SDC ✓ VERIFIER

Tutorial

Modelling with SDC Verifier

Updated on: November 15th 2023

Tested with: SDC Verifier 2023 R2

SDC Verifier is a powerful all-in-one software solution for structural design, FEA analysis, and verification according to standards.

This step-by-step tutorial is designed to **get** you **started** with the main SDC Verifier features.
You will learn how to:

- Launch SDC Verifier;

- Create new project;

- Getting familiar with the graphical interface, the functionality for modelling and meshing;

- Create Properties;

- Define Materials;

- Build a Mesh;

- Refer to Icons Functionality of Geometry and Mesh Ribbons (slides 54-55)

Open a Project

1 Launch SDC Verifier 2023 R2.1
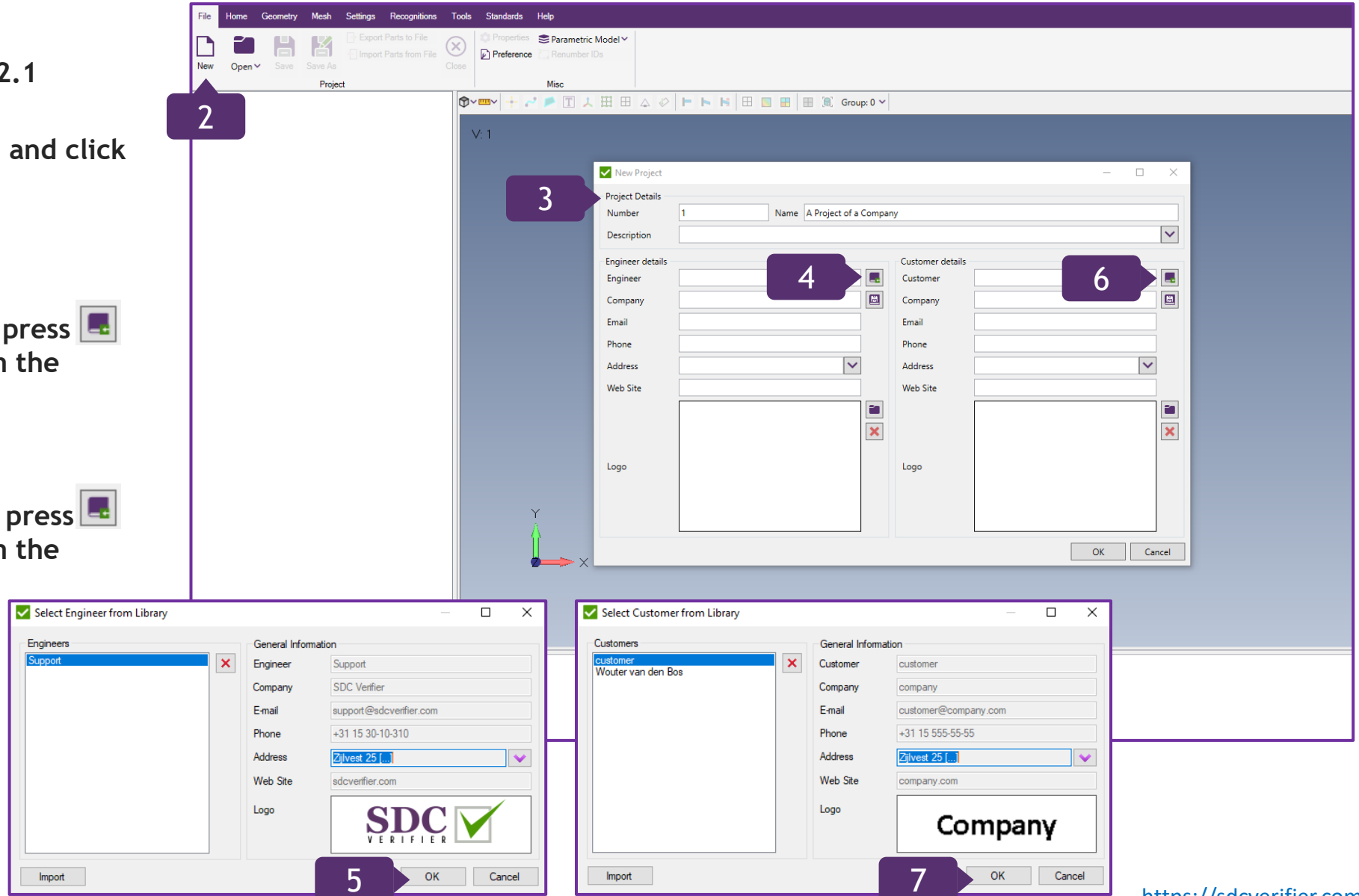
2 Go to *File* tab on the ribbon and click on *New*

3 Fill in the Project Details

**4** In *Engineer details* section, press to load the information from the Library

5 Press *OK*

6 In Customer details section, press to load the information form the Library

7 Press *OK*

3

Icons Functionality Reference (Geometry)

SDC VERIFIER

Geometry Ribbon is broken down into six separate sections with expanded sets of options and commands.
The sets are grouped together by functionality.

Surface: to create plated structural elements.
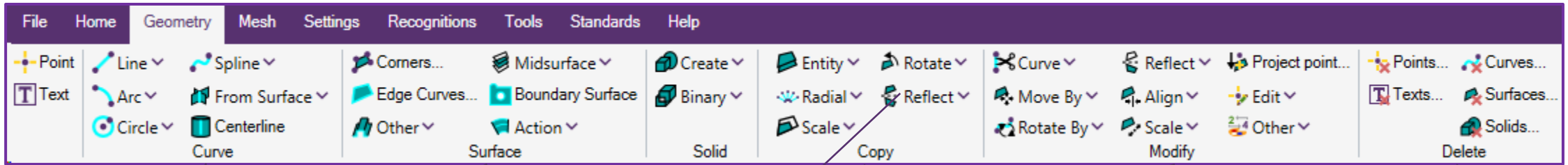
Curve: to form the basis from which you can create surfaces, and they can also be generated from surfaces. They reference points to define their location.

Solid: to create and modify three-dimentional solid elements.
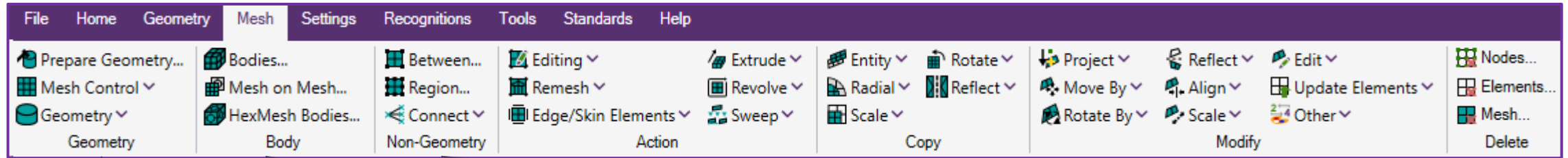
Modify: to implement modifications of existing geometry.

Copy: these operations can be performed with any geometry (points, curves, surfaces, volumes, and solids). When the copied geometry is comprised of other geometry (such as surfaces which are comprised of curves), SDC Verifier will automatically copy these "framework" entities, and then connect them properly to form the new copies.

Delete: to remove entities from the model.

4

Icons Functionality Reference (Mesh)

SDC VERIFIER

> Mesh Ribbon is broken down into seven separate sections with expanded sets of options and commands. The sets are grouped together by functionality.
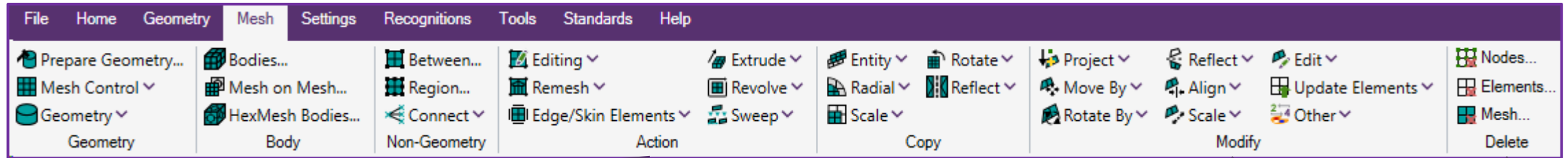
File | Home | Geometry | **Mesh** | Settings | Recognitions | Tools | Standards | Help

Geometry	Body	Non-Geometry	Action	Copy	Modify	Delete
Prepare Geometry...	Bodies...	Between...	Editing ⌄	Entity ⌄ / Rotate ⌄	Project ⌄ / Reflect ⌄ / Edit ⌄	Nodes...
Mesh Control ⌄	Mesh on Mesh...	Region...	Remesh ⌄	Radial ⌄ / Reflect ⌄	Move By ⌄ / Align ⌄ / Update Elements ⌄	Elements...
Geometry ⌄	HexMesh Bodies...	Connect ⌄	Edge/Skin Elements ⌄ / Sweep ⌄	Scale ⌄	Rotate By ⌄ / Scale ⌄ / Other ⌄	Mesh...
			Extrude ⌄ / Revolve ⌄			

Geometry: contains a command for *Preparing complex Geometry* for meshing; *Mesh Control* drop down menu allows to customize the meshing procedure, and to specify the mesh size; *Geometry* menu serves for production the actual mesh on the selected geometry.

Body: *Bodies*…creates a high quality mesh by imposing less restrictions when meshing geometric surfaces. It is used to create a 2D surface mesh on a connected sheet solid or general body or a 3D solid tetrahedral mesh in a solid part.
Mesh on Mesh…incorporates existing triangular or quadrilateral elements as the starting point instead of geometric entities.
HexMesh Bodies…puts away the necessity of splitting the geometry. After the geometric entities are selected, the Hex-Dominant Automesh Bodies dialog box will be opened.

Non-Geometry: *Between*…is applied when creating a mapped mesh of nodes and elements between corner locations, which a person specifies.
Region…is used to create a ruled region of nodes and elements between patterns of existing nodes. These patterns can take any shape, but should be compatible with the type of elements that a person tries to generate between the patterns.
Connect is helpful when automatically creating connections in the model with rigid and line elements, or constraint equations between sets of nodes.

Mesh Ribbon is broken down into seven separate sections with expanded sets of options and commands.
The sets are grouped together by functionality.

| File | Home | Geometry | **Mesh** | Settings | Recognitions | Tools | Standards | Help |

Geometry	Body	Non-Geometry	Action	Copy	Modify	Delete
Prepare Geometry...	Bodies...	Between...	Editing	Entity / Rotate	Project / Reflect	Edit
Mesh Control	Mesh on Mesh...	Region...	Remesh	Radial / Reflect	Move By / Align	Update Elements
Geometry	HexMesh Bodies...	Connect	Edge/Skin Elements / Extrude / Revolve / Sweep	Scale	Rotate By / Scale	Elements...
					Other	Mesh...

Action: *Editing* has a number of commands for modifying different types of existing elements. The commands: Interactive...Split...Element Refine...Edge Split etc.
Remesh commands allow taking existing nodes and produce a new mesh with the same boundaries.
Edge/Skin Elements menu is used to create line elements along the free edges of selected planar elements or planar elements on the free faces of solids.
Extrude allows to select curves, elements or element faces, and then move along a specified vector to form new elements.
Revolve commands take existing curves, elements or element faces and create additional planar or solid elements.
Sweep menu is handy when selecting curves, elements or element faces, and then sweeping them along one or more curves to form new elements.

Copy: the menu presents commands for duplicating existing nodes or elements.
Radial commands use either a single location or a defined vector, along with an offset distance.
Scale commands create one or more copies of selected entities, offset from a center location.
Rotate menu creates duplicate copies of model entities. They rotate the duplicate copies around a vector, using the specified options.
Reflect command create duplicate copies of model entities, and they reflect any portion of the model across a plane by flipping existing nodes or elements.

Modify: the menu contains commands for modifying a mesh. They serve to refine, unrefine, smooth, or remesh particular elements. These tools can be very useful to smooth or remesh small areas of meshes, especially if the original mesh was produced over a large area. In general, smoother meshes will be obtained, and remeshing will produce excellent results when small areas of the mesh are connected.

Delete: the commands allow to reduce the amount of Nodes, Elements and Mesh.

Geometry Functionality and Creating Points

1 In *Geometry* tab, press on ⊕ to create a Point with Coordinates: X=0,Y=0,Z=0

2 Press on *Methods* to expand the functionality of the *Menu*; select *Locate*

3 Press *OK* and close the window

4 Press on ⊕ to create second point with Coordinates: X=0,Y=0,Z=3 m

5 Press *OK*

6 Close the window

Geometry tabs allow to create points, lines, curves, surfaces, solids and many more. Also, it is possible to adjust them, copy, modify and remove them.

The Locate – Enter Coordinates or Select with Cursor menu allows to create points by coordinates.

The Coordinate System is unitless, so the geometry can be modelled with meters as a value.

Note: In this case, Z is a vertical axis.
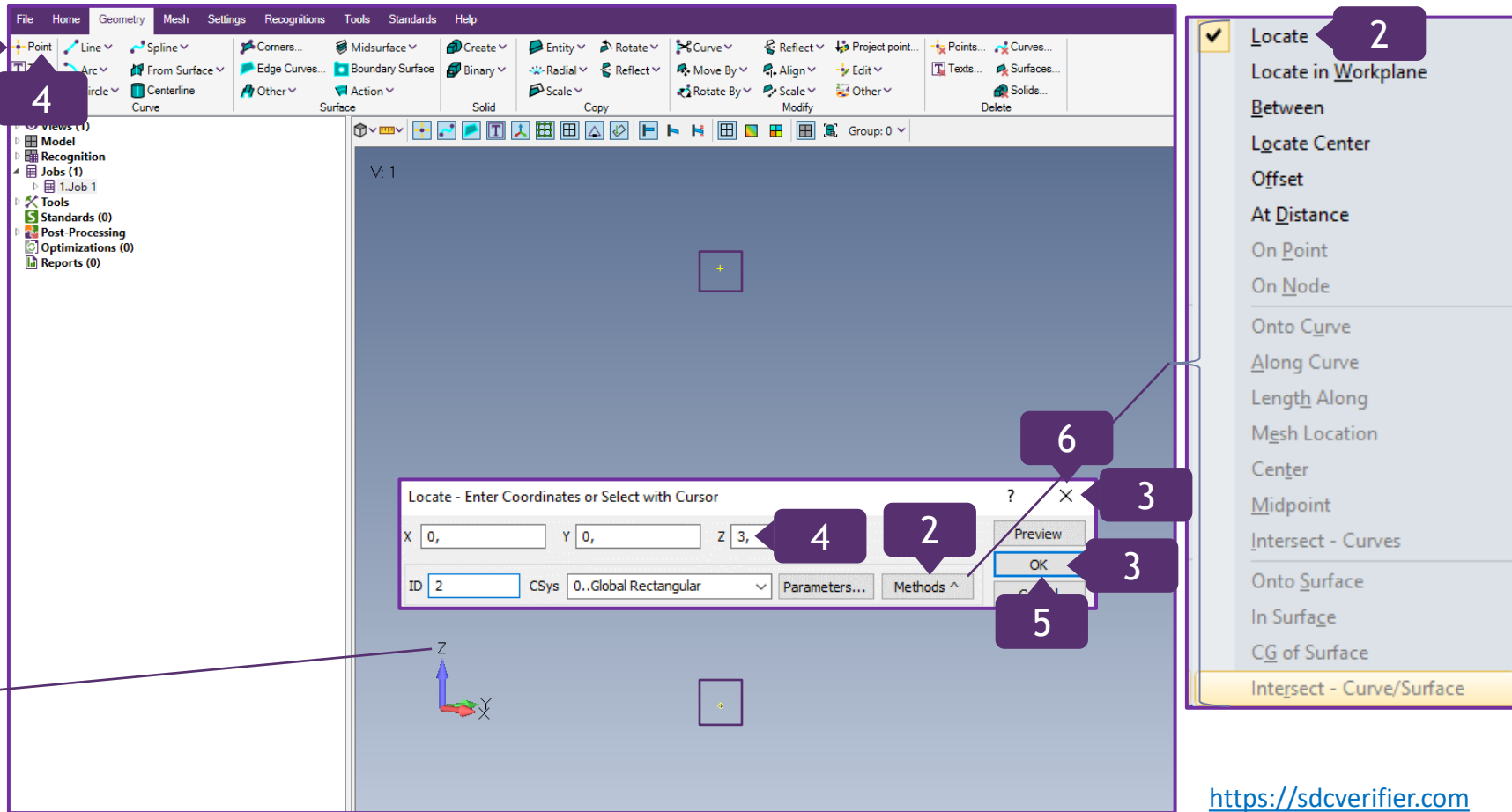
Create a Line in Geometry

1 In *Curve* section, press on 📏 *Line* and in the menu, select *Coordinates...*

2 Click on *Methods* and select *Locate*

3 On the graphical interface, execute right click and select *Snap to Point*

> Snapping to Point option allows to pick snaps to the nearest point in relation to the cursor positioning on the graphical interface.

Note: An alternative option of creating a line is to type in the coordinates values in the Locate – Enter First Location for Line
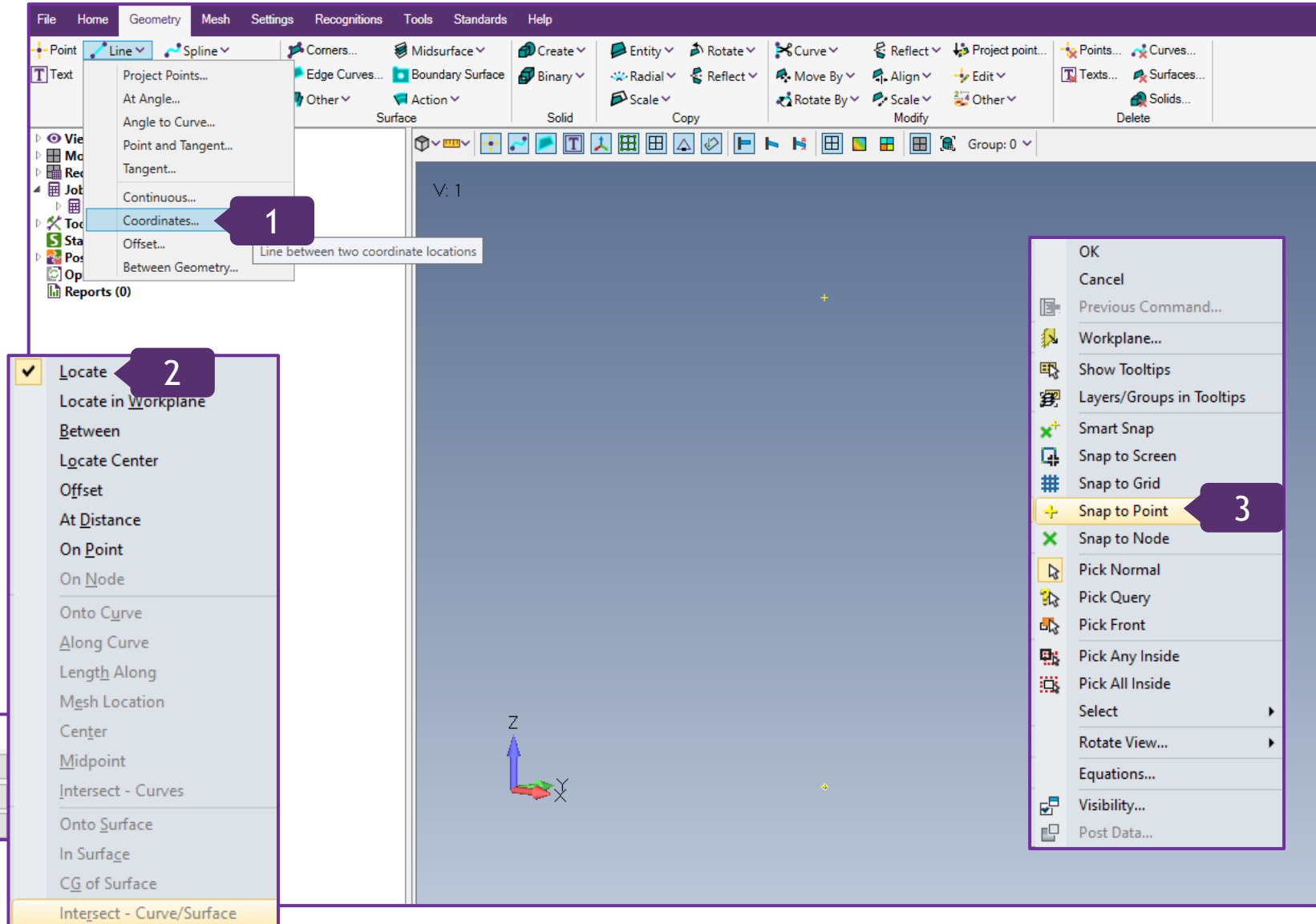
Create a Line in Geometry (Continuation)

4 On the graphical interface, select the Point with ID=3. Press *OK*

5 Select another Point and press *OK*

6 Close the window
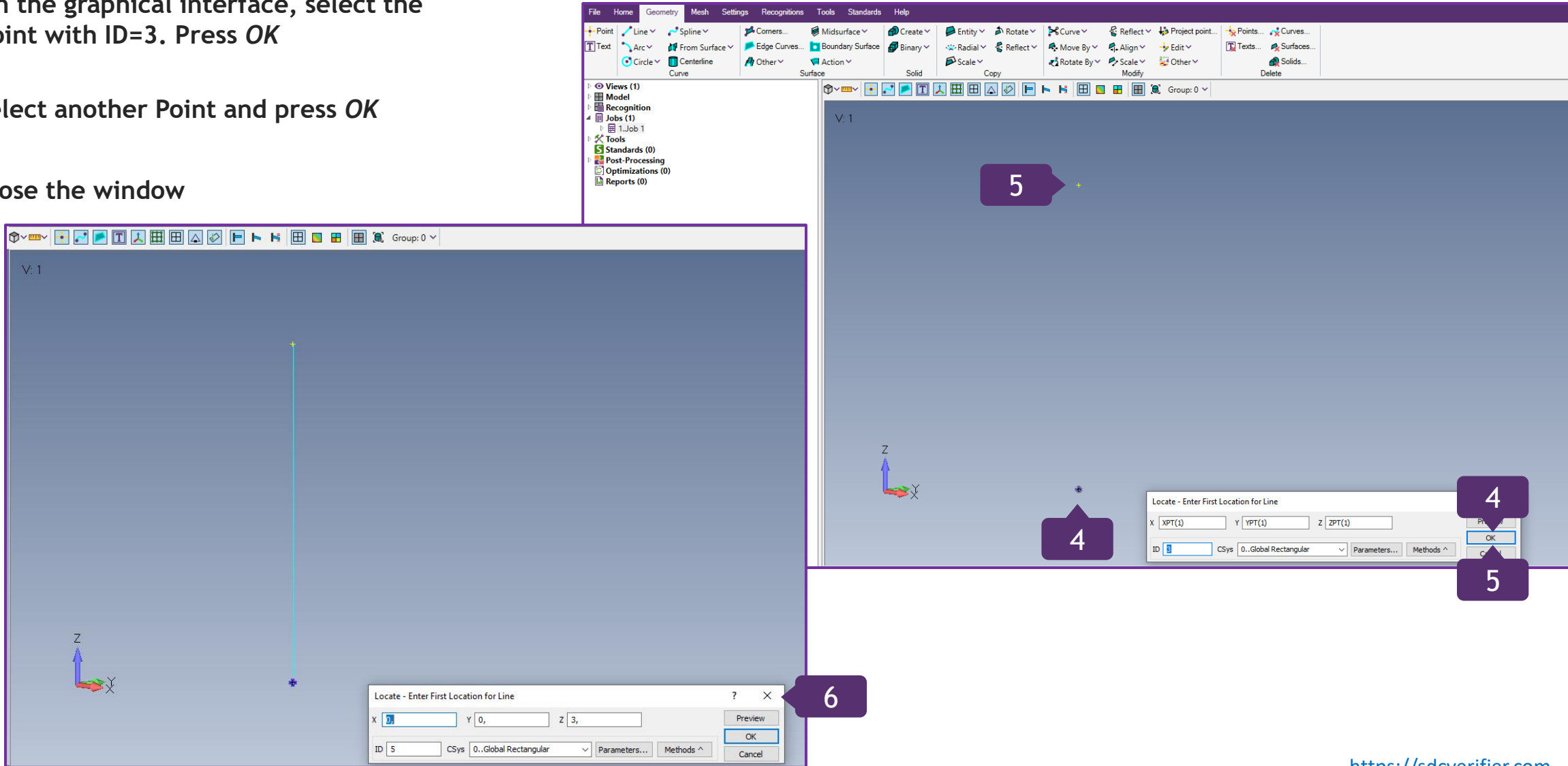
Create another Line by Coordinate System

1 In *Curve* section, press on 🖊 *Line* and in the menu, select *Coordinates…*

2 Fill in the coordinates: X=3, Y=0, Z=0

3 Press *OK*

4 Fill in the coordinates: X=3,Y=0,Z=3

5 Press *OK*

6 Close the window
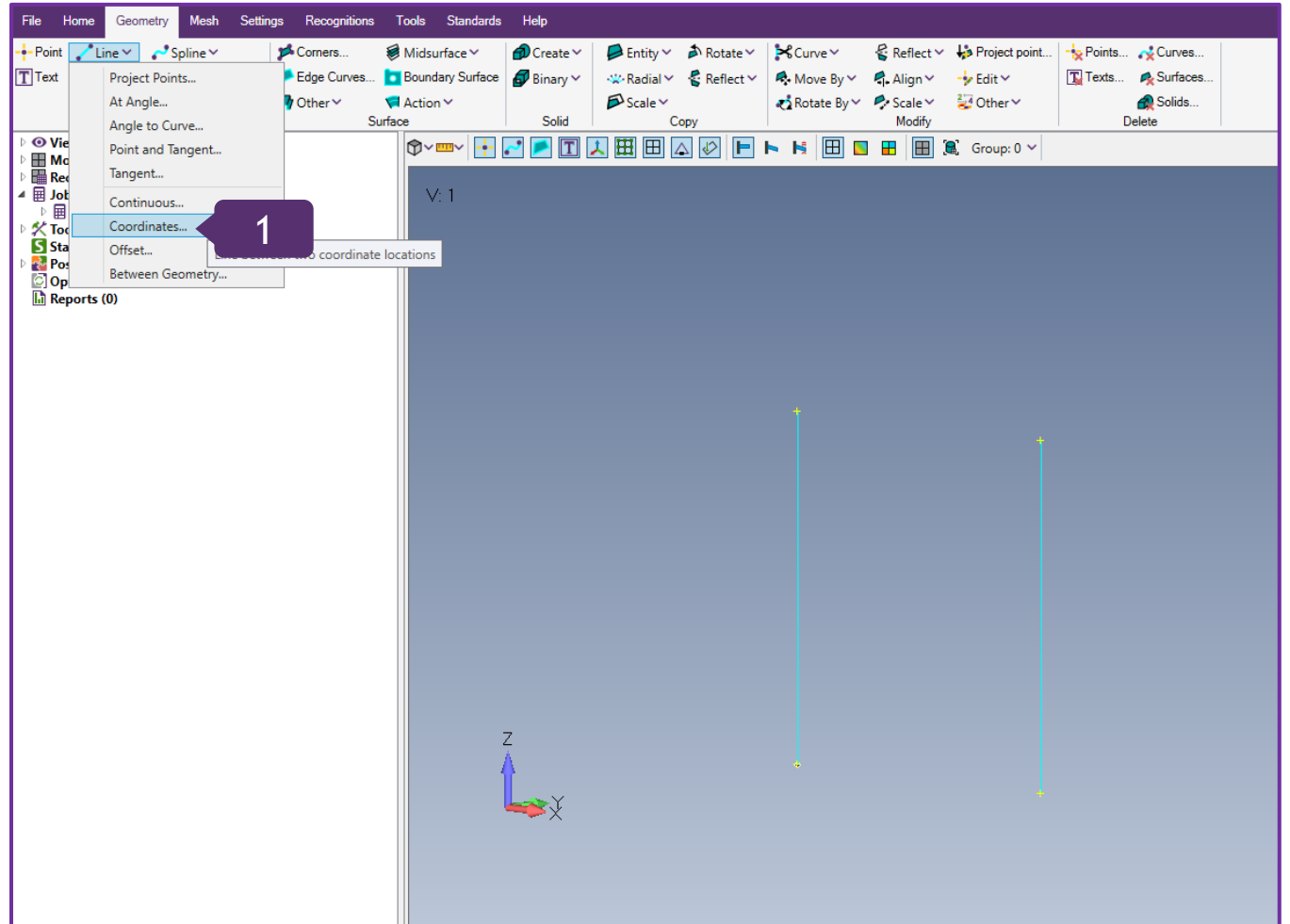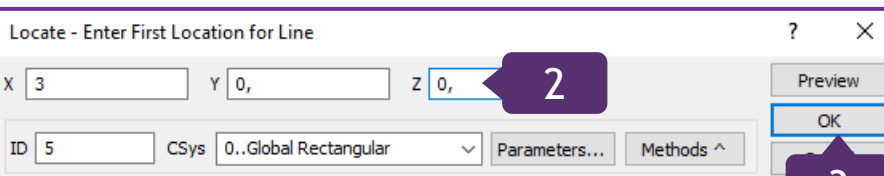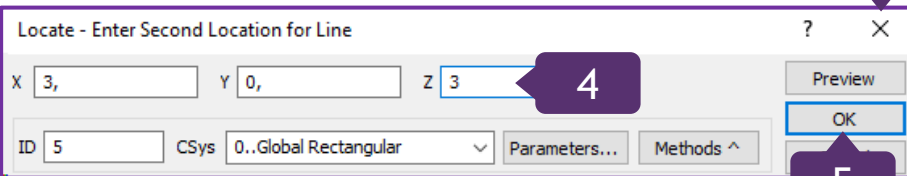
Note: The second line has been created 3 meters away from X-direction.

