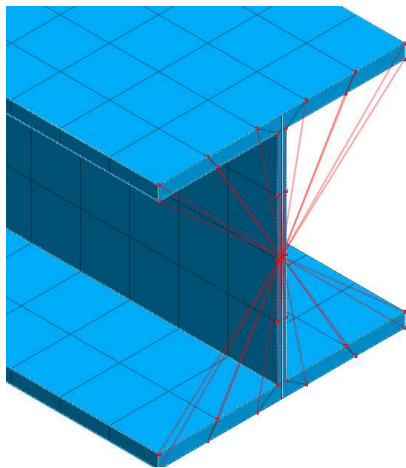
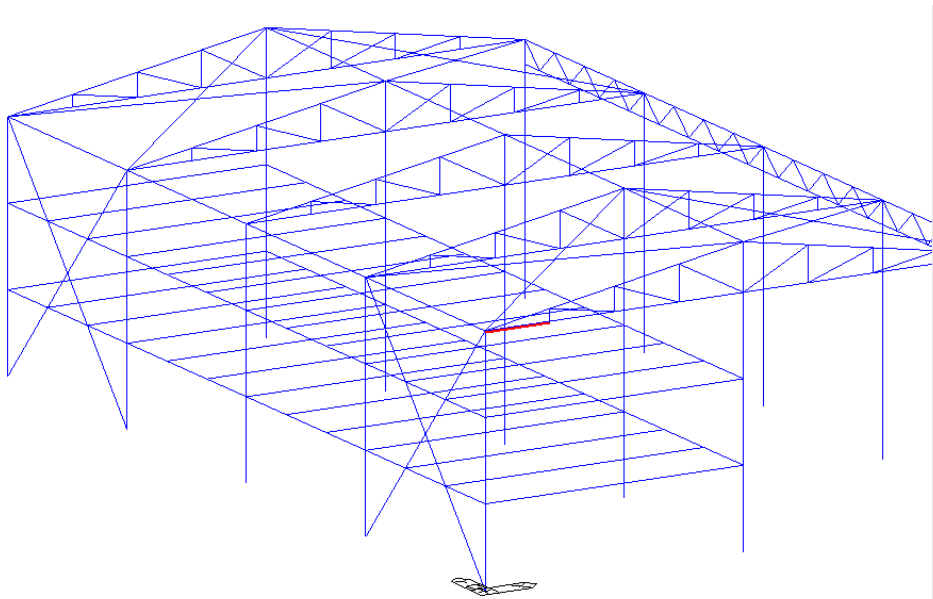


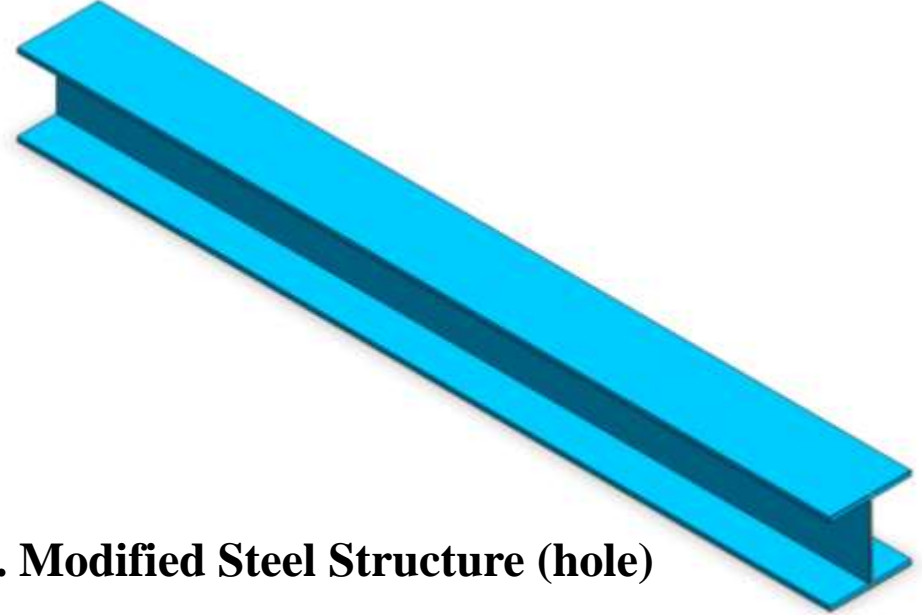
# FEA NX Simulation Platform

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

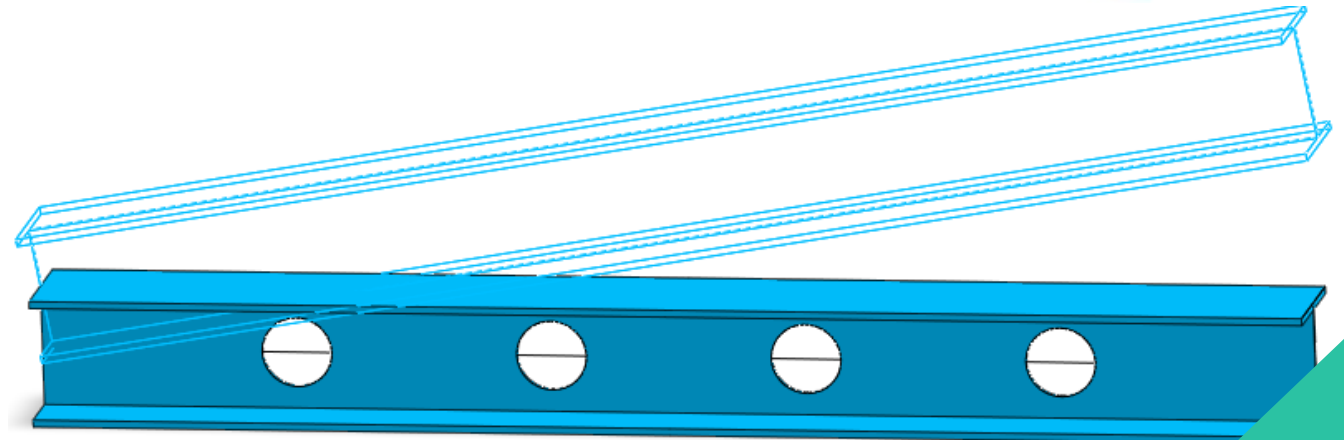
# CASE ANALYSIS OVERVIEW



## Part1. Solid Steel Structure



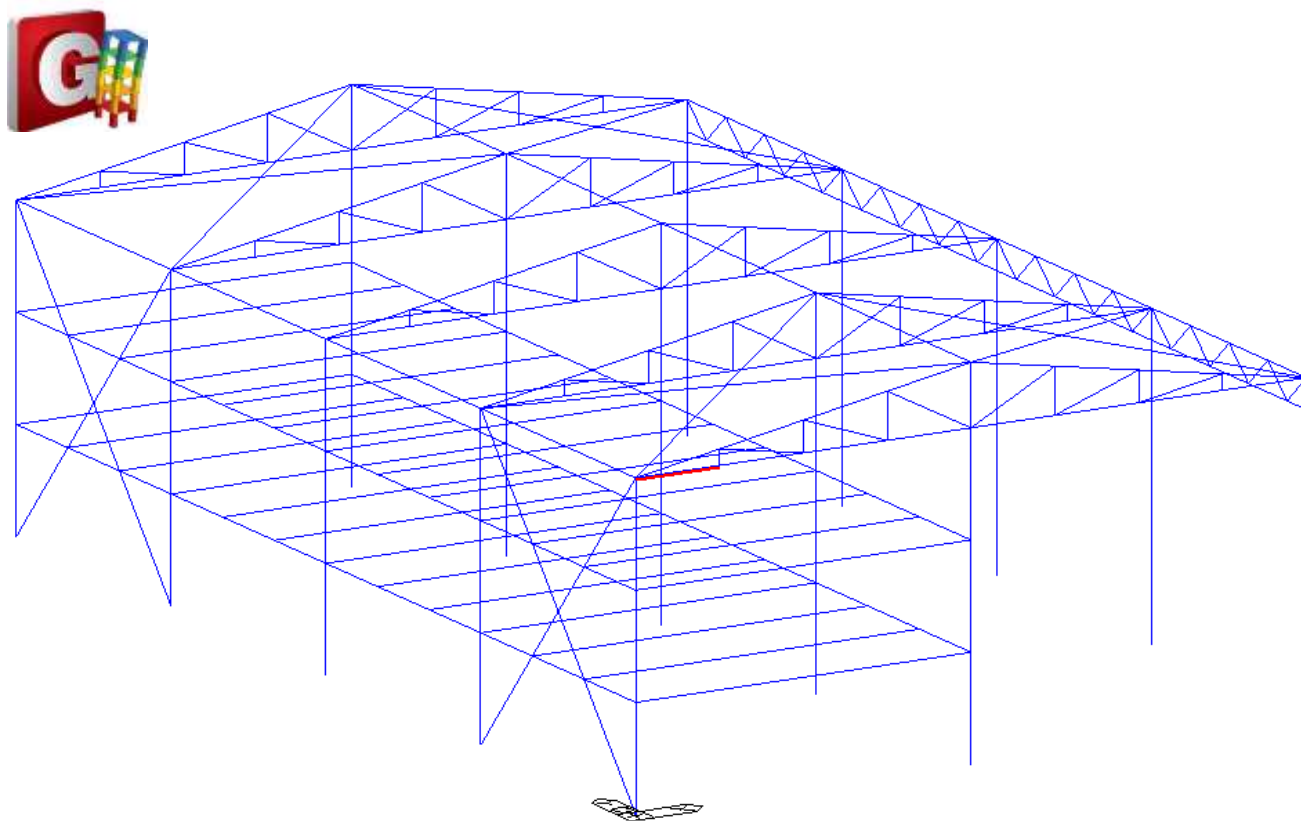
## Part2. Modified Steel Structure (hole)



# Part 0

## GEN Simulation and Results

# MIDAS GEN MODEL and RESULTS

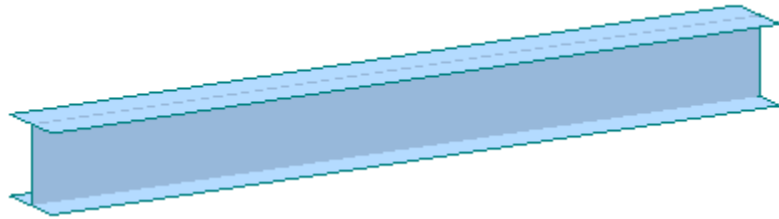
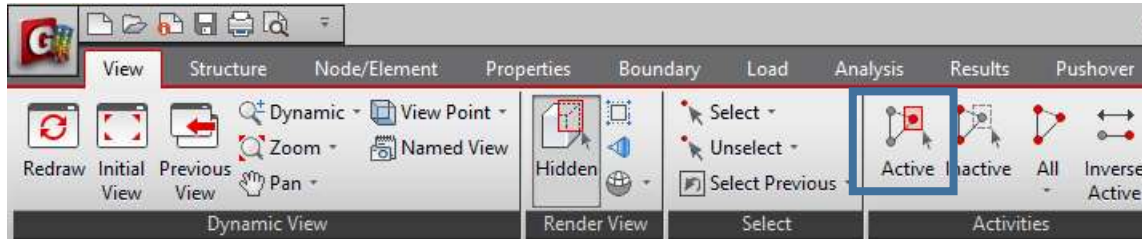
A screenshot of a Windows File Explorer window. The address bar shows the path: Local Disk (C:) > Program Files > MIDAS > midas Gen > Tutorial. The file list contains various tutorial files, with 'Tut2' highlighted. The table below represents the data shown in the file list.

Name	Date modified	Type	Size
App5_EC2 Design-start model	7/31/2008 11:46 AM	MIDAS/GENw Doc...	285 KB
App6_Pushover_2D RC Structure	7/31/2008 11:36 AM	MIDAS/GENw Doc...	329 KB
App7_EC3 design_final model	12/22/2008 5:58 PM	MIDAS/GENw Doc...	346 KB
App7_EC3 design_start model	12/22/2008 5:58 PM	MIDAS/GENw Doc...	277 KB
App8_Steel Pushover Analysis_start	12/24/2008 11:04 AM	MIDAS/GENw Doc...	198 KB
App8_Steel Pushover Analysis_final	12/26/2008 10:40 AM	MIDAS/GENw Doc...	248 KB
App9_Meshed Slab and Wall Design	9/10/2010 3:58 PM	MIDAS/GENw Doc...	59 KB
App9_Meshed Slab and Wall Design_final	9/10/2010 4:31 PM	MIDAS/GENw Doc...	10,272 KB
App10_Cracked Section Analysis	11/5/2010 11:47 AM	MIDAS/GENw Doc...	1,748 KB
App10_Cracked Section Analysis_final	11/5/2010 11:44 AM	MIDAS/GENw Doc...	2,514 KB
App11_Capacity Design	2/28/2011 5:51 PM	MIDAS/GENw Doc...	503 KB
App11_Capacity Design_final	2/28/2011 5:51 PM	MIDAS/GENw Doc...	508 KB
Tut1	5/4/2006 8:18 AM	MIDAS/GENw Doc...	50 KB
<b>Tut2</b>	5/26/2006 3:55 AM	MIDAS/GENw Doc...	188 KB
Tut3	5/4/2006 8:18 AM	MIDAS/GENw Doc...	173 KB
Tut4	5/4/2006 8:17 AM	MIDAS/GENw Doc...	904 KB

## Warehouse Frame Example

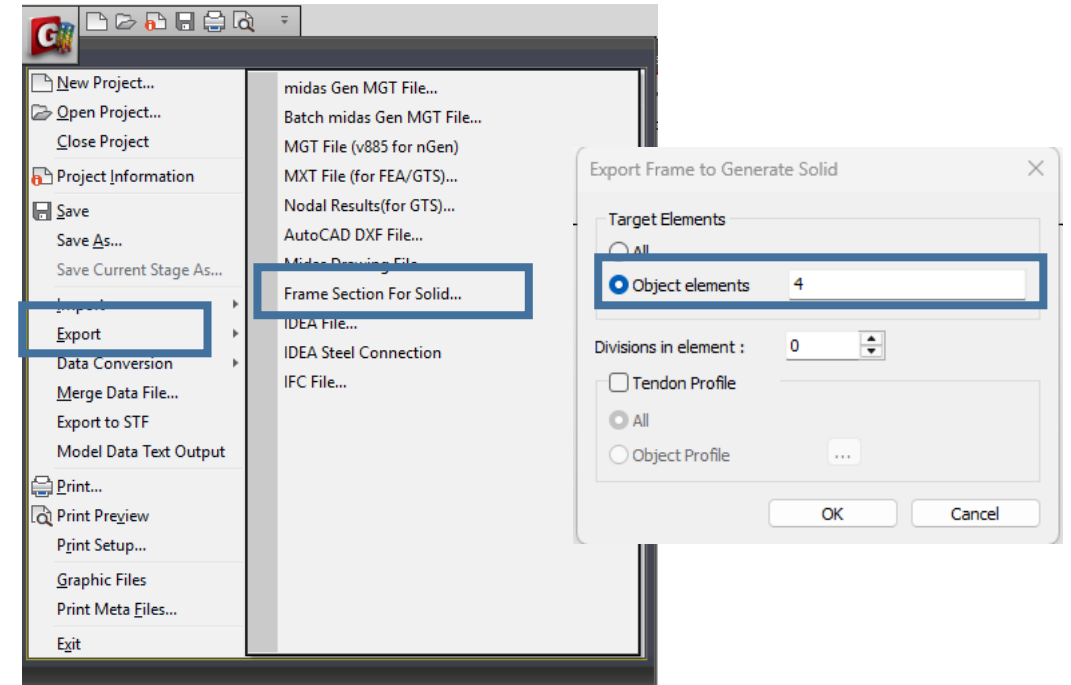
- Retrieve setting file from your MIDAS Gen Installation Folder
- Local Disk (C:) > Program Files > MIDAS > Midas Gen > Tutorial > **Tut2**

# CASE ANALYSIS 1: GEN Tutorial 2 Example



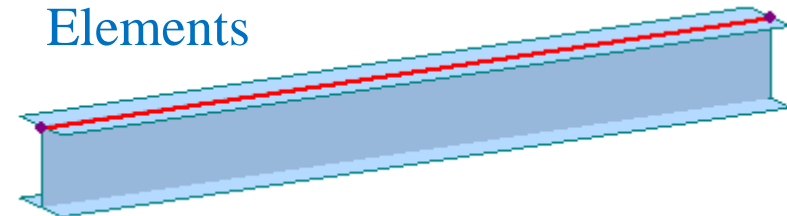
Isolating the element (beam)

- Highlight the beam
- Use Active

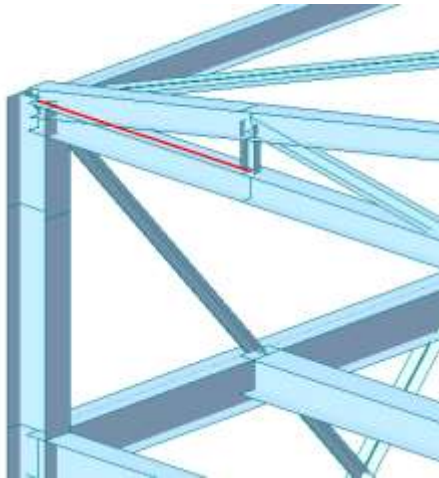
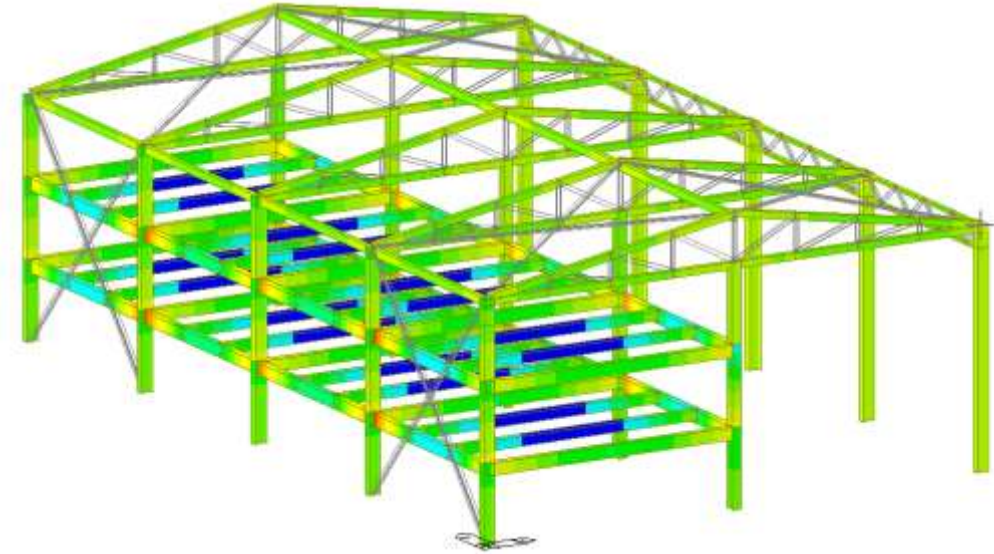
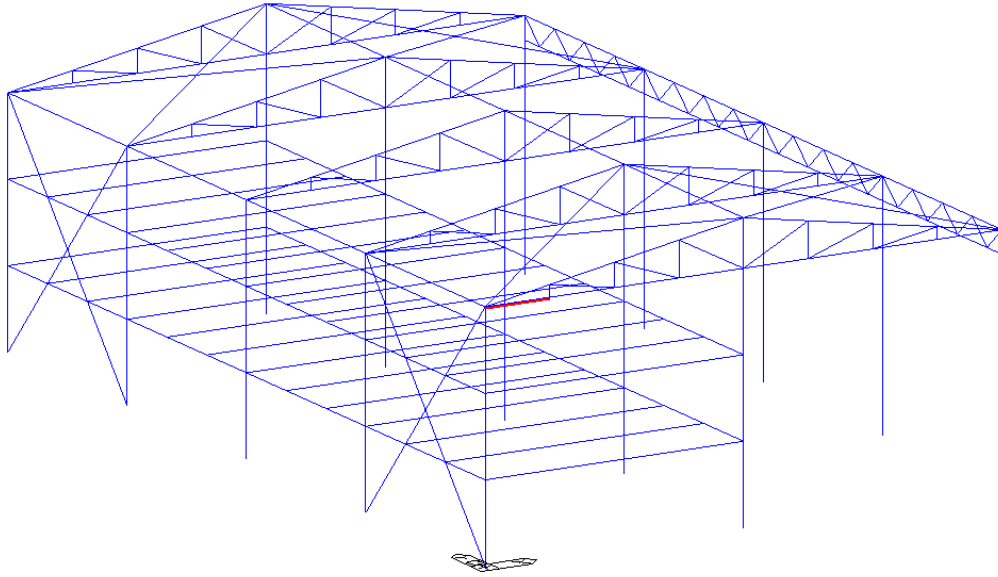


Extracting the element (beam)

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

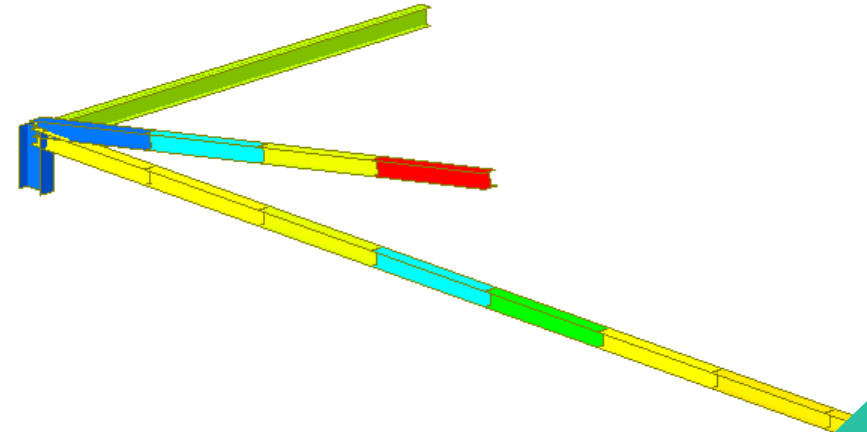
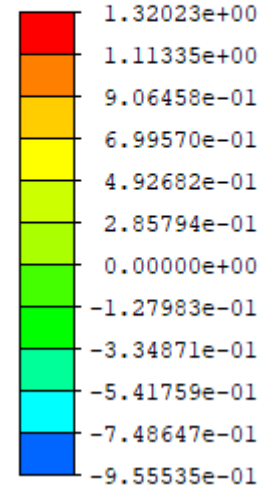


# MIDAS GEN MODEL and RESULTS



BEAM FORCE

AXIAL

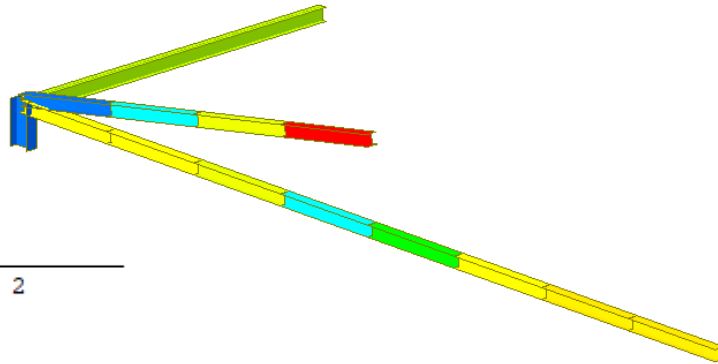




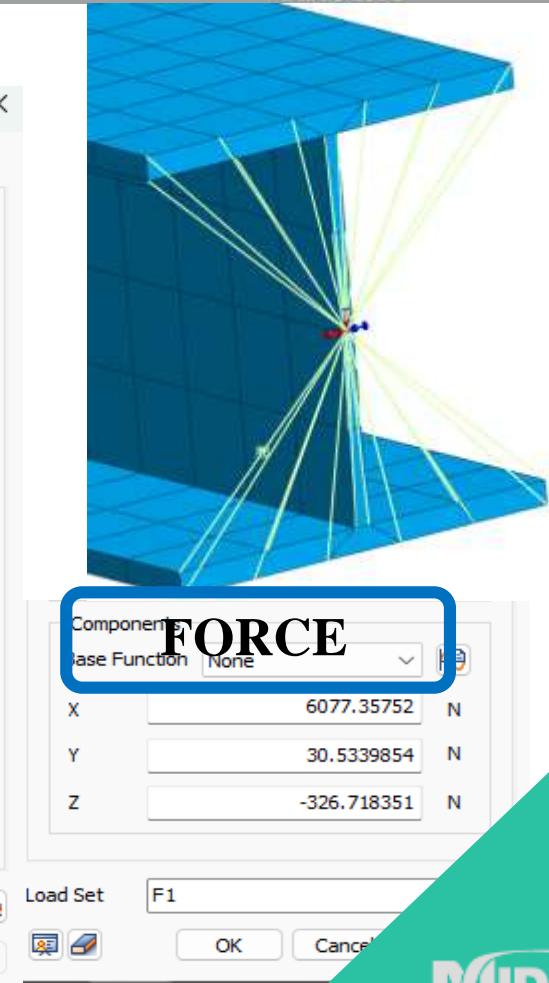
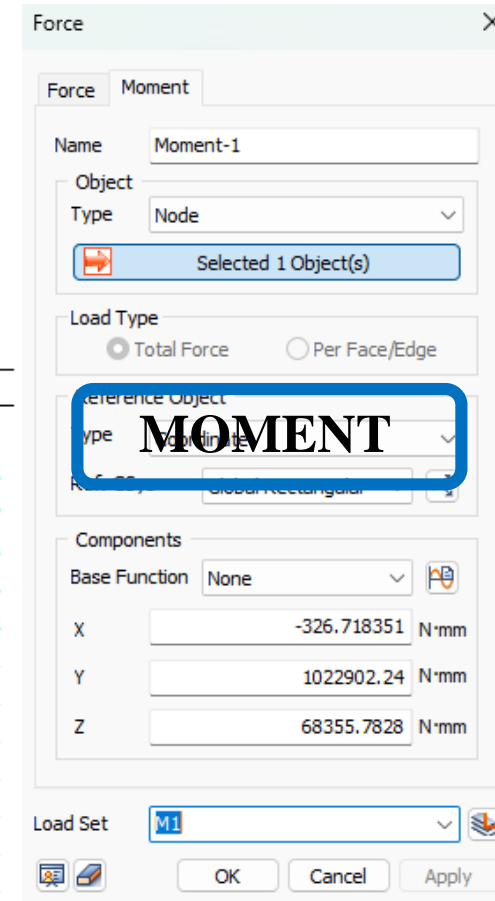
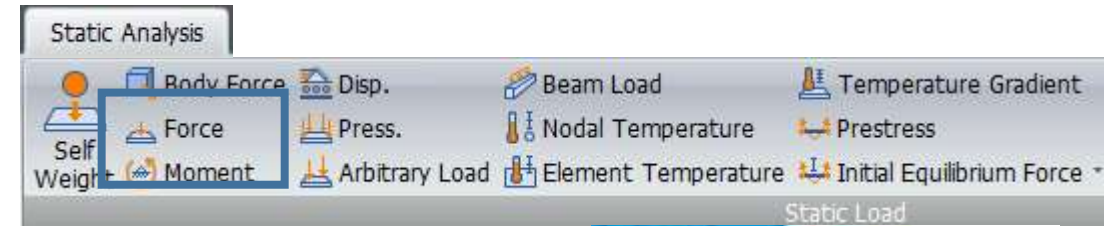
# LOADS and CONSTRAINTS

**LOAD: FORCE and MOMENT**

**CONSTRAINT: Pinned connection on the joint**



midas Gen POST-PROCESSOR BEAM FORCE	midas Gen POST-PROCESSOR BEAM FORCE	midas Gen POST-PROCESSOR BEAM FORCE	midas Gen POST-PROCESSOR BEAM FORCE	midas Gen POST-PROCESSOR BEAM FORCE	midas Gen POST-PROCESSOR BEAM FORCE
AXIAL	SHEAR-y	SHEAR-z	TORSION	MOMENT-y	MOMENT-z
1.32023e+00	2.28597e-01	2.29354e-01	3.46312e-05	4.02255e-01	1.82931e-02
1.11335e+00	2.07535e-01	1.86228e-01	2.17157e-05	3.37138e-01	0.00000e+00
9.06458e-01	1.86473e-01	1.43102e-01	8.80026e-06	2.72022e-01	-3.28748e-02
6.99570e-01	1.65411e-01	9.99755e-02	0.00000e+00	2.06906e-01	-5.84588e-02
4.92682e-01	1.44349e-01	5.68491e-02	-1.70307e-05	1.41789e-01	-8.40428e-02
2.85794e-01	1.23287e-01	0.00000e+00	-2.99462e-05	7.66728e-02	-1.09627e-01
0.00000e+00	1.02225e-01	-2.94035e-02	-4.28616e-05	0.00000e+00	-1.35211e-01
-1.27983e-01	8.11635e-02	-7.25299e-02	-5.57771e-05	-5.35601e-02	-1.60795e-01
-3.34871e-01	6.01016e-02	-1.15656e-01	-6.86926e-05	-1.18676e-01	-1.86379e-01
-5.41759e-01	3.90397e-02	-1.58783e-01	-8.16081e-05	-1.83793e-01	-2.11963e-01
-7.48647e-01	0.00000e+00	-2.01909e-01	-9.45235e-05	-2.48909e-01	-2.37547e-01
-9.55535e-01	-3.08411e-03	-2.45035e-01	-1.07439e-04	-3.14026e-01	-2.63131e-01



# Part 1

# **Solid Steel Structure**



# GEOMETRY

The screenshot displays the MIDAS software interface for creating a section geometry. The 'Section Data' dialog box is open, showing the 'DB/User' tab. The 'Section ID' is 6, and the 'Name' is 'RH 250x250x9x14'. The 'Sect. Name' is 'RH 250x250x9x14'. The 'DB Name' is 'AISC10(US)'. The 'Sect. Name' is empty. The 'Offset' is 'Center-Center'. The 'Consider Shear Deformation' checkbox is checked. The 'Consider Warping Effect(7th DOF)' checkbox is unchecked. The 'Show Calculation Results...' button is visible.

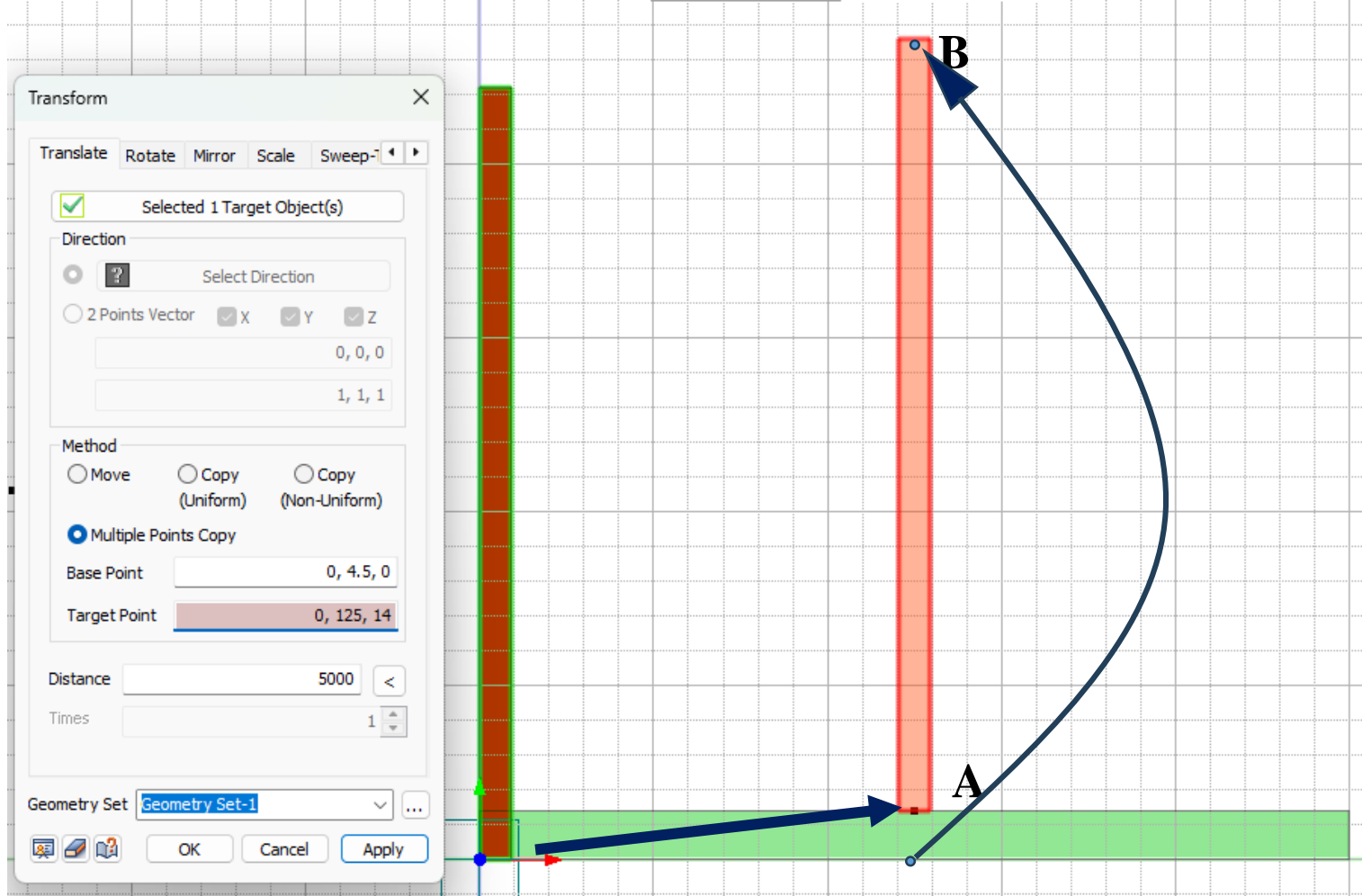
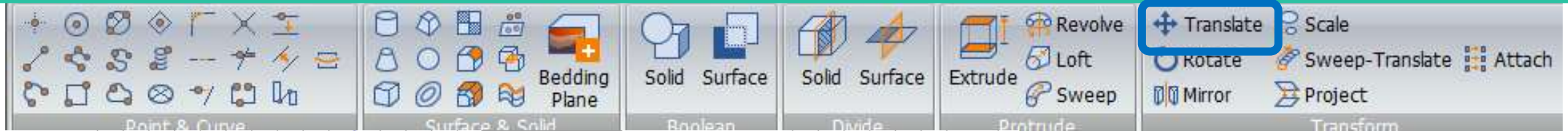
The 'Rectangle' dialog box is also open, showing the '2D' tab. The 'Method' is 'Input One Corner'. The 'Location' is '250,14'. The 'Method' is 'ABS x, y'. The 'Make Face' checkbox is checked. The 'Geometry Set' is 'Geometry Set-1'. The 'OK' button is visible.

