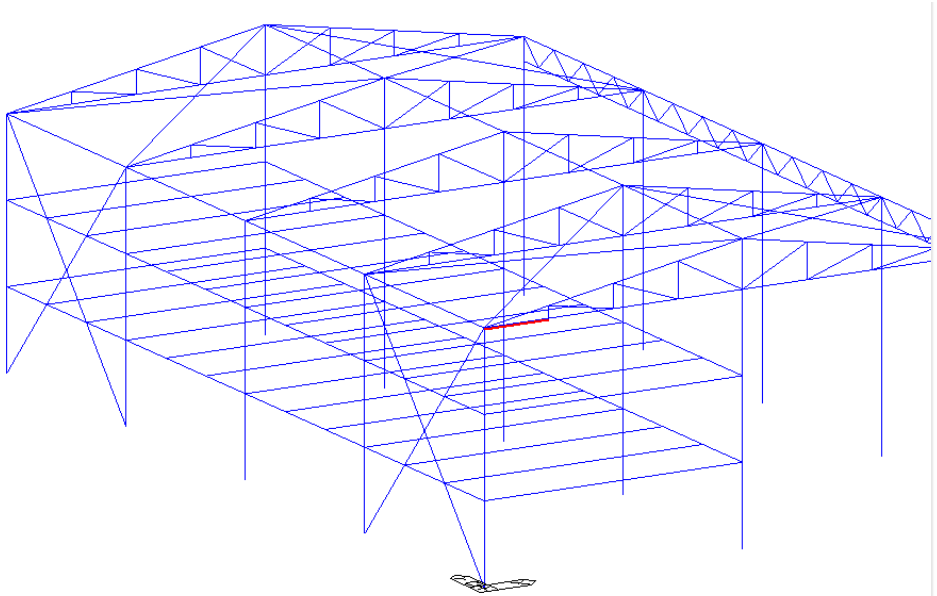
FEA NX Simulation Platform

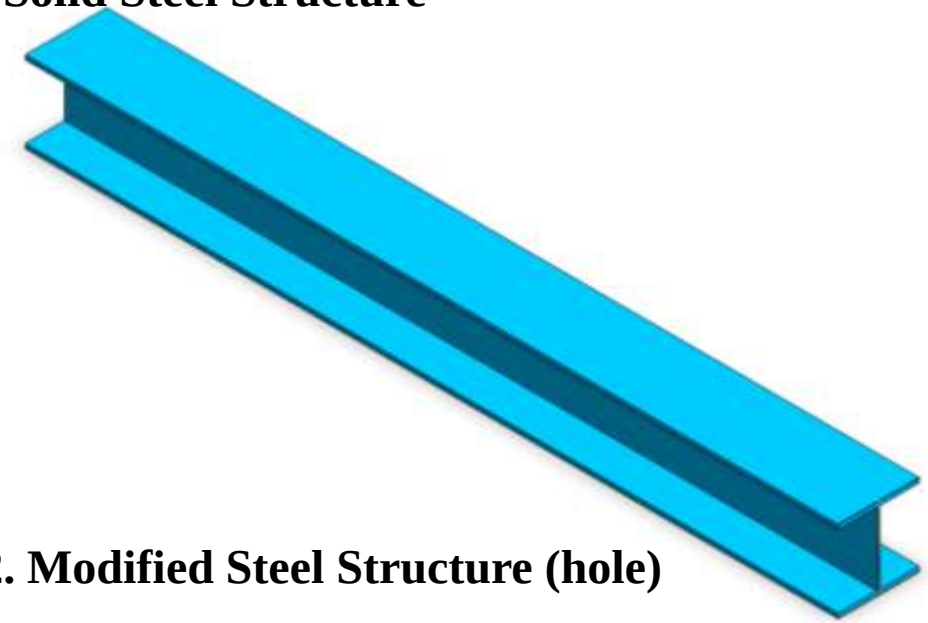
Integration of MIDAS **GEN** and **FEA NX** Series Structural Element Detailed Analysis (Simplified Approach)

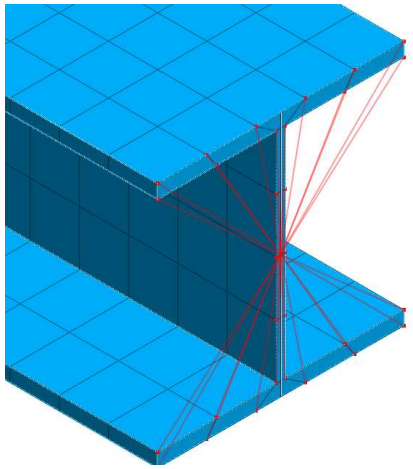
沈約翰
john@midasuser.com.tw

CASE ANALYSIS OVERVIEW

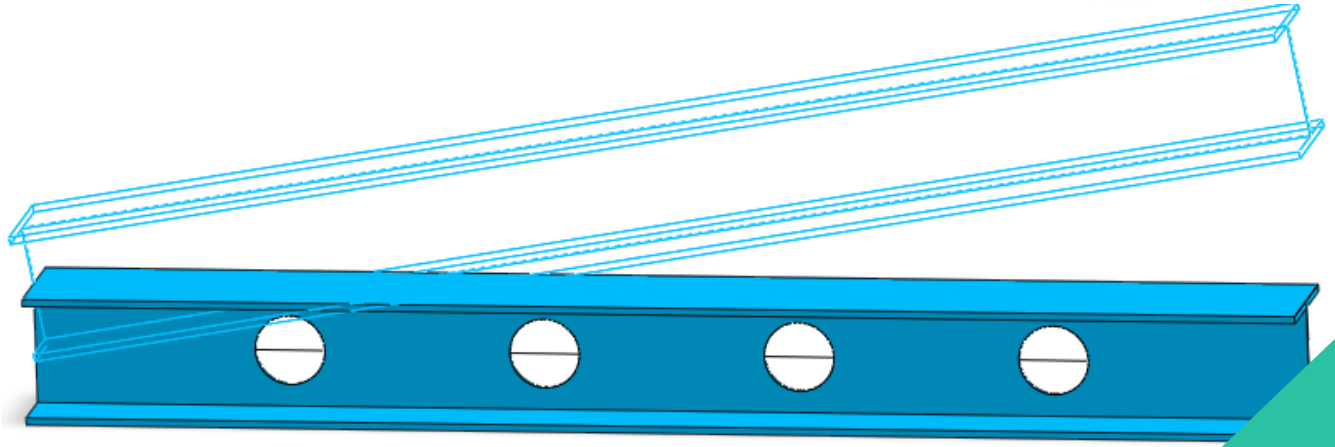


Part1. Solid Steel Structure





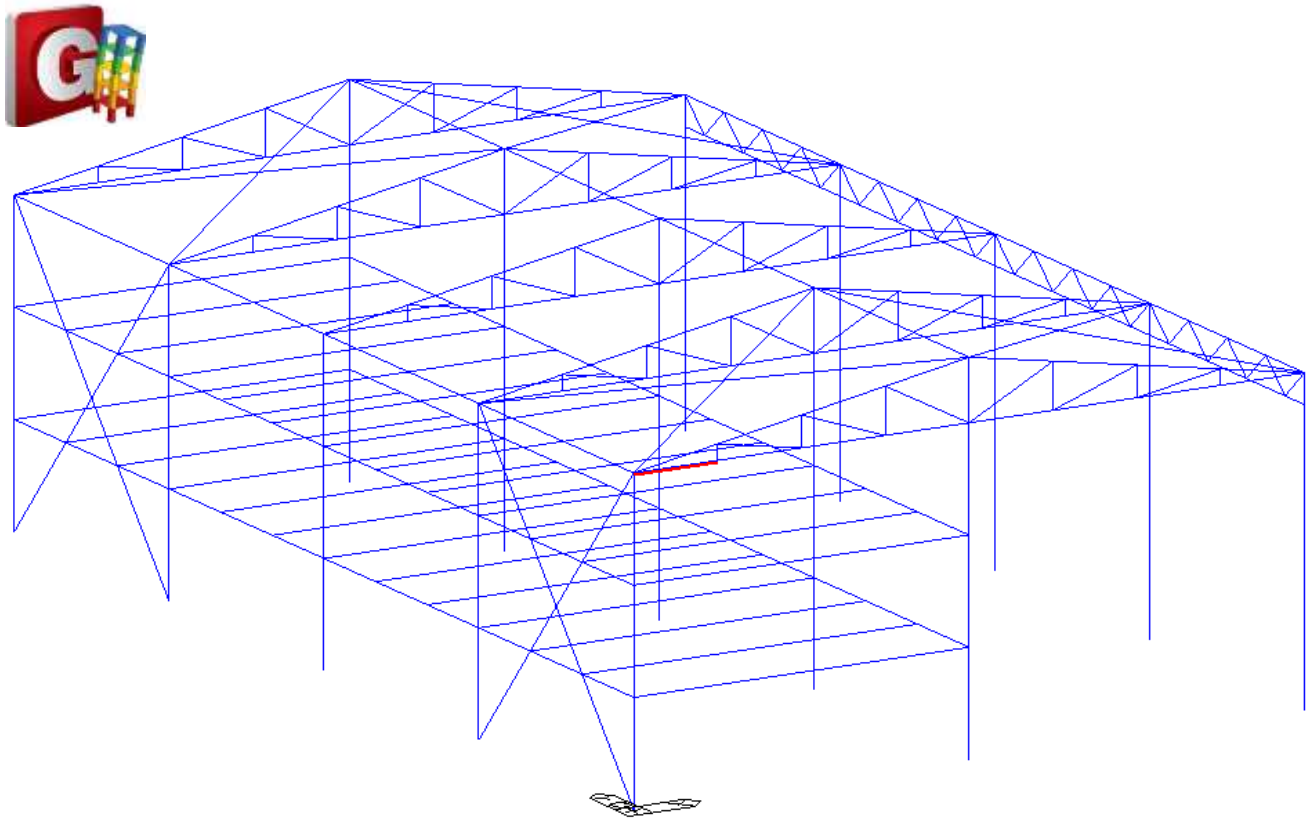
Part2. Modified Steel Structure (hole)

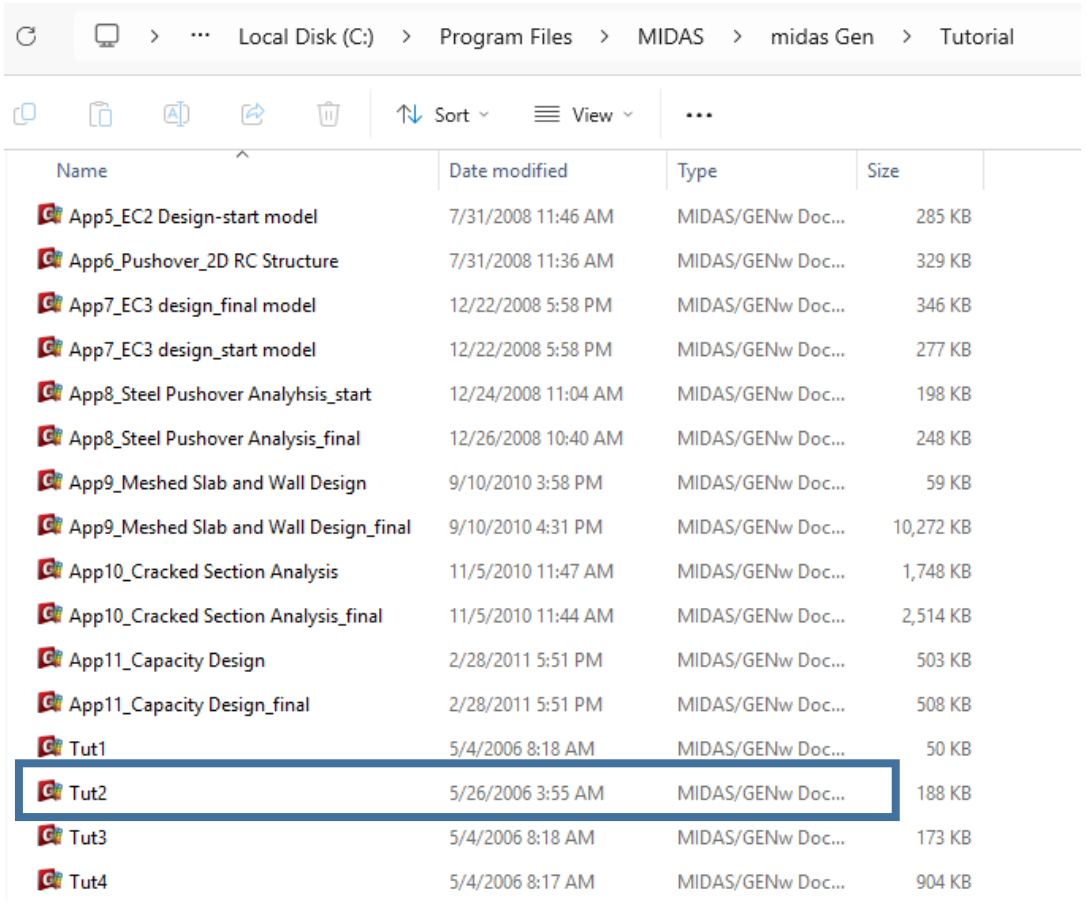


Part 0

GEN Simulation and Results

MIDAS GEN MODEL and RESULTS

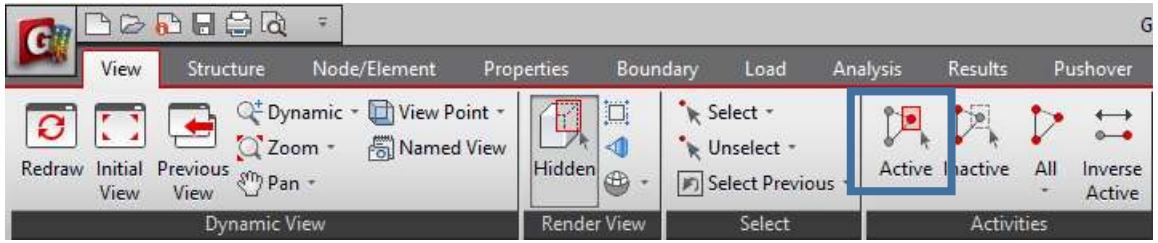


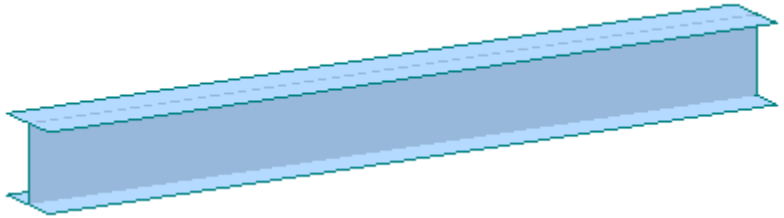


Warehouse Frame Example

- Retrieve setting file from your MIDAS Gen Installation Folder
- Local Disk (C:) > Program Files > MIDAS > Midas Gen > Tutorial > Tut2

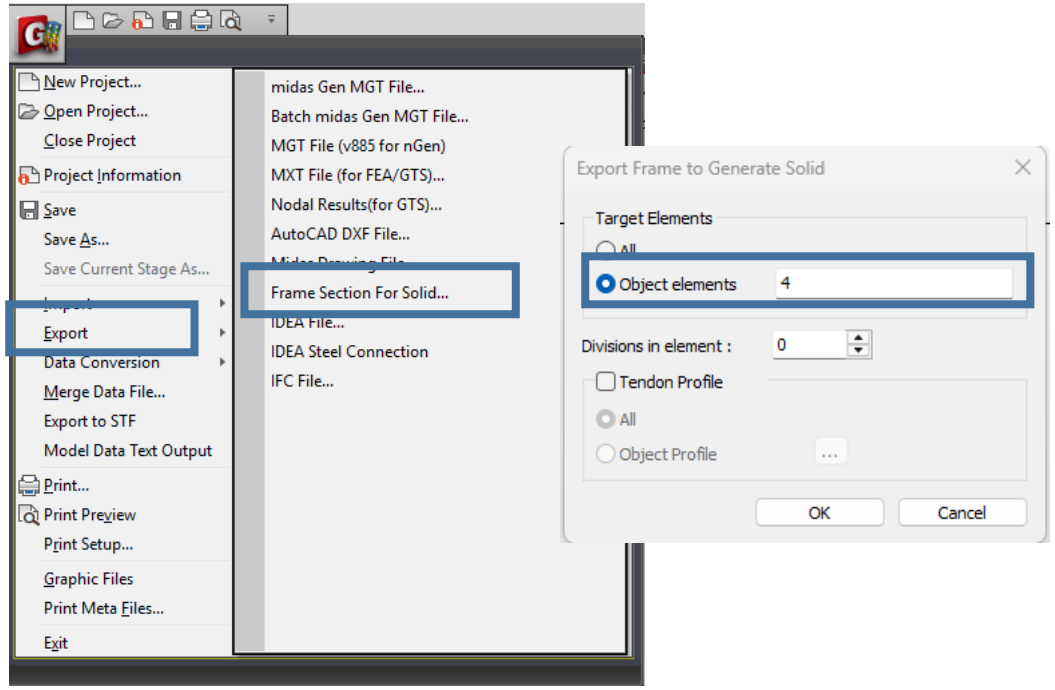
CASE ANALYSIS 1: GEN Tutorial 2 Example





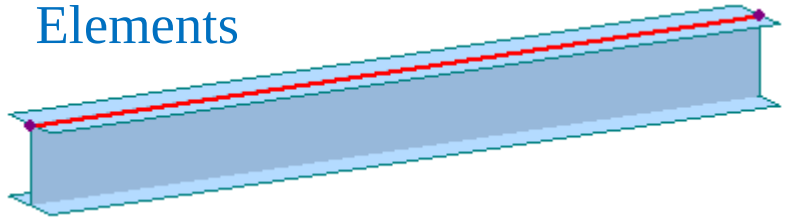
Isolating the element (beam)

- Highlight the beam
- Use Active

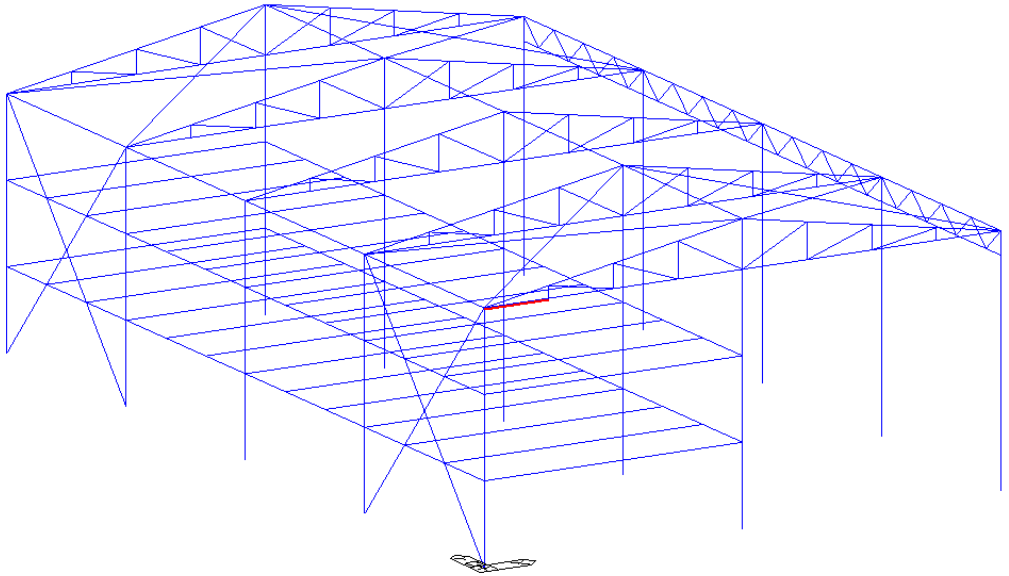


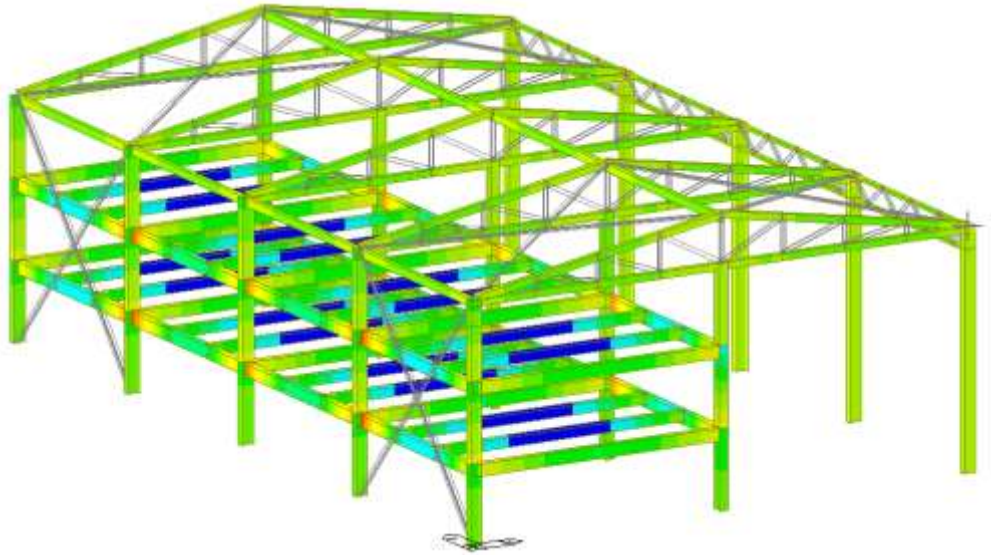
Extracting the element (beam)

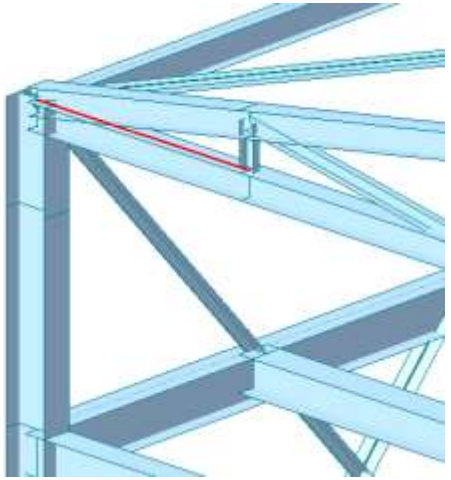
Export > FrameSection For Solid > Object Elements > Highlight the Elements

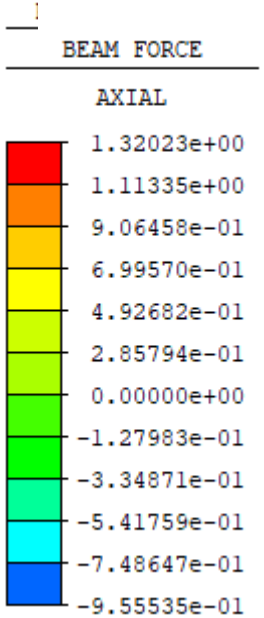


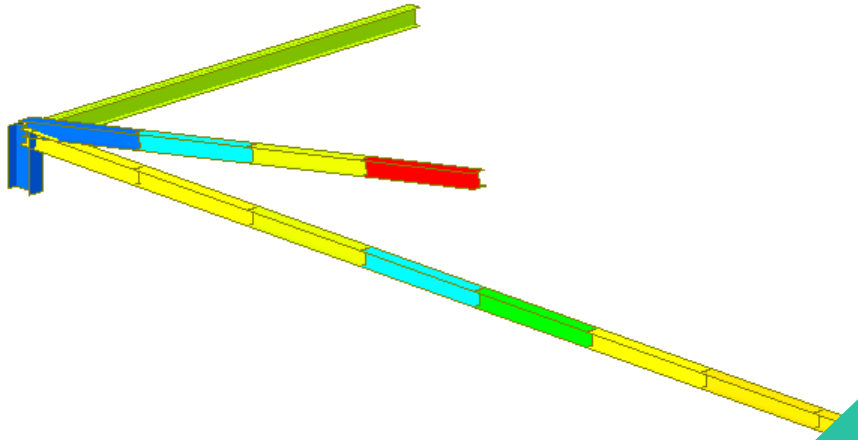
MIDAS GEN MODEL and RESULTS







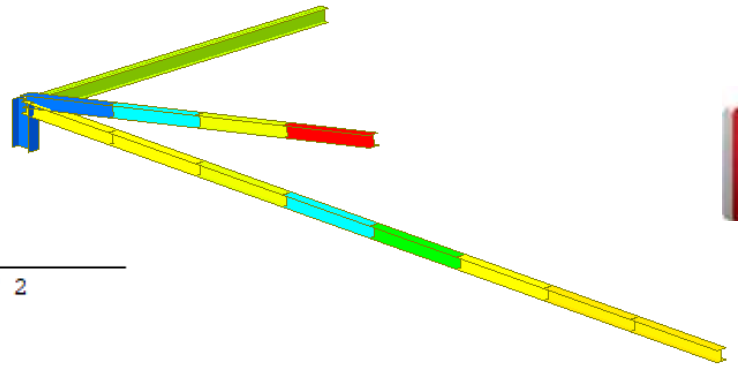


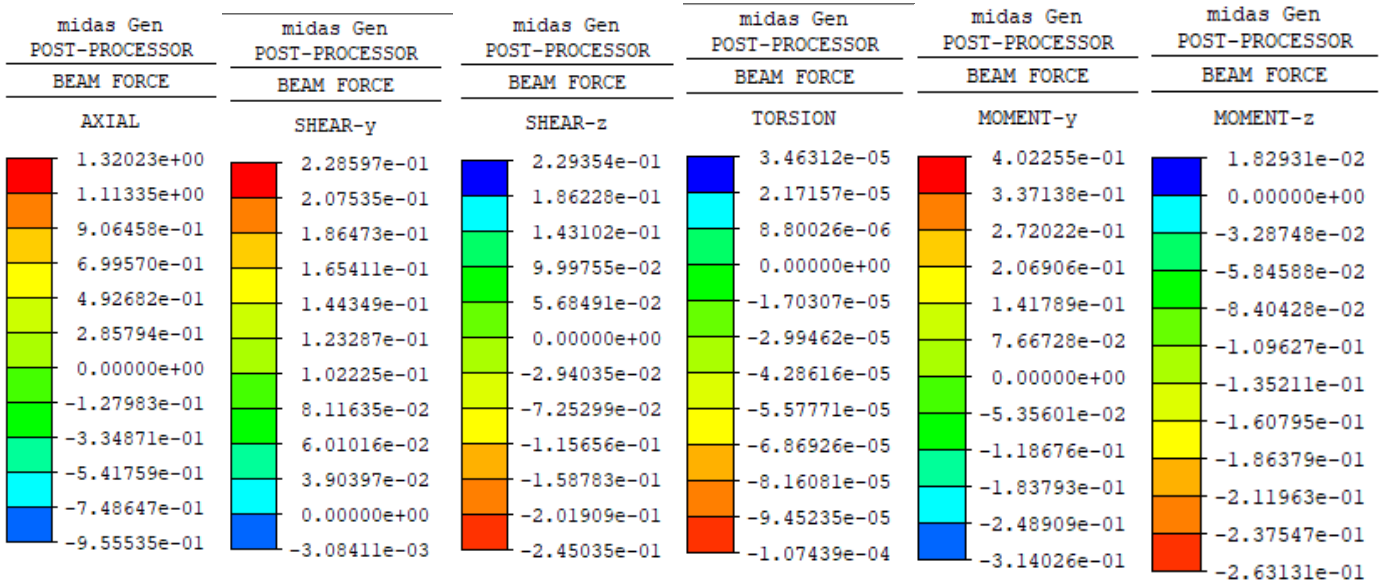


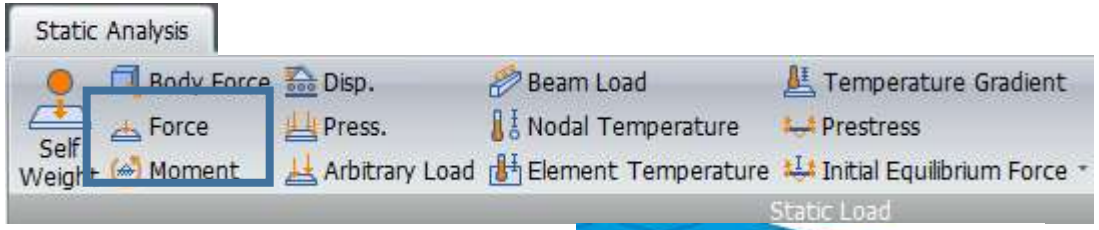
LOADS and CONSTRAINTS

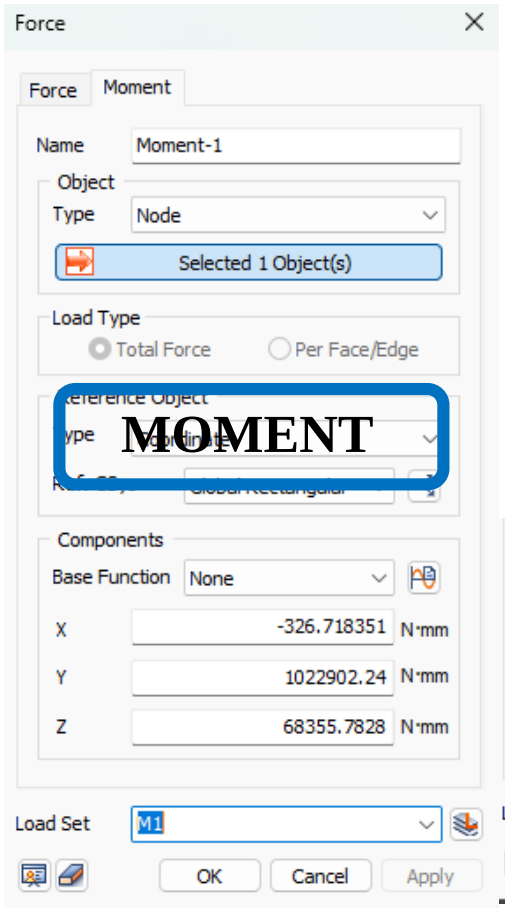
LOAD: FORCE and MOMENT

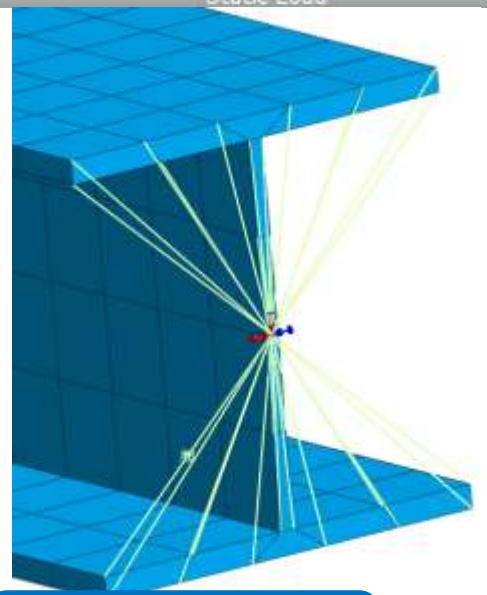
CONSTRAINT: Pinned connection on the joint



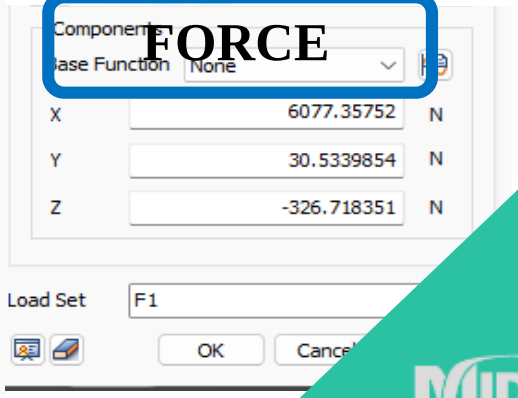








MOMENT



FORCE

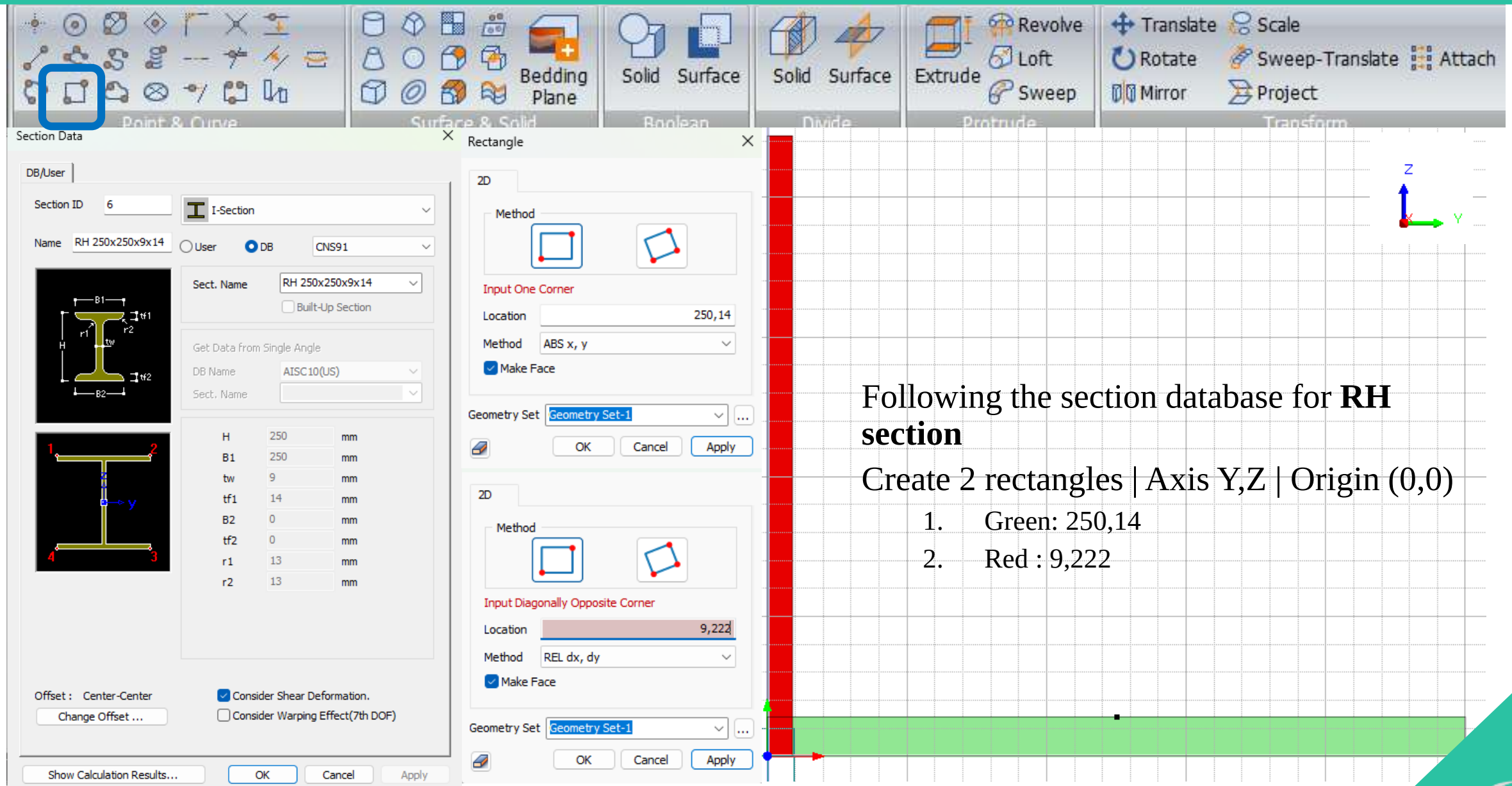
MIDAS

FEA

Part 1

Solid Steel Structure

GEOMETRY

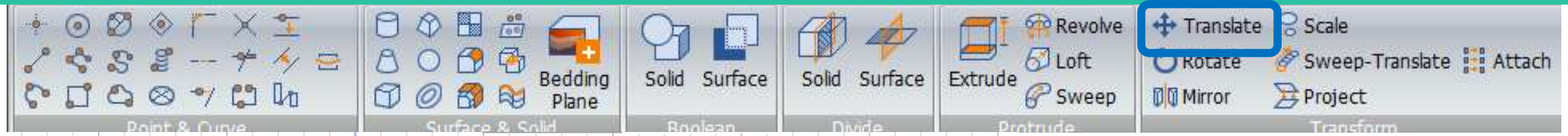


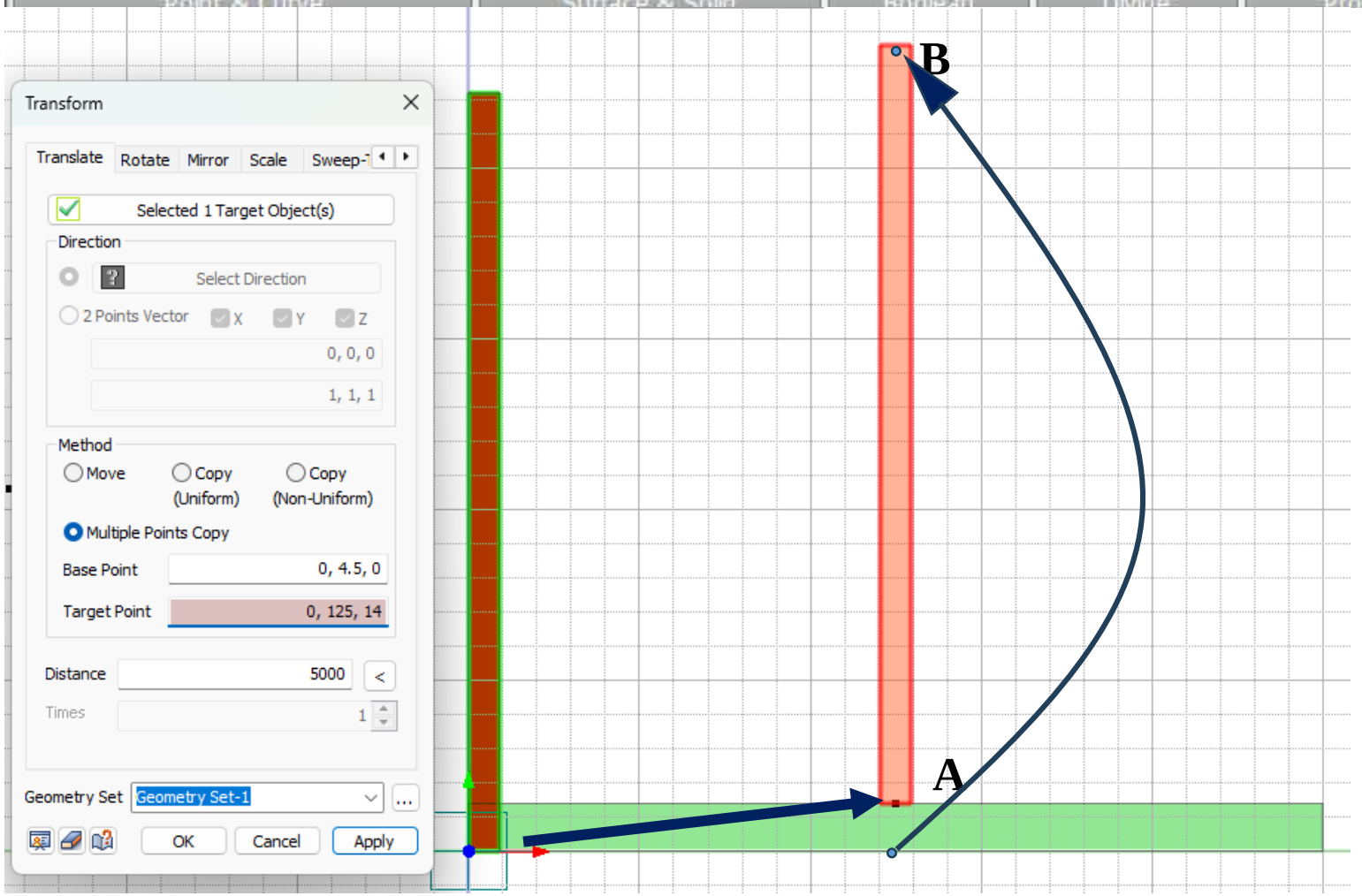
Following the section database for **RH section**

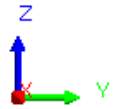
Create 2 rectangles | Axis Y,Z | Origin (0,0)

1. Green: 250,14
2. Red : 9,222

GEOMETRY



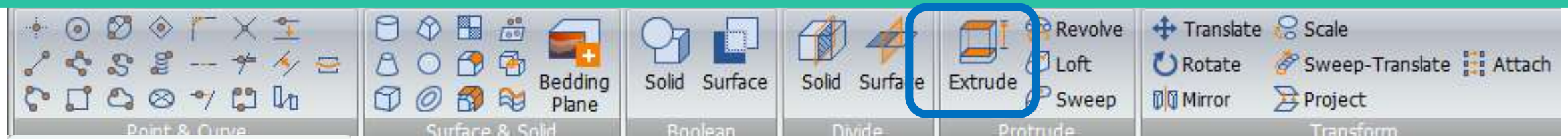


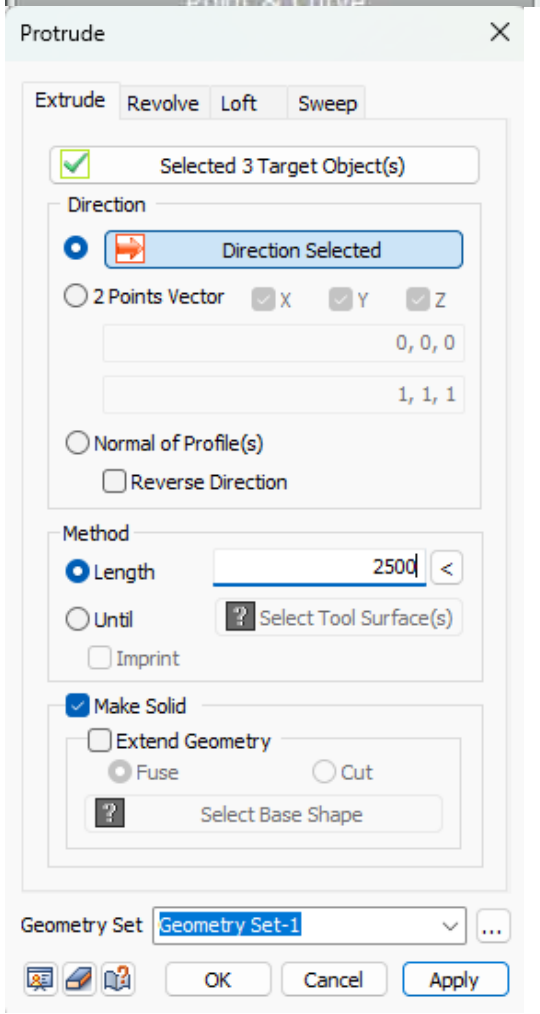


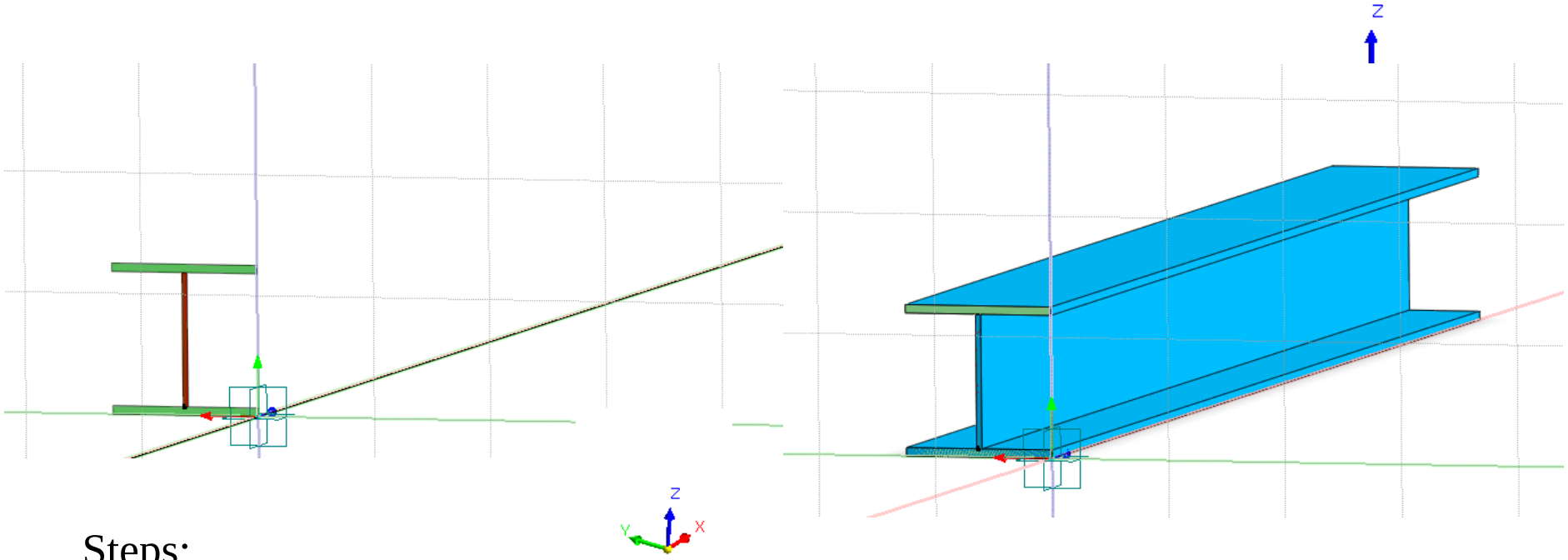
Steps

1. Using Translate function, move the red rectangle to point A.
2. Then copy the green rectangle to point B

GEOMETRY





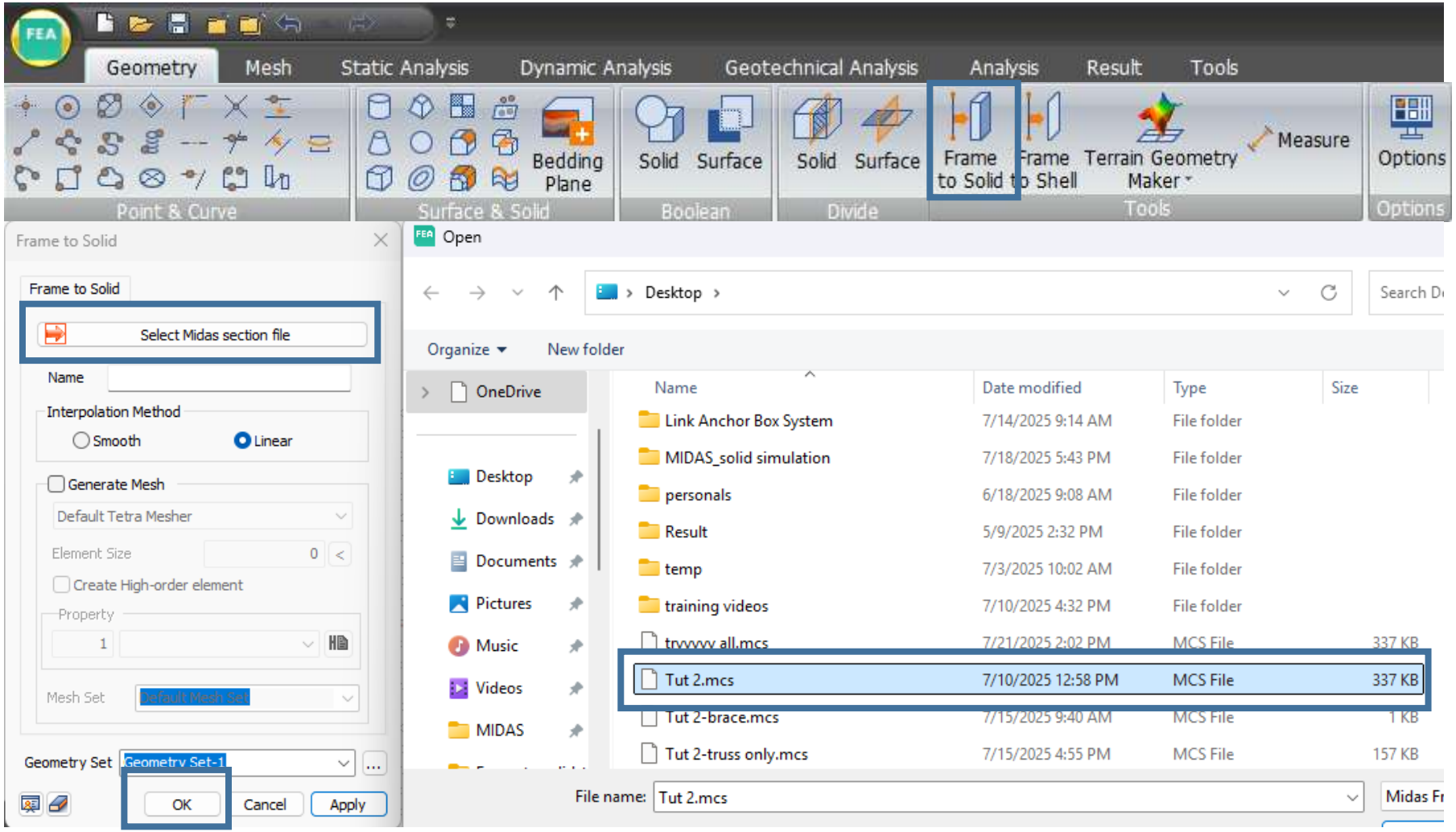


Steps:

Select all rectangles

1. Direction : X-axis
2. Length of beam 2.5m

IMPORTED FILE



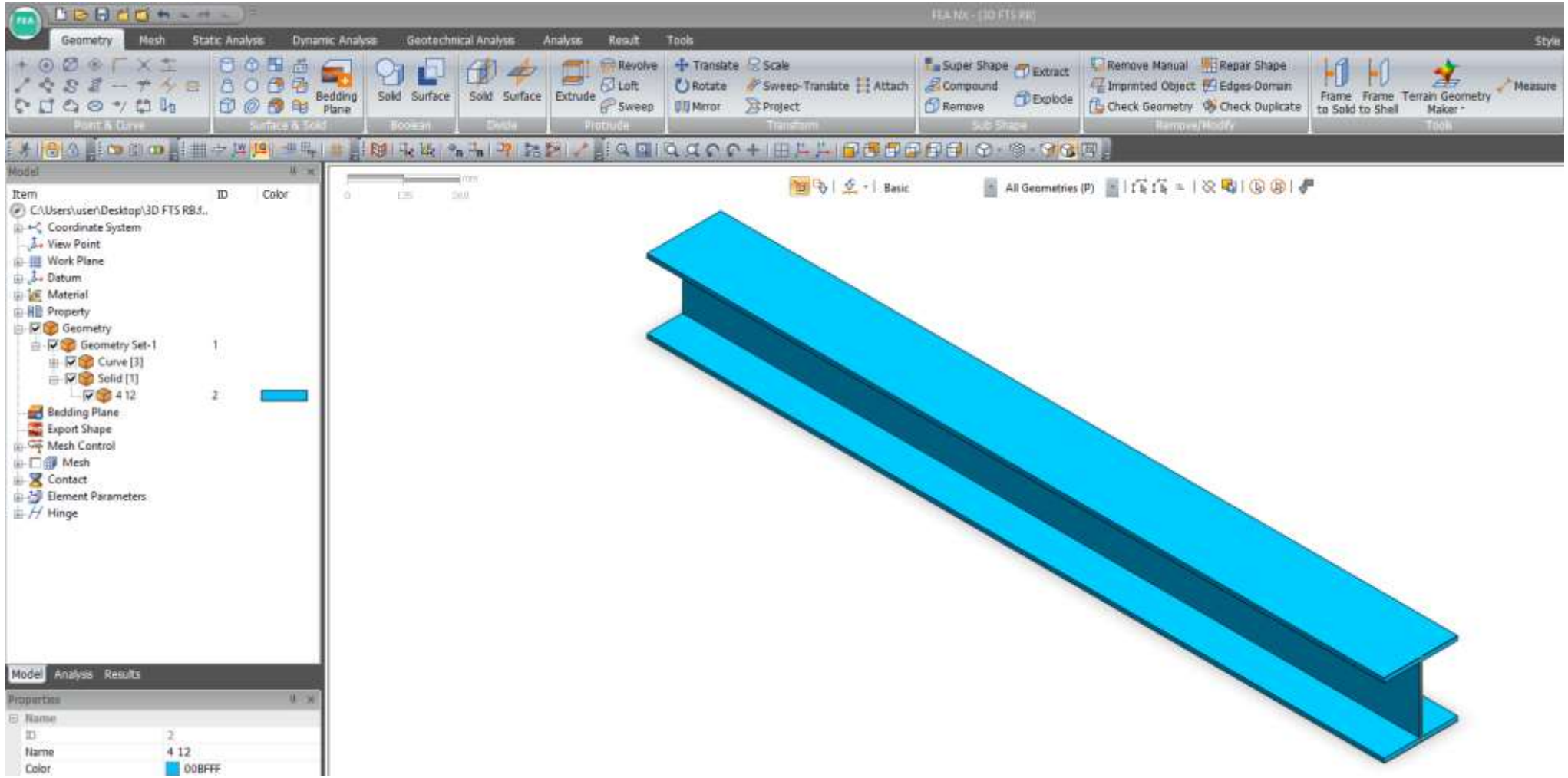
Importing to FEA NX

- Frame to Solid
- Select Midas Section File
- Choose File
- Choose OK

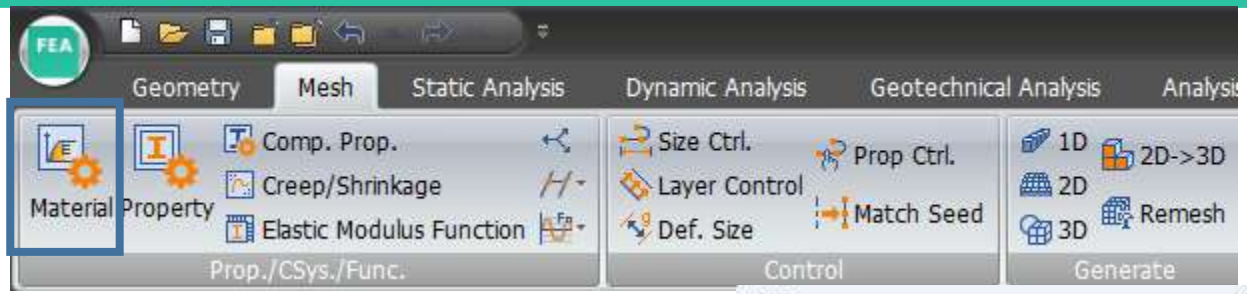
NOTE:

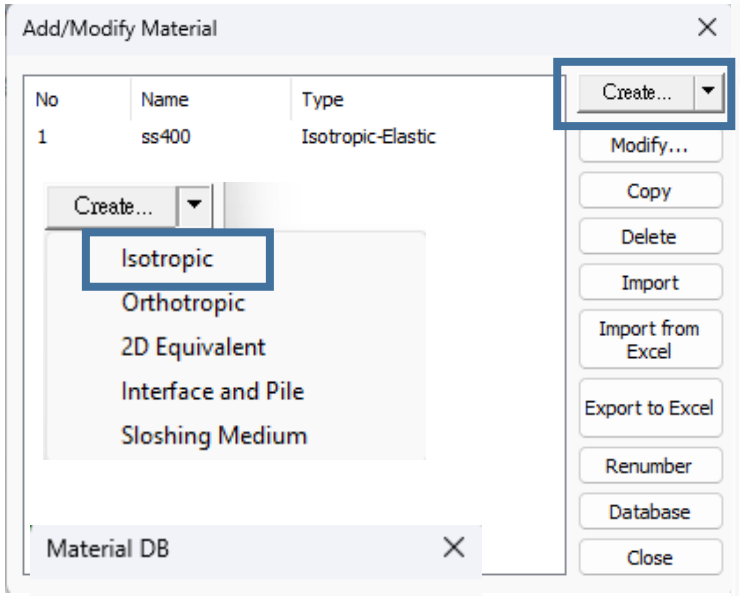
File Type : (.mcs)

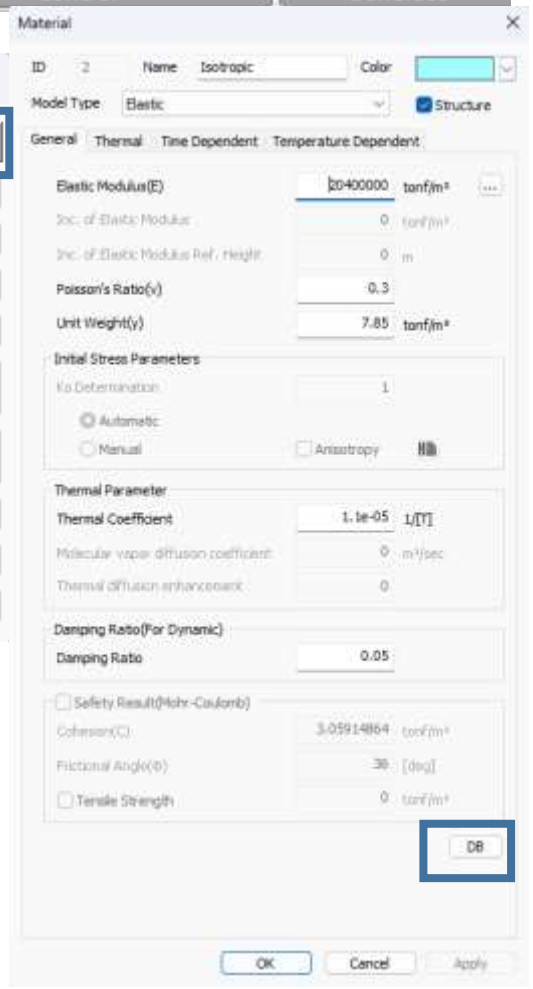
IMPORTED FILE

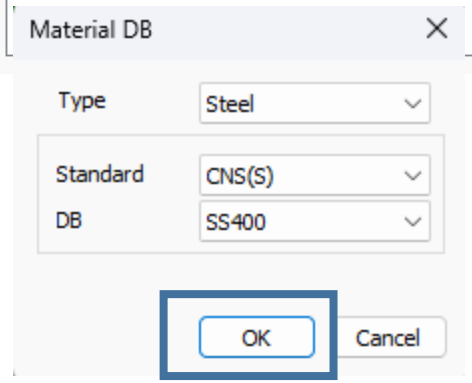


MATERIALS AND PROPERTIES





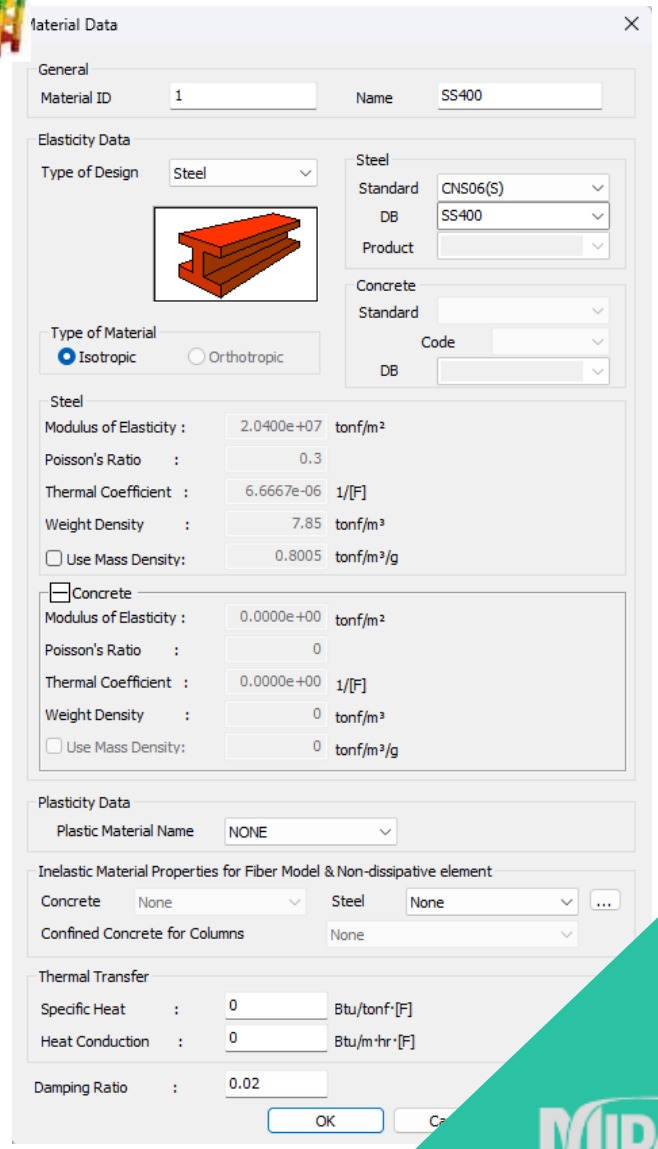




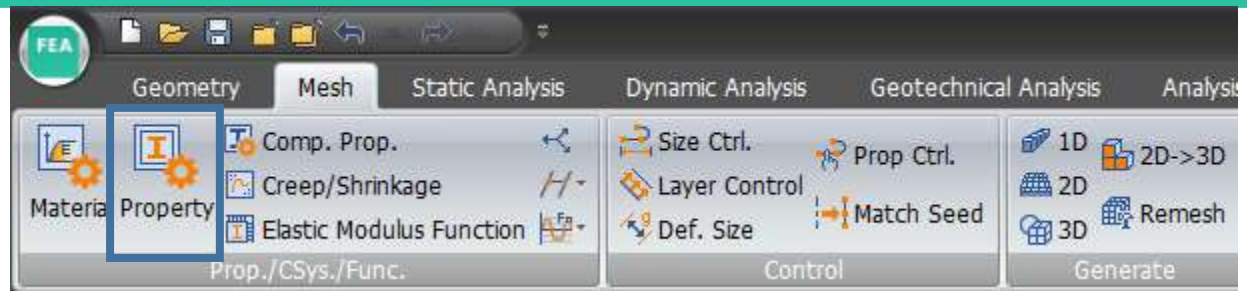
NOTE:

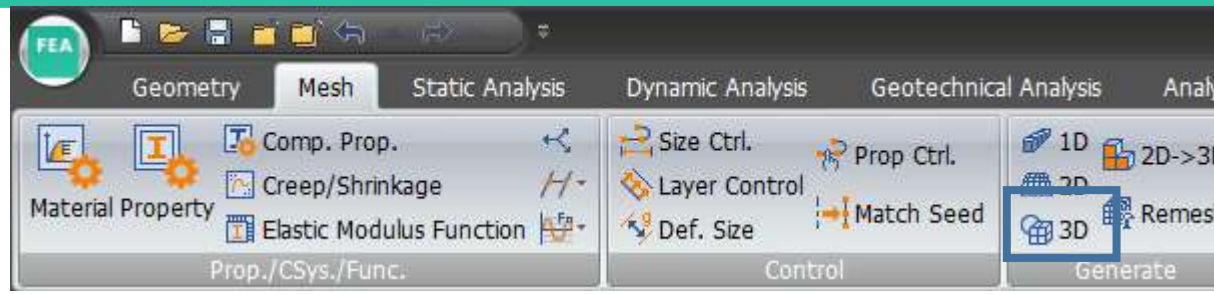
Material Parameters are modifiable.