Copy the previously created Lines

1 In *Copy* section, press on *Entity* and select *Curves*

2 With the left clicks of the mouse, select two lines on the graphical interface

3 Press *OK*
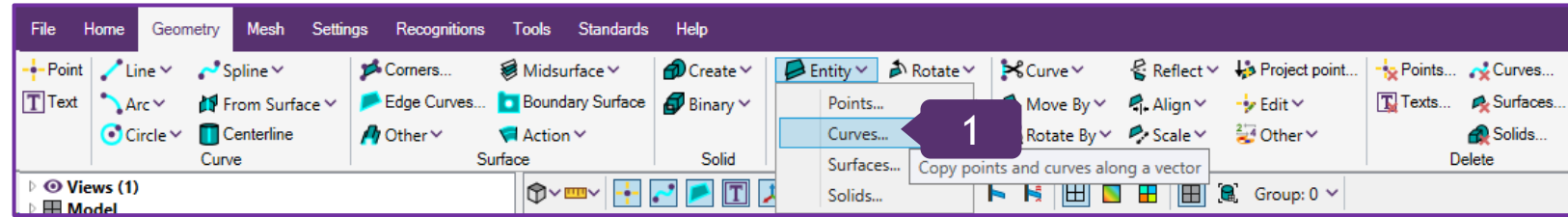
Note: As an option, curves can be selected by ID. Also, the selection can be defined by clicking on *Method*.
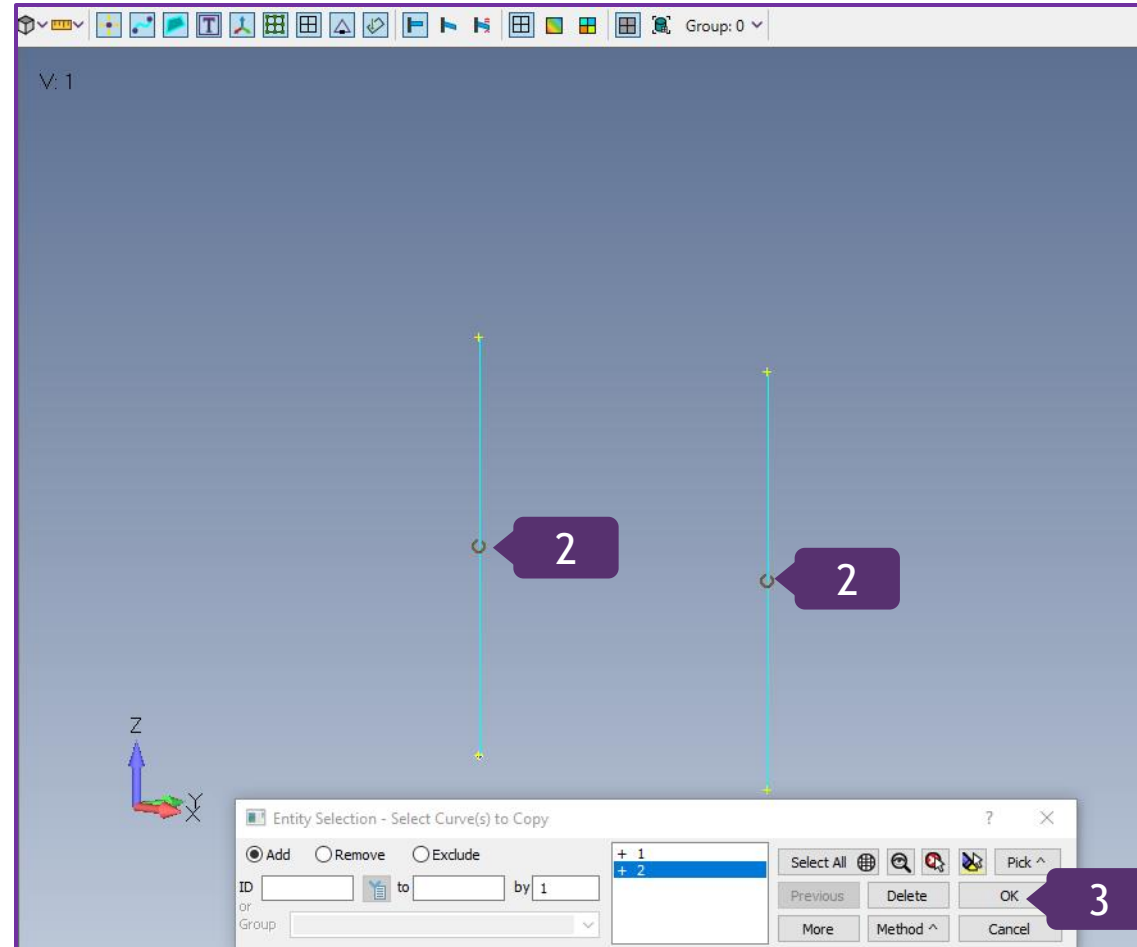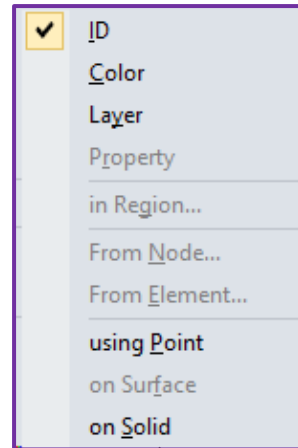
Copy the previously created Lines (Continuation)

4 In *Orient/Transform* section, *Move Along vector* is *ON*

5 *Approach => Single/Multiple Copies => Repetitions: 1*

6 Press *OK*

7 To specify the Vector =>
Base: X=0,Y=0,Z=0
Tip: X=0,Y=3,Z=0

8 Press *OK*

Note: the Vector can also be specified by clicking on the points.

The result
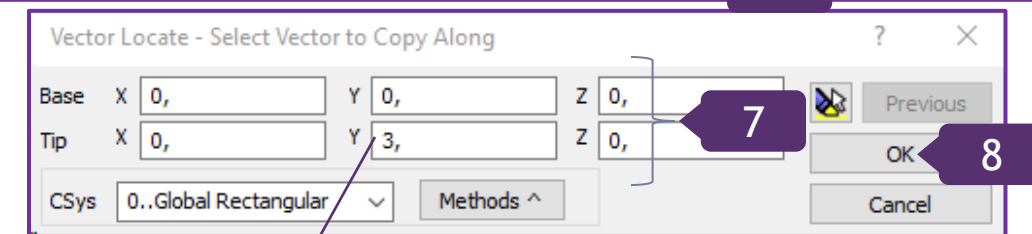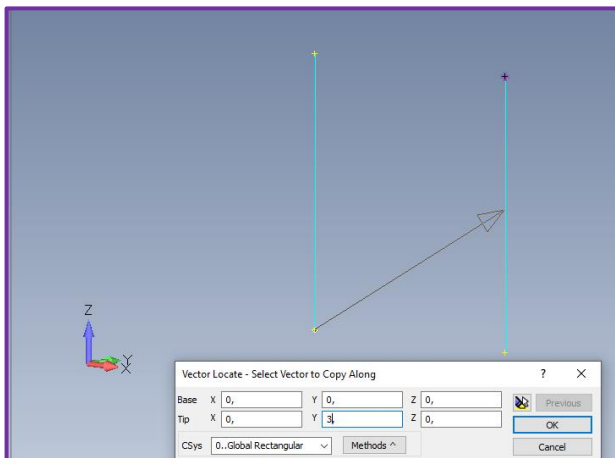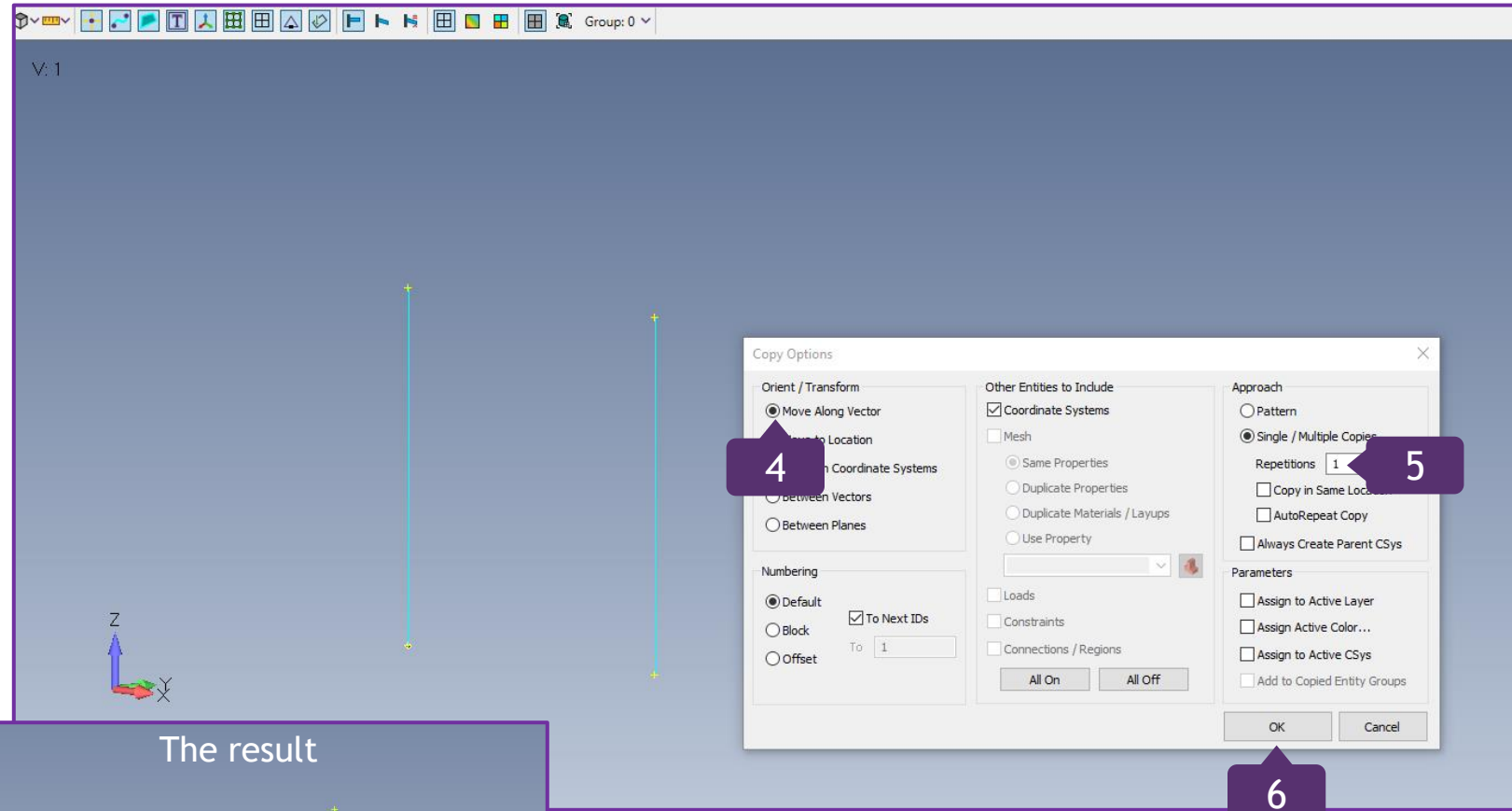
Note: Tip Y=3 refers to the lines, copied 3 meters to the side in Y-direction.

https://sdcverifier.com

Model a Tower

1 In *Curve* section, press on 📈 *Line* and in the menu, select *Coordinates…*

2 Create *Lines by Coordinates*, as it has been shown on 5-6 slides

3 Press *OK*

4 To make repetitions, go to *Copy* section, press on *Entity* and select *Curves*

5 Pressing CTRL + left click of the mouse, select all Lines

6 Press *OK*

13
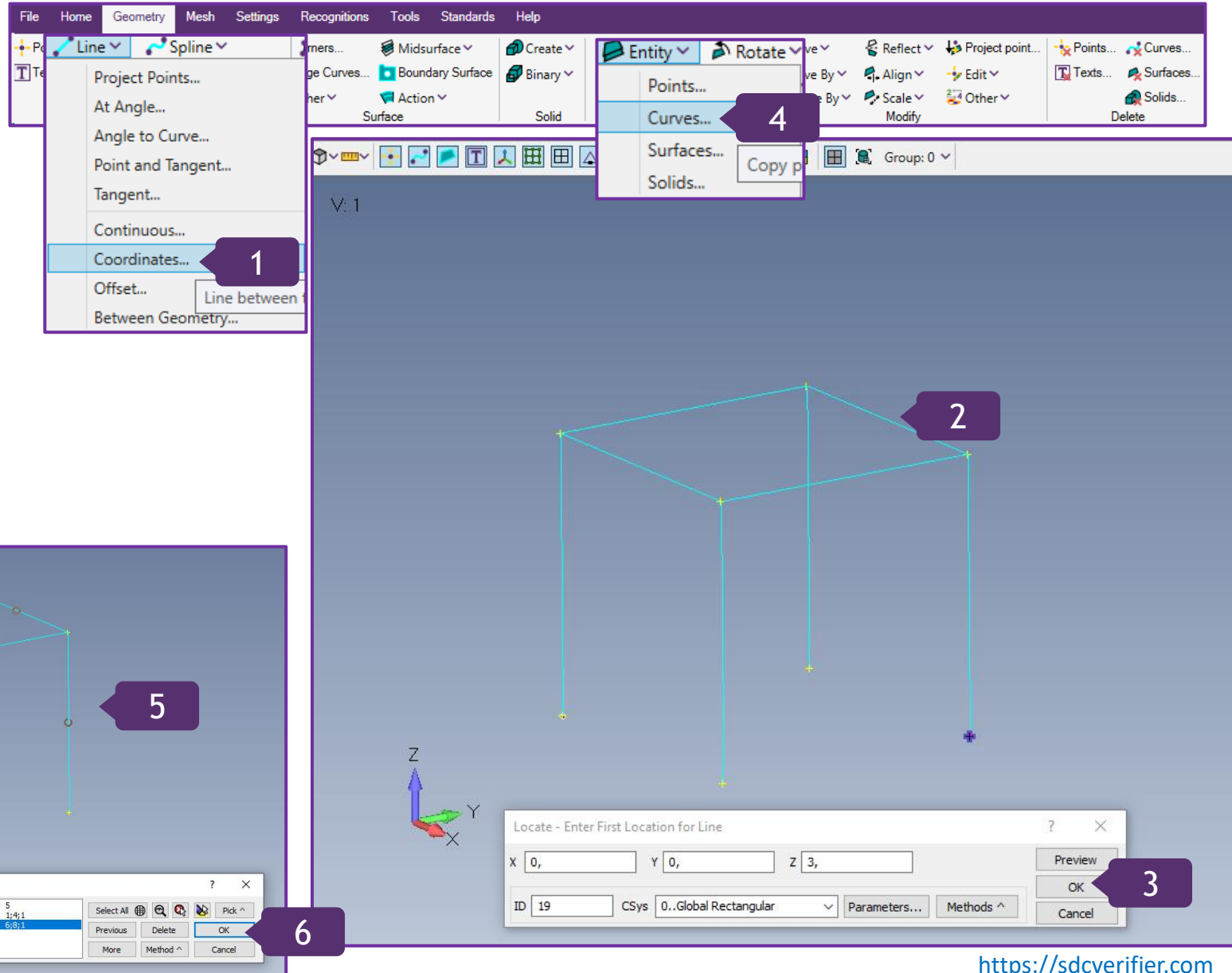
Model a Tower (Continuation)

7 In *Approach* section => *Single/Multiple Copies* => *Repetitions: 3*

8 Press *OK*

9 To copy the curves up, press on the Point and drag the cursor up, reaching the nearest Point with a click. Press OK

The result

https://sdcverifier.com
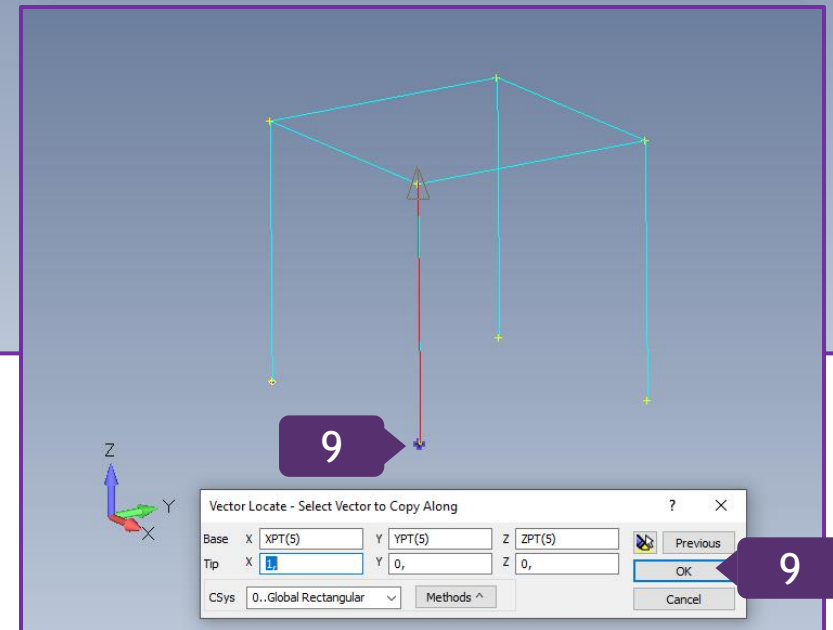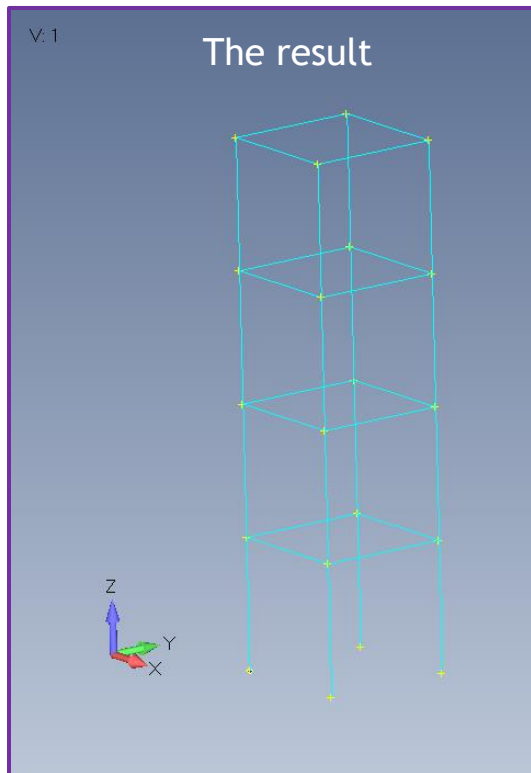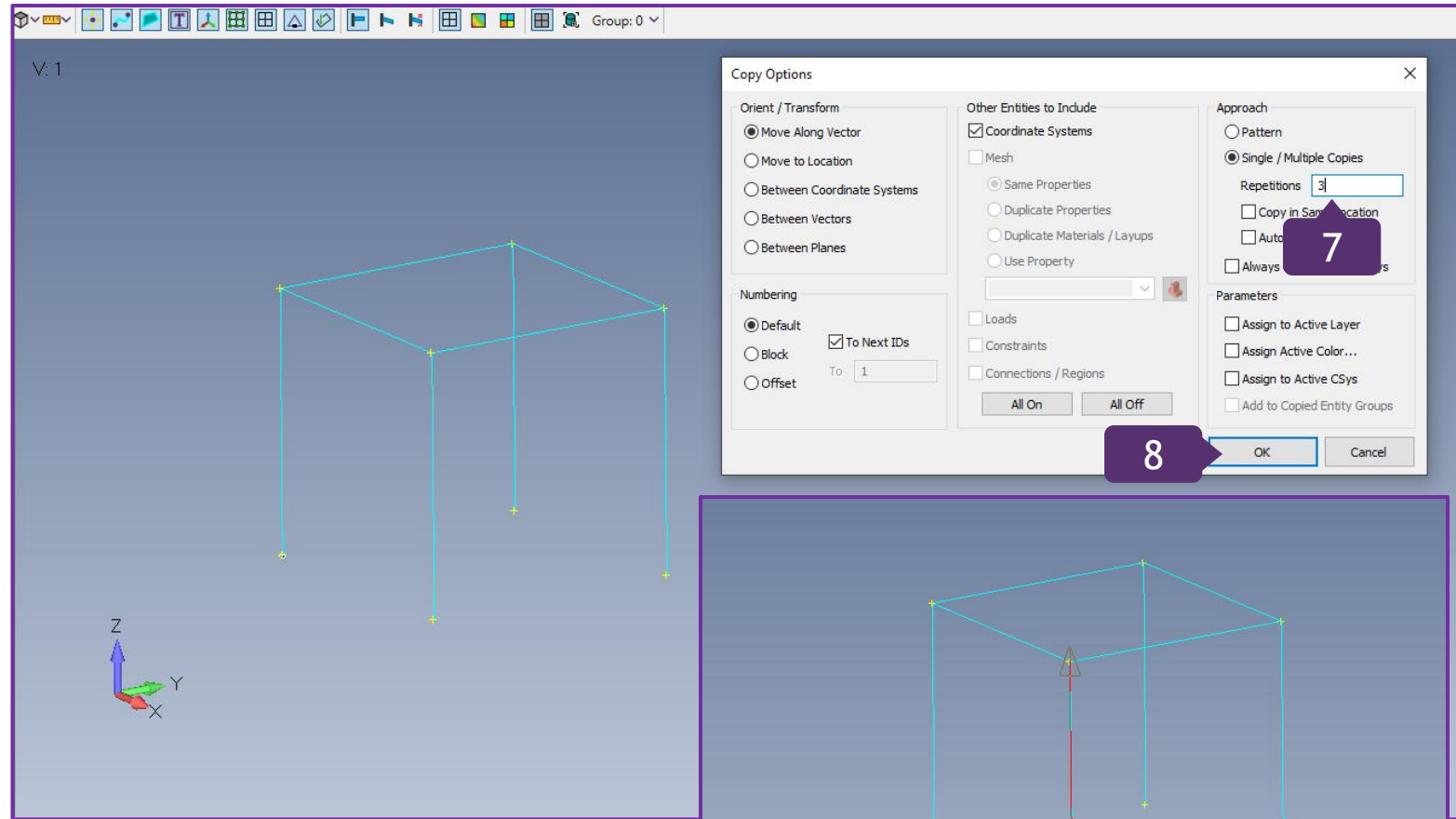
Model Diagonal Stiffeners and split them in the middle

1 In *Curve* section, press on ✏ *Line* and in the menu, select *Coordinates…*

2 Create diagonal *Lines by Coordinates*

Note: As the diagonal members are continuous lines, the mesh needs to be split, making nodes in the middle.

3 In *Modify* section, press on ✂ *Curve* and in the menu, select *Break at Location…*

4 Select the Lines by clicking on them and press *OK*
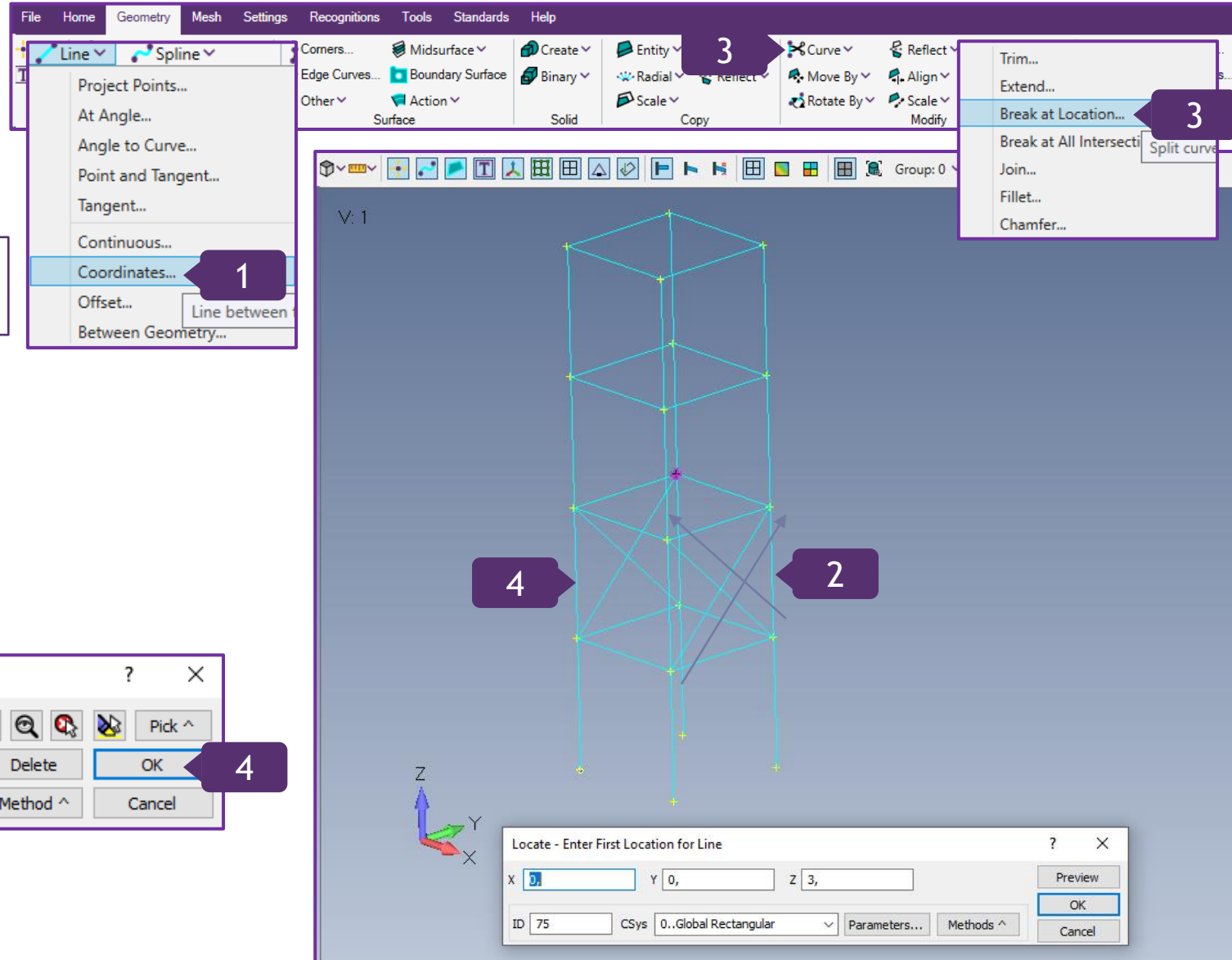
Model Diagonal Stiffeners and split them in the middle (Continuation)

5 — Click on *Methods*

6 — Select *Intersect - Curves*

7 — Select *Curve ID 1* and *Curve ID 2*
Press *OK*

The result

Intersect - Enter Location to Break At

Curve ID 1 42   Curve ID 2 41   **7**

Nearest To   X   Y   Z

ID   CSys 0..Global Rectangular   Parameters...   Methods ^   Cancel

OK **7**

Locate
Locate in Workplane
Between
Locate Center
Offset
At Distance
On Point
On Node
Onto Curve
Along Curve
Length Along
Mesh Location
Center
Midpoint
Intersect - Curves   **6**
Onto Surface
In Surface
CG of Surface
Intersect - Curve/Surface

Locate - Enter Location to Break At

X 0,   Y 3,   Z 6,   **5**

ID   CSys 0..Global Rectangular   Parameters...   Methods ^

Preview
OK
Cancel

Reinforce the Structure

1 In *Copy* section, press on *Entity* and select *Curves*

2 With the left clicks of the mouse, select the lines on the graphical interface

3 Press *OK*

4 In *Orient/Transform* section, *Move Along vector* is *ON*

5 *Approach => Single/Multiple Copies => Repetitions: 1*
Press *OK*

6 To specify the Vector, dragging it up =>
Tip: X=1,Y=0,Z=0

7 Press *OK*
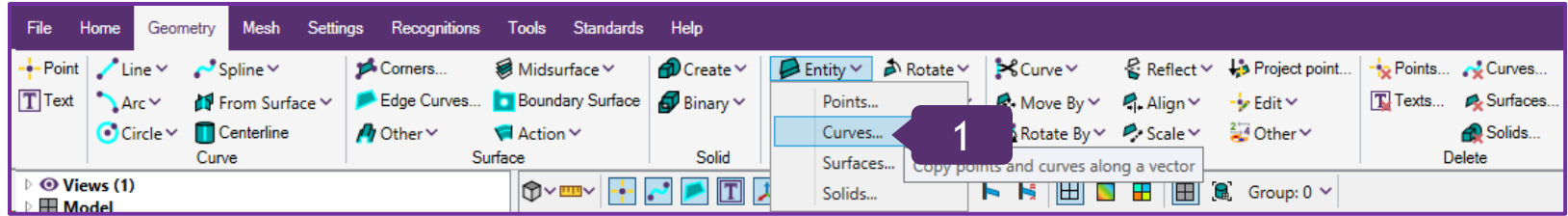
Aiming at reinforcing the Tower, additional symmetrical structural elements have been added, using Copying Curves functionality.

