6. FE mesh is sent back from core to the ATENA Studio PRE for preview and checking.
7. Materials are created and assigned to building and reinforcing elements.
8. Data of structural analysis are prepared and assigned to objects of geometric model.
9. Complete analysis model can be exported as ATENA INP text file for future use or directly sent to the ATENA Studio by data channel 2 for immediate computing.
10. Final ATENA Studio PRE project can be exported as BIM project (IFC file) or CAD project (IGES file)

As you can see, the ATENA Studio PRE is the only module that provide interoperability with other BIM applications.

2.13 IFC file import to the ATENA Studio PRE

The IFC file used for import test contains bottom of elevator shaft with 1 footing (IfcFooting), 4 walls (IfcWallStandarCase) and many reinforcing bars (IfcReinforcingBar) represented as mapped items (IfcMappedItem) of a few reinforcing bars prototypes. Each instance of reinforcing bar is created as cartesian transformation of it's prototype and represented as swept solid (IfcSweptDiscSolid) defined by rebar radius and polyline (IfcPolyline) directrix.

The IFC file was created in the **Nemetschek Allplan** application and exported as IFC2x3 Coordination View. This information and information about author you can see in the next picture.

```

Soubor Upravit Možnosti Kódování Nápověda 97 %
#21693= IFCCARTESIANPOINT((#21694,#21696,#21698);
#21694= IFCCARTESIANPOINT((29980.00000022435,22145.,820.8207377754645));
#21696= IFCDIRECTION((0.,0.,1.));
#21698= IFCDIRECTION((1.,0.,0.));
#21700= IFCLocalPLACEMENT(#71,#21693);
#21701= IFCPRODUCTDEFINITIONSHAPE($,$,(#21819));
#21702= IFCSHAPEREPRESENTATION(#34,'Body','AdvancedSweptSolid',(#21704)
);
#21704= IFCSWEPTDISKSOLID(#21705,6.,$,0.,1.);
#21705= IFCPOLYLINE((#21707,#21709,#21711,#21713,#21715,#21717,#21719,
#21721,#21723,#21725,#21727,#21729,#21731,#21733,#21735,#21737,#21739,
#21741,#21743,#21745,#21747,#21749,#21751,#21753,#21755,#21757,#21759,
#21761,#21763,#21765,#21767,#21769,#21771,#21773,#21775,#21777,#21779,
#21781,#21783,#21785,#21787,#21789,#21791,#21793,#21795,#21797,#21799,
#21801,#21803,#21805,#21807,#21809,#21811,#21813,#21815,#21817));

```

Fig. 2-40: Header of IFC file used for import

```

Soubor Upravit Možnosti Kódování Nápověda 0 %
ISO-10303-21;
HEADER;FILE_DESCRIPTION(('IFC2x3 Coordination View'),'2;1');
FILE_NAME('export.ifc',
'2014-04-14T14:30:50',('Hahnebach Bernd'),(''),
'The EXPRESS Data Manager Version 5.02.0100.02 : 29 May 2013',
'Allplan 2014.0 23.01.2014 - 15:01:43', '');
FILE_SCHEMA(('IFC2X3'));
ENDSEC;

DATA;
#1= IFCCARTESIANPOINT(#32,$,$);
#4= IFCOWNERHISTORY(#20,#23,$,ADDED,$,$,$,1397478648);
#5= IFCPROJECT('1KvpqAbxiHuu6qT4QWih1b',#4,
',,$,$,$,($11,
#38),#42);
#11= IFCGEOMETRICREPRESENTATIONCONTEXT($,'Model',3,1.000000000000000E-5,
#1,$);
#14= IFCPERSON($,'Bernd','Hahnebach',$,$,$,$);
#16= IFCORGANIZATION($,'Nemetschek Allplan GmbH',$,$,$);

```

Fig. 2-41: Definition of reinforcing bar prototype

```

Soubor Upravit Možnosti Kódování Nápověda 98 %
#21819= IFCSHAPEREPRESENTATION(#34, 'Body', 'MappedRepresentation', (#21823
, #21834, #21844, #21854, #21864, #21874, #21884, #21894, #21904, #21914, #21924,
#21934, #21944, #21954, #21964));
#21821= IFCREPRESENTATIONMAP(#1, #21702);
#21823= IFCMAPPEDITEM(#21821, #21825);
#21825= IFCCARTESIANTRANSFORMATIONOPERATOR3D(#21828, #21830, #21826, $,
#21832);
#21826= IFCCARTESIANPOINT((0., 0., 0.));
#21828= IFCDIRECTION((1., 0., 0.));
#21830= IFCDIRECTION((0., 1., 0.));
#21832= IFCDIRECTION((0., 0., 1.));
#21834= IFCMAPPEDITEM(#21821, #21835);
#21835= IFCCARTESIANTRANSFORMATIONOPERATOR3D(#21838, #21840, #21836, $,
#21842);
#21836= IFCCARTESIANPOINT((150., 0., 0.));
#21838= IFCDIRECTION((1., 0., 0.));
#21840= IFCDIRECTION((0., 1., 0.));
#21842= IFCDIRECTION((0., 0., 1.));
#21844= IFCMAPPEDITEM(#21821, #21845);
#21845= IFCCARTESIANTRANSFORMATIONOPERATOR3D(#21848, #21850, #21846, $,
#21852);
    
```

Fig. 2-42: Definition of reinforcing bar as mapped item

Tested IFC file was previewed in the **xBIM Xplorer** viewer. The xBIM Xplorer is a part of the **xBIM Toolkit** which libraries are used by ATENA Studio PRE for IFC input and output. Therefore the geometric representations of IFC model in the xBIM Xplorer and in the ATENA Studio PRE must exactly match.

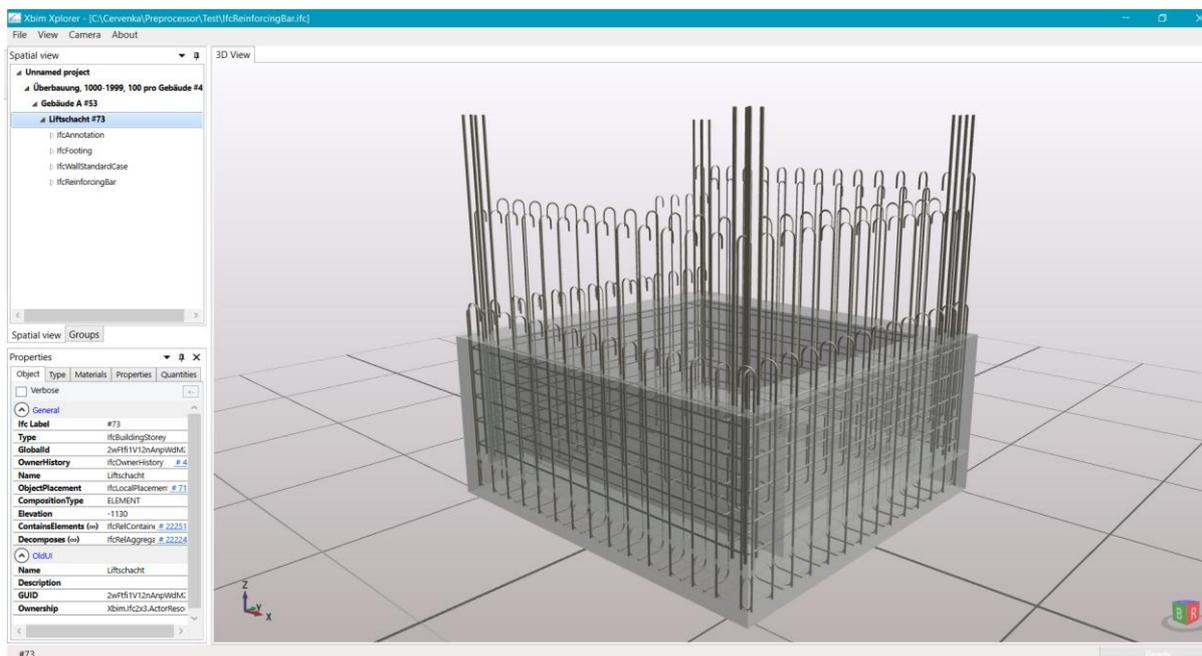


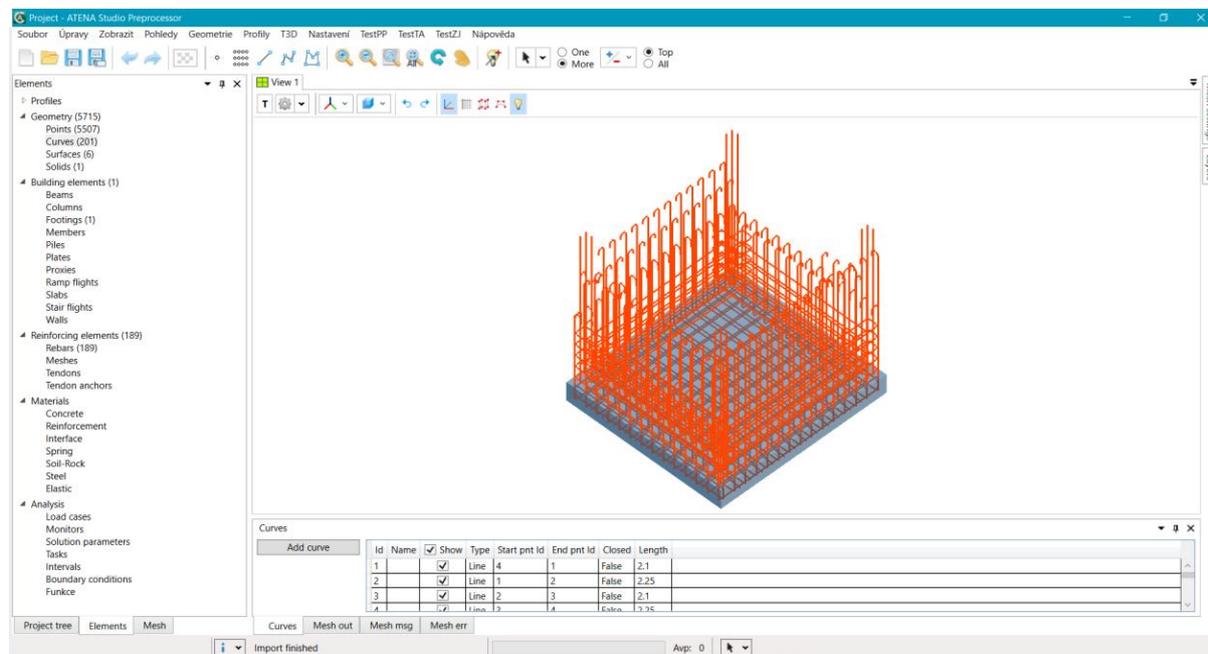
Fig. 2-43: IFC test file previewed in the xBIM Xplorer

IFC was successfully imported to the ATENA Studio PRE. All 189 reinforcing bars are present and correctly displayed.

Note 1: The walls were excluded during import.

Note 2: Reinforcing bars are displayed as curves. Displaying them as solids is optional in ATENA Studio PRE.

Some details of imported rebars are shown in the next picture. This is the test version of reinforcing bars details used mainly for testing of nominal and geometric diameter and cross



section area. While both diameters match, the cross section areas computed from diameter shall be 2.54E-04 and 1.131E-4. There was detected incorrect usage of area units in imported IFC.

Name, steel grade and length values are optional in IFC and none of them were defined. Reinforcing bar type must be one of IfcReinforcingBarRoleEnum values. Here all types are set to NotDefined.

Id	Name	Steel Grade	Nominal D	Geometric D	CS Area	Length	Type
13			0.018	0.018	2.54E-06		NotDefined
14			0.018	0.018	2.54E-06		NotDefined
15			0.018	0.018	2.54E-06		NotDefined
16			0.018	0.018	2.54E-06		NotDefined
17			0.012	0.012	1.131E-06		NotDefined
18			0.012	0.012	1.131E-06		NotDefined
19			0.012	0.012	1.131E-06		NotDefined

Fig. 2-45: Details of imported reinforcing bars

2.14 IFC data compatibility in ATENA Studio PRE

ATENA Studio PRE module is developed with respect to compatibility with the IFC2x3 Coordination View and the IFC4 Design Transfer View. Internal data structures enable IFC import and export of ATENA Studio PRE project as BIM project.

Some internal structures (mainly geometric entities) not included in these MVDs are enabled to be exported as converted to entities included in these MVDs without loss of data.

For import from and export to IFC files ATENA Studio PRE uses software libraries of open source **xBIM Toolkit**, which fully implements the IFC2x3 and IFC4 standards.

The most important IFC data groups used in ATENA are:

- materials
- building elements
- reinforcing elements
- profiles
- geometric entities

2.14.1 Materials

Data structures of materials used in ATENA are very complex and enhanced in comparison with IFC materials. For that reason there is no direct compatibility between ATENA and IFC materials and can't be enabled not even by using IFC property sets. So that only base material properties can be imported or exported.

In oposite to IFC where materials can be assigned to profiles, in the ATENA Studio PRE materials can be assigned only to building or reinforcing elements.

2.14.2 Building elements

Building elements defined in IFC are supported by ATENA Studio PRE with following exceptions:

- Voids filling elements (IfcDoor, IfcWindow...) and non-load bearing elements (IfcCovering, IfcCurtainWall...) are not yet implemented, but can be added for the purpose of thermal analysis in the future.
- "StandardCase" building elements are defined in IFC as elements of single solid geometry and internal layers of different materials defined by thickness. In ATENA these elements must be converted to one or more single instances of base type by materials.
- Complex elements (IfcRoof...) are not supported as whole, but only as their single components.

In the ATENA Studio PRE project each building element must have single geometric representation (unique or mapped) and must have material assigned.

2.14.3 Reinforcing elements

Reinforcing bars, tendons, meshes and anchors are fully supported by ATENA Studio PRE. Anchors can be applied as point elements with anchor properties or as fully defined elements including it's geometry.

In the ATENA Studio PRE project each reinforcing elements (as subtype of building element) must have single geometric representation (unique or mapped) and must have material assigned.