The 'Rectangle' dialog box is also open, showing the '2D' tab. The 'Method' is 'Input Diagonally Opposite Corner'. The 'Location' is '9,222'. The 'Method' is 'REL dx, dy'. The 'Make Face' checkbox is checked. The 'Geometry Set' is 'Geometry Set-1'. The 'OK' button is visible.

Following the section database for **RH section**  
Create 2 rectangles | Axis Y,Z | Origin (0,0)

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

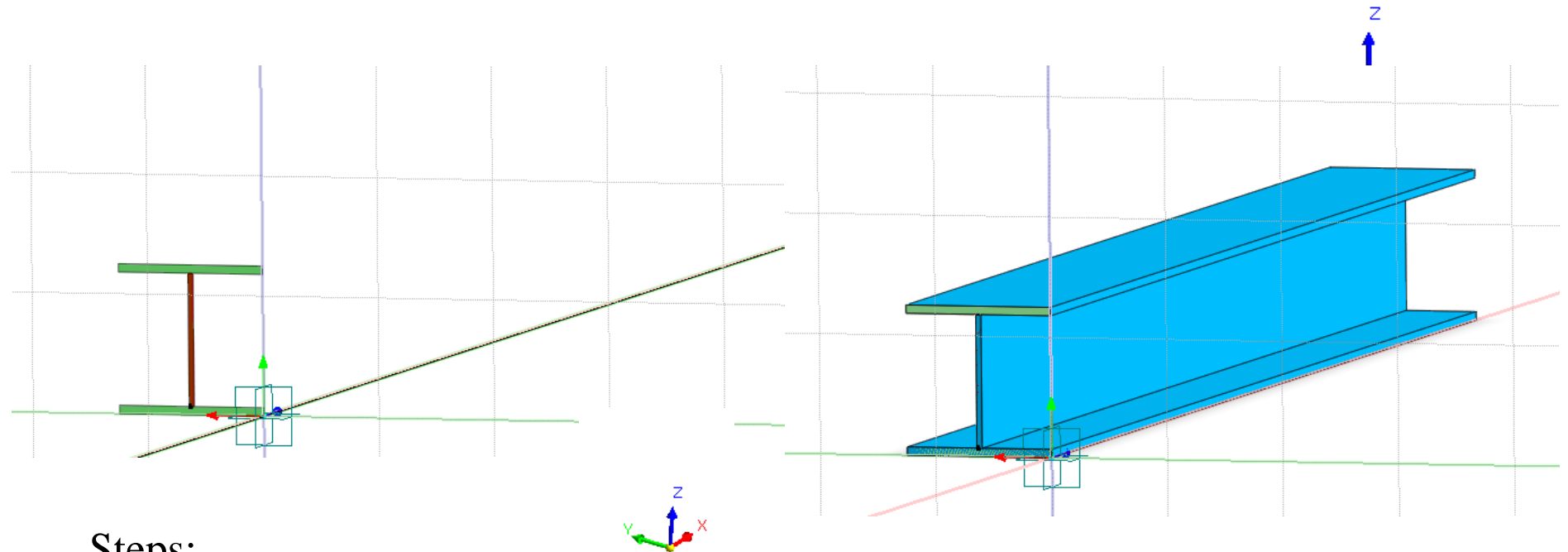
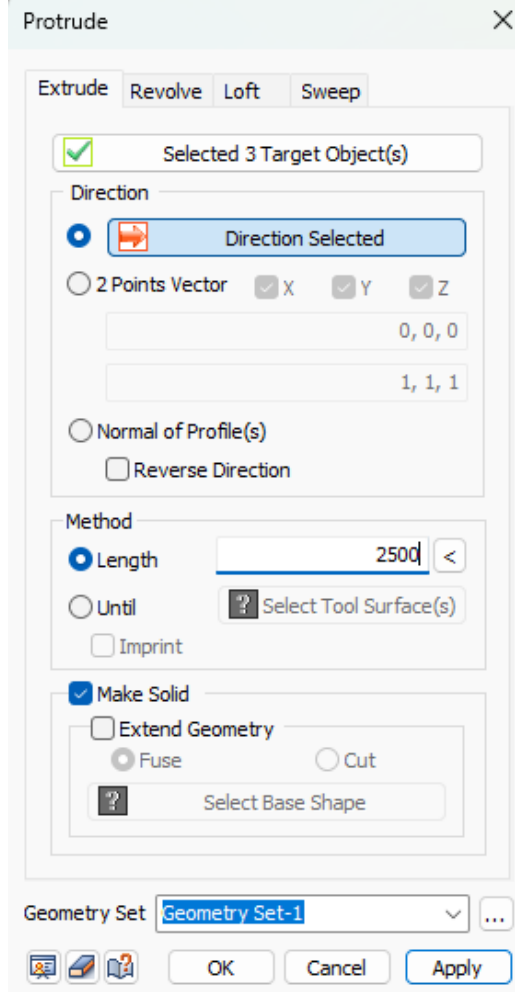
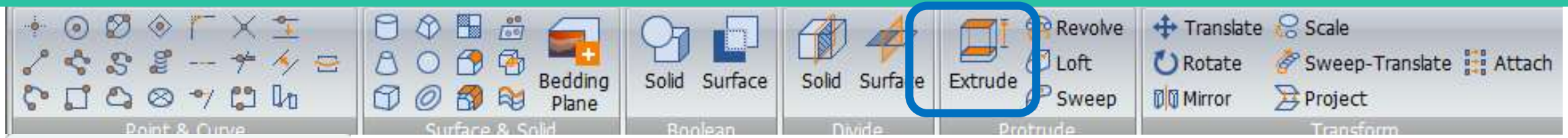
# GEOMETRY



## Steps

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

# GEOMETRY

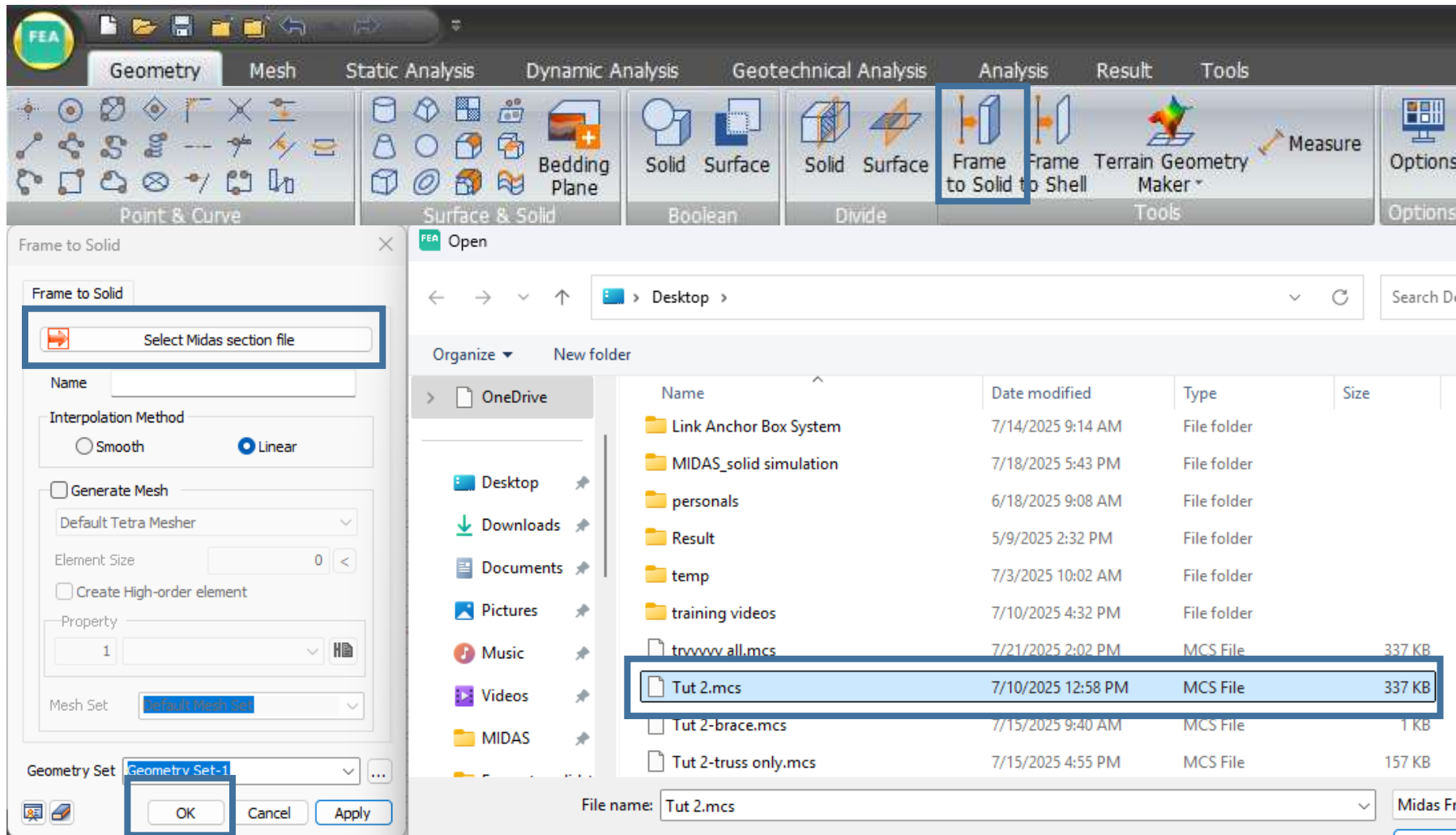


Steps:

Select all rectangles

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

# IMPORTED FILE



## Importing to FEA NX

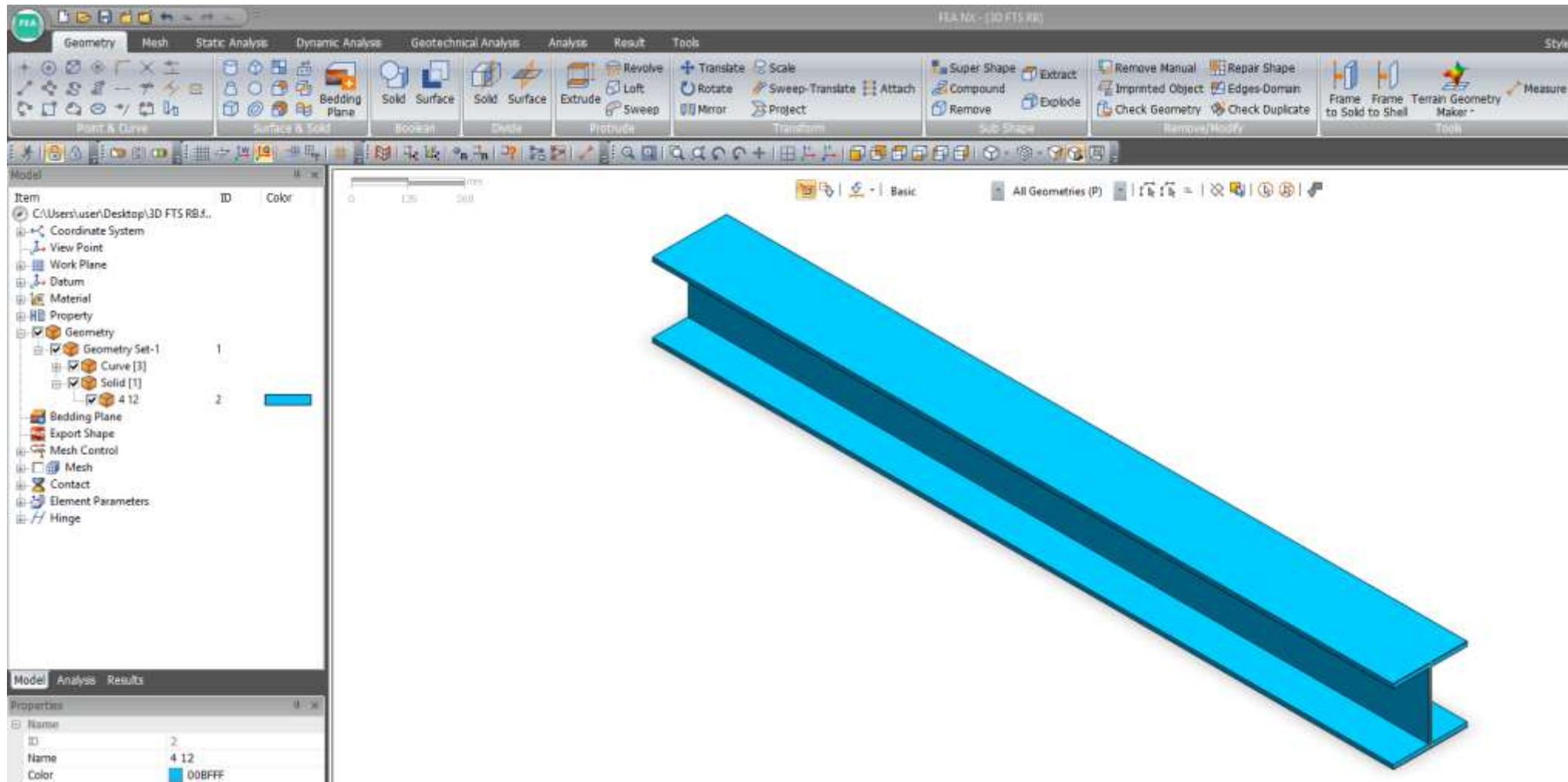
- Frame to Solid
- Select Midas Section File
- Choose File
- Choose OK

NOTE:

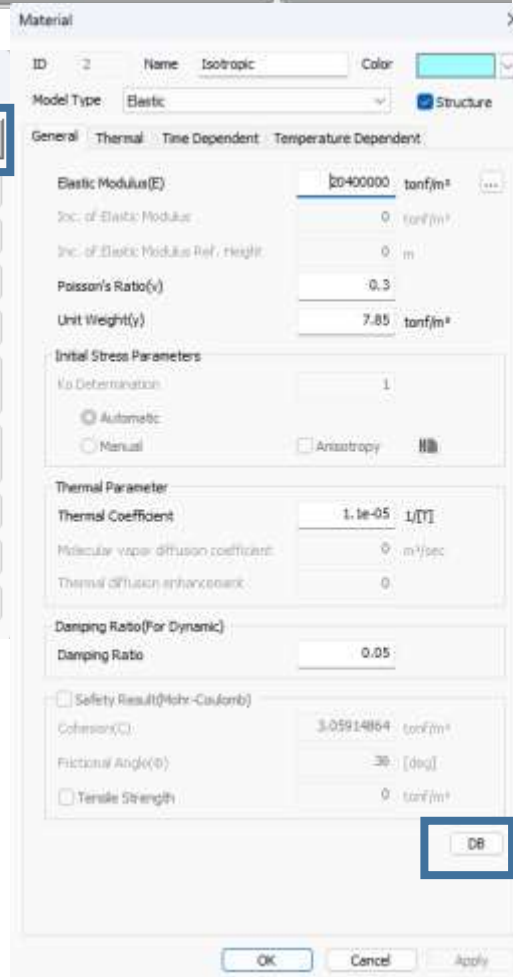
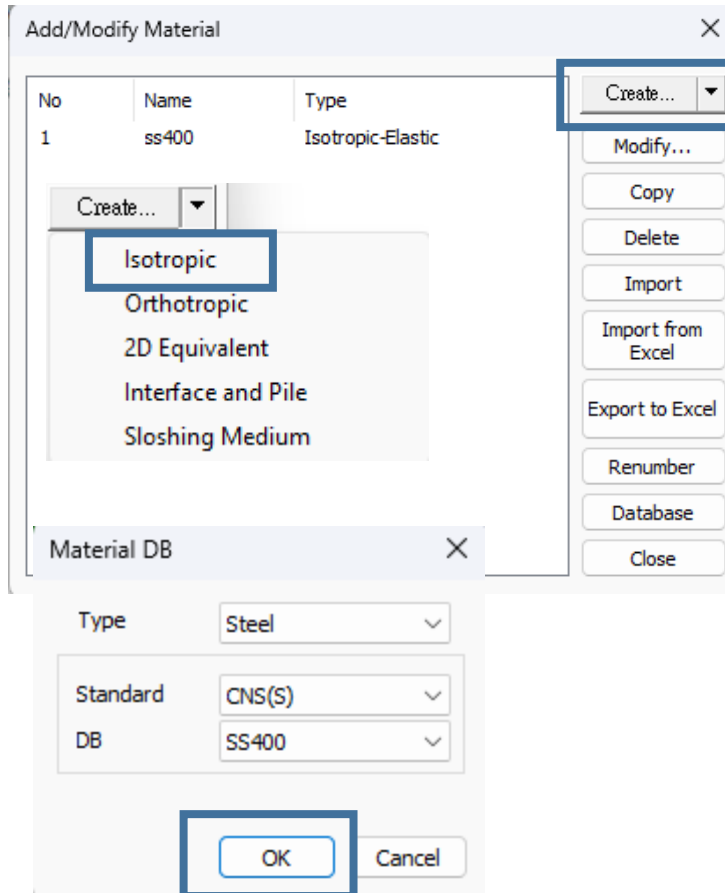
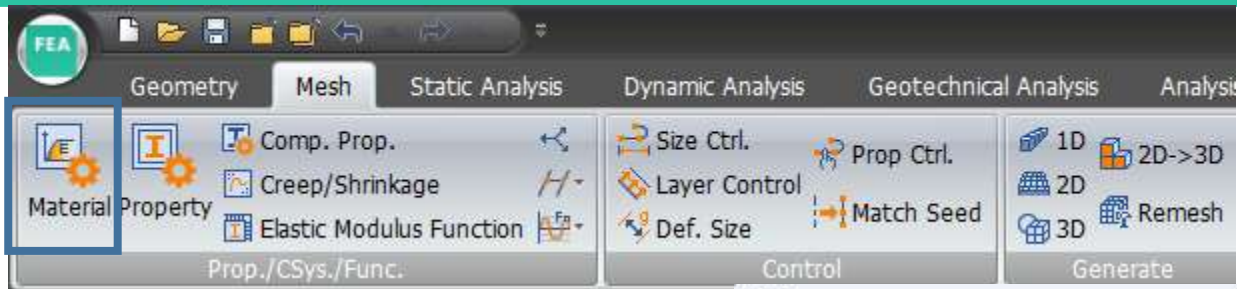
File Type : (.mcs)



# IMPORTED FILE



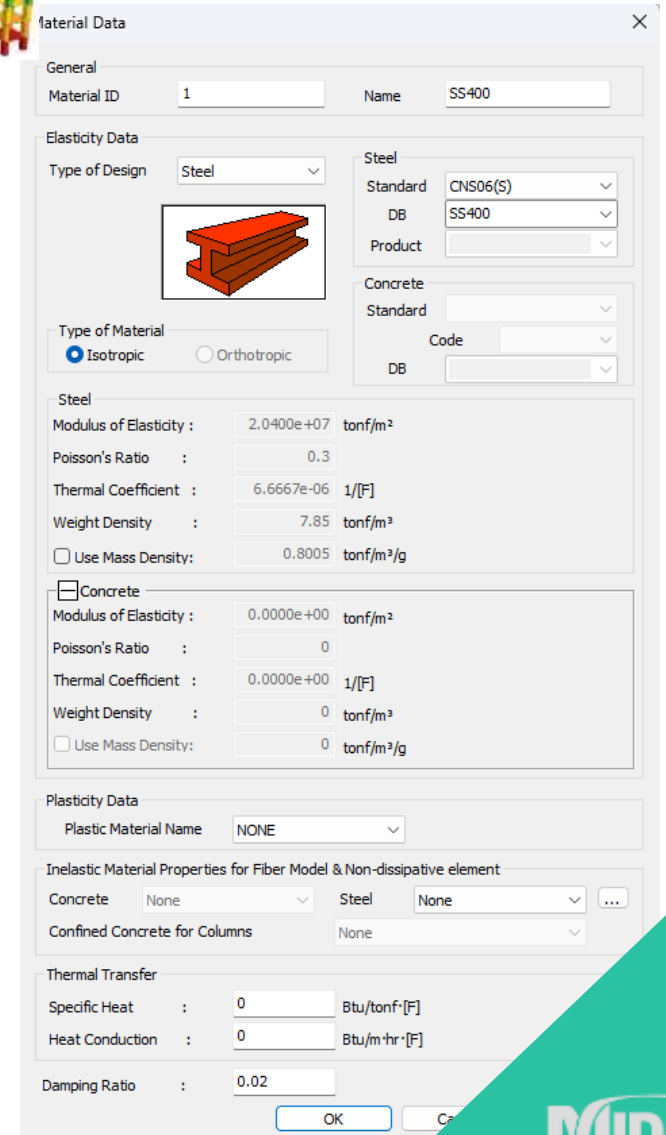
# MATERIALS AND PROPERTIES



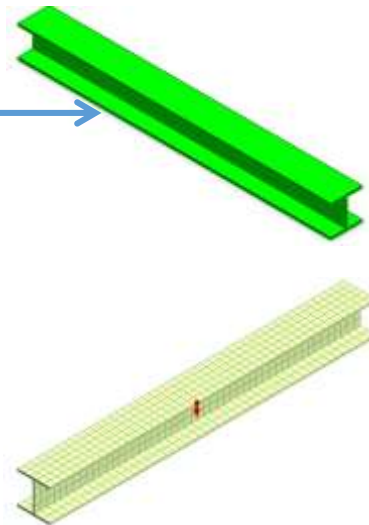
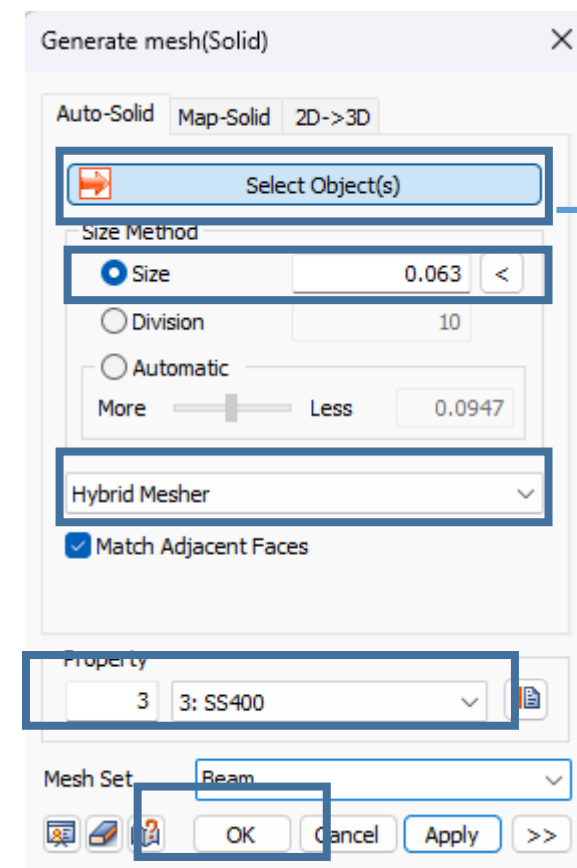
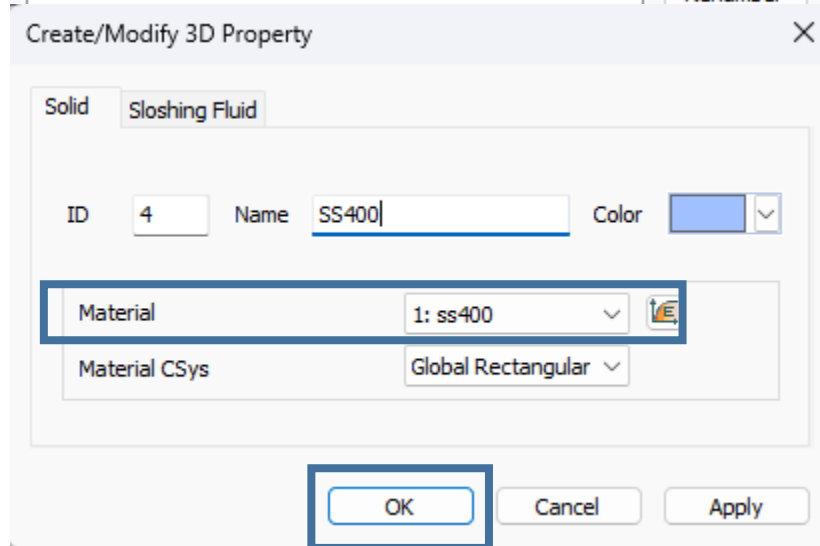
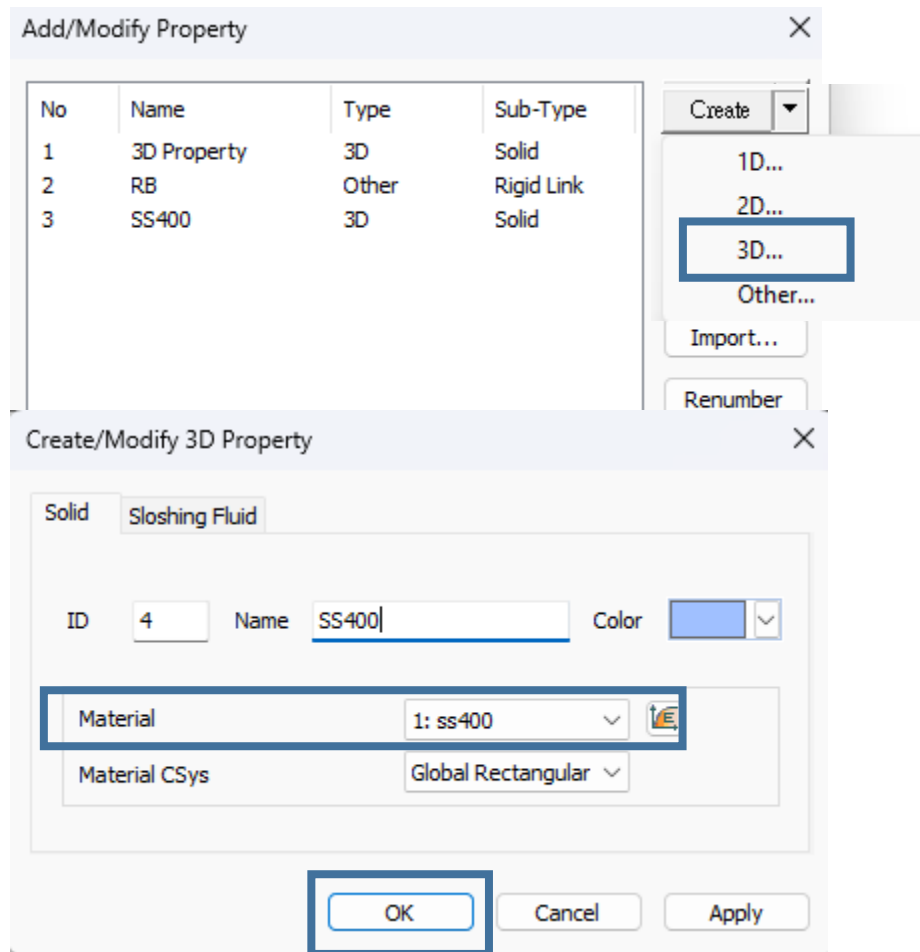
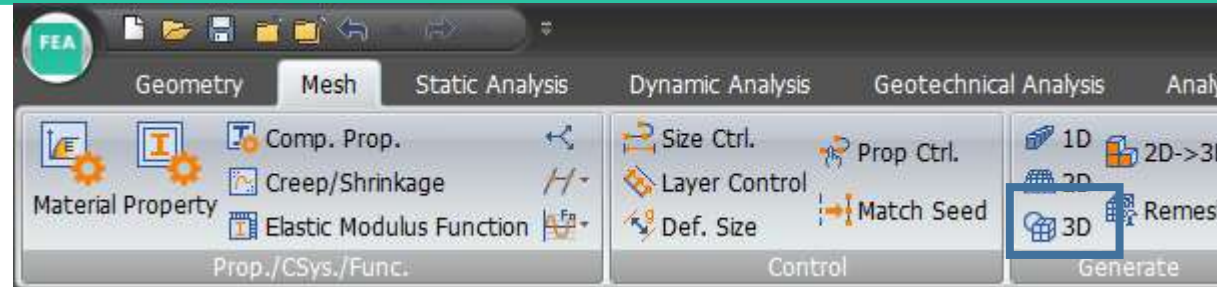
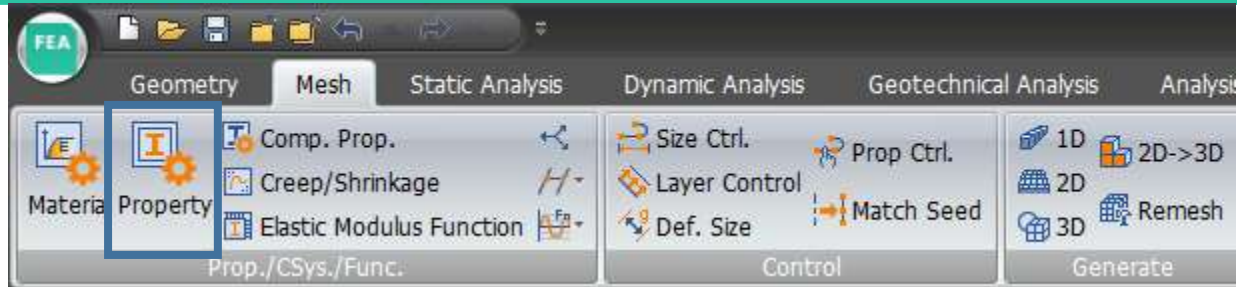
NOTE:  
Material Parameters  
are modifiable.



From MIDAS GEN



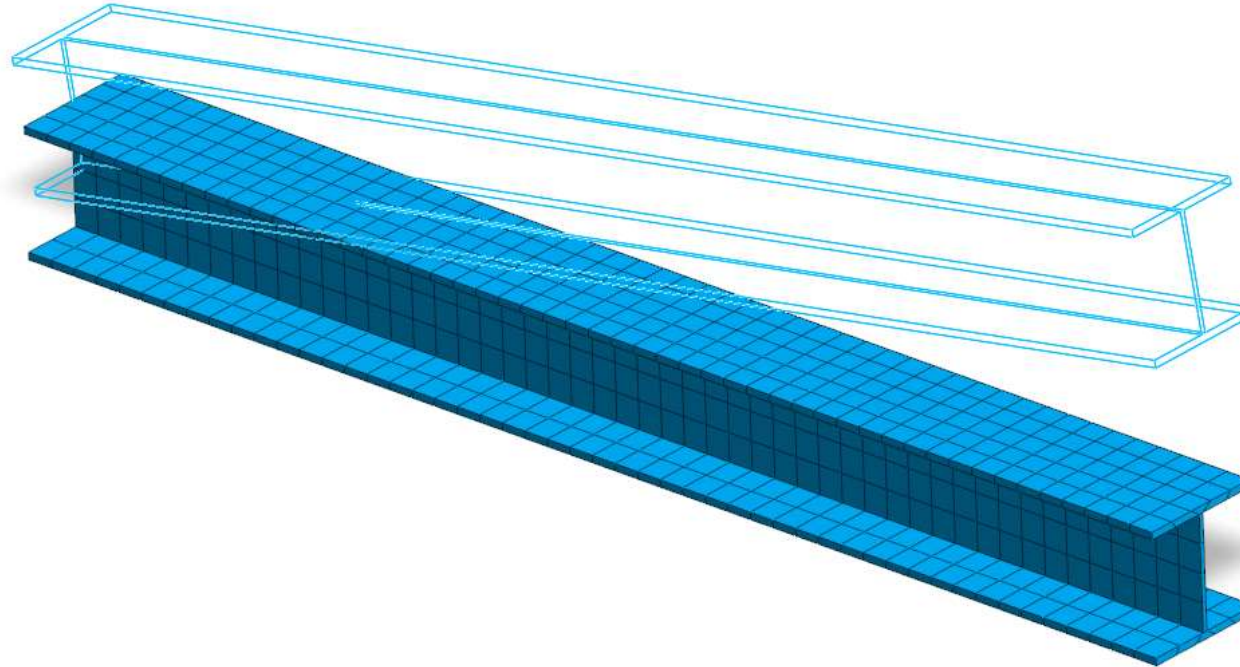
# MESHING





# MESHING

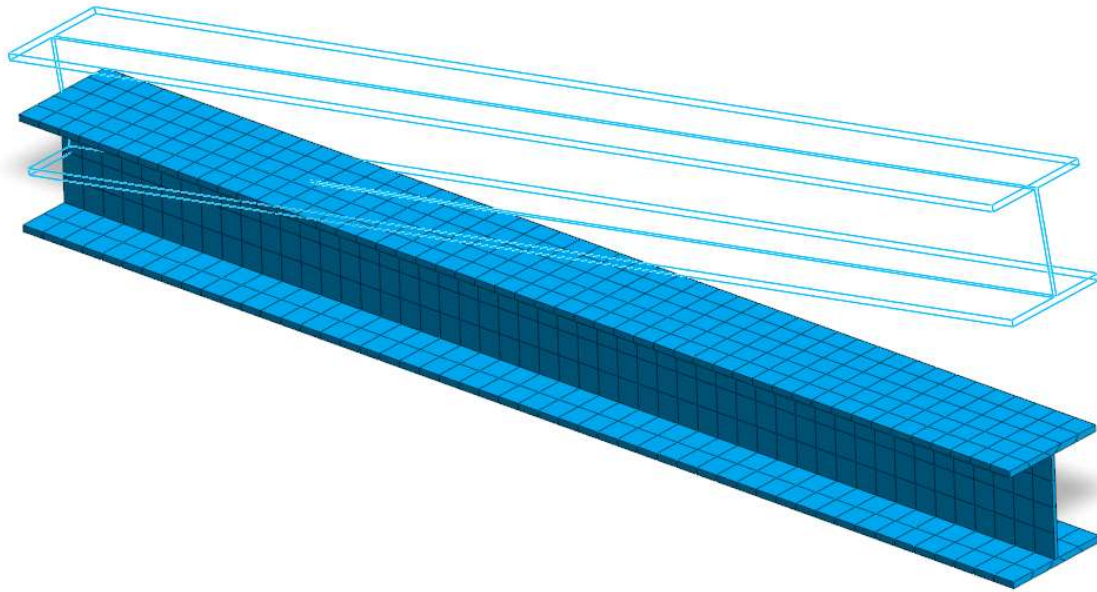
Item	ID	Color
C:\Users\user\Desktop\3D FTS RB.f..		
Coordinate System		
View Point		
Work Plane		
Datum		
Material		
Property		
Geometry		
Geometry Set-1	1	
Curve [3]		
Solid [2]		
4 12	2	
4 12	1	
Bedding Plane		
Export Shape		
Mesh Control		
Mesh		
Default Mesh Set	1	
Beam	2	



## NOTE:

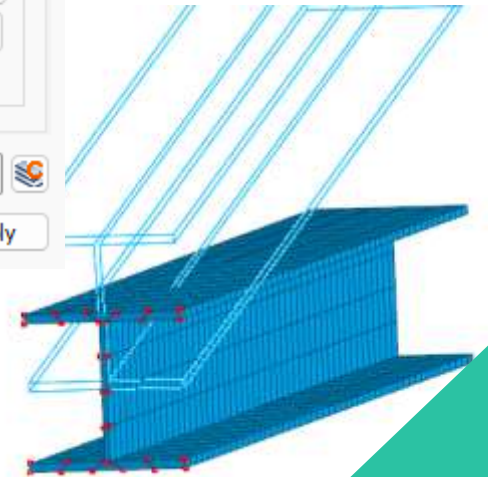
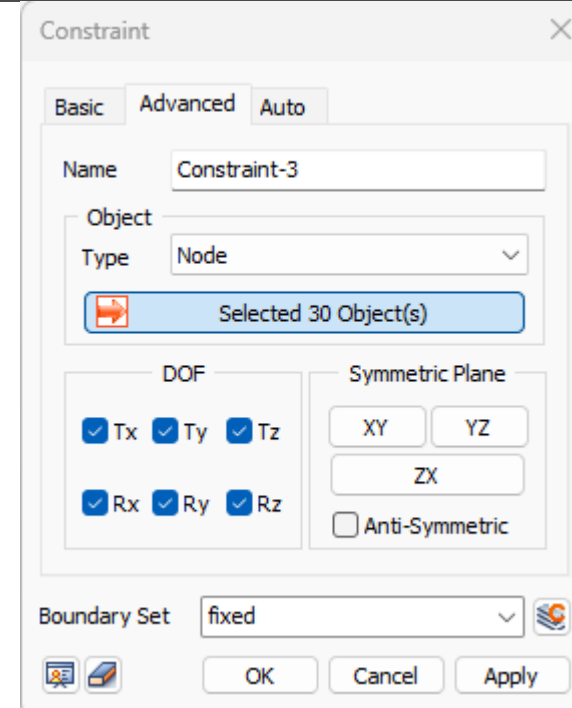
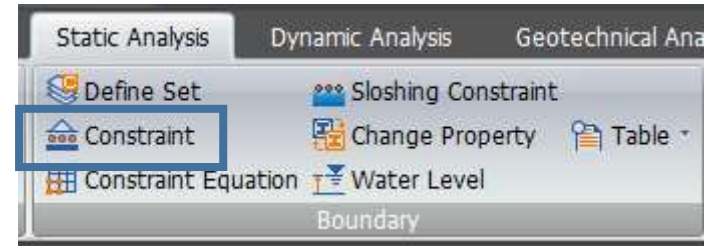
The smaller the mesh, the more accurate the results are.

