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

ATENA2025
Program Documentation

Tutorial for ATENA Engineering

Written by
Jan Červenka, Tomáš Altman, Zdeněk Janda,
Jiří Rymeš, Michaela Herzfeldt,
Pavel Pálek and **Radomír Pukl**

Prague, April 15, 2024

Acknowledgements:

Various ATENA software modules were developed with partial support of various national and international funding programs:

ATENA 2024 Engineering development has been supported by the Technology Agency of the Czech Republic under the project TM01000059 "Reducing material demands and enhancing structural capacity of multi-spiral reinforced concrete columns – advanced simulation and experimental validation" within the DELTA Programme.

ATENA3DPrint module has been developed within the project TF04000051, "digiCON2 – Simulation Service for Digital Concrete Production" funded by TA CR under the DELTA Programme.

ATENA TwinBridge module containing the support for TwinBridge platform integration and models for long time material degradation: chloride ingress, carbonation, reinforcement and bond corrosion model, alkali-silica reaction material degradation was developed under the project CK03000023 "Digital twin for increased reliability and sustainability of concrete bridges" within the DOPRAVA 2020+ Programme.

T A

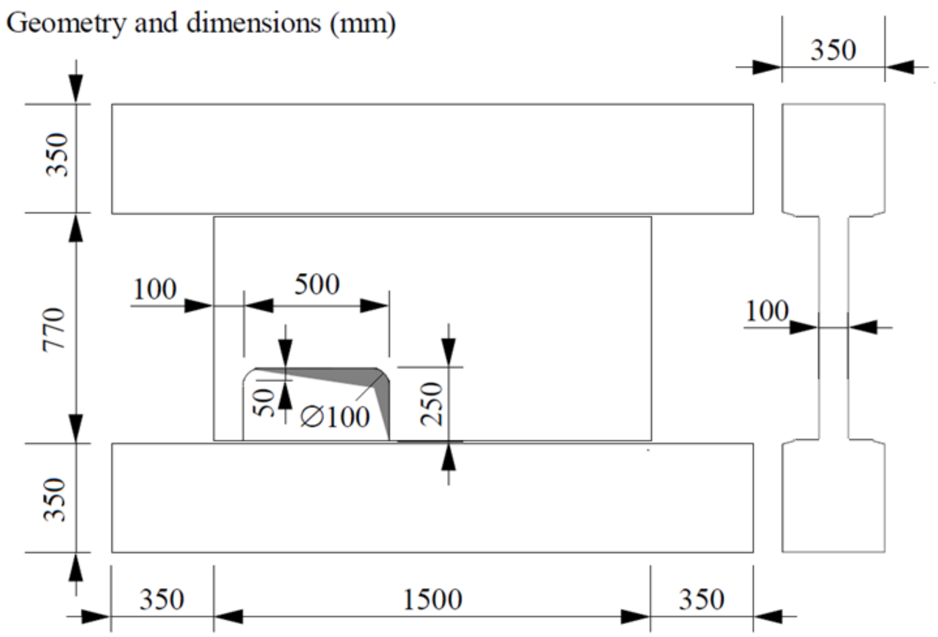
Č R

CONTENTS

1 INTRODUCTION

This tutorial introduces **ATENA-PRE** as a preprocessor for **ATENA-Studio**. ATENA software is developed for nonlinear finite element analyses of reinforced concrete structures [1]. The following text is an example of model development in ATENA-PRE where user can define model geometry, materials, boundary conditions etc. Such an example is presented in following text to show how to prepare the model for simulation in step-by-step manner.

An example of modelling in **ATENA-PRE** is shown on shear wall with opening which has round corners, loaded by lateral force. The model is based on the blind test calculation organized by French institute CEBTP [2]. Model itself consists of loaded top beam and supporting bottom beam. Middle panel and the beams are reinforced with mesh of steel bars while the edges of panel and opening are reinforced with extra bars. Geometry along with reinforcement arrangement is shown in Fig. 1-1.



Discrete reinforcement

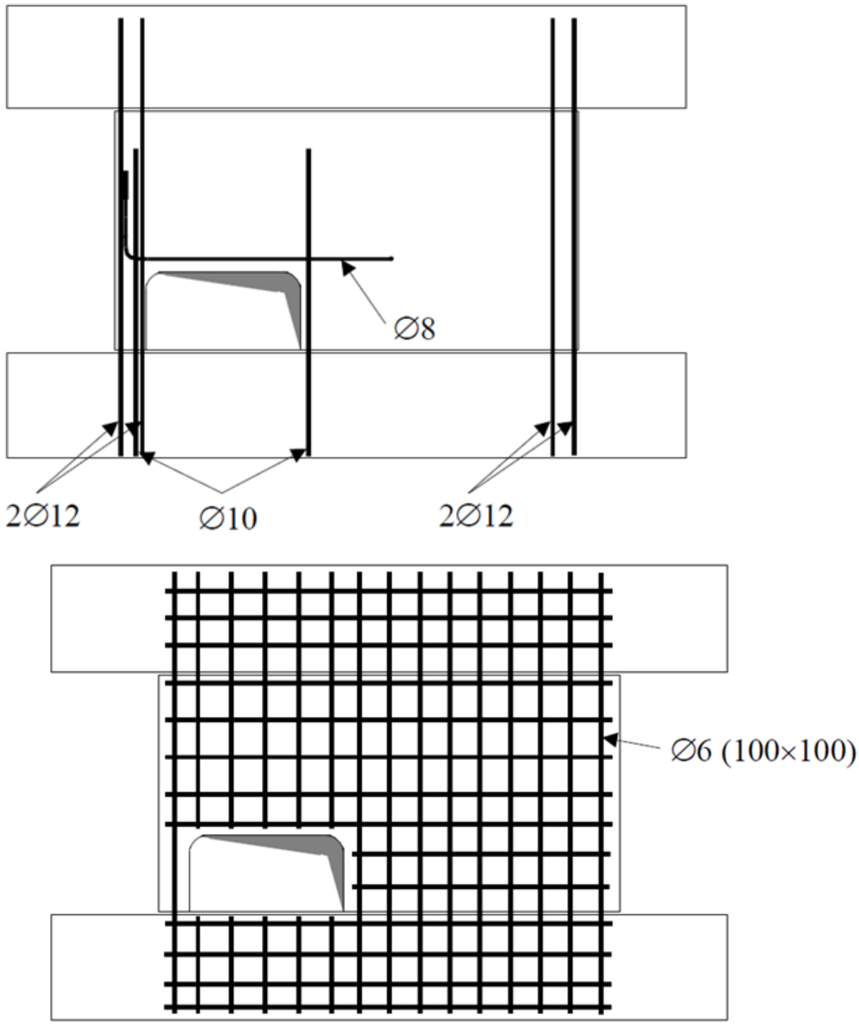


Fig. 1-1: Geometry and reinforcement of the model.

2 STARTING PROGRAM

At first the program has to be installed. The latest version of ATENA 2024 is available on the www.cervenka.cz website in section Downloads. When the downloaded archive is unpacked and installed, **ATENA Center 2024** can be found using Windows search bar for applications (Windows logo key + S) or in default directory:

C:\Program Files\CervenkaConsulting\ATENA 2024\AtenaCenter\ATENA Center.exe

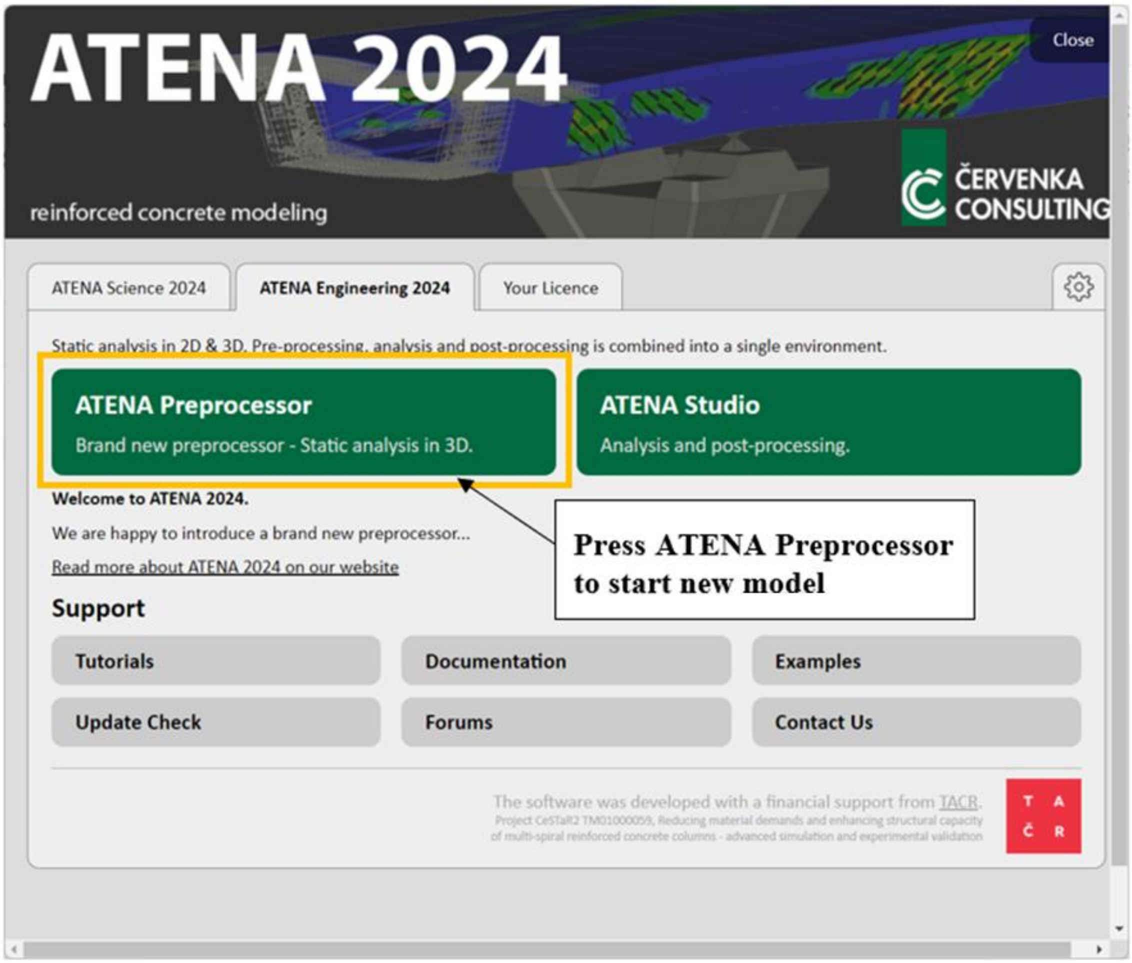


Fig. 2-1: ATENA Center interface.

ATENA Center is user-friendly environment that gives users access to all ATENA software, documentation, examples etc.

Otherwise ATENA-PRE is located directly in:

C:\Program Files\CervenkaConsulting\ATENA 2024\AtenaStudio.Preprocessor.exe

In order to clarify the terms which are used in following text to describe the parts of **ATENA-PRE** environment, see the Fig. 2-2 to identify respective parts such as Main menu, GUI toolbar, Input data tree, Workspace view, Layers window, Data table (input review, modelling info), Script history, Log and Coordinates and snapping. General information about ATENA-PRE can be found in [3].

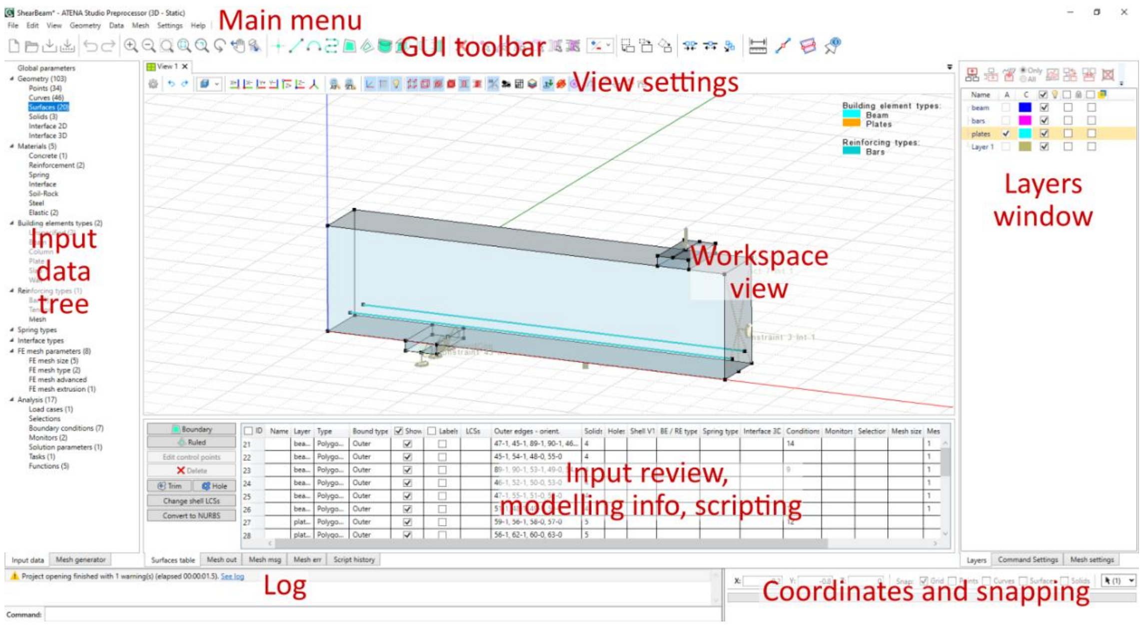


Fig. 2-2: ATENA-PRE interface.

New project button is at the beginning of the Main menu or it can be selected in the **File | New project** (Ctrl + N). The New project dialogue appears and the name of the project with its location is adjusted.

Parameters input:	
Project name	SWO_round

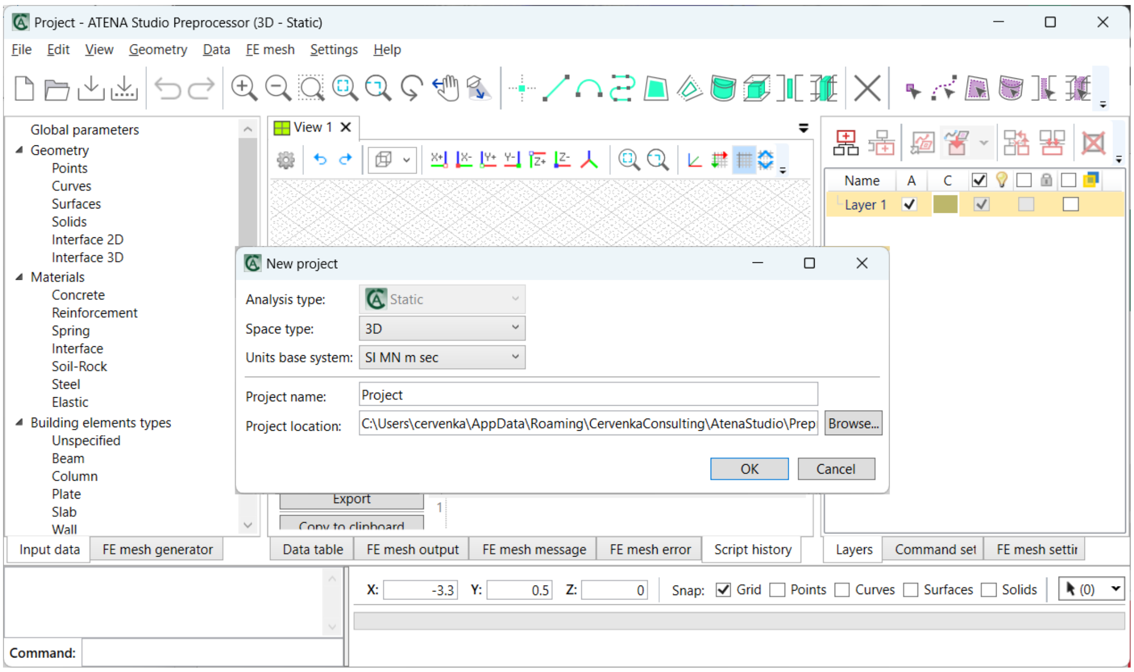


Fig. 2-3: New project in ATENA-PRE.

3 PREPROCESSING

3.1 Geometry

The geometry of the model can be prepared in various ways but one of them is going to be presented below including the information of other possible approach.

3.1.1 Concrete structure

It might be very helpful to prepare the grid line distance in Workspace view related to the size of structural parts. The first part of the structure that is created is the middle panel which has sizes with common multiple 50 mm. Therefore the grid size of 50 mm is set to create the points of the middle panel. In **View settings** there is **Viewport settings** | **Viewports** | **Grid** (Fig. 3-1) and then it is set:

Parameter input:
|Uncheck Automatic adjustment to predefined views
Distance between lines: 0.05 m
Major line every: 10 lines

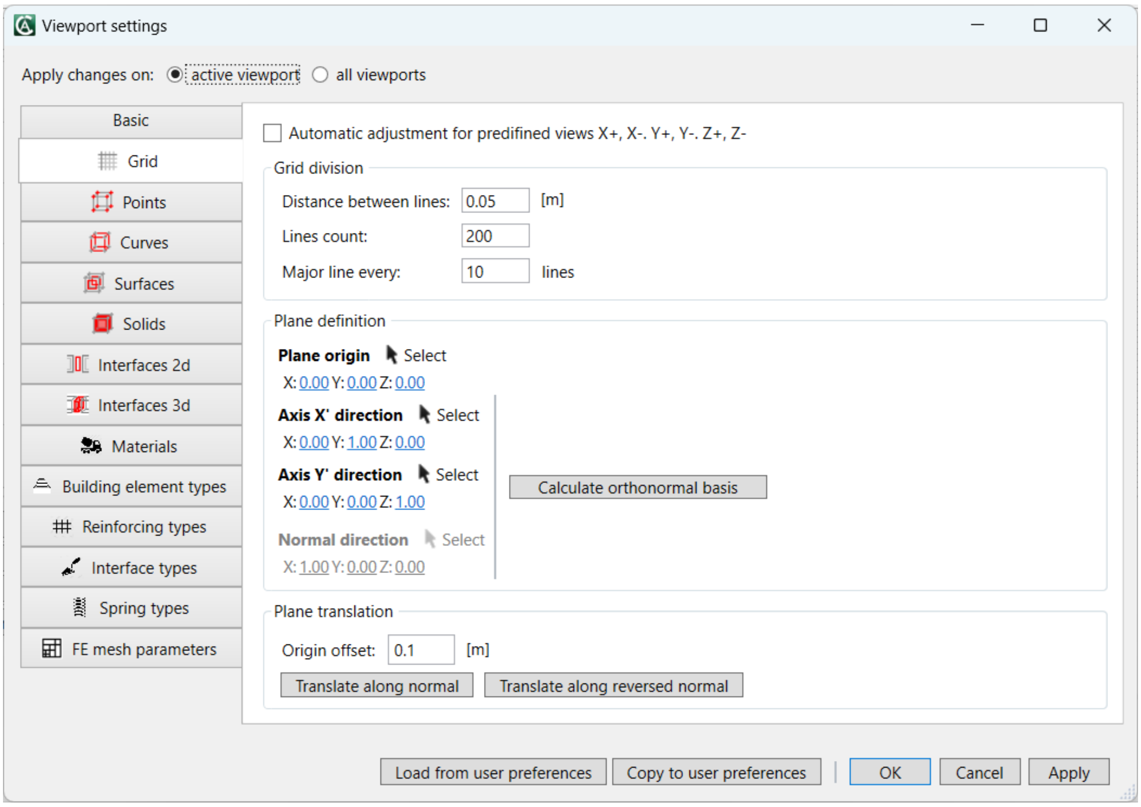


Fig. 3-1: User preferences-Grid settings.

When the new grid is ready clicking on **Create points** icon in **GUI toolbar** and **Point create setting** tab gets activated in **Layers** window location, see Fig. 3-2. Then the points' coordinates can be input or using mouse cursor the points can be inserted by clicking mouse. The first point has coordinates 0,0,0 which you can enter in **Point create**

setting tab or using the **View settings** button **Show grid axes** to see the origin. Other points can be added just by clicking on grid with corresponding distance. Pressing ESC the inserting of points is concluded.

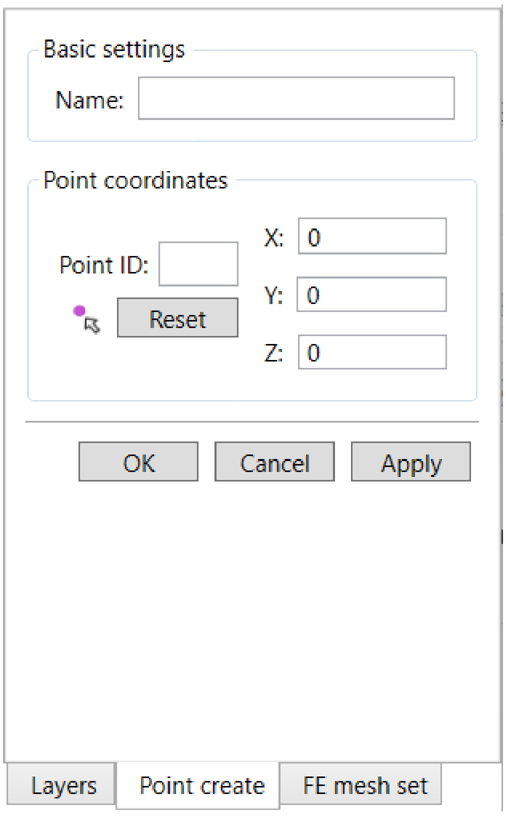


Fig. 3-2: Point create settings.

Parameters input:

Label	Coordinates X,Y,Z
1	[0, 0, 0]
2	[0, 0.1, 0]
3	[0, 0.6, 0]
4	[0, 1.5, 0]
5	[0, 0.6, 0.2]
6	[0, 0.1, 0.2]
7	[0, 0.15, 0.25]
8	[0, 0.55, 0.25]

Then in **Points table** points‘ labels might be activated in the **Workspace** by selecting checkbox **Labels** shown in Fig. 3-3, points that have been created and their labels.

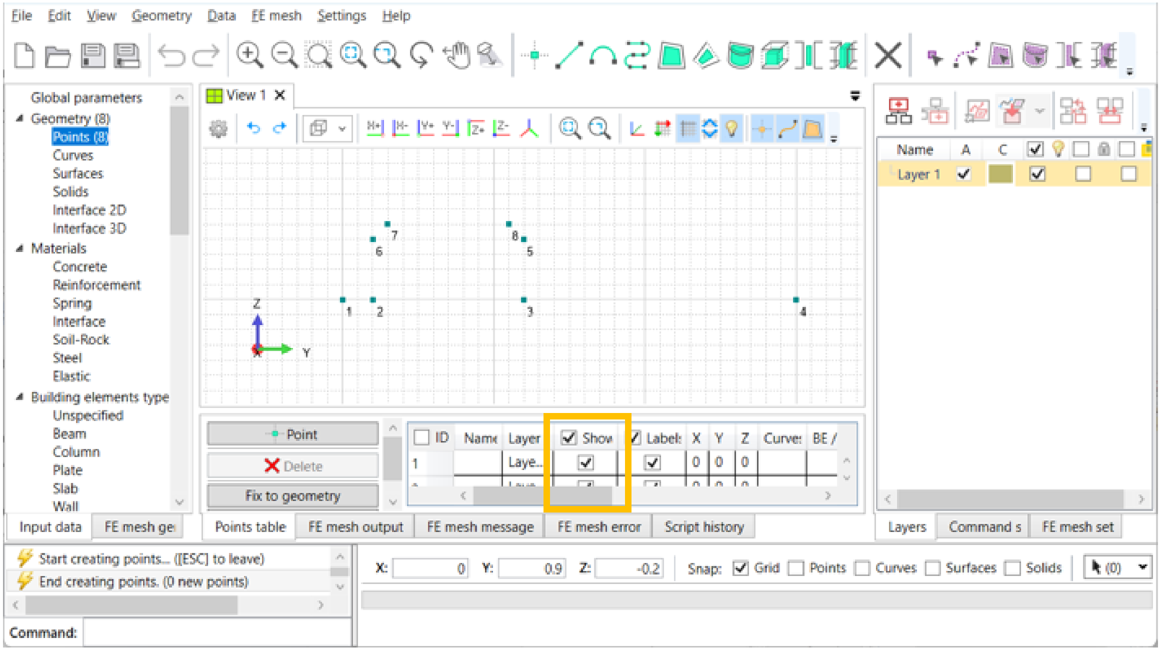


Fig. 3-3: Created points with labels.

Last two points of middle panel are not directly grid points so they can be added just by their coordinates or by translation of existing points. In this case the points 1 and 4 are translated as a copy in Z direction by 0.77 m. To translate points the tool **Translate geometry**  is used to translate and copy the original points with settings shown in Fig. 3-4. Copy checkbox must be ticked otherwise the original points would be just moved.

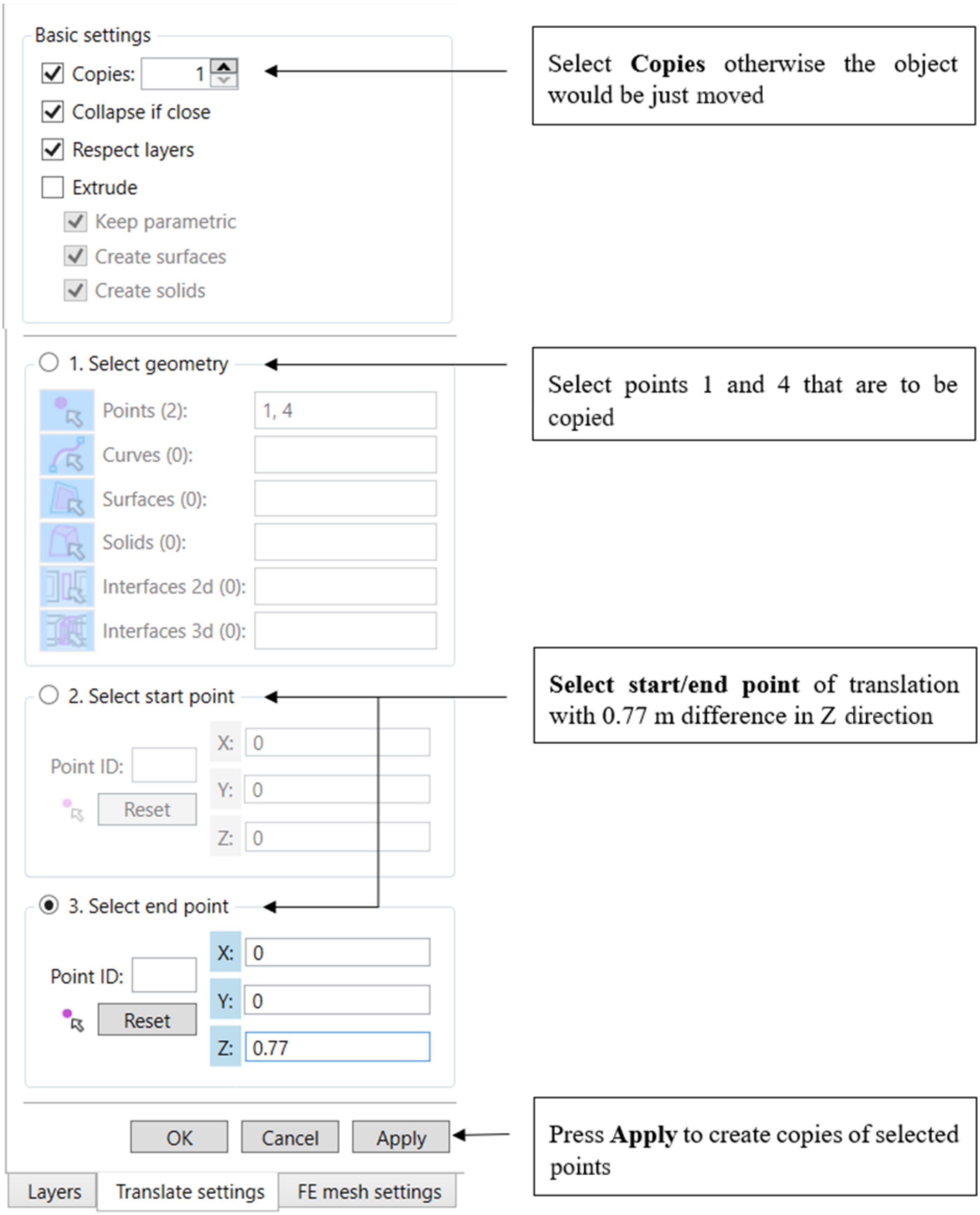


Fig. 3-4: Translate geometry – Copy settings.

When all points of middle panel are created, tool **Create line segments** is used for connecting them with lines, Fig. 3-5.

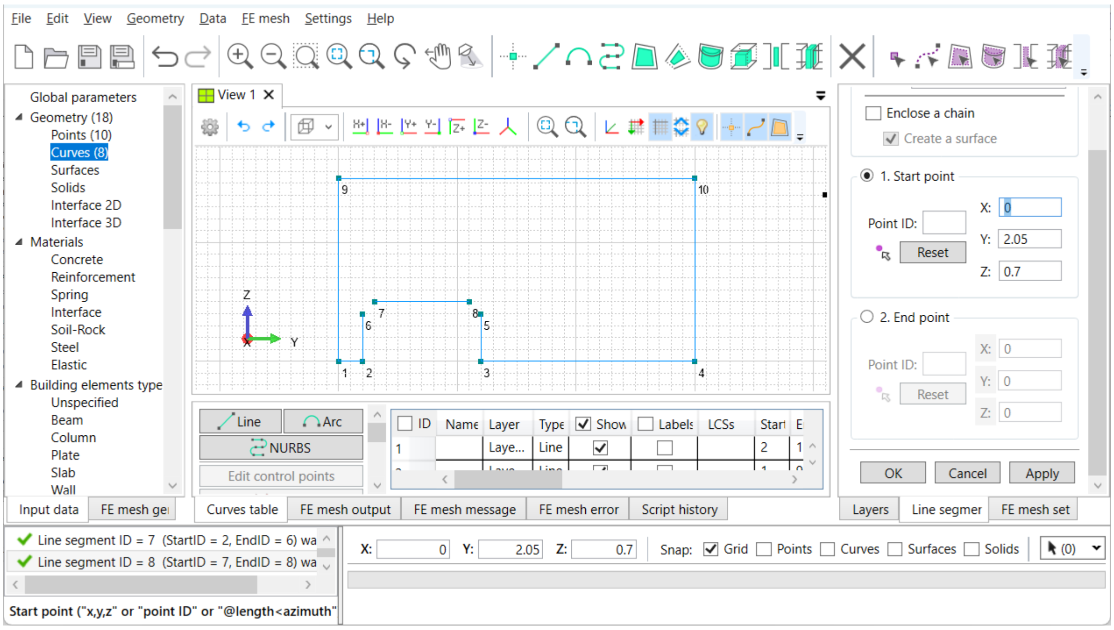
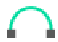


Fig. 3-5: Points connected with lines.

Last part of middle panel are arcs at the corners of opening. **Create circular arcs**  button opens **Circular arcs setting** (Fig. 3-6) where method **Start, end, radius** is selected along with start (point 6) and ending point (point 7) of arc directly in Workspace view or in **Circular arc settings** and the radius 0.05 m is defined. The mouse cursor is used to choose orientation of arc or coordinates of point that belongs to same half-plane as center of arc is directly typed in, see Fig. 3-7. Repeating the procedure for point 5 and 8 the other arc of the opening corner is created.

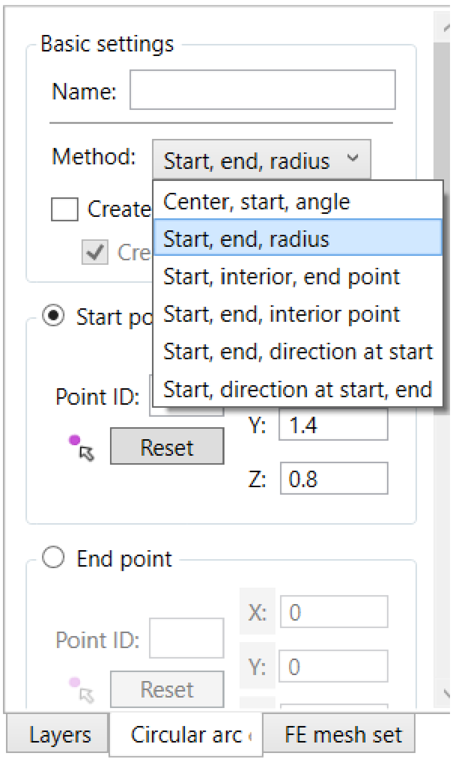


Fig. 3-6: Circular arc settings.

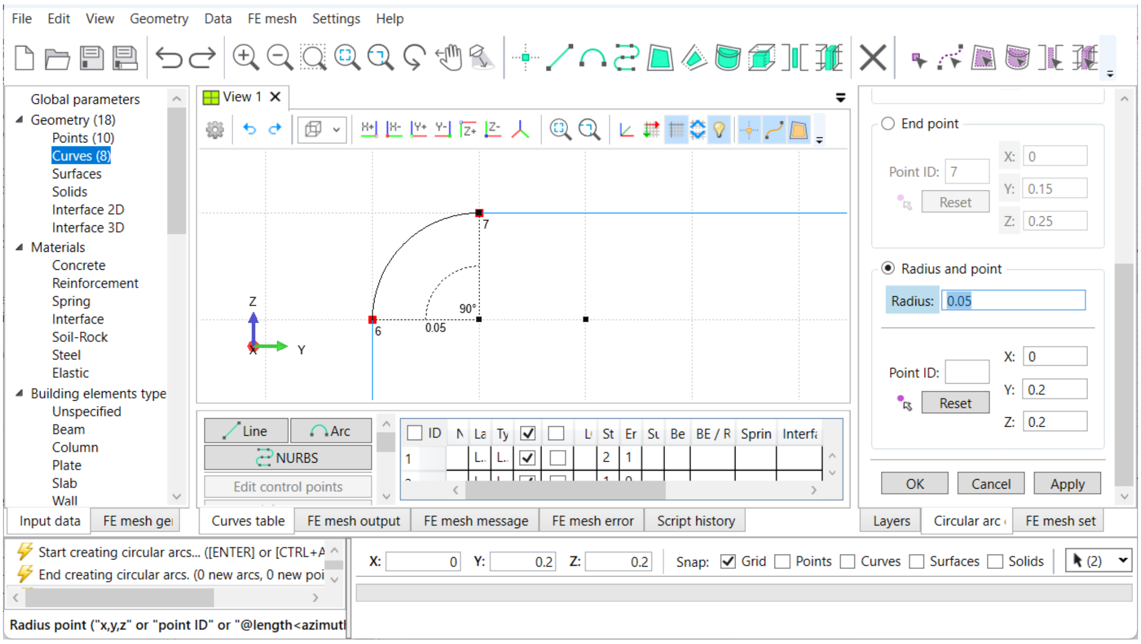
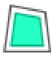


Fig. 3-7: Create circular arc with settings.

Then the surface is created using **Create surface from boundary curves**  and all lines in model are selected as surface boundary finishing with ENTER. Once the surface is created the purple line along the surface boundaries appears, see Fig. 3-8.

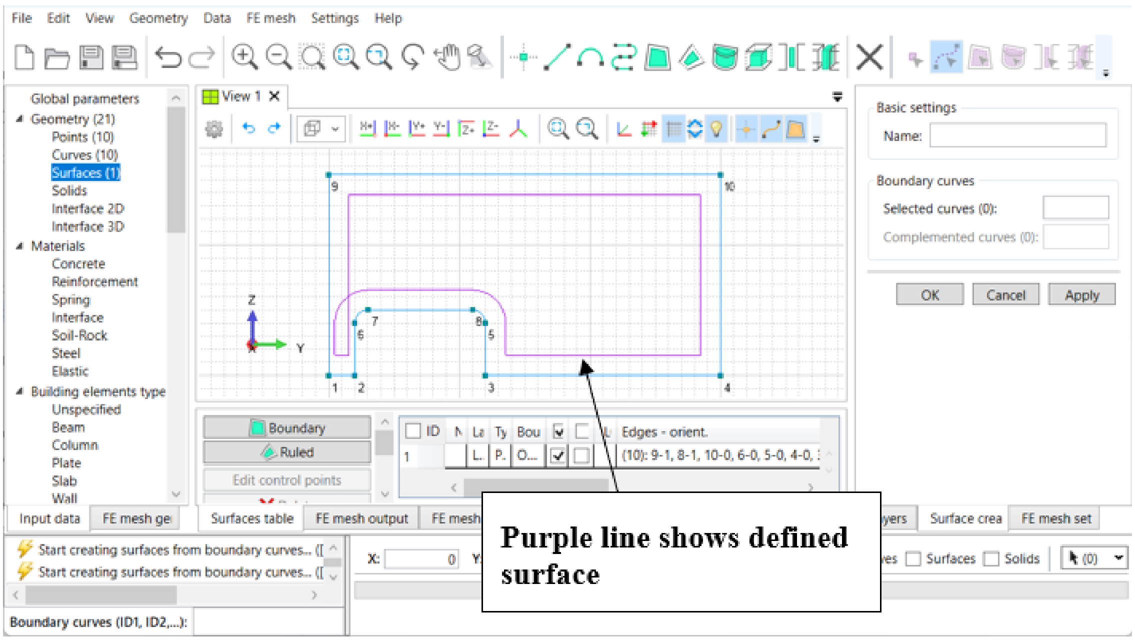


Fig. 3-8: Surface with boundary lines.

The surface of the panel is then extruded to create solid using tool Translate geometry  with Extrude checkbox ticked (Fig. 3-9), surface number 1 selected (Fig. 3-10), starting point and end point with difference 0.1 m in X direction defined. Pressing Apply the newly created solid is outlined in turquoise color as shown in Fig. 3-11.

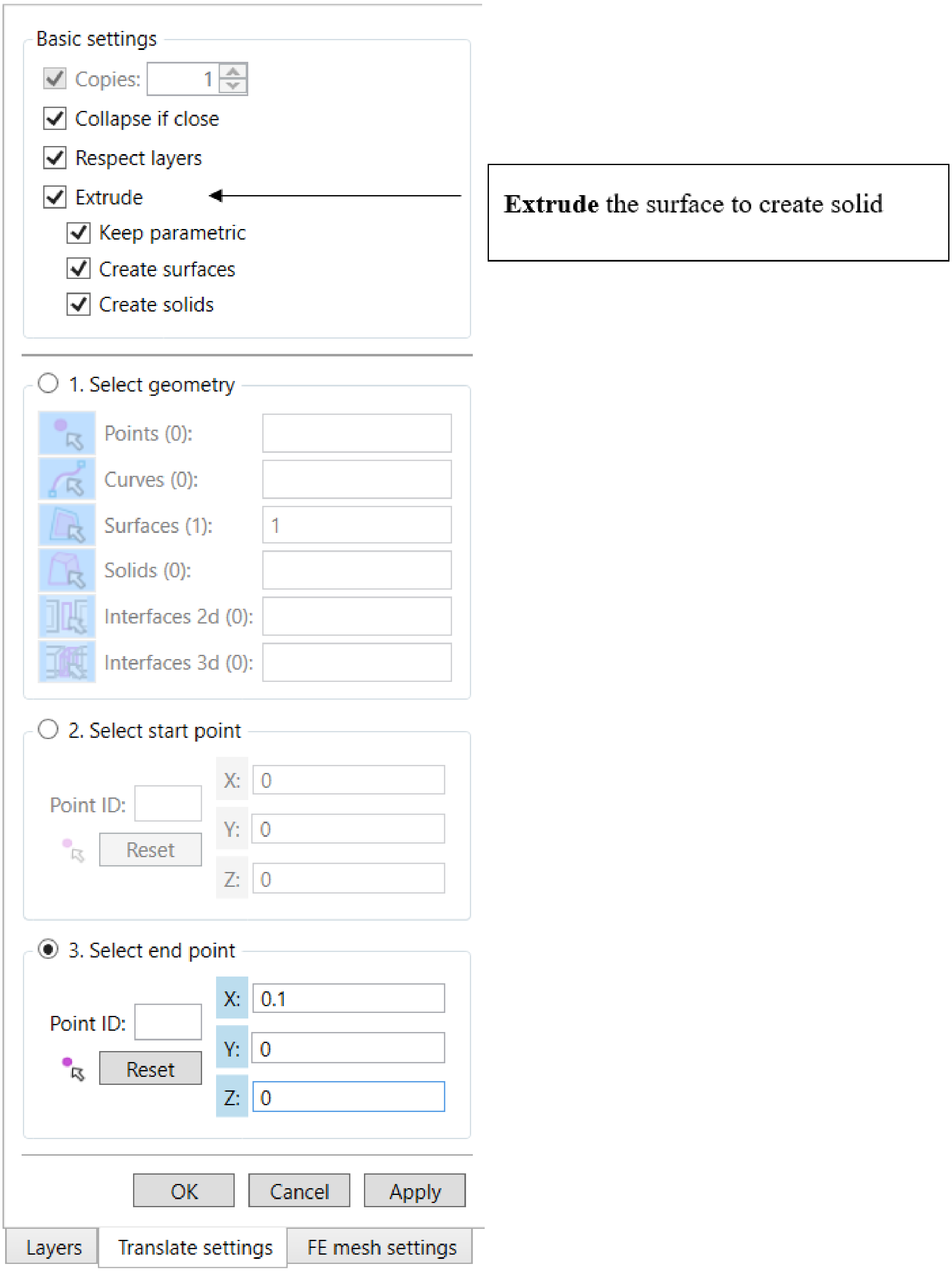


Fig. 3-9: Translate geometry – Extrude settings.

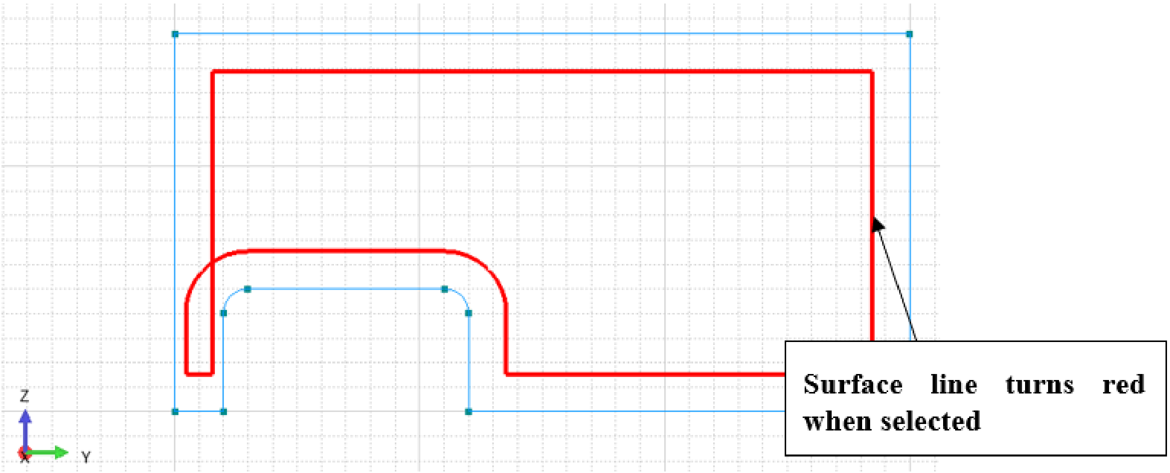


Fig. 3-10: Selected surface with red outline.

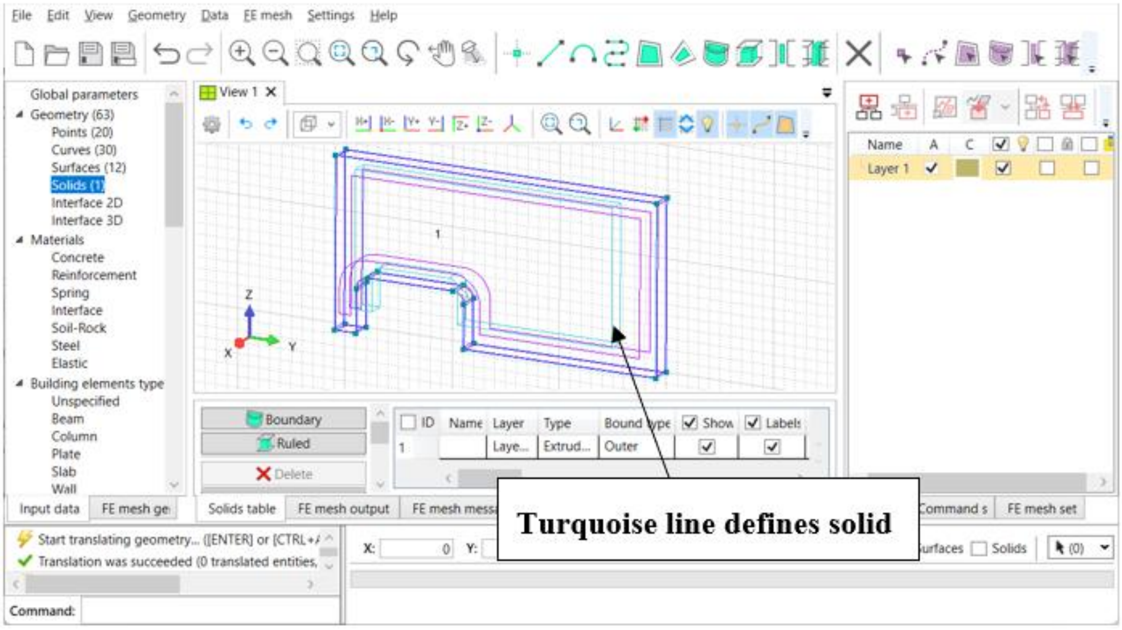


Fig. 3-11: Solid made with extrusion of surface.

Next part to create is the bottom beam. First point added to geometry using **Create point** is:

Parameters input:	
Label	Coordinates X,Y,Z
21	[-0.125, -0.35, 0]

This point is extruded to line in X direction with length 0.35 m, Fig. 3-12. Translating the point with checked Extrusion and start point in origin [0,0,0] and end point in [0.35,0,0], the line 31 is made. In **Surface table** the specific curve can be selected so that it is visible in Workspace view and the label directly for one or more curves can be switched on as well.