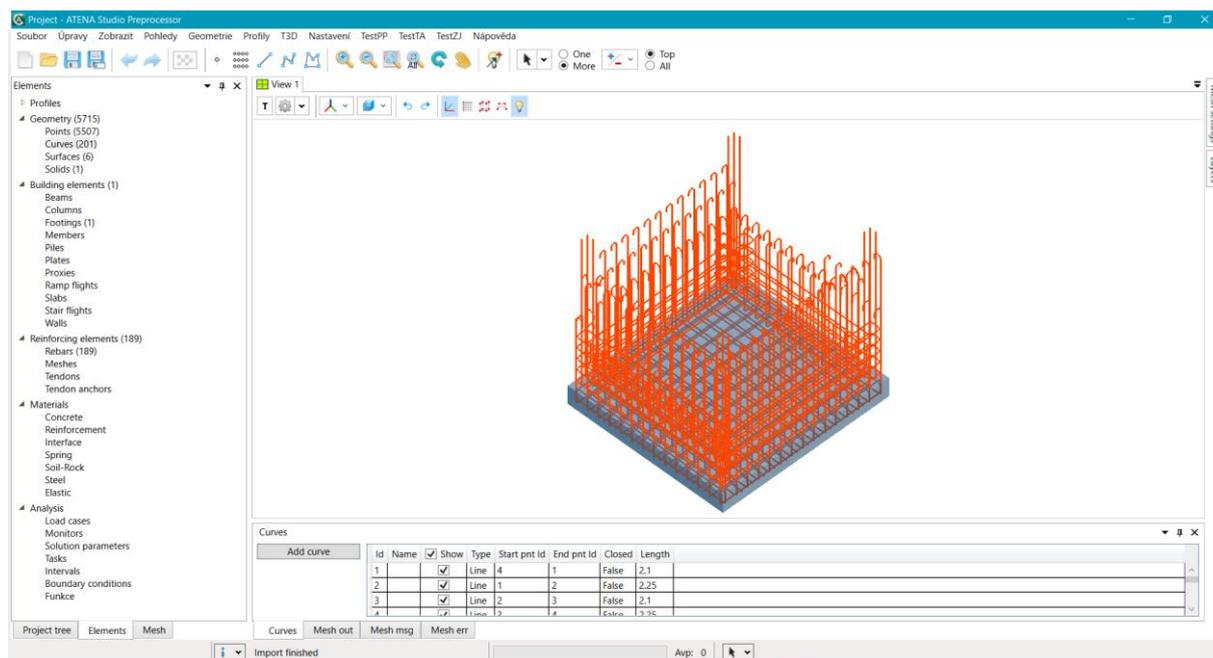


Fig. 2-46 Reinforcing bars imported from IFC to ATENA Studio PRE

2.14.4 Profiles

Profiles are one of the core parts of IFC. All swept solids use area profiles and all swept surfaces use curve profiles.

There are two groups of profiles: parametric and user defined.

Parametric profiles are supported in ATENA Studio PRE by its built-in profile generator. Besides IFC defined profiles this generator supports some specific profiles. ATENA Studio PRE specific profiles can be simply exported as user defined profiles.

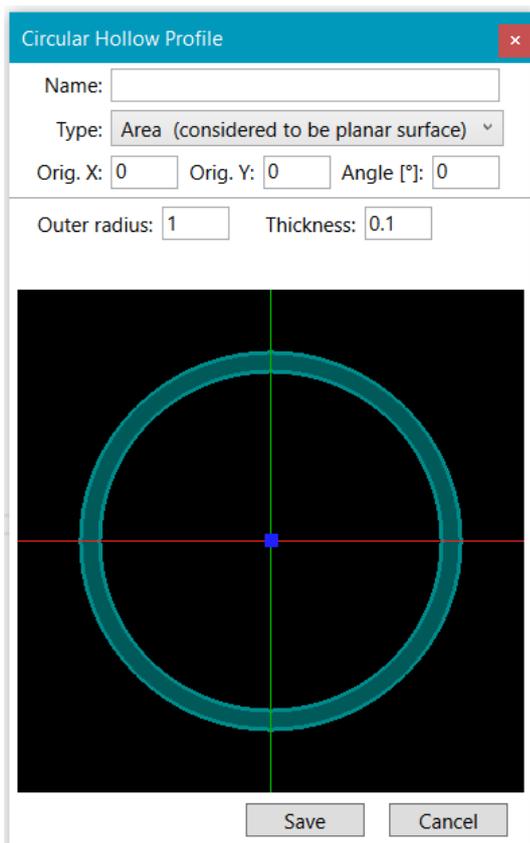


Fig. 2-48: Profile generator. IFC standard profile

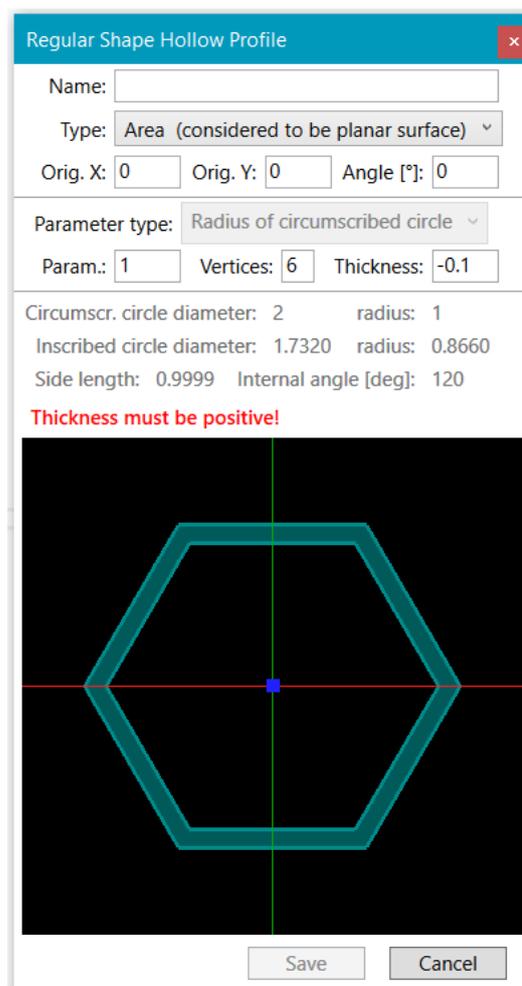


Fig. 2-47: Profile generator. ATENA Studio PRE parametric profile

Besides of parametric or user defined profiles ATENA Studio PRE supports direct sweeping of curves or areas. This way generated geometric objects are also enabled to be exported as based on user defined profiles.

There are plans to enable store and retrieve libraries of predefined or user defined profiles in the future.

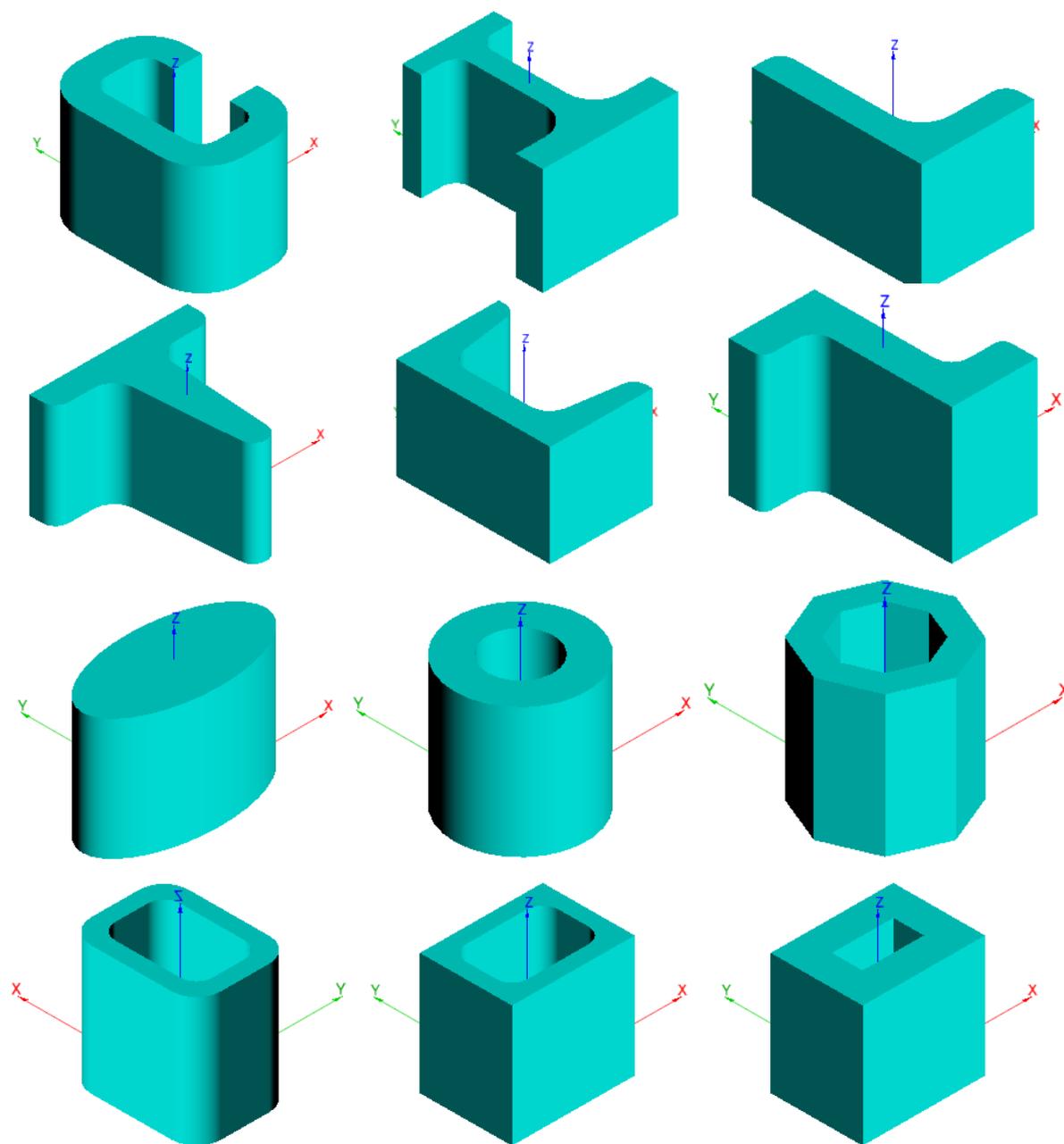


Fig. 2-49: Examples of parametric profiles supported by ATENA Studio Pre

2.14.5 Geometry

In IFC, spatial objects can be defined by their geometric or topologic representation. ATENA Studio PRE uses only geometric representations. Topologic representations are automatically converted to geometric during import of IFC.

The design of geometric data structures of the ATENA Studio PRE had to take into consideration entirely different requirements of ATENA Core and IFC. ATENA Core require all solids defined as BREPs (boundary representations) with limited set of curves used as edges. So that the ATENA Studio PRE save all geometric objects as they were created or imported (as BREP, by parameters, swept solids or surfaces etc.) and BREPs required by ATENA Core are generated just before sending data to it by ATENA Studio PRE.

The ATENA Studio PRE internally utilizes for most of graphic data the open source **OpenNURBS** library. This library will enable in the future import of CAD models created in the **Rhinoceros** application in it's native format.

Geometry – curves

The ATENA Core module supports only two types of curves:

- rational Bézier curve
- NURBS curve

All other curves must be converted to one of these types. For instance a line is interpreted as linear Bézier curve.

ATENA Studio PRE internally differentiates these groups of curves:

- directly usable as edges in ATENA Core (line, Bézier curve, NURBS curve)
- directly convertible to Bézier curve (circular, elliptic, parabolic and hyperbolic arc)
- closed conic sections (circle, ellipse), whose must be for ATENA Core divided into segments represented by Bézier curves
- complex curves (polyline, polycurve), whose are sent to the ATENA Core as segments of supported types and can be exported as simple or complex curves in dependence of the target file format

The ATENA Studio PRE implements all types of IFC curves.

ATENA Studio PRE implements all types of curves contained in MVDs. Additional curves used internally (parabolic or hyperbolic arc, Bézier curve excluded from IFC4) can be exported to IFC2x3 as rational Bézier curves and to IFC4 as NURBS curves.

Geometry – surfaces

ATENA Studio PRE implements almost all types of surfaces contained in MVDs.

An important exception are surfaces derived from `IfcTessellatedItem` entity. These surfaces are defined by their surface mesh and there is no possibility of automatic conversion to other surface type. Import and visualization of these surfaces is possible and not difficult, but there are serious problems to use them in the ATENA Core. Limited support will be added to the ATENA Studio PRE in some of future versions. In that case all tessellated surfaces will need to be manually “redrawn” to some of simplest surfaces.



Fig. 2-50: Testing objects – parametric elementary surfaces

Geometry – solids

IFC enable to define solids by many different ways – as parametric definition of 3D primitives, as BREPs (Boundary representations), as set of surfaces, and as swept area profiles. Most of them is supported by ATENA Studio PRE.

Boolean operations with solids are not supported in current version of ATENA Studio PRE and this support will be added in the future.

Other problems may occur with more complex solids, whose can be successfully imported form IFC, but their conversion to the BREP representation acceptable by ATENA Core may not be possible.

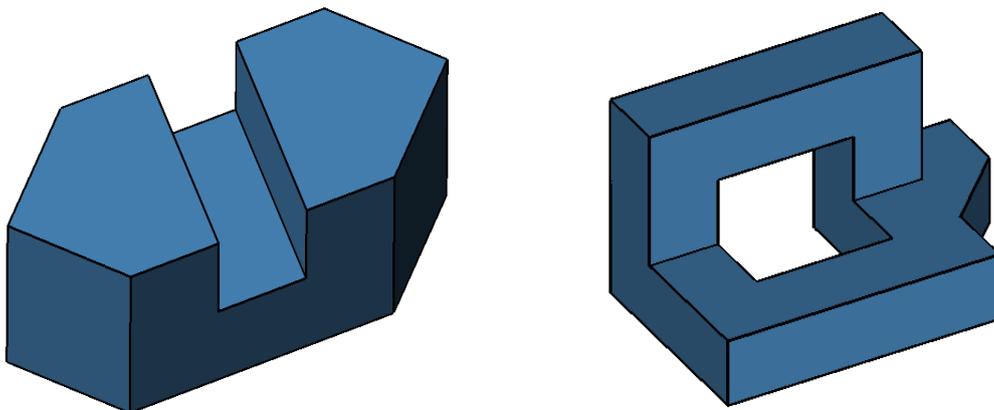


Fig. 2-51 Testing objects – solids representing as faceted breps

3. FRC Model Parameters Identification

The tensile response of FRC differs from normal concrete not only in the values like tensile strength and especially fracture energy, but also in the shape of tensile softening branch. The original exponential function valid for normal concrete can be used as a first approach, but preferably would be to use more realistic form of the tensile constitutive law. Therefore, special material models at macroscopic level are needed for modelling of fibre reinforced concrete.

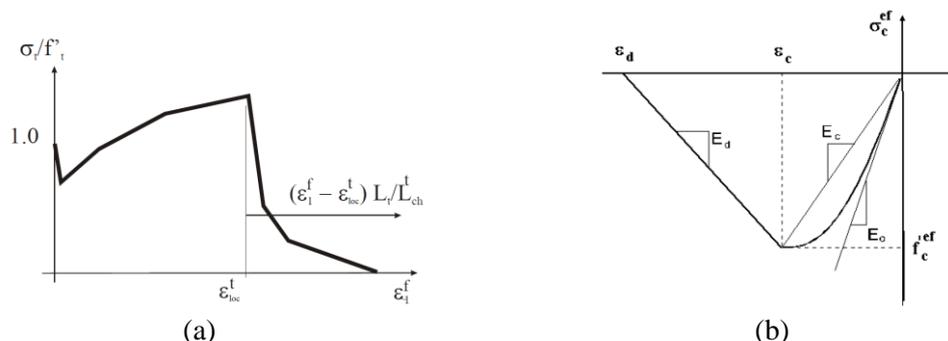


Fig. 3-1: (a) User defined tensile behaviour, (b) compressive stress-strain law

The most sophisticated and most general model of FRC material represents an extension to the fracture-plastic constitutive law [15] called 3D NLC2 User model. It describes the tensile behaviour (Fig. 3-1) according to the material response measured in tests point-wise in terms of the stress-strain relationship. The first part of the diagram is the usual stress-strain constitutive law. After exceeding the localization strain ϵ_{loc} the material law assumed for the characteristic crack band width L_{ch} is adjusted to the actual crack band width L_t . The characteristic crack band width (characteristic length) is the size (length) for which the defined material law is valid. The same procedure (with eventually different characteristic length) is used for the compression part of the material law. Compressive stress-strain law of mentioned material models is described in Fig. 3-1. The softening law in compression is linearly descending and the end point of the softening curve is defined by plastic displacement w_d (corresponding to ϵ_d in Fig. 3-1b). By increasing material parameter w_d the contribution of the fibres to the compressive behaviour of concrete is considered. Another important compressive parameter for FRC modelling is reduction of compressive strength due to cracks which says how the strength is reduced while the material is subjected to lateral tension.