# LOADS and CONSTRAINTS



## CONSTRAINT:

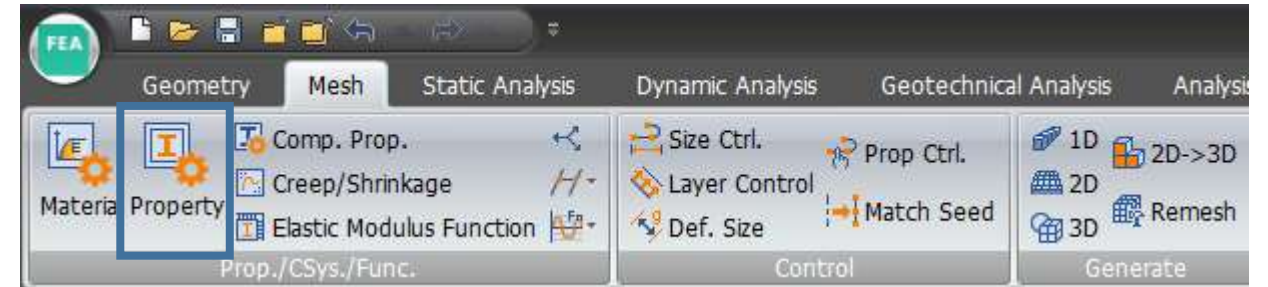
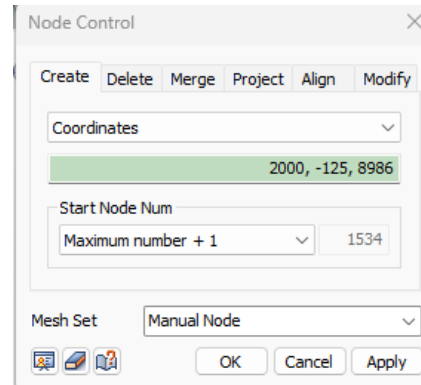
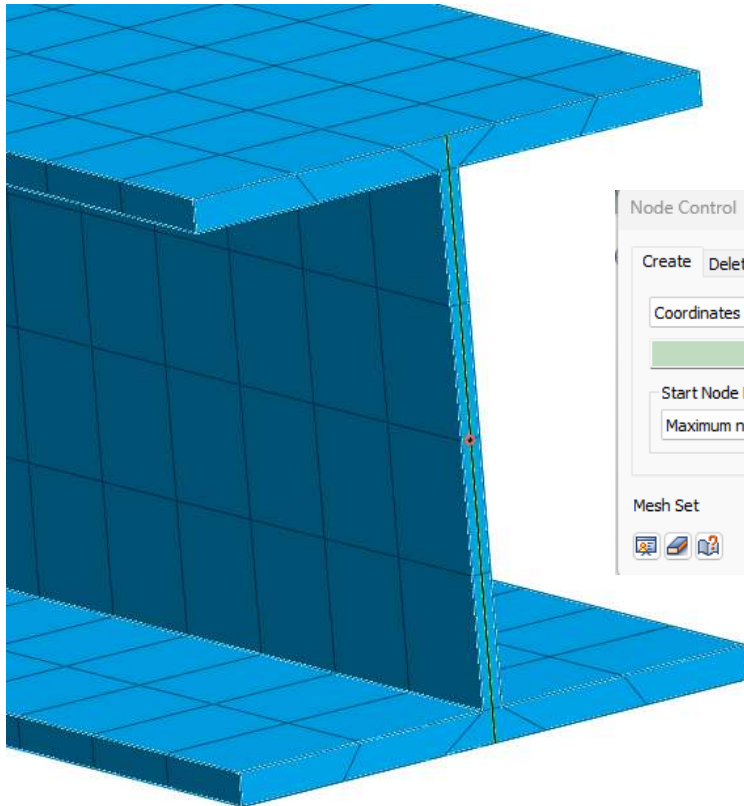
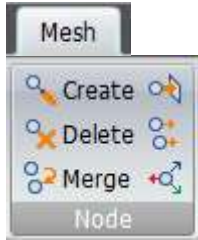
Pinned support on the joint



# LOADS and CONSTRAINTS

## RIGID LINK

### STEP 1: CREATE A MASTER NODE



### STEP 2: CREATE PROPERTY - RIGID LINK

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...

Create/Modify Other Property

Point Spring  
Matrix Spring  
Elastic Link  
Rigid Link  
Embedded Axisymmetric Point  
Embedded Axisymmetric Line  
Interface  
Shell Interface  
User Supplied Behavior for Shell Interface  
Pile Tip  
Infinite  
Fluid Boundary  
Free Field  
Seepage Cut Off

ID 3 Name RB Color

Properties  
☒ DX ☒ DY ☒ DZ ☒ RX ☒ RY ☒ RZ

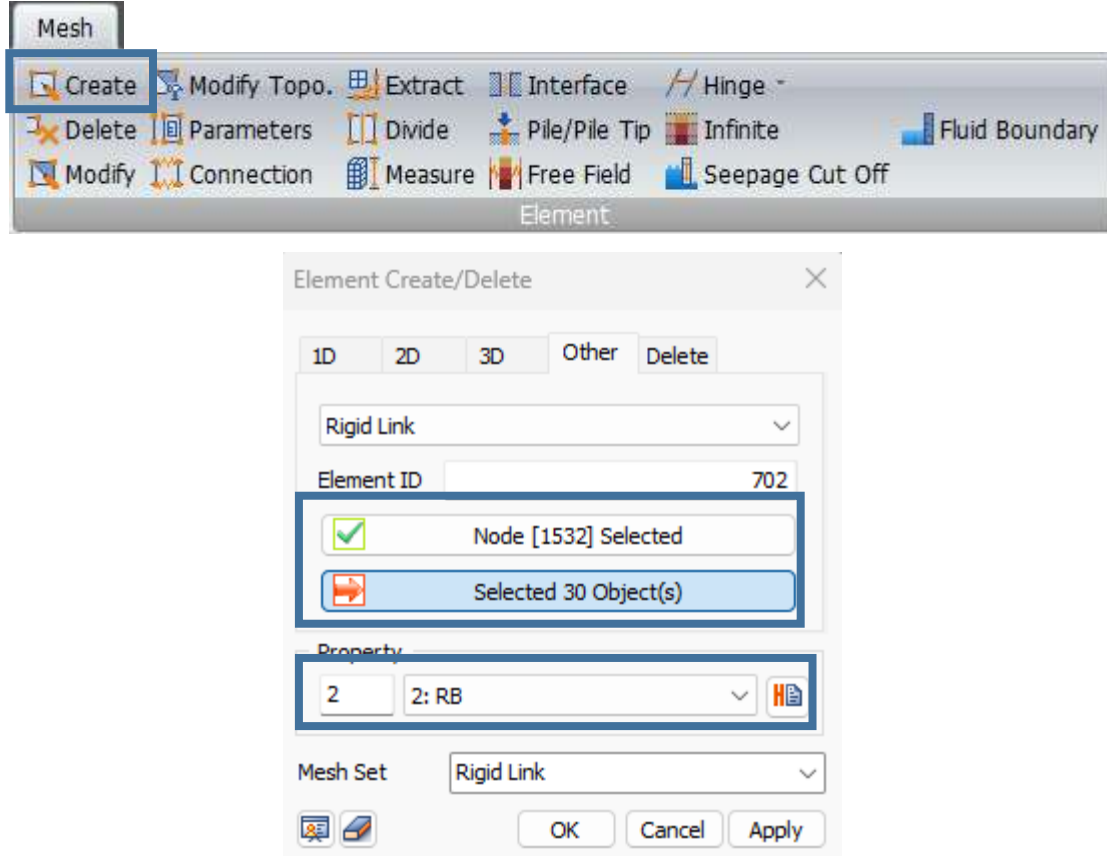
Typical Types  
Rigid Body  
Plane X-Y  
Plane Y-Z  
Plane Z-X

☐ Seepage Flow DOF ☐ Temperature

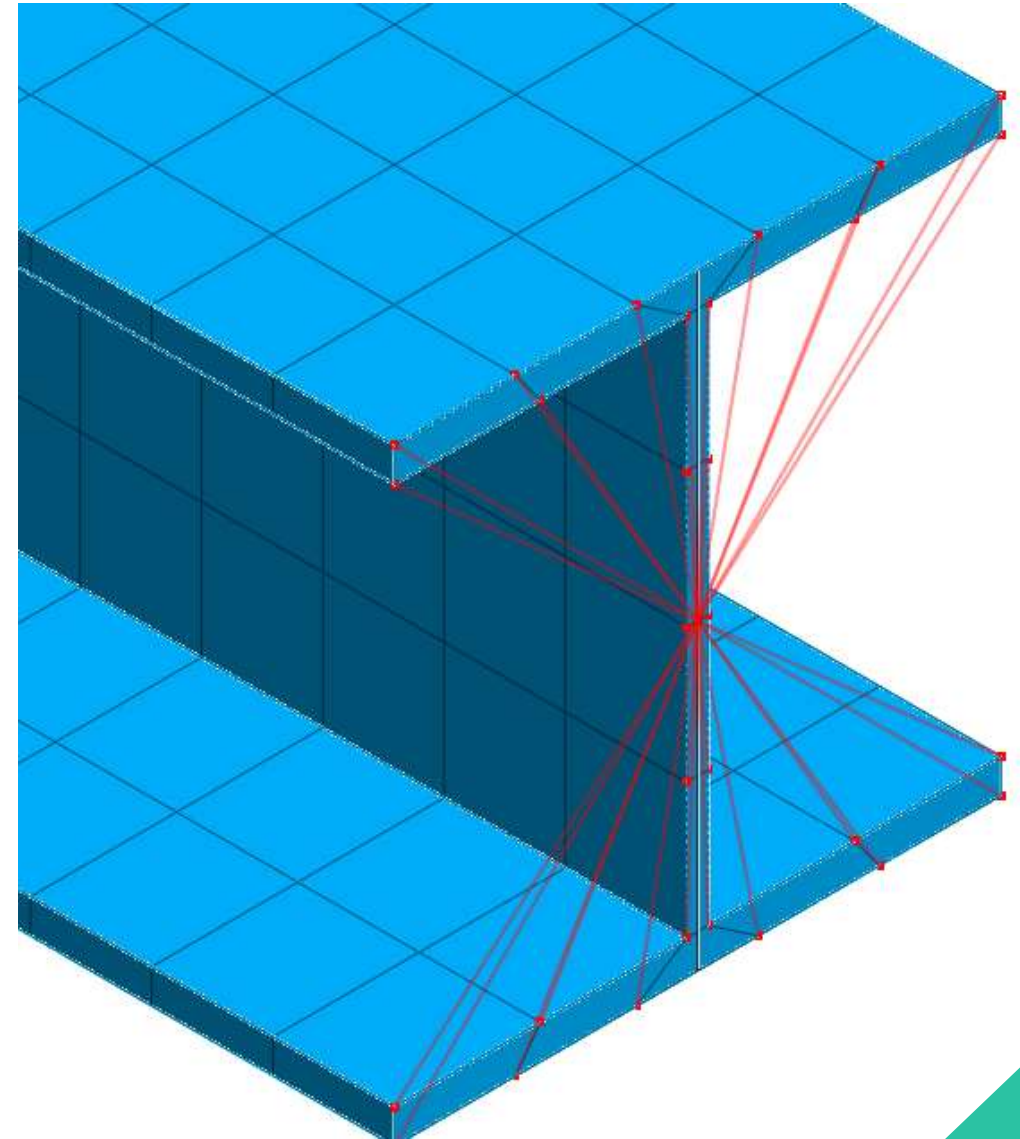


# LOADS and CONSTRAINTS

## RIGID LINK



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



# ANALYSIS CASE

Add/Modify Analysis Case

Analysis Case Setting

Title

Description

Solution Type **Nonlinear Static**

Construction Stage Set

Analysis Case Model

All Sets

- Mesh
  - Beam
  - Default Mesh Set
  - Manual Node
  - Rigid Link
- Boundary Condition
  - T only
  - fixed
- Static Load
  - Default Self-Weight
  - Disp Tz=-0.088
  - Displacement\_fromGEN
  - Fz -2000
- Combined Load Sets
- Contact Pair

Active Sets

- Mesh
  - Beam
  - Default Mesh Set
  - Manual Node
  - Rigid Link
- Boundary Condition
  - fixed
- Static Load
  - Displacement\_fromGEN
- Combined Load Sets
- Contact Pair

☐ Solve Each Load Set Independently

Sorting Name

OK Cancel Apply

Analysis Control

General Nonlinear

Geometry Nonlinearity

☐ Consider Geometric Nonlinear Effects

☐ Update Pore Pressure with Deformation

Basic Nonlinear Parameters

☒ Load Steps

Number of Increments 10

Intermediate Output Request Last Increment

☐ Manual with User-Defined Steps Load Step...

☐ Time Steps

Time(Duration) 86400 sec

Number of Increments 1

Intermediate Output Request Last Increment

☐ Manual with User-Defined Steps Time Step...

Iterative Scheme

☒ General ☐ Enhanced Init Stress

Convergence Criteria / Error Tolerance

☐ Displacement(U) 0.001

☒ Load(P) 0.001

☒ Work(W) 1e-06

☐ Use Iteration Method

☒ Arc-Length Method ☐ Displacement-Control Method

Min. Arc-Length Adjustment Ratio 0.25

Max. Arc-Length Adjustment Ratio 4

Max Arc-Length Increments 20

Advanced Nonlinear Setting...

OK Cancel

Output Control

Output Type Output Option

☒ Write Results of All Active Mesh Sets

Nodal Results

☒ Displacement Mesh Set...

☒ Applied Load Mesh Set...

☒ Reaction Force Mesh Set...

☒ Grid Point Force Mesh Set...

☒ Contact Mesh Set...

Element Results

☒ Force Mesh Set...

☒ Stress Mesh Set...

☒ Strain Mesh Set...

☒ Status Mesh Set...

☒ Damaged Index Mesh Set...

☒ Ductility Mesh Set...

☒ Multi-layered Grid Mesh Set...

Element Output Location

☒ Element Corner Results

☐ Shell Mid-Plane Results

☒ Composite Shell Mid-Plane Results

Number of Beam Output Segments 4

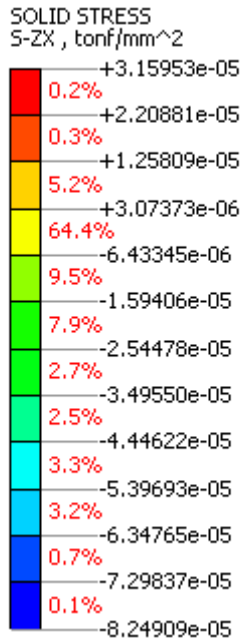
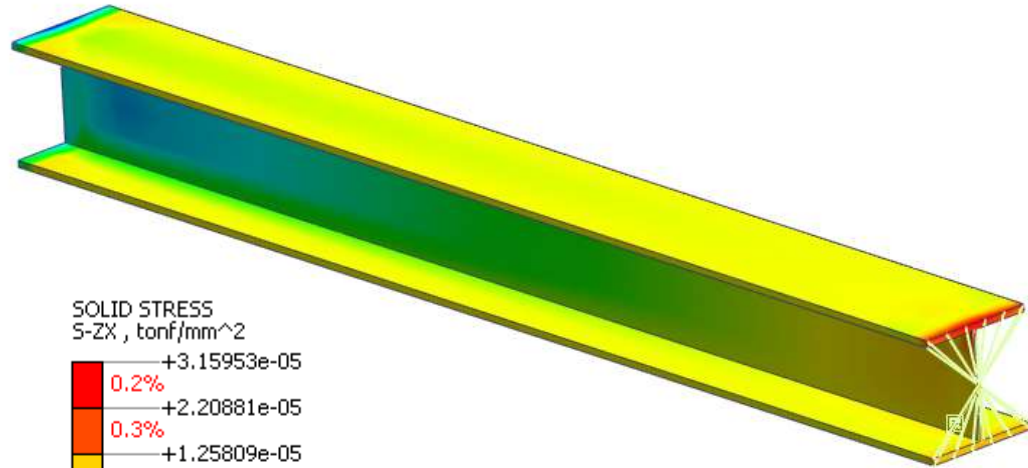
Output Option

☒ Binary ☐ Binary and Text

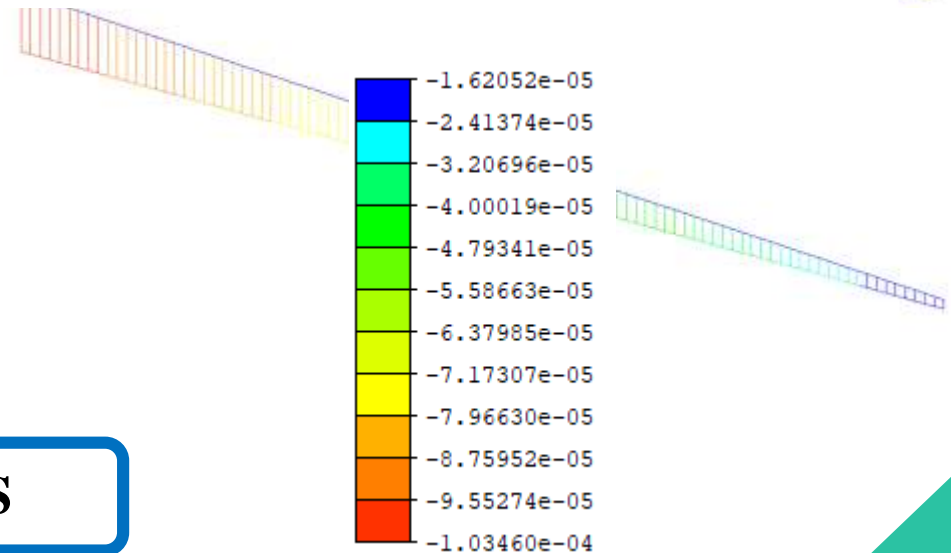
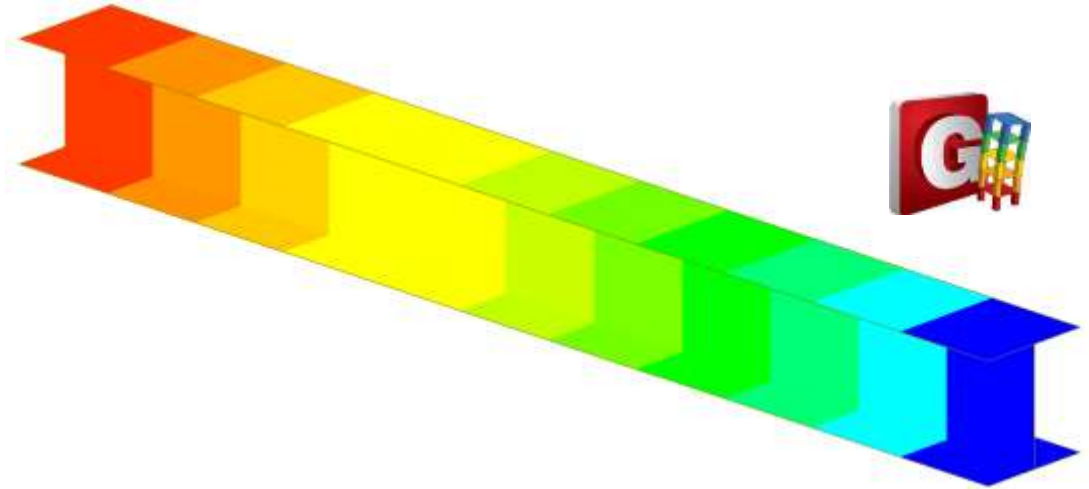
OK Cancel

# RESULTS - Beam Stress Comparison

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

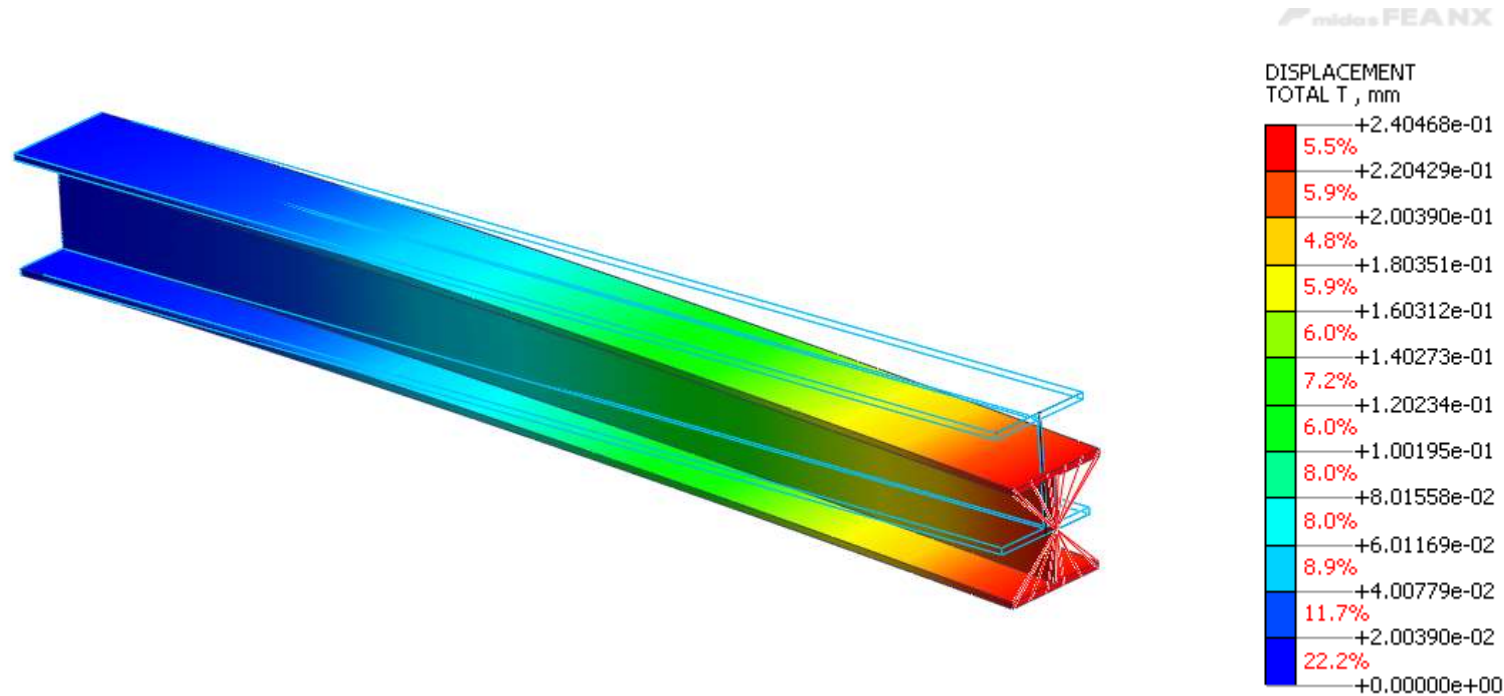


**MIDAS GEN Analysis in between two nodes : Linear Interpolation**



**SHEAR Z-AXIS**

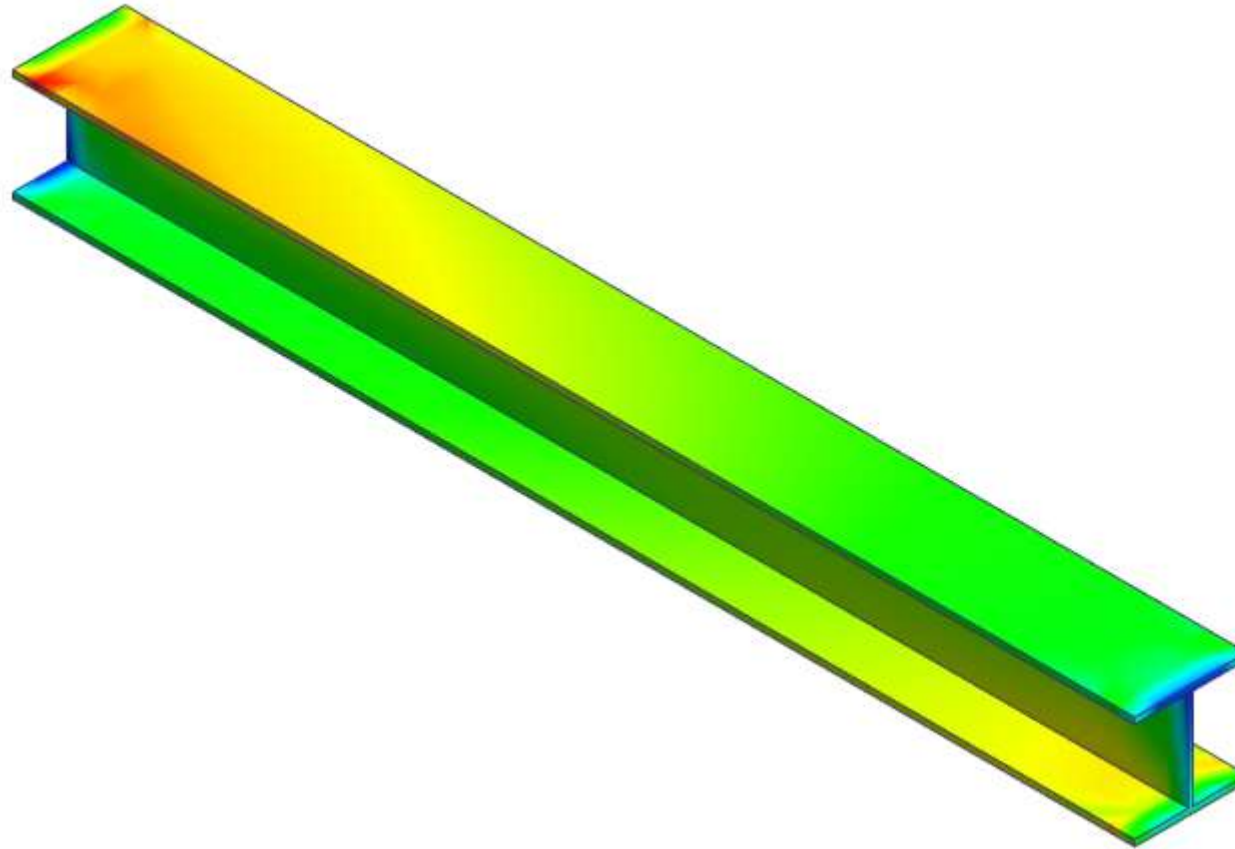
# RESULTS - Beam Deformation



**Deformation**



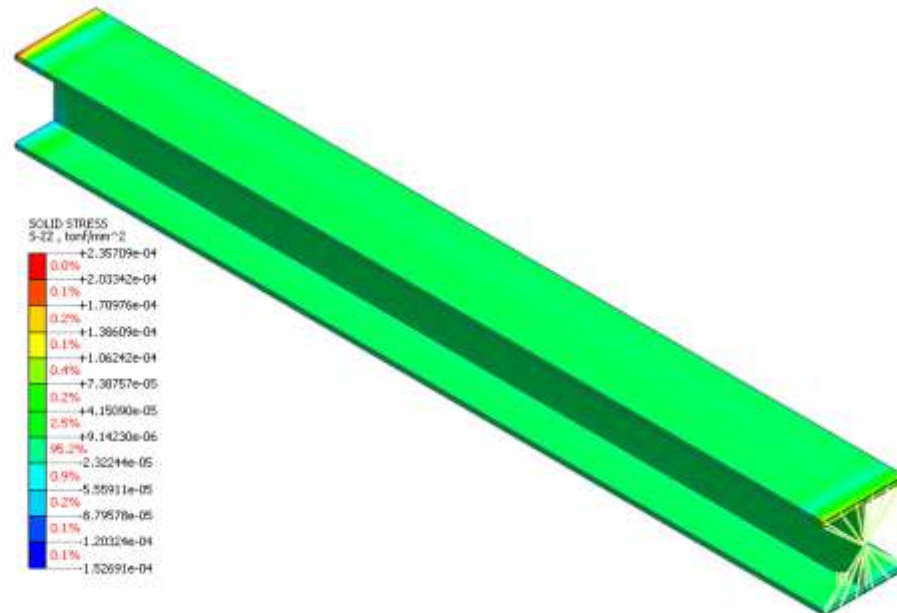
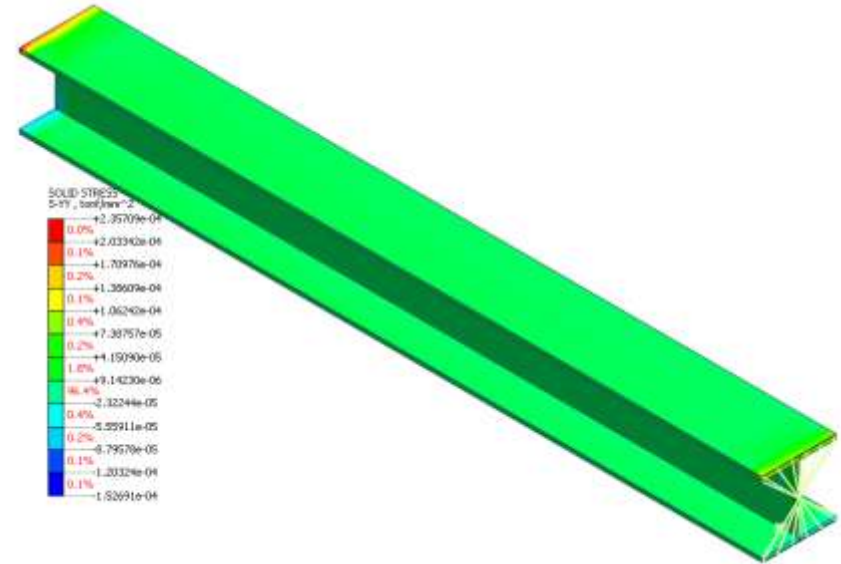
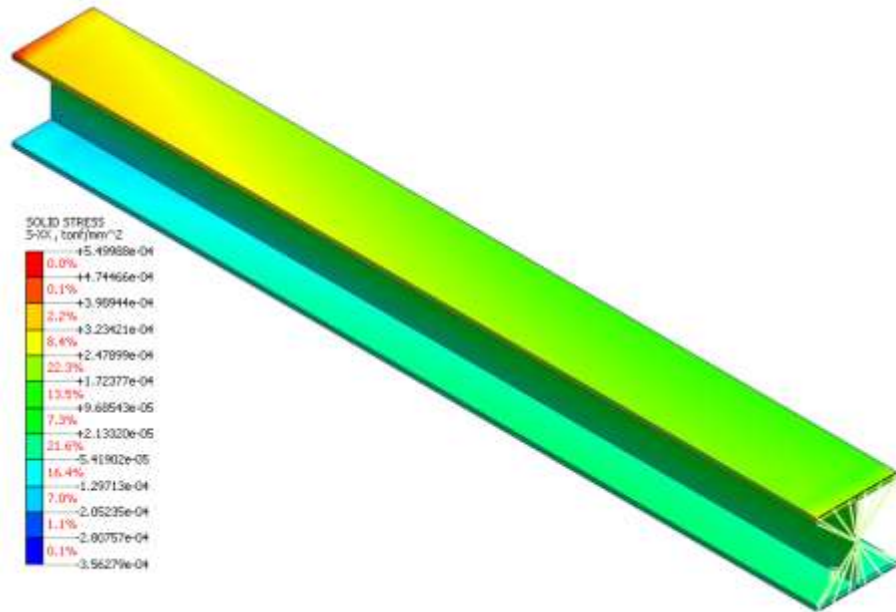
# RESULTS - STRESS VON MISES