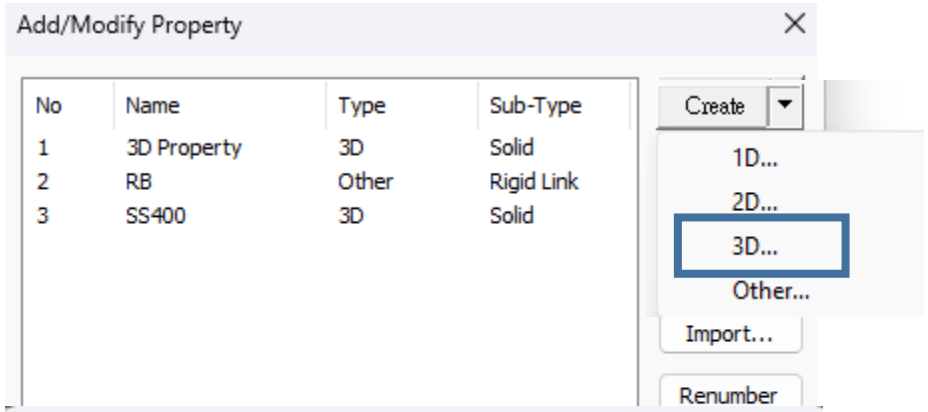
From **MIDAS GEN**

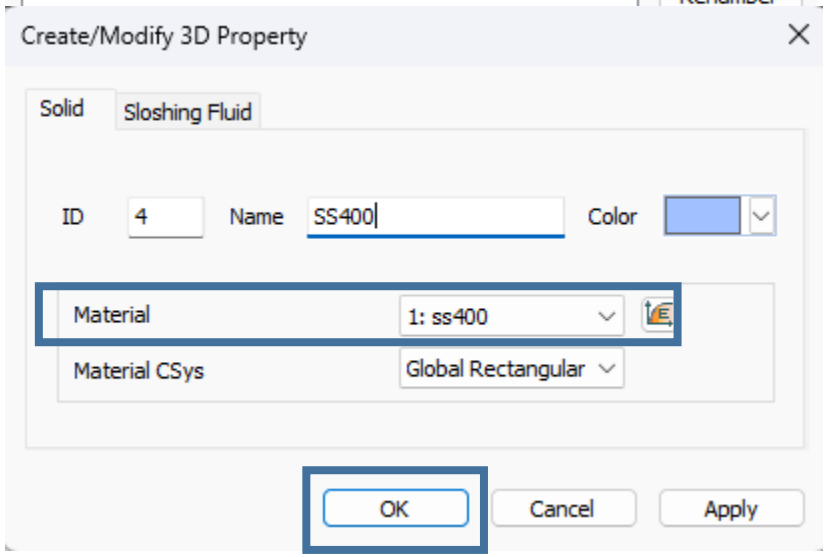


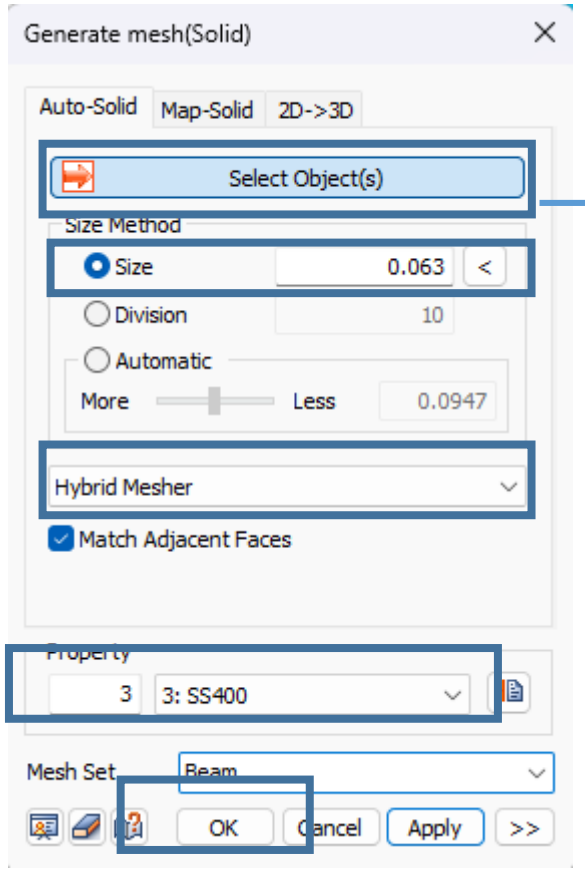
MESHING

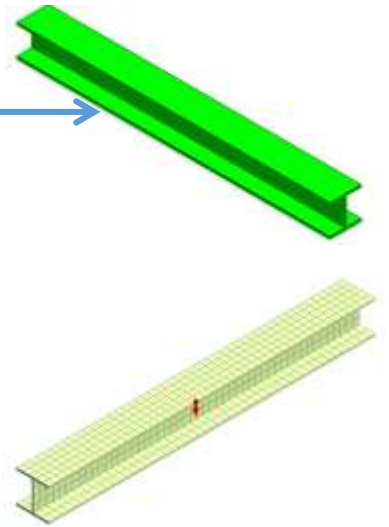

Geometry
Mesh
Static Analysis
Dynamic Analysis
Geotechnical Analysis
Materia
Property
Comp. Prop.
Creep/Shrinkage
Elastic Modulus Function
Prop./CSys./Func.
Size Ctrl.
Layer Control
Def. Size
Prop Ctrl.
Match Seed
Control
1D
2D
3D
2D->3D
Remesh
Generate


Geometry
Mesh
Static Analysis
Dynamic Analysis
Geotechnical Analysis
Material Property
Comp. Prop.
Creep/Shrinkage
Elastic Modulus Function
Prop./CSys./Func.
Size Ctrl.
Layer Control
Def. Size
Prop Ctrl.
Match Seed
Control
1D
3D
Generate

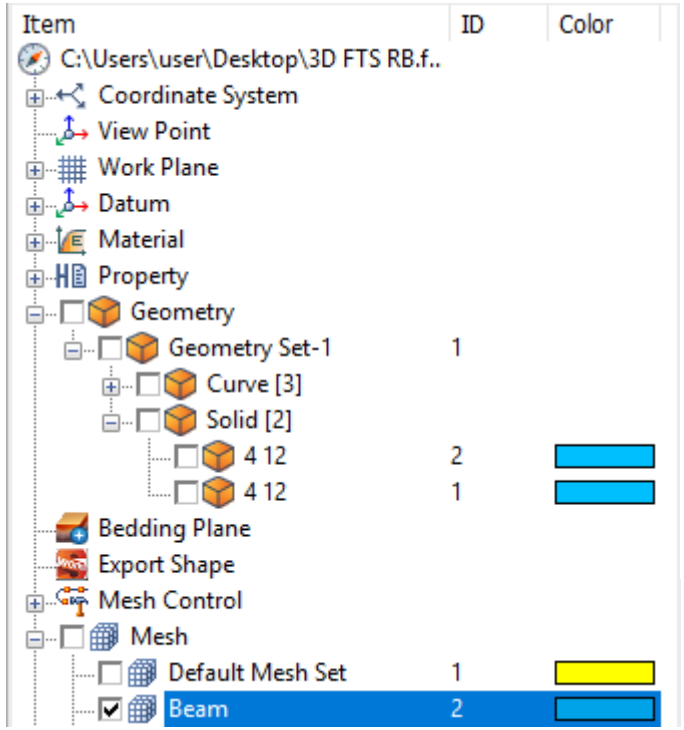

Add/Modify Property
No
Name
Type
Sub-Type
1
3D Property
3D
Solid
2
RB
Other
Rigid Link
3
SS400
3D
Solid
Create
1D...
2D...
3D...
Other...
Import...
Renumber

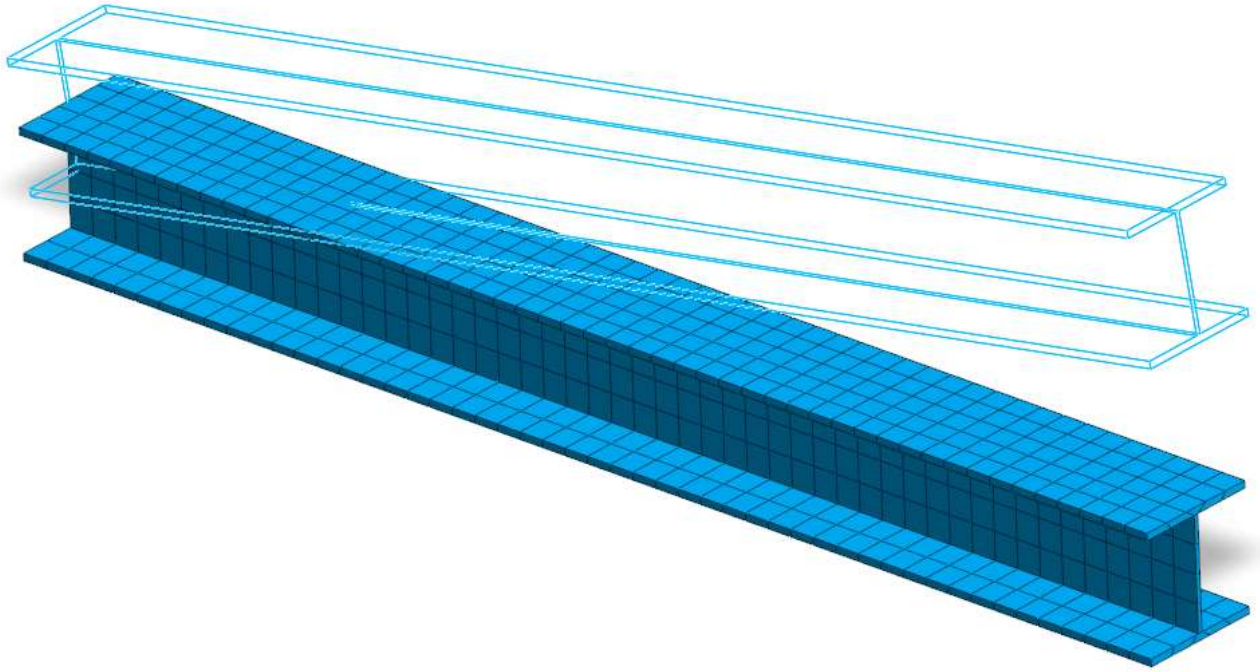

Create/Modify 3D Property
Solid
Sloshing Fluid
ID
4
Name
SS400
Color
Material
1: ss400
Material CSys
Global Rectangular
OK
Cancel
Apply

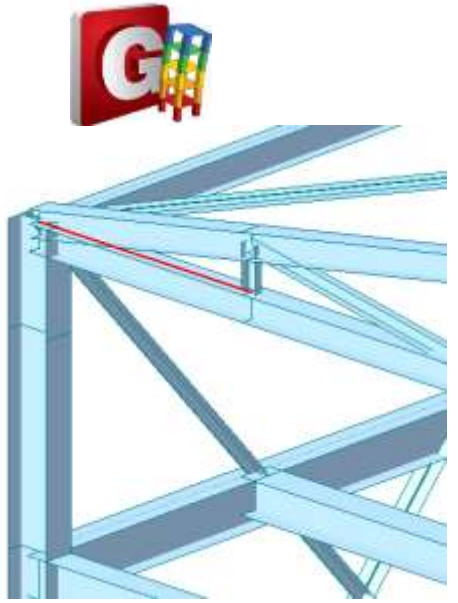

Generate mesh(Solid)
Auto-Solid
Map-Solid
2D->3D
Select Object(s)
Size Method
Size
0.063
Division
10
Automatic
More
Less
0.0947
Hybrid Mesher
Match Adjacent Faces
Property
3
3: SS400
Mesh Set
Beam
OK
Cancel
Apply
>>



MESHING



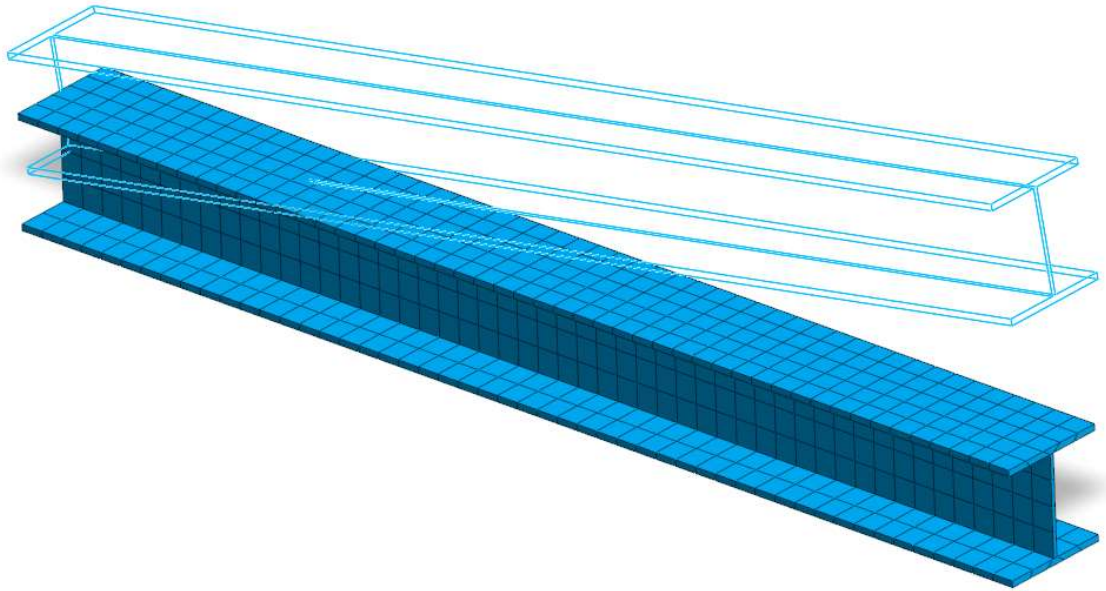


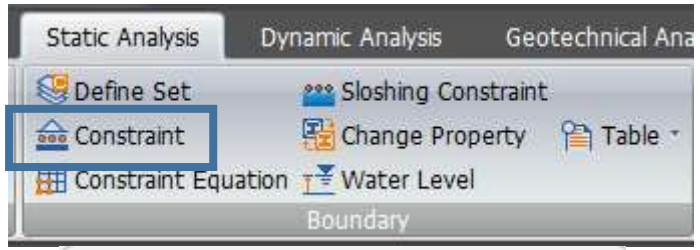


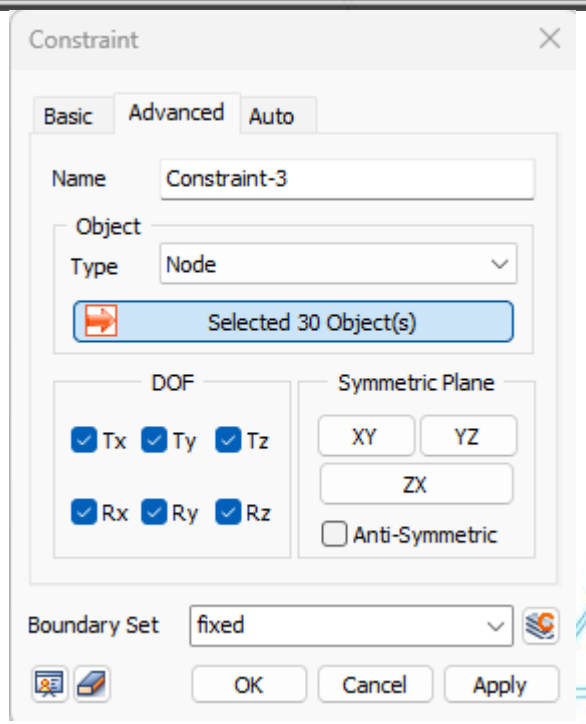
NOTE:

The smaller the mesh, the more accurate the results are.

LOADS and CONSTRAINTS

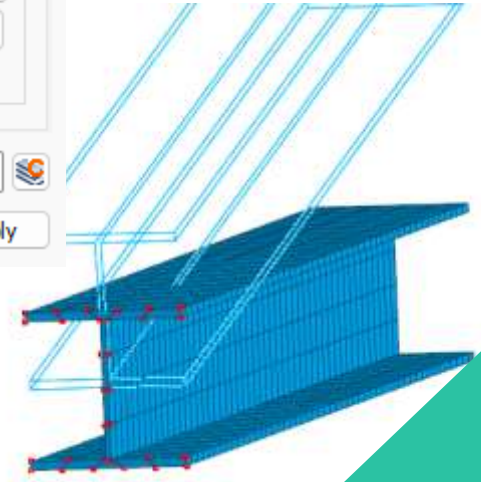






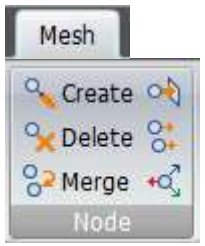
CONSTRAINT:

Pinned support on the joint

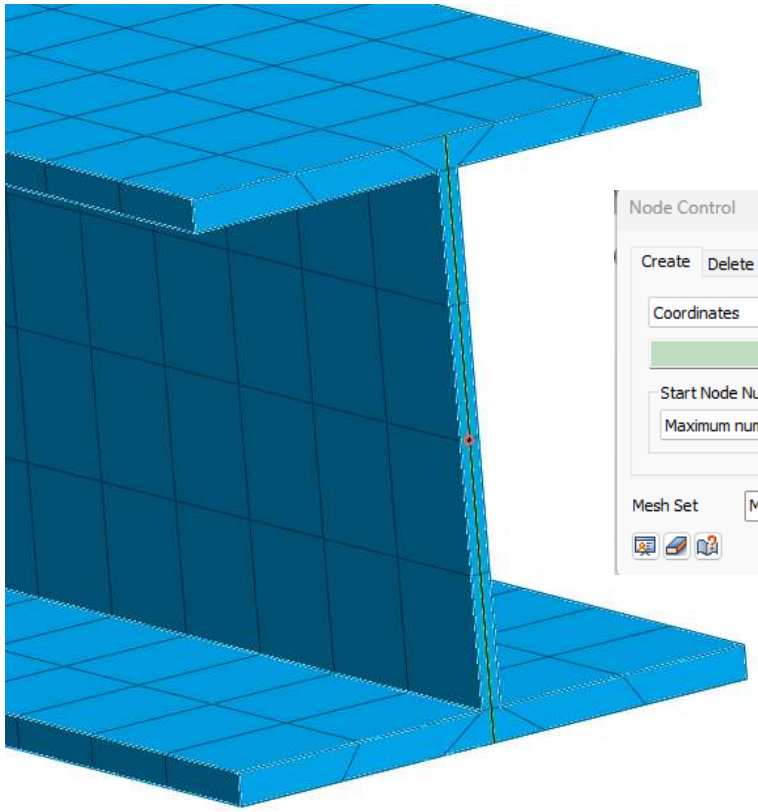


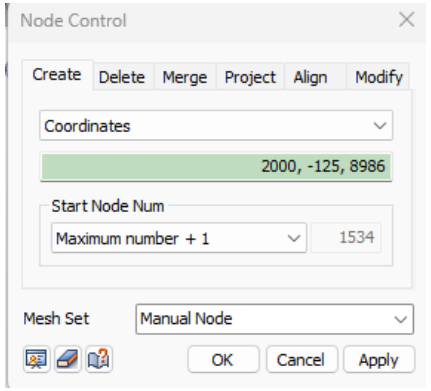
LOADS and CONSTRAINTS

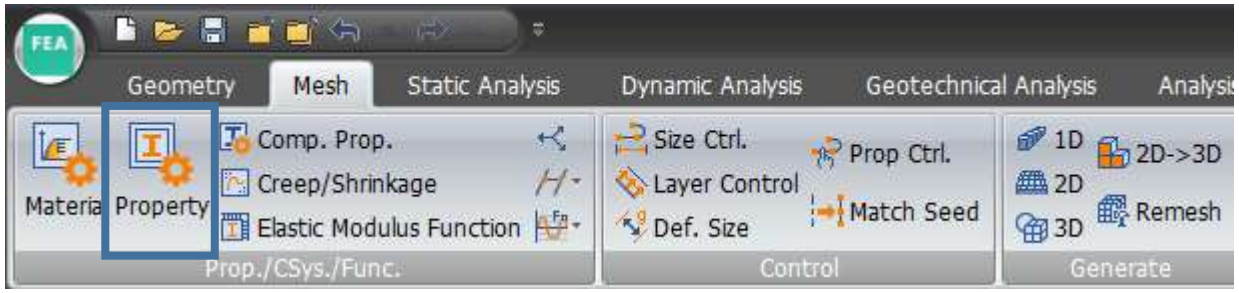
RIGID LINK

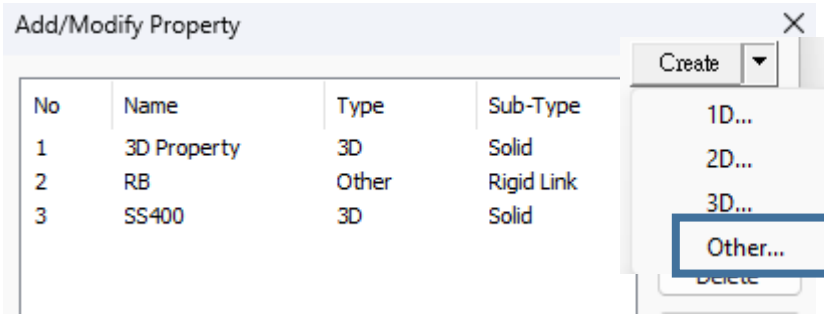


STEP 1: CREATE A MASTER NODE

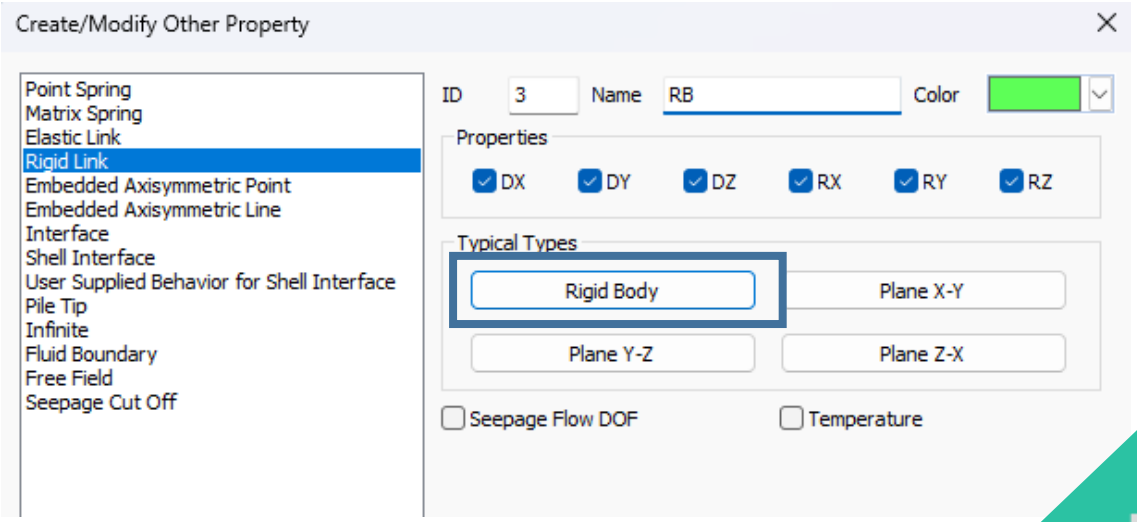






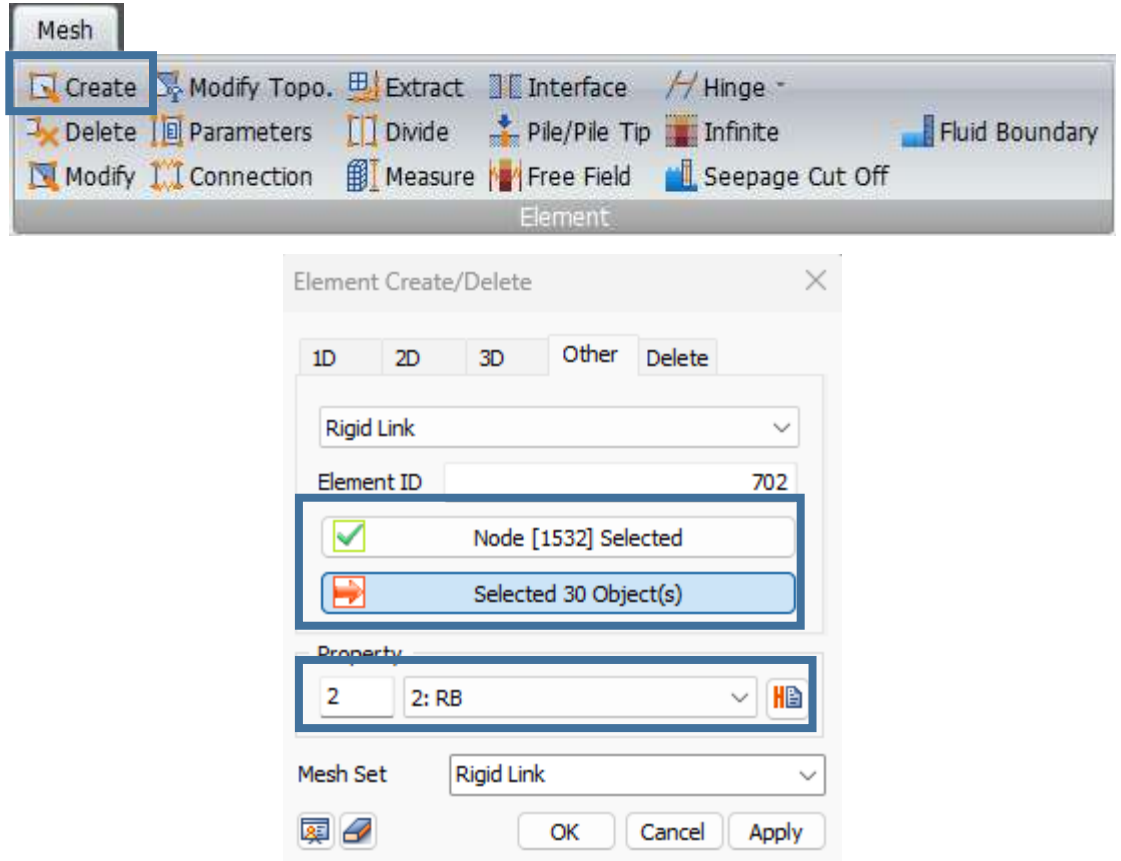


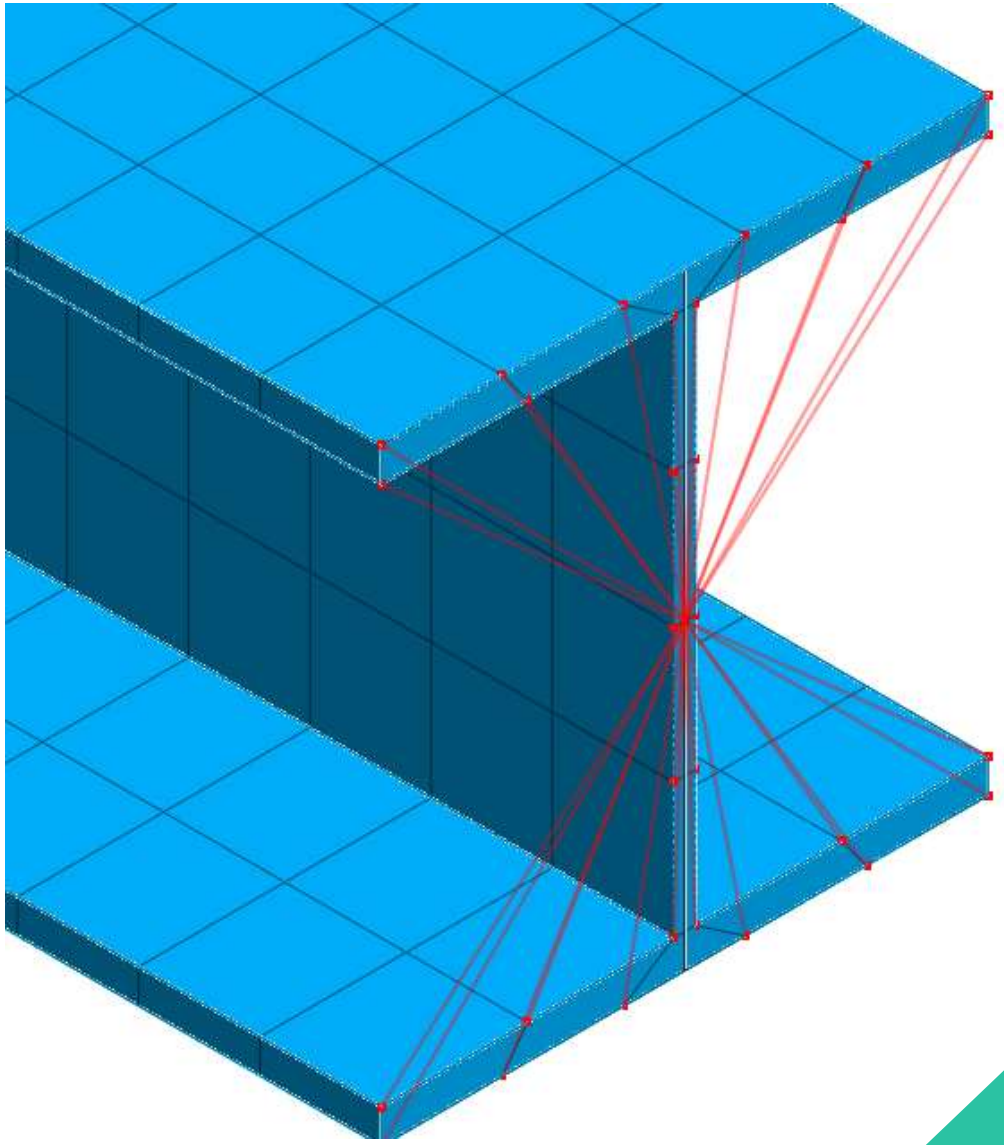
STEP 2: CREATE PROPERTY - RIGID LINK



LOADS and CONSTRAINTS

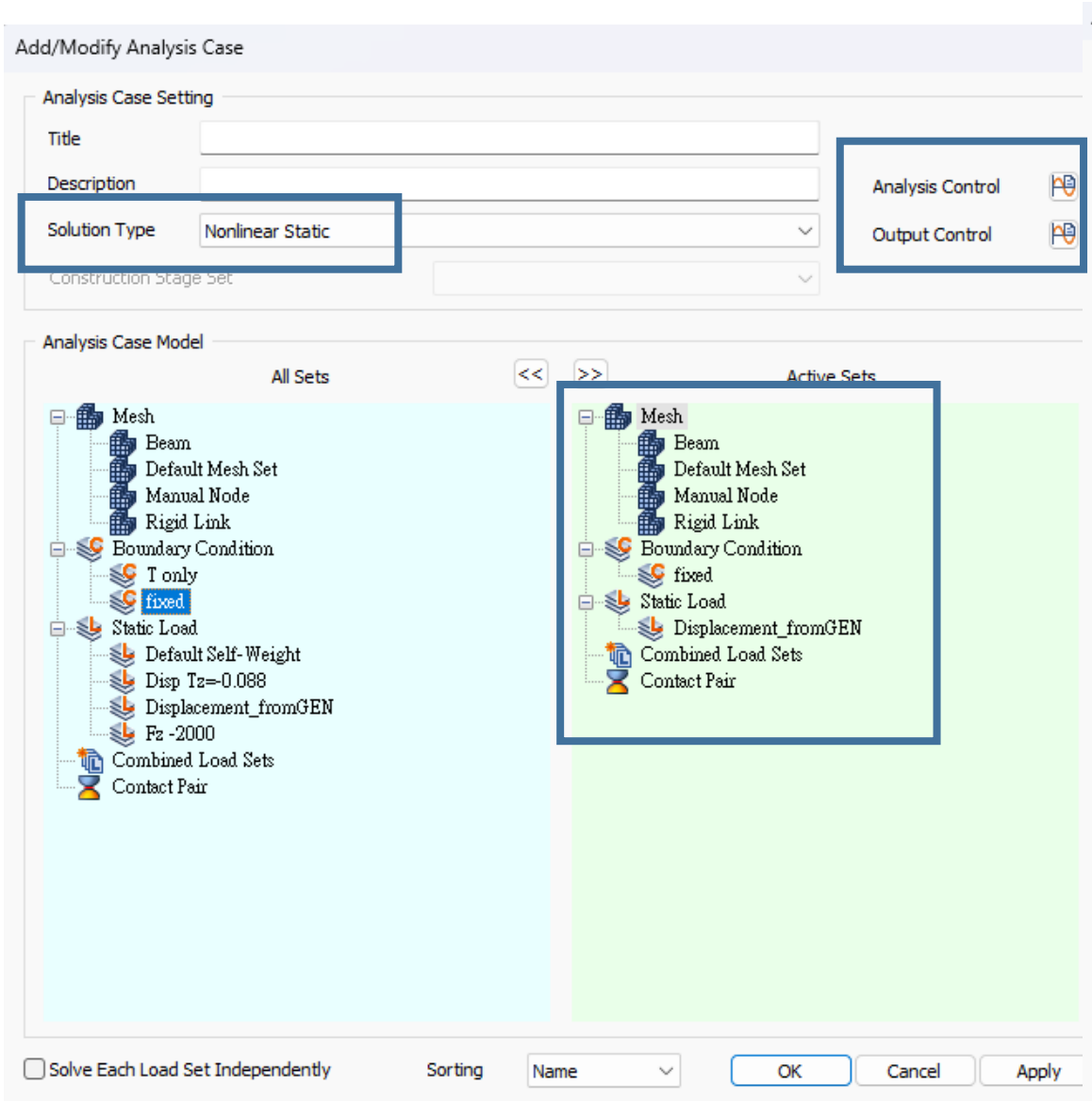
RIGID LINK

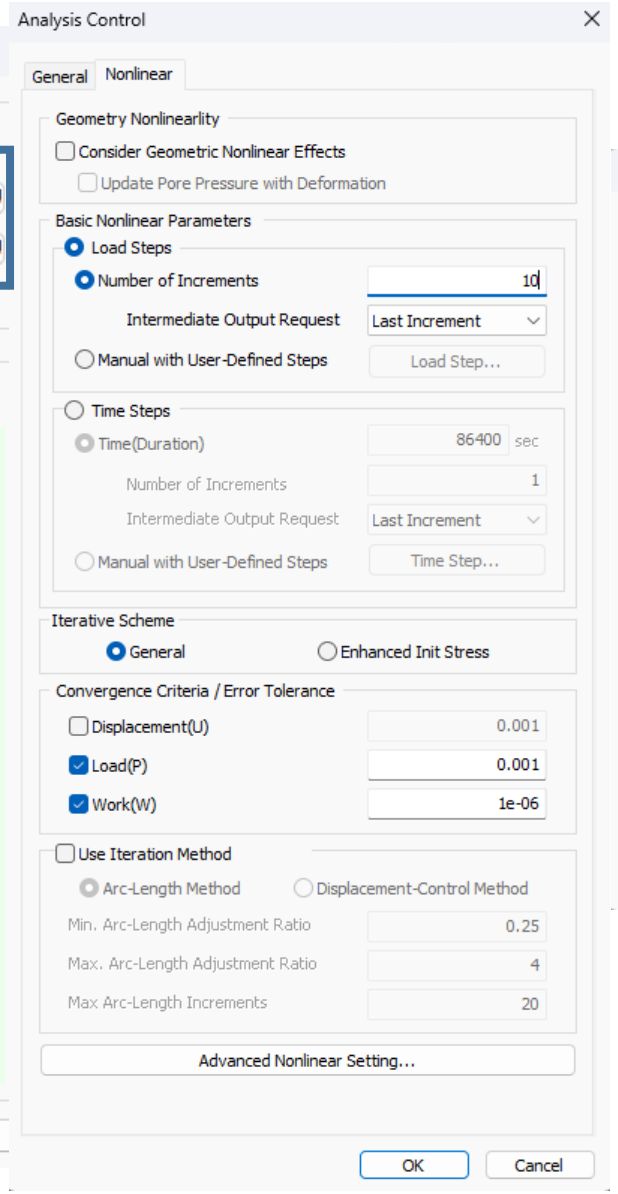


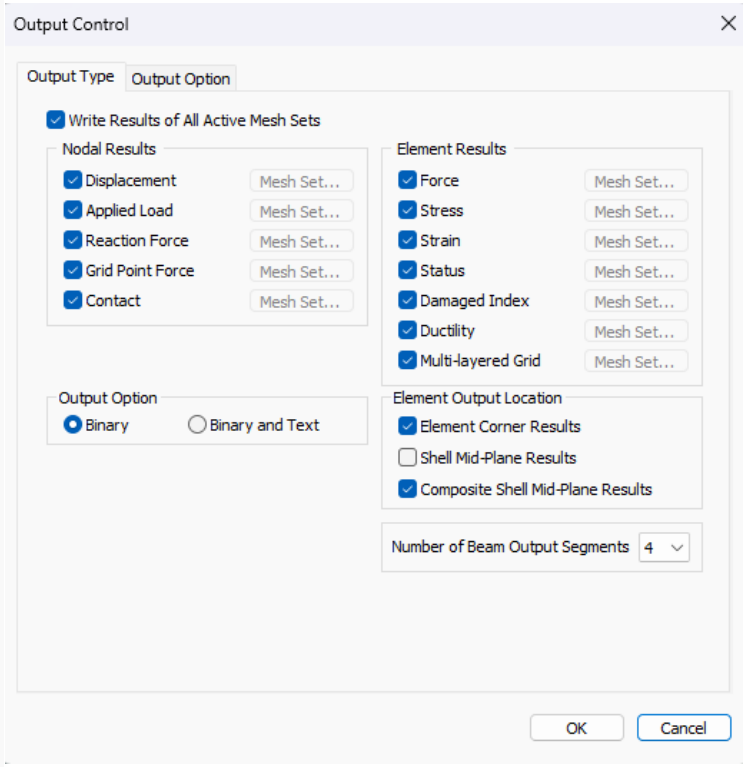


NOTE: After establishing the rigid link, you can apply the internal forces on the master node.

ANALYSIS CASE

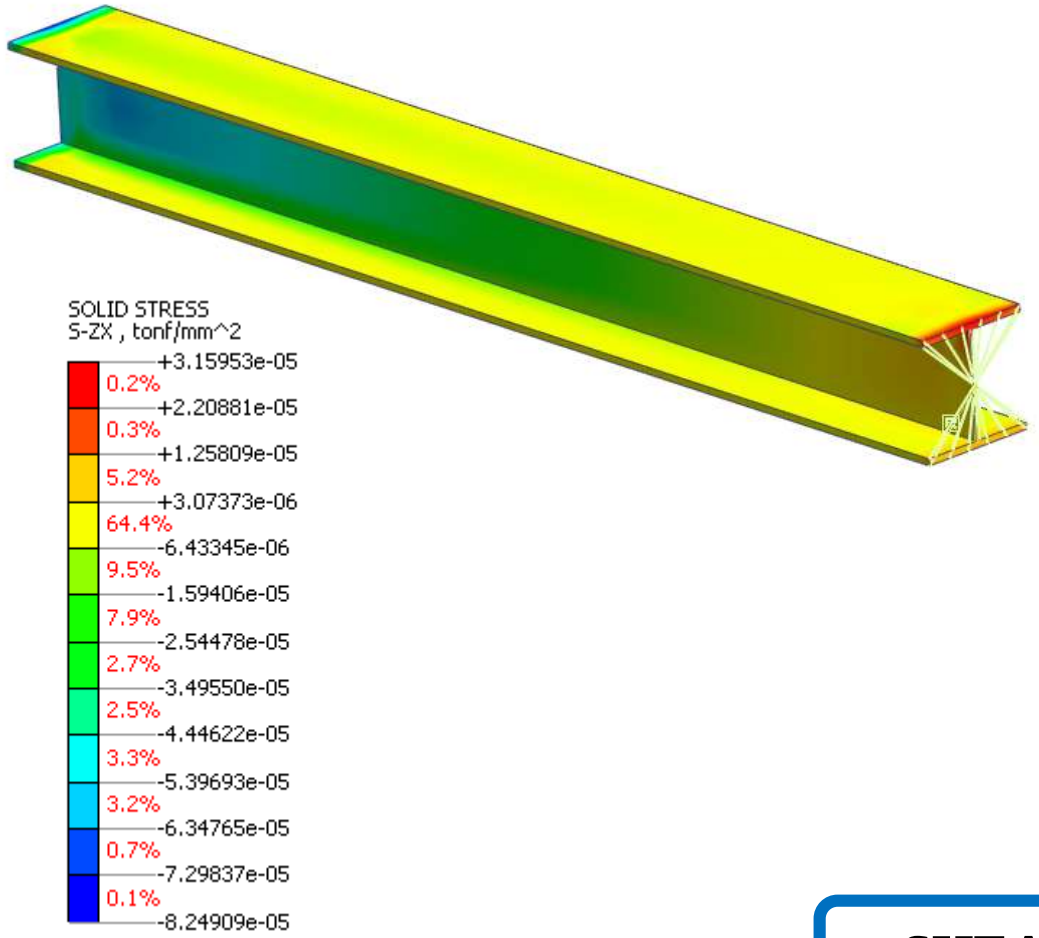




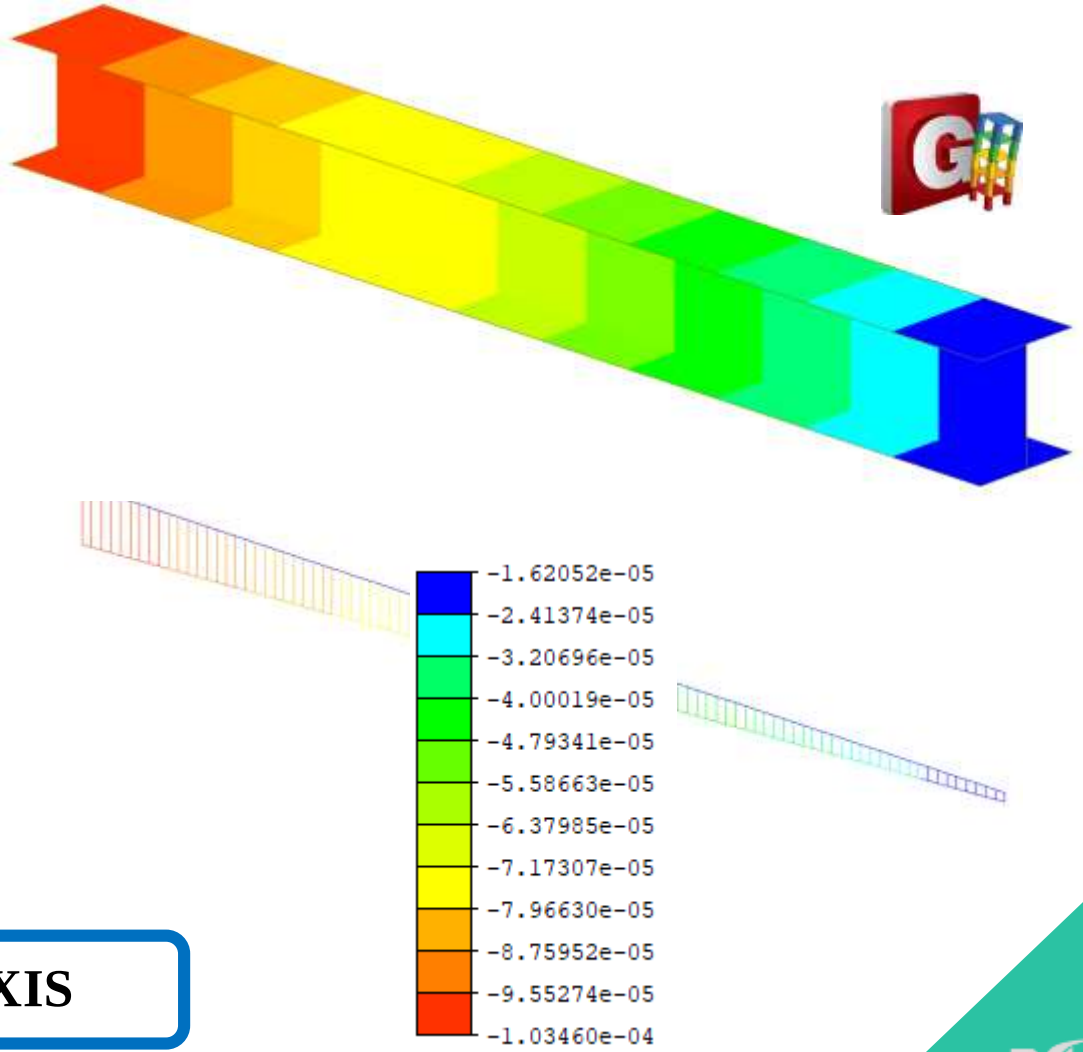


RESULTS - Beam Stress Comparison

MIDAS FEA NX Analysis in between two nodes : Finite element analysis

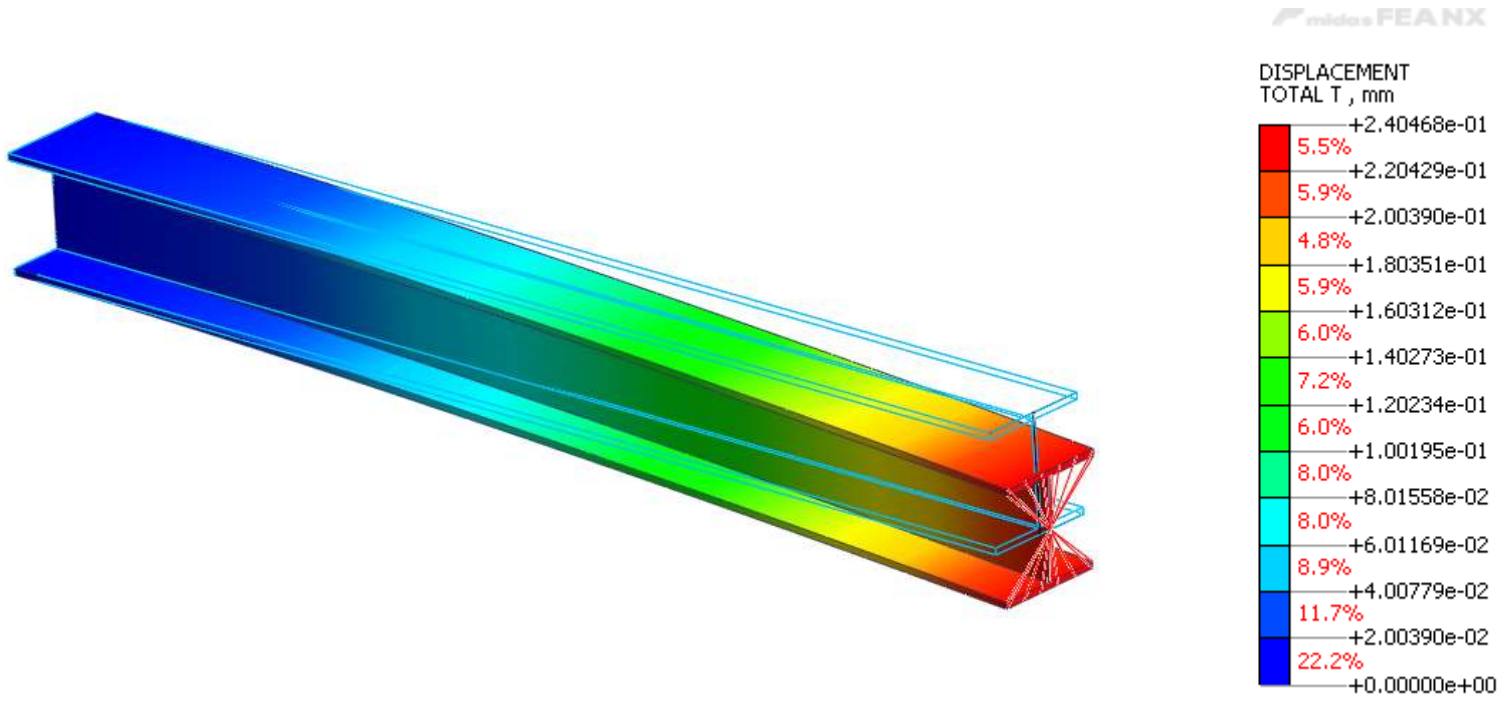


MIDAS GEN Analysis in between two nodes : Linear Interpolation



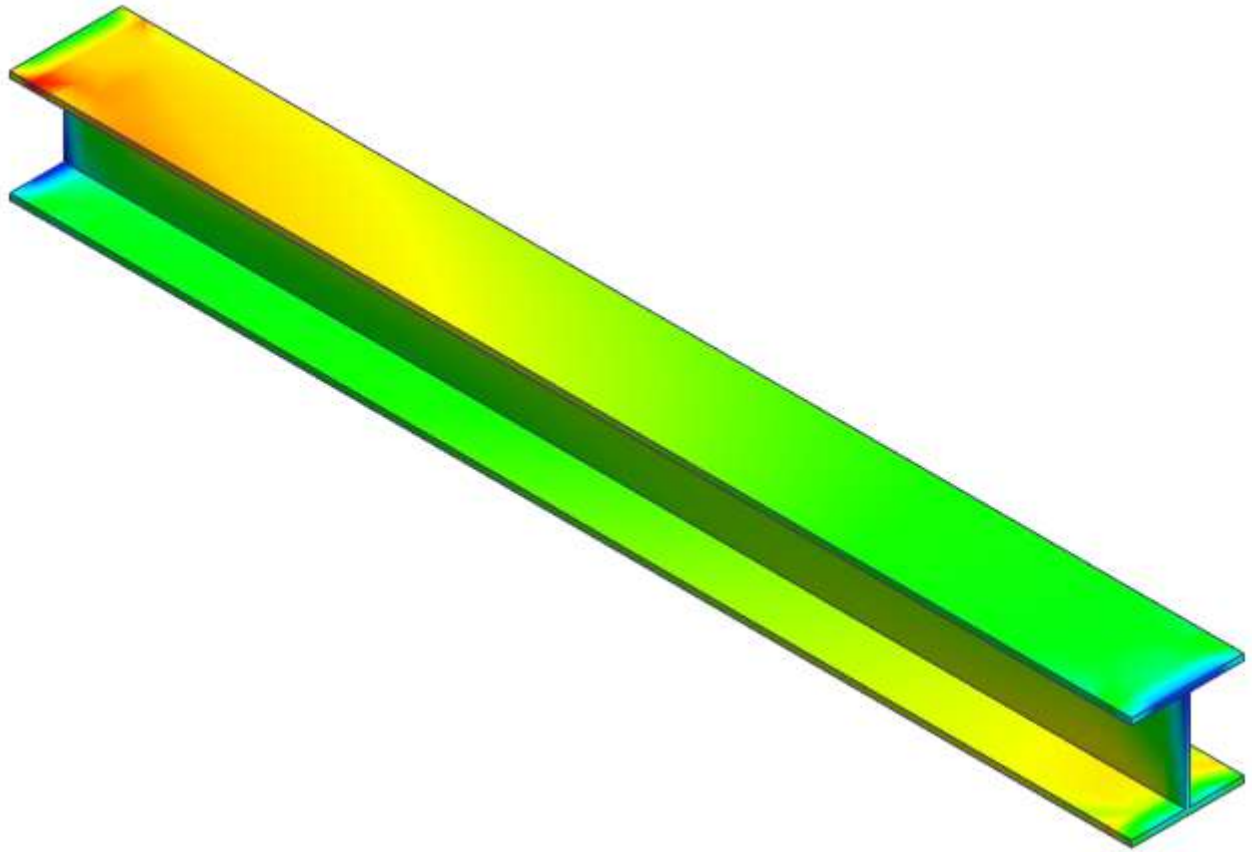
SHEAR Z-AXIS

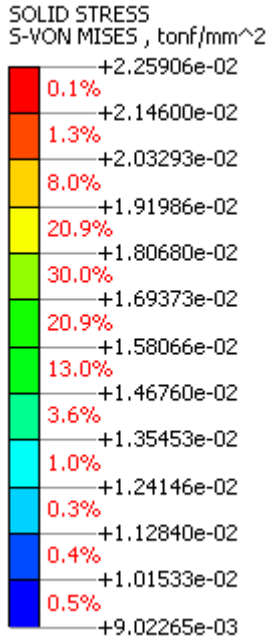
RESULTS - Beam Deformation



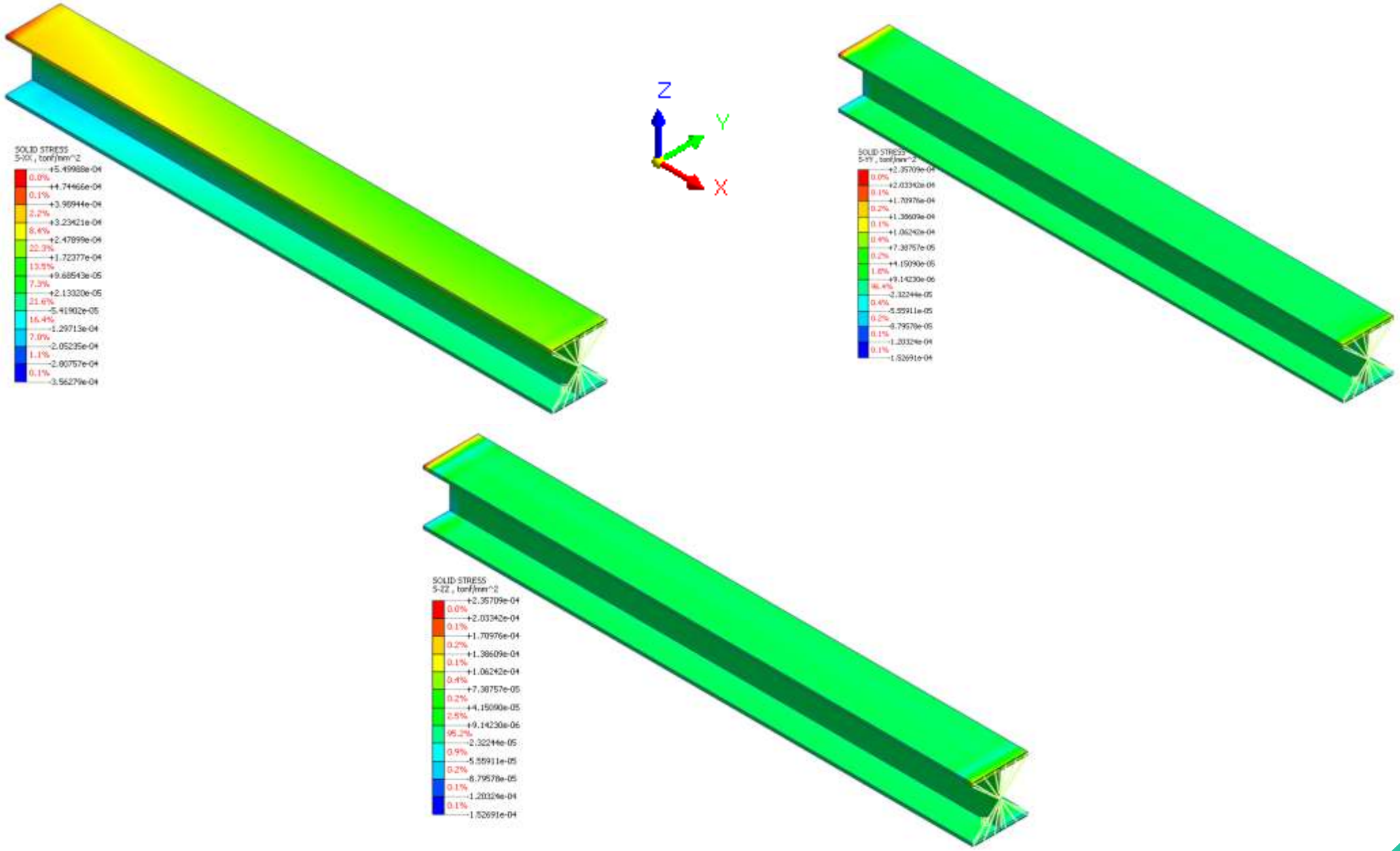
Deformation

RESULTS - STRESS VON MISES

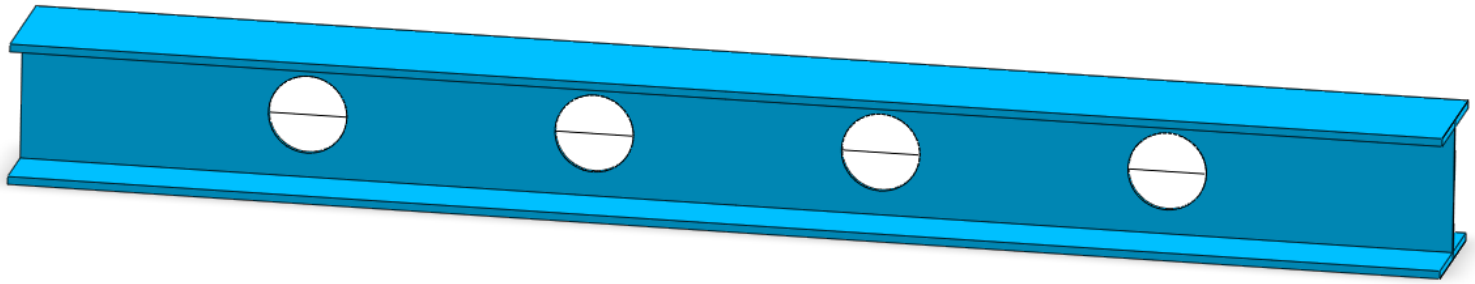




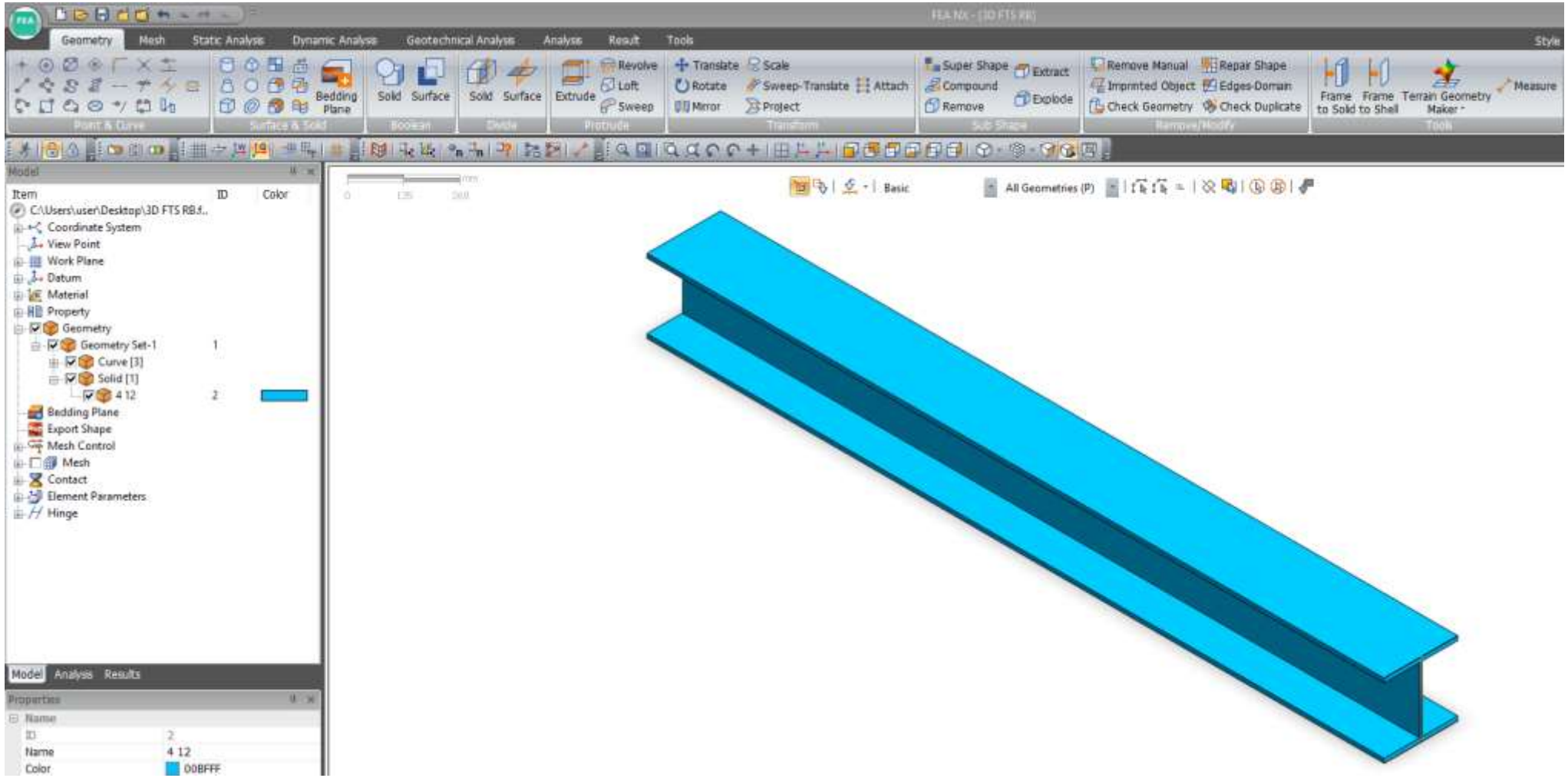
RESULTS - Beam Strains Normal



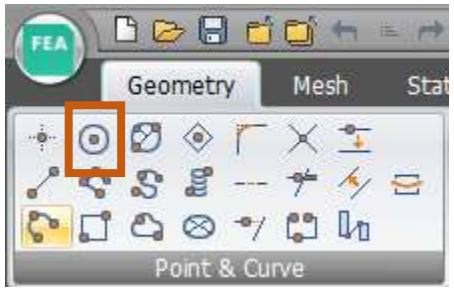
Part 2
Modified Steel Structure

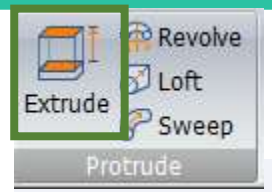


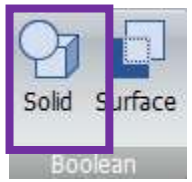
IMPORTED FILE

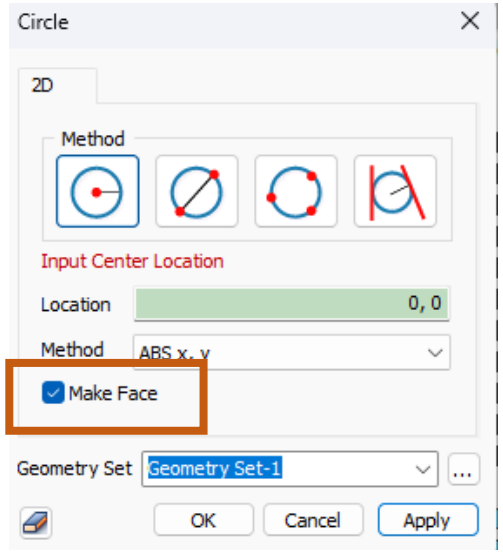


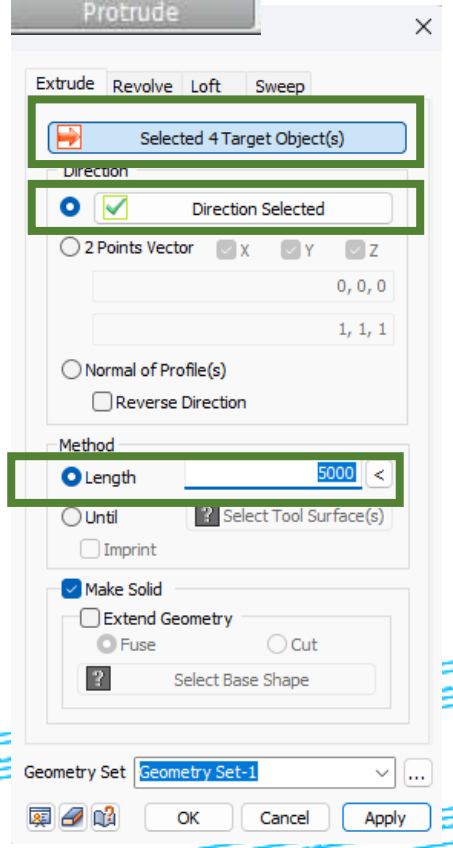
GEOMETRY

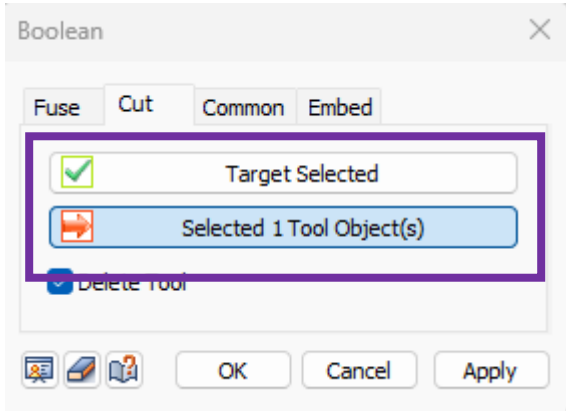


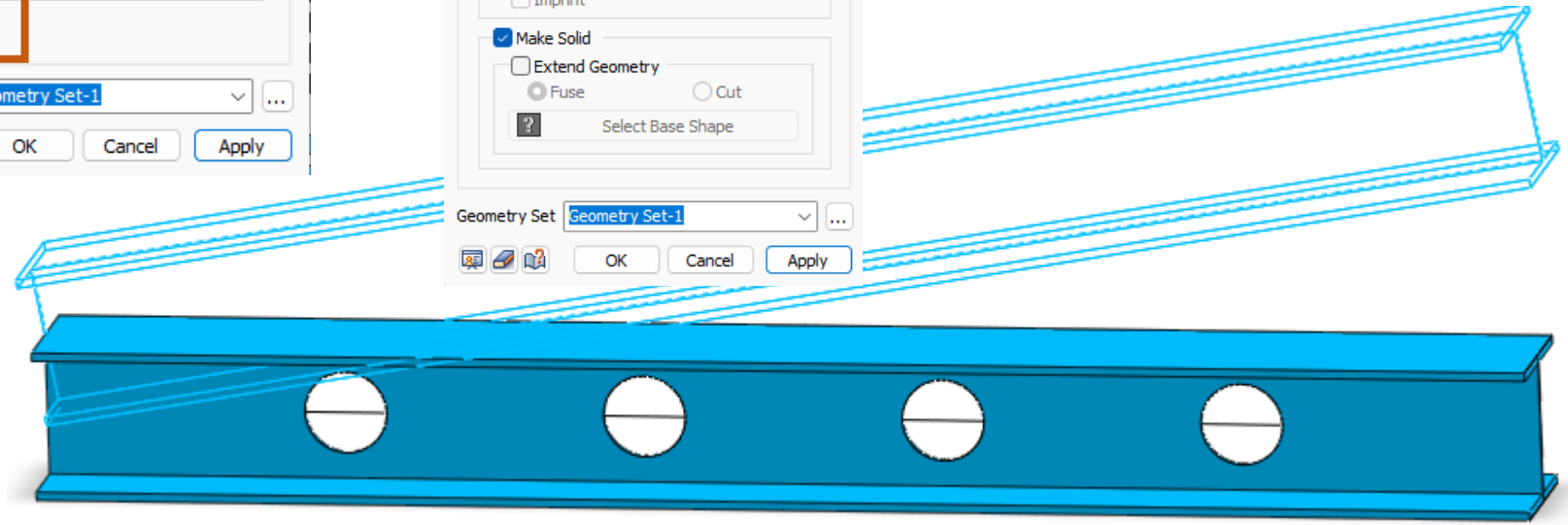










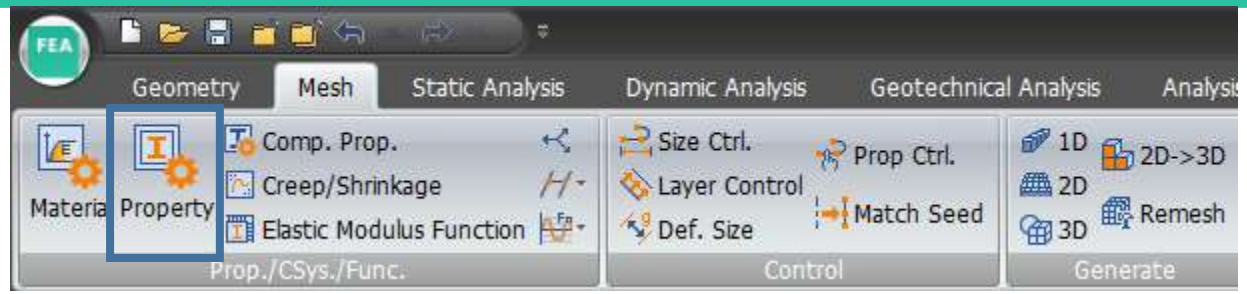


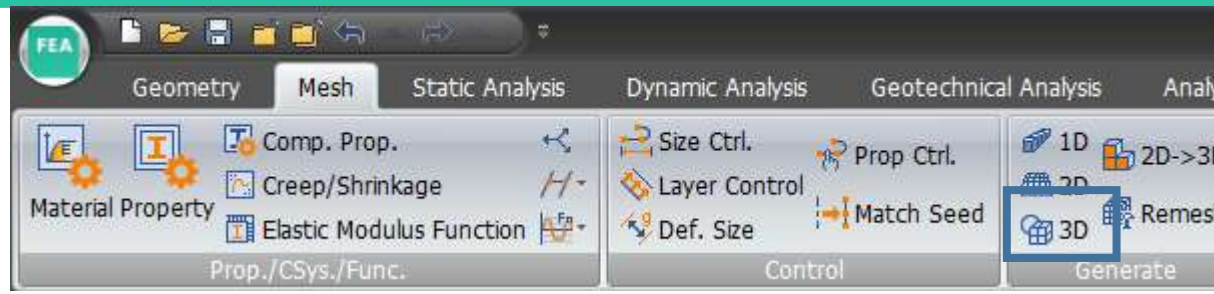
MODIFYING BEAM ELEMENT

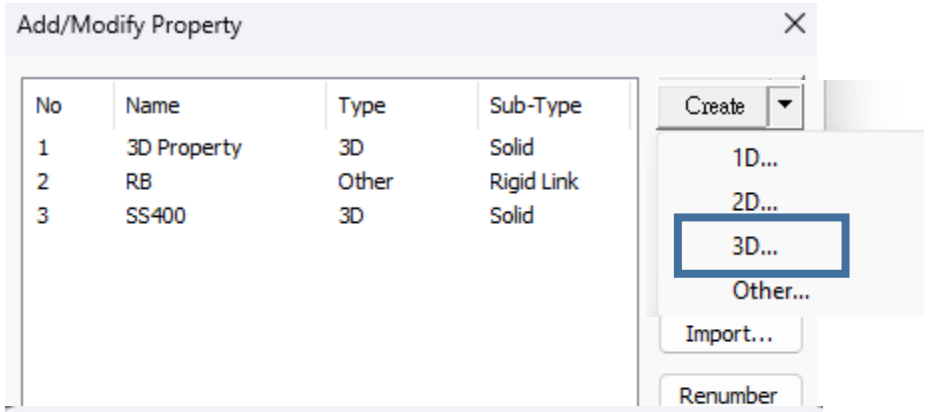
(Sample Case: There are holes in the beam)

