

Virtual Lab for Fibre Reinforced Concrete Design by Simulation Prototyping

FibreLAB

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



D2.2 Modelling Software

Responsible author: Jan Cervenka

Co-authors: Zdenek Janda,

Tomas Altman,

Pavel Palek,

Peter Schaul,

Tereza Sajdlová

Status: final

Type: software/pilot

Access: public

Version: 3.0

Date: 29.1.2019

Project website:

<http://www.fibrelab.eu>



Executive summary:

This project document is developed during the project FibreLab to describe the prototyping and modelling software developed during the third year of the project. This documents extends the document developed in the second project year, i.e. deliverable D2.1. This document has been extended by new features developed during the third project year. The new features and development are mainly summarized in Section 2, 2.5, 2.6, 2.7 and 2.8.

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 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.

Revision history:

date	author	status	changes
09.11.2018	J. Červenka	1.0	draft
18.1.2019	J. Červenka	2.0	First complete draft
20.1.2019	T. Altman	3.0	Documentation of the fast prototyping features.

Table of contents:

REVISION HISTORY:	3
1. GENERAL	5
2. SOFTWARE OVERVIEW	6
2.1 ATENA STUDIO PRE – OVERVIEW	6
2.2 GEOMETRICAL AND NUMERICAL MODELS	7
2.3 MAIN WINDOW	7
2.4 MAIN DATA TREE	10
2.5 SCRIPT HISTORY	11
2.6 CONSTRUCTION COMMANDS	14
2.7 DELETE COMMAND	15
2.8 TRANSFORMATION COMMANDS	16
3. MATERIAL DEFINITION	18
4. MODELLING OF STRUCTURAL ELEMENTS USING BIM MODEL	23
4.1 ATENA IN RELATION TO BIM	24
4.2 IFC FILE IMPORT TO THE ATENA STUDIO PRE	25
4.3 IFC DATA COMPATIBILITY IN ATENA STUDIO PRE	29
4.3.1 Materials	29
4.3.2 Building elements	29
4.3.3 Reinforcing elements	29
4.3.4 Profiles	31
4.3.5 Geometry	33
5. REFERENCES	35

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 modeling 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. Software Overview

2.1 ATENA Studio PRE – Overview

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:

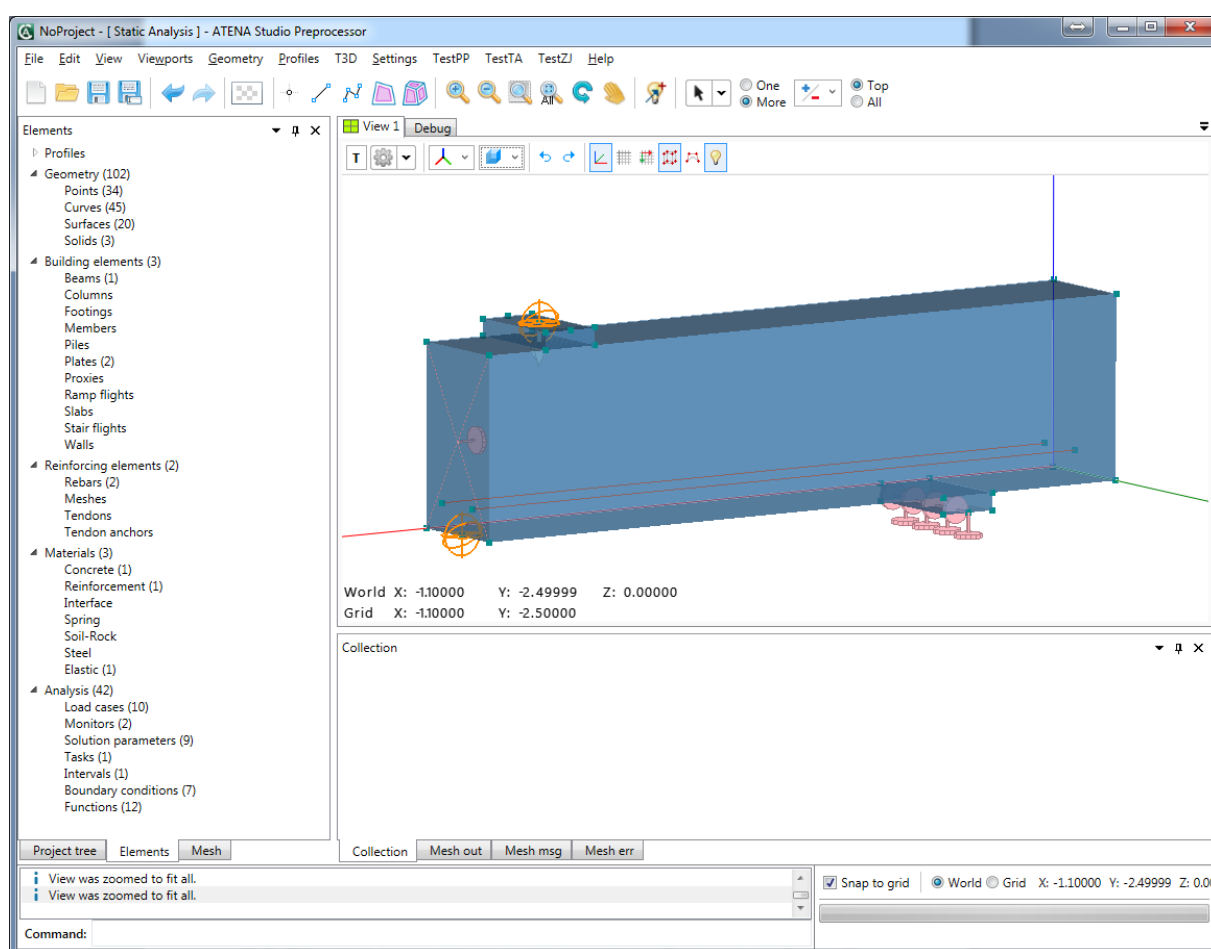


Fig. 1: Main window of ATENA Studio PRE

2.2 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 discretisation 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.

2.3 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. 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