SOLID STRESS  
S-VON MISES, tonf/mm<sup>2</sup>

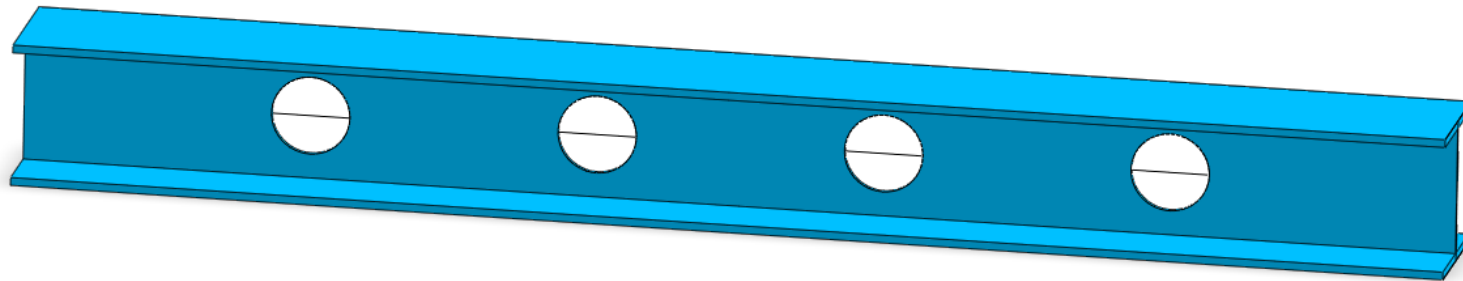
	+2.25906e-02
0.1%	+2.14600e-02
1.3%	+2.03293e-02
8.0%	+1.91986e-02
20.9%	+1.80680e-02
30.0%	+1.69373e-02
20.9%	+1.58066e-02
13.0%	+1.46760e-02
3.6%	+1.35453e-02
1.0%	+1.24146e-02
0.3%	+1.12840e-02
0.4%	+1.01533e-02
0.5%	+9.02265e-03

# RESULTS - Beam Strains Normal

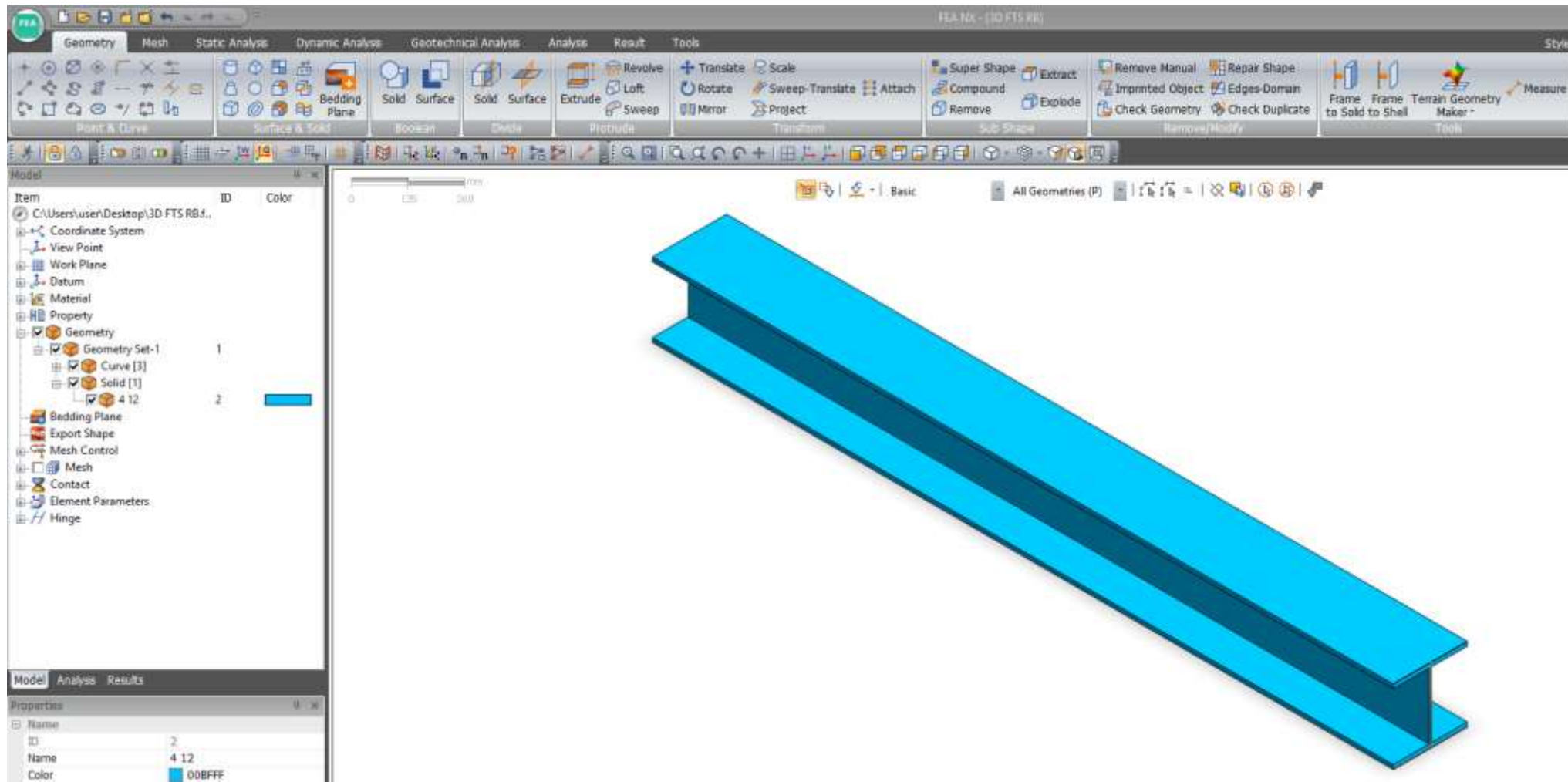


# Part 2

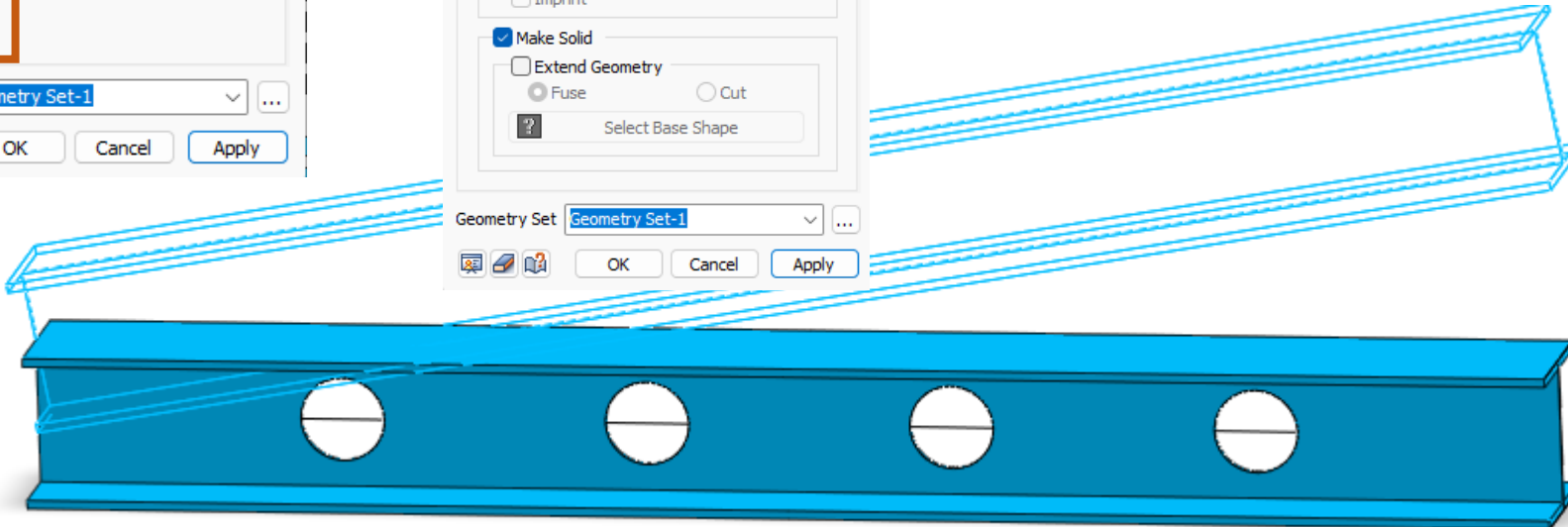
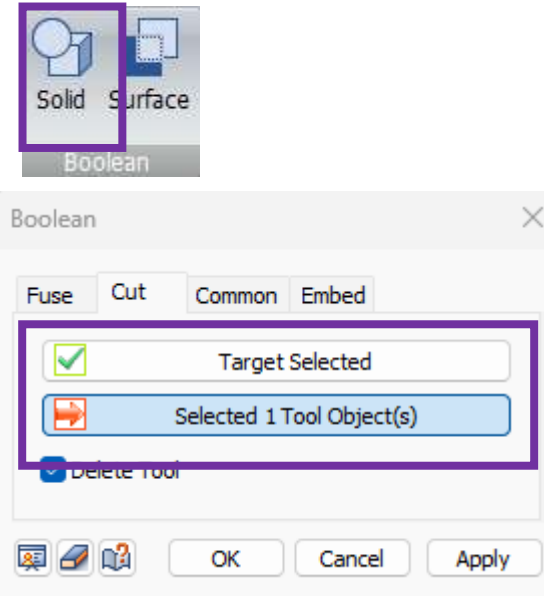
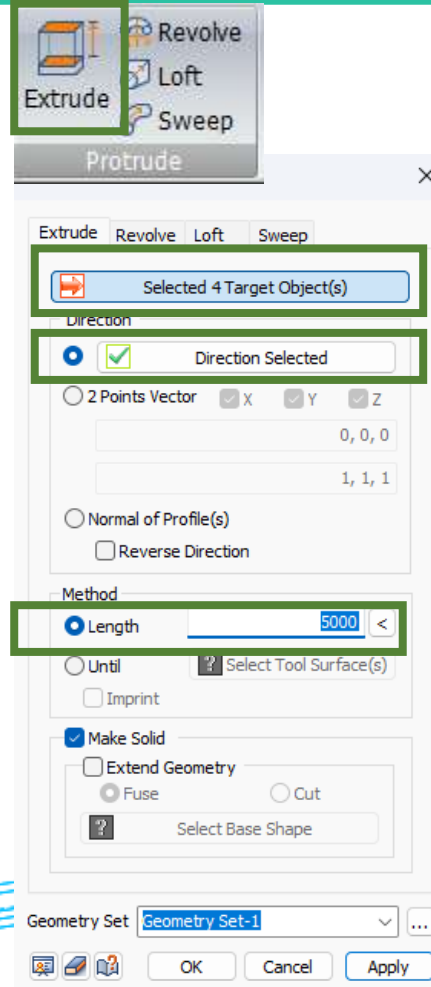
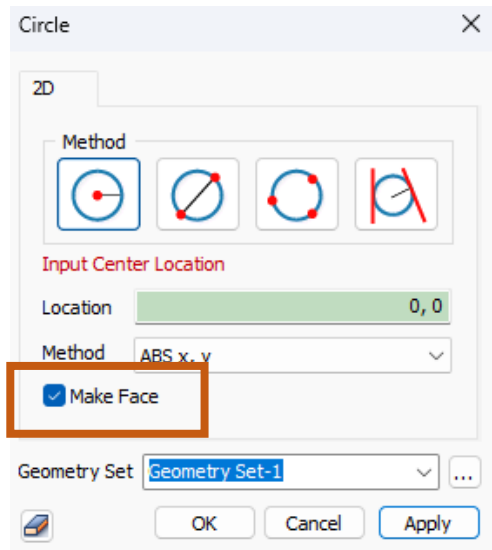
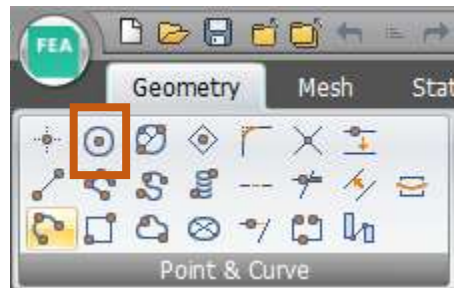
# Modified Steel Structure



# IMPORTED FILE



# GEOMETRY



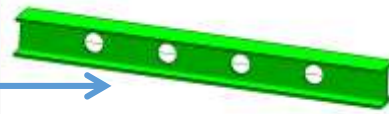
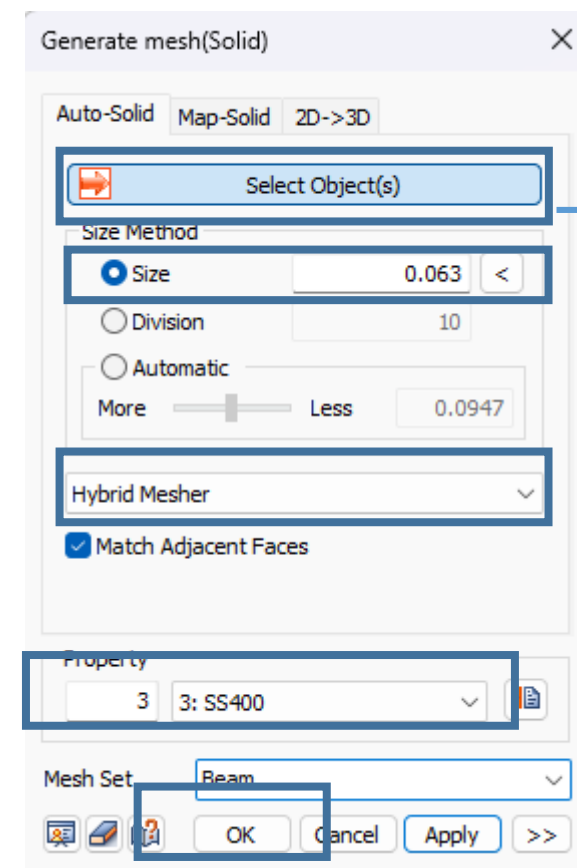
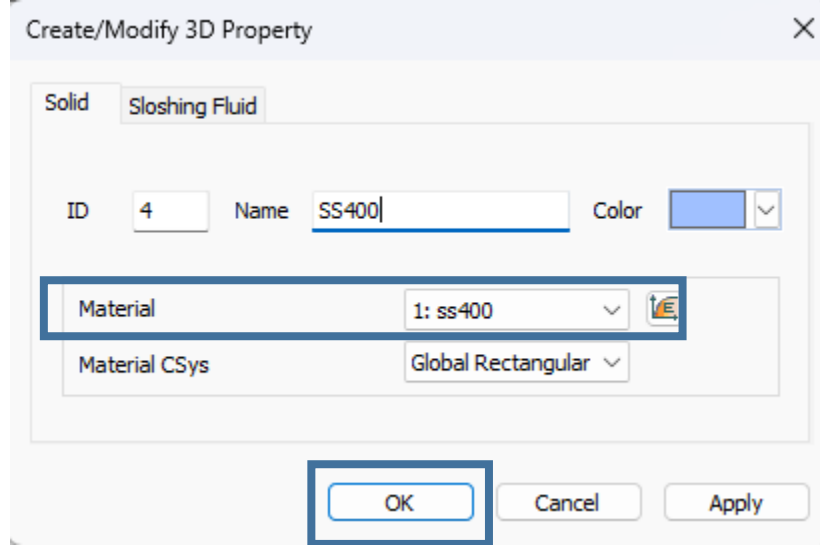
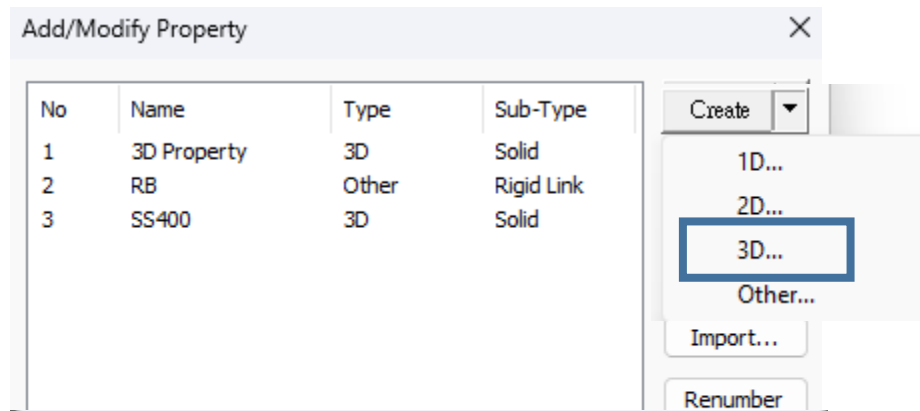
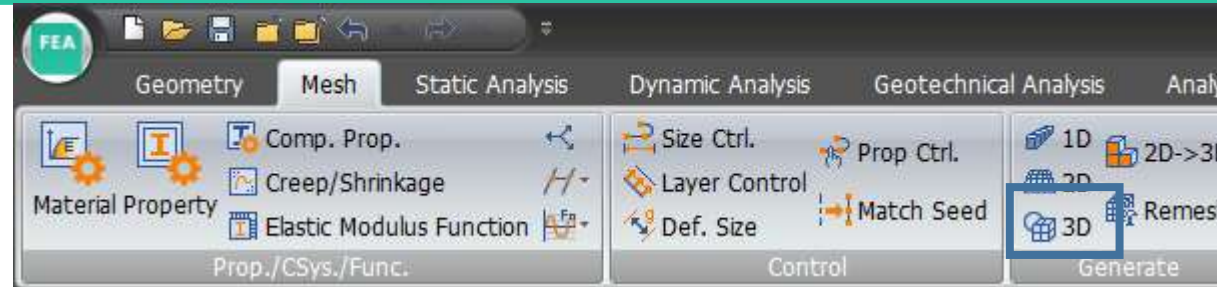
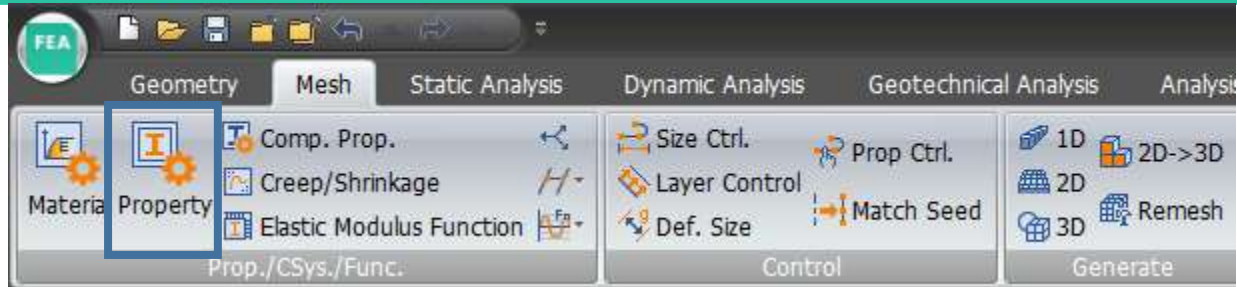
## MODIFYING BEAM ELEMENT

(Sample Case: There are holes in the beam)

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

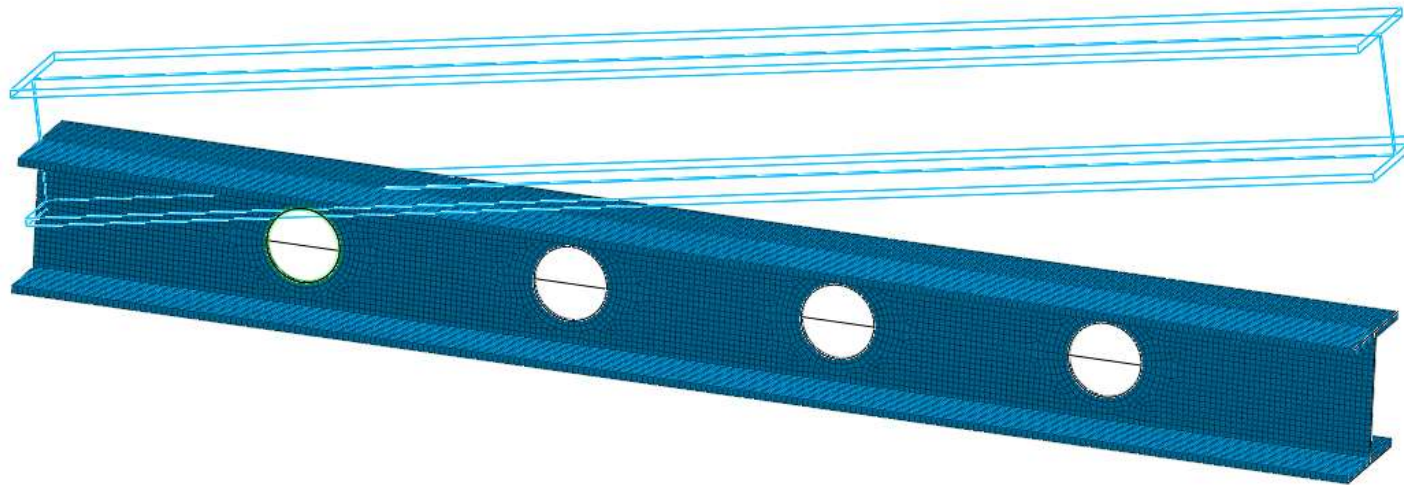


# MESHING



# MESHING

Item	ID
C:\Users\user\Desktop\3D FTS RB.f..	
Coordinate System	
View Point	
Work Plane	
Datum	
Material	
Property	
Geometry	
Geometry Set-1	1
Curve [3]	
Solid [2]	
4 12	2
4 12	1
Bedding Plane	
Export Shape	
Mesh Control	
Mesh	
Default Mesh Set	1
Beam	2

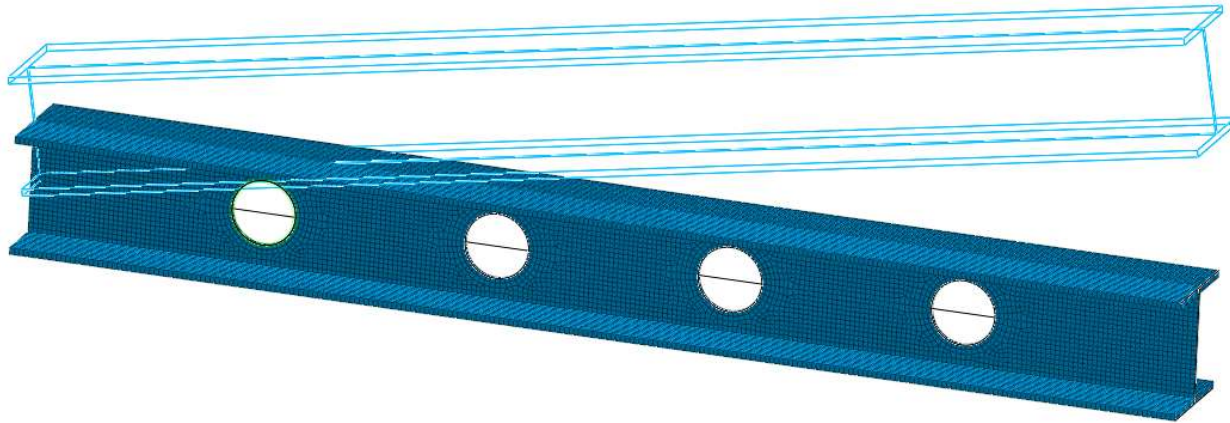


## NOTE:

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

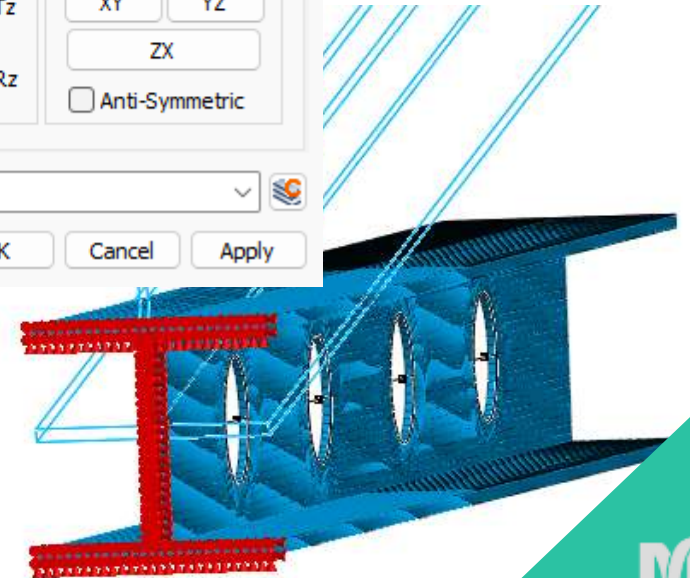
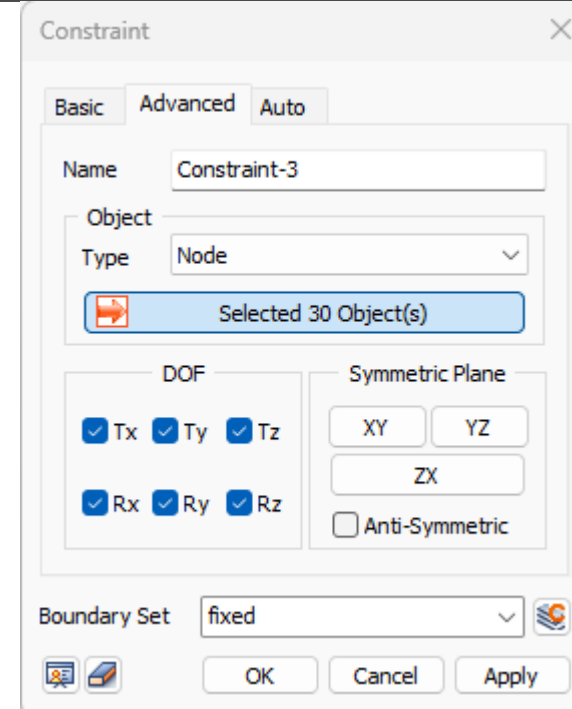
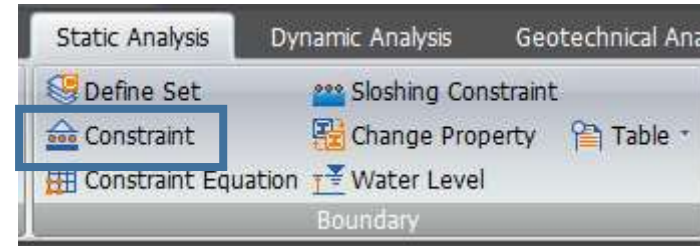


# LOADS and CONSTRAINTS



## CONSTRAINT:

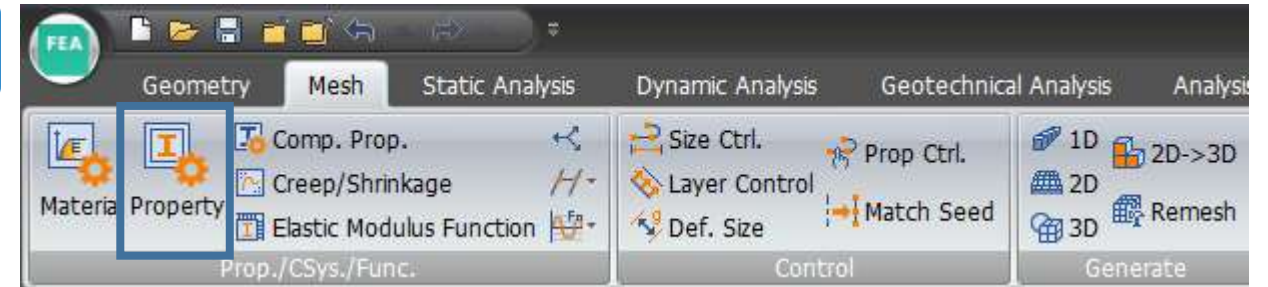
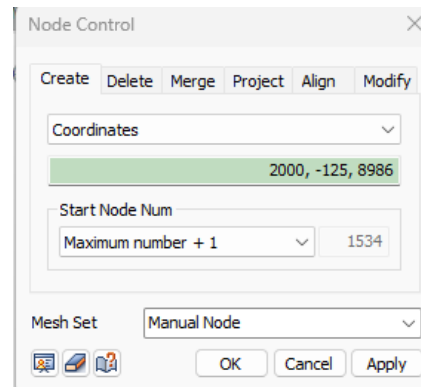
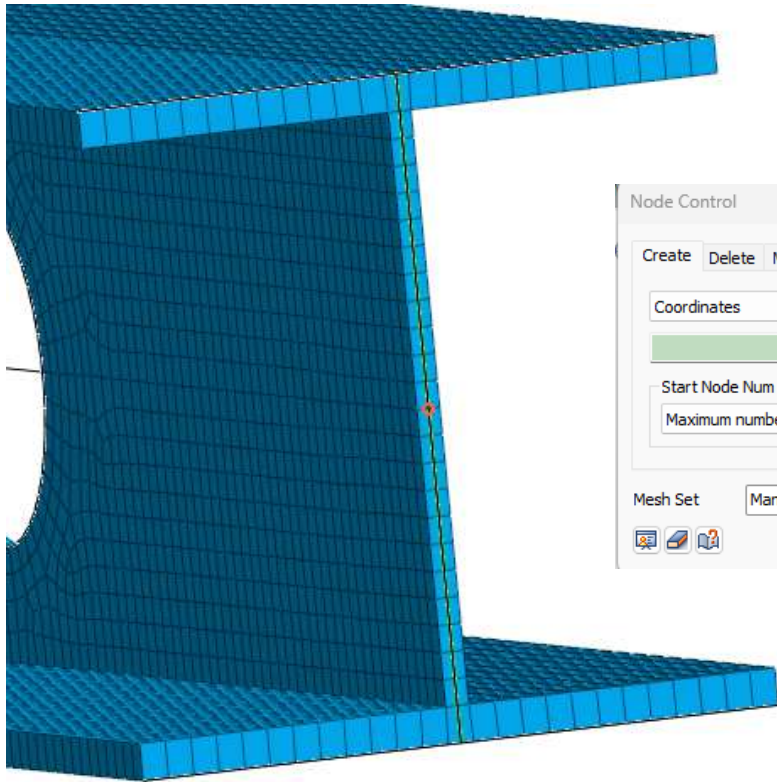
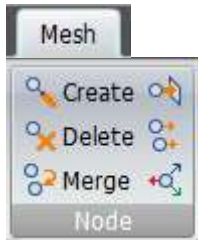
Pinned support on the joint



# LOADS and CONSTRAINTS

## RIGID LINK-BEAM END FOR LOADING

### STEP 1: CREATE A MASTER NODE



### STEP 2: CREATE PROPERTY - RIGID LINK

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...