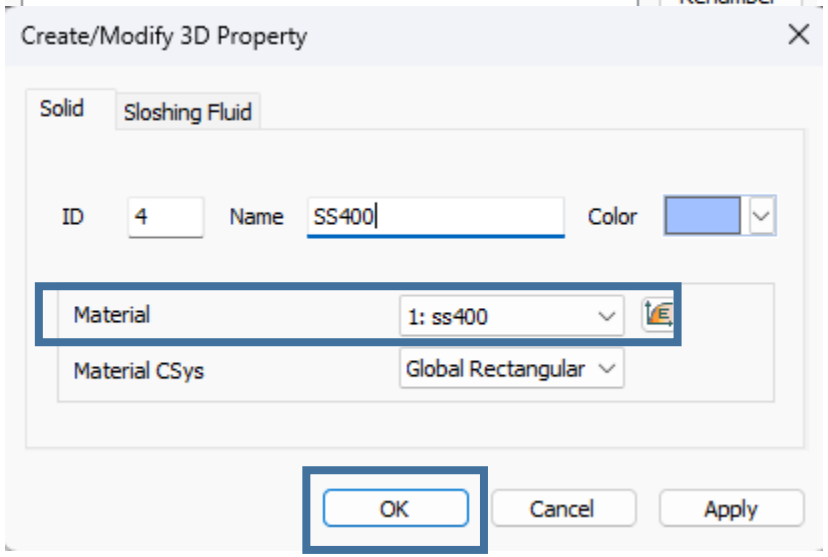
1. Create a circle face
2. Extrude to make a cylinder
3. Cut Solid

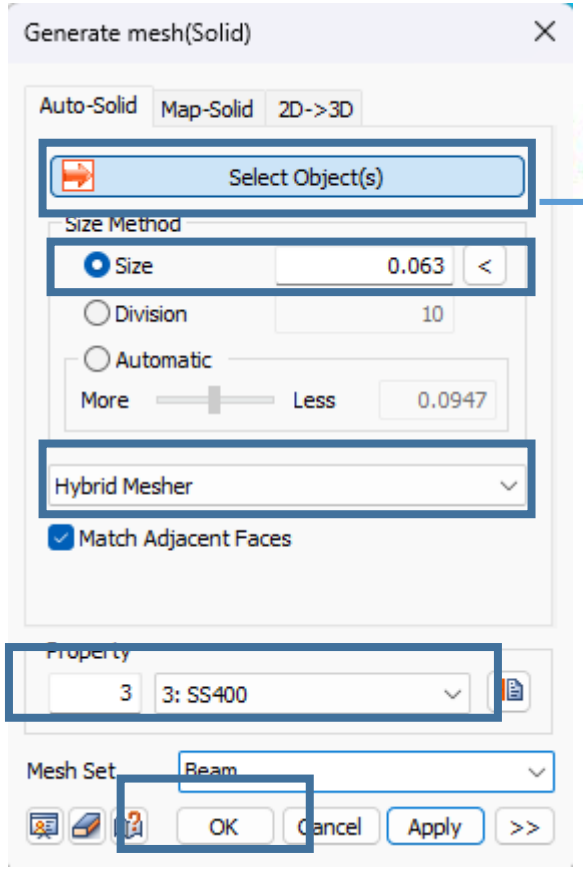
MESHING

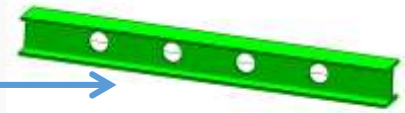

Geometry
Mesh
Static Analysis
Dynamic Analysis
Geotechnical Analysis
Materia
Property
Comp. Prop.
Creep/Shrinkage
Elastic Modulus Function
Prop./CSys./Func.
Size Ctrl.
Layer Control
Def. Size
Prop Ctrl.
Match Seed
Control
1D
2D
3D
2D->3D
Remesh
Generate


Geometry
Mesh
Static Analysis
Dynamic Analysis
Geotechnical Analysis
Material Property
Comp. Prop.
Creep/Shrinkage
Elastic Modulus Function
Prop./CSys./Func.
Size Ctrl.
Layer Control
Def. Size
Prop Ctrl.
Match Seed
Control
1D
3D
Generate

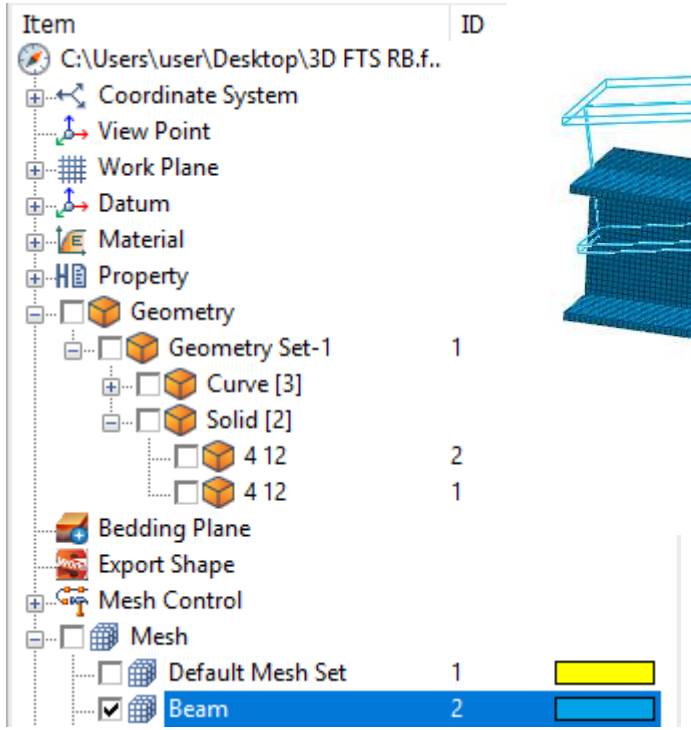

Add/Modify Property
No
Name
Type
Sub-Type
1
3D Property
3D
Solid
2
RB
Other
Rigid Link
3
SS400
3D
Solid
Create
1D...
2D...
3D...
Other...
Import...
Renumber

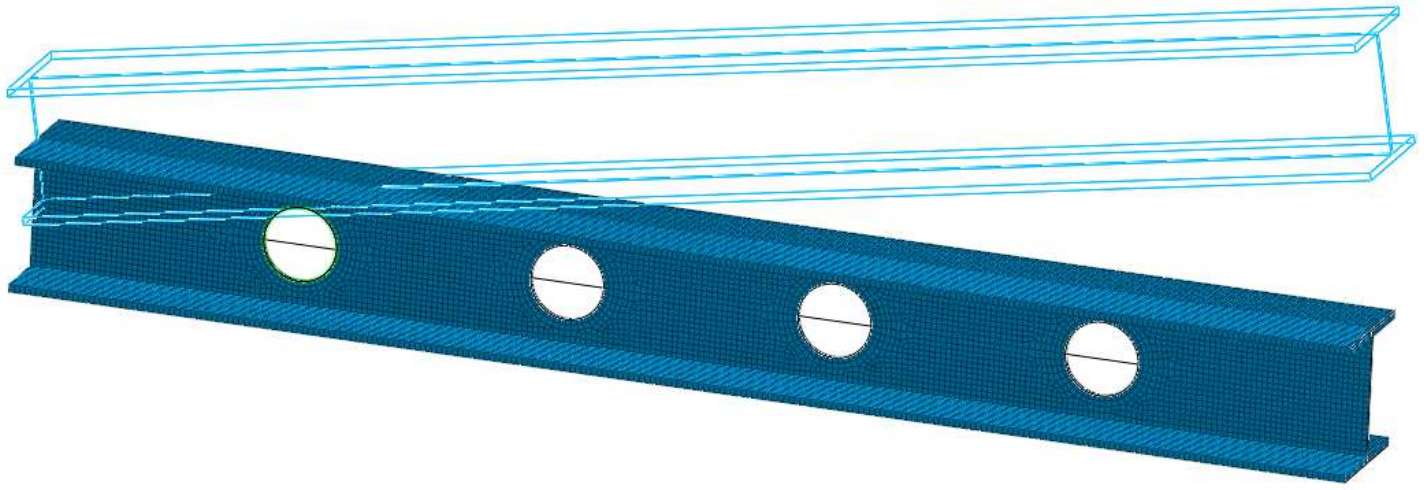

Create/Modify 3D Property
Solid
Sloshing Fluid
ID
4
Name
SS400
Color
Material
1: ss400
Material CSys
Global Rectangular
OK
Cancel
Apply

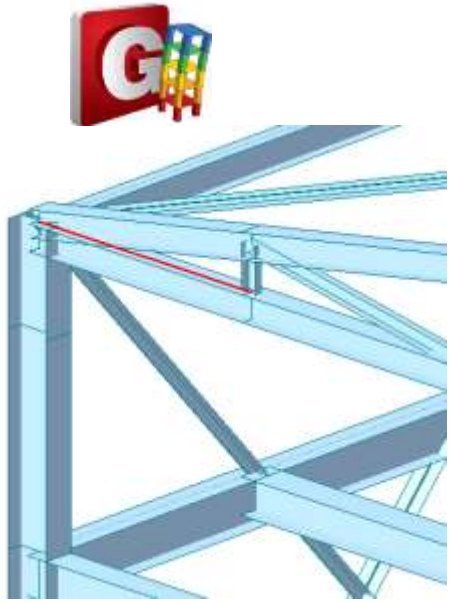

Generate mesh(Solid)
Auto-Solid
Map-Solid
2D->3D
Select Object(s)
Size Method
Size
0.063
Division
10
Automatic
More
Less
0.0947
Hybrid Mesher
Match Adjacent Faces
3
3: SS400
Mesh Set
Beam
OK
Cancel
Apply
>>



MESHING



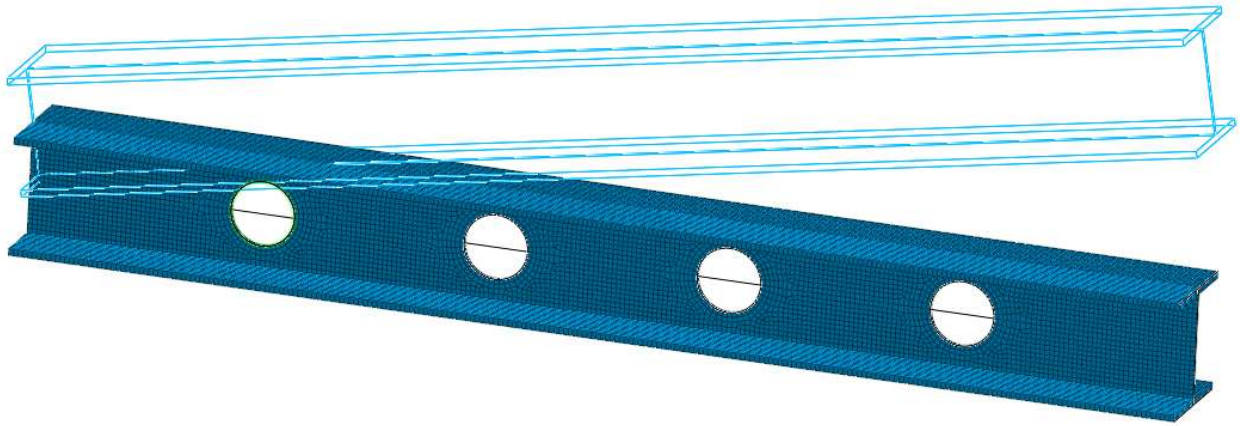


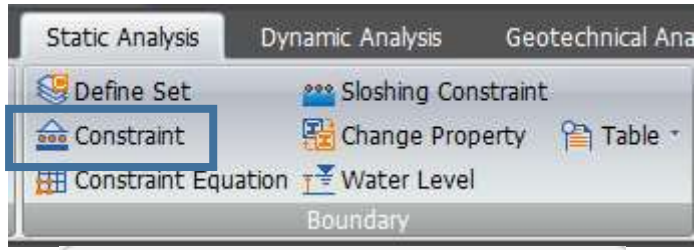


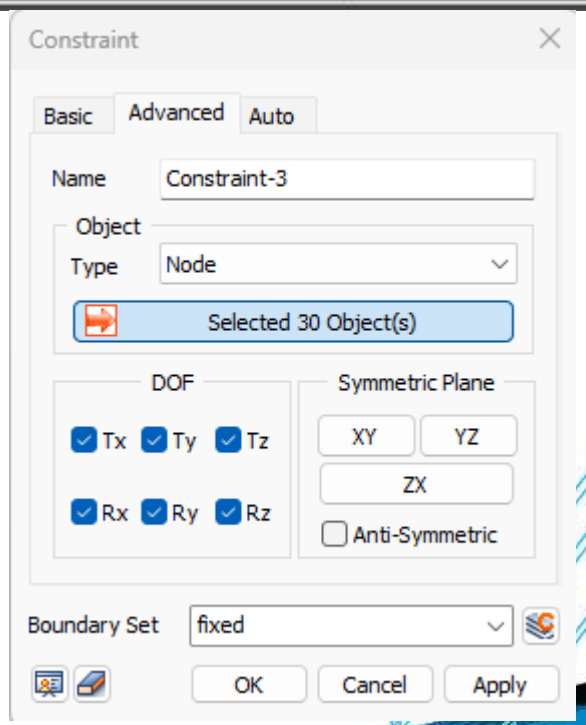
NOTE:

The smaller the mesh, the more accurate the results are.

LOADS and CONSTRAINTS

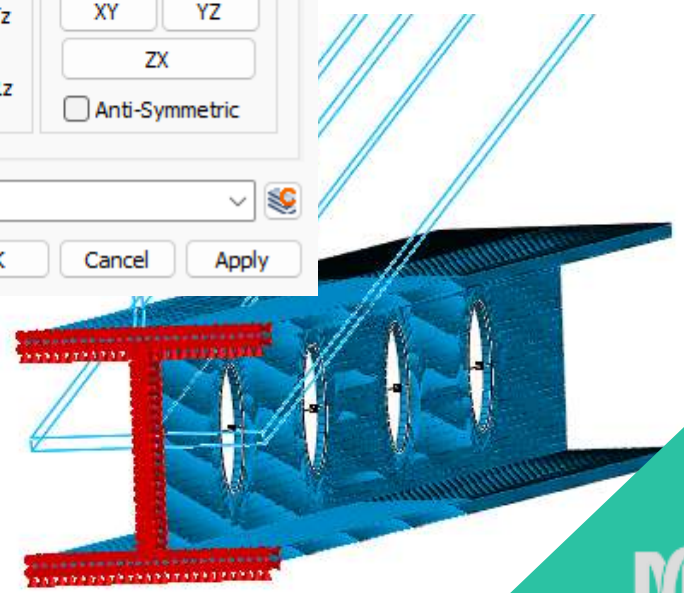






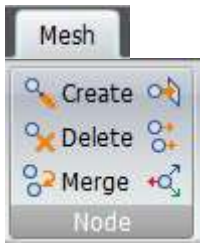
CONSTRAINT:

Pinned support on the joint

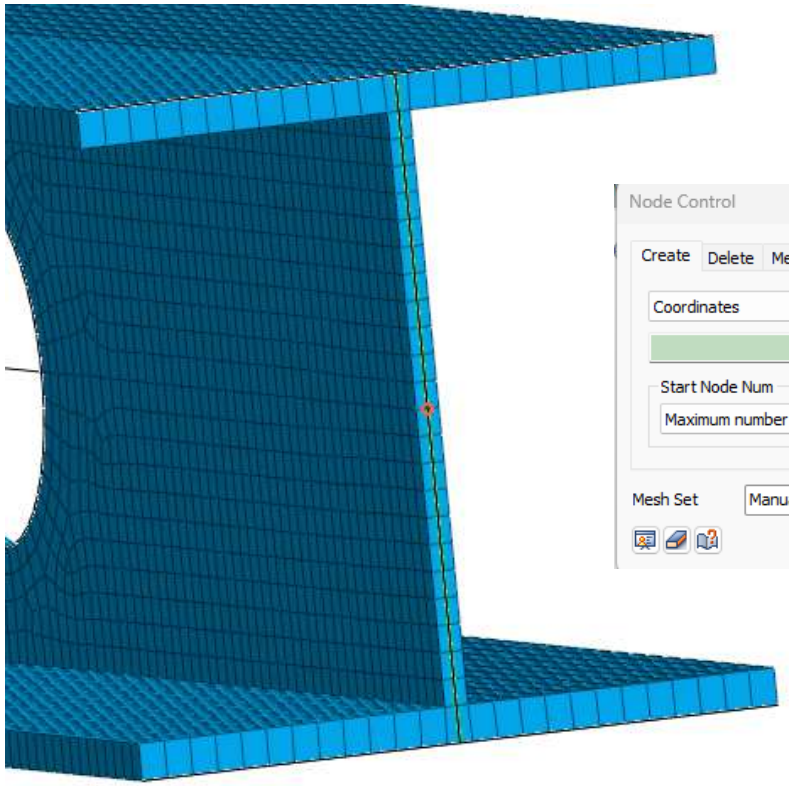


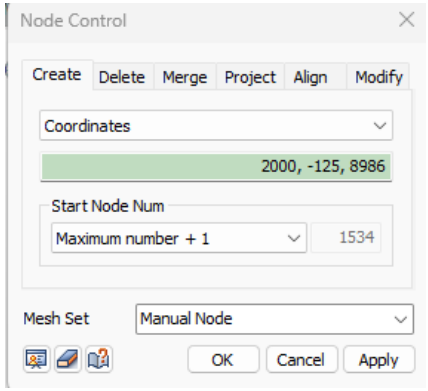
LOADS and CONSTRAINTS

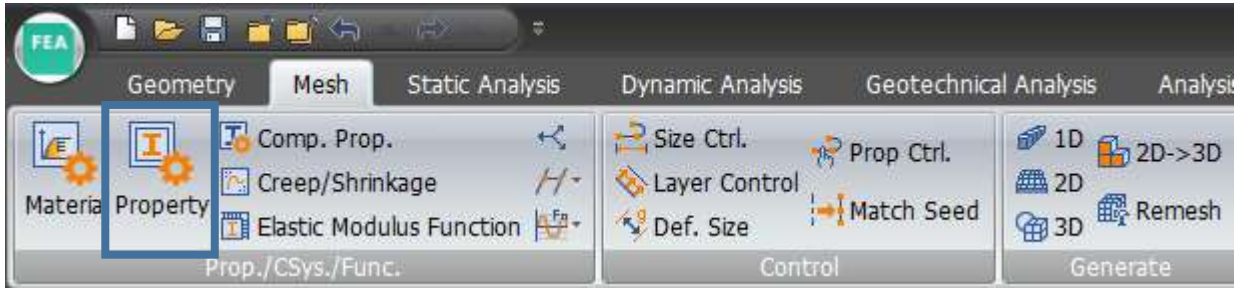
RIGID LINK-BEAM END FOR LOADING

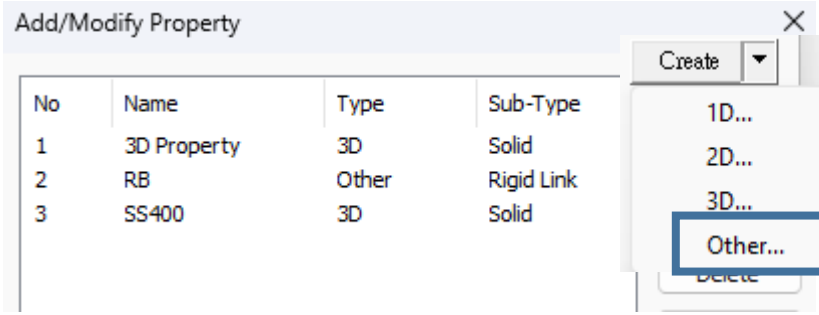


STEP 1: CREATE A MASTER NODE

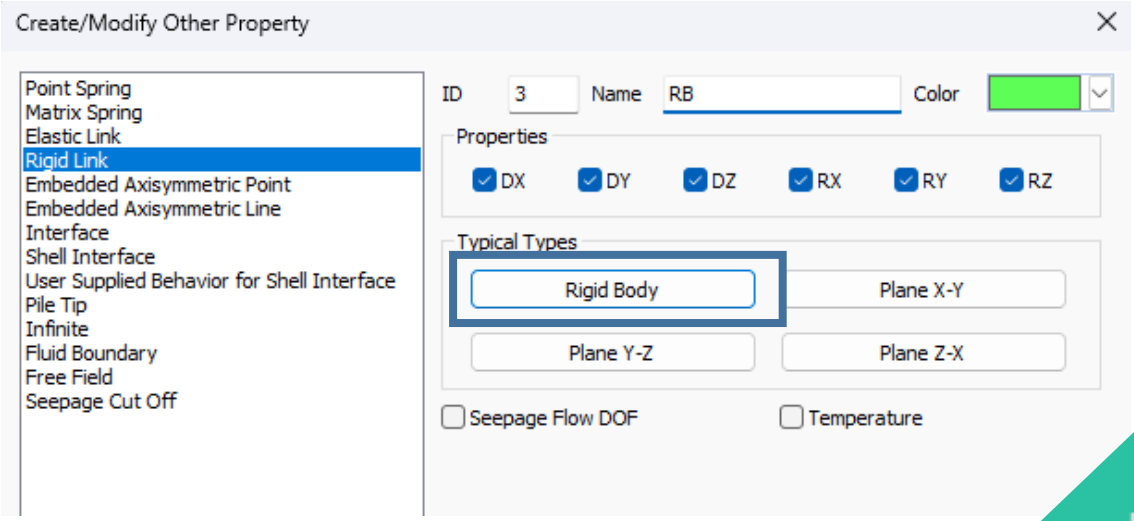






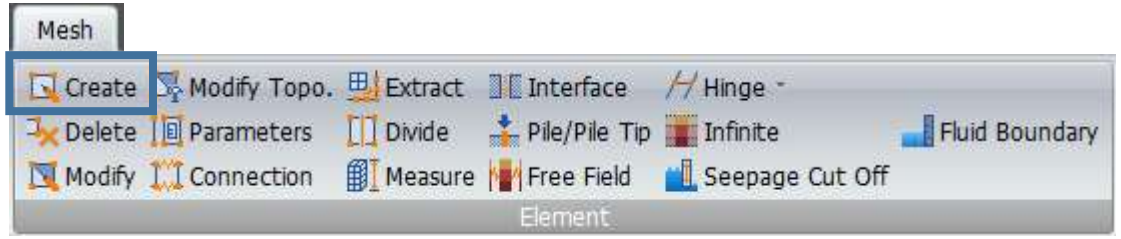


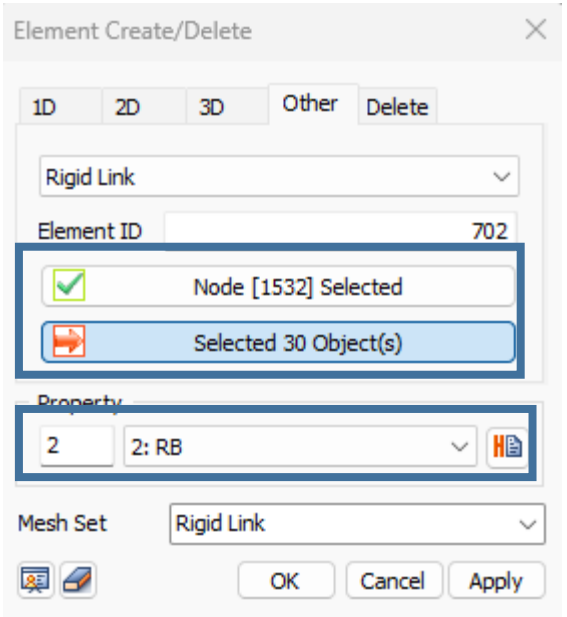
STEP 2: CREATE PROPERTY - RIGID LINK



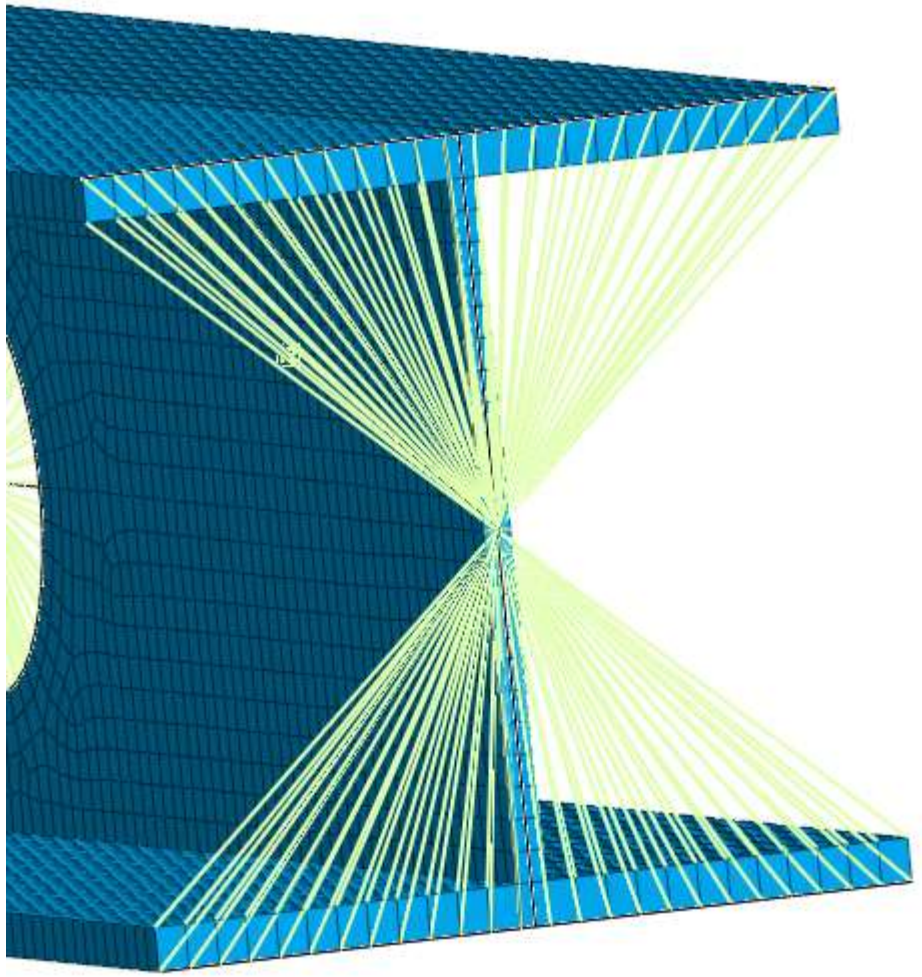
LOADS and CONSTRAINTS

RIGID LINK-BEAM END FOR LOADING



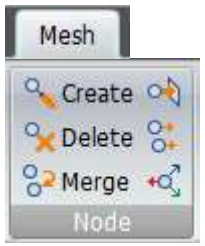


STEP 3: DEFINE RIGID LINK

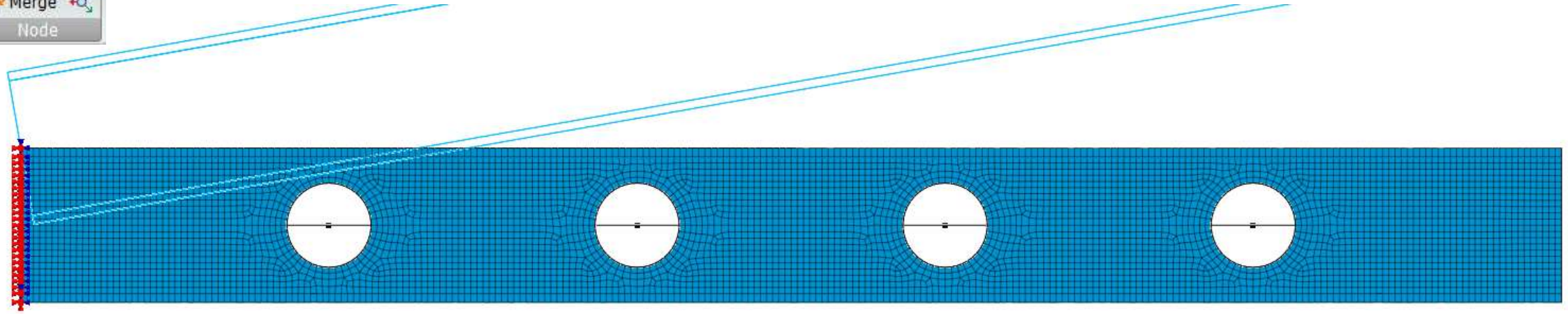


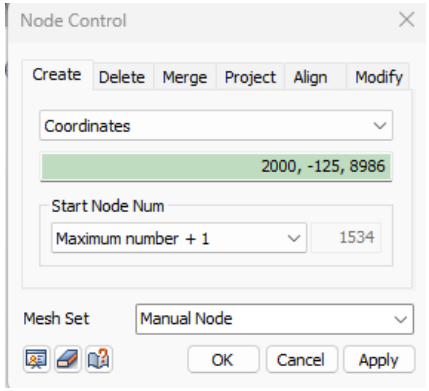
LOADS and CONSTRAINTS

RIGID LINK-CREATE LINKS FOR HOLES



STEP 4: CREATE A MASTER NODE

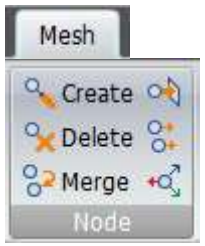




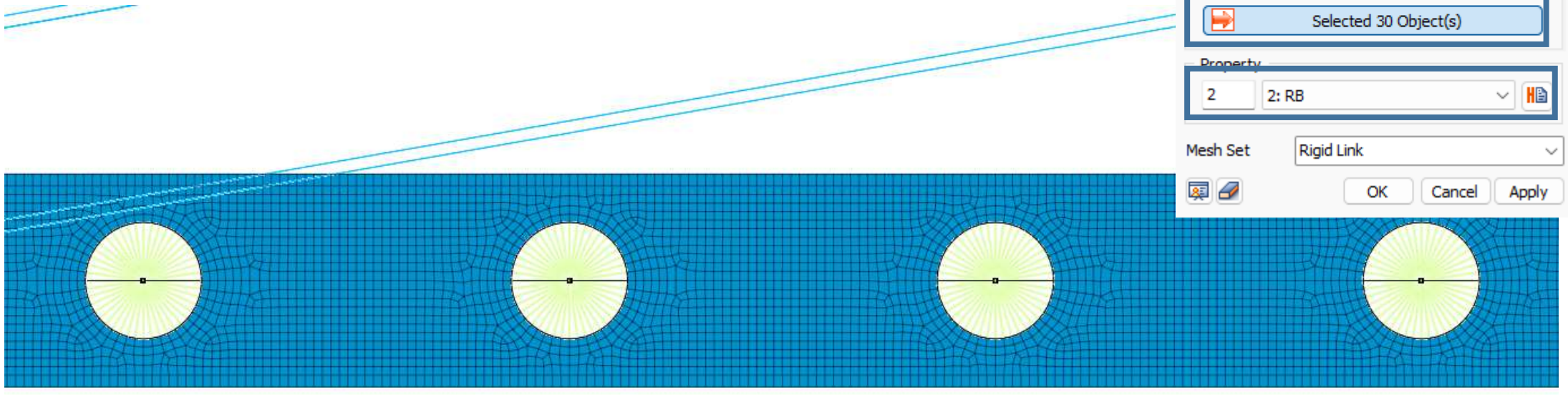
NOTE: This is to define the rigid link of the surface surrounding the hole which would ensure its rigid movement.

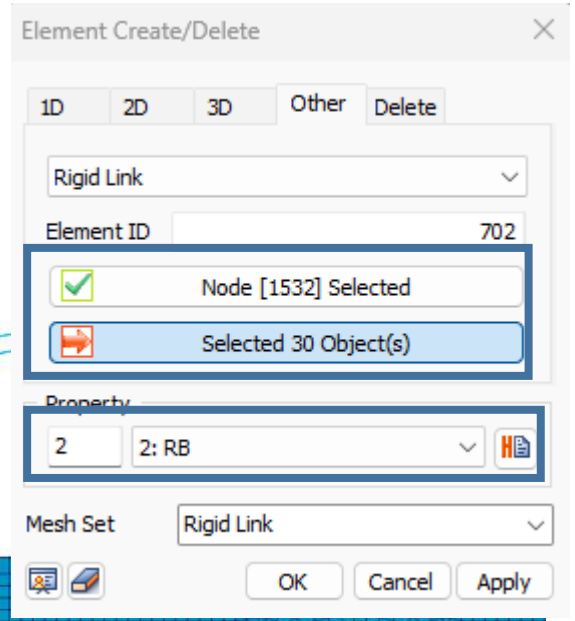
LOADS and CONSTRAINTS

RIGID LINK-CREATE LINKS FOR HOLES



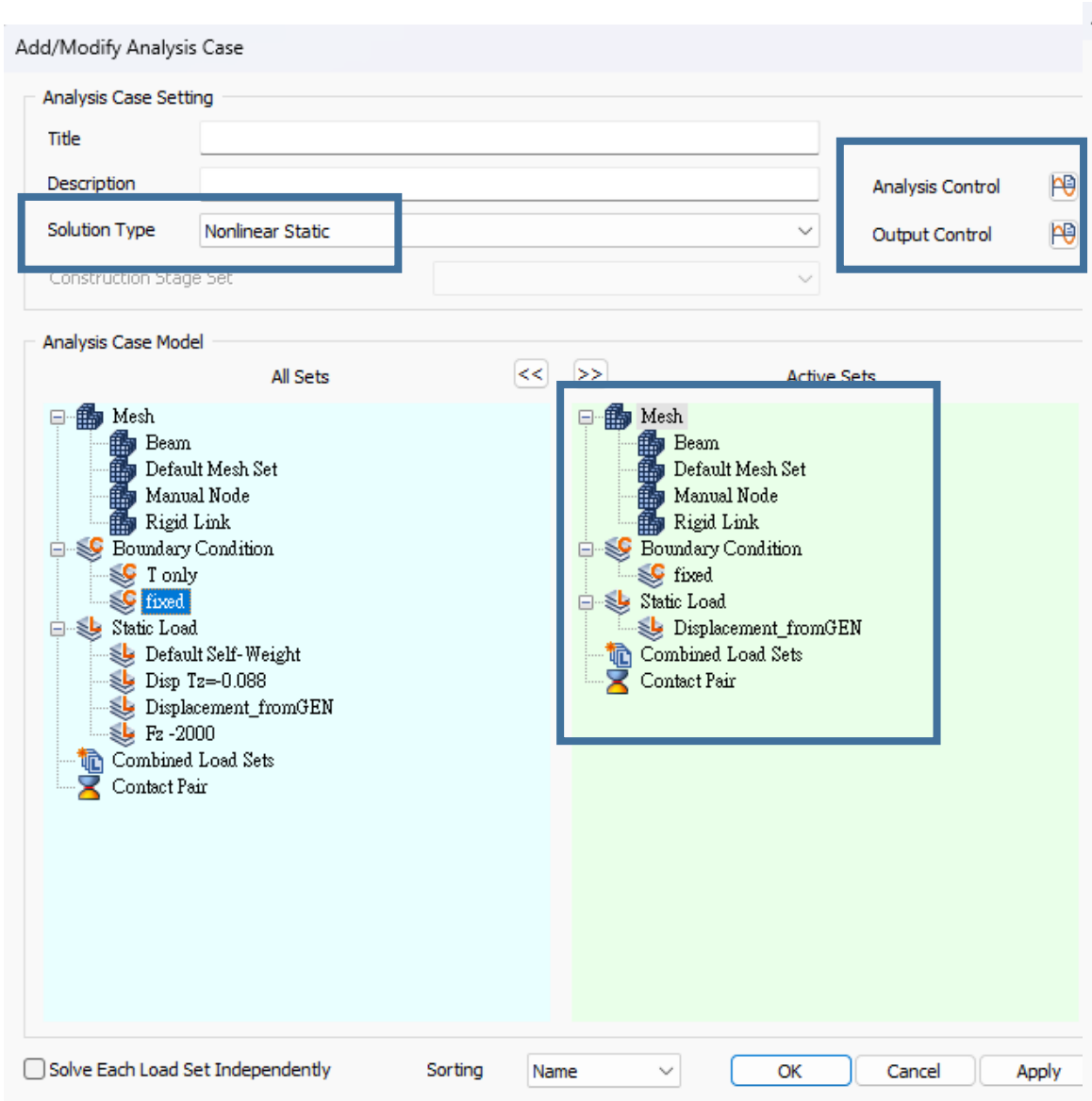
STEP 5: DEFINE RIGID LINK FOR HOLES

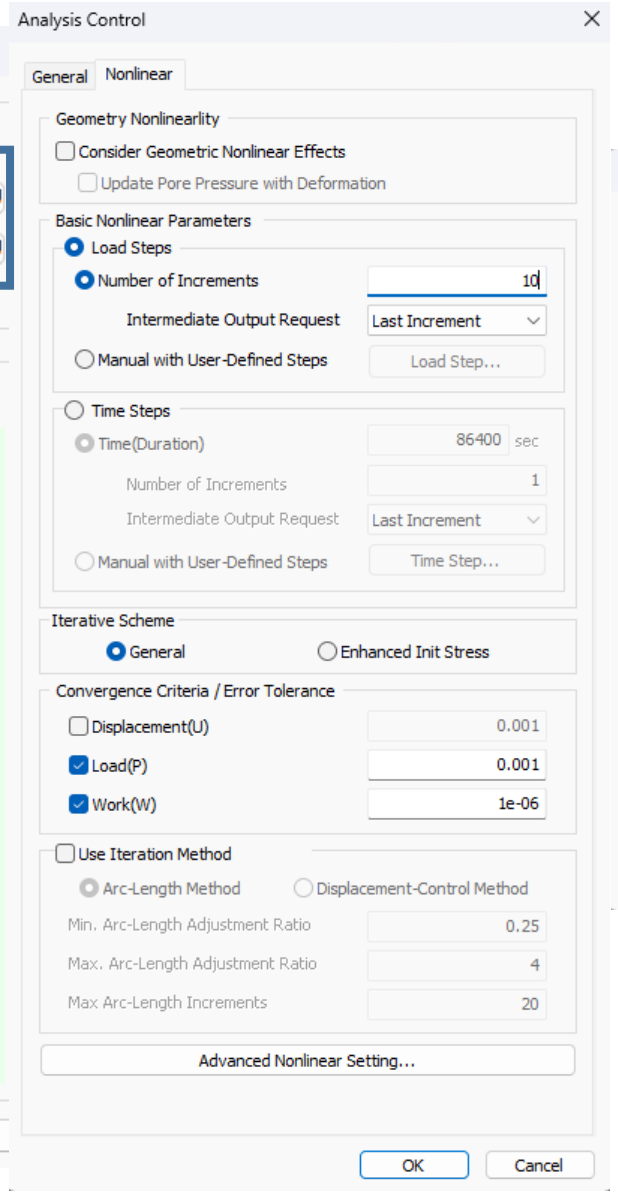


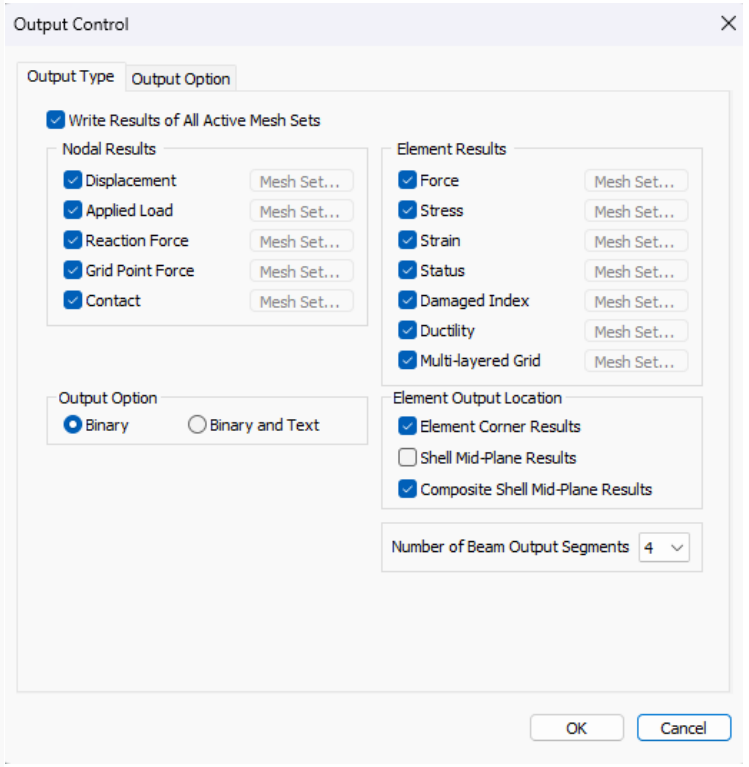


NOTE: Define the rigid link 1 by 1 (per hole).

