Tutorial

Analysis of the Structure

Updated on: November 30th, 2023

Tested with: SDC Verifier 2023 R2

Content

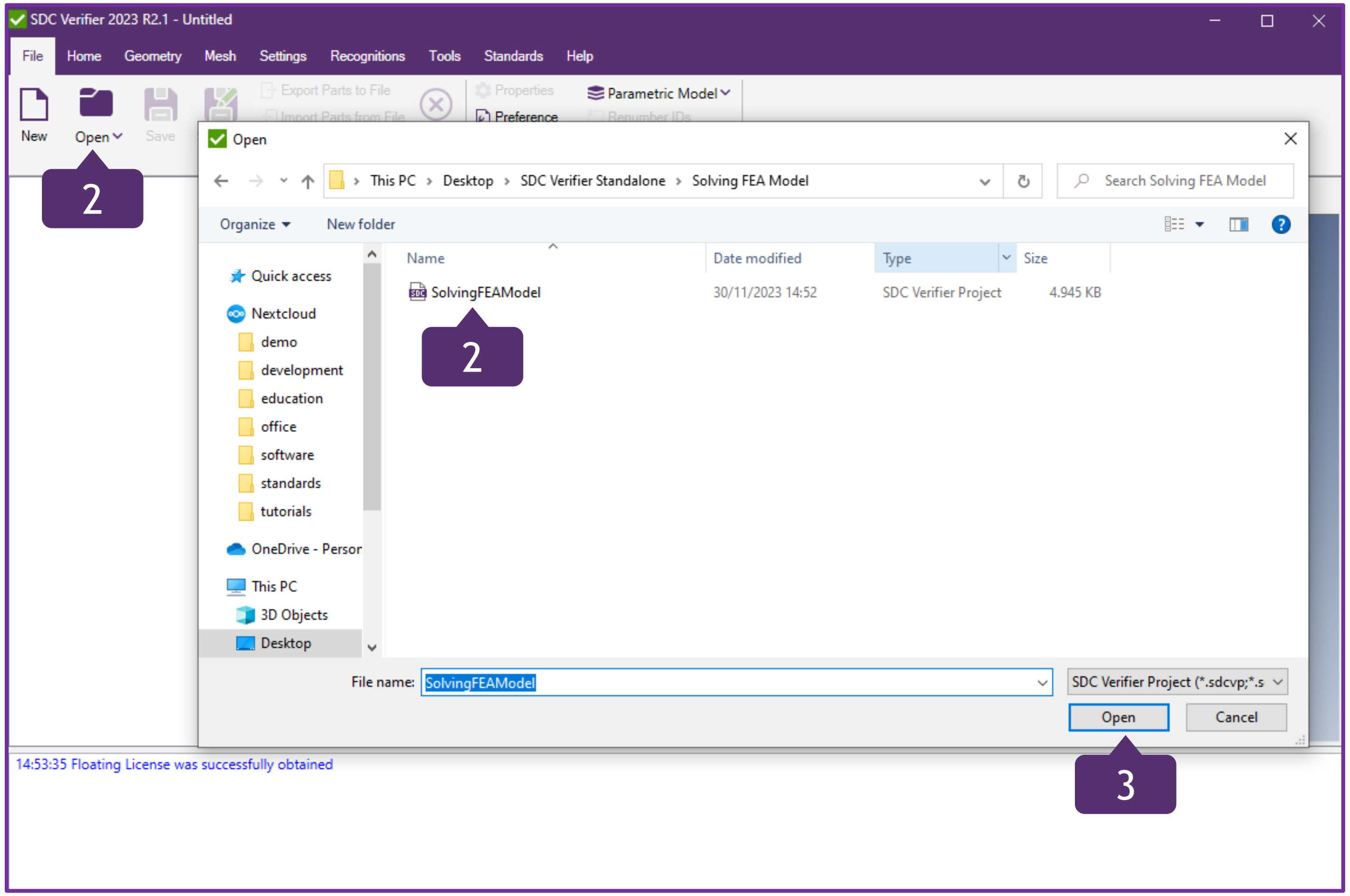
SDC Verifier is a powerful all-in-one software solution for structural design, FEA analysis, and verification according to standards.

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 boundary conditions and constraints;
- Apply FEM Loads;
- Create Individual Loads, Load Sets and Load Groups;
- Run the Analysis;
- Preview the Plot

Open the Starter Model

1. Launch SDC Verifier 2023 R2.1
2. Press Open and select project *SolvingFEAModel*
3. Press *Open*



Create Constraints

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

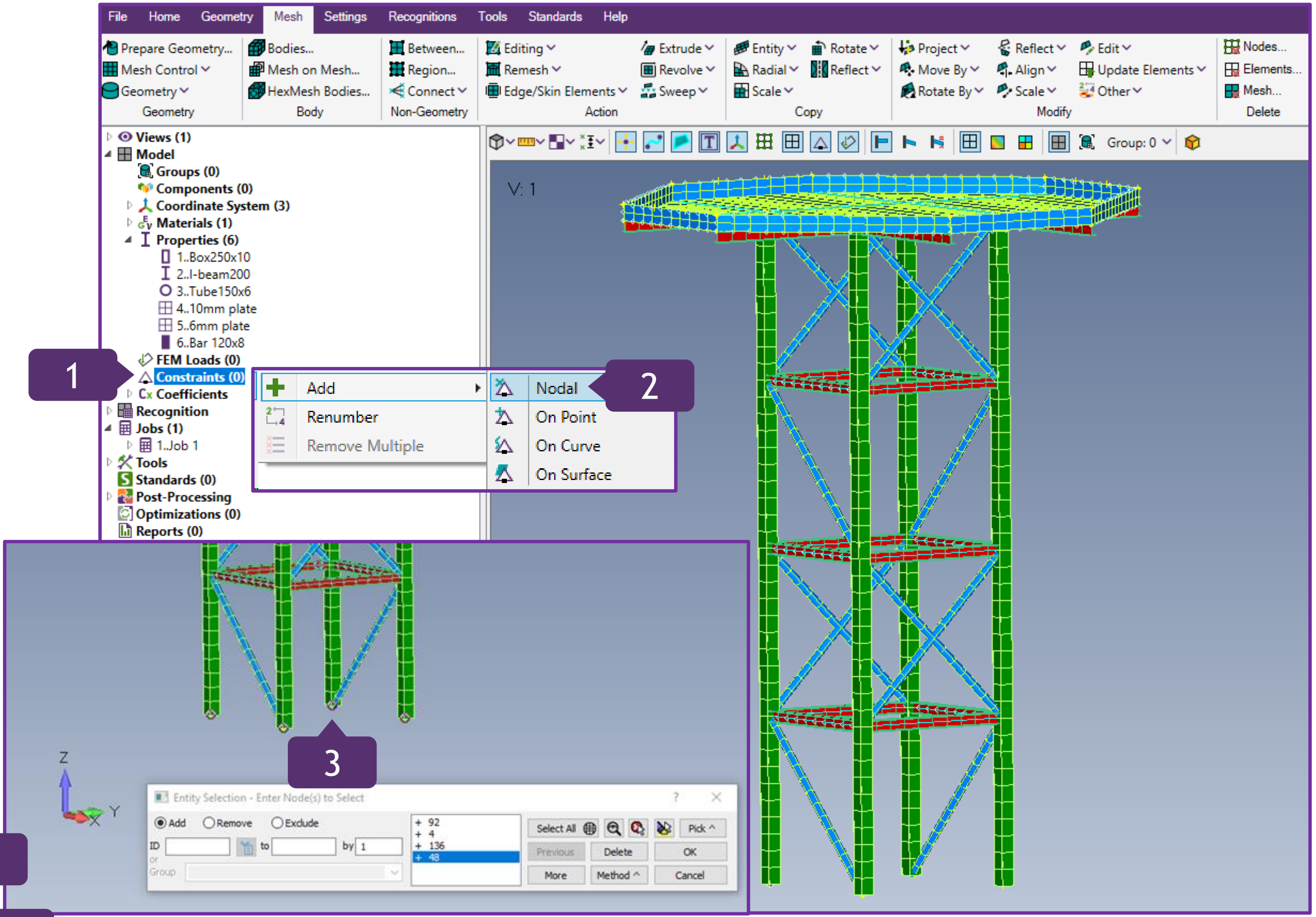
2 Execute right click on *Constraints* => *Add* and select *Nodal*

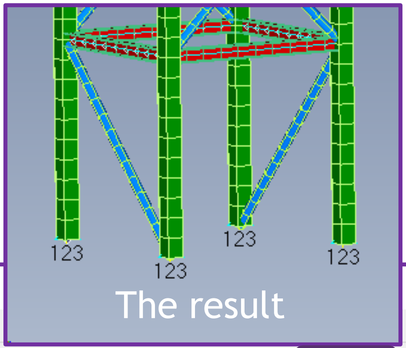
3 With the left clicks of the mouse, select the nodes that have to be constrained

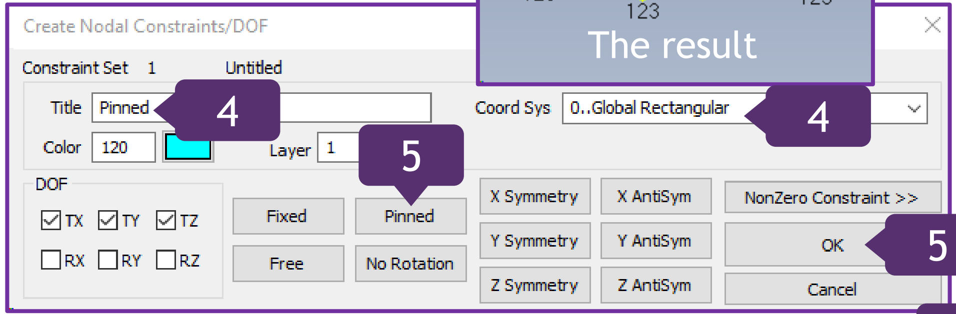
4 Title: *Pinned*; Coord Sys: *0..Global Rectangular*

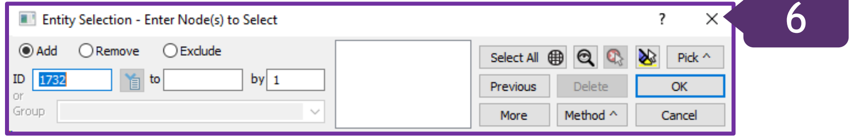
5 Press *Pinned* Press *OK*

6 Close the window







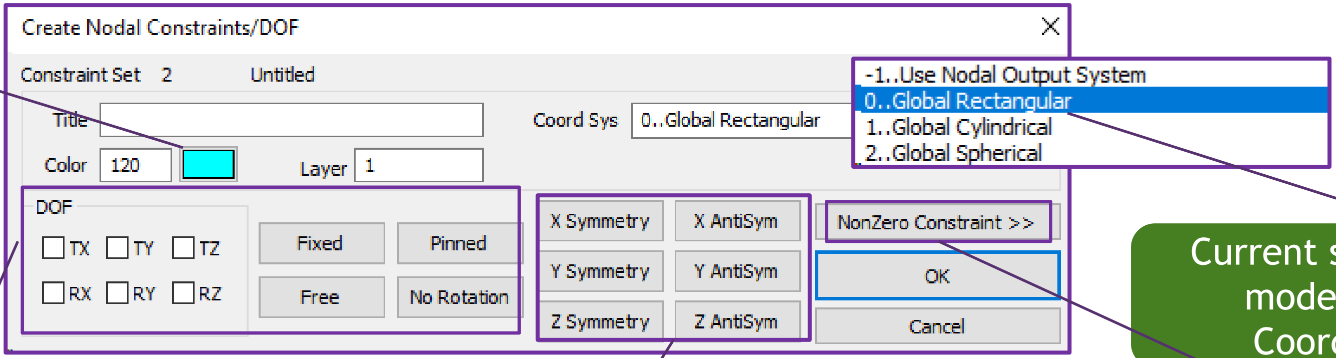


For this type of a model, it is normally a bottom part that has to be constrained.

https://sdcverifier.com

Expanded Functionality of Create Nodal Constraints/DOF Menu

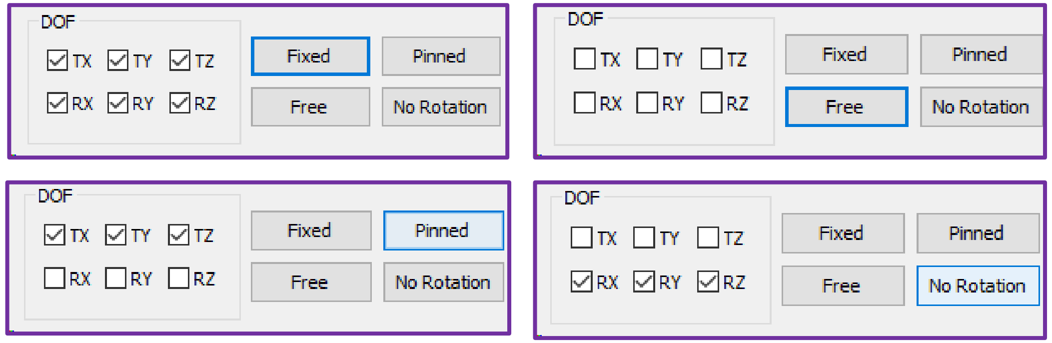
Constraint Sets can be defined by Color selection; press on the button to open the menu.

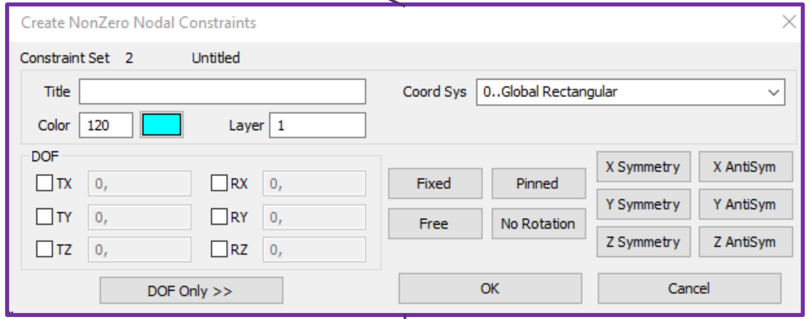


The menu presents two options of creating Constraints:
1. The Degree Of Freedom (DOF) section, in which they can be selected manually;
2. Clicking on the buttons: Fixed, Pinned, Free, No Rotation

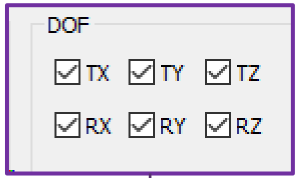
Current structure has been modelled within one Coordinate System;

If there is only a part of a model with a symmetry being developed, Symmetry Constraints may be incorporated. These options allow to define them.





NonZero Constraint >> option allow to create NonZero Nodal Constraints.