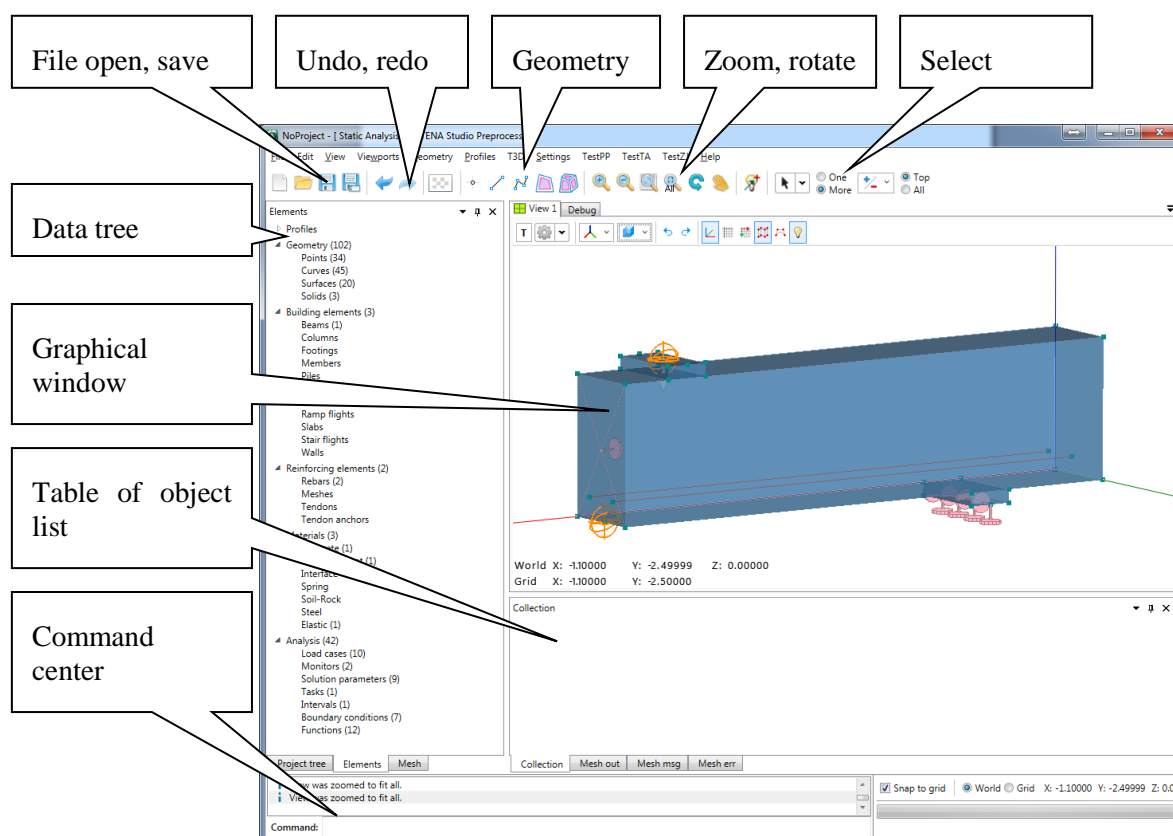


Fig. 2: Main window of ATENA Studio PRE – description

Main Menu

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

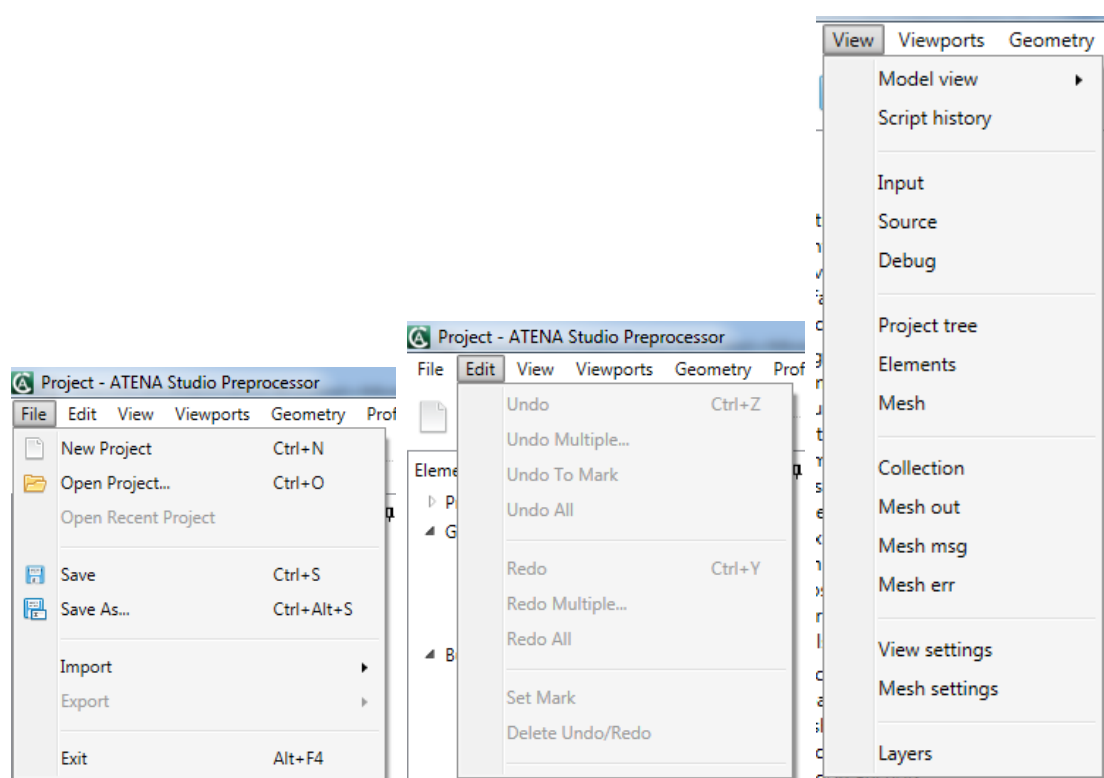


Fig. 3 Main menu a) File, b) Edit, c) View

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.

2.4 Main Data tree

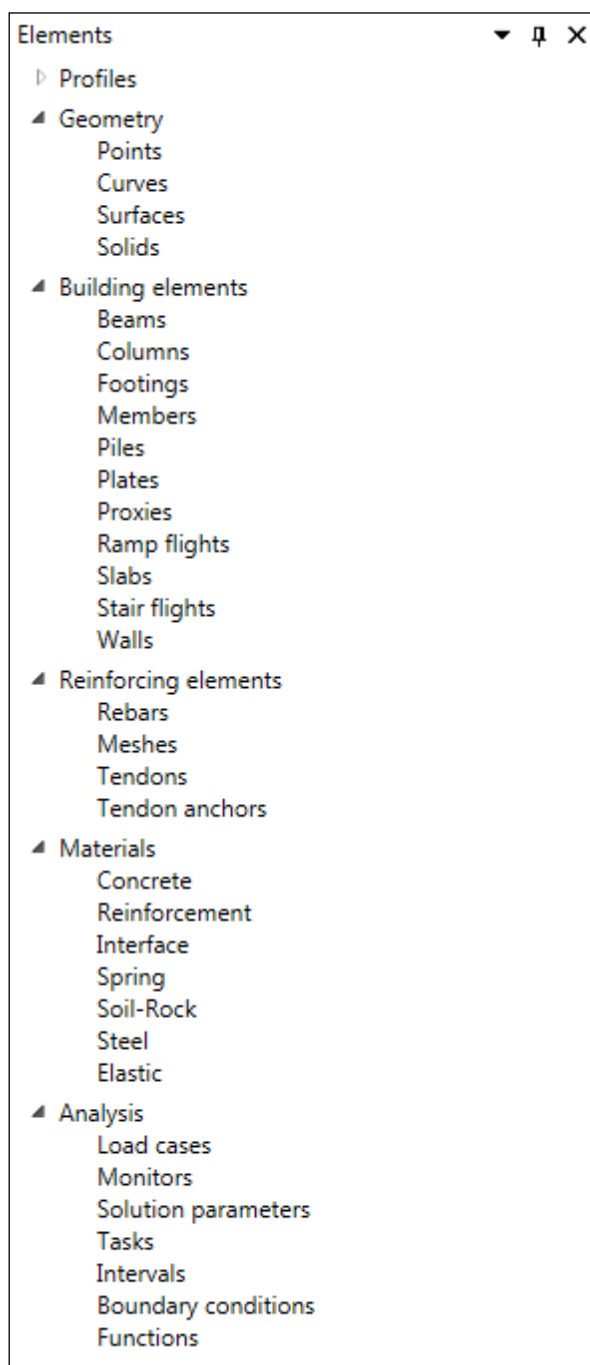


Fig. 4: main data tree

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 simulate one experiment. Each task have several intervals, and each intervals have combination of load cases. So you can set different boundary conditions in different part of experiment.

2.5 Script history

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

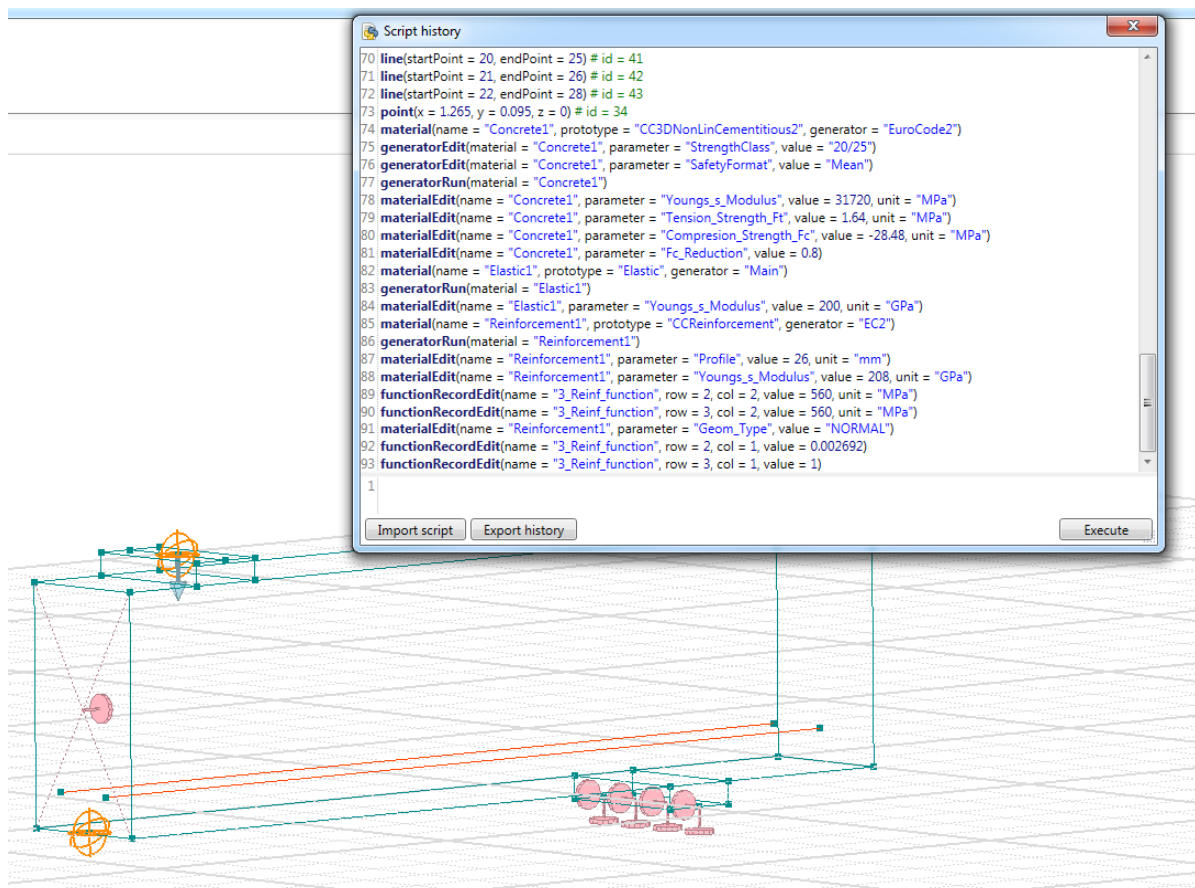


Fig. 5: Example of script history for model Shear beam

The script history can be a good tool for creating model parametrically. If geometry relationships are known, then global parameters can be separated and with that created the model, see Fig. 6 and Fig. 7 (global parameters are size, copies count and model origin)