Create/Modify Other Property

Point Spring  
Matrix Spring  
Elastic Link  
Rigid Link  
Embedded Axisymmetric Point  
Embedded Axisymmetric Line  
Interface  
Shell Interface  
User Supplied Behavior for Shell Interface  
Pile Tip  
Infinite  
Fluid Boundary  
Free Field  
Seepage Cut Off

ID 3 Name RB Color

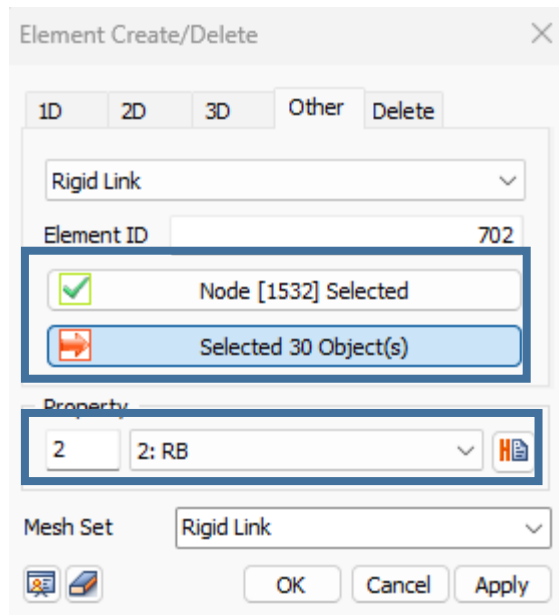
Properties  
☒ DX ☒ DY ☒ DZ ☒ RX ☒ RY ☒ RZ

Typical Types  
Rigid Body  
Plane X-Y  
Plane Y-Z  
Plane Z-X

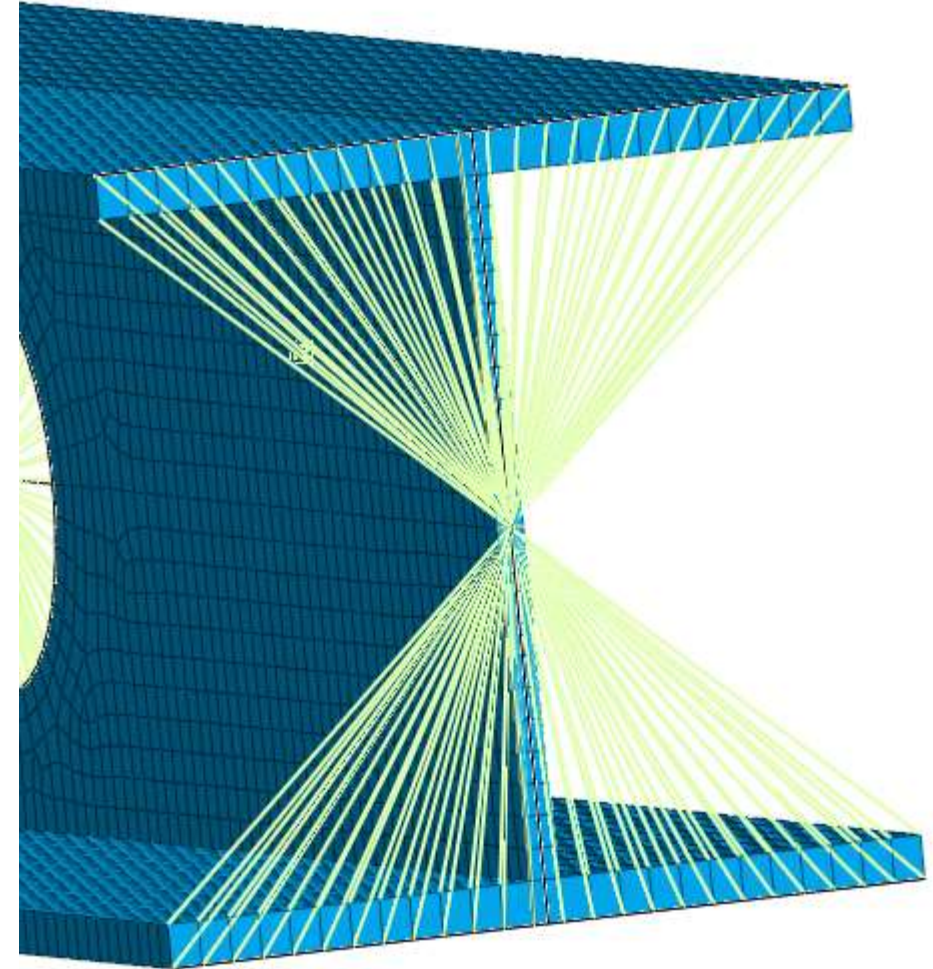
☐ Seepage Flow DOF ☐ Temperature

# LOADS and CONSTRAINTS

## RIGID LINK-BEAM END FOR LOADING



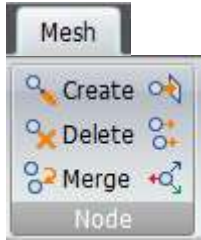
### STEP 3: DEFINE RIGID LINK



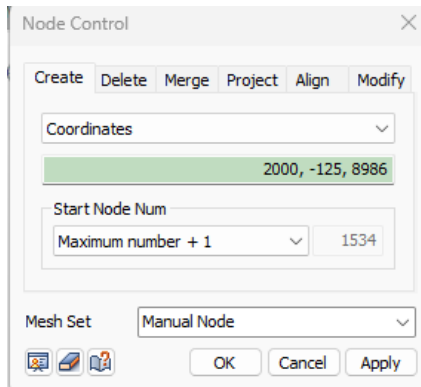
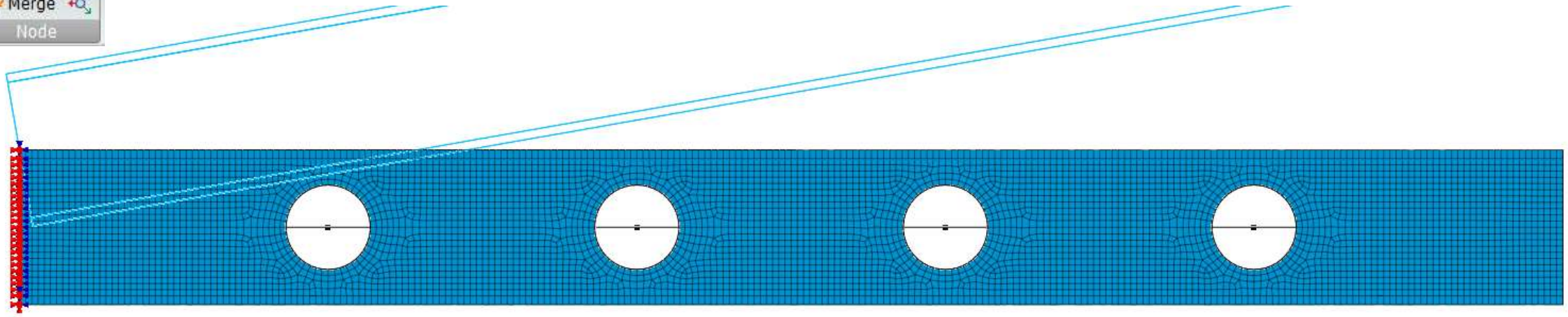


# LOADS and CONSTRAINTS

## RIGID LINK-CREATE LINKS FOR HOLES



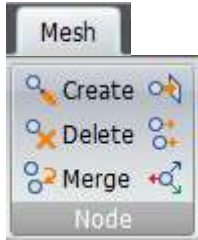
### STEP 4: CREATE A MASTER NODE



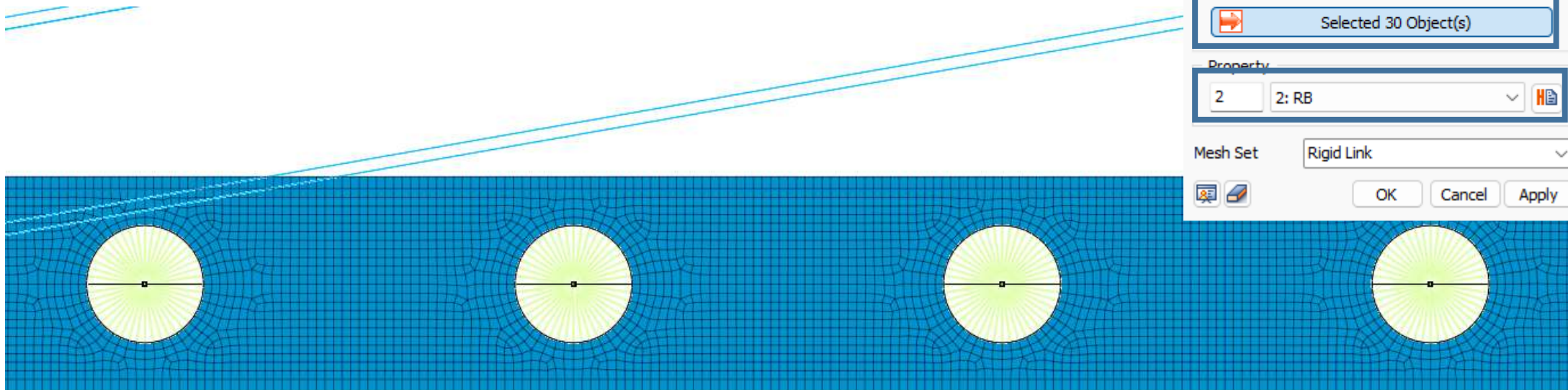
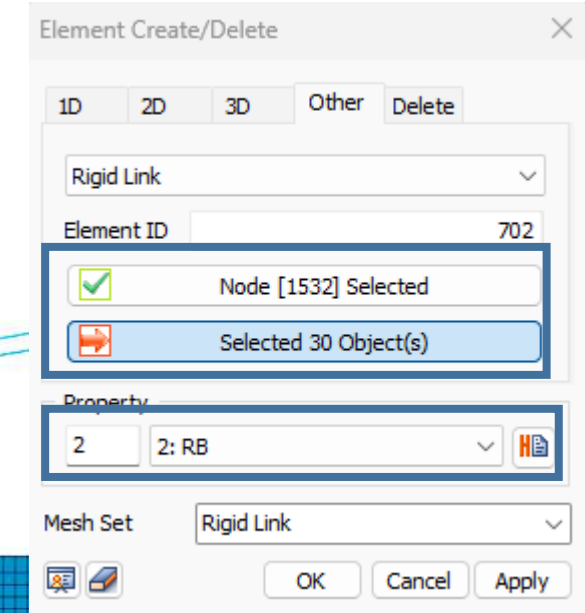
**NOTE:** This is to define the rigid link of the surface surrounding the hole which would ensure its rigid movement.

# 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

Add/Modify Analysis Case

Analysis Case Setting

Title

Description

Solution Type **Nonlinear Static**

Construction Stage Set

Analysis Case Model

All Sets

- Mesh
  - Beam
  - Default Mesh Set
  - Manual Node
  - Rigid Link
- Boundary Condition
  - T only
  - fixed
- Static Load
  - Default Self-Weight
  - Disp Tz=-0.088
  - Displacement\_fromGEN
  - Fz -2000
- Combined Load Sets
- Contact Pair

Active Sets

- Mesh
  - Beam
  - Default Mesh Set
  - Manual Node
  - Rigid Link
- Boundary Condition
  - fixed
- Static Load
  - Displacement\_fromGEN
- Combined Load Sets
- Contact Pair

☐ Solve Each Load Set Independently

Sorting Name

OK Cancel Apply

Analysis Control

General Nonlinear

Geometry Nonlinearity

☐ Consider Geometric Nonlinear Effects

☐ Update Pore Pressure with Deformation

Basic Nonlinear Parameters

☒ Load Steps

Number of Increments 10

Intermediate Output Request Last Increment

☐ Manual with User-Defined Steps Load Step...

☐ Time Steps

Time(Duration) 86400 sec

Number of Increments 1

Intermediate Output Request Last Increment

☐ Manual with User-Defined Steps Time Step...

Iterative Scheme

☒ General ☐ Enhanced Init Stress

Convergence Criteria / Error Tolerance

☐ Displacement(U) 0.001

☒ Load(P) 0.001

☒ Work(W) 1e-06

☐ Use Iteration Method

☒ Arc-Length Method ☐ Displacement-Control Method

Min. Arc-Length Adjustment Ratio 0.25

Max. Arc-Length Adjustment Ratio 4

Max Arc-Length Increments 20

Advanced Nonlinear Setting...

OK Cancel

Output Control

Output Type Output Option

☒ Write Results of All Active Mesh Sets

Nodal Results

☒ Displacement Mesh Set...

☒ Applied Load Mesh Set...

☒ Reaction Force Mesh Set...

☒ Grid Point Force Mesh Set...

☒ Contact Mesh Set...

Element Results

☒ Force Mesh Set...

☒ Stress Mesh Set...

☒ Strain Mesh Set...

☒ Status Mesh Set...

☒ Damaged Index Mesh Set...

☒ Ductility Mesh Set...

☒ Multi-layered Grid Mesh Set...

Element Output Location

☒ Element Corner Results

☐ Shell Mid-Plane Results

☒ Composite Shell Mid-Plane Results

Number of Beam Output Segments 4

Output Option

☒ Binary ☐ Binary and Text

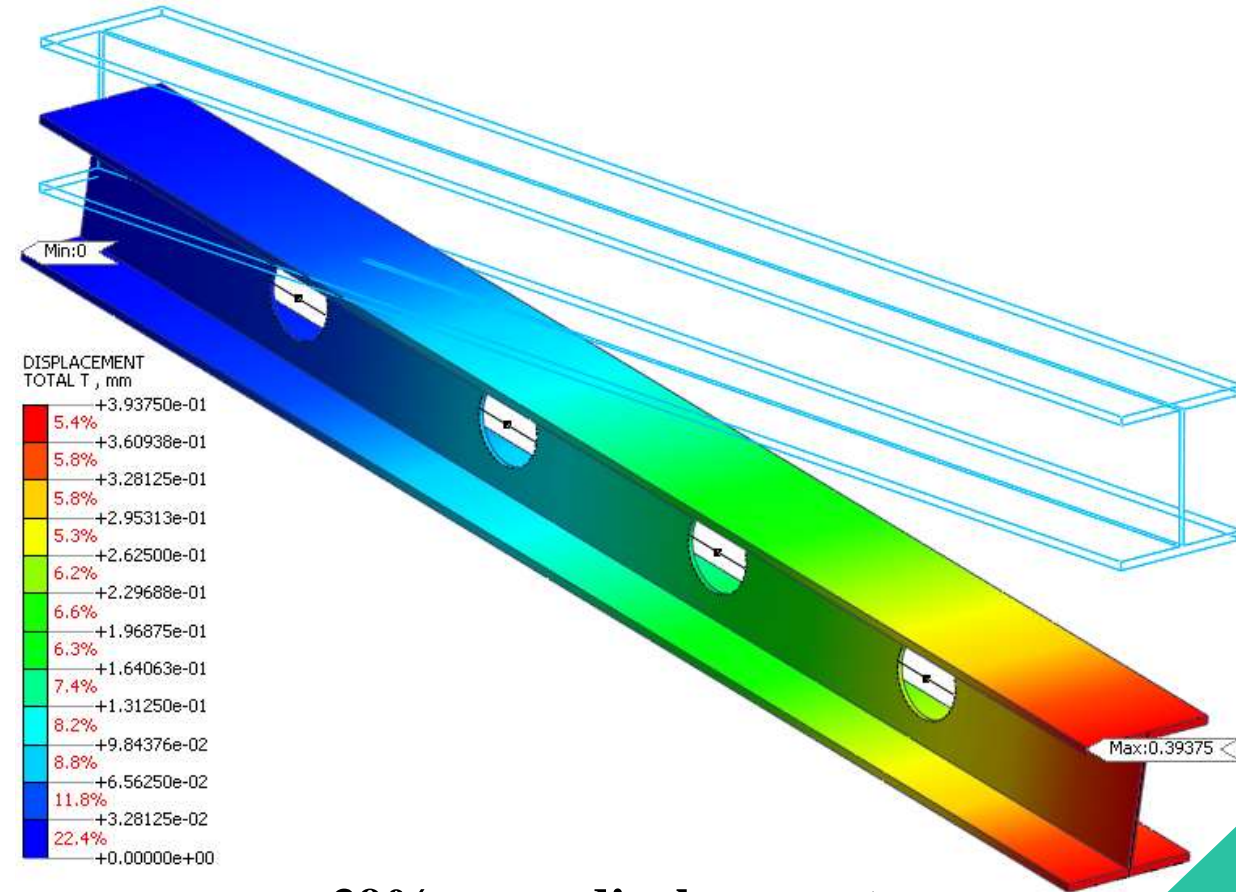
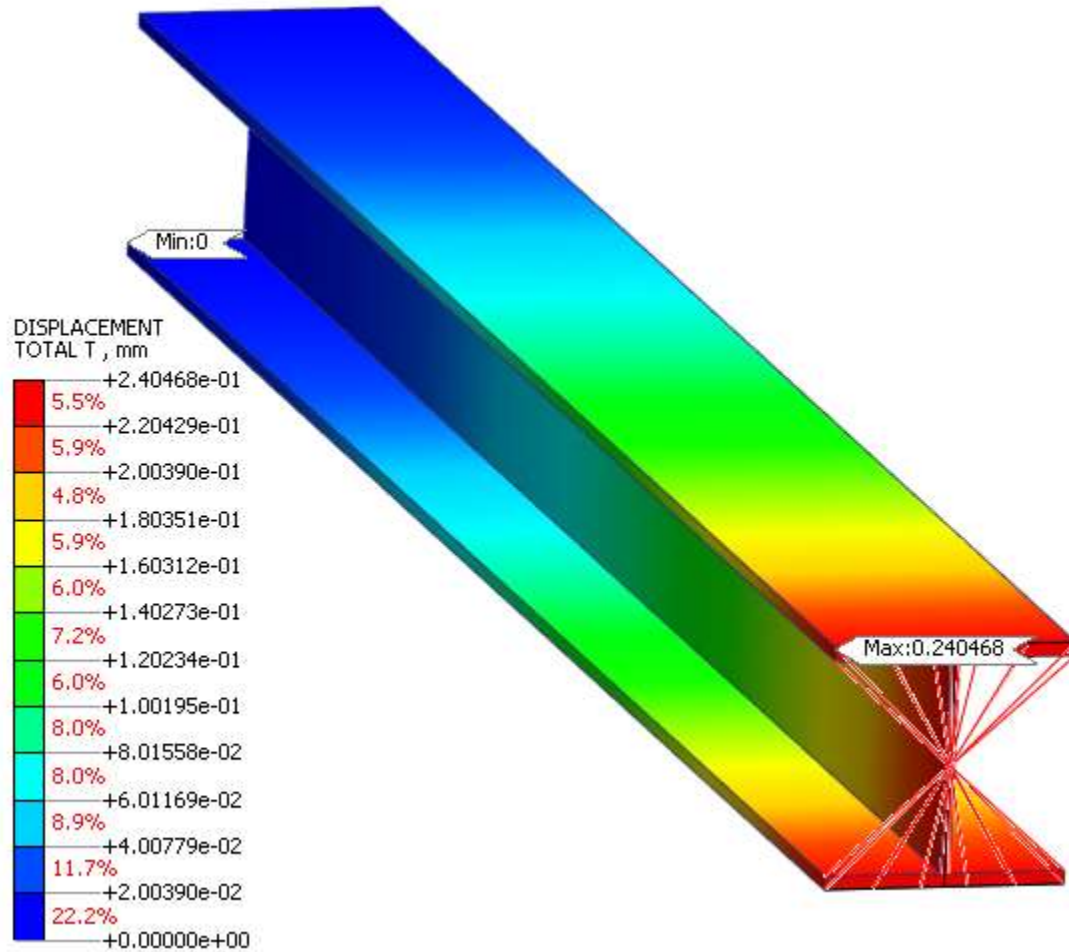
OK Cancel

# Part 3

## **Solid and Modified Beam Result Comparison**

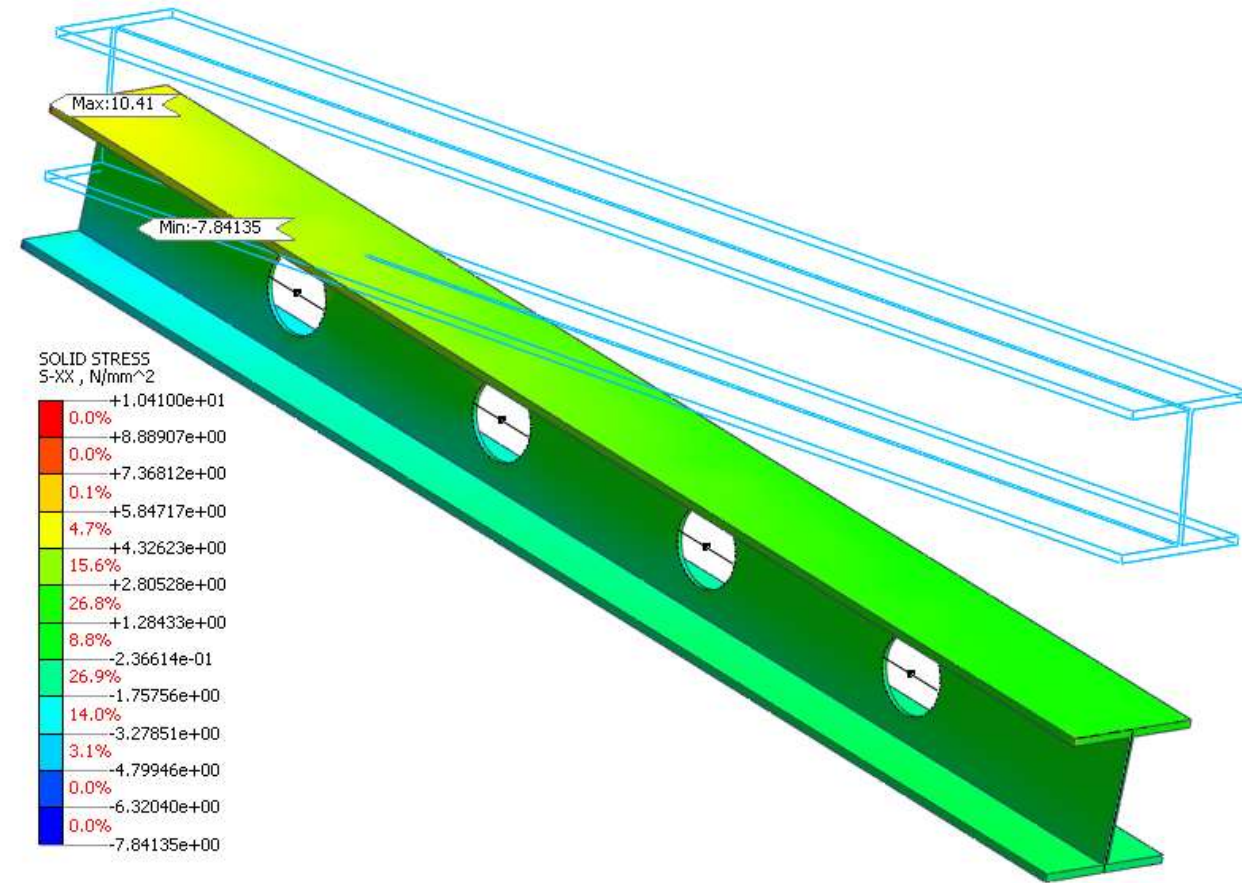
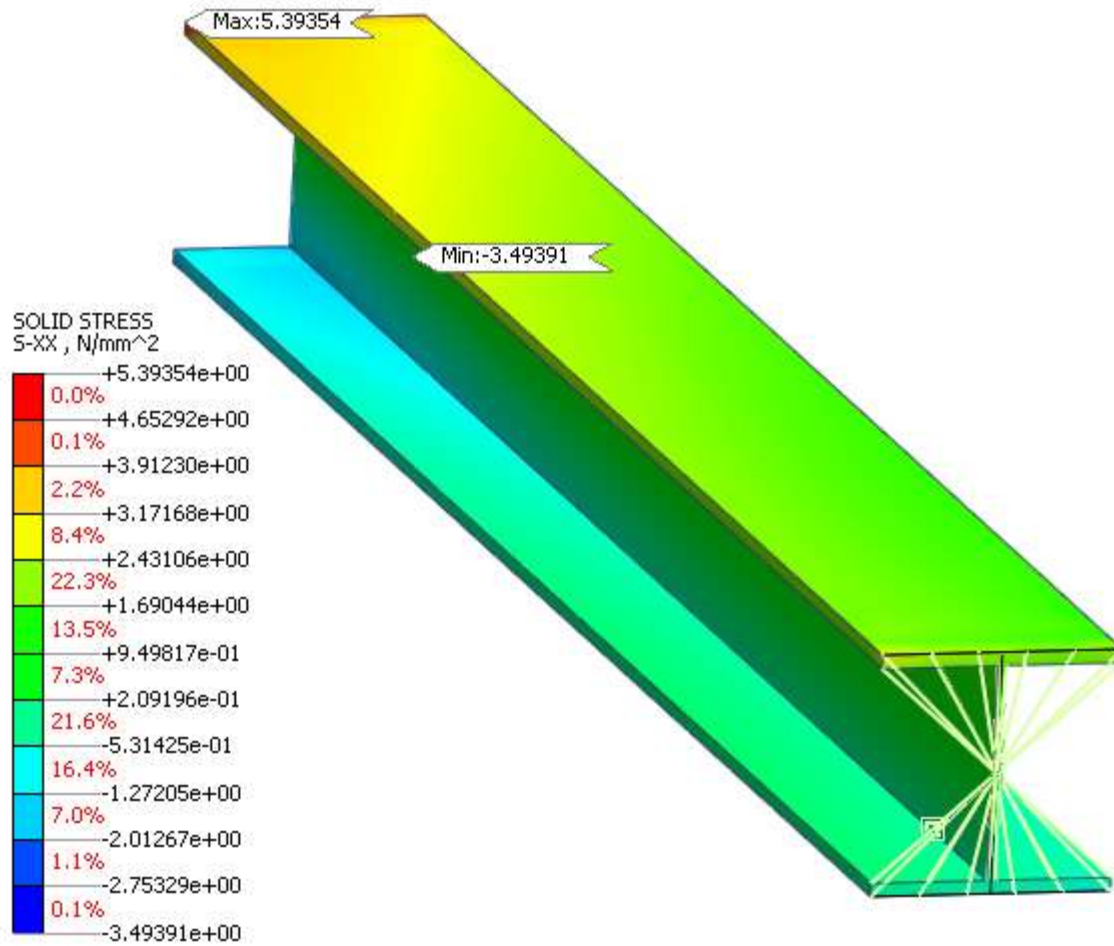


# RESULTS - BEAM DEFLECTION COMPARISON



~39% more displacement

# RESULTS - BEAM STRESS COMPARISON

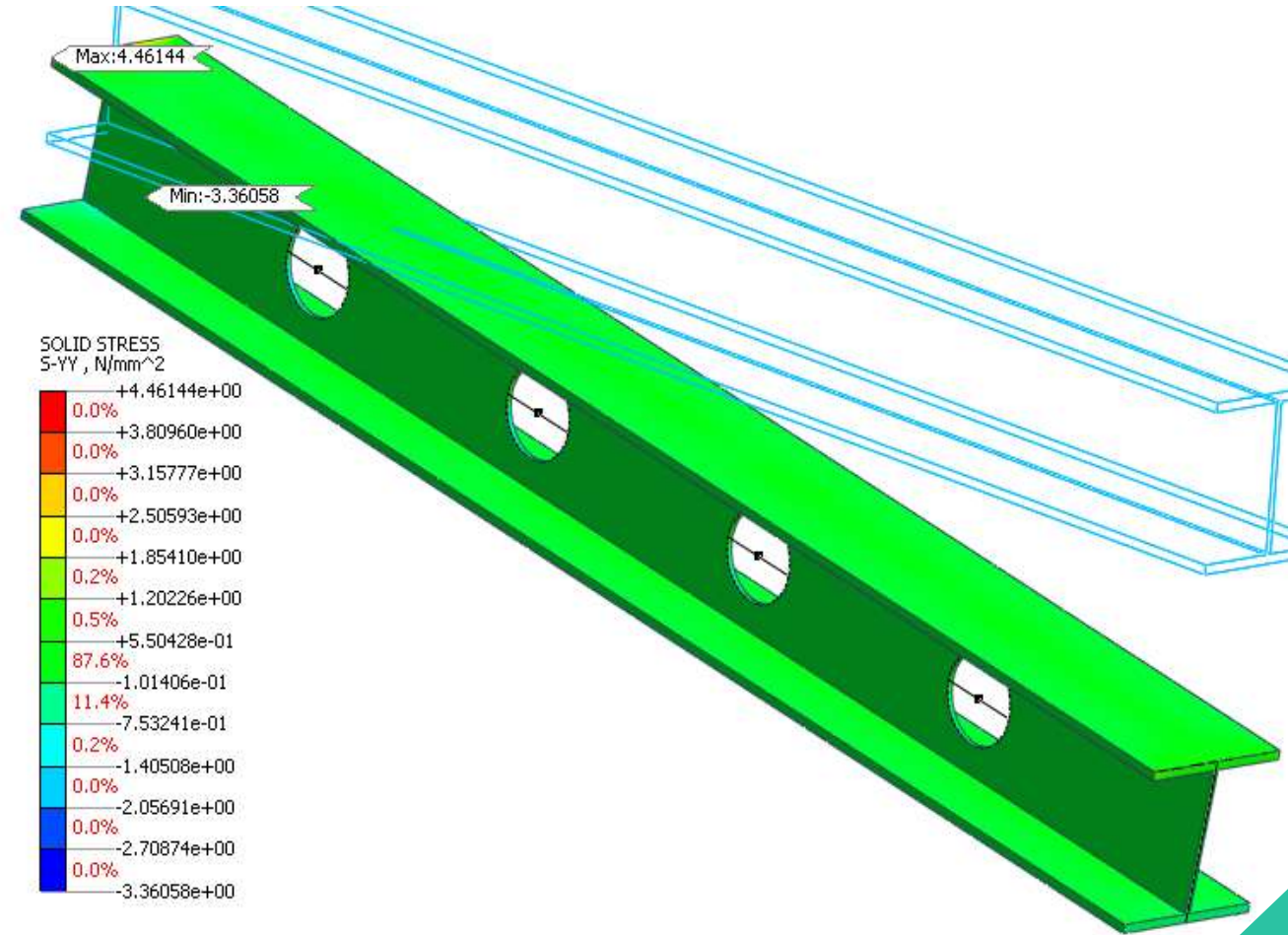
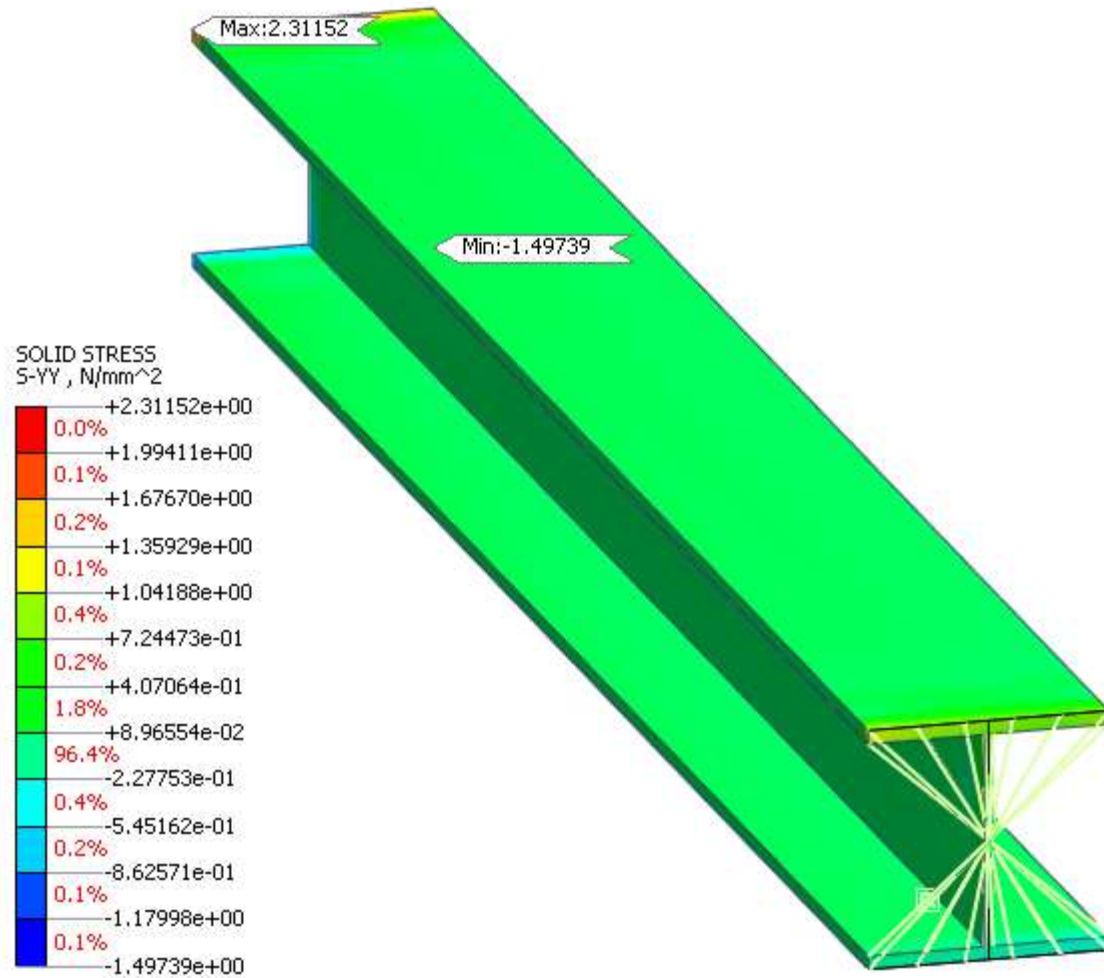


**Max S-XX:** ~93% increase

**Primary region S-XX:** ~15% increase



# RESULTS - BEAM STRESS COMPARISON

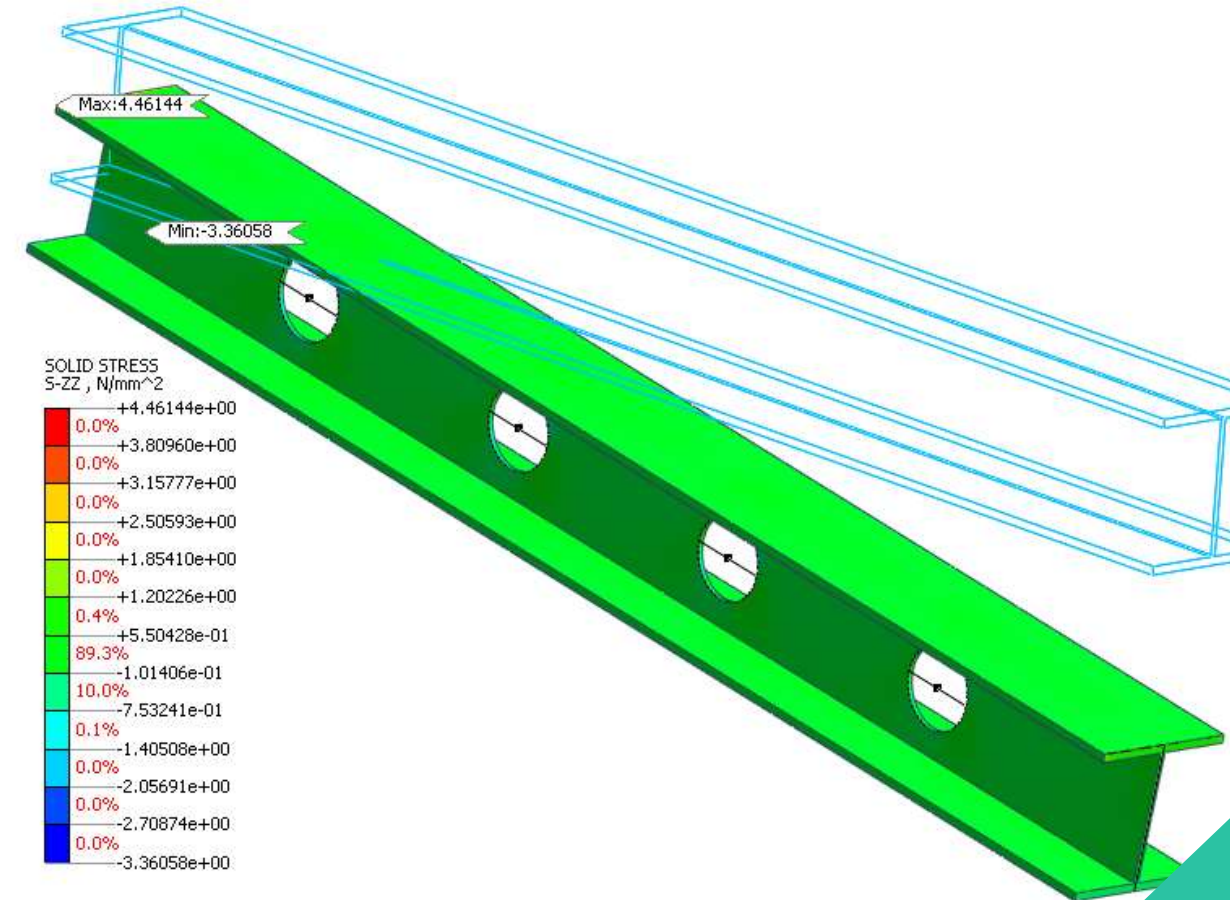
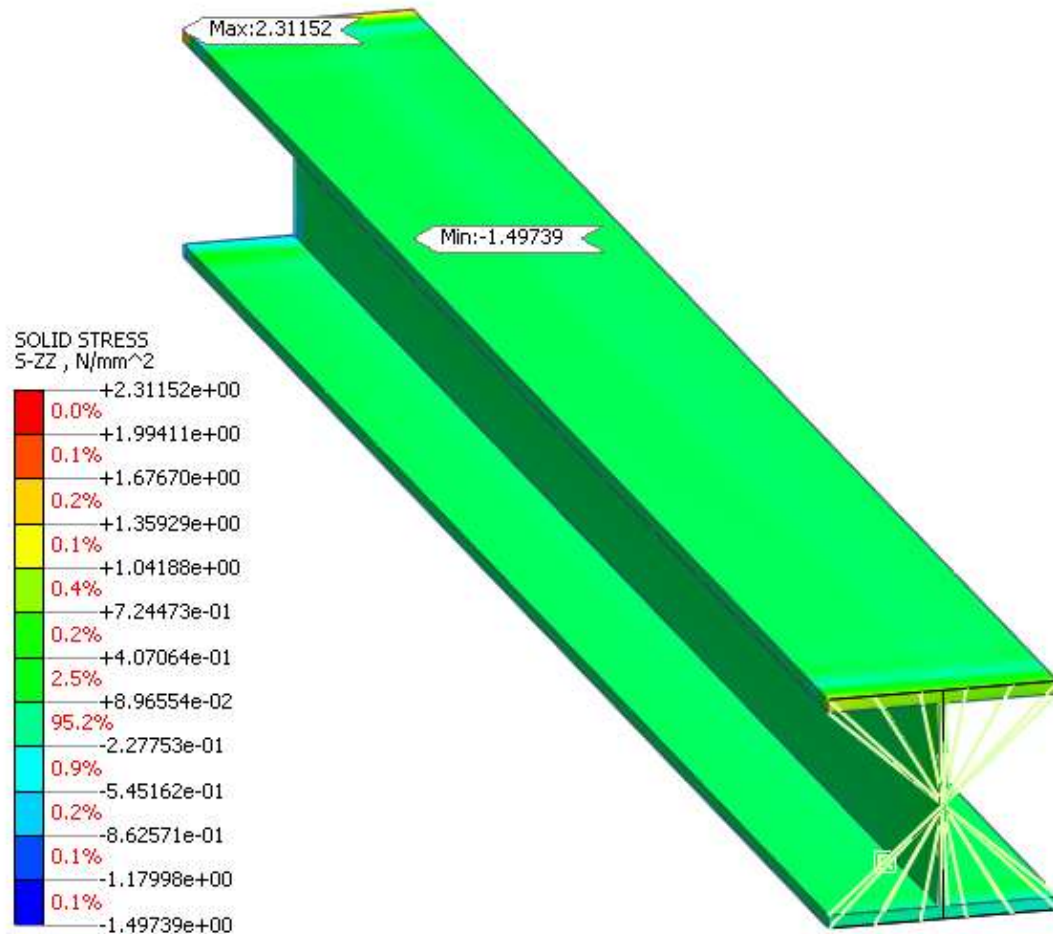