The result
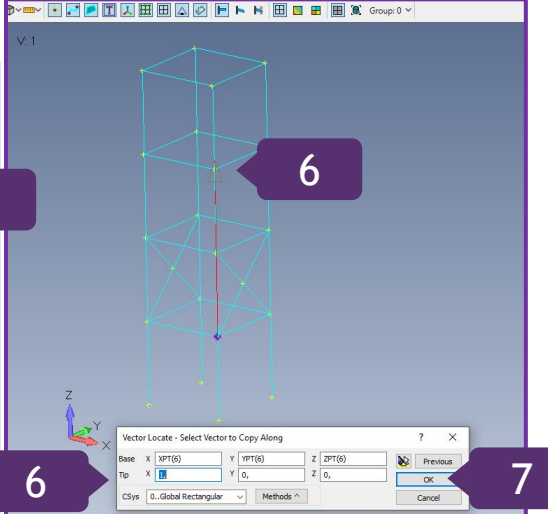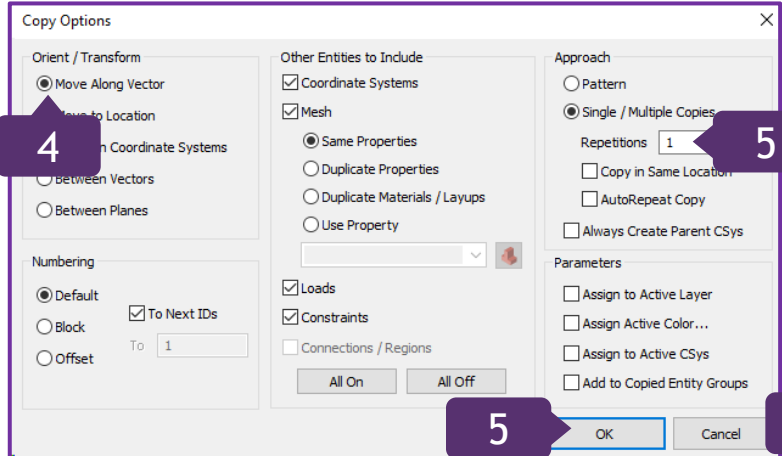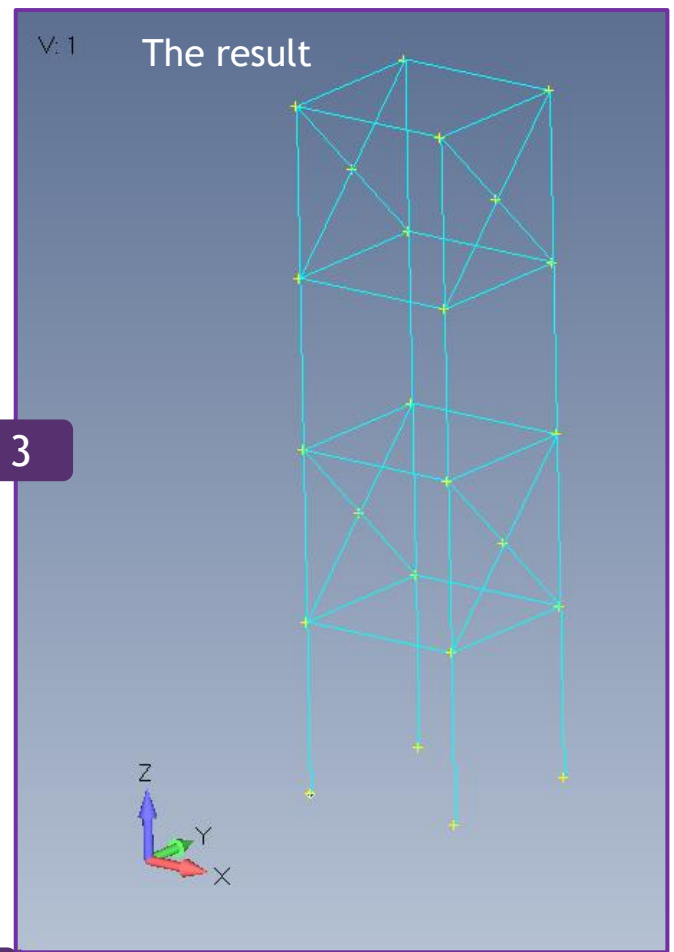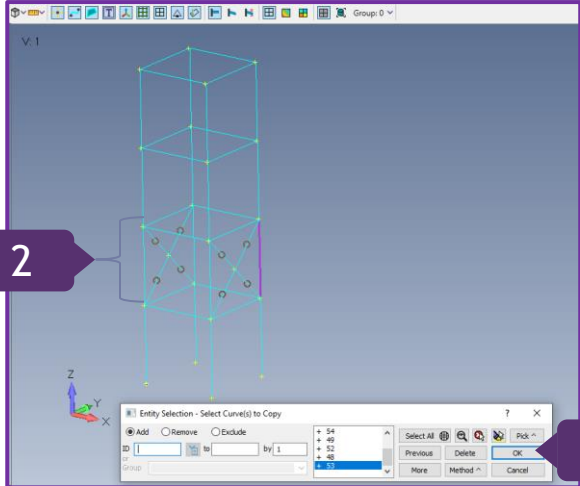
Add Additional Diagonals

For the sake of demonstrative purposes of Deleting Functionality (it will be presented on the next slide), more diagonal members have been added.

Note: as the previously used Method Intersect – Curves is on, it is required to switch it to Locate, when adding more diagonals.

Deleting Functionality
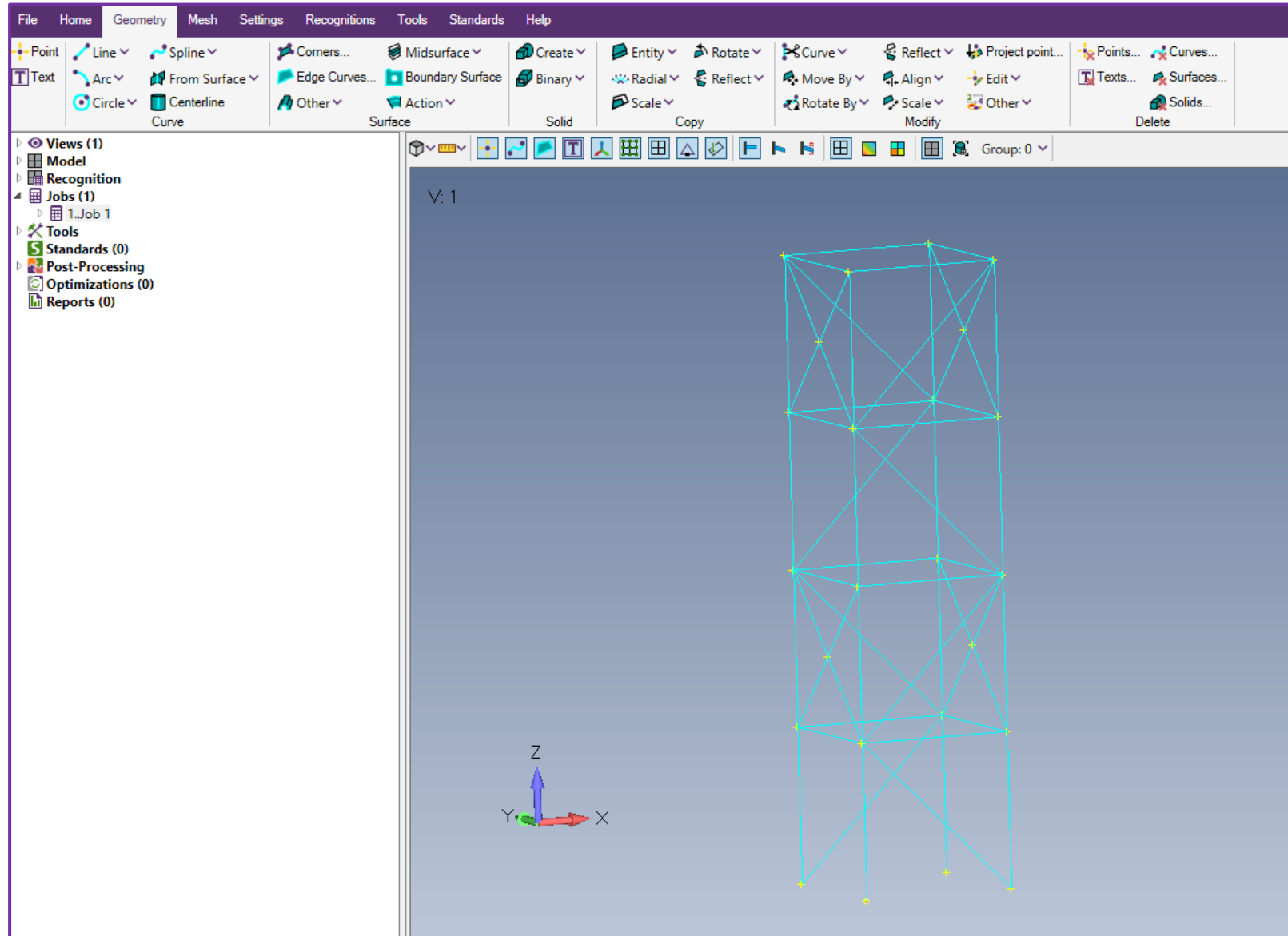
1 In *Delete* section, press on *Curves*

2 With a left click of the mouse, select curves. Press *OK*

3 Press *OK*

The result
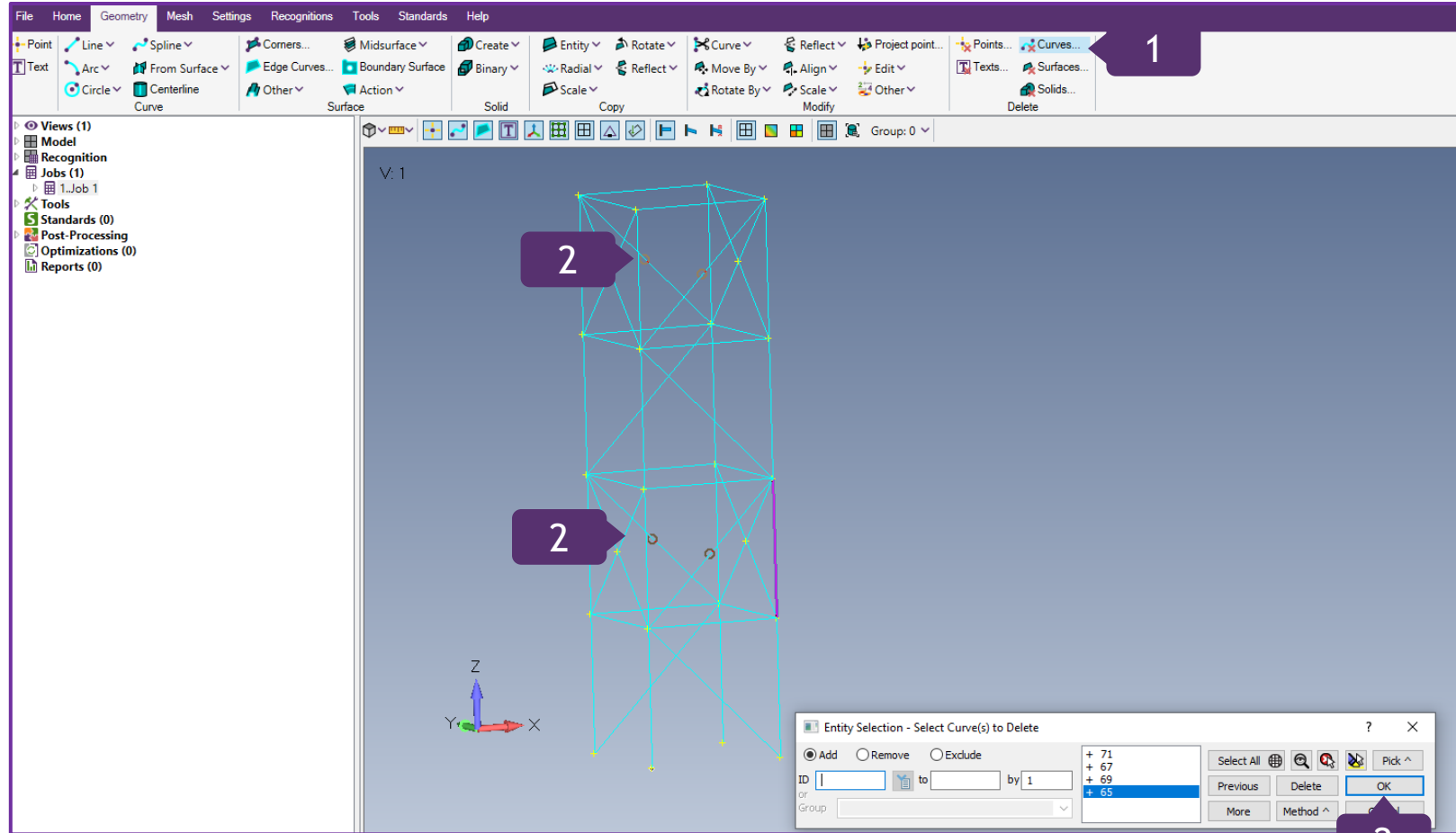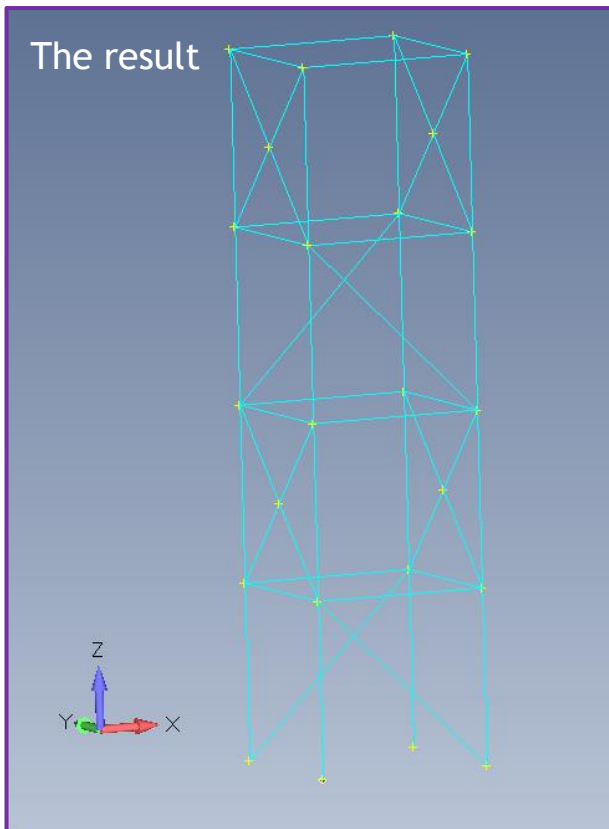
It is possible to disable the Confirm Delete window, that emerges every time the structural member deletion is being carried out. To activate it, select *Don't confirm delete again* option.

Confirm Delete

OK to Delete 4 Selected Curve(s)?

☑ Update Views   OK   **3**

☐ Don't confirm delete again   Cancel

19

https://sdcverifier.com
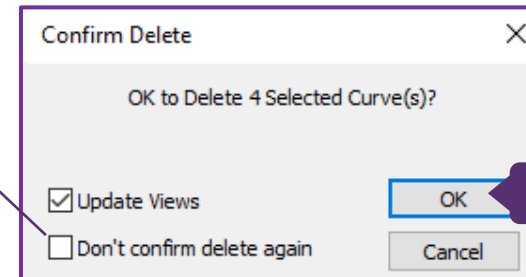
Model a Surface of the Structure

1 In *Copy* section, press on *Entity* and select *Curves*

2 Select the Curves by clicking on them and press *OK*

3 In *Approach* section => *Single/Multiple Copies* => *Repetitions: 1* Press *OK*

4 Click on the Point and drag the cursor, reaching the nearest Point. Press OK

The result

Previous Command Functionality
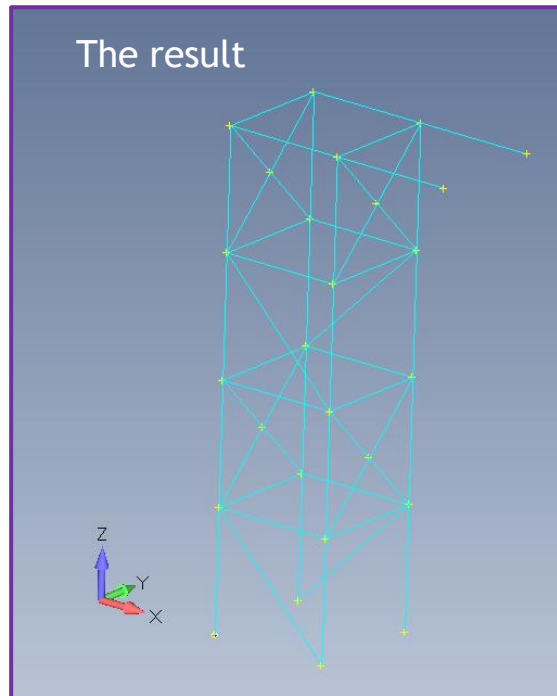
1 To execute previous command, right click on the graphical interface

2 Select *Previous Command…*

3 Repeat the steps 2-4 from the slide 17

This functionality is useful for a routine of modelling with identical structural elements, as the process doesn't require a constant incorporation of the Ribbon functionality.

https://sdcverifier.com

Model Plates
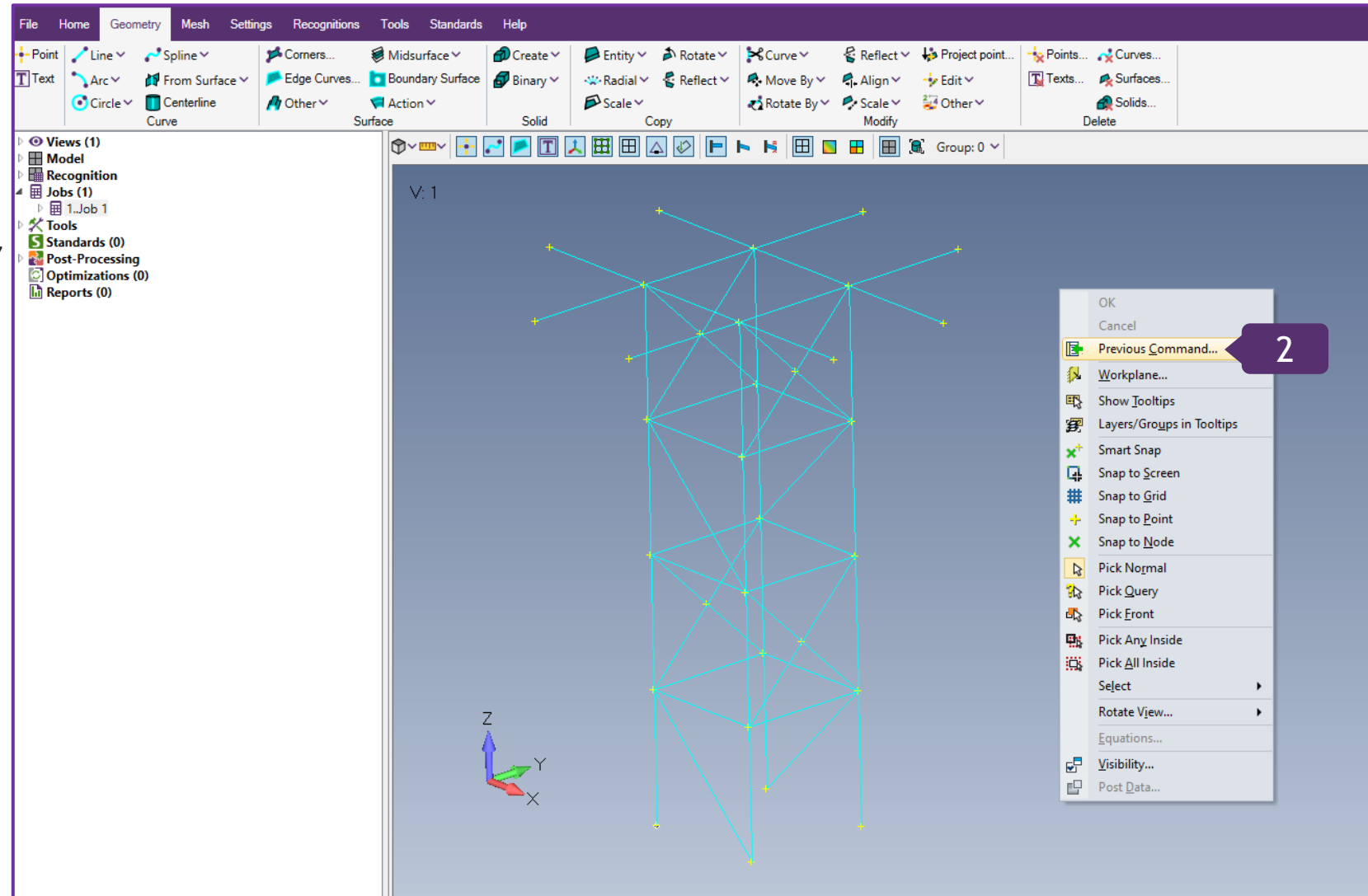
1 In *Surface* section, press on *Edge Curves...*

2 Select *3-sided* Shape

3 Select a field, which requires an ID or Coordinate

4 Select the Lines by clicking on them and press *OK*

The result
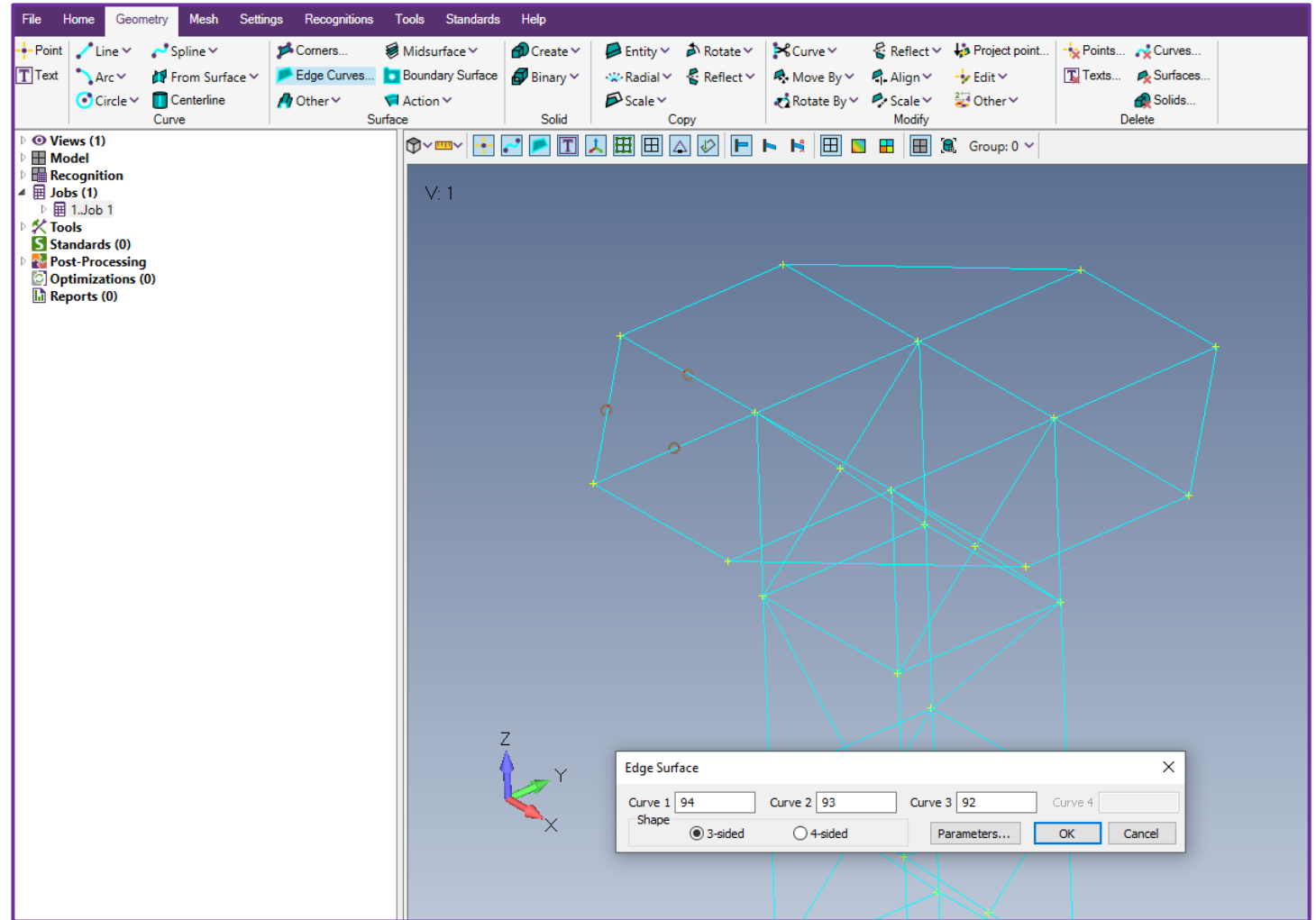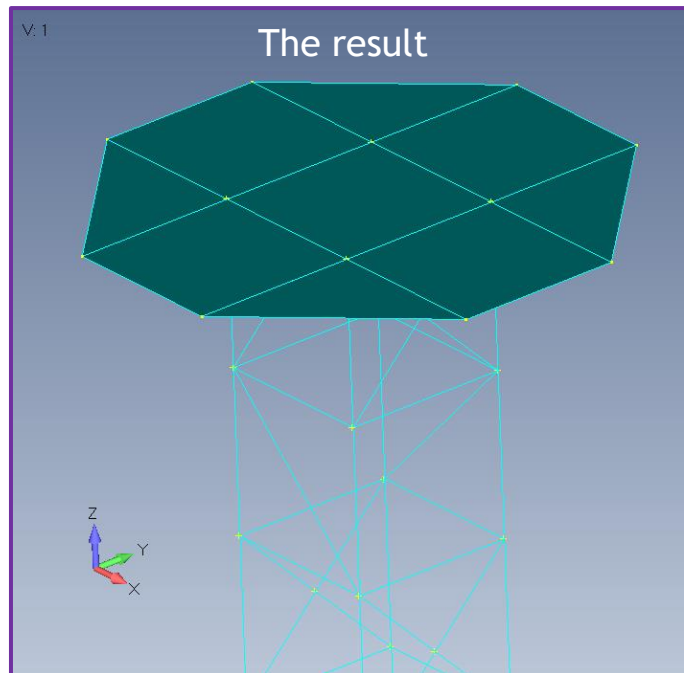
Depending on the number of sides a particular Shape has, it has to be selected before modelling a Plate. By default, 4-sided option is ON.

https://sdcverifier.com

Create Vertical Edges

1 In *Copy* section, press on *Entity* and select *Curves*

2 Select the Curves by clicking on them and press *OK*

3 In *Approach* section => *Single/Multiple Copies* => *Repetitions: 1*
Press *OK*

4 Fill in the coordinates:
Tip: X=0,Y=0,Z=0.3
Press *OK*

The result

23
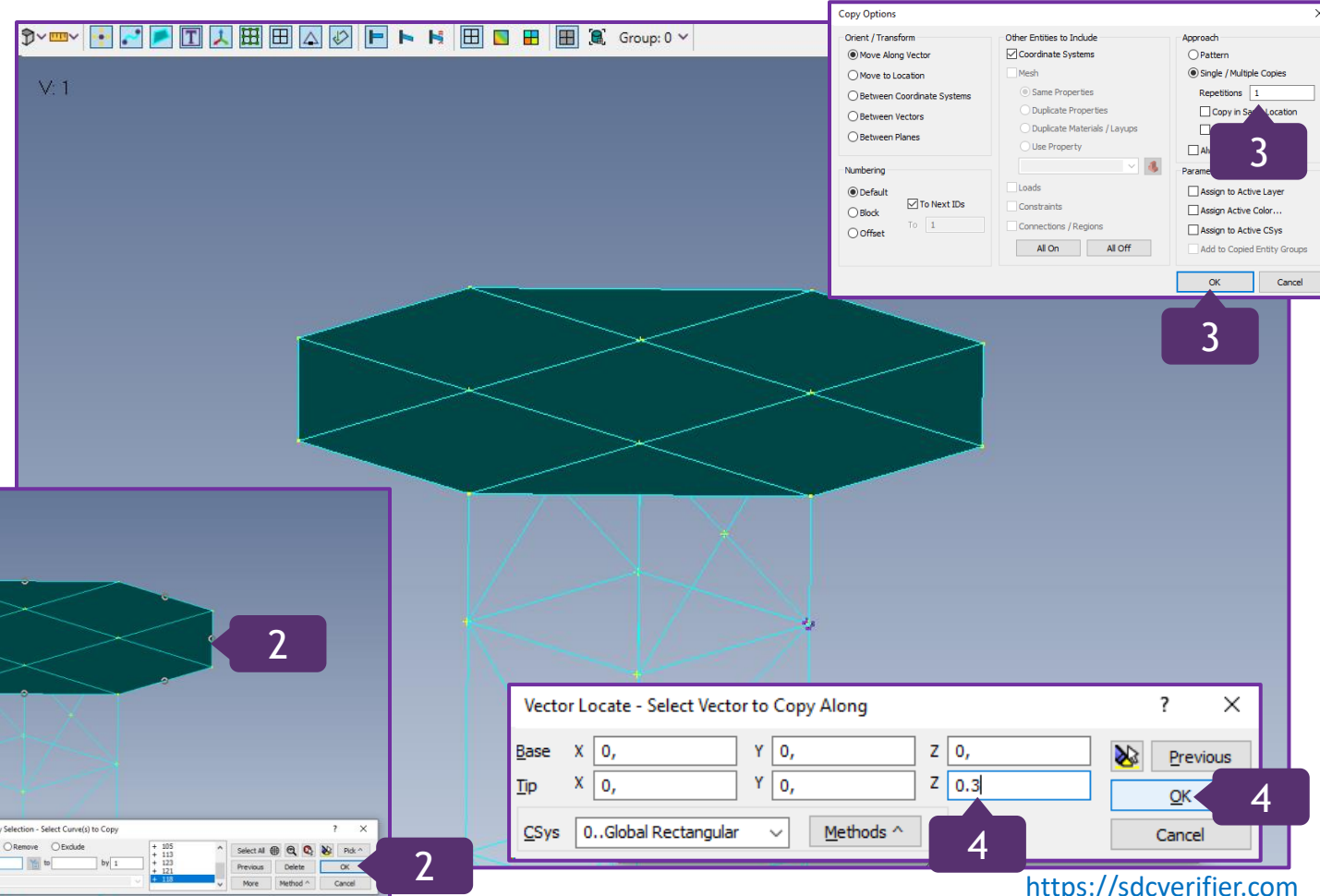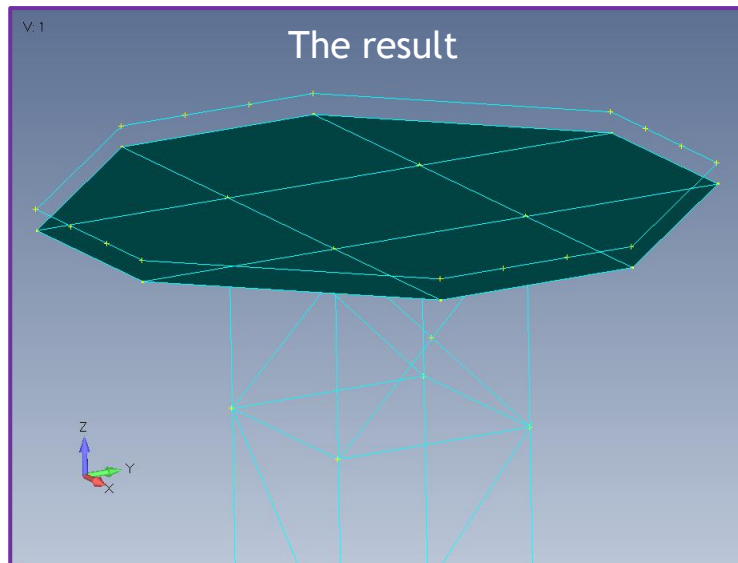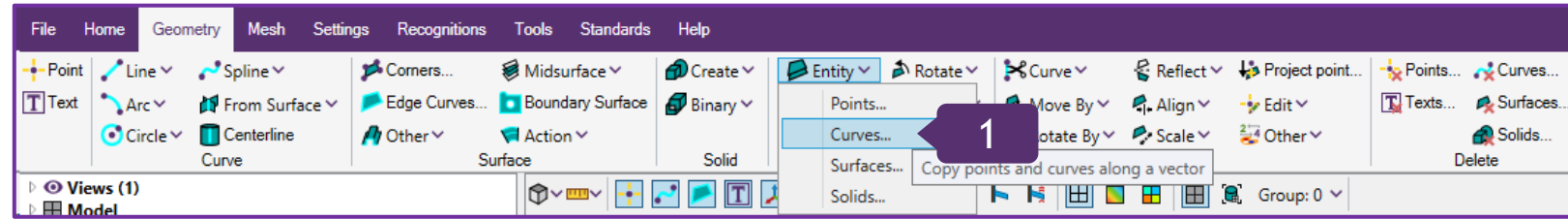
Create Vertical Surfaces by Corners