ANALYSIS CASE



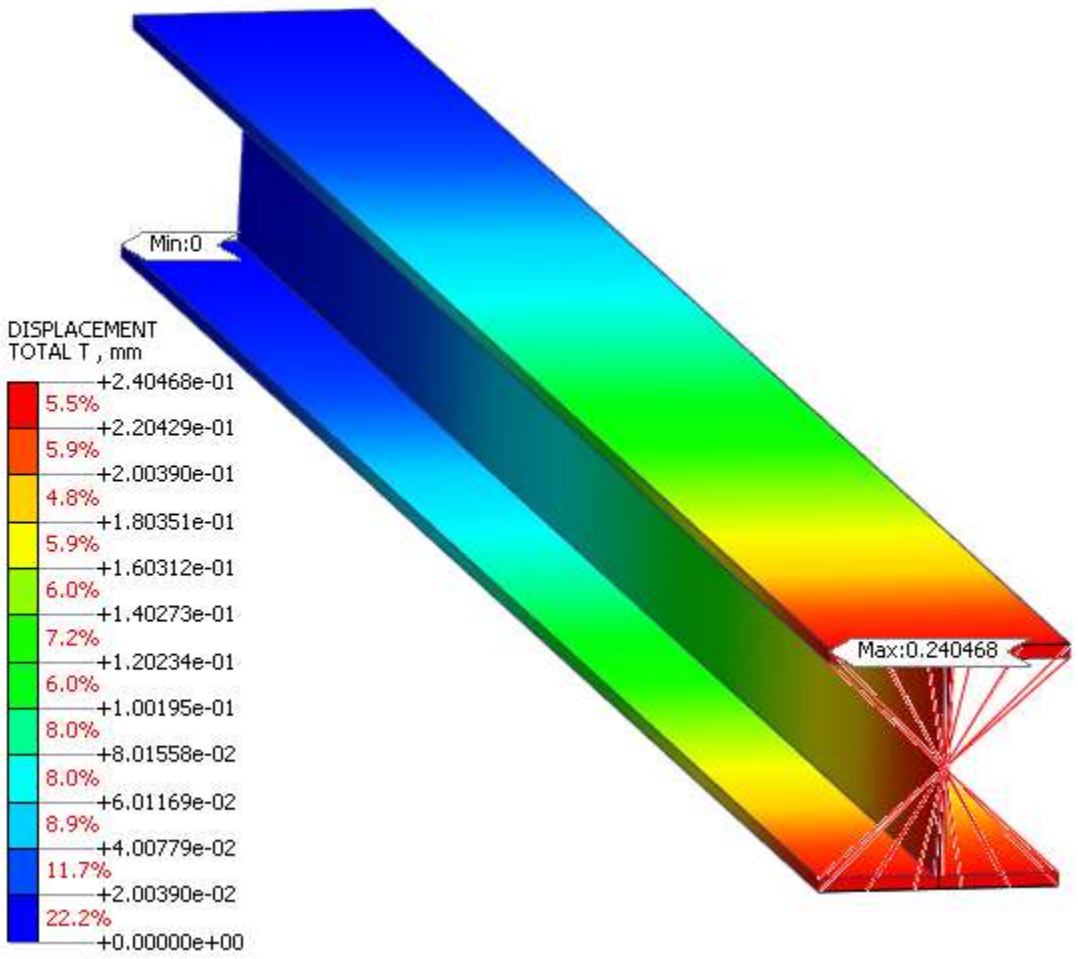


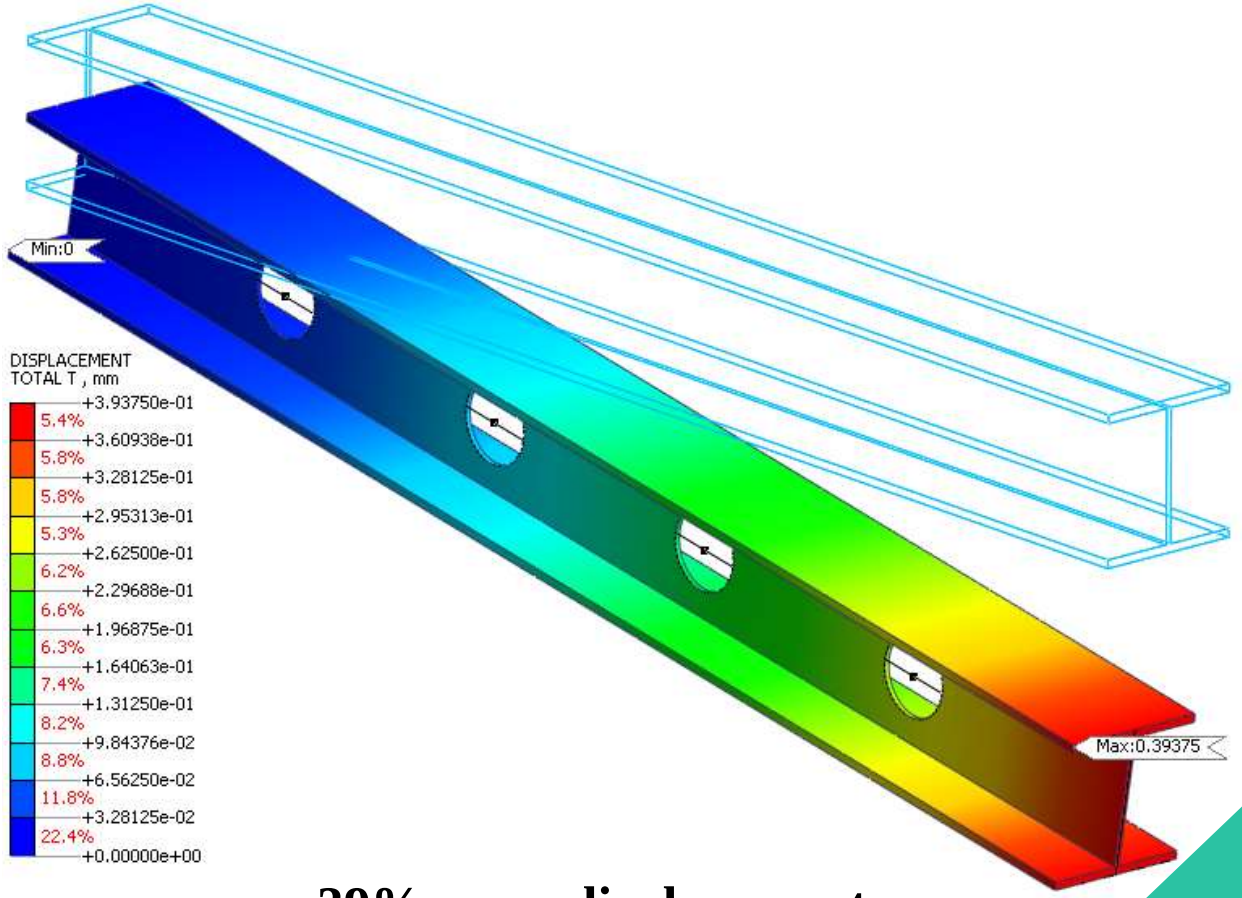


Part 3

Solid and Modified Beam Result Comparison

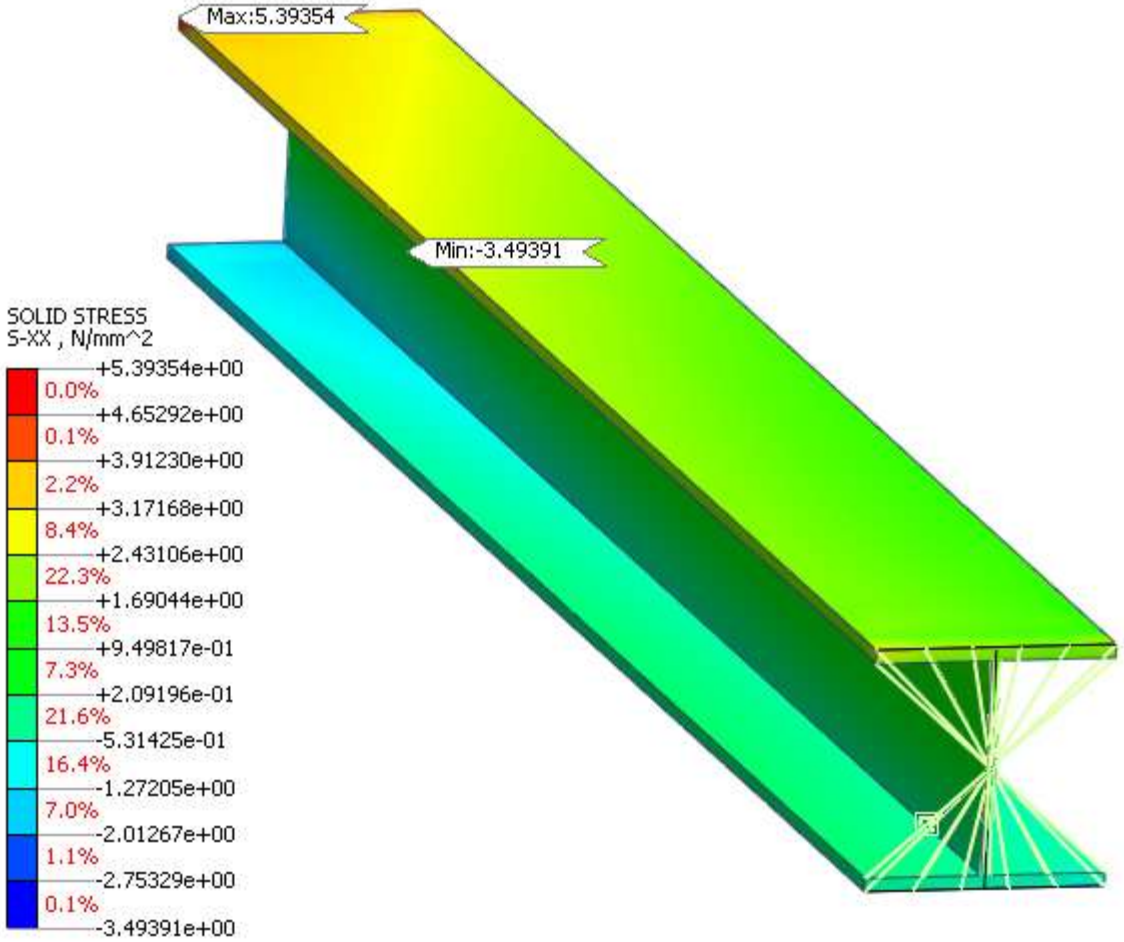
RESULTS - BEAM DEFLECTION COMPARISON

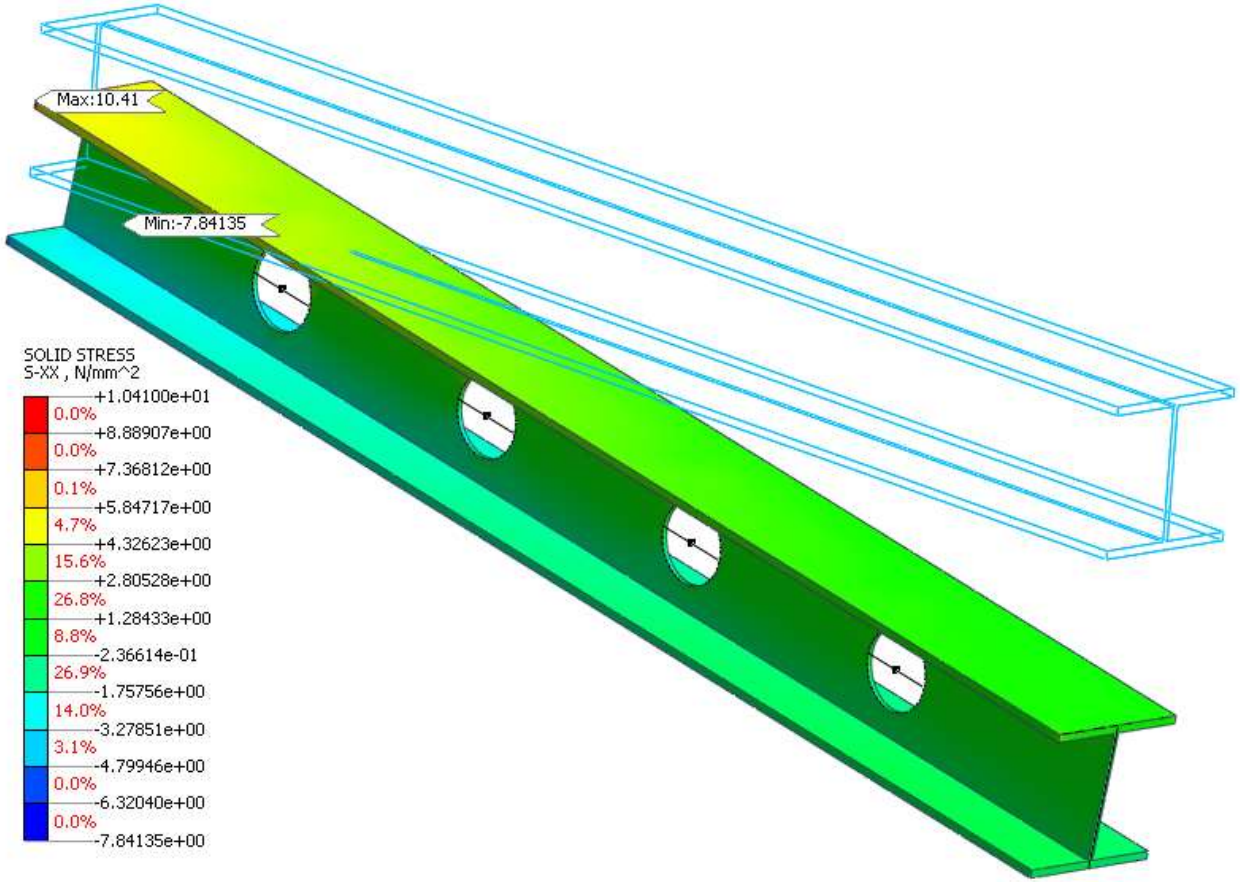




~39% more displacement

RESULTS - BEAM STRESS COMPARISON

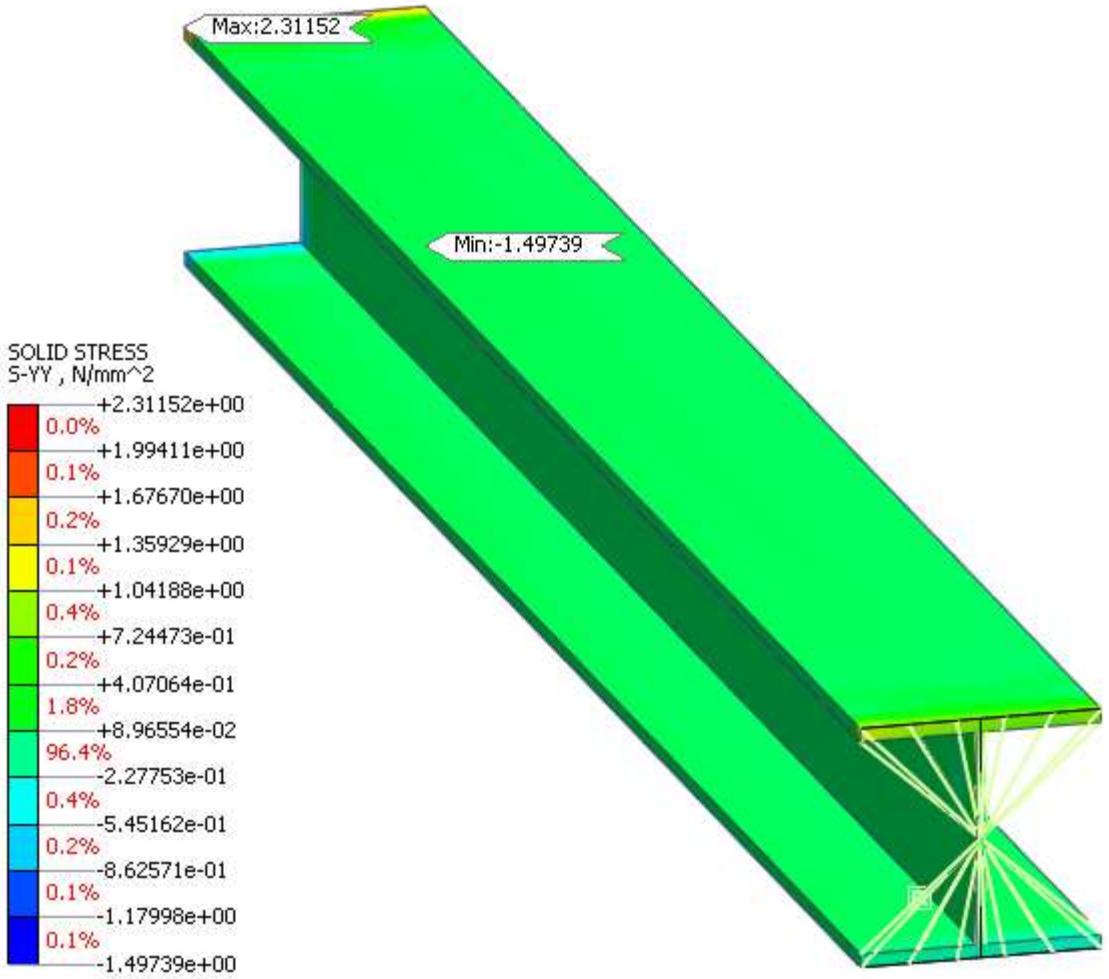


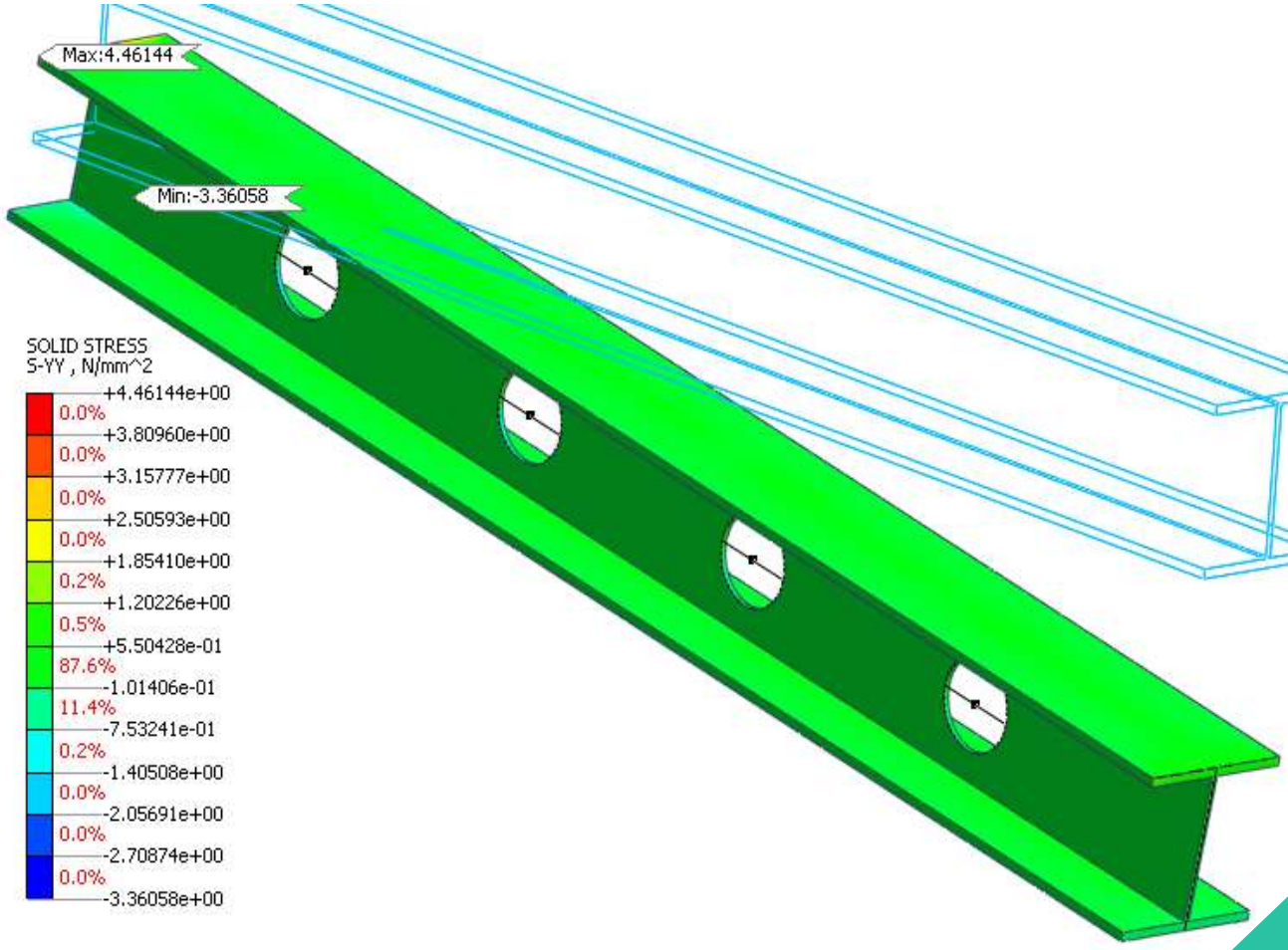


Max S-XX: ~93% increase

Primary region S-XX: ~15% increase

RESULTS - BEAM STRESS COMPARISON



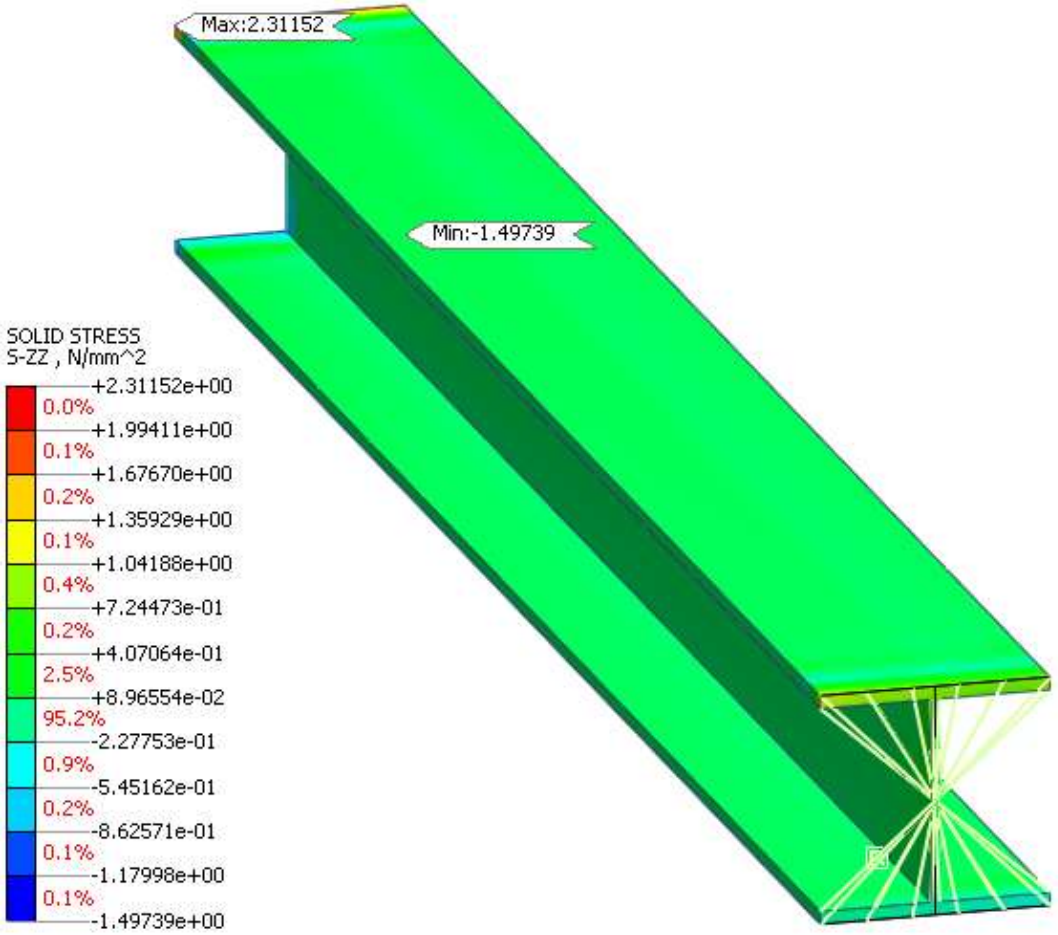


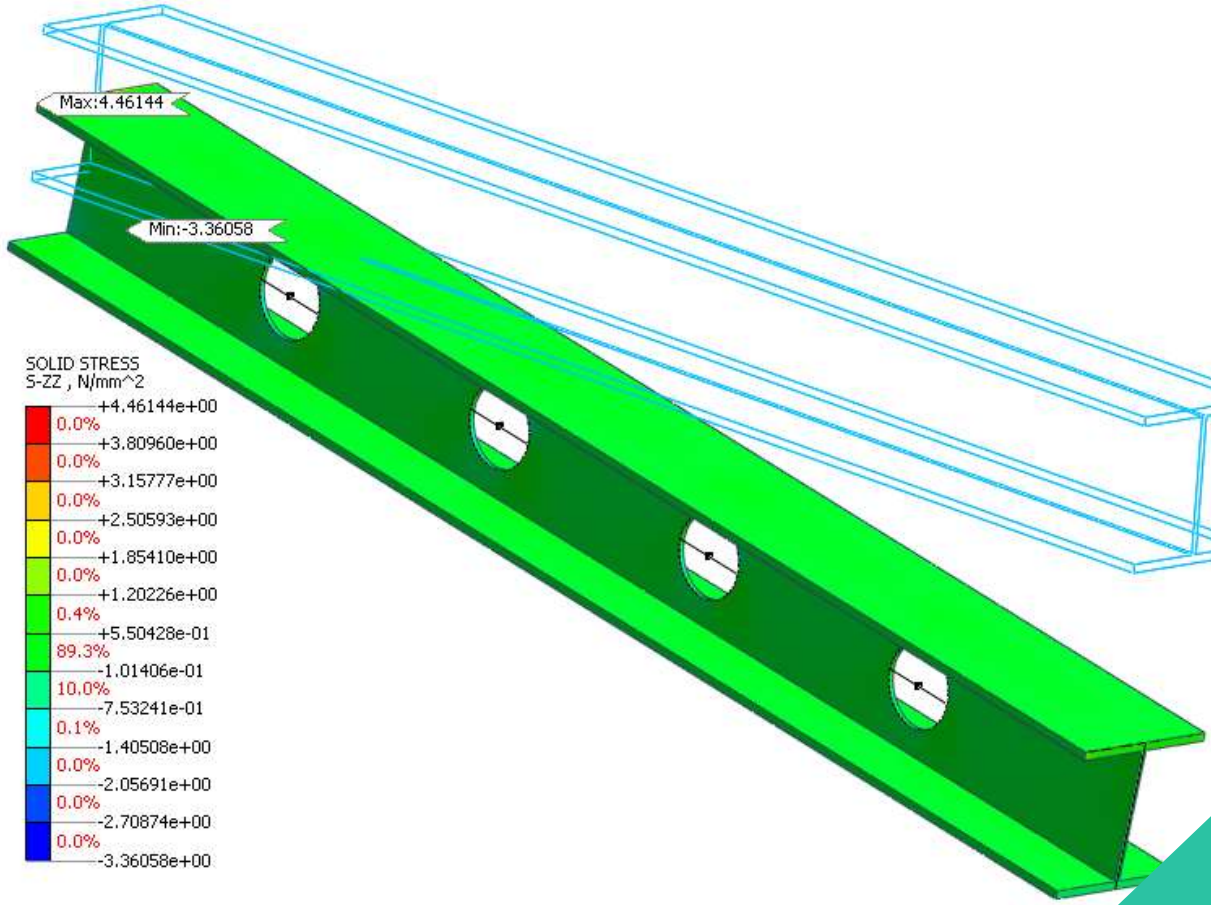
Max S-YY: ~93% increase

Primary region S-YY: 5x increase

Compression Region: 0.8% --> 11.6%

RESULTS - BEAM STRESS COMPARISON



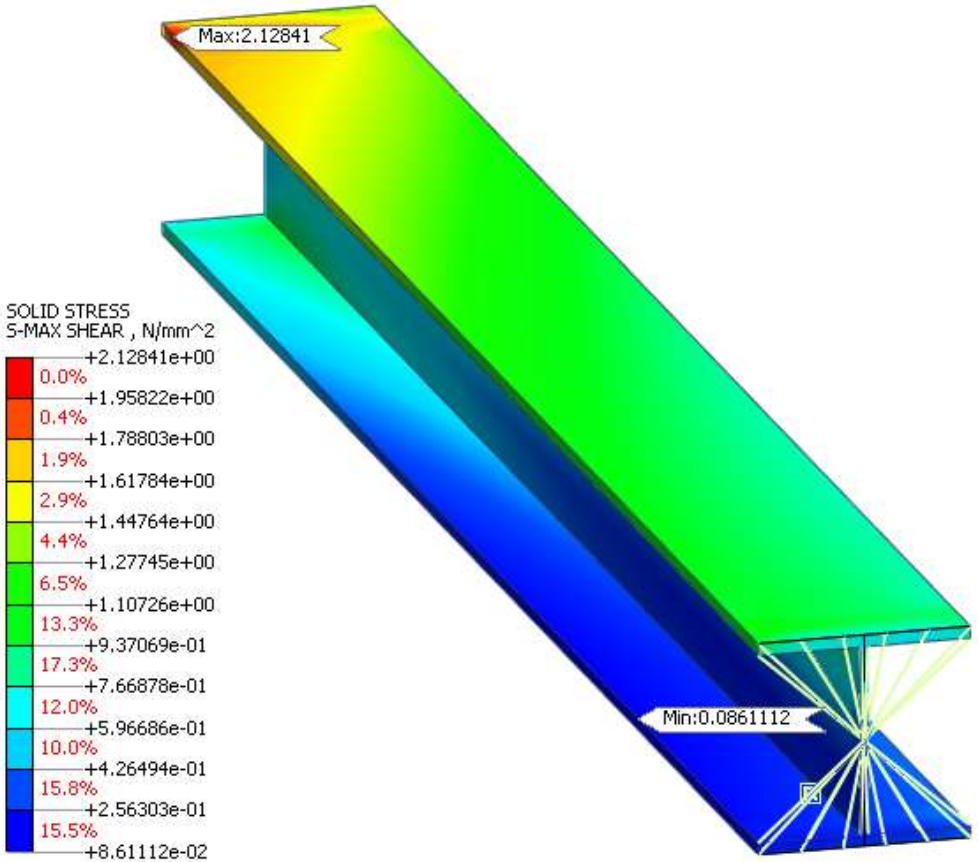


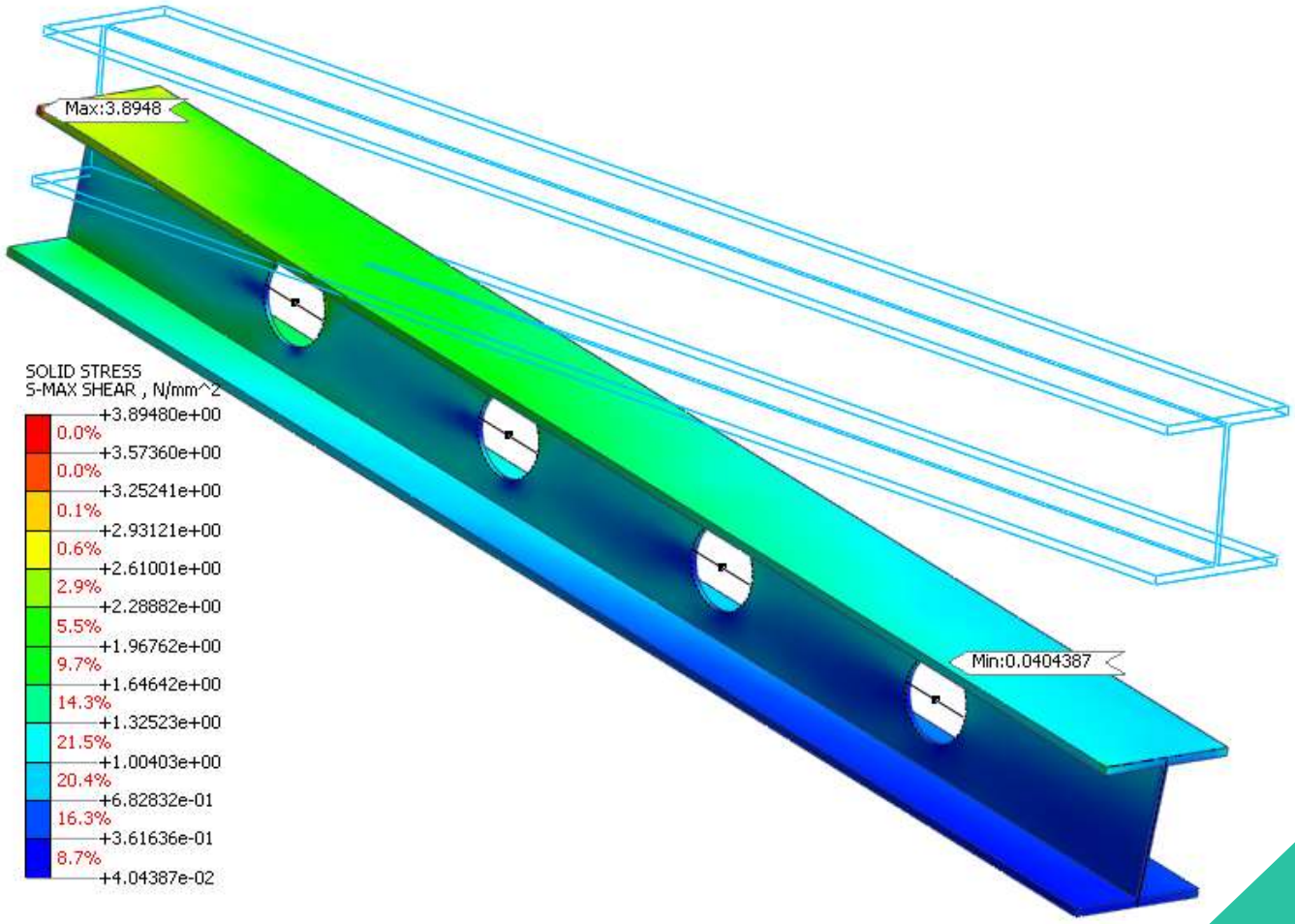
Max S-ZZ: ~93% increase

Primary region S-ZZ: 5x increase

Compression Region: 0.13% --> 11.1%

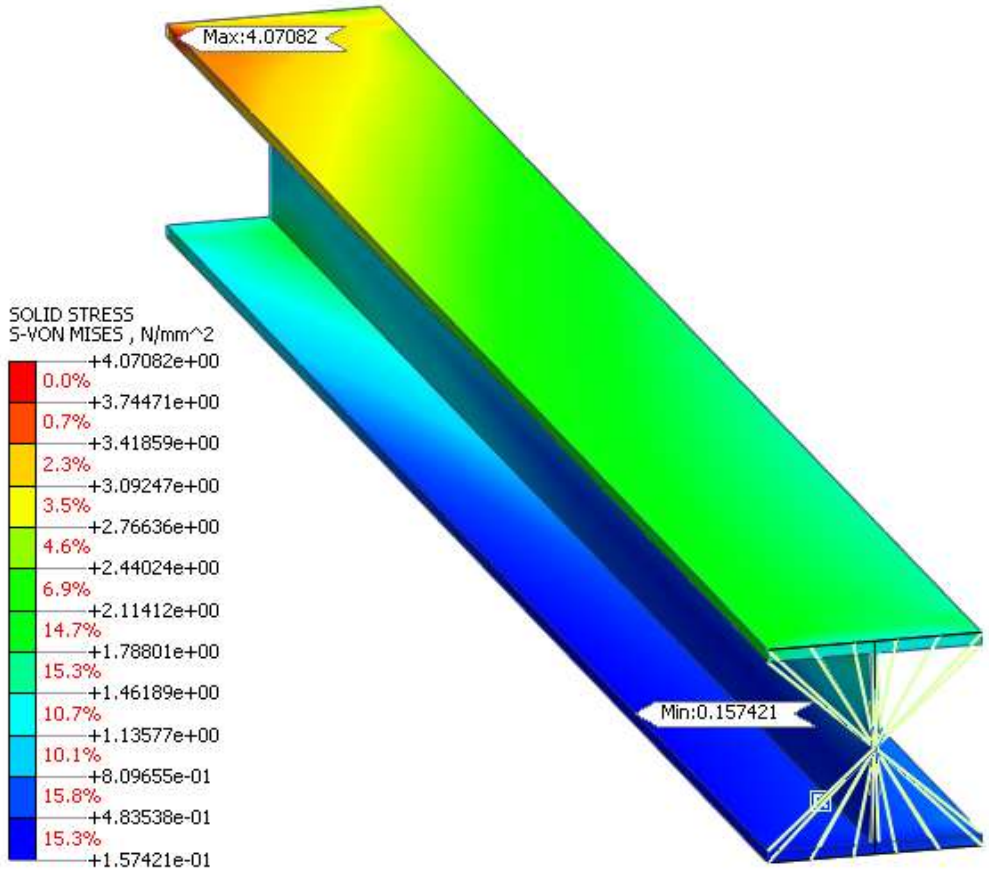
RESULTS - BEAM STRESS COMPARISON

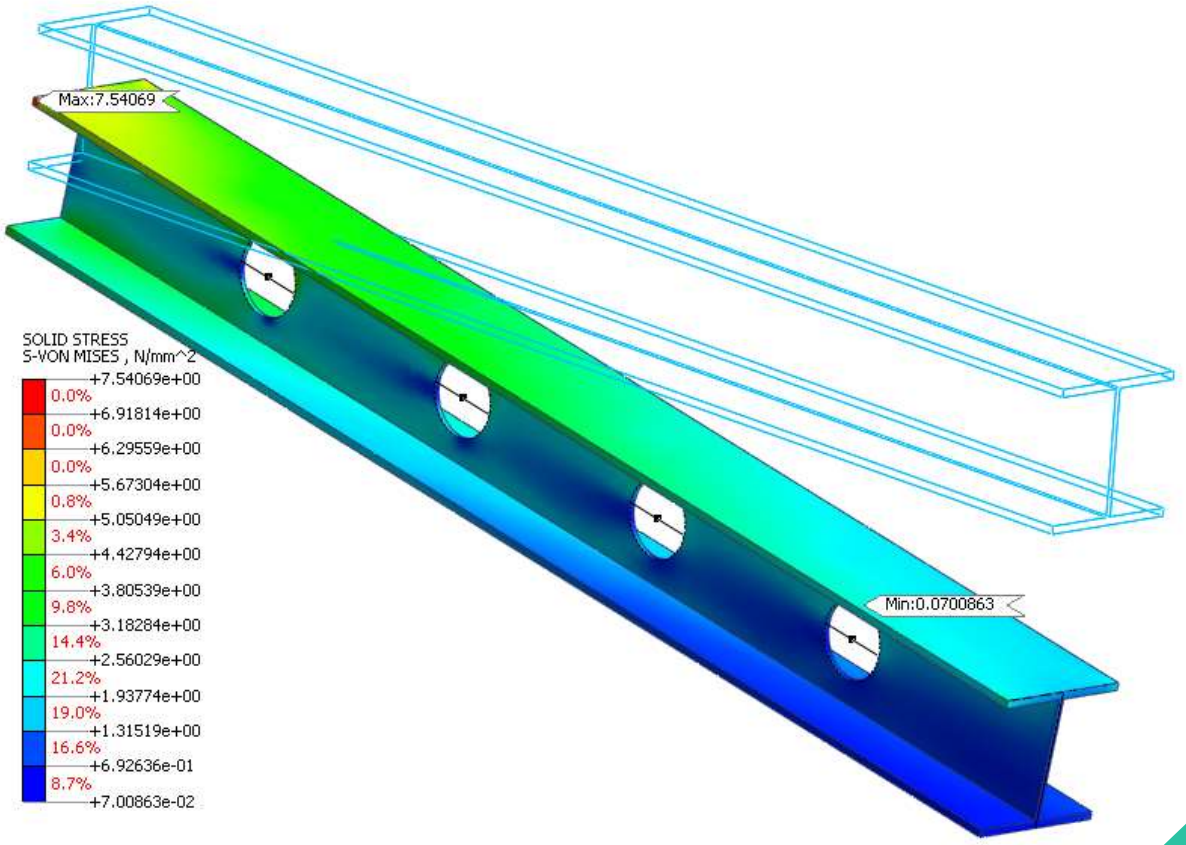




Max Shear: ~77.8% increase

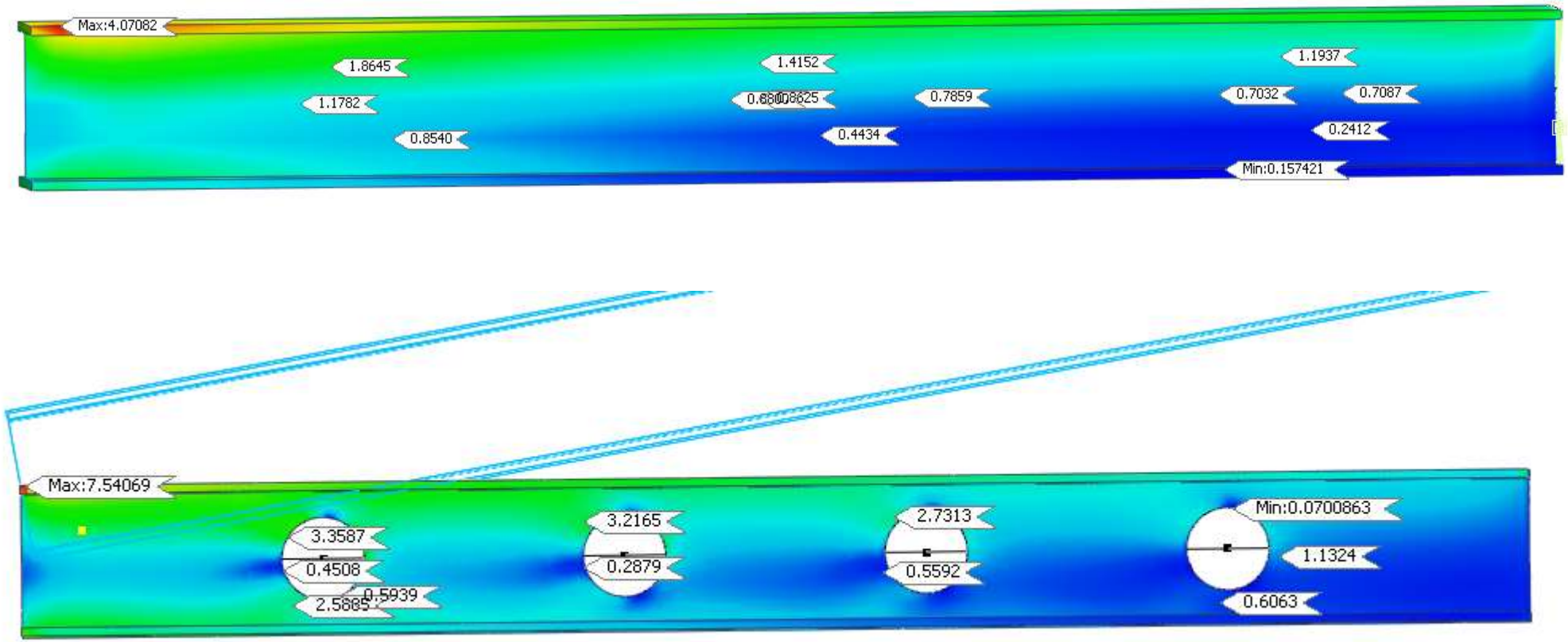
RESULTS - BEAM STRESS COMPARISON





Max S-Von mises: ~85.2% increase

RESULTS - BEAM STRESS COMPARISON



Max S-Von mises: Created high stress points near the hole, about 2x increase

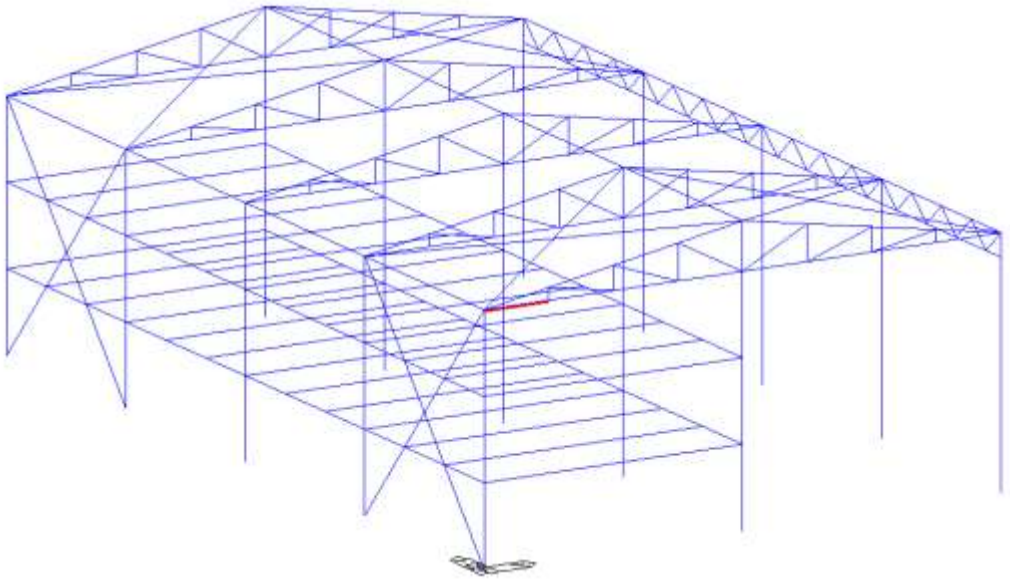
FEA NX Simulation Platform

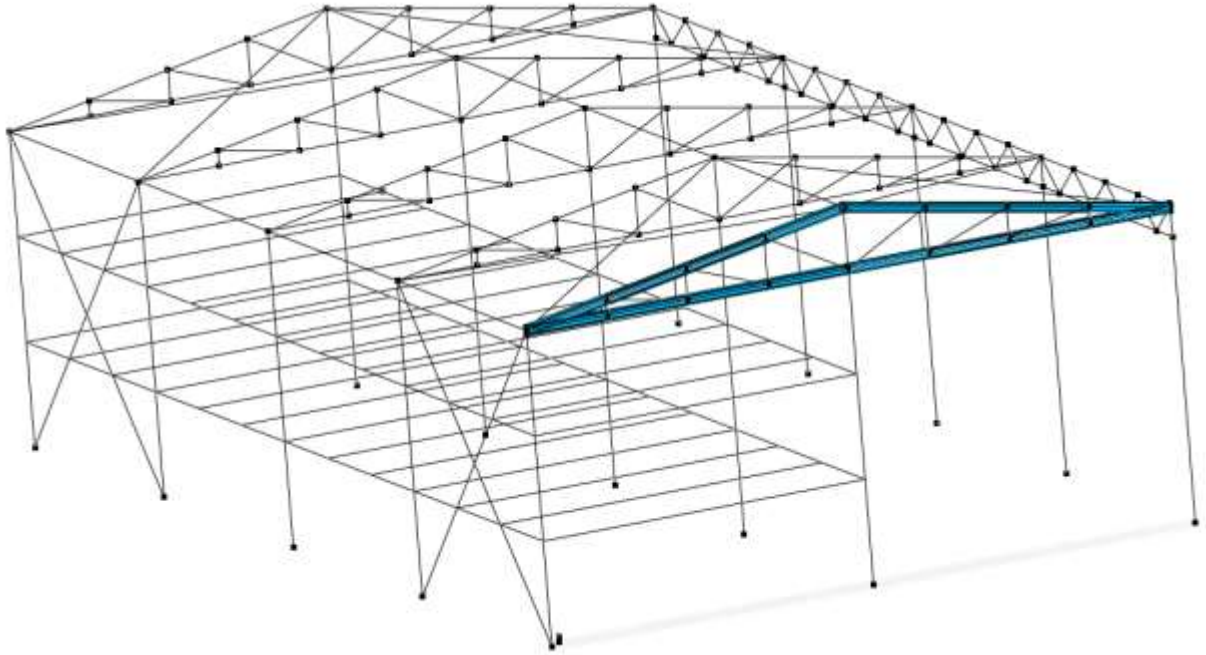
Integration of MIDAS **GEN** and **FEA NX** Series Structural Element Detailed Analysis (Full Frame Approach)

沈約翰
john@midasuser.com.tw

CASE ANALYSIS OVERVIEW

3D ANALYSIS OF A BEAM ELEMENT FROM A 1D FRAME

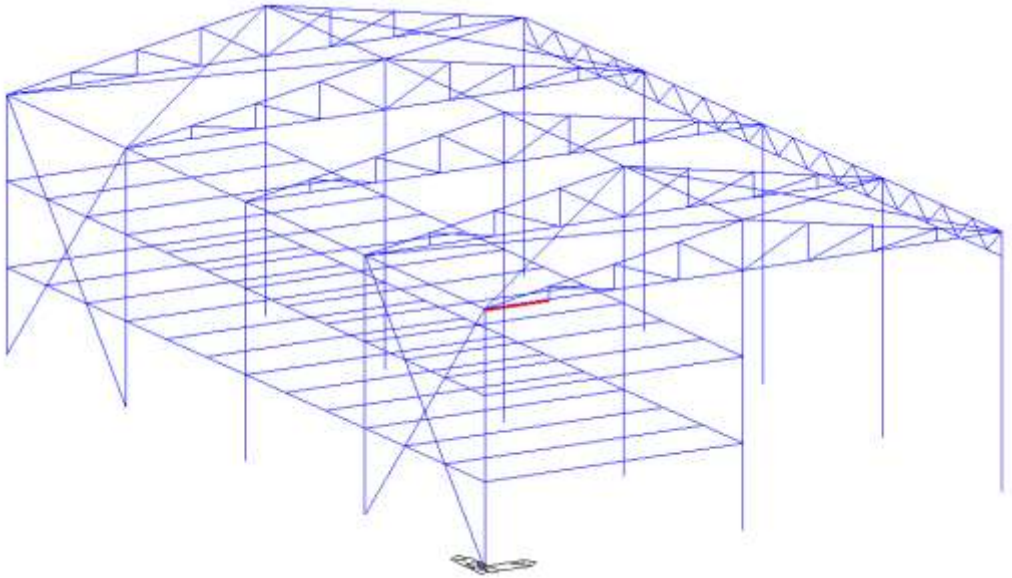


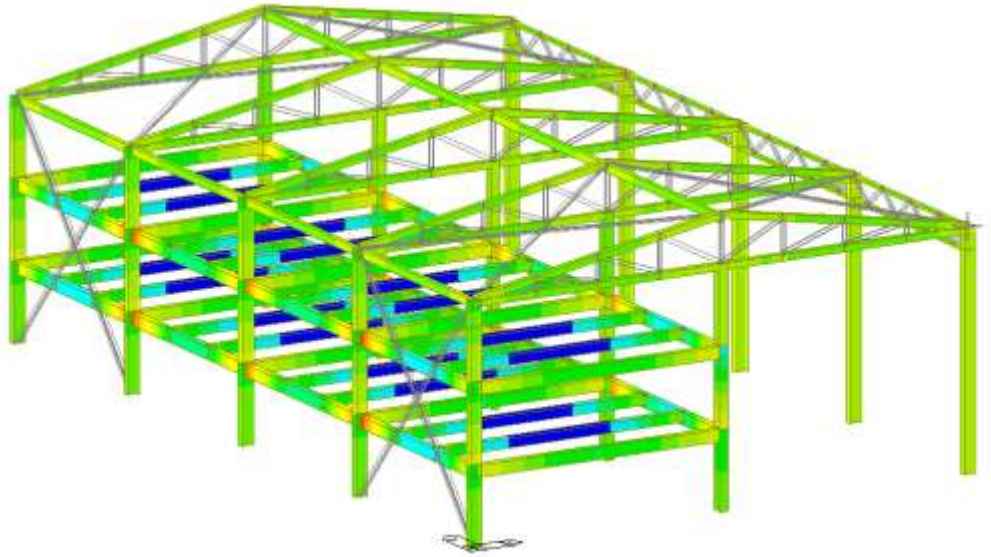


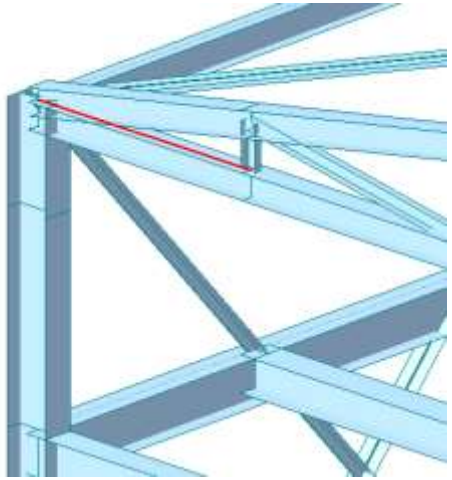
Part 0

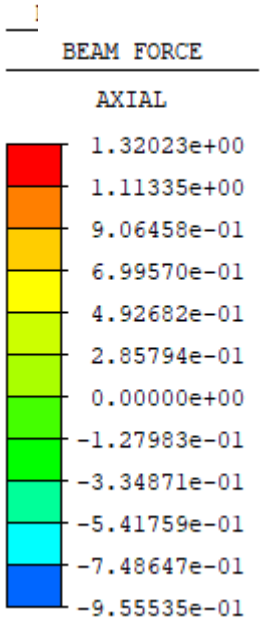
GEN Simulation and Results

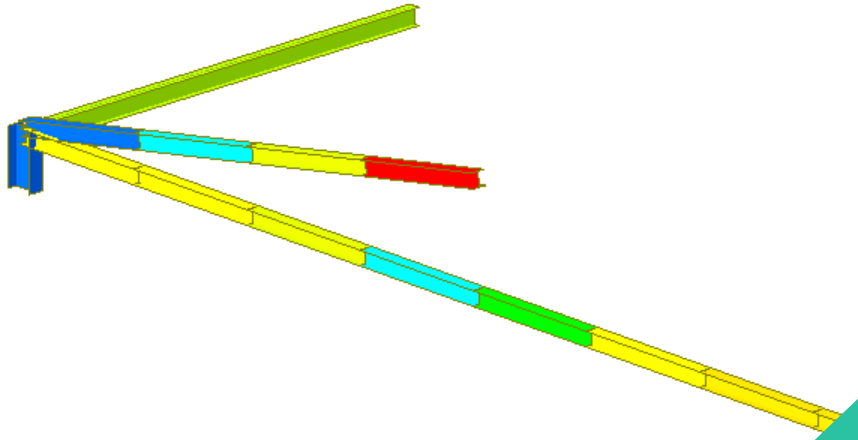
MIDAS GEN MODEL and RESULTS





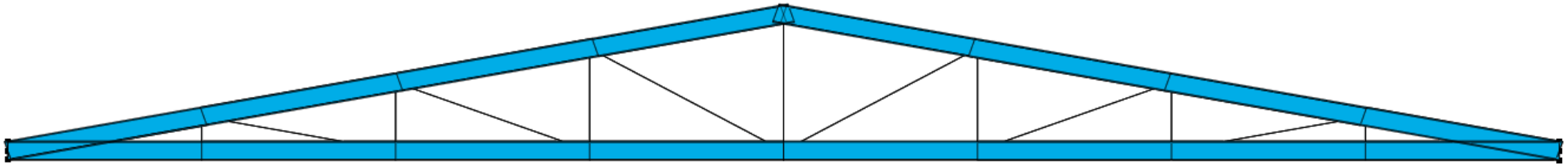




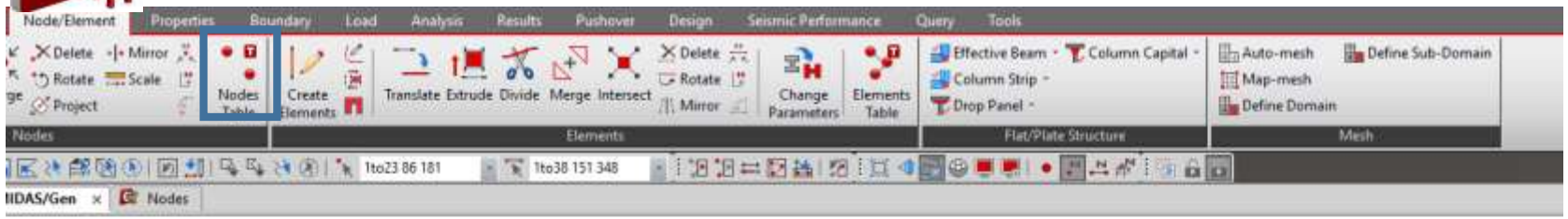


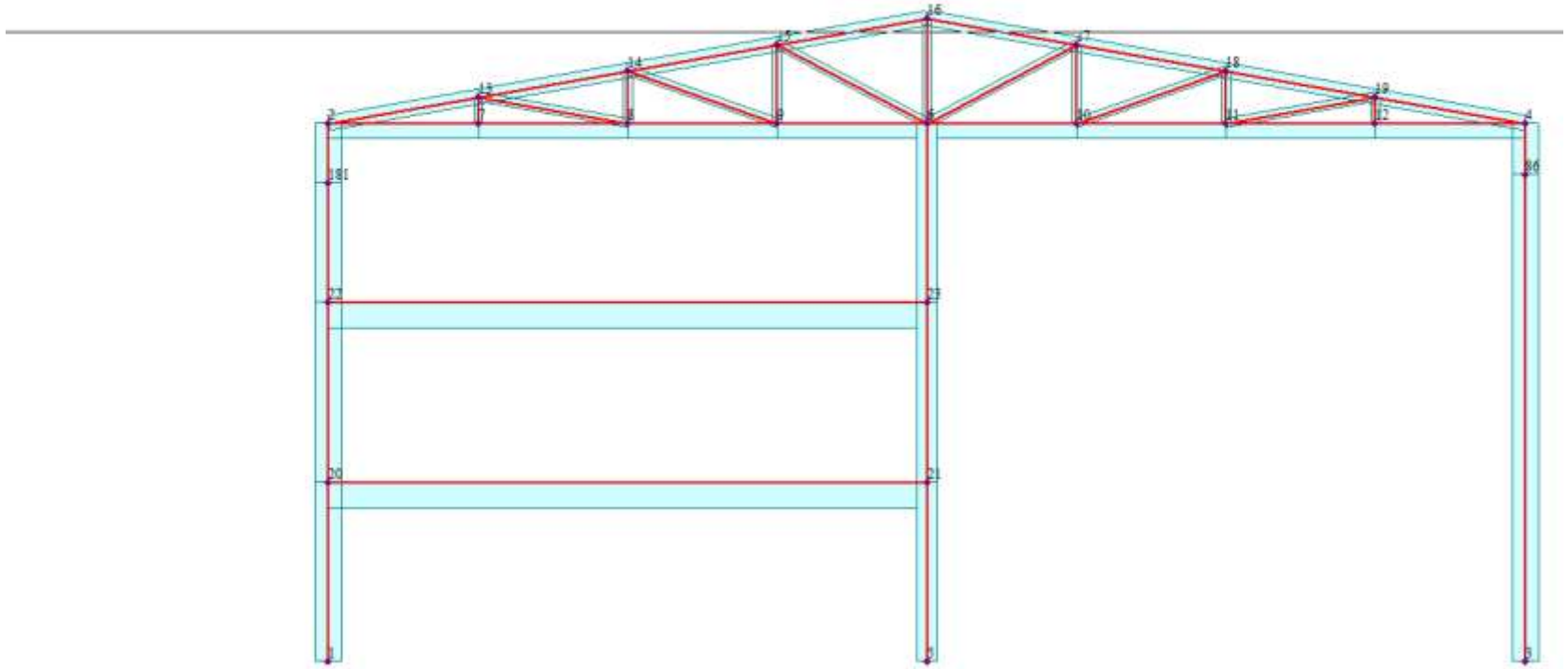
Part 1

Frame Detailed Analysis



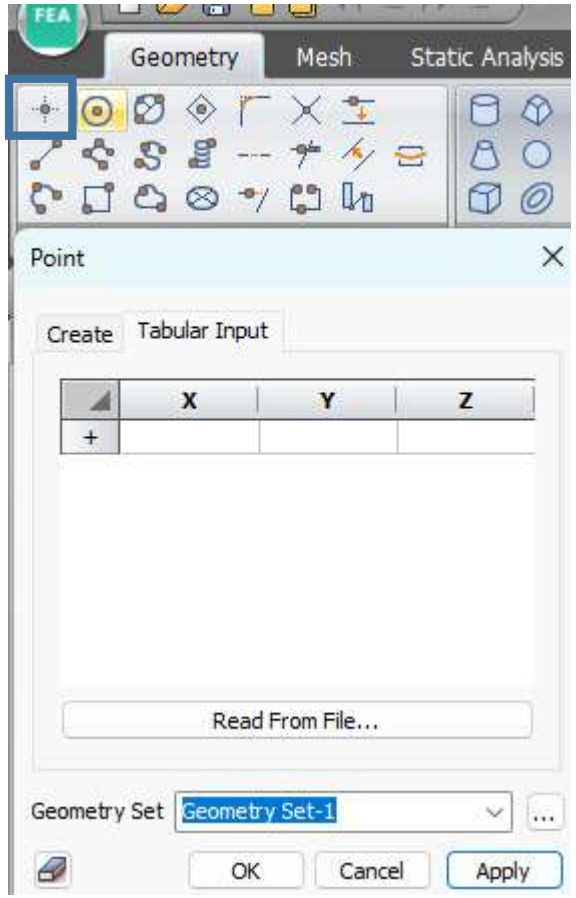
FRAME GUIDE





	Node	X(m)	Y(m)	Z(m)
	1	0.000000	0.000000	0.000000
▶	2	0.000000	0.000000	9.000000
	3	20.000000	0.000000	0.000000
	4	20.000000	0.000000	9.000000
	5	10.000000	0.000000	0.000000
	6	10.000000	0.000000	9.000000
	7	2.500000	0.000000	9.000000
	8	5.000000	0.000000	9.000000
	9	7.500000	0.000000	9.000000
	10	12.500000	0.000000	9.000000
	11	15.000000	0.000000	9.000000
	12	17.500000	0.000000	9.000000
	13	2.500000	0.000000	9.438107
	14	5.000000	0.000000	9.876214
	15	7.500000	0.000000	10.314321
	16	10.000000	0.000000	10.752427
	17	12.500000	0.000000	10.314321
	18	15.000000	0.000000	9.876214
	19	17.500000	0.000000	9.438107
	20	0.000000	0.000000	3.000000
	21	10.000000	0.000000	3.000000
	22	0.000000	0.000000	6.000000
	23	10.000000	0.000000	6.000000

FRAME GUIDE

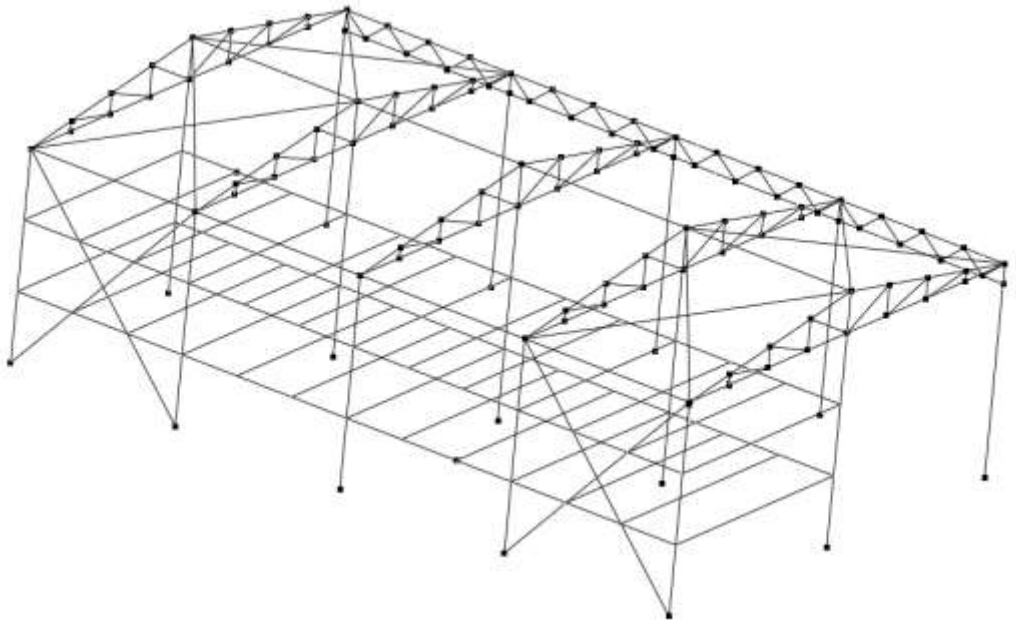


NOTE:

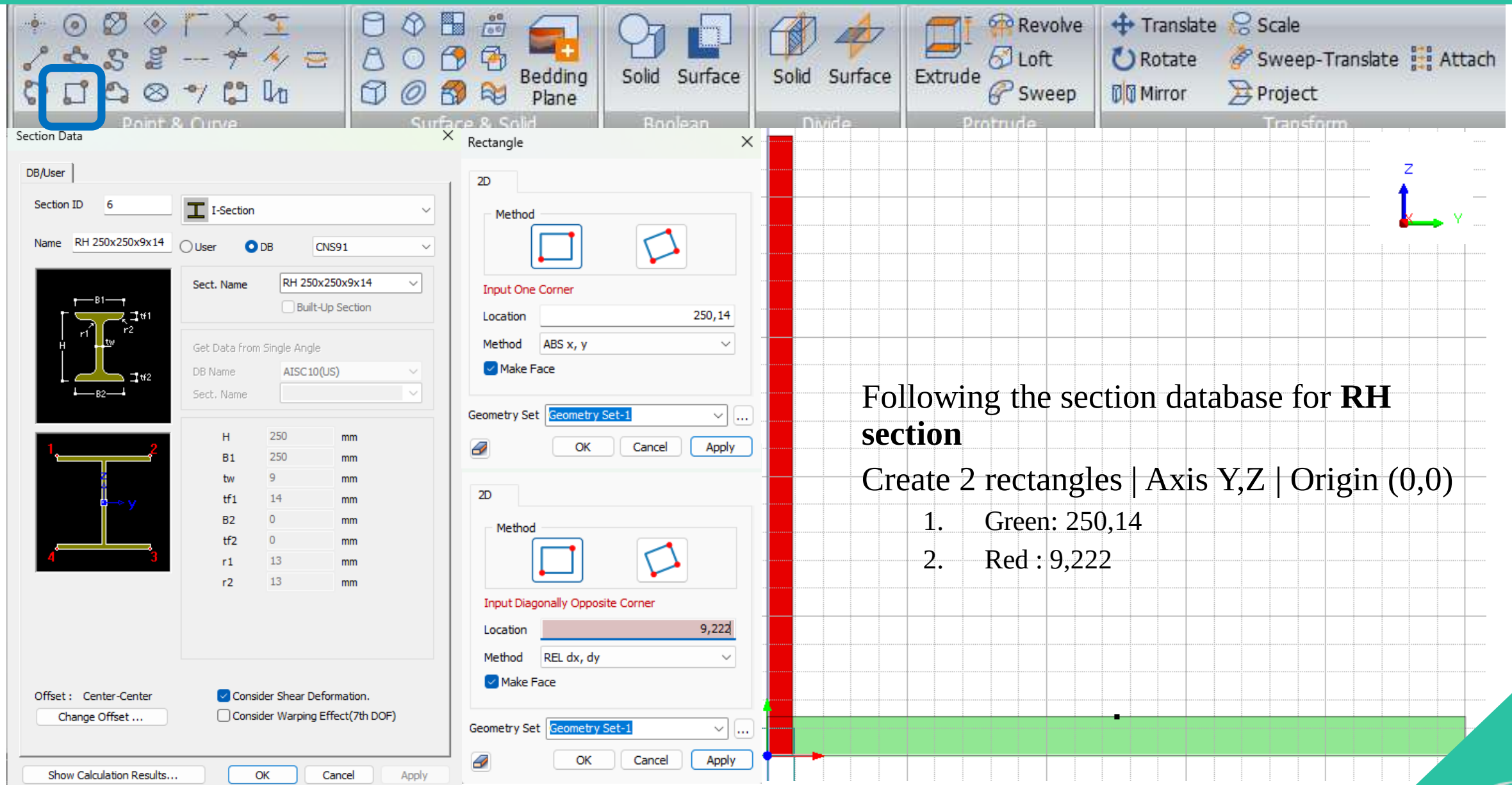
Extract the nodes from GEN and import it to FEA.

This will create guidepoints to construct the frame.

Then add the other elements to create the entire **1D structure**.



ADDITIONAL GEOMETRY (3D)

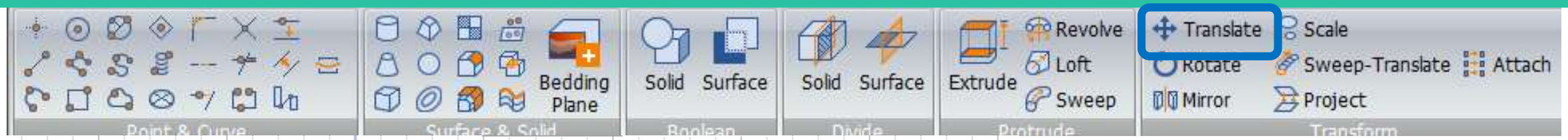


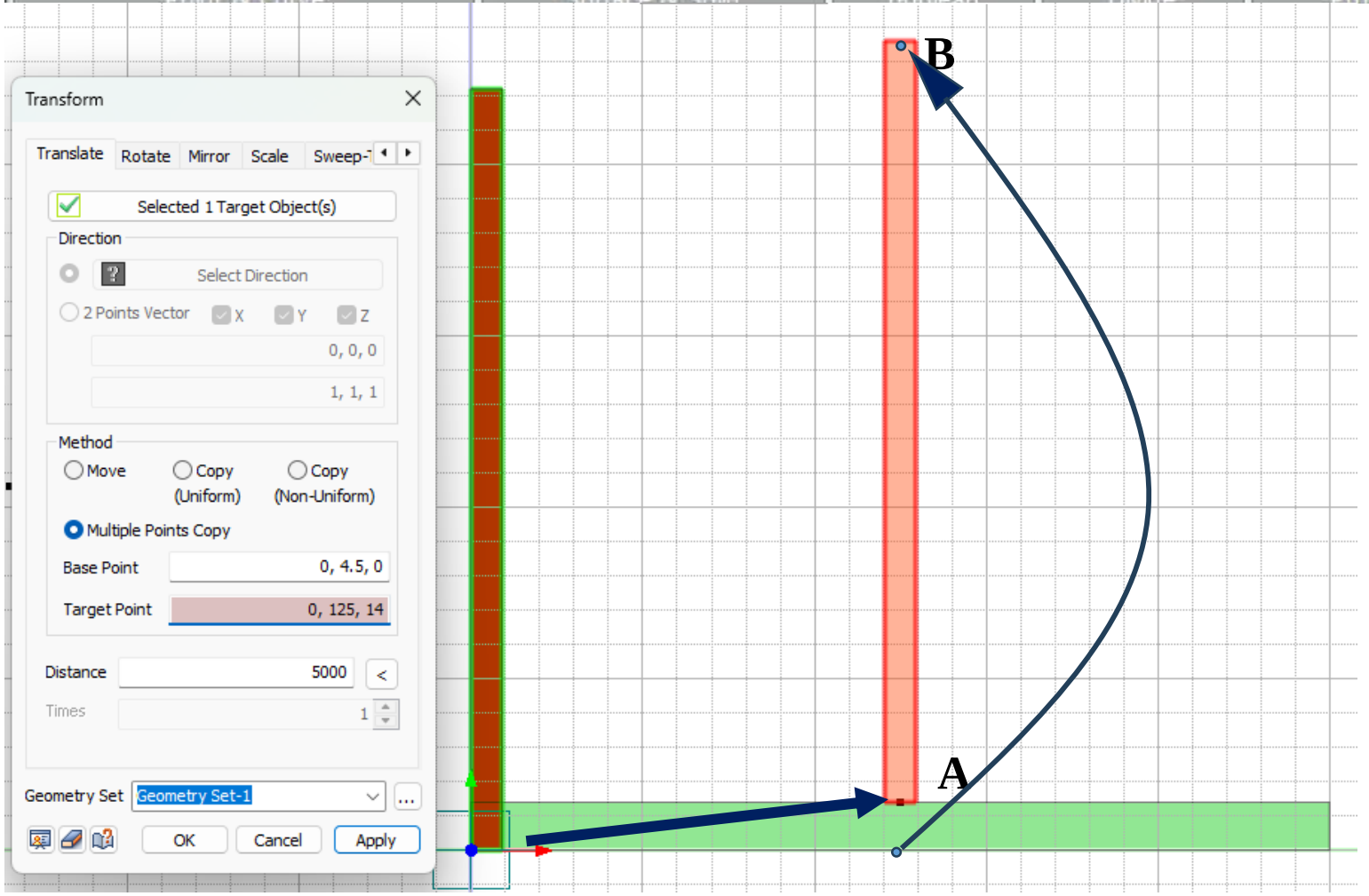
Following the section database for **RH section**

Create 2 rectangles | Axis Y,Z | Origin (0,0)

1. Green: 250,14
2. Red : 9,222

ADDITIONAL GEOMETRY (3D)

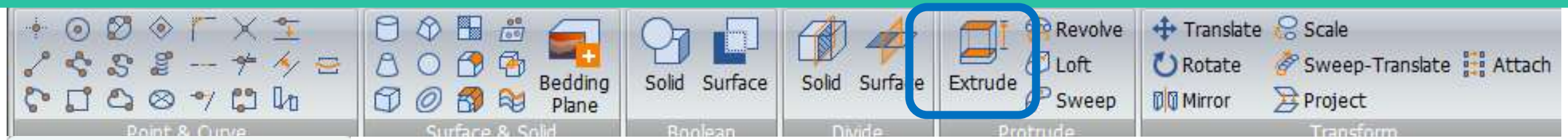


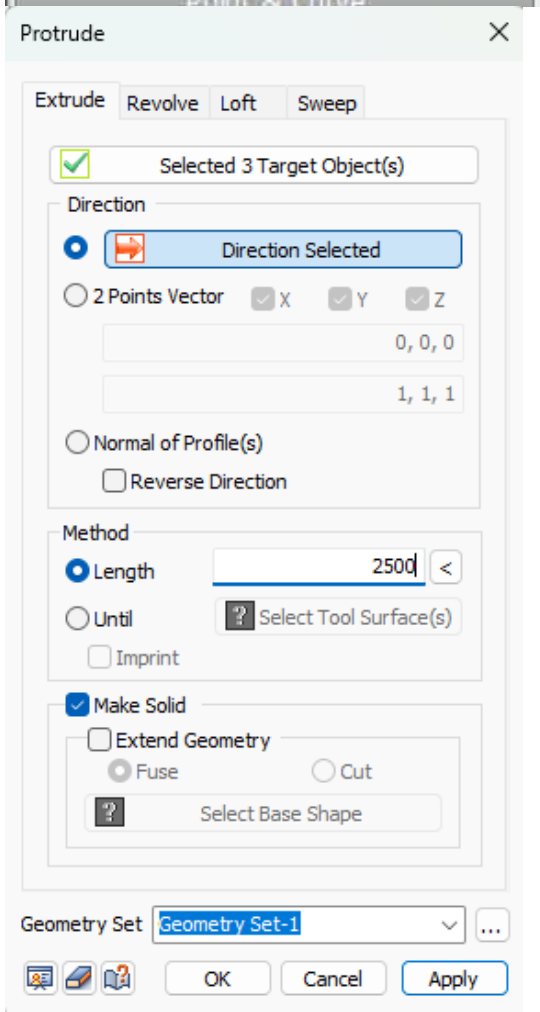


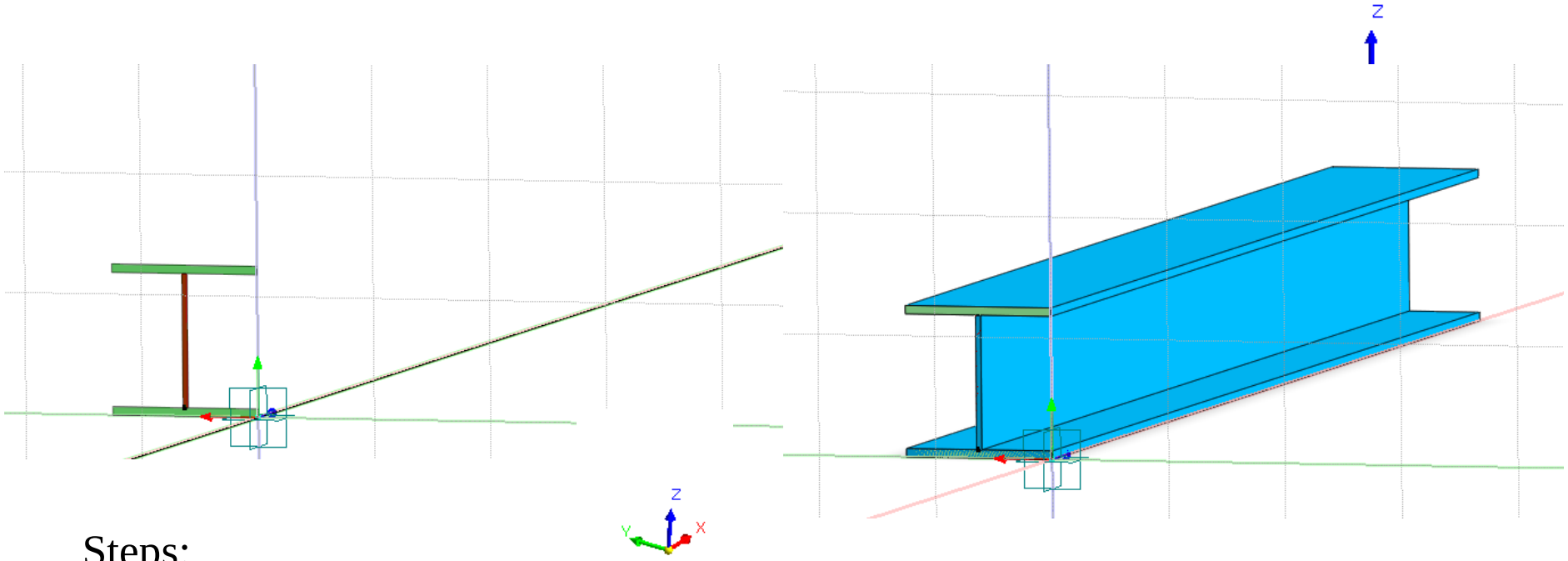
Steps

1. Using Translate function, move the red rectangle to point A.
2. Then copy the green rectangle to point B

ADDITIONAL GEOMETRY (3D)





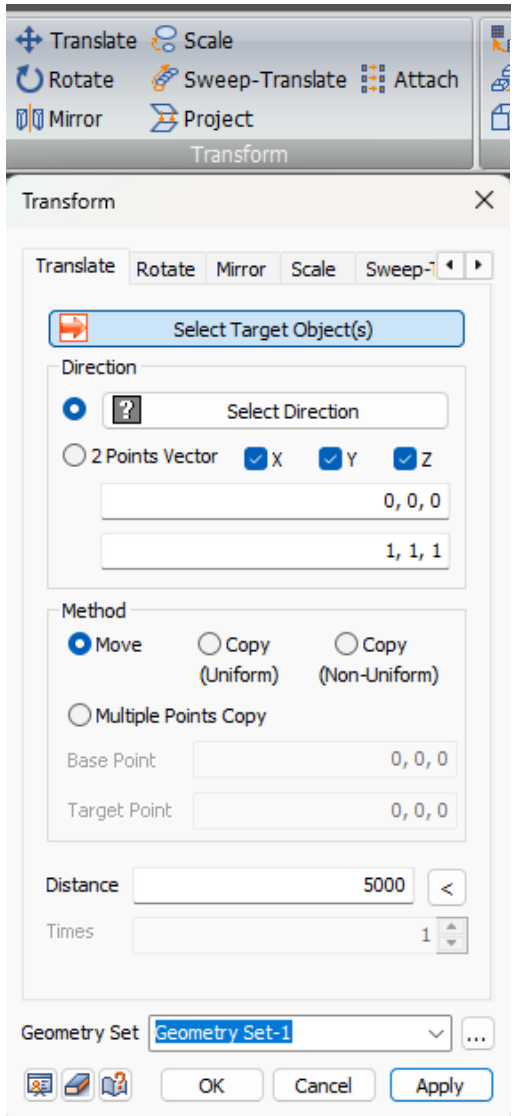


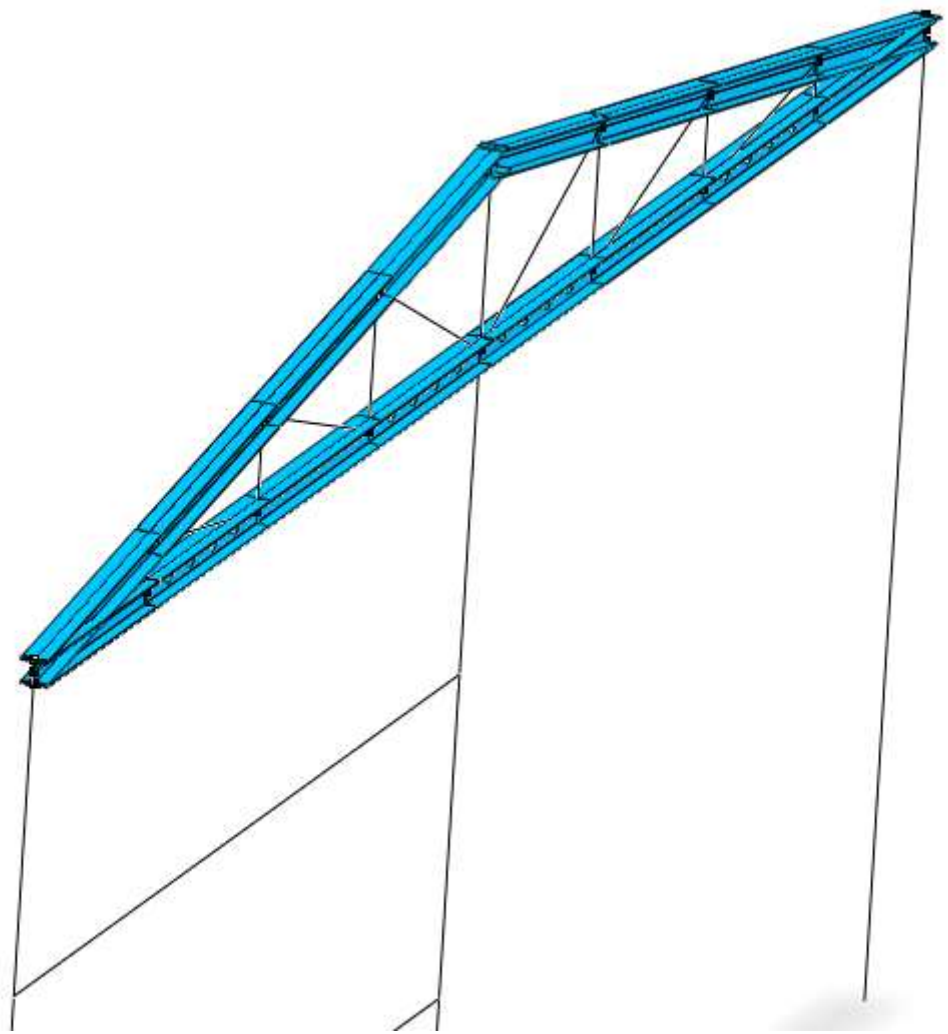
Steps:

Select all rectangles

1. Direction : X-axis
2. Length of beam 2.5m

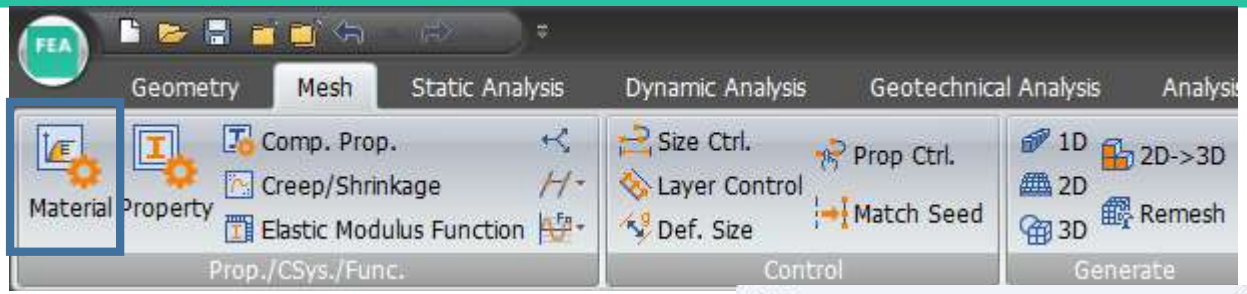
FRAME GUIDE

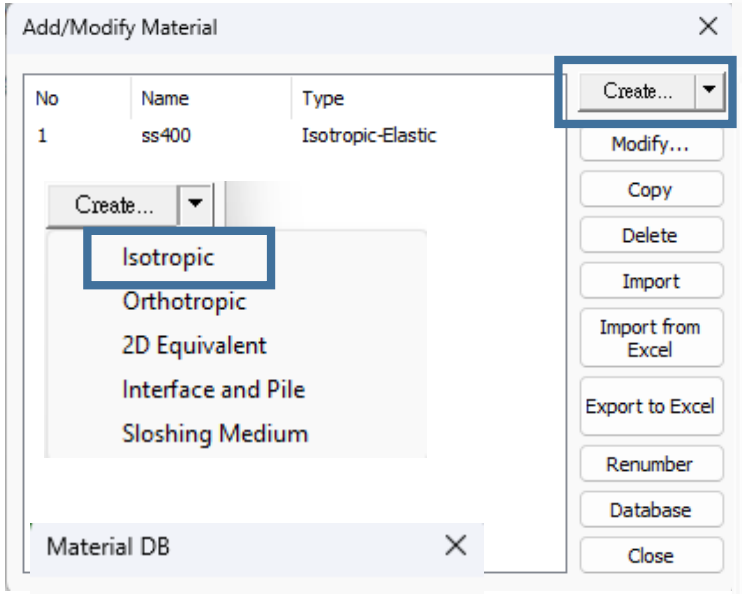


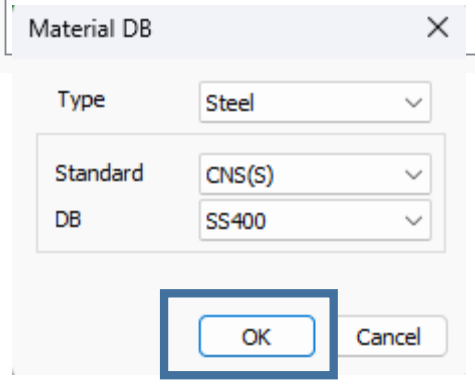


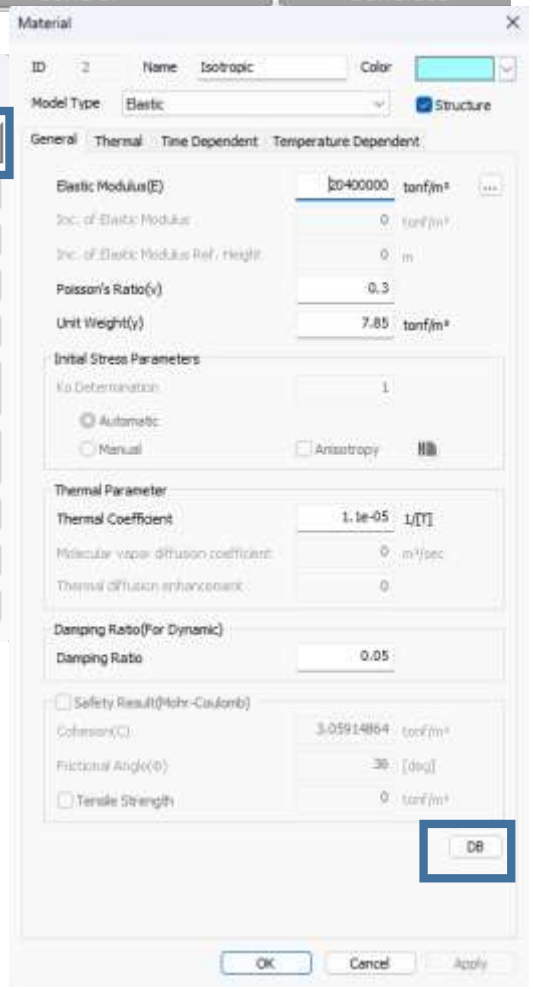
NOTE: Translate the 3D beam to the point where detailed analysis is to be done then reorient using rotate function.