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.

Example data from four point bending test (load-displacement diagrams, see Fig. 3-3), compression test (compressive strength) and Young's modulus are usually available and should be used as a basis for the determination of FRC parameters for ATENA analysis. Usually also a concrete class and fibre types are known. For the sake of clear presentation the following parameters will be assumed in the subsequent examples and figures in this chapter:

concrete class C110/130 and steel fibers BASF Masterfiber 482 with volume fraction 1.5 %. The fibers length is 13 mm and the diameter is 0.2 mm.

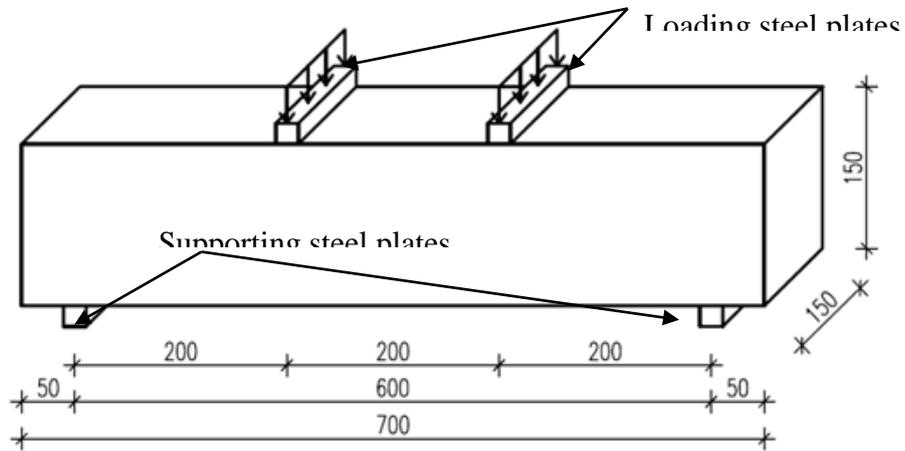


Fig. 3-2: Geometry of the structure for inverse analysis of material model parameters

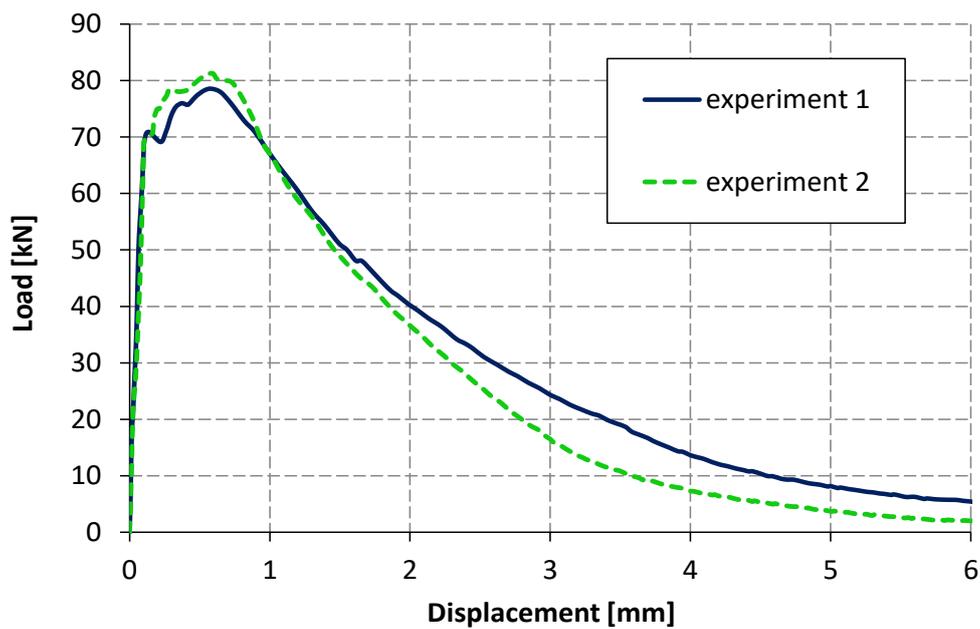


Fig. 3-3: Measured load-displacement diagrams of laboratory tests

Tab. 3-1: Available material parameters

Material parameter [unit]	Value
Cylindrical compressive strength [MPa]	125
Young' s modulus [MPa]	45000

3.1 Simplified approach for FRC parameter identification

As a representative document, RILEM TC162-TDF is chosen for determination of the material law. Experimental programme described in previous chapter involves the same test procedure and specimen geometry that is described in RILEM. Flexural tensile strengths $f_{R,i}$ are determined Based on Bernoulli-Navier hypothesis by expression:

$$f_{R,i} = \frac{3F_{R,i}L}{2bh_{sp}^2} \quad (1)$$

where b is width of the specimen,
 h_{sp} is distance between tip of the notch and top of cross section,
 L is span of the specimen,
 $F_{R,i}$ is load recorded at $CMOD_i$.

Maximal flexural strength $f_{f_{cm,fl}}$ and residual flexural strengths for crack mouth opening displacement $CMOD$ 0.5 mm ($f_{R,1}$) and 3.5 mm ($f_{R,4}$) are used for the determination of material law, see Fig. 3-4left. Stress-strain diagram defined by RILEM is trilinear, for the numerical model part after the peak is important. Final diagram utilized in the numerical analysis contains bilinear softening as is shown in Fig. 3-4right.

The finite element model for bending test is made for a plane stress simplification, with low order quadrilateral elements with 2x2 integration scheme, with the square elements shape and size of 5 mm, i.e. 30 elements through the height (25 elements above notch), see Fig. 3-5a. The loading is applied by force on the top loading plate. $CMOD$ is calculated as difference between horizontal displacement of the right and left bottom part of the notch. Characteristic length for tensile stress-strain diagram is equal to the element size, i.e. 5 mm.

Comparison of the load-displacement diagram from test and numerical simulation is shown in Fig. 3-5b. Model can correctly describe behavior on the tail of the diagram but there are differences after the crack localization. Model according to RILEM underestimate the flexural strength of the material.

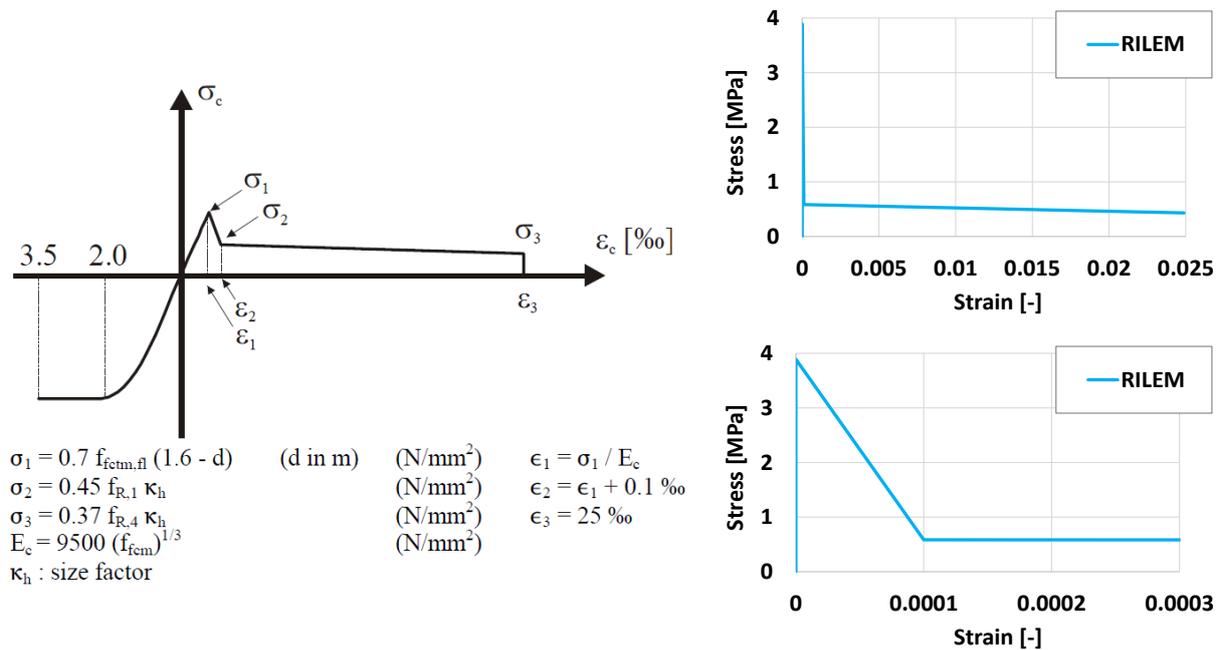


Fig. 3-4: (left) Stress-strain diagram according to RILEM TC162-TDF [7], (right top) diagram for material with fibre dosage 2 kg/m³, (right bottom) detail of the diagram until strain 0.0003

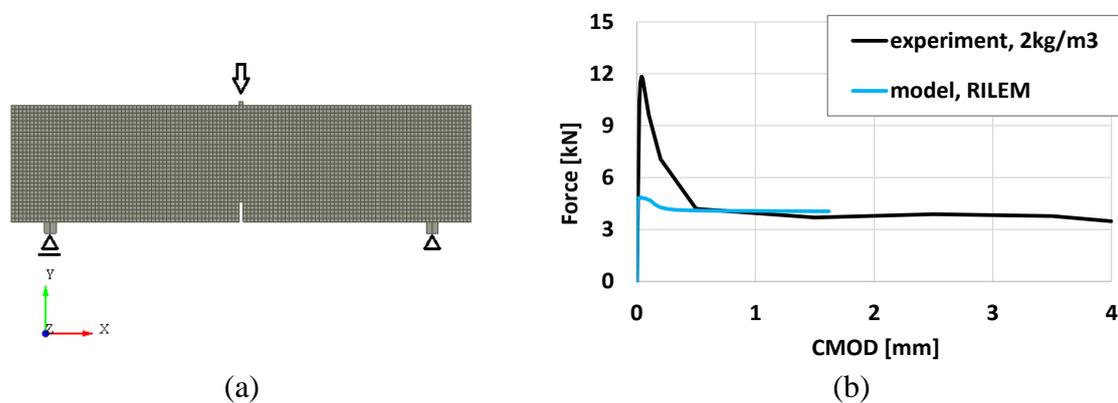


Fig. 3-5: (a) FEM model of three point bending test, (b) comparison of experimental result for fibre dosage 2 kg/m³ and model with RILEM material law (characteristic length 5 mm)

3.2 Manual inverse analysis for FRC parameter identification

This process is describe in more detail in the tutorial [24]. Here only the main principles are presented and described. The material modification starts by selecting the **Define SOLID**

Concrete material icon  or with the command **Data | Materials | SOLID Concrete** in main menu of ATENA GiD preprocessor. After selecting this command, the window for the definition of the **SOLID Concrete** appears (see Fig. 3-6). The prepared material called FRC beam should be selected (see Fig. 3-7) and then its parameters can be changed.