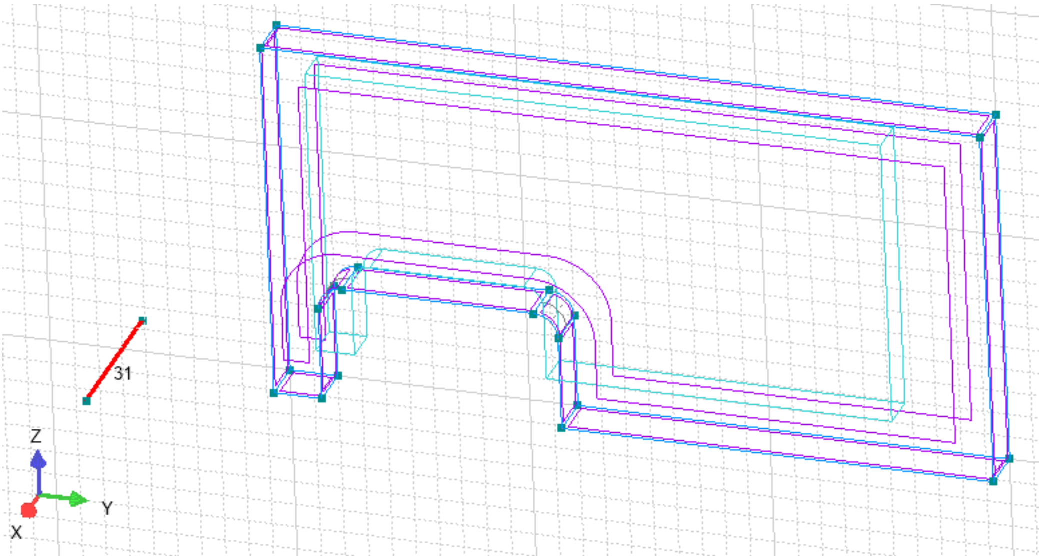


Fig. 3-12: Line extruded from point.

Line 31 is then extruded in direction Z with length 0.35 m to create the surface number 13 (Fig. 3-13) and the surface is extruded by 2.2 m in Y direction which produces solid number 2 (Fig. 3-14).

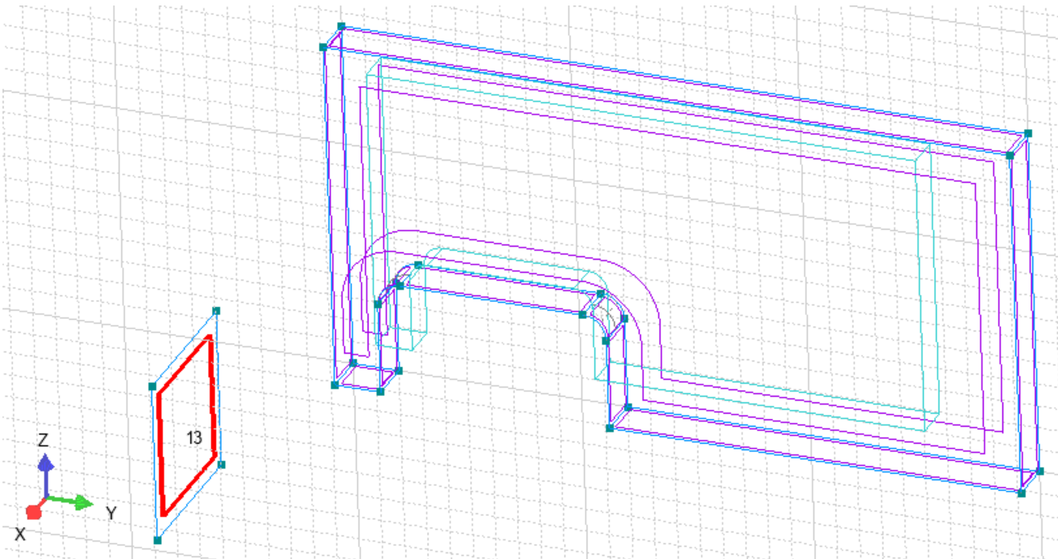


Fig. 3-13: Surface made from line.

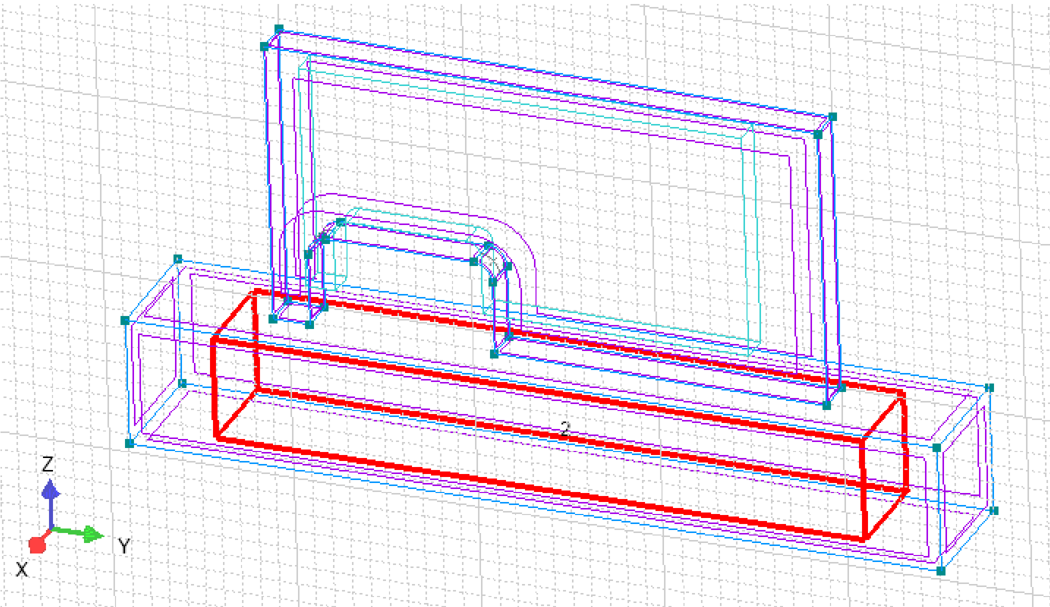


Fig. 3-14: Solid made out of surface extrusion.

The top beam is identical with bottom therefore the bottom one, volume number 3, is copied, see Fig. 3-15, using **Translate geometry** in direction Z with start point in origin [0,0,0] and end point [0,0,1.12].

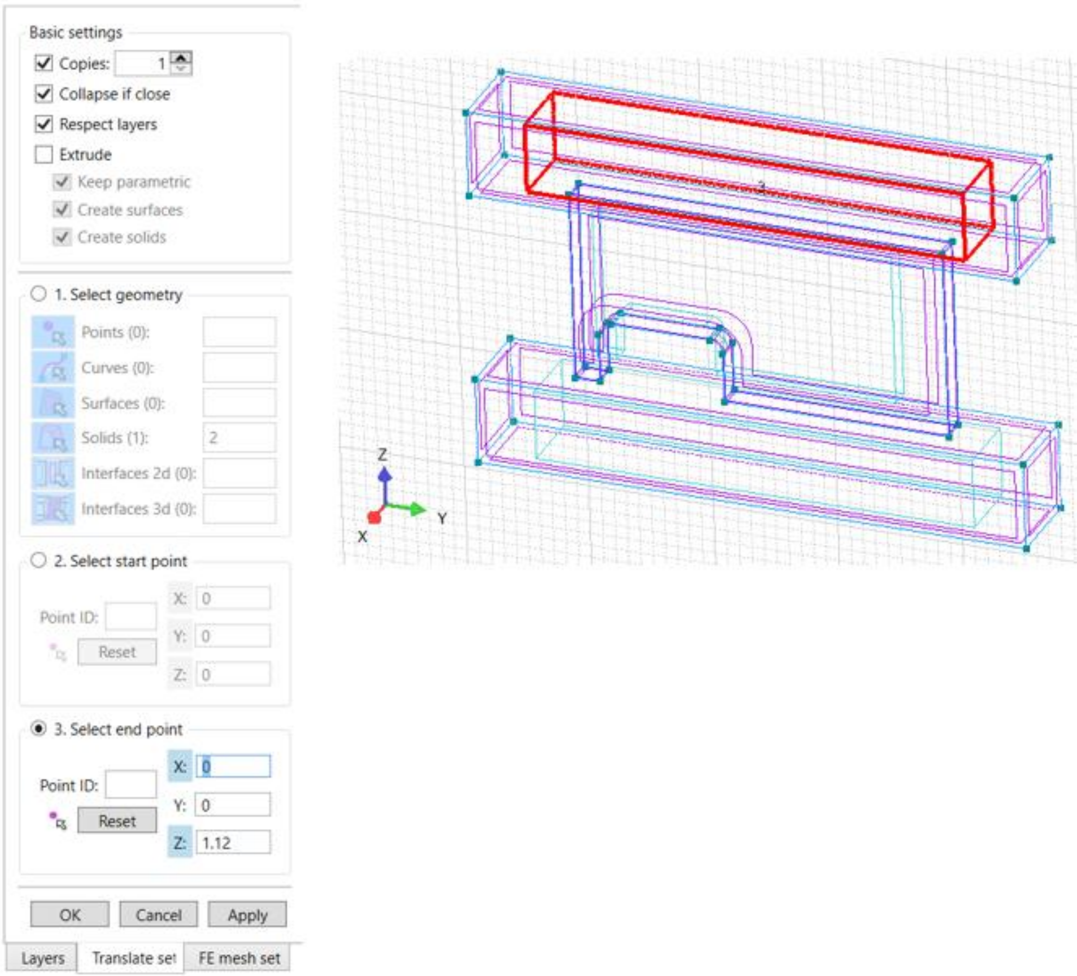


Fig. 3-15: Translate geometry – Copy of solid.

Finally, the concrete structure is moved to new layer which is helpful later on during the model development especially material assignment. New layer is created via button **Add new root layer**  with name ‘wall’ and it is activated right away as shown in Fig. 3-16.

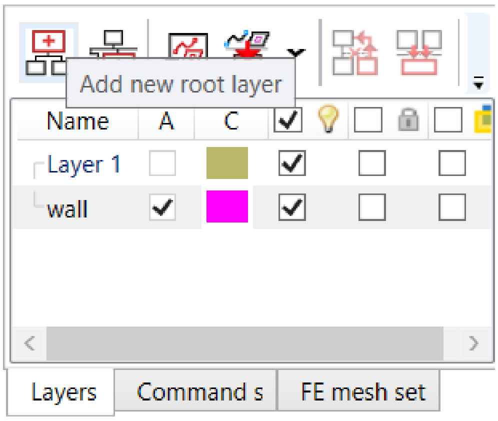


Fig. 3-16: Create new layer.

In order to move structure in certain layer, the structure must be selected using the tool **Select solid**  and then **Add selected geometry to selected layer**  in **Layers** window with selection of all lower entities as a part of the solid geometry, see Fig. 3-17.

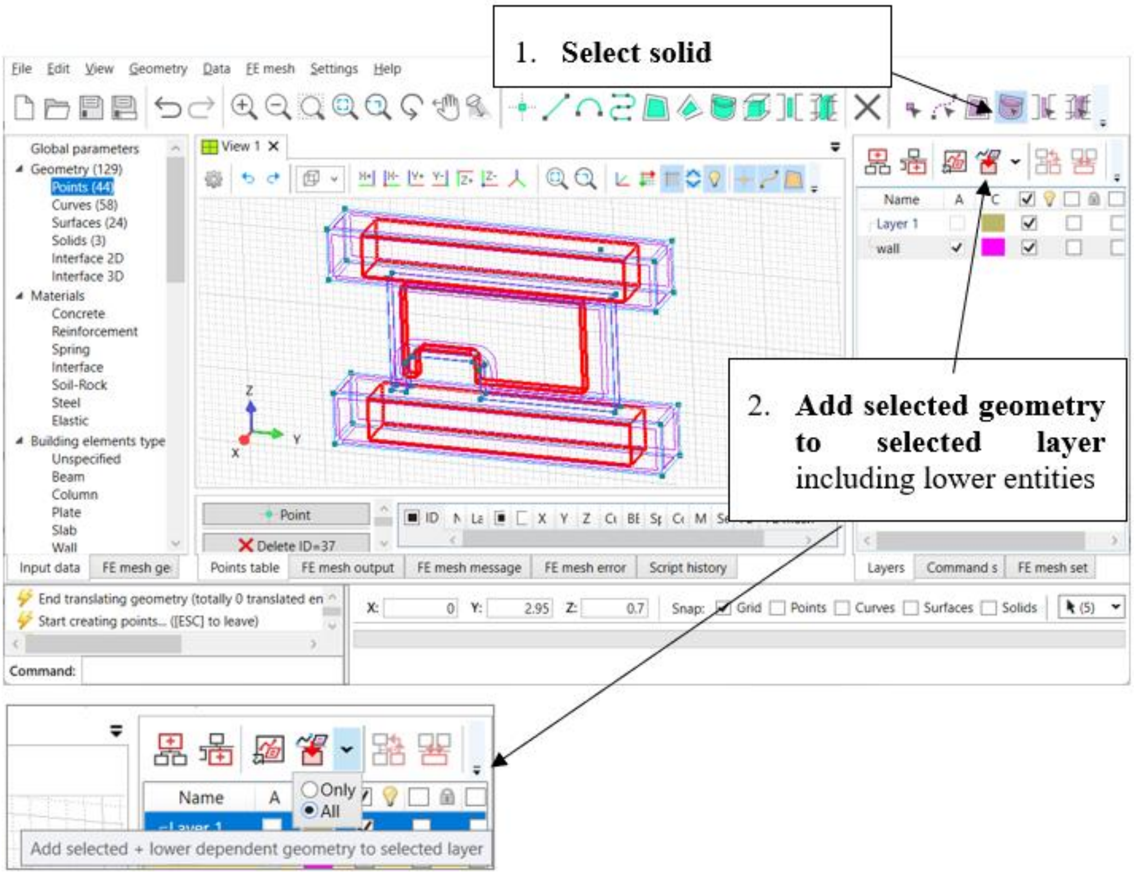


Fig. 3-17: Move entities into selected layer.

3.1.2 Discrete reinforcement

The geometry of concrete volumes is complete. Now the reinforcement needs to be added. First bars added to structure are the discrete bars on the edge of the middle panel with diameter 12 mm. Adding points (Fig. 3-18):

Parameters input:	
Label	Coordinates X,Y,Z
37	[0.05, 0.03, -0.32]
38	[0.05, 0.07, -0.32]
39	[0.05, 1.43, -0.32]
40	[0.05, 1.47, -0.32]

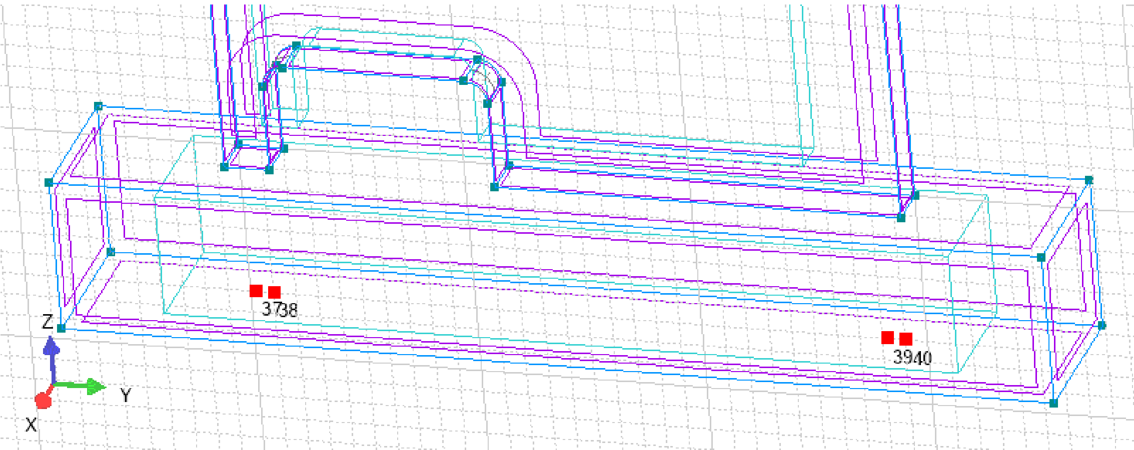


Fig. 3-18: Start points of discrete reinforcement.

The points are extruded to four new lines with length 1.41 m, see Fig. 3-19.

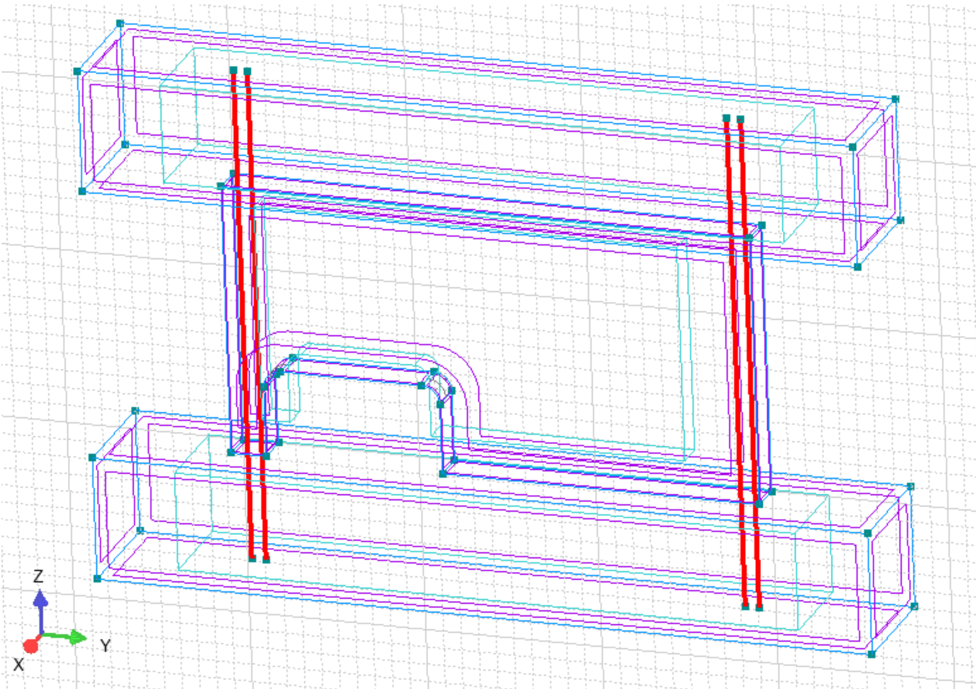


Fig. 3-19: New discrete reinforcement.

Two more vertical bars are added along the opening with starting points:

Parameters input:	
Label	Coordinates X,Y,Z
45	[0.05, 0.06, -0.32]
46	[0.05, 0.64, -0.32]

And then extruded to lines 1 m long, see Fig. 3-20.

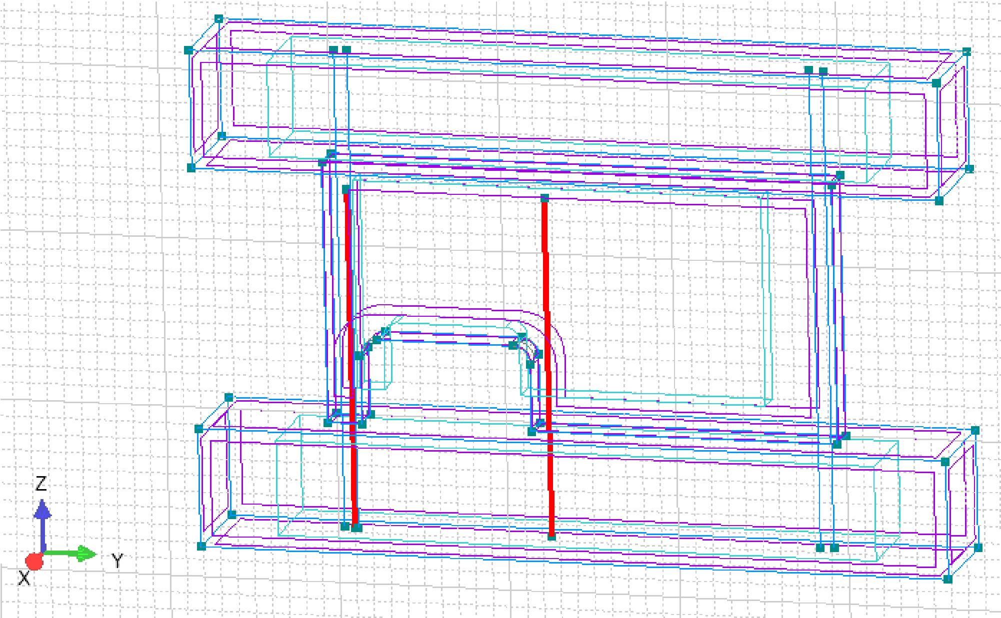


Fig. 3-20: New reinforcement on the side of opening.

Last discrete bar added to model is the one above the opening. Four points that define this reinforcement are:

Parameters input:	
Label	Coordinates X,Y,Z
49	[0.05, 0.95, 0.3]
50	[0.05, 0.09, 0.3]
51	[0.05, 0.04, 0.35]
52	[0.05, 0.04, 0.65]

The connection between point 50 and 51 is arc with radius 0.05 m, in Fig. 3-21.

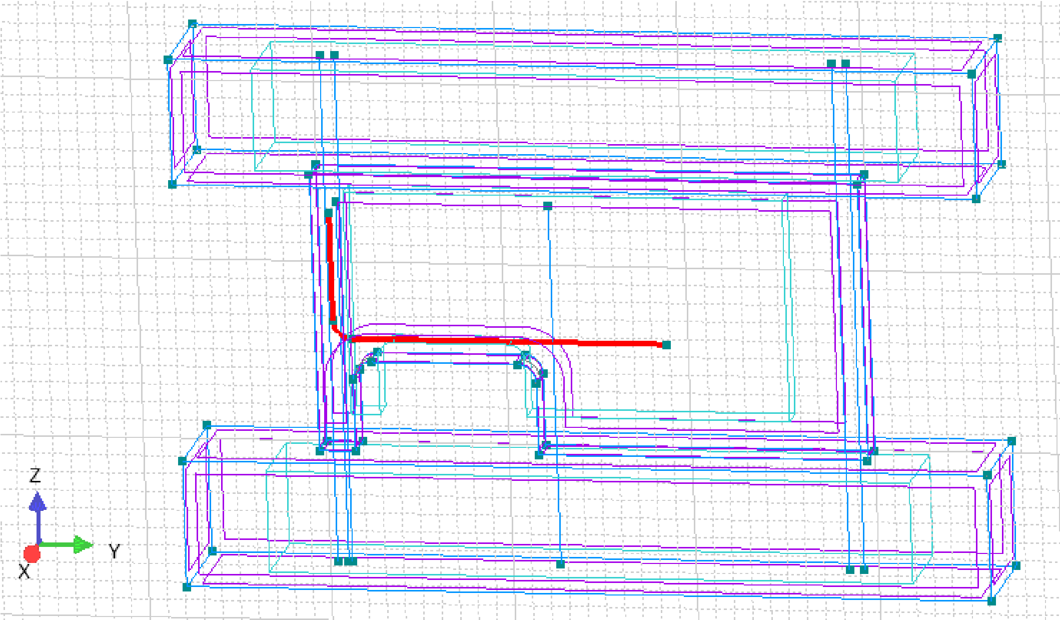


Fig. 3-21: Reinforcing bar above opening.

All discrete bars are moved together to layer called ‘bars’, Fig. 3-22.

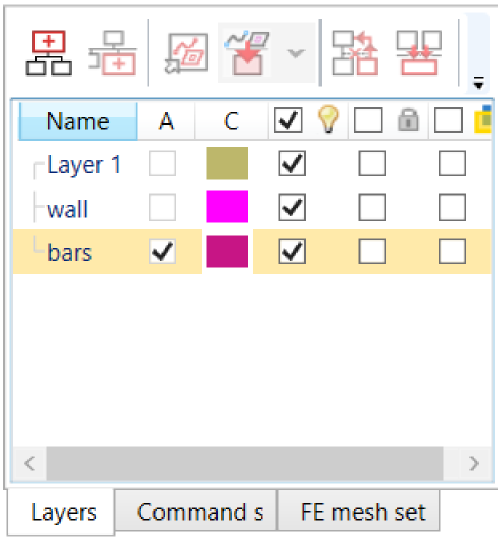


Fig. 3-22: New layer ‘bars’.

Final part of the geometry is the reinforcement mesh. The horizontal mesh bars are created at first. First point is number 53 and its coordinates are [0.05, 0.03. -0.025], the point is extruded in Y direction by 1.41 m (Fig. 3-23). Then the bar is copied 3 times bellow opening in Z direction with distance -0.1 m (Fig. 3-24).

Parameters input:	
Label	Coordinates X,Y,Z
53	[0.05, 0.03, -0.025]
Extrude	Point: 53
	Start point: [0, 0, 0]
	End point: [0, 1.41, 0]

Copy	Number of copies: 3 Curve: 64 Start point: [0, 0, 0] End point: [0, 0, -0.1]

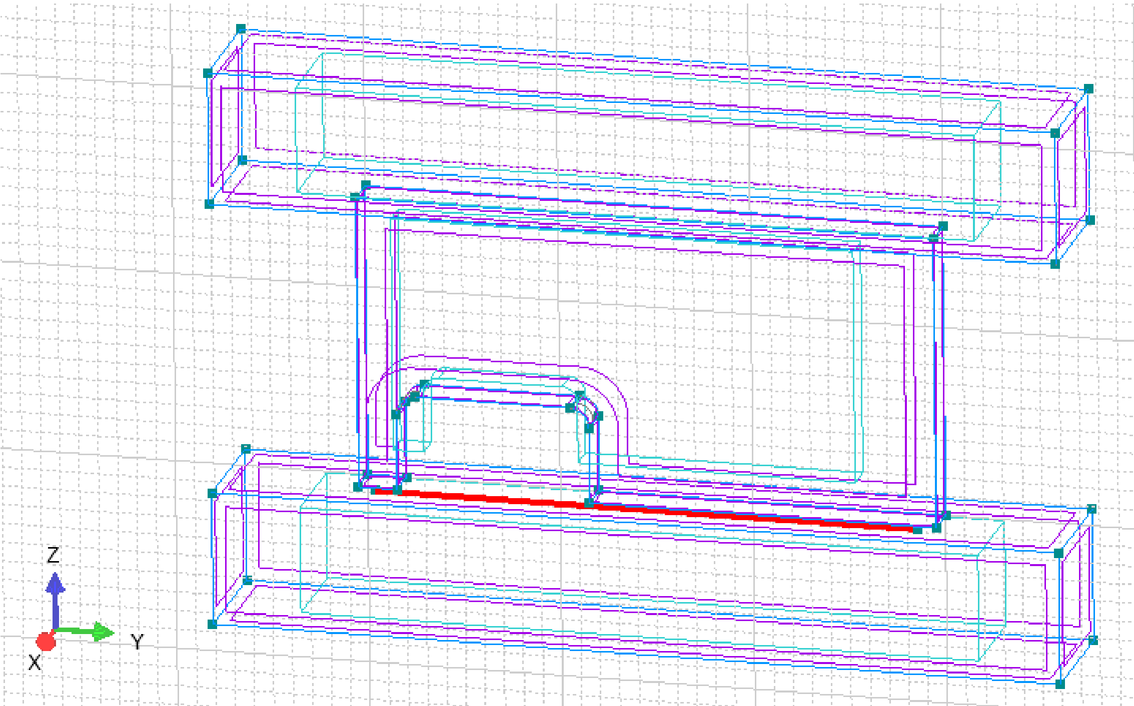


Fig. 3-23: Reinforcement mesh horizontal bar below opening.

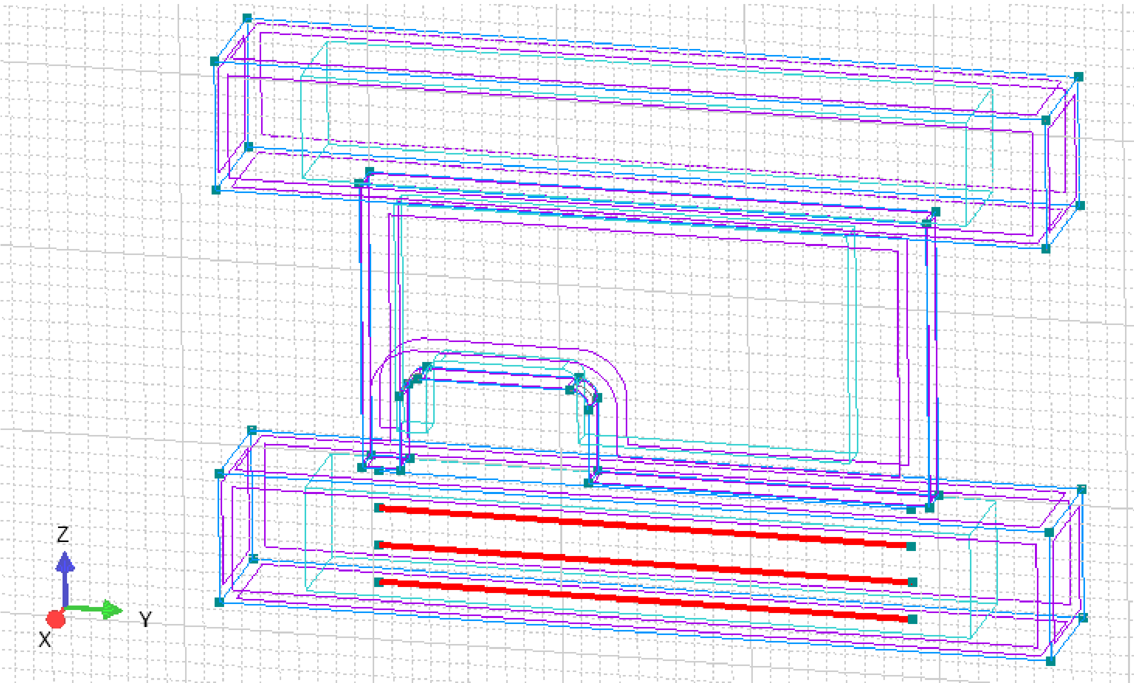


Fig. 3-24: Copies of initial horizontal bar.

One bar is copied above opening with distance 0.325 in Z direction (Fig. 3-25) and then copied 7 more time in direction Z with distance 0.1 m (Fig. 3-26).

Parameters input:	
Copy	Number of copies: 1
	Curve: 64
	Start point: [0, 0, 0]
	End point: [0, 0, 0.325]
Copy	Number of copies: 7
	Curve: 68
	Start point: [0, 0, 0]
	End point: [0, 0, 0.1]

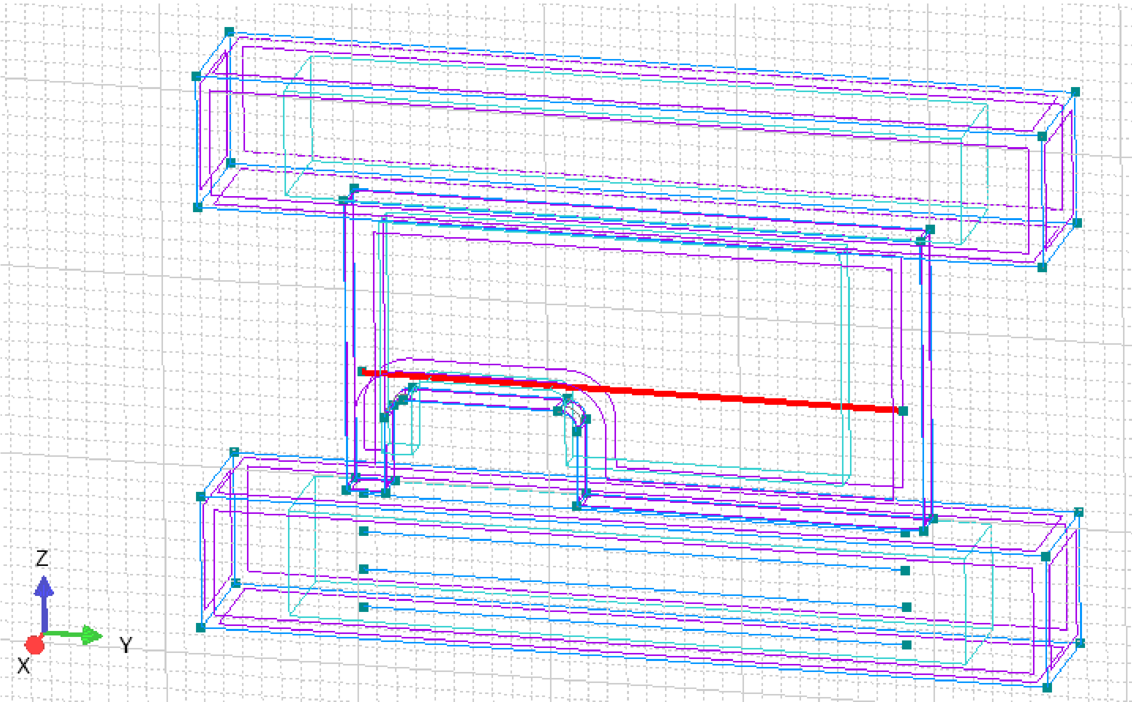


Fig. 3-25: Reinforcement mesh horizontal bar above opening.

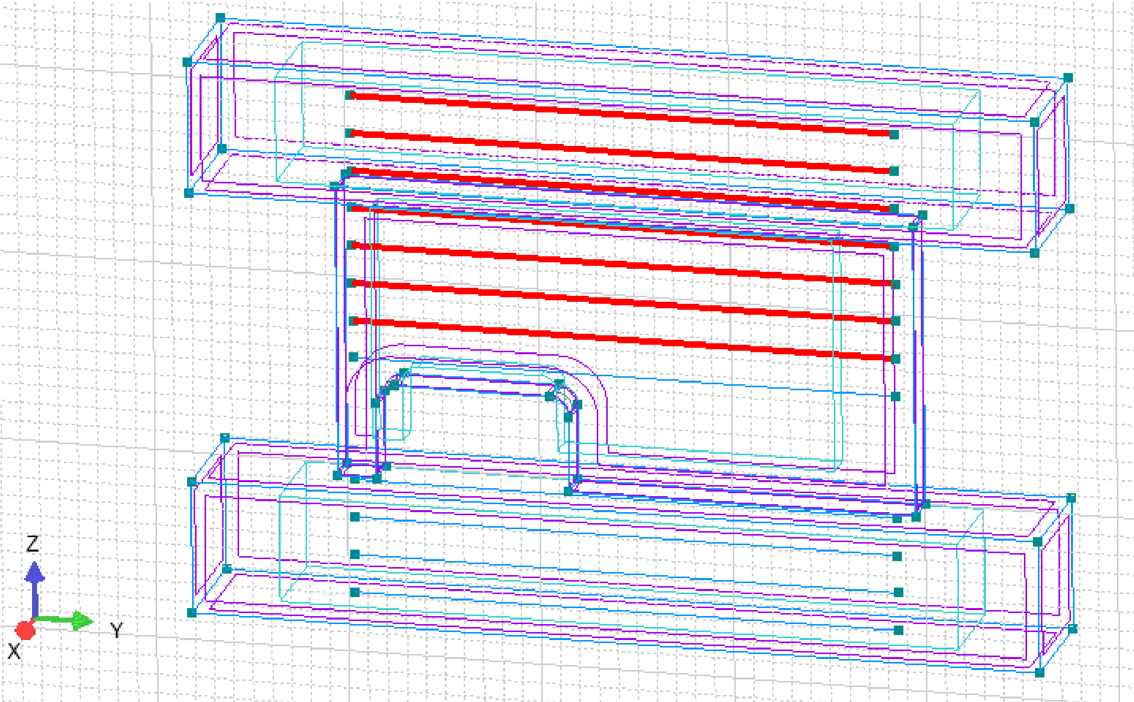


Fig. 3-26: Copies of horizontal bar above opening.

Point 62 is copied 2 times in direction Z by -0.1 m and then extruded by -0.82 m in direction Y (Fig. 3-27).

Parameters input:	
Label	Coordinates X,Y,Z
62	[0.05, 1.44, 0.3]
Copy	Number of copies: 2
	Point: 64
	Start point: [0, 0, 0]
	End point: [0, 0, -0.1]
Extrude	Point: 77,78
	Start point: [0, 0, 0]
	End point: [0, -0.82, 0]

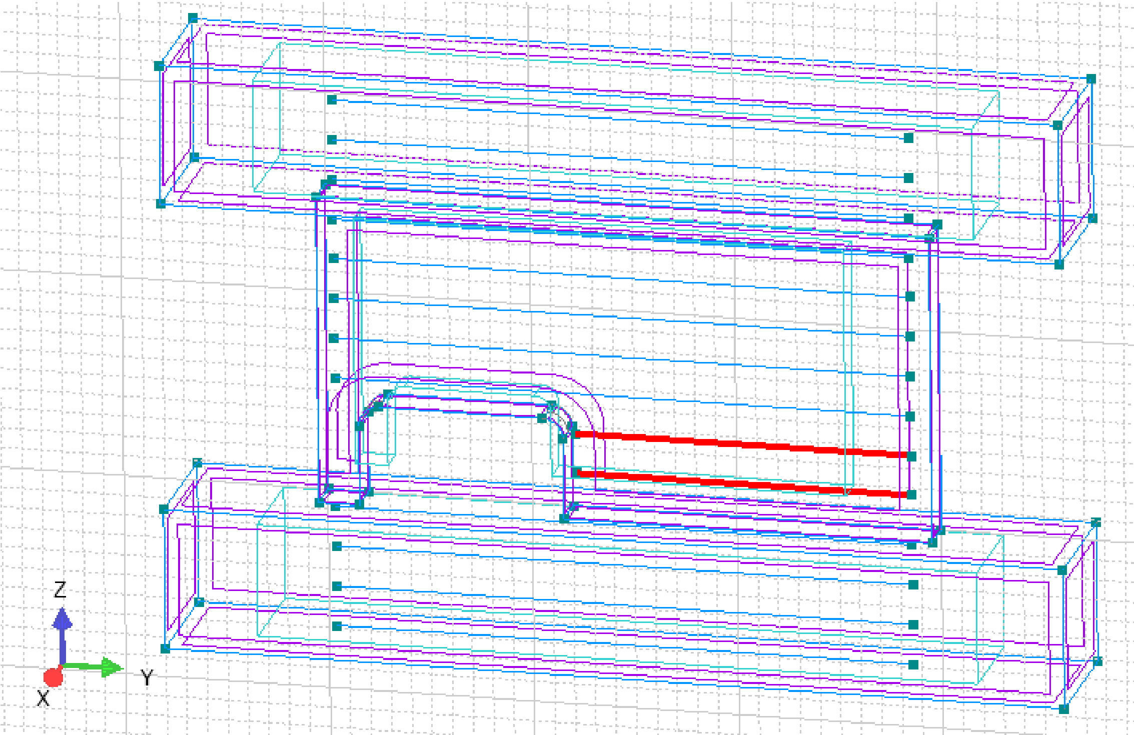


Fig. 3-27: Horizontal bars on the side of opening.

Same procedure is applied on the vertical bars of reinforcing mesh. Starting with point number 81 and extruding the point into line 1.4 m long in Z direction which is copied 8 times in direction Y with distance -0.1 m, see Fig. 3-28.

Parameters input:	
Label	Coordinates X,Y,Z
81	[0.05, 1.42, -0.33]
Extrude	Point: 81
	Start point: [0, 0, 0]
	End point: [0, 0, 1.4]
Copy	Number of copies: 8
	Curve: 78
	Start point: [0, 0, 0]
	End point: [0, -0.1, 0]

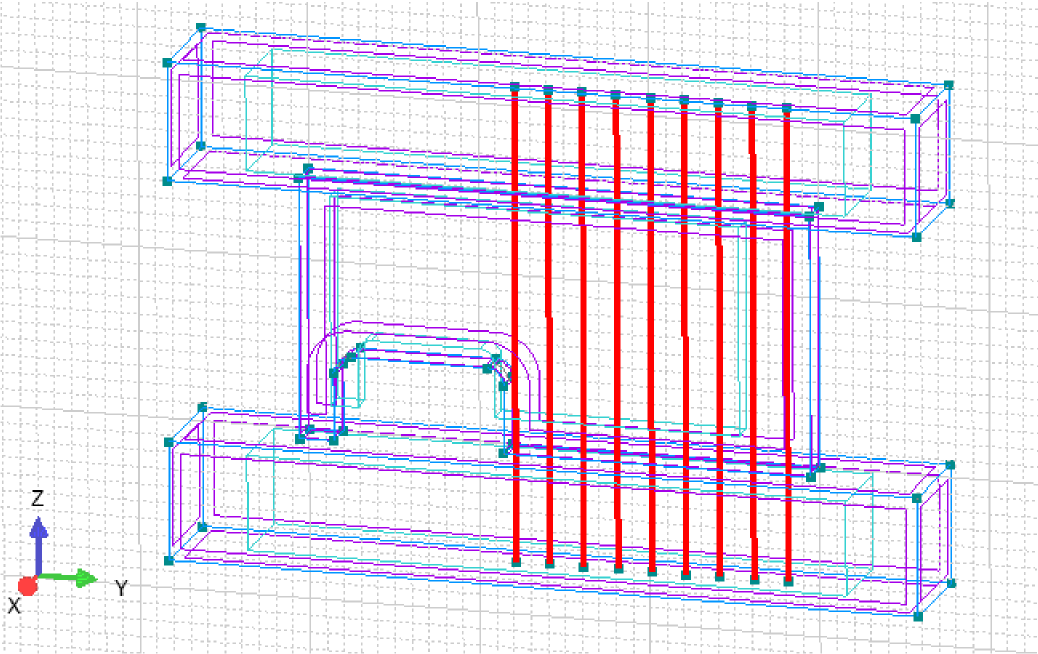


Fig. 3-28: Reinforcement mesh vertical bars.

More points are copied in the same direction to make shorter bars below and above opening. Two points 97 and 98 are copied by -0.1 m in direction Y and extruded by 0.31 m in Z direction bellow opening and -0.79 m in Z direction above opening, Fig. 3-29.

Parameters input:	
Copy	Number of copies: 1
	Point: 97, 98
	Start point: [0, 0, 0]
	End point: [0, -0.1, 0]
Extrude	Point: 99
	Start point: [0, 0, 0]
	End point: [0, 0, 0.31]
Extrude	Point: 100
	Start point: [0, 0, 0]
	End point: [0, 0, -0.79]

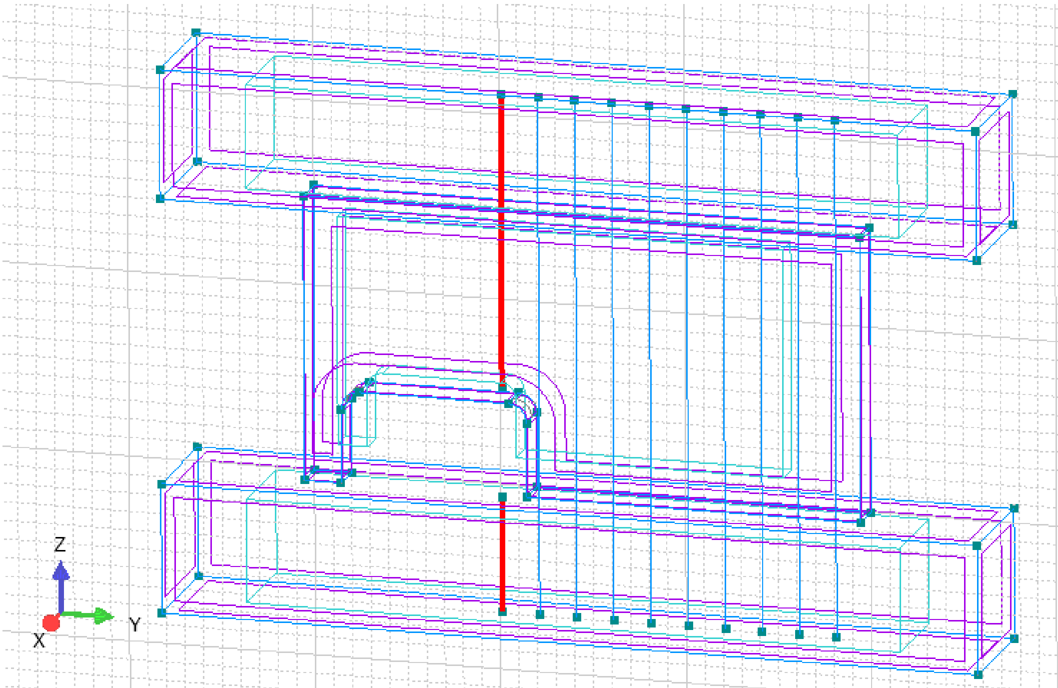


Fig. 3-29: Vertical bars below and above opening.

By copying the two bars (87 and 88) 4 times in Y direction by -0.1 m the vertical bars of mesh are nearly finished, Fig. 3-30.

Parameters input:	
Copy	Number of copies: 4
	Curve: 87, 88
	Start point: [0, 0, 0]
	End point: [0, -0.1, 0]

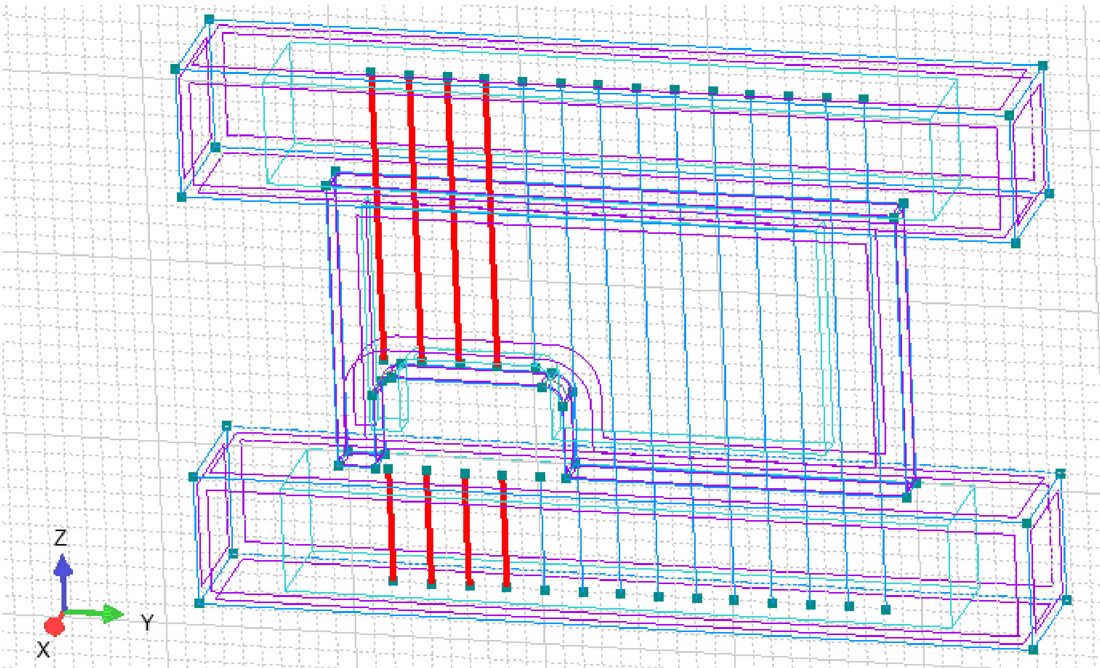


Fig. 3-30: Copies of vertical bars below and above opening.