MATERIALS AND PROPERTIES





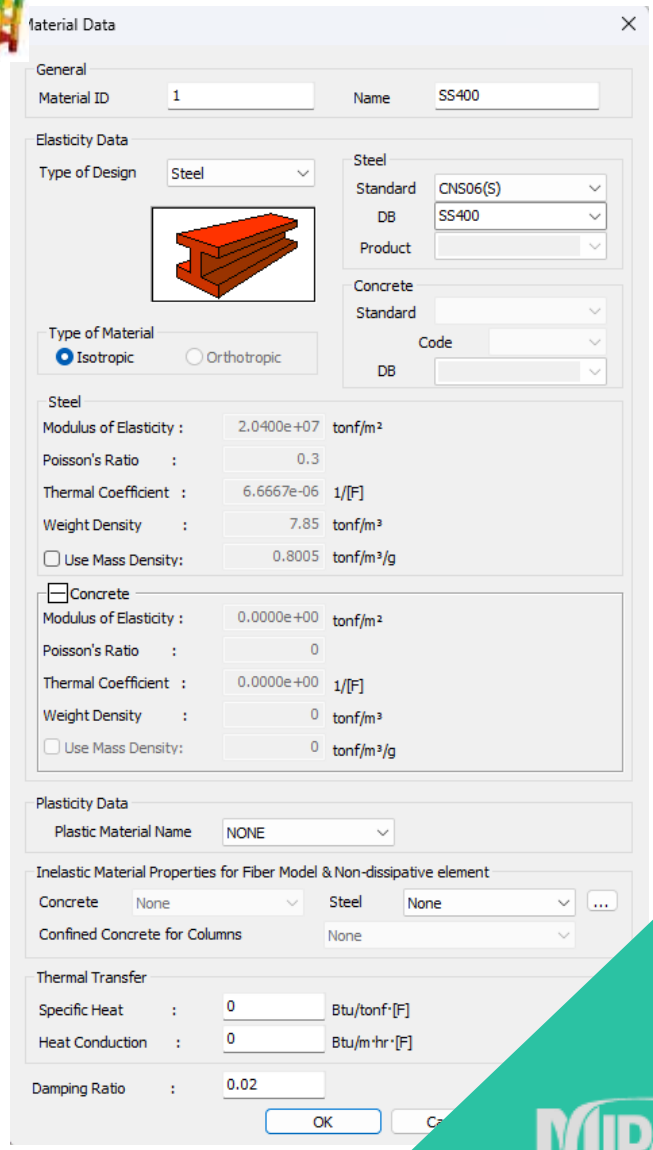




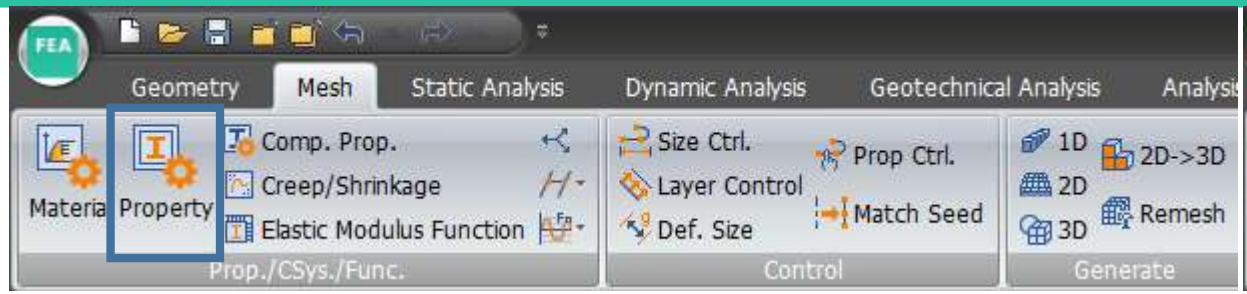
NOTE:

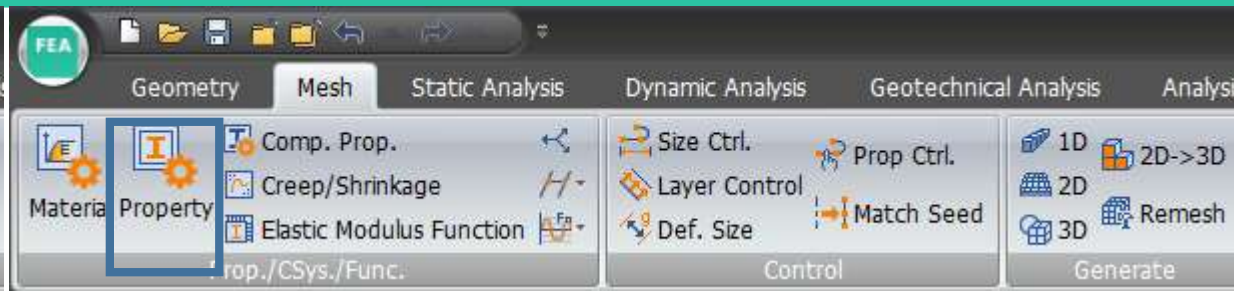
Material Parameters are modifiable.

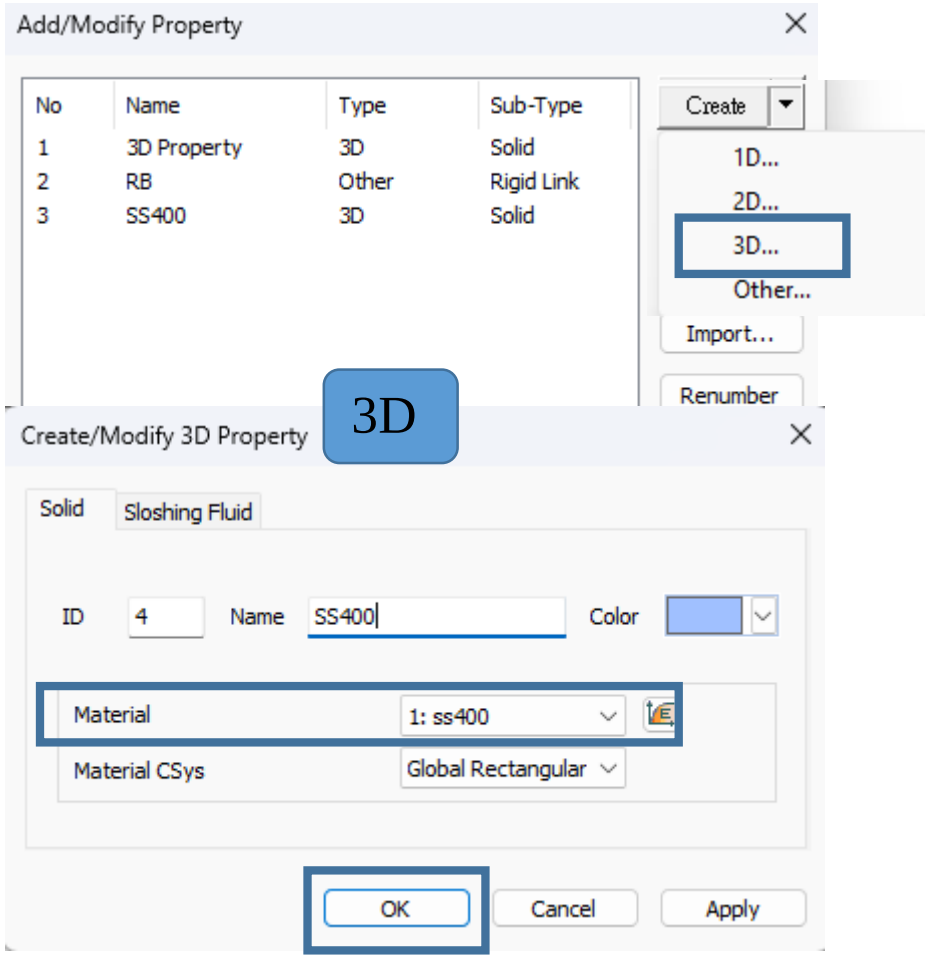
From **MIDAS GEN**

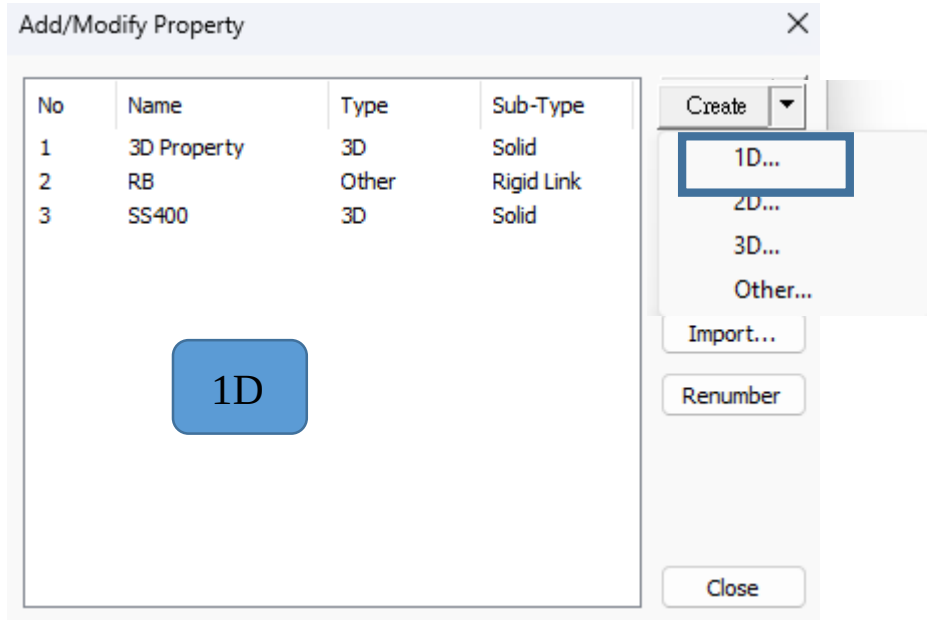


MATERIALS AND PROPERTIES

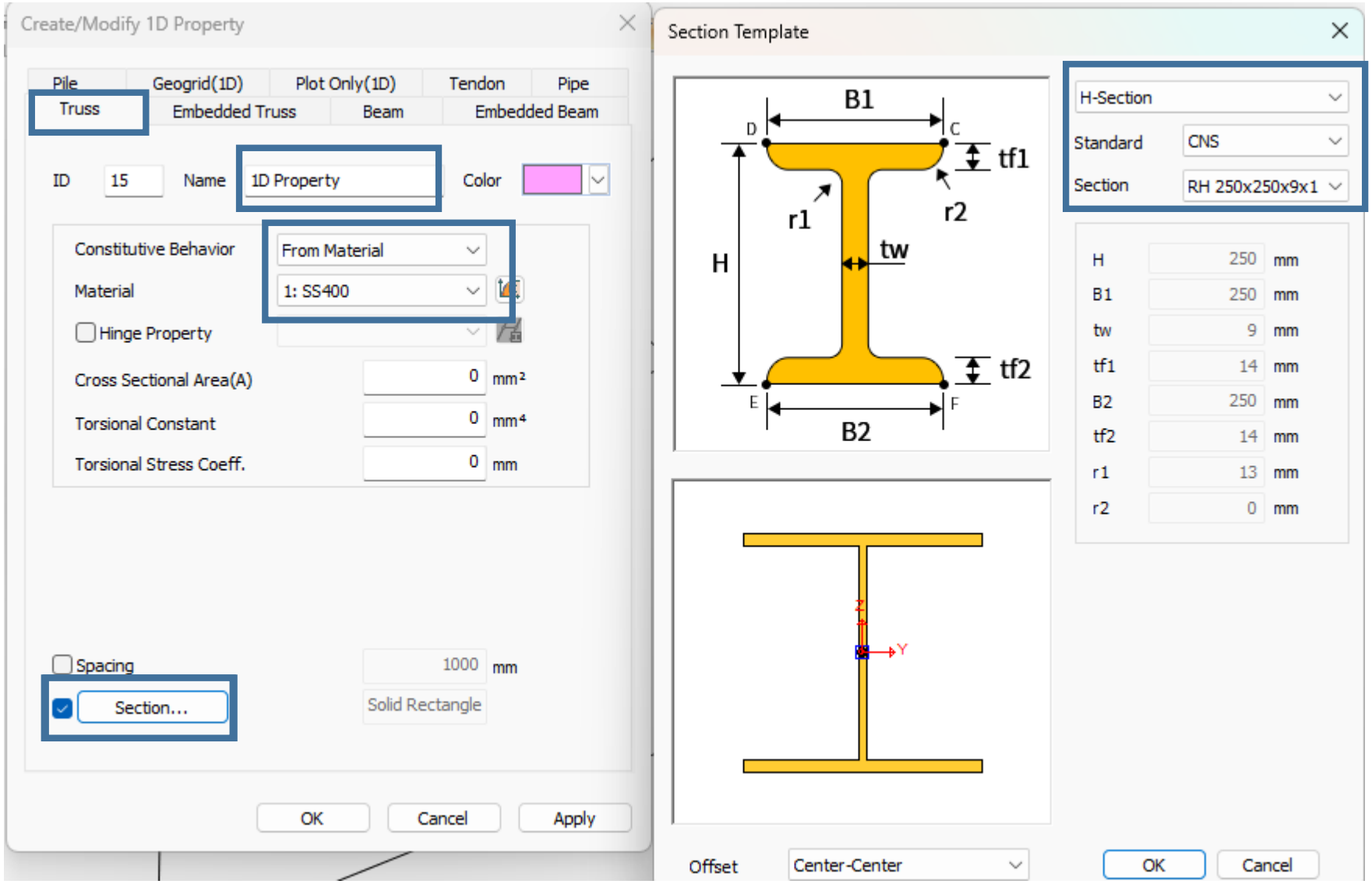








MATERIALS AND PROPERTIES



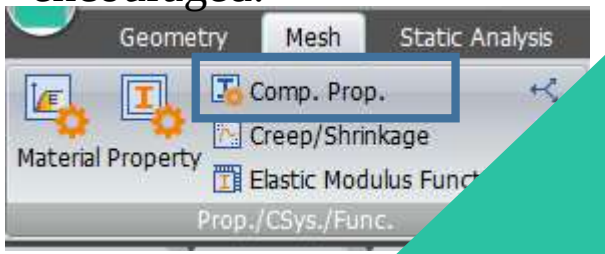
For 1D elements, the beam section database is recommended.

Take note of the following:

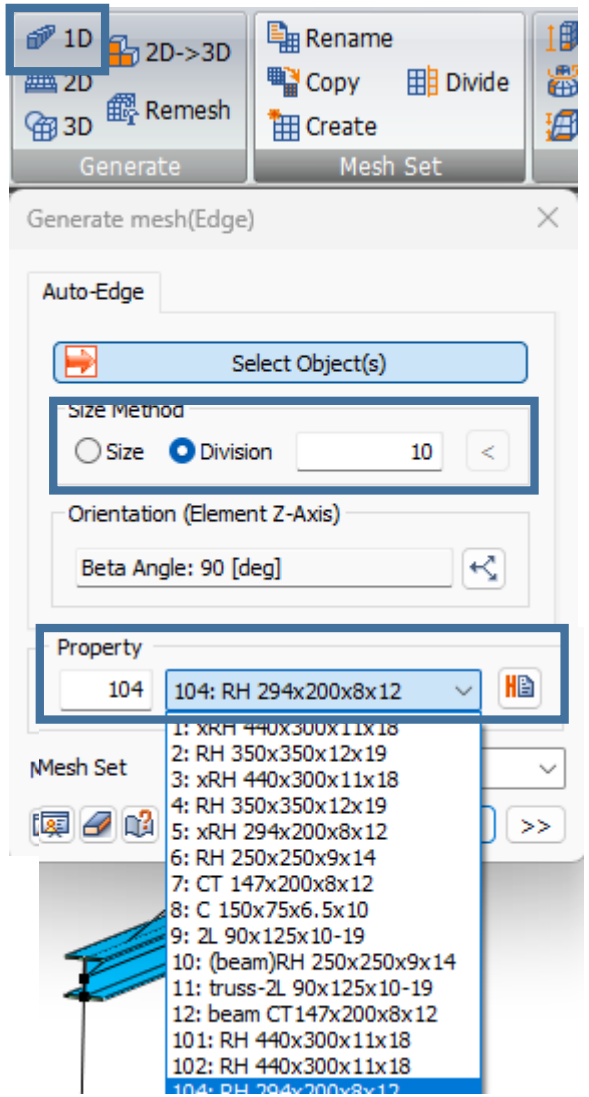
1. 1D Type
2. Material
3. Section
   - Standard
   - Section Shape

Additional Note:

If the section is not within the database, **Complex Section Function** is encouraged.



MESHING (1D and 3D)



For **1D elements**, define the appropriate section property.

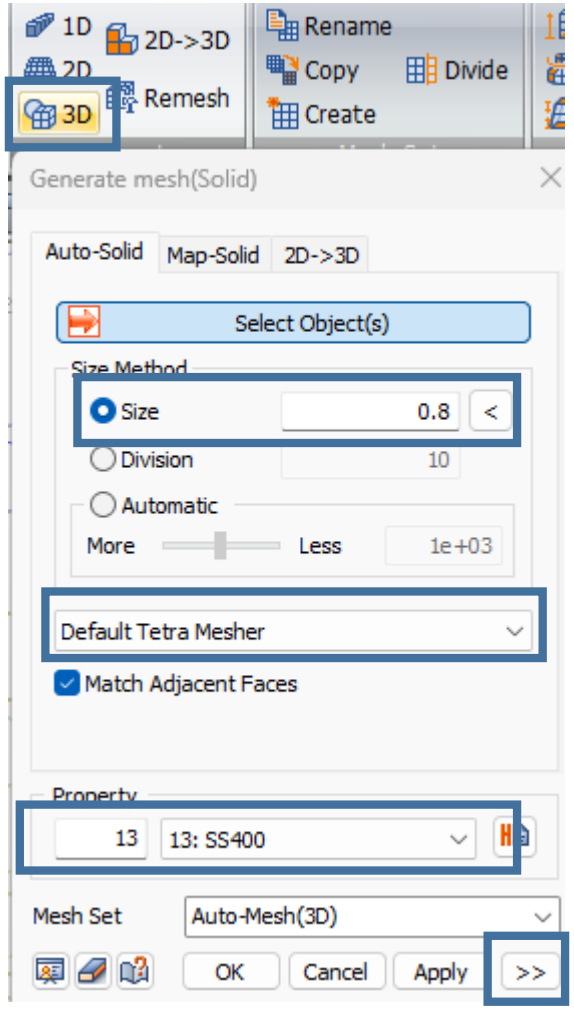
NOTE: Ensure that the geometry lines are divided into the correct intersections.

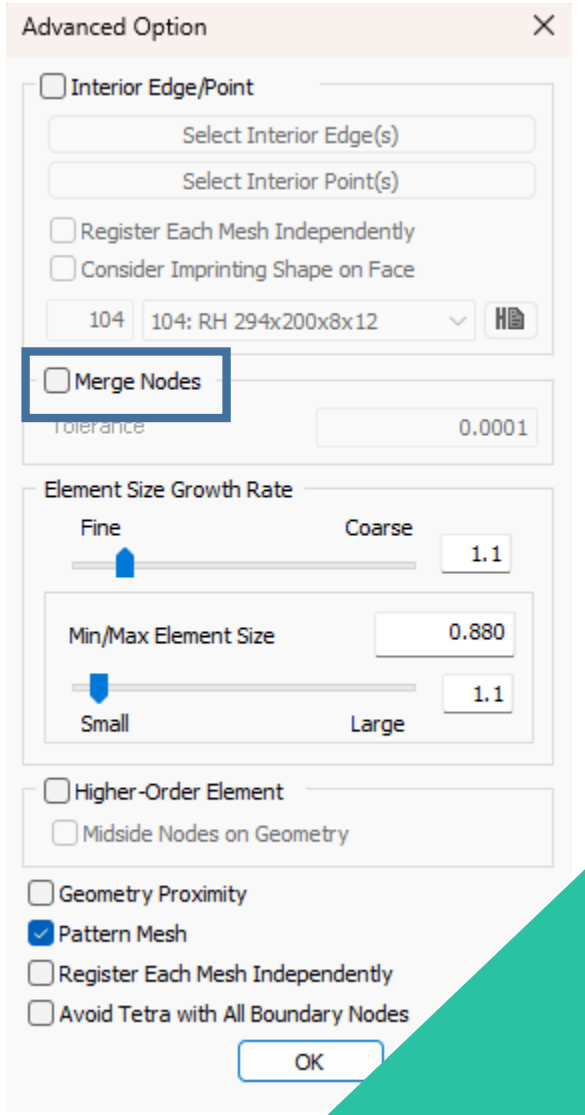
For **3D elements**, define the mesh size with the appropriate materials.

NOTE:

1. For future GEN export, use Tetra Mesher as mesh type

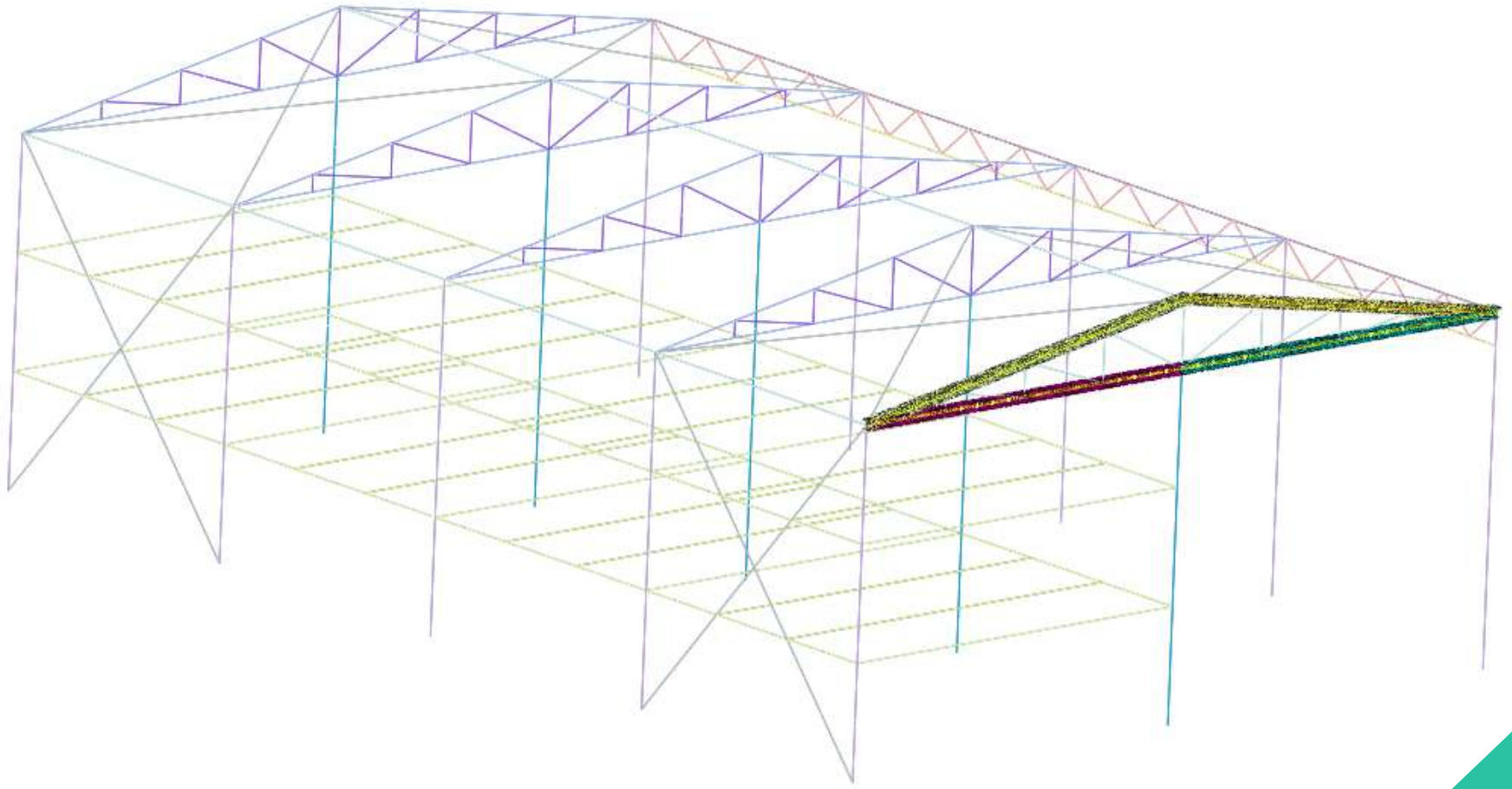
2. Deselect Merge nodes to create a segmented parts of the truss.



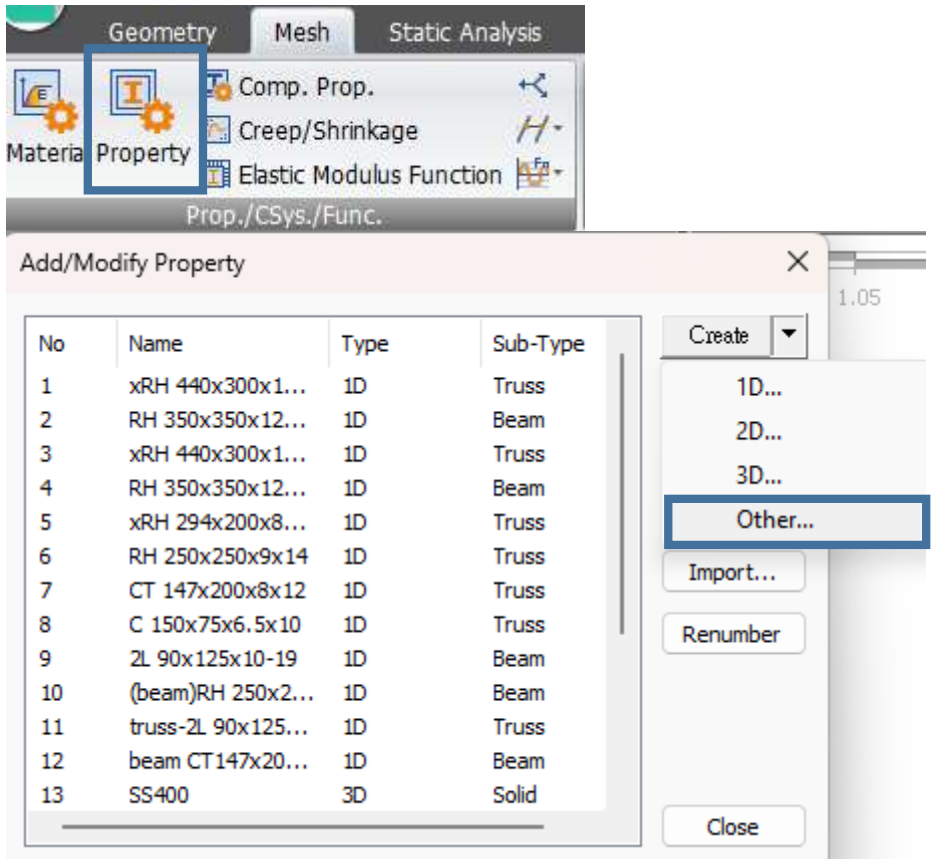


MODEL

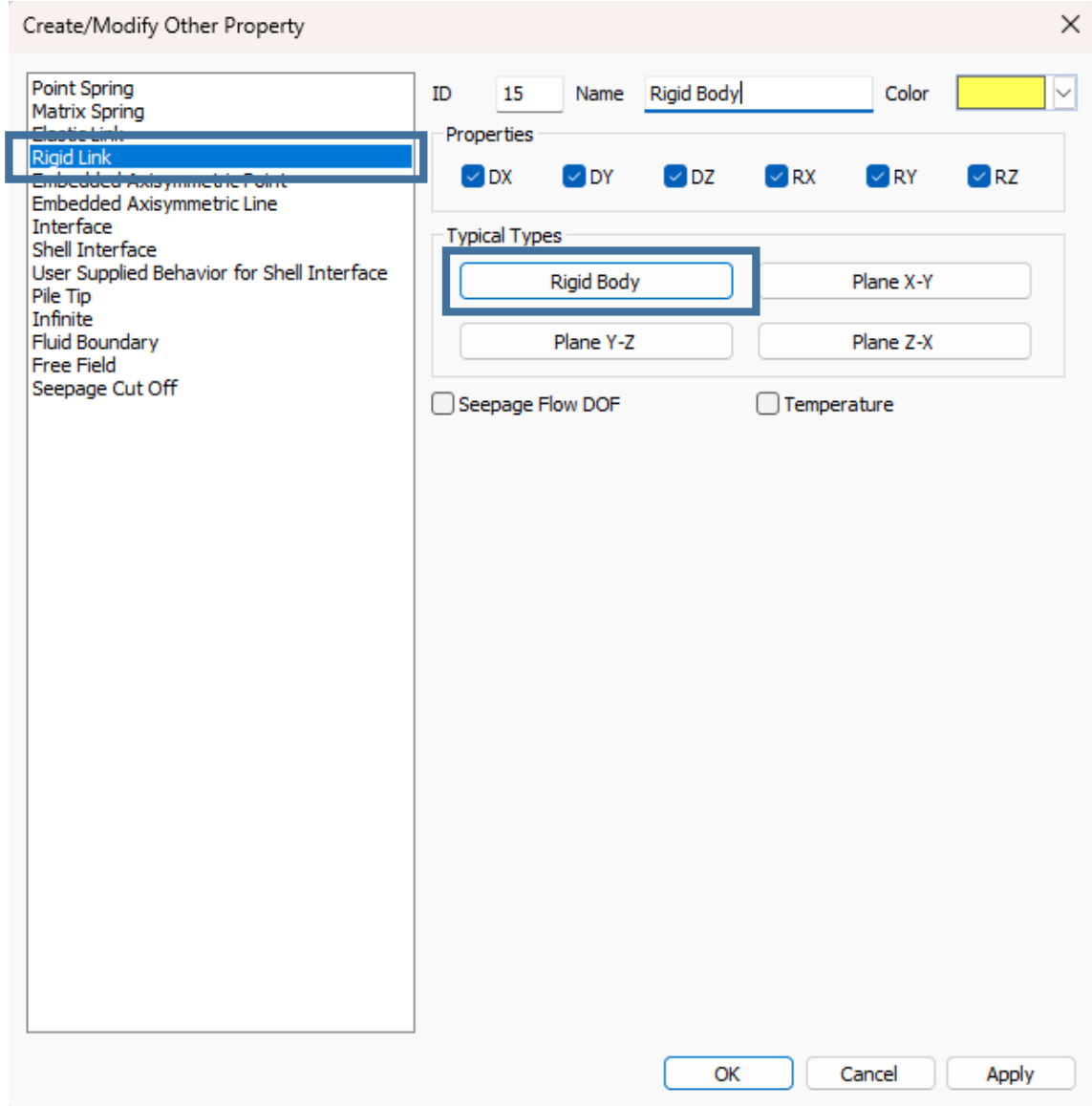
Item	ID	Color
modify (holes)	9	
Bedding Plane		
Export Shape		
Mesh Control		
Mesh		
Default Mesh Set	1	
main columns (outside)	63	
main column (middle)	64	
main girders	65	
slab girders	66	
roof girders	67	
truss chords	68	
side truss girder top	69	
side truss web	70	
main truss web	71	
brace	72	
slab girders 1D	75	
side truss girder bot	76	
(extract)side truss girder t.	77	
3D Truss 1	172	
3D Truss 2	173	
3D Truss web	175	
3D Truss 3	199	
3D Truss 4	200	



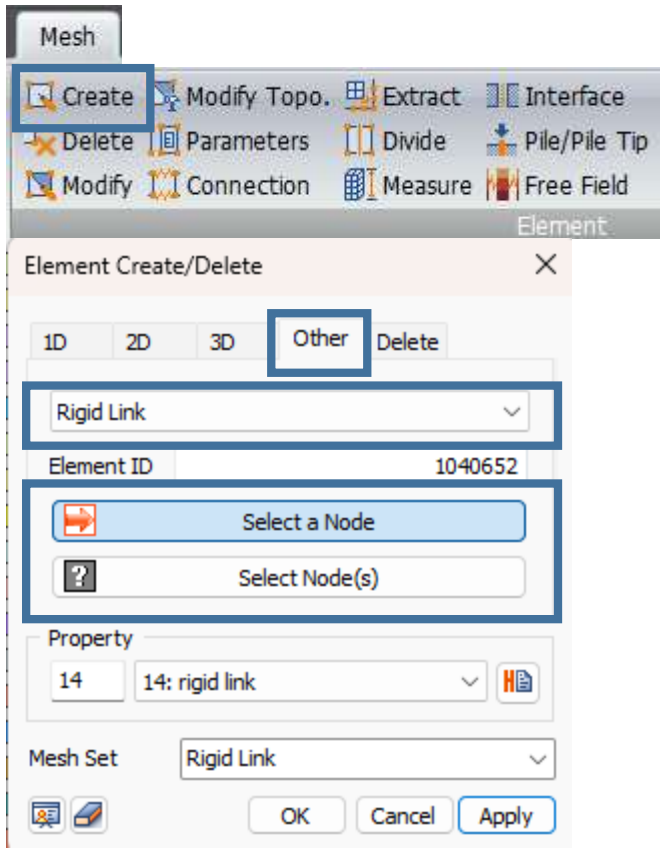
RIGID LINKS

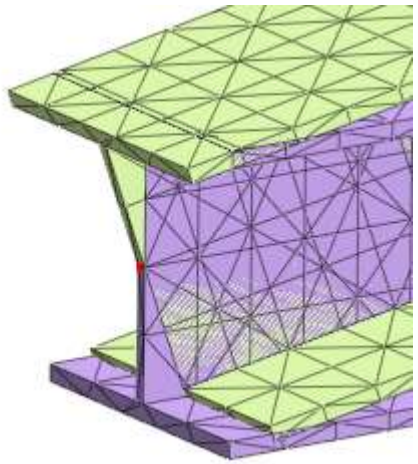


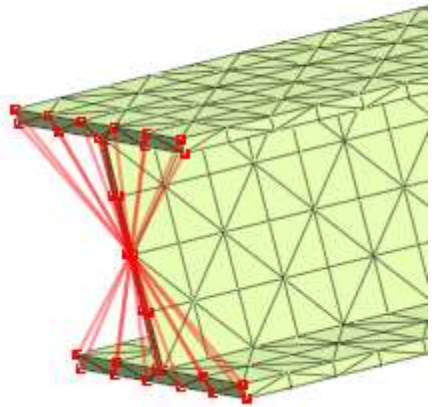
Define a rigid body in the property to create a rigid link.
NOTE: No material definition needed for rigid body.

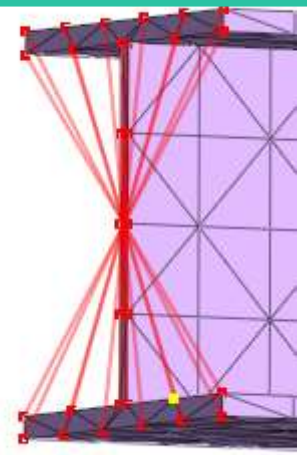


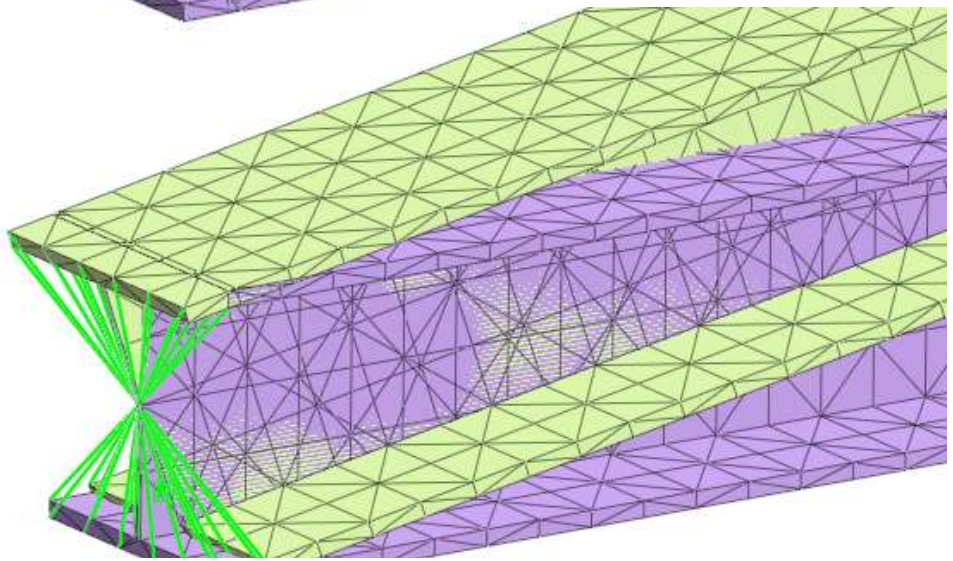
RIGID LINKS



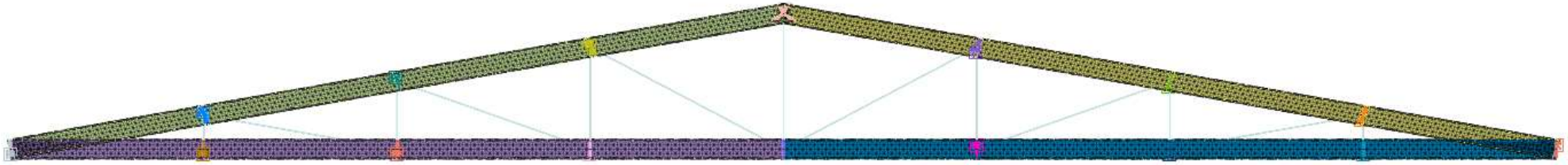




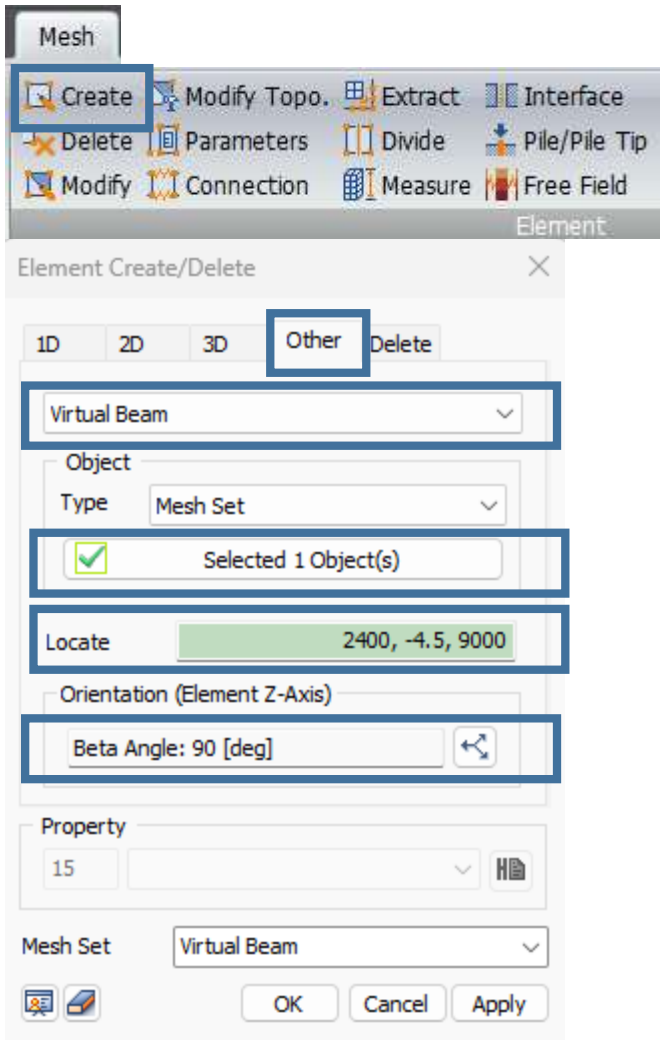


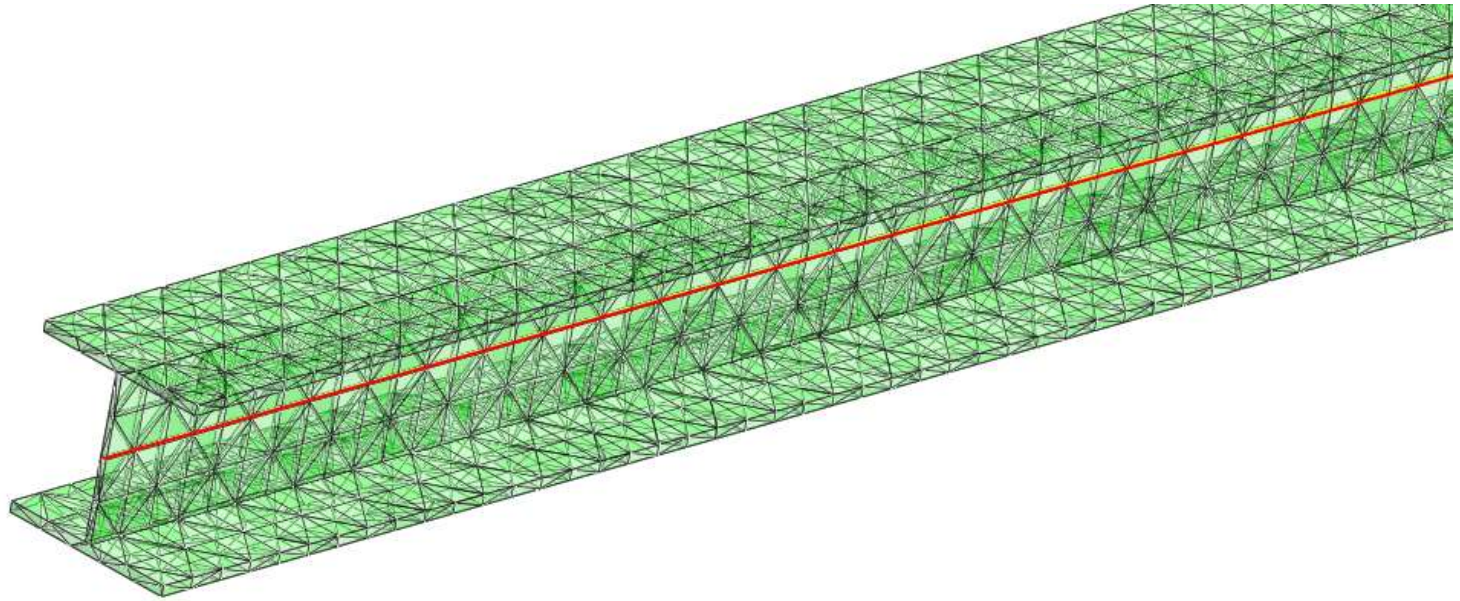


NOTE: Define rigid link to all joints connected to the 3D element



VIRTUAL BEAM

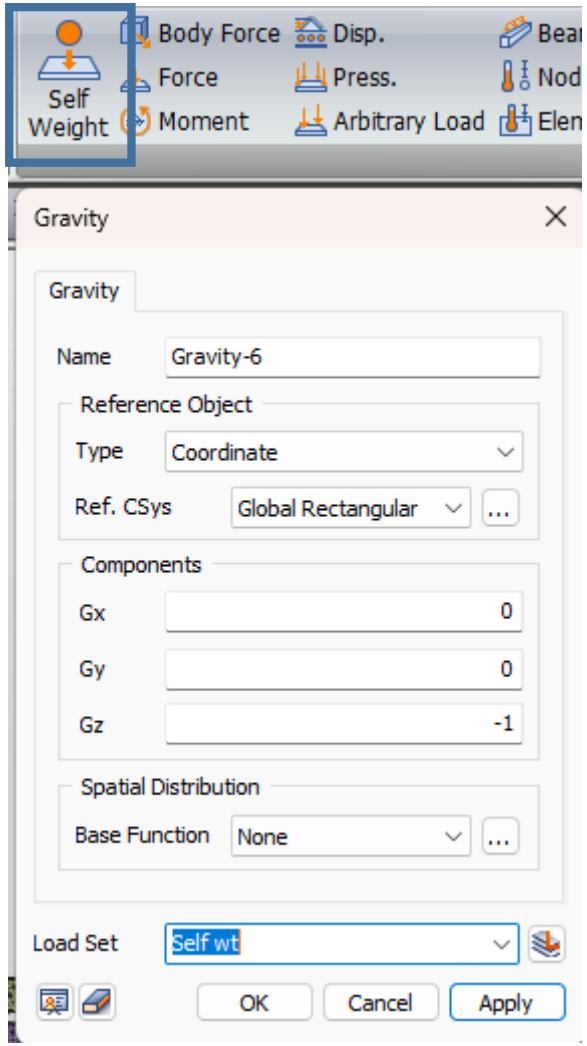


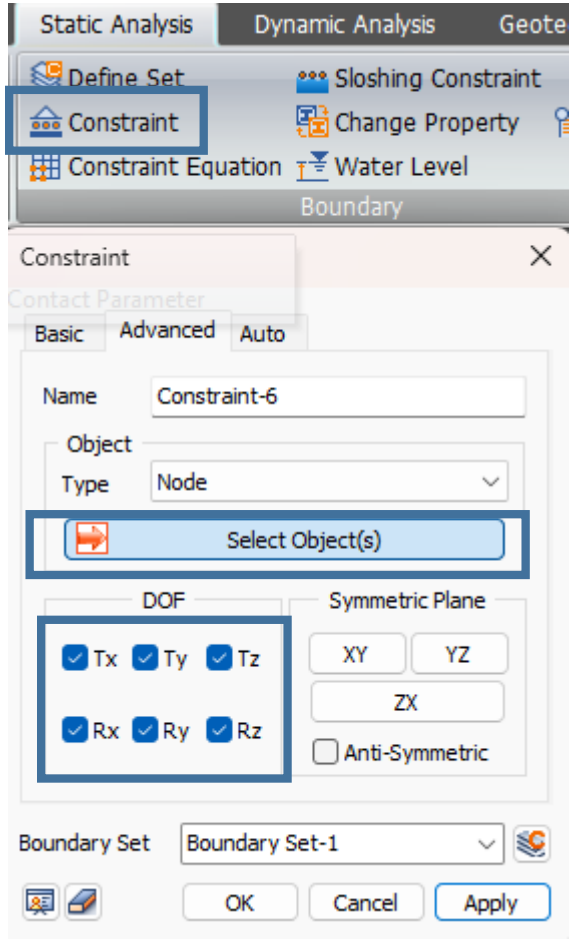


STEPS:

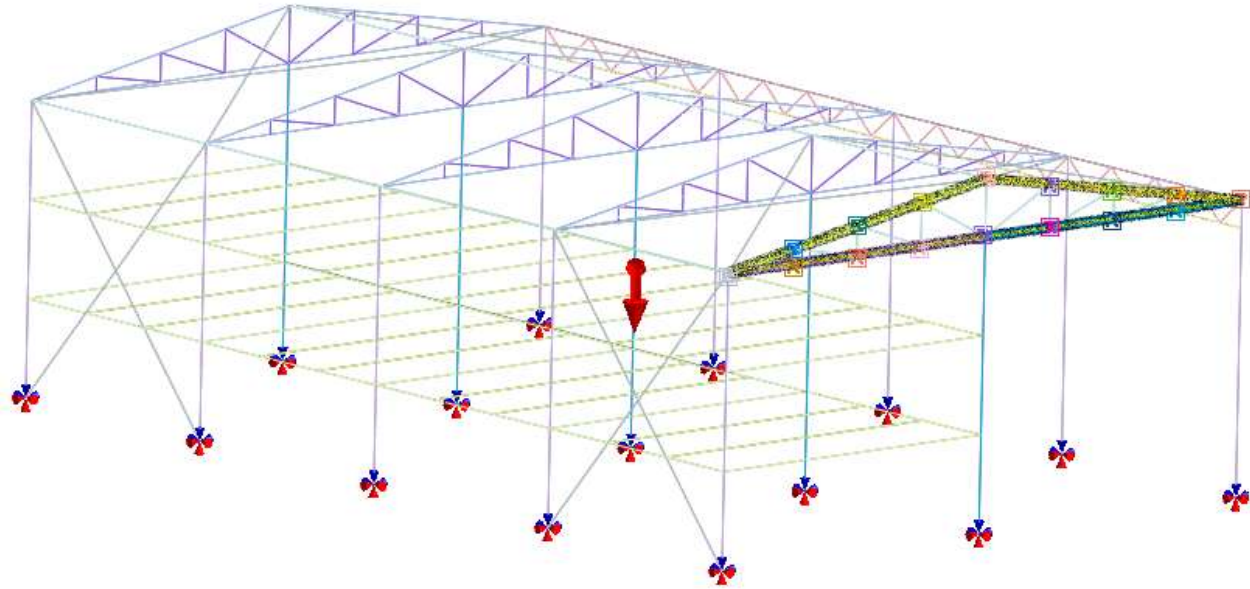
1. Select mesh set.
2. Create a line along the geometric center of the mesh.
3. Choose the proper orentation.

LOADS and CONSTRAINTS





NOTE: Set constraint on the column to be fixed

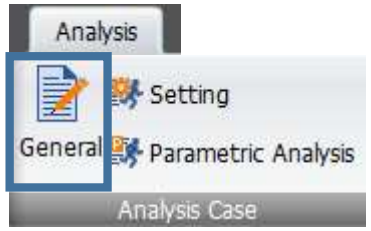


NOTE:

1. Initially, set to simplest load case (self weight)

2. Set constraint on the column base to be fixed (DOF: Txyz, Rxyz)

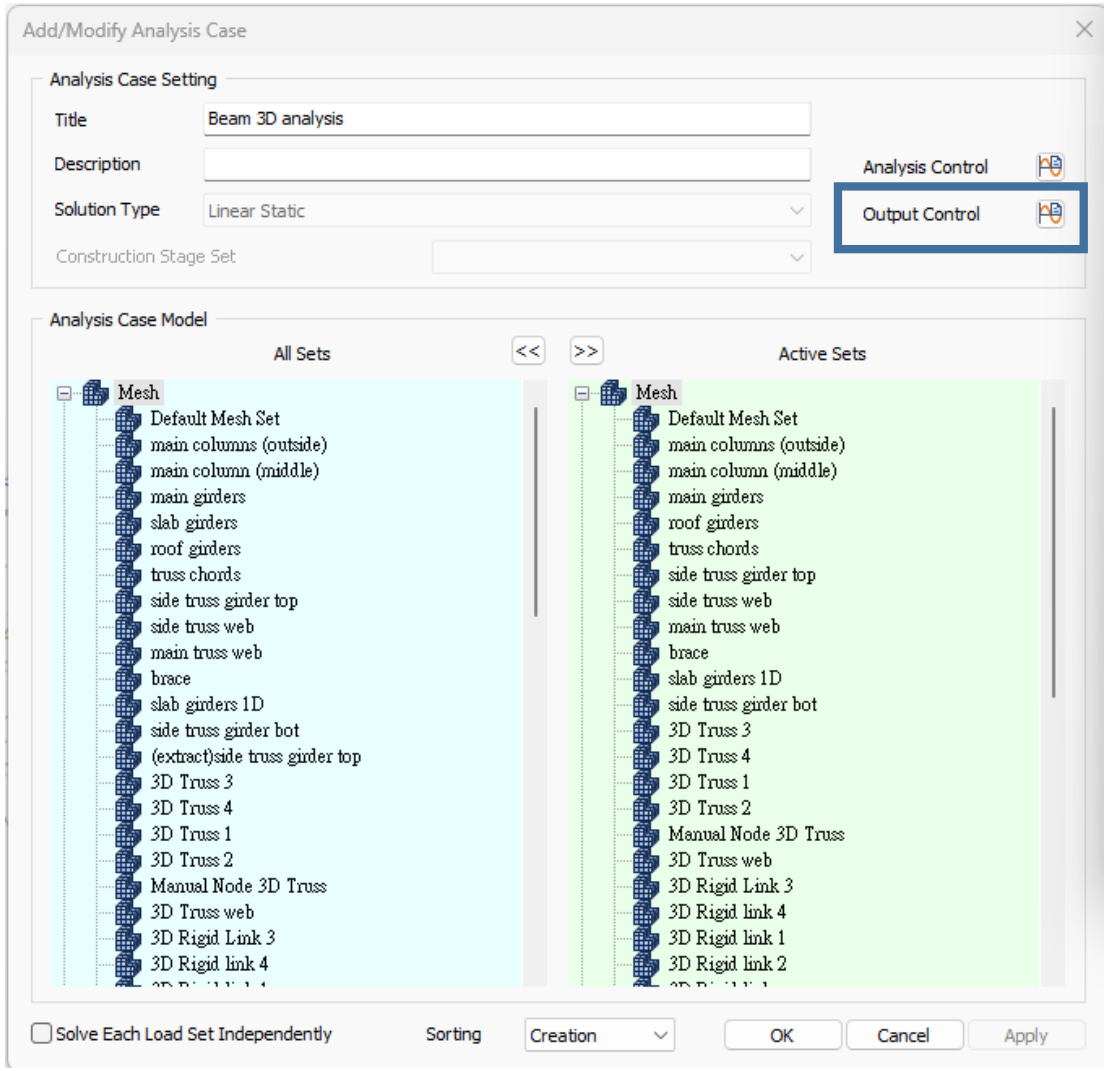
ANALYSIS CASE

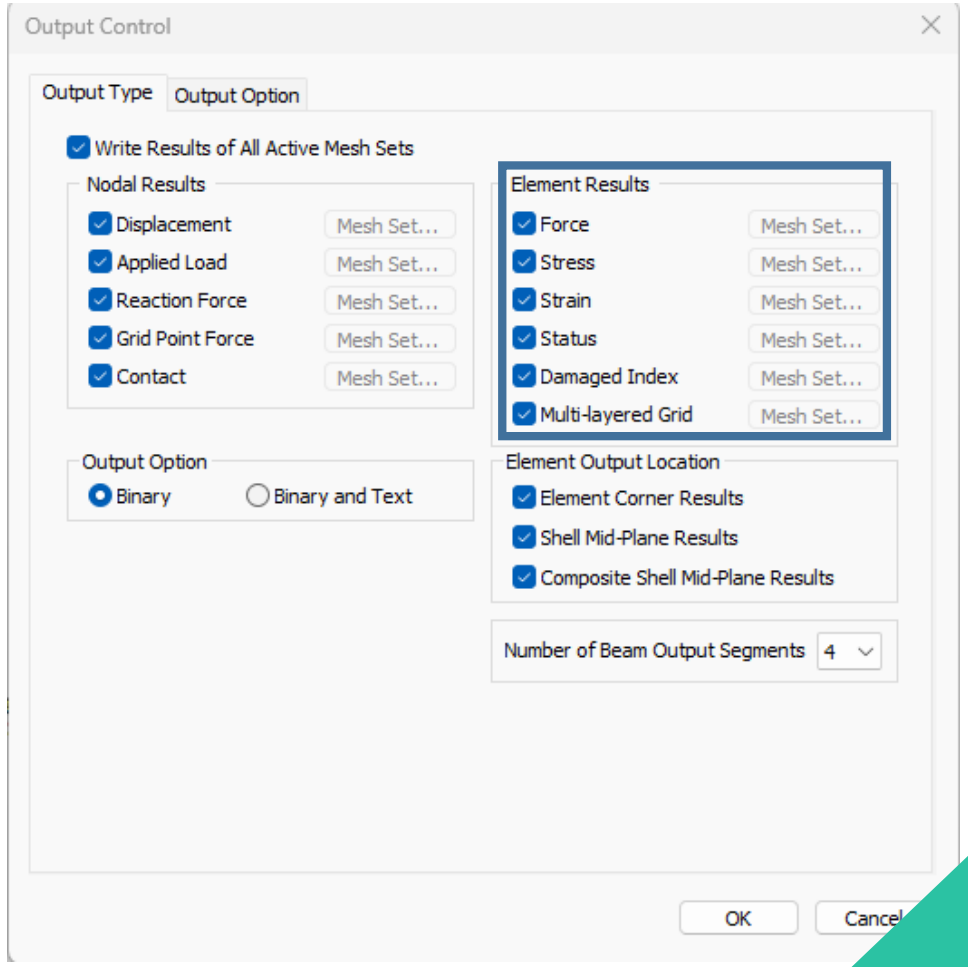


NOTE:

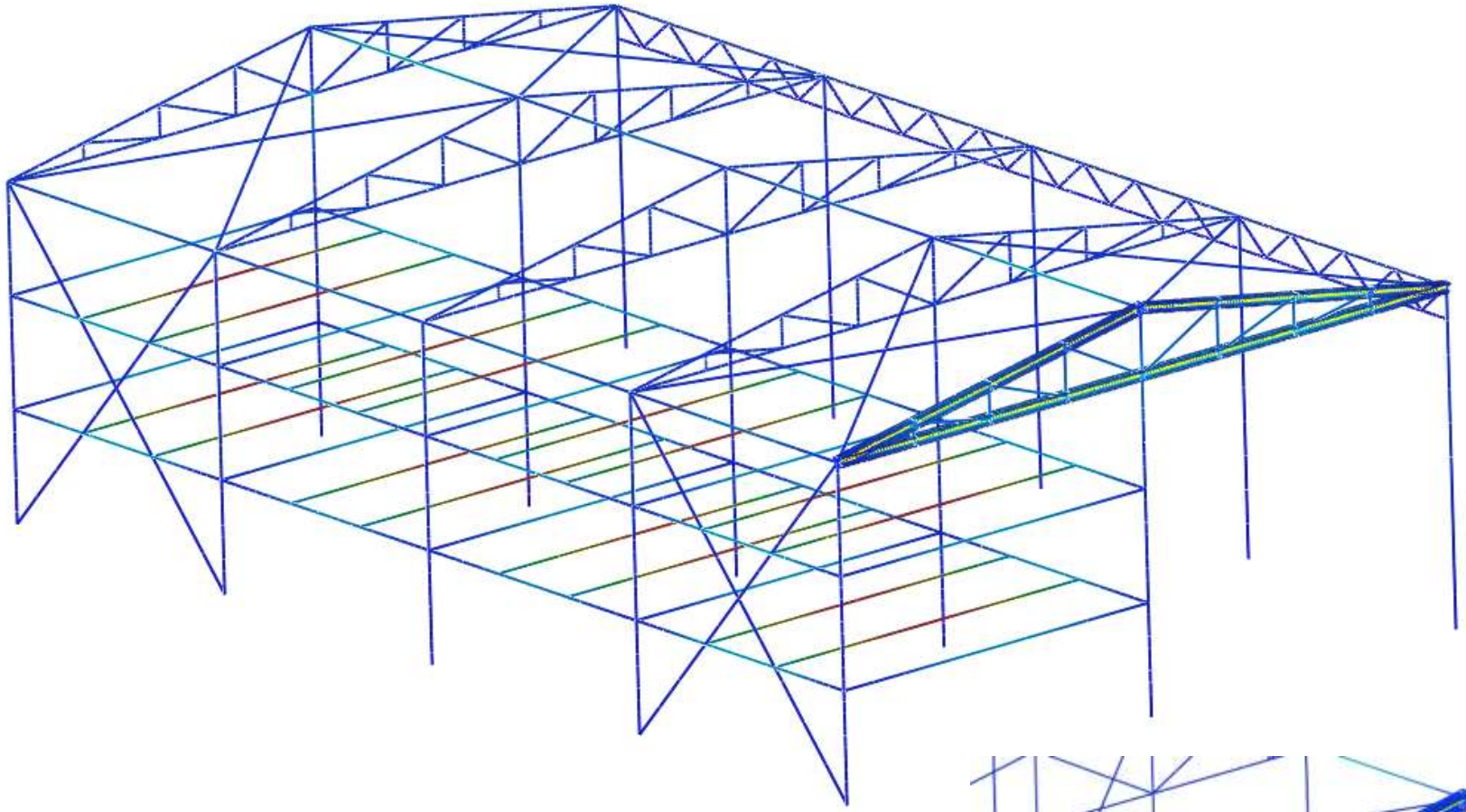
1. Include all the necessary meshes, constraints, and loads.