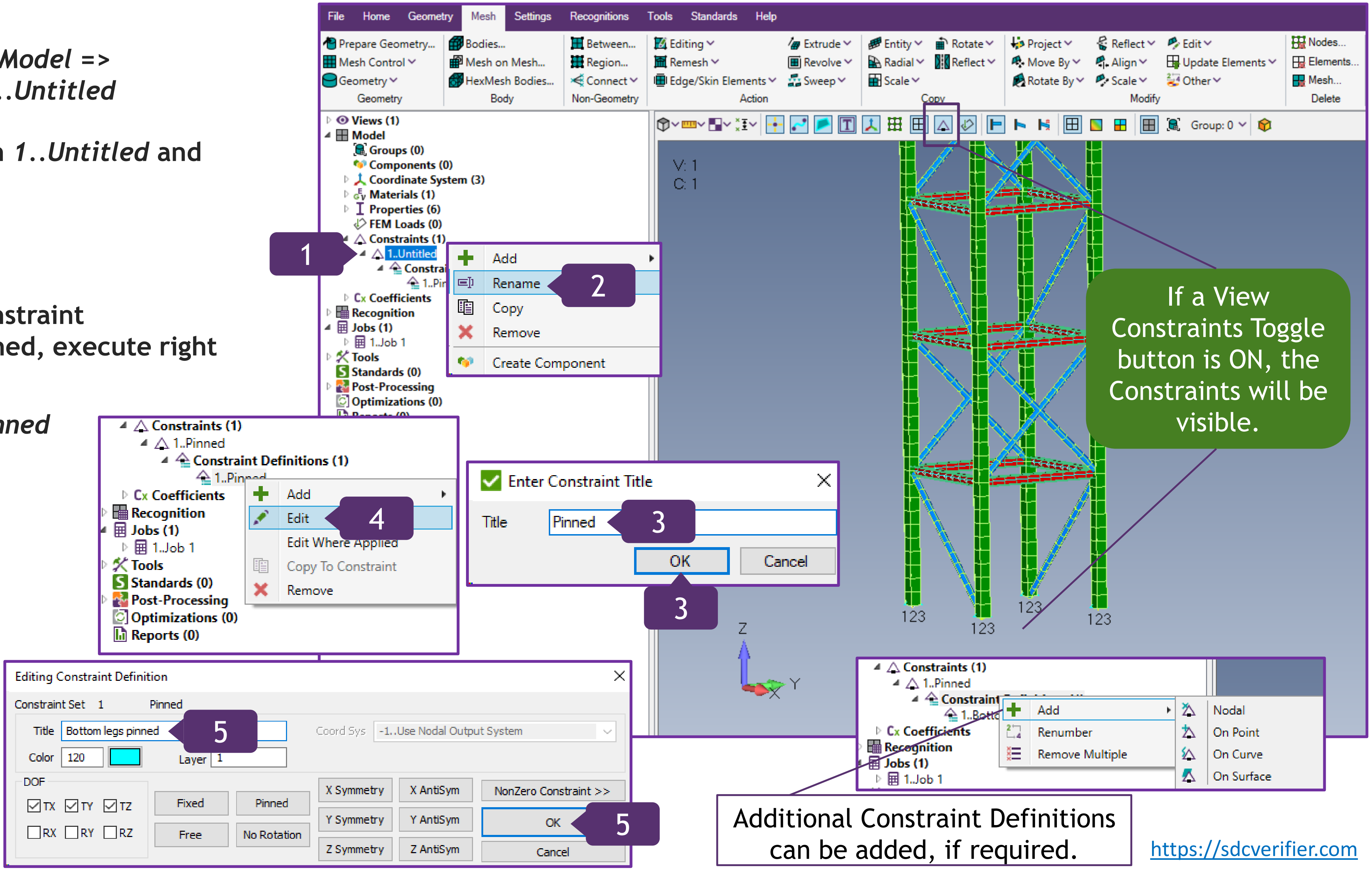


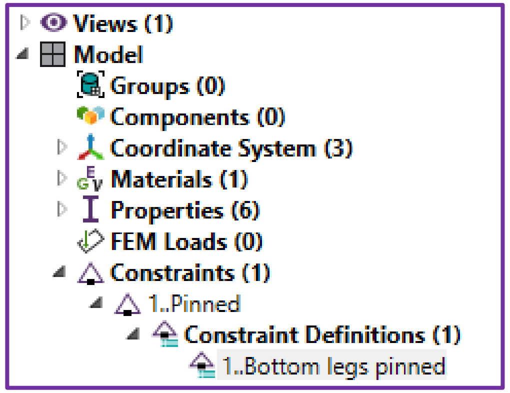
TX – pinned translation in X direction;
TY – pinned translation in Y direction;
TZ – pinned translation in Z direction;
RX – pinned rotation around X axis;
RY – pinned rotation around Y axis;
RZ – pinned rotation around Z axis.

If to select all the options, all zones of Constraints will be fixed.

Rename the Set of Constraints

1. In the Model Tree => *Model* => *Constraints*, select *1..Untitled*
2. Execute right click on *1..Untitled* and select *Rename*
3. Title: *Pinned*
   Press *OK*
4. In *Constraints* => Constraint Definitions => 1..Pinned, execute right click and select Edit
5. Title: *Bottom legs pinned*
   Press *OK*

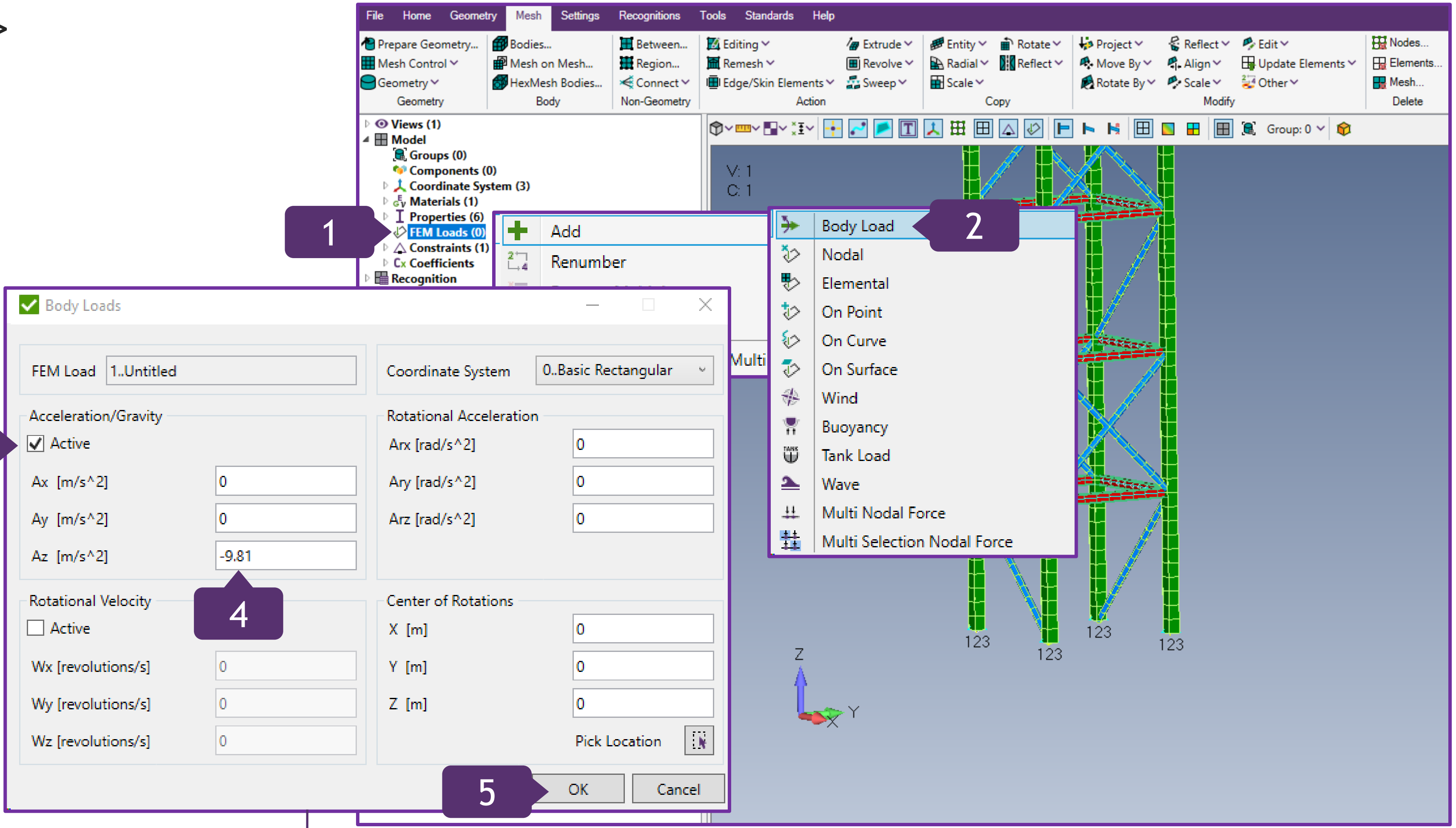




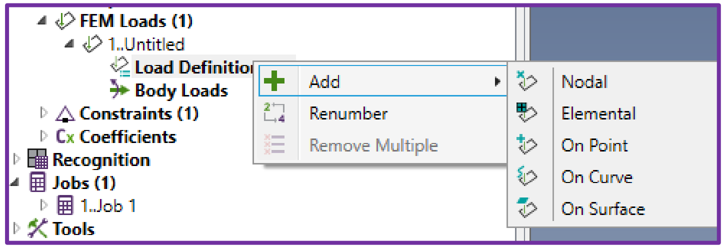
https://sdcverifier.com

Apply Loads (Gravity)

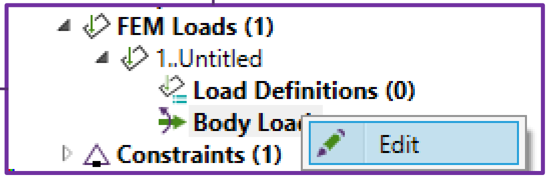
1. In *Mesh* tab, go to the Model Tree => *Model* and select *FEM Loads*
2. Execute right click on FEM Loads => *Add* and select *Body Load*
3. In *Body Loads* menu => *Acceleration/Gravity*, select *Active*
4. Az [m/s^2]: -9.81
5. Press *OK*



More Load Definitions can be set.



To check the result of applying Gravity, execute right click on Body Loads, and select Edit.

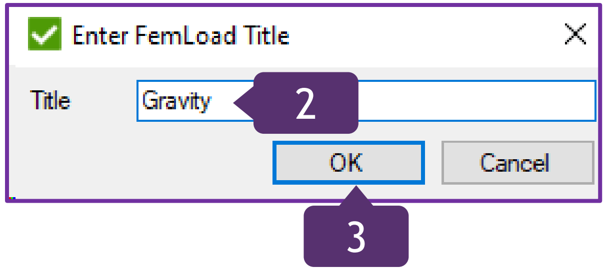


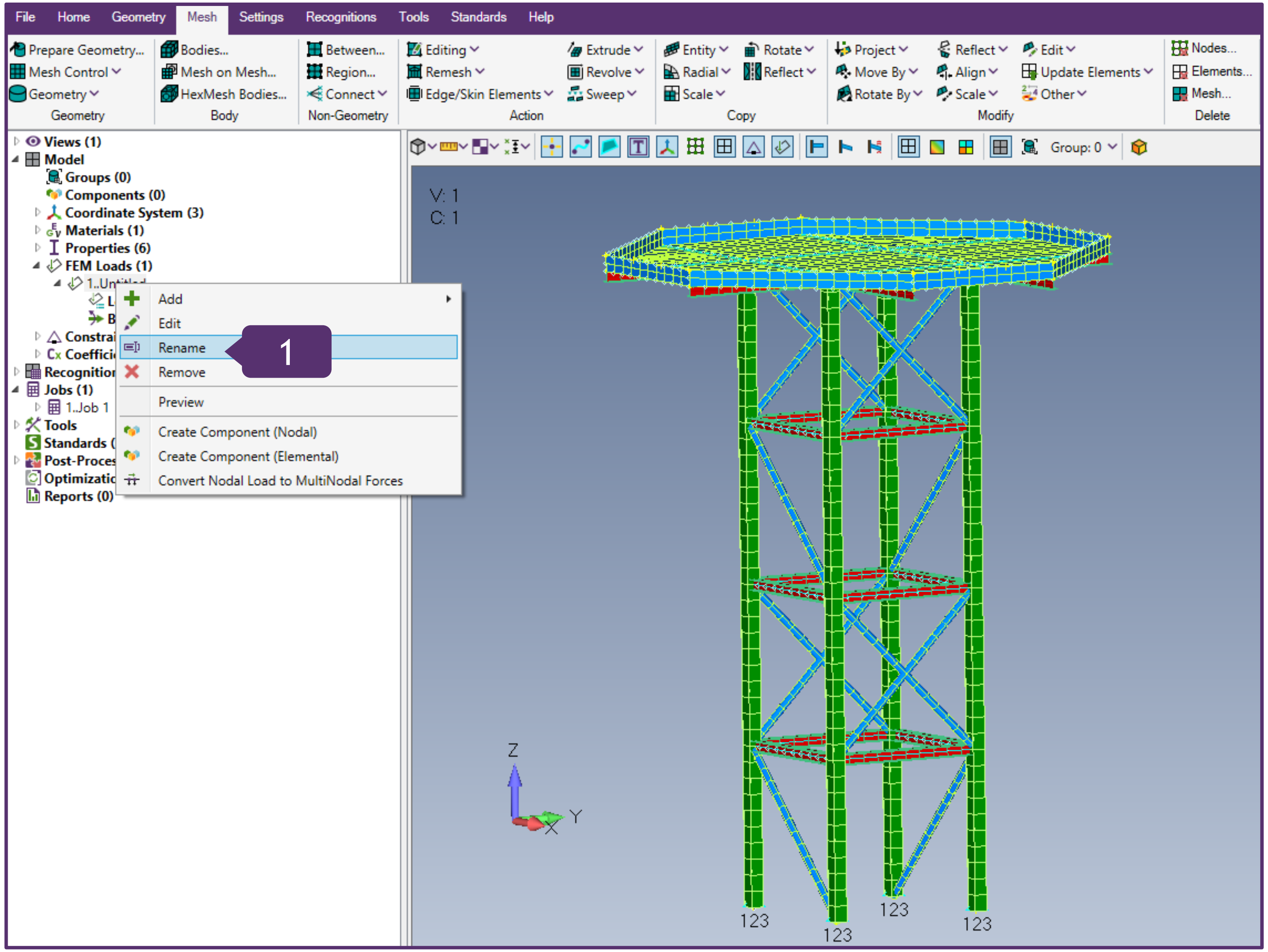
Note: as the vertical vector is Z, the standard earth gravity in negative (minus) is applied.