The last vertical bar is placed in the middle of the narrow part of frame next to opening (Fig. 3-31) and the point that define this bar have coordinates:

Parameters input:	
Label	Coordinates X,Y,Z
103	[0.05, 0.05, -0.33]
104	[0.05, 0.05, 1.07]

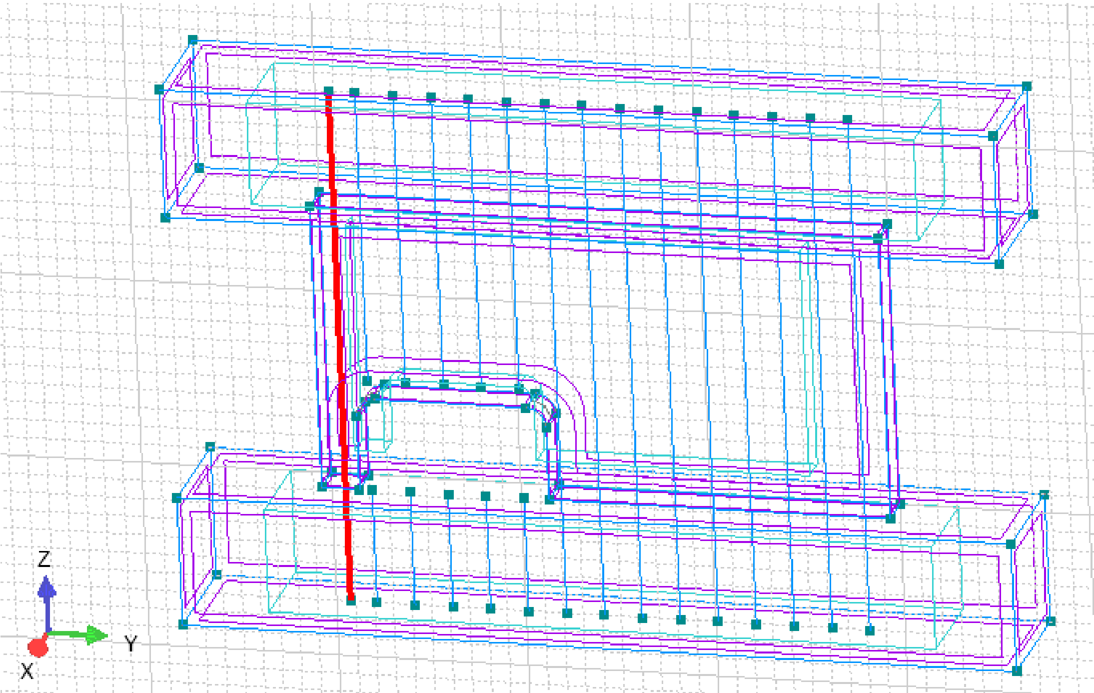


Fig. 3-31: Vertical bar in the narrow part of frame around opening.

All the vertical and horizontal bars of the mesh are selected and moved to new layer called 'mesh', Fig. 3-32.

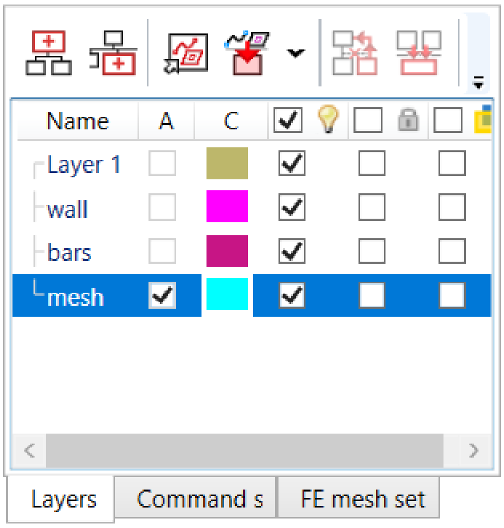


Fig. 3-32: New layer 'mesh'.

3.2 Materials

The wall that consists of middle panel and top and bottom beam is made of concrete which is reinforced with discrete bars of various diameters. This part covers the material generation and properties adjustment for concrete wall and reinforcing bars.

3.2.1 Concrete material

For the concrete part the material CC3DNonLinCementitious2 is used. In **Input data tree** the **Materials | Concrete** is selected which activates **Concrete materials table**, Fig. 3-33. Pressing button **Concrete** 

the **Generate material** menu appears or in the Main menu can be selected the path **Data | Materials | Concrete**, Fig. 3-34.

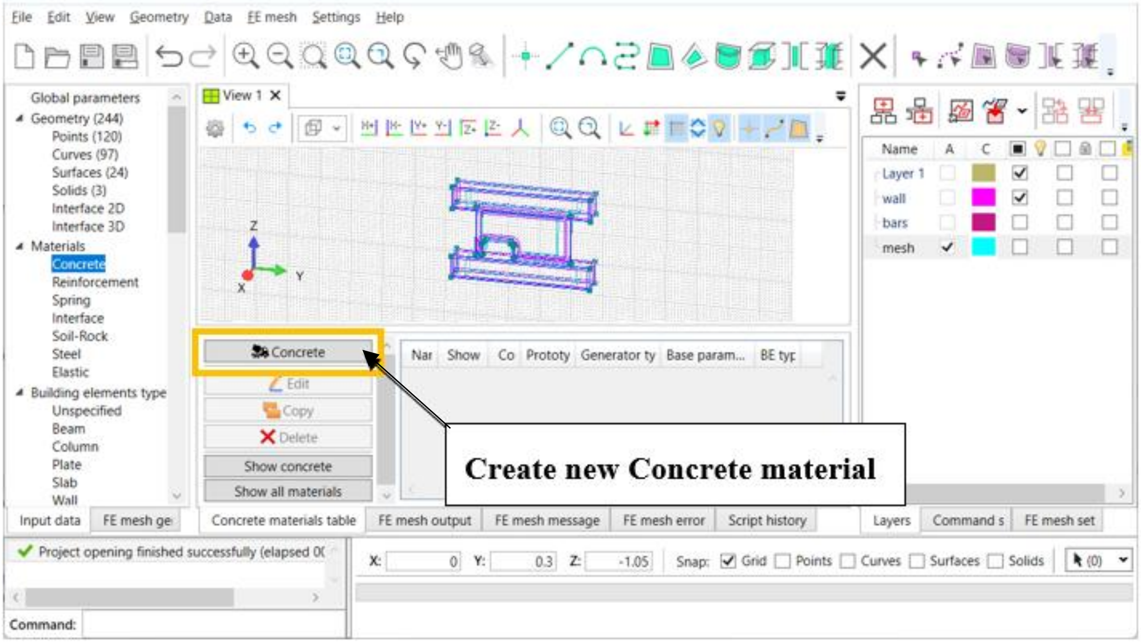


Fig. 3-33: Create new Concrete material.

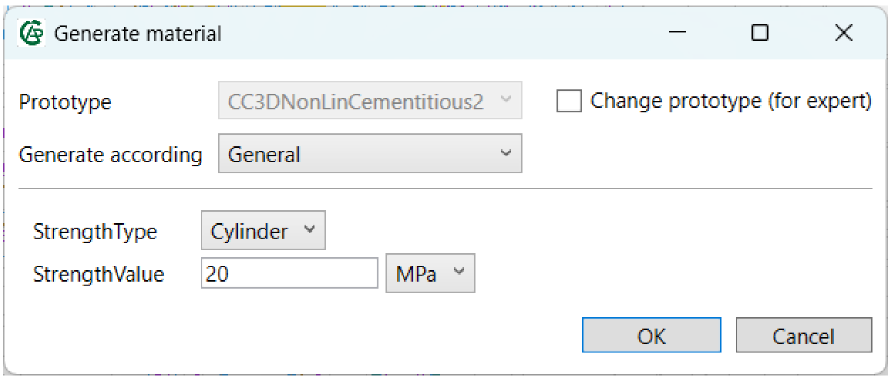


Fig. 3-34: Generate material – Concrete default.

Standard used for material parameters generation is ModelCode. Then the Strength type is Cylinder-Mean with strength 30 MPa is selected along with Safety format – Mean, Fig. 3-35. **Ok** button generates the material parameters according to selected data.

Parameters input:

Generate according	ModelCode
StrengthType	Cylinder-Mean
StrengthValue	30 MPa
Generate for	Mean

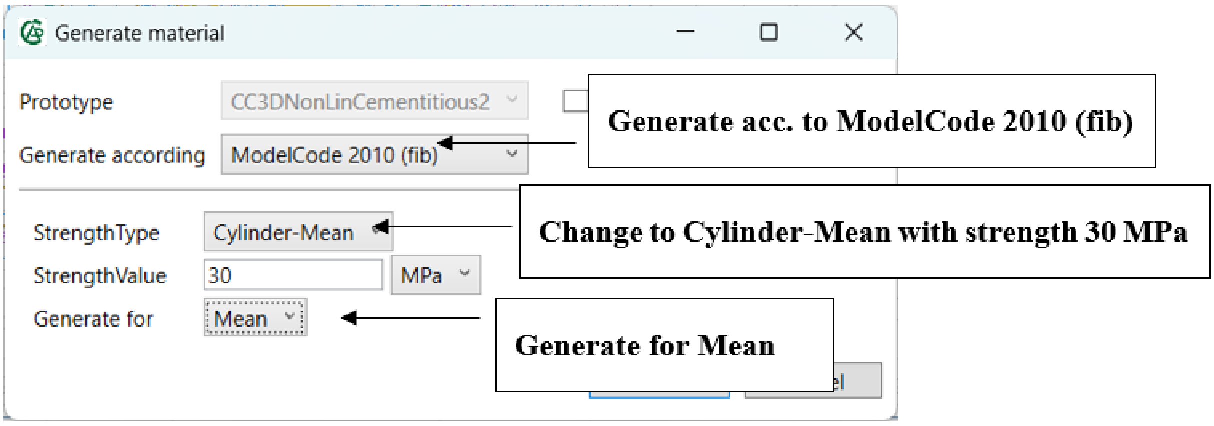


Fig. 3-35: Generate material – Concrete adjusted.

The generated material parameters are shown in Fig. 3-36. Those are adjusted according to data about concrete material from experiment which are shown in Tab. 3-1.

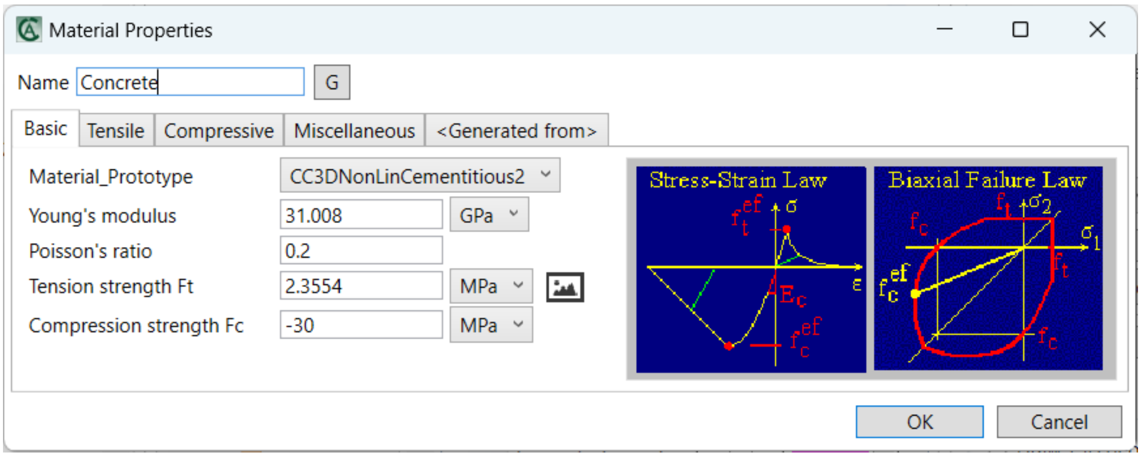


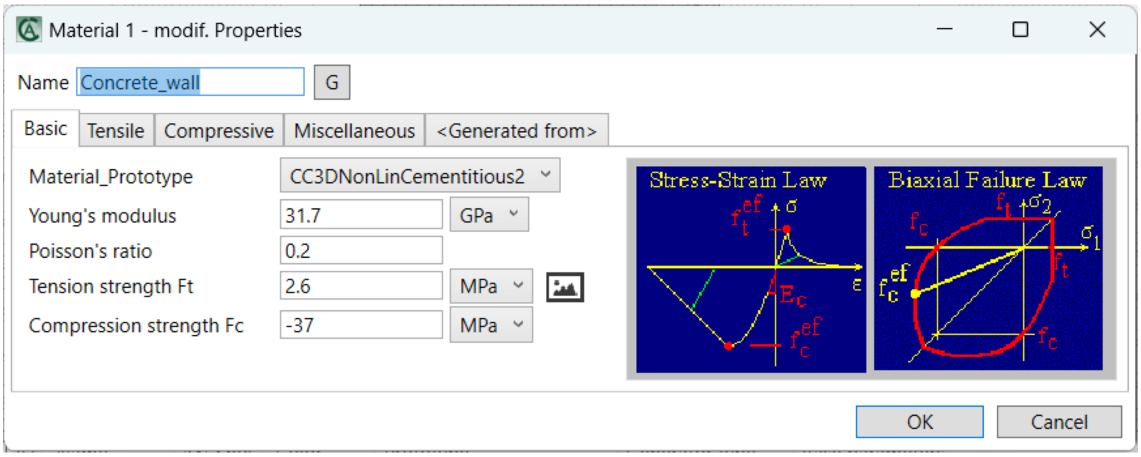
Fig. 3-36: Material properties - generated.

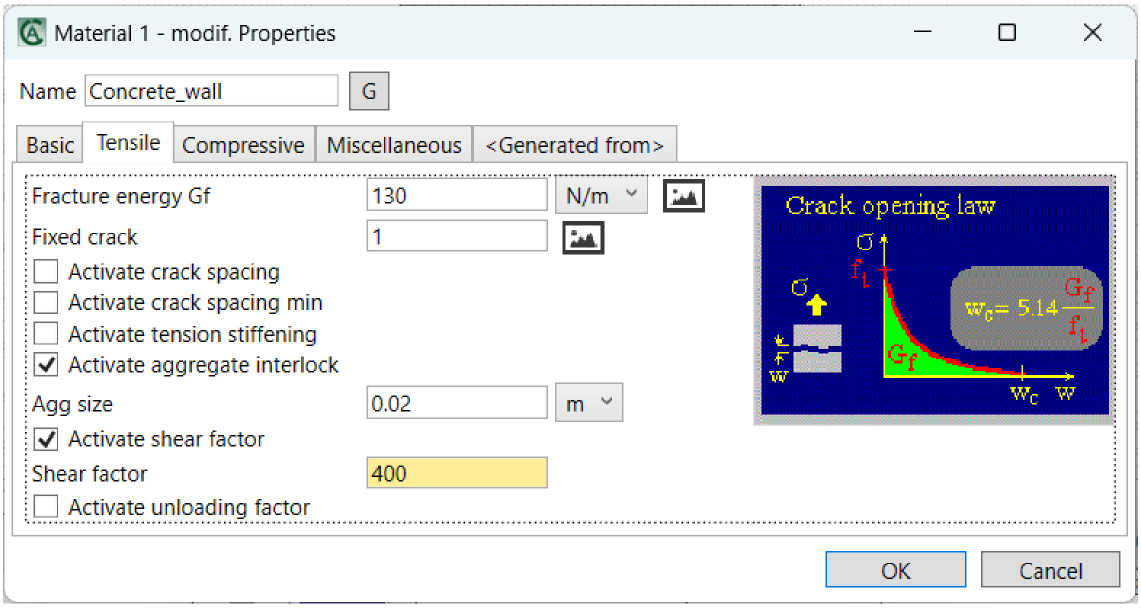
Tab. 3-1: Concrete material properties.

Material parameters for concrete structure			
Elastic modulus	E_c	31.7	GPa
Poisson’s ratio	ν	0.2	-
Compressive strength	f_c	37	MPa
Tensile strength	f_t	2.6	MPa
Fracture energy	G_f	130.0	N/m
Crack model		Fixed	

The material parameters specified for the model in table are adjusted in Material – modif. Properties (Fig. 3-37) and the name of material is changed to **Concrete_wall**.

Parameters input:	
Name	Concrete_wall
Basic	Young's modulus: 31.7 GPa
	Tension strength Ft: 2.6 MPa
	Compression strength Fc: -37 MPa
Tensile	Fracture energy Gf: 130 N/m
	Shear factor: 400
Compressive	Plastic strain EPS CP: -0.0017
	Onset of crushing Fc0: -20 MPa
	Critical comp disp Wd: -0.0005 m
	Activate crush band min.: 0.1 m





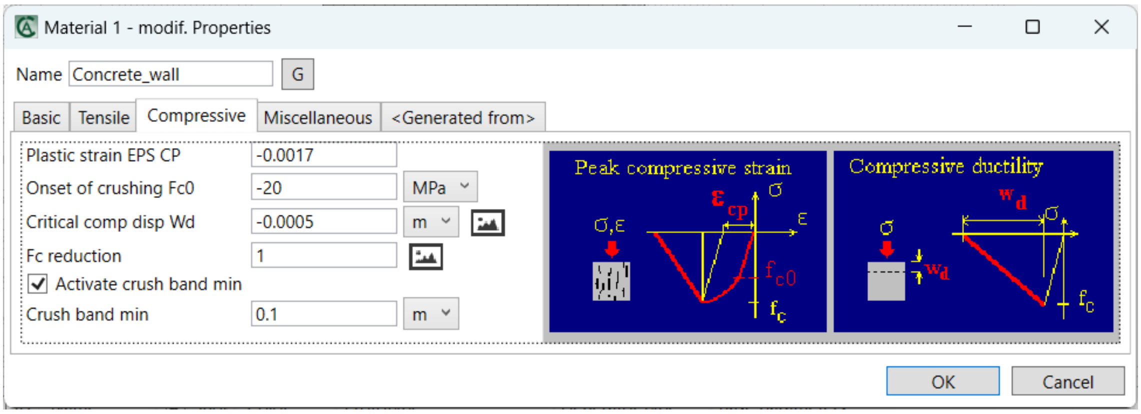


Fig. 3-37: Material properties adjusted.

When the name and properties are change by pressing Ok the new material appears in **Concrete material table**, see Fig. 3-38.

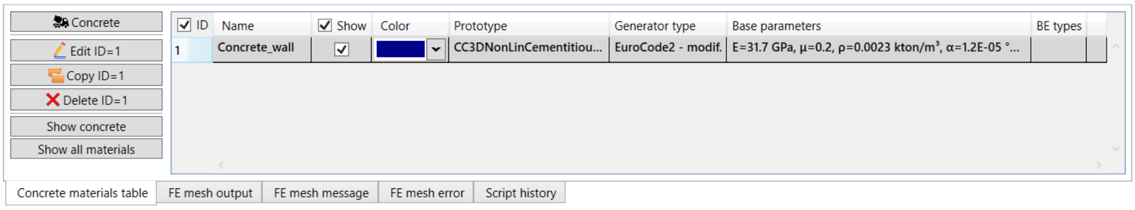
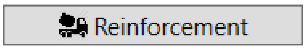


Fig. 3-38: Concrete material table with material Concrete_wall.

3.2.2 *Reinforcement material*

Then materials for reinforcement are created. There are 4 different types of bars regarding its diameter and material properties therefore 4 different material models for reinforcement are made. To create new reinforcement, **Materials | Reinforcement** in **Input tree** is selected and **Reinforcement material table** appears, Fig. 3-39, or it can be open via **Data | Materials | Reinforcement** in **Main menu**.

Then pressing button Reinforcement 

opens dialog Generate material for reinforcement.

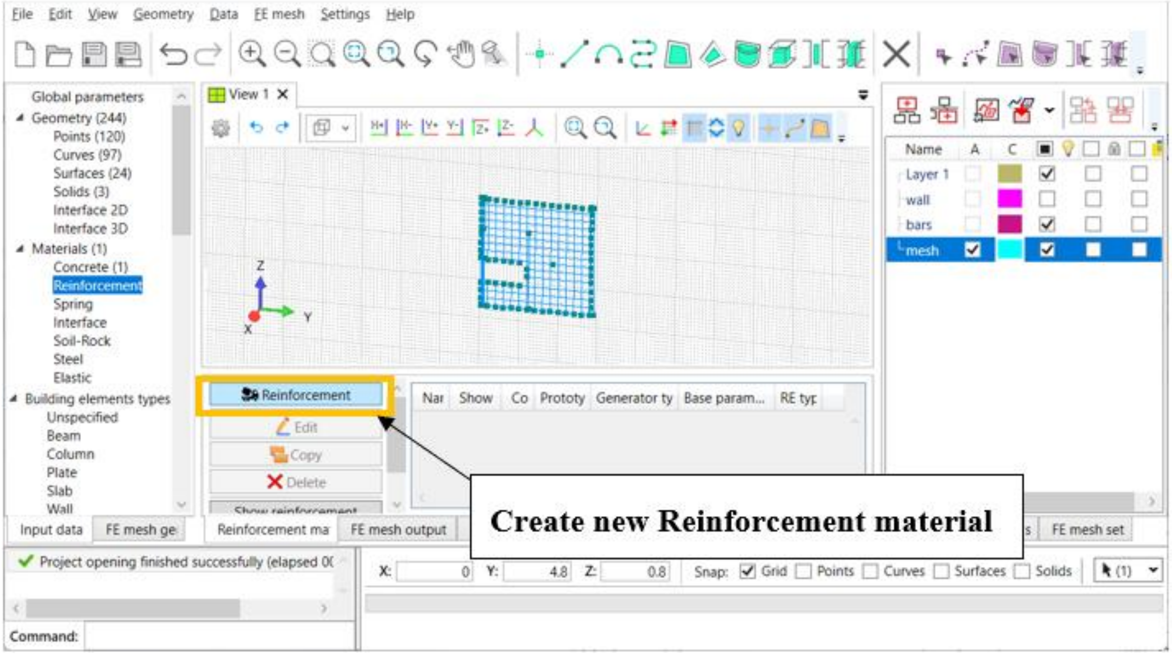


Fig. 3-39: Create new Reinforcement material.

In the Generate material dialog (Fig. 3-40) the Characteristic yield strength is changed to 570 MPa and Characteristic Safety format is selected (Fig. 3-41). The material parameter are taken from Tab. 3-2.

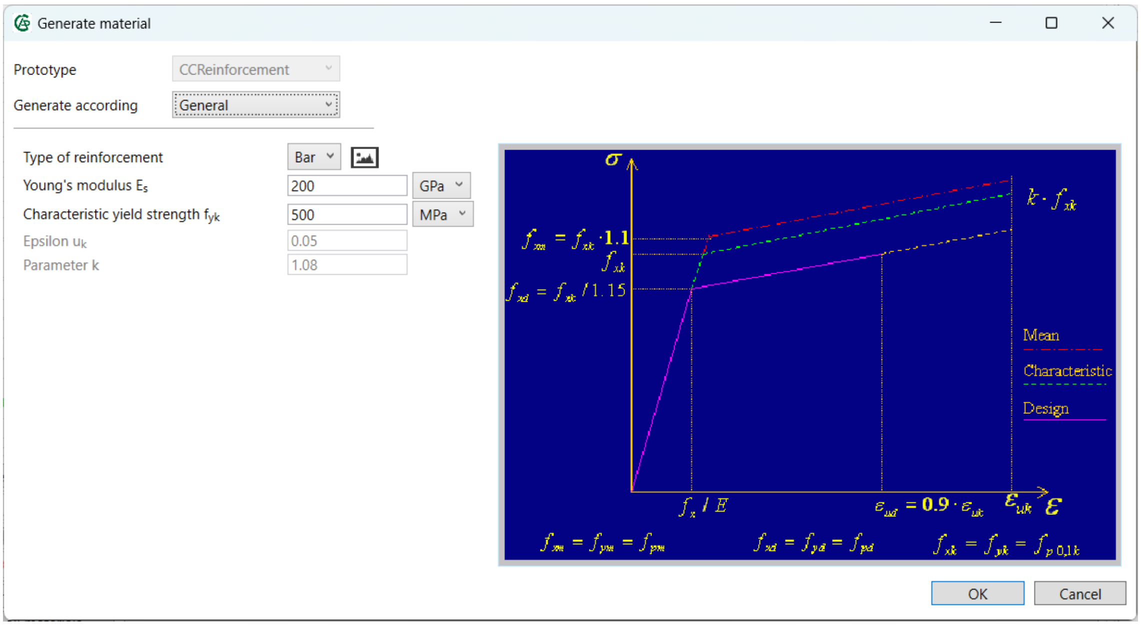


Fig. 3-40: Generate material – Reinforcement default.

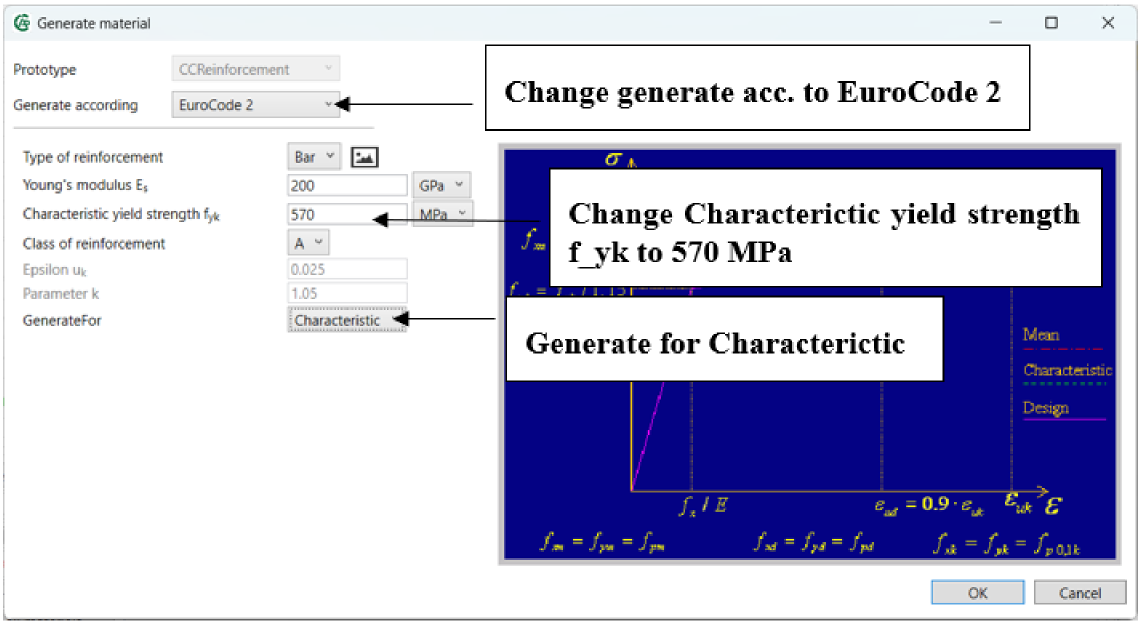


Fig. 3-41: Generate material – Reinforcement adjusted.

Tab. 3-2: Reinforcement material D_6.

Material parameters of reinforcement D_6			
Elastic modulus	E	200	GPa
Yield strength	σ_y	570	MPa
Hardening		linear	

Parameters input:

Name D_6

Basic Young's modulus: 200 GPa

Characteristic yield strength f_yk: 570 MPa

When the material is generated, in Fig. 3-42, by clicking on Edit of Reinf_function the stress-strain diagram of reinforcement is shown in Edit function dialogue, in Fig. 3-43. The material is created when you close Edit dialogue and press Ok. Then the material is listed in **Reinforcement materials table**, Fig. 3-44.

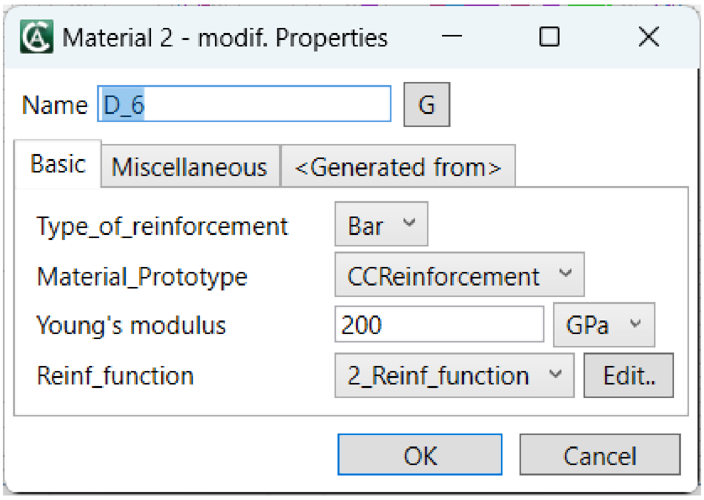


Fig. 3-42: Modification of Reinforcement material properties.

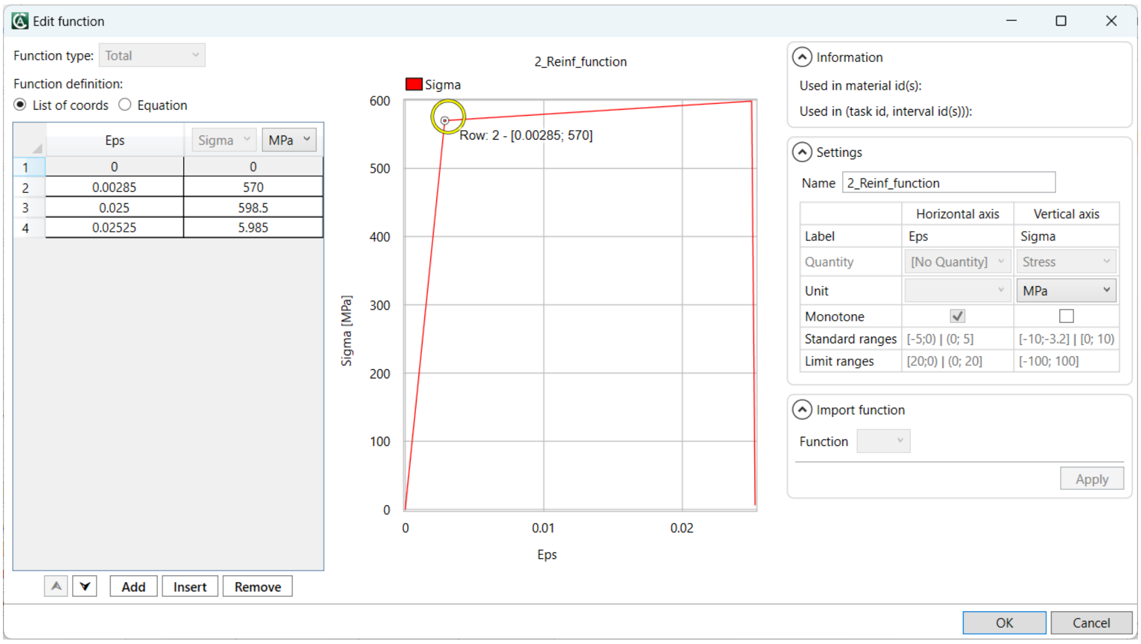


Fig. 3-43: Reinforcement function edit dialogue.

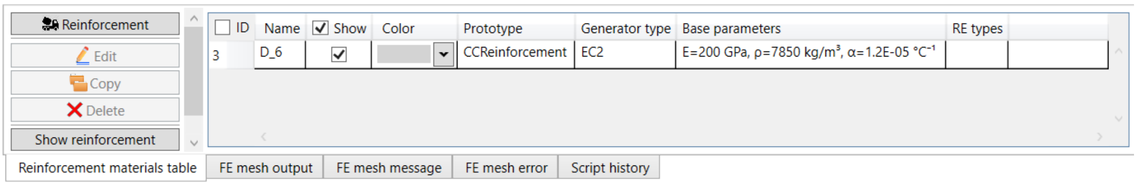


Fig. 3-44: Reinforcement materials table with new material D_6.

The procedure of creating new reinforcement is repeated 3 more times with respective properties specified in following tables, Tab. 3-3 - Tab. 3-5.

Tab. 3-3: Reinforcement material D12.

Material parameters of reinforcement D12			
Elastic modulus	E	200	GPa
Yield strength	σ_y	480	MPa
Hardening		linear	

Tab. 3-4: Reinforcement material D10.

Material parameters of reinforcement D10			
Elastic modulus	E	200	GPa
Yield strength	σ_y	470	MPa
Hardening		linear	

Tab. 3-5: Reinforcement material D_8.

Material parameters of reinforcement D_8			
Elastic modulus	E	200	GPa
Yield strength	σ_y	620	MPa
Hardening		linear	

Question about non-standard value of Characteristic yield strength pops up when material D_8 is generated which is shown in Fig. 3-45. By pressing Ok it is possible to finish material definition.

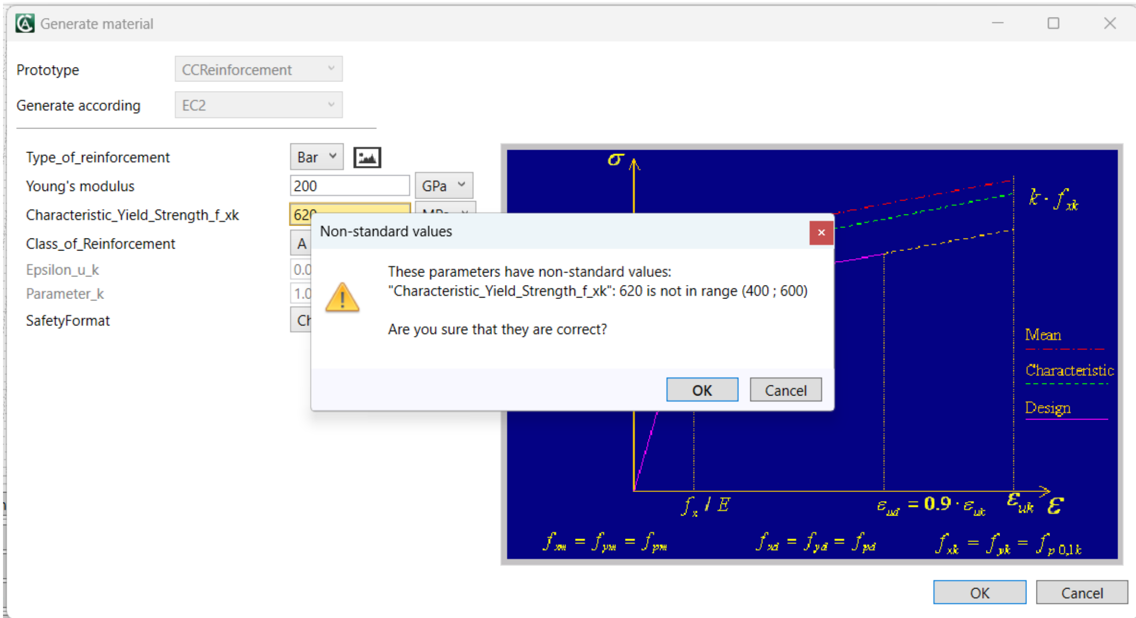


Fig. 3-45: Non-standard value check.

All reinforcement materials which are created are listed in Reinforcement materials table, Fig. 3-46.

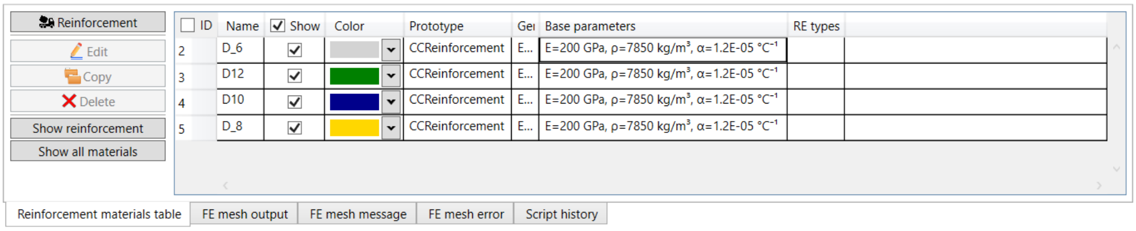


Fig. 3-46: Reinforcement materials table with list of defined materials.

3.2.3 *Beams' material*

Since the beams are part that is going to be loaded and supported, the material assigned to these solids is going to be improved. Otherwise the local damage due to the load type would occur in numerical simulation.

Concrete base material for beams is based on Model code model for concrete with compressive strength on cylinder 80 MPa, Fig. 3-47.

Parameters input:	
Generate acc.	ModelCode
StrengthType	Cylinder-Mean
StrengthValue	80 MPa
Generate for	Mean

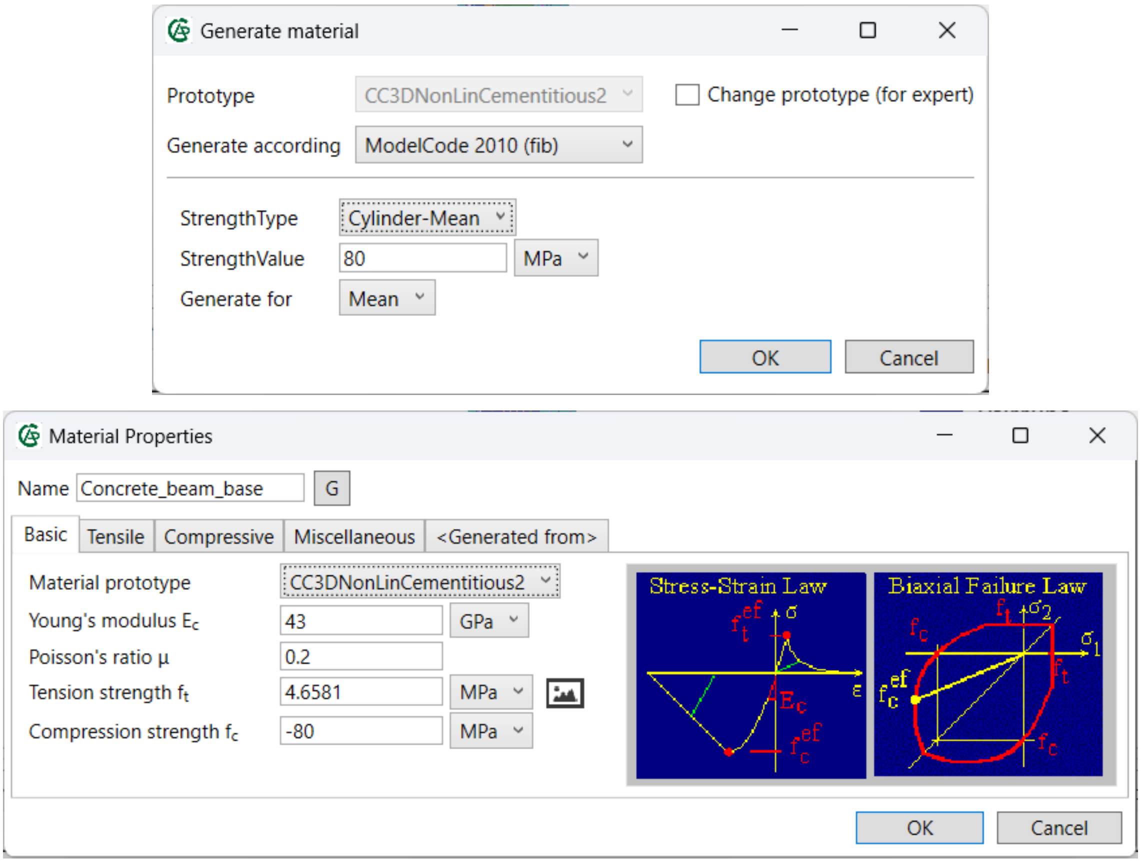


Fig. 3-47: Concrete beam base material generated from ModelCode.

Concrete_beam_base material has been generated and is used in Combined material which is concrete with smeared reinforcement.

In Concrete generate material the check of Change prototype is checked so the Prototype can be changed, Fig. 3-48. The material CCCombinedMaterial is chosen and the Ok is pressed.

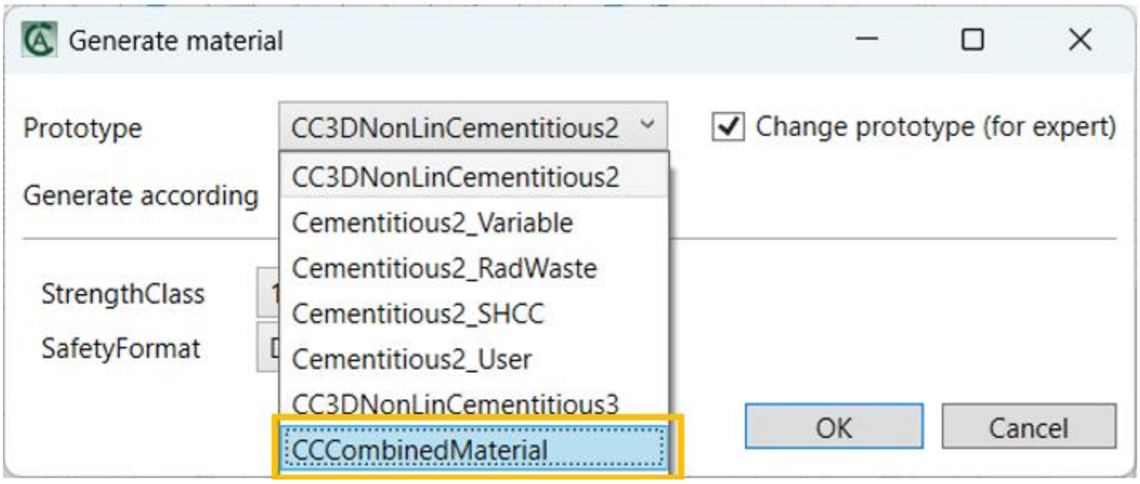


Fig. 3-48: Concrete beam material with smeared reinforcement.

Then the dialogue for modification of material properties is opened. Solid material is added – Concrete_beam_base with ratio 1 and has to be activated via checkbox. Then the Reinforcement is added in two perpendicular direction which are specified by direction vector. Ratio of reinforcement is changed and the reinforcements are activated, see Fig. 3-49.

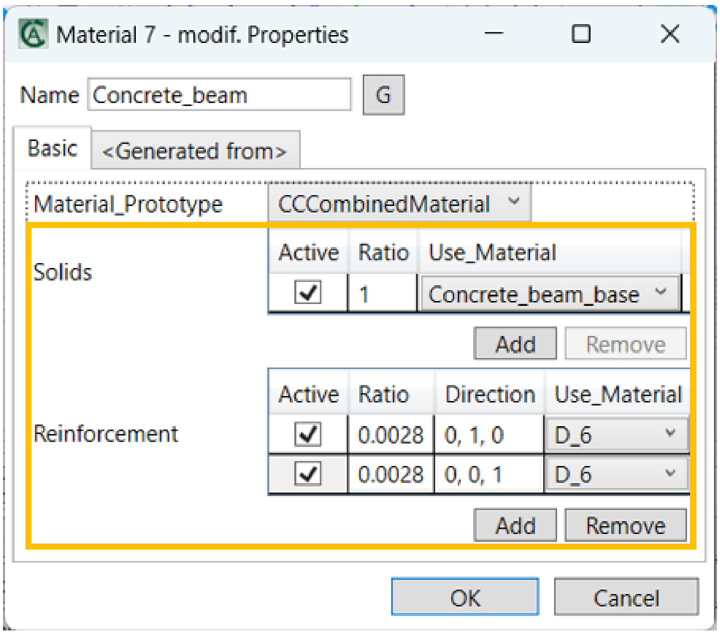


Fig. 3-49: Properties of CCCombinedMaterial.

List of all Concrete materials is visible in the table, Fig. 3-50.

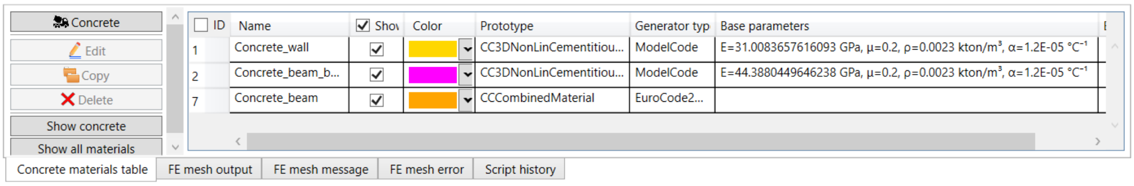


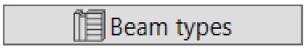
Fig. 3-50: List of concrete materials.

3.3 Material assignment to geometry

Materials that have been created in previous chapter are not assigned to any geometry. At this point the materials are assigned to geometrical entities.

3.3.1 Building element types

Two Building element types are made for two types of geometry: beams and wall. The view in Workspace is adjusted so that selecting of geometrical entities is easier. Therefore all the layers except the 'wall' layer are switched off in Layers window.

First the new building element type is created by selecting **Building element type | Beam** (Fig. 3-51) with double click in Input tree which opens the dialogue for building element type specification, or with just one click the BE beam table is activated and the button Beam types 

is pressed, Fig. 3-52. In the dialogue are chosen the properties of the geometric entity, mesh parameters and material:

Parameters input:	
Geometric entity	solid
Static model	3D (solid)
Mesh type	linear
Assigned material	Concrete_beam
BE type	Beam types

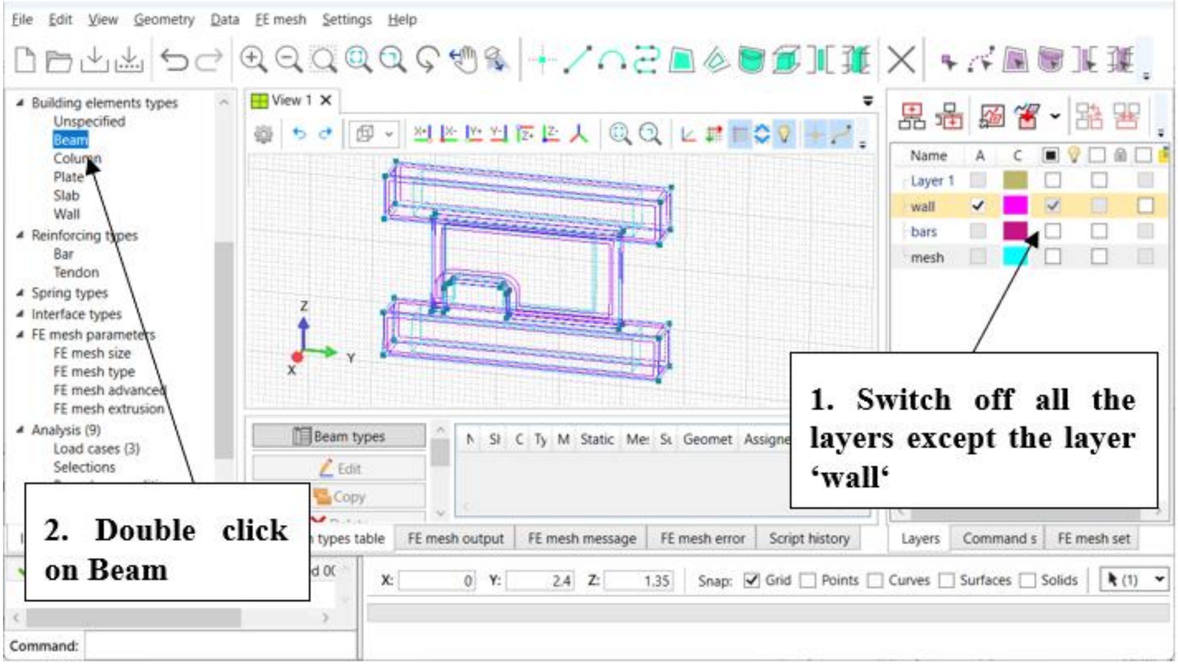


Fig. 3-51: New BE types - Beam.