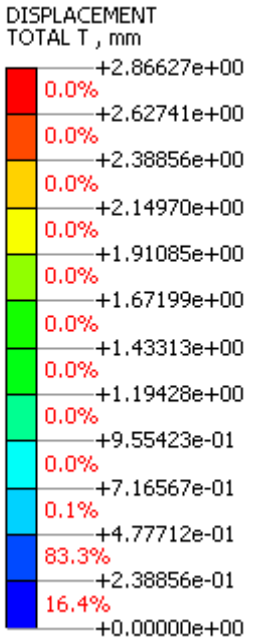
2. In the output control, include in the output the strain results since the analysis includes a 3D element.

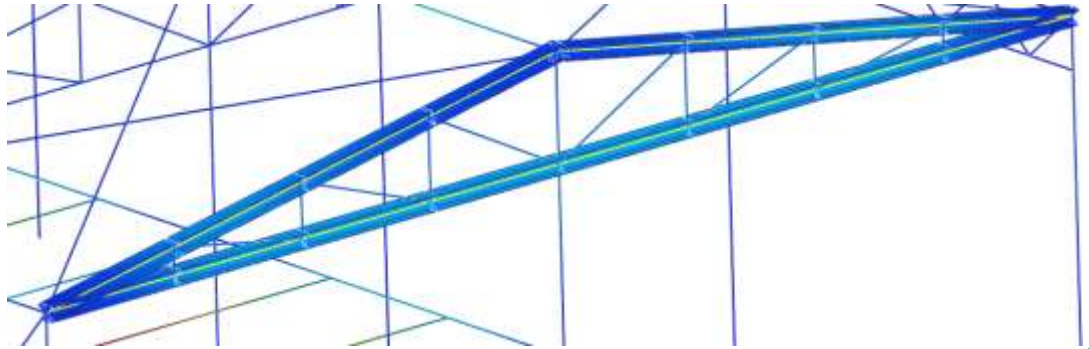




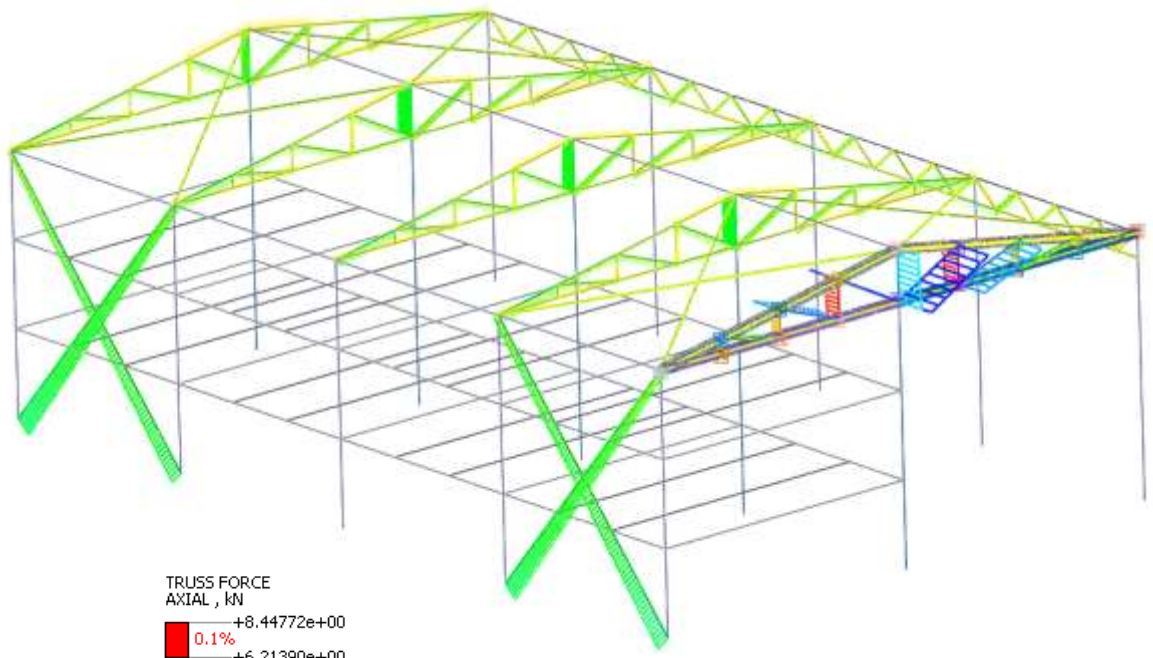
RESULTS - DISPLACEMENT



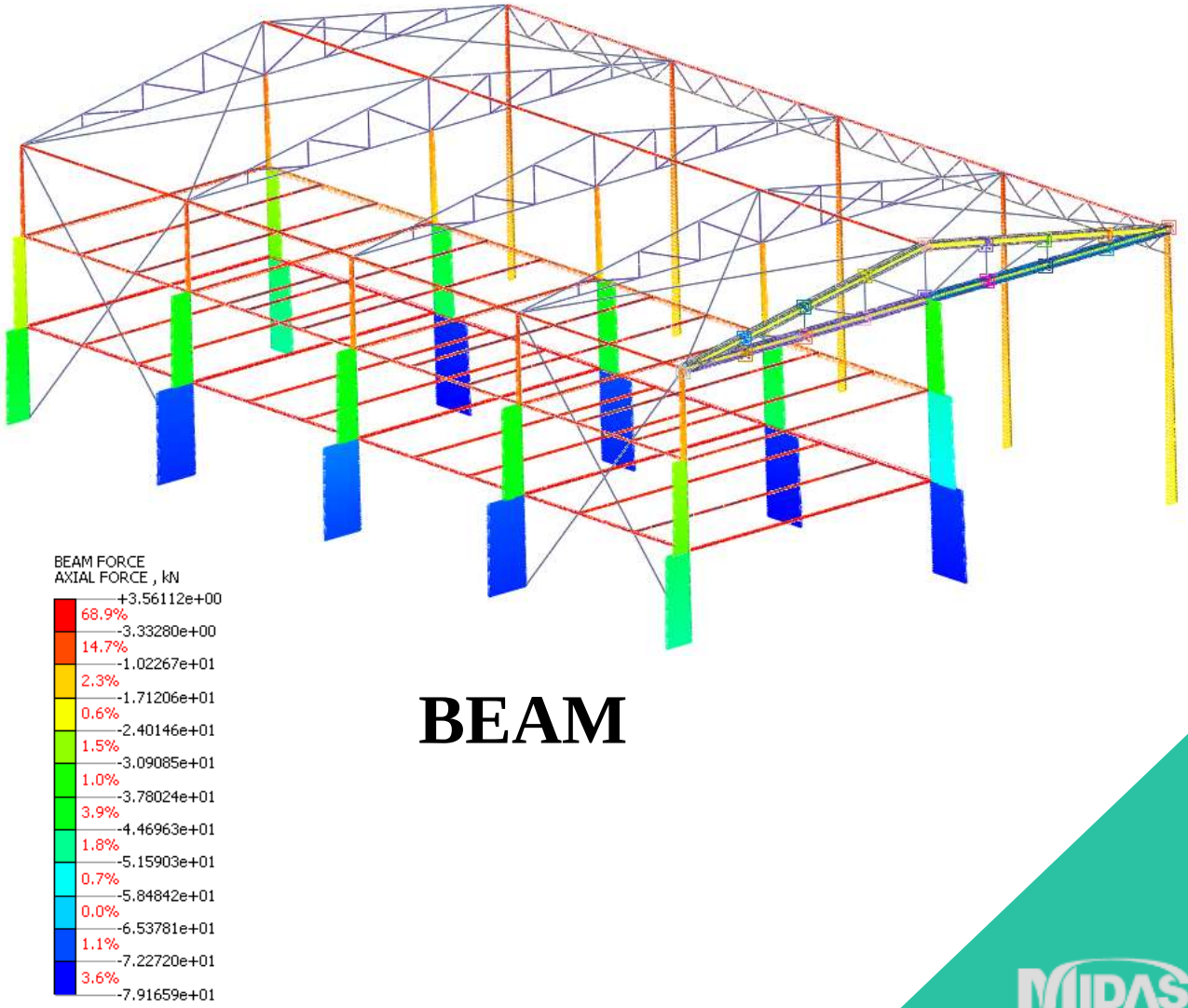




RESULTS - 1D AXIAL FORCES

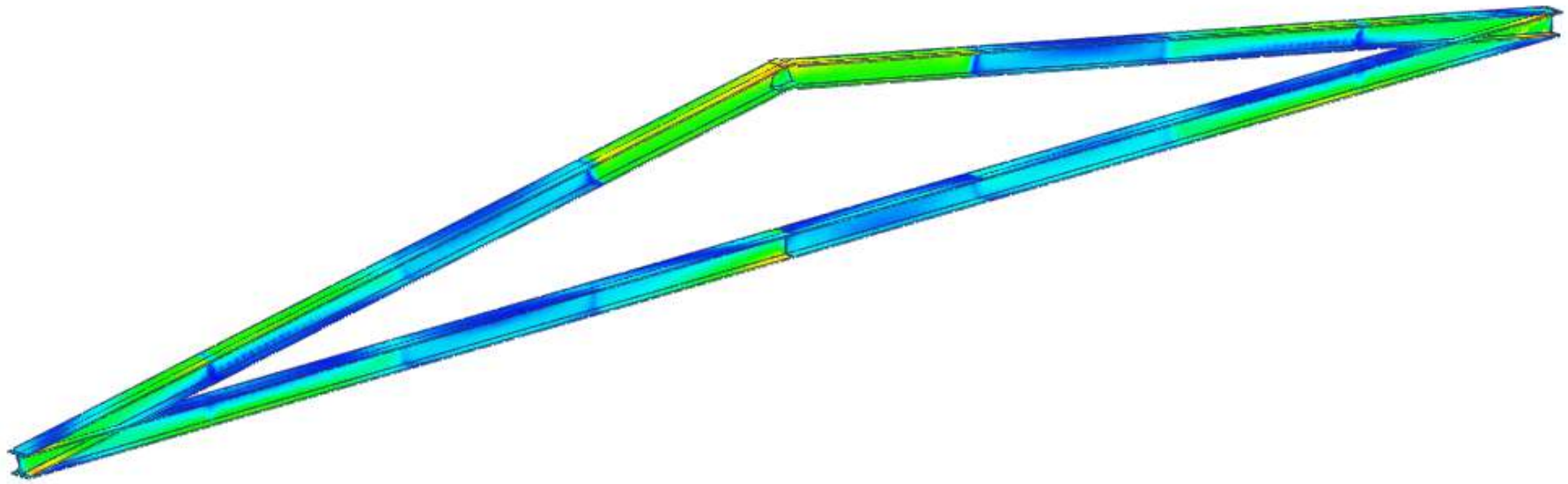


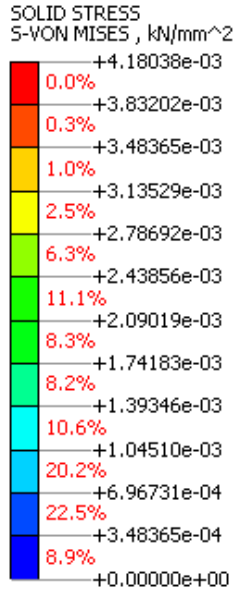
TRUSS



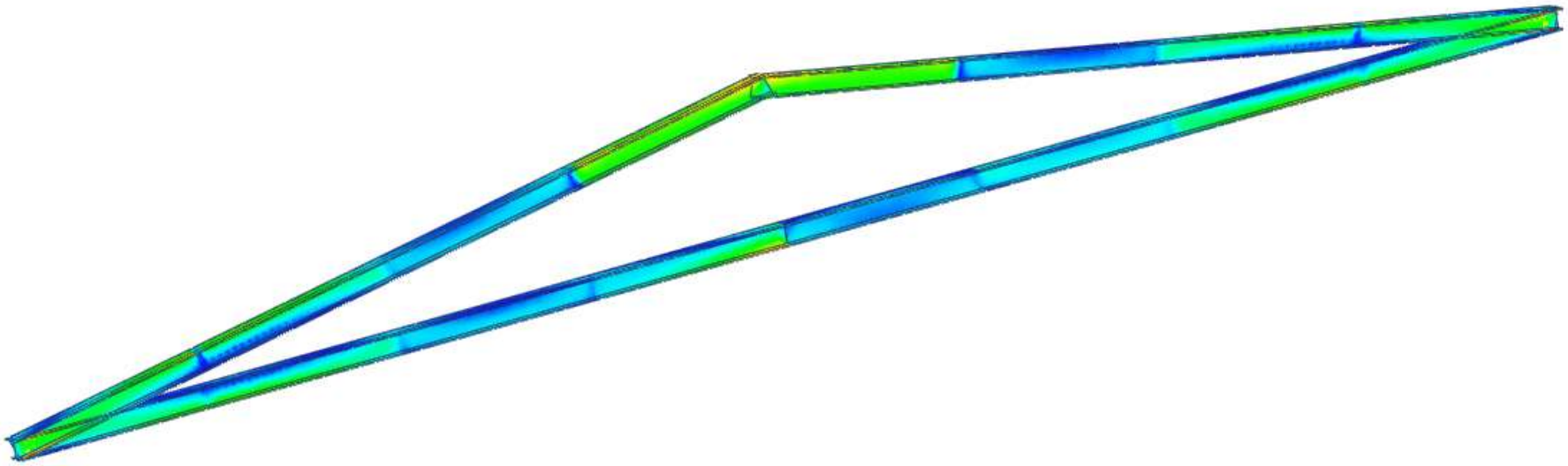
BEAM

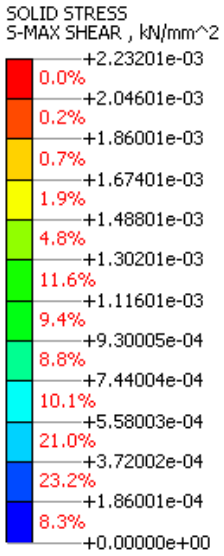
RESULTS - 3D STRESS VON MISES





STRESS
Von Mises

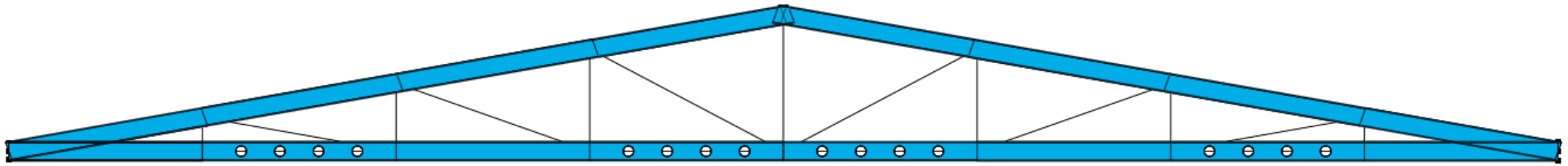




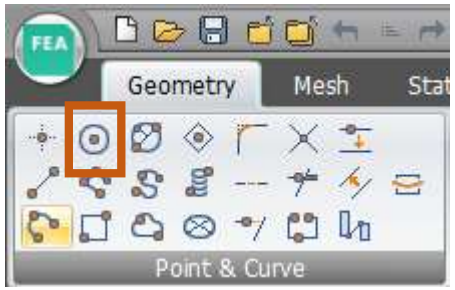
STRESS
Max Shear

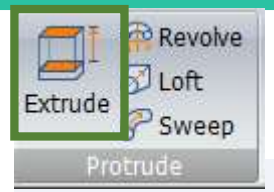
Part 2

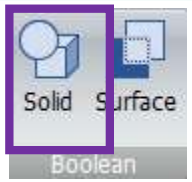
Modified Frame Detailed Analysis

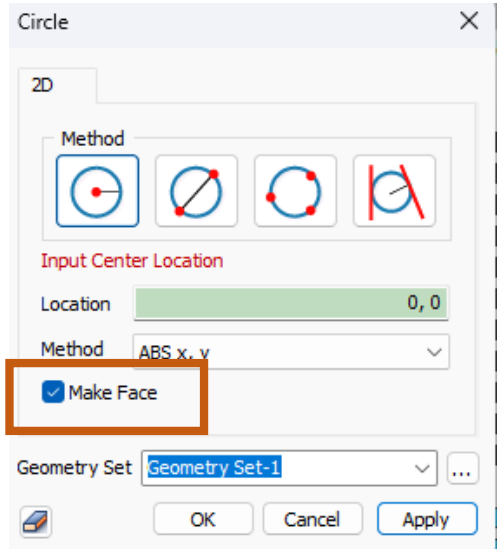


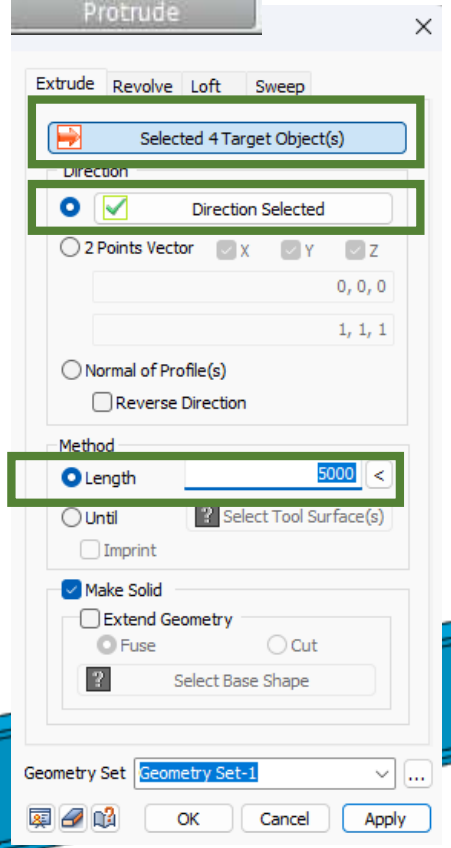
GEOMETRY

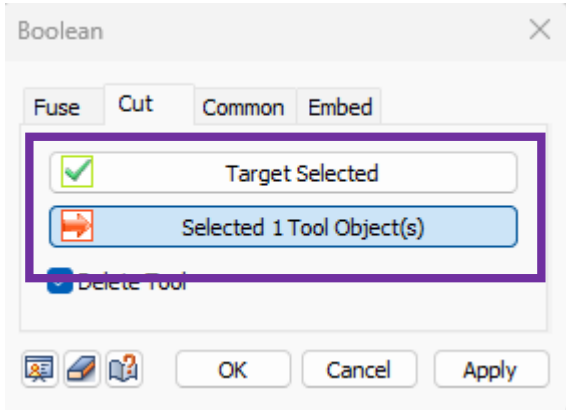


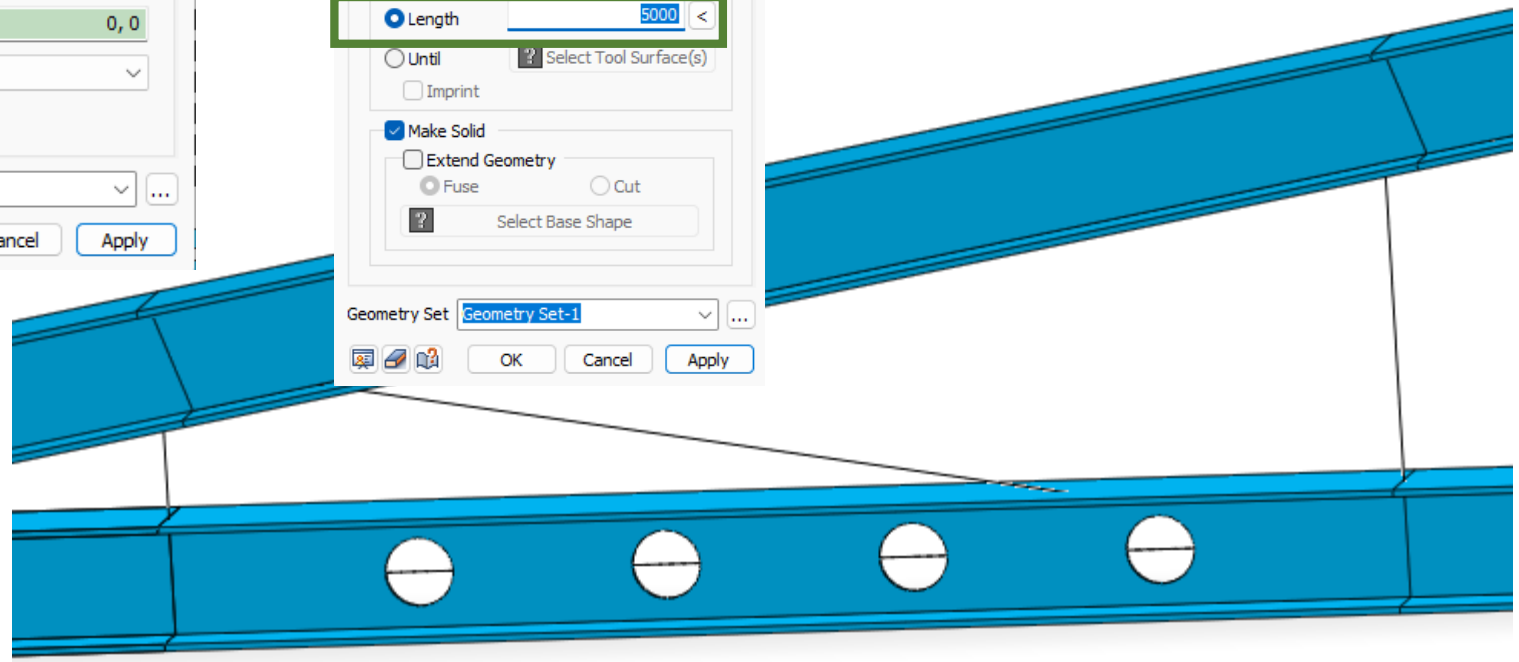










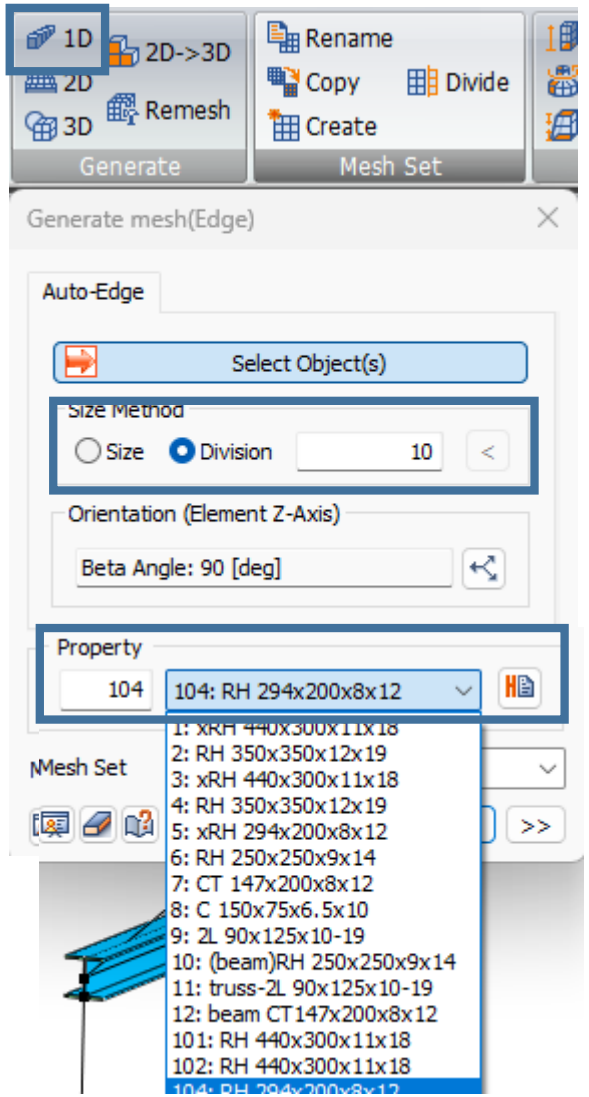


MODIFYING BEAM ELEMENT

(Sample Case: There are holes in the beam)

1. Create a circle face
2. Extrude to make a cylinder
3. Cut Solid
4. Repeat to all concerned elements

MESHING (1D and 3D)



For **1D elements**, define the appropriate section property.

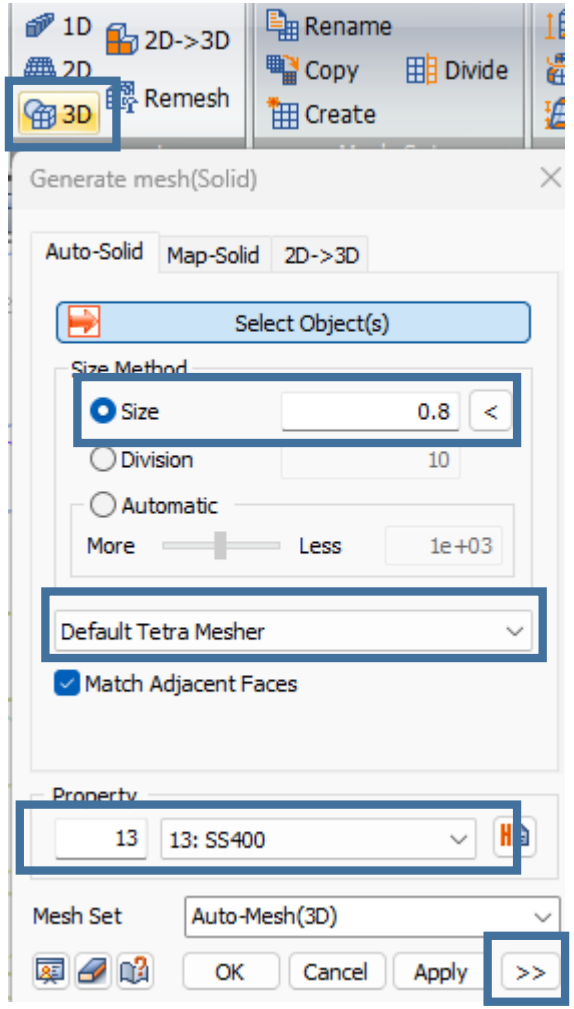
NOTE: Ensure that the geometry lines are divided into the correct intersections.

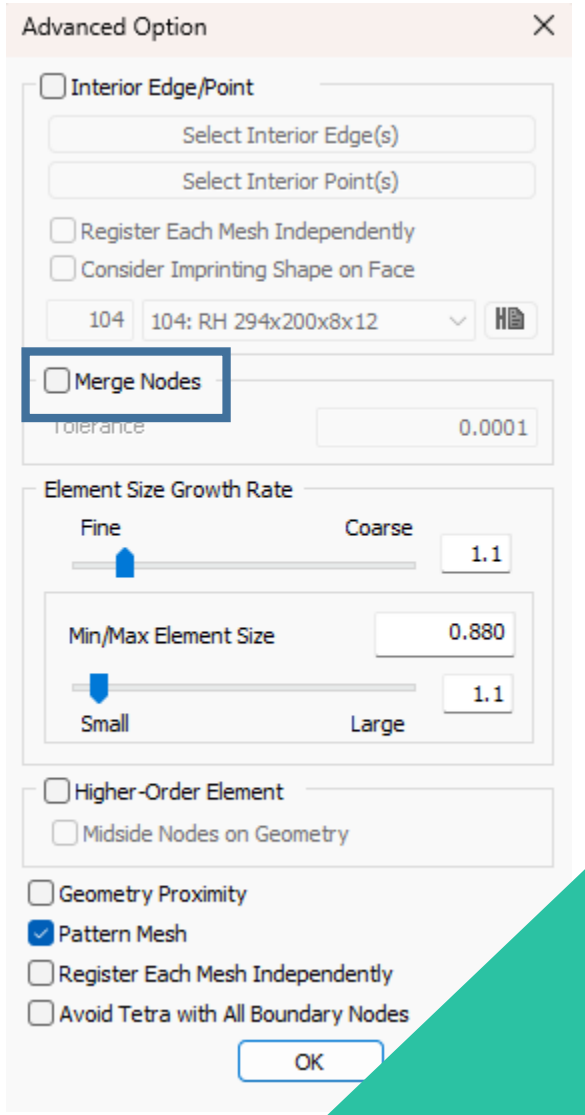
For **3D elements**, define the mesh size with the appropriate materials.

NOTE:

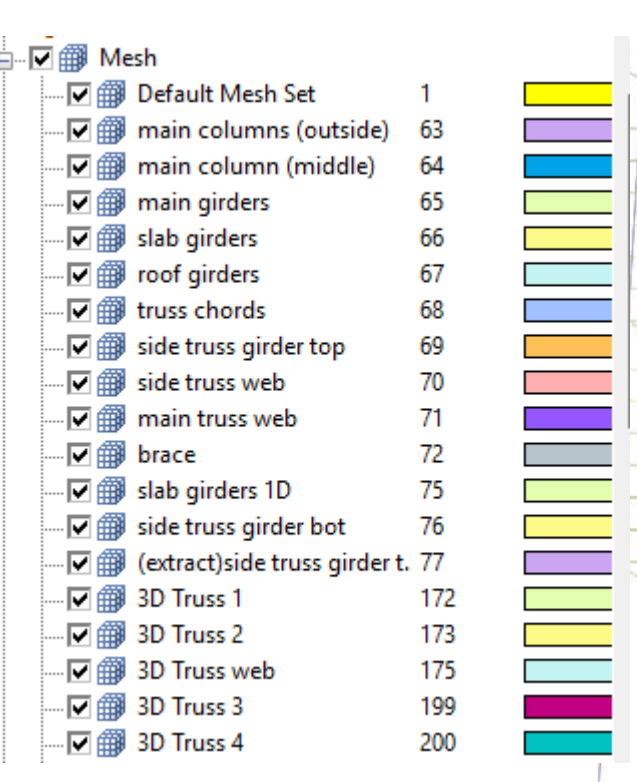
1. For future GEN export, use Tetra Mesher as mesh type

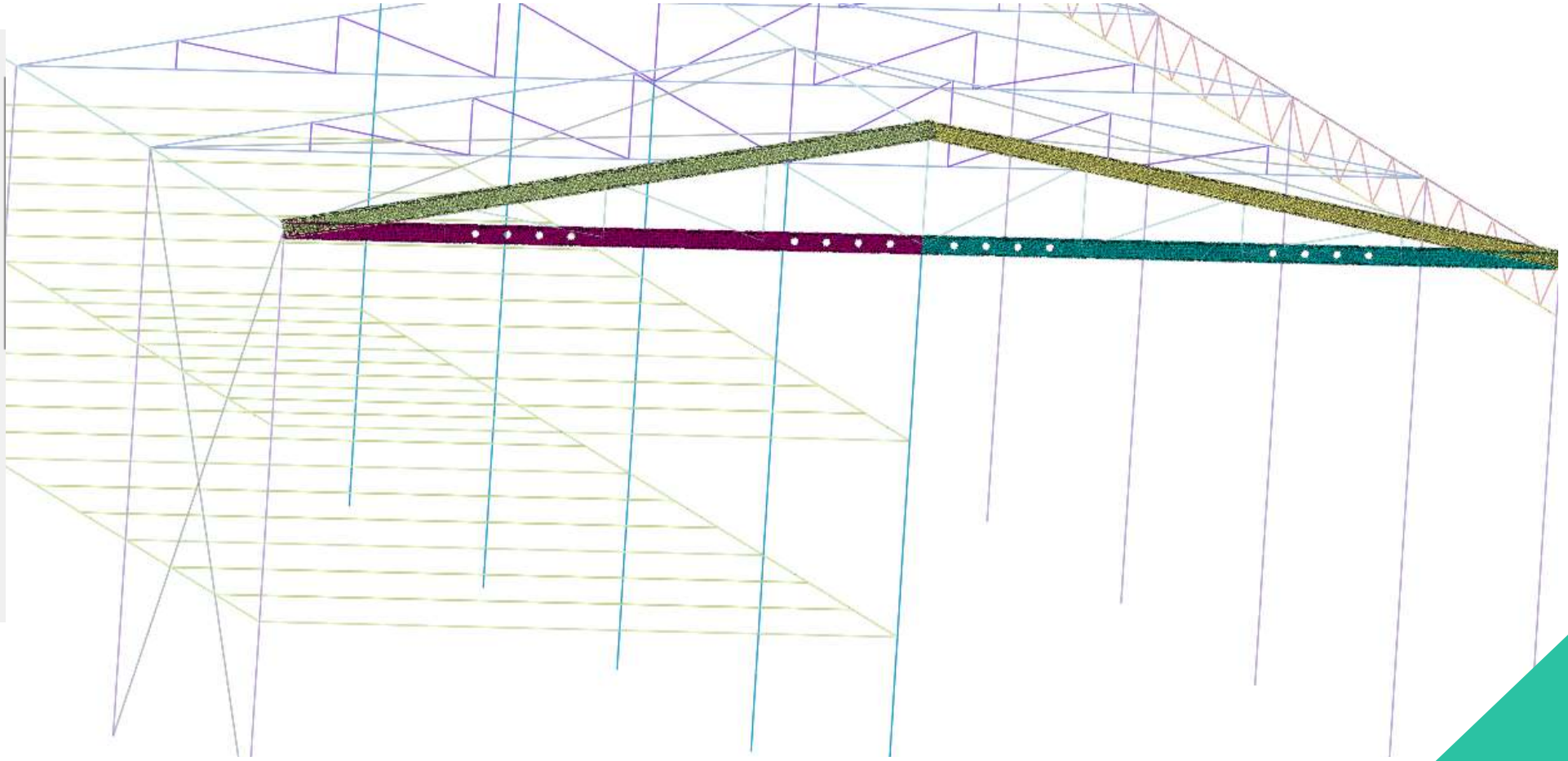
2. Deselect Merge nodes to create a segmented parts of the truss.



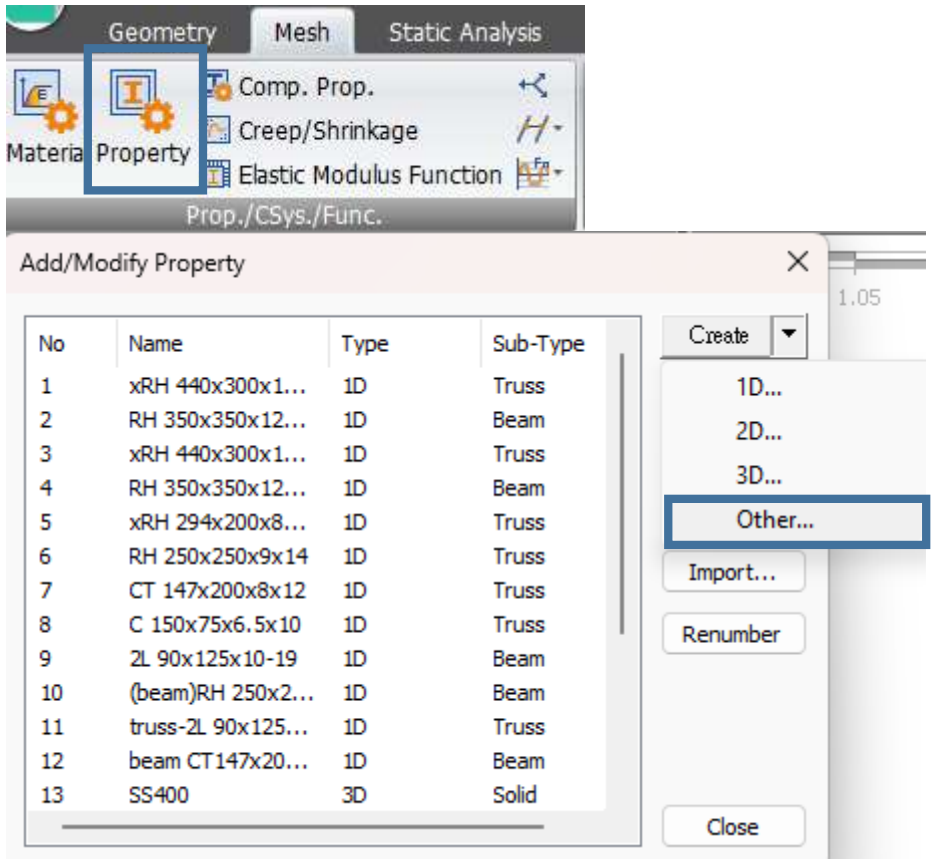


MODEL

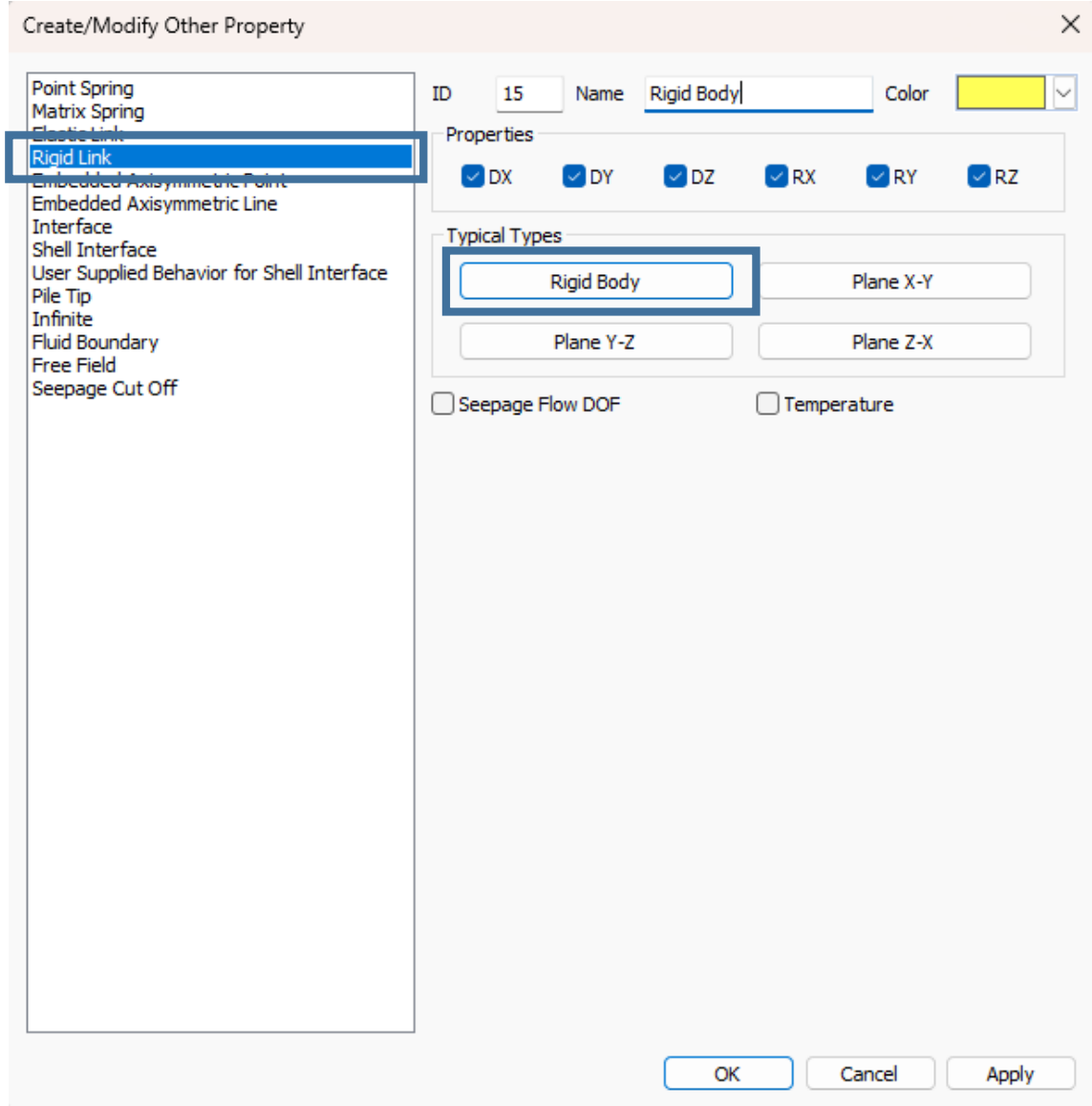

Mesh
Default Mesh Set 1
main columns (outside) 63
main column (middle) 64
main girders 65
slab girders 66
roof girders 67
truss chords 68
side truss girder top 69
side truss web 70
main truss web 71
brace 72
slab girders 1D 75
side truss girder bot 76
(extract)side truss girder t. 77
3D Truss 1 172
3D Truss 2 173
3D Truss web 175
3D Truss 3 199
3D Truss 4 200



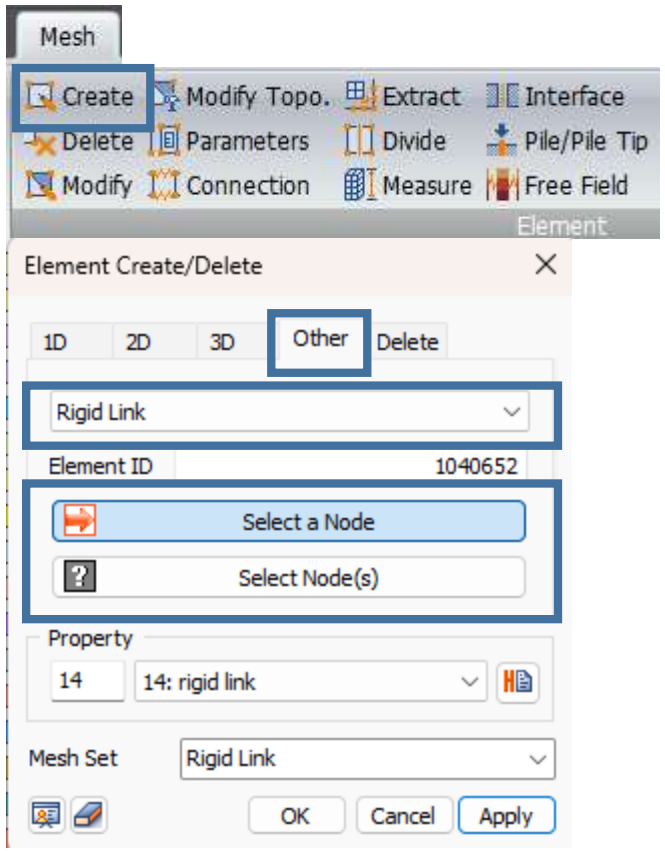
RIGID LINKS

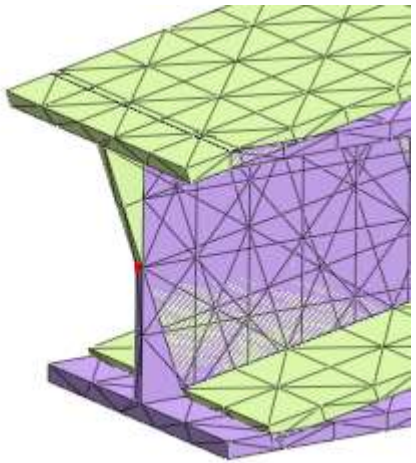


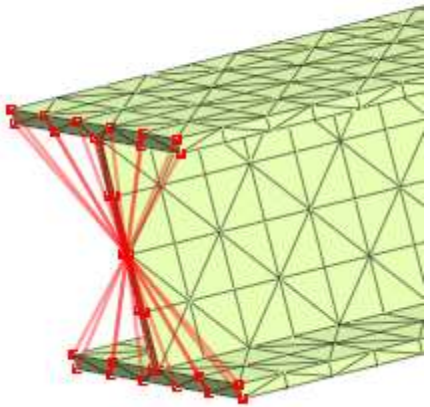
Define a rigid body in the property to create a rigid link.
NOTE: No material definition needed for rigid body.

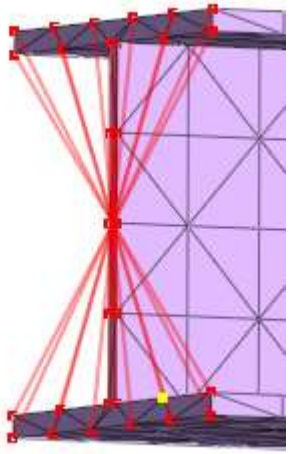


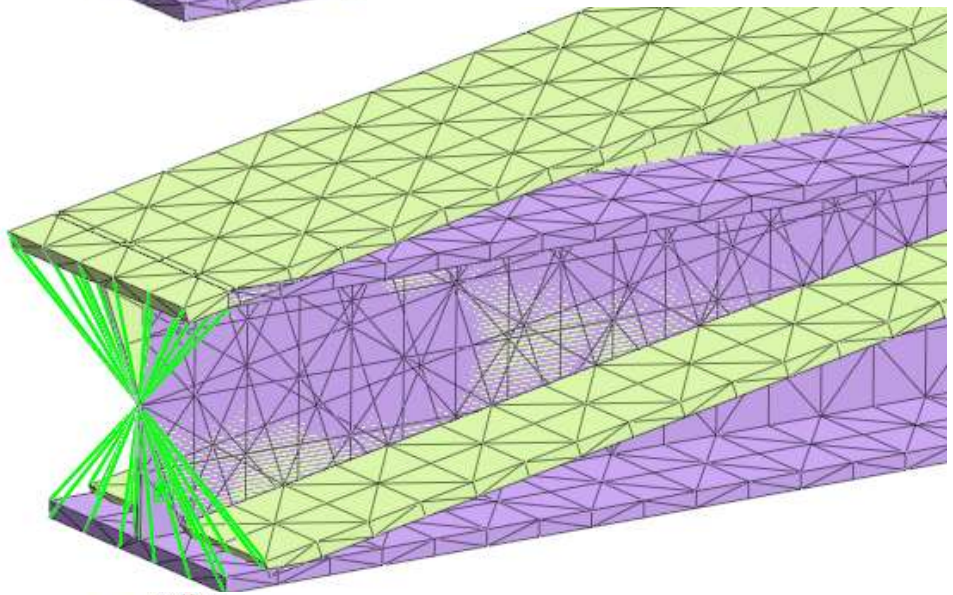
RIGID LINKS



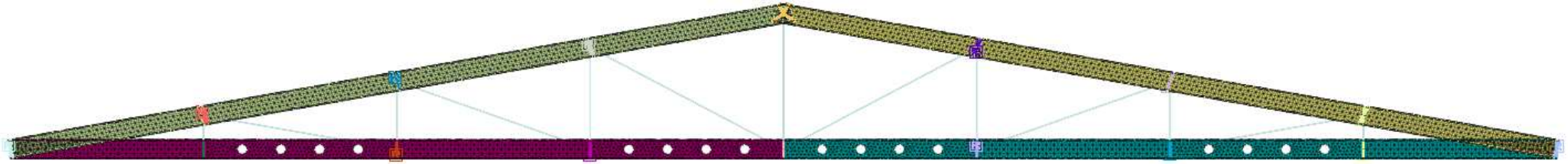






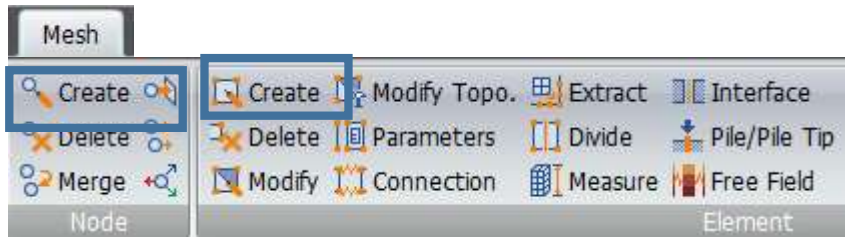


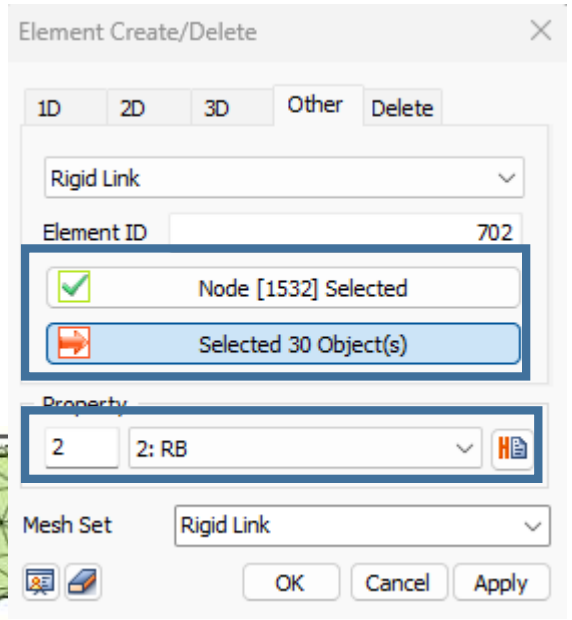
NOTE: Define rigid link to all joints

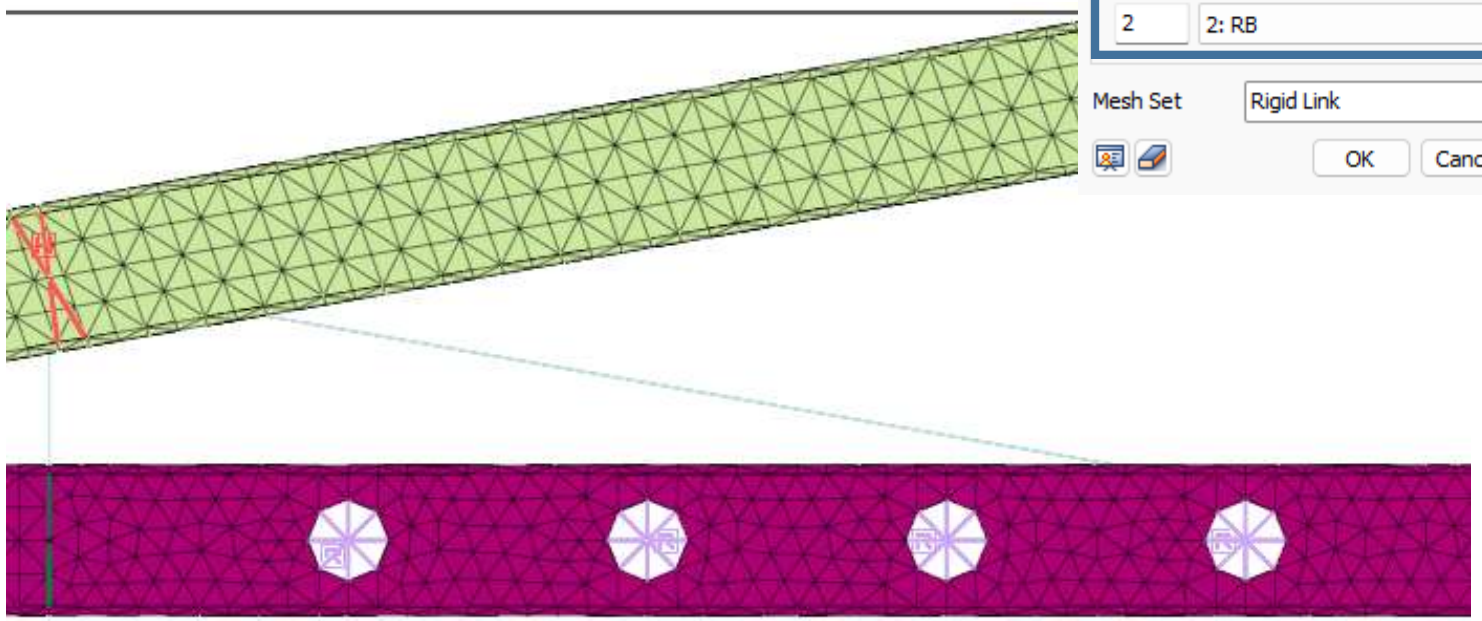


RIGID LINK

RIGID LINK-CREATE LINKS FOR HOLES

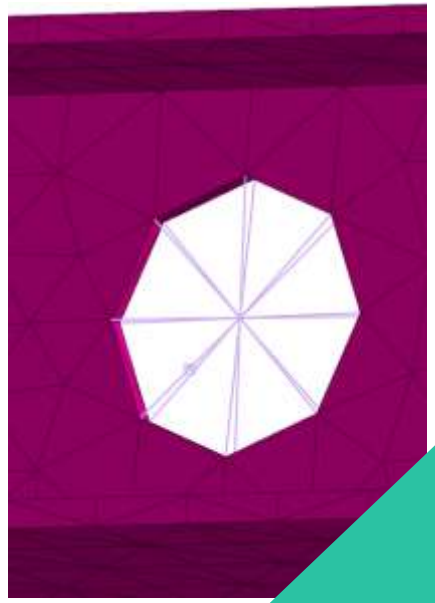






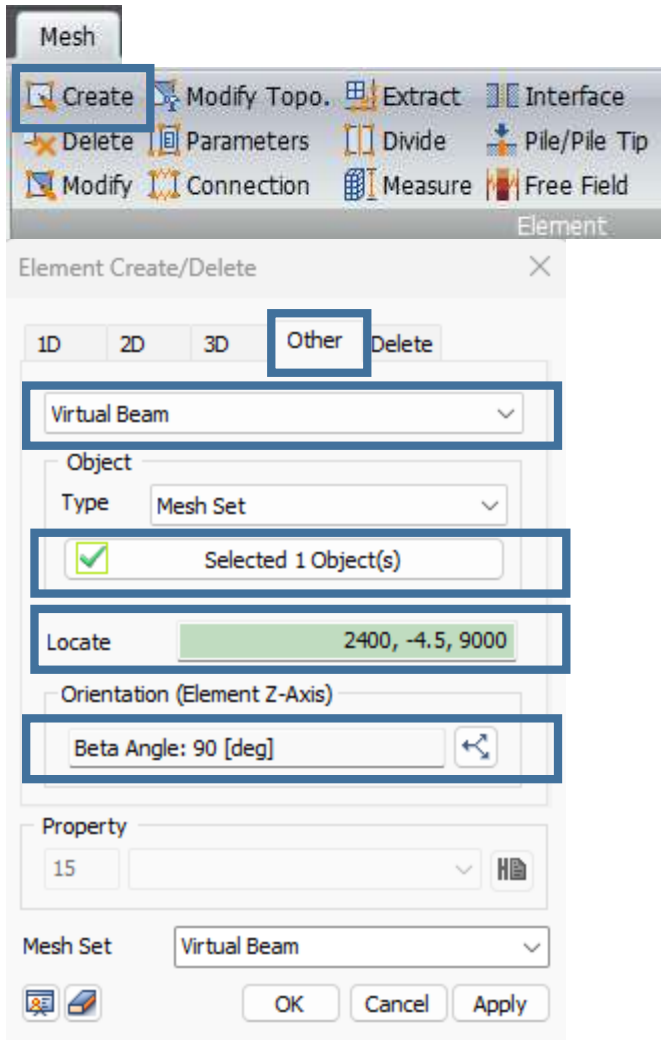
STEPS:

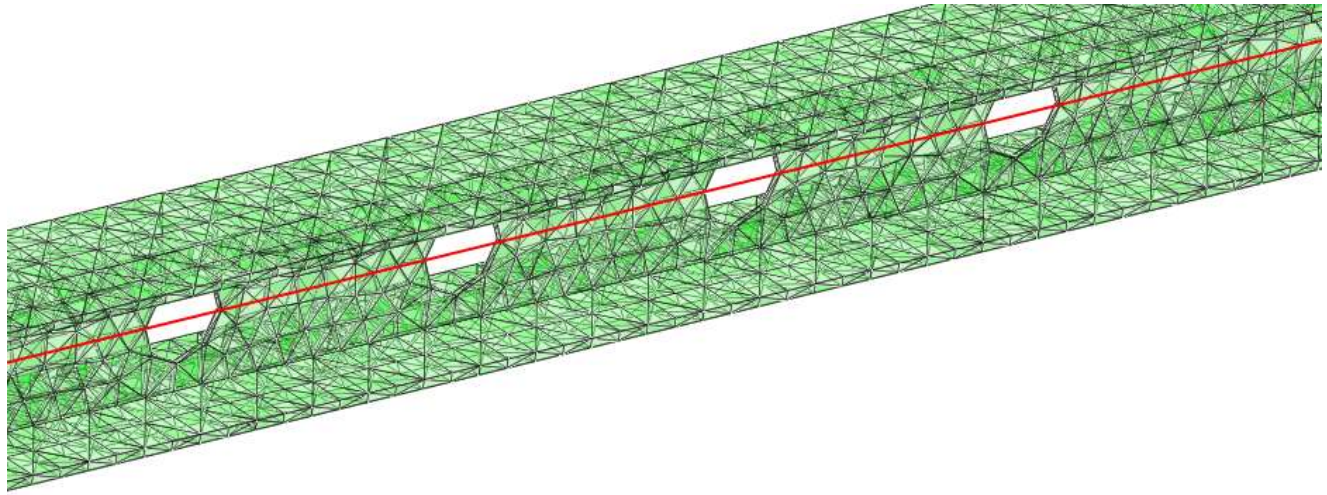
1. Define the master node in the center of the holes.
2. Link the edge of the hole to the master node.



NOTE: Define the rigid link 1 by 1 (per hole).

VIRTUAL BEAM





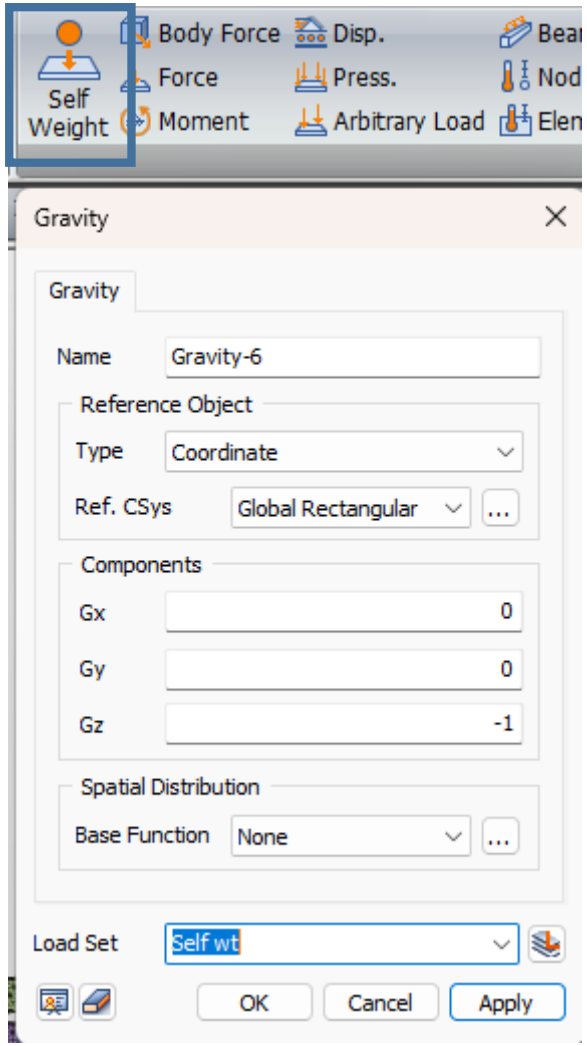
STEPS:

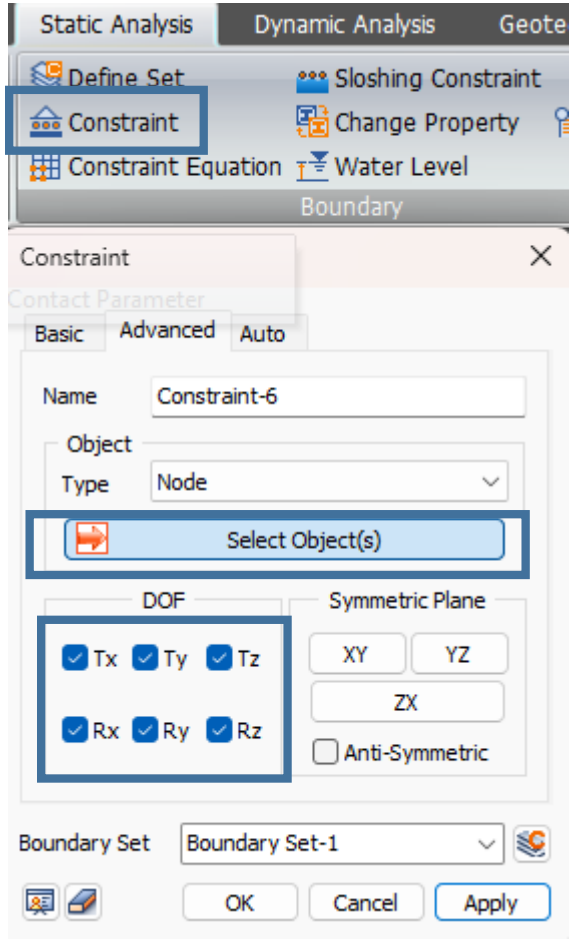
1. Select mesh set.

2. Create a line (end-to-end) along the geometric center of the mesh.

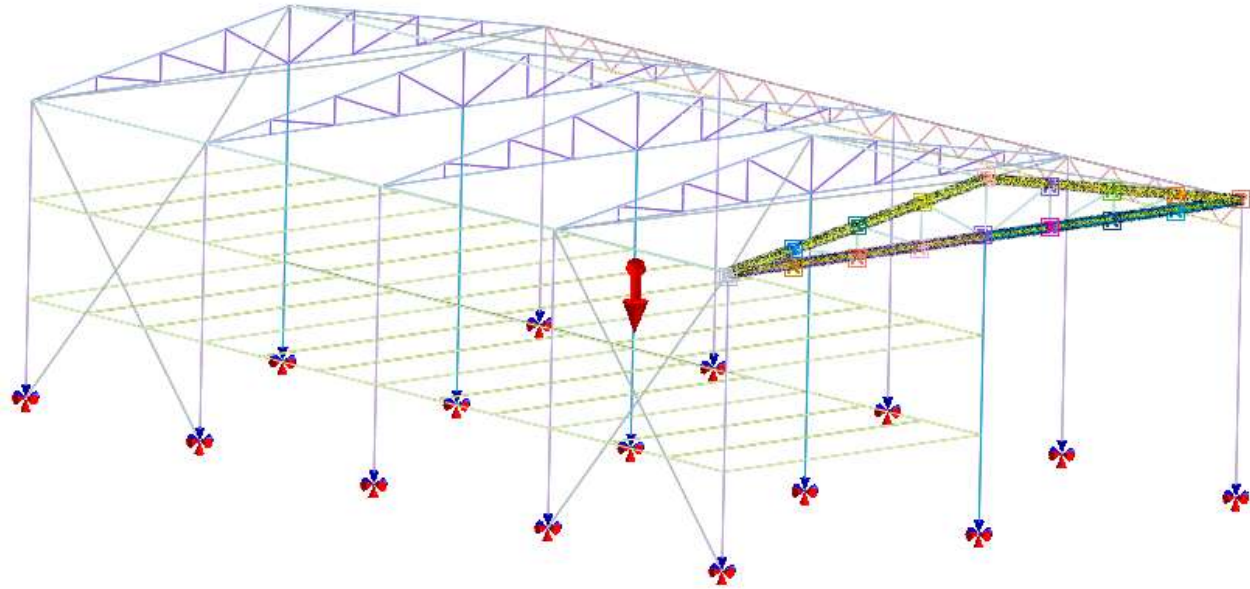
3. Choose the proper orentation.