https://sdcverifier.com

Rename FemLoad

1. Under FEM Loads, execute right click on *1..Untitled* and select *Rename*
2. Title: *Gravity*
3. Press *OK*

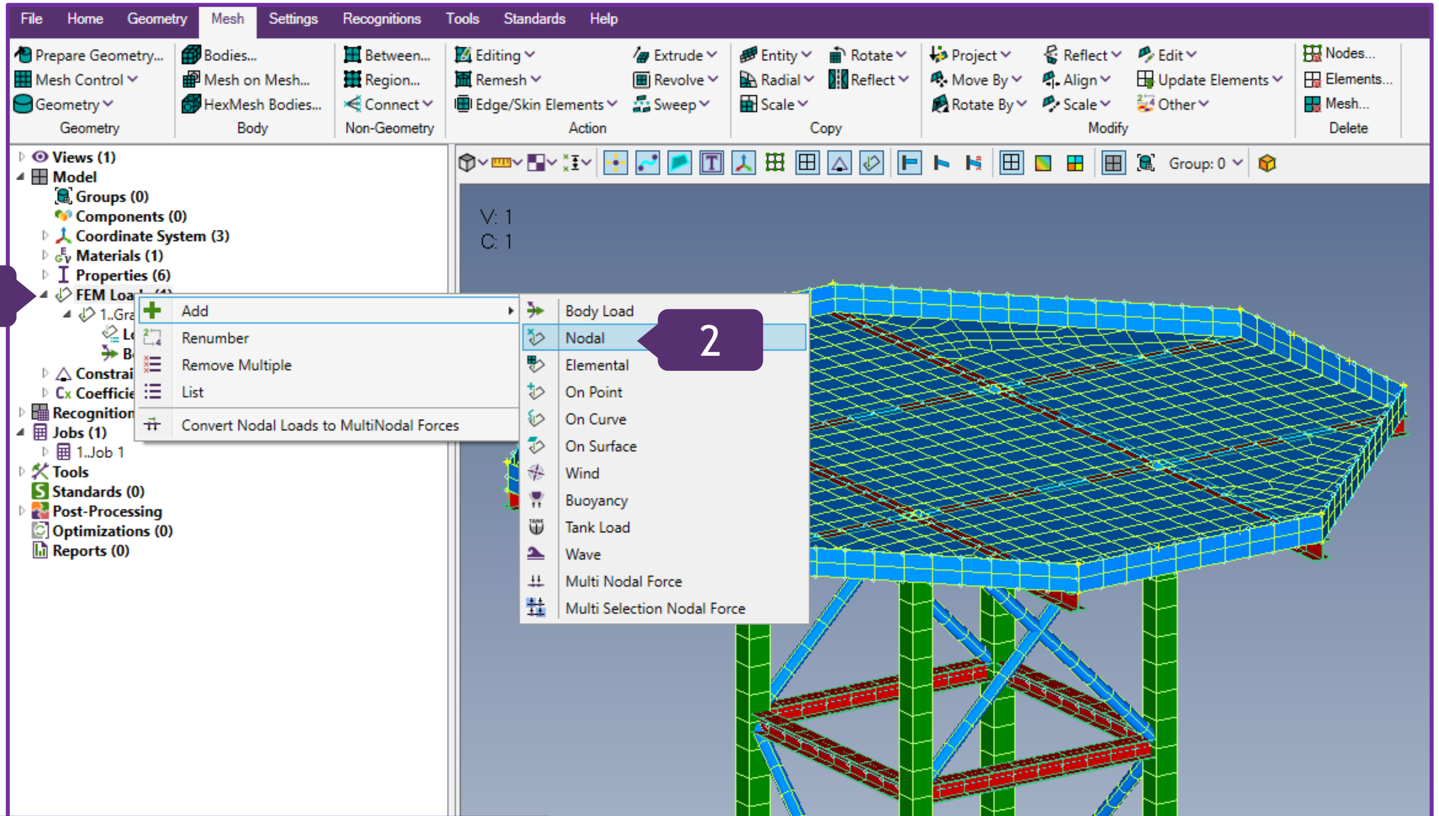


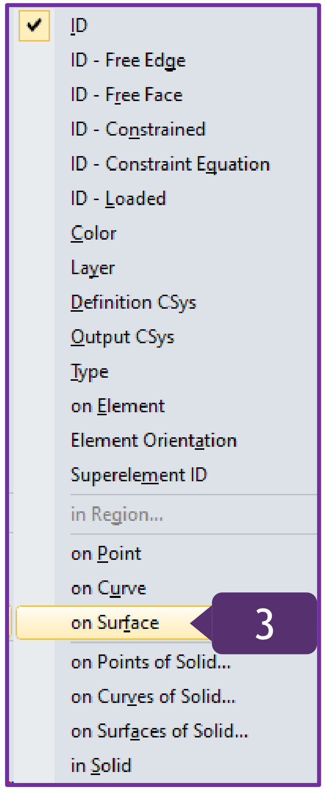


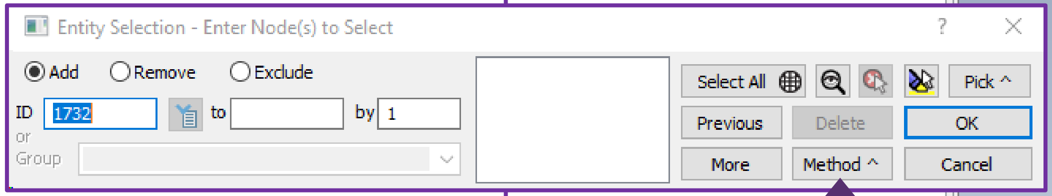
Create Load on all Nodes of the Surface

1. In *Mesh* tab, go to the Model Tree => *Model* and select *FEM Loads*
2. Execute right click on FEM Loads => *Add* and select *Nodal*
3. Press on *Method* and select: *on Surface*

The aim of applying Nodal Load on every node is to add distributed weight over the structure.





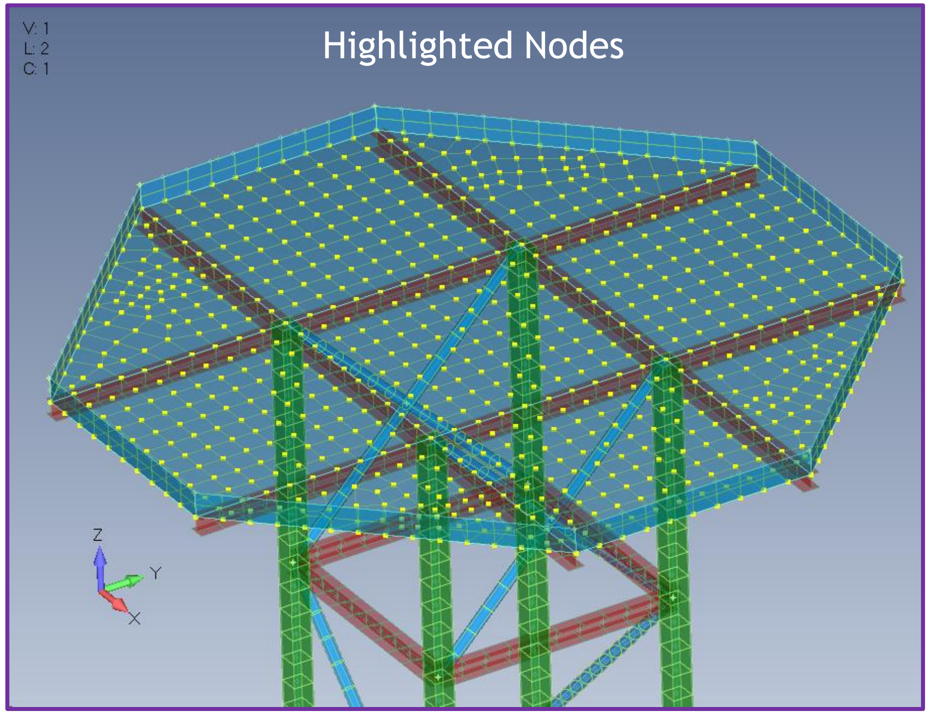


Create Load on all Nodes of the Surface (Continuation)

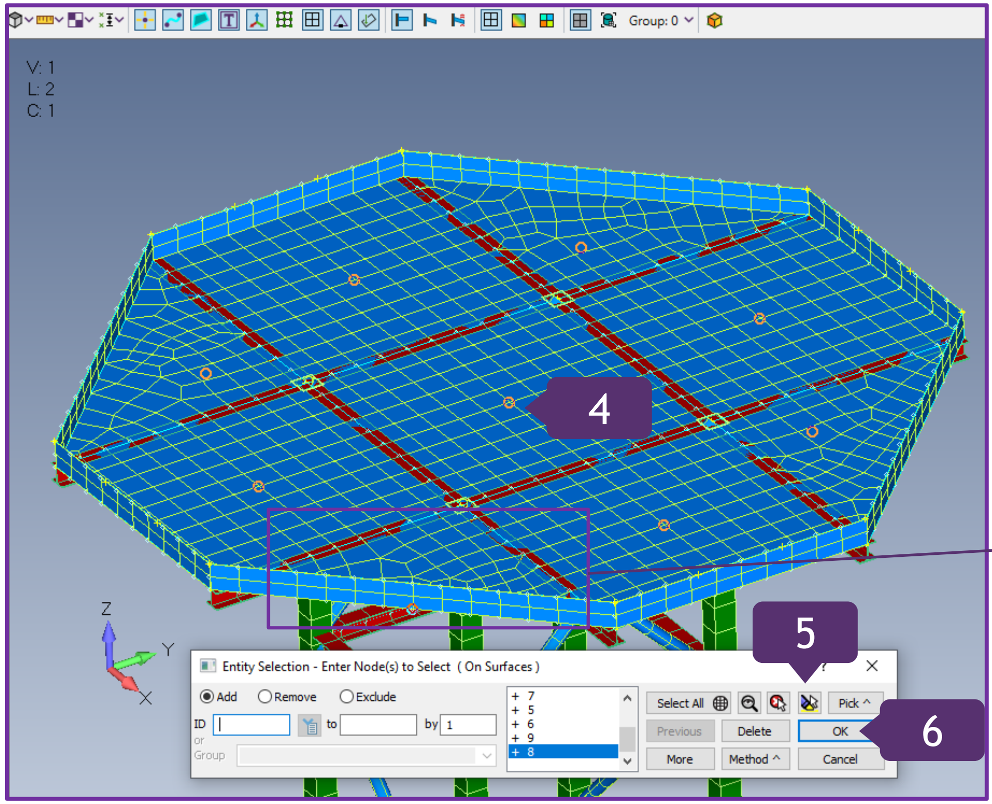
4 With the left clicks of the mouse, select the Nodes of the surface

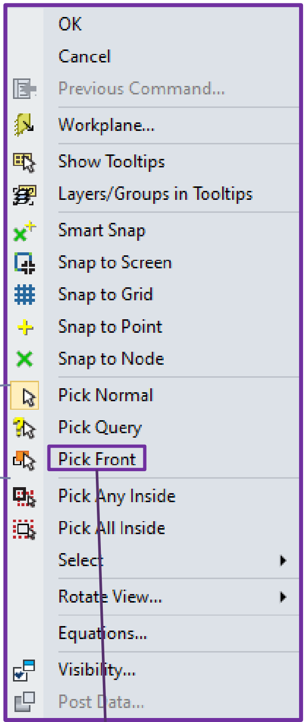
5 Press to check if all Nodes have been selected

6 Press *OK*



In order to conveniently select the required sections of the structure, there are options: Pick Normal, Pick Query and Pick Front. To use these functions, execute right click on the graphical interface.

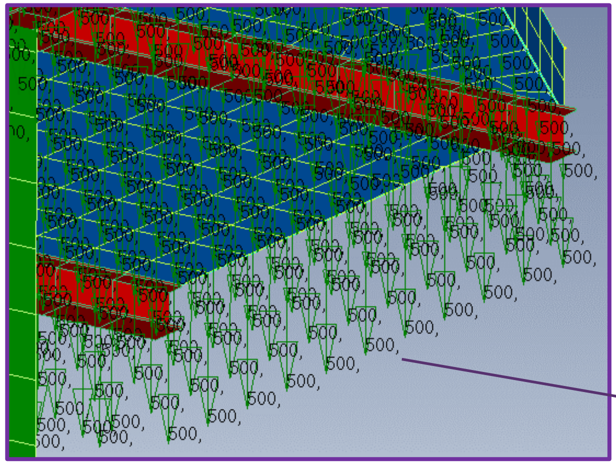


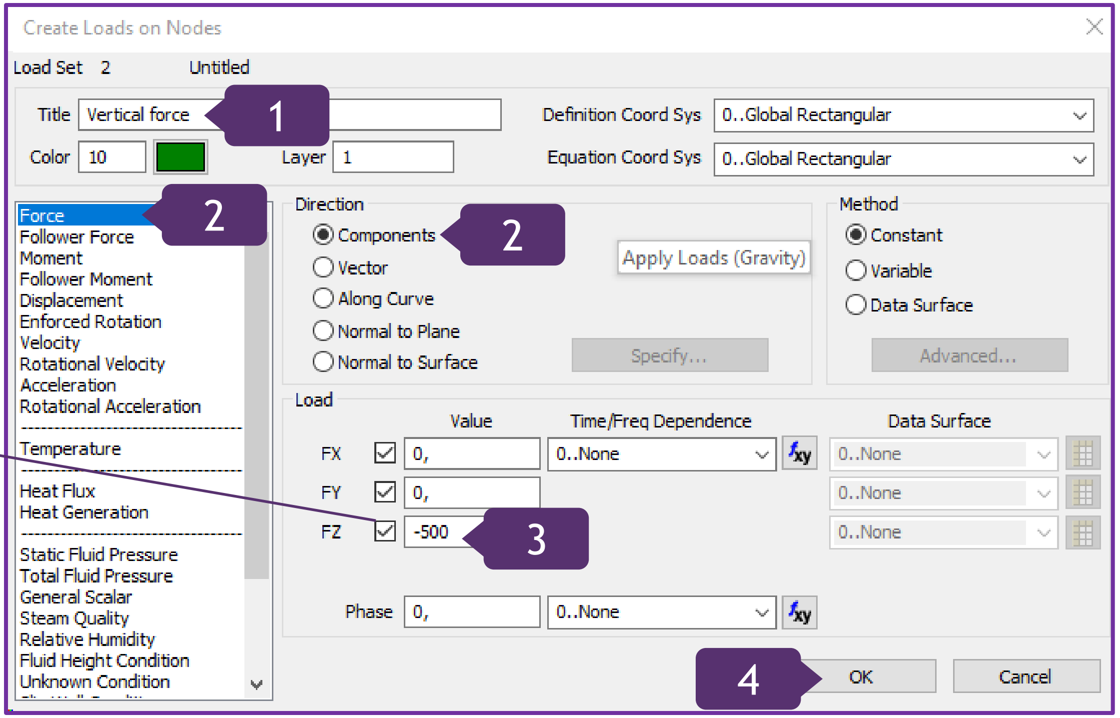


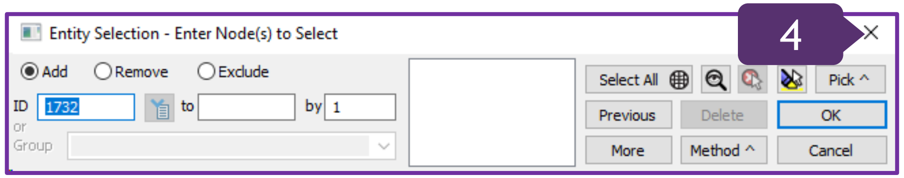
Note: For the front sections of the structure, select Pick Front option.