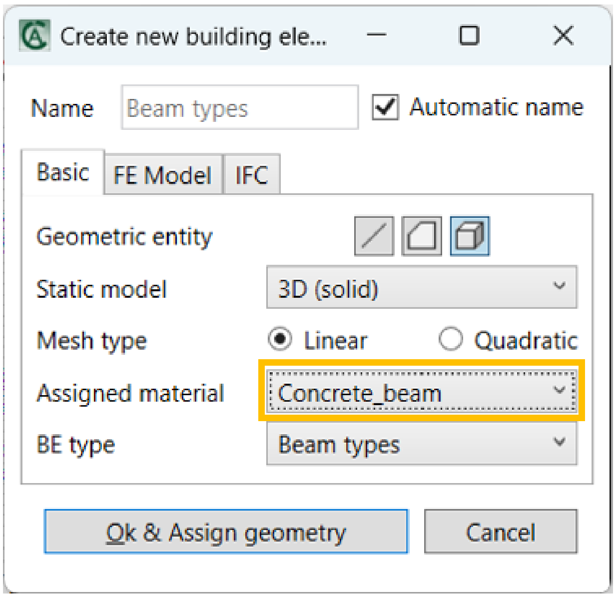


Fig. 3-52: BE types properties.

When the parameters are selected the Ok button should be pressed which triggers the selection mode to choose the geometry. Both beam solids are selected with mouse cursor changing the beam color into red, seeFig. 3-53. Then the Enter button completes the selection process and materials and mesh properties are assigned to geometrical entities.

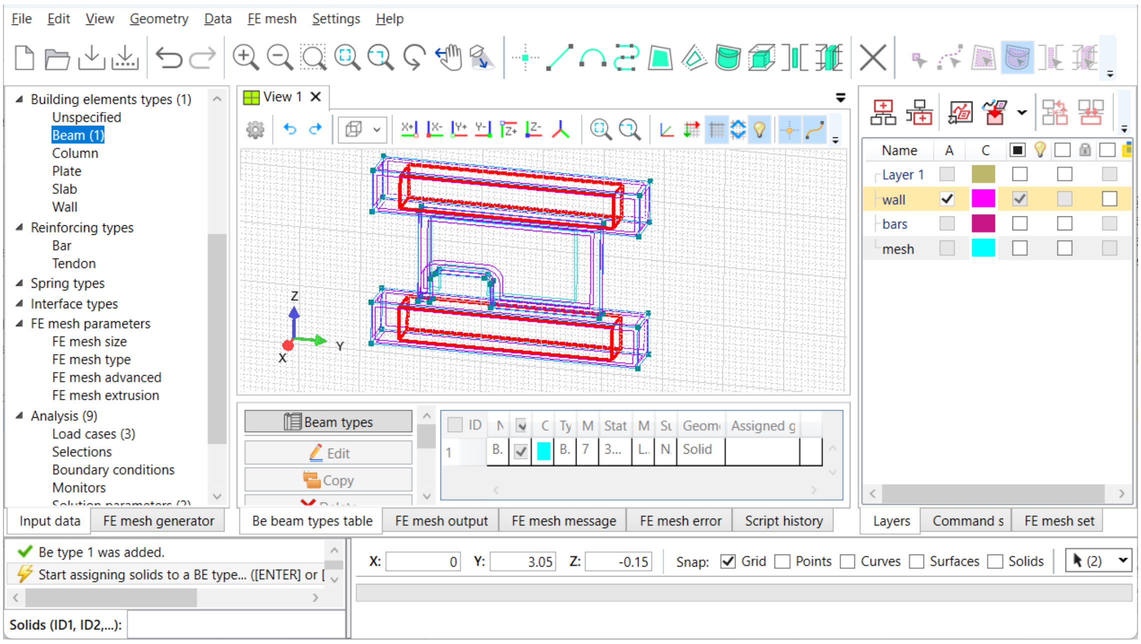


Fig. 3-53: Assignment of BE type to geometry.

When the color of the building element type is similar to colors which are already used in the Workspace view, it is helpful to change the color of BE types in **BE beam types table** so that the new color is clearly visible in the model, in Fig. 3-54.

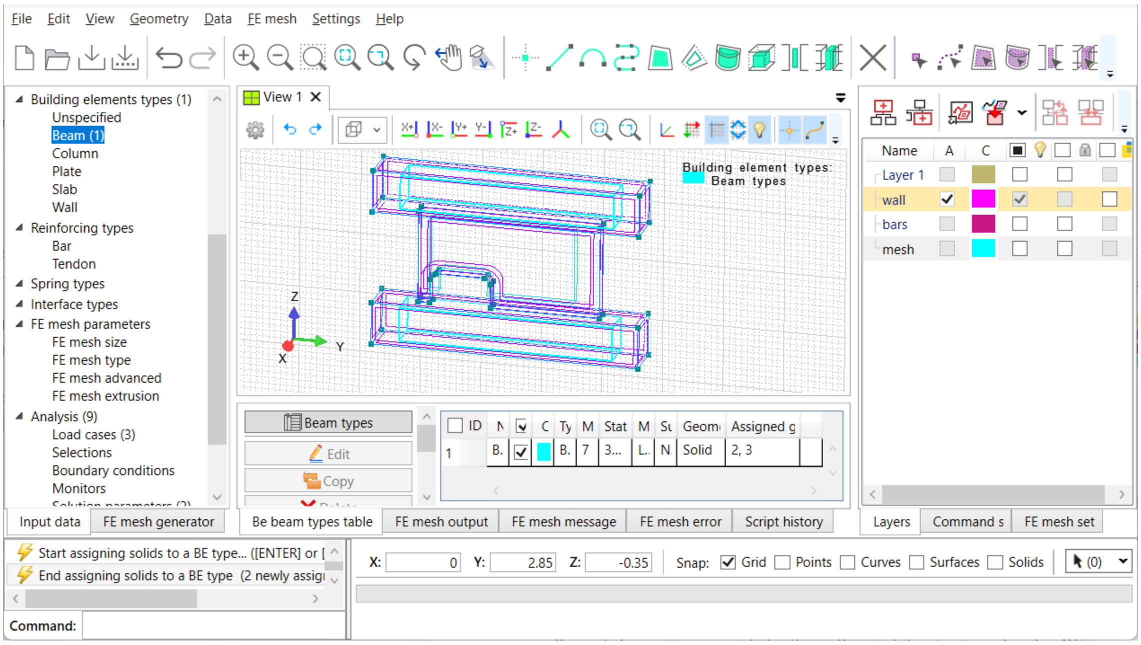


Fig. 3-54: BE type assigned to geometry.

Second element type is made for the wall part of the structure. Another BE type is created via BE type dialogue which pops up after double click on **Building element types | Wall** in Input tree, Fig. 3-55. The parameters of the Wall are adjusted as in previous case for solid entity with linear mesh. Then the solid (wall part of the structure) is selected and with Enter the assigning of properties for chosen entity is finished, Fig. 3-56.

Parameters input:

Geometric entity	solid
Static model	3D (solid)
Mesh type	linear
Assigned material	Concrete_wall
BE type	Wall types

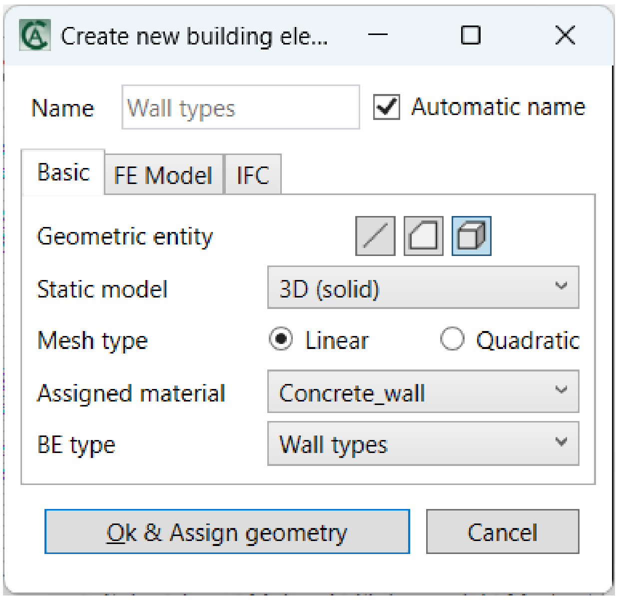


Fig. 3-55: BE type properties.

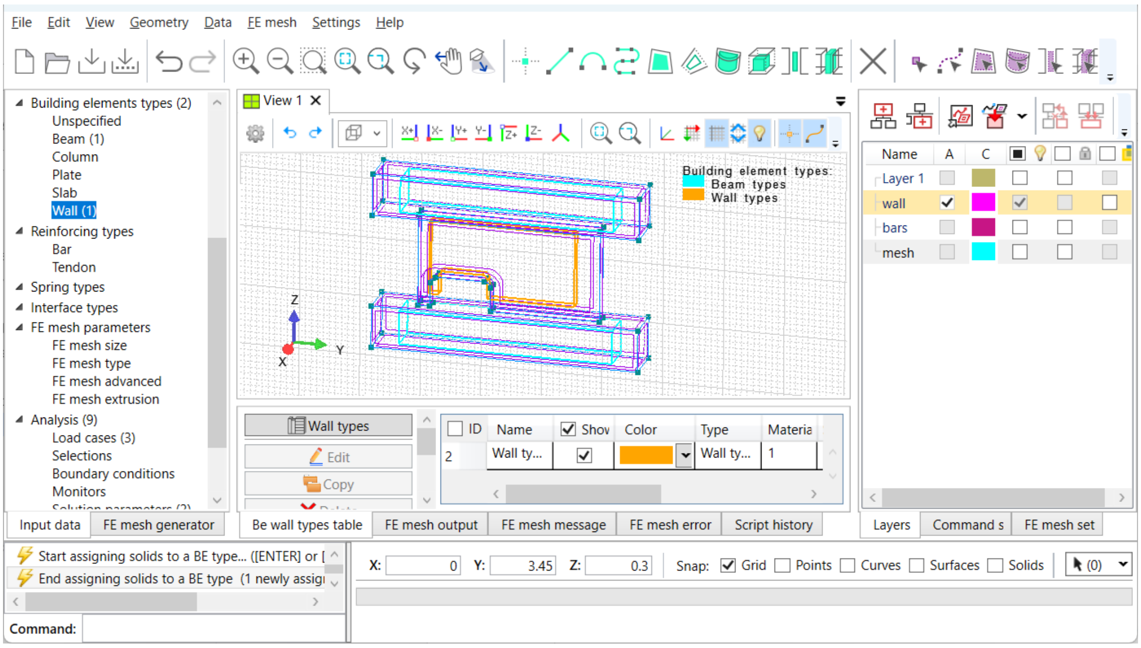


Fig. 3-56: BE types assigned to solids.

3.3.2 Reinforcing types

There are 4 different materials for discrete reinforcement and for each of them Reinforcing types are going to be created. At first the layer with mesh geometry which are bars with diameter 6 mm and material D_6 is only one activated, see Fig. 3-57.

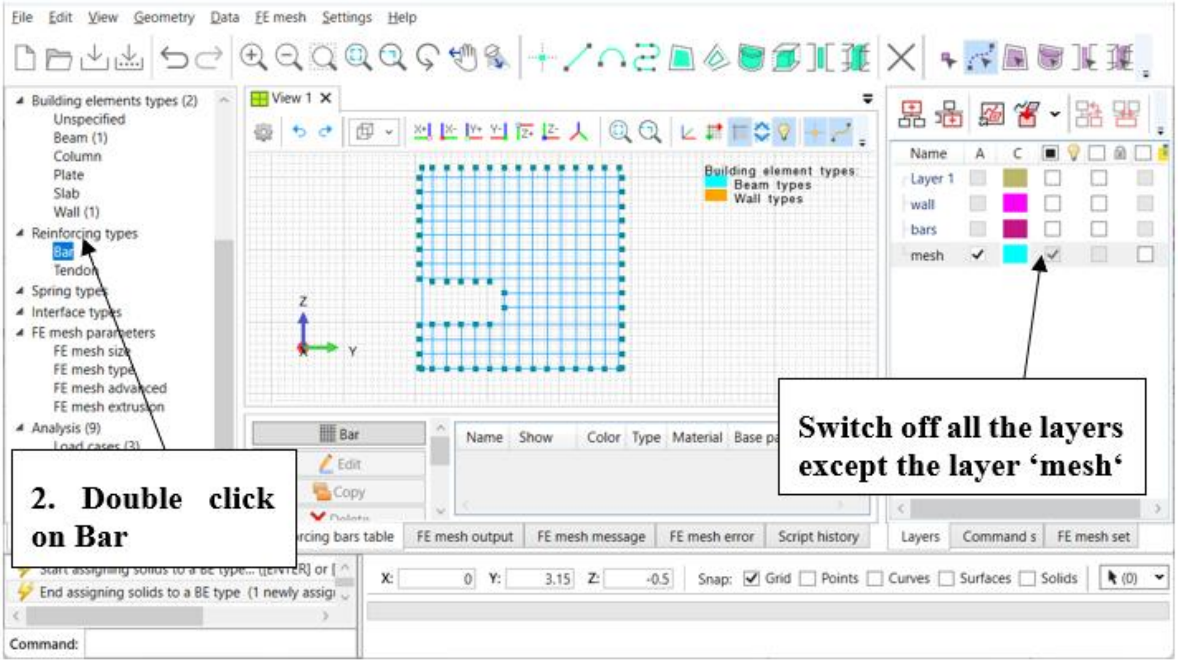


Fig. 3-57: Preparing to create Reinforcing types.

Then, reinforcing element type dialogue opens double click on Bar in Input tree **Reinforcing types | Bar**, or the simple click on Bar that activates Reinforcing bars table where is button Bar .

Parameters input:	
Name	Bar_D6
RE type	Bar
Bond type	Perfect bond
Assigned material	D_6
Bar diameter	6 mm

The input parameters specified above are entered into Create reinforcing element type dialogue (Fig. 3-58 and Fig. 3-59). To activate the change of the name the checkbox Automatic name has to be unchecked. Also when the bar diameter is changed with mouse click outside of the Bar diameter box the profile area and perimeter are updated.

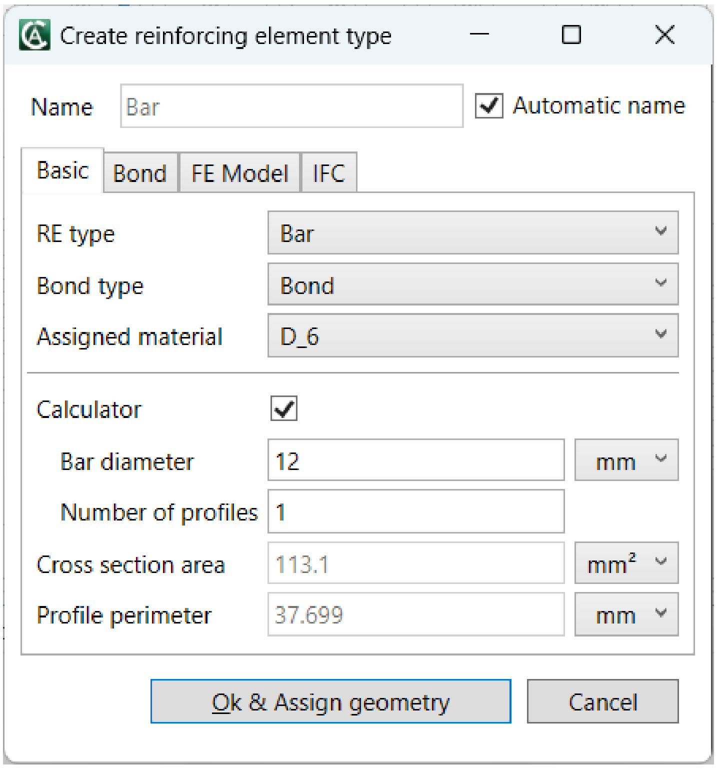


Fig. 3-58: Reinforcing type properties - default.

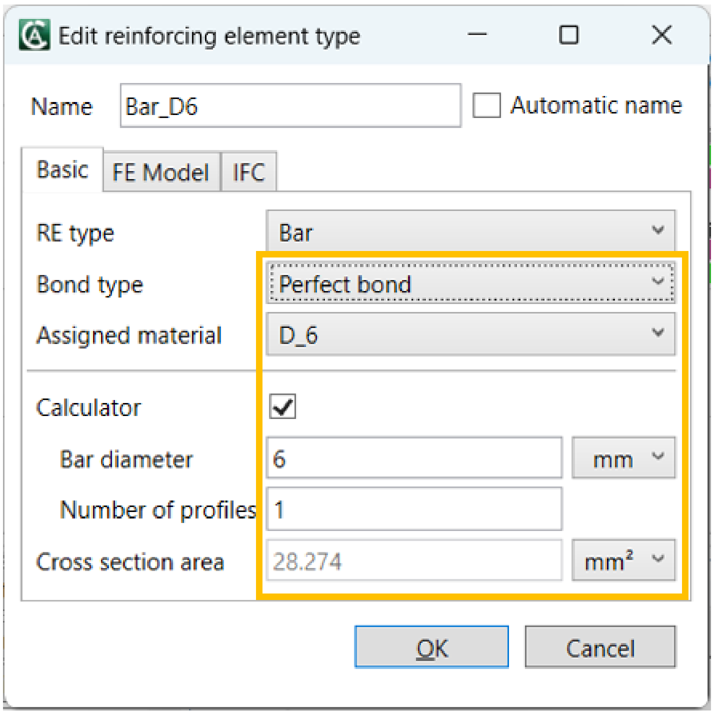


Fig. 3-59: Reinforcing type properties – adjusted for material D_6.

Once the parameters are set, the button Ok should be pressed which turns program in selection mode and the lines with described parameters are chosen, Fig. 3-60. Selection of all lines in layer 'mesh' changes their color into red and is visible in Workspace view. Then Enter button completes the selection and assignment of reinforcing types.

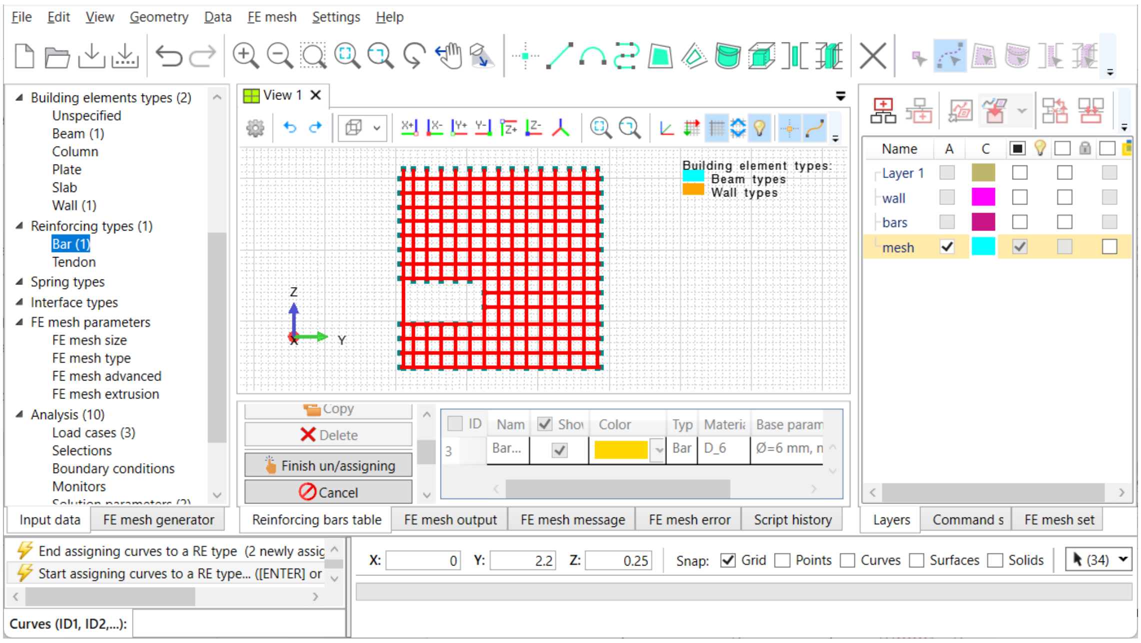


Fig. 3-60: Selected geometry of Reinforcement type.

Reinforcing type which has just been made appears in the Reinforcing bars table and lines in Workspace view change into corresponding color, Fig. 3-61.

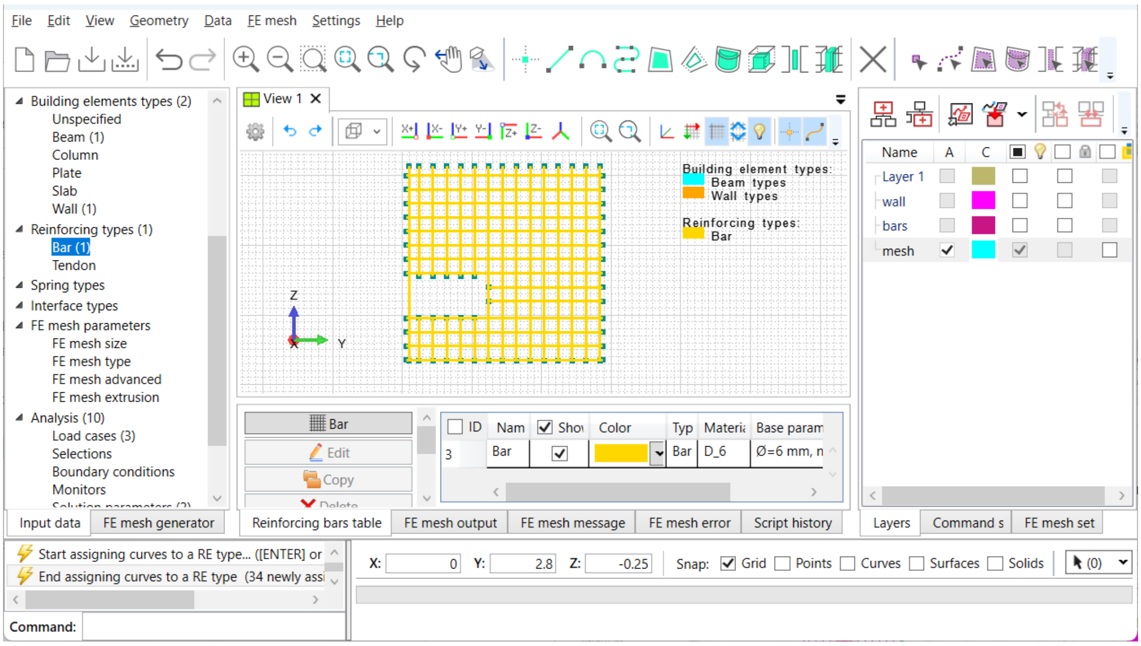


Fig. 3-61: Geometry with assigned Reinforcing type.

When mesh reinforcement is finished, the layer ‘mesh’ is deactivated and layer ‘bar’ is activated, Fig. 3-62.

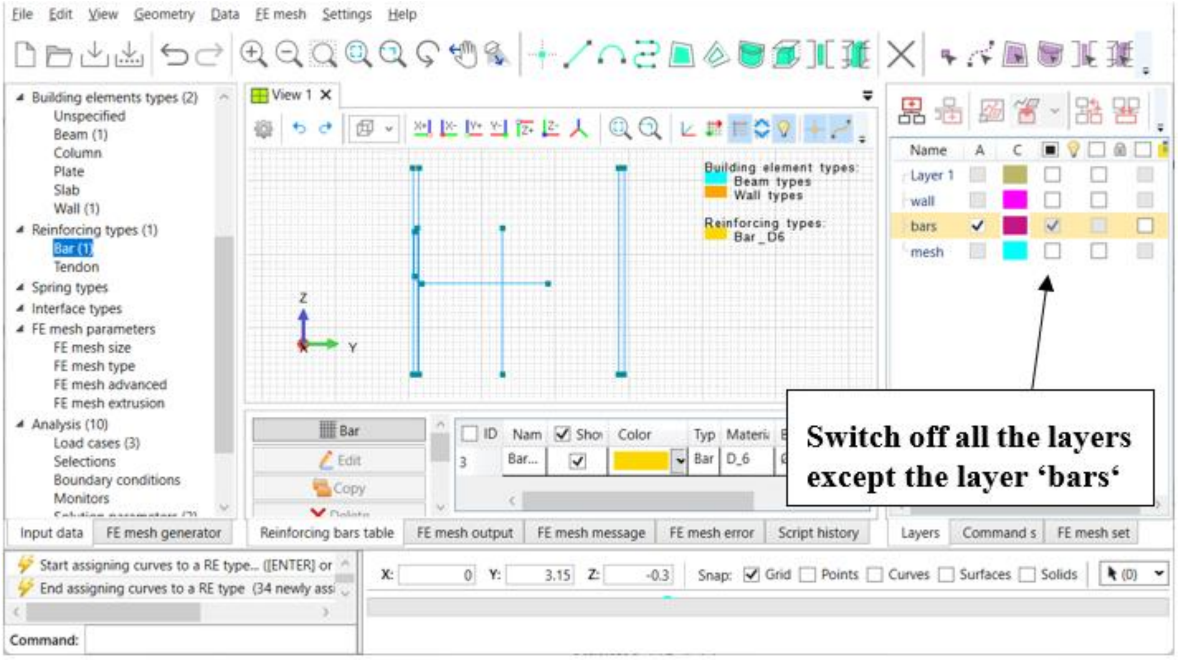


Fig. 3-62: Preparation for new Reinforcing types assignment.

Horizontal bar above opening that is bent into vertical direction and has diameter 8 mm. Reinforcing type named Bar_D8 with material D_8 and respective diameter is assigned to the geometry, see Fig. 3-63.

Parameters input:	
Name	Bar_D8
RE type	Bar
Bond type	Perfect bond
Assigned material	D_8
Bar diameter	8 mm

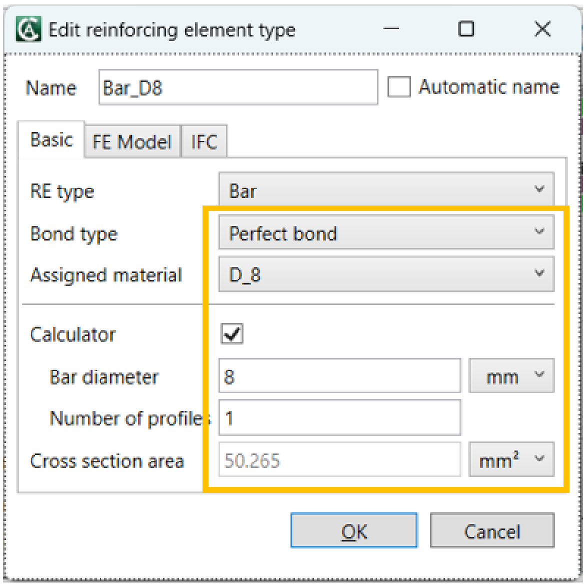


Fig. 3-63: Reinforcing type for material D_8.

The procedure is same as in previous case, the Reinforcing type is created then the corresponding lines are selected and assigned to Bar_D8. The lines with assigned Reinforcing type are visible in Workspace view and new Reinforcing type is listed in table below, Fig. 3-64.

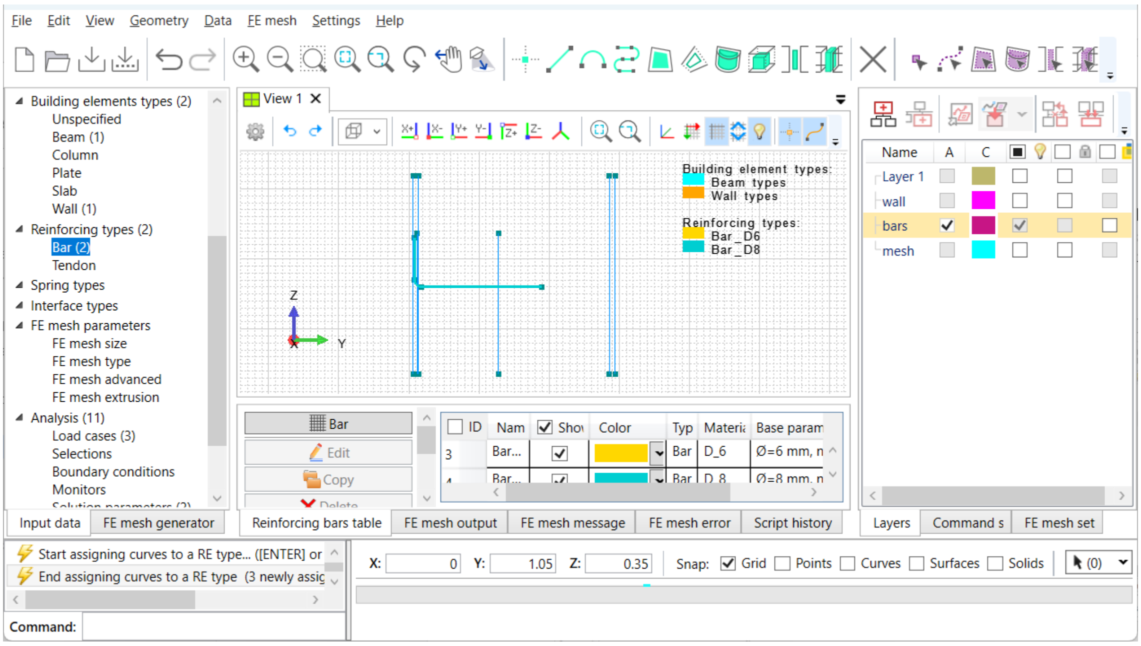


Fig. 3-64: Reinforcing type with material D_8 assigned to geometry.

Two shorter vertical bars along the opening have diameter 10 mm and therefore material D10 should be assigned to them, Fig. 3-65.

Parameters input:	
Name	Bar_D10
RE type	Bar
Bond type	Perfect bond
Assigned material	D10
Bar diameter	10 mm

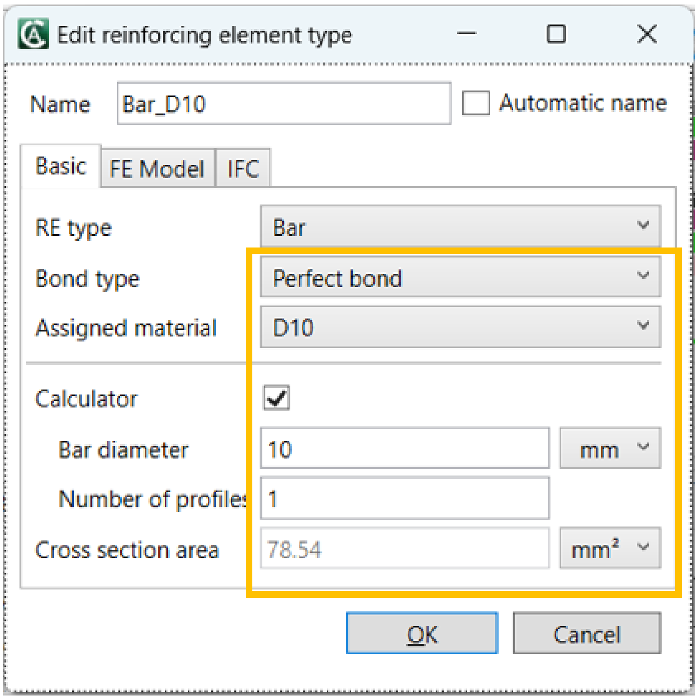


Fig. 3-65: Reinforcing type for material D10.

Reinforcing type Bar_D10 is made with parameters above and assigned to short vertical lines. The lines change color accordingly and Reinforcing type appears in table of Reinforcing bars, Fig. 3-66.

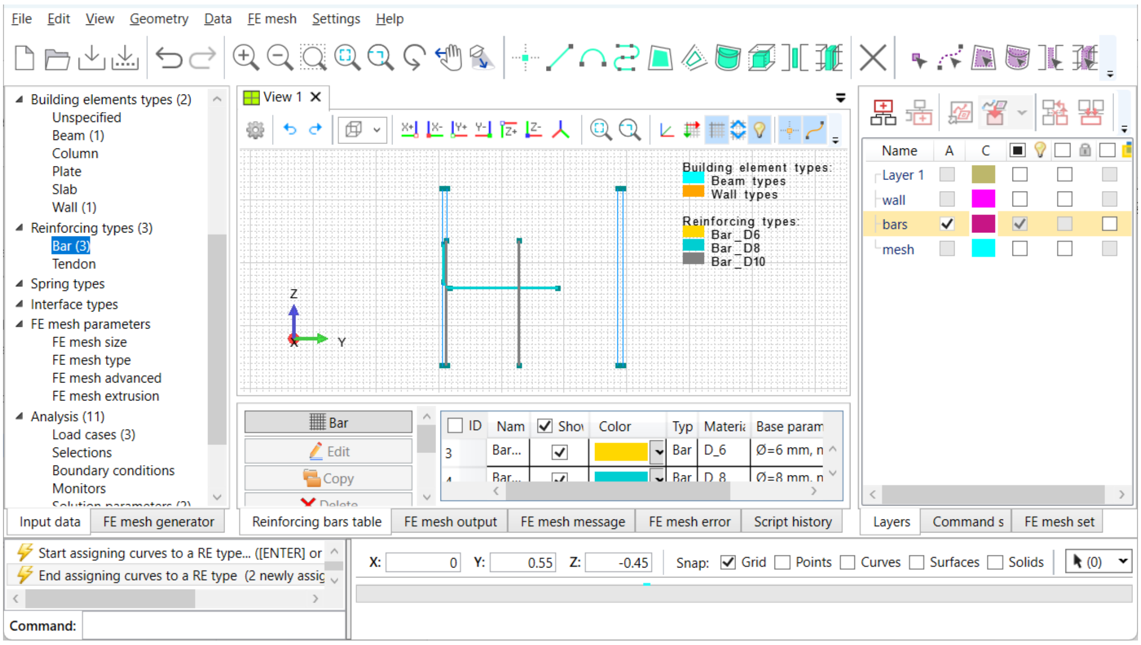


Fig. 3-66: Reinforcing type with material D10 assigned to geometry.

Remaining discrete vertical bars have diameter 12 mm and their material is D12. Corresponding Reinforcing type is created with parameters below and assigned to remaining geometry, see Fig. 3-67.

Parameters input:	
Name	Bar_D12
RE type	Bar
Bond type	Perfect bond
Assigned material	D12
Bar diameter	12 mm

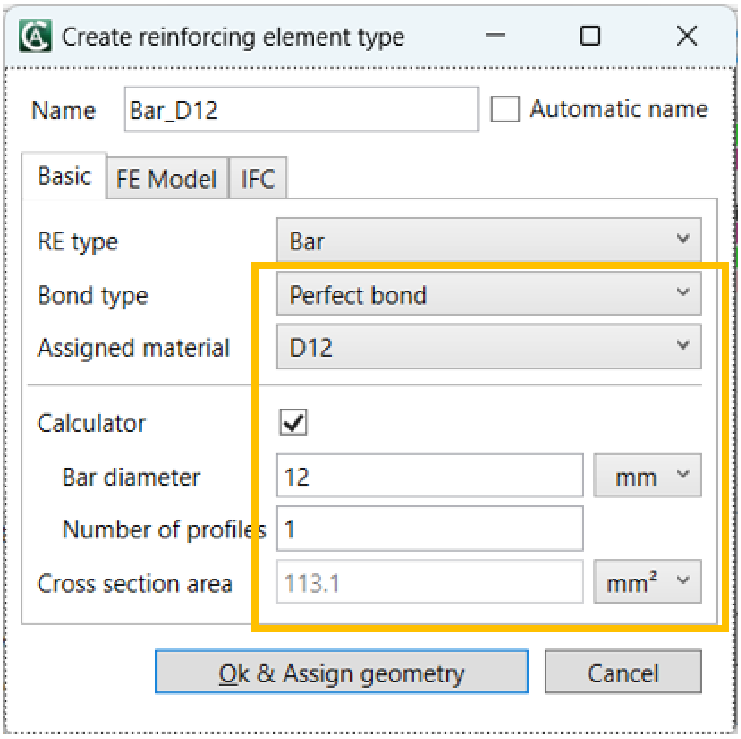


Fig. 3-67: Reinforcing type for material D12.

Fourth Reinforcing type named Bar_D12 is made and the last four vertical lines are selected to complete assignment of corresponding material, bond or diameter.

All Reinforcing types are listed in table and the corresponding geometry is visible in Workspace view with descriptive legend as shown in Fig. 3-68.

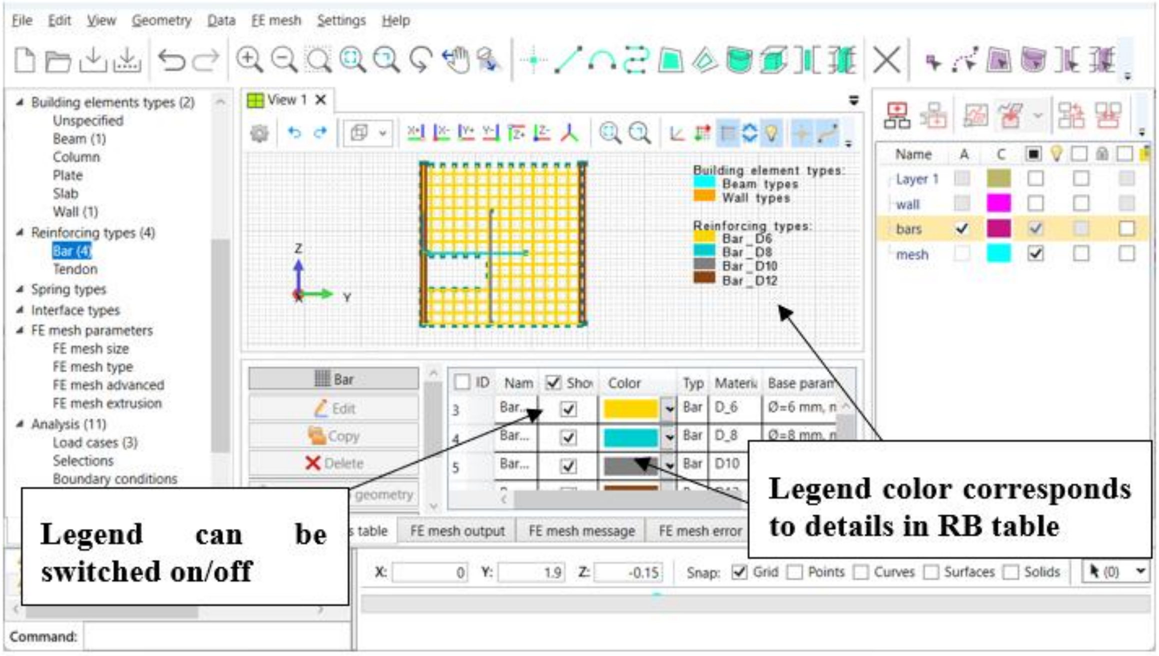


Fig. 3-68: Reinforcing types for all reinforcement material assigned and listed in table below.

3.4 Finite Element Mesh

By now the geometry of the model has been built up, material models have been set up and assigned to geometry. Therefore at this point the FEM mesh can be prepared.

Default FEM mesh is generated in **FE mesh generator** tab within Input tree section, see Fig. 3-69. Then when the user scrolls down in the generator the button **Preview FE mesh** Preview FE mesh generates and shows mesh in the Workspace view.

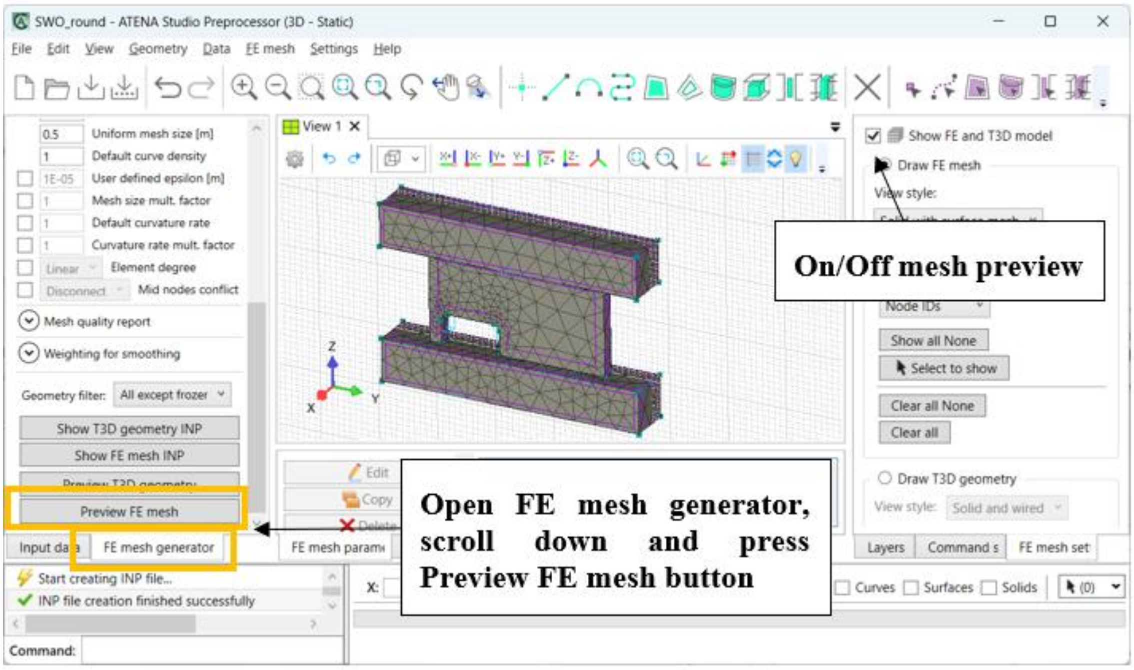


Fig. 3-69: Preview default FE mesh.

Quality of the mesh is crucial for FE analysis and its results therefore the mesh needs to be improved. Before the mesh parameters are changed, the FE mesh preview is switched off (in **FE mesh settings**) and the 'wall' layer is activated and the BE types with Reinforcing types are deactivated in the Workspace view via checkbox Show in the respective tables.

The size of the mesh is adjustable in Input tree **FE mesh parameters | FE mesh size**.

Then in **FE mesh parameters table** the solids  are chosen which pops up a window with Size definition, Fig. 3-70. The User defined size definition is selected and size is set to 0.1 m, Fig. 3-71.

Parameters input:	
Size definition	User defined
Mesh size	0.1 m
Equidistant	check

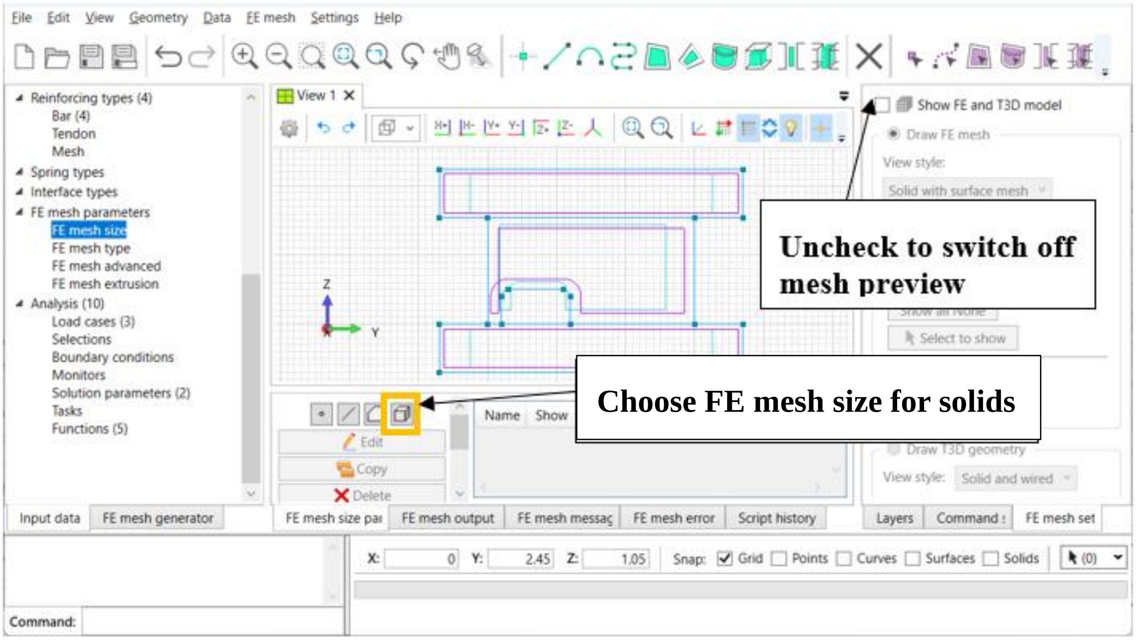


Fig. 3-70: Change FE mesh size for solid (beams).

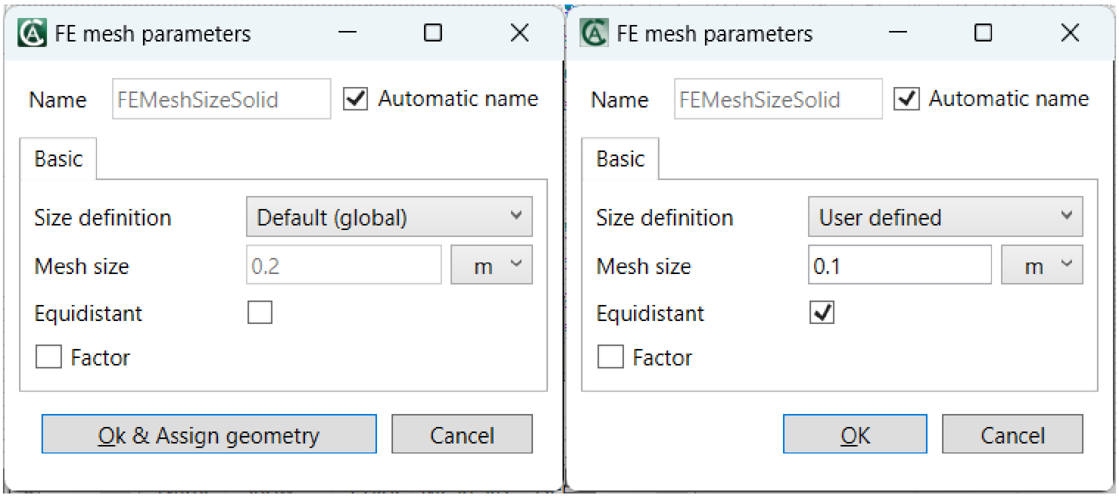


Fig. 3-71: FE mesh size parameters – default and then adjusted.

After pressing button **Ok & Assign geometry**, the selection mode is directly activated and the beams (top and bottom) are marked (Fig. 3-72), the selection process is concluded with **Enter**.