1 In *Surface* section, press on *Corners*…

2 Click on the Points of the Corners, following the logic of modelling Lines (slides 5-6)

3 After all Surfaces by Corners have been created, close the window

The result
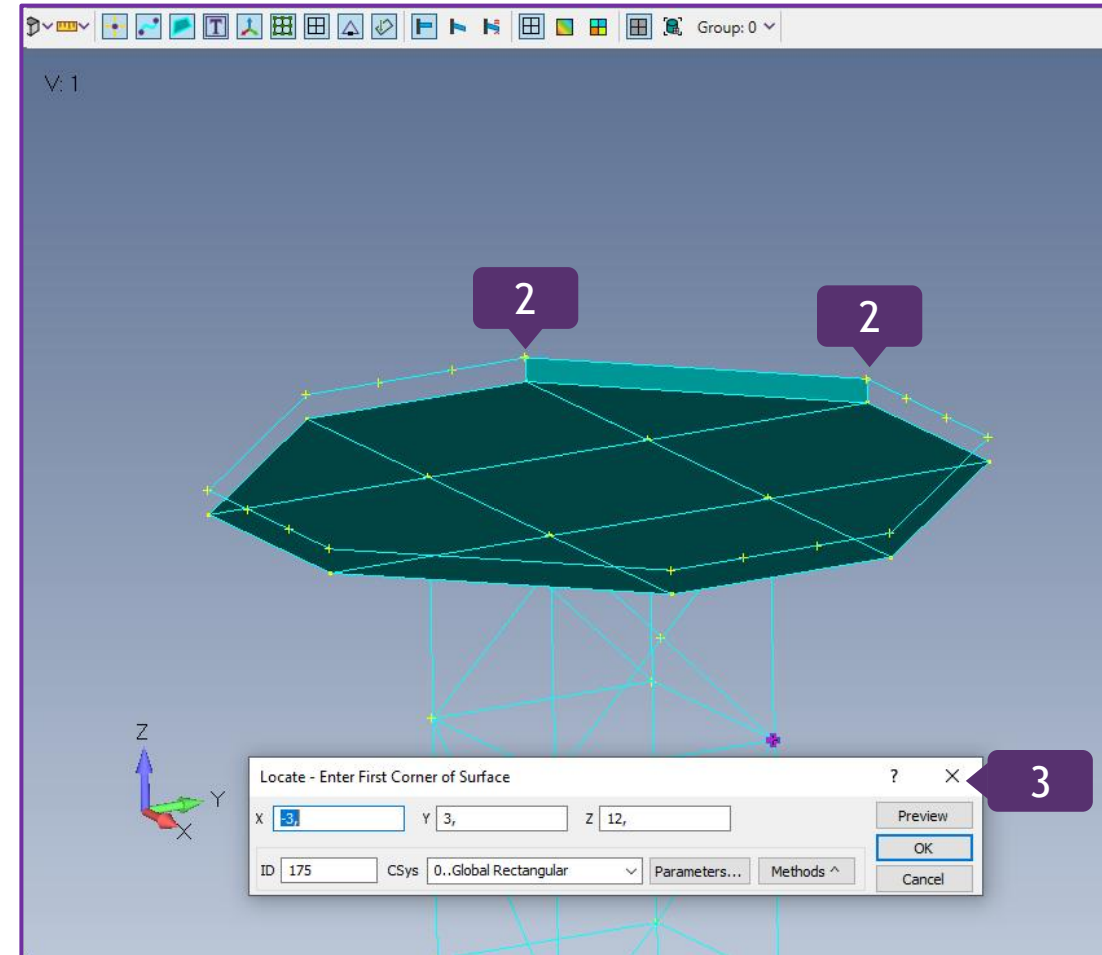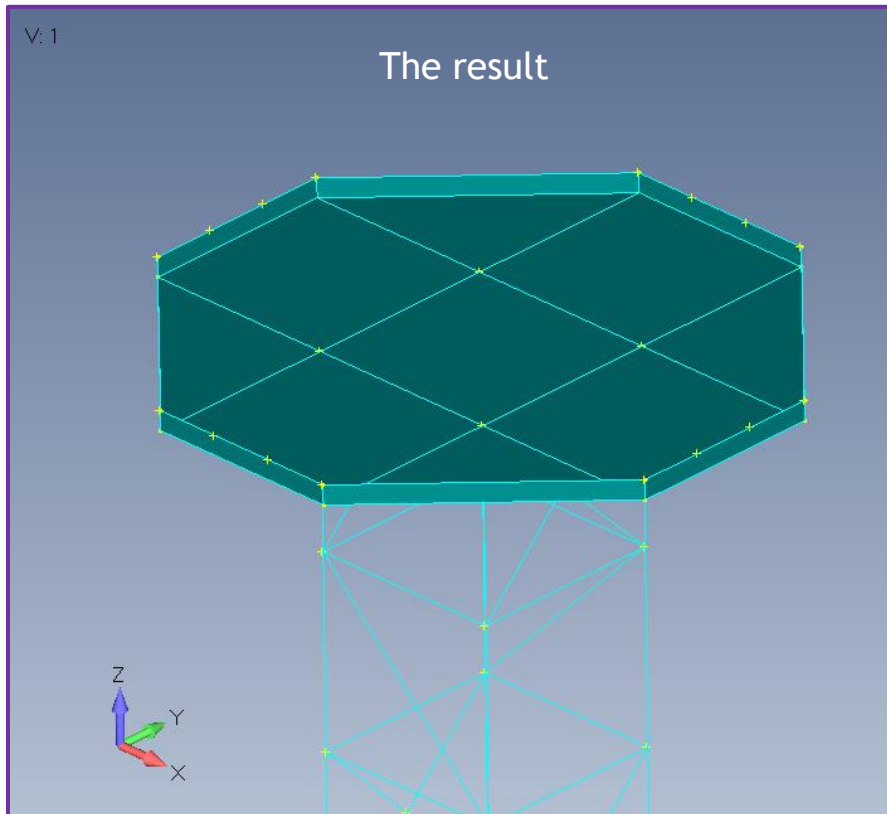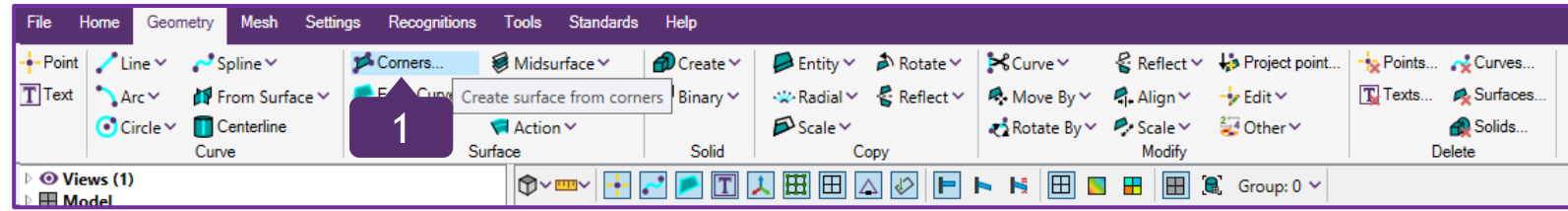
Define Materials for a Mesh

1 In *Mesh* tab, go to the Model Tree => *Model* and select *Materials*

2 Execute right click on *Materials* and select *Add*

3 Title: *Steel S240*

4 Young Modulus [Pa]: *210e9*

5 Poisson Ratio: *0.33*

6 Mass Density [kg/m^3]: *7850*

7 Tensile Strength [Pa]: *355e6*
Yield Stress [Pa]: *240e6*

8 Press *OK*

Add Properties for a Mesh

1 In *Mesh* tab, go to the Model Tree => *Model* and select Properties

2 Execute right click and select *Add*

3 In Material section, select previously defined *1..Steel S240*

4 To create *New Material*, press ⊞. If not currently relevant, press *Cancel*

> Define Material – ISOTROPIC menu allows to define even more parameters.

https://sdcverifier.com
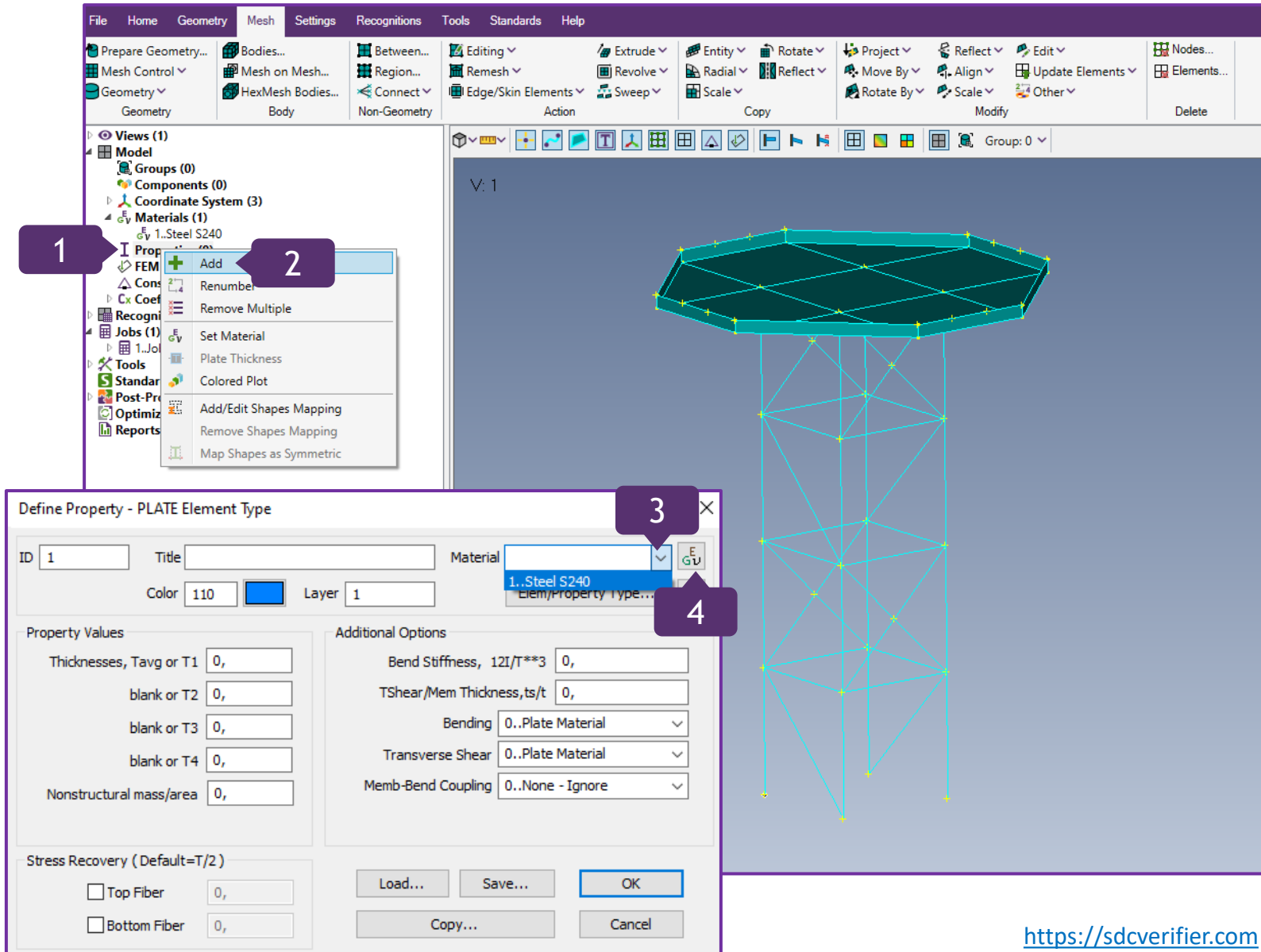
Define Beam Properties for a Mesh

1 Press on *Elem/Property Type...*

2 In Line Elements section, select *Beam* and press *OK*

3 Press on *Shape...* to create *Property*
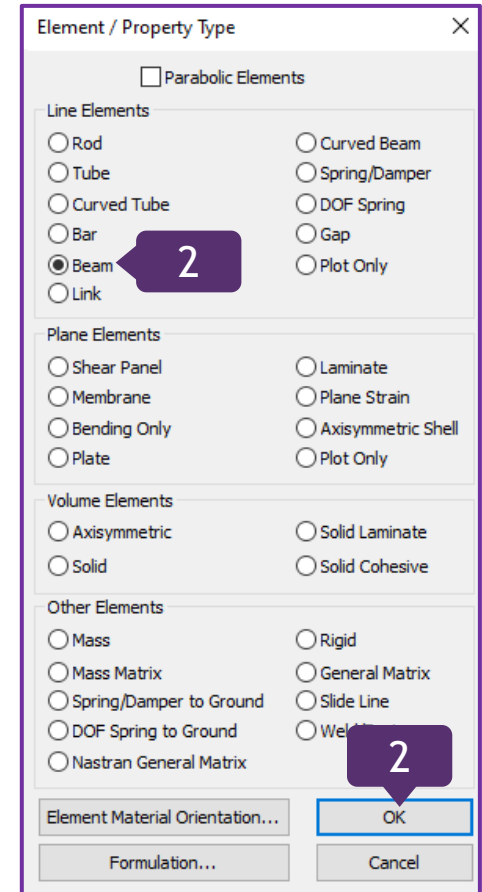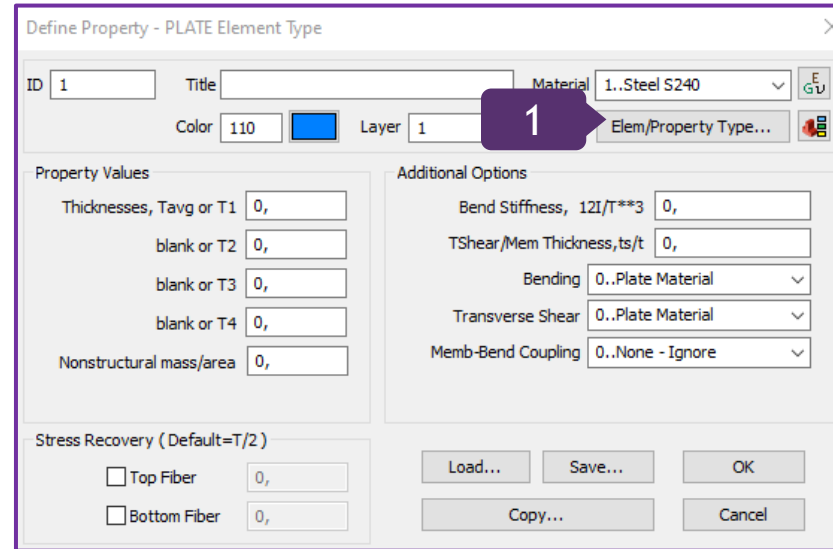
Note: All the parameters in Property Values and Stress Recovery (2 to 4 Blank=Square) can be defined manually.

https://sdcverifier.com

4 In *Shape* section, select *Rectangular Tube*

5 *Height* and *Width: 0.25*;
Thickness: 0.01
Press *OK*

6 Title: *Box250x10*
Press *OK*

7 Close *Define Property – BEAM Element Type* window

Note: All the parameters can be visible by clicking on one of the fields.

The calculation has been carried out.

The Property has been added.
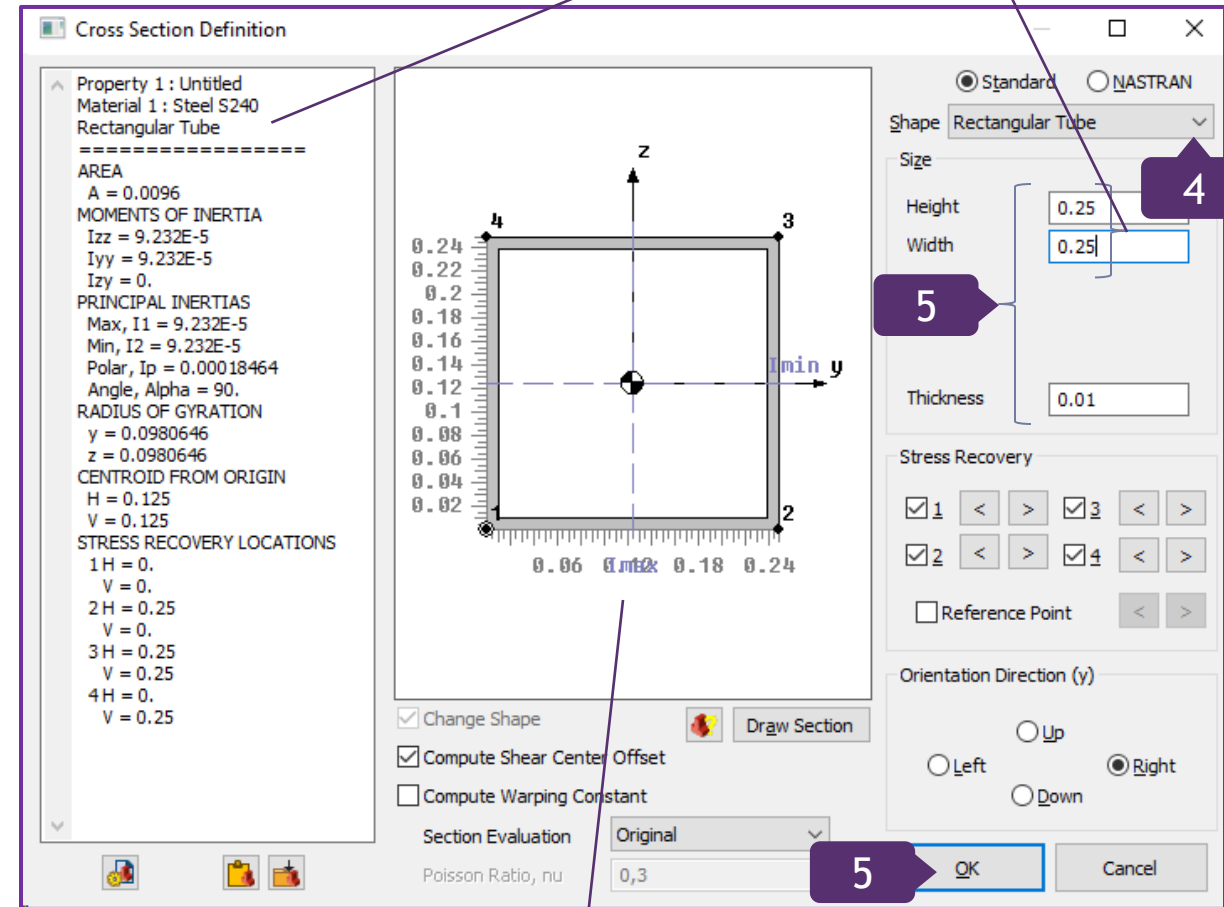
The cross-section can also be displayed.

Define the Number of Elements per Length

1 In *Mesh* tab, go to Mesh Control and select Size Along Curve...

2 *Select All* Curves.
Press *OK*

3 Press on *Element Size* and define **it**:
0.3 meters

4 Press *OK*.
Close the menu

The mesh size is displayed
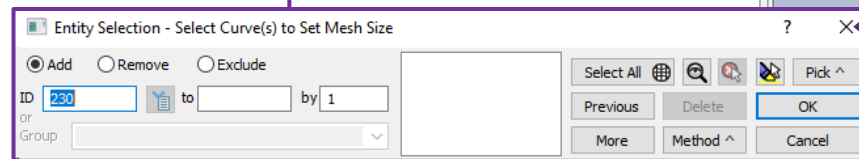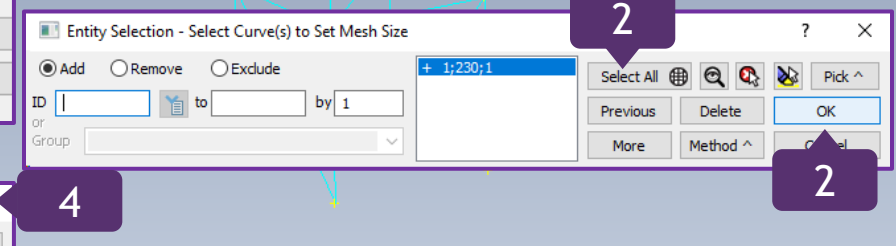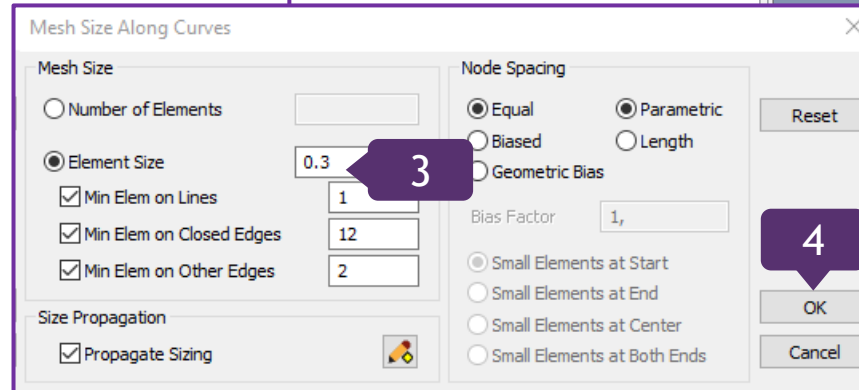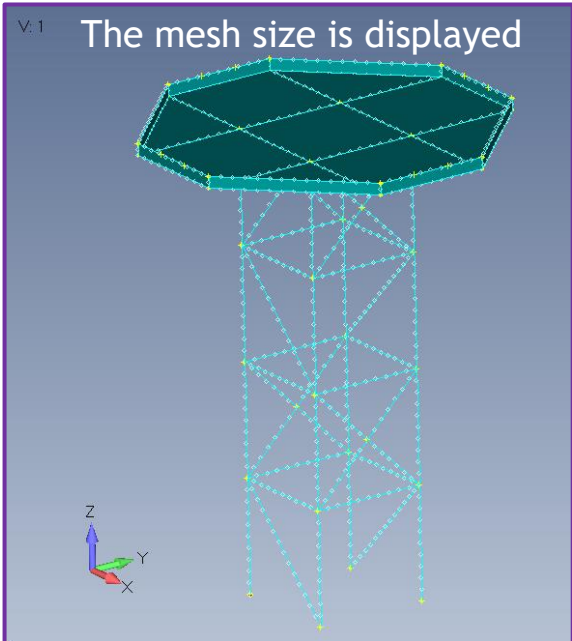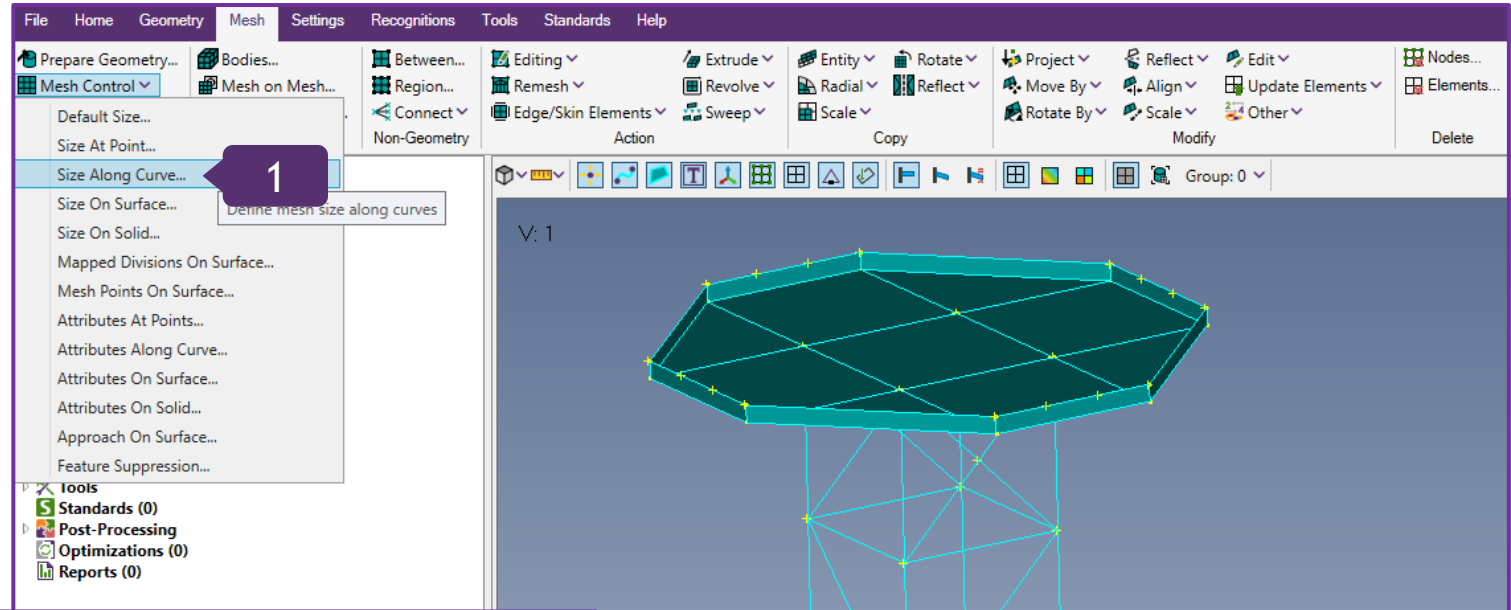
Mesh the created Lines

1 In *Mesh* tab, go to Geometry and select Curve…

2 With the left clicks of the mouse, select the lines

3 To highlight the selected lines, press

Highlighted selected Lines
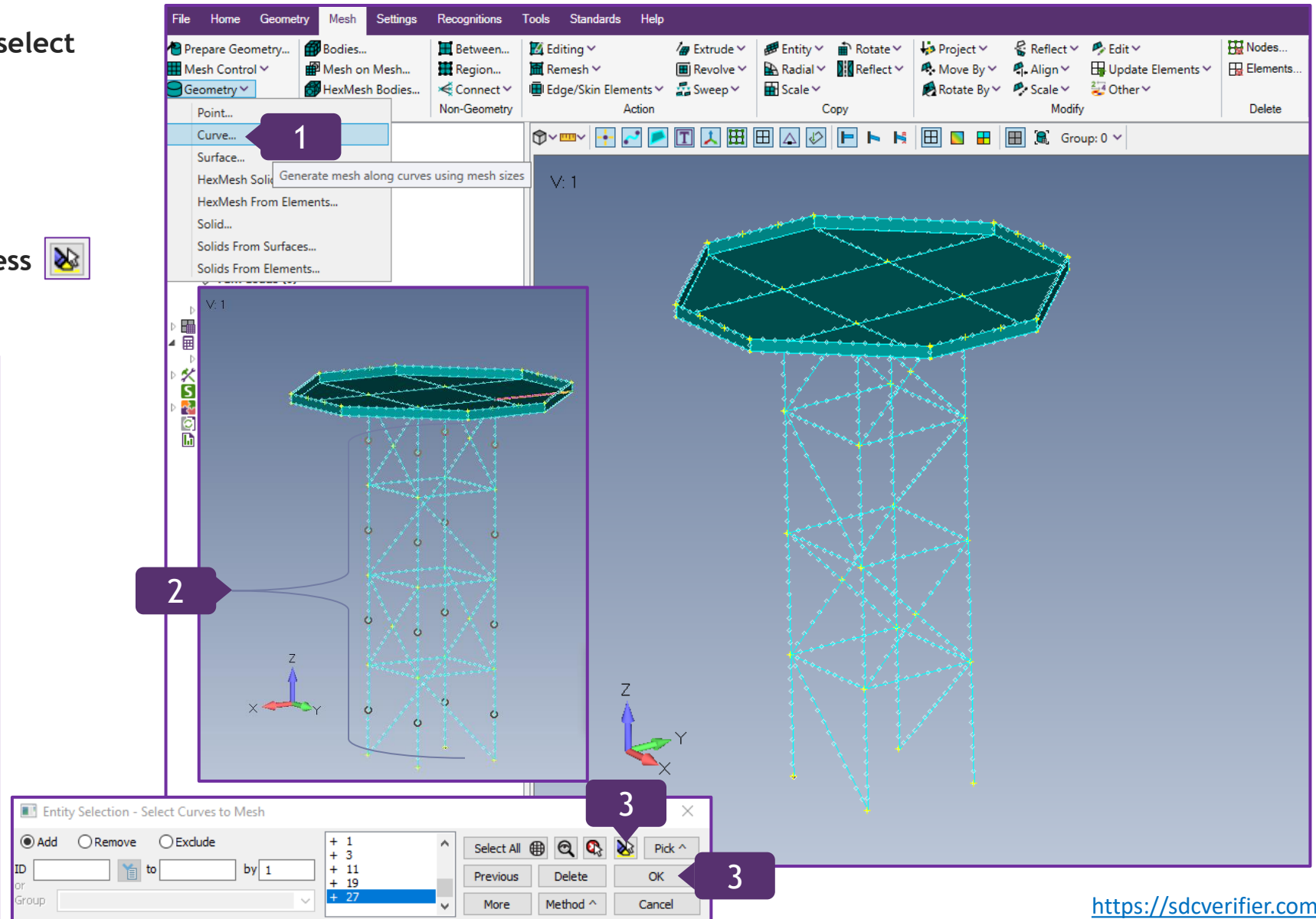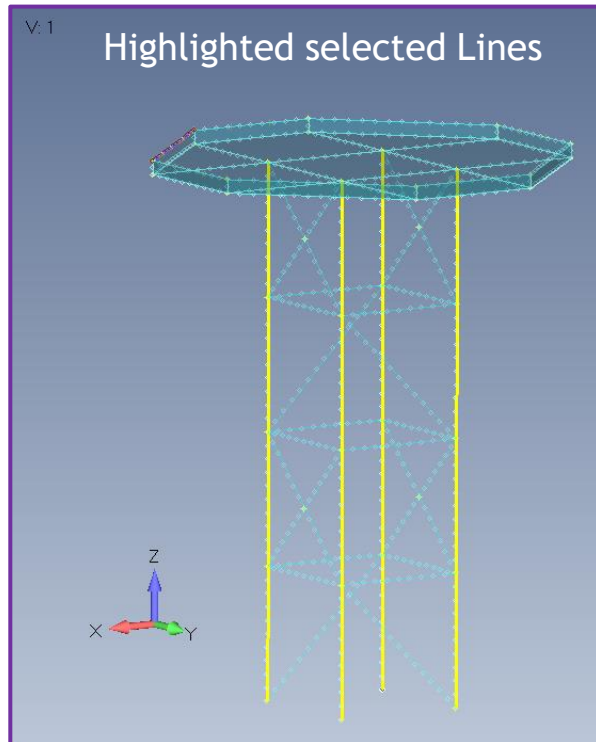
Mesh the created Lines (Continuation)

SDC VERIFIER

From this list, the Property with which the model will be meshed, can be created.