Fig. 6: Example of parametric script for generating rails. Solid view

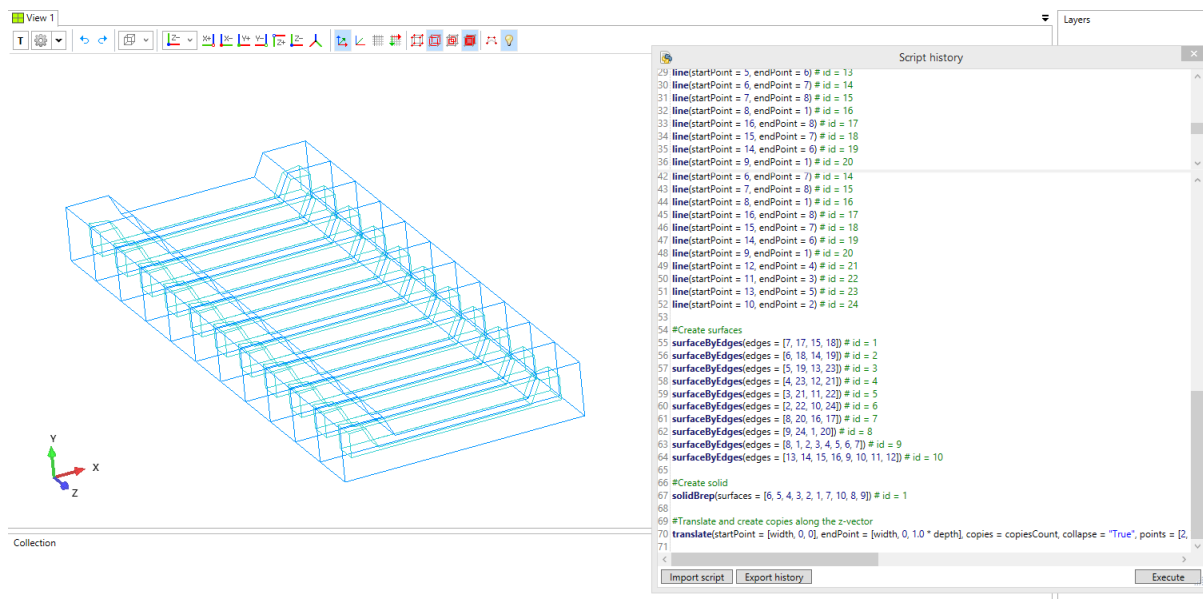


Fig. 7: Example of parametric script for generating rails. Wireframe view with displayed lines and solids.

2.6 Construction Commands

In Preprocessor geometry can be imported from IGES or IFC, created by python script or created by user self. If the user decides to create a new or some part of model, there are four base buttons which serve to create points, lines, polygonal surfaces and brep solids dynamically. Each of these commands has appropriate settings window placed in layout, where it can be seen which entities are selected or which coordinates are written. See Fig. 8

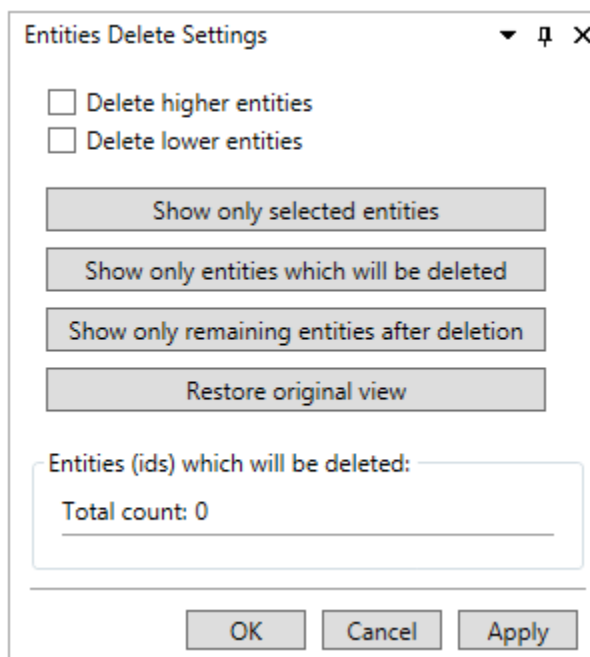
The figure displays four separate dialog boxes for creating different geometric entities in a software application. Each dialog box has a title bar with a dropdown arrow, a maximize icon, and a close icon (X).

- Points Create Settings:** This dialog has a 'Name' field and a 'Point coordinates' section with three input fields: 'X' (value: -1.8), 'Y' (value: 2), and 'Z' (value: 0). It includes 'OK', 'Cancel', and 'Apply' buttons at the bottom.
- Line Segments Create Settings:** This dialog has a 'Name' field and two radio button options: 'Start point' (selected) and 'End point'. The 'Start point' section has a 'Select' button (with a point icon) and three input fields: 'X' (value: -2.9), 'Y' (value: 0.1), and 'Z' (value: 0). The 'End point' section also has a 'Select' button and three empty input fields for 'X', 'Y', and 'Z'. It includes 'OK', 'Cancel', and 'Apply' buttons at the bottom.
- Surface Create by Edges Settings:** This dialog has a 'Name' field and a 'Boundary of surface' section with two input fields: 'Selected edges' and 'Complemented edges'. It includes 'OK', 'Cancel', and 'Apply' buttons at the bottom.
- Solid Create by Polygonal Surfaces Settings:** This dialog has a 'Name' field and a 'Boundary of solid' section with two input fields: 'Selected surfaces' and 'Complemented surfaces'. It includes 'OK', 'Cancel', and 'Apply' buttons at the bottom.

Fig. 8: Points, lines, polygonal surfaces, BREP solids creating settings

2.7 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. 9.



Entities Delete Settings

☐ Delete higher entities

☐ Delete lower entities

Show only selected entities

Show only entities which will be deleted

Show only remaining entities after deletion

Restore original view

Entities (ids) which will be deleted:

Total count: 0

OK Cancel Apply

Fig. 9: Entity deleting settings

2.8 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. 10). Transformed geometry can be copied and collapsed with existing geometry too. 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 holes.

The figure displays three software dialog boxes for geometry transformations: Translation, Scale, and Rotate. Each dialog box contains a 'Basic settings' section at the top with a 'Copies' input field (set to 1) and a 'Collapse if close' checkbox (checked). Below this, each dialog has a '1. Select geometry' section with a 'Select' button and input fields for Points, Curves, Surfaces, and Solids. The Translation dialog includes '2. Select start point' and '3. Select end point' sections, each with a 'Select' button and X, Y, Z coordinate fields. The Scale dialog includes a '2. Select scale center' section with a 'Select' button and X, Y, Z coordinate fields. The Rotate dialog includes '2. Select center of rotation', '3. Select start point', and '4. Select end point' sections, each with a 'Select' button and X, Y, Z coordinate fields. All three dialogs feature 'OK', 'Cancel', and 'Apply' buttons at the bottom.

Fig. 10: Geometry transformation settings

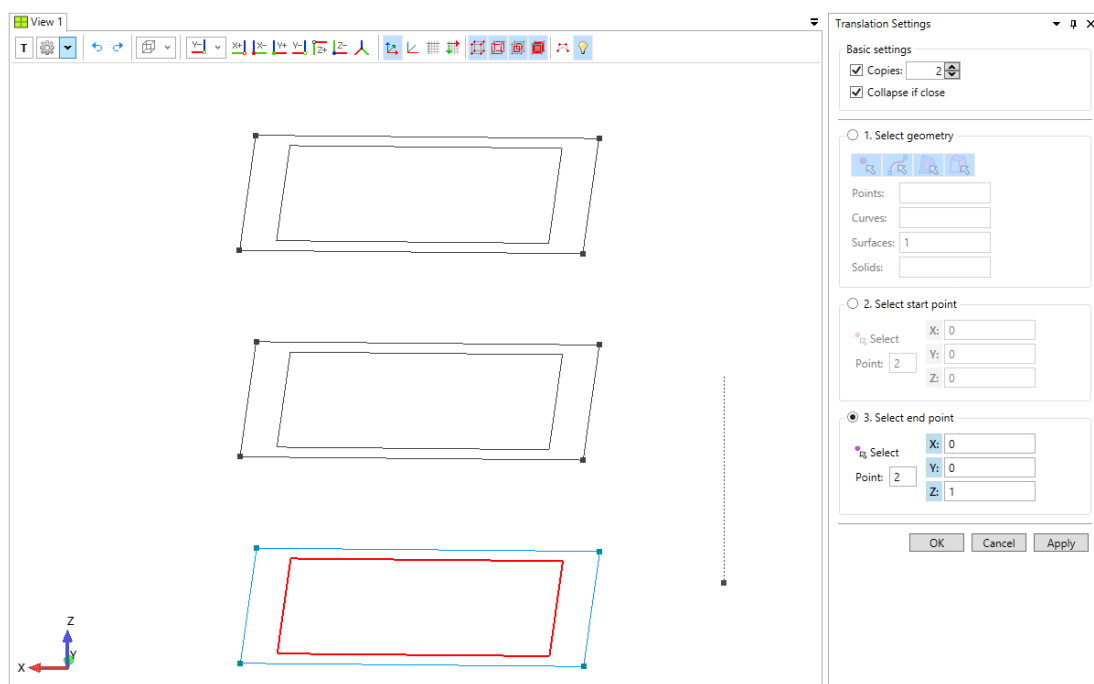


Fig. 11: Example of copying a surface during the translation.

3. 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, Iterface, 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, ...) Each material prototype has a specific material parameters, which is generated by this automatic tool.

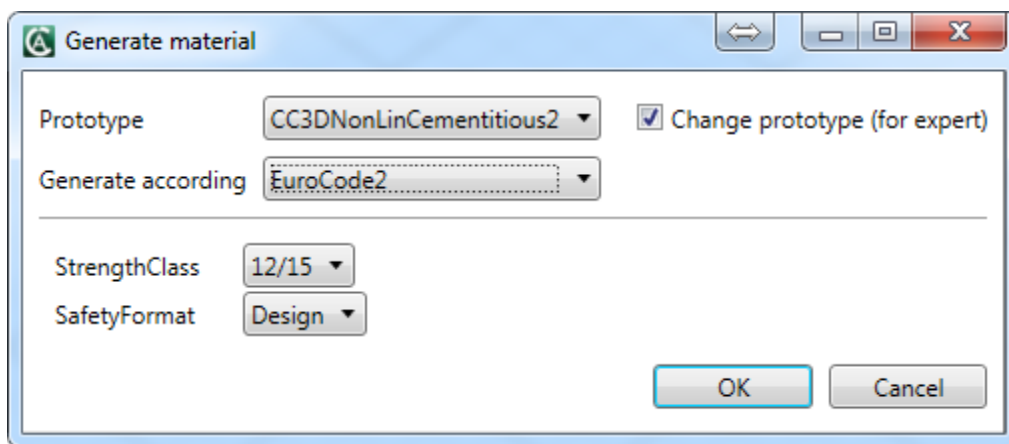


Fig. 12: Generator dialog for concrete

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.