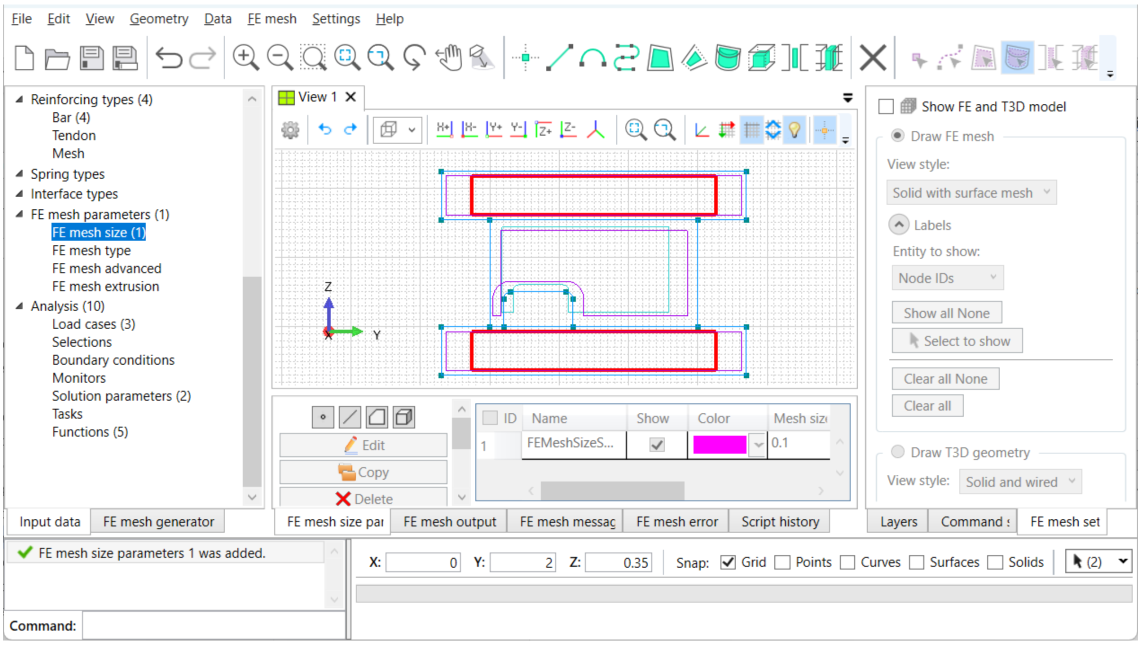


Fig. 3-72: Selected solids with modified mesh size.

In the **FE mesh size parameters table** can be checked the correct assignment to geometry, Fig. 3-73.

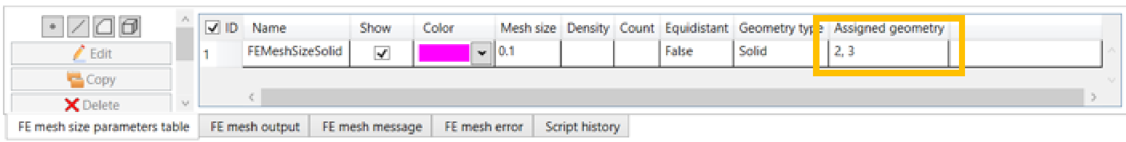


Fig. 3-73: Assigned geometry to new FE mesh size.

Other mesh size is prescribed for the middle panel. Again, **Create new FE mesh for solid**  is chosen and the mesh size for middle panel is set to 0.05 m, Fig. 3-74.

Parameters input:	
Size definition	User defined
Mesh size	0.03 m

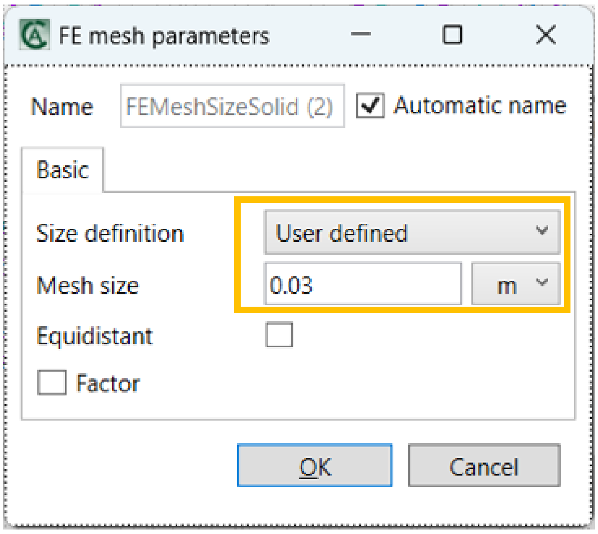


Fig. 3-74: FE mesh size for middle panel.

After pressing **Ok & Assign geometry** middle panel solid is selected and the **Enter** finishes the task, Fig. 3-75.

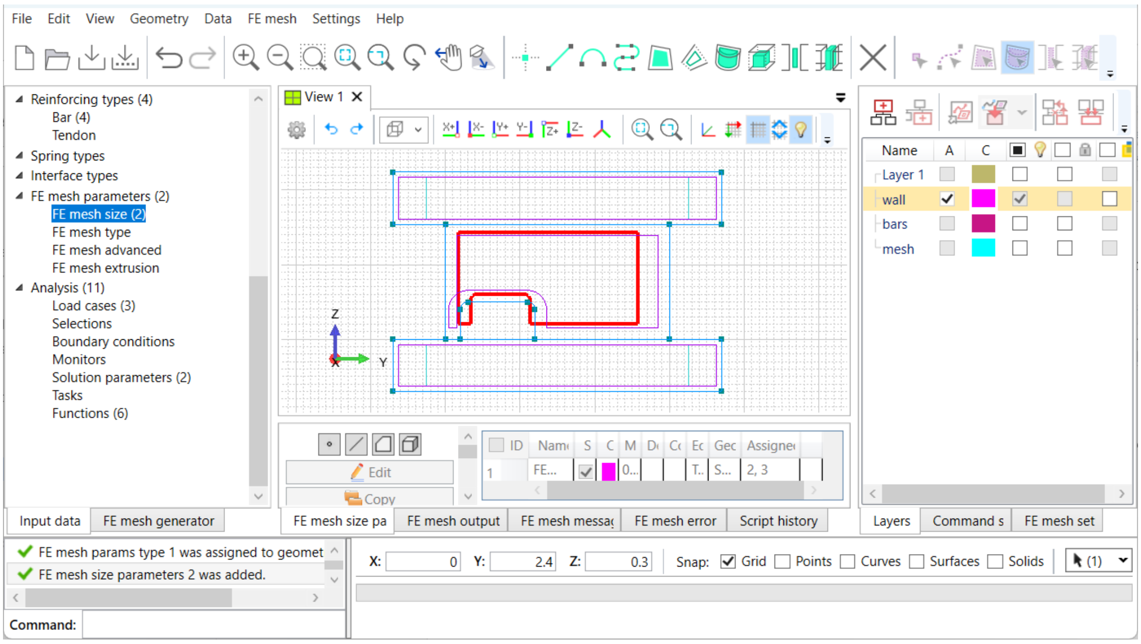


Fig. 3-75: Selected geometry for FE mesh size modification.

The assigned geometry of respective FE mesh size is displayed in FE mesh size parameters table, see Fig. 3-76.

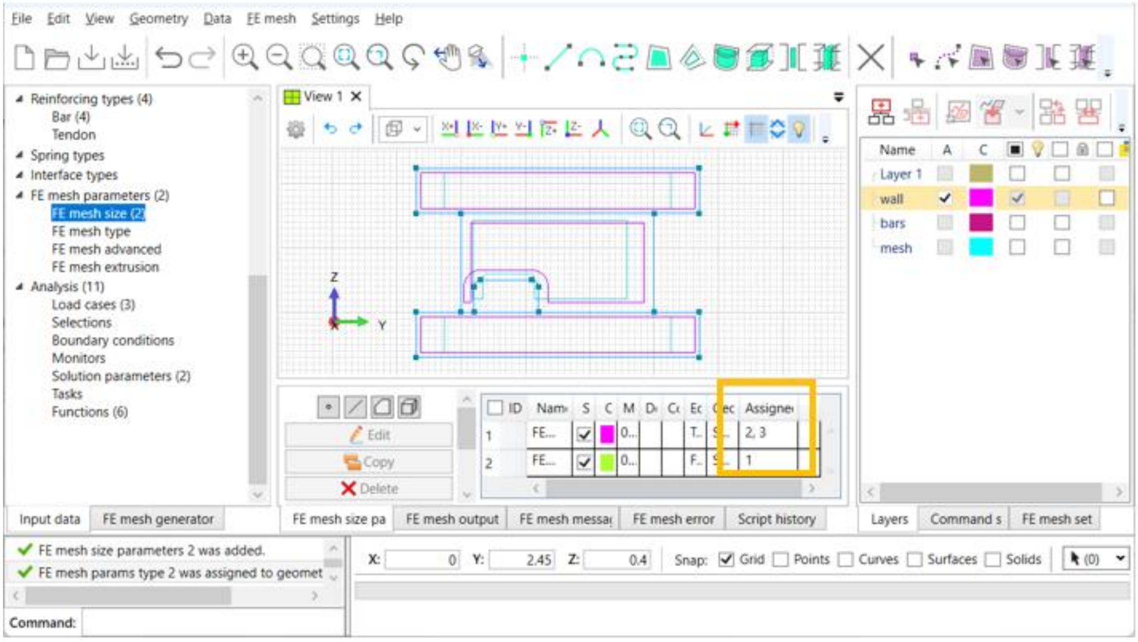


Fig. 3-76: FE mesh sizes assigned to different solids.

The adjusted size of mesh can be checked in **FE mesh generator** when **Preview FE mesh** is applied, Fig. 3-77.

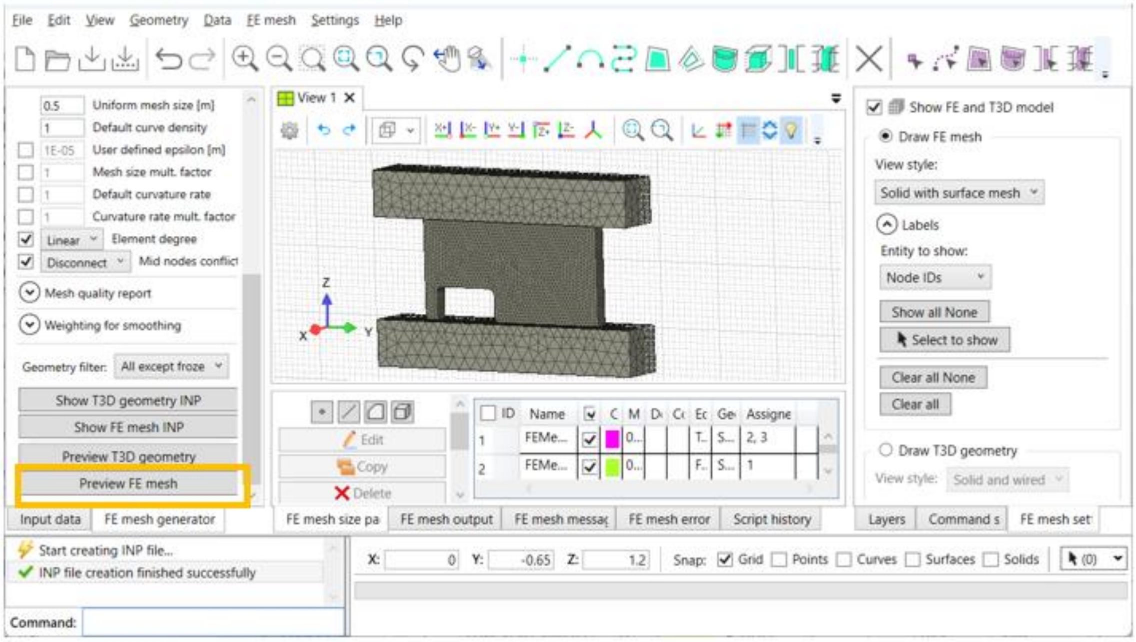


Fig. 3-77: FE mesh preview with adjusted mesh size.

The size of the mesh is set nevertheless the unstructured FE type – tetrahedra is going to be modified. Before the FE mesh type is created, the FE mesh preview needs to be switched off in **FE mesh settings**, otherwise the selection of geometry is not visible in Workspace view.

The beams are solids with 6 surfaces that are prism-shaped therefore the structured mesh using hexahedrons can be easily applied. In **FE mesh parameters | FE mesh types** the **Create new FE mesh for solid** (Fig. 3-78) is selected in **FE mesh type parameters table**. The dialogue of FE mesh type appears, Fig. 3-79. The FE mesh structured is selected (Fig. 3-80) then the **Ok & Assign geometry** button is pressed, and beams geometry is selected (selected geometry turns red as in previous cases). The process is confirmed and executed by pressing **Enter**. Assigned geometry can be checked in table below, see Fig. 3-81.

Parameters input:	
FE mesh structured	Yes

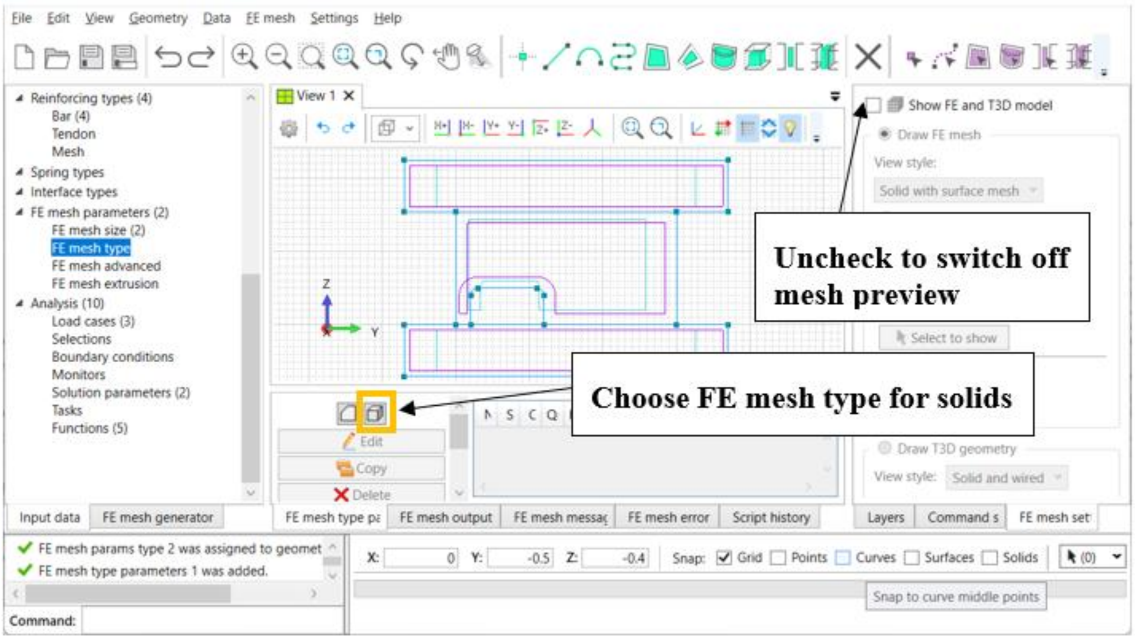


Fig. 3-78: Create new FE mesh type for solid.

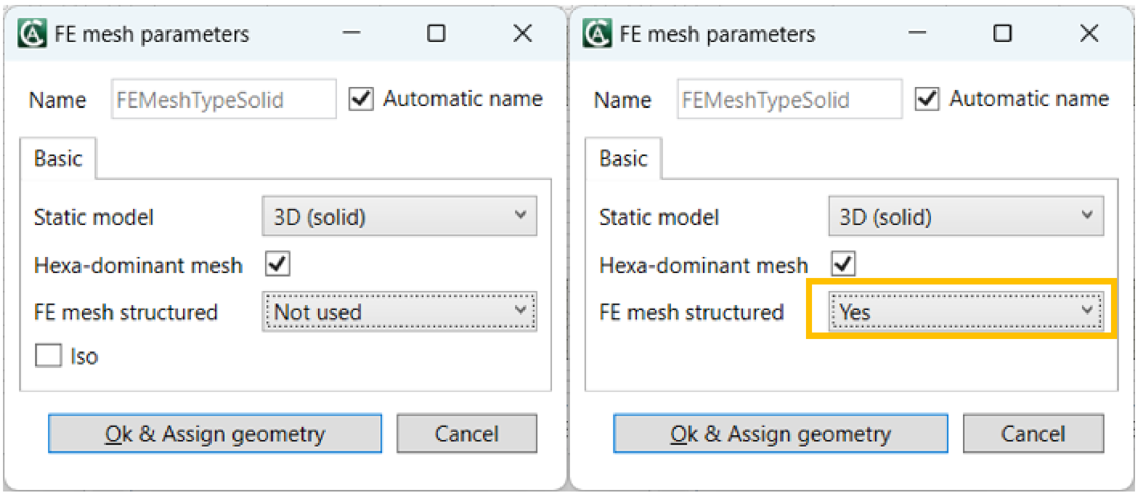


Fig. 3-79: FE mesh type parameters – default and then adjusted to structured.

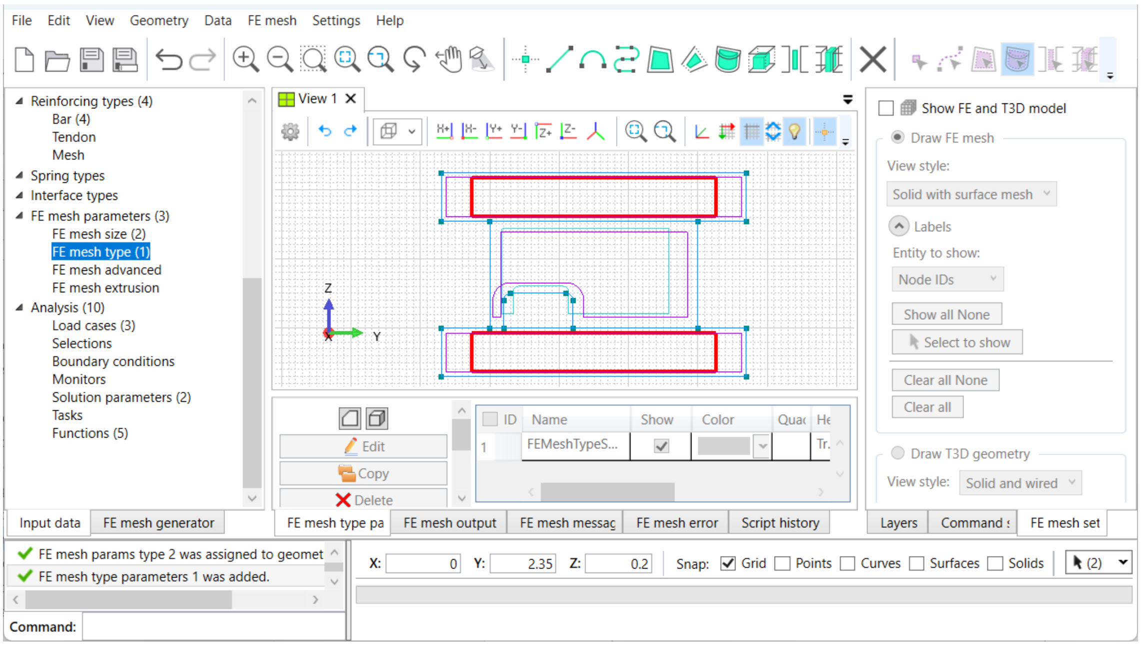


Fig. 3-80: Selection of geometry for new FE mesh type.

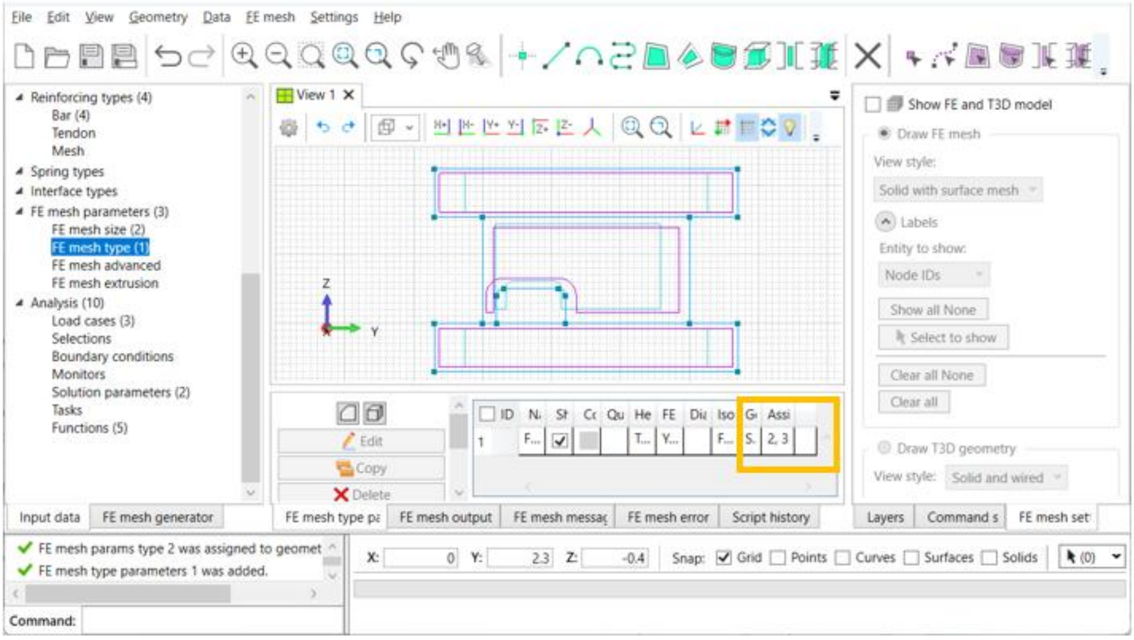


Fig. 3-81: New FE mesh type listed in table with assigned geometry.

Geometry which is assigned to newly created FE mesh type is displayed in **FE mesh type parameters table**.

Preview of the changes in mesh are visible in **FE mesh generator** after clicking on **Preview FE mesh** button, Fig. 3-82.

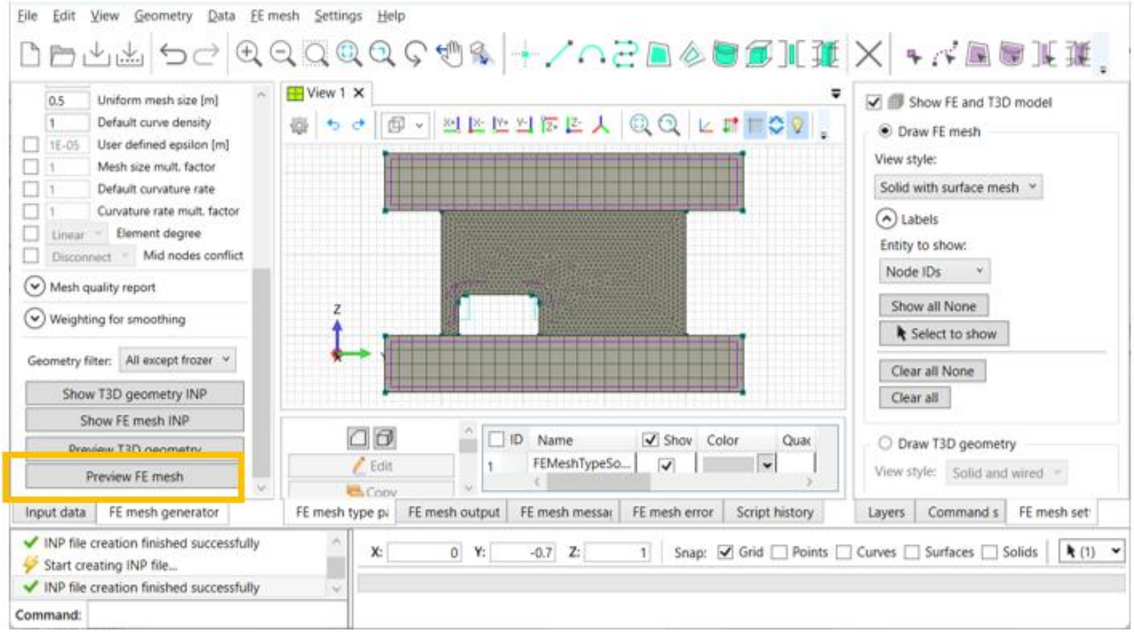


Fig. 3-82: Preview of FE mesh with structured elements within beams.

Finally the shape of elements in middle panel are edited. Adding the new FE mesh type for surface changes triangular mesh into quadrilateral. As in previous cases, the Workspace view is set to geometry instead of FE mesh, Fig. 3-83.

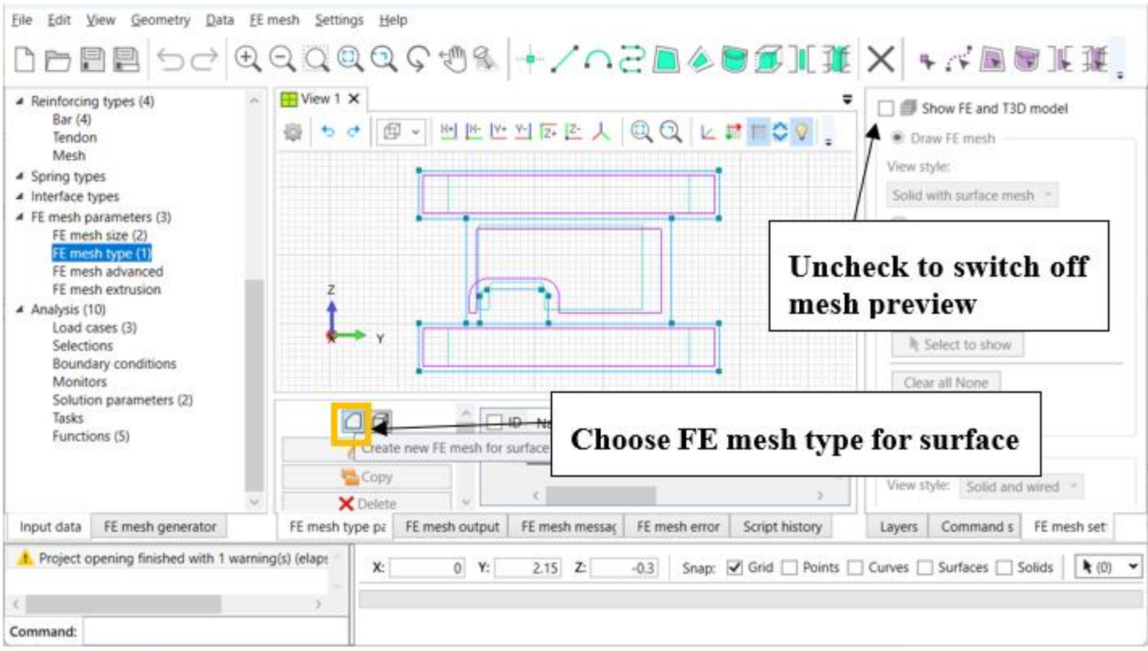


Fig. 3-83: New FE mesh type for surface.

In dialogue of FE mesh type for surface the structured mesh is chosen and by clicking on the Ok & Assign geometry button the surface can be selected. The main surface of middle panel (number 1 or 2) is selected in order to assign quad-dominant mesh, Fig. 3-84 and Fig. 3-85.

Parameters input:

FE mesh structured   Yes

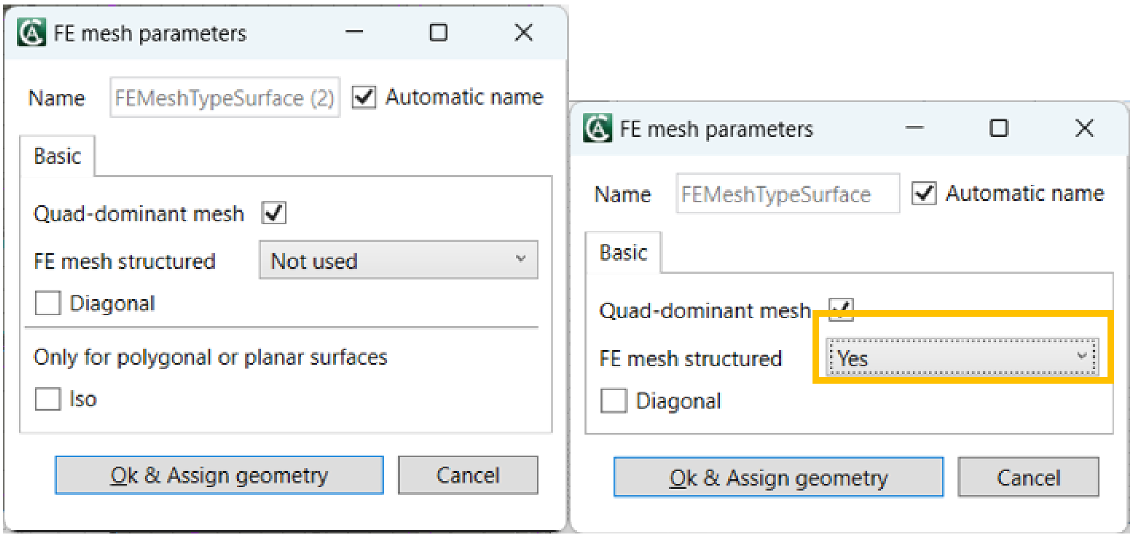


Fig. 3-84: FE mesh type parameters – default and adjusted with surface structured mesh.

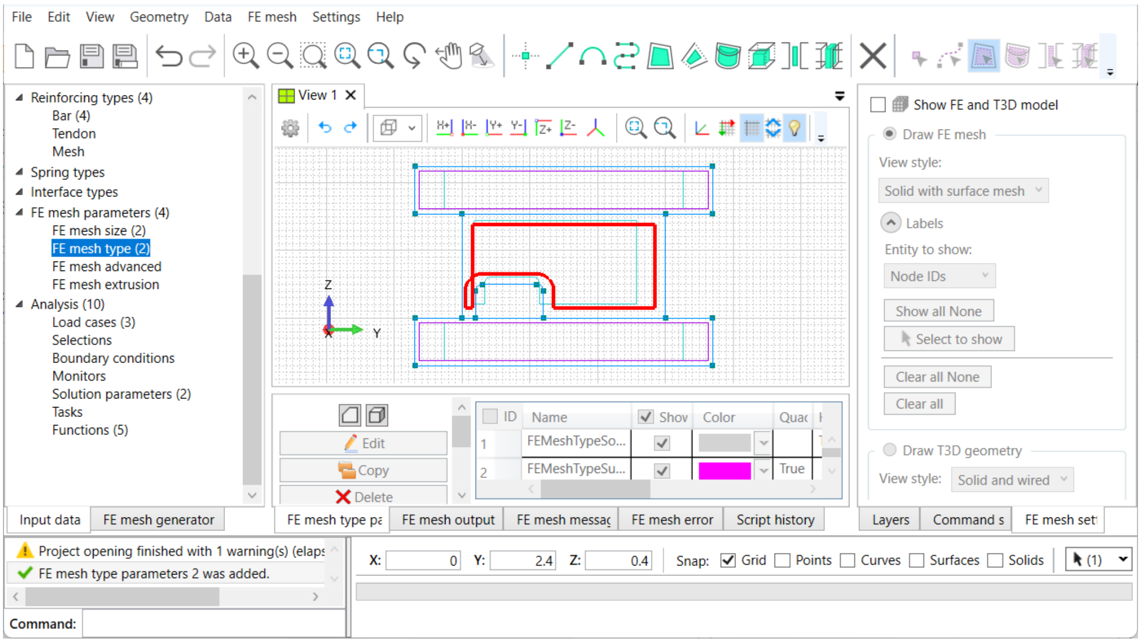


Fig. 3-85: Assigning of FE mesh type to selected geometry.

Quadrilateral mesh is assigned to surface number 2 which translates to parallel surface number 1 as well, Fig. 3-86.

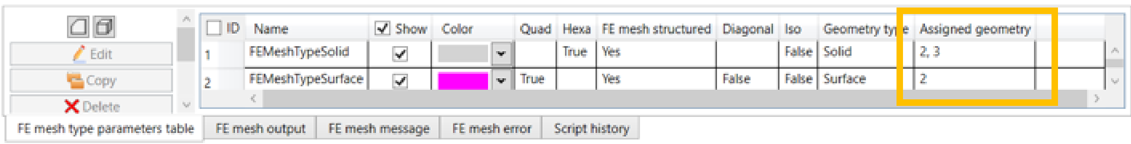


Fig. 3-86: List of FE mesh types and its assigned geometry.

The changes in mesh are previewed in mesh generator by clicking on **Preview FE mesh**, Fig. 3-87.

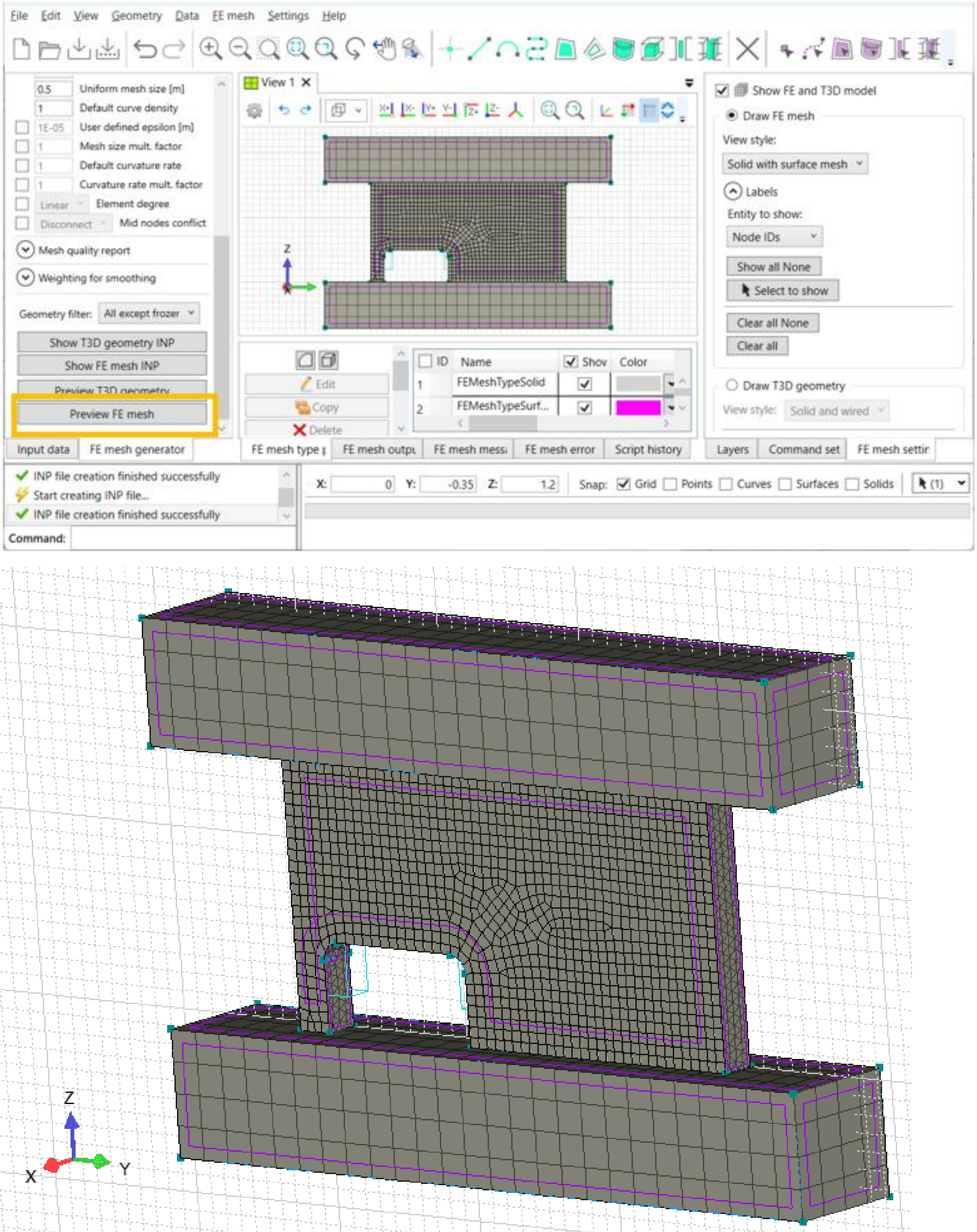


Fig. 3-87: Preview of structured mesh on the surface.

As it is visible on figure above, the quadrilateral mesh on the main surface of middle panel changed the type of elements into wedges. In the final step of mesh modification, the wedge elements turn into hexahedra elements.

The view in Workspace is set to geometry by unchecking the **Show FE and T3D model** within the FE mesh settings. Then in **FE mesh parameters | FE mesh extrusion** the new

FE mesh extrusion for solid is made, see Fig. 3-88. The dialog which appears does not require adjustment so the default settings is assigned to middle panel geometry solid, Fig. 3-89. Just by clicking on **Ok & Assign geometry** the selection of solid is activated and the middle panel is chosen, Fig. 3-90. Pressing **Enter** completes the assignment.

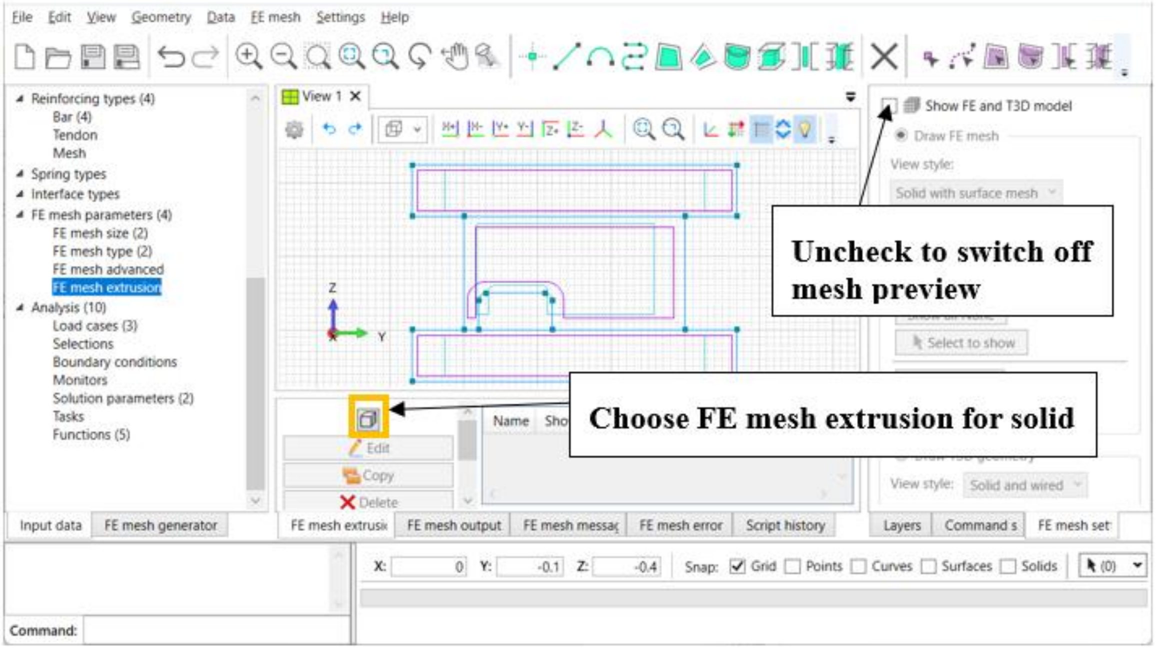


Fig. 3-88: New FE mesh extrusion.

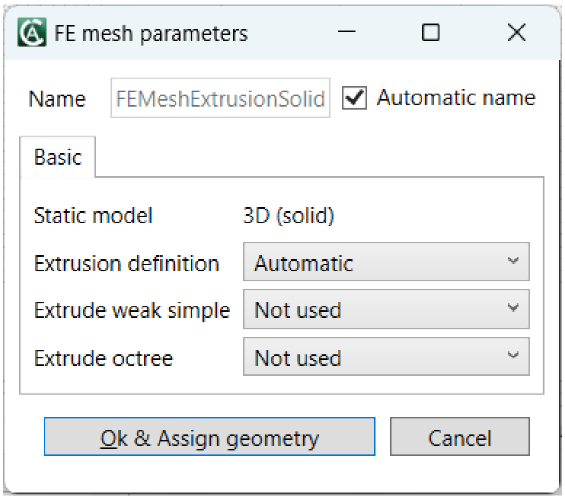


Fig. 3-89: FE mesh extrusion parameters.

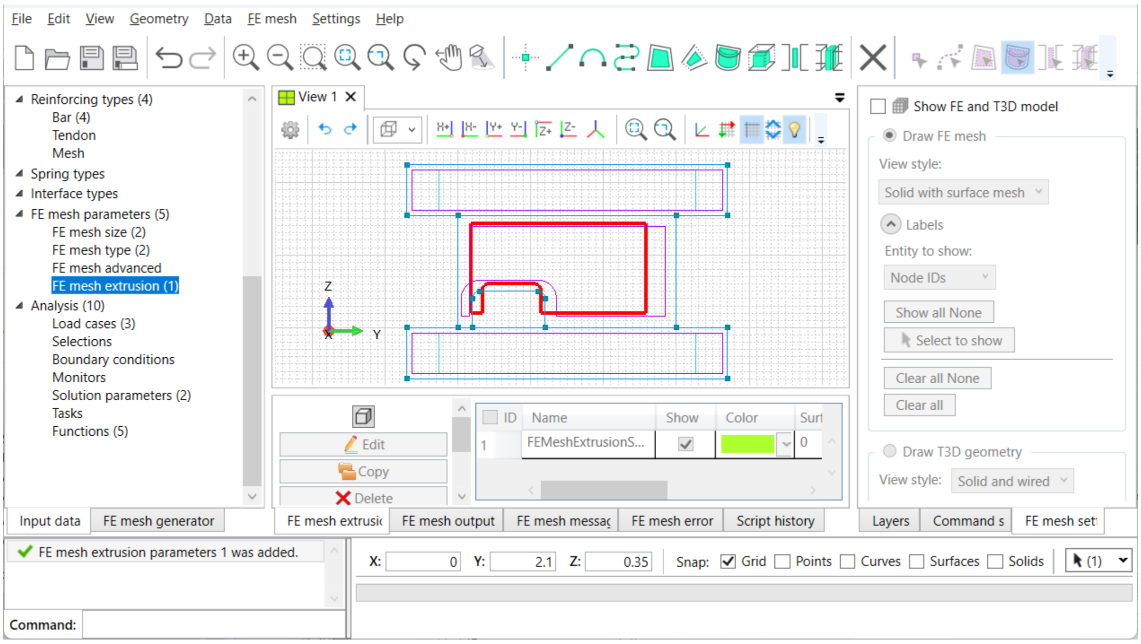


Fig. 3-90: Assignment of FE mesh extrusion.

Extrusion of mesh is assigned to geometry according to the selection and can be reviewed in table of **FE mesh extrusion parameters table**, Fig. 3-91.

The completed FE mesh is previewed in FE mesh generator when Preview FE mesh button is used. The hexahedra elements are now within the whole geometry of the model, Fig. 3-92.

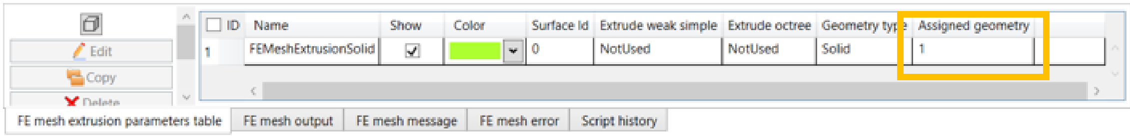


Fig. 3-91: FE mesh extrusion listed in table with assigned geometry.

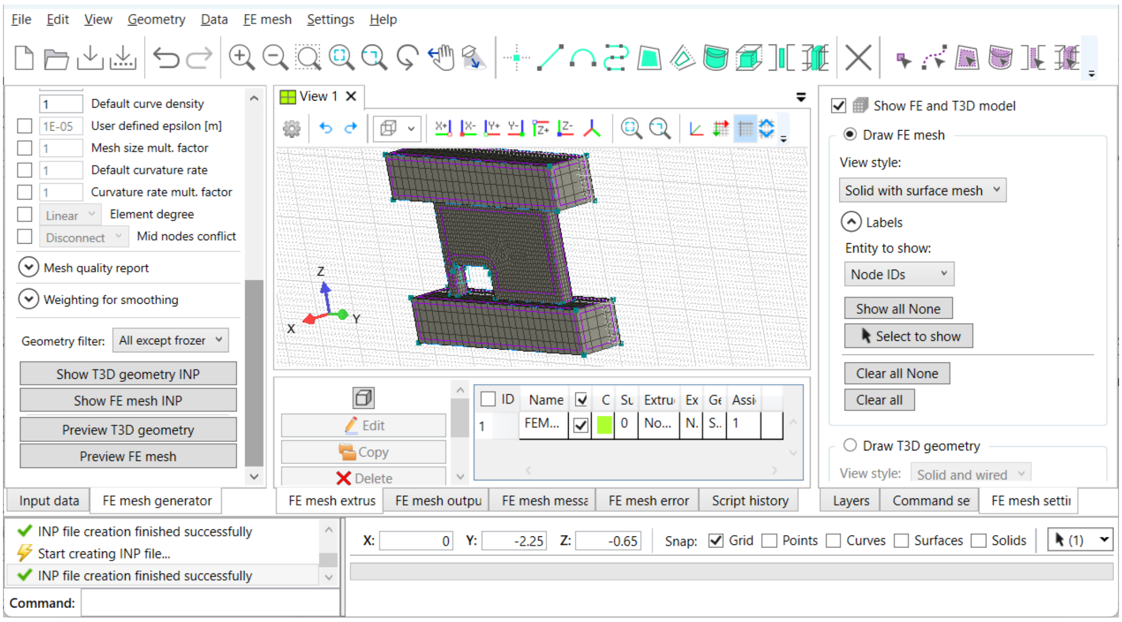


Fig. 3-92: Structured mesh in the whole model.

3.5 Supports

In this chapter the Support is assigned and the model parts with incompatible mesh are connected with Fixed contact.

3.5.1 Support

At the bottom part of Input tree is last category item Analysis. Clicking on **Analysis | Boundary conditions** (BC) the Boundary conditions table appears. The model is going to be supported at the bottom surface of the bottom beam therefore the new Surface boundary condition is selected, see Fig. 3-93.

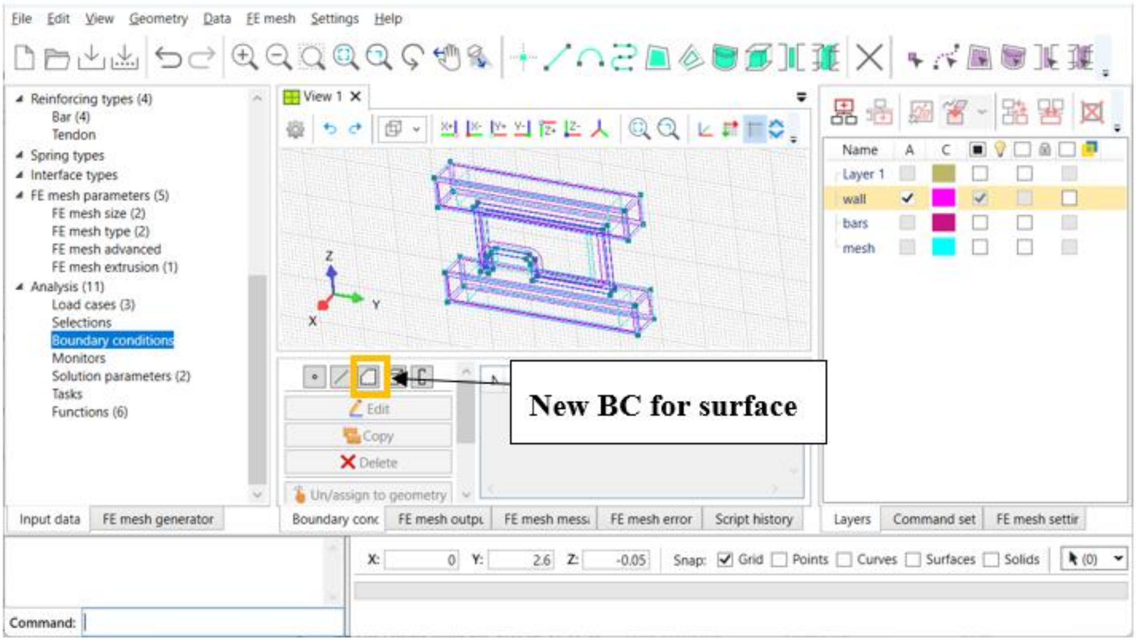


Fig. 3-93: New Boundary condition.