**Max S-YY: ~93% increase**

**Primary region S-YY: 5x increase**

**Compression Region: 0.8% --> 11.6%**

# RESULTS - BEAM STRESS COMPARISON



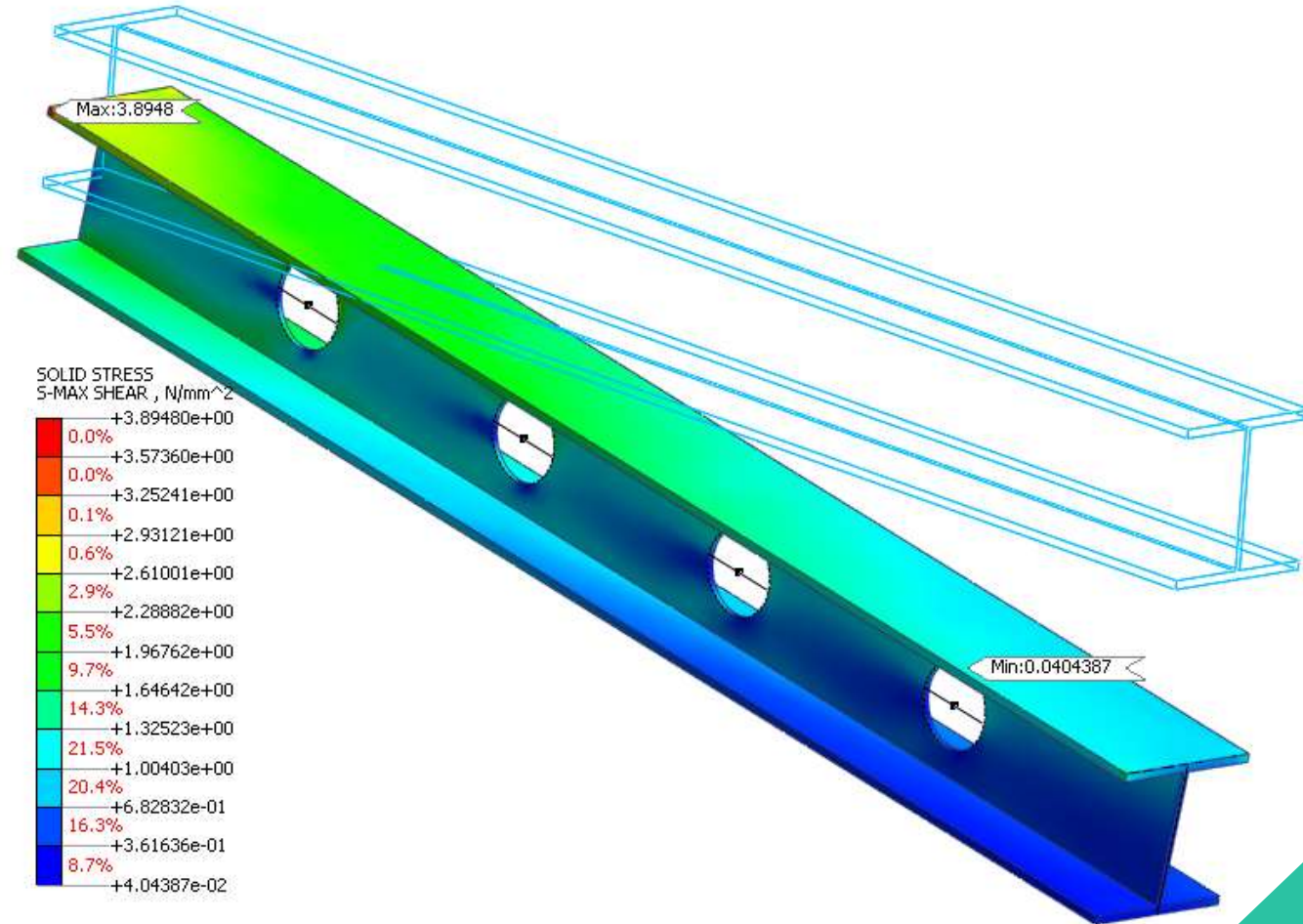
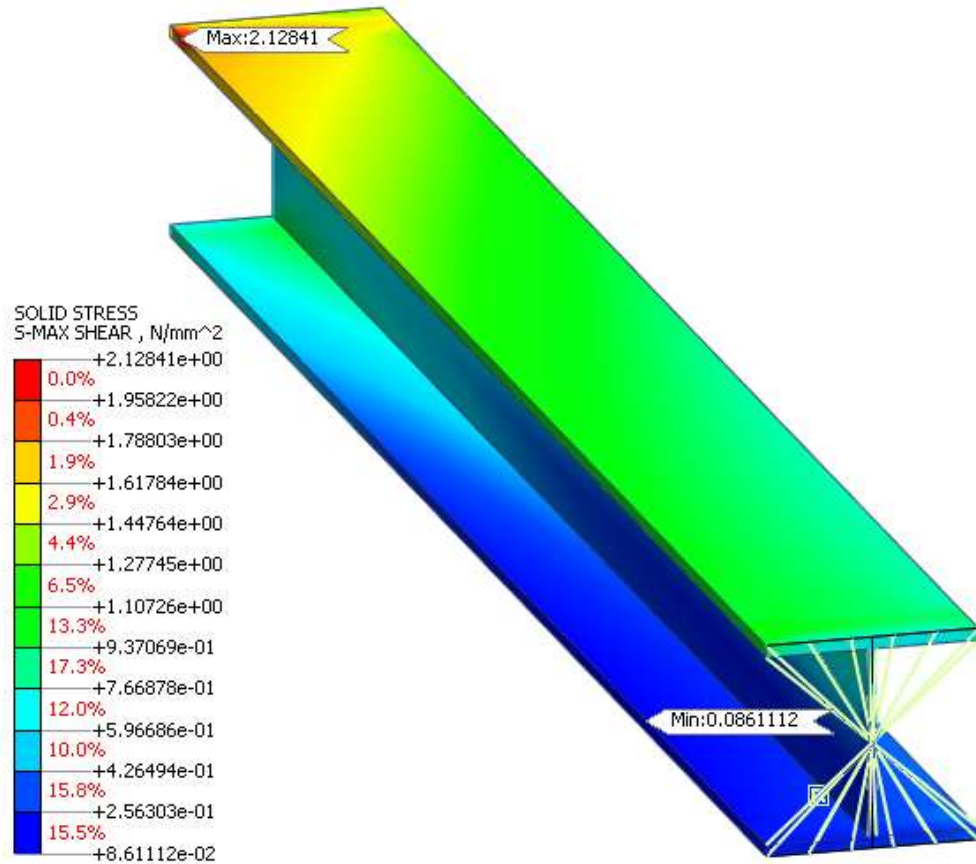
**Max S-ZZ:** ~93% increase

**Primary region S-ZZ:** 5x increase

**Compression Region:** 0.13% --> 11.1%



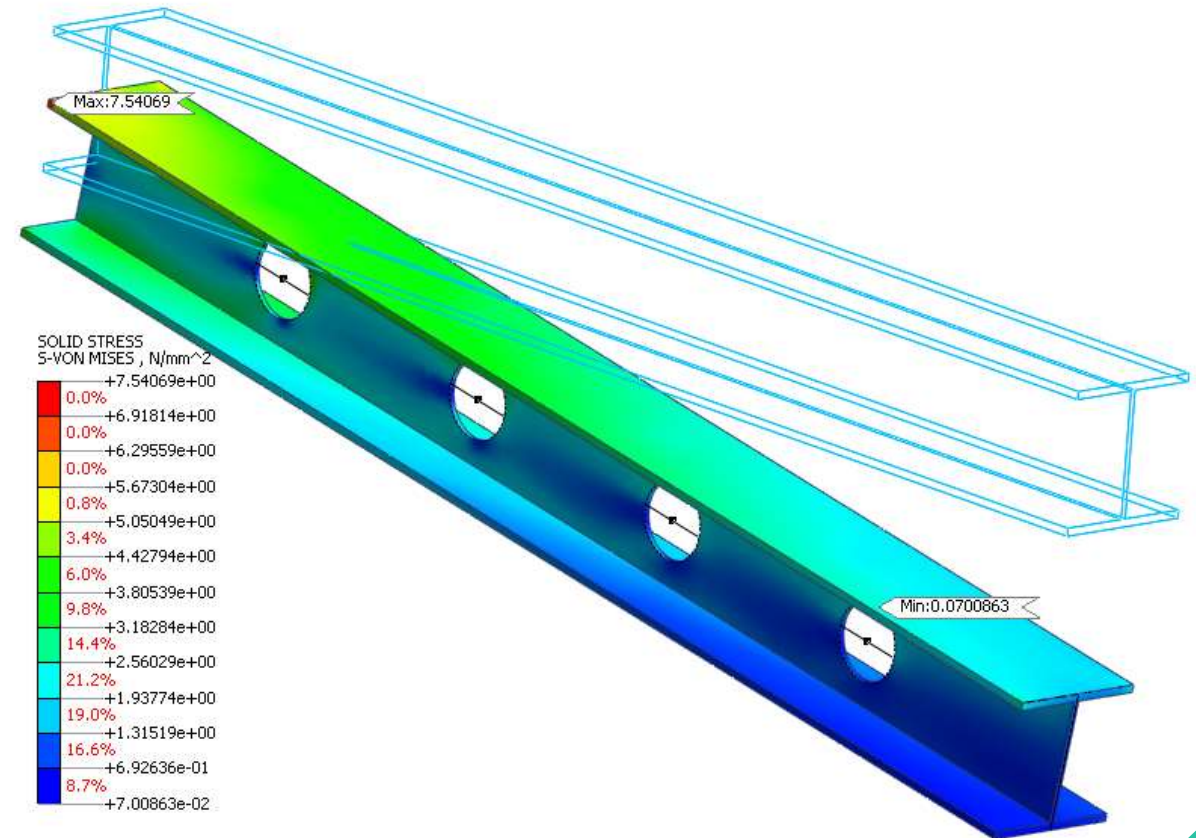
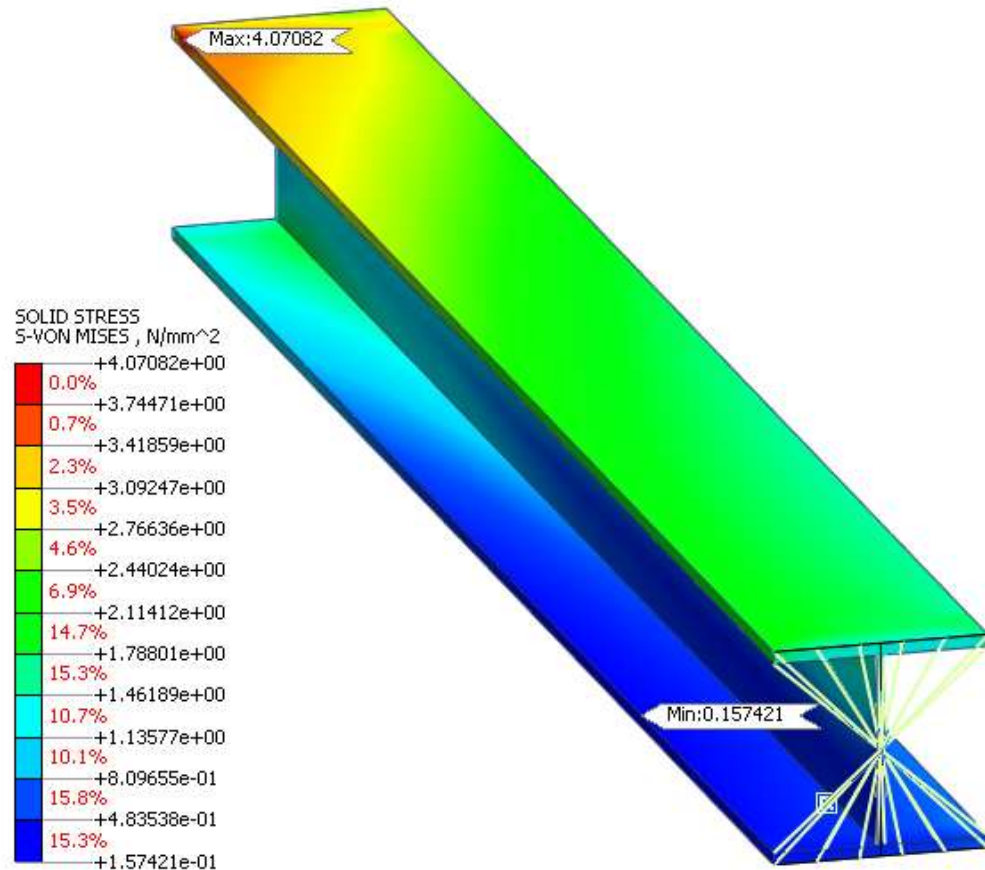
# RESULTS - BEAM STRESS COMPARISON



**Max Shear: ~77.8% increase**

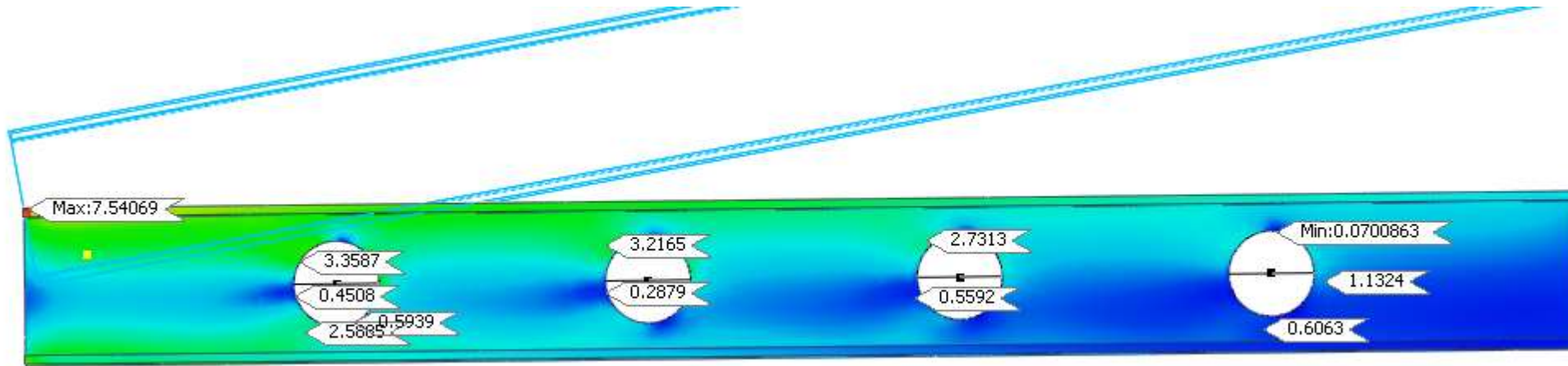
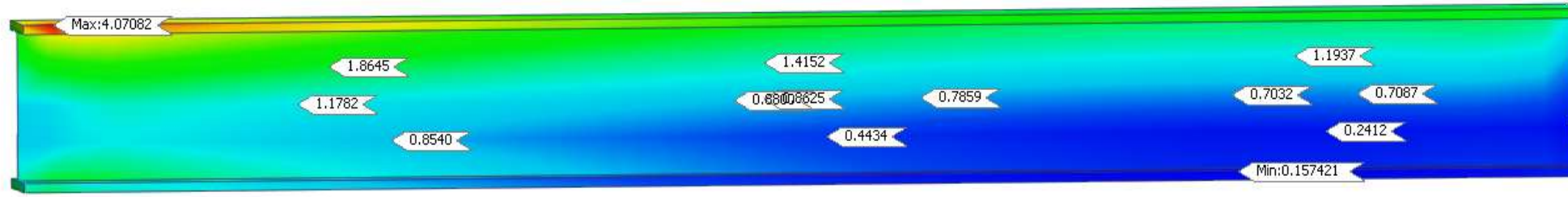


# RESULTS - BEAM STRESS COMPARISON



**Max S-Von mises: ~85.2% increase**

# RESULTS - BEAM STRESS COMPARISON



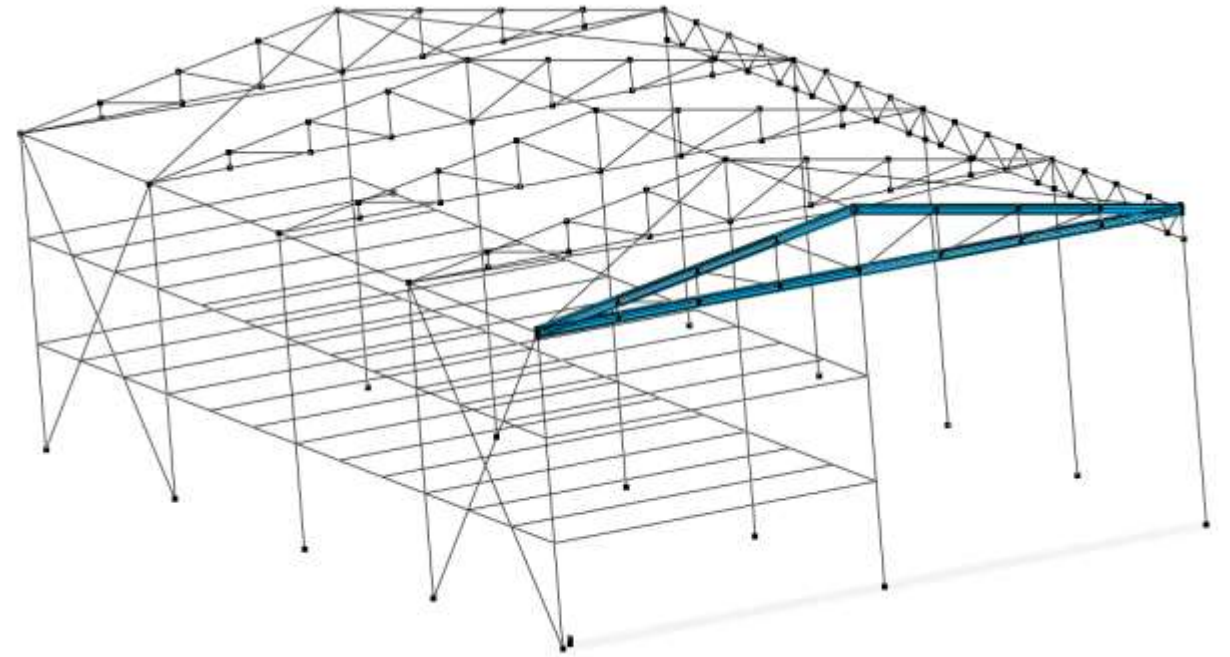
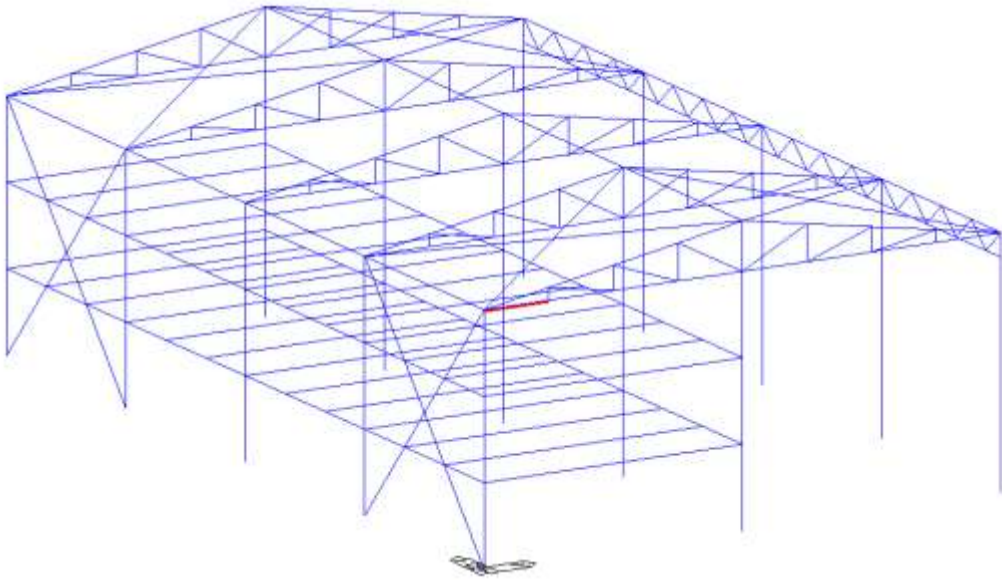
**Max S-Von mises:** Created high stress points near the hole, about 2x increase

# FEA NX Simulation Platform

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

# CASE ANALYSIS OVERVIEW

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

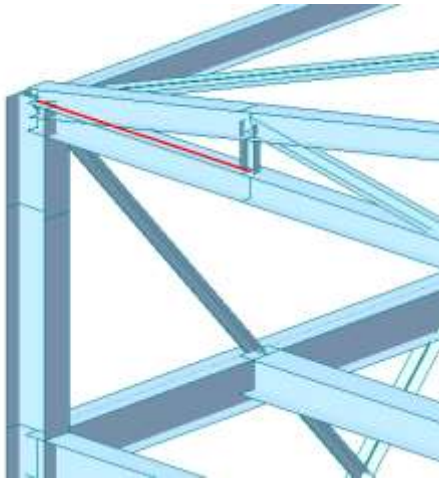
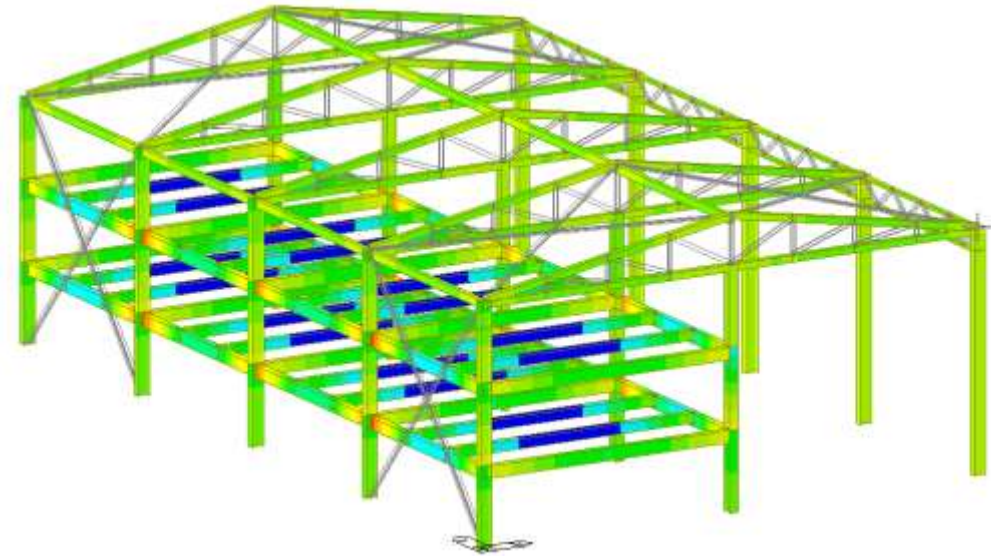
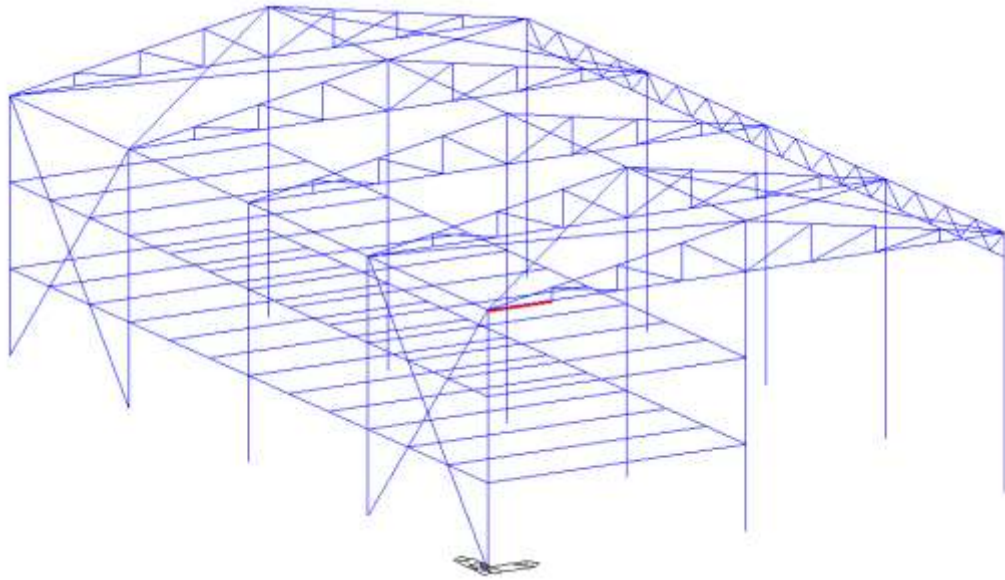


# Part 0

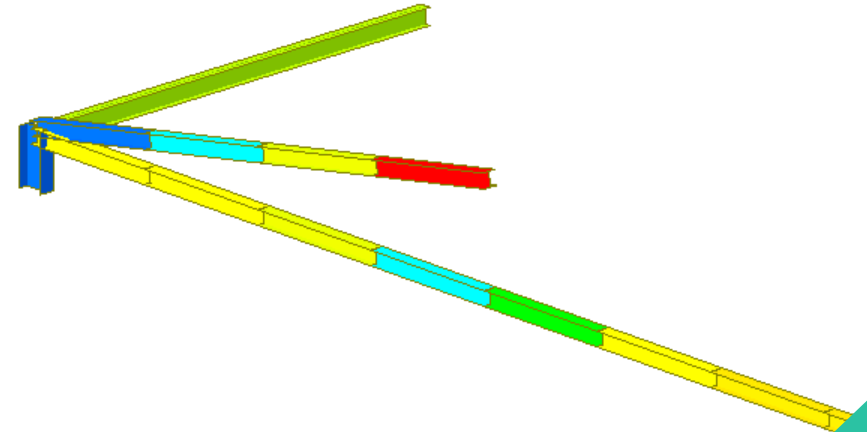
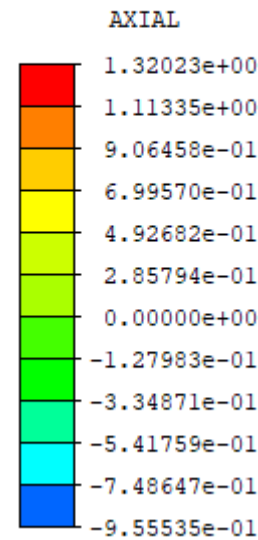
## GEN Simulation and Results



# MIDAS GEN MODEL and RESULTS

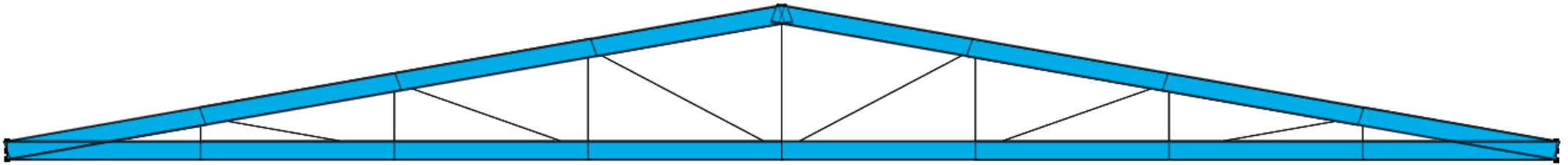


BEAM FORCE

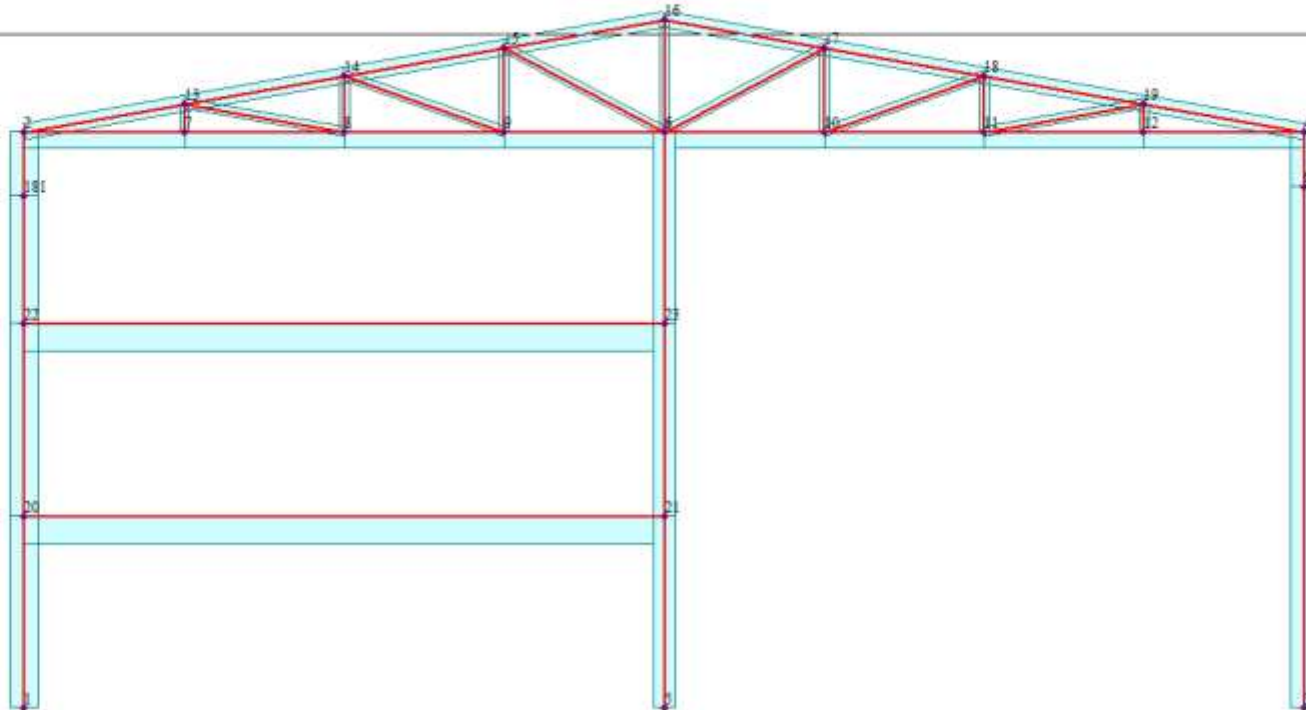
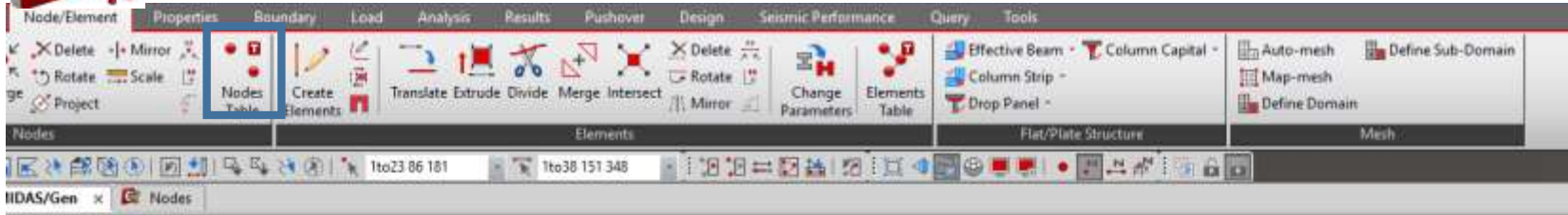


# Part 1

## Frame Detailed Analysis

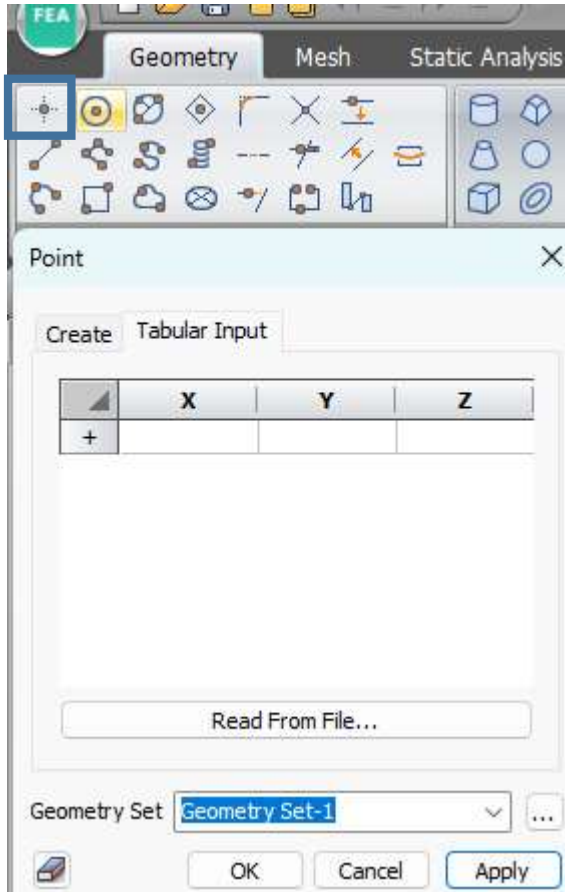


# FRAME GUIDE



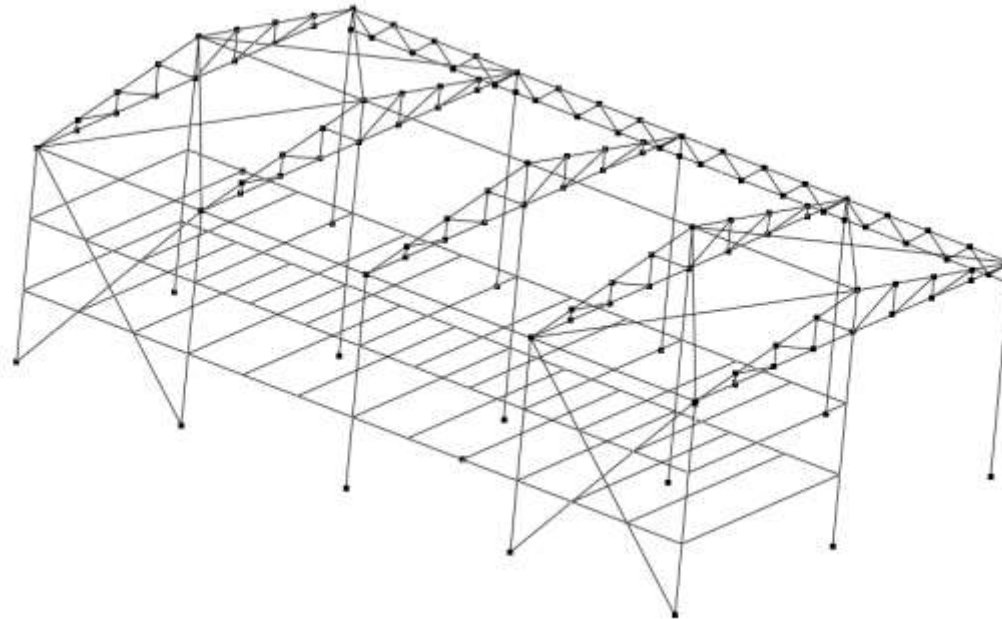
	Node	X(m)	Y(m)	Z(m)
	1	0.000000	0.000000	0.000000
▶	2	0.000000	0.000000	9.000000
	3	20.000000	0.000000	0.000000
	4	20.000000	0.000000	9.000000
	5	10.000000	0.000000	0.000000
	6	10.000000	0.000000	9.000000
	7	2.500000	0.000000	9.000000
	8	5.000000	0.000000	9.000000
	9	7.500000	0.000000	9.000000
	10	12.500000	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)

The image displays the software interface for creating 3D geometry, specifically focusing on the 'Section Data' and 'Rectangle' dialog boxes.