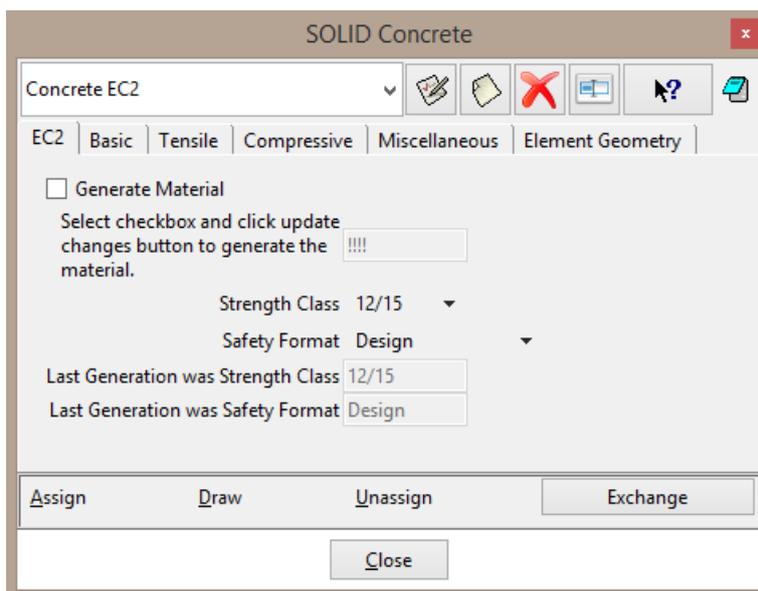


Fig. 3-6: The window for the definition of the SOLID Concrete

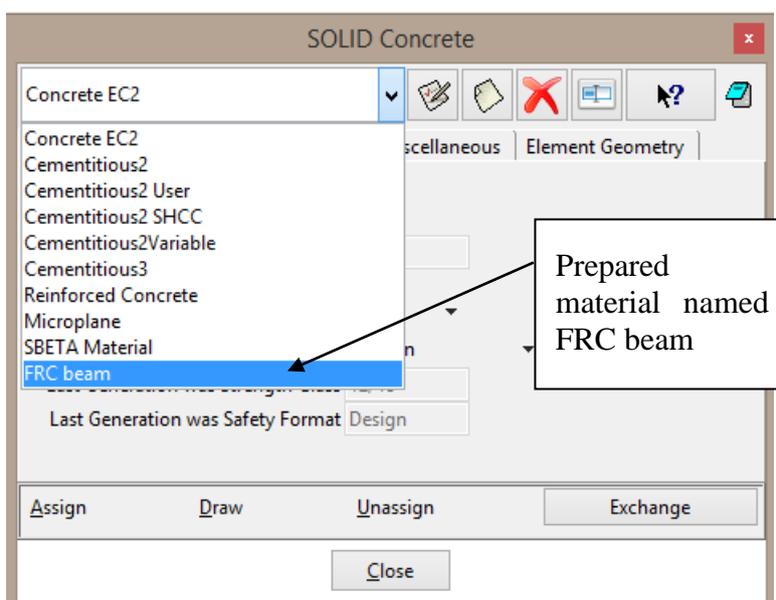
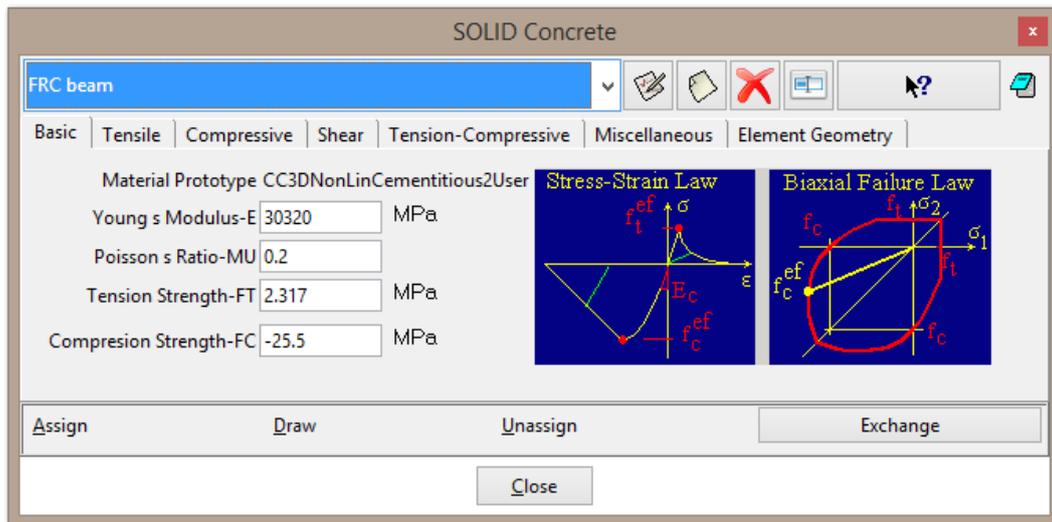
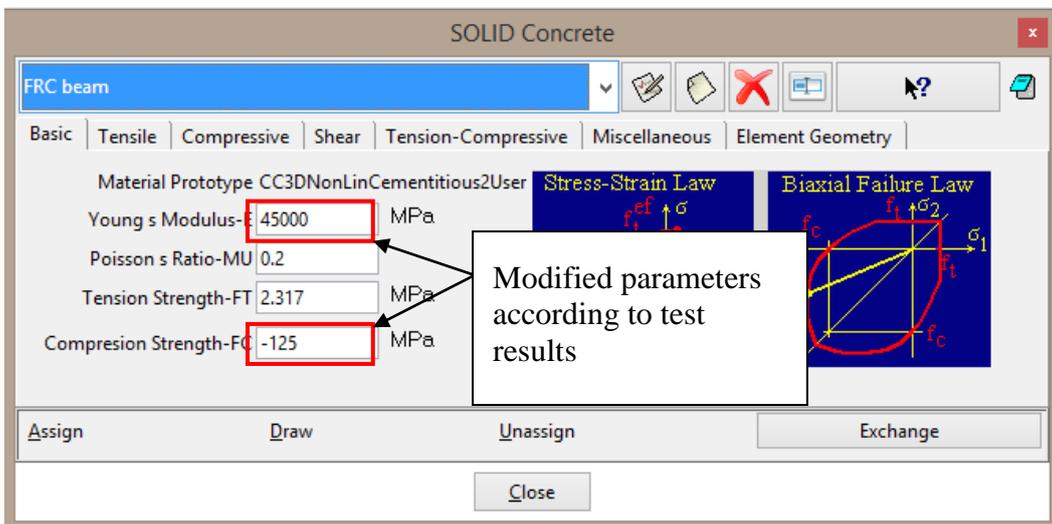


Fig. 3-7: The selection of the FRC beam material

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 Tab. 3-1. The data before and after the modification are depicted in Fig. 3-8. The modified parameters are saved by clicking on the **Update Changes** icon .



(a)



(b)

Fig. 3-8: 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 σ :

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

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 [1] 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 [11]. 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 [1]. It 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 Fig. 3-3. The cross section dimensions are 150 x 150 mm and span of the beam is 600 mm (Fig. 3-2). The tensile strength calculated by formula (4) is 12.4 MPa and this value is used in the model, see Fig. 3-7. The tensile strength is quite high but considering high compressive strength 125 MPa it is possible.

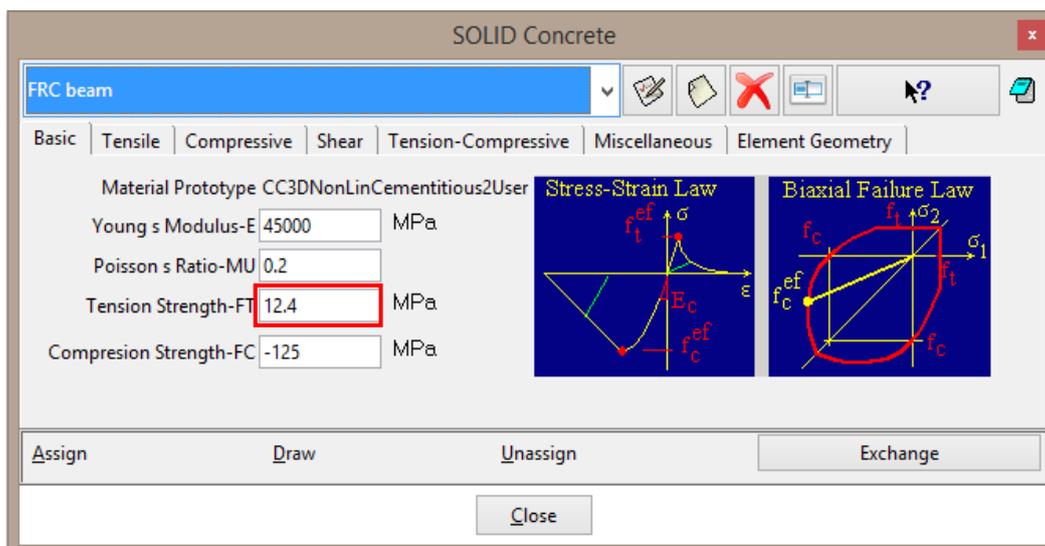


Fig. 3-9: The modification of the tensile strength

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:

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

where ε_f is fracture strain, w_c is crack opening and L_t is characteristic length. More information can be found at ATENA Theory Manual [15]. The characteristic length is a material parameter and it should be equal to the size of the element, in this case 0.015 m (Fig. 3-11). Then, the tensile function describes the evolution of tensile stresses in one finite element after crack formation.

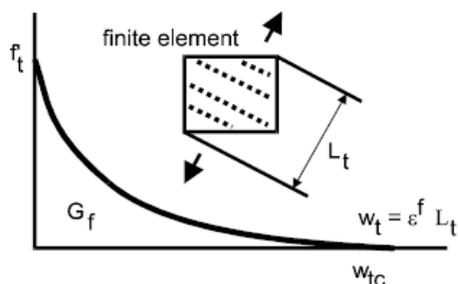


Fig. 3-10: Tensile softening and characteristic length **Chyba! Nenalezen zdroj odkazů.**

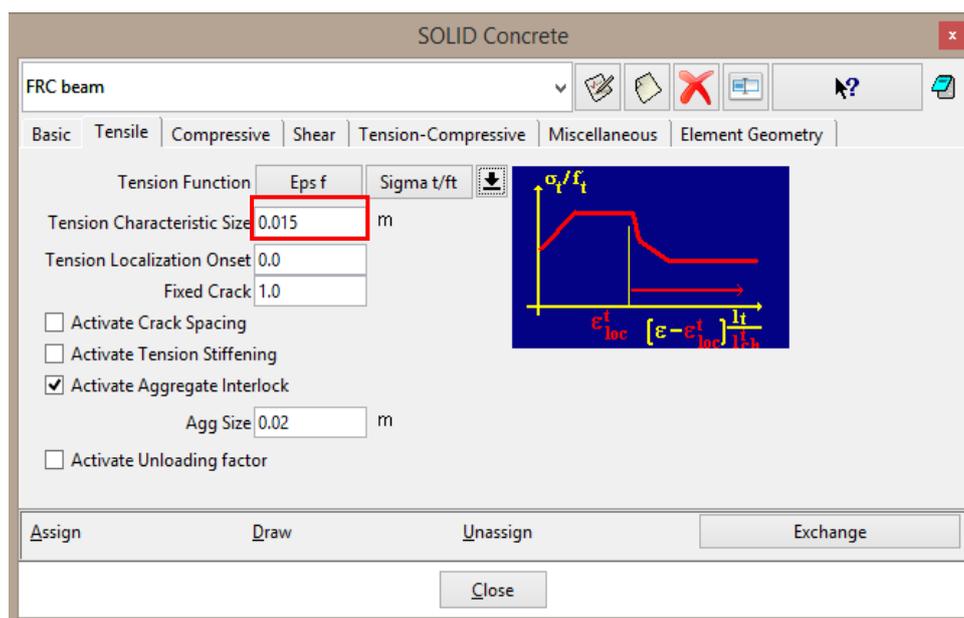


Fig. 3-11: Modification of tension characteristic length

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 (Fig. 3-13). The horizontal values are maintained because it is not clear now how to modify them.

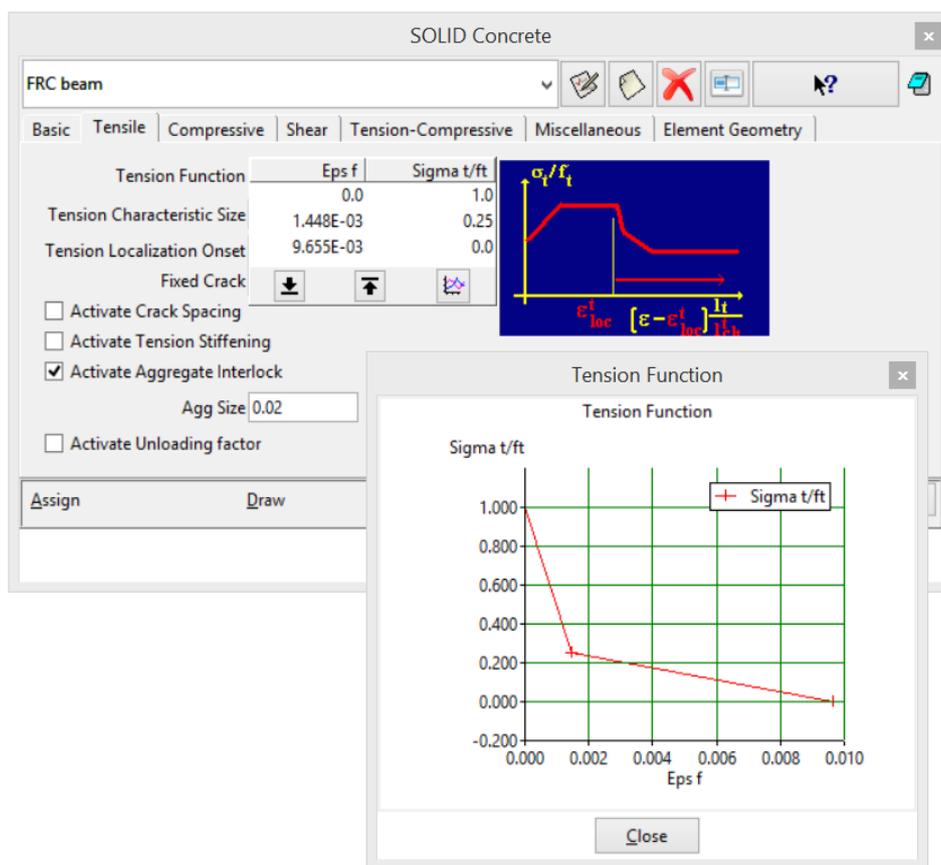


Fig. 3-12: Default tensile function

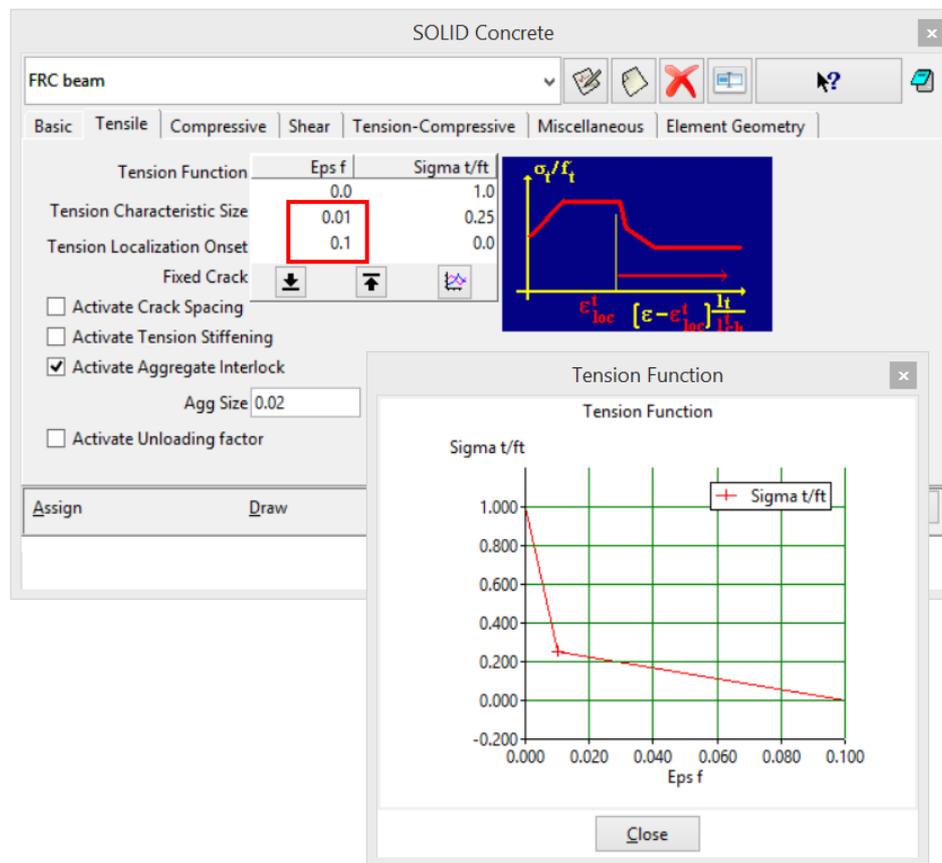


Fig. 3-13: Modified tensile function

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 6. 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 Fig. 3-14.