The default dialog of boundary condition offers first load case which is Body force by default, Fig. 3-94. Switching the load case to Supports the condition type Support is selected, Fig. 3-95. Then the support I defined in all directions of coordinate system X, Y and Z and the button **Save & Assign to geometry** switches the program to selection mode and the surface to be supported is selected, Fig. 3-96. By pressing **Enter** the command is concluded.

Parameters input:	
Load case	2 Supports
Condition type	Support
Coordinate system	Support in X
	Support in Y
	Support in Z

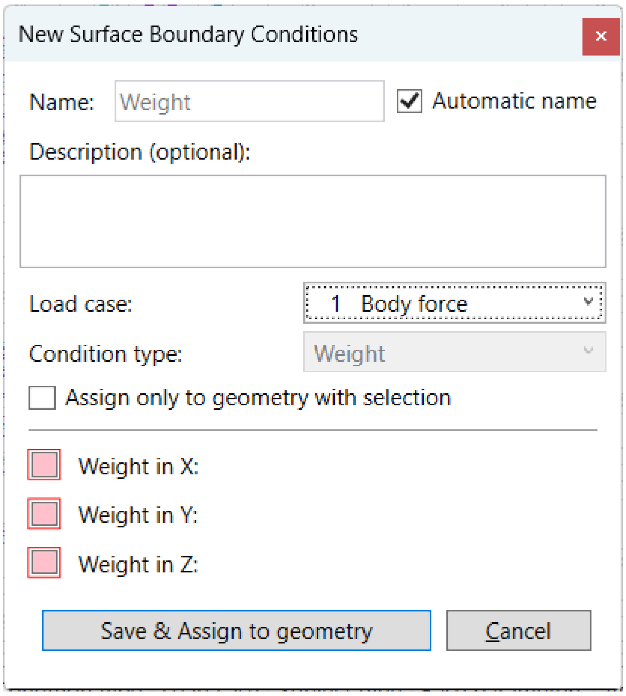


Fig. 3-94: Boundary condition - default.

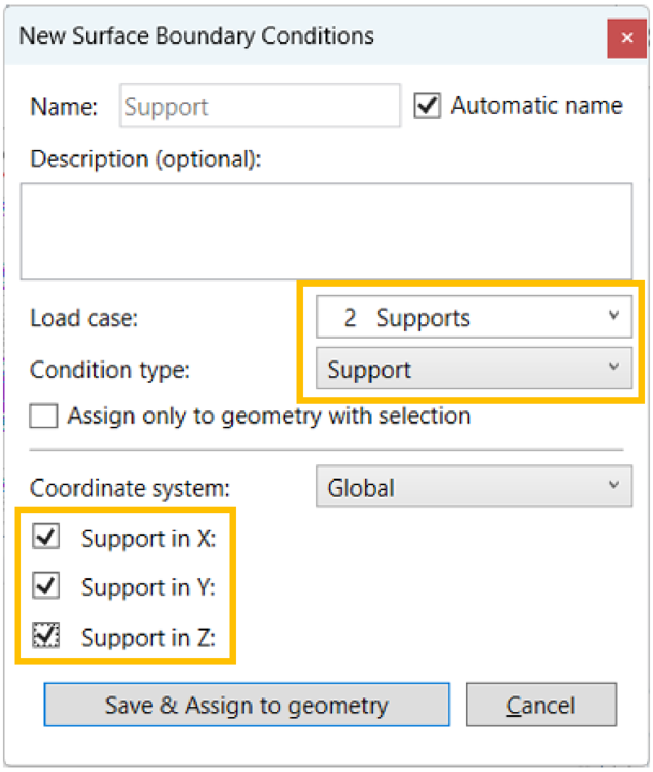


Fig. 3-95: Boundary condition - Support.

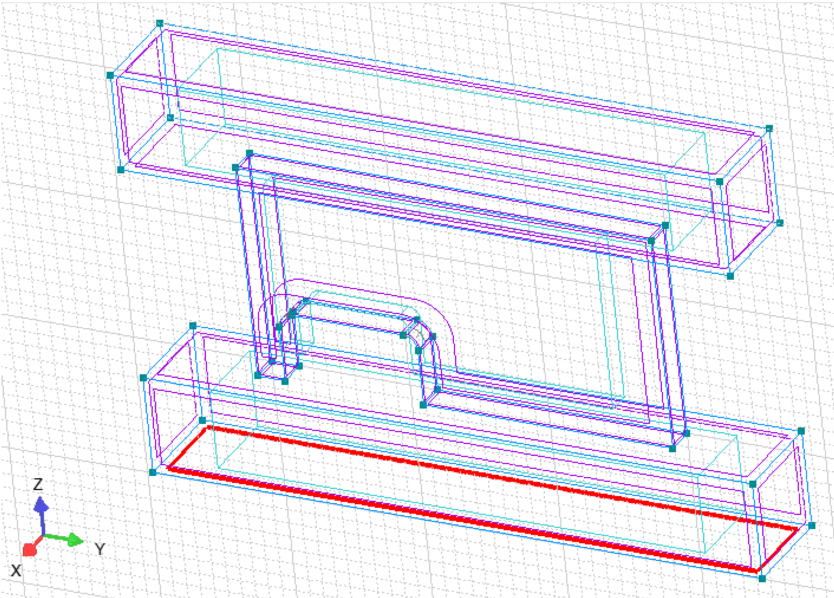


Fig. 3-96: Selected surface for assigning support.

The surface with support is displayed in Workspace view, Fig. 3-97.

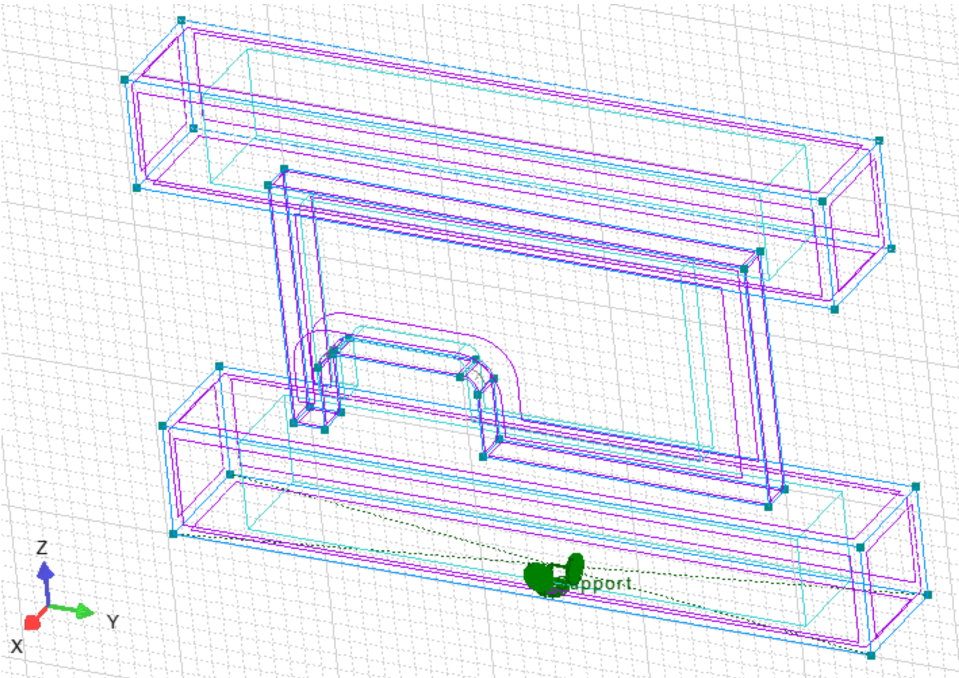


Fig. 3-97: Surface with support.

3.5.2 Fixed contact

Two different types of mesh are assigned to beams and wall respectively which results in two incompatible meshes that are not connected. Hence the beams and wall need to be connected. For this purpose the fixed contact is assigned.

New Surface boundary condition is created with Load case Supports as in previous case but this time the condition type is switched to Fixed contact, Fig. 3-98. The Slave condition type is assigned as first, clicking on Save & Assign to geometry the selection mode is triggered. The three surfaces at the top and bottom of middle layer are selected, Fig. 3-99. Then pressing Enter pops up the dialogue window for Fixed contact with the Master type, Fig. 3-100, so the **Save & Assign to geometry** is pressed once more and the bottom surface of top beam with top surface of bottom beam are selected, see Fig. 3-101, and with **Enter** the whole process of assigning the Fixed contact is concluded, Fig. 3-102.

Parameters input:	
Load case	2 Supports
Condition type	Fixed contact

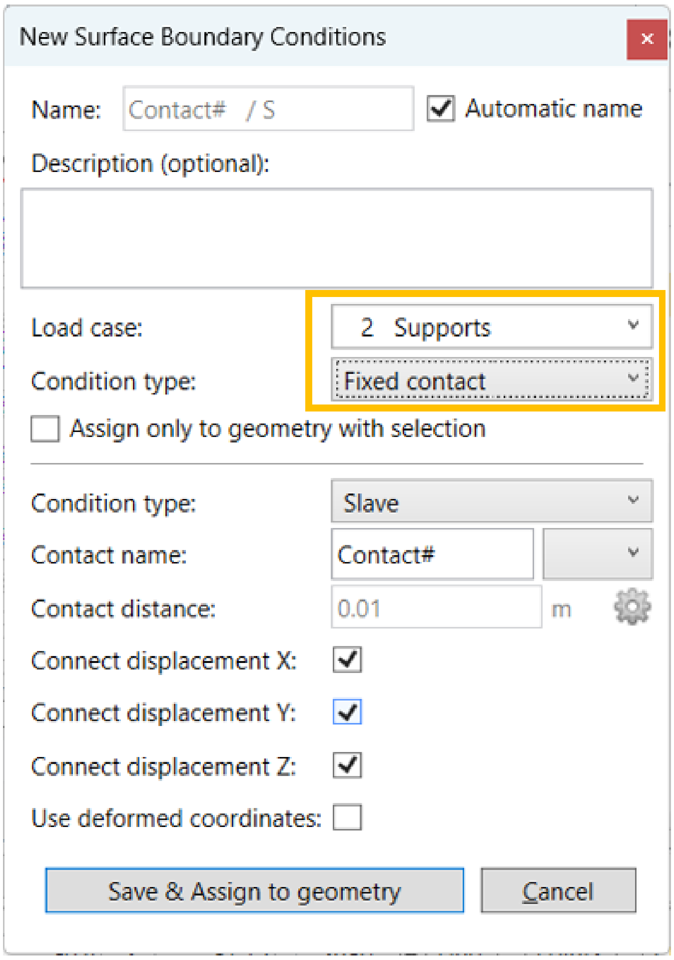


Fig. 3-98: Create new fixed contact - Slave.

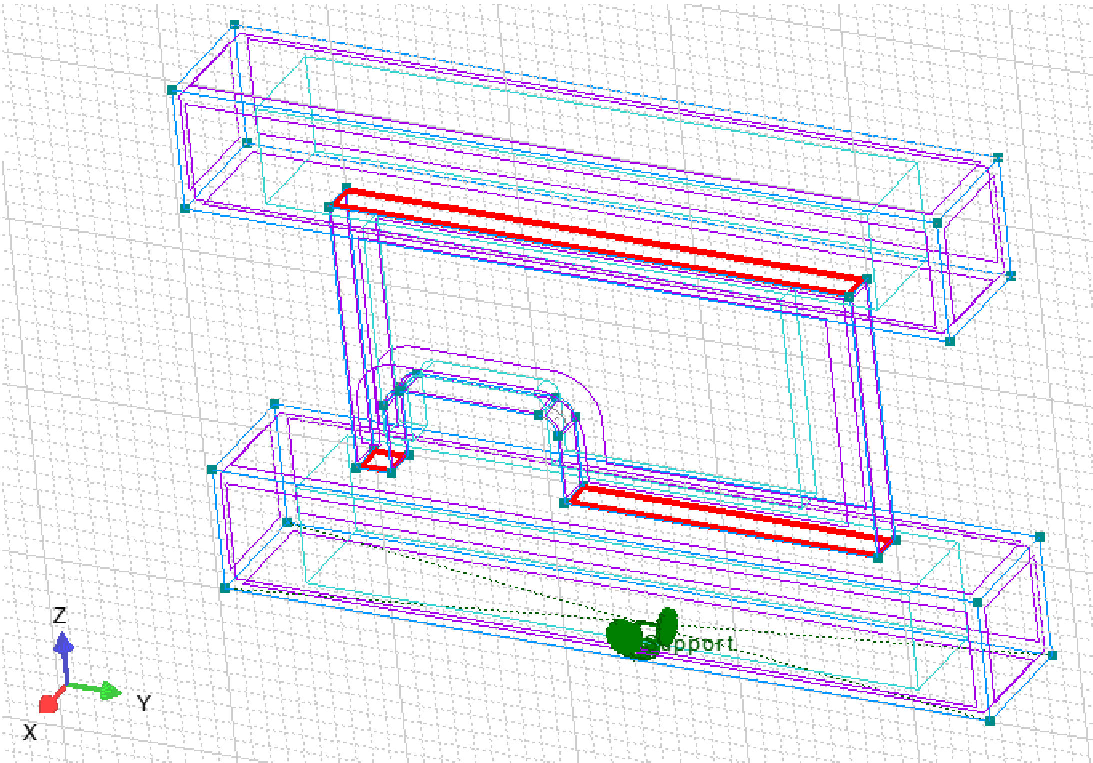


Fig. 3-99: Surfaces selected to assign fixed contact - Slave.

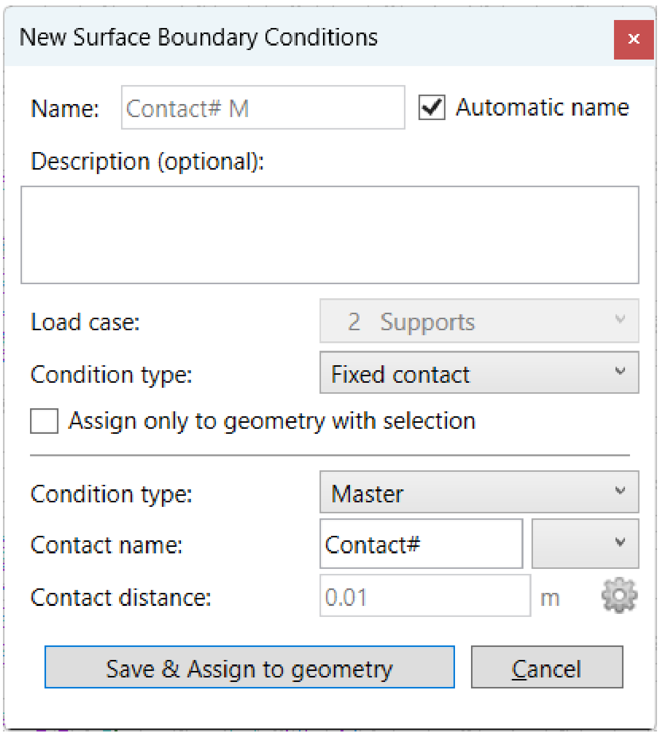


Fig. 3-100: Create new fixed contact - Master.

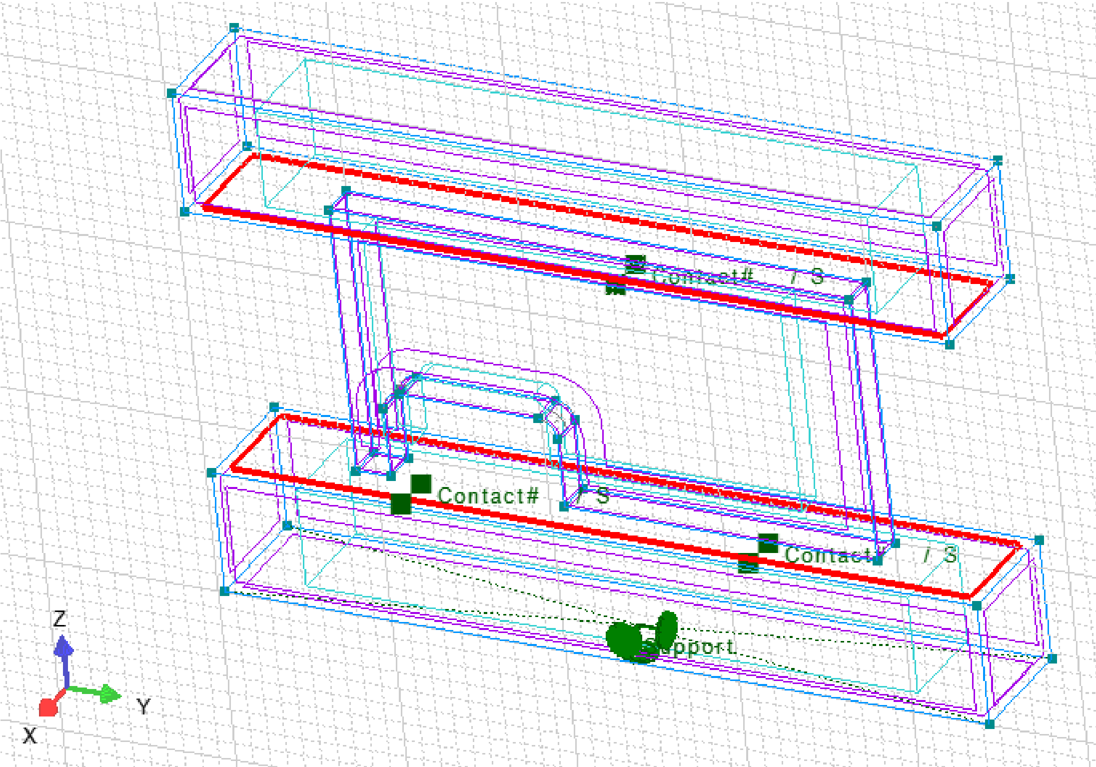


Fig. 3-101: Surfaces selected to assign fixed contact - Master.

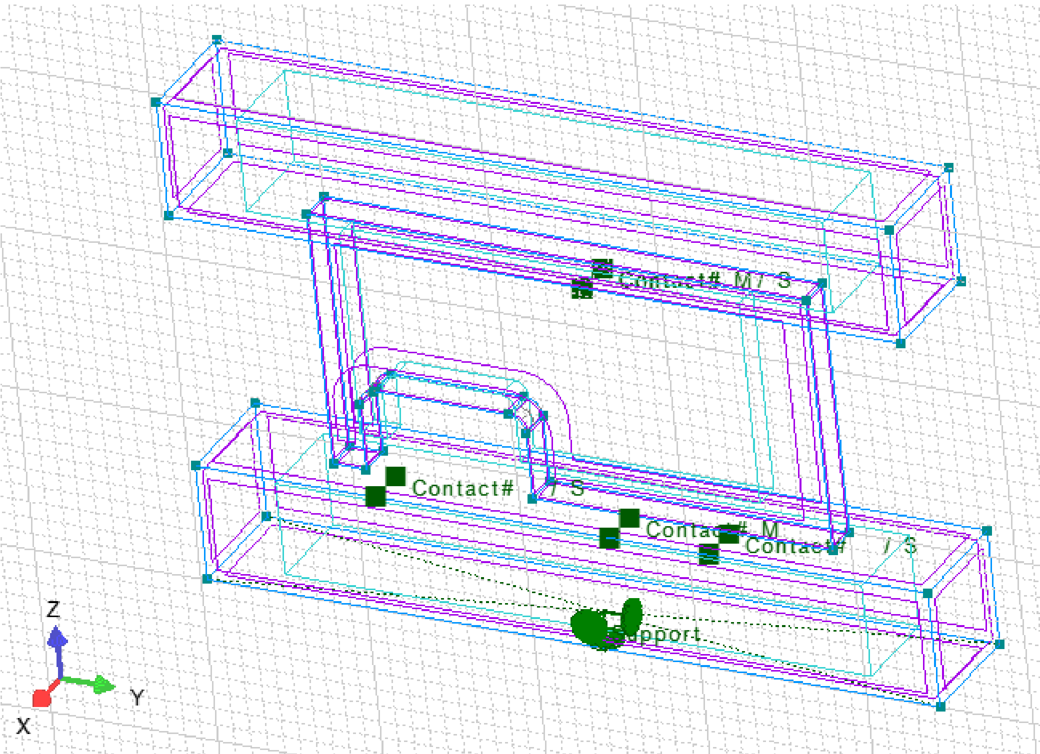


Fig. 3-102: Surfaces with fixed contact.

All the boundary conditions created in this section are listed in table, Fig. 3-103.

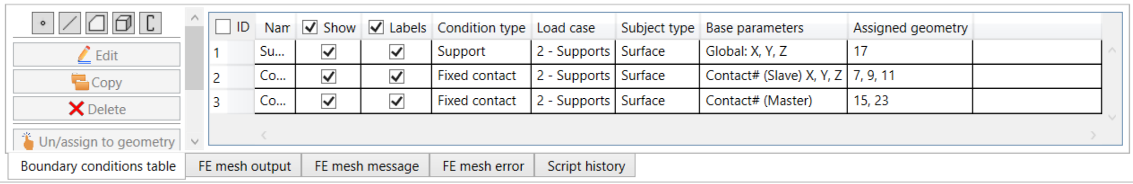


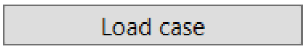
Fig. 3-103: List of boundary conditions in table.

3.6 Load and Monitors

The previous part covered the supporting of the model so the structure can be loaded. In this section the load is prescribed along with monitors in order to survey the behavior of the structure as whole.

3.6.1 Load

The shear wall has bottom beam supported and the top beam is loaded. The beam is loaded by prescribed deformation.

At first the Load case for the prescribed deformation needs to be created since the default load cases cover Body forces, Supports and Forces, not deformation. In **Analysis | Load cases** new load case is created by clicking on Load case button  as shown in Fig. 3-104.

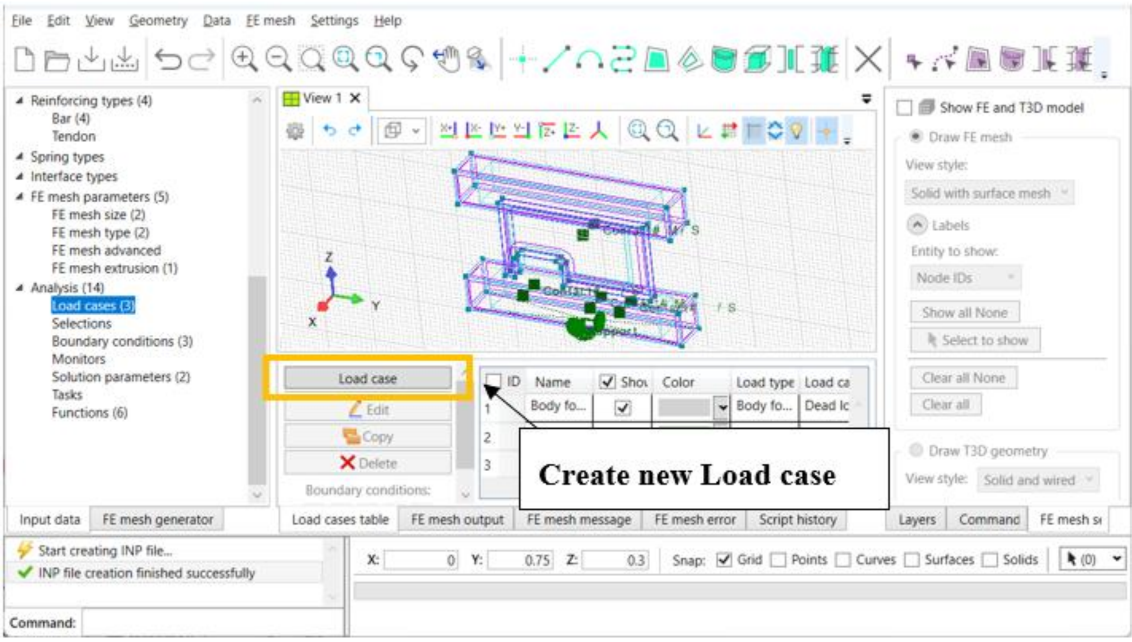


Fig. 3-104: Create new Load case.

New load case dialogue offers various load types, Load category and multiplier, Fig. 3-105. In this case the Load type is changed to Prescribed deformation and saved with button **Save**.

Parameters input:	
Load type	Prescribed deformation

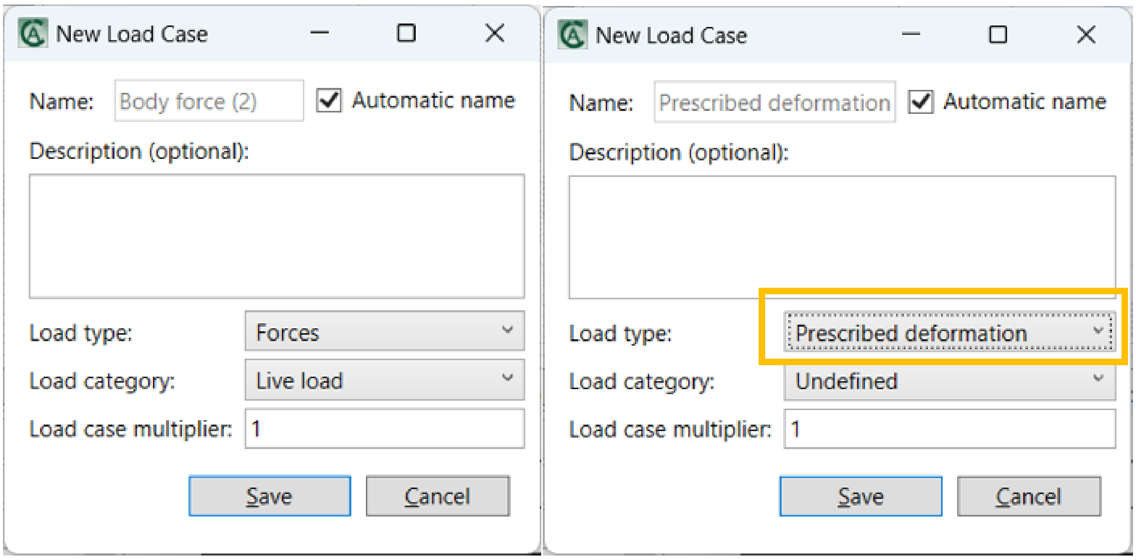


Fig. 3-105: New Load case – Prescribed deformation.

New Load case is listed in the table, Fig. 3-106.

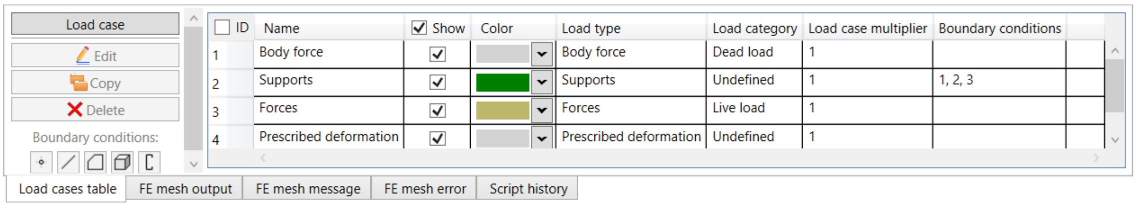


Fig. 3-106: List of Load cases in table.

When the Load case is created the actual load is made in **Analysis | Boundary conditions** as a new BC for curve, see Fig. 3-107.

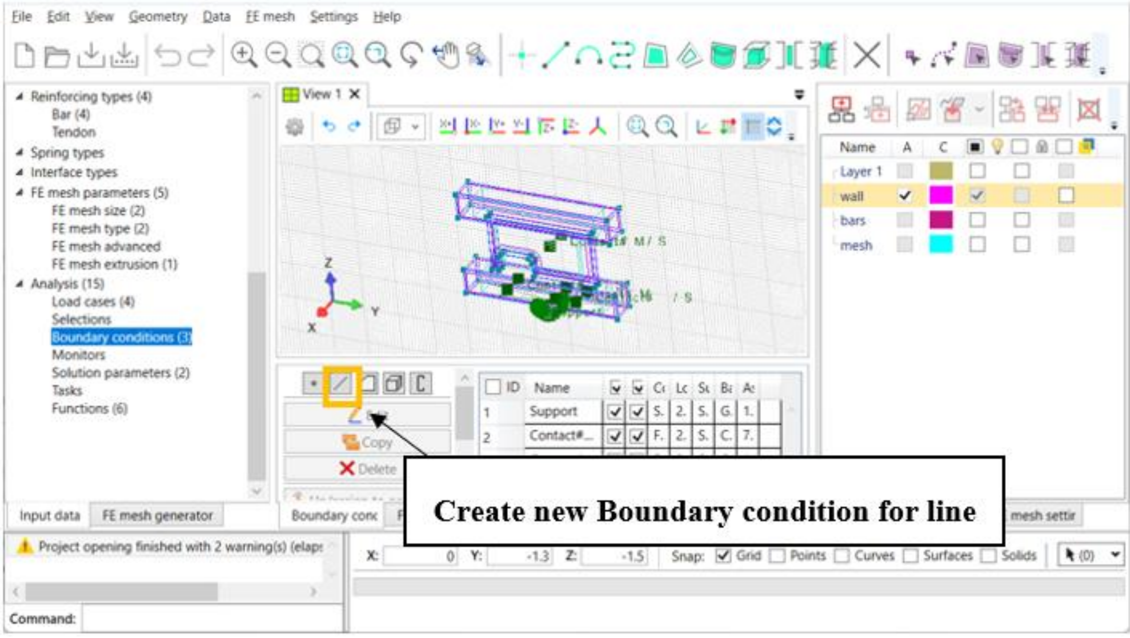


Fig. 3-107: New Boundary condition – Prescribed deformation.

In New boundary condition dialogue select load case Prescribed deformation and set the displacement in Y direction with magnitude -0.1 mm, Fig. 3-108. Button Save & Assign to geometry starts selection mode where the line to be loaded is chosen, see Fig. 3-109.

Parameters input:	
Load case	Prescribed deformation
Coordinate system	Displacement in Y: -0.0001 m (-0.1 mm)

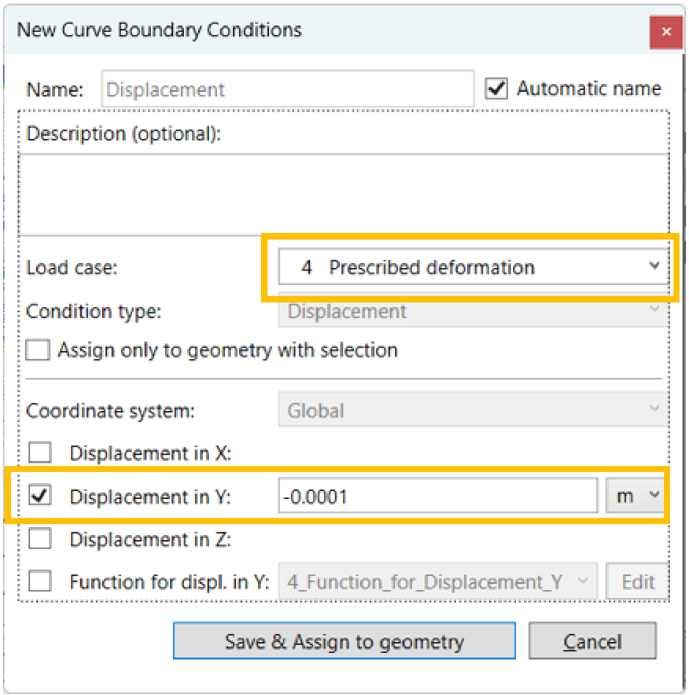


Fig. 3-108: Prescribed deformation properties.

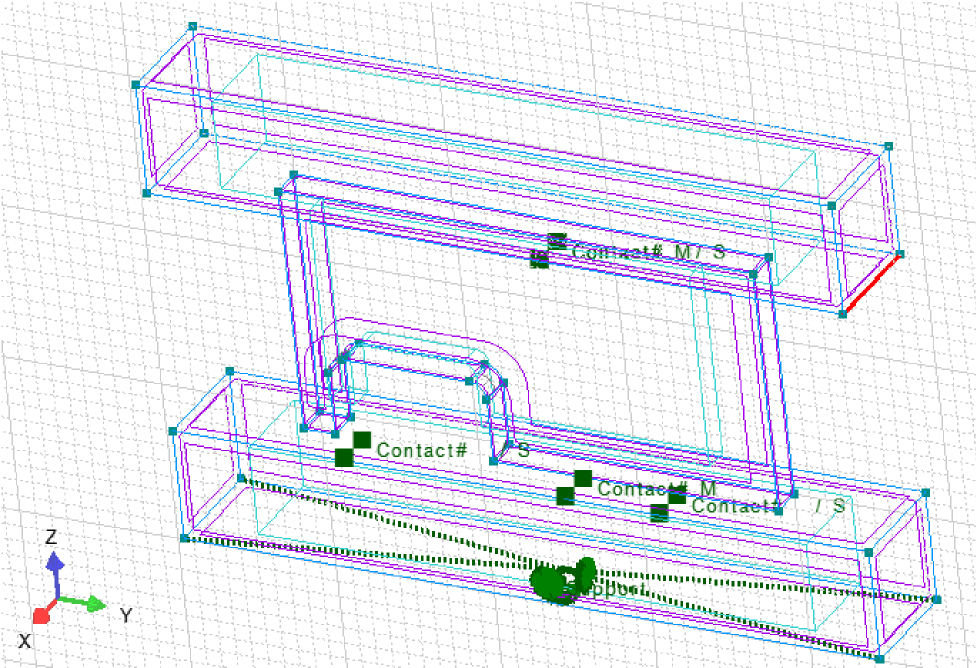


Fig. 3-109: Selected line to be loaded.

The loaded line is displayed in Workspace view with arrows, Fig. 3-110.

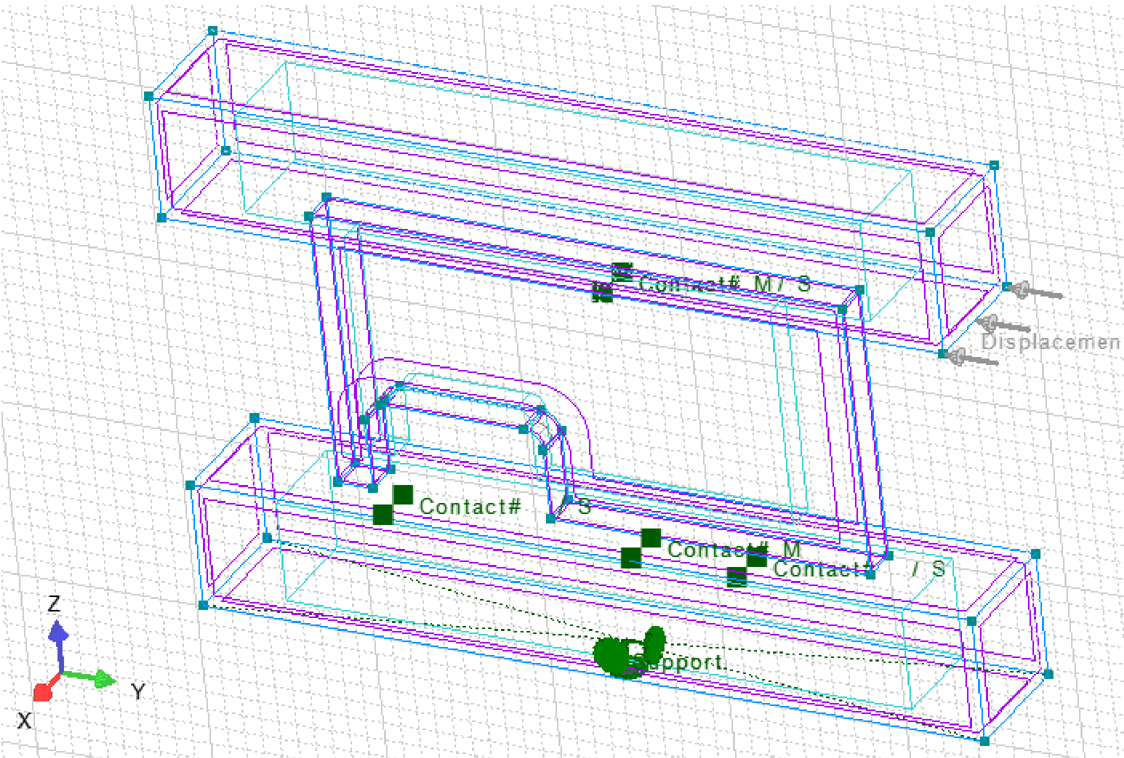


Fig. 3-110: Line loaded by Prescribed deformation.

3.6.2 Monitors

The first monitor assigned to the model is reaction monitor that gathers vertical reaction of the load in support zone. New monitor for Surface is selected in **Analysis | Monitors**, Fig. 3-111.

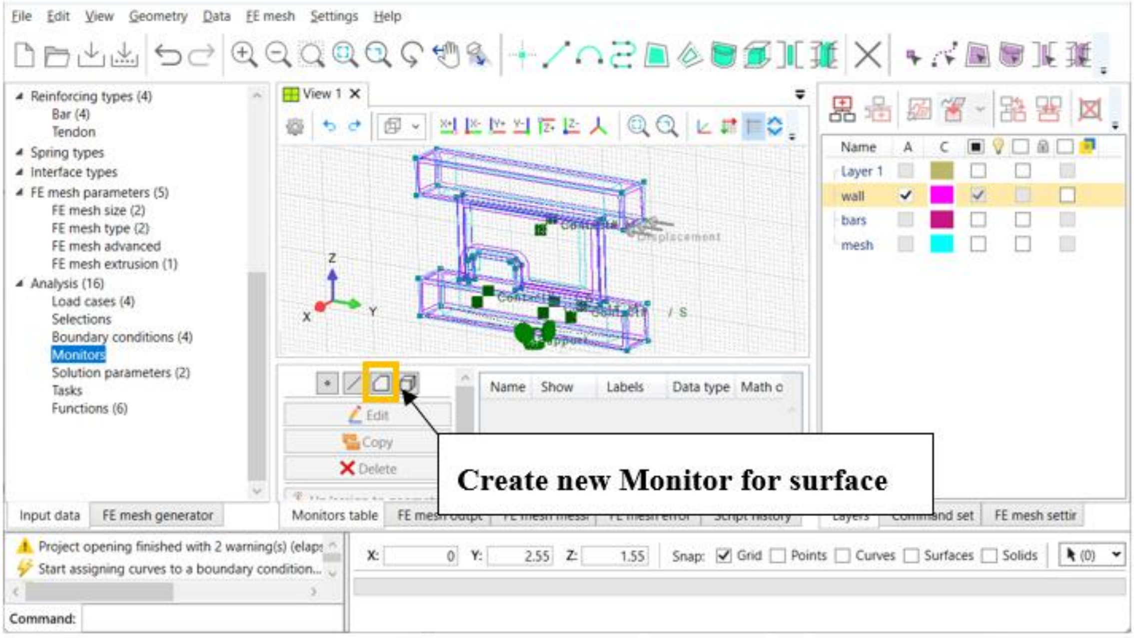


Fig. 3-111: Create new monitor.

New surface monitor parameters are changed to Reactions data type in direction Y and the math operation of this monitor is Summation of node values on the surface, see Fig. 3-112.

Parameters input:	
Name	Reaction_Y_sum
Output data type	Reactions
	Direction Y
Math operation	Summation

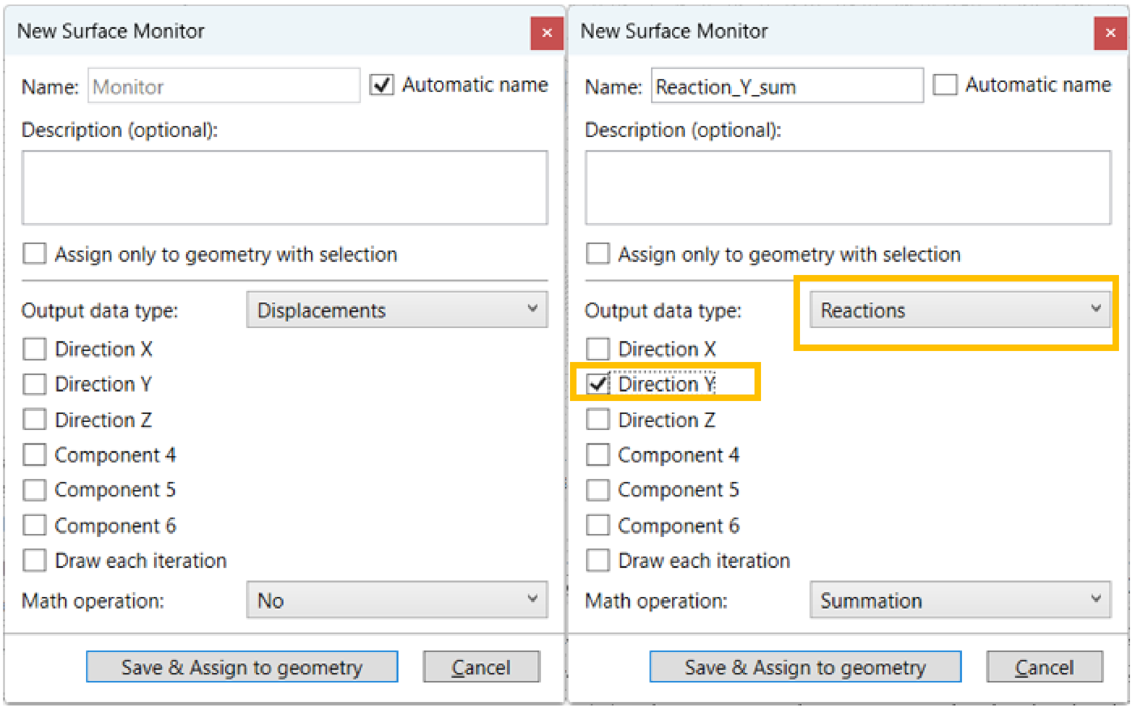


Fig. 3-112: Surface monitor – default and edited.

When the monitor is edited the **Save & Assign to geometry** is pressed and the geometry of the support surface is selected. Fig. 3-113, **Enter** concludes the assignment. And the monitor appears in the table below., Fig. 3-114.

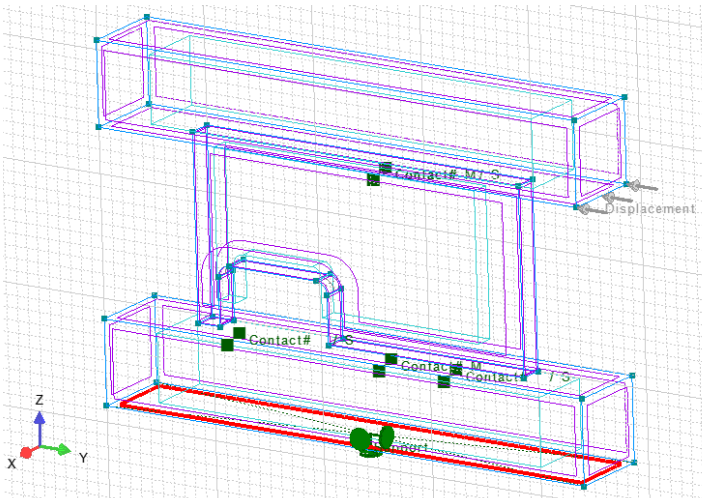


Fig. 3-113: Reaction monitor surface during selection.

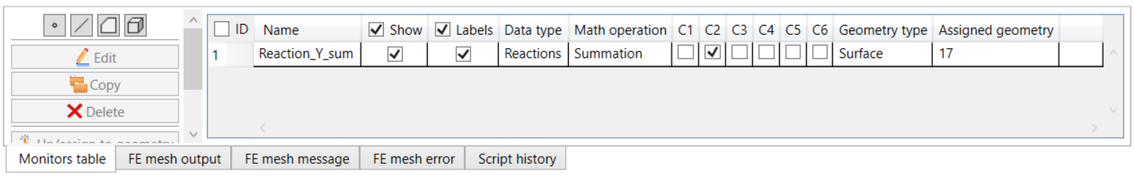


Fig. 3-114: Surface monitor - reaction.

Second monitor is assigned to the line with prescribed displacement loading. Displacement of the line is monitored in direction Y and the value is average of node values on the selected line, Fig. 3-115 and Fig. 3-116.

Parameters input:	
Name	Displacement
Output data type	Displacement
	Direction Y
Math operation	Average

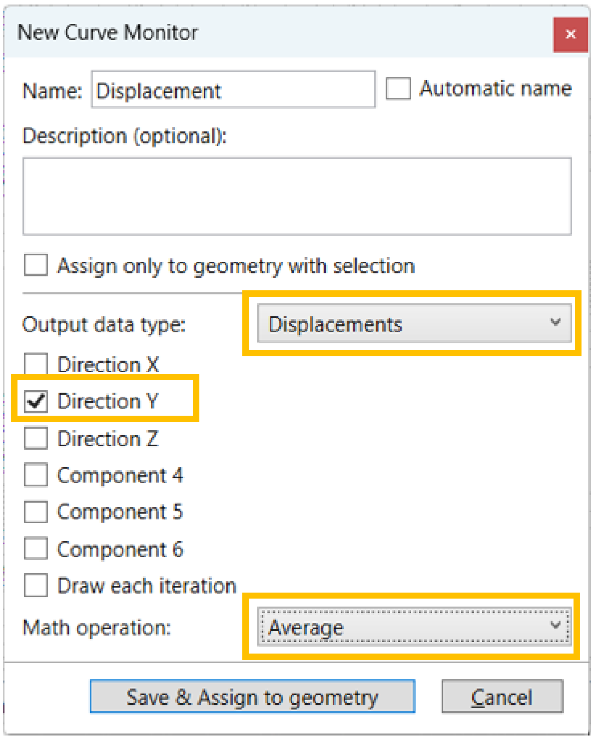


Fig. 3-115: Displacement monitor on line.