LOADS and CONSTRAINTS





NOTE: Set constraint on the column to be fixed

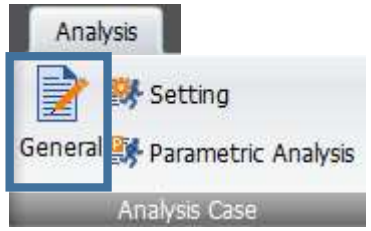


NOTE:

1. Initially, set to simplest load case (self weight)

2. Set constraint on the column base to be fixed (DOF: Txyz, Rxyz)

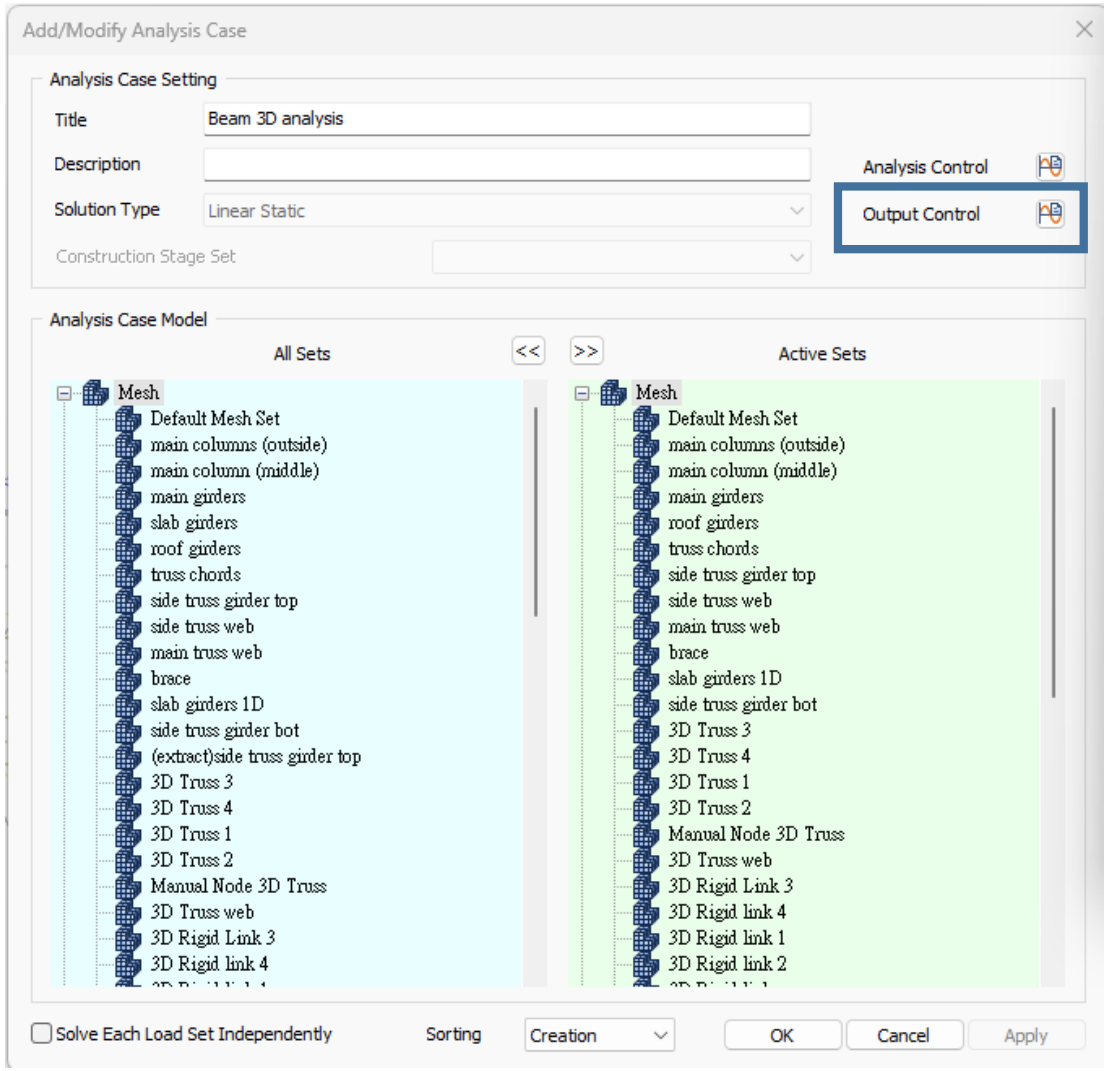
ANALYSIS CASE

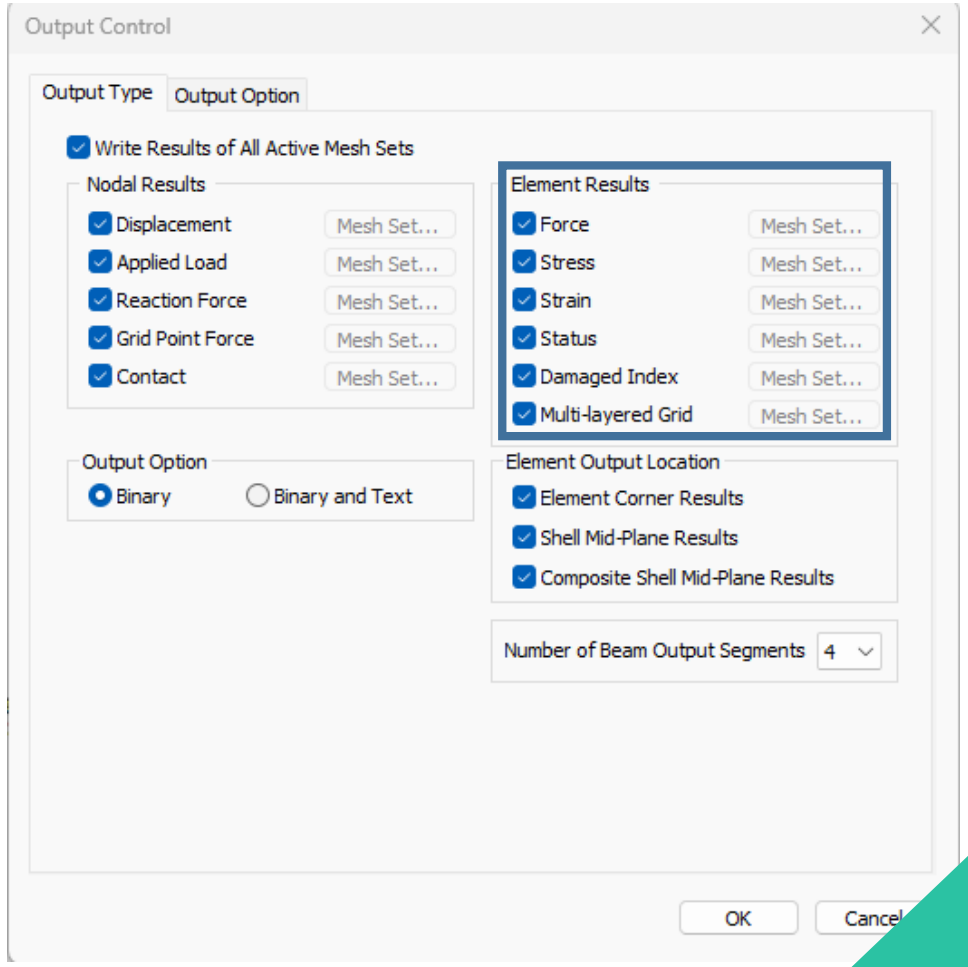


NOTE:

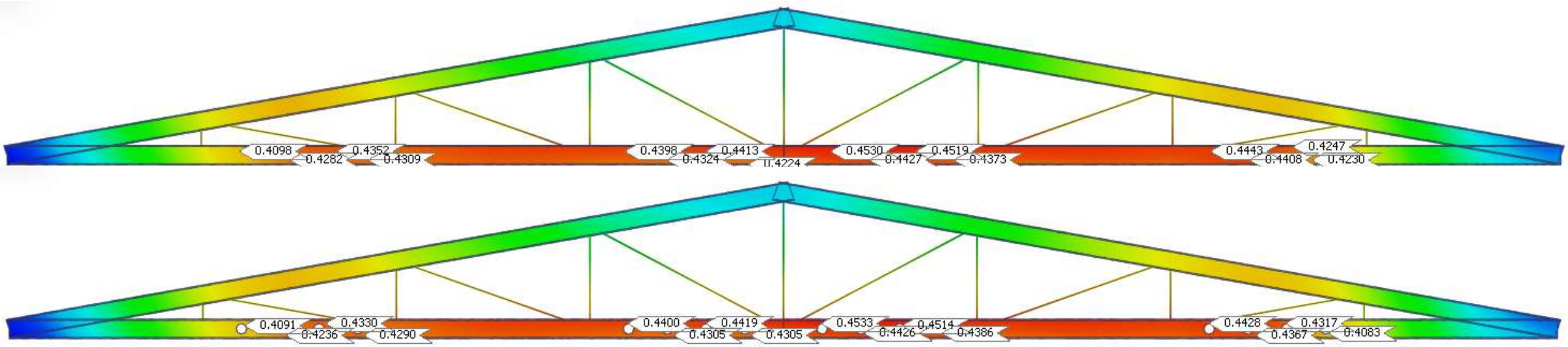
1. Include all the necessary meshes, constraints, and loads.

2. In the output control, include in the output the strain results since the analysis includes a 3D element.

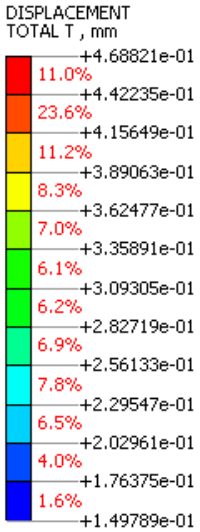




RESULT COMPARISON - DISPLACEMENT

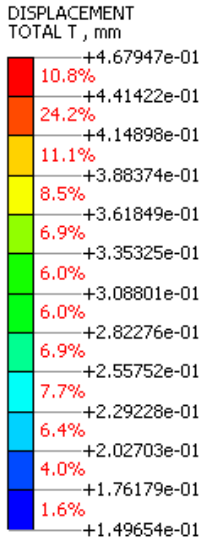


STANDARD

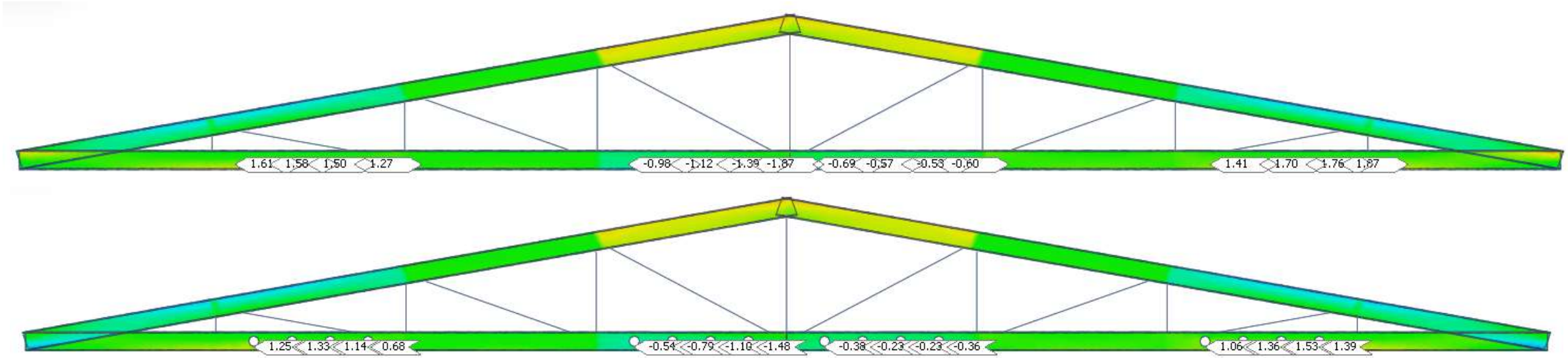


Overall, there is a small difference between the displacement after modifying the beams.

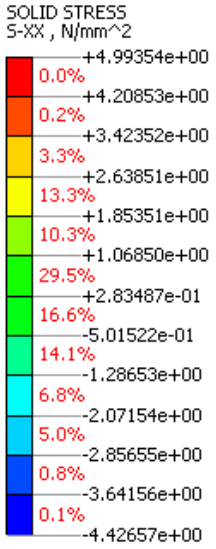
MODIFIED



RESULT COMPARISON - STRESS VON MISES

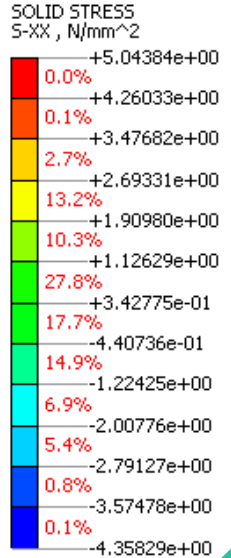


STANDARD

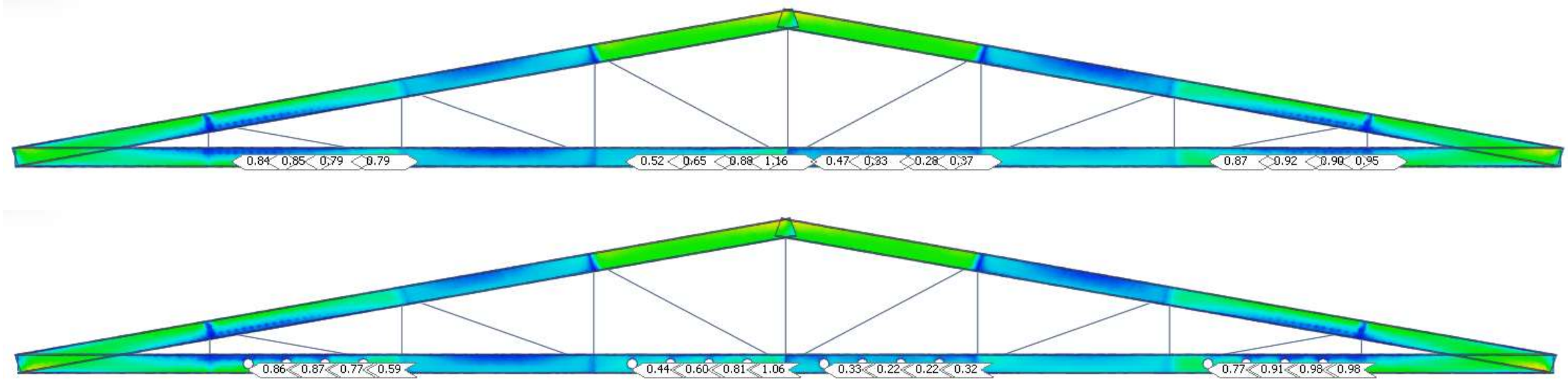


The modification created an S-XX increase of 13% - 60% on the hole areas.

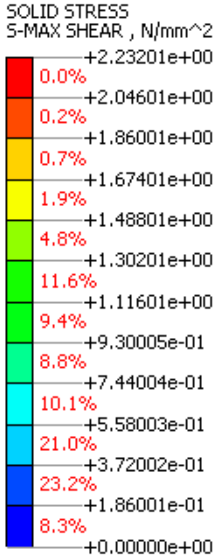
MODIFIED



RESULT COMPARISON - STRESS MAX SHEAR

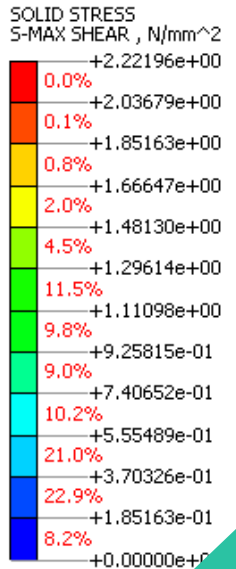


STANDARD

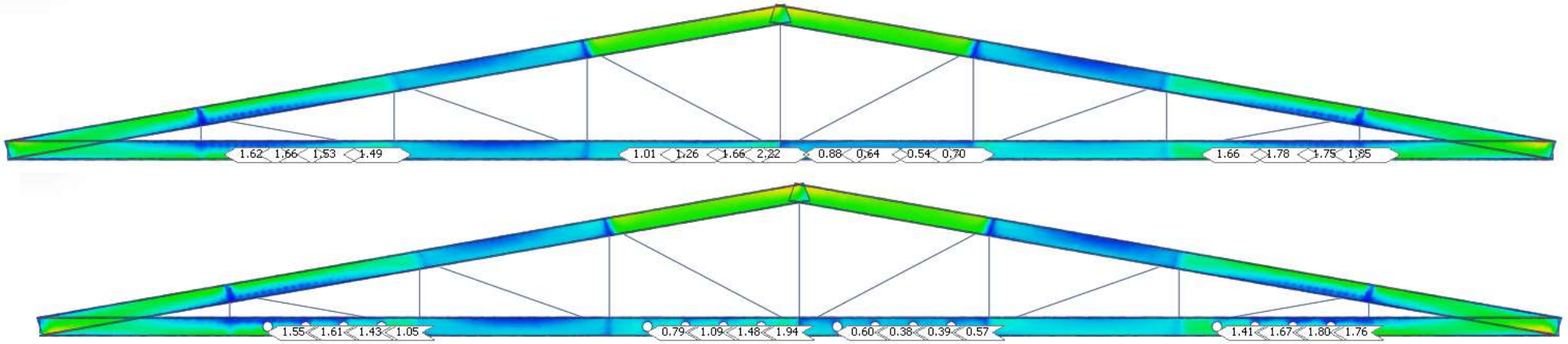


The modification created a max shear decrease in the midspan by 17%.

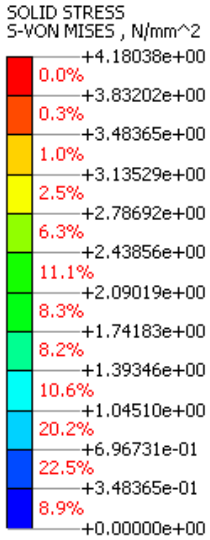
MODIFIED



RESULT COMPARISON - STRESS VON MISES

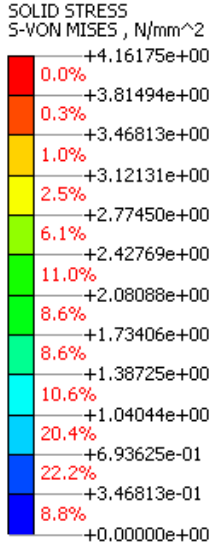


STANDARD

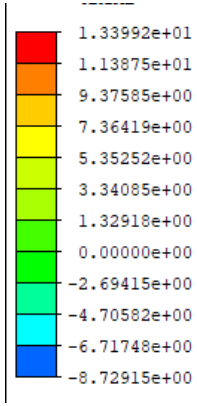


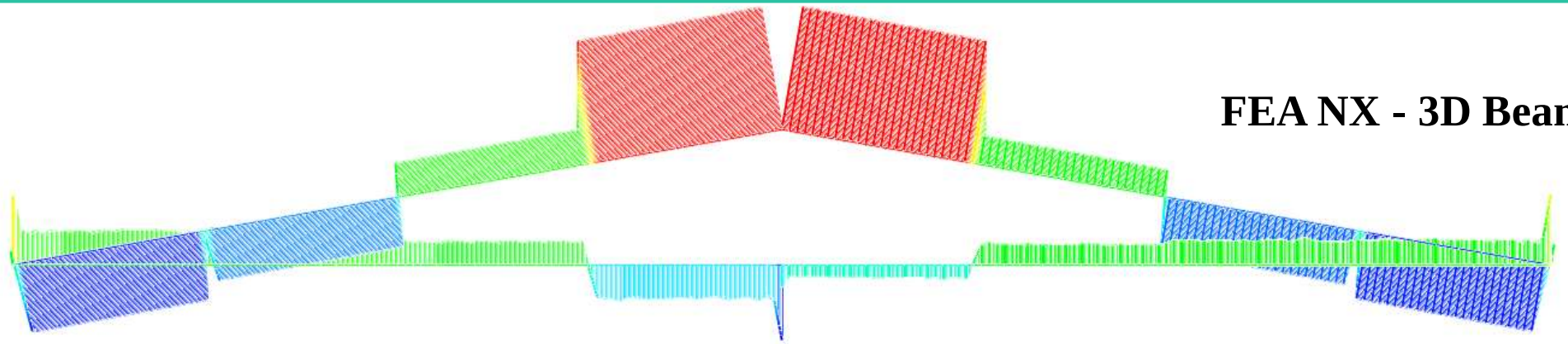
The modification created a stress von mises decrease in the midspan by 22.2%.

MODIFIED

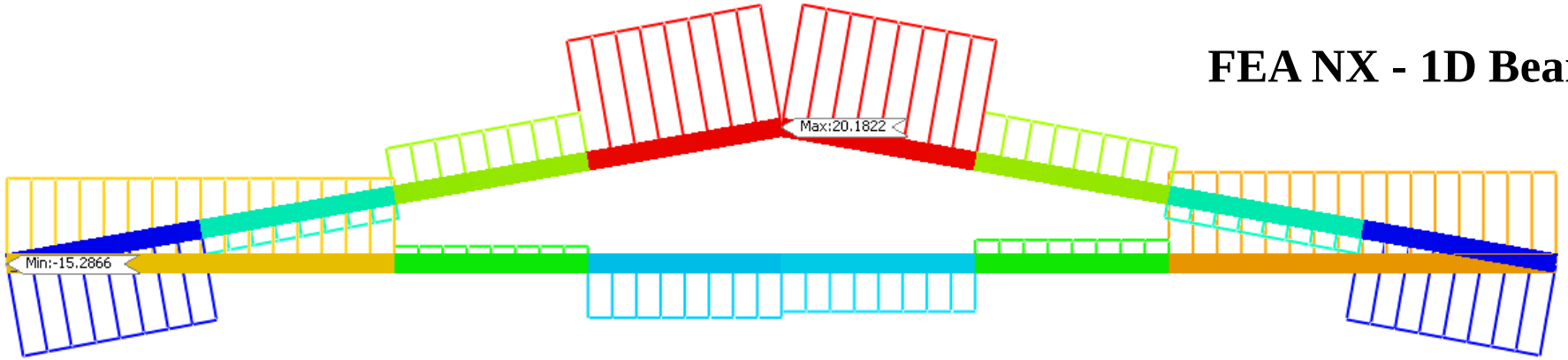


RESULT COMPARISON - AXIAL X (Fx)

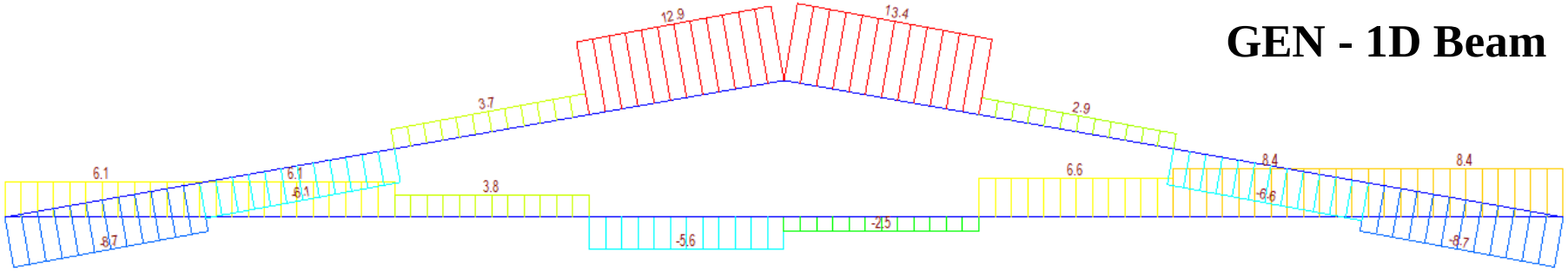




FEA NX - 3D Beam

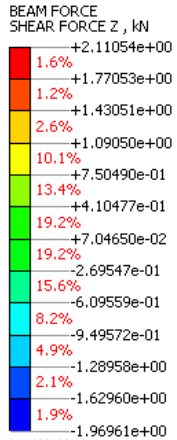


FEA NX - 1D Beam



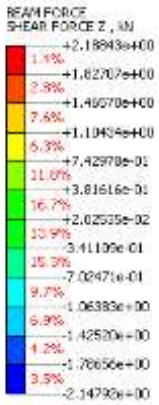
GEN - 1D Beam

RESULT COMPARISON - AXIAL Z (Fz)



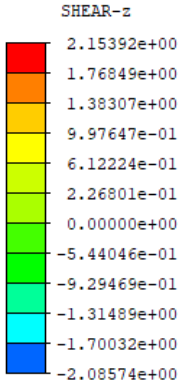
FEA NX - 3D Beam

FEA

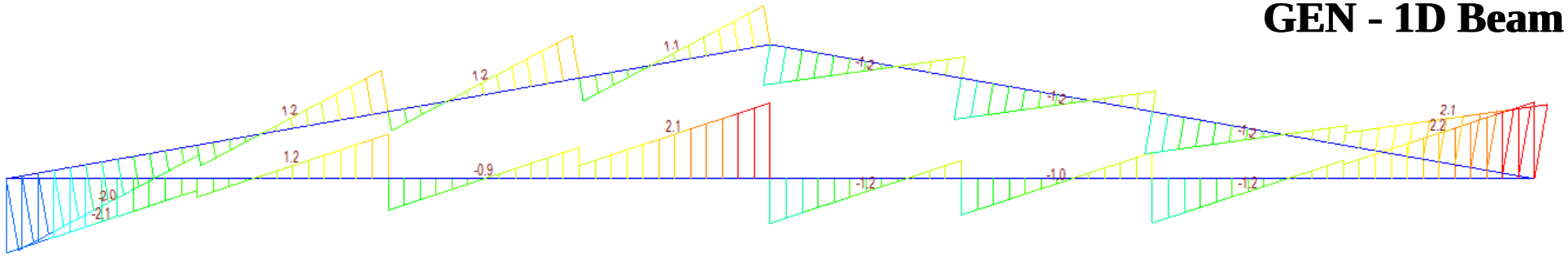


FEA NX - 1D Beam

FEA

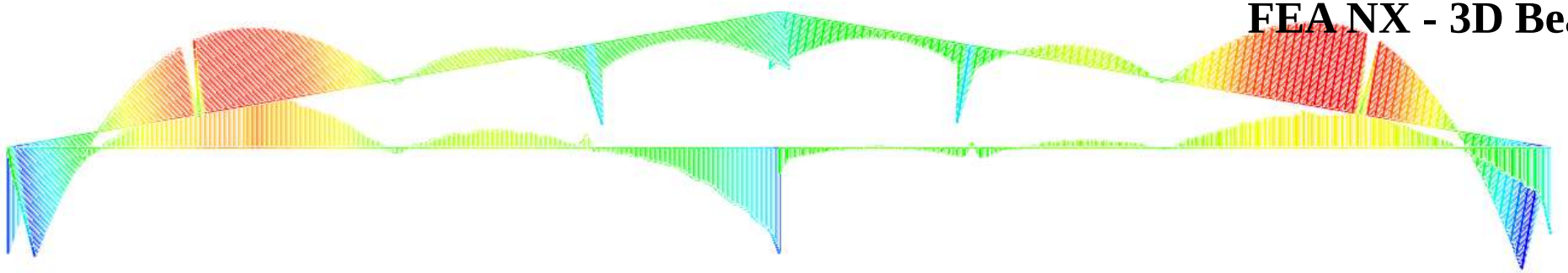


GEN - 1D Beam

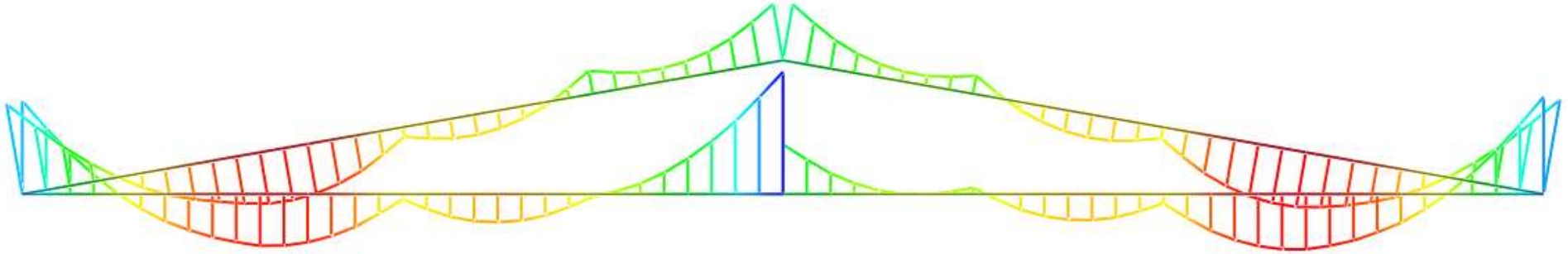


RESULT COMPARISON - MOMENT Y (My)

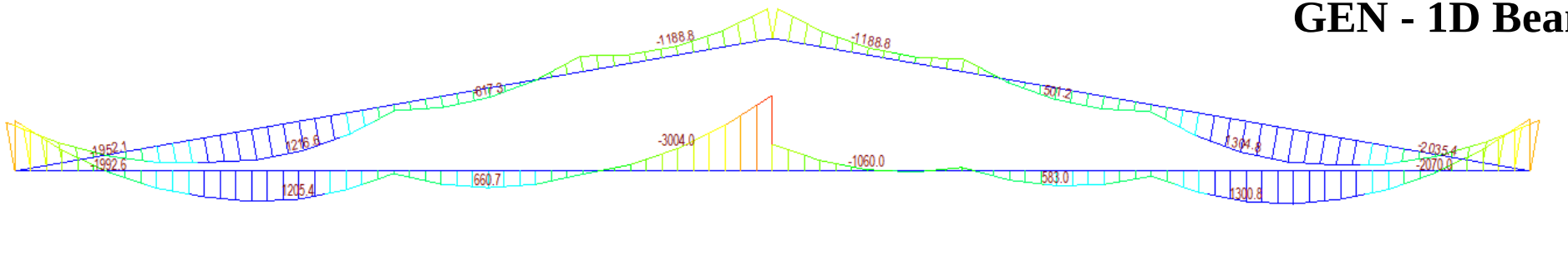
FEA NX - 3D Beam



FEA NX - 1D Beam



GEN - 1D Beam

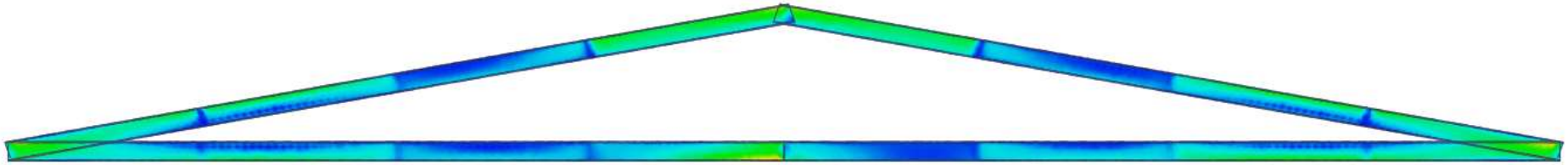


RESULT COMPARISON - STRESS VON MISES

SOLID STRESS
S-VON MISES , kN/mm^2

	+3.98693e-03
0.0%	
	+3.65468e-03
0.2%	
	+3.32244e-03
0.5%	
	+2.99019e-03
0.9%	
	+2.65795e-03
1.9%	
	+2.32571e-03
5.5%	
	+1.99346e-03
9.4%	
	+1.66122e-03
11.7%	
	+1.32898e-03
13.2%	
	+9.96731e-04
19.0%	
	+6.64488e-04
22.7%	
	+3.32244e-04
15.1%	
	+0.00000e+00

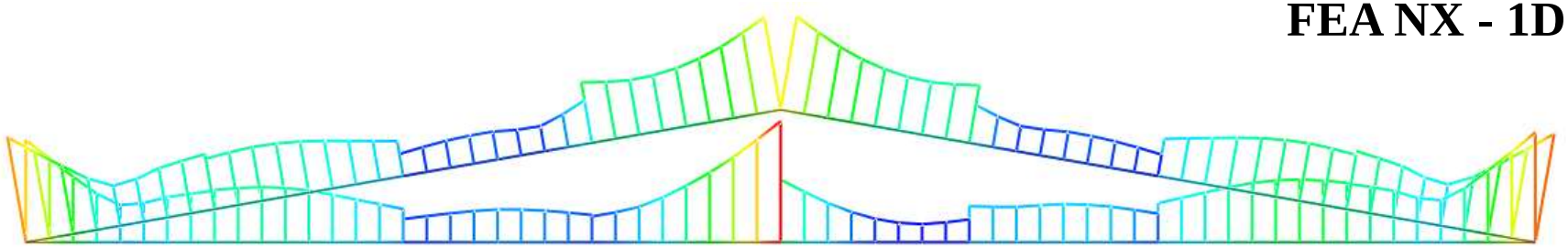
FEA NX - 3D Beam



BEAM STRESS
S-VON MISES , kN/mm^2

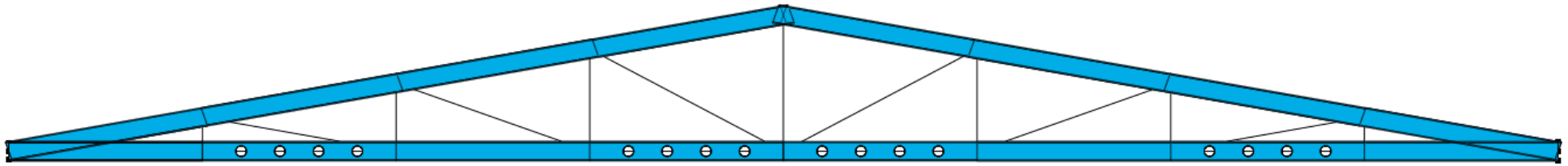
	+4.75315e-03
0.7%	
	+4.41655e-03
2.8%	
	+4.07995e-03
1.4%	
	+3.74336e-03
1.4%	
	+3.40676e-03
1.4%	
	+3.07016e-03
5.6%	
	+2.73356e-03
6.9%	
	+2.39696e-03
20.1%	
	+2.06037e-03
16.0%	
	+1.72377e-03
19.4%	
	+1.38717e-03
11.8%	
	+1.05057e-03
12.5%	
	+7.13975e-04

FEA NX - 1D Beam

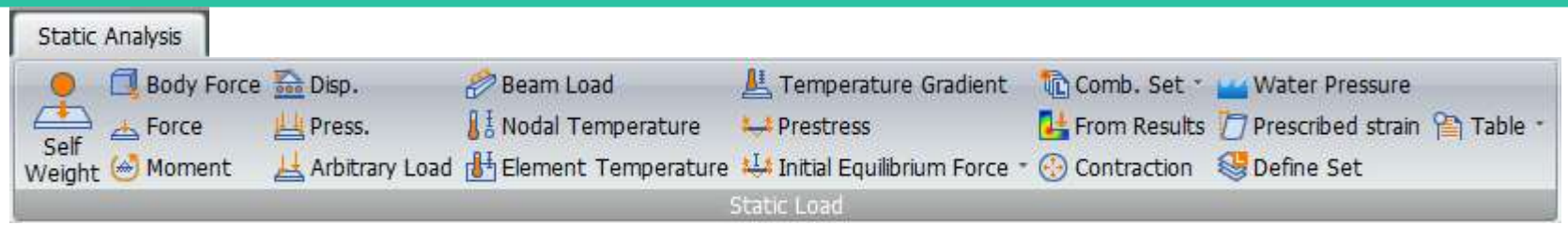


Part 3

Load Applications



LOAD



Dead Load

Slab = -0.4 tonf/m

Roof Girder = -0.1 tonf/m

Purlins = -1.3 tonf/m

Side Truss = -0.1 tonf/m

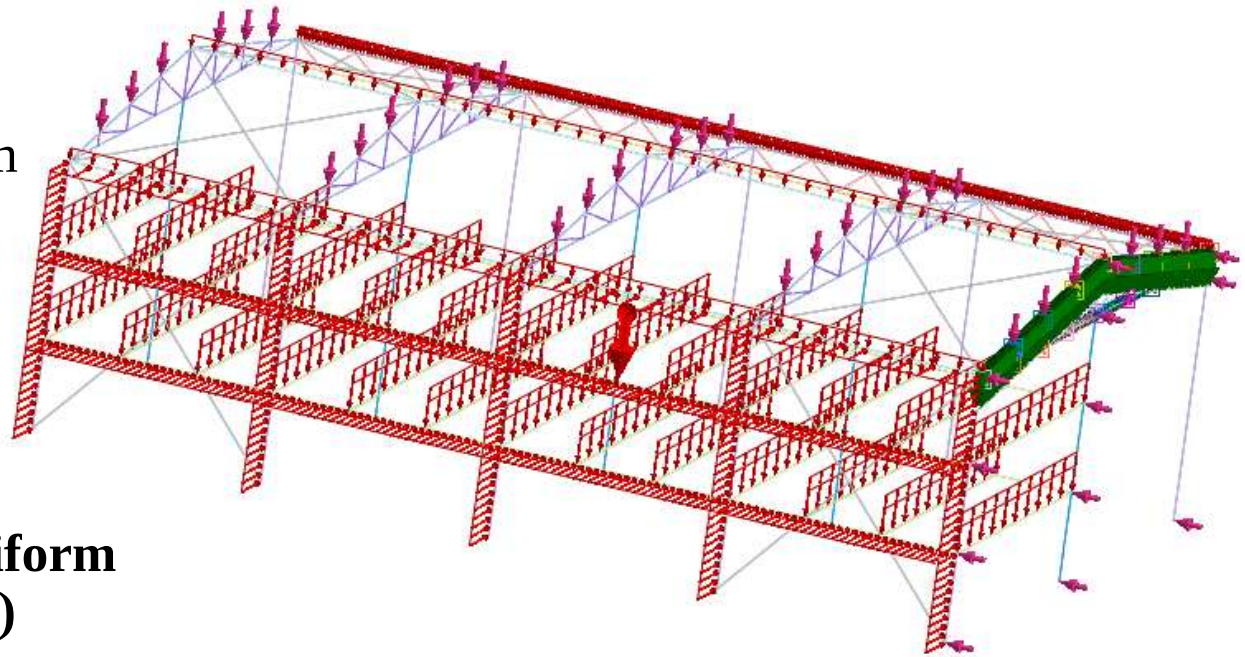
Live Load

Slab = -0.7 tonf/m

Roof Girder = -0.1 tonf/m

Purlins = -1.0 tonf/m

Side Truss = -0.1 tonf/m



Wind Load

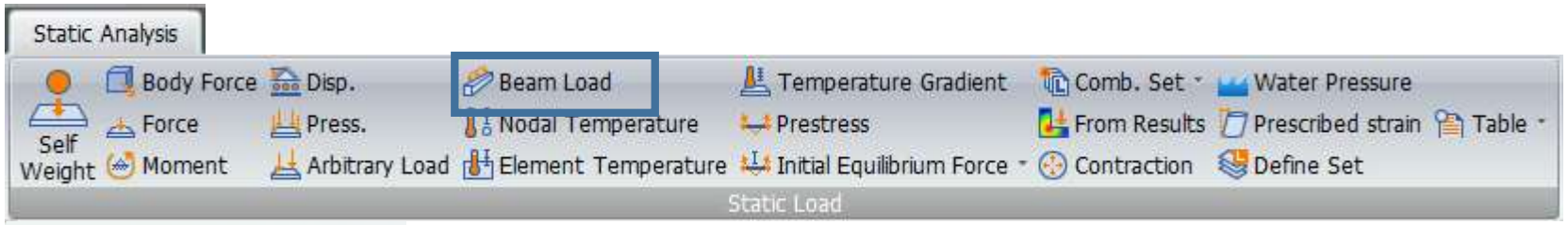
Wx= 0.88 tonf/m

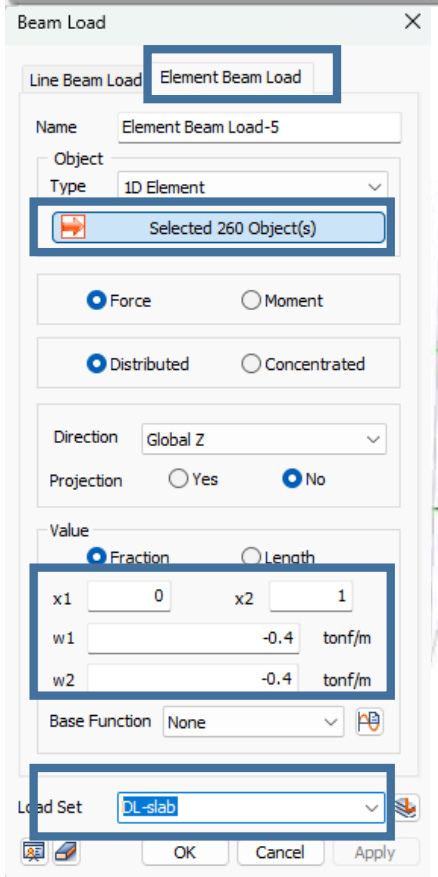
Wy = 0.92 tonf

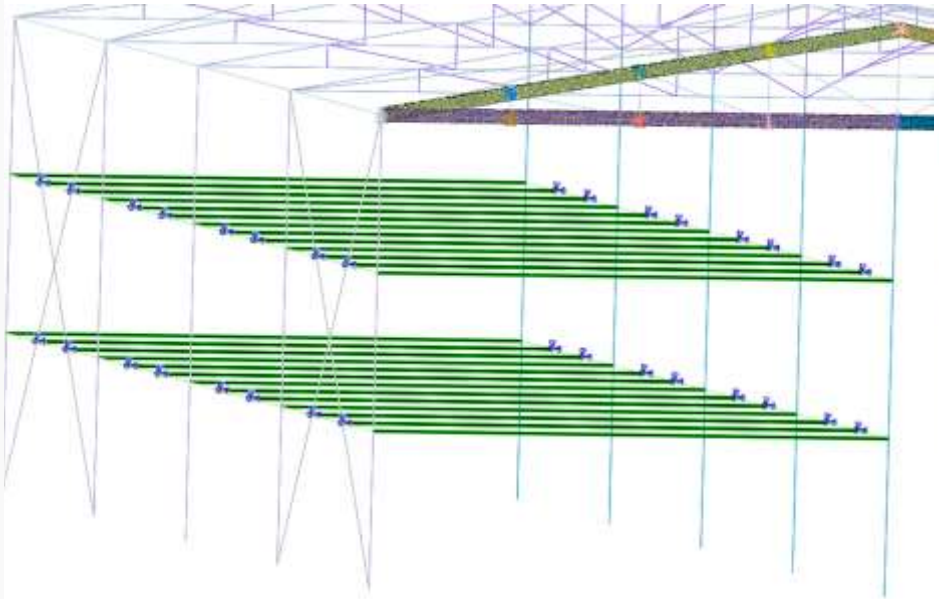
Additional Load (Uniform Load front Truss)