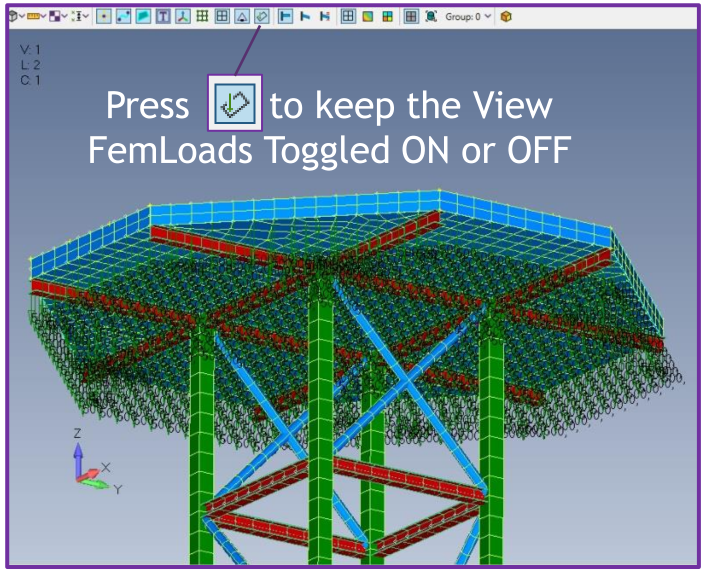
Define Vertical Force Load

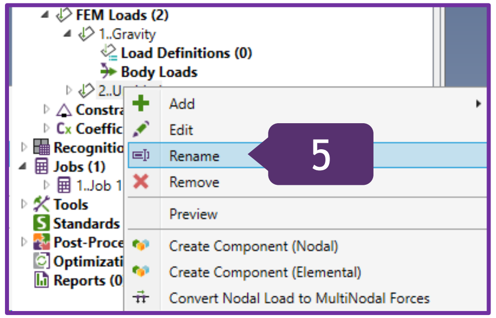
1. Title: *Vertical force*
2. Select: *Force*;
   Direction: *Components*
3. In *Load* section, *Value FZ: -500 N*
4. Press *OK*;
   Close the menu
5. Under FEM Loads, execute right click on *2..Untitled* and select *Rename*
6. Title: *Vertical force*
7. Press *OK*

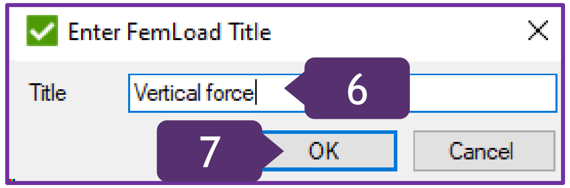






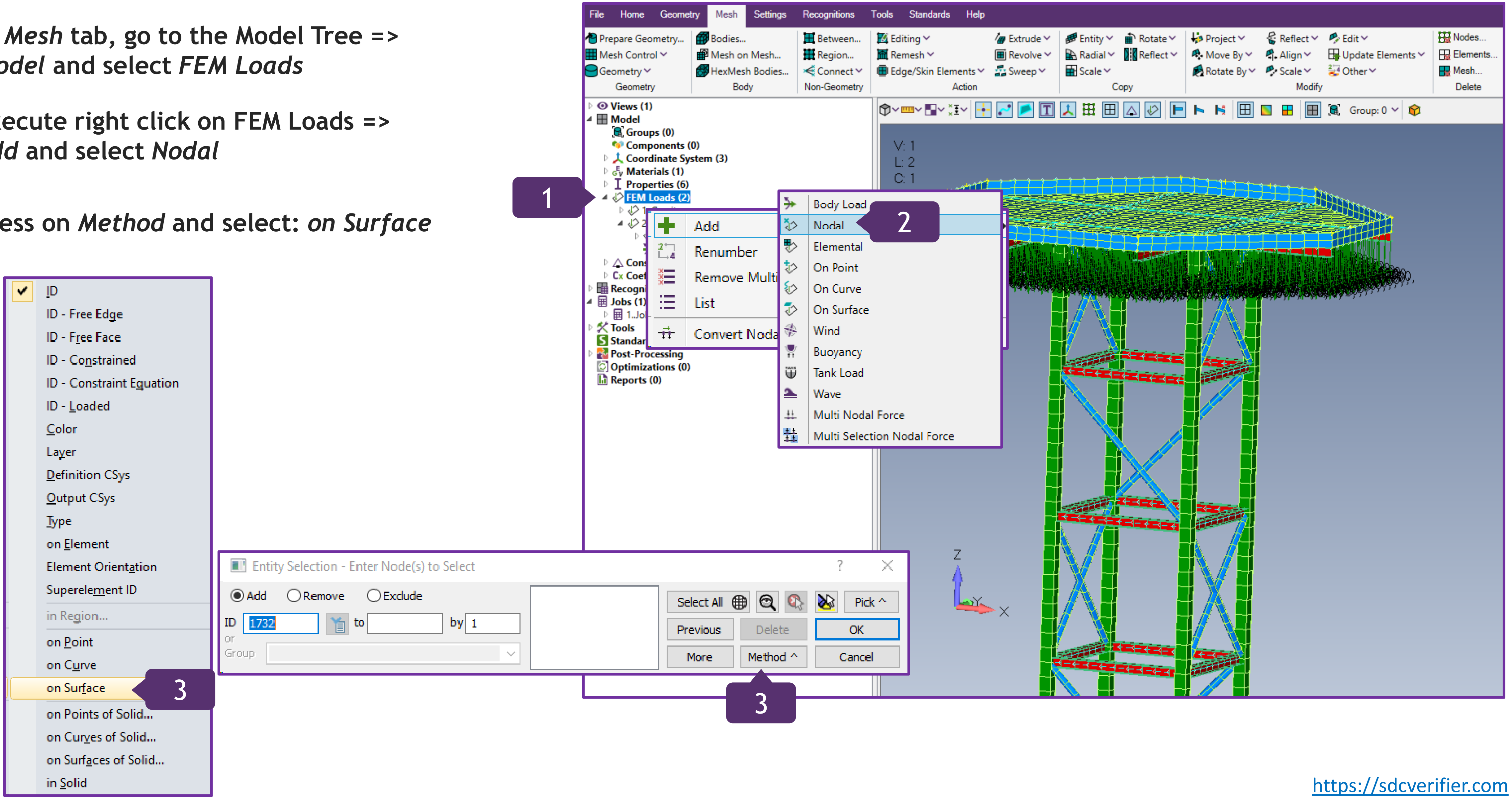






Create Side Load

1. In *Mesh* tab, go to the Model Tree => *Model* and select *FEM Loads*
2. Execute right click on FEM Loads => *Add* and select *Nodal*
3. Press on *Method* and select: *on Surface*



Create Side Load (Continuation)

SDC VERIFIER

4 With the left clicks of the mouse, select the Nodes of one side surface

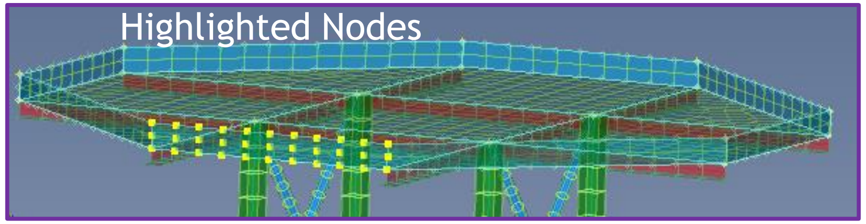
5 Press [icon] to check if all Nodes have been selected;
Press *OK*

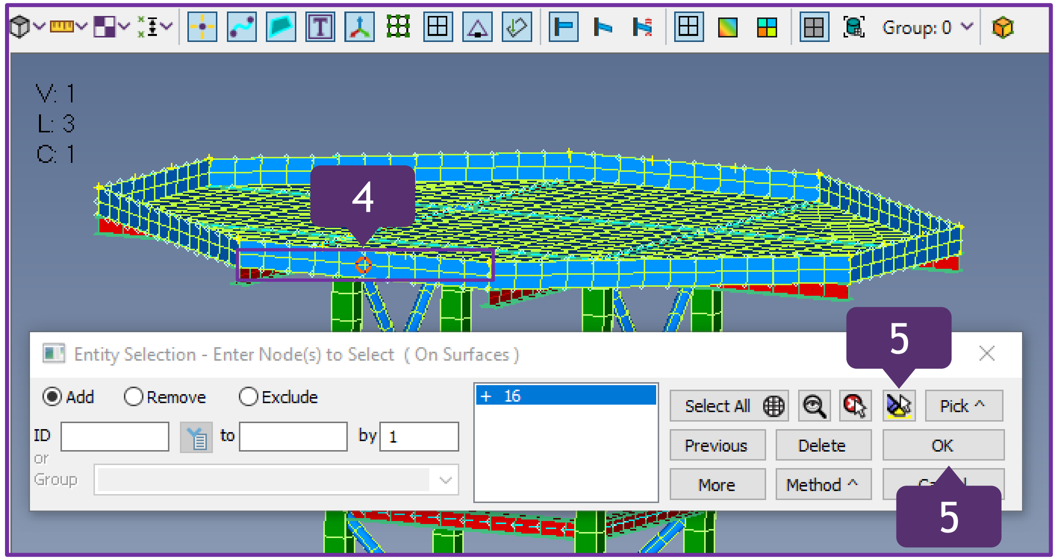
6 Title: *Side force*

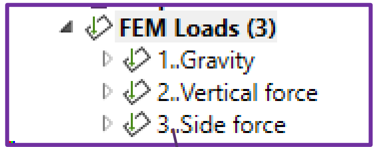
7 Select: *Force*;
Direction: *Components*

8 In *Load* section,
Value FY: -300 N

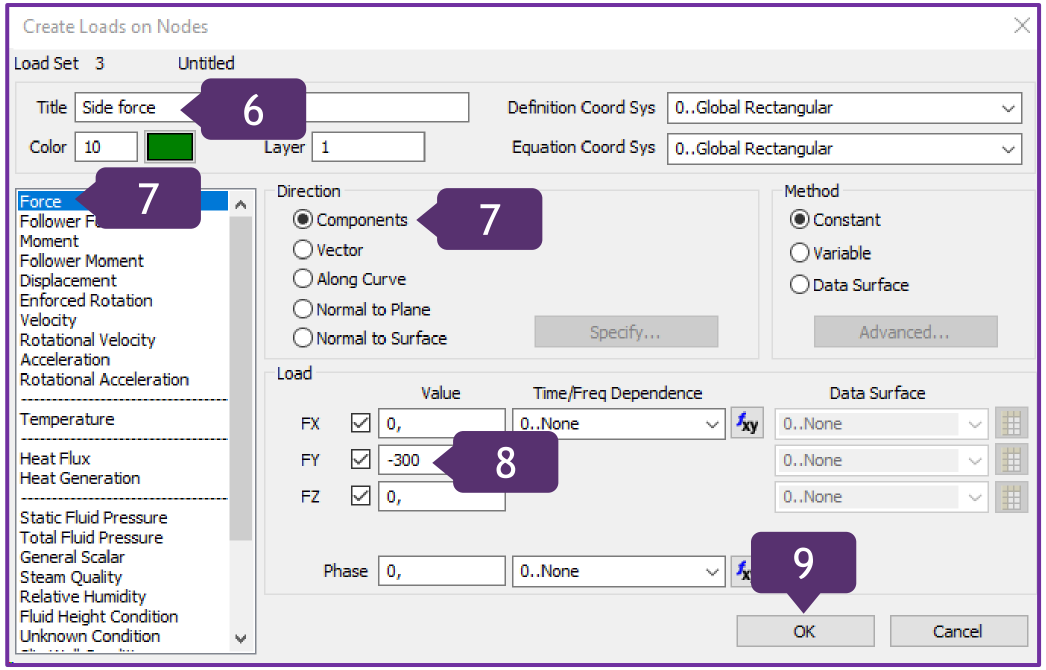
9 Press *OK*;
Close the menu

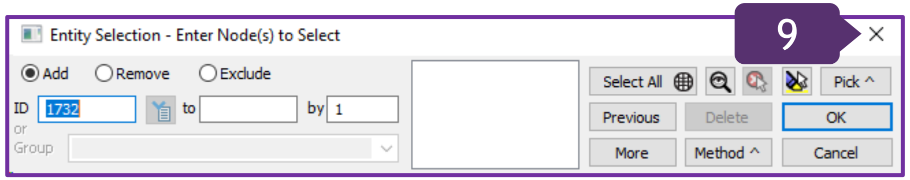


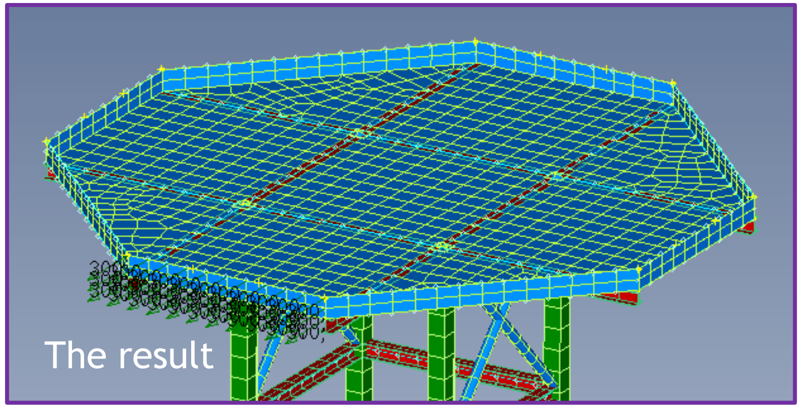




Renamed FEM Load





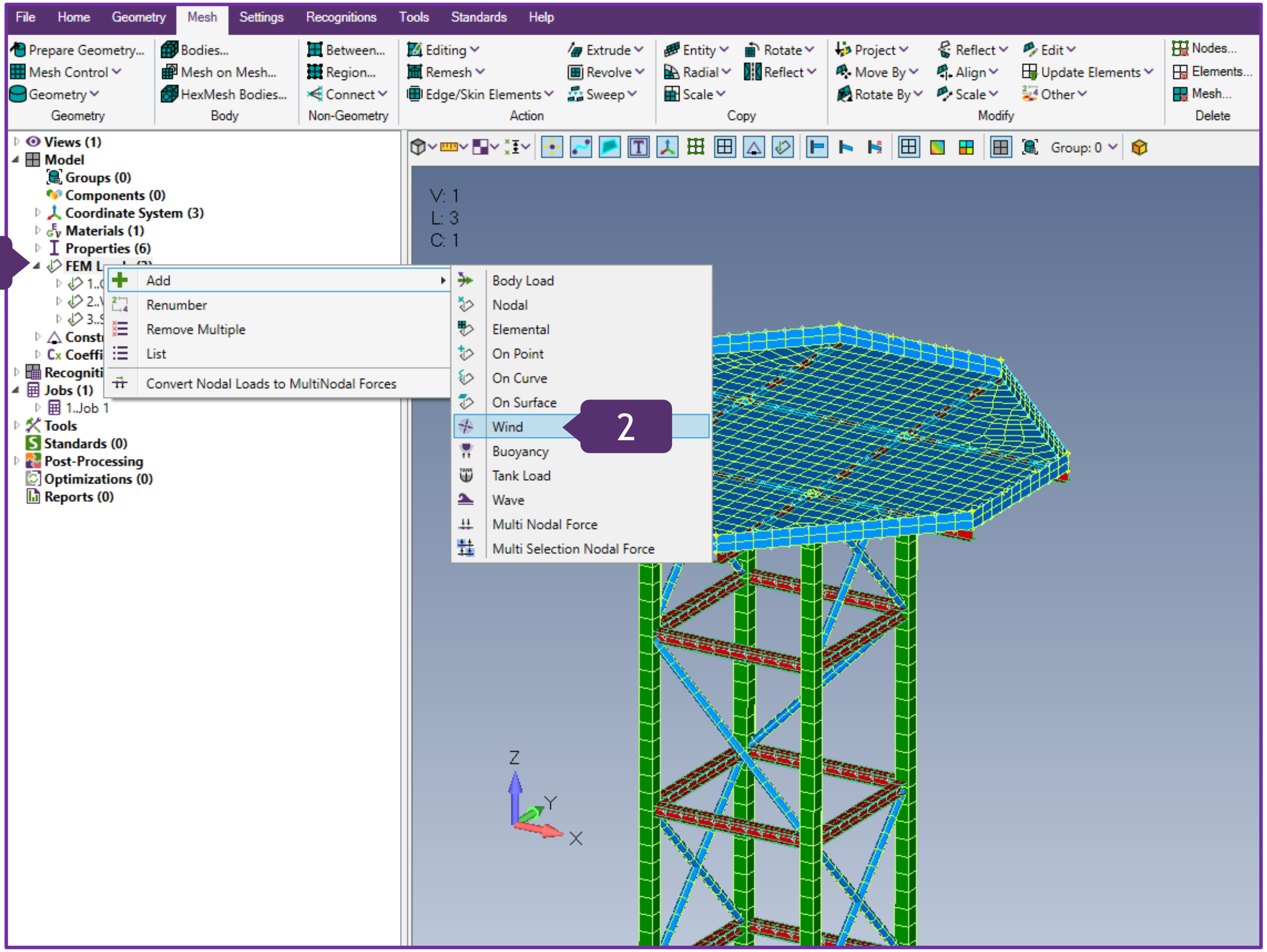


https://sdcverifier.com

Create Wind Load

1. In *Mesh* tab, go to the Model Tree => *Model* and select *FEM Loads*
2. Execute right click on FEM Loads => *Add* and select *Wind*

Only one Wind Load will be applied in X Direction.