4   Press *OK*

5   *Tip:* X=1,Y=3,Z=0,
Press *OK*

6   To switch the thickness ON, press

7   Select *Thickness/Cross section*

Geometry Mesh Options

Node and Element Options
- Node ID: 1   CSys: 0..Global Rectangular   ☑ Merge Nodes
- Elem ID: 1   Property: 1..Box250x10

Generate
- ○ Nodes   ☐ Offset to Reference Point
- ⦿ Elements and Nodes

OK **4**
Cancel

Menu options:
- Hidden
- Solid
- ☑ Fill
- ☑ Filled Edges
- Feature Line
- Curve Connectivity
- ☑ Shading
- Transparency
- ☑ Mesh Size/Locations
- Shrink
- ☑ Offsets
- ☐ Orientation
- ☑ Thickness/Cross Section **7**
- Color With
- Workplane
- Clipping Plane
- Options

The created elements

V: 1

Vector Locate - Define Element Orientation Vector
- Base   X 0,   Y 0,   Z 0,
- Tip   X 1,   Y 0,   Z 0, **5**
- CSys 0..Global Rectangular   Methods

OK **5**
Cancel

Create Properties for Floors

1 In *Mesh* tab, go to the Model Tree => *Model* and select Properties

2 Execute right click and select *Add*

3 Press on *Shape...* to create *Property*

4 In *Shape* section, select *I-Beam or Wide Flange (W)*

5 Insert *the Size properties*, as shown on the screen. Press *OK*

6 Title: *I-beam200*. Press *OK*
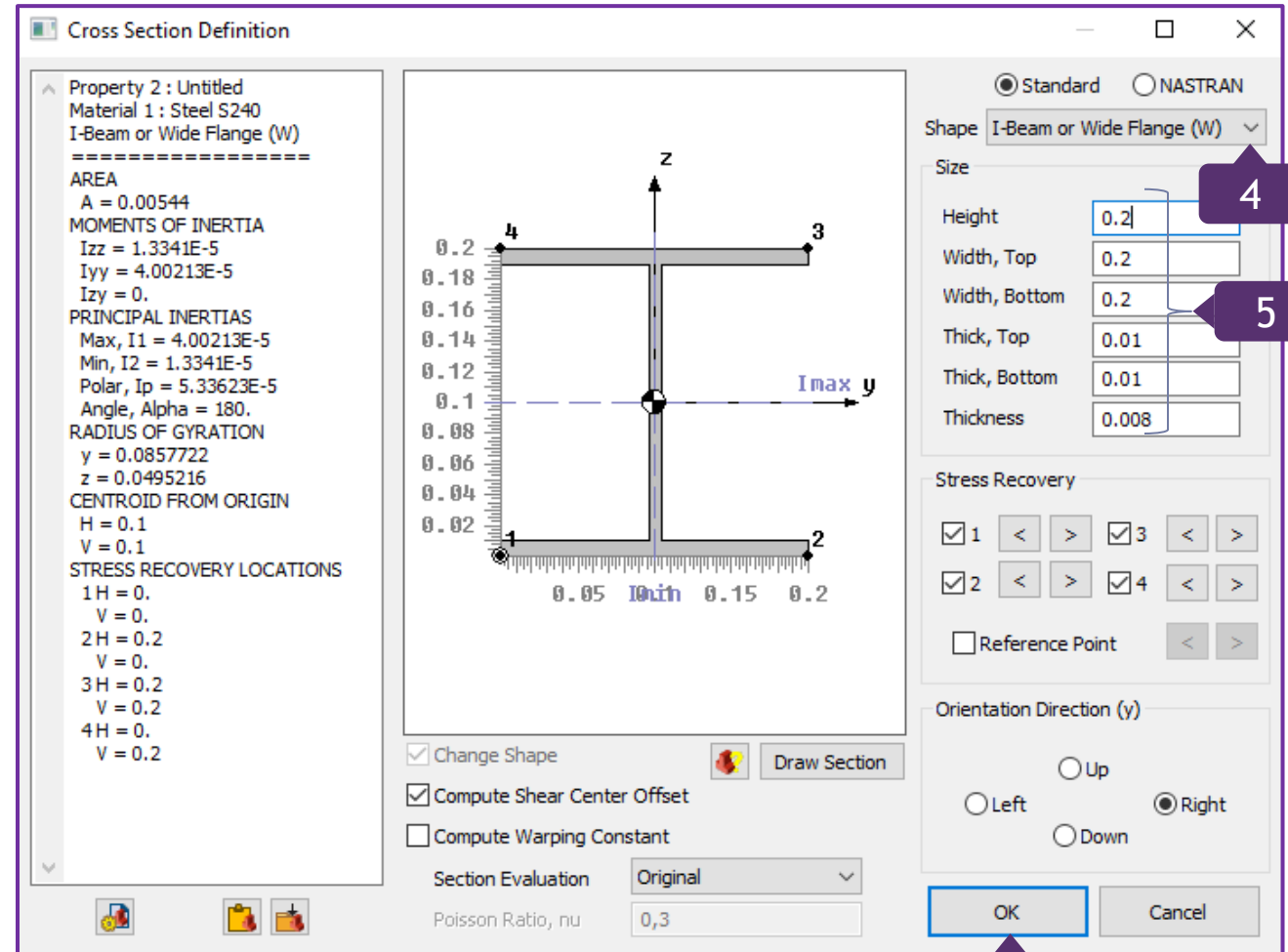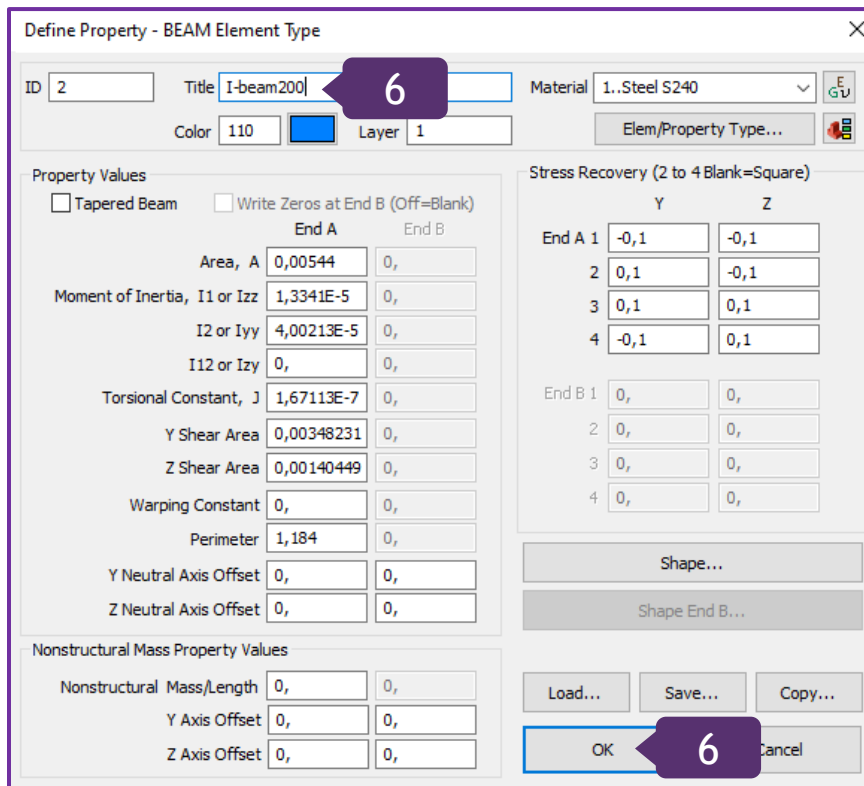
Create Properties for Diagonals

1 Press on *Shape*… to create *Property*

2 In *Shape* section, select *Circular Tube*

3 *Radius: 0.075;*
Thickness: 0.006
Press *OK*

4 Title: *Tube150x6*
Press *OK*

5 Close the Menu
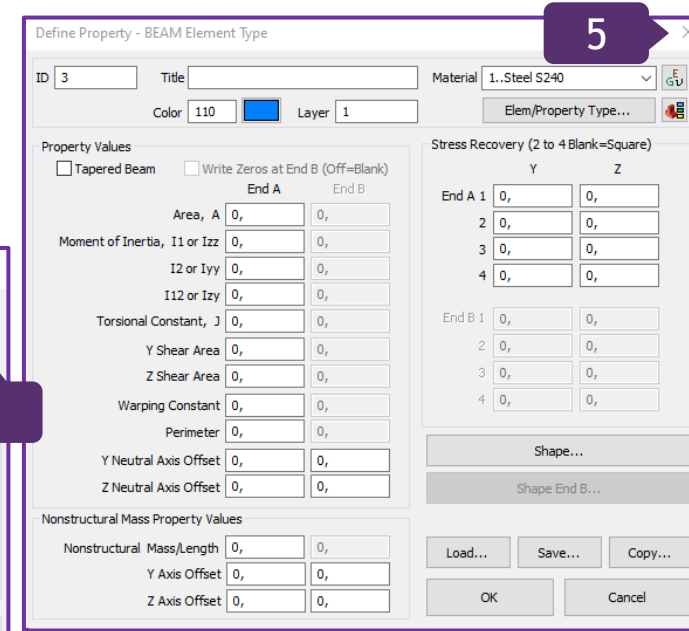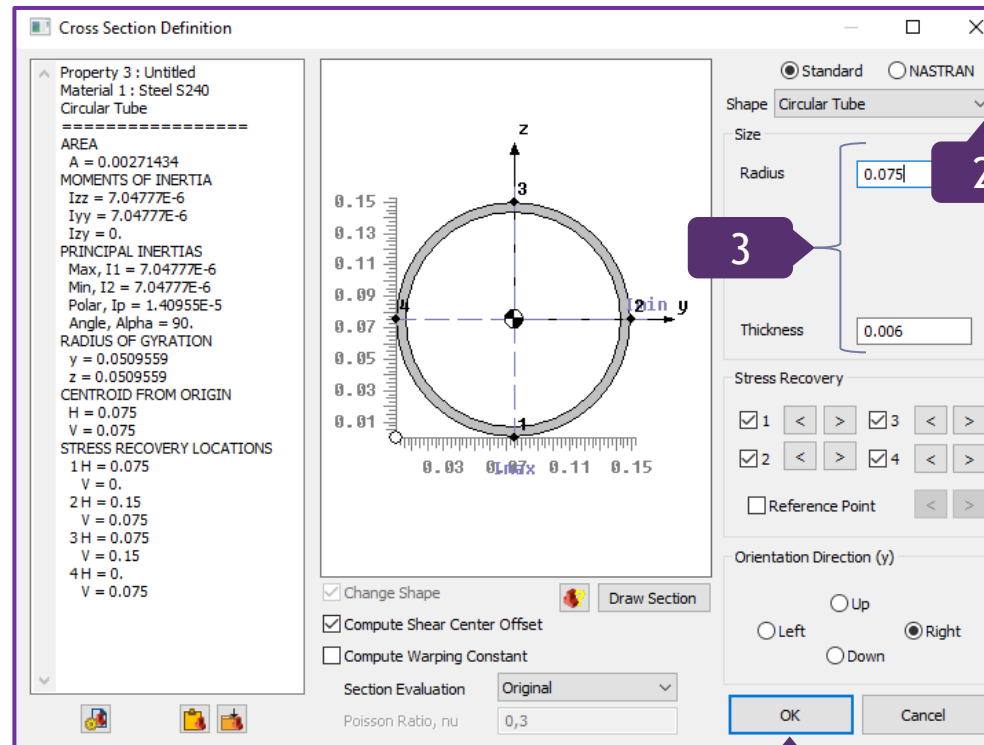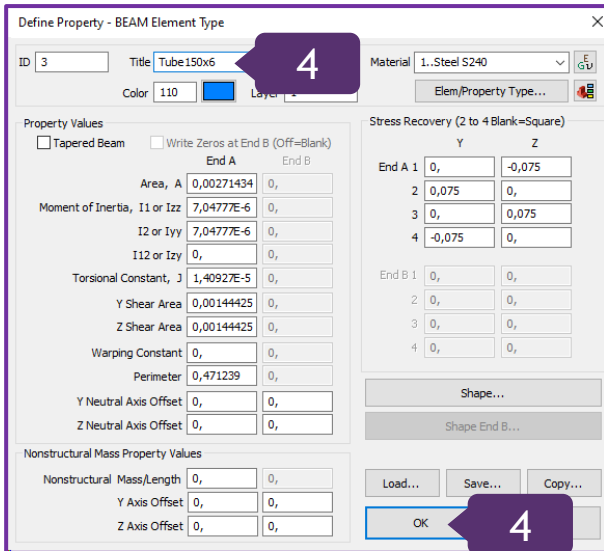
The Properties have been added.
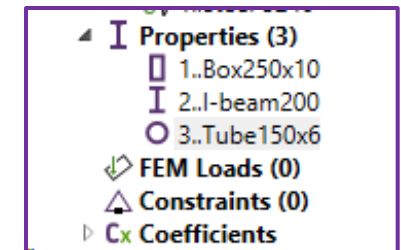
Mesh the Lines in Y Direction

1 In *Mesh* tab, go to Geometry and select Curve…

2 With the left clicks of the mouse, select the lines, that are aligned to X direction

3 Press *OK*

4 Property: *2..I-beam200*
Press *OK*

5 *Vector Locate: Tip:* X=0,Y=1,Z=0
Press *OK*

The reason for the Tip in Vector Location: X0,-Y1,-Z0, is that it is required the Vector to be perpendicular in Y direction.

The result

Mesh the Lines in X Direction

1 Right click on the graphical interface and select *Previous Command*…

2 With the left clicks of the mouse, select the lines, that are aligned to Y direction

3 Press *OK*

4 Property: *2..I-beam200*
Press *OK*

5 *Vector Locate: Tip:* X=1,Y=0,Z=0,
Press *OK*

The result

V: 1

1
OK
Cancel
1 Previous Command...
Workplane...
Show Tooltips
Layers/Groups in Tooltips
Smart Snap
Snap to Screen
Snap to Grid
Snap to Point
Snap to Node
Pick Normal
Pick Query
Pick Front
Pick Any Inside
Pick All Inside
Select
Rotate View...
Equations...
Visibility...
Post Data...

2

Geometry Mesh Options

Node and Element Options
Node ID: 249   CSys: 0..Global Rectangular   ☑ Merge Nodes
Elem ID: 225   Property: 2..I-beam200   **4**

Generate
○ Nodes
◉ Elements and Nodes

4 OK
Cancel

Entity Selection - Select Curves to Mesh
◉ Add   ○ Remove   ○ Exclude
ID [   ] to [   ] by [1]
or
Group
+ 5
+ 15
+ 13
+ 23
+ 21

Select All   Previous   More   Method ^   Pick ^   Delete   **3** OK

Vector Locate - Define Element Orientation Vector
Base   X [0,]   Y [0,]   Z [0,]   Previous
5 Tip   X [1,]   Y [0,]   Z [0,]   **5** OK
CSys   0..Global Rectangular   Methods ^   Cancel

Mesh the Diagonals

1 In *Mesh* tab, go to Geometry and select Curve...

2 With the left clicks of the mouse, select the diagonal lines, including the ones, divided by nodes. Press *OK*

3 Property: *3..Tube150x6*
Press *OK*

4 *Vector Locate: Tip:* X=1,Y=0,Z=0,
Press *OK*

The result

37

View Style Functions
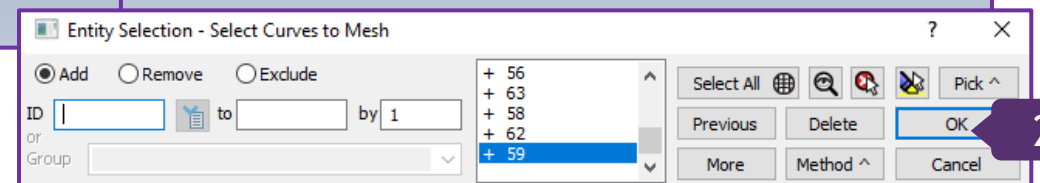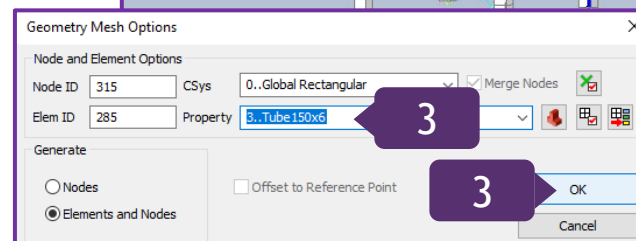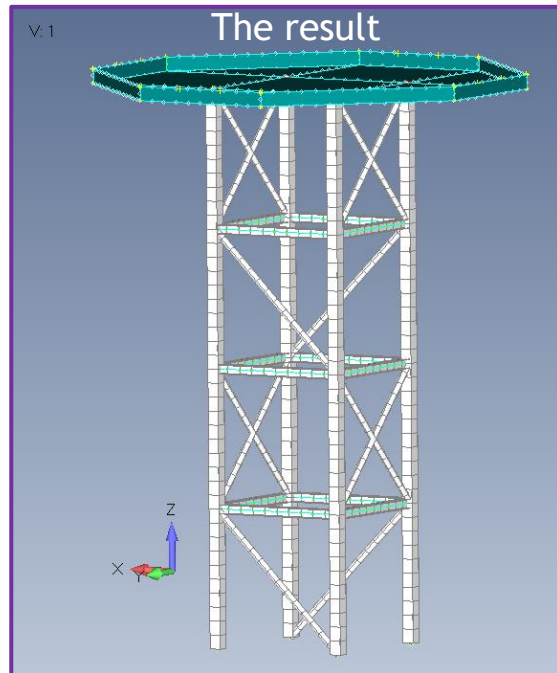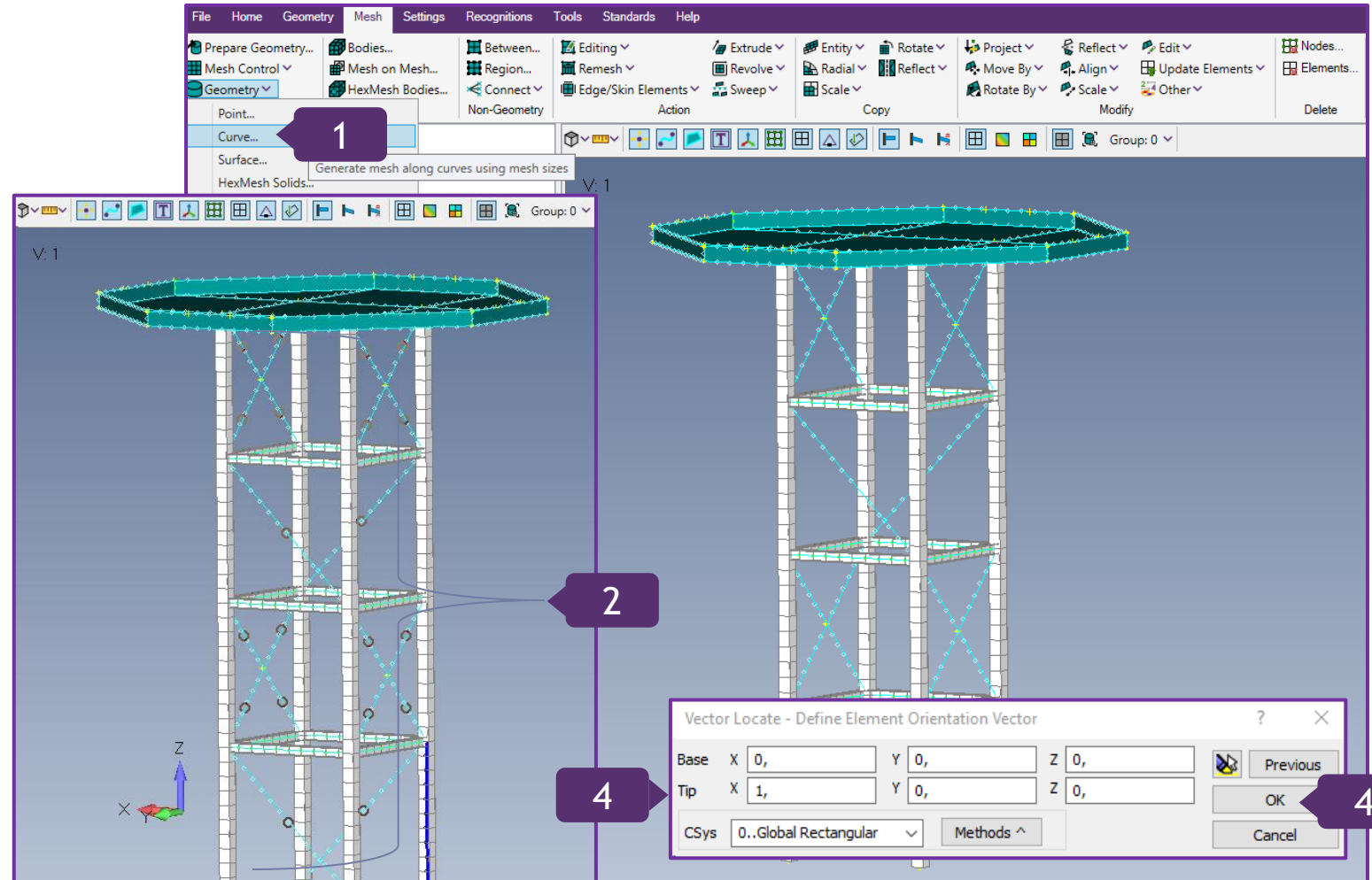
1 Press on [icon] => **Color With and select** *Property Colors*
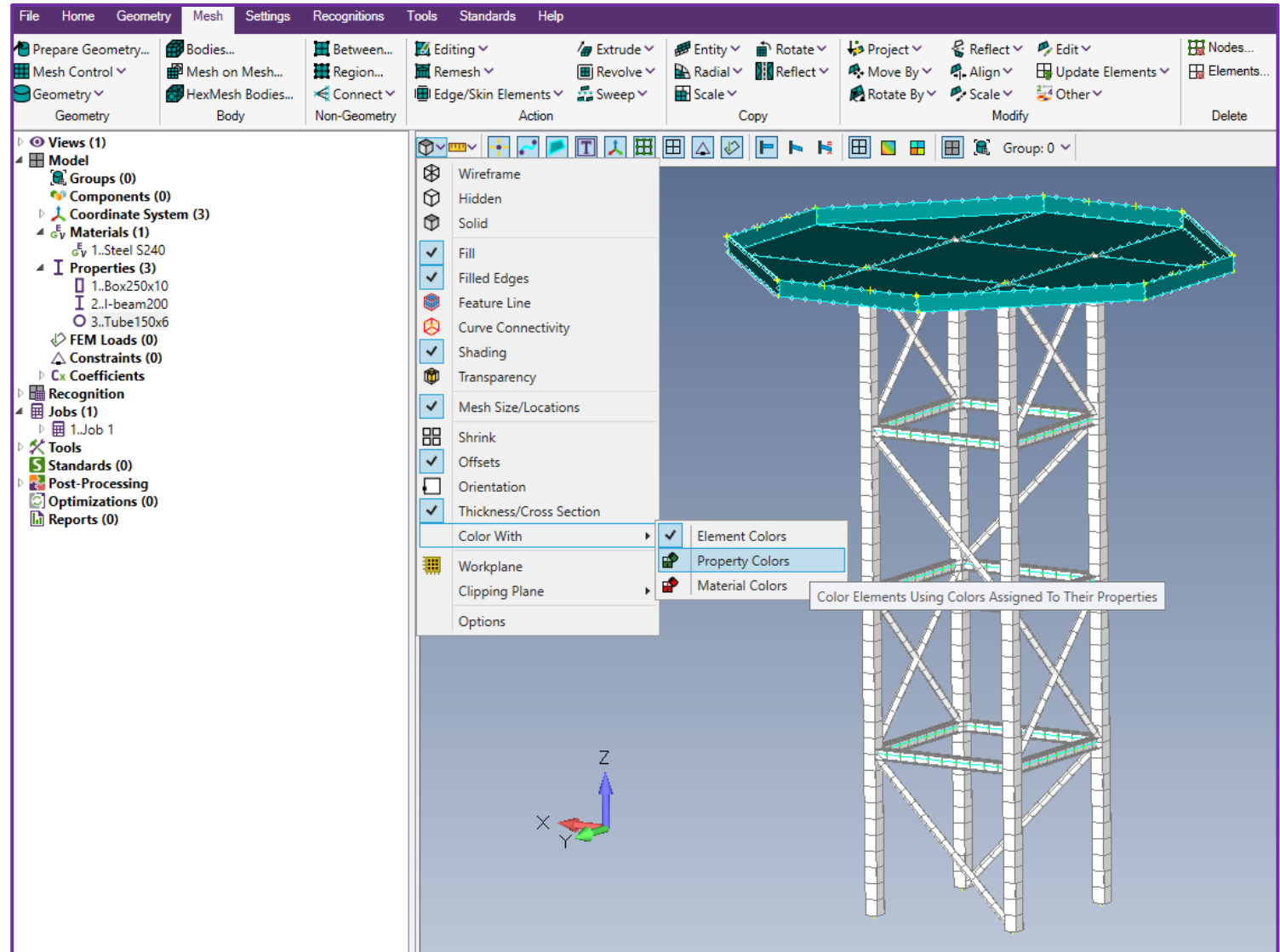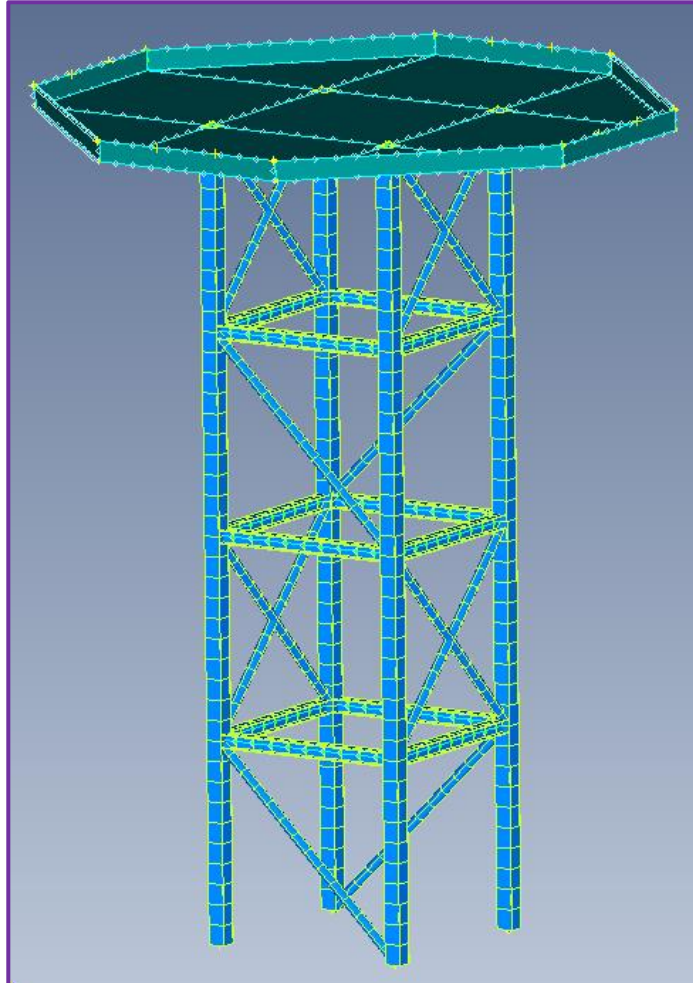
Edit the Properties Colors

Note: any color of the choice may be used, when editing the Properties Colors.