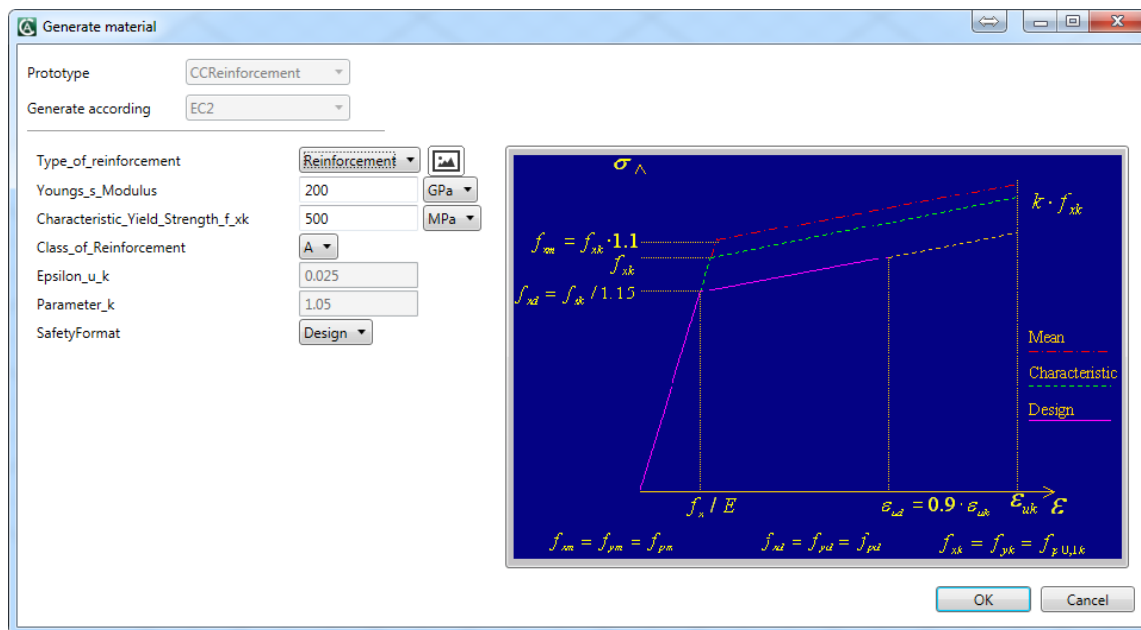


Fig. 13: Generator dialog for reinforcement

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).

```

concreteMaterial_SHCC.xml - Poznámkový blok
Soubor Úpravy Formát Zobrazení nápověda
<InputPython>
from cStringIO import StringIO
result = StringIO()

Function_for_COMP_SOFT_HARD = 22
Function_for_TENSION_SOFT_HARD = 21
Function_for_TENSILE_STRENGTH_RED = 24
result.write(function_param_to_inp("Compressive_Function", Function_for_COMP_SOFT_HARD))
result.write(function_param_to_inp("Tension_Function", Function_for_TENSION_SOFT_HARD))
result.write(function_param_to_inp("Ft_Reduction-COMPRED", Function_for_TENSILE_STRENGTH_RED))
mat_id = materialProxy.Id

if (materialProxy.MaterialParameters("Material_Prototype").GetValue() == "CC3DNonLinCementitious2SHCC"):
    result.write(header_to_inp("CC3DNonLinCementitious2SHCC"))
elif (materialProxy.MaterialParameters("Material_Prototype").GetValue() == "CC3DNonLinCementitious2SHCCWithTempDepPro"):
    Function_for_E_id = 1
    Function_for_FT_id = 2
    Function_for_FC_id = 3
    Function_for_EPS_T_id = 8
    result.write(function_param_to_inp("Function_for_E", Function_for_E_id))
    result.write(function_param_to_inp("Function_for_FT", Function_for_FT_id))
    result.write(function_param_to_inp("Function_for_FC", Function_for_FC_id))
    result.write(function_param_to_inp("Function_for_EPS_T", Function_for_EPS_T_id))
    result.write(header_to_inp("CC3DNonLinCementitious2SHCC", 10000, "Base for "))
else:
    result.write(header_to_inp("CC3DNonLinCementitious2SHCC", 1))
    
```

Fig. 14 : Example of python scripting of material Concrete SHCC

Concrete SHCC – Strain Hardening Cementitious Composite for Fibre Reinforced Concrete