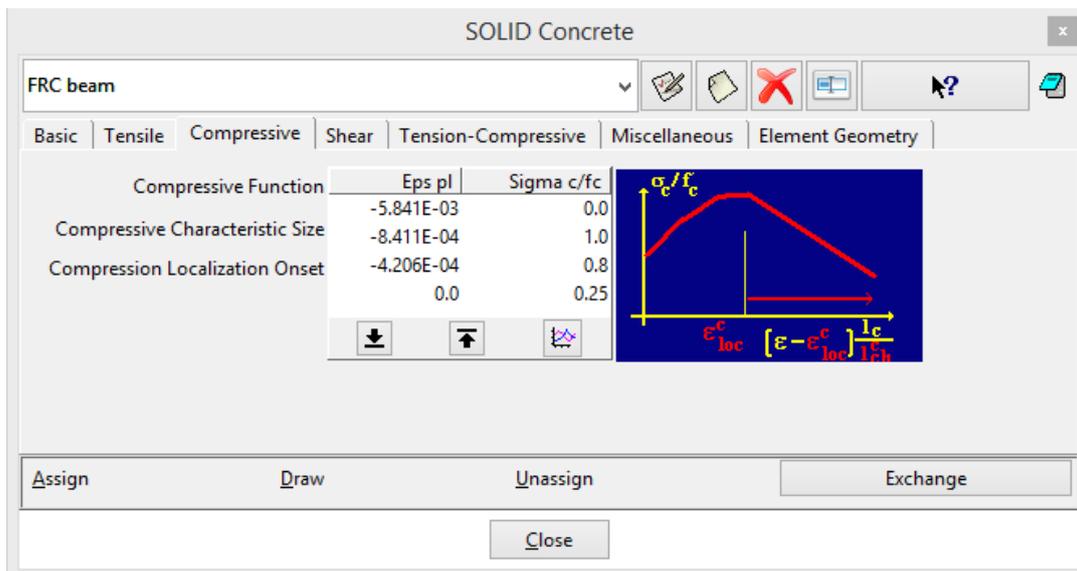


Figure 6: Default compressive function

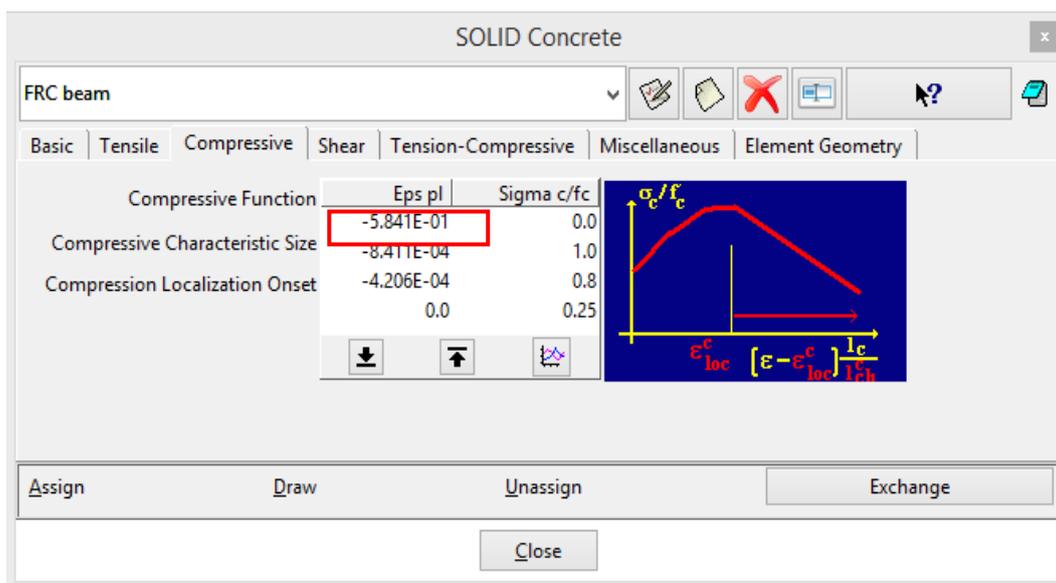


Fig. 3-14: Modified compressive function

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 Fig. 3-15. Based on the developers experience, this reduction should be deactivated by adjusting the function to the constant value 1, see Fig. 3-16.

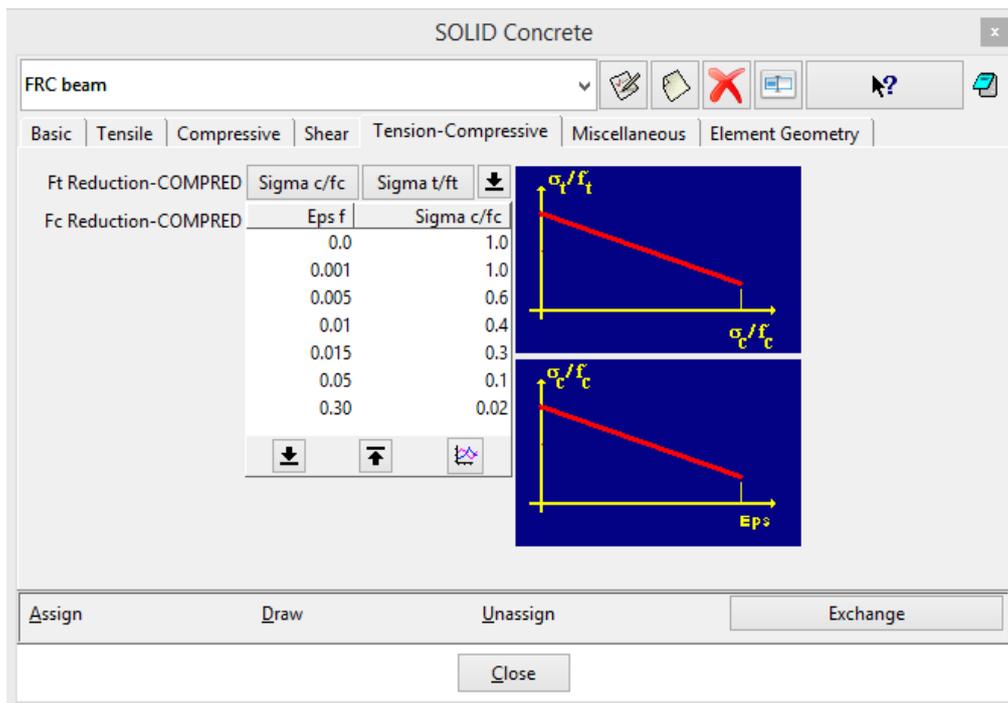


Fig. 3-15: Default parameter Fc Reduction-COMPRED

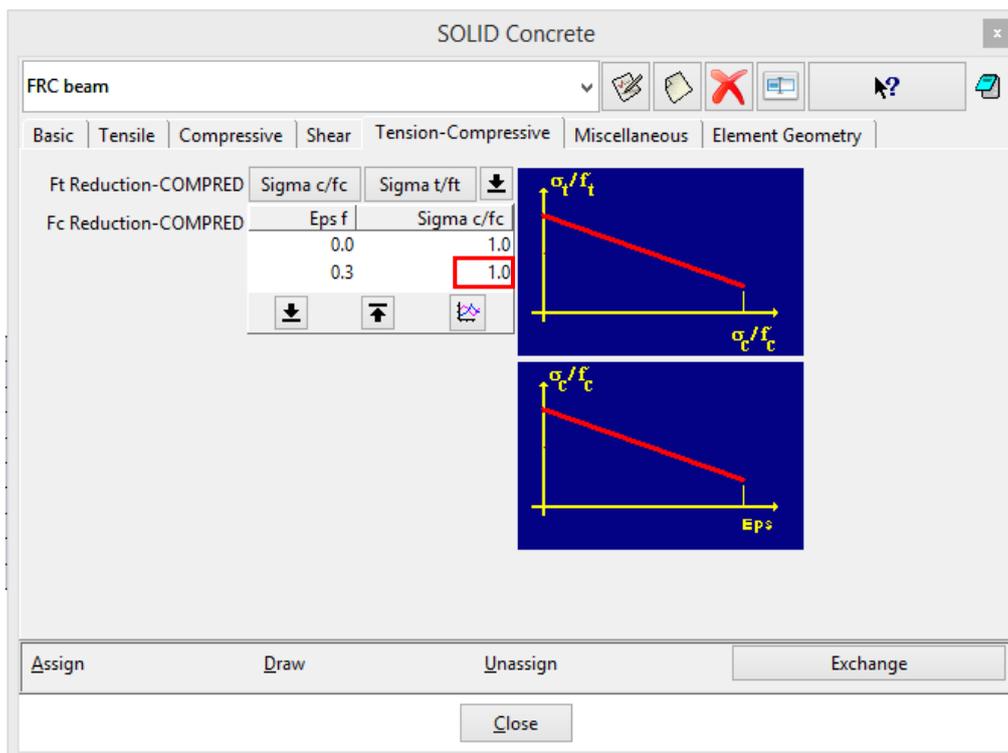


Fig. 3-16: Modified parameter Fc Reduction-COMPRED

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.

The manual inverse analysis is described in more detail in the tutorial [24]. The final softening curve after the manual fitting process according to the tutorial [24] is shown in Fig. 3-17. Fig. 3-18 shows the final response from the numerical analysis of the four-point bending experiment and its comparison to the experimental data. During the inverse analysis, the user should understand how to work with the material model and how modification of each parameter affects the results. The most 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]-[5] or contact the support (following ATENA Troubleshooting [5], 2.1.1).

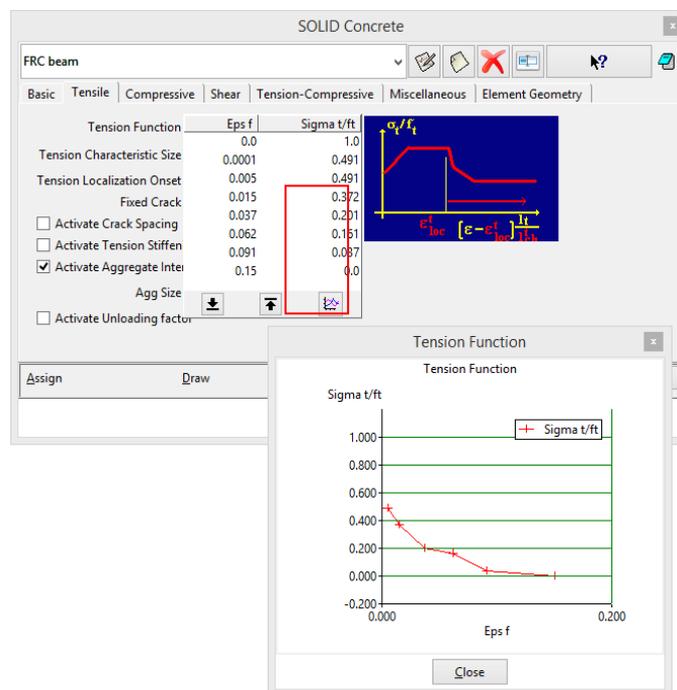


Fig. 3-17: Final shape of the tensile function

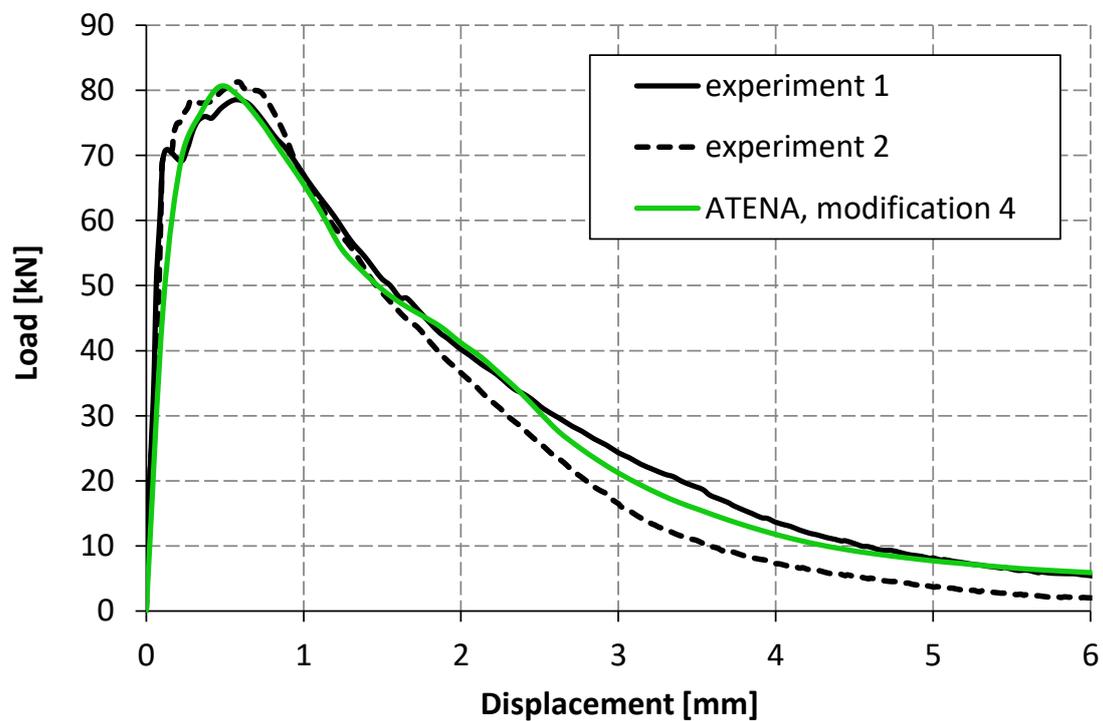


Fig. 3-18: Load-displacement diagram with the final result

3.3 Rule-based inverse analysis for FRC parameter identification

SARA Science software has been extended within the FibreLAB project to include support for model optimization by means of inverse analysis. This feature extends the existing capabilities of conducting statistical evaluations and enables users to fine tune their analytical ATENA model according to a measured structural behaviour (experimental test results / on site monitoring data).

The general flow of SARA optimization process:

1. Identification of specific parameters to be adjusted in the numerical model.
2. Iterative modifications of one or more of these parameters according to a set of predefined rules.
3. Future releases will include extended capabilities allowing users to define their own optimization rules and methods, as well as a range of predefined options for automatic and more robust optimization.

This chapter describes a specific optimization flow of the formerly mentioned tensile function to best fit the measured data. This SARA user flow is similar to the manual one described in 'ATENA-Science-GiD_Tutorial_FRC' [24]:

1. Define an initial ATENA model including the following:
 - i . A max-crack monitor.
 - ii . A load/reaction monitor.
 - iii . A displacement monitor.
 - iv . An initial tensile function.
2. Define a SARA project based on the initial ATENA model:
 - i . Click 'New project...' and fill in the information in the dialog box.

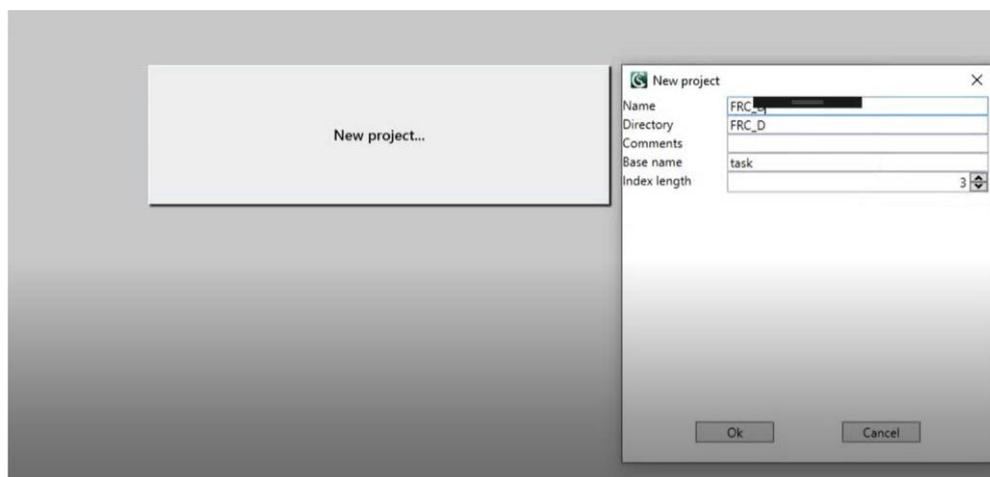


Fig. 3-19: New project definition in SARA Studio

- ii. Click 'import basic .inp...' and select the model file in the dialog.