1 To edit the Colors, in Model Tree go to *Properties => 1..Box250x10* and with a right click, select *Edit*

2 Press on

3 In *Color Palette* menu, select Green #128
Press *OK*

4 Press *OK*

For 2..I-beam200 Property, execute steps 1-4 from the current slide. The selected color is Red #192

The result

39

SDC VERIFIER
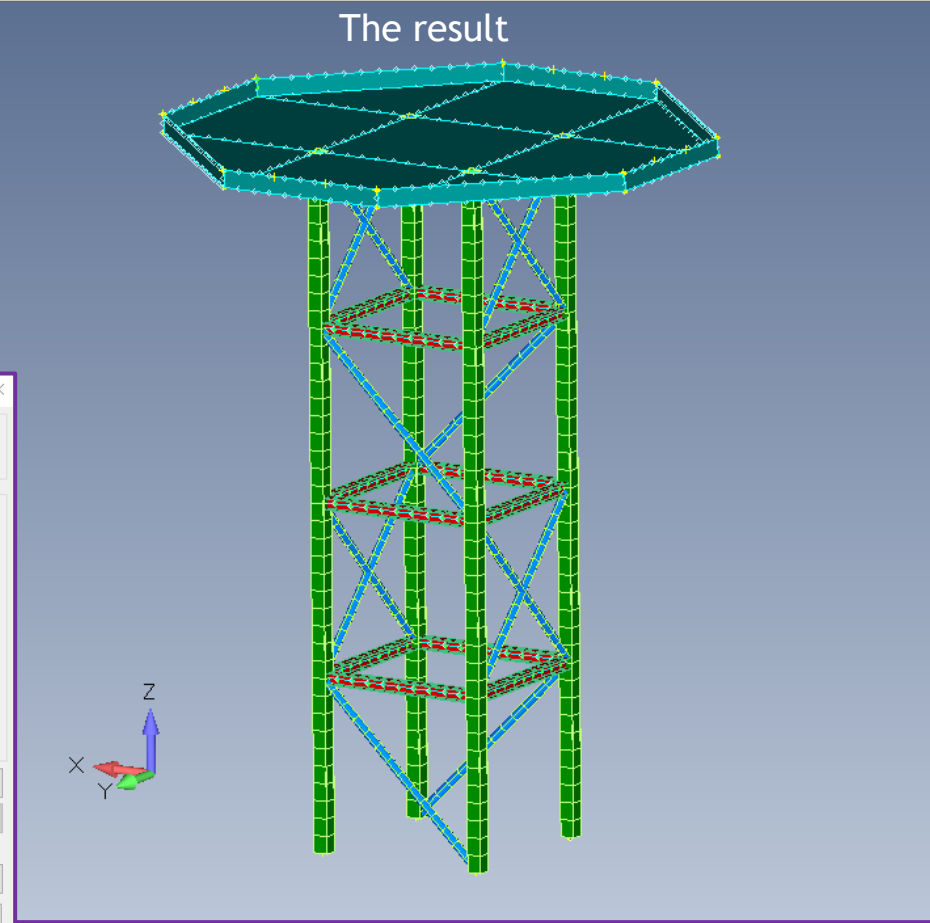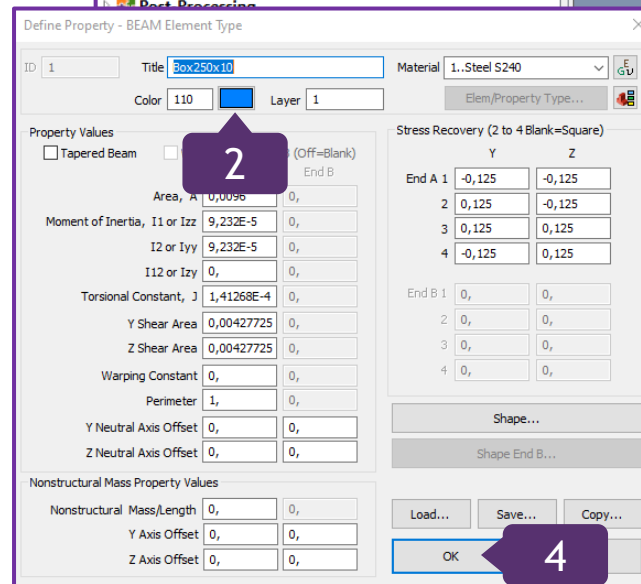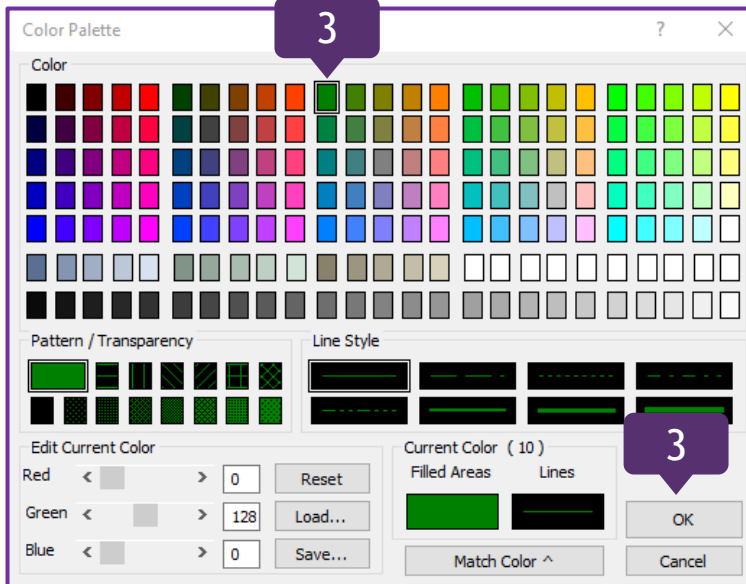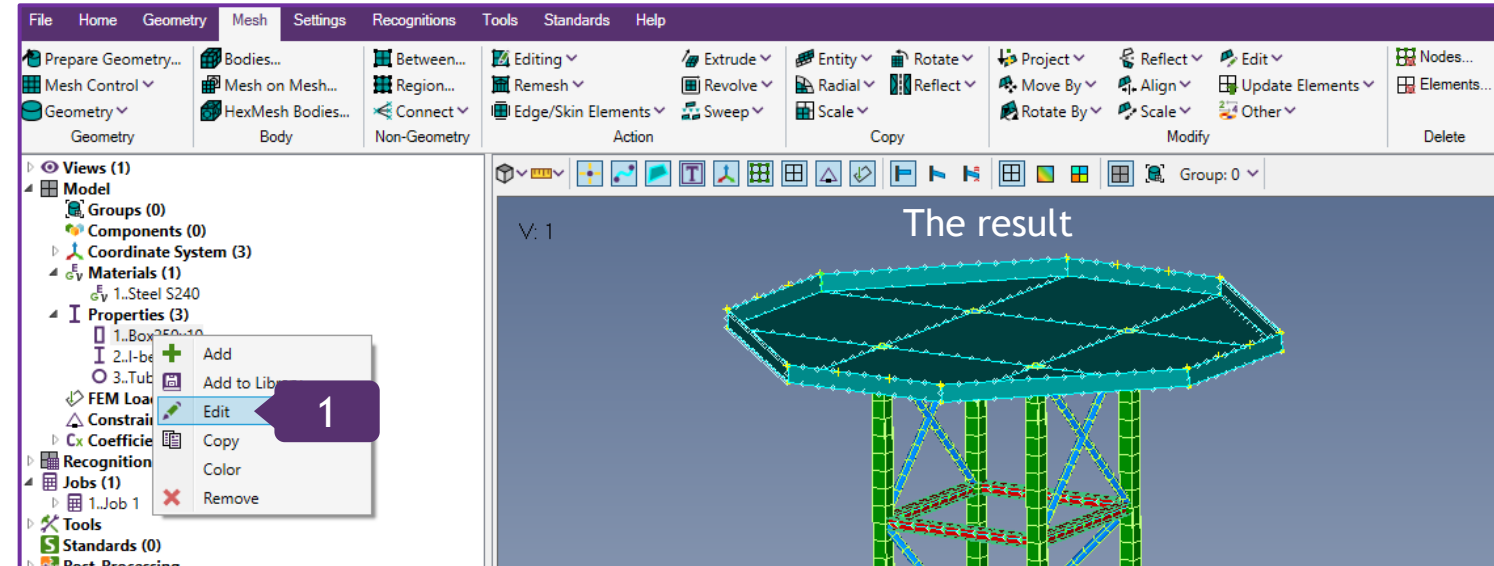
1 Press on [⬡⌄] and learn some of the
View Style functions

1

Icon	Function
⬡	Wireframe
⬡	Hidden
⬡	Solid
✓	Fill
✓	Filled Edges
⬡	Feature Line
⬡	Curve Connectivity
✓	Shading
⬡	Transparency
✓	Mesh Size/Locations
▦	Shrink
✓	Offsets
☐	Orientation
✓	Thickness/Cross Section
	Color With ▸
▦	Workplane
	Clipping Plane ▸
	Options

Filled Edges
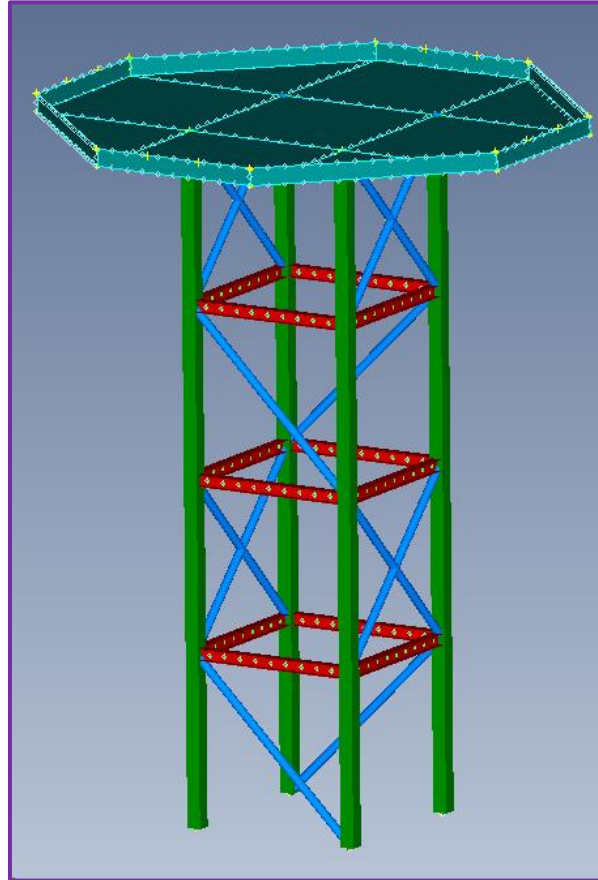
Transparency
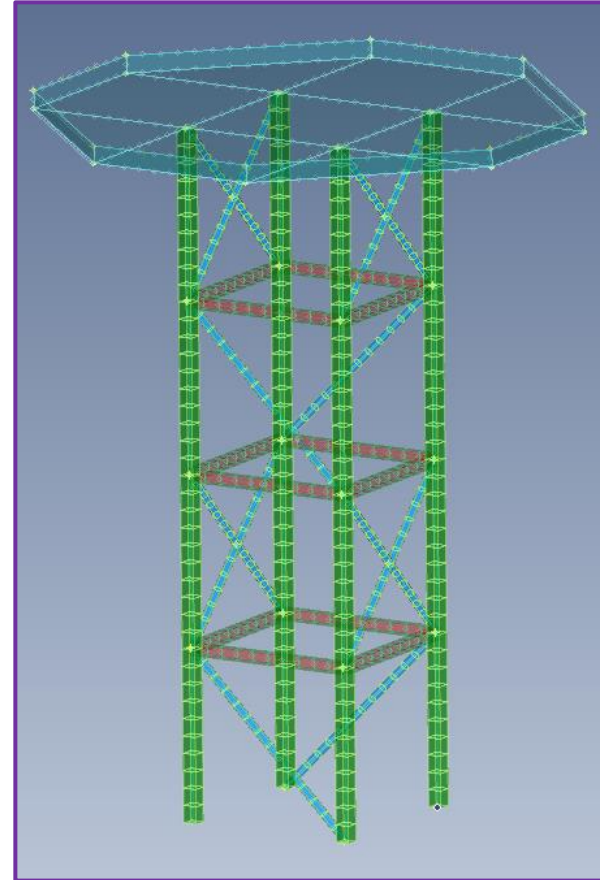
Wireframe
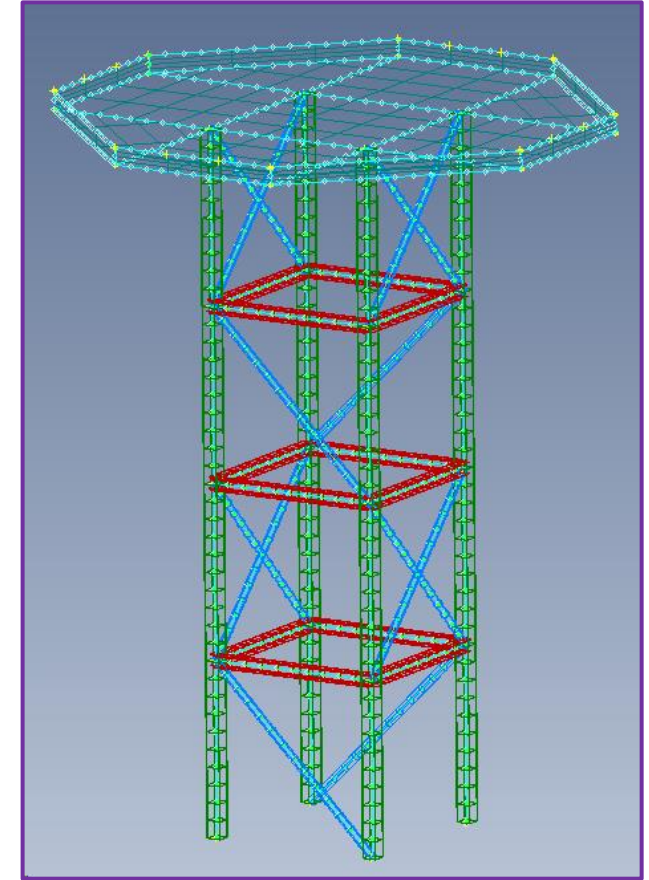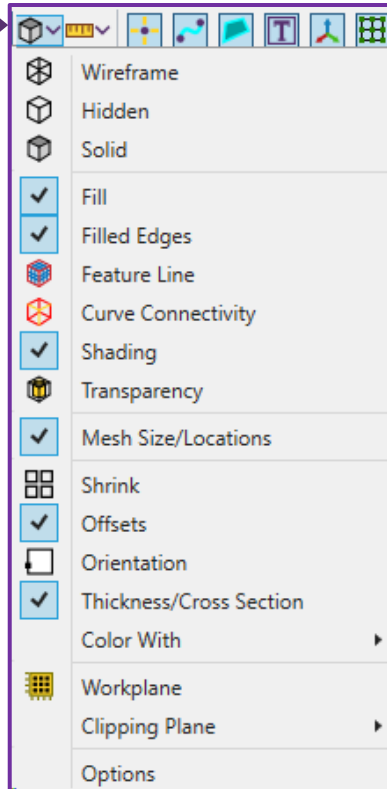
Mesh the Plates

1 In *Mesh* tab, go to Geometry and select *Surface…*

2 With the left clicks of the mouse, select the plates of the surface

3 Press ▨ to check if all necessary Plates have been selected

4 Press *OK*

Highlighted Plates

V: 1
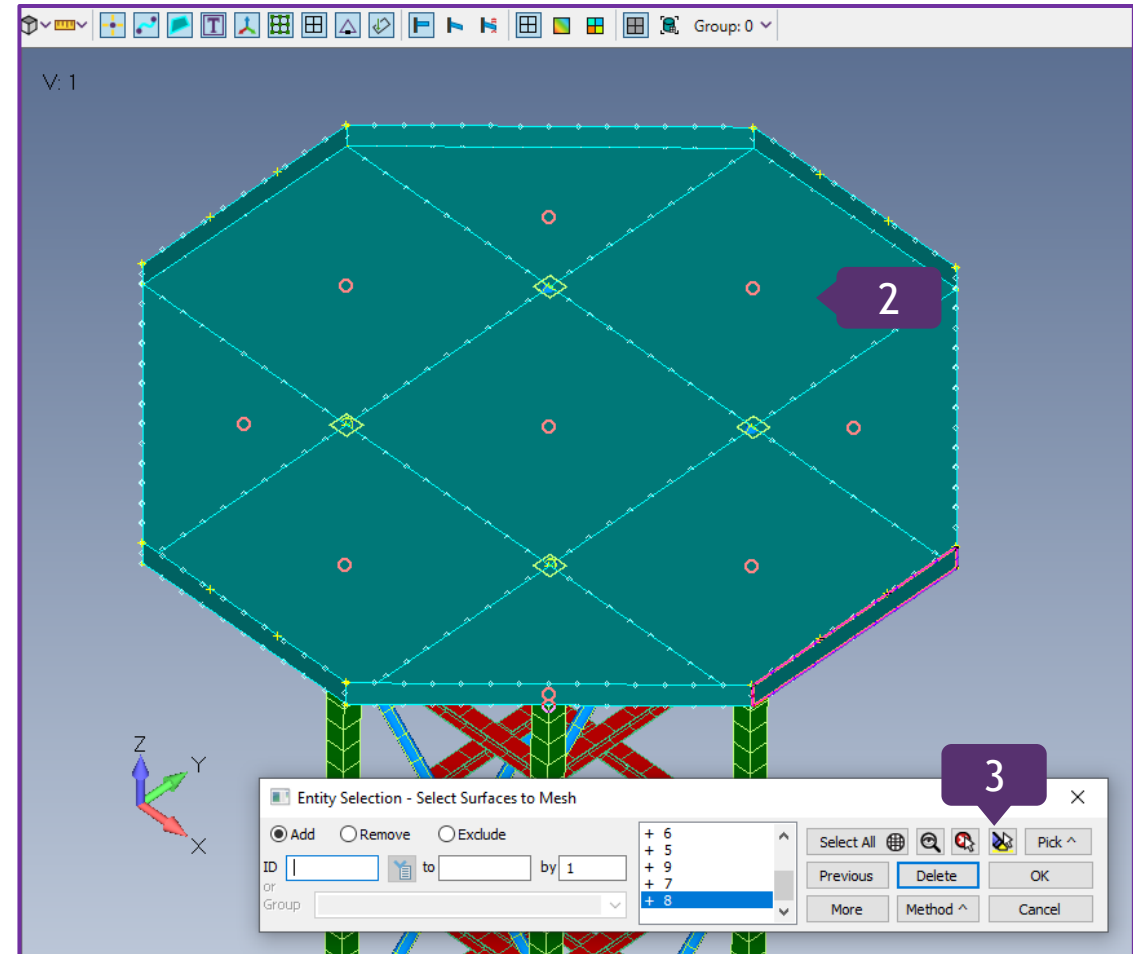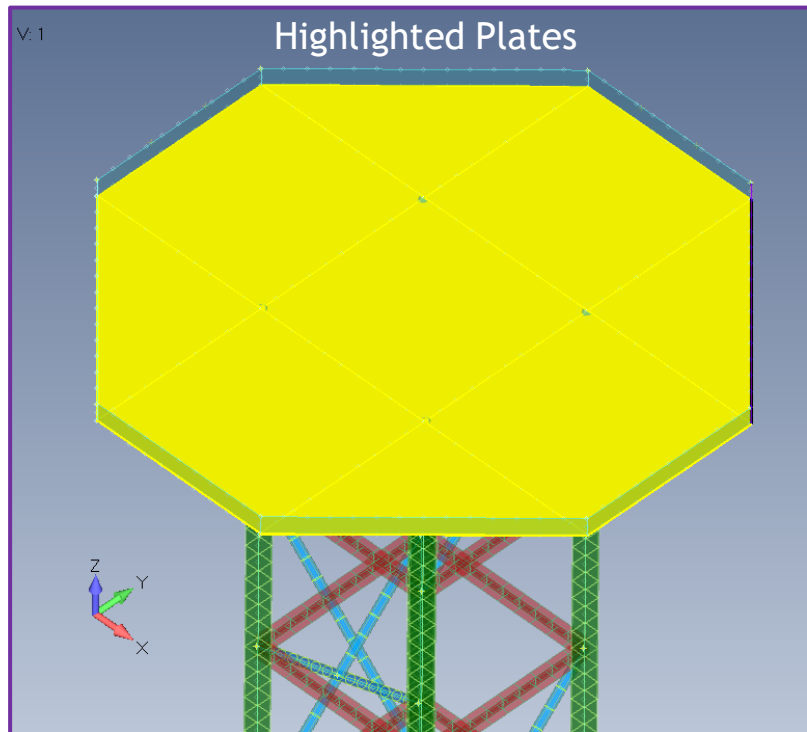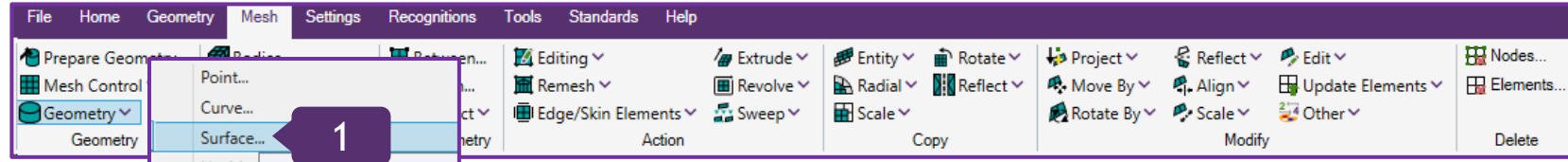
Alternative Method of Creating New Property

SDC VERIFIER

1 Click on [icon] to create new *Property*

2 Title: *10mm plate*
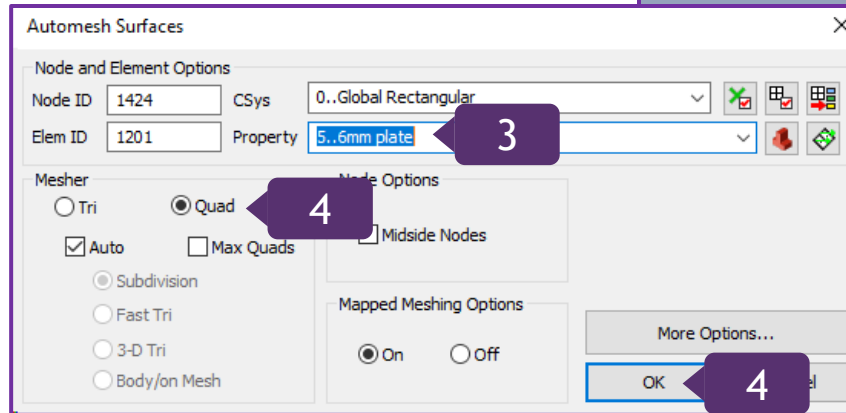Press *OK*

3 Thickness, Tavg or T1: 0.01;
Press *OK*

The result

Automesh Surfaces

Node and Element Options
Node ID: 503   CSys: 0..Global Rectangular
Elem ID: 453   Property: [blank]   **1**

Mesher
○ Tri   ● Quad
☑ Auto   ☐ Max Quads
○ Subdivision
○ Fast Tri
○ 3-D Tri
○ Body/on Mesh

Node Options
☐ Midside Nodes

Mapped Meshing Options
● On   ○ Off

More Options...
OK   Cancel

Define Property - PLATE Element Type
ID 4   Title: 10mm plate   **2**   Material: 1..Steel S240
Color: 110   [blue]   Layer: 1   Elem/Property Type...

Property Values
Thicknesses, Tavg or T1: 0.01   **3**
blank or T2: 0,
blank or T3: 0,
blank or T4: 0,
Nonstructural mass/area: 0,

Additional Options
Bend Stiffness, 12I/T**3: 0,
TShear/Mem Thickness,ts/t: 0,
Bending: 0..Plate Material
Transverse Shear: 0..Plate Material
Memb-Bend Coupling: 0..None - Ignore

Stress Recovery (Default=T/2)
☐ Top Fiber: 0,
☐ Bottom Fiber: 0,

Load...   Save...   OK   **2**
Copy...   Cancel

Automesh Surfaces

Node and Element Options
Node ID: 503   CSys: 0..Global Rectangular
Elem ID: 453   Property: 4..10mm plate

Mesher
○ Tri   ● Quad
☑ Auto   ☐ Max Quads
○ Subdivision
○ Fast Tri
○ 3-D Tri
○ Body/on Mesh

Node Options
☐ Midside Nodes

Mapped Meshing Options
● On   ○ Off

More Options...
OK   Cancel   **3**

Update the Model

1   In *Home* tab, press Update Model

2   Select *Continue with existing results* option

3   Press *OK*
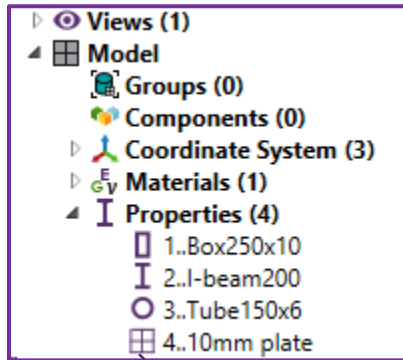
The Property has been added

The Update Model command is used to update parameters which are referenced by existing results.

Update Model

SDC Verifier will update model data from SDC Verifier 2023 R2.

If mesh was changed in your model it is recommended to clear results.