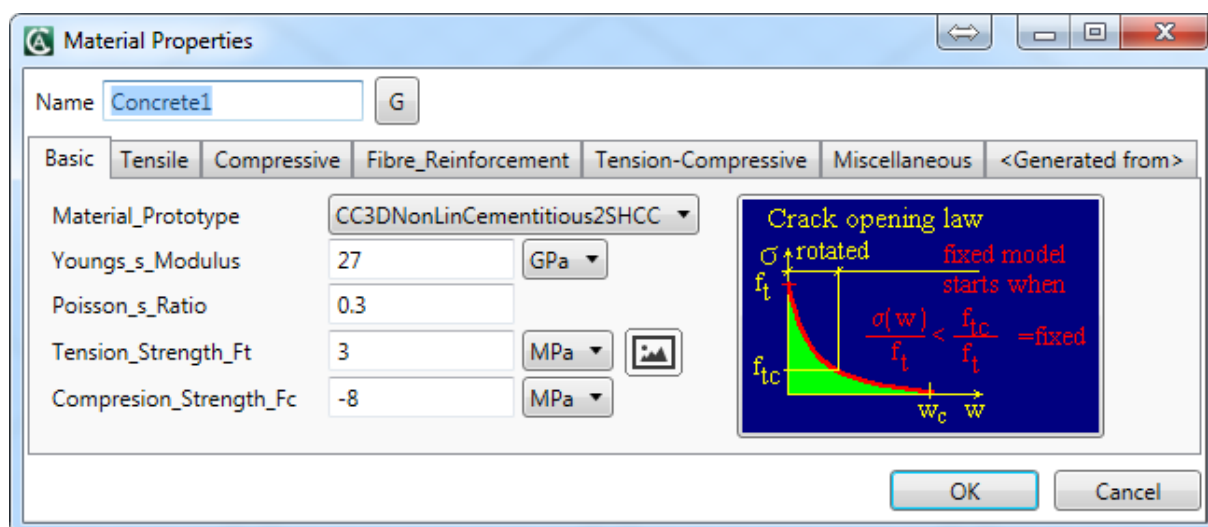


Fig. 15: Dialog Concrete SHCC – Basic parameters

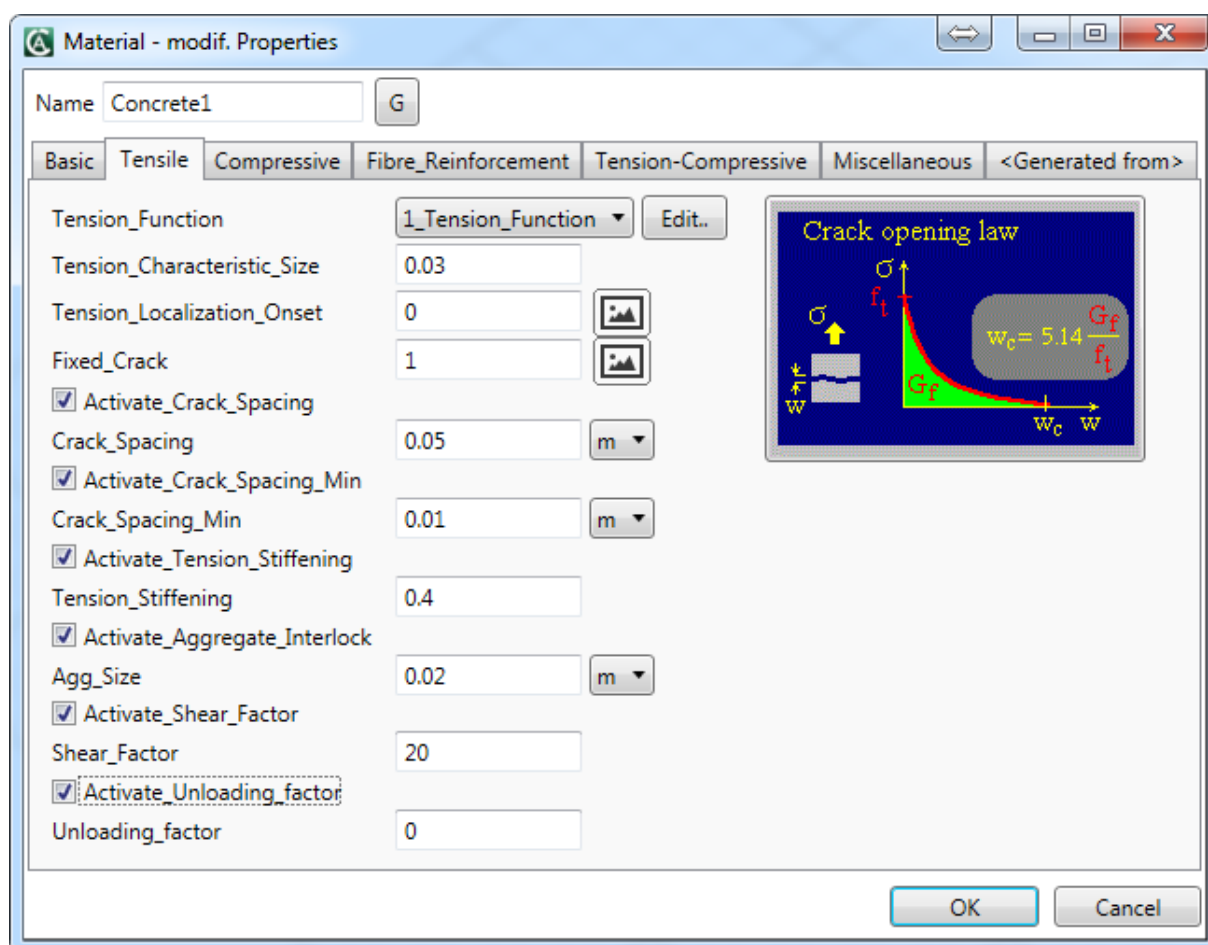


Fig. 16: Dialog Concrete SHCC – Tensile parameters

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

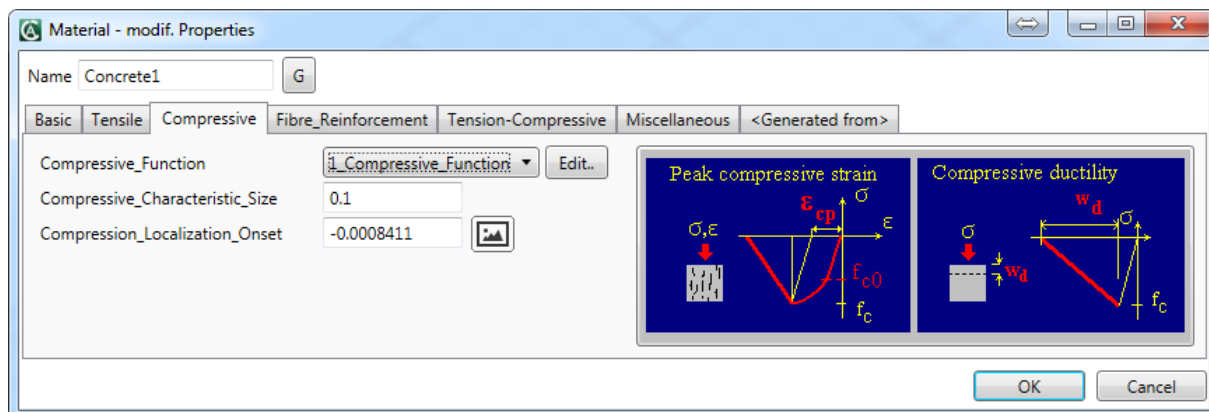


Fig. 17: Dialog Concrete SHCC – Compressive parameters

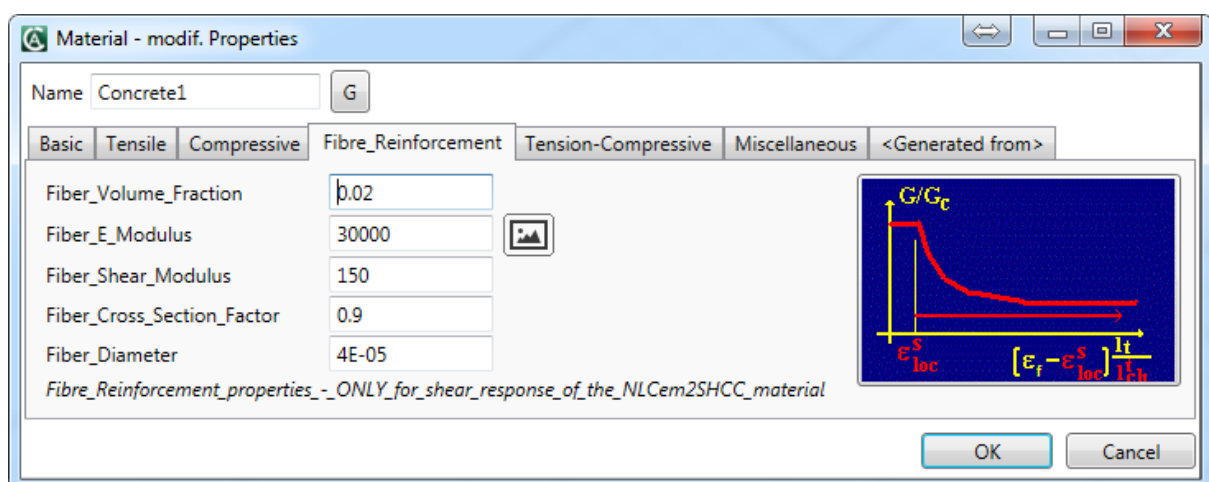


Fig. 18: Dialog Concrete SHCC – Fibre reinforcement parameters

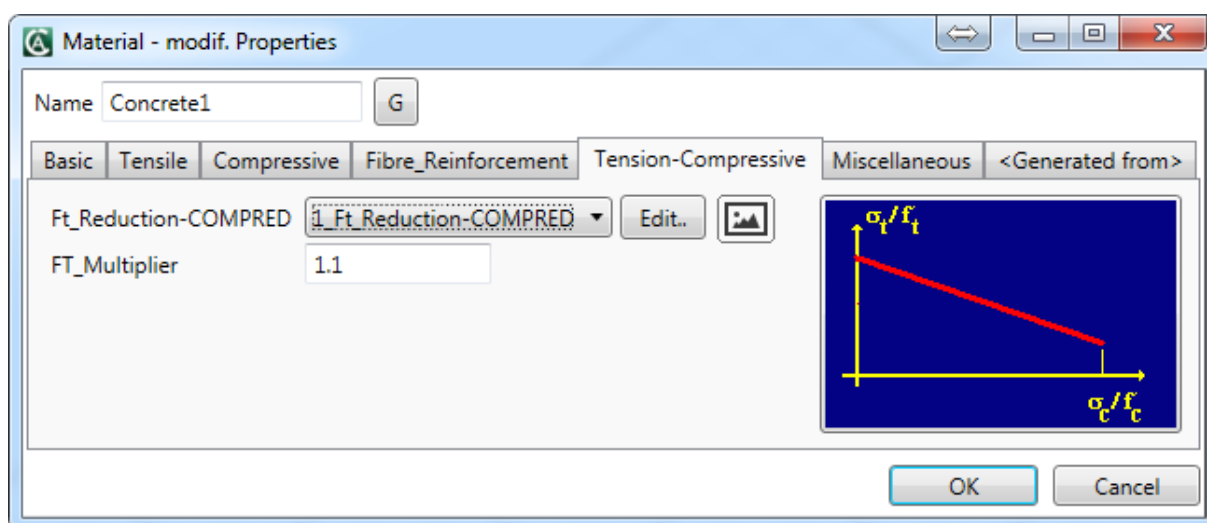


Fig. 19: Dialog Concrete SHCC – Tension-Compressive parameters

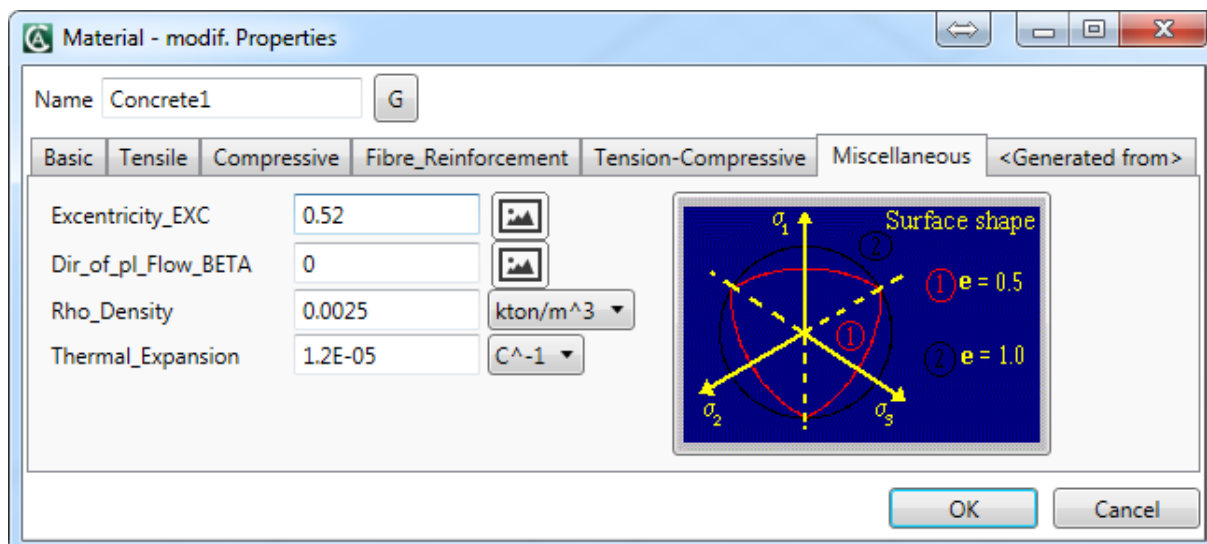


Fig. 20: Dialog Concrete SHCC – Miscellaneous parameters

4. 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.