Create Wind Load (Continuation)

3 Title: *Wind X*

4 In Settings, Vertical Direction: *Z*; Direction: *X*

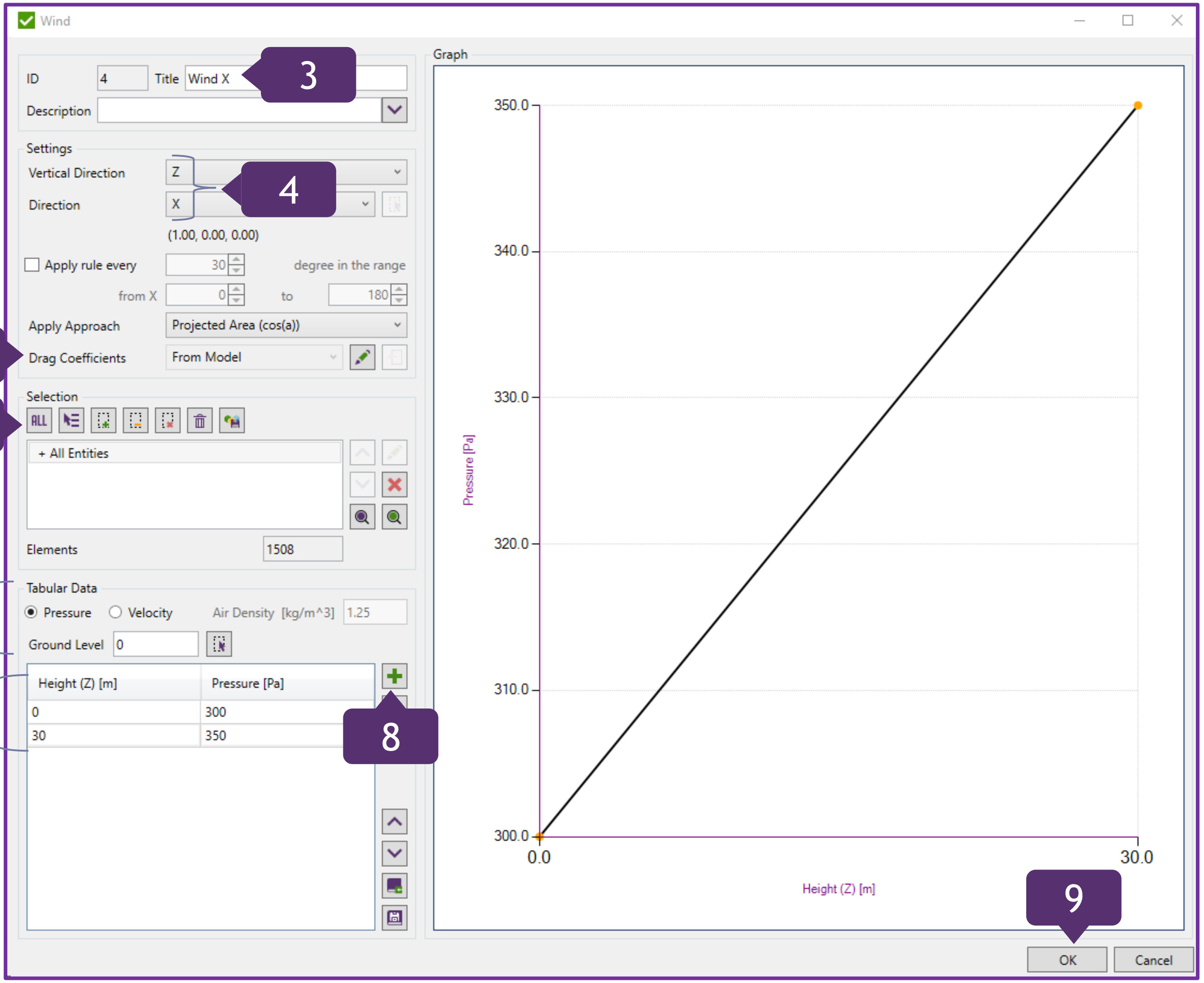
5 Drag Coefficients: *From Model*

6 In Selection, press ALL to add *All Entities*

7 In Tabular Data, select: *Pressure*; Ground Level: *0*

8 To define Pressure Profile, click on + two times

9 Height (Z) [m]: *0* - Pressure [Pa]: *300*; Height (Z) [m]: *30* - Pressure [Pa]: *350*; Press *OK*



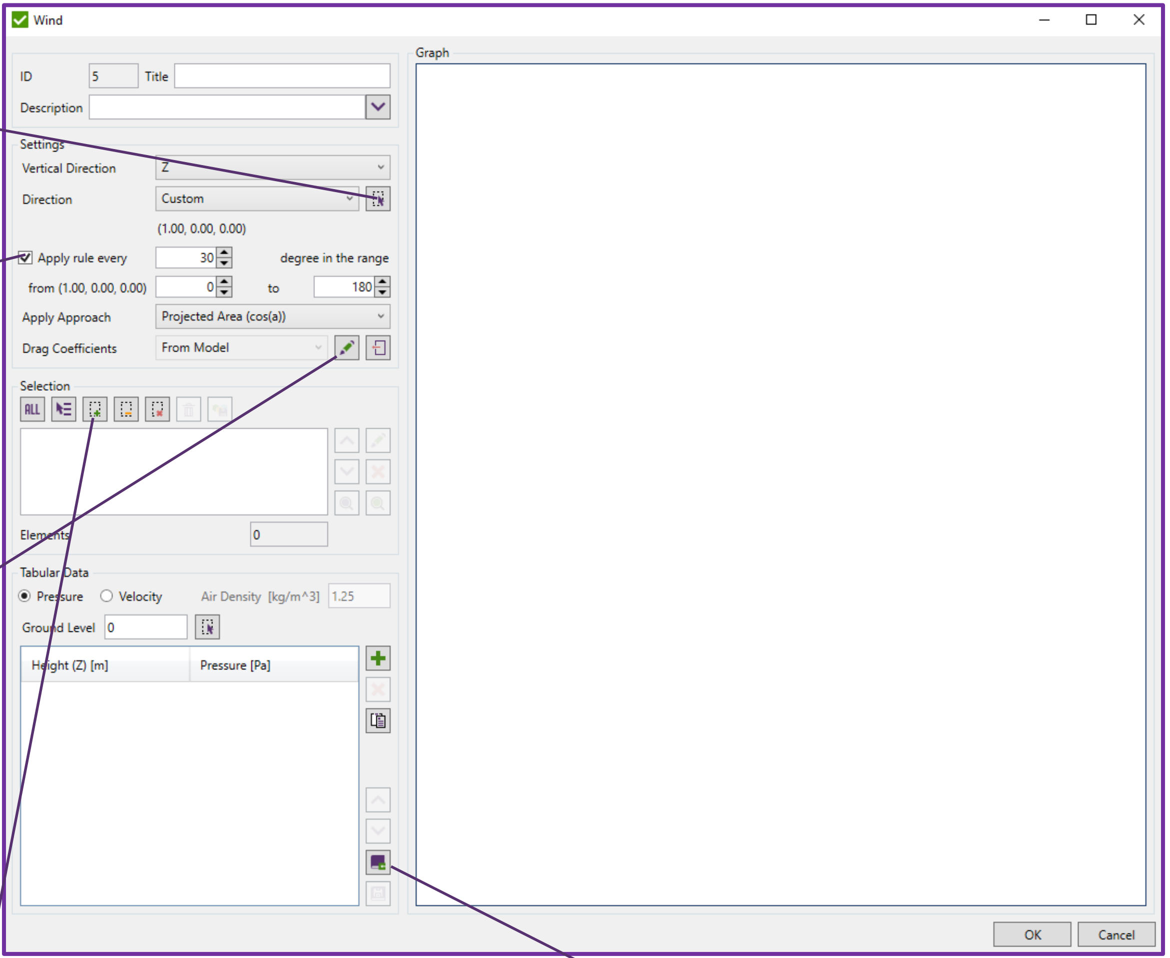
The graph demonstrates the wind pressure over height.

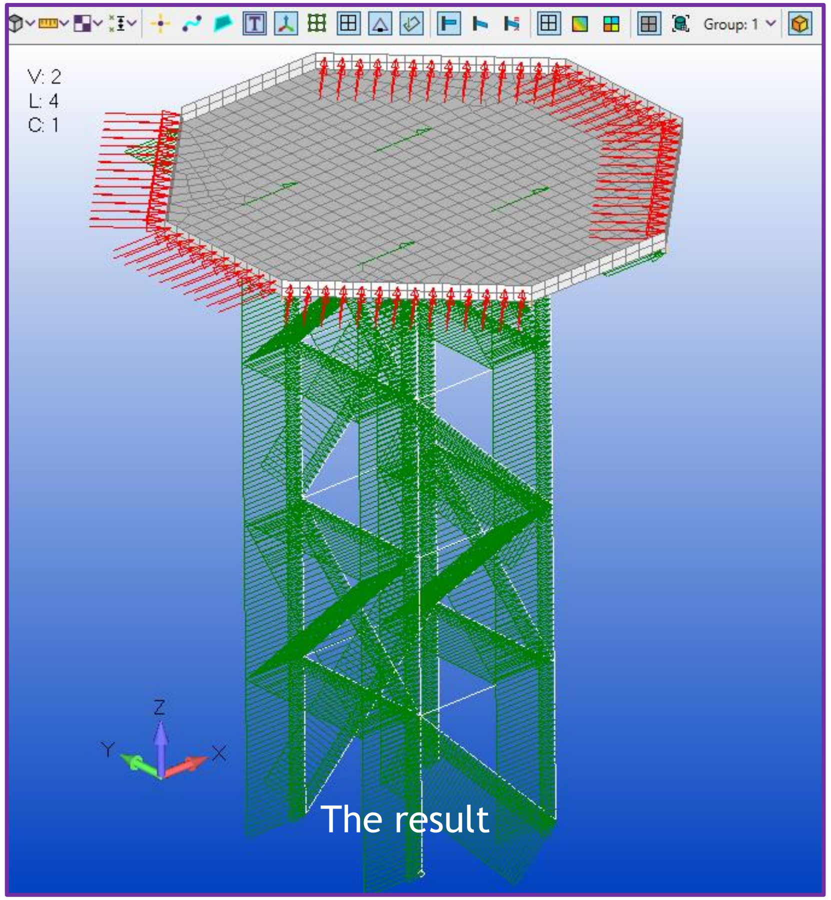
Expanded Functionality of Creating Wind Load

The Direction can also be customized by selecting the Custom option, and then pressing

If it is required to apply Wind Load along a particular vector for a certain number of degrees, activate Apply rule every degree from 0 to a selected degree in the range.

Drag Coefficients can also be defined by pressing





The selection of parts of the model that will be subjected to Wind Load, can also be selected manually by pressing

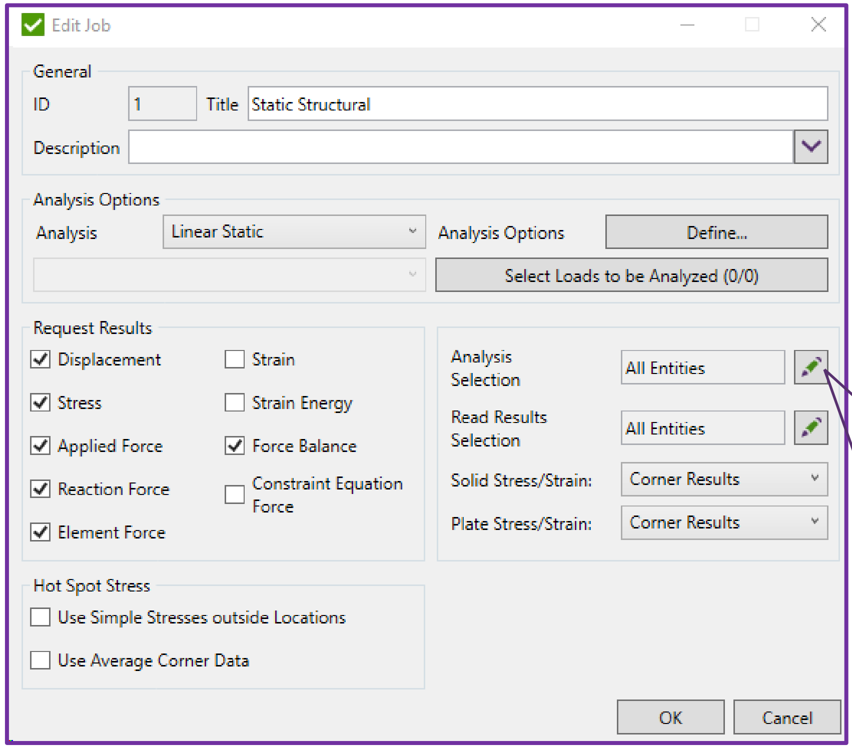
As an option, the profile for Tabular Data can be uploaded from Wind Library for different Loads by pressing

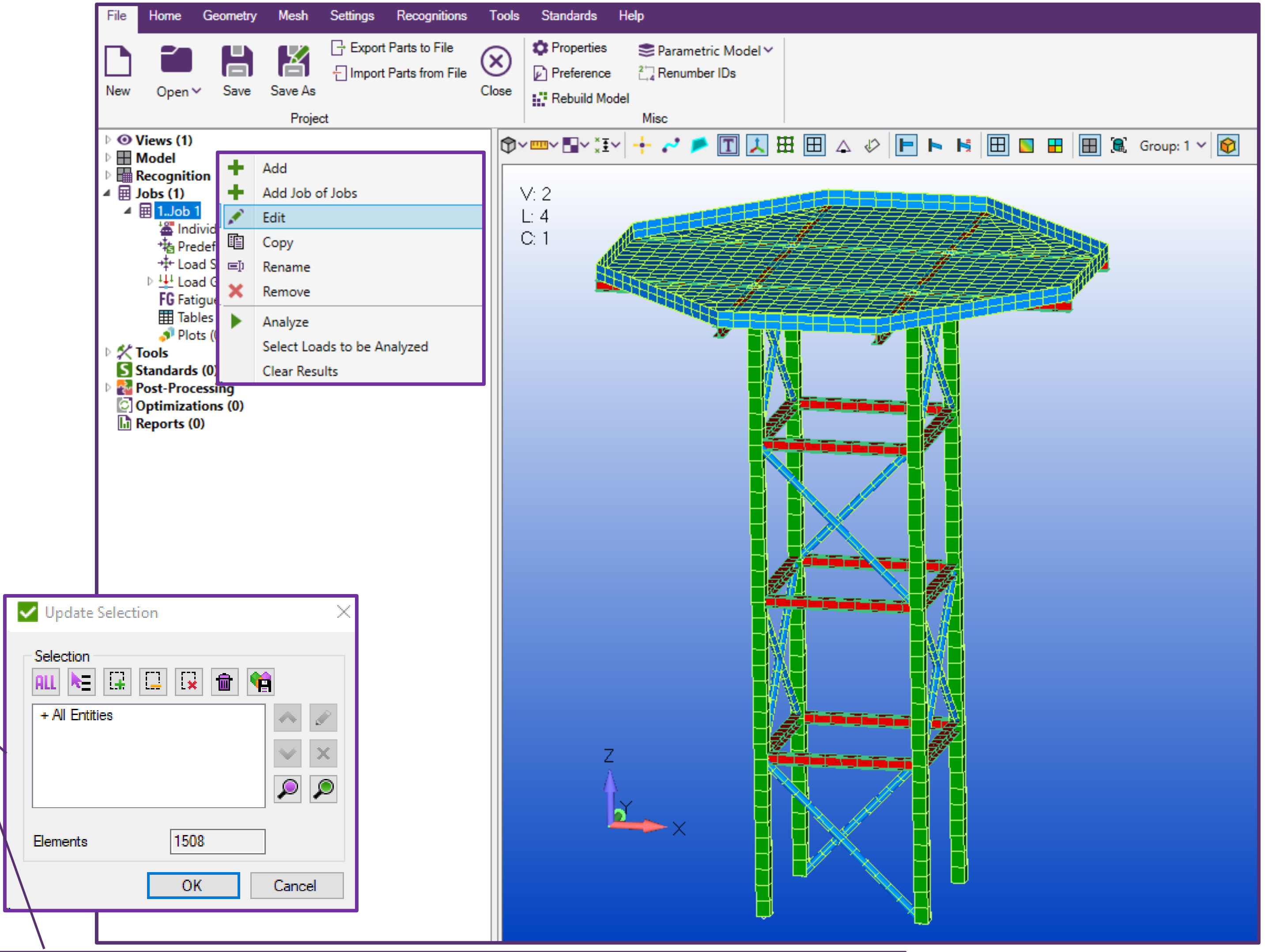
The Wind Load will be applied to all entities, which are subjected to wind. Yet, entities that are along X direction, will not be influenced by any Wind Load.