○ Clear calculated results

○ Remove links to Output Sets and clear (import results manually or reanalyze is required)

● Continue with existing results (e.g. new loads / constraints were added)

OK   Cancel
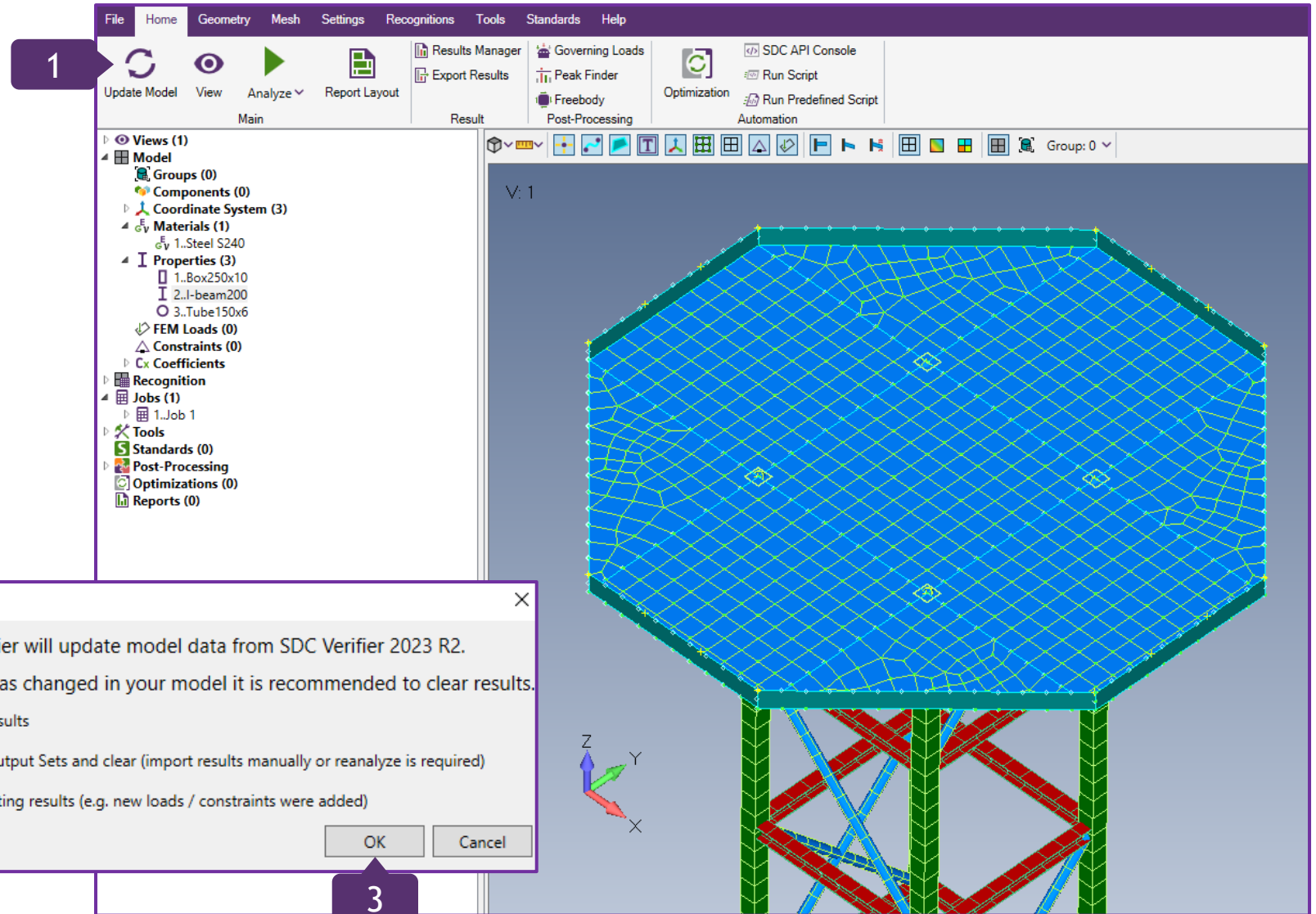
Create Property for Side Plates

1 In *Mesh* tab, go to the Model Tree => *Model* and select Properties

2 Execute right click and select *Add*

3 Title: *6mm plate*

4 Thicknesses, Tavg or T1: 0.006

5 Press *OK* and close the window

The alternative method of creating Properties from slide 39 may also be incorporated.
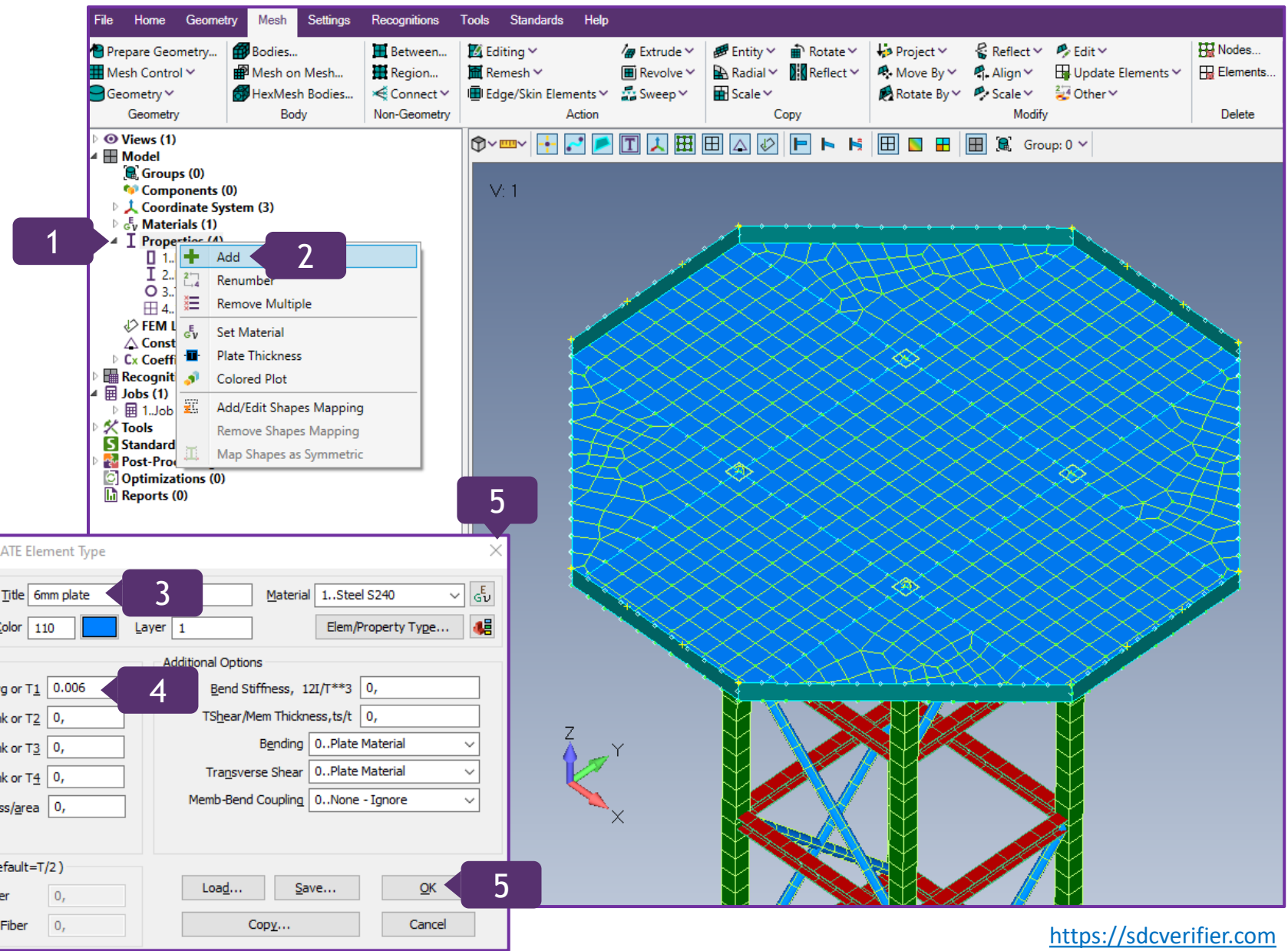
Mesh the Side Plates

1 In *Mesh* tab, go to Geometry and select *Surface…*

2 With the left clicks of the mouse, select all plates of the side surface. Press *OK*

3 Select Property: *5..6mm plate*

4 In *Mesher section*, *Quad* is *ON*. Press *OK*

The result
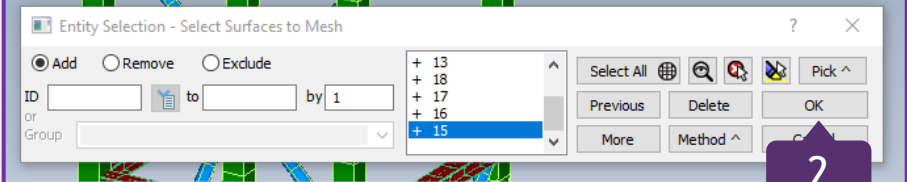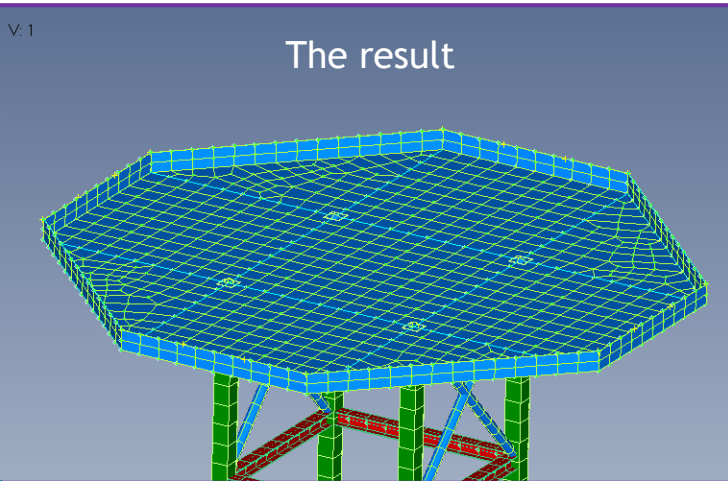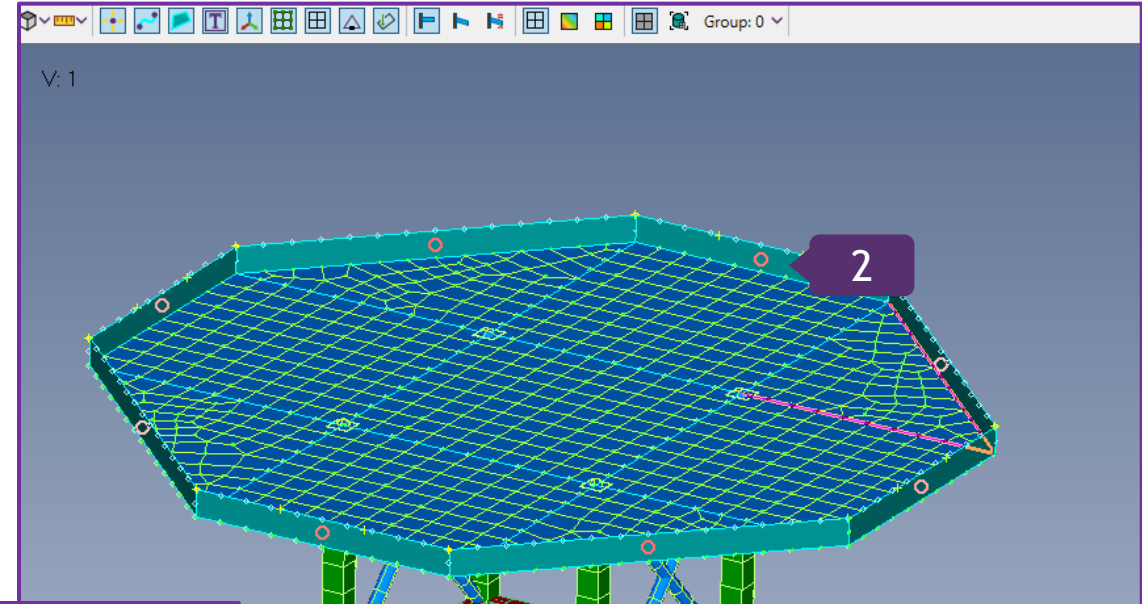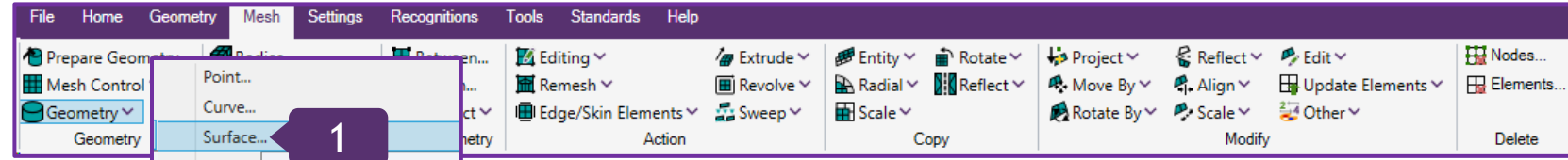
Add Stiffeners; create another Property

1 In *Mesh* tab, go to the Model Tree => *Model* and select Properties

2 Execute right click and select *Add*

3 Press on *Elem/Property Type...*

4 In Line Elements section, select *Beam* and press *OK*
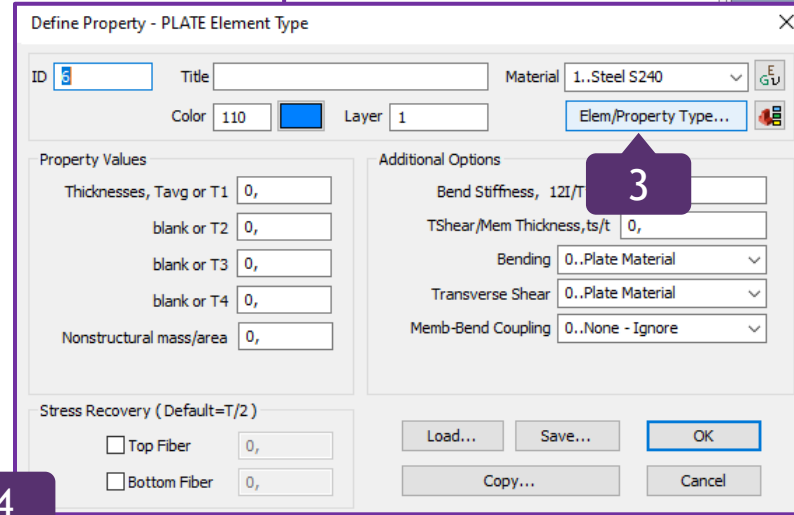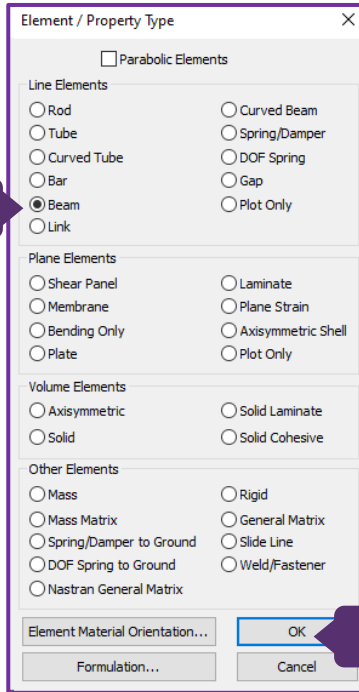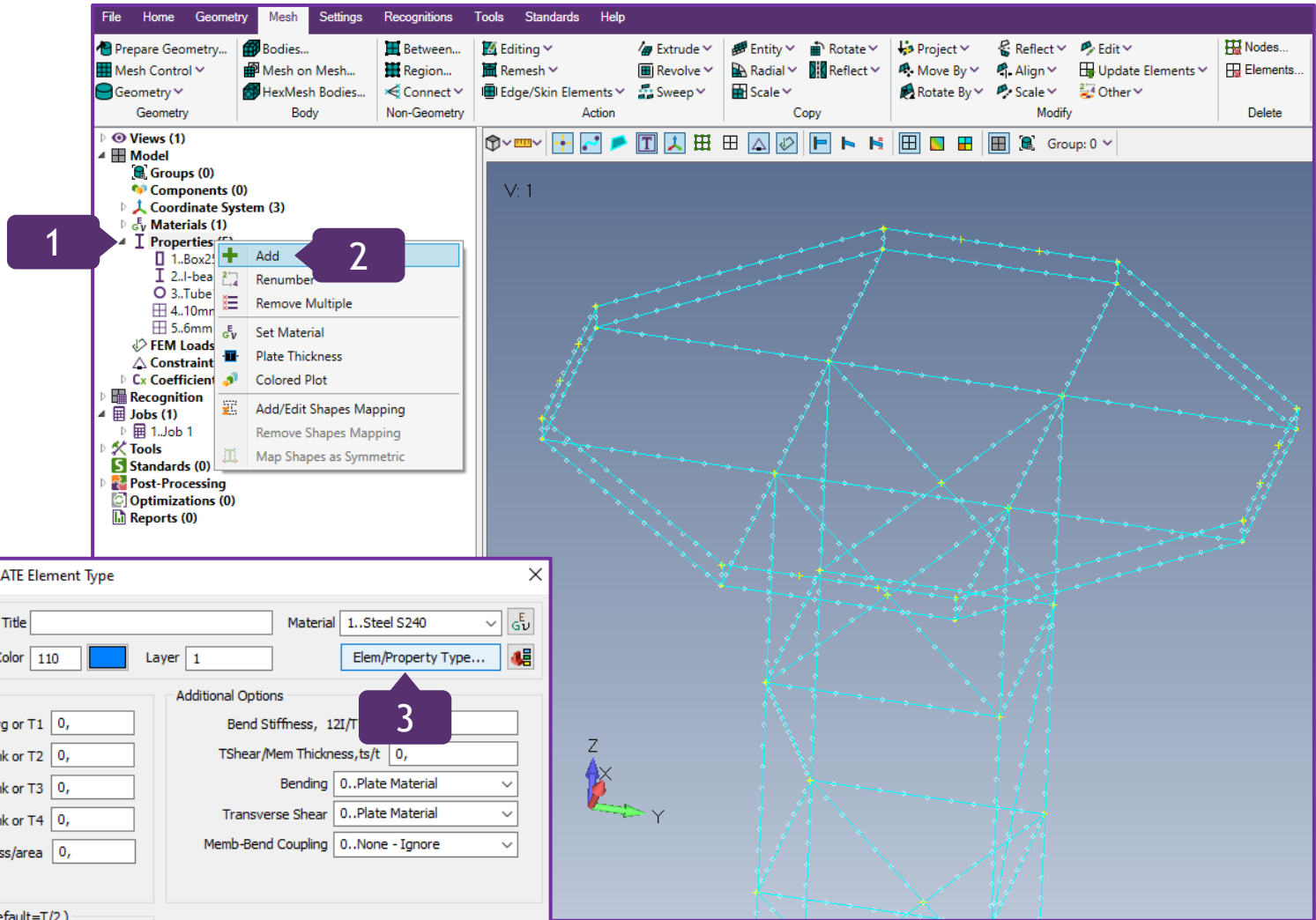
Add Stiffeners; create another Property (Continuation)

5 *In Define Property – PLATE Element Type* menu, press on *Shape…*

6 In *Shape* section, select *Rectangular Bar*

7 Height: 0.12;
Width: 0.008
Press *OK*

8 Title: Bar 120x8

9 Press on ▢ and select the color 24 Yellow. Press *OK*
Close the window

Mesh Stiffeners

1 In *Mesh* tab, go to *Geometry* and select *Curve...*

2 With the left clicks of the mouse, select the parallel lines. Press *OK*

3 Property: *6..Bar120x8*
Press *OK*
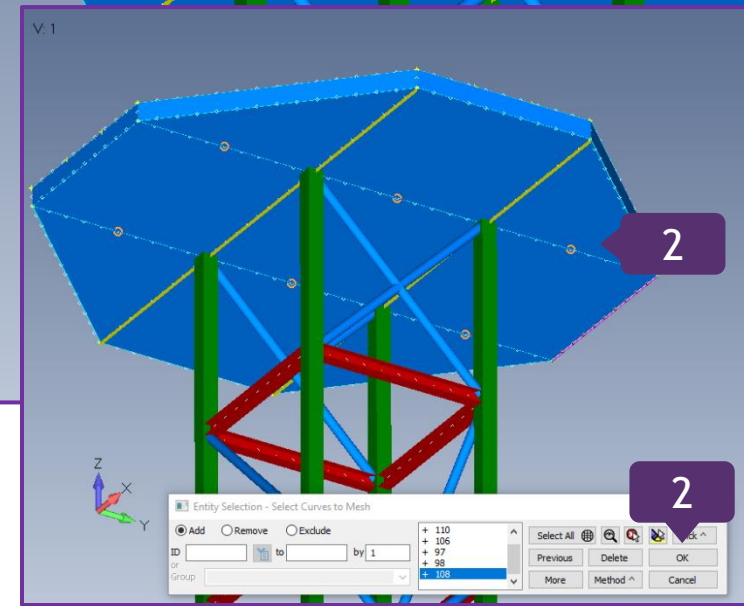
4 *Vector Locate: Tip:* X=0,Y=1,Z=0
Press *OK*

Highlighted selected lines
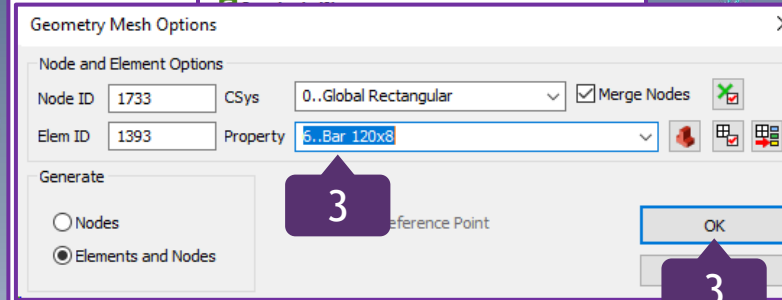
The Y-direction will be Global Y
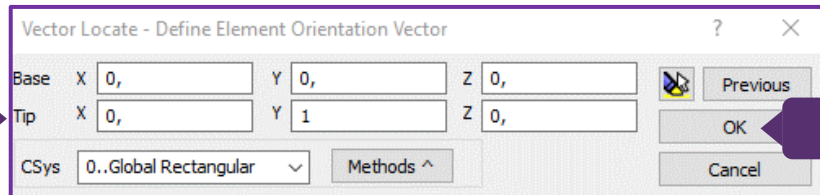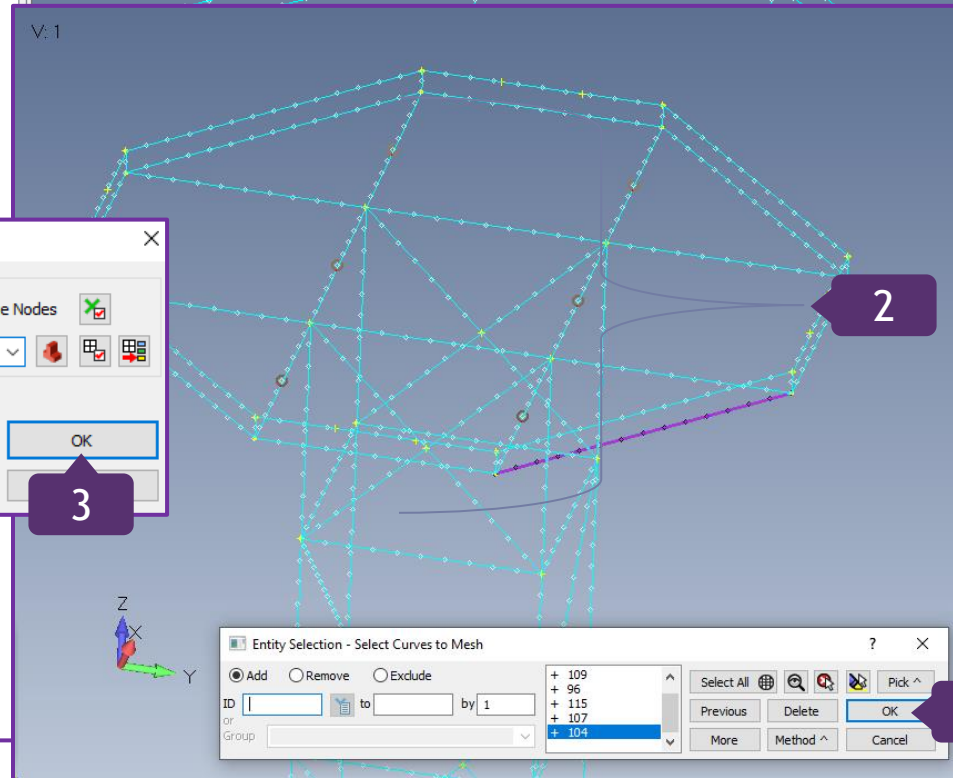
Mesh Stiffeners (Continuation)

1 Execute previous command with the right click on the graphical interface

2 With the left clicks of the mouse, select the parallel lines. Press *OK*

3 Property: *6..Bar120x8*
Press *OK*

4 *Vector Locate: Tip:* X=1,Y=0,Z=0,
Press *OK*
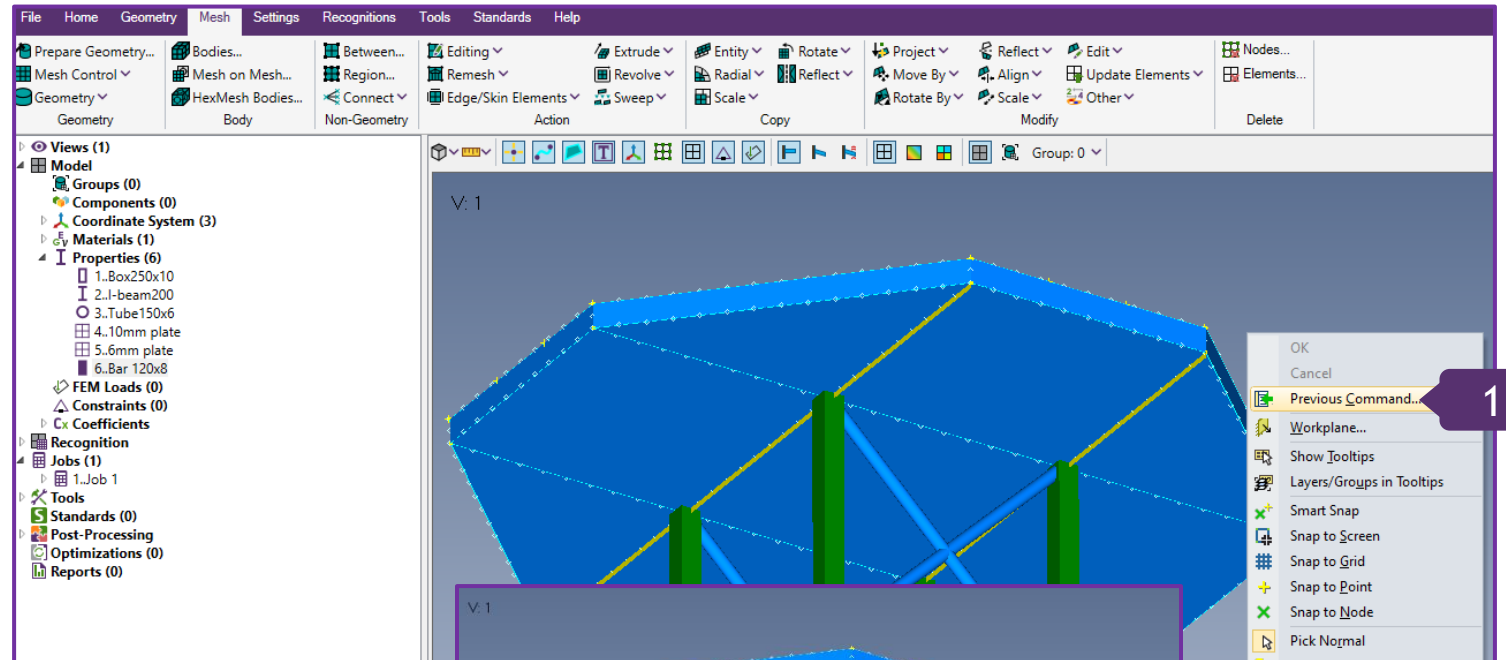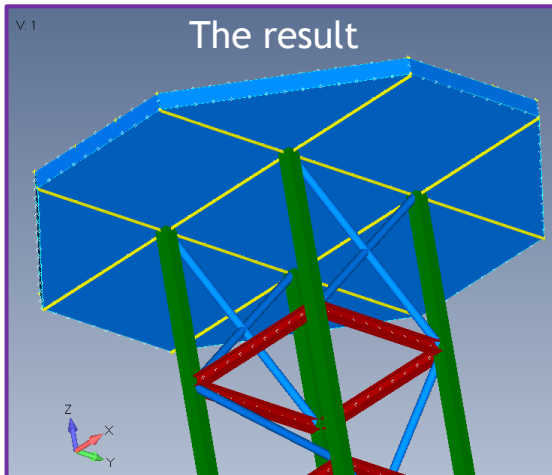
The result
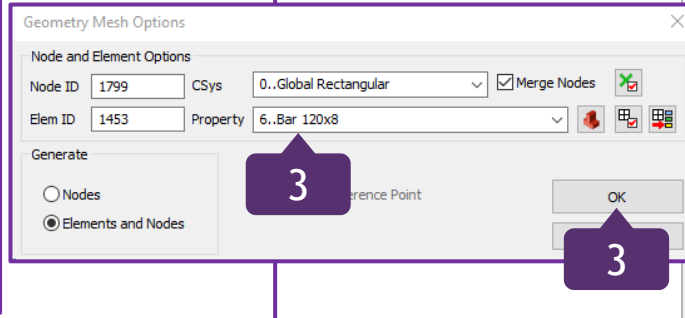
The X-direction will be Global X

https://sdcverifier.com
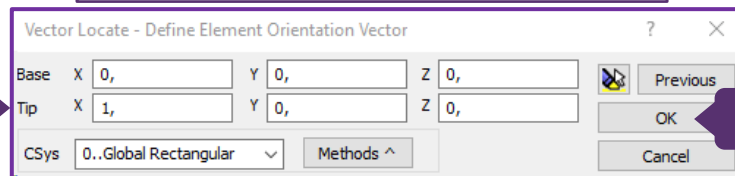
Replace some Properties