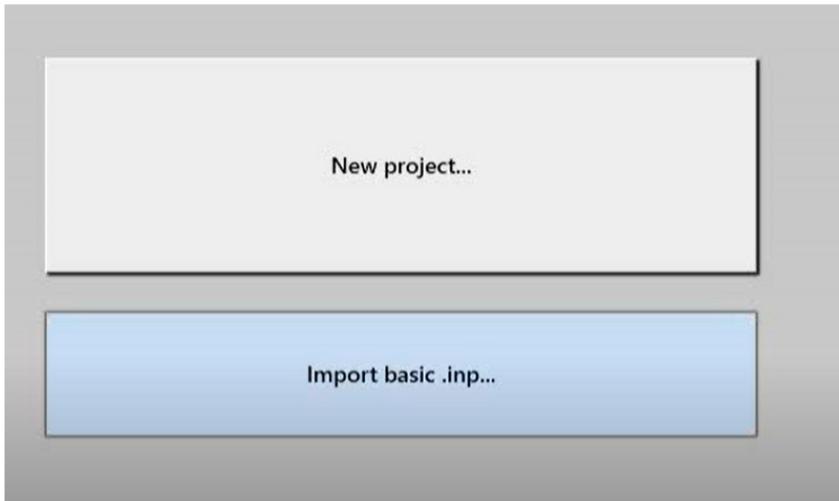


Fig. 3-20: Import of basic numerical model, i.e. input file with the extension *.inp

- iii. Click 'Deterministic → Run ATENA'.

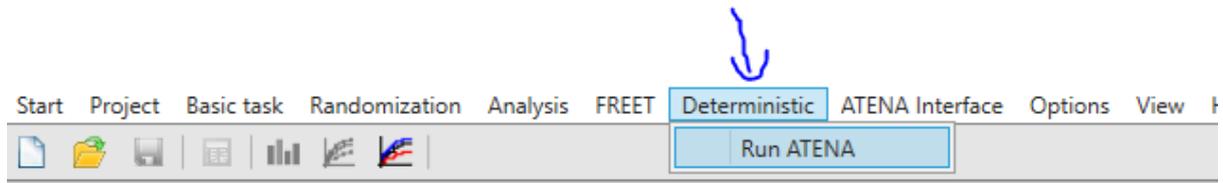


Fig. 3-21: Execution of deterministic initial ATENA analysis

3. Upload experimental results:

- i. SARA currently supports loading results in csv format. Other formats such as xlsx and vendor specific data representation formats will be supported in future releases. The user may load one or more files describing the load-displacement behaviour. SARA will use the averaged values for the optimization.
- ii. Data preparation: export the experimental results as a double column csv file (first column for displacement, second column for load). Make sure the units and directions coincide with the modeled monitored data.
- iii. Click 'import basic .csv...' or simply place your csv file in the SARA project directory.

4. Run SARA Function fitter:

- i. Click the function fitter button on the top toolbar.

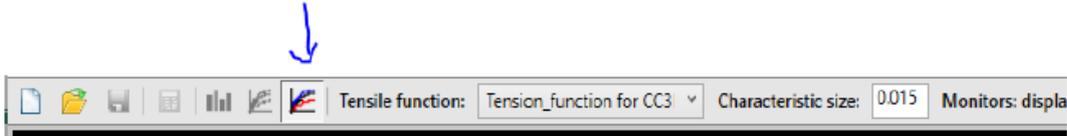


Fig. 3-22: Function filter button for displaying the load-displacement diagrams

- ii. Select the tensile function to be adjusted from the drop down list. This initial function will be redefined before each iteration until the desired load-displacement diagram result is achieved.

- iii. Fill in the characteristic size of the modeled FE. Along with the monitored crack width, this value is used to estimate the strain axis of the tensile function, assuming a linear relation between the crack width and tensile strain of the material.

- iv. Select the appropriate monitors from the drop down lists (displacement, load, max-crack). Before each iteration, the measured load-displacement curve will serve as reference data for recalculating the tensile function.

- v. Select the points on the calculated curve by which to adjust the tensile function or skip this step to auto select. It is recommended to pick several points along the curve where there is a significantly large difference between the measured value and the calculated value. By default, SARA will pick and attempt to adjust the peak point of the calculated curve and set the last point of the tensile function by determining the max strain. If the user does not select any points, SARA will automatically pick additional points in several intervals where the calculated curve significantly deviates from the measured values.

- vi. Click 'Run curve fitter...'. Results will automatically refresh every few seconds.

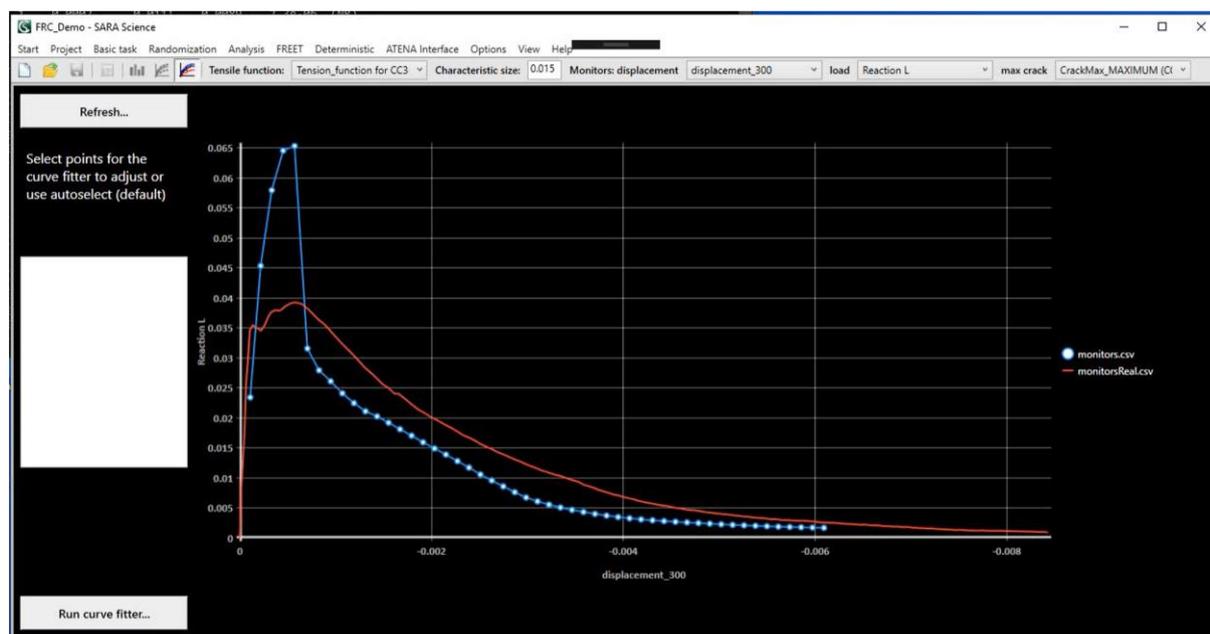


Fig. 3-23: Execution and visualization of the fitting process

5. Iterations 2-n:

- i. When the 'Iteration completed' popup appears, click 'Yes' if the results are satisfactory or to manually repeat steps 4.v. - 4.vi. using the modified function as baseline.
- ii. Click 'No' to run another iteration on the modified function using an approximation of the original selection of points. As the modifications are made with respect to the previous calculation, several iteration may be required to reach the desired fit.

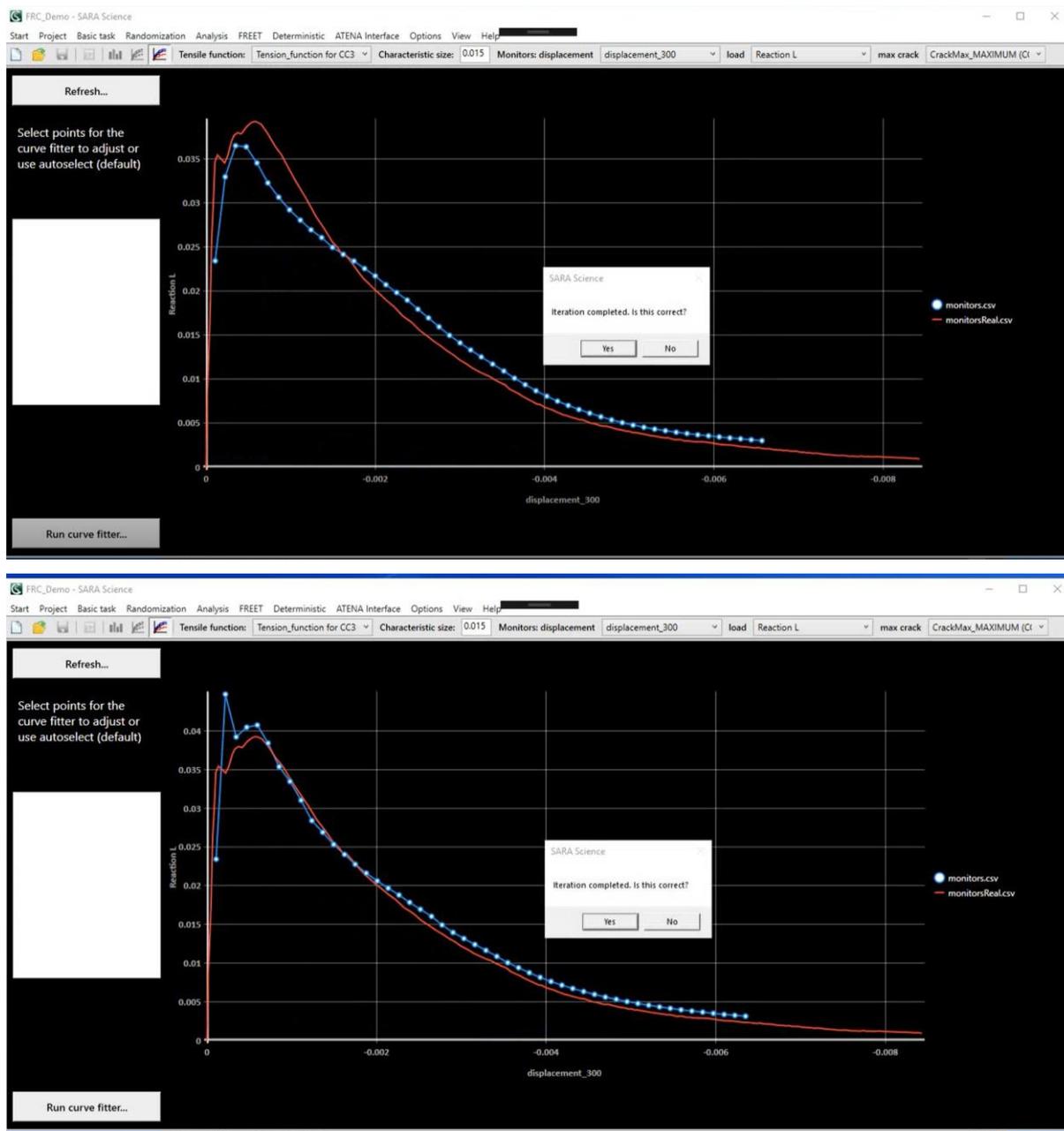


Fig. 3-24: Two stages of the iterative fitting process

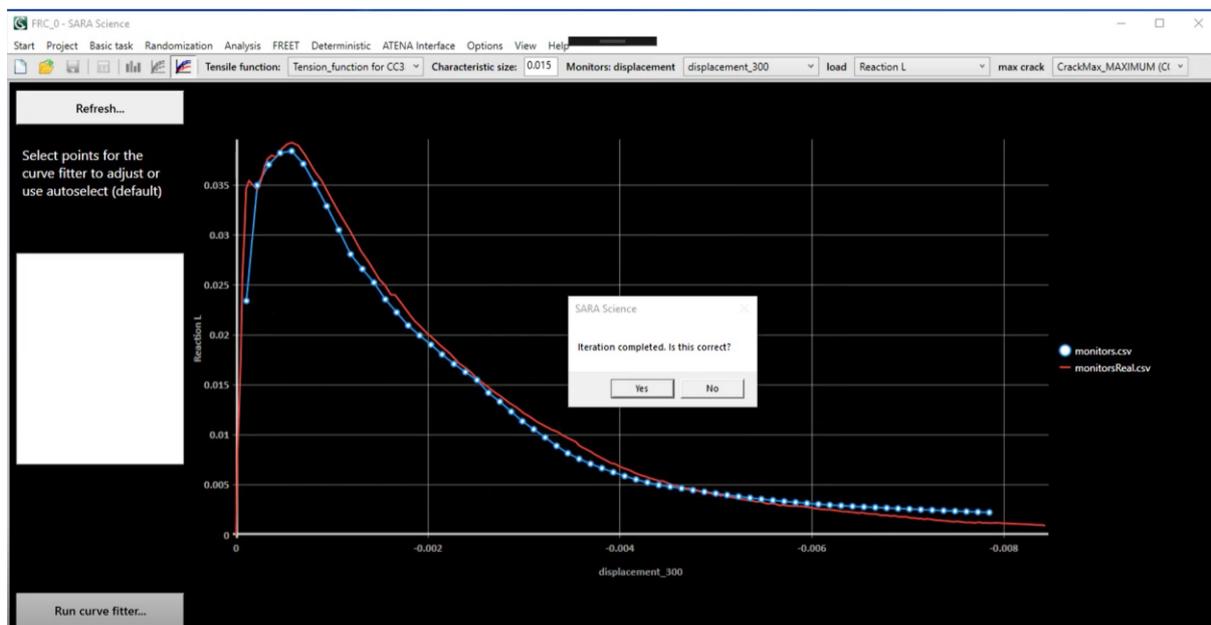


Fig. 3-25: Final fitted response using the automated SARA Studio fitting process

4. FRC Design using Simulation Prototyping

Since the design codes for FRC structures are generally not available, computer simulation based on nonlinear finite element analysis has a big potential in investigation and design of these structures. This chapter is focused on assessment of these structures by safety formats published in the new *fib* Model Code 2010 78, where rational safety assessment approach is presented, which reflects new developments in safety formats based on probabilistic methods. By these approaches, numerical results can be introduced into a suitable engineering safety concept. This document supported the results of authors' research project presented in [14],[18], where comparison of global safety determined by several methods was presented. The investigation described in [14],[18] included typical ordinary reinforced concrete structures with bending, shear-bending or compression failure modes and confirmed that simplified global safety formats can be used for design and structural verification. In this chapter, previous calculations for RC structures are extended by the study of safety formats of the two fibre reinforced concrete structural elements, such as tunnel lining precast segment.

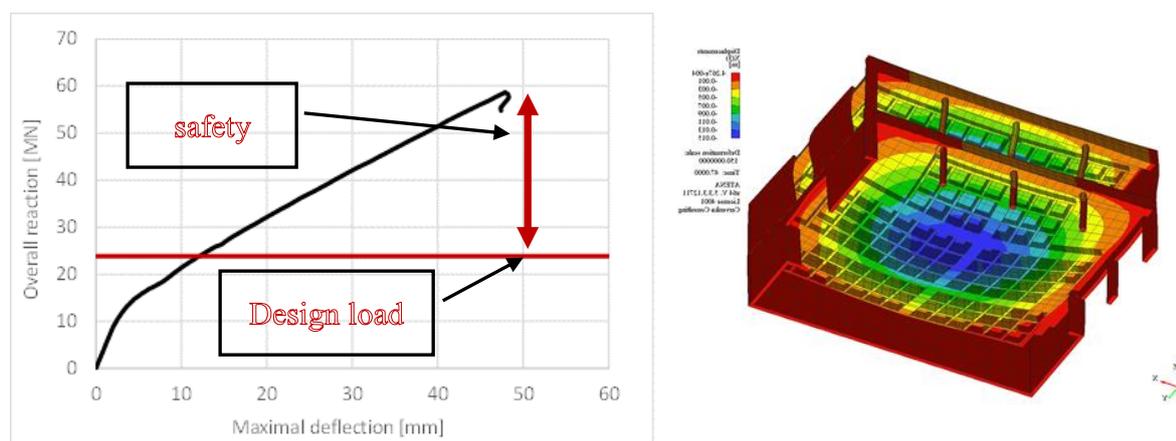


Figure 7: Global assessment approach

4.1 Assessment of Structural Safety and Reliability

Reliability and safety of reinforced concrete structures can be verified by several safety formats calculated by nonlinear analysis that are proposed in *fib* Model Code 2010:

- Full Probabilistic Analysis,
- ECOV Method – Estimate of Coefficient of Variation,
- Method based on EN 1992-2,
- Partial Safety Factors (PSF).