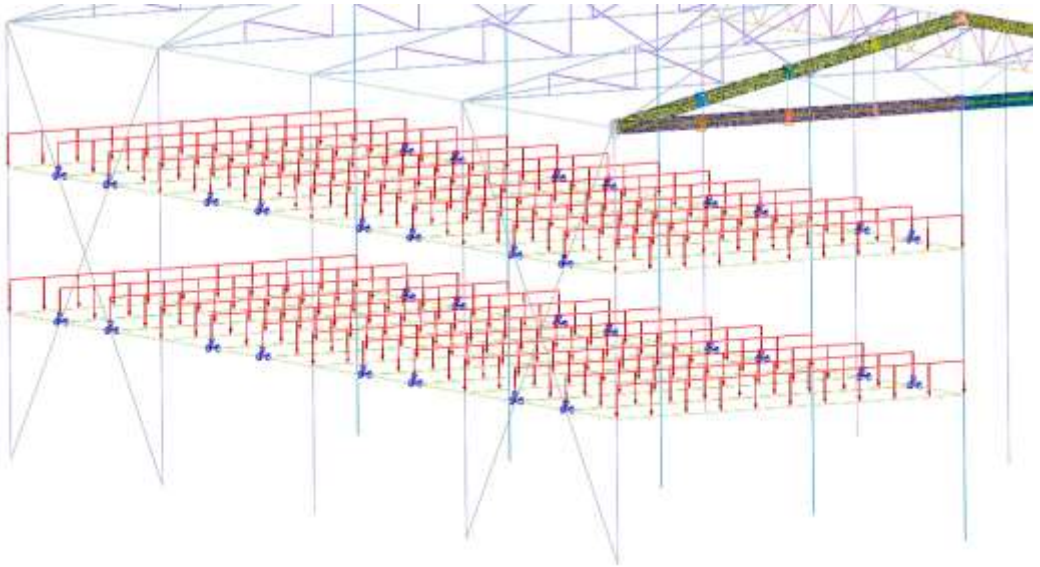
Uniform load = 2 tonf/m

DEAD LOAD - SLAB

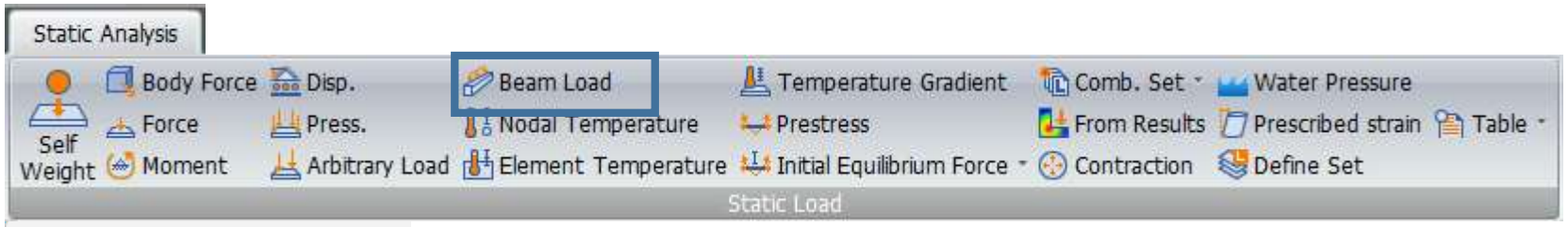


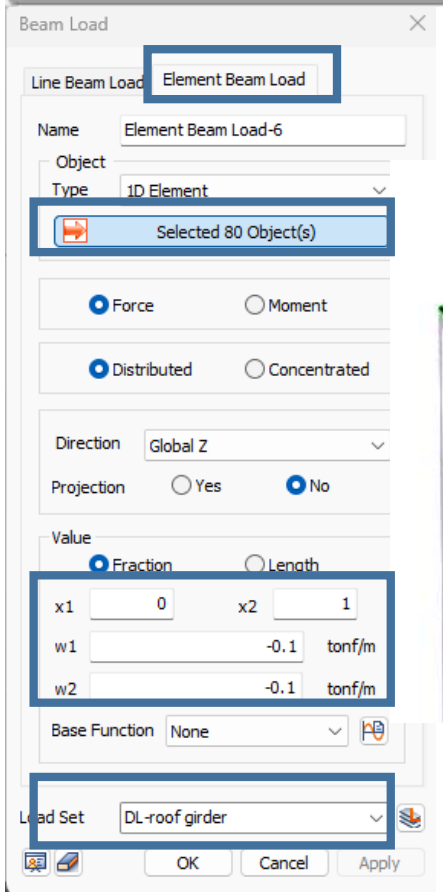


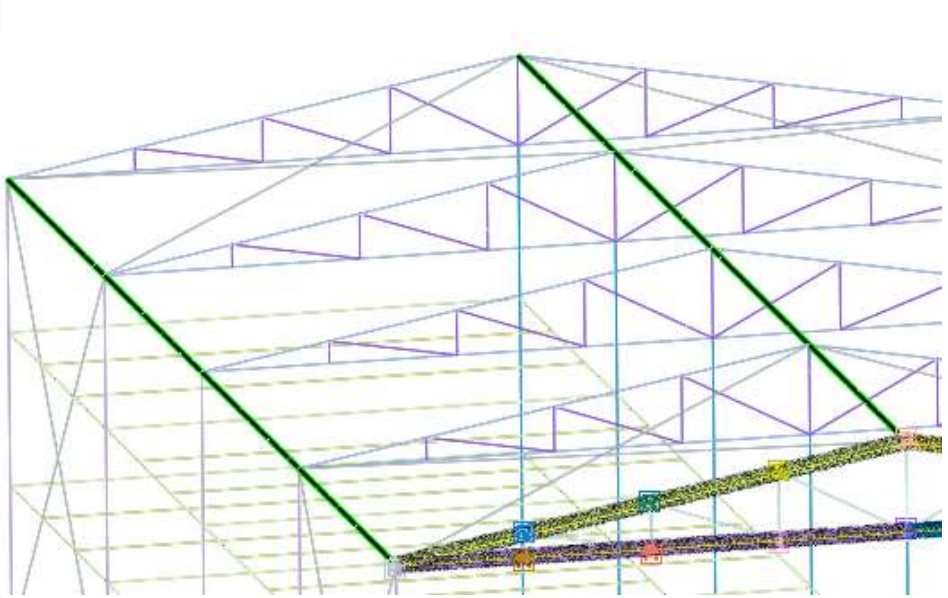


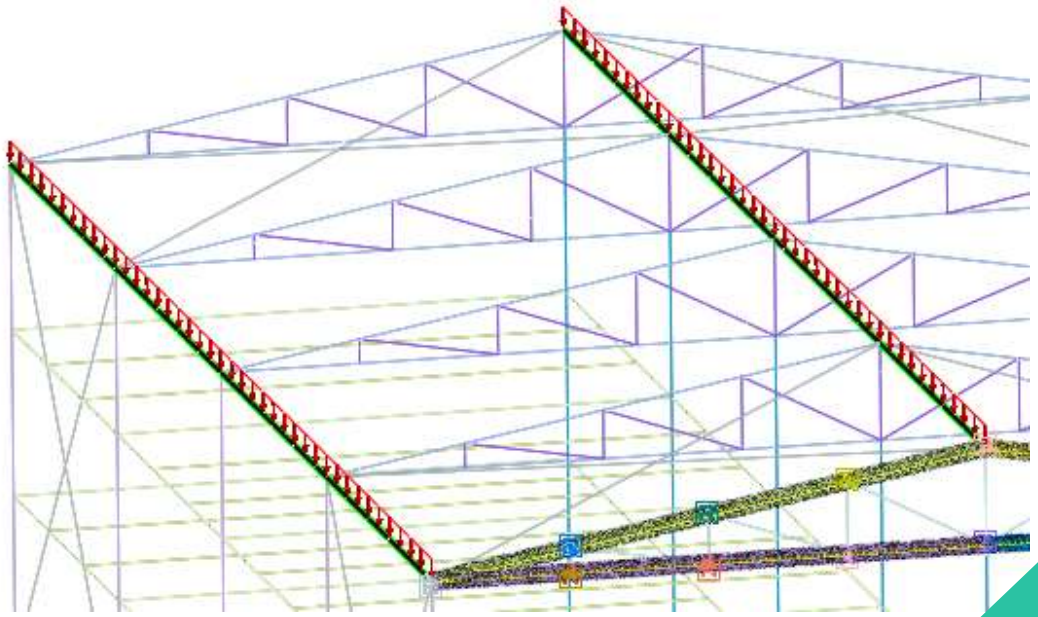


DEAD LOAD - ROOF GIRDER

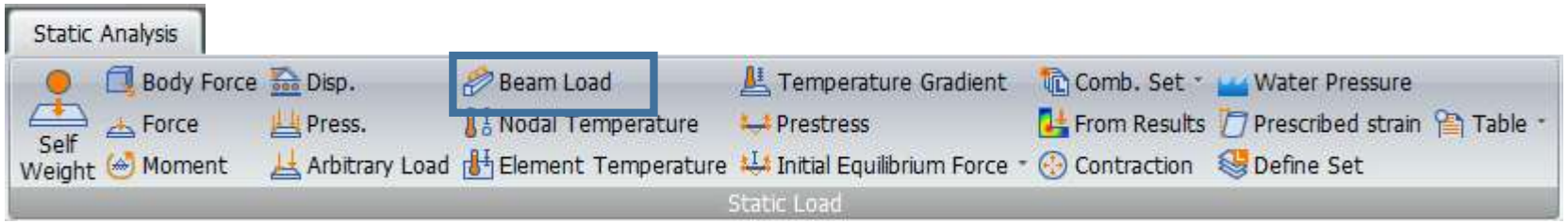


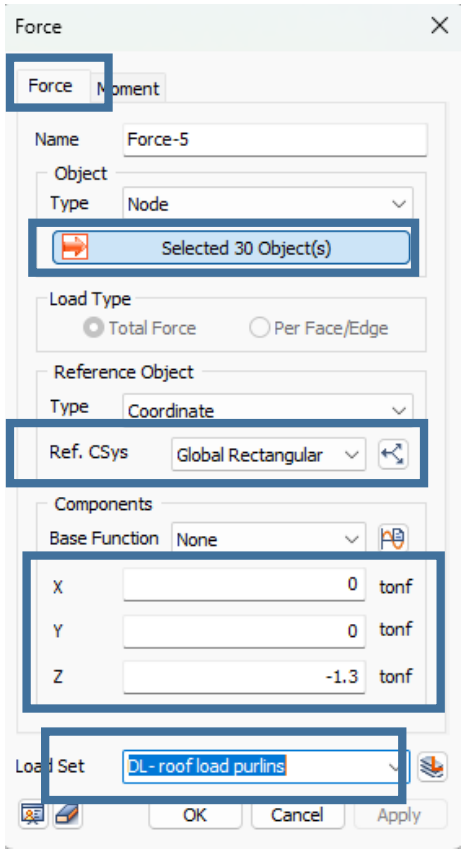


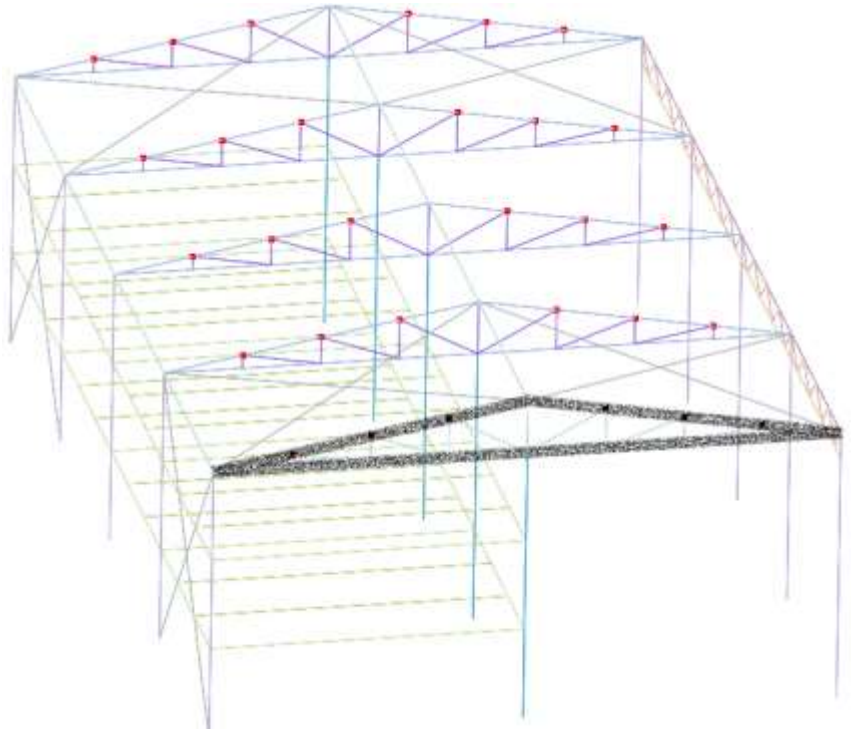


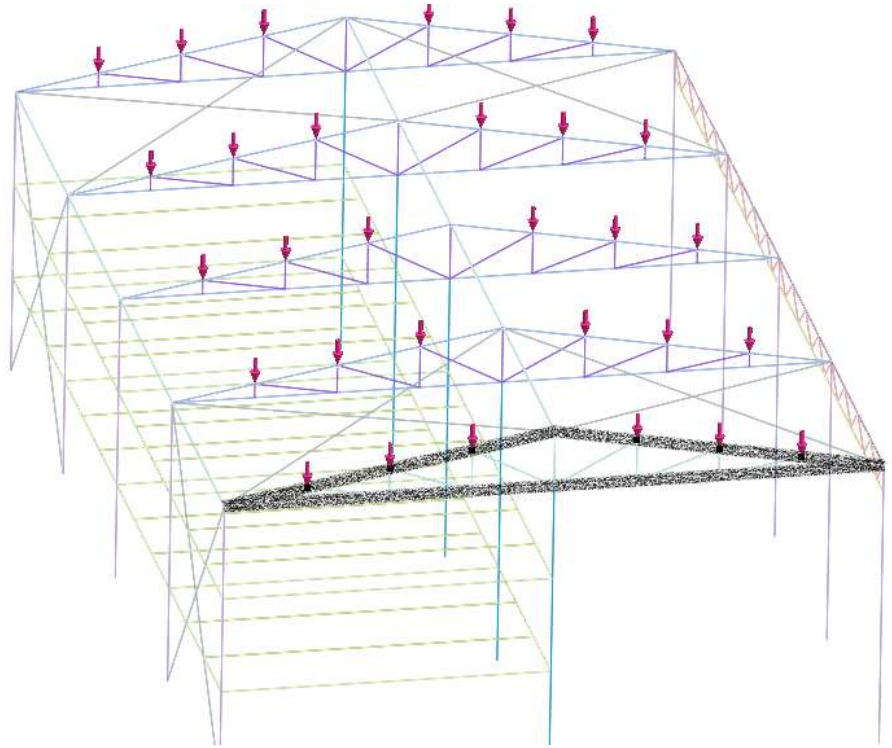


DEAD LOAD - PURLINS

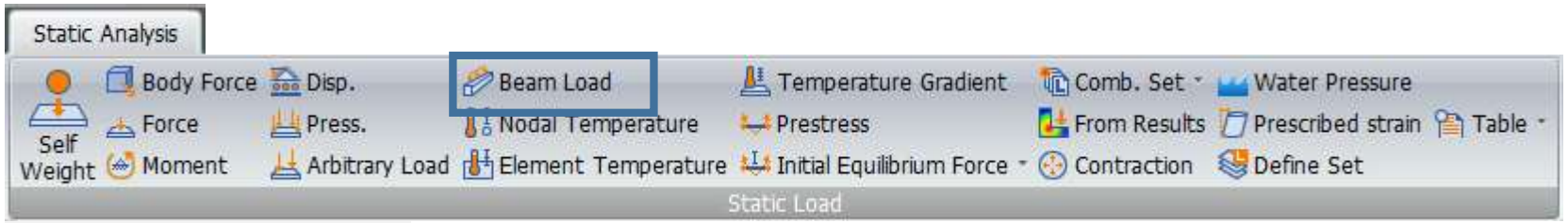


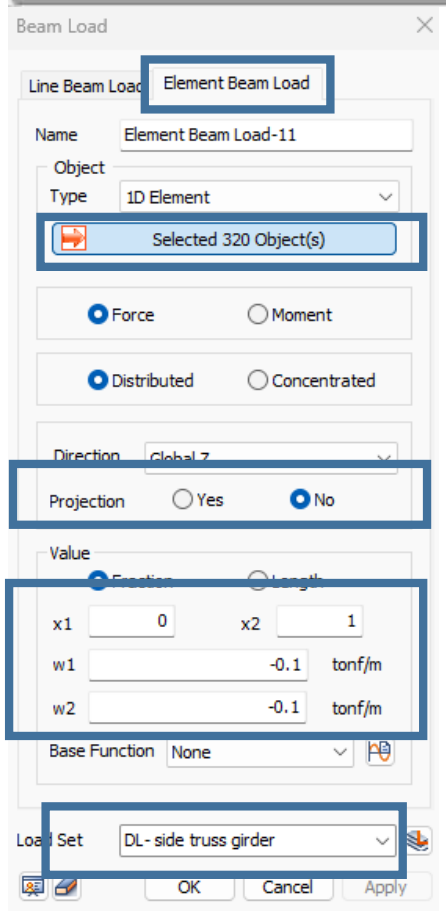


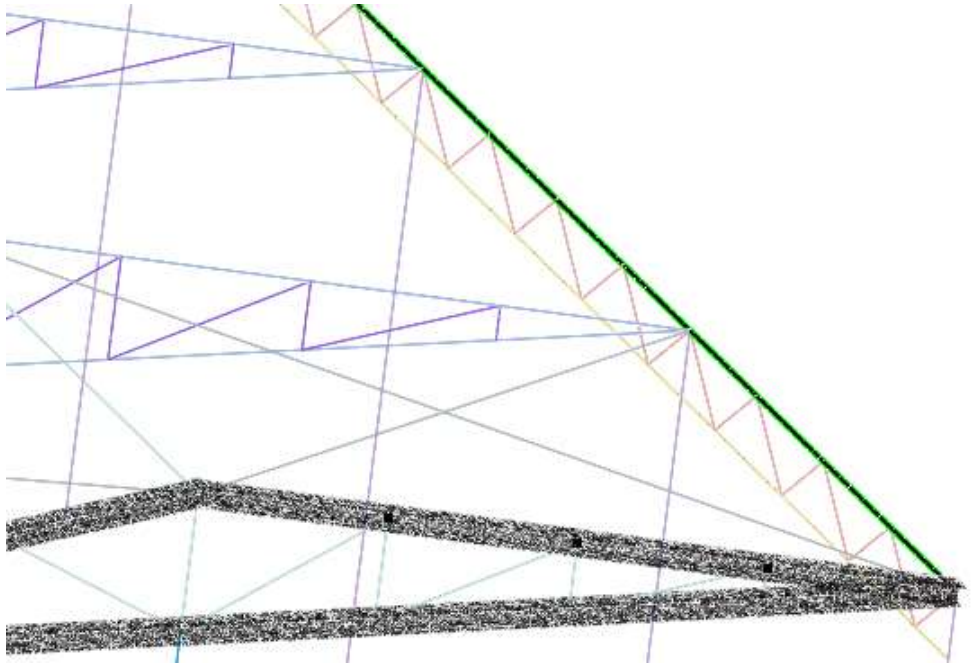


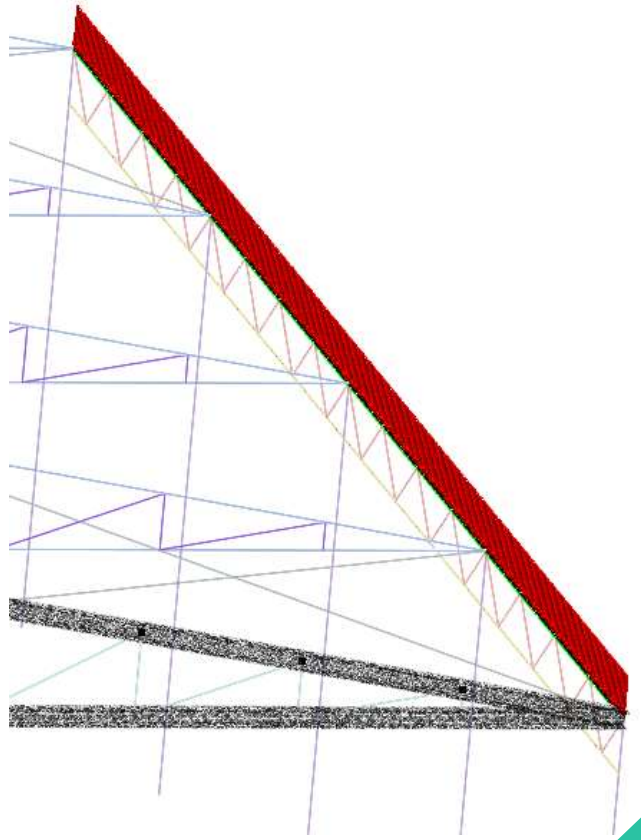


DEAD LOAD - SIDE TRUSS

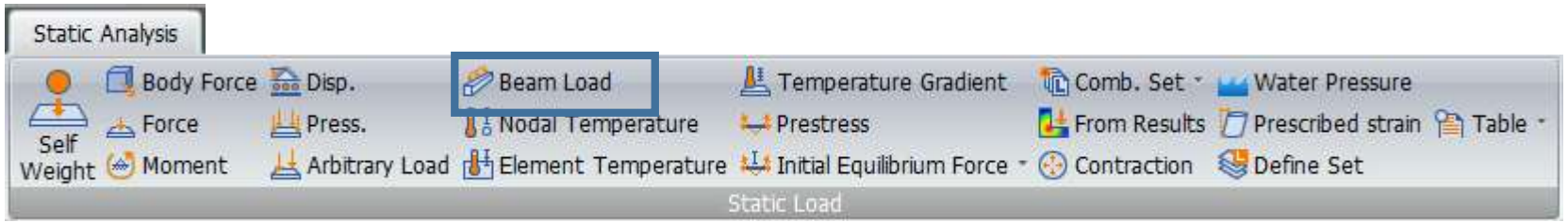


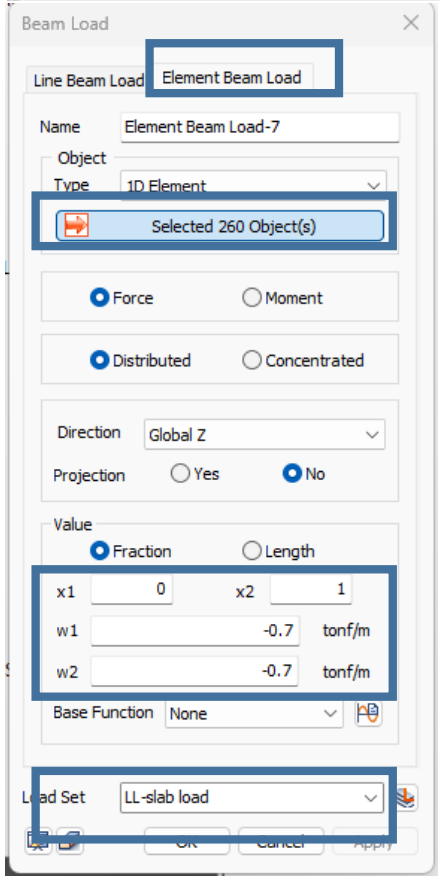


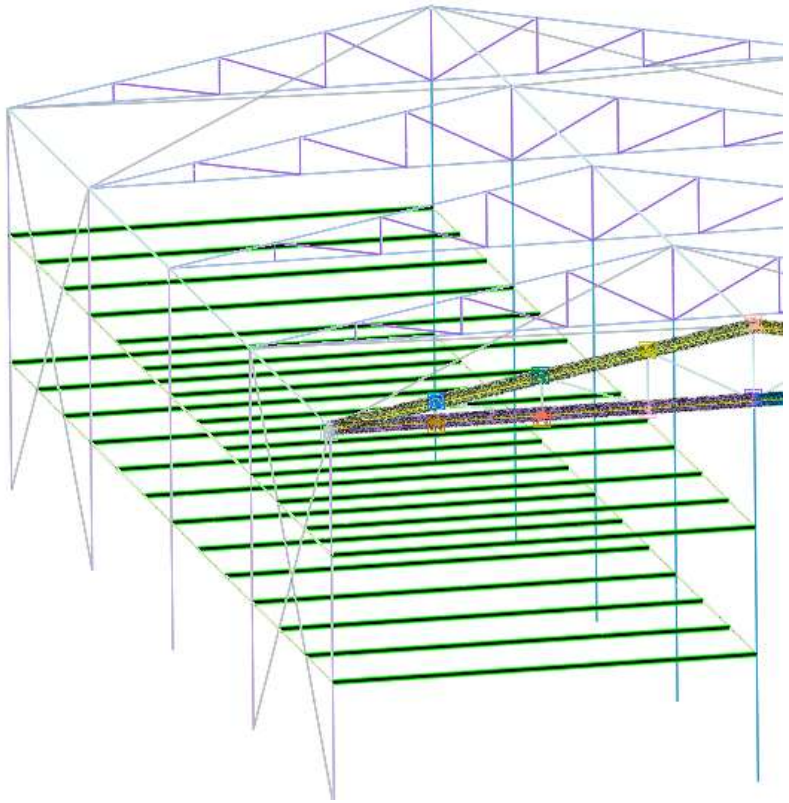


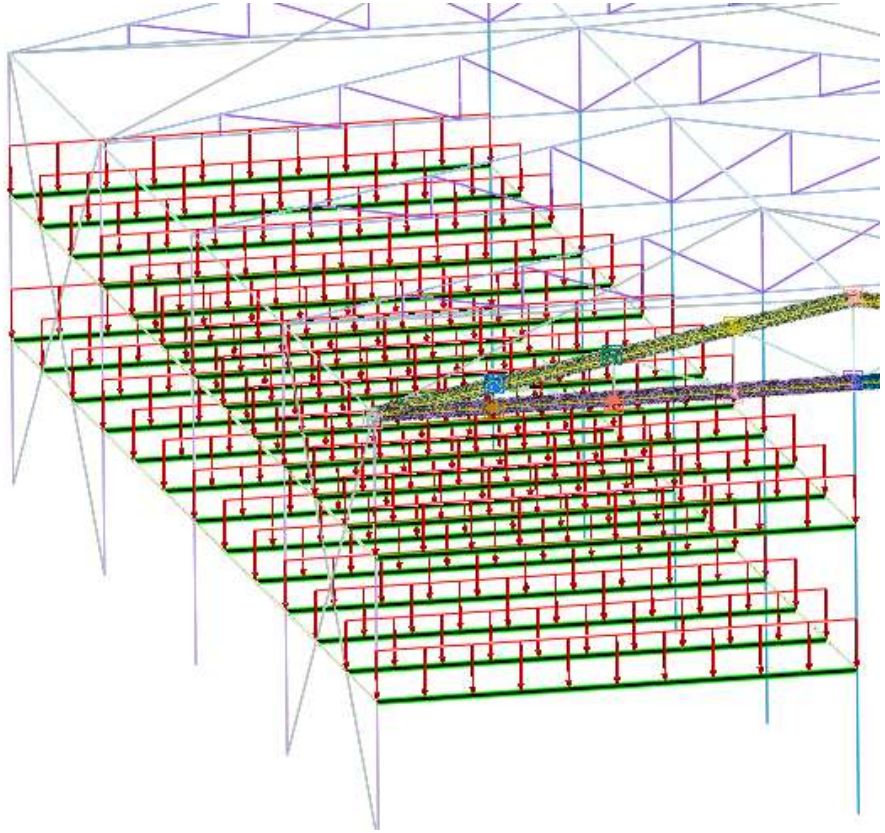


LIVE LOAD - SLAB

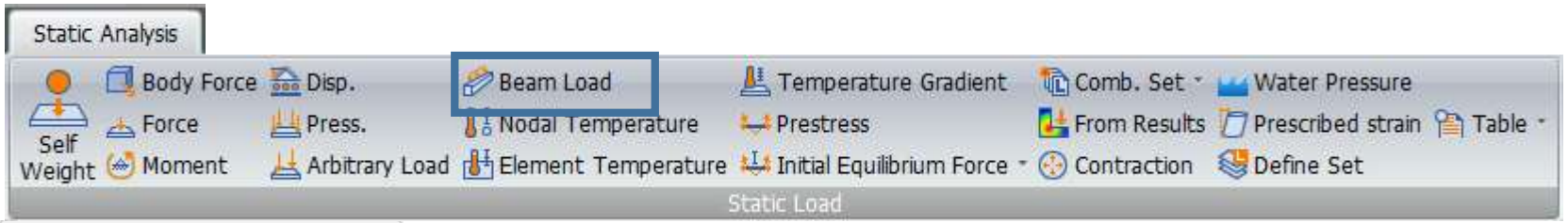


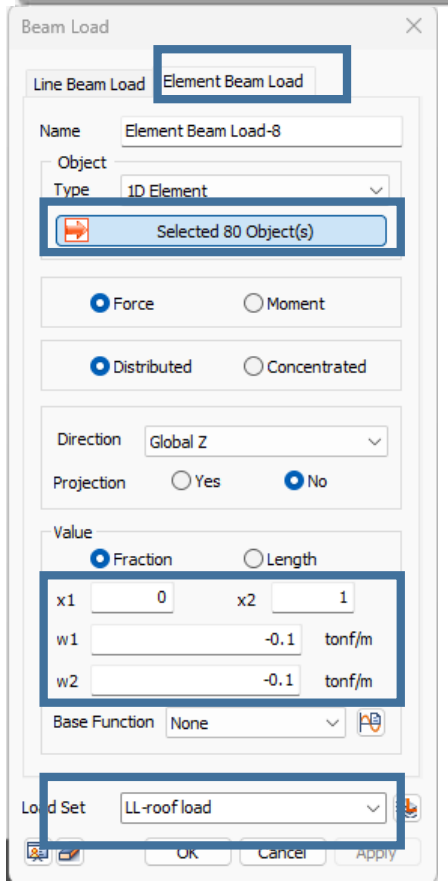


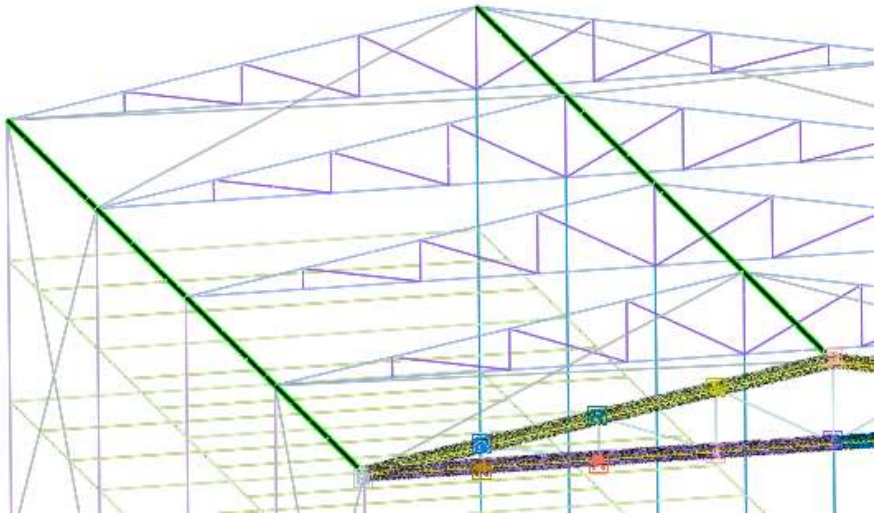


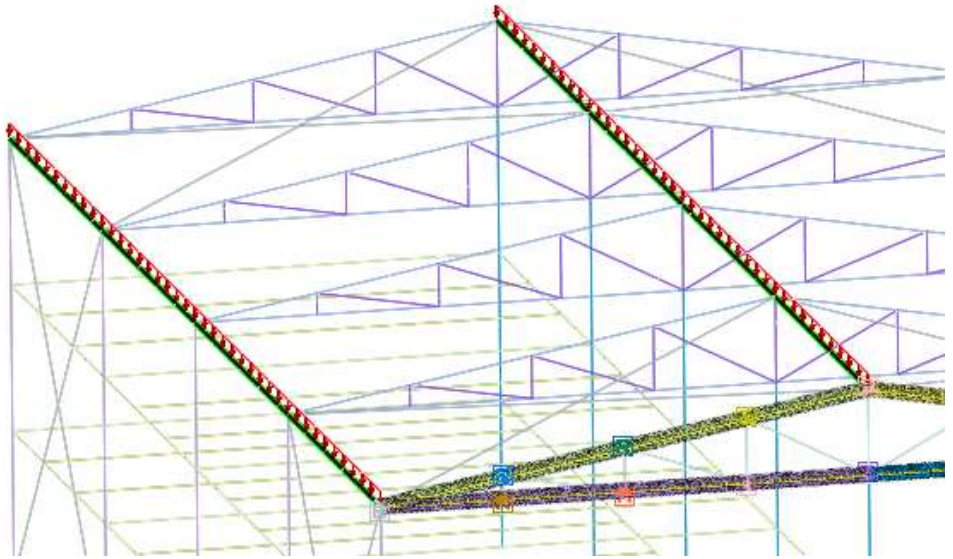


LIVE LOAD - ROOF LOAD

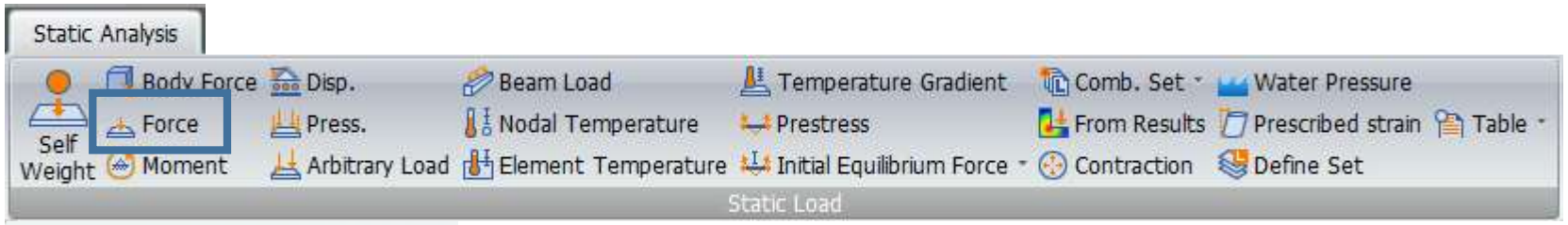


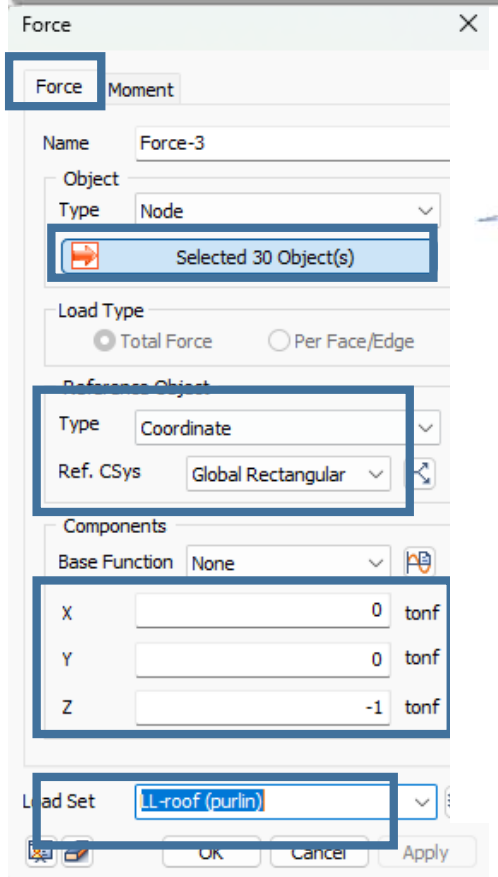


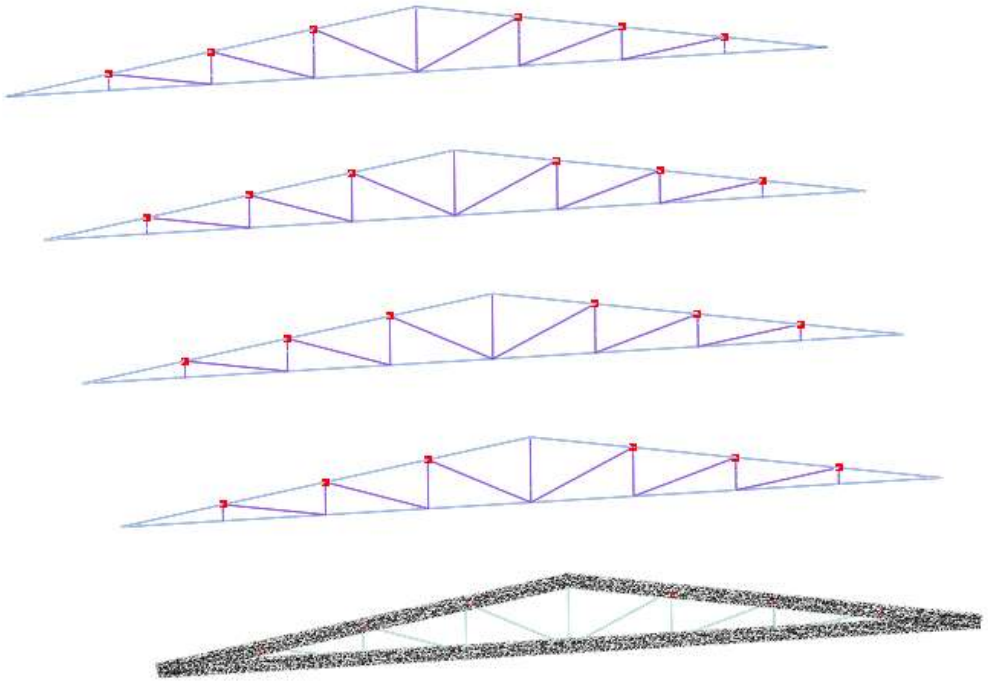


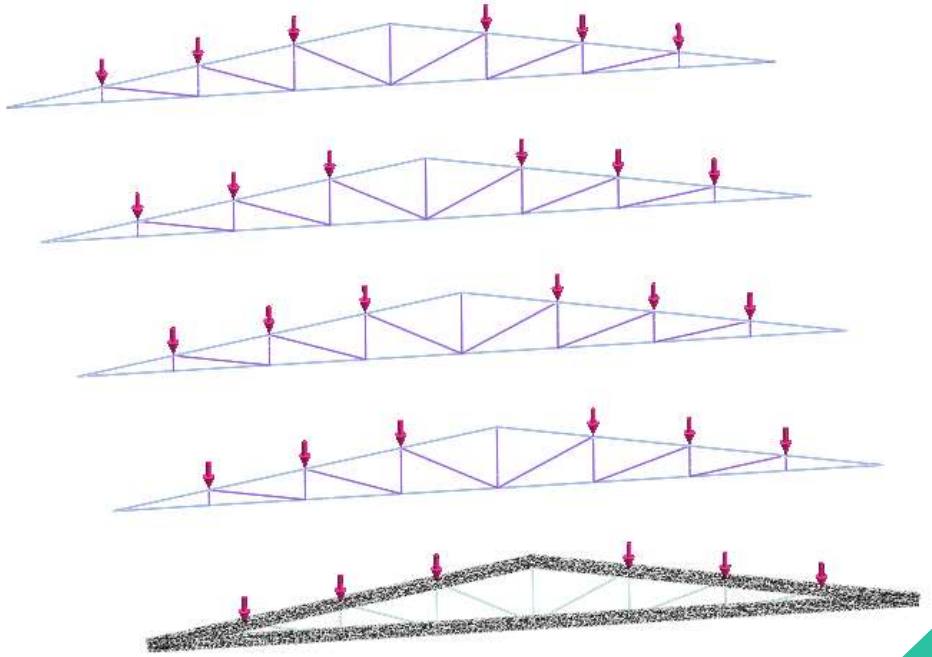


LIVE LOAD - PURLINS

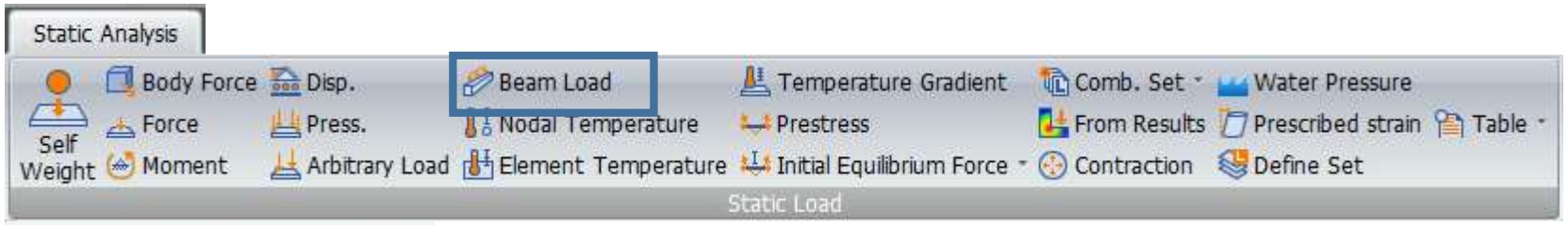


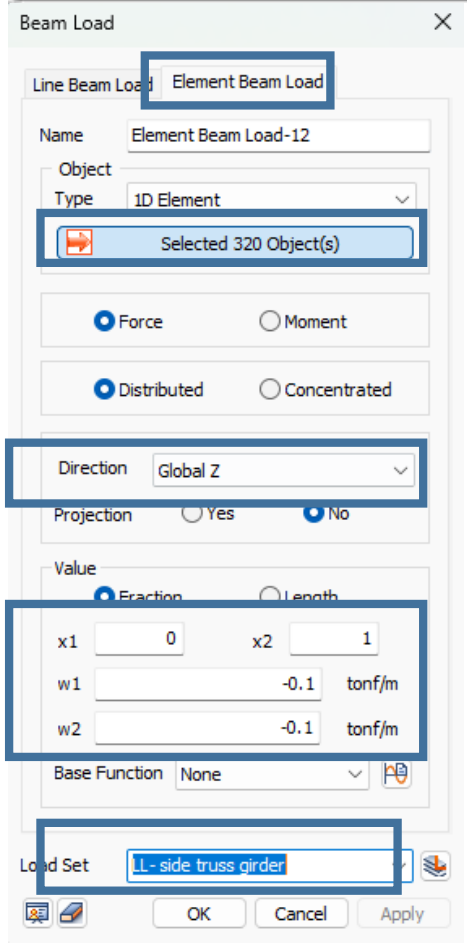


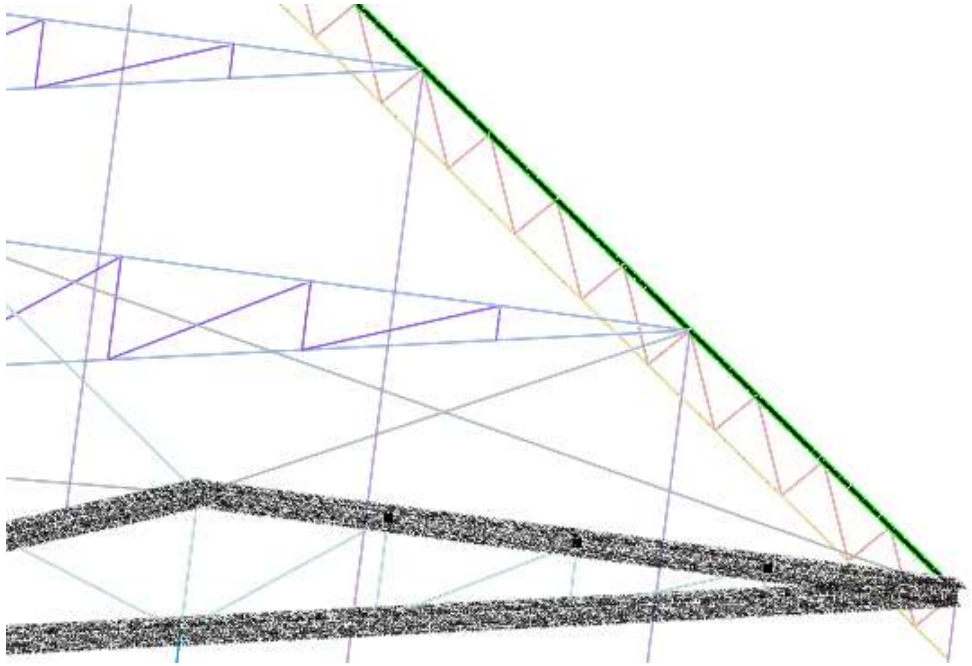


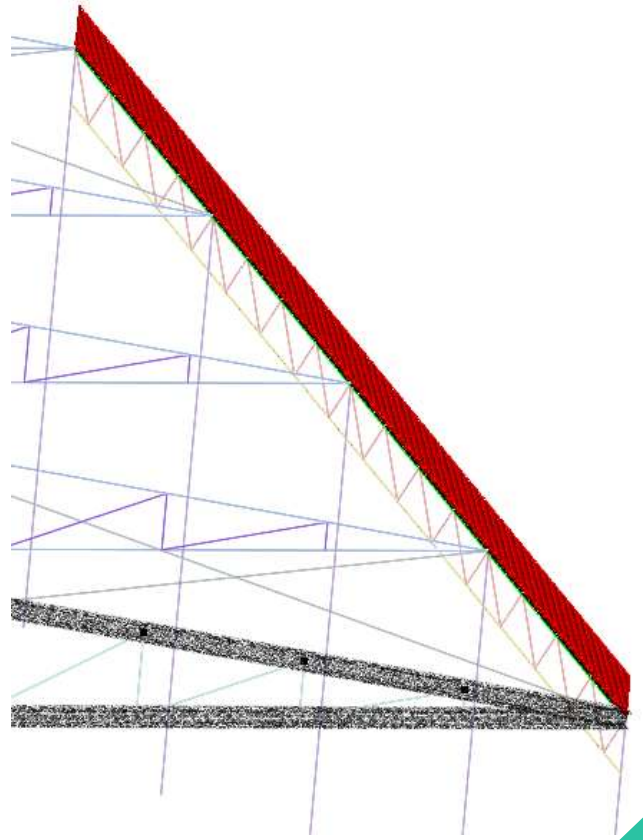


LIVE LOAD - SIDE TRUSS

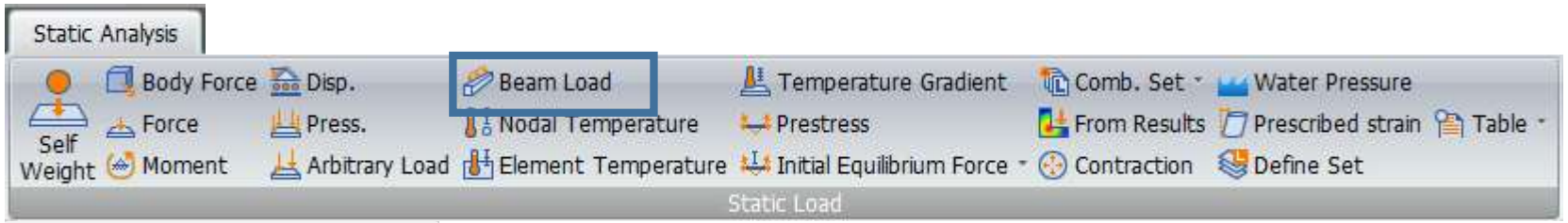


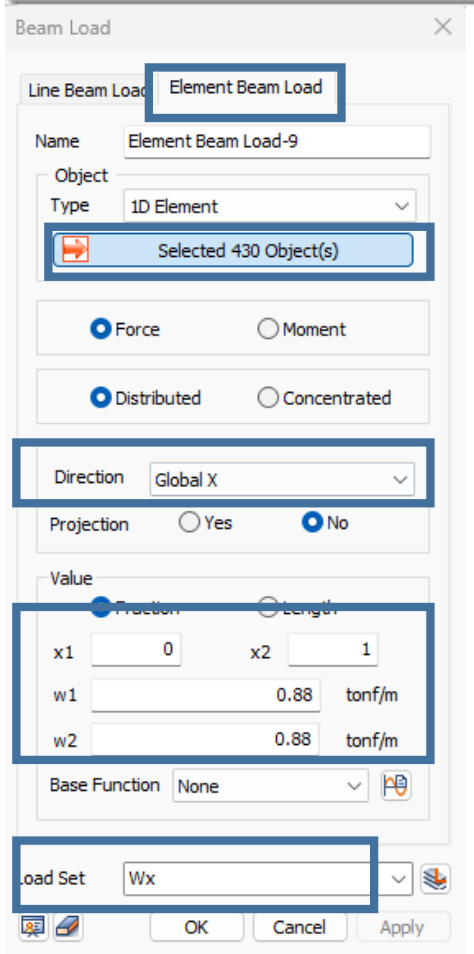


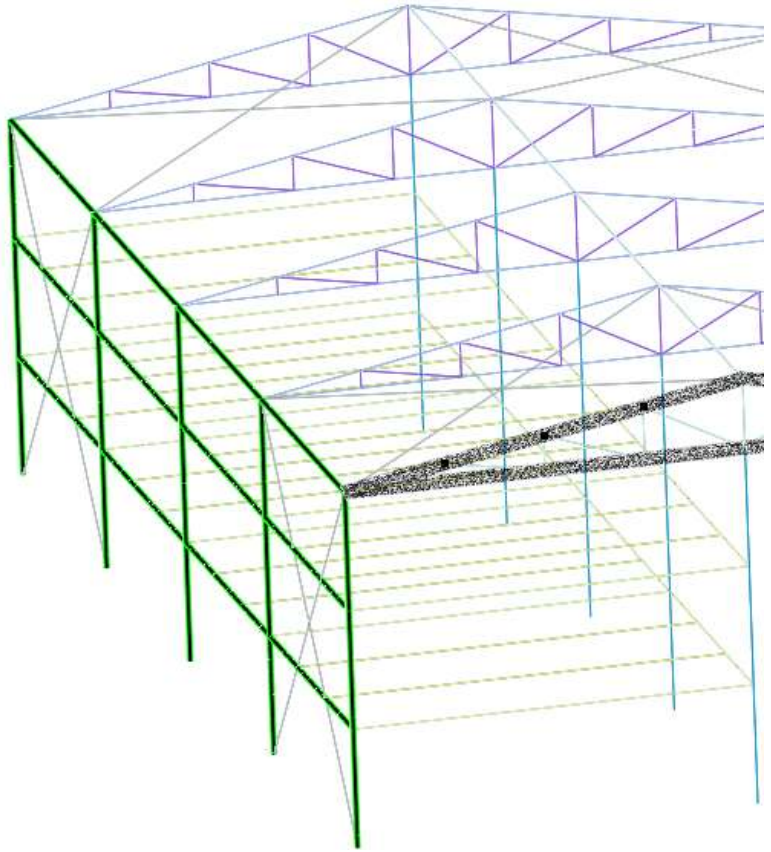


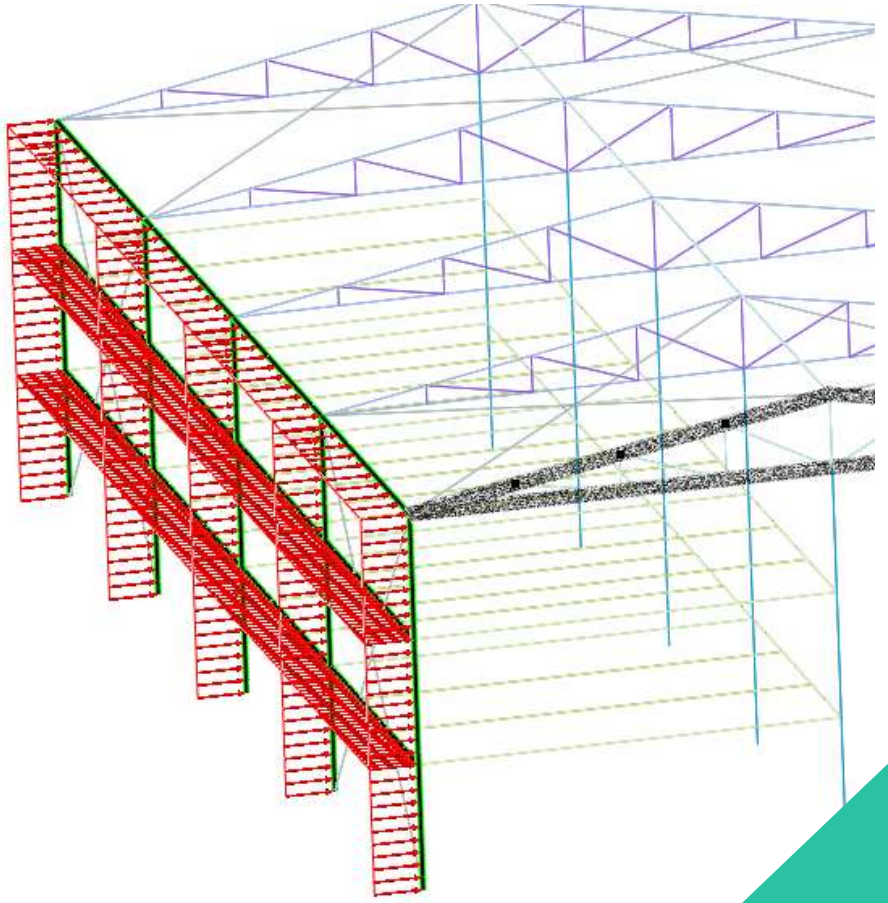


WIND LOAD Wx

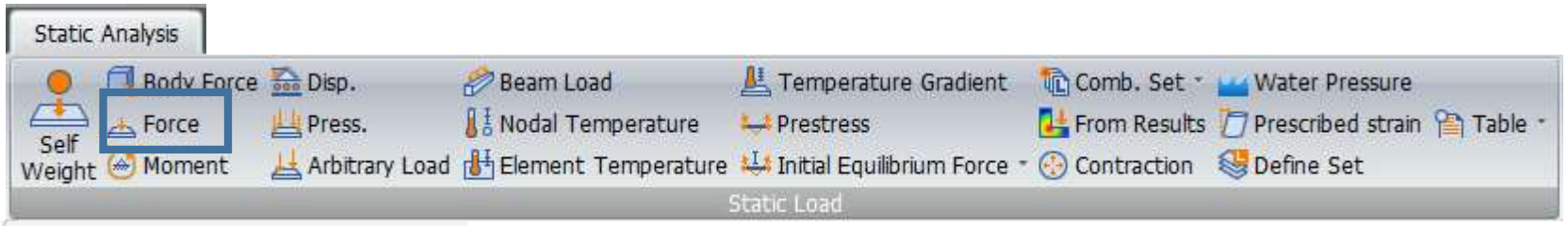

Static Analysis
Body Force
Disp.
Beam Load
Temperature Gradient
Comb. Set
Water Pressure
Force
Press.
Nodal Temperature
Prestress
From Results
Prescribed strain
Table
Self Weight
Moment
Arbitrary Load
Element Temperature
Initial Equilibrium Force
Contraction
Define Set
Static Load

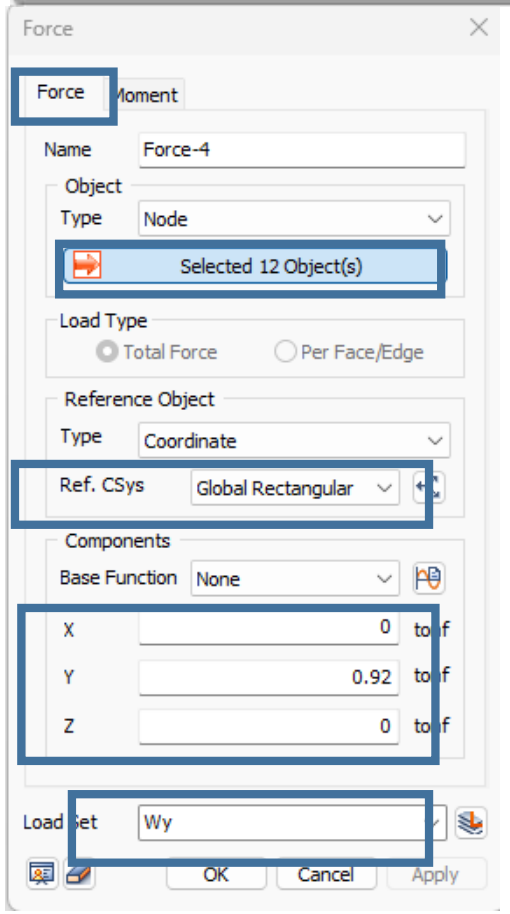

Beam Load
Line Beam Load
Element Beam Load
Name
Element Beam Load-9
Object
Type
1D Element
Selected 430 Object(s)
Force
Moment
Distributed
Concentrated
Direction
Global X
Projection
Yes
No
Value
x1
0
x2
1
w1
0.88
tonf/m
w2
0.88
tonf/m
Base Function
None
Load Set
Wx
OK
Cancel
Apply

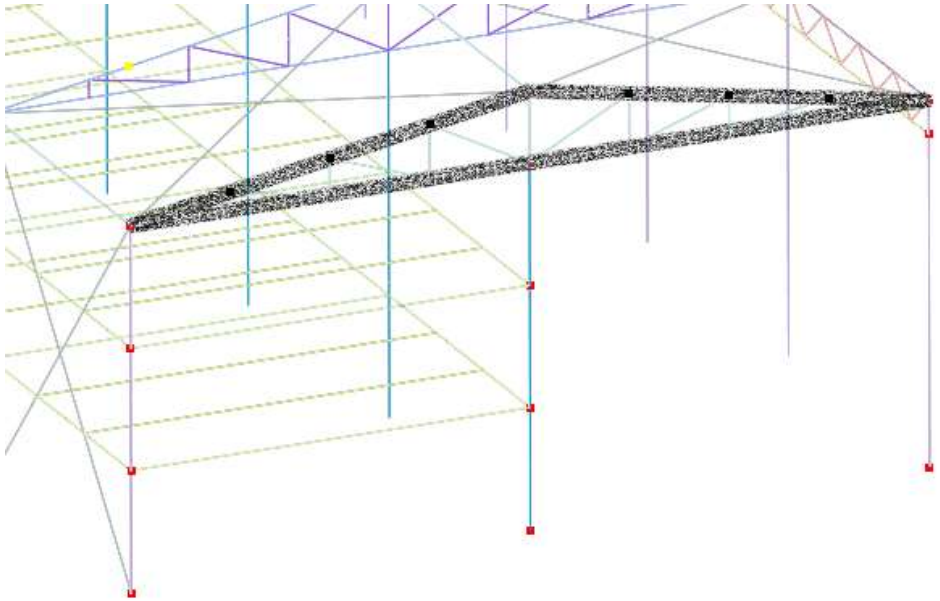


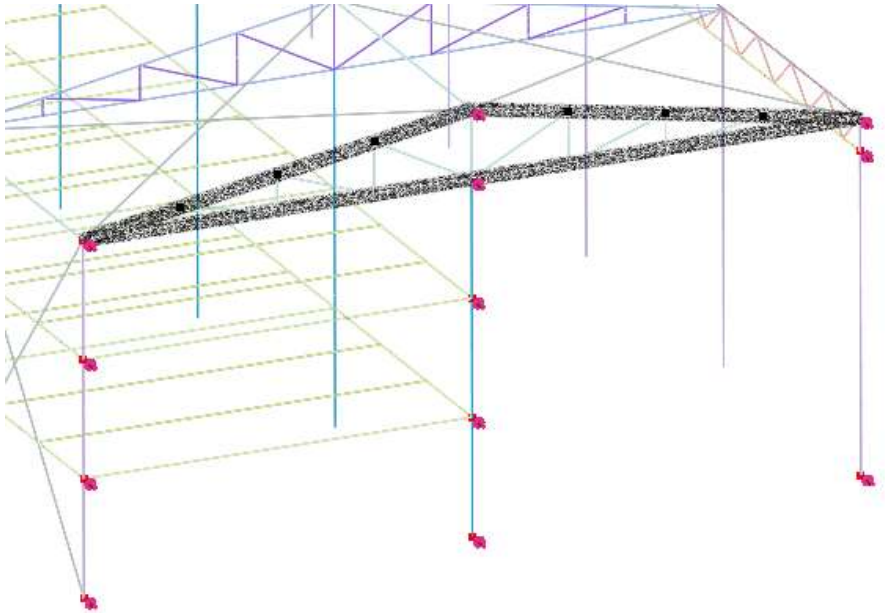


WIND LOAD Wy

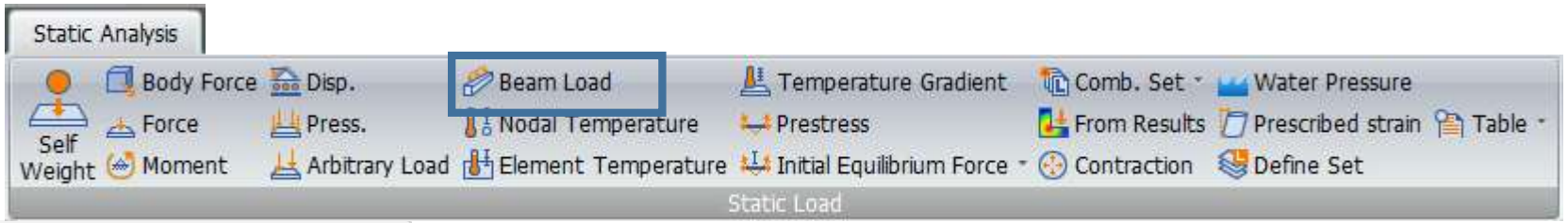

Static Analysis
Body Force
Disp.
Beam Load
Temperature Gradient
Comb. Set
Water Pressure
Self Weight
Force
Press.
Nodal Temperature
Prestress
From Results
Prescribed strain
Table
Moment
Arbitrary Load
Element Temperature
Initial Equilibrium Force
Contraction
Define Set
Static Load

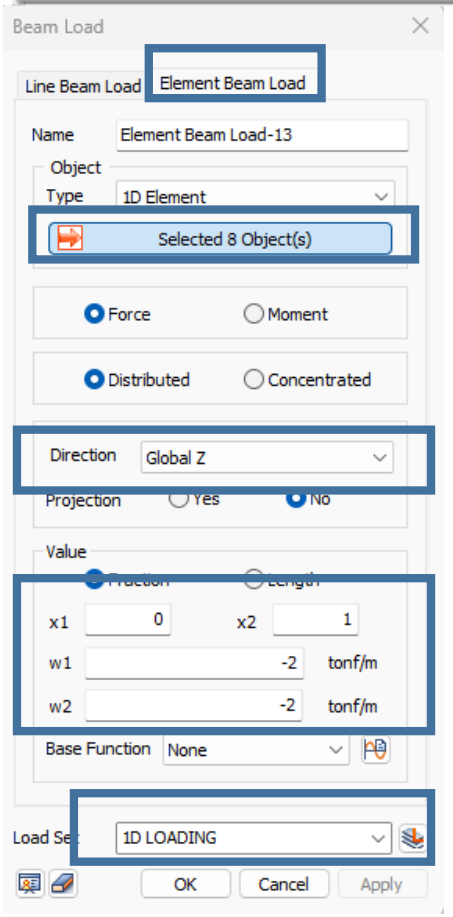

Force
Force
Moment
Name
Force-4
Object
Type
Node
Selected 12 Object(s)
Load Type
Total Force
Per Face/Edge
Reference Object
Type
Coordinate
Ref. CSys
Global Rectangular
Components
Base Function
None
X
0
Y
0.92
Z
0
Load Set
Wy
OK
Cancel
Apply

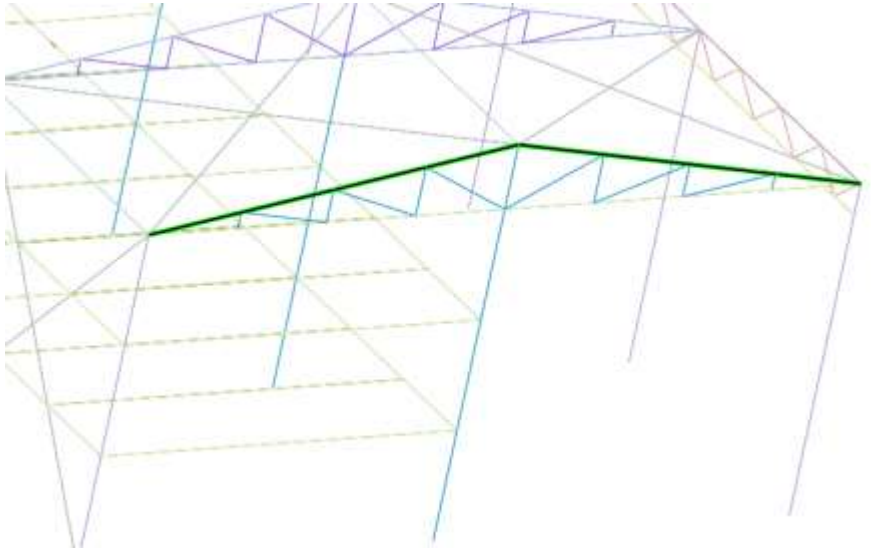


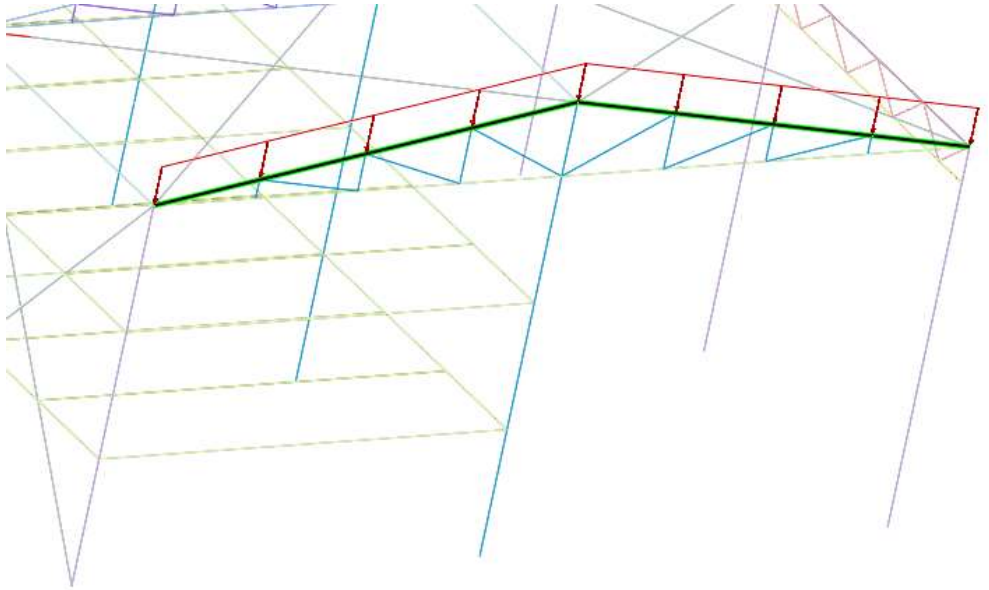


1D LOADING

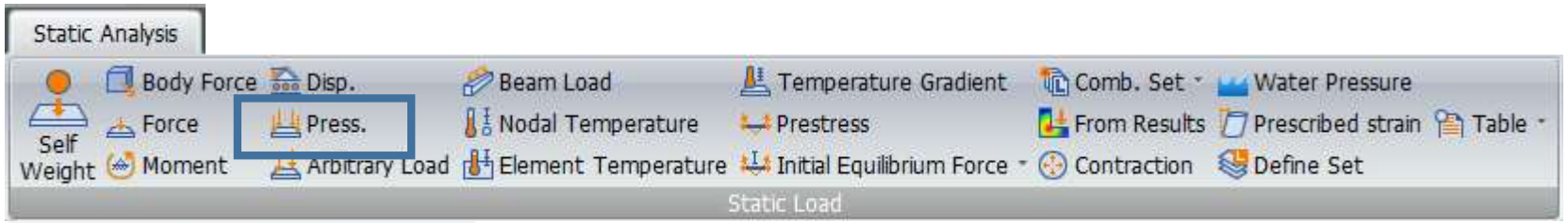

Static Analysis
Body Force
Disp.
Beam Load
Temperature Gradient
Comb. Set
Water Pressure
Force
Press.
Nodal Temperature
Prestress
From Results
Prescribed strain
Table
Self Weight
Moment
Arbitrary Load
Element Temperature
Initial Equilibrium Force
Contraction
Define Set
Static Load

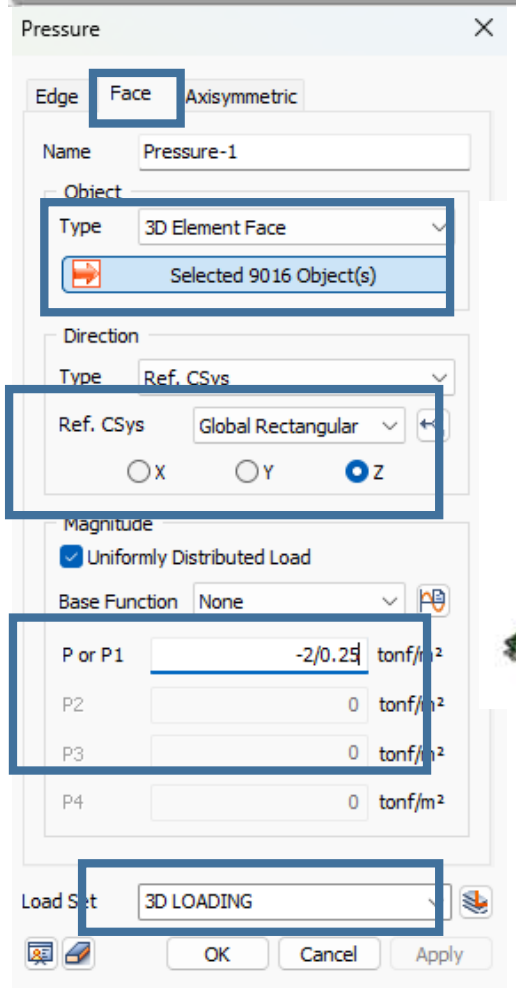

Beam Load
Line Beam Load
Element Beam Load
Name
Element Beam Load-13
Object
Type
1D Element
Selected 8 Object(s)
Force
Moment
Distributed
Concentrated
Direction
Global Z
Projection
Yes
No
Value
x1
0
x2
1
w1
-2
tonf/m
w2
-2
tonf/m
Base Function
None
Load Set
1D LOADING
OK
Cancel
Apply

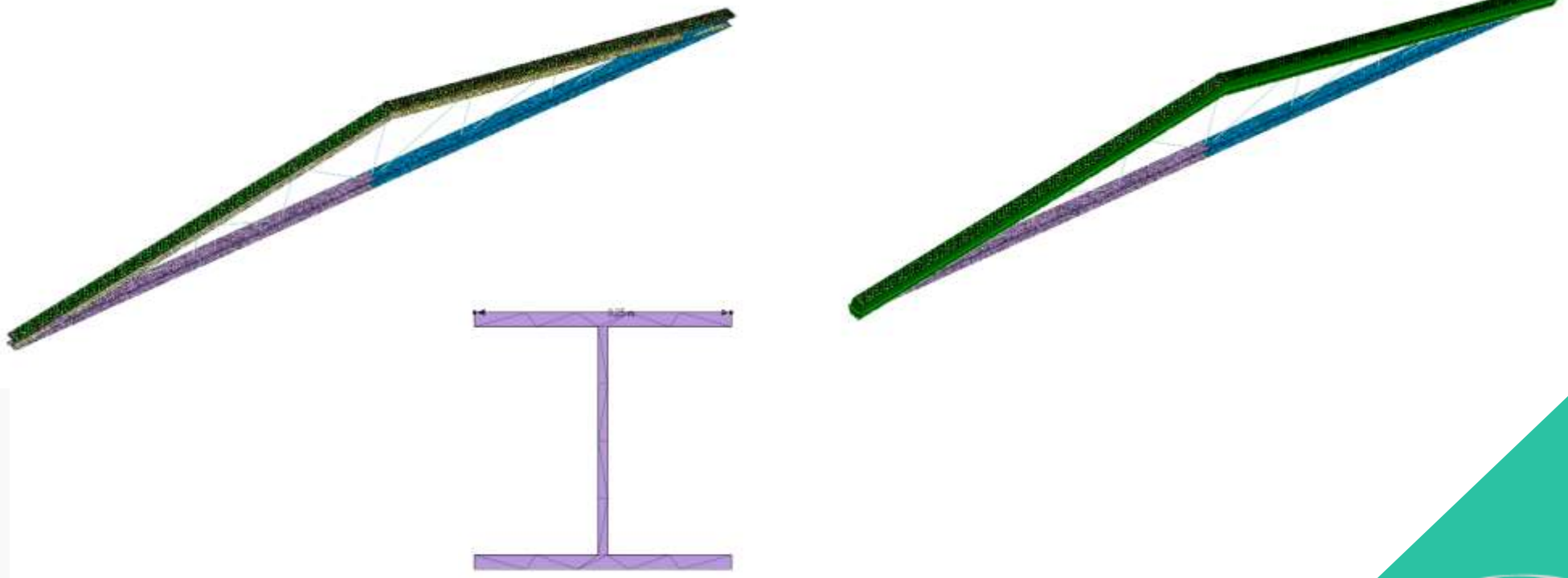




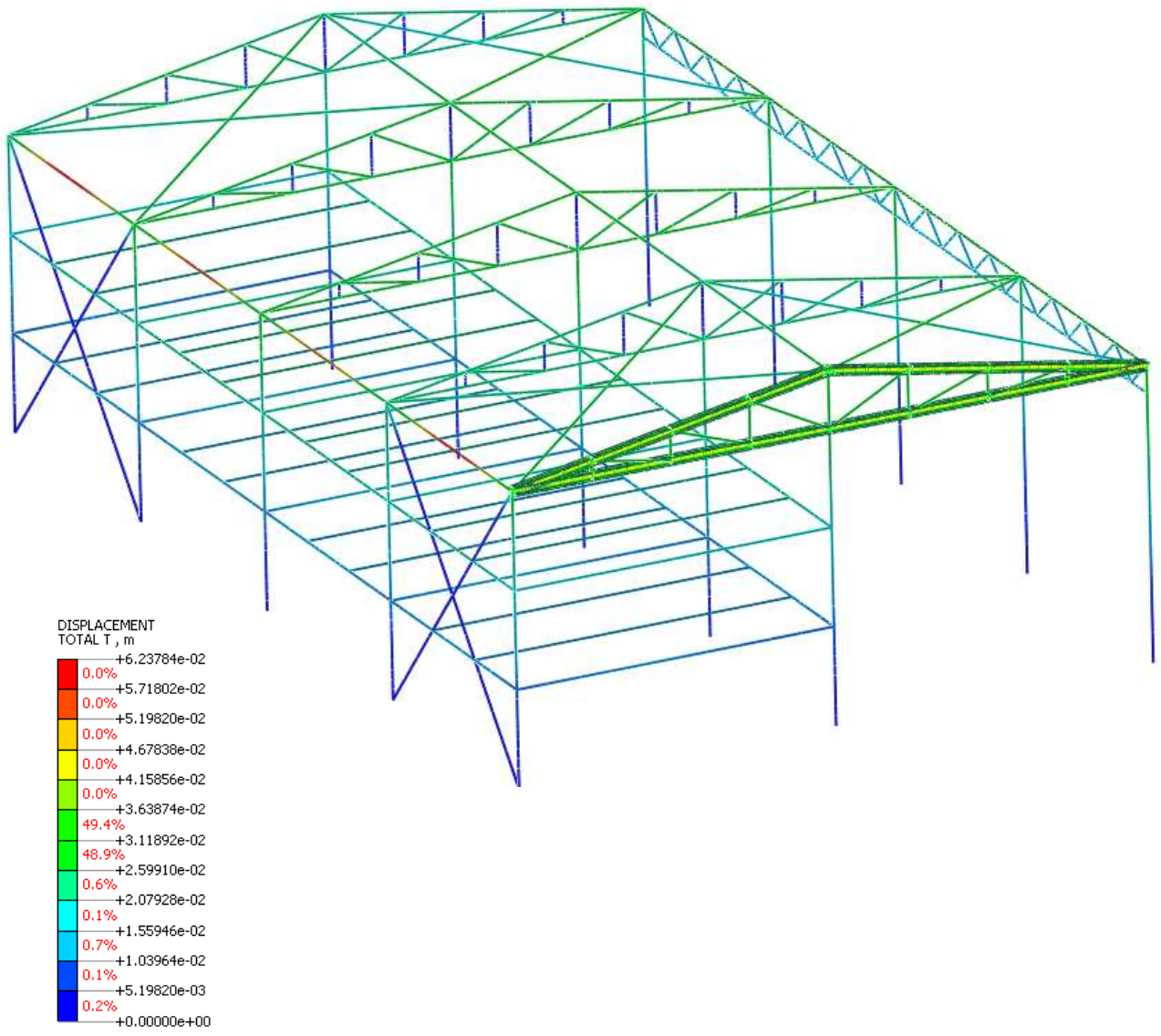
3D LOADING

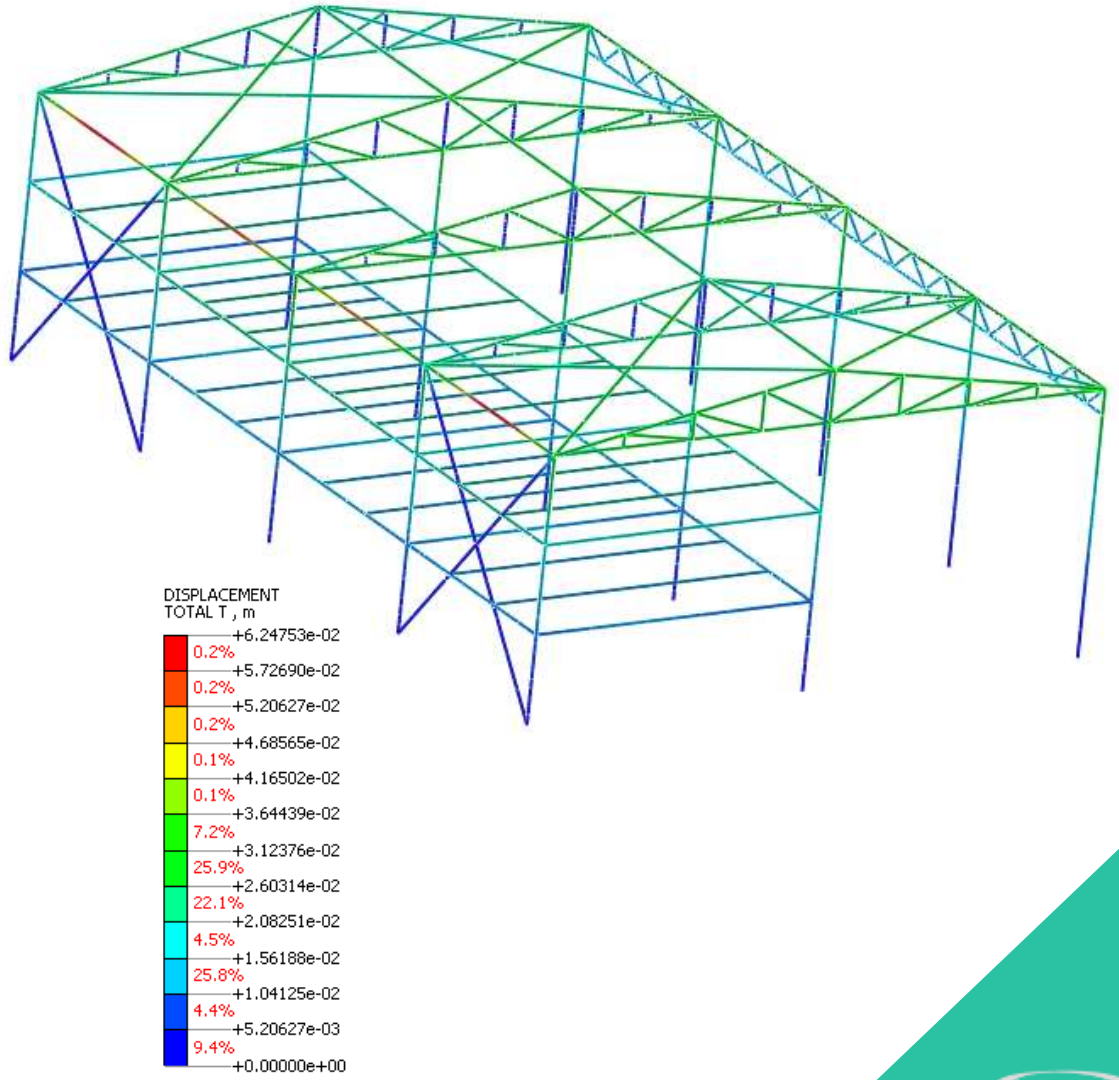

Static Analysis
Body Force
Disp.
Beam Load
Temperature Gradient
Comb. Set
Water Pressure
Force
Press.
Nodal Temperature
Prestress
From Results
Prescribed strain
Table
Self Weight
Moment
Arbitrary Load
Element Temperature
Initial Equilibrium Force
Contraction
Define Set
Static Load


Pressure
Edge
Face
Axisymmetric
Name
Pressure-1
Object
Type
3D Element Face
Selected 9016 Object(s)
Direction
Type
Ref. CSys
Ref. CSys
Global Rectangular
X
Y
Z
Magnitude
Uniformly Distributed Load
Base Function
None
P or P1
-2/0.25
tonf/m²
P2
0
tonf/m²
P3
0
tonf/m²
P4
0
tonf/m²
Load Set
3D LOADING
OK
Cancel
Apply



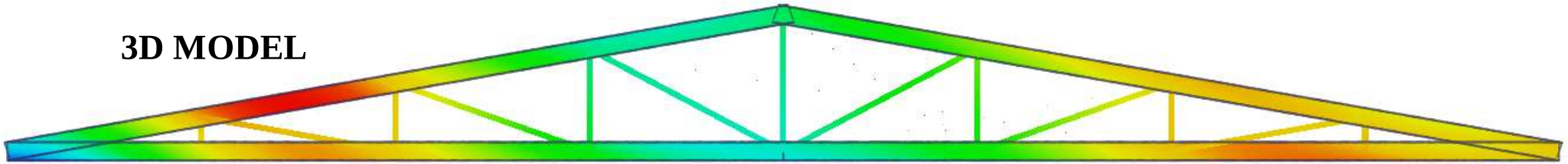
RESULT COMPARISON



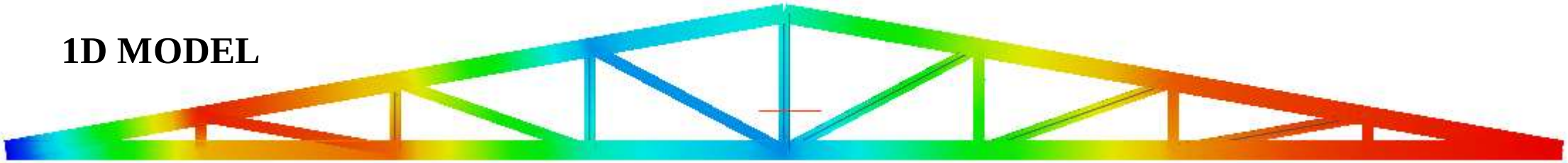


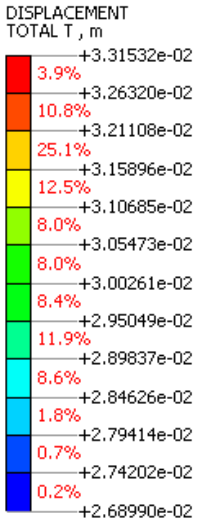
RESULT COMPARISON

3D MODEL

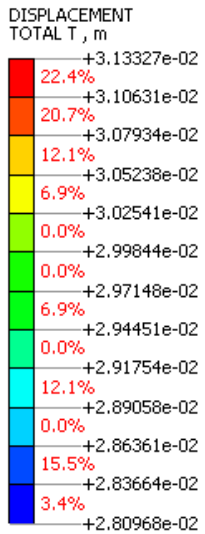


1D MODEL





DISPLACEMENT

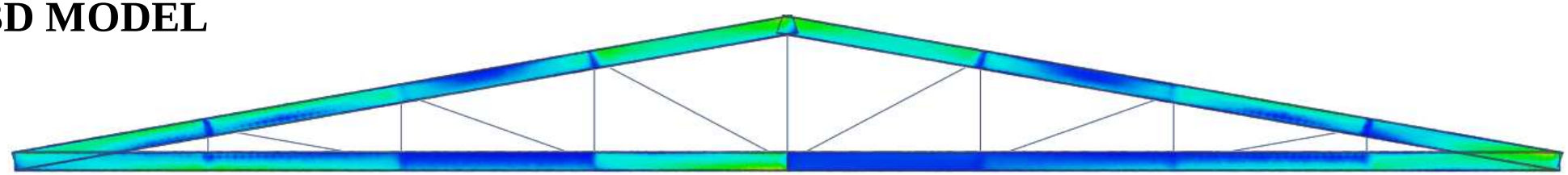


3D MODEL

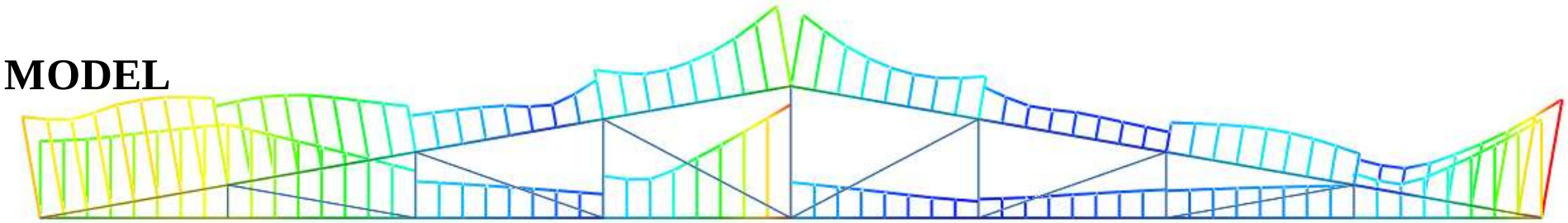
1D MODEL

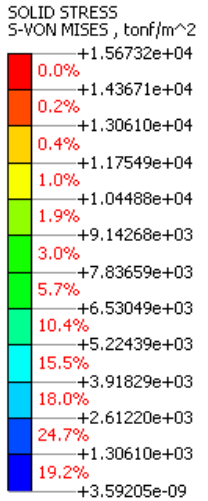
RESULT COMPARISON

3D MODEL

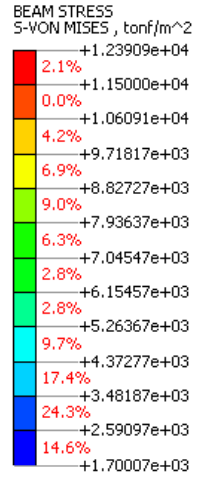


1D MODEL





STRESS VON MISES



3D MODEL

1D MODEL