1 In *Mesh* tab, go to *Modify => Update Elements* and select *Property ID...*

2 Press on *Methods* and select *Property*

3 With the left clicks of the mouse, select the Stiffener Property in Yellow. Press *OK*

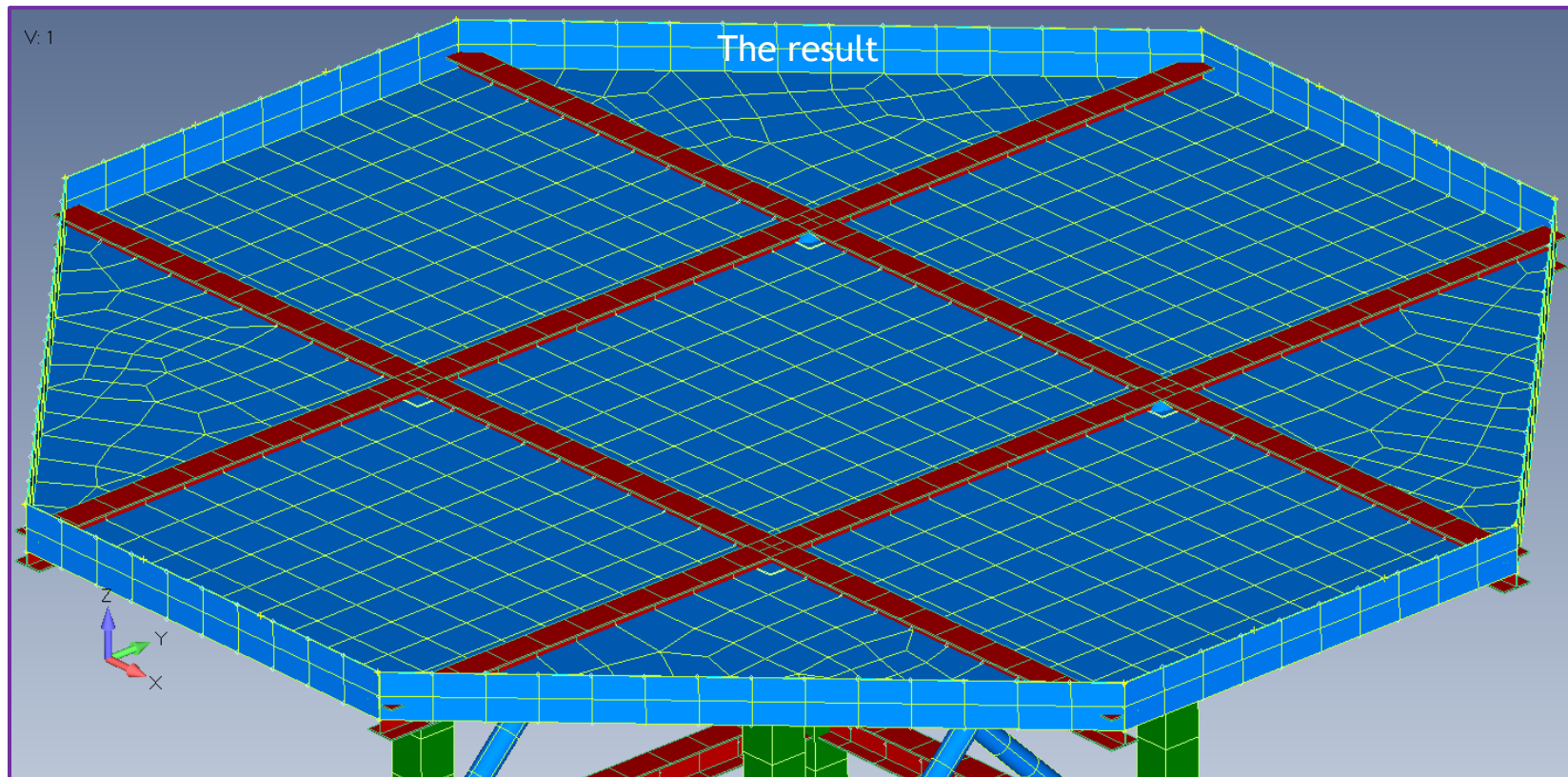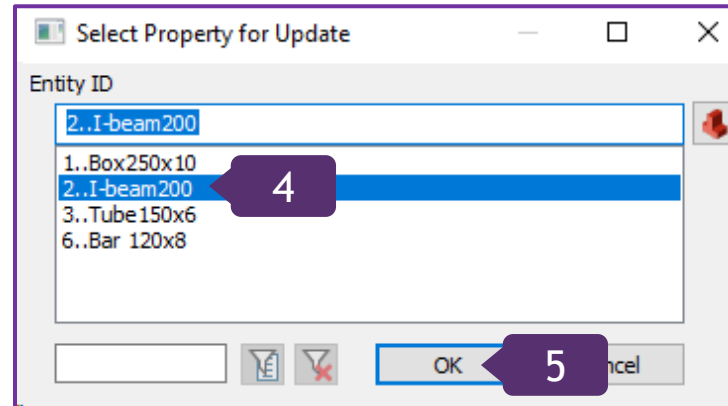# Replace some Properties (Continuation)

4 *In Select Property for Update* menu, select *2..I-beam200*

5 Press *OK*

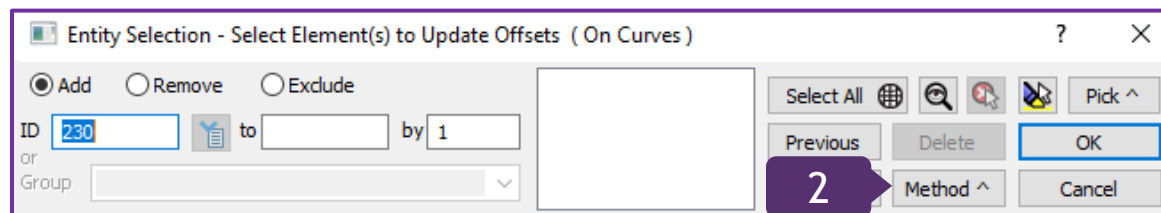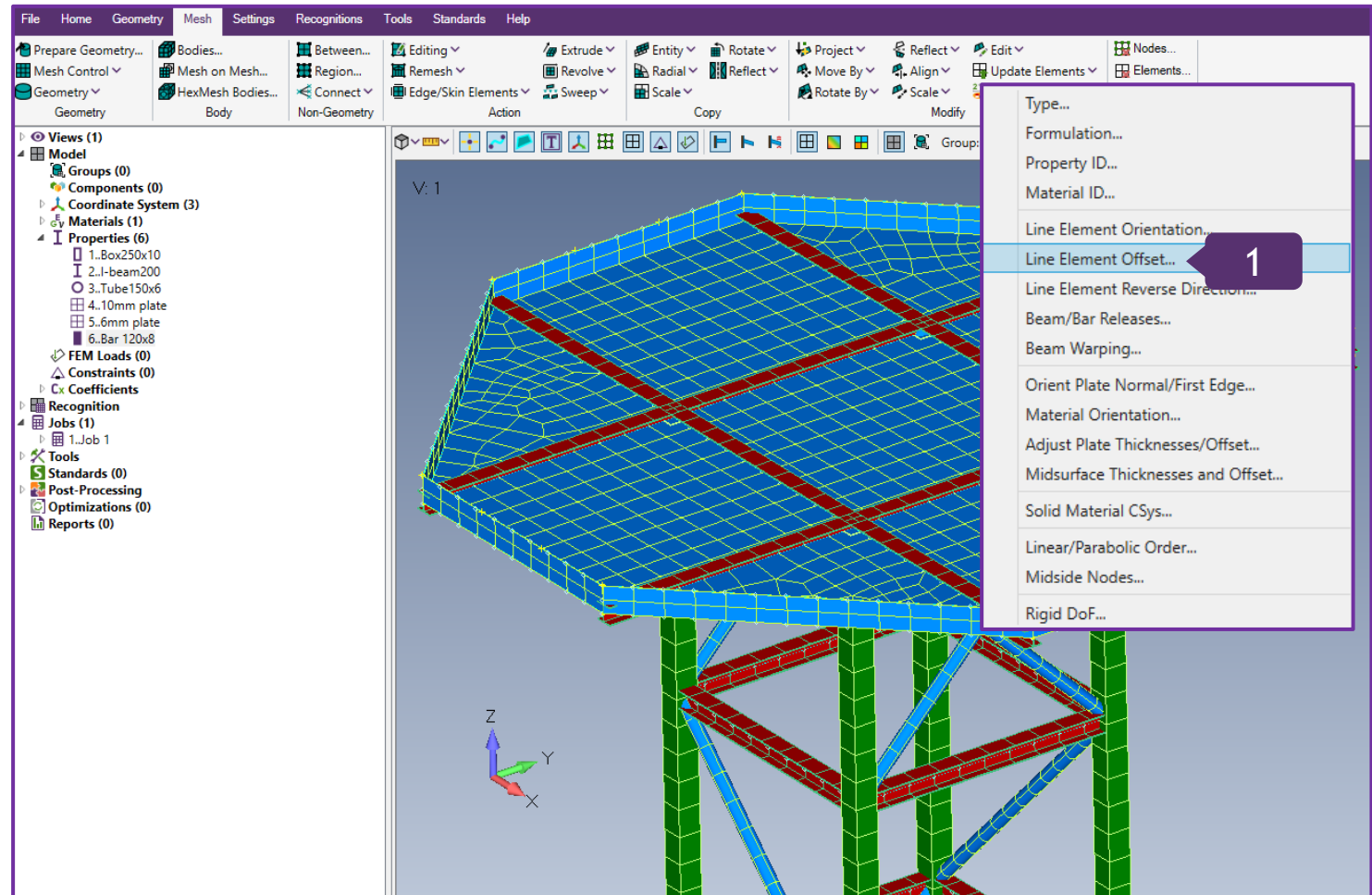**6** Execute left double click on the model to refresh it

Select Property for Update

Entity ID

2..I-beam200

1..Box250x10
2..I-beam200 **4**
3..Tube150x6
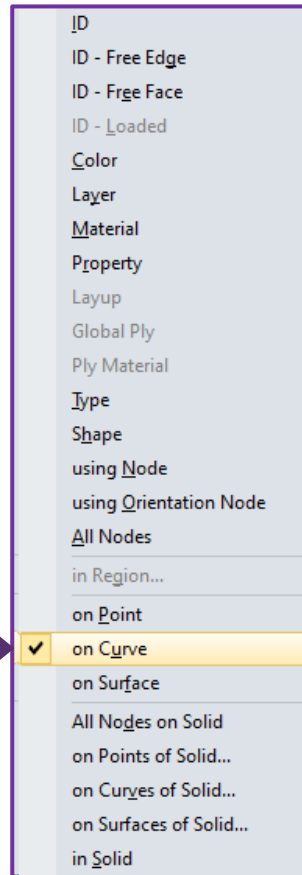6..Bar 120x8

OK **5** Cancel

V: 1

The result

Add the Offset

1 In *Mesh* tab, go to *Modify => Update Elements* and select Line Element Offset…

2 Press on *Method* and select on *Curve*

52

Add the Offset (Continuation)

SDC VERIFIER

3 With the left clicks of the mouse, select all the curves.

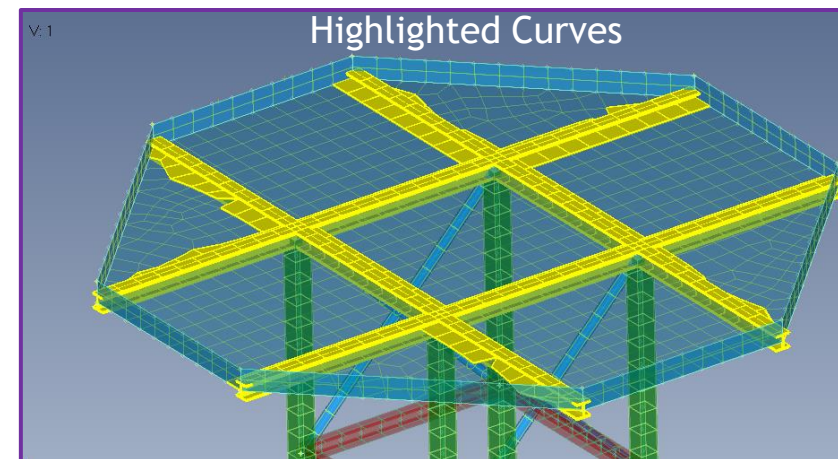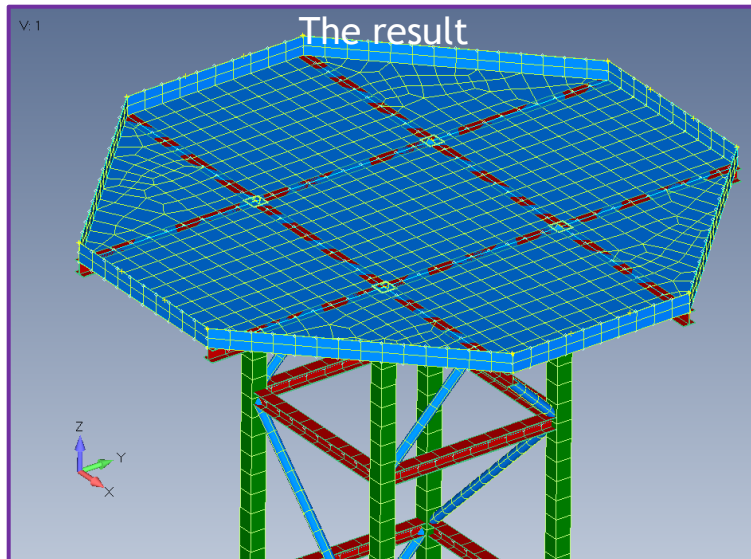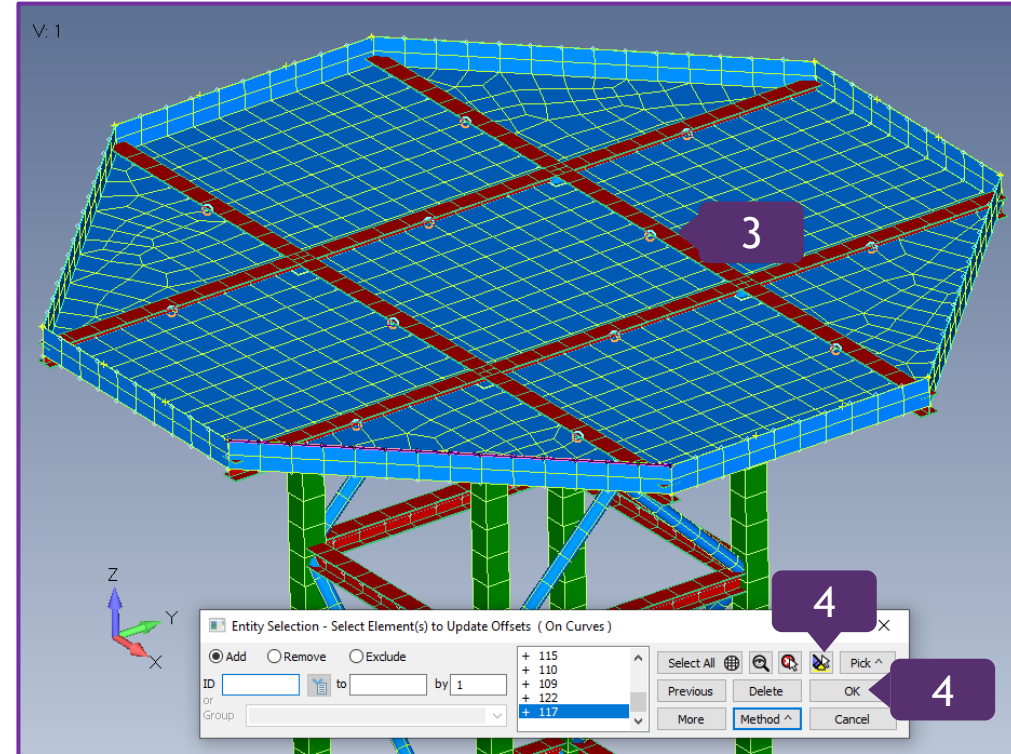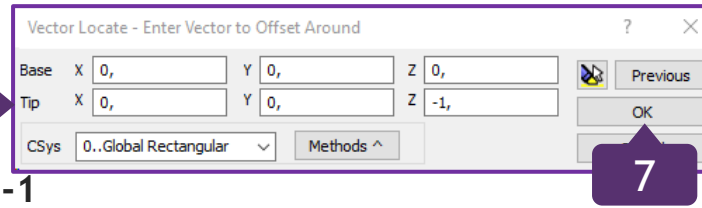**4** Press [icon] to check if all necessary Curves have been selected. Press *OK*
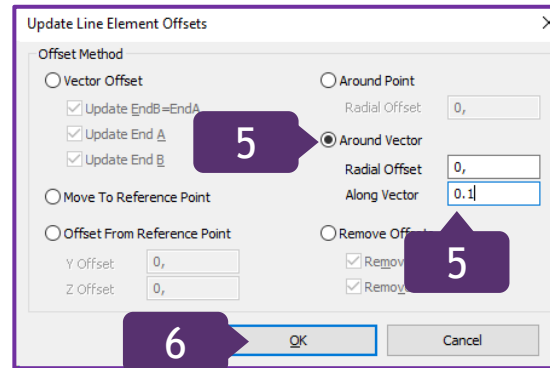
5 Select *Around Vector*; *Along Vector: 0.1*

6 Press *OK*

7 *Vector Locate: Tip:* X=0,Y=0,Z= -1 Press *OK* and double click to update the model

---

Update Line Element Offsets

Offset Method
- ○ Vector Offset
  - ☑ Update EndB=EndA
  - ☑ Update End A
  - ☑ Update End B
- ○ Move To Reference Point
- ○ Offset From Reference Point
  - Y Offset: 0,
  - Z Offset: 0,

- ○ Around Point
  - Radial Offset: 0,
- **5** ● Around Vector
  - Radial Offset: 0,
  - Along Vector: 0.1
- ○ Remove Off...
  - ☑ Remov
  - ☑ Remov

6 OK   Cancel

Vector Locate - Enter Vector to Offset Around

	X	Y	Z	
Base	0,	0,	0,	Previous
7 Tip	0,	0,	-1,	OK

CSys: 0..Global Rectangular   Methods ^

7

V: 1

3

4

Entity Selection - Select Element(s) to Update Offsets (On Curves)

● Add   ○ Remove   ○ Exclude

ID: [   ] to [   ] by 1
or Group: [   ]

115
110
109
122
117

Select All   Pick ^
Previous   Delete   OK   **4**
More   Method ^   Cancel

The result

Highlighted Curves

53

https://sdcverifier.com

Measuring Distance Functionality
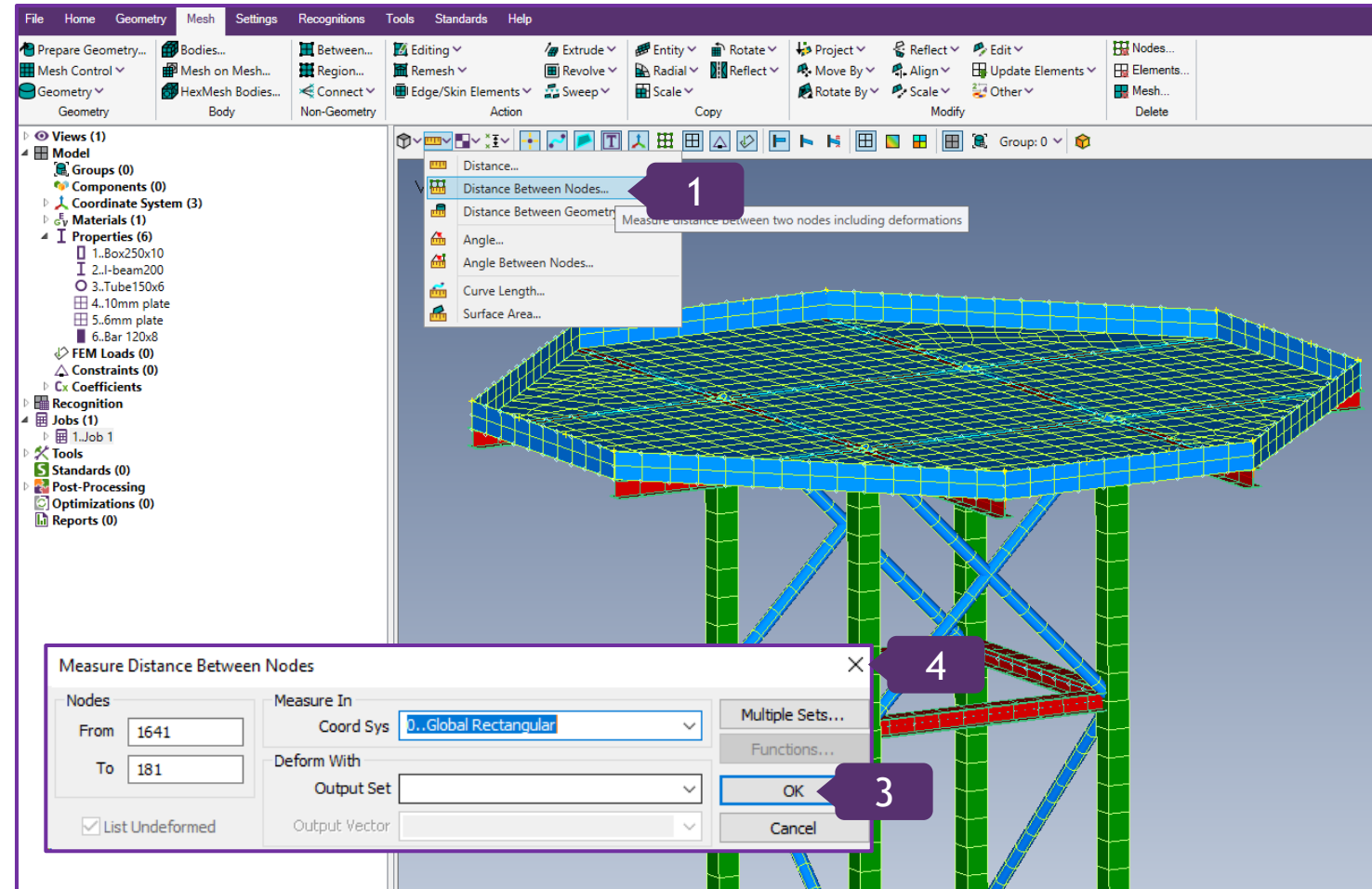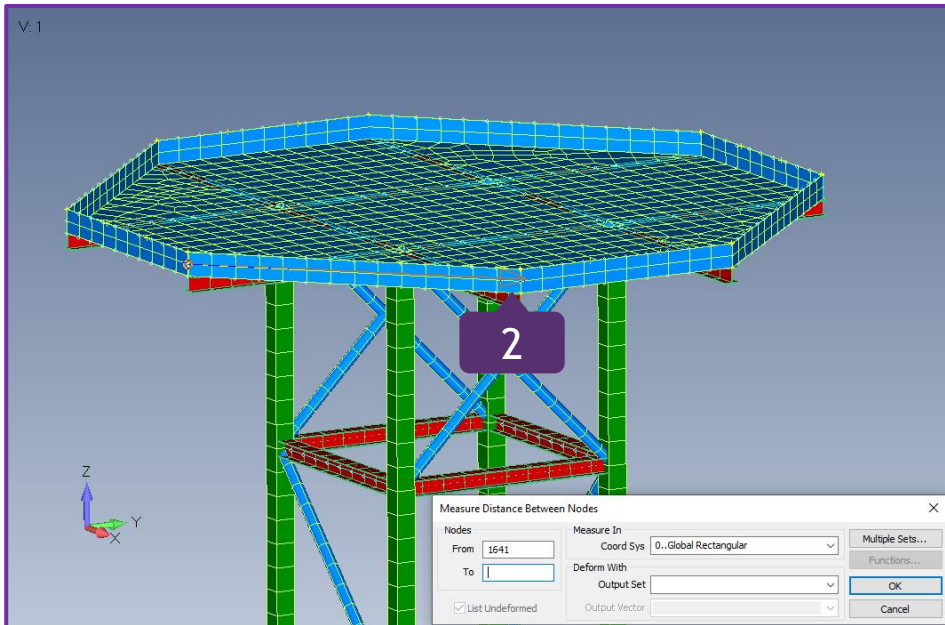
1 Press on ⊞⊞⊞ and select *Distance Between Nodes…*

2 With the left click of the mouse, select the node and drag the Vector to the second one

3 Press *OK*

4 Close the window

The result
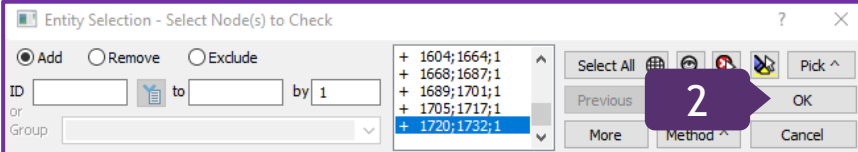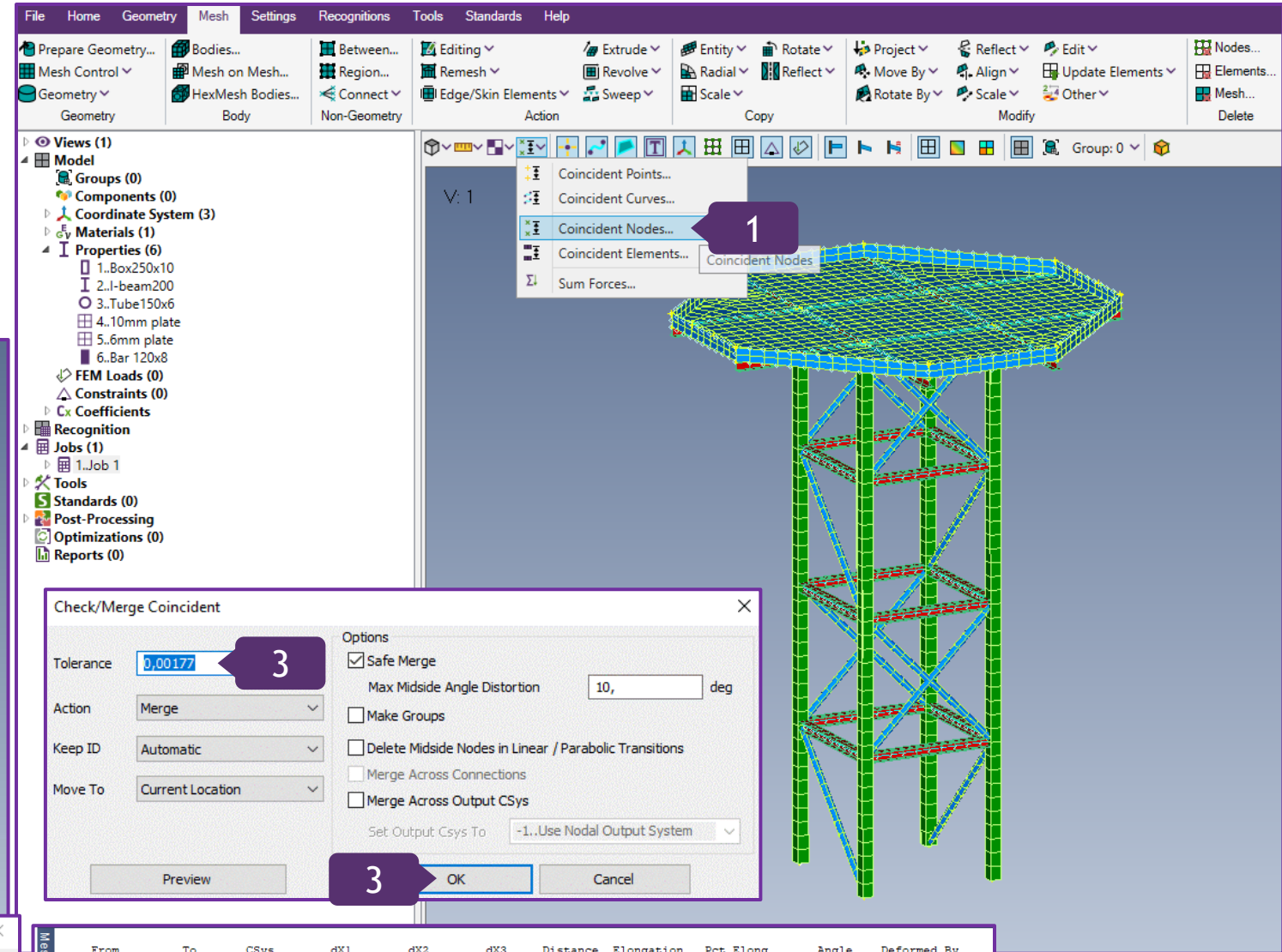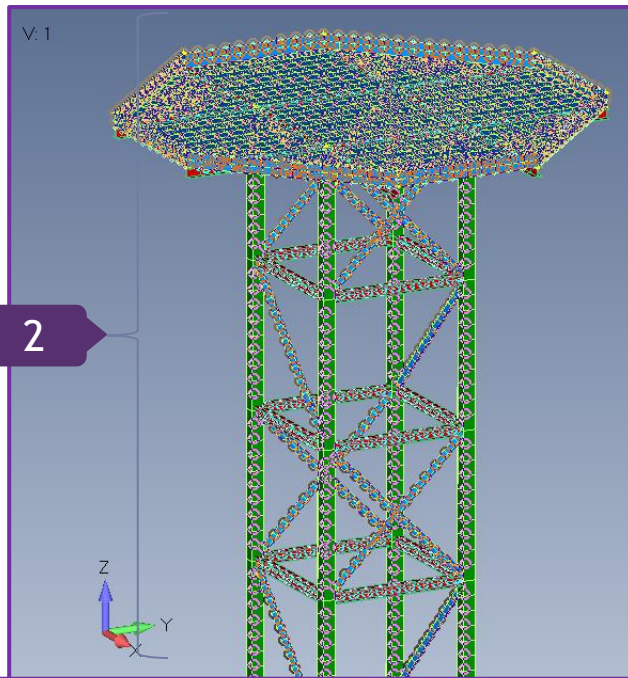
Check the Model for Coincident Nodes

1 Press on ⬚ and select *Coincident Nodes…*

2 With the combination of CTRL and left click, select all nodes.
Press *OK*

3 Tolerance: 0,00177 and press *OK*

There are no coincident nodes.

Meshing Toolbox Functionality

1 On the Toolbar, press on

2 Click on *Mesh Sizing* to expand its functionality

3 Sizing Option: *Size Curve(s)*;
Operation: *Increase*;
Number of elements: *1*

4 Press on and select curves to re-mesh them, by adding extra elements

Meshing Toolbox helps to control and adjust the Mesh.

Re-meshed

https://sdcverifier.com