The same safety formats can be applied also for fibre reinforced concrete structures. Design combining numerical and experimental investigations together with safety formats is appropriate method how to obtain safe and reliable structure.

4.1.1 Full Probabilistic Analysis

The full probabilistic approach represents the most accurate method for the safety assessment of civil engineering structures. The accuracy of this approach is much higher if the nonlinear structural analysis is used as a limit state function. The numerical simulation resembles a real testing of structures by considering a representative group of samples, which can be statistically analysed for the assessment of safety. This concept is supported by the new *fib* Model Code 2010 [1] where rational safety assessment approach is presented, which reflects new developments in safety formats based on probabilistic methods. In the Chapter 4 on *Principles of structural design* the probabilistic safety format is introduced as a general and rational basis of safety evaluation. In addition to the partial factor format (which remains as the main safety format for most practical cases) a global resistance format is recommended for nonlinear analysis.

In case of fibre concrete structures the design codes are generally not available or not sufficient. Therefore, the computer simulation based on advanced nonlinear finite element analysis has a big potential in design of these structures. The numerical results can be introduced into a suitable engineering safety concept, from which the fully probabilistic analysis is the ultimate tool for design and safety assessment. It is superior to simplified methods because it provides information on the variability of resistance. However, it is computationally demanding and requires good information about random properties of input variables. Therefore, it is applied in special cases, where consequences of failure substantiate the increased effort.

The probabilistic analysis is performed here using software SARA, which integrates ATENA [12] and FReET program tools. The variability of basic properties is described by distribution functions and its parameters (mean, standard deviation, etc.). Probabilistic analysis of the resistance is performed by numerical method such as Latin hypercube sampling (LHS) method. Resulting set of resistance values is approximated by a probability distribution function (PDF) of global resistance, and describes the random properties of the resistance. Finally, for a required reliability index β , or failure probability P_f , a value of the design resistance R_d shall be calculated.

Probabilistic analysis is an efficient tool for safety assessment of civil engineering structures, in particular of concrete or fibre concrete structures. In the probabilistic nonlinear approach the structural resistance R_d is calculated by means of the probabilistic nonlinear analysis. The classical statistical and reliability approach is to consider material parameters as random variables with prescribed distribution function. The stochastic response requires repeated analyses of the structure with these random input parameters, which reflects randomness and uncertainties in the input values (see e.g. [12]). In this approach, the resistance function $r(r)$ is represented by nonlinear structural analysis and loading function $s(s)$ is represented by action model. Safety can be evaluated by the reliability index β , or alternatively by failure probability P_f taking into account all uncertainties due to random variation of the input values – material properties, dimensions, loading, and other.

Probabilistic analysis based on the nonlinear numerical simulation includes following steps:

- *Numerical model based on the nonlinear finite element analysis.* This model describes the resistance function $r(r)$ and performs a deterministic analysis of resistance for given set of input variables.
- *Randomization of input variables* (material properties, dimensions, boundary conditions, etc.). This can also include some effects of actions, which are not in the action function $s(s)$ (for example pre-stressing, dead load etc.). Random properties are defined by random distribution type and its parameters (mean standard deviation, etc.). They describe the uncertainties due to statistical variation of resistance properties.
- *Probabilistic analysis of resistance and action.* This can be performed by stratified method of Monte Carlo-type of sampling, such as LHS sampling method. Results of this analysis provide random parameters of resistance and actions, such as mean, standard deviation, etc. and the type of distribution function for resistance.
- *Evaluation of safety* using reliability index β or failure probability P_f .
- Probabilistic analysis can be also used for *determination of design value of resistance function* $r(r)$ expressed as R_d . Such analysis involves repeatedly the first three steps above, and R_d is determined for required reliability β or failure probability P_f .

In order to make the application of the probabilistic nonlinear analysis user-friendly, special software tool has been developed by the authors and their co-workers. The resulting software SARA (Structural Analysis and Reliability Assessment) integrates software tools ATENA and FReET. It is equipped with a user-friendly shell called *SARA Studio* (), which leads the user interactively through the modelling and randomization process of the solved problem as described above. All features (or just selected ones) of the involved programs including modelling of deterioration/degradation phenomena can be utilized also in the reliability analysis and performance-based assessment of concrete structures. For this purpose, the interconnectivity between ATENA Engineering as well as ATENA Science (i.e. ATENA input file), with the probabilistic modules was achieved. The program control and data exchange is organized by an efficient and user friendly shell interface ([12]).

Integration of ATENA and FREET

Structural Analysis and Reliability Assessment

data exchange, program control, user interaction

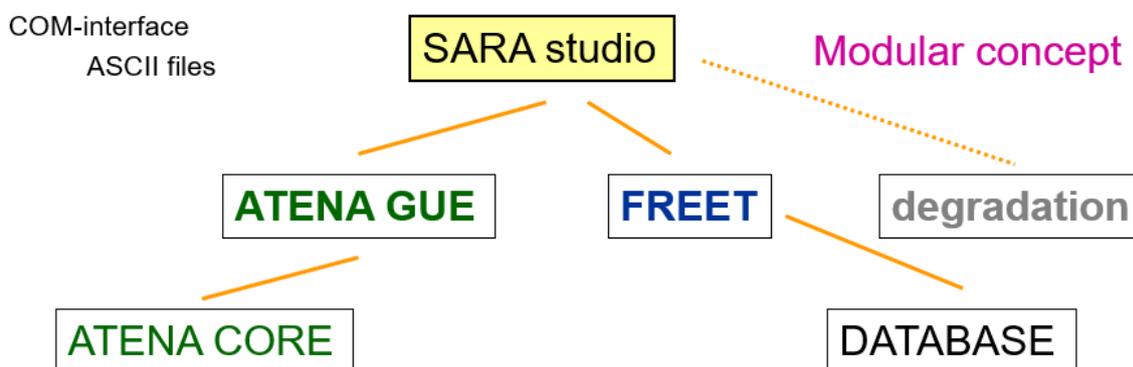


Fig. 4-1: Integration of ATENA and FREET in SARA Studio

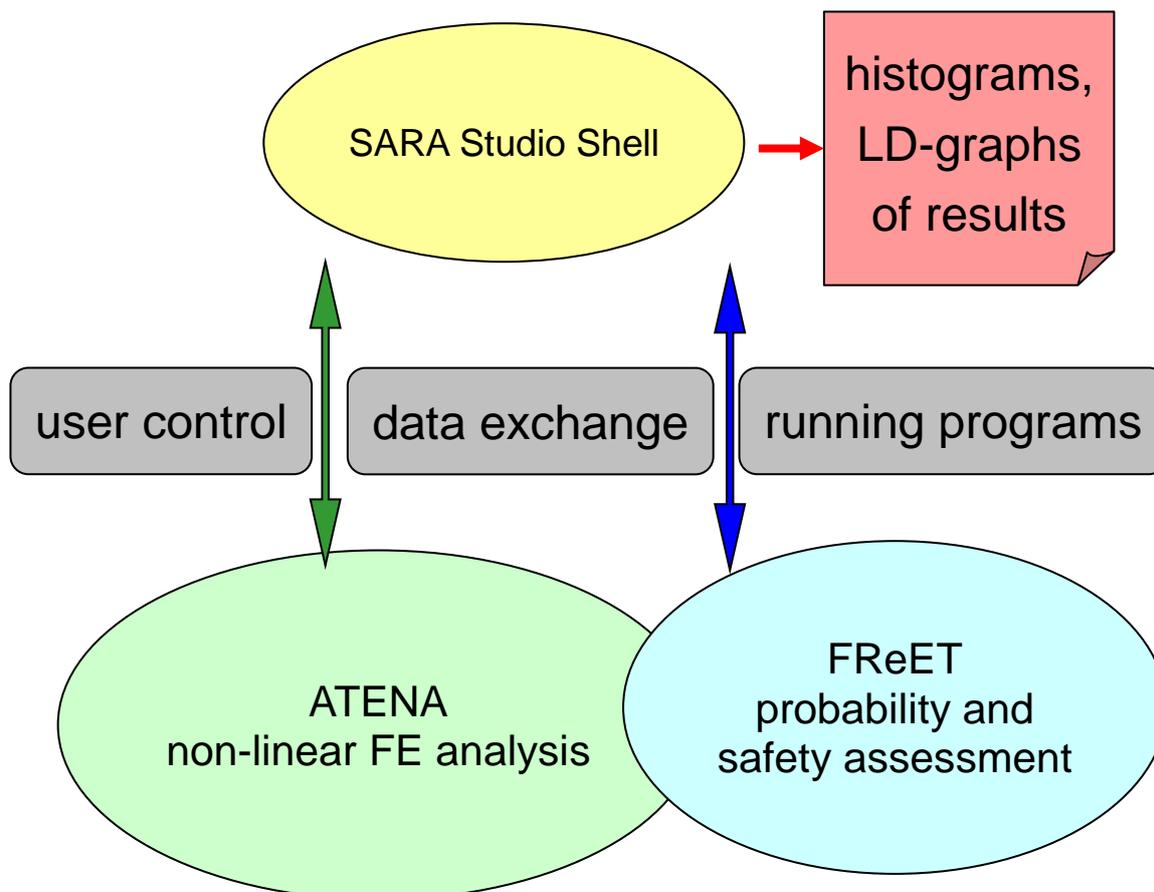


Fig. 4-2: Structure and data flow in the SARA system

4.1.1.1 Stochastic analysis e

The stochastic analysis is based on repeated analysis of the prepared model with randomly generated properties. Since each of these samples represent a demanding nonlinear finite element analysis, the number of samples should be kept on moderate level. In the same time the applied methodology should be sufficiently accurate and representative.

The probabilistic software FReET has been developed for stochastic and probabilistic analysis of computationally intensive problems such as nonlinear finite analysis. Stratified simulation technique LHS is used to keep the number of required simulations at an acceptable level. This technique can be used for both random variables and random fields levels. Statistical correlation is imposed by the stochastic optimization technique called simulated annealing. Sensitivity analysis of the input parameters to resulting values is based on nonparametric rank-order correlation coefficients. Procedure can be briefly outlined:

- random input parameters are generated according to their PDF using LHS sampling
- samples are reordered by the Simulated Annealing approach in order to match the required correlation matrix as closely as possible
- generated realizations of random parameters are used as inputs for the analyzed function (computational model)

- solution is performed many times and the results (structural response) are saved
- simulation process the resulting set of structural responses is statistically evaluated

Main results from the stochastic analysis are:

- estimates of the mean value
- variance
- coefficient of skewness and kurtosis
- empirical cumulative probability density function estimated by an empirical histogram of structural response

This basic statistical assessment is visualized through histograms. This is followed by reliability analysis based on several approximation techniques:

- estimate of reliability by the Cornell safety index β
- curve fitting approach applied to the computed empirical histogram of response variables
- estimate of probability of failure based on the ratio of failed trials to the total number of simulations

State-of-the-art probabilistic algorithms are implemented to compute the probabilistic response and reliability. The main features of the FReET software are:

(a) stochastic model (inputs)

- direct connectivity to the nonlinear analysis input data
- friendly Graphical User Environment (GUE)
- 30 probability distribution functions (PDF), mostly 2-parametric, some 3-parametric, two 4-parametric (Beta PDF and normal PDF with Weibullian left tail)
- unified description of random variables optionally by statistical moments or parameters or a combination of moments and parameters
- PDF calculator
- statistical correlation (also weighting option)
- categories and comparative values for PDFs
- basic random variables visualization, including statistical correlation in both Cartesian and parallel coordinates

(b) probabilistic techniques

- Crude Monte Carlo simulation
- Latin Hypercube Sampling (3 alternatives)
- First Order Reliability Method (FORM)
- Curve fitting
- Simulated Annealing
- Bayesian updating

(c) response/limit state function

- numerical form directly connected to the results of nonlinear FE analysis
- multiple response functions assessed in same simulation run

4.1.2 ECOV Method – Estimate of Coefficient of Variation

ECOV method proposed by Cervenka in [18] is based on the idea, that the random distribution of resistance due to material, which is described by the coefficient of variation V_m , can be estimated from mean R_m and characteristic values R_k of resistance. The underlying assumption is that random distribution of resistance is according to the lognormal distribution, which is however typical for the structural resistance. Considering these assumptions the coefficient of variation V_m can be expressed as:

$$V_m = \frac{1}{1.65} \ln \left(\frac{R_m}{R_k} \right) \quad (1)$$

Then the global safety factor γ_R of resistance can be calculated as:

$$\gamma_R = \exp(\alpha_R \beta V_R), \quad V_R = \sqrt{V_m^2 + V_{Rd}^2} \quad (2)$$

where α_R is the sensitivity factor for resistance (as defined by FORM) and β is the reliability index. V_{Rd} is the model uncertainty. The above procedure enables to estimate the safety of resistance in a rational way, based on the principles of reliability accepted by the codes. Appropriate code provisions can be used to identify these parameters. For instance in the case of Eurocode EN 1990, typical values are $\beta = 3.8$ (50 years) and $\alpha_R = 0.8$, which corresponds to the failure probability $P_f = 0.001$. The global resistance factor is then:

$$\gamma_R \cong \exp(3.04 V_R) \quad (3)$$

and the design resistance is calculated as:

$$R_d = R_m / \gamma_R. \quad (4)$$

The main task in this method is the determination of the mean and characteristic values of resistance R_m, R_k . They can be calculated by two separate nonlinear analyses using mean and characteristic values of the input material parameters, respectively. The method is very general and reliability level β and distribution type can be changed if required. It can capture different types of failure and the sensitivity to a random variation of the material parameters is automatically included. The slight disadvantage of this method is the need for two separate non-linear analyses.

4.1.3 Method based on EN 1992-2

Eurocode for bridges introduced a concept for global safety verification based on nonlinear analysis. In this approach, the design resistance is calculated from

$$R_d = R(f_{ym}, \tilde{f}_{cm} \dots) / \gamma_R \quad (5)$$

where f_{ym}, \tilde{f}_{cm} are mean values of material parameters of steel reinforcement and concrete, $f_{ym} = 1.1 f_{yk}$ and $\tilde{f}_{cm} = 0.843 f_{ck}$. The concrete mean value is reduced to account for the higher variability of concrete property. This method is described in more detail in [18]. The global factor of resistance is then $\gamma_R = 1,27$. The evaluation of the resistance function is done by a single nonlinear analysis assuming the material parameters according to the above rules.