**Section Data Dialog:**

- Section ID: 6
- Name: RH 250x250x9x14
- DB Name: AISC10(US)
- Sect. Name: RH 250x250x9x14
- Dimensions (mm):
  - H: 250
  - B1: 250
  - tw: 9
  - tf1: 14
  - B2: 0
  - tf2: 0
  - r1: 13
  - r2: 13
- Offset: Center-Center
- Consider Shear Deformation: ☒
- Consider Warping Effect(7th DOF): ☐

**Rectangle Dialog:**

- Method: ☒ Make Face
- Location: 250, 14
- Method: ABS x, y
- Geometry Set: Geometry Set-1
- Location: 9, 222
- Method: REL dx, dy
- Geometry Set: Geometry Set-1

The background shows a 3D grid with a red vertical rectangle and a green horizontal rectangle, illustrating the created geometry. A coordinate system (X, Y, Z) is visible in the top right corner.

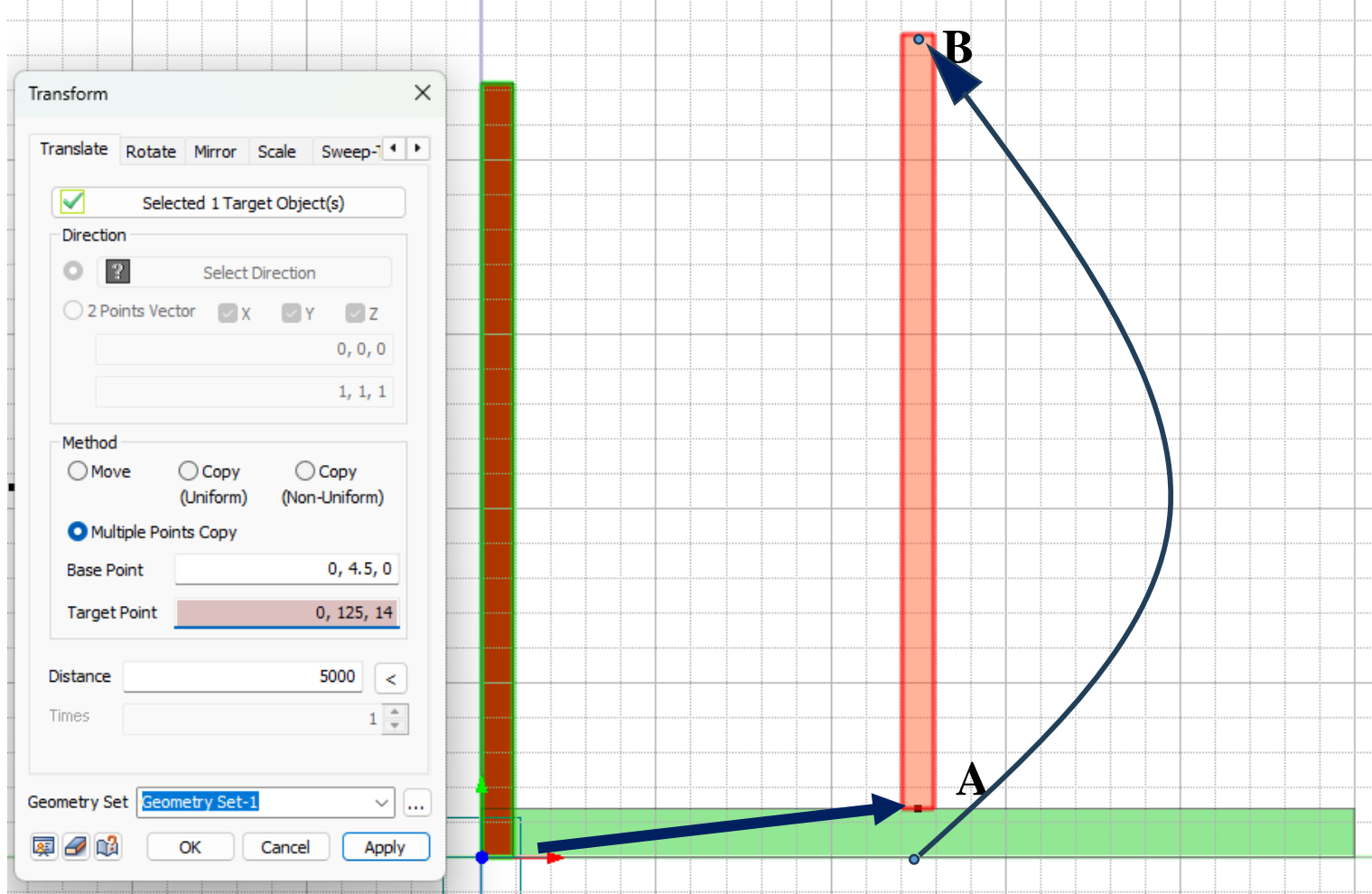
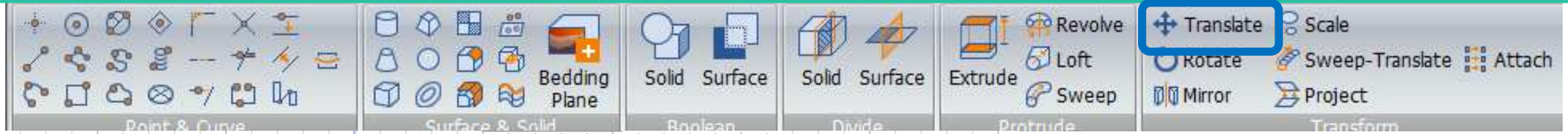
Following the section database for **RH** section

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

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



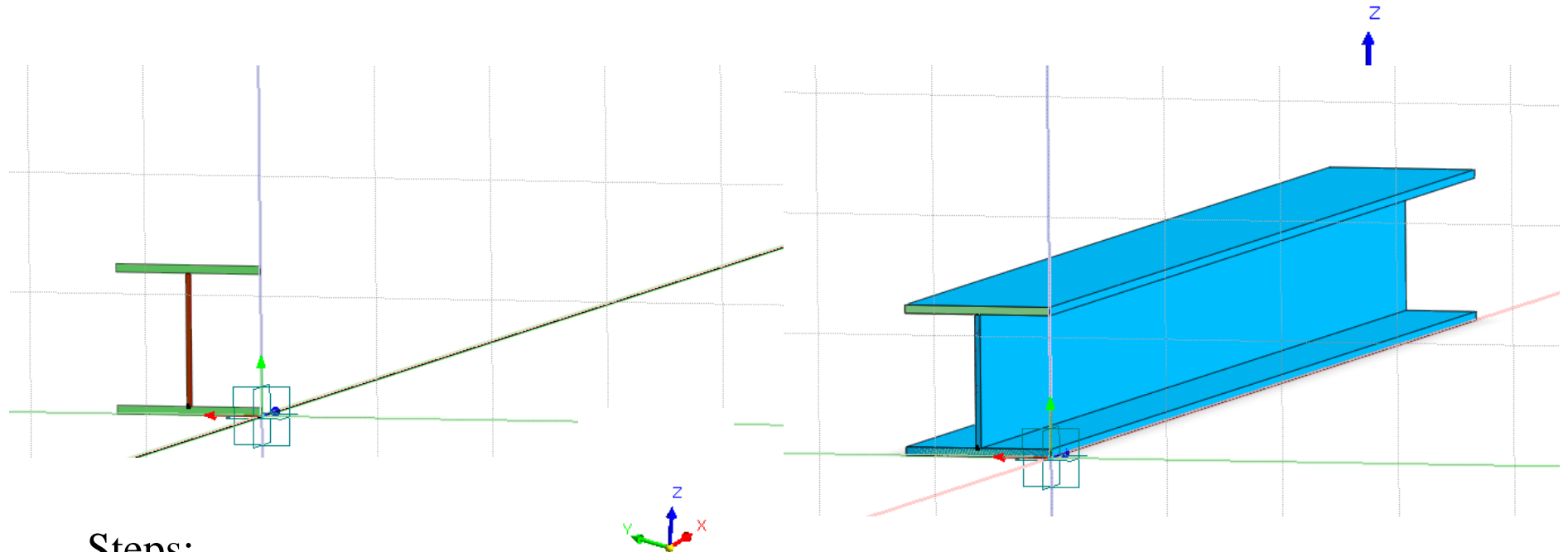
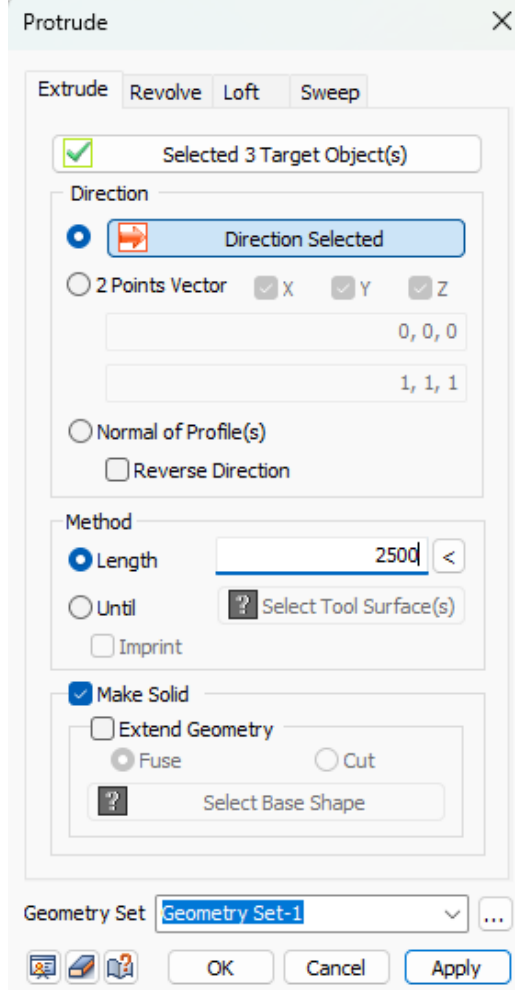
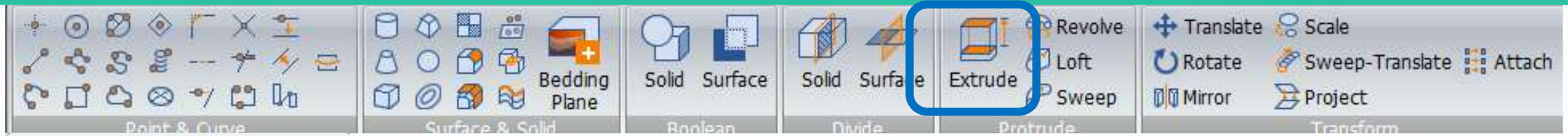
# ADDITIONAL GEOMETRY (3D)



## Steps

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

# ADDITIONAL GEOMETRY (3D)



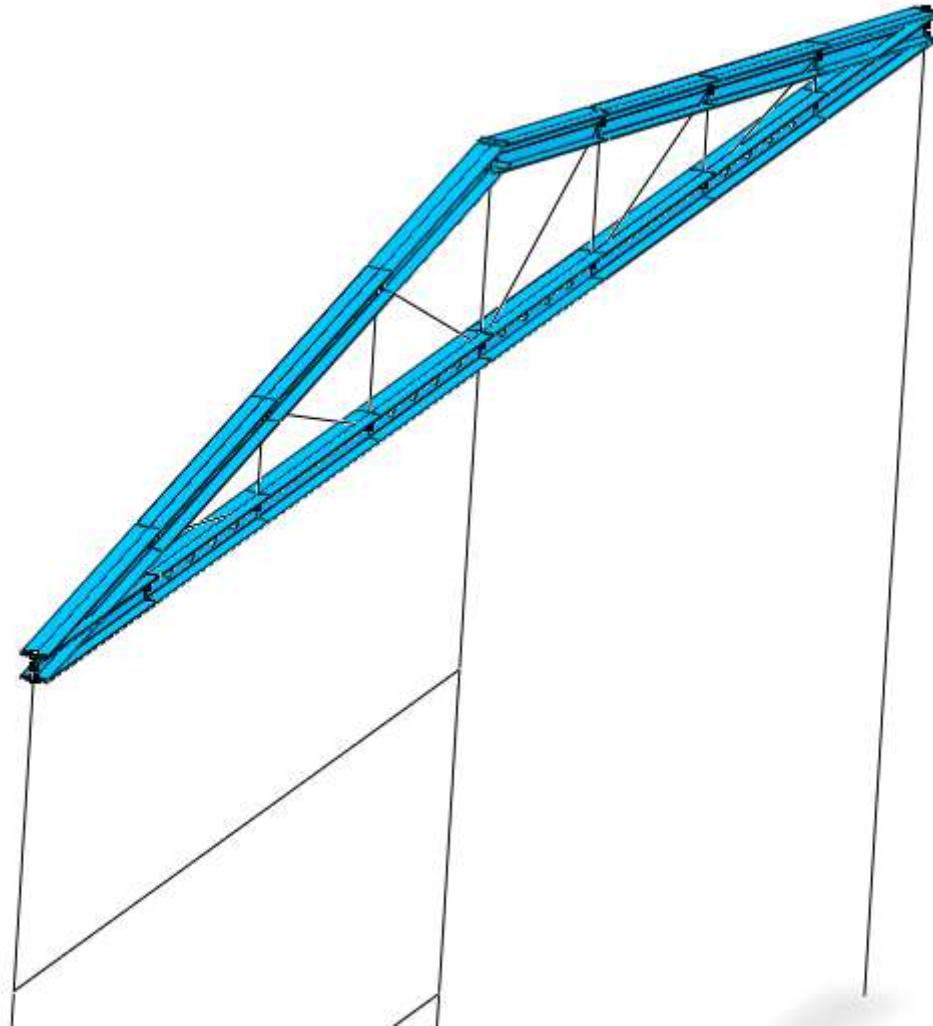
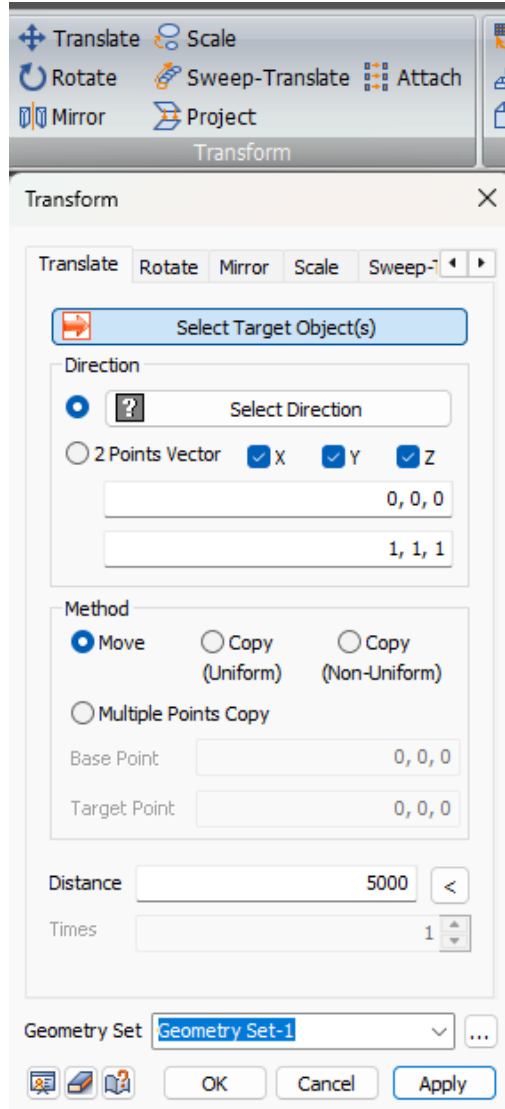
Steps:

Select all rectangles

1. Direction : X-axis

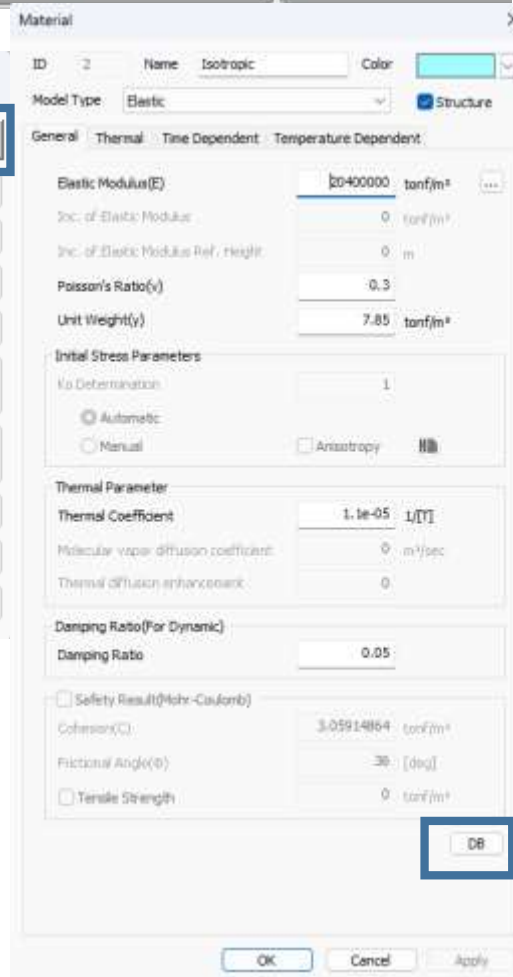
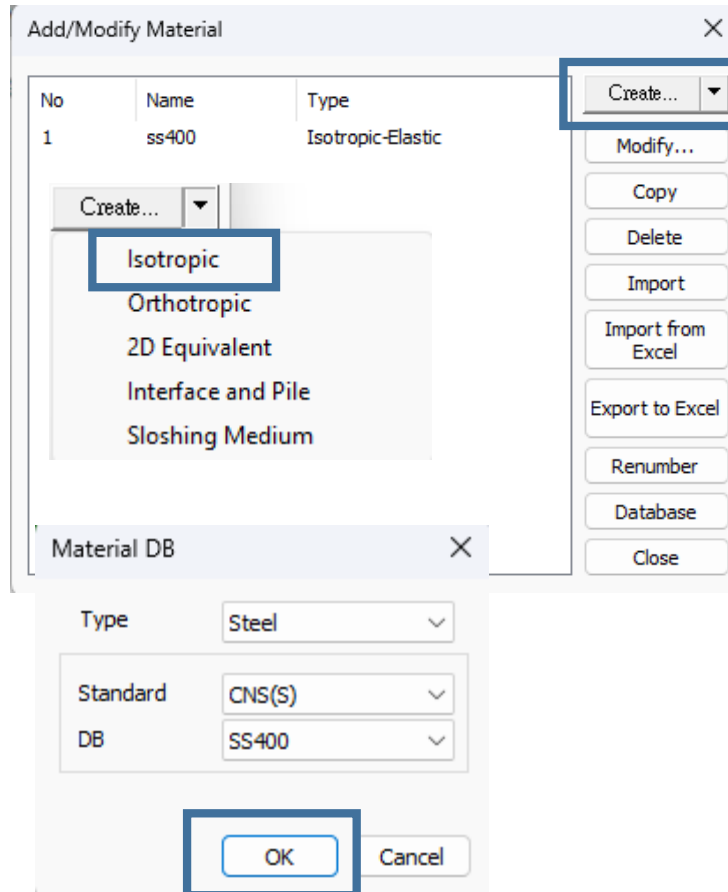
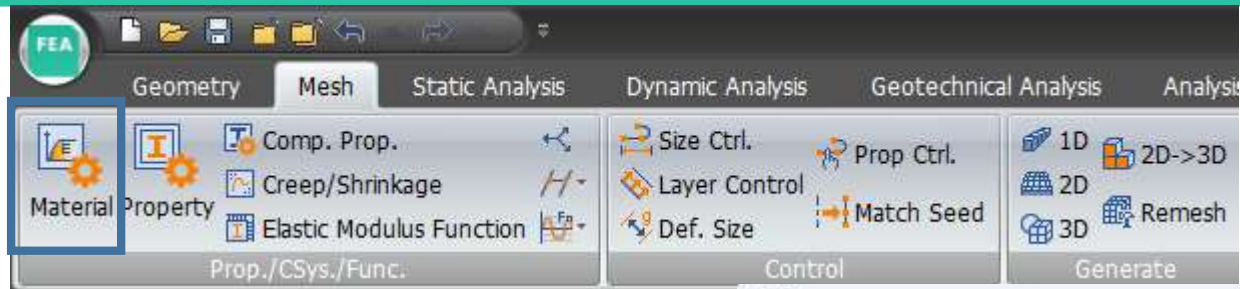
2. Length of beam 2.5m

# FRAME GUIDE



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

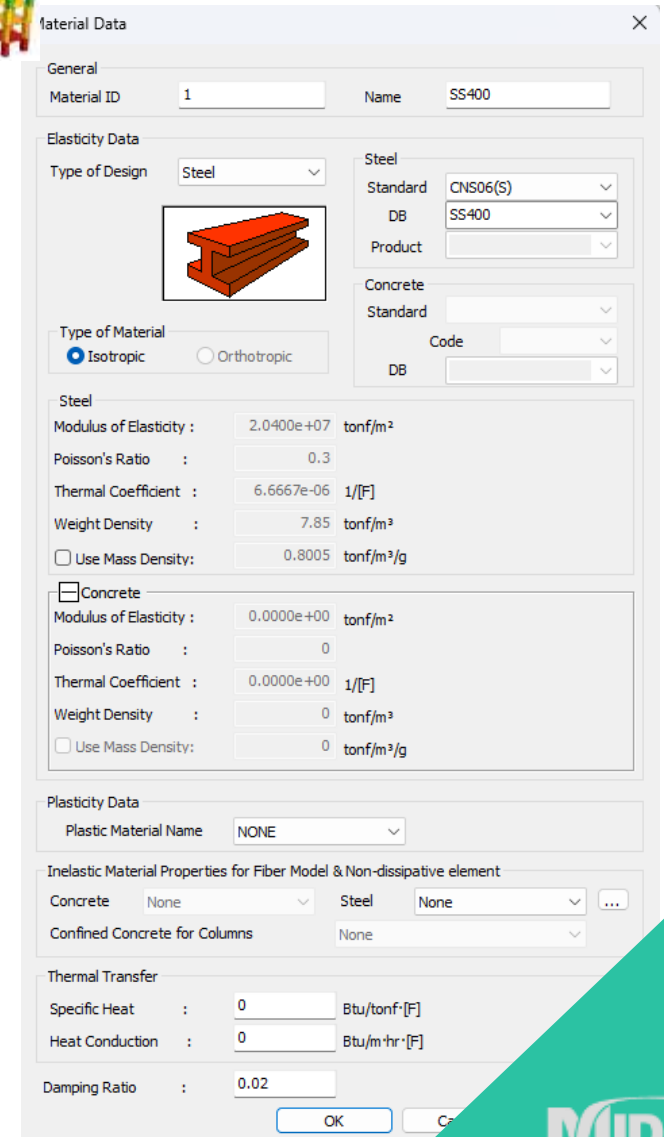
# MATERIALS AND PROPERTIES



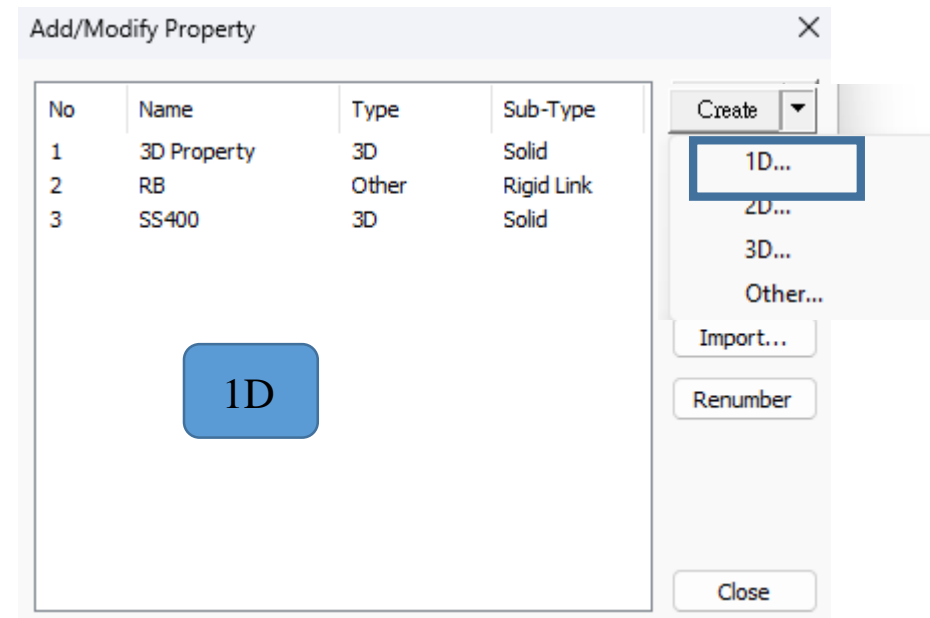
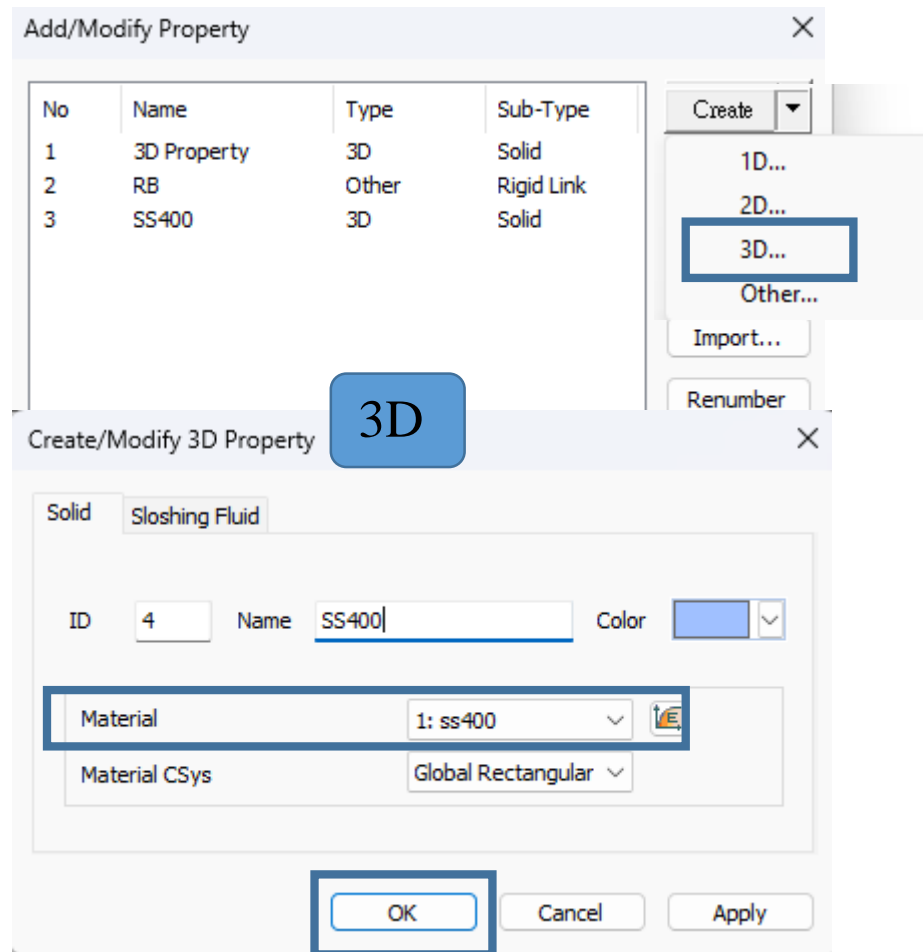
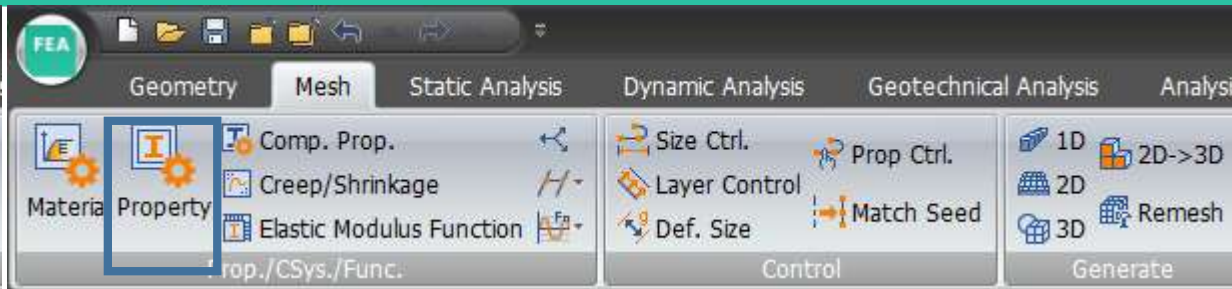
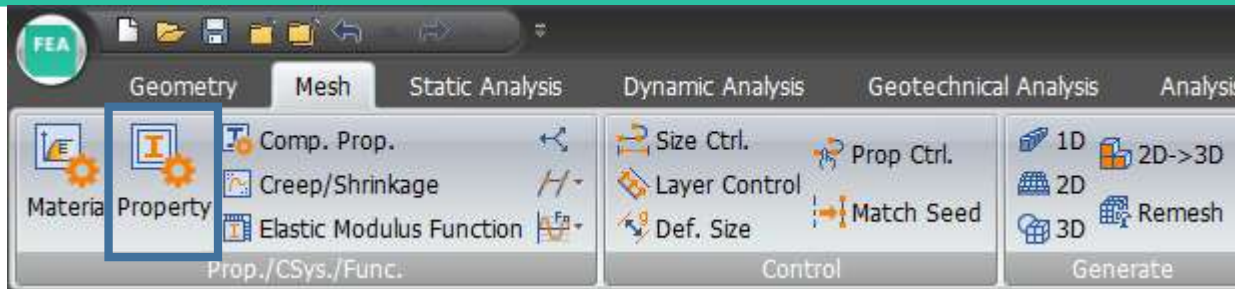
**NOTE:**  
Material Parameters  
are modifiable.



From MIDAS GEN

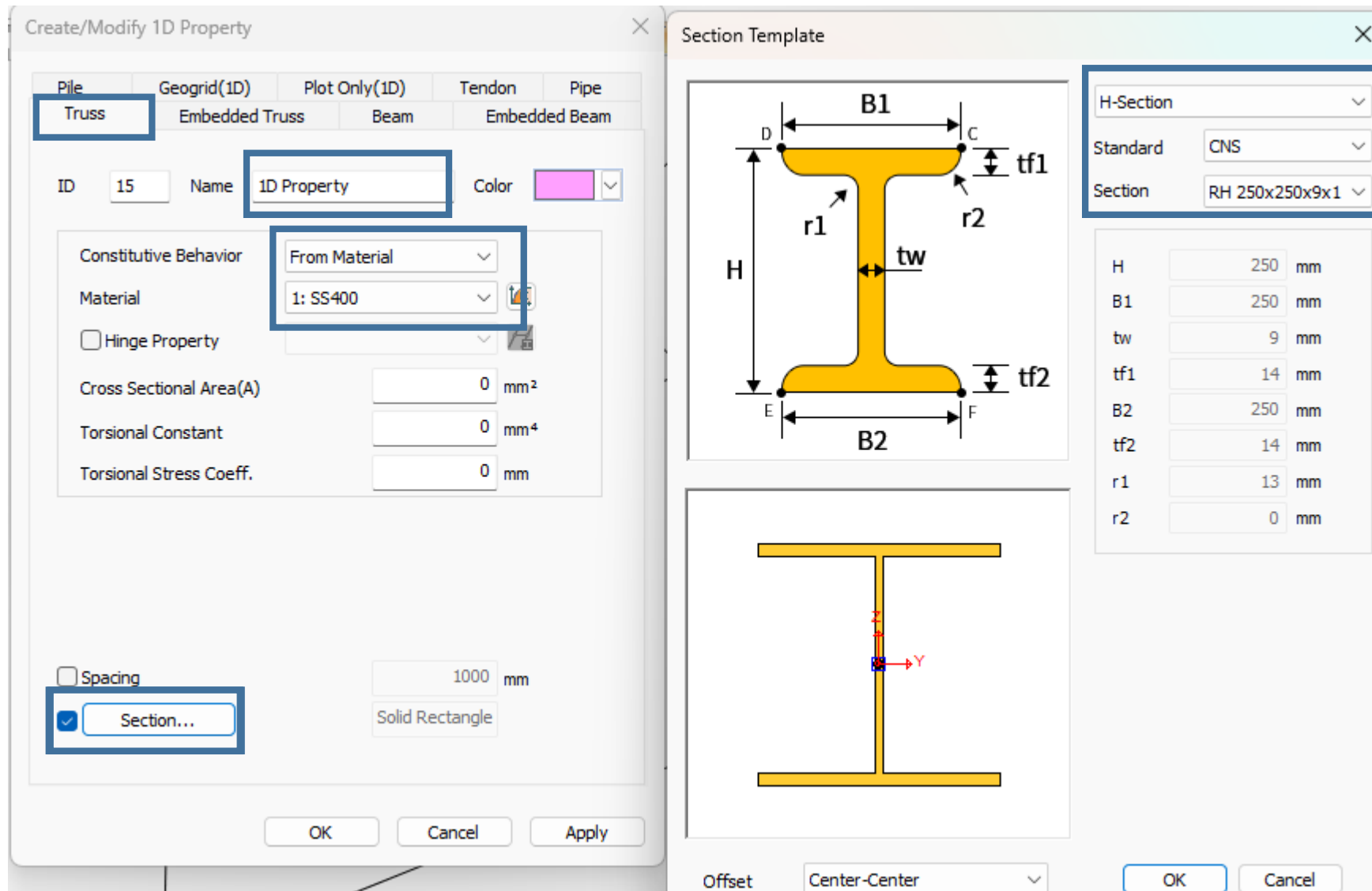


# MATERIALS AND PROPERTIES





# MATERIALS AND PROPERTIES



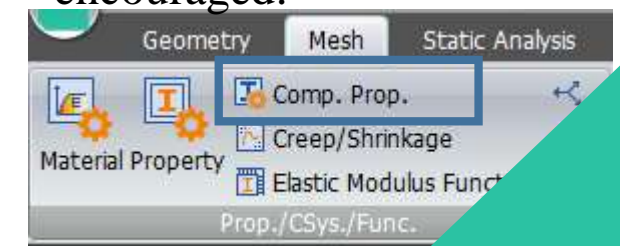
For 1D elements, the beam section database is recommended.