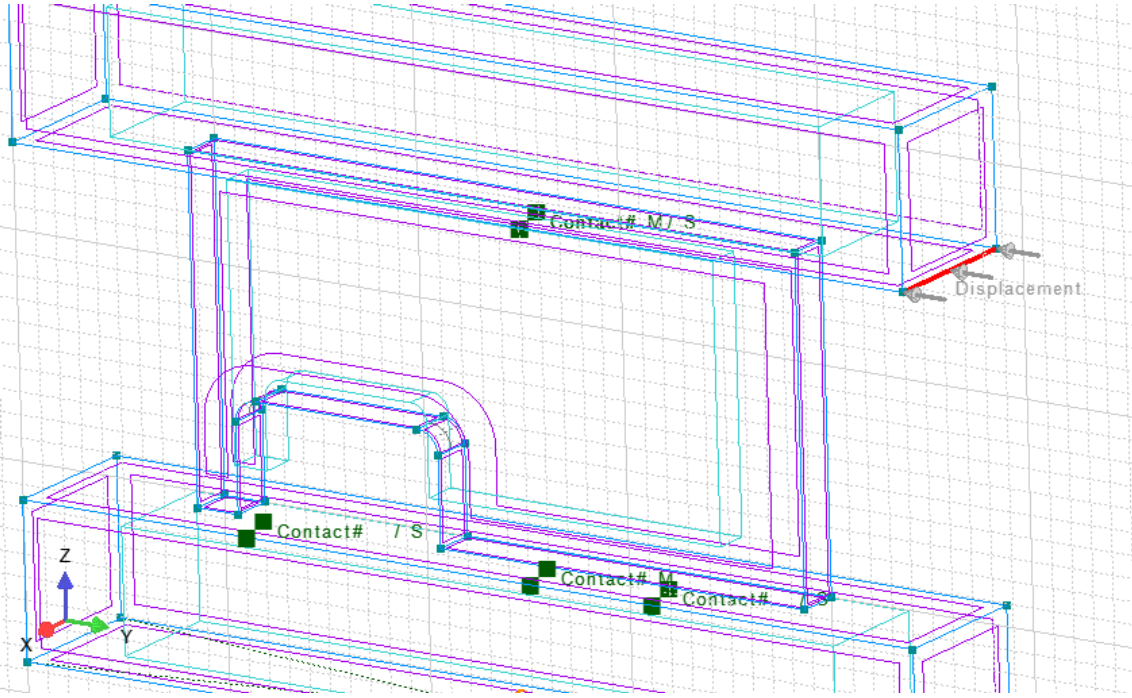


Fig. 3-116: Line selected to assign displacement monitor.

The monitors are listed in table, Fig. 3-117.

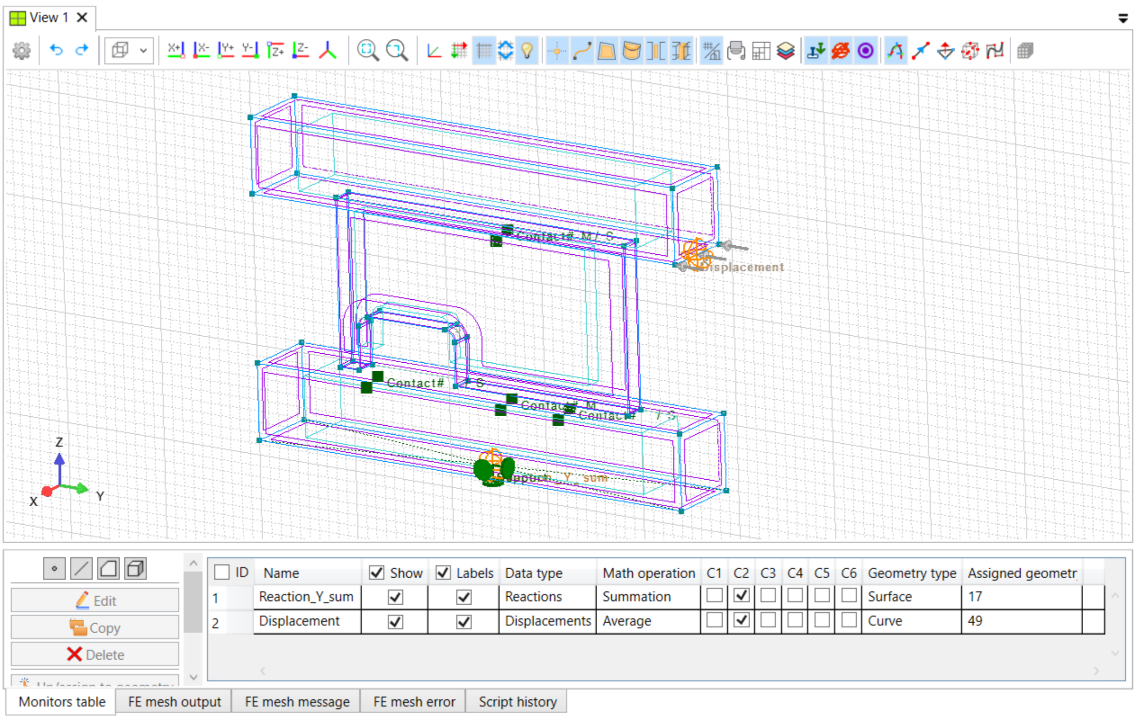


Fig. 3-117: List of monitors in the table.

3.7 Task – Analysis Settings

The nonlinear analysis in ATENA modelled incrementally therefore the history of loading is of essence. The goal of most of analyses is to gradually reach the load level when failure of the structure occurs.

There are two parameters of the gradual loading that are crucial for appropriate analyses results. The first one is final load level that is reached at the end of interval which is comparable to the failure load and is defined by Final load multiplier given from Load case multiplier and Interval multiplier. This final load can be estimated from hand calculation or from the test analysis with few steps and load level to see the development of stresses within the structure. The second parameter is increment of load which is given by final load level and the number of steps (load is divided into number of steps).

In this section the loading history of the Shear wall is described. The loading history is defined in **Analysis | Tasks** (Fig. 3-118). Generally, the different types of load can be divided into different Intervals however the one Interval for different loads can be used too, it all depends on the desired effect of load or combination of loads acting on the structure.

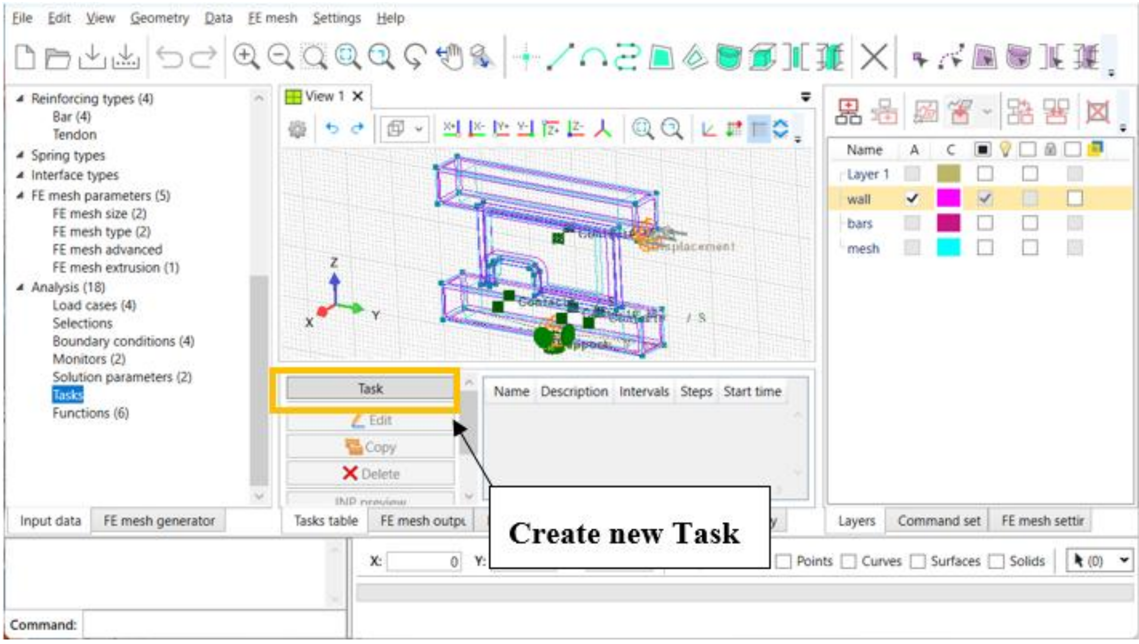


Fig. 3-118: Create new Task with Interval parameters and loading history.

In case of Shear wall only one Interval is used since only one load is applied, see Interval dialogue in Fig. 3-119. Within the interval the load defined in BC section (Prescribed deformation) is multiplied by Interval multiplier and then it is divided into specified number of interval steps. The experimental data show the peak of the load when the displacement of the top reaches about 3 mm. Therefore the Interval multiplier is set to 30. Then the number of steps is set to 200 and only every 10th step is saved in order to save memory, in Fig. 3-120.

Parameters input:	
Number of steps	200
Interval multiplier	30
Saving mode	Every 10th step

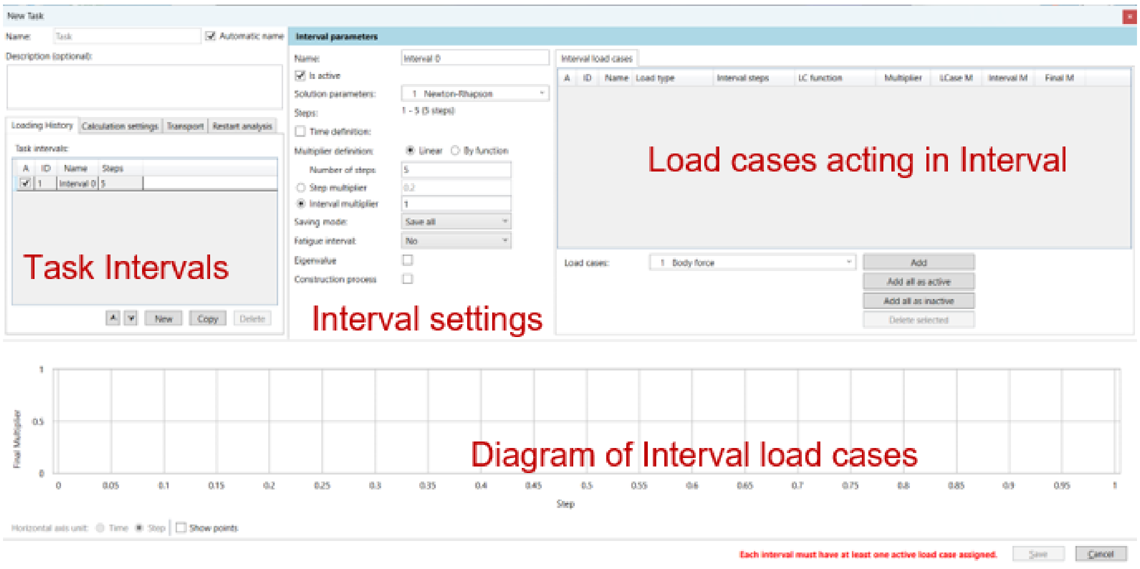


Fig. 3-119: Task dialogue interface.

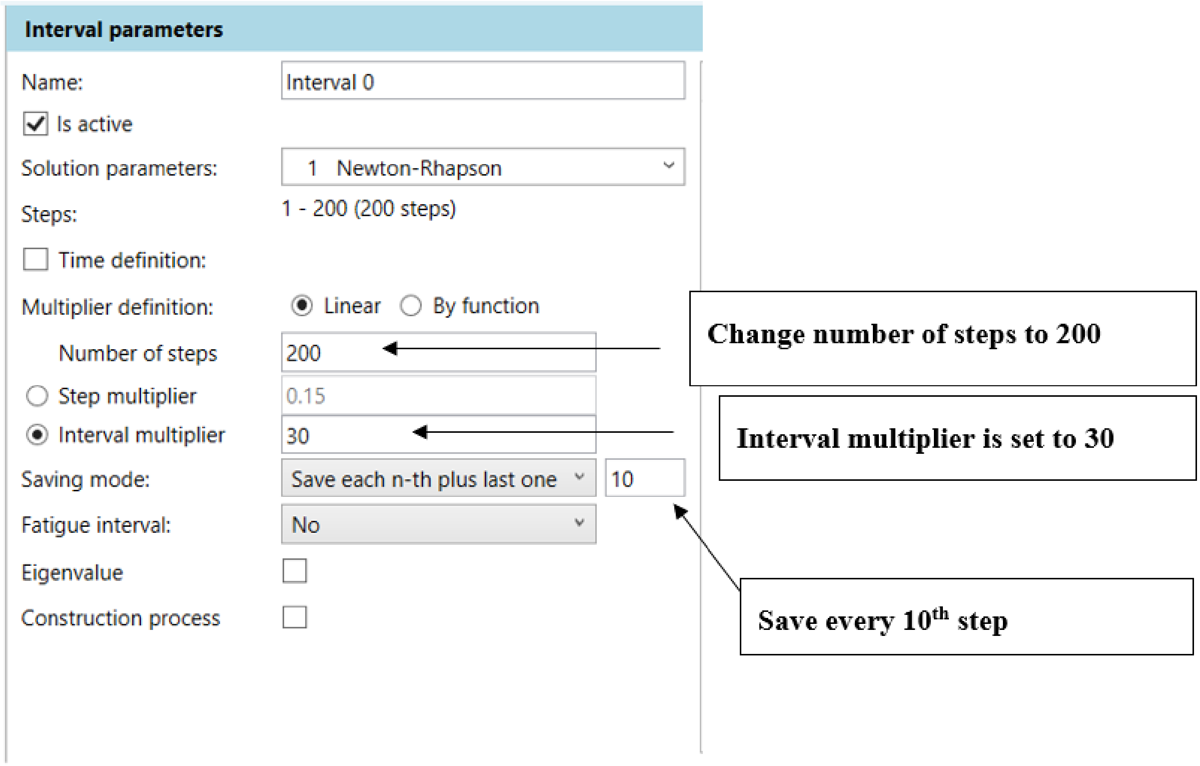


Fig. 3-120: Interval settings.

Also, the Load cases acting during the Interval need to be added into Input parameters. In the Interval load cases window, the Load cases are selected one at a time (Supports and Prescribed deformation) and clicking on **Add** button the load cases is activated in Interval, Fig. 3-121.

Parameters input:	
Load cases	Supports
	Prescribed deformation

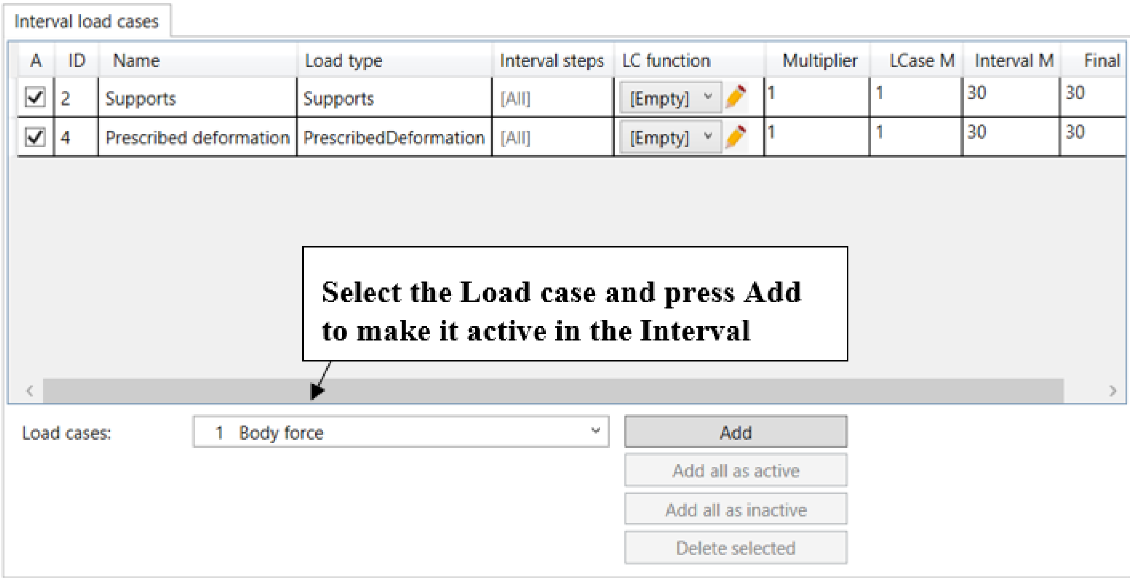


Fig. 3-121: Activation of load cases in Interval.

When the Input parameters are prepared (Fig. 3-122) and the Task is saved then the analysis can be executed by pressing **Run** while the Task is selected. The Atena Studio appears and analysis starts to run, see Fig. 3-123.

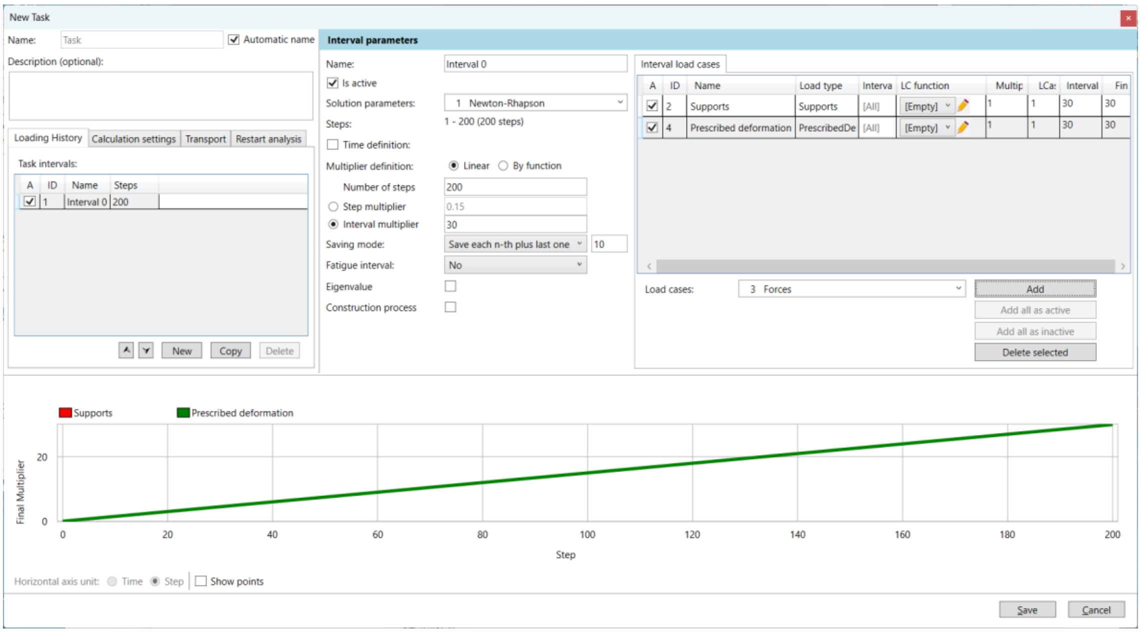


Fig. 3-122: Ready Task to run analysis.

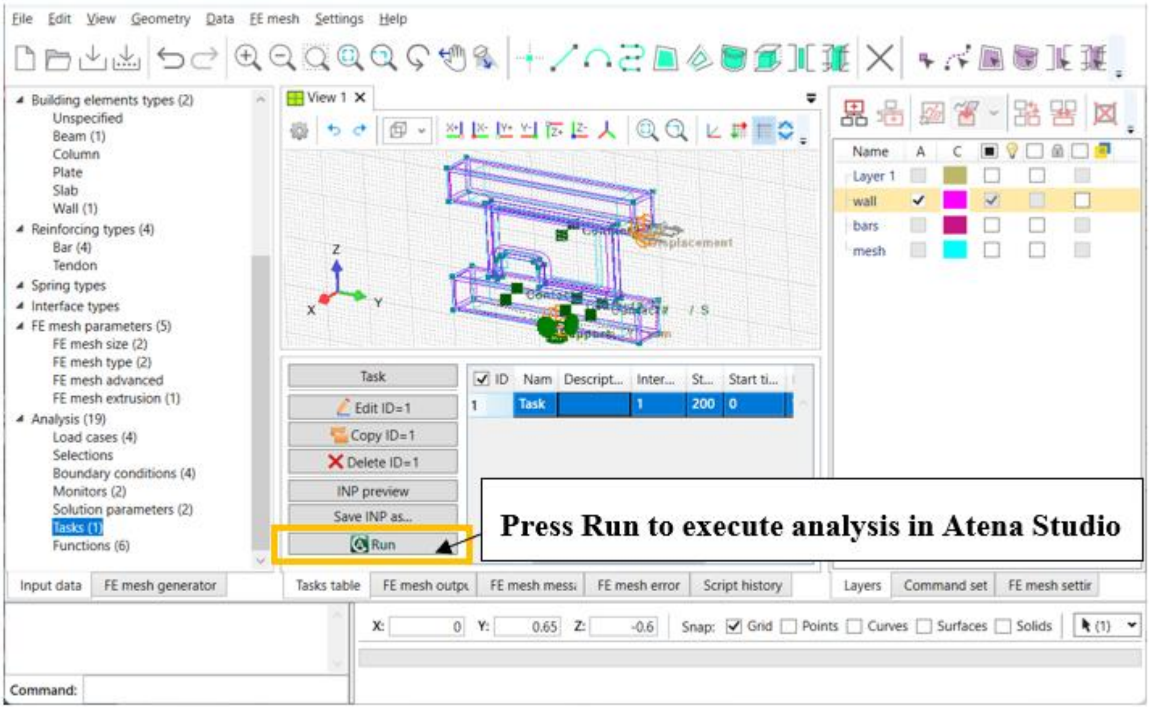


Fig. 3-123: Execution of analysis.

4 FE ANALYSIS

When the analysis is executed in ATENA-PRE by pressing Run, the ATENA Studio appears and runs the simulation, as in Fig. 4-1.

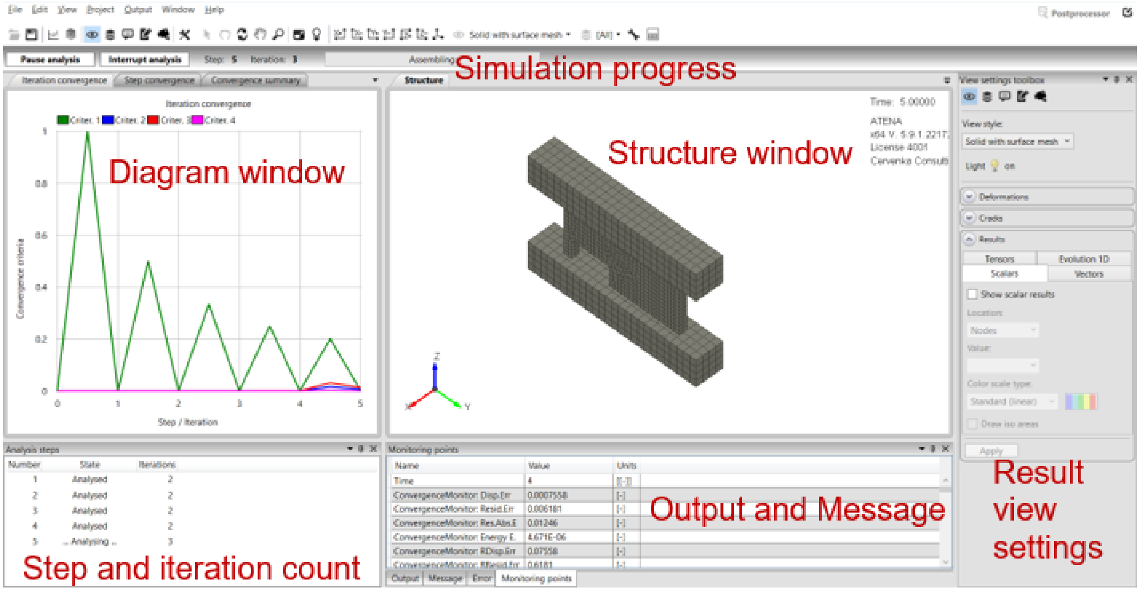


Fig. 4-1: Launching analysis in ATENA Studio.

4.1 L-D diagram

While the simulation is running, user can display L-D diagram to assess the progress of the calculation. There is a **New diagram** button within toolbar menu that open dialogue for setting the monitors to axes on diagram, Fig. 4-2. Then user chooses displacement monitor to label horizontal axis and reaction in case of vertical axis, Fig. 4-3 and Fig. 4-4. When user presses **OK** the new diagram is shown in diagram window.

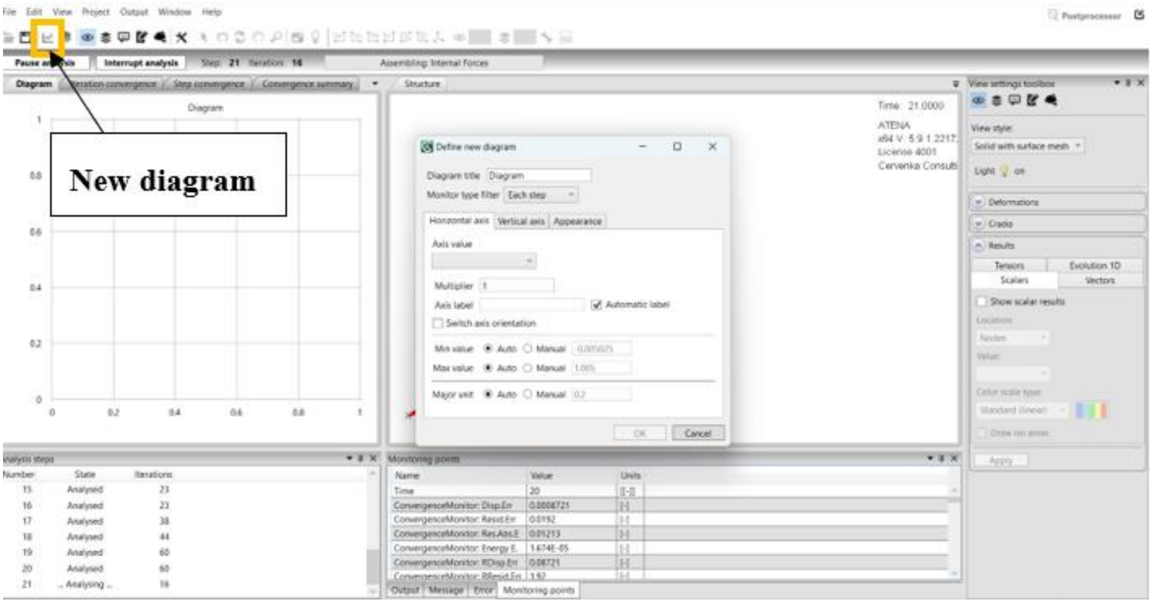


Fig. 4-2: Creating new diagram.

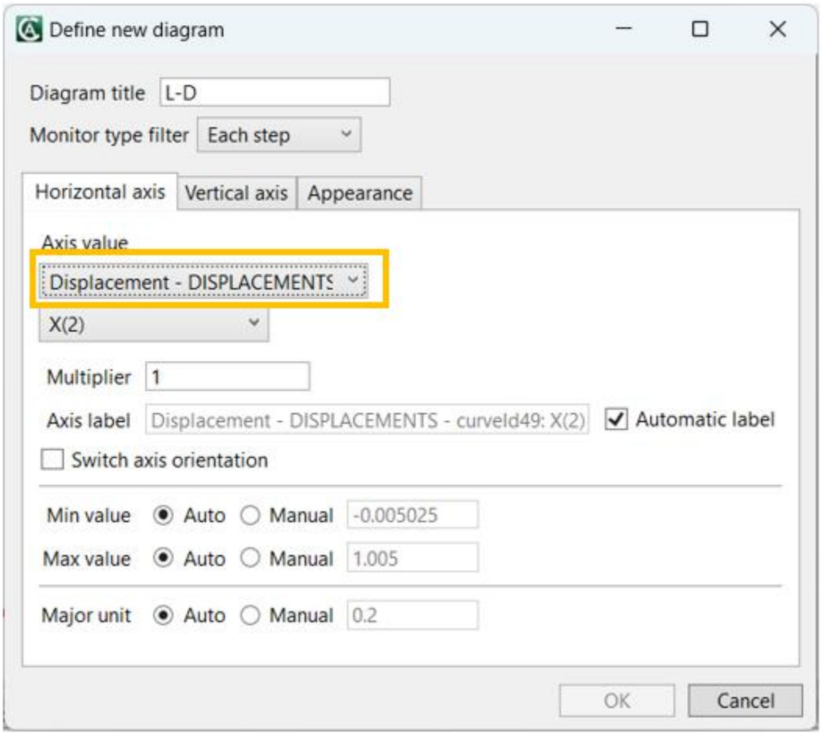


Fig. 4-3: Horizontal axis properties.

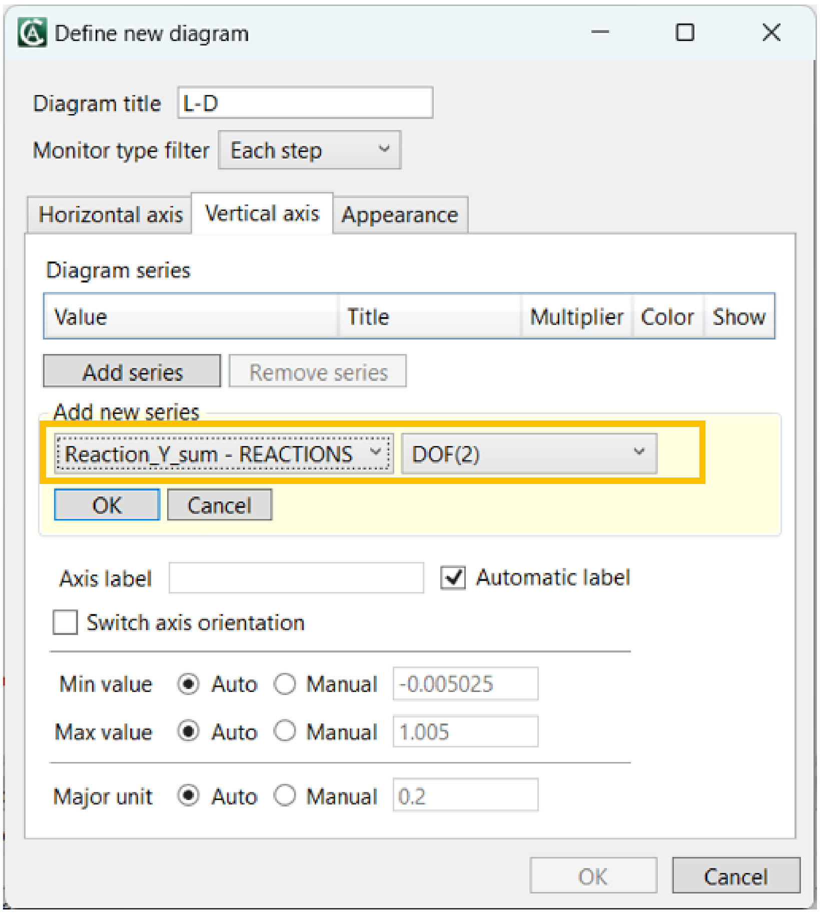


Fig. 4-4: Vertical axis properties.

Since the displacement is negative which absolute value is growing, the diagram is plotted leftwards, Fig. 4-5. Therefore the it is handy to switch the horizontal axis orientation to have the rightwards orientation of the L-D diagram plot.

The settings of the diagram is open by using button **Properties of active document** (Fig. 4-6) and in the tab Horizontal axis the Switch axis orientation checkbox is ticked, Fig. 4-7. Then the L-D diagram gets proper orientation, Fig. 4-8.

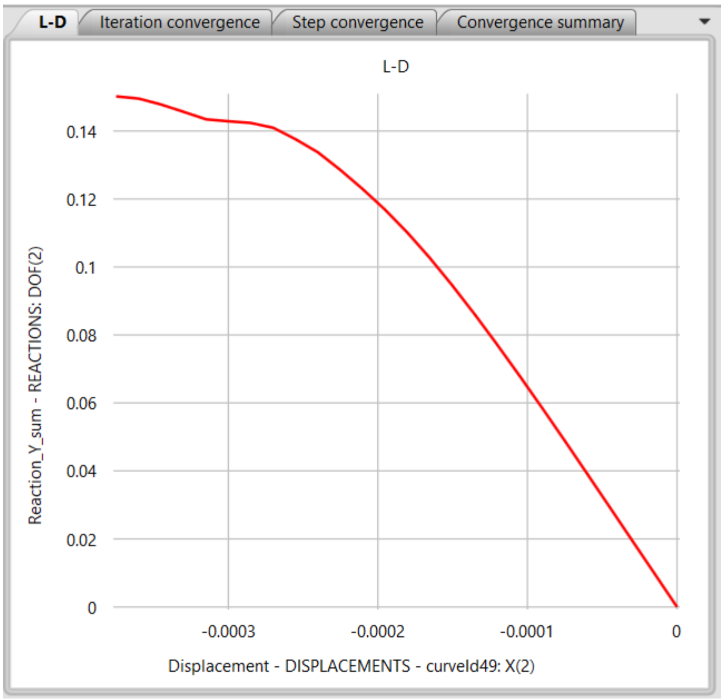


Fig. 4-5: Leftwards oriented diagram.

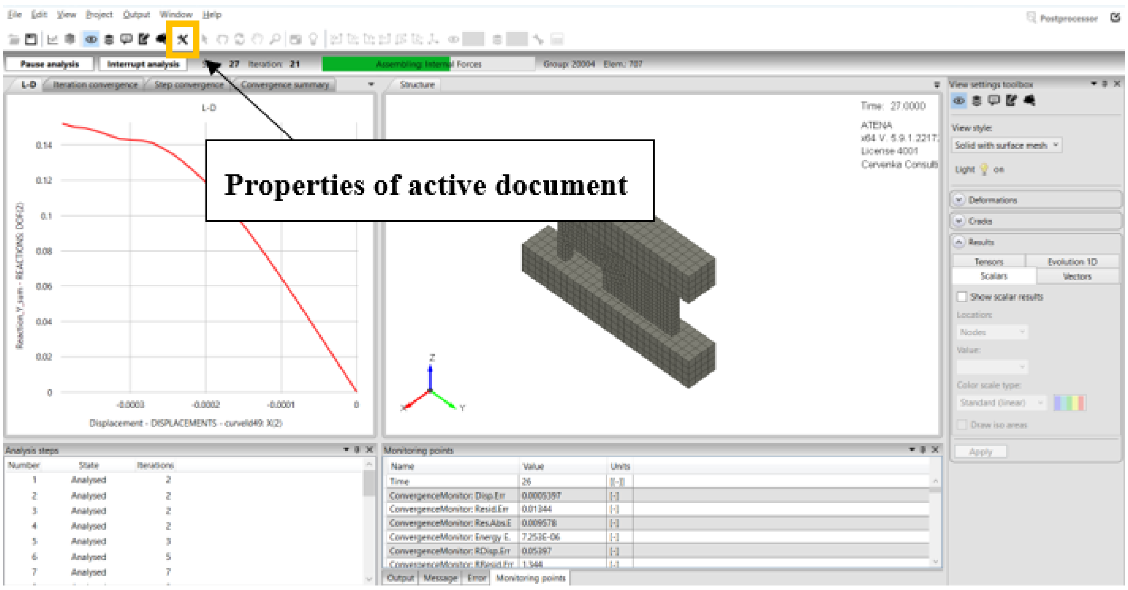


Fig. 4-6: Opening properties of diagram.

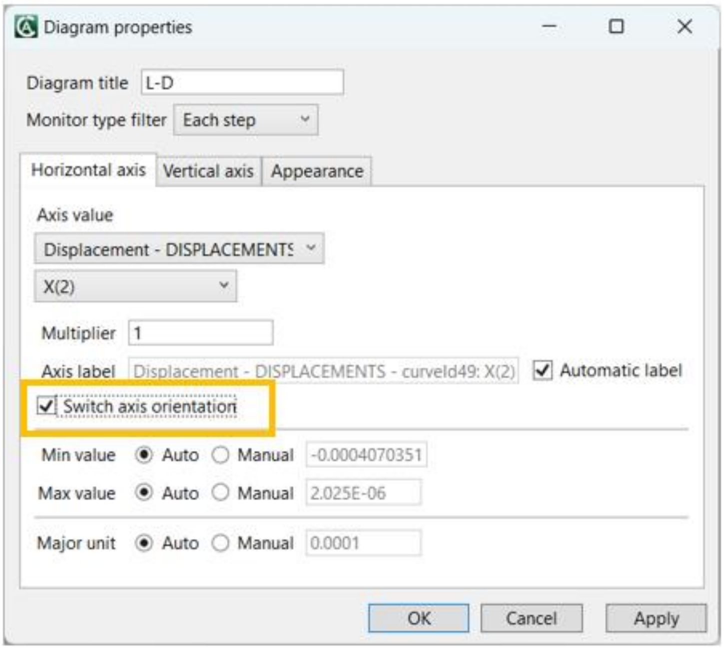


Fig. 4-7: Switching axis orientation.

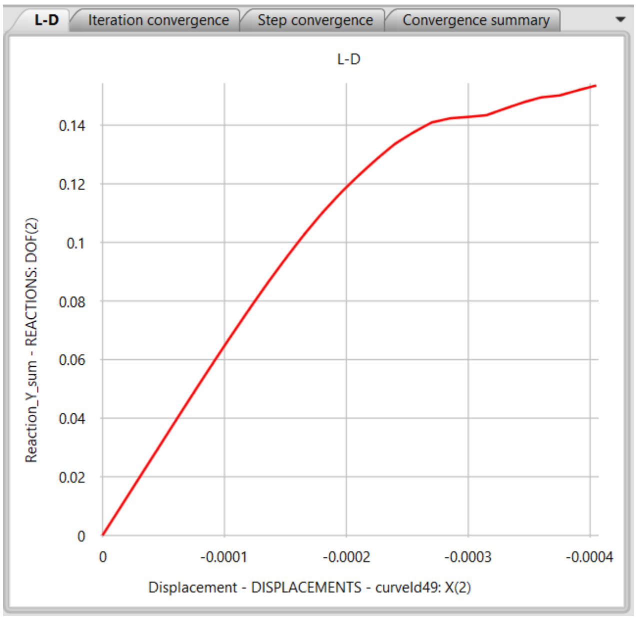


Fig. 4-8: Rightwards oriented diagram.

4.2 Visible cracks

Another handy feature to see the progress of the damage in the structure is crack view which can be showed using Crack settings for visible cracks:

Parameters input:	
View style	Solid with outline
Show cracks	Yes
Max crack level	1
Max crack width	0.0001

The structure view may be adjusted using Shift+Left mouse button, see Fig. 4-9.

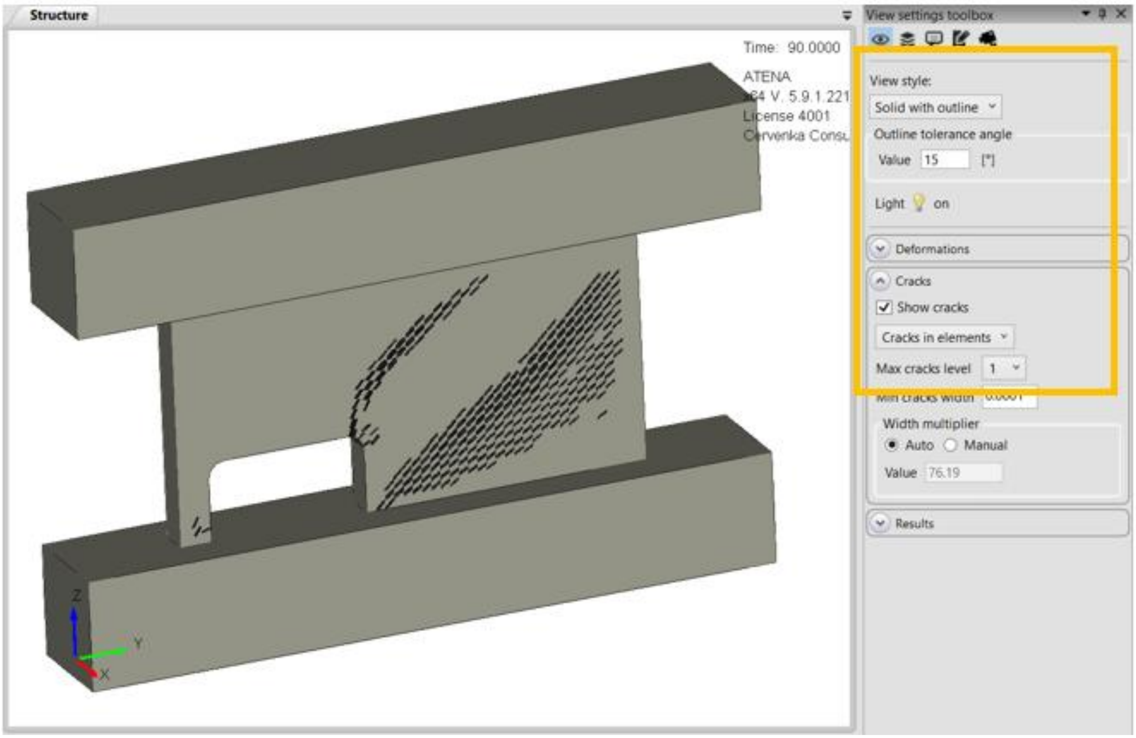


Fig. 4-9: Visible cracks properties.

Below Crack settings is Result setting where scalar results might be displayed while the simulation is running, Fig. 4-10.

Parameters input:	
Show scalar results	Yes
Value	Crack width
Item	Cod1
Draw iso areas	Yes

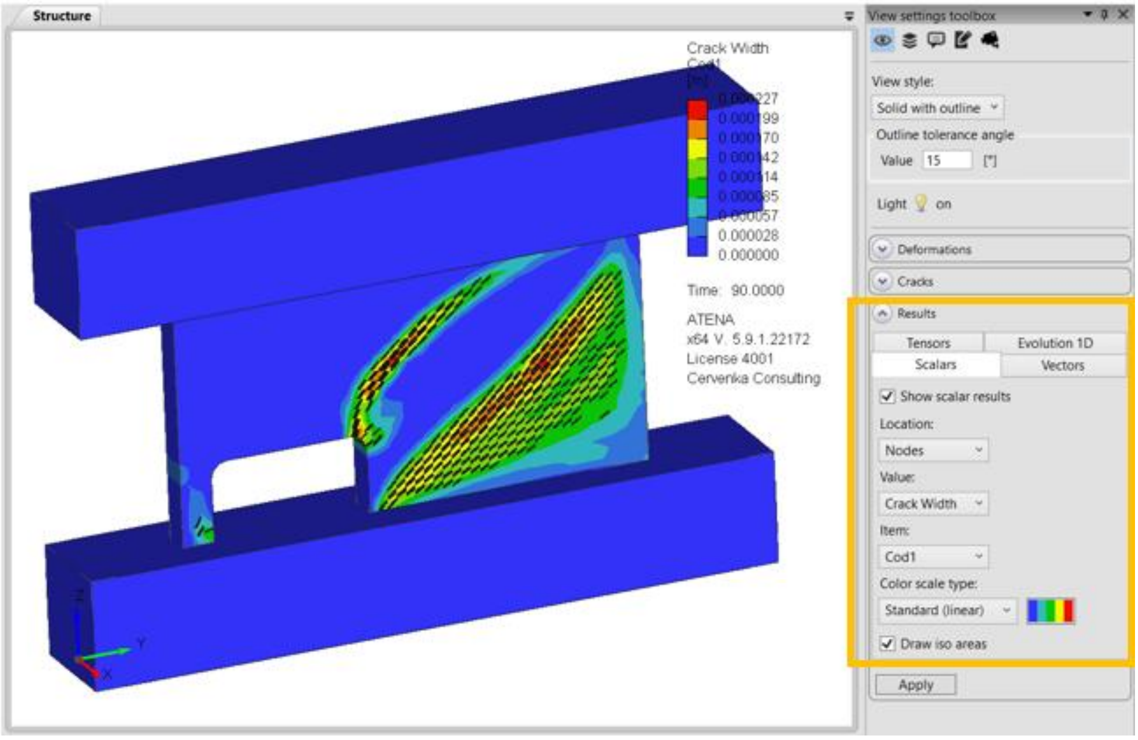


Fig. 4-10: Scalar results settings.

Since the solid elements are non-transparent, the cracks are visible only on the surface and also the reinforcement is hidden. That might be remedied by switching on transparency in **Properties of active document** as shown in Fig. 4-11 resulting in transparent structure in Fig. 4-12.

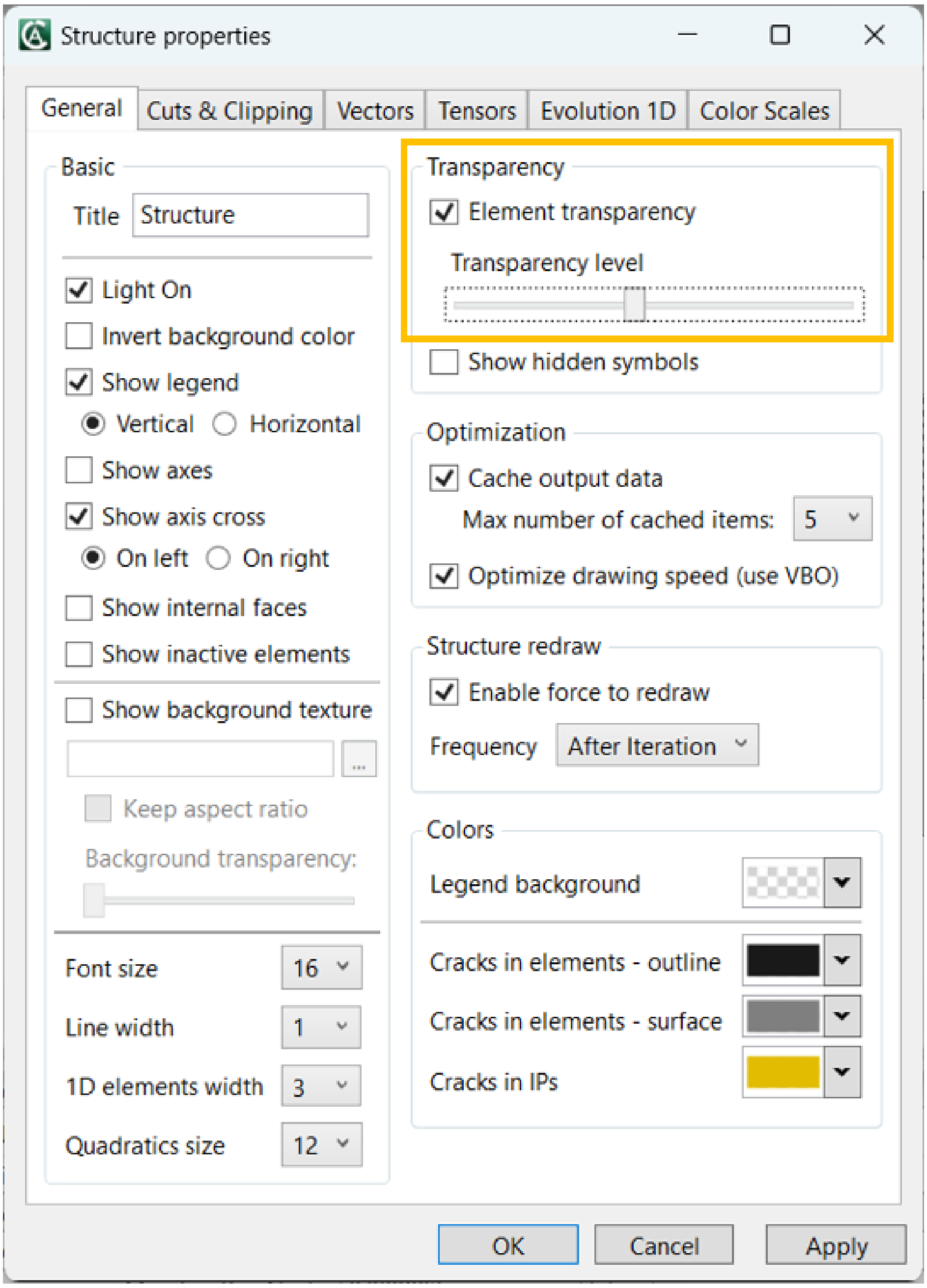


Fig. 4-11: Transparency settings.