4.1 ATENA in relation to BIM

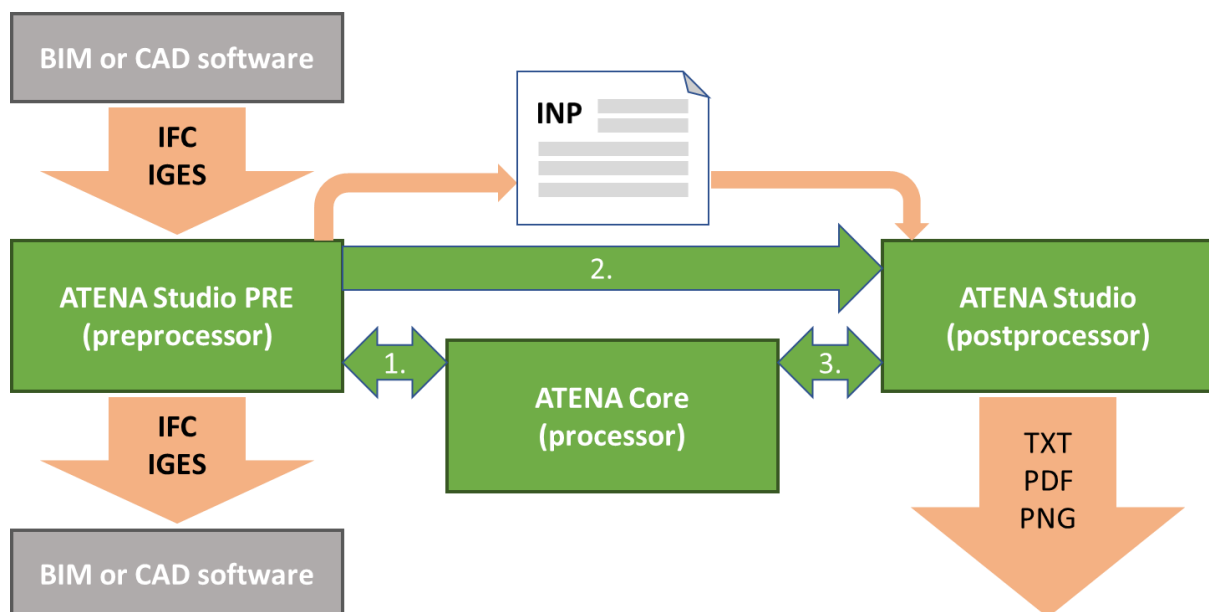


Fig. 21: Main ATENA modules and dataflow

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 it's CAD functions.

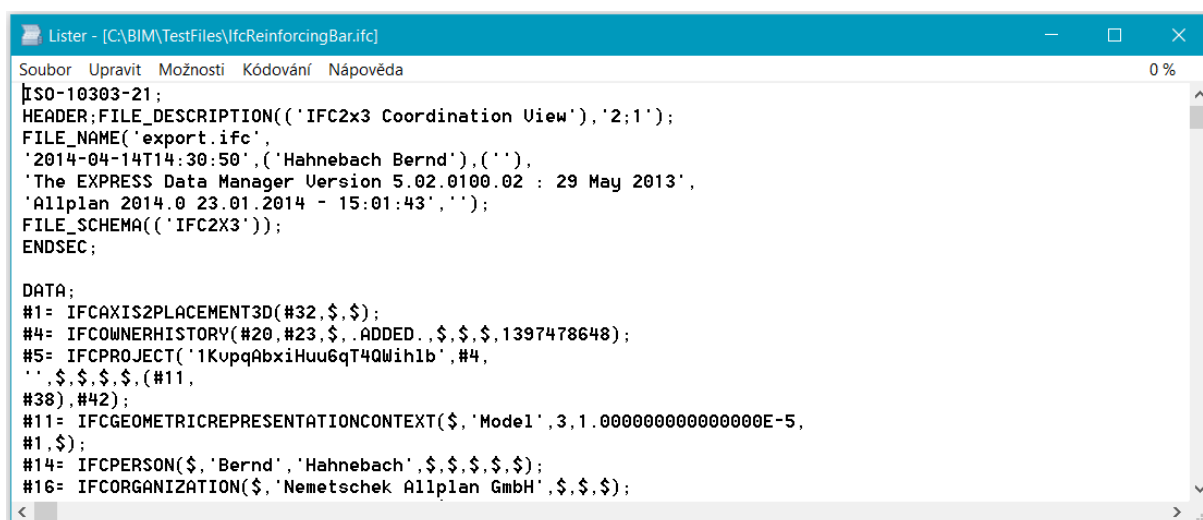
3. CAD model is converted to BIM model by creating building and reinforcing elements (non-graphical properties only) and assigning them to graphical objects.
4. FE mesh parameters are set globally or can be refined for individual elements of geometric model.
5. Geometric model is sent through data channel 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.

4.2 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.

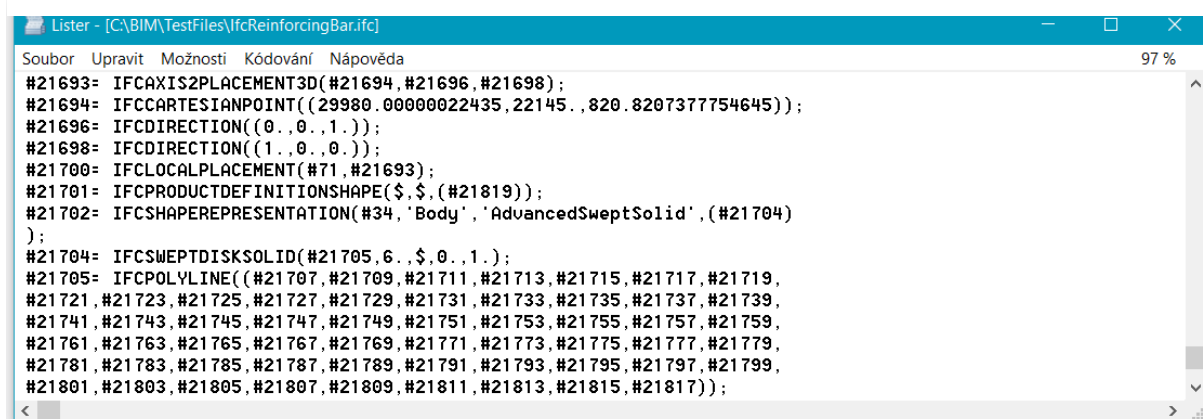
The IFC file was created in the **Nemetschek Allplan** application and exported as IFC2x3 Coordination View. This information and information about author you can see in the next picture.



```
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= IFCAXIS2PLACEMENT3D(#32,$,$);
#4= IFCOWNERHISTORY(#20,#23,$,ADDED,$,$,1397478648);
#5= IFCPROJECT('1KupqAbxiHuu6qT4QWih1b',#4,
'', $,$,$,$, (#11,
#38),#42);
#11= IFCGEOMETRICREPRESENTATIONCONTEXT($,'Model',3,1.000000000000000E-5,
#1,$);
#14= IFCPERSON($,'Bernd','Hahnebach',$,$,$,$);
#16= IFCORGANIZATION($,'Nemetschek Allplan GmbH',$,$,$);
```