4.1.4 Partial Safety Factors

This approach is the natural extension of the standard partial safety factor method, which is used in the most design codes. The design condition is formulated as

$$E_d < R_d \quad (6)$$

The design action $E_d = E(F, \gamma_G, \gamma_Q, \gamma_P, \dots)$ is a function of the representative load F , which is multiplied by the partial safety factors $\gamma_G, \gamma_Q, \gamma_P, \dots$ for permanent loads, live loads, prestressing, etc. The resistance $R_d = R(f_d)$ is calculated by a nonlinear analysis using design values of the material parameters $f_d = f_k / \gamma_M$, where f_k are characteristic values and γ_M partial safety factors of materials. The verification of safety by the condition (6) in the standard design practice is applied to cross sections and actions that are obtained by a linear analysis. It is well known that this concept is not consistent, since different methods are used for the calculation of actions (linear analysis) on one side, and for the resistance of cross sections (nonlinear) on the other. Furthermore, only local safety check is exercised and a global safety assessment is not performed, and is unknown. The action E_d in condition (6) is considered on the global level (for example live load intensity) and the resistance R_d is an ultimate load intensity obtained by a nonlinear analysis, in which design values of material parameters f_d are used.

4.2 Validation of Safety Formats for Ordinary Reinforced Concrete Structures

Validation of the safety formats described in the previous chapter can be found in the studies [14] and [18], where the calculated examples are described in more detail. The summary of results is shown in Tab. 4-1. This table shows the resistance for the four presented methods and 7 examples are listed and compared. To enable the comparison the results are normalized with respect to the resistance obtained by the PSF method. It should be noted that the study does not reflect the model uncertainty in a consistent way. The methods PSF and EN1992-2 include a model uncertainty, while in the ECOV and full probabilistic analysis the model uncertainty was considered, i.e. $V_{Rd} = 0$ which will enable an evaluation of the model uncertainty. This can explain the average results of ECOV method being slightly higher than the other two methods.

Tab. 4-1: Case study of safety formats for RC structures (R_d is normalized with respect to PSF)

	R_d/R_d^{PSF}			
	PSF	Probabilistic	ECOV	EN 1992-2
Example 1 - bending	1	0.96	1	0.95
Example 2 - deep shear beam	1	0.98	1.02	0.98
Example 3 - bridge pier, geom. nonlinear	1	1.02	1.06	0.98
Example 4 - bridge frame	1	1.01	0.97	0.93
Example 5 - shear beam w/o ties	1	0.97	0.95	0.99
Example 6 - shear beam w. ties	1	1.23	1.28	1.01
Example 7 - long-span girder	1	1.19	1.13	1.04
Average	1	1.05	1.06	0.98

4.3 Performance of FRC Structures

The examples presented in chapter 4.2 were extended in FibreLAB project to include the validation of the proposed safety formats also for fibre-reinforced structures. Two new example structures were analysed as shown in. The first example is a four point bending beam (see Fig. 4-3).

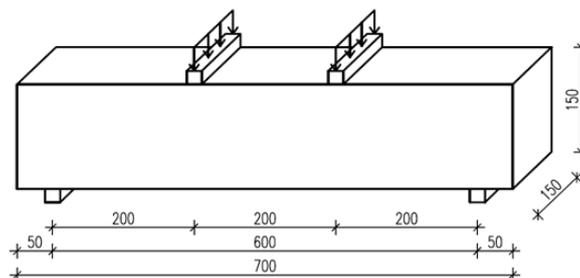


Fig. 4-3: Geometry of the FRC bending beam

The mean material parameters were first calibrated by inverse analysis described in the previous chapter 3 and then the global safety approaches were used to calculate the corresponding design resistances. The second example represents a tunnel lining segment (see Fig. 4-4a) installed by TBM (tunnel boring machine) during tunnel excavation. Full size laboratory tests of both RC and SFRC segments have been performed in the Klokner Institute of CTU in order to check their resistance under various loading conditions [23]. This experimental and analytical program was part of an engineering project in Prague, Czech Republic [25]. Maximum load measured in experiments with SFRC segments was around 500 kN. The numerical model was first validated by experimental data for a loading scenario simulating the action of the TBM machine during the installation and assembly of segments (see Fig. 4-4b, c, d). After that the global safety formats above were applied to calculate the design resistance (Fig. 4-5), see chapter 4.1.

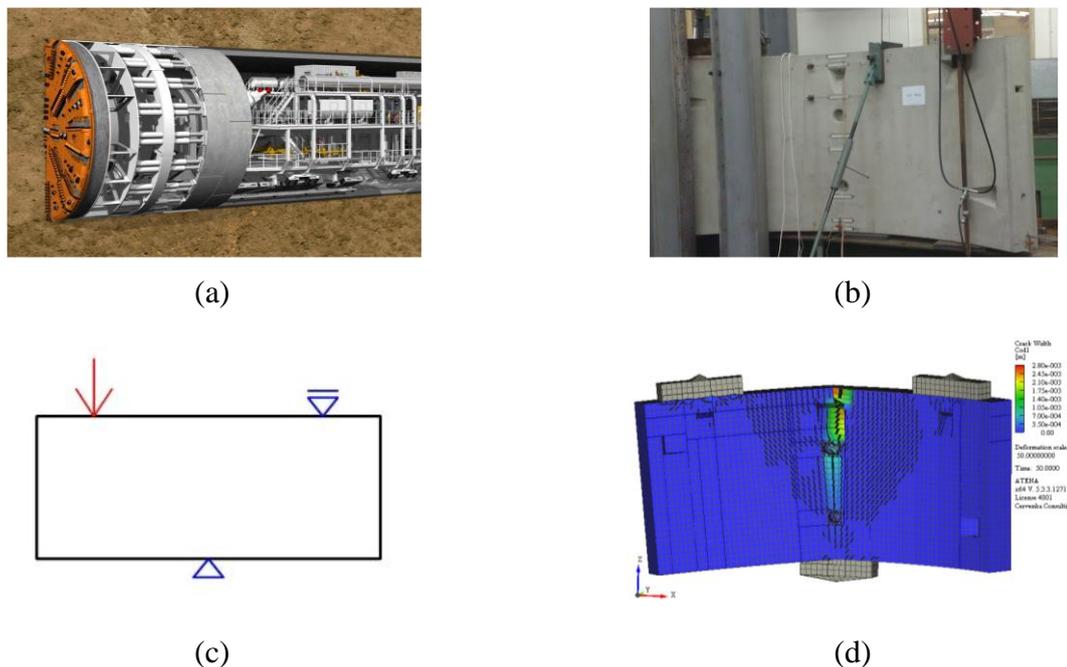


Fig. 4-4: Case study example of a tunnel tubing made of fibre reinforced concrete

Results in Tab. 4-2 indicate that the global safety formats for nonlinear analysis are applicable to fibre reinforced concrete as well as for ordinary reinforced concrete as presented in [16] and [18]. Similarly to the previously published results, there does not appear to be a significant difference between the design resistances calculated by the investigated methods. It is observed that the EN 1992-2 approach gives in most cases more conservative results than the other methods. On the contrary the probabilistic and ECOV approaches give mostly slightly higher resistance values. This can be however attributed to the fact that they do not include model uncertainties, which are partially included in the PSF and EN 1992-2 methods. Due to high variability of FRC material properties it can be recommended to utilize preferably the stochastic analysis based methods, i.e. full probabilistic analysis or ECOV method, where the actual material variability can be accounted for the evaluation of structural performance, safety and reliability under severe conditions.

Tab. 4-2: Safety format results for FRC structures (R_d is normalized with respect to PSF)

	R_d/R_d^{PSF}			
	PSF	Probabilistic	ECOV	EN 1992-2
FRC bending	1	1.16	1.23	0.97
FRC tubing	1	1.22	1.27	1.00
Average	1	1.19	1.25	0.99

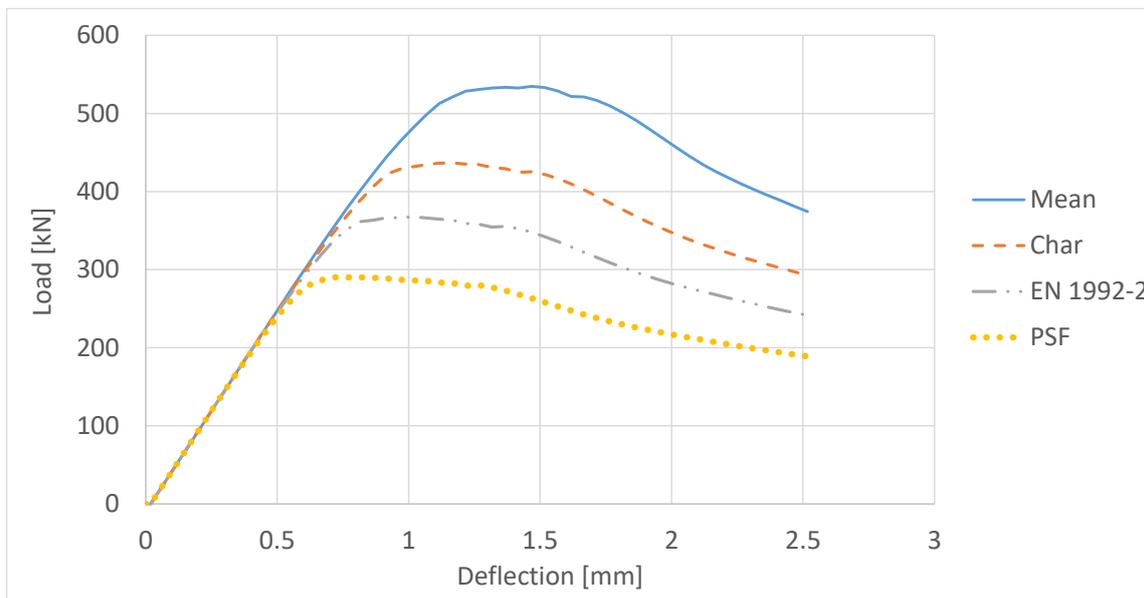


Fig. 4-5: Calculated load-displacement curves for FRC tunnel tubing example for various safety formats

5. Summary

This documents provides the basic information how to apply the software tools developed in the research and development project FibreLAB for the analysis, design and performance evaluation of structures or structural components made of fibre-reinforced concrete. The document is divided into three main chapters which corresponds to the main three products developed during FibreLAB research project:

Chapter 2 - Rapid Prototyping for Parametric Modelling of FRC Products/Structures – ATENA Studio Pre

This chapter describes the main application of the prototyping and modelling software ATENA Pre. This software allows the parametric description and modelling of FRC products and structures.

Chapter 3 - FRC Model Parameters Identification

This chapter provides step by step description of the FRC model parameter identification based on experimental data using the procedures and software tools developed during FibreLAB project.

Chapter 4 - FRC Design using Simulation Prototyping

This chapter provides a guidance how to perform the global structural assessment and design using the new ATENA software developed during FibreLAB project for fibre-reinforced structures and products.

6. References

- [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] *fib* Model Code for Concrete Structures 2010. Wilhelm Ernst & Sohn, Berlin, Germany, (2013), ISBN 978-3-433-03061-5.
- [7] RILEM, 'Technical Recommendations for the Testing and Use of Construction Materials' (E & FN SPON, London, 1992).
- [8] Richtlinie Faserbeton, Österreichische Bautechnik Vereinigung (ÖVB), Wien, 2008.
- [9] ACI 544.8R-16, 'Report on Indirect Method to Obtain Stress-Strain Response of Fiber-Reinforced Concrete (FRC)', ACI Committee 544, 2016.
- [10] CNR-DT 204/2006, 'Guide for the design and construction of fibre reinforced concrete structures', CNR - Advisory Committee on Technical Recommendations for Construction, Rome, 2007.
- [11] 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
- [12] Bergmeister, K., Novák, D., Pukl, R., Červenka, V., Structural assessment and reliability analysis for existing engineering structures, theoretical background, *Structure and Infrastructure Engineering*, Vol. 5, Issue 4, August 2009, pp. 267-275.
- [13] Bertagnoli, G., Giordano L., Mancini, G., Safety format for the nonlinear analysis of concrete structures, *STUDIES AND RESEARCHES –V.25*, Politech. di Milano, (2004) Italy
- [14] Červenka, V., Reliability-based non-linear analysis according to fib Model Code 2010, *Structural Concrete*, Journal of the *fib*, Vol. 14, March 2013, ISSN 1464-4177, (2013) 19-28, DOI: 10.1002/suco.201200022.
- [15] Červenka V., Červenka J., Jendele L., ATENA Program Documentation, Part 1 Theory, Prague, (2016), Červenka Consulting, www.cervenka.cz.
- [16] Červenka, V., Simulating a response, *Concrete Engineering International*, Vol. 4, No. 4, (2002) 45-49.

- [17] Cervenka, J., Pappanikolaou, V., Three dimensional combined fracture-plastic material model for concrete. *Int. J. of Plasticity*, Vol. 24, 12, (2008) 2192-2220, ISSN 0749-6419, doi:10.1016/j.ijplas.2008.01.004.
- [18] Cervenka, V., Global Safety Format for Nonlinear Calculation of Reinforced Concrete. *Beton- und Stahlbetonbau* 103, Special Edition, Ernst&Sohn, (2008) 37-42.
- [19] Consoft - software documentation (2015). Available at: <http://bauwesen.htwk-leipzig.de/nc/de/fakultaet-bauwesen/personen/name/slowik/>
- [20] Kolísko, J., Tichý, J., Kalný, M., Huňka, P., Hájek, P. & Trefil, V., Development of ultra-high performance concrete (UHPC) on the basis of raw materials available in the Czech Republic. In: *Betonové konstrukce 21. století: betony s přidanou hodnotou*. Praha: Beton TKS (2012).
- [21] Lehký, D., Keršner, Z. and Novák, D., 'Determination of statistical material parameters of concrete using fracture test and inverse analysis based on FraMePID-3PB tool', in *Proc. of Reliable Engineering Computing - REC 2012*, June 2012, Brno, Czech Republic, ISBN 978-80-214-4507-9 261-270
- [22] Novák, D., Lehký, D., ANN Inverse Analysis Based on Stochastic Small-Sample Training Set Simulation, *Engineering Application of Artificial Intelligence* 19 (2006) 731-740.
- [23] Pukl, R., Havlásek, P., Vítek, P., Vítek, J.L., Vokáč, M., Bouška, P., Hilar, M., Numerical and experimental investigation of structural members made from RC and SFRC, *fib Symposium, Tel Aviv 2013*, (2013) 241-244.
- [24] Sajdlová, T., ATENA Program Documentation, Part 4-7, ATENA Science – GiD FRC Tutorial, Červenka Consulting s.r.o., 2016, www.cervenka.cz/downloads
- [25] Stehlik, E., Cyron D., Prague Metro's return to TBMs. *Tunnels & Tunnelling International*, August 2012.

Acknowledgements

This document was created within the Eurostars project FibreLAB E!10316. The financial support of the Eurostars program and national funding agencies in Czech Republic and Hungary are greatly appreciated.

The partners in the project are:

Cervenka Consulting s.r.o. CER
JKP Static Ltd..... JKP

This report owes to a collaborative effort of the above organizations.

More information

Public FibreLAB reports are available through:

FibreLAB public web site http://www.fibrelab.eu

Copyright

© FibreLAB Consortium 2019

© Cervenka Consulting s.r.o. 2019

© JKP Static 2019