Take note of the following:

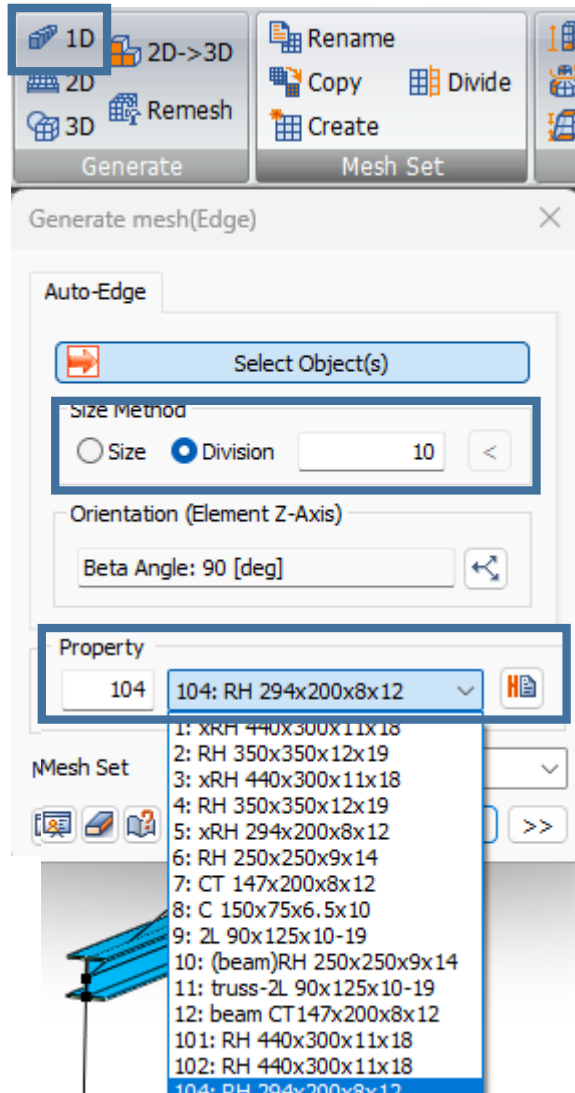
1. 1D Type
2. Material
3. Section
  - Standard
  - Section Shape

Additional Note:

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



# MESHING (1D and 3D)



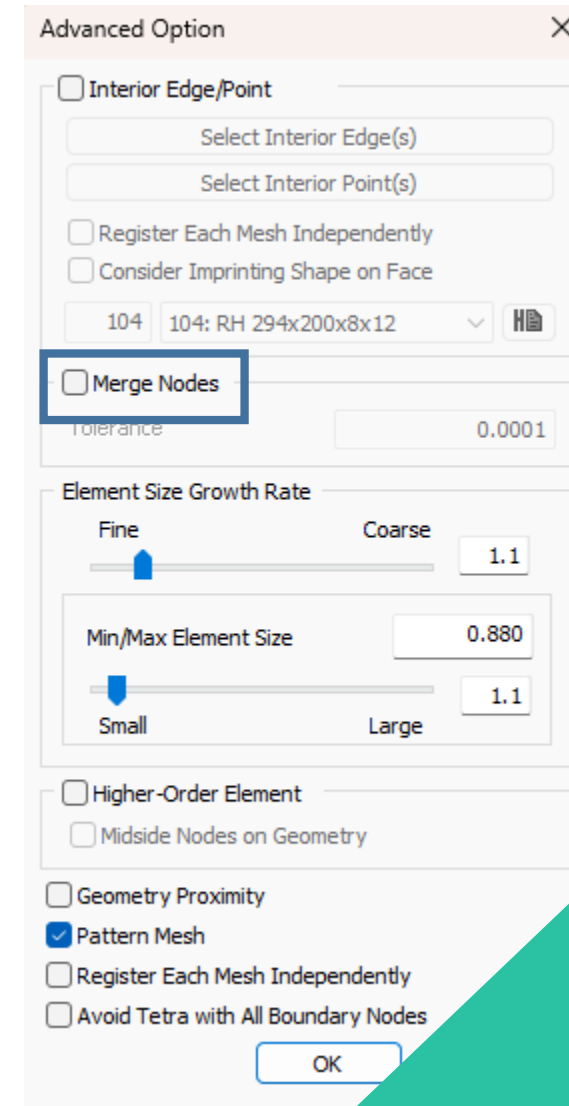
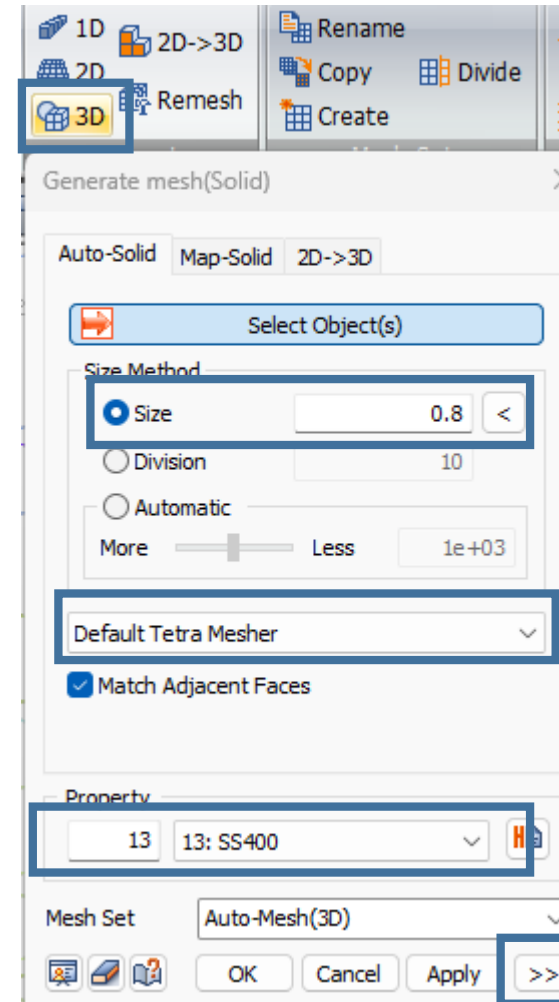
For **1D elements**, define the appropriate section property.

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

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

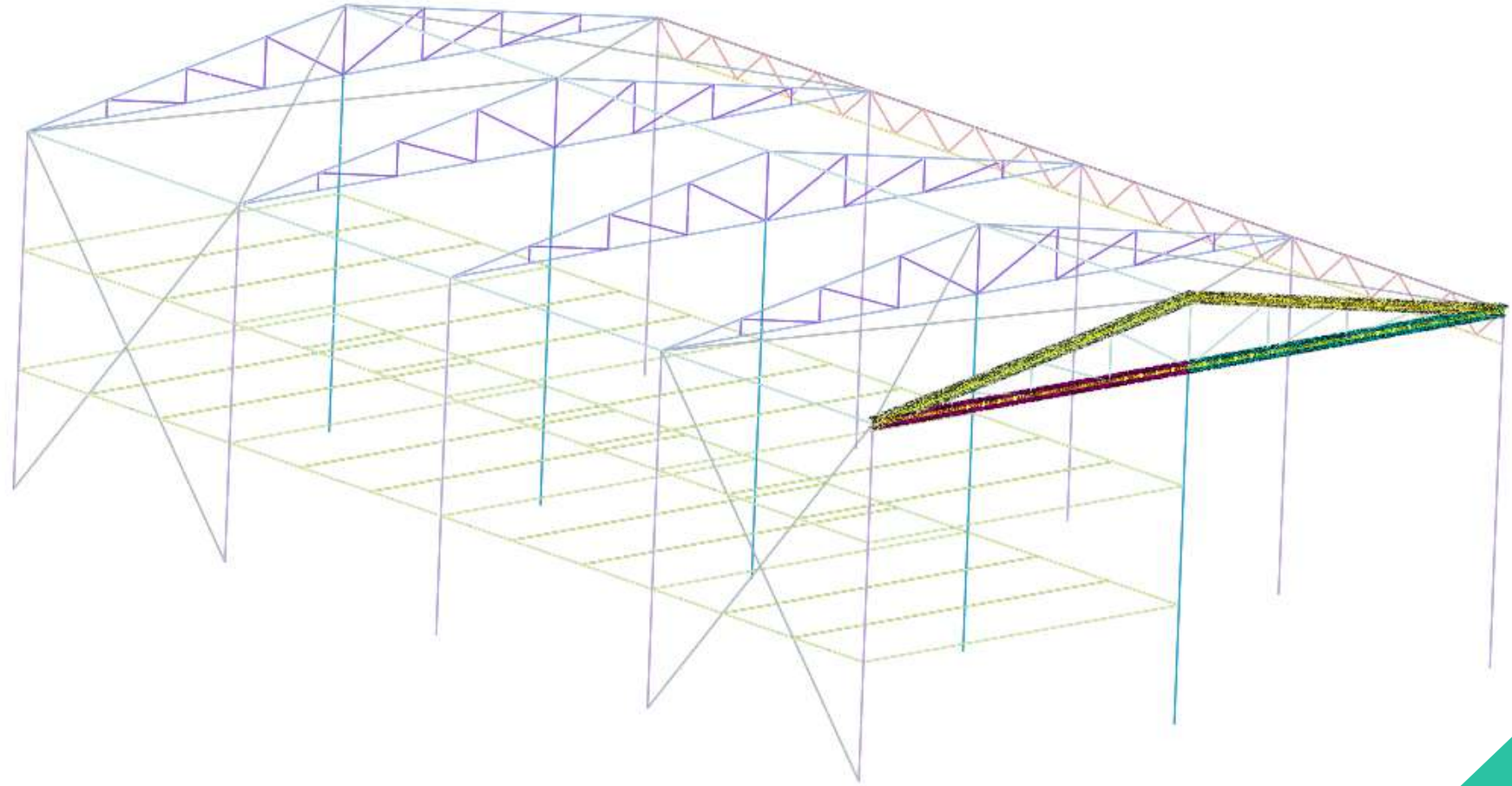
NOTE:

1. For future GEN export, use Tetra Mesher as mesh type
2. Deselect Merge nodes to create a segmented parts of the truss.

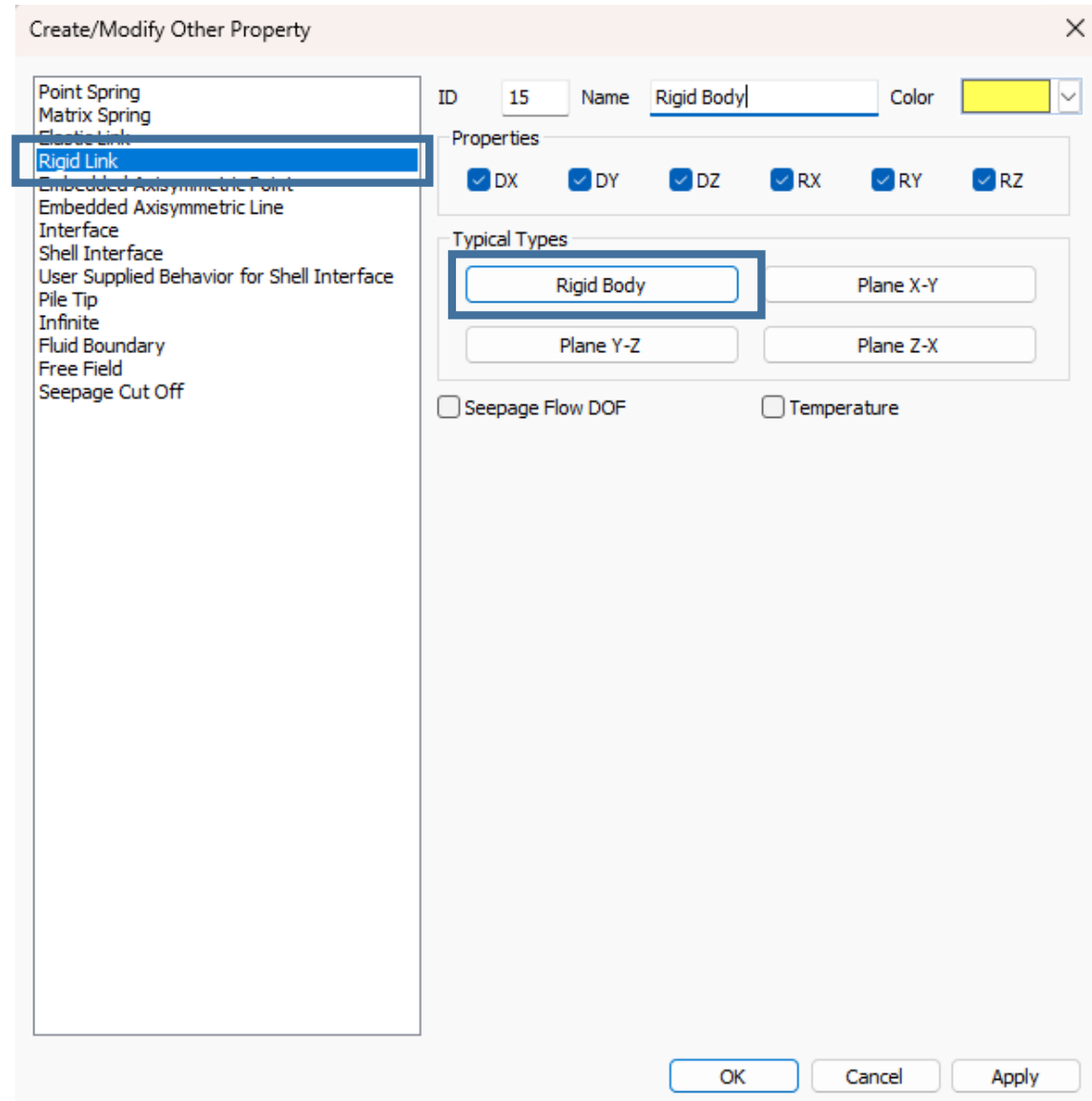
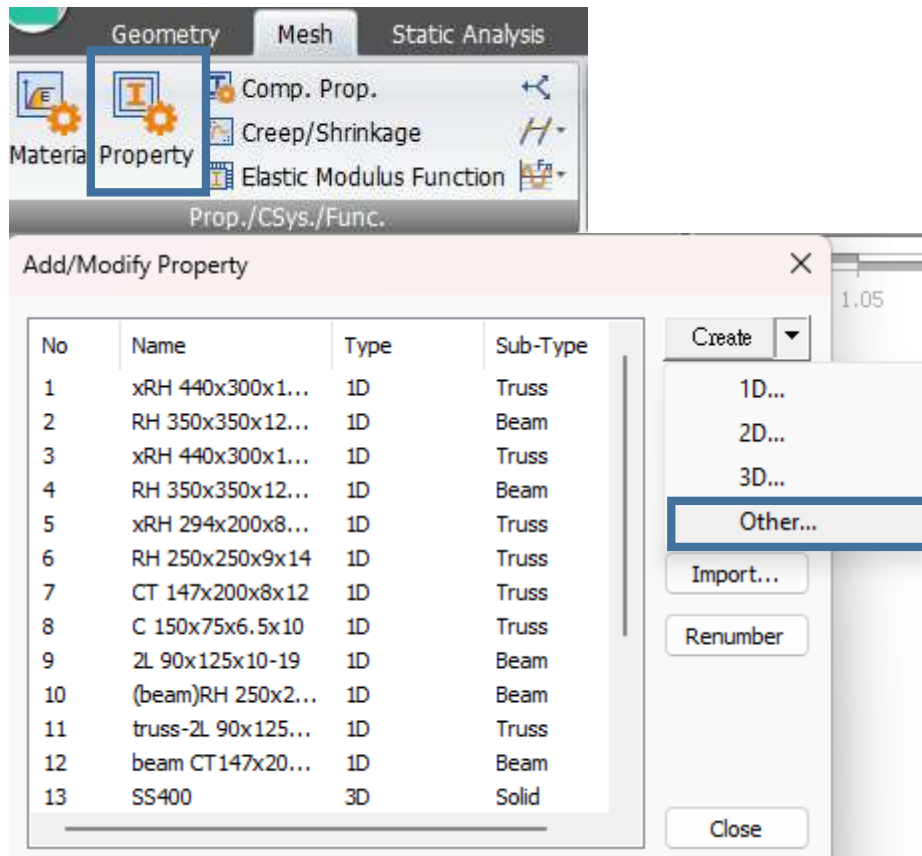


# MODEL

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



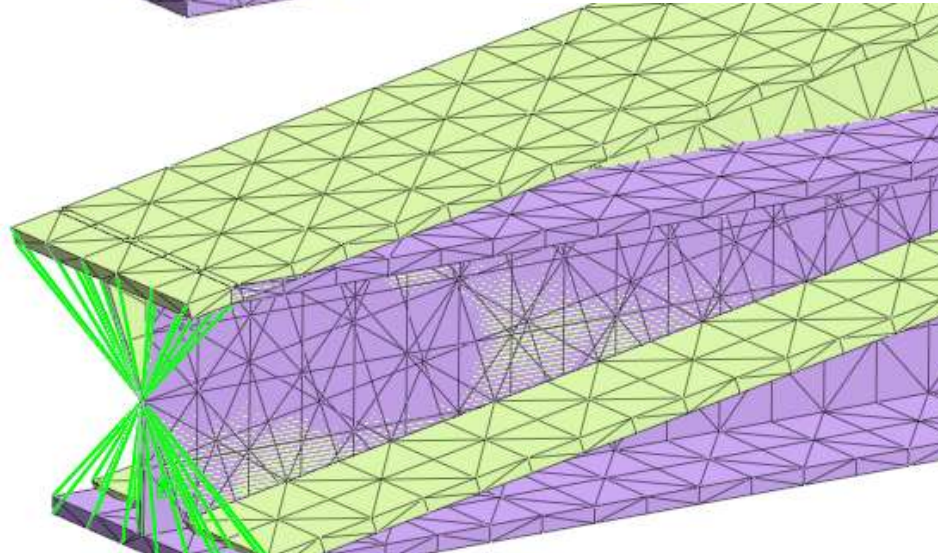
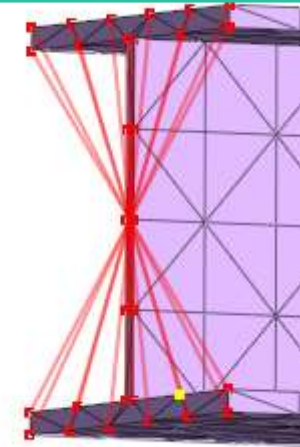
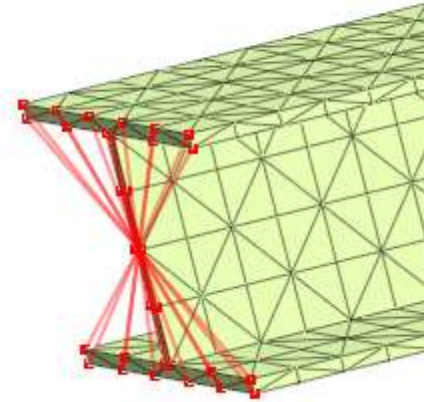
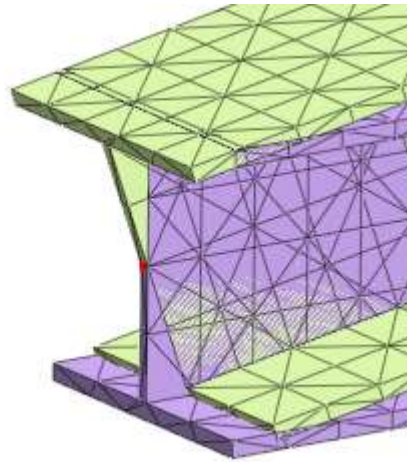
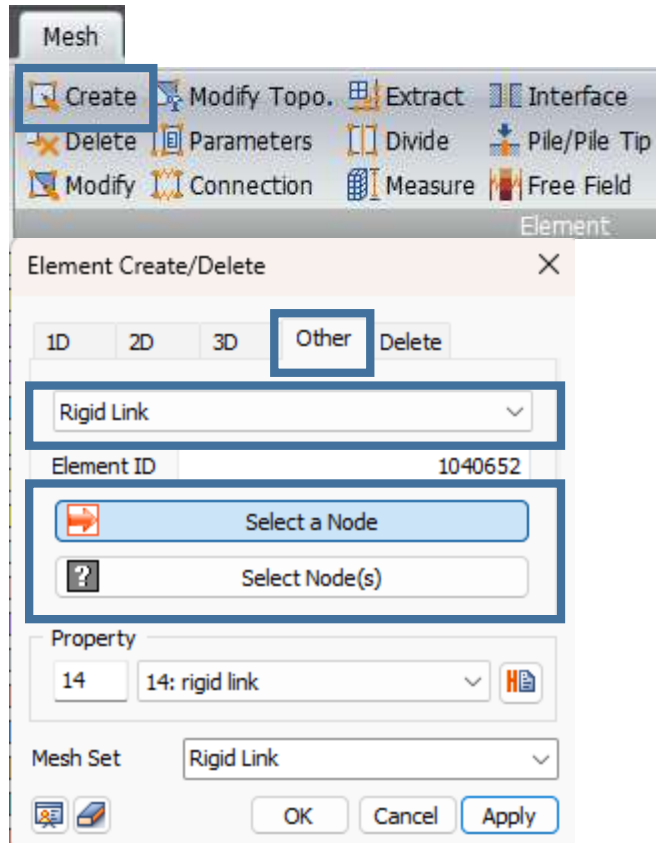
# RIGID LINKS



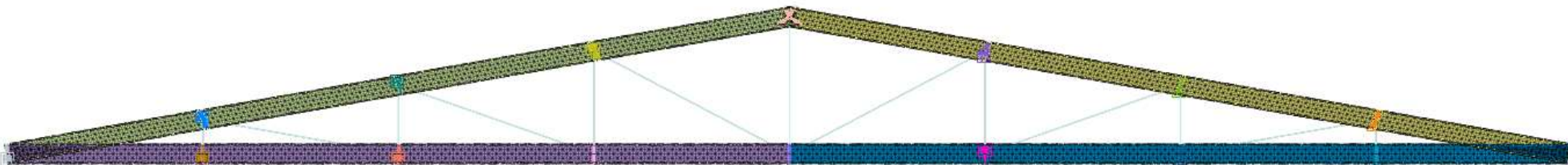
Define a rigid body in the property to create a rigid link.  
NOTE: No material definition needed for rigid body.



# RIGID LINKS

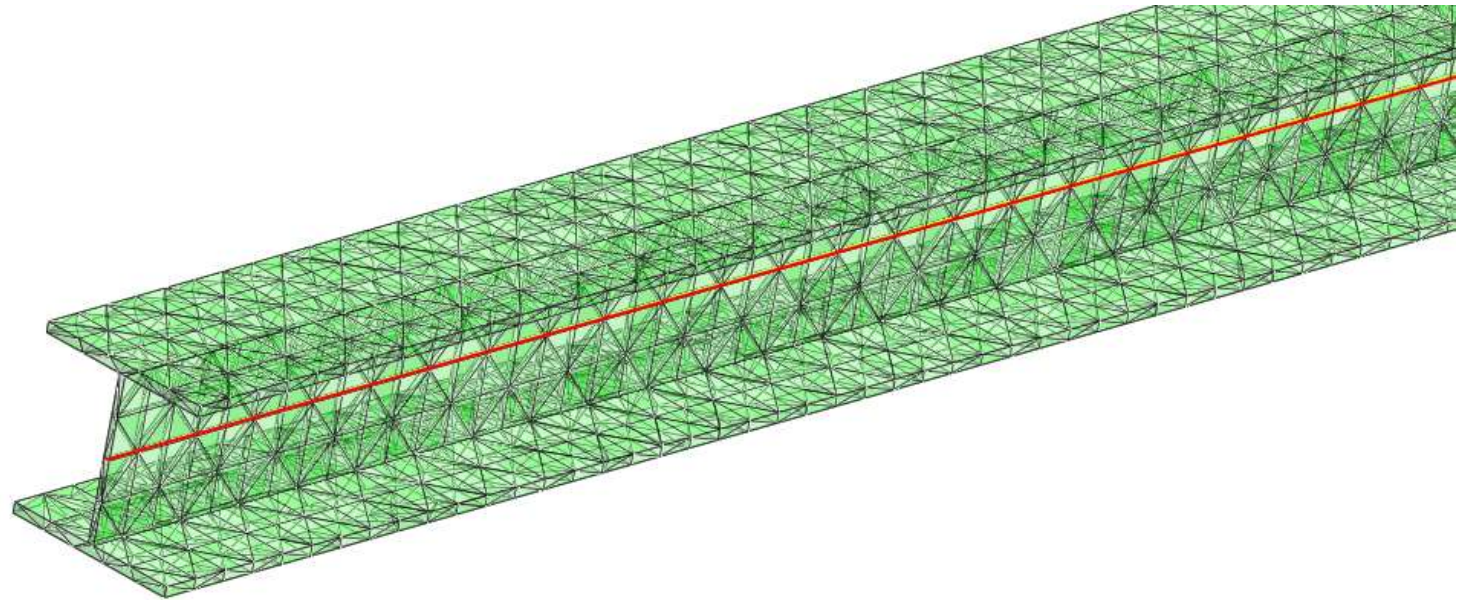
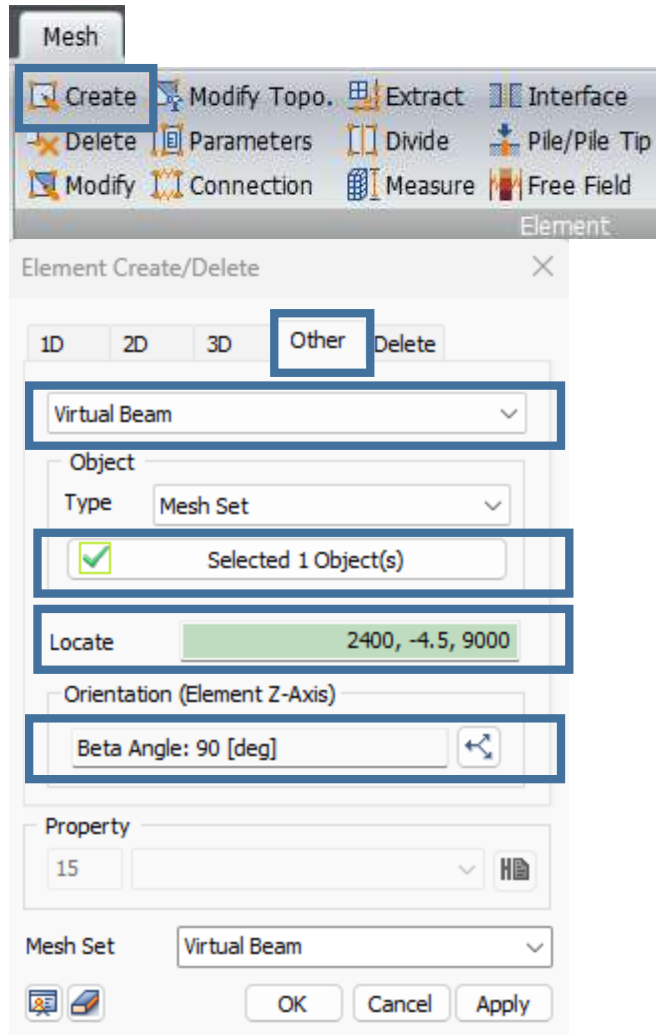


NOTE: Define rigid link to all joints connected to the 3D element





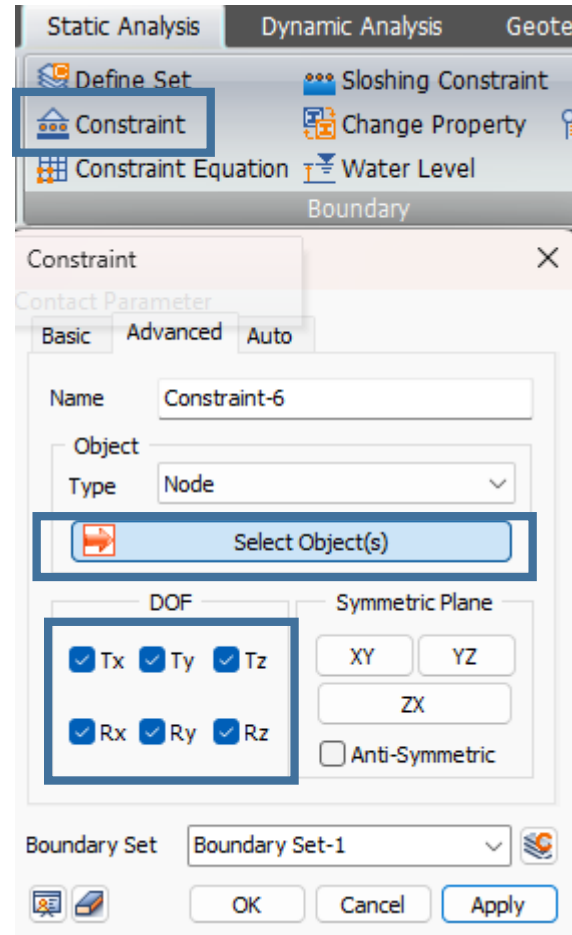
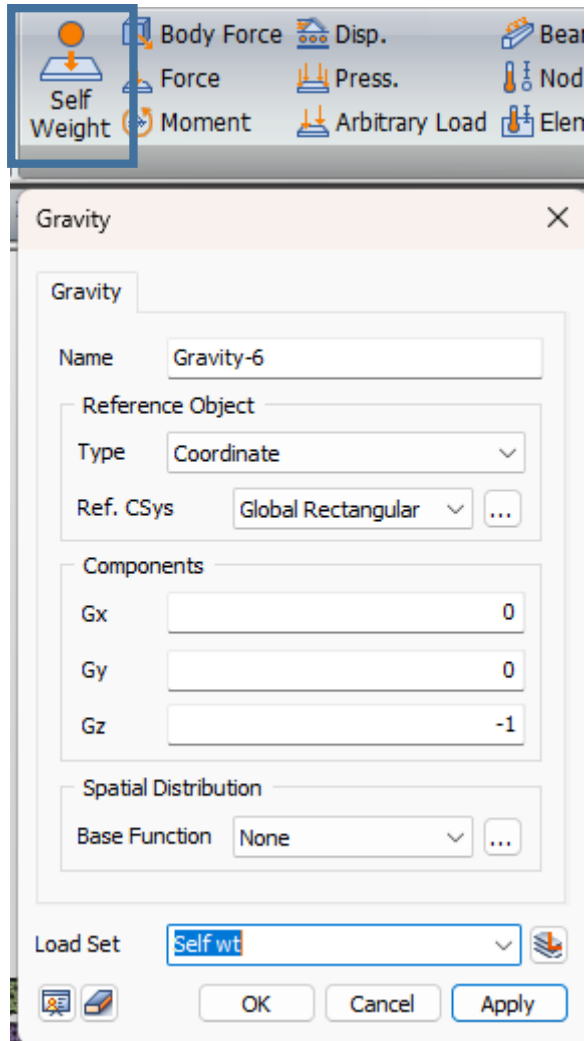
# VIRTUAL BEAM



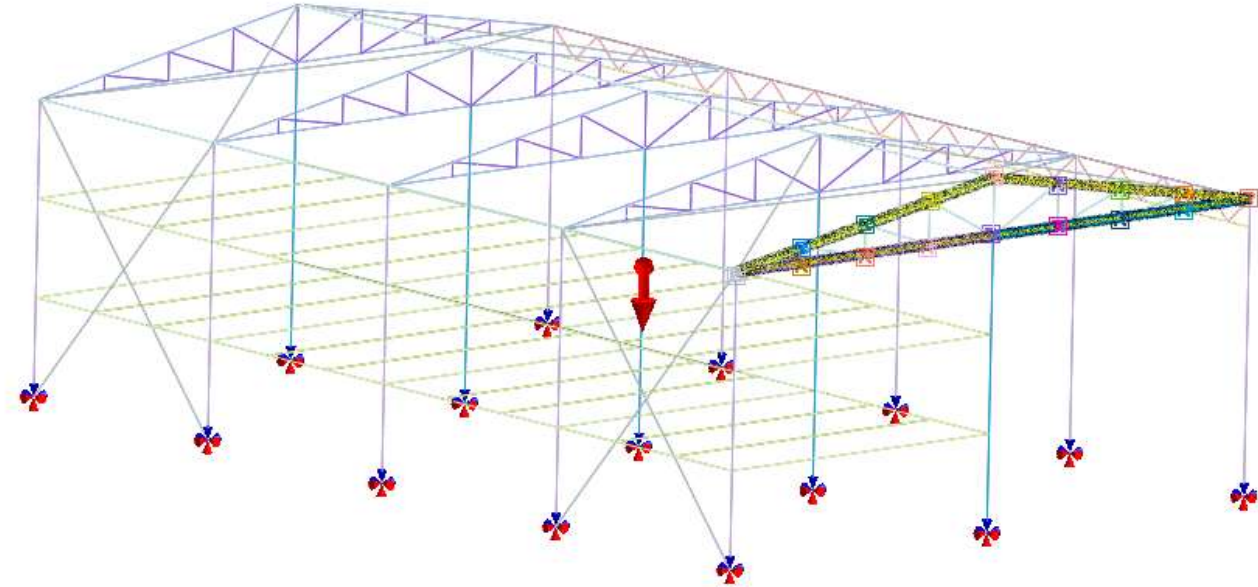
## STEPS:

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

# LOADS and CONSTRAINTS



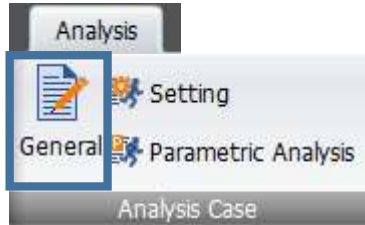
NOTE: Set constraint on the column to be fixed



NOTE:

1. Initially, set to simplest load case (self weight)
2. Set constraint on the column base to be fixed (DOF: Txyz, Rxyz)