Fig. 23: Header of IFC file used for import



```
#21693= IFCAXIS2PLACEMENT3D(#21694,#21696,#21698);
#21694= IFCARTESIANPOINT((29980.00000022435,22145.,820.820737754645));
#21696= IFCDIRECTION((0.,0.,1.));
#21698= IFCDIRECTION((1.,0.,0.));
#21700= IFCLOCALPLACEMENT(#71,#21693);
#21701= IFCPRODUCTDEFINITIONSHAPE($,$, (#21819));
#21702= IFCSHAPE REPRESENTATION(#34,'Body','AdvancedSweptSolid', (#21704)
);
#21704= IFCWEPTDISKSOLID(#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. 22: Definition of reinforcing bar prototype

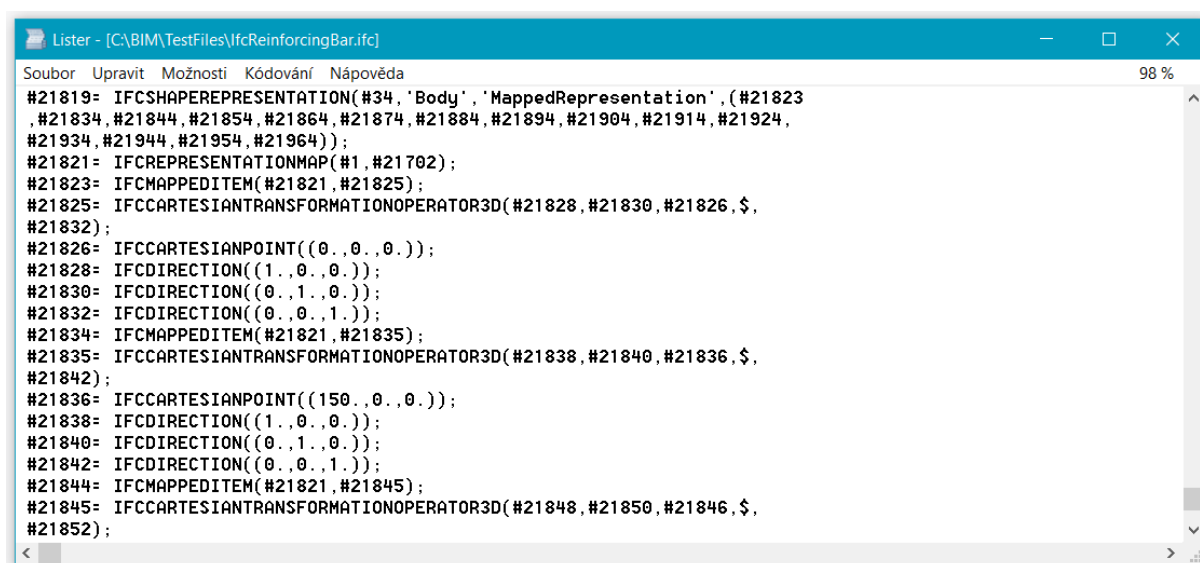


Fig. 24: Definition of reinforcing bar as mapped item

Tested IFC file was previewed in the **xBIM Explorer** viewer. The xBIM Explorer 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 Explorer and in the ATENA Studio PRE must exactly match.

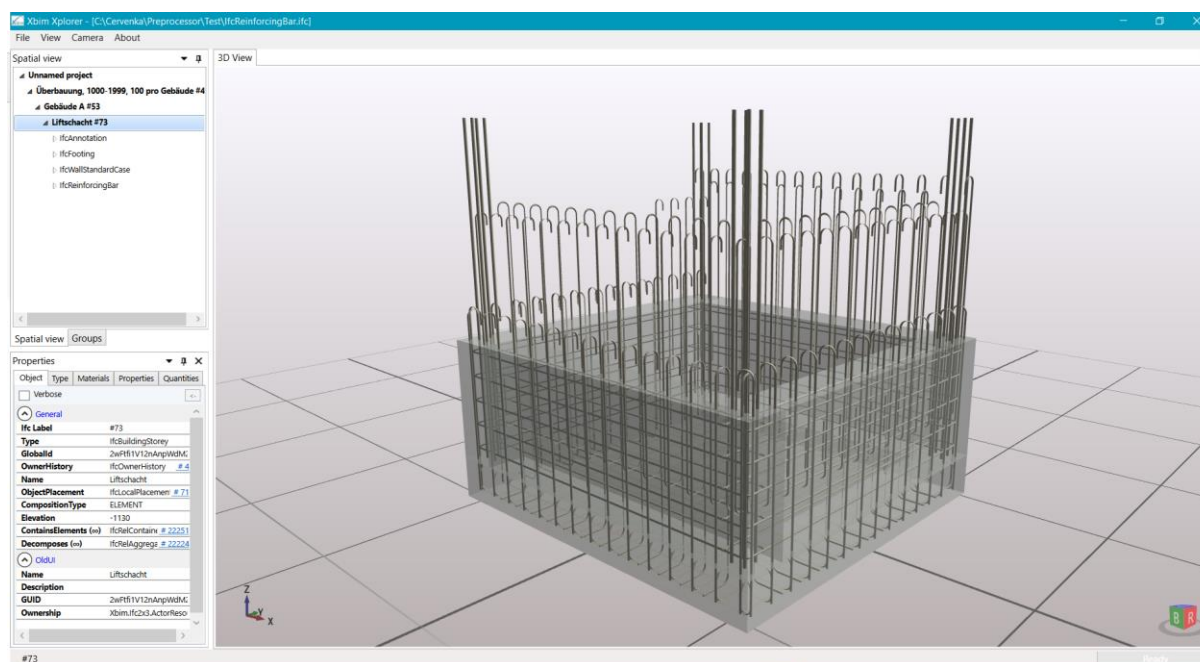


Fig. 25: IFC test file previewed in the xBIM Explorer

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.

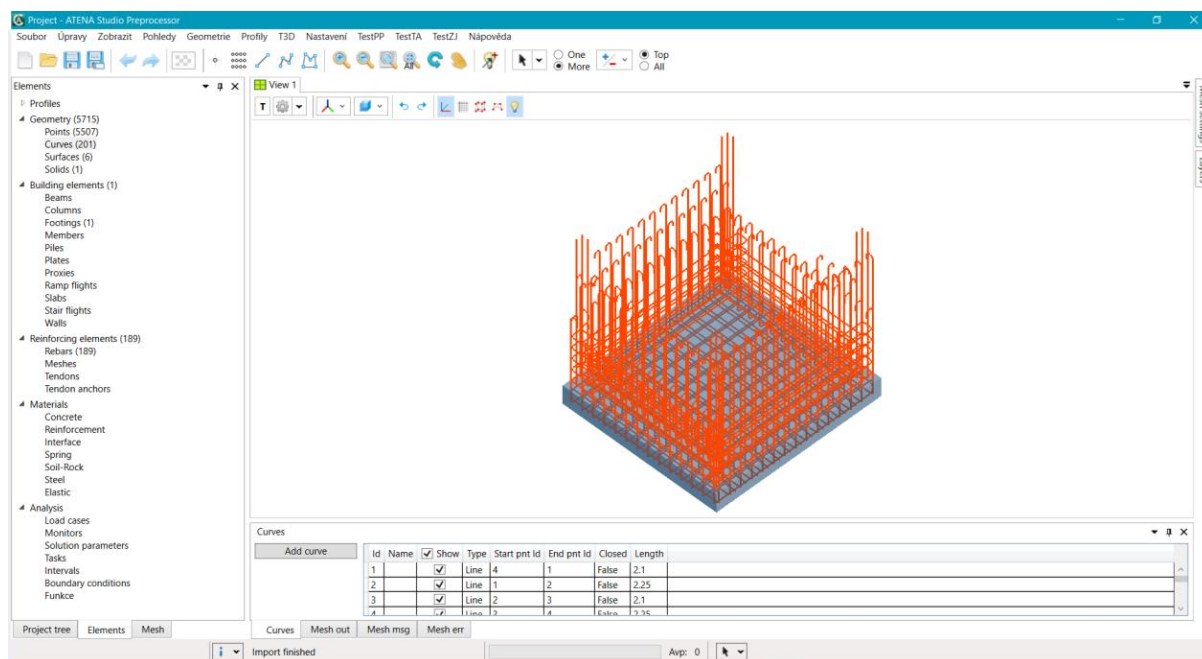


Fig. 26: IFC imported to the ATENA Studio PRE

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 $2.54\text{E-}04$ and $1.131\text{E-}4$. 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. 27: Details of imported reinforcing bars

4.3 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

4.3.1 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.

4.3.2 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.

4.3.3 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.

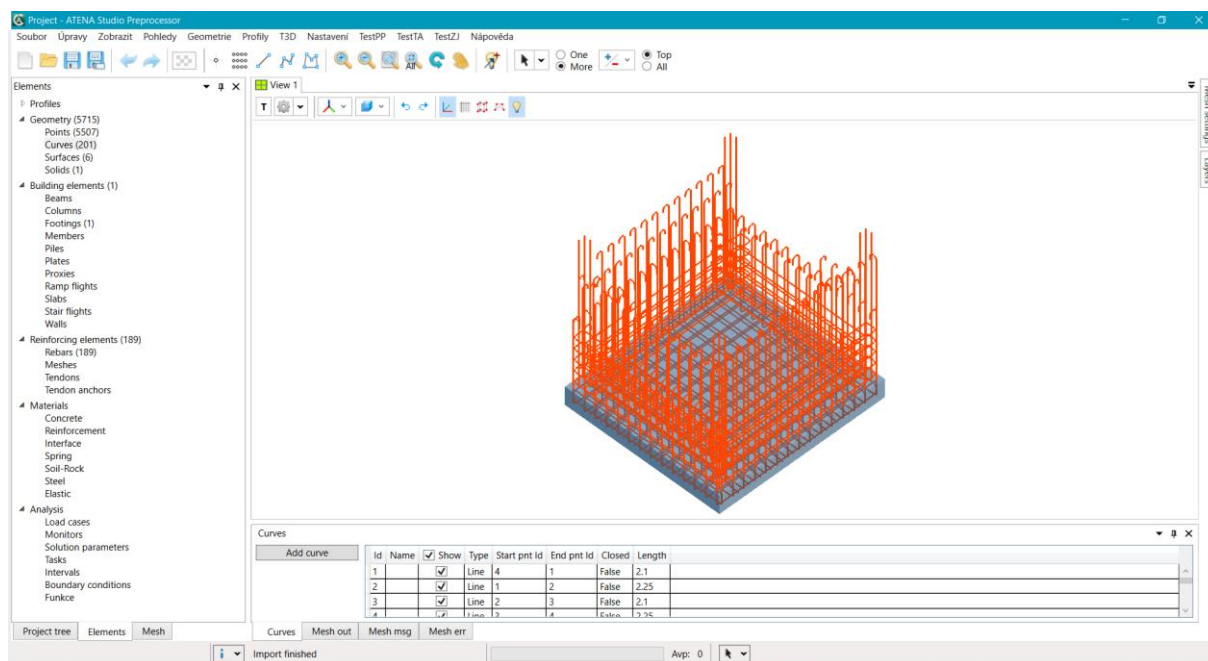


Fig. 28 Reinforcing bars imported from IFC to ATENA Studio PRE

4.3.4 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.

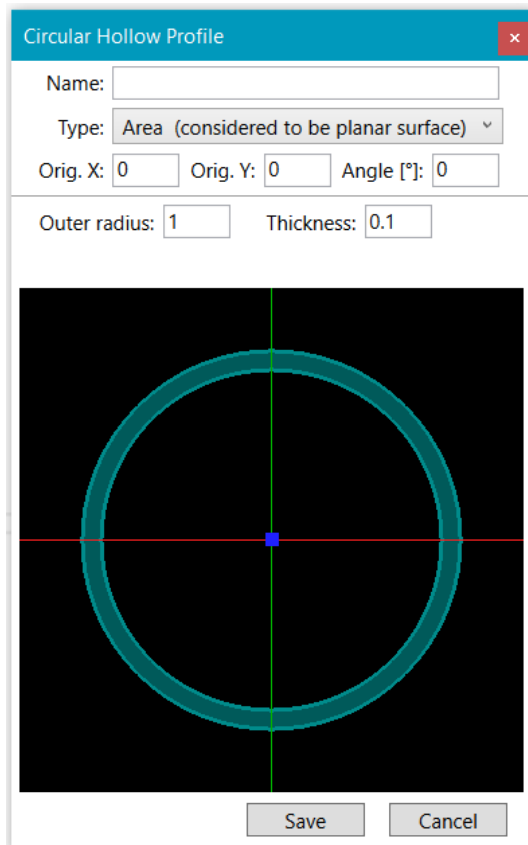


Fig. 30: Profile generator. IFC standard profile

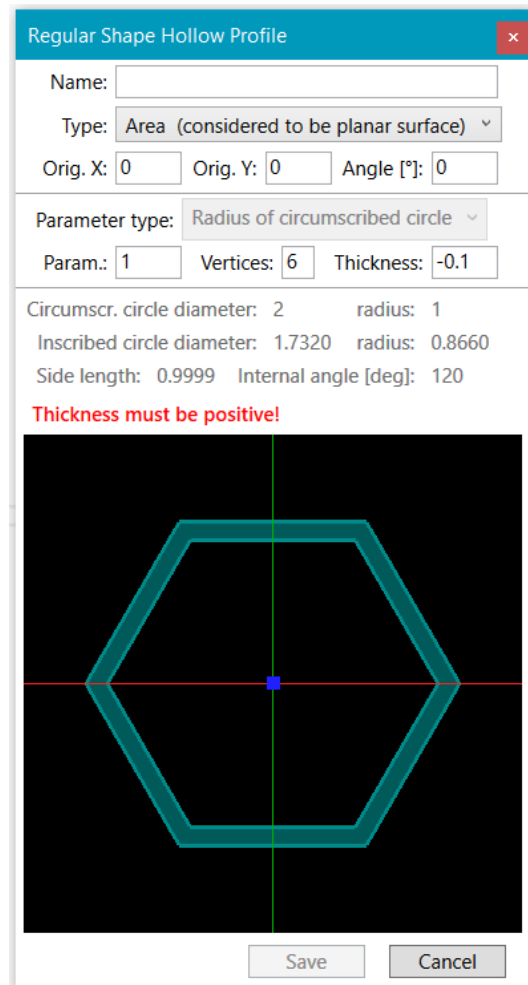


Fig. 29: Profile generator. ATENA Studio PRE parametric profile

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.