Edit Job

SDC VERIFIER

1. In *File* tab, go to the Model Tree => *Jobs* and select *1..Job 1*
2. Execute right click on *1..Jobs 1* and select *Edit*
3. Title: *Static Structural*
   Press *OK*

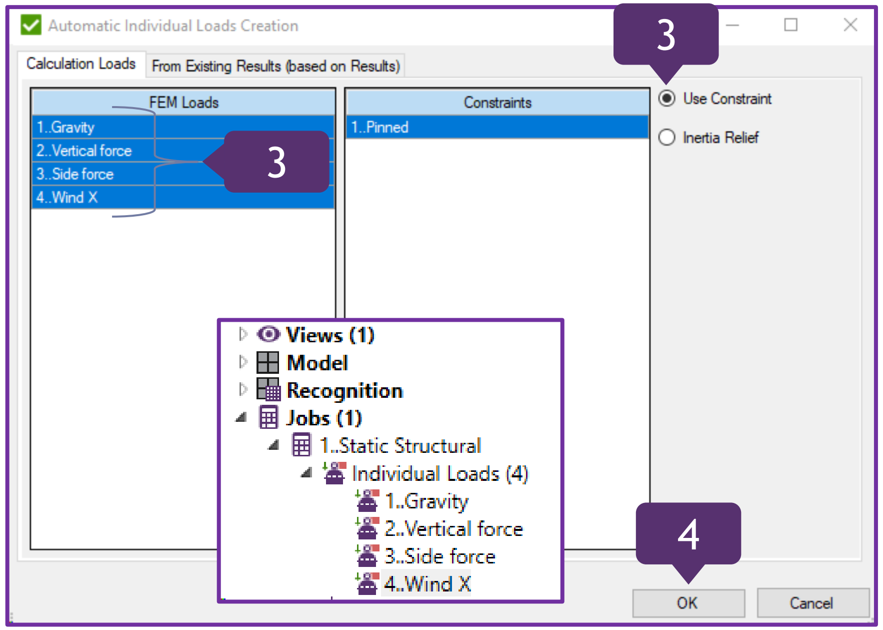




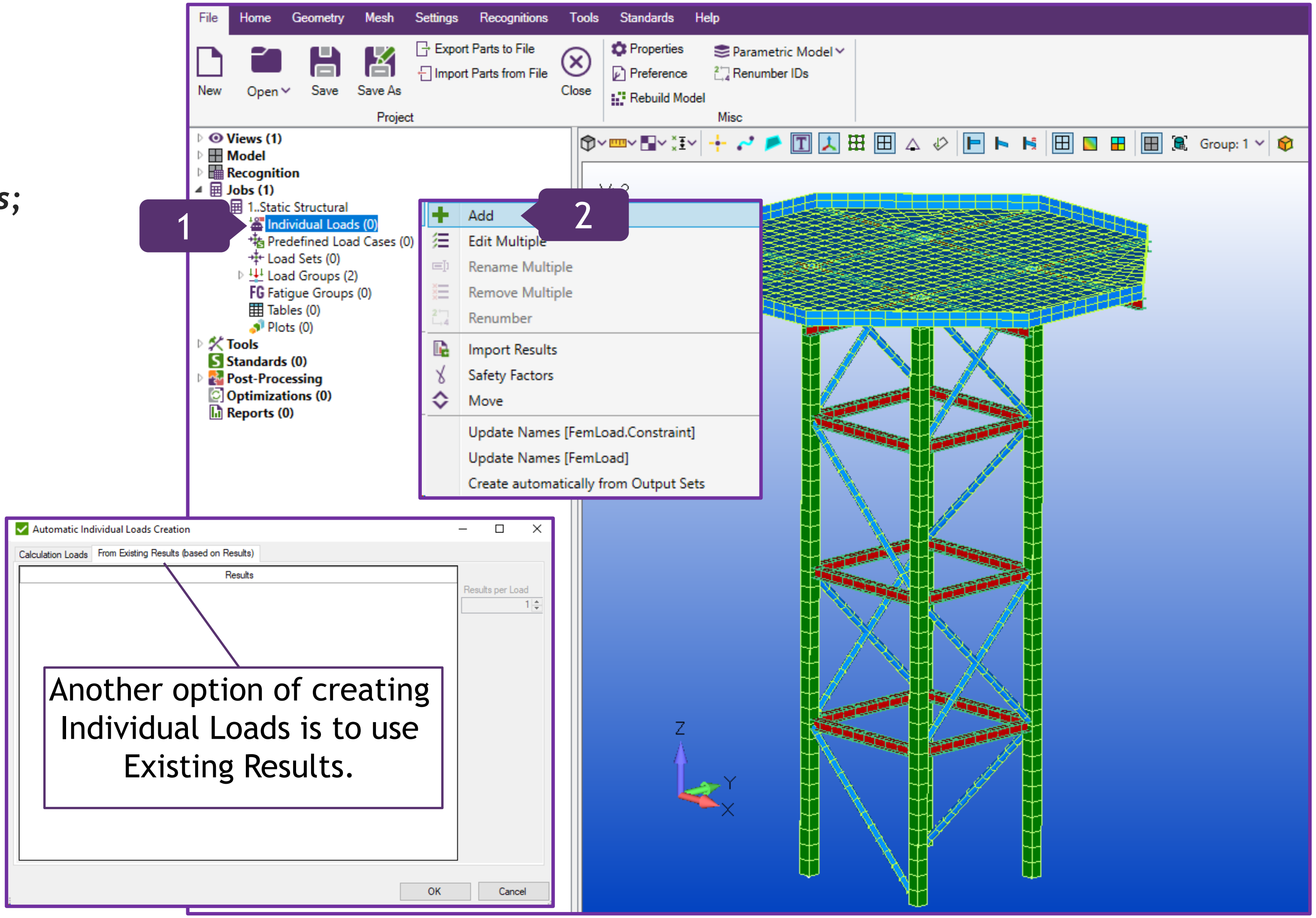
Additionally, a portion of a model can be analysed. If in the model under the analysis there are plenty of different groups or selections, they can be edited by clicking

Create Individual Loads

1. In *File* tab, go to the Model Tree => *Jobs* and select *Individual Loads (0)*
2. Execute right click on *Individual Loads (0)* and select *Add*
3. *In Calculation Loads* tab, select all *FEM Loads*; *Use Constraint* is ON
4. Press *OK*



Individual Load in SDC Verifier is a combination of FEM Load and Constraint.



Create Load Sets

1. In *File* tab, go to the Model Tree => *Jobs* and select Load Sets *(0)*

2. Execute right click on Load Sets *(0)* and select *Create/Edit Multiple*

3. In *Add load Sets* section, *Count: 4*; Press

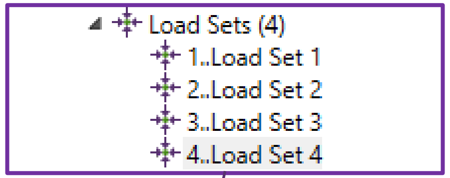
4. *IL1..Gravity:* select all *Load Sets; Factor: 1;* Press *Set*

5. *IL2..Vertical force: Load Set 1: 1; Load Set 3: 1.1*

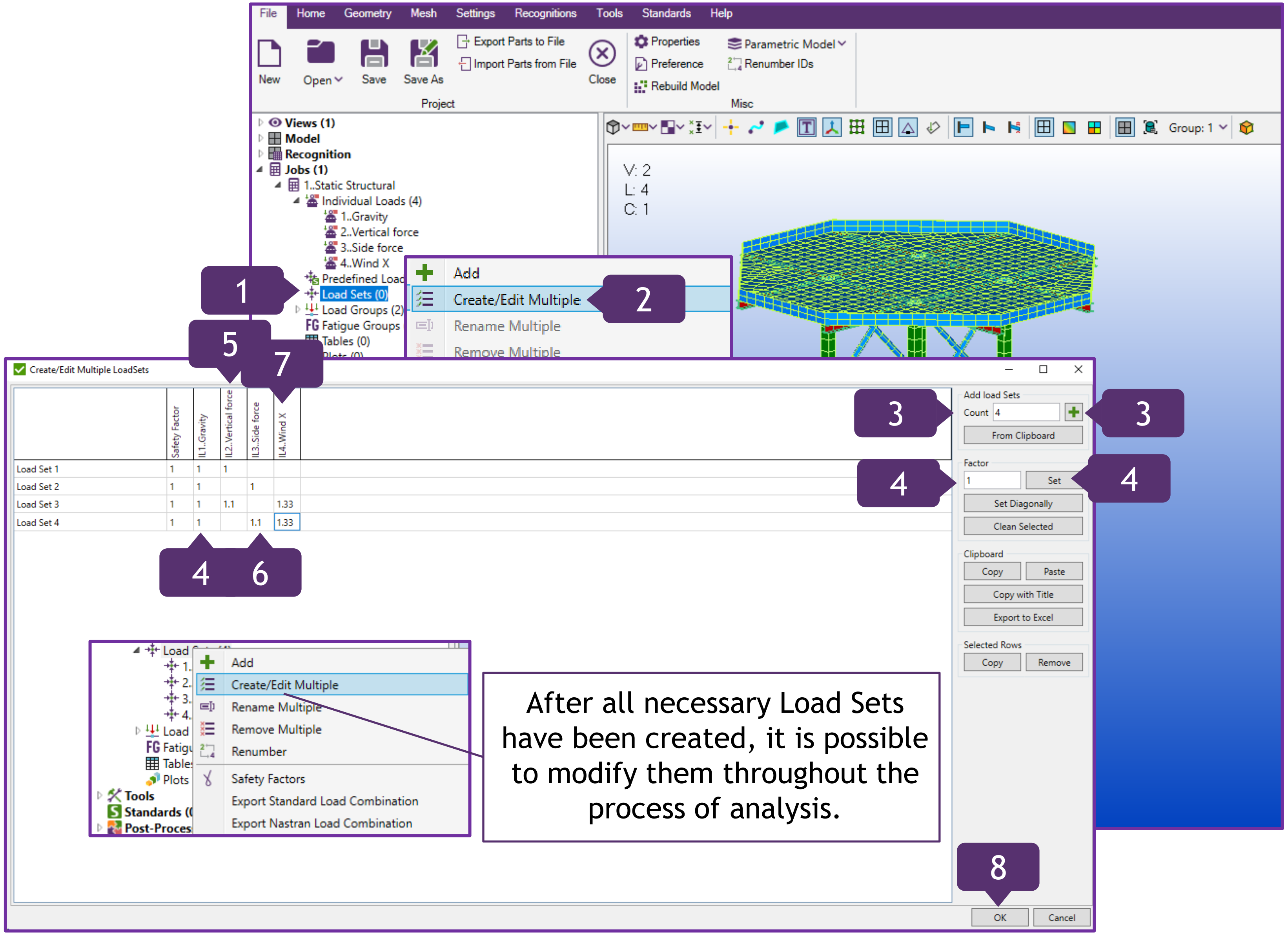
6. *IL3..Side force: Load Set 2: 1; Load Set 4: 1.1*

7. *IL4..Wind X: Load Set 3* and *Load Set 4: 1.33*

8. Press *OK*



Combinations have been created.



After all necessary Load Sets have been created, it is possible to modify them throughout the process of analysis.

Load Set is a linear combination of Individual Loads and a Safety Factor.