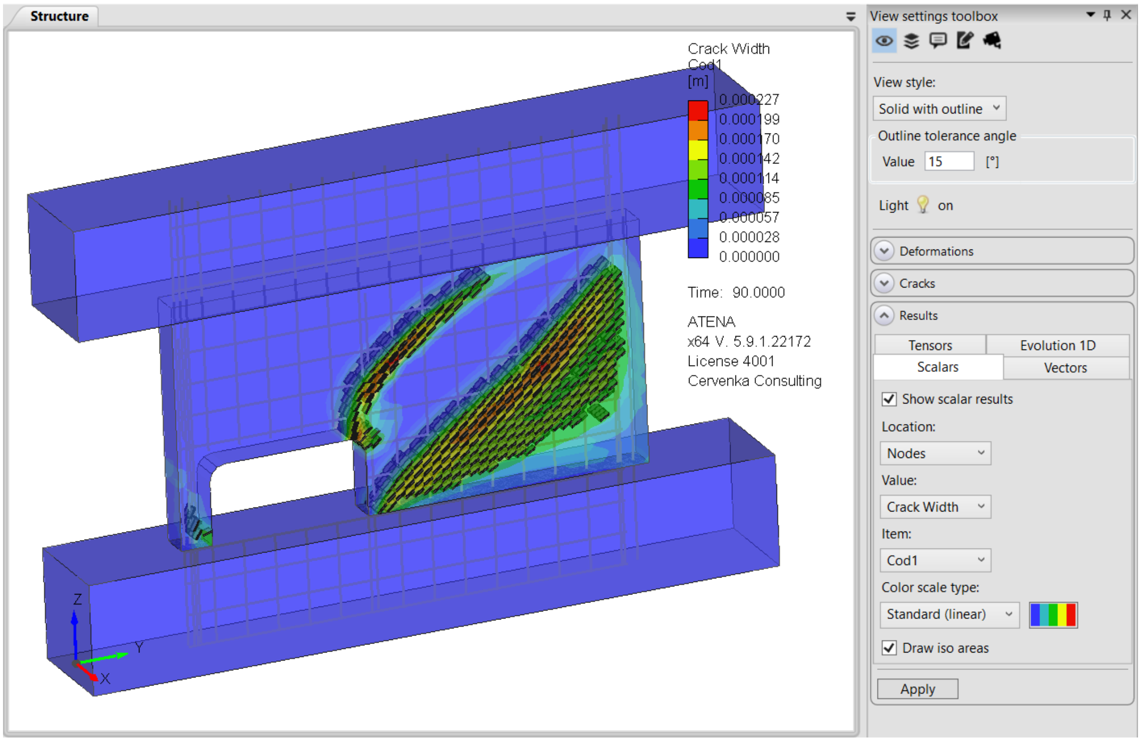


Fig. 4-12: Launching analysis in ATENA Studio.

5 POSTPROCESSING

Once analysis is completed, the user switches from Runtime to Postprocessing when clicking on button **Postprocessor** . Then the Progress bar disappears and it is possible to switch to results in any of the saved steps, see Fig. 5-1.

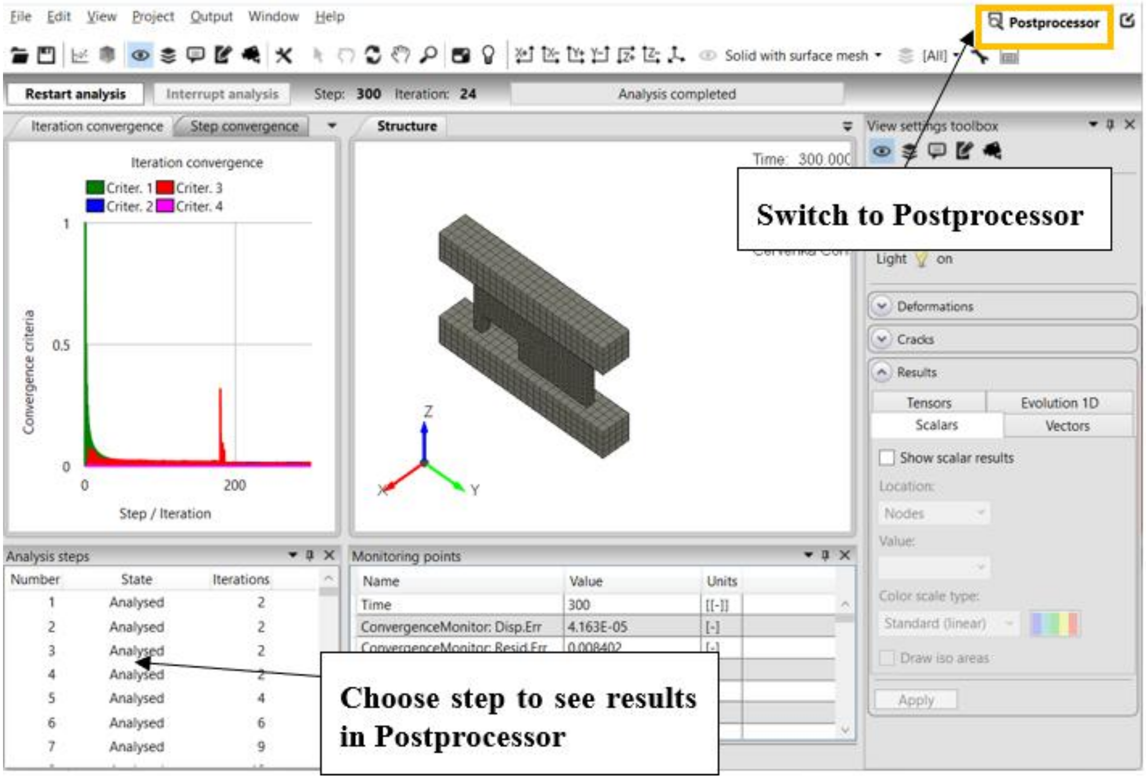


Fig. 5-1: Switching Runtime to Postprocessor in ATENA Studio.

5.1 Activities

Useful tool to display only part of the structure to show behavior in somehewere particular is Activity in **Visible domain**. Default Activities available to choose from are 1D and 3D. Otherwise it is possible to make another Activity in **Edit Activities** to create desired combination of structure parts. Just to show the advantage of Activities, Current Activity is switched to 1D and then the stresses can be viewed on the reinforcement directly, Fig. 5-2.

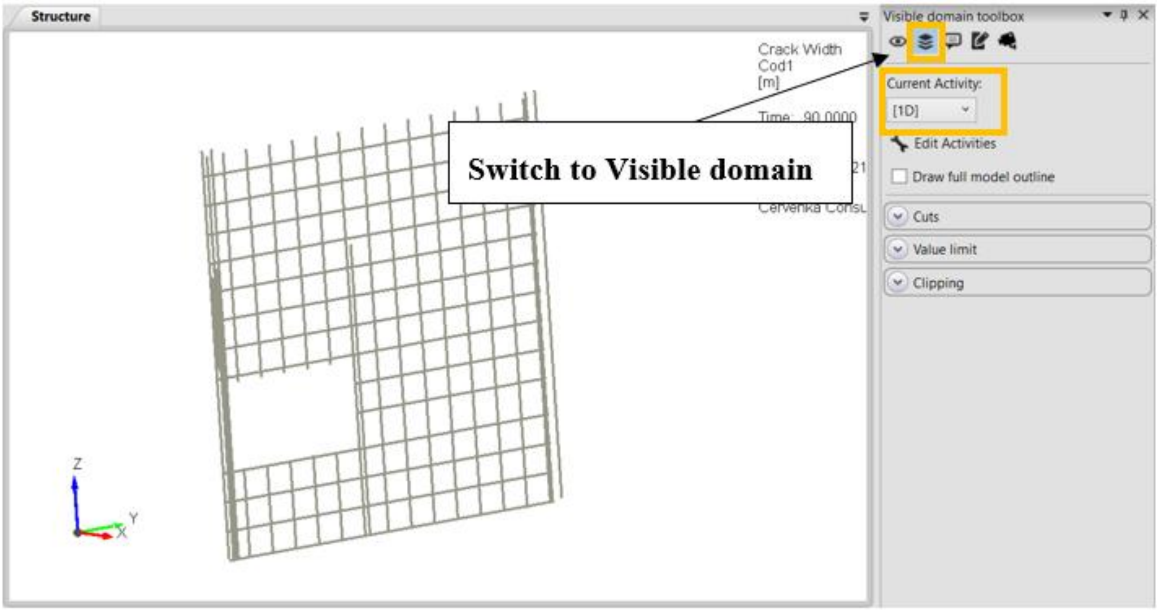


Fig. 5-2: Activities settings.

Back in the View settings, the Scalar results are shown when values are selected. It the example the Stress values in X direction which is direction of bars' longitudinal axis, see Fig. 5-3.

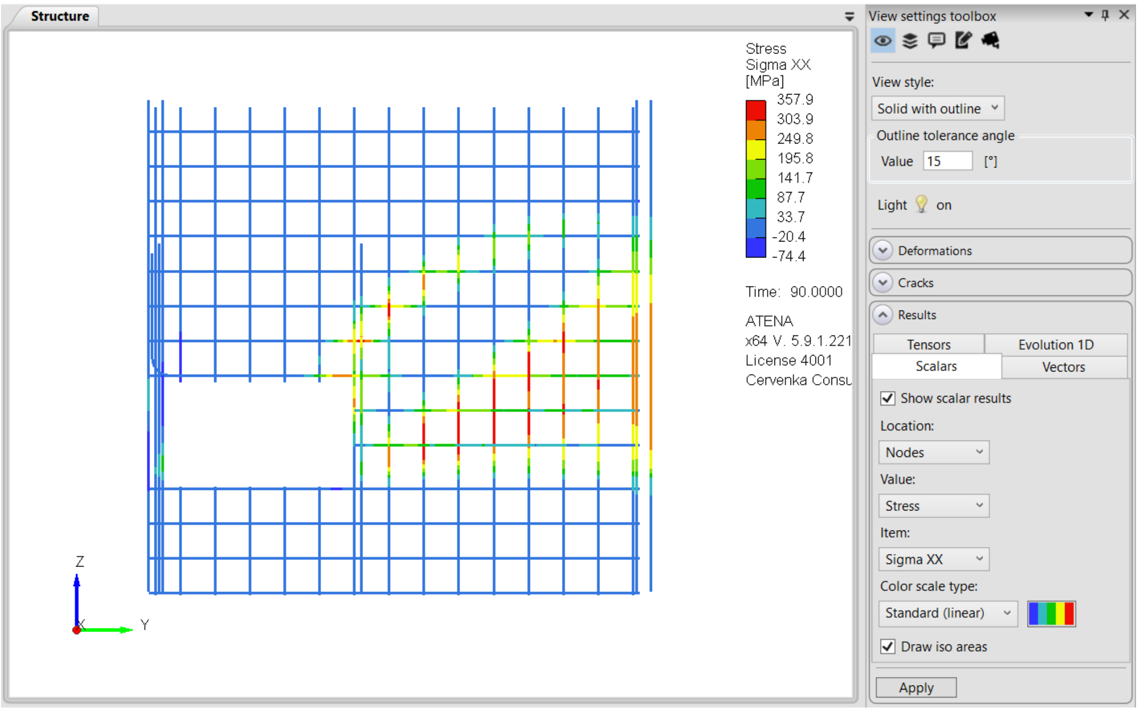


Fig. 5-3: Axial stresses in reinforcement.

5.2 Copy diagram to clipboard

L-D diagram shown in the diagram window can be easily transformed into .csv or .txt file. The mouse cursor has to be located within the diagram window then by left button click the window is activated and simple clipboard shortcut Ctrl + C is used to store the diagram data, Copy to clipboard dialogue appears aby pressing Ok the data are stored, see Fig. 5-4. Then the software such as simple Notepad or Excel can be opened and the data inserted by using shortcut Ctrl + V.

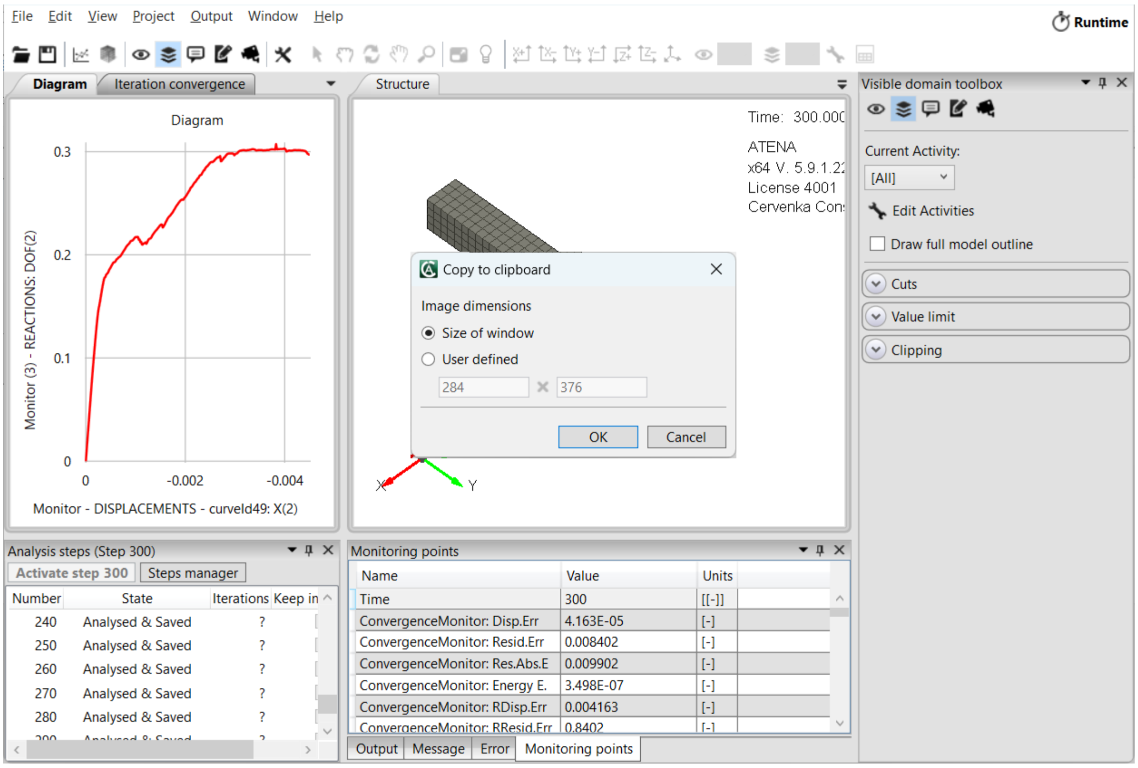


Fig. 5-4: Copy data to clipboard.

The results of the simulation are then compared to experimental data as shown in Fig. 5-5 where the results of analysis correlate with experiment very well.

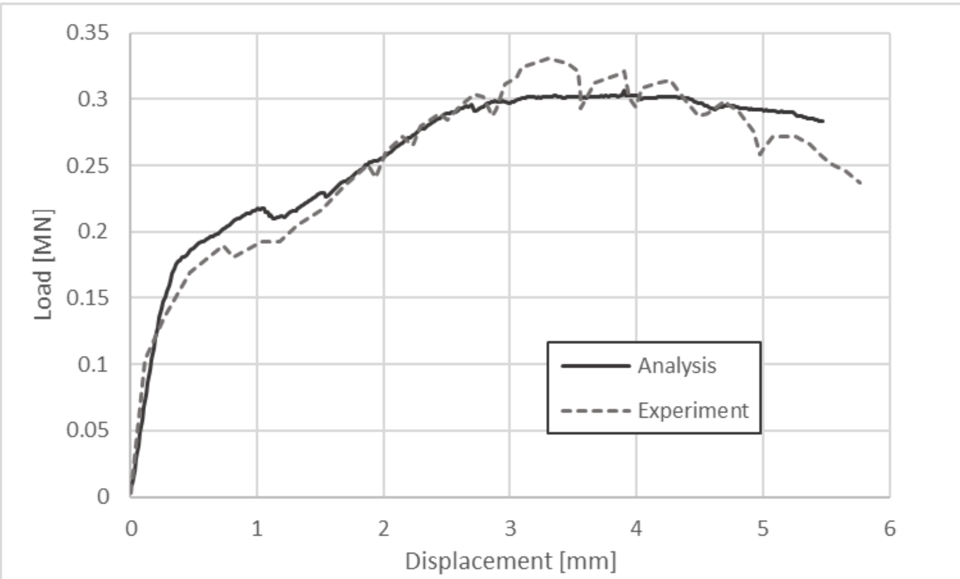


Fig. 5-5: Analysis results compared to experiment.

5.3 Selection and Output data request

In this section is shown how to use combination of two very helpful features such as Selection and Output data request.

The Selection in post-processing is used to separate part or parts of the model from the rest of it. Then the Selection has various ways of utilization e.g., visualization of the Selection through activities which have been introduced above, data or monitor request of the structure part included in Selection. Output data request is shown later in the text once the Selection is prepared.

The process how to create Selection is shown on the part of the structure that is severely crushed during the loading of the model. Specifically, the bottom part of the column which makes frame of the wall opening is the pressed most intensely as shown in Fig. 5-6 with visible scalar results of minimal principal stresses.

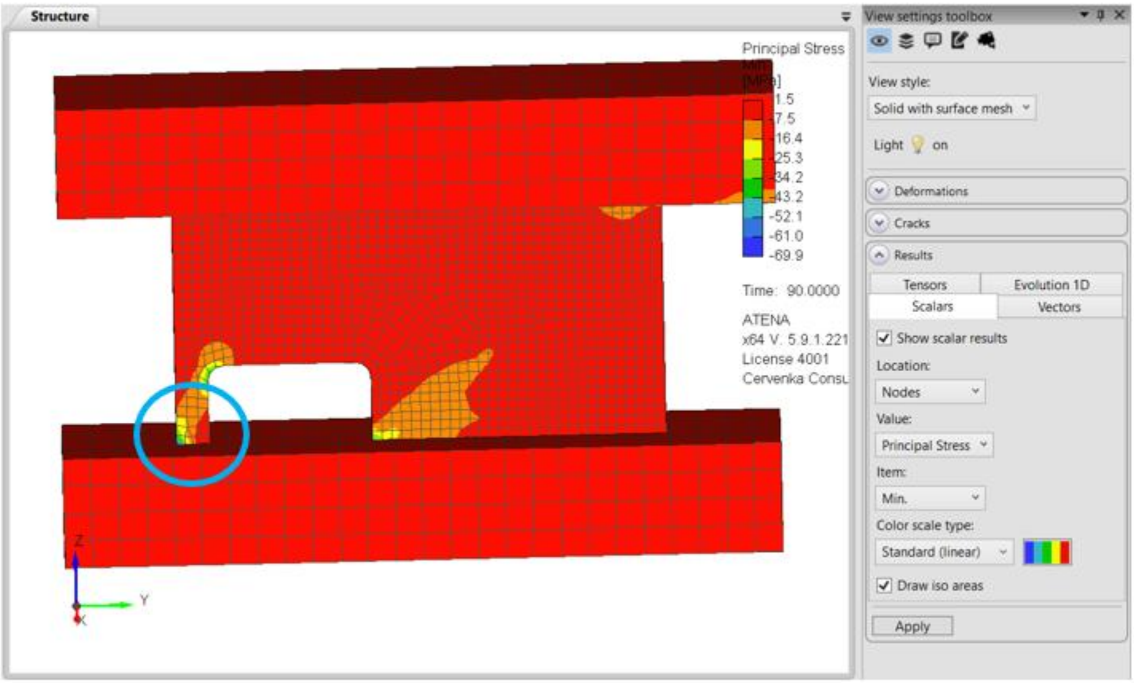


Fig. 5-6: Minimal principal stresses within the structure.

The principal stresses in the crushing zone are relevant only for 3D concrete material therefore the 1D reinforcement is switched off in current activity by using button **Edit activities** (Fig. 5-7) which opens the dialogue where user creates new activity when pressing **Add new activity**. Within new **Activity 1** only parts made of material Concrete_wall are selected to be visible, see Fig. 5-8, by clicking button **Ok** the new activity is activated right away. Then the new Selection is created in Selection manager by clicking on button Create new selection, see Fig. 5-9.

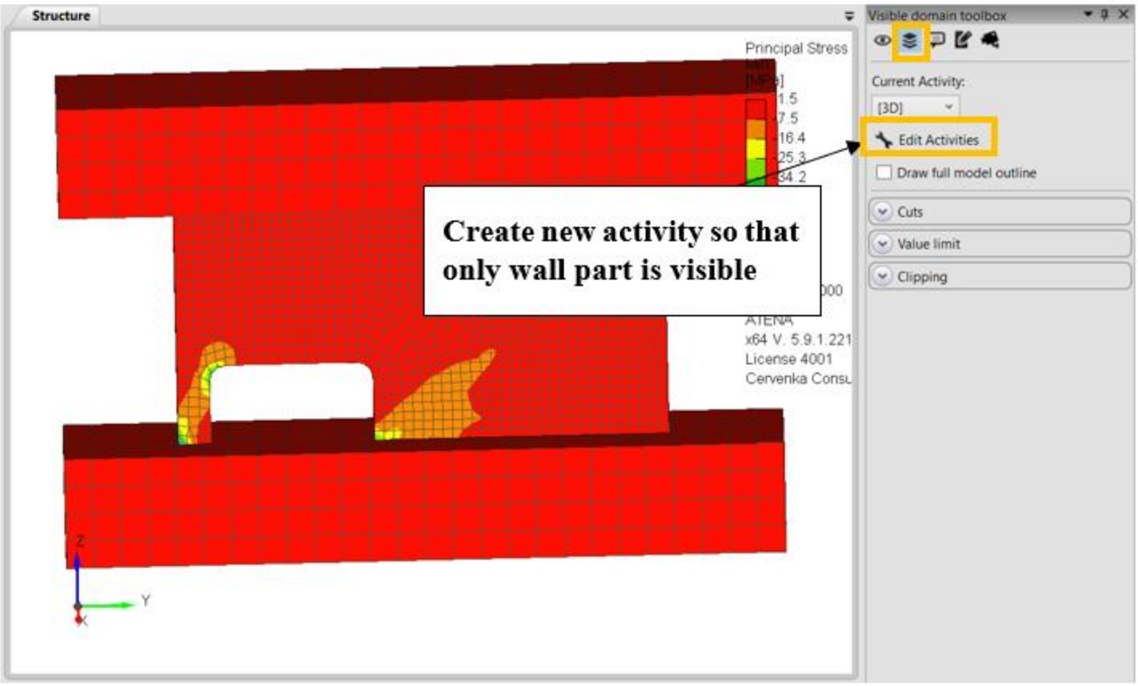


Fig. 5-7: Current activity – Edit activities.

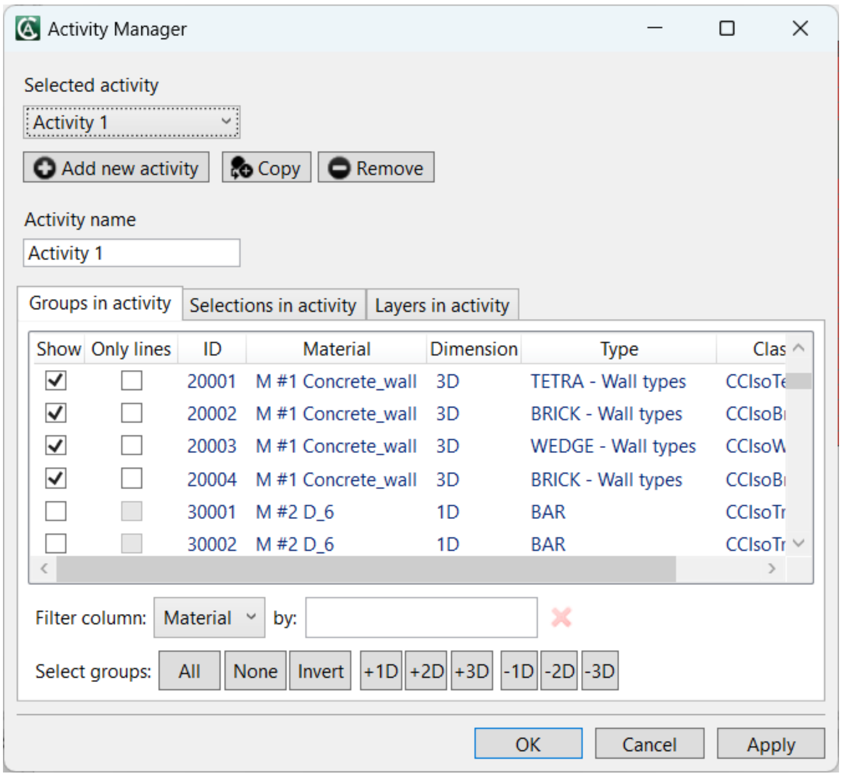


Fig. 5-8: Activity manager – Create new activity.

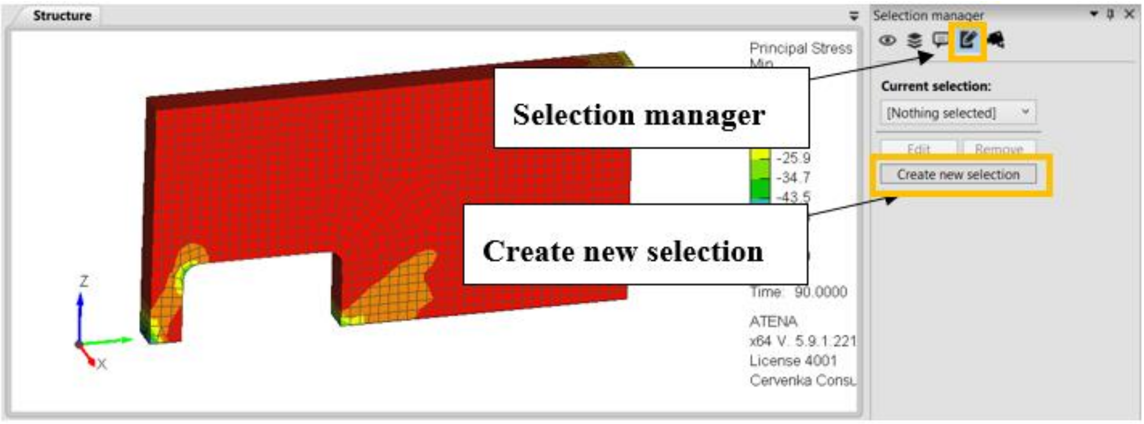


Fig. 5-9: Selection tab – Create new selection.

Current selection settings of new selection are shown in Fig. 5-10. The new name of selection is entered as Frame_corner, then in selection options the Nodes are chosen to be selected (otherwise user can select Elements or Element Ips, however they do not offer same possibilities of Output data request as in case of Nodes), Add to selection and Select internal entities checkboxes are checked. Then the mouse selection on the structure part is done in Structure view. The selected nodes are marked with red balls, shown in Fig. 5-11. In case of selection some extra nodes, it is possible to remove them using checkbox Remove from selection.

Parameters input:	
Selection name	Frame_corner
Entities to select	Nodes
Add to selection	Yes
Select internal entities	Yes

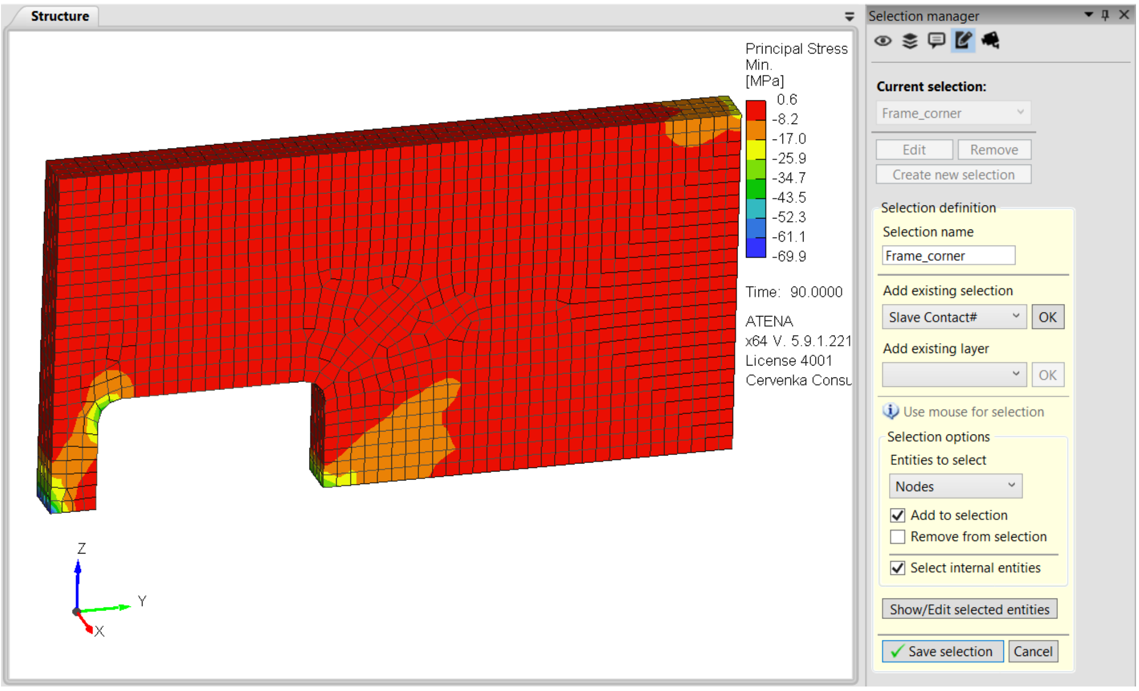


Fig. 5-10: Current selection settings.

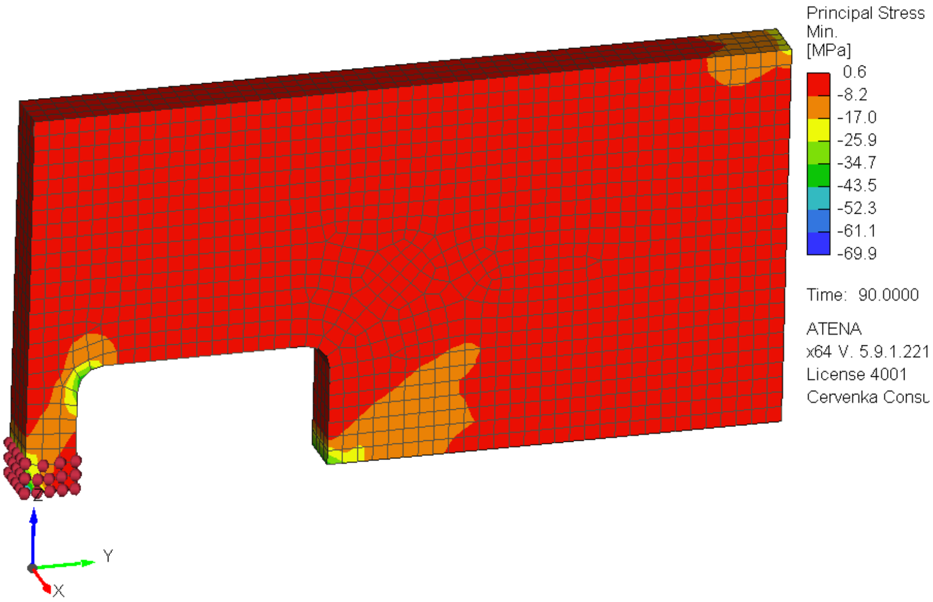


Fig. 5-11: Marked nodes to be added to selection.

Once the selection of Frame_corner is created the data request can be applied on the piece of the structure. Since the Frame_corner is highly compressed part of the structure the Minimal principal stresses are of interest in this region. In order to find out what are the values of Principal stress exactly the user can go to Output | Data request … or just press F8 (Fig. 5-12) and the dialogue of Output data request appears, Fig. 5-13.

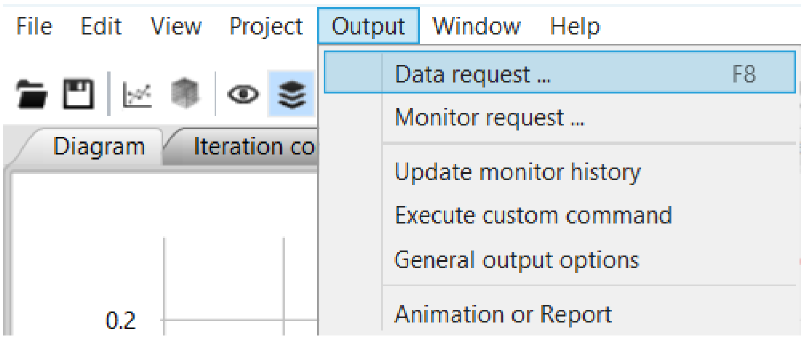


Fig. 5-12: Opening - Output | Data request ...

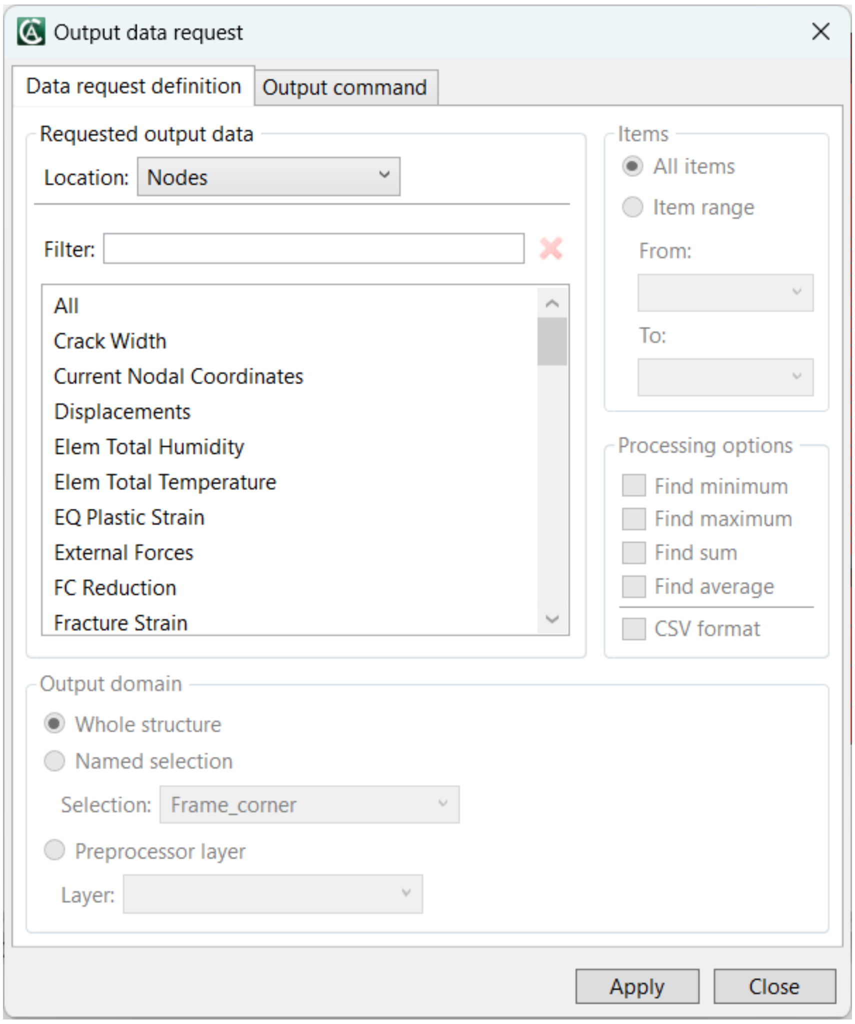


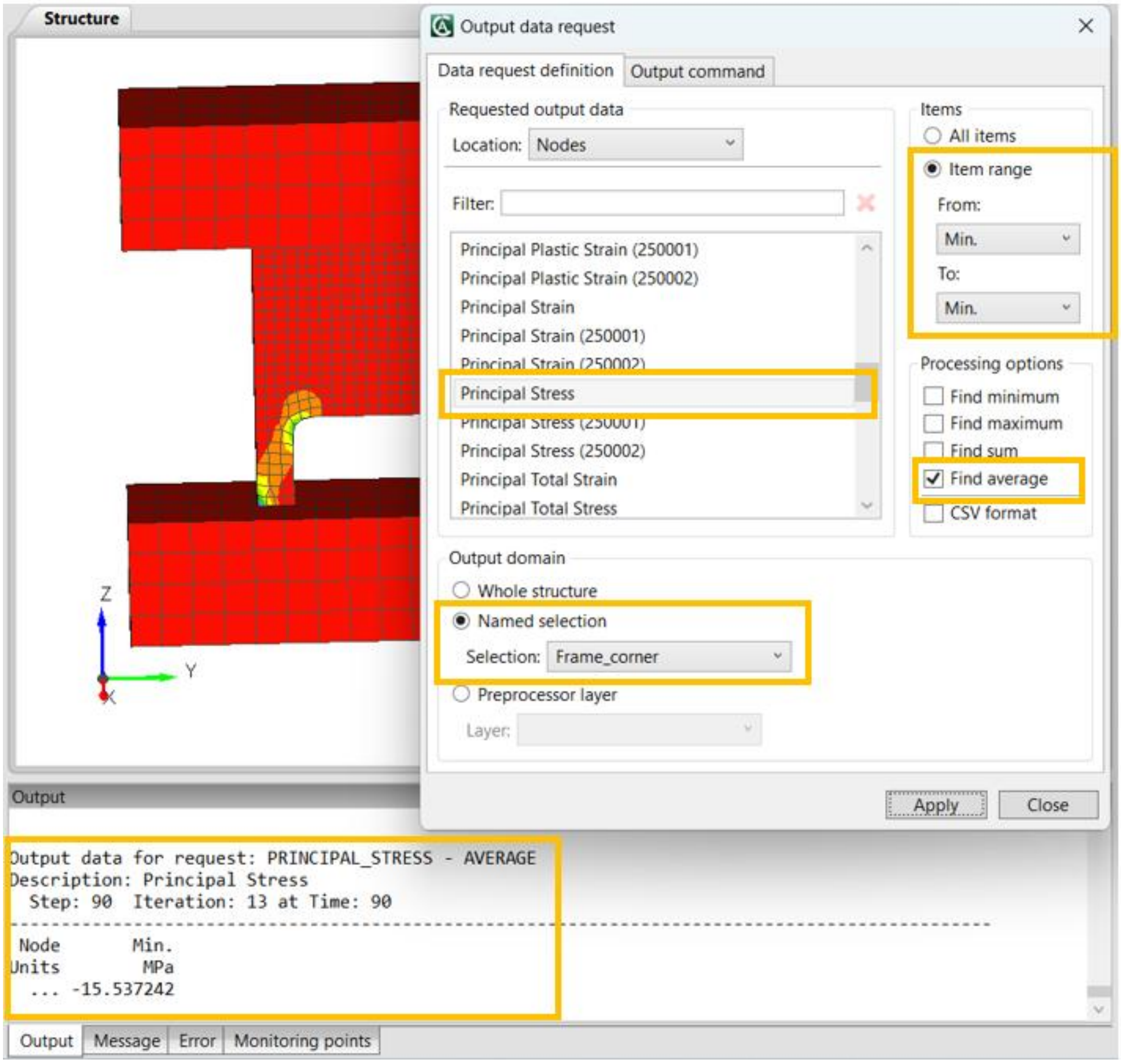
Fig. 5-13: Output data request dialogue.

In the **Output data request** dialogue are set the specifics of results the user is interested in. This example shows how to obtain average Minimal principal stress in the selection that includes bottom part of compressed column. Location of the requested data are in Nodes just like selection which is created from nodes, using Filter the user can easily find Principal Stresses. Then the domain of requested data is the selection created earlier – Frame_corner. Finally, out of all types of principal stresses the Item range is set to Min which gives the minimal stresses representing the maximal compressive stresses and the Processing option is set to Find average that returns the mean values of compressive principal stress within the region of selection, see Fig. 5-14. The result of the data request appears in **Output console** once the user presses **Apply**.

Parameters input:	
Location	Nodes
Results	Principal Stress
Output domain	Named selection | Frame_corner
Items	Item range | Min to Min
Processing options	Find average

Fig. 5-14: Output data settings with result in Output console – Average compressive stress within selection.

Another example shows the output of minimum of Min. Principal Stresses which is actually the maximal compressive stress located in the region of specified selection, see Fig. 5-15.


Structure
Output data request
Data request definition
Output command
Requested output data
Location: Nodes
Filter:
Principal Plastic Strain (250001)
Principal Plastic Strain (250002)
Principal Strain
Principal Strain (250001)
Principal Stress
Principal Stress (250002)
Principal Total Strain
Principal Total Stress
Items
All items
Item range
From:
Min.
To:
Min.
Processing options
Find minimum
Find maximum
Find sum
Find average
CSV format
Output domain
Whole structure
Named selection
Selection: Frame_corner
Preprocessor layer
Layer:
Apply
Close
Output
Output data for request: PRINCIPAL_STRESS - AVERAGE
Description: Principal Stress
Step: 90 Iteration: 13 at Time: 90
Node Min.
Units MPa
... -15.537242
Output
Message
Error
Monitoring points
Z
Y
X

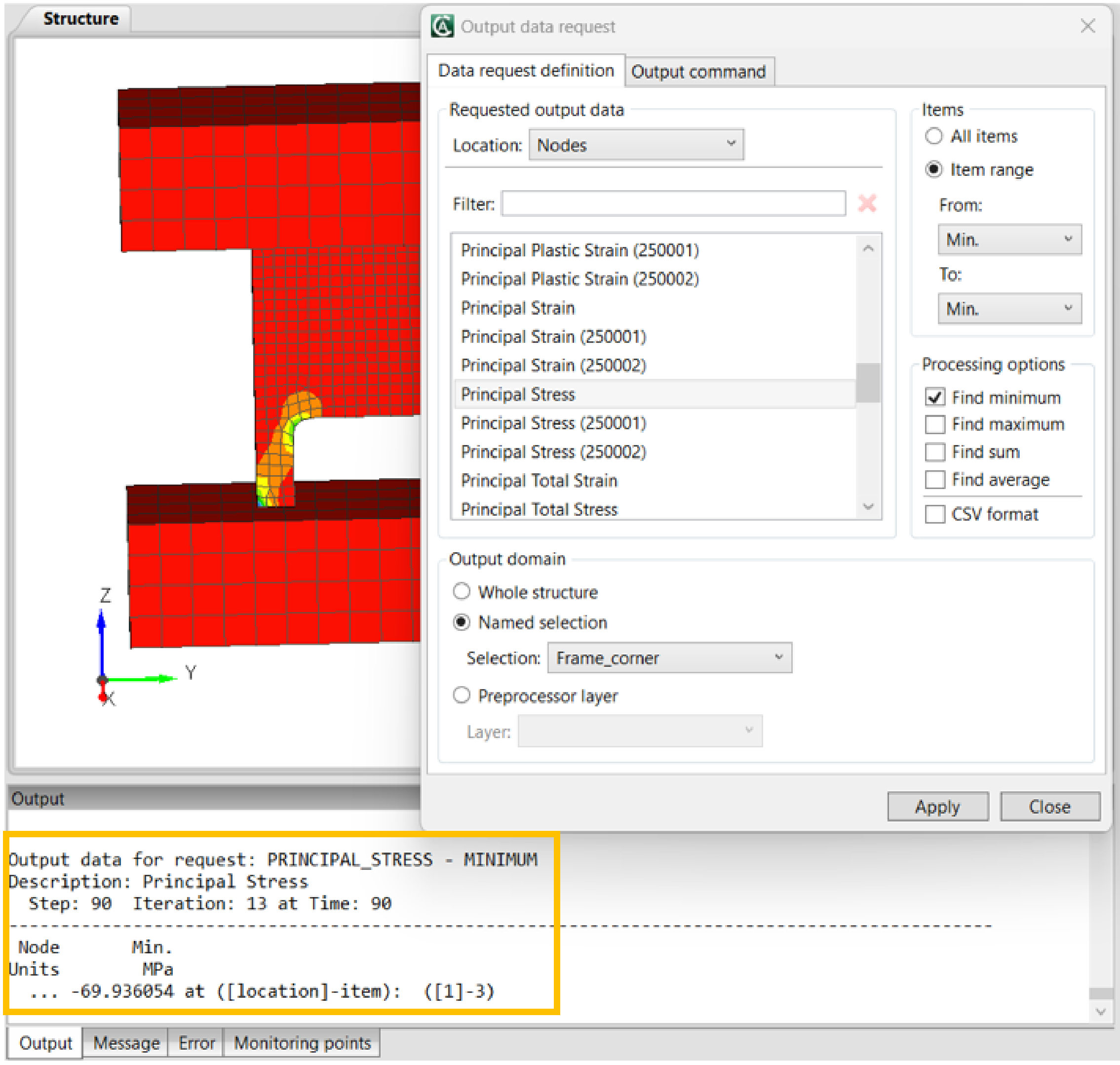


Fig. 5-15: Output data settings with result in Output console – Maximal compressive stress within selection.

6 CONCLUSION

This tutorial documents the way to develop model of structure in ATENA-PRE and then execute FE nonlinear analysis in ATENA Studio.

The topology of the problem is defined in the ATENA-PRE step-by-step so the user can follow the procedure and grasp understanding of the flow of work in preprocessor. Once the problem is developed in preprocessor, the analysis is shown in Atena Studio including the postprocessing part showing some of the useful features.

For more information about ATENA, please refer to ATENA Theory manual [1]. More information about the meshing method used in ATENA software is in detail described in T3D manual [5].

Our team is ready to help you should you have any questions. Also some troubles might be resolved with Troubleshooting manual [6].

7 DISTRIBUTORS AND DEVELOPERS

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

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

References

[1] Cervenka, V., Jendele, L, Cervenka, J., (2022), *ATENA Program Documentation, Part 1, Theory*, Cervenka Consulting, 2022

[2] Foure, B., Eléments finis appliqués au béton armé ou precontraint (comportement non-linéaire jusqu'á rupture), Test de logiciels de calcul, CEBTP (no. 9641006), Fevrier 1998

[3] Cervenka, J., Altman, T., Janda, Z., Rymes, J., Herzfeldt, M., Palek, P., Pukl, R. *ATENA 2024 Program documentation, User's Manual for ATENA-PRE*, Cervenka Consulting, 2024

[4] Benes, S., Mikolaskova, J., Altman, T. *ATENA Program Documentation, Part 12, User's Manual for ATENA Studio*, Cervenka Consulting, 2021

[5] Rypl, D. (2016), *Triangulation of 3D domains – User guide*, CTU in Prague, 2016

[6] Pryl, D. and Cervenka, J., (2022), *ATENA Program Documentation Part 11, ATENA Troubleshooting*, Cervenka Consulting, 2022