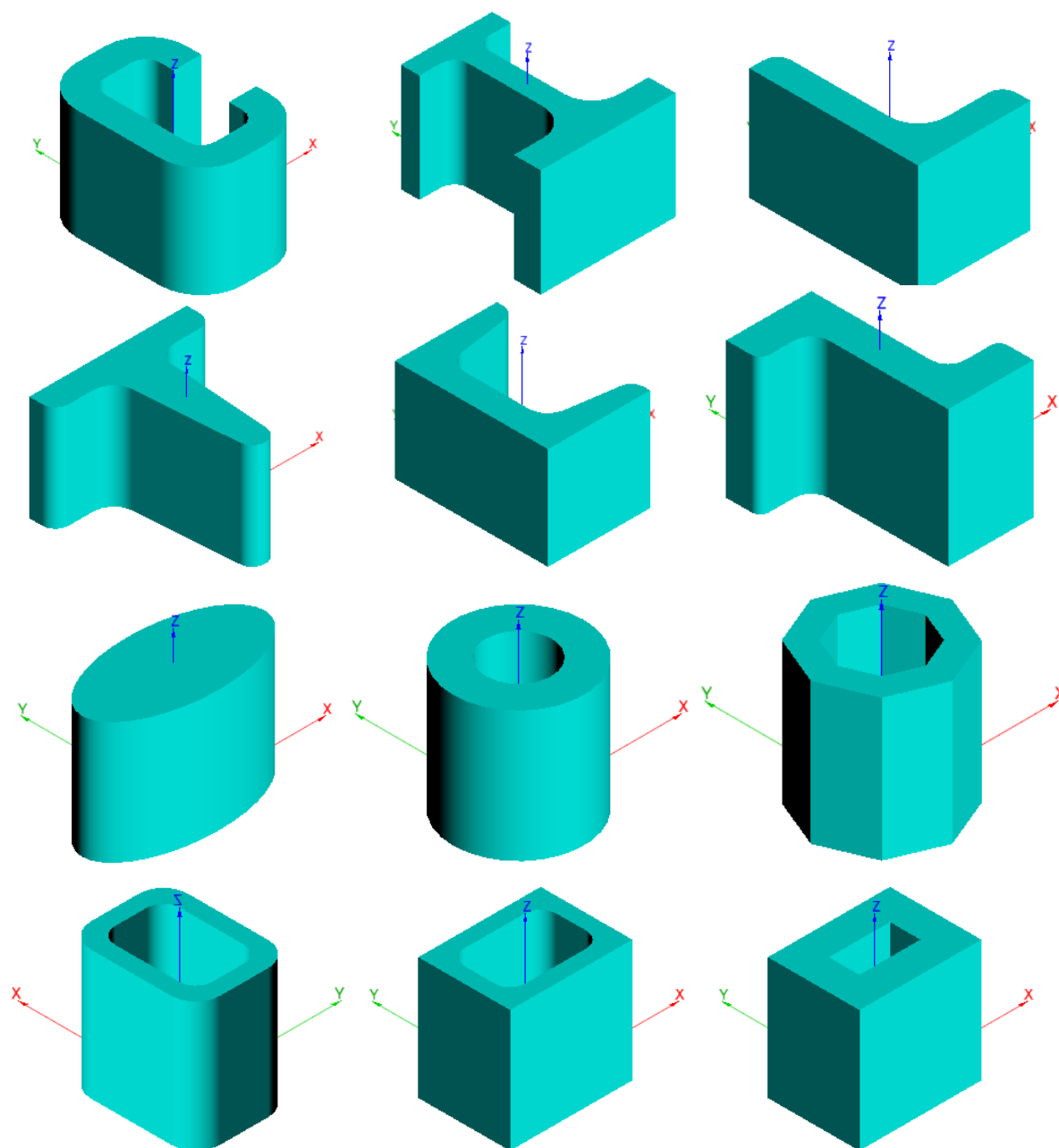


Fig. 31: Examples of parametric profiles supported by ATENA Studio Pre

4.3.5 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.

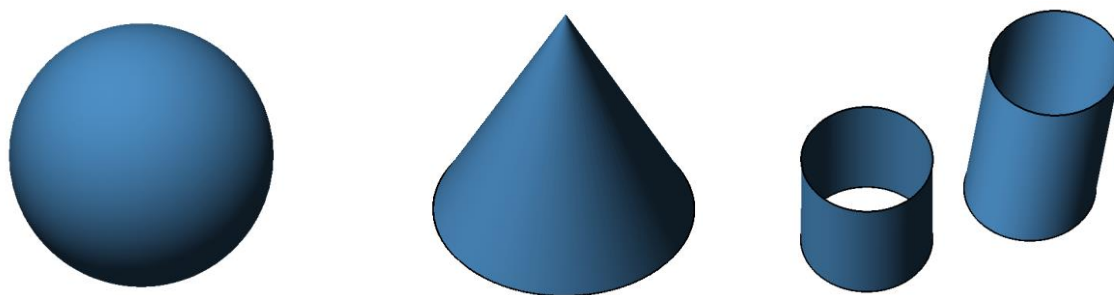


Fig. 32: Testing objects – parametric elementary surfaces

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.

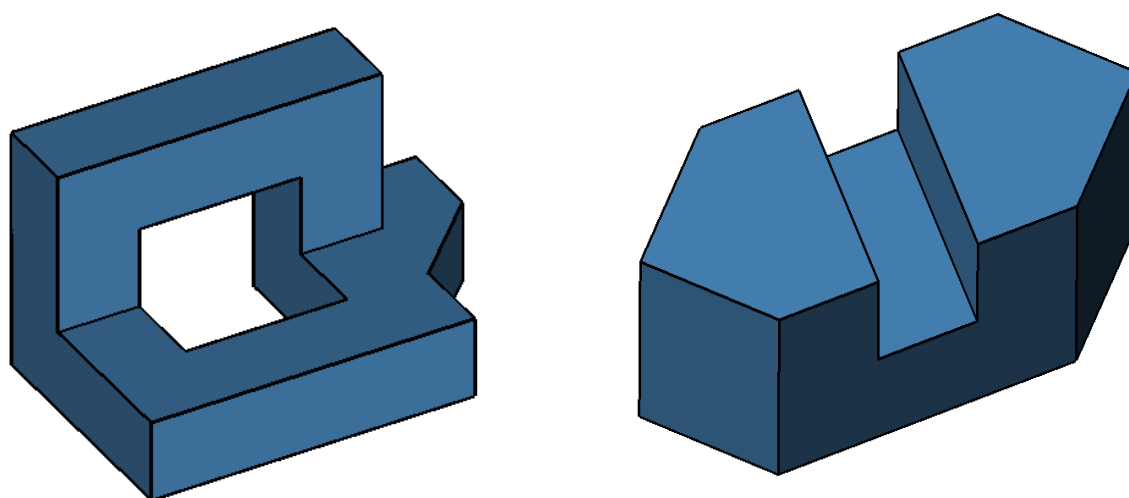


Fig. 33 Testing objects – solids representing as faceted breps

5. References

- [1] *fib* Model Code for Concrete Structures 2010. Wilhelm Ernst & Sohn, Berlin, Germany, (2013), ISBN 978-3-433-03061-5.
- [2] Cervenka, 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.
- [3] Cervenka V., Cervenka J., Jendele L., ATENA Program Documentation, Part 1 Theory, Prague, (2016), Cervenka Consulting, www.cervenka.cz.
- [4] Cervenka, V., Simulating a response, *Concrete Engineering International*, Vol. 4, No. 4, (2002) 45-49.
- [5] 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.
- [6] 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.
- [7] Cervenka, V., Global Safety Format for Nonlinear Calculation of Reinforced Concrete. Beton- und Stahlbetonbau 103, Special Edition, Ernst&Sohn, (2008) 37-42.
- [8] 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.
- [9] Consoft - software documentation (2015). Available at: <http://bauwesen.htwk-leipzig.de/nc/de/fakultaet-bauwesen/personen/name/slowik/>
- [10] 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).
- [11] Pukl, R., Havlásek, P., Vitek, P., Vitek, 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.
- [12] 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
- [13] D. Novák, D. Lehký, ANN Inverse Analysis Based on Stochastic Small-Sample Training Set Simulation, Engineering Application of Artificial Intelligence 19 (2006) 731-740.

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 2017, 2018

© Cervenka Consulting s.r.o. 2017, 2018

© JKP Static 2017, 2018