Load Groups Information

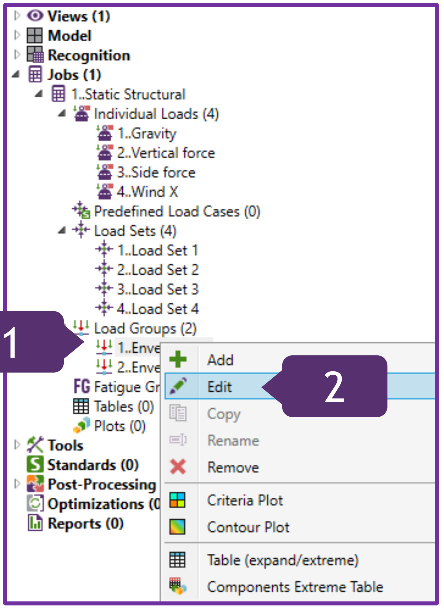
1 In *File* tab, go to the Model Tree => *Load Groups* and select 1..Envelope (IL)

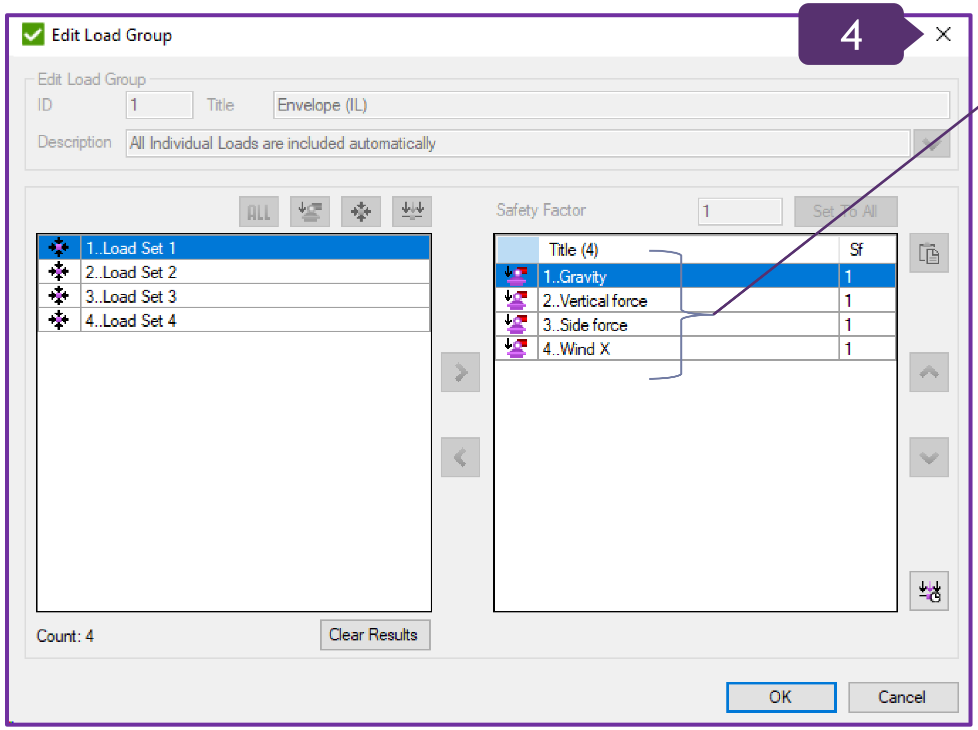
2 Execute right click on *Envelope (IL)* and select *Edit*

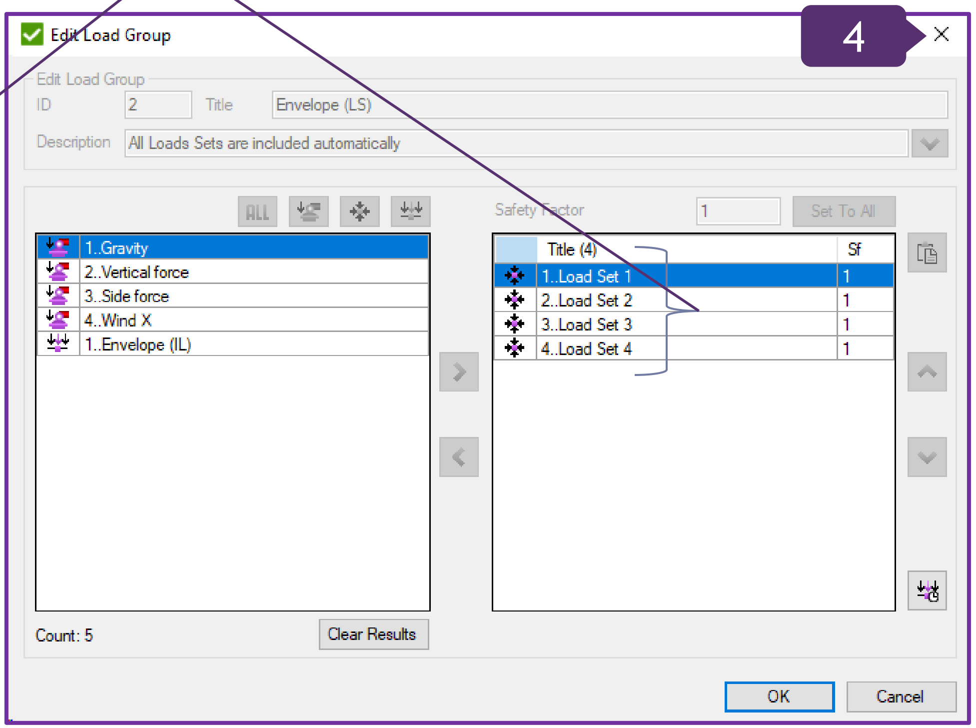
3 Repeat steps 1-2 for *2..Envelope (IS)*;

4 Close the menu

Envelope is a group of worst results of both Individual Loads and Load Sets.



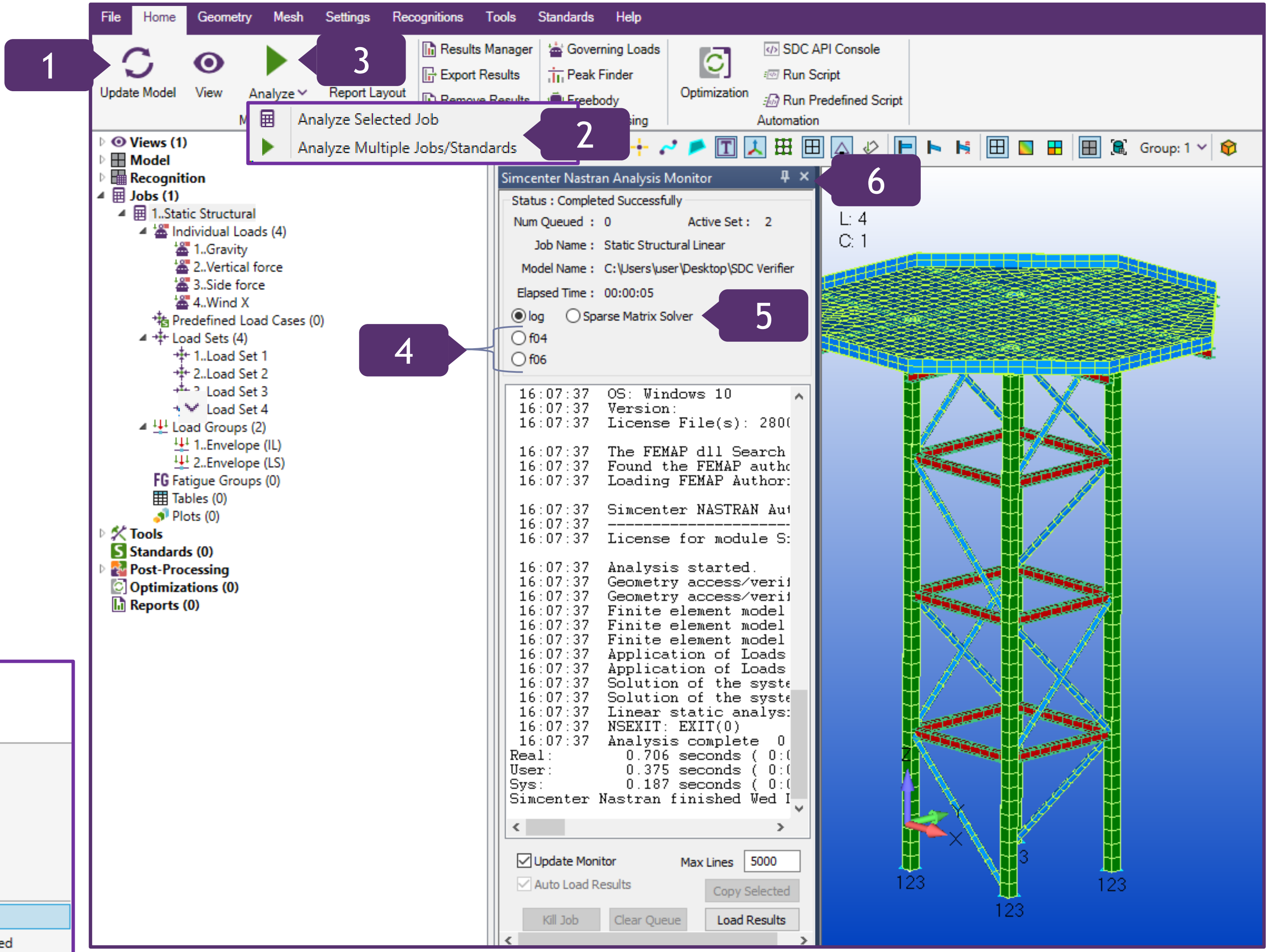




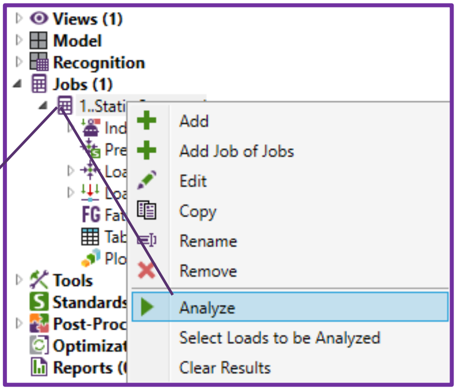
Run the Analysis

SDC VERIFIER

1. In *Home* tab, press *Update Model*
2. If the project contains multiple Jobs, press Analyze and select the required option.
3. Press *Analyze*
4. Activate *f04* or *f06* to check the log file
5. Activate *Sparse Matrix Solver* to check the Graphs of Equations
6. Close *Simcenter Nastran Analysis Monitor*



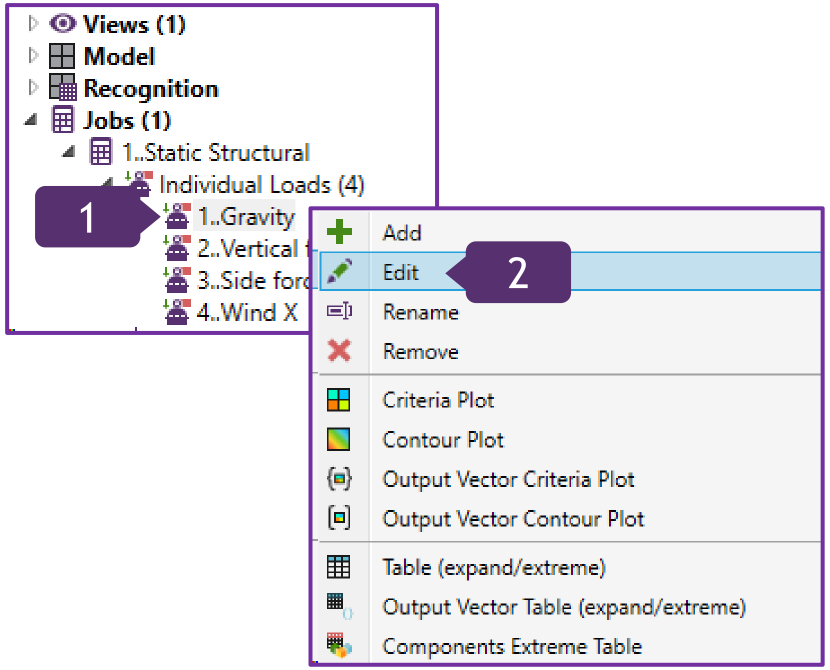
An alternative way of the Analysis start is to select 1..Static Structural; execute right click and select Analyze.

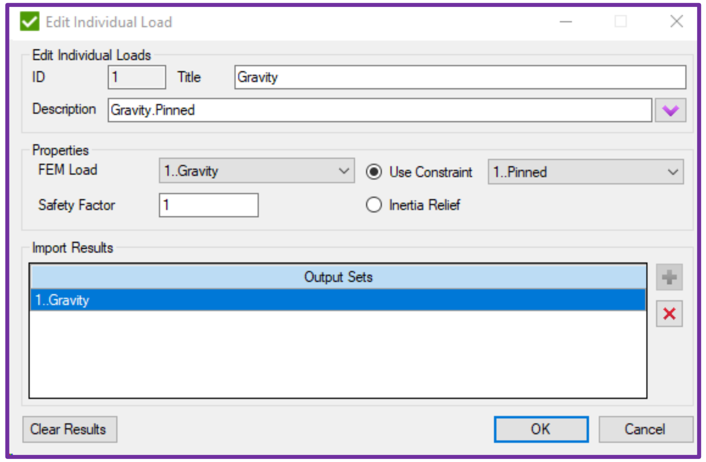


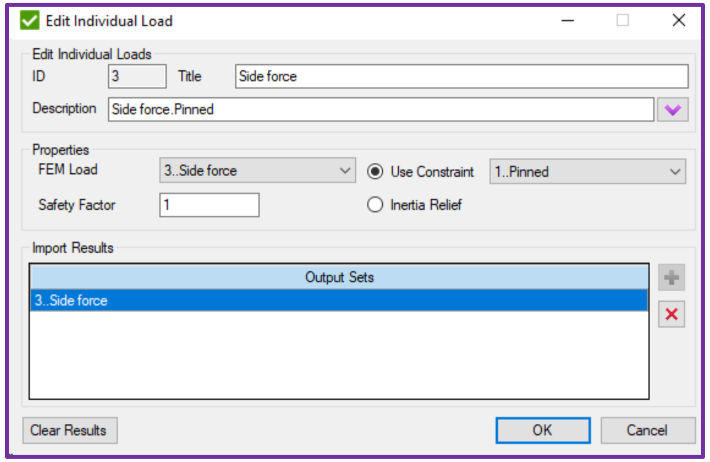
Check the Output Sets

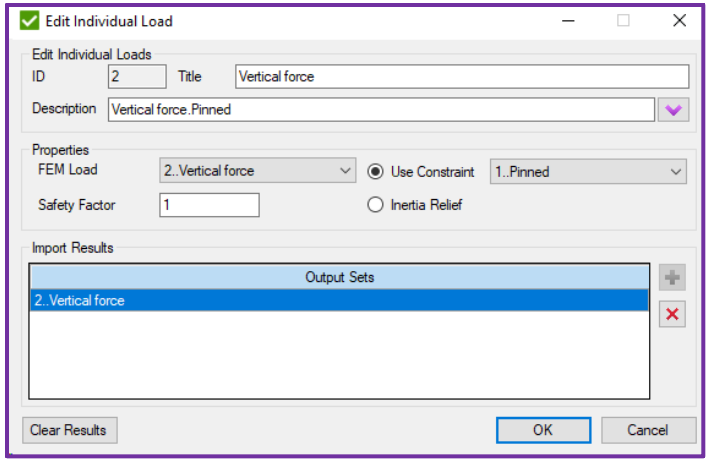
1. Go to the Model Tree => *Jobs* => Individual Loads and select *1..Gravity*
2. Execute right click on *1..Gravity* and select *Edit*

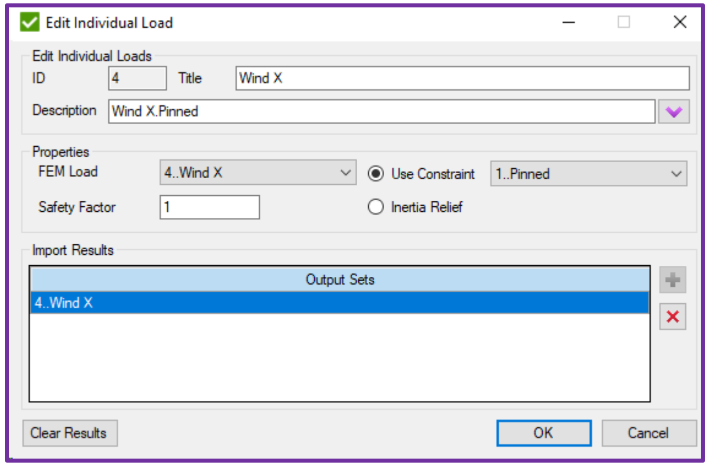
After the analysis, the Output Sets on each Individual Load have been created. In order to check if the results have been linked, any of Individual Loads can be edited by following the steps 1-2.





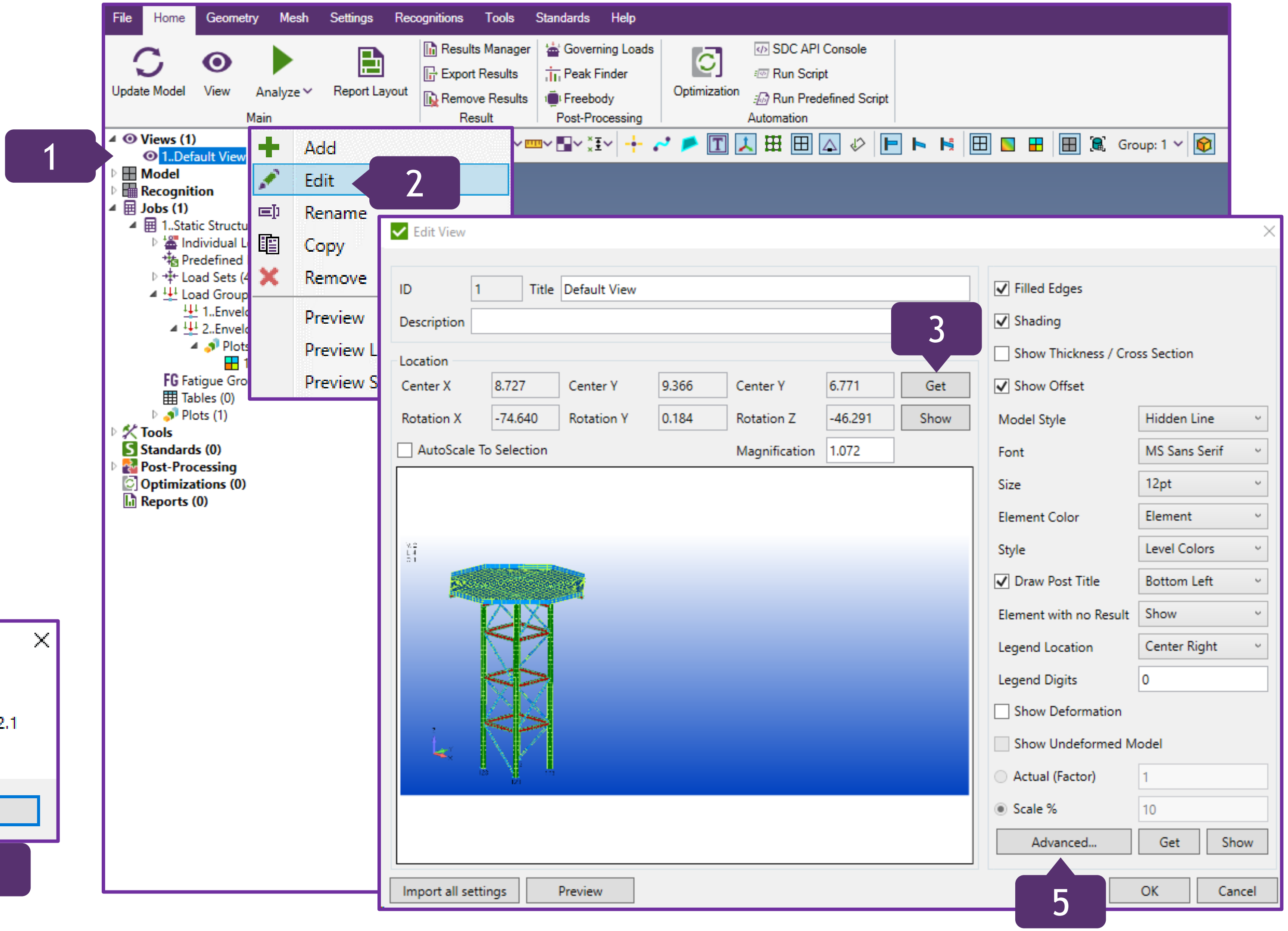


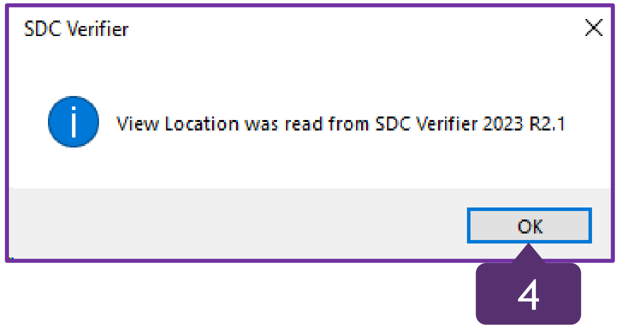




Set Views

1. In Views, select *1..Default View*
2. Execute right click on *1..Default View* and select *Edit*
3. Press *Get*
4. In the pop-up window, press *OK*
5. Press on *Advanced…*





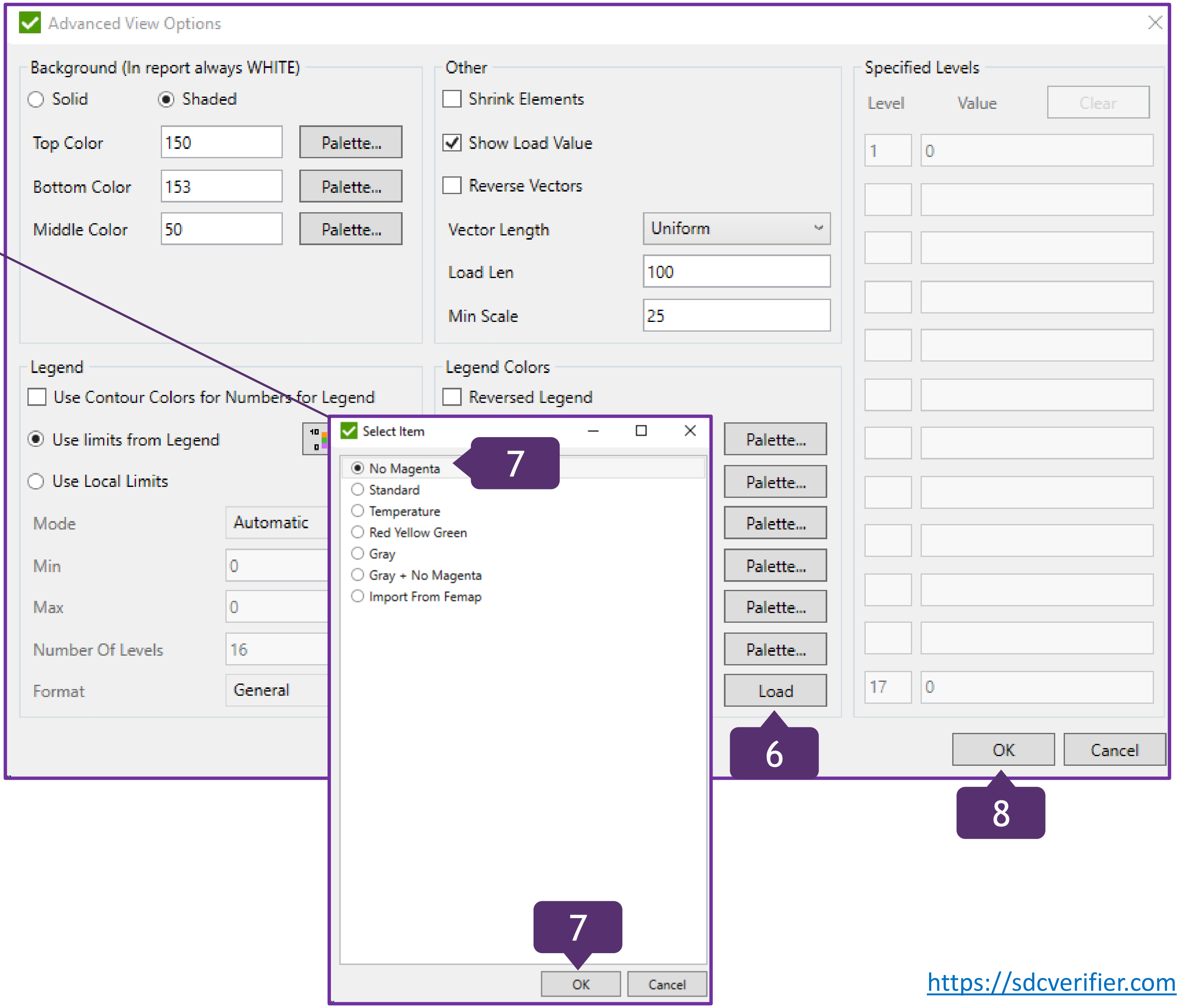
Set Views (Continuation)

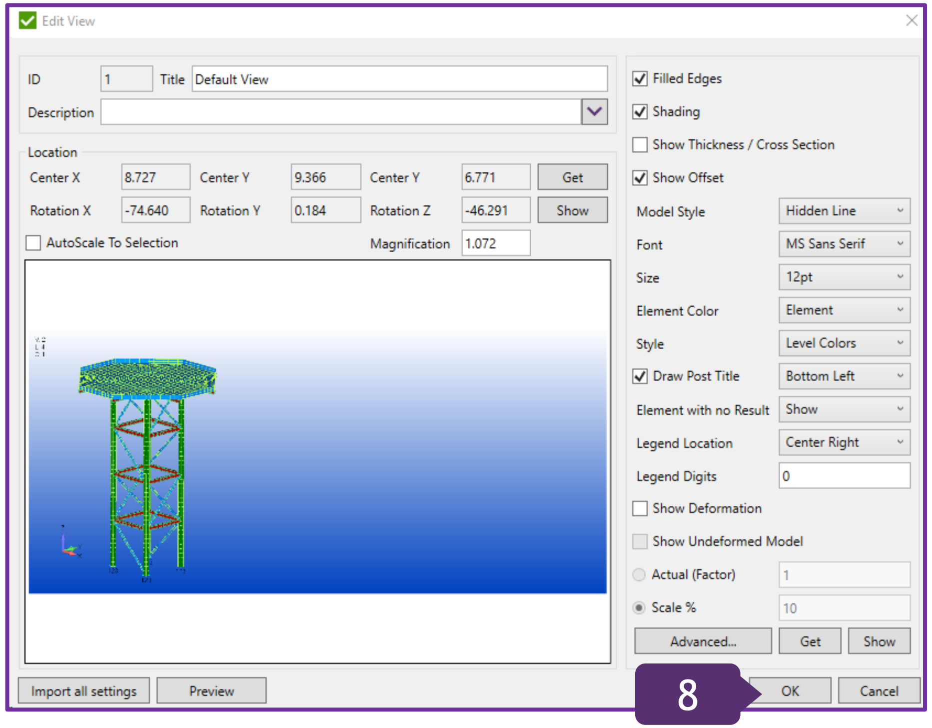
6 Press on *Load*

7 No Magenta *ON*; Press *OK*

8 Press *OK*

The selection of preferences in Advanced Settings is based on the requirements of the project or personal choice.

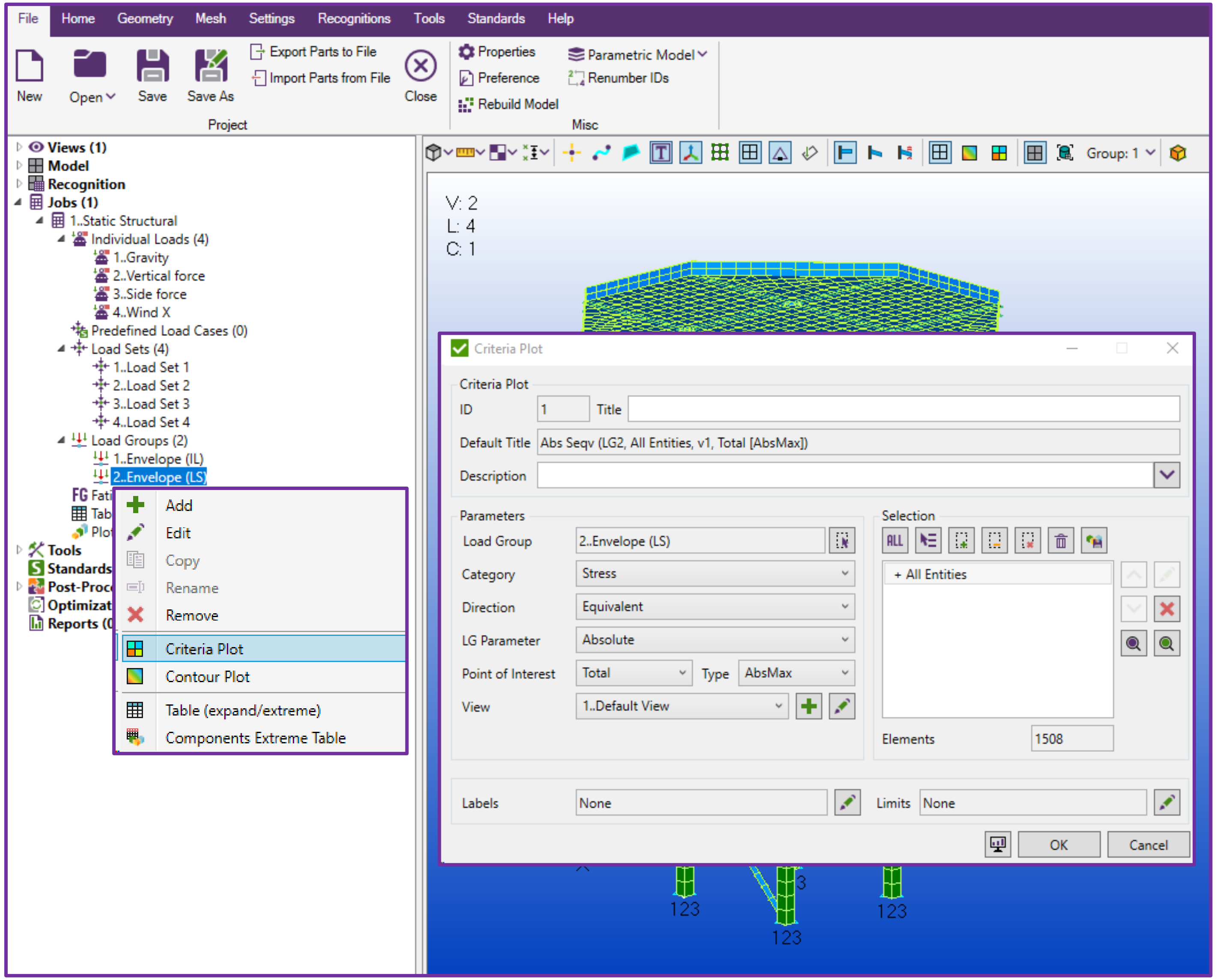




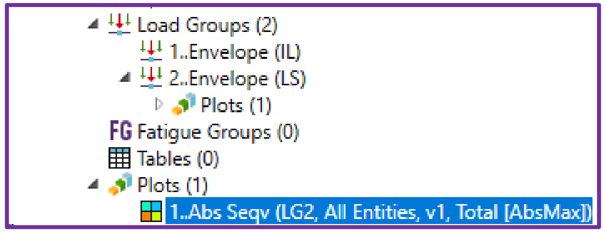
Define Criteria Plot

SDC VERIFIER

1. To preview results, go to the Model Tree => *Load Groups* and select *2..Envelope (LS)*
2. Execute right click on *2..Envelope (LS)* and select *Criteria Plot*
3. In Parameters, Load Group: *2..Envelope (LS)*
4. Category: *Stress*
5. Direction: *Equivalent*
6. Point of Interest: *Total*; Type: *AbsMax*
7. Selection: + *All Entities*
8. Press *OK*



The Plot has been added.



Preview the Plot

1. To preview the *Plot*, go to *Jobs => Plots =>1..Abs Seqv (LG2, All Entities,v1, Total [AbsMax])*
2. Execute right click on *1..Abs Seqv (LG2, All Entities,v1, Total [AbsMax])* and select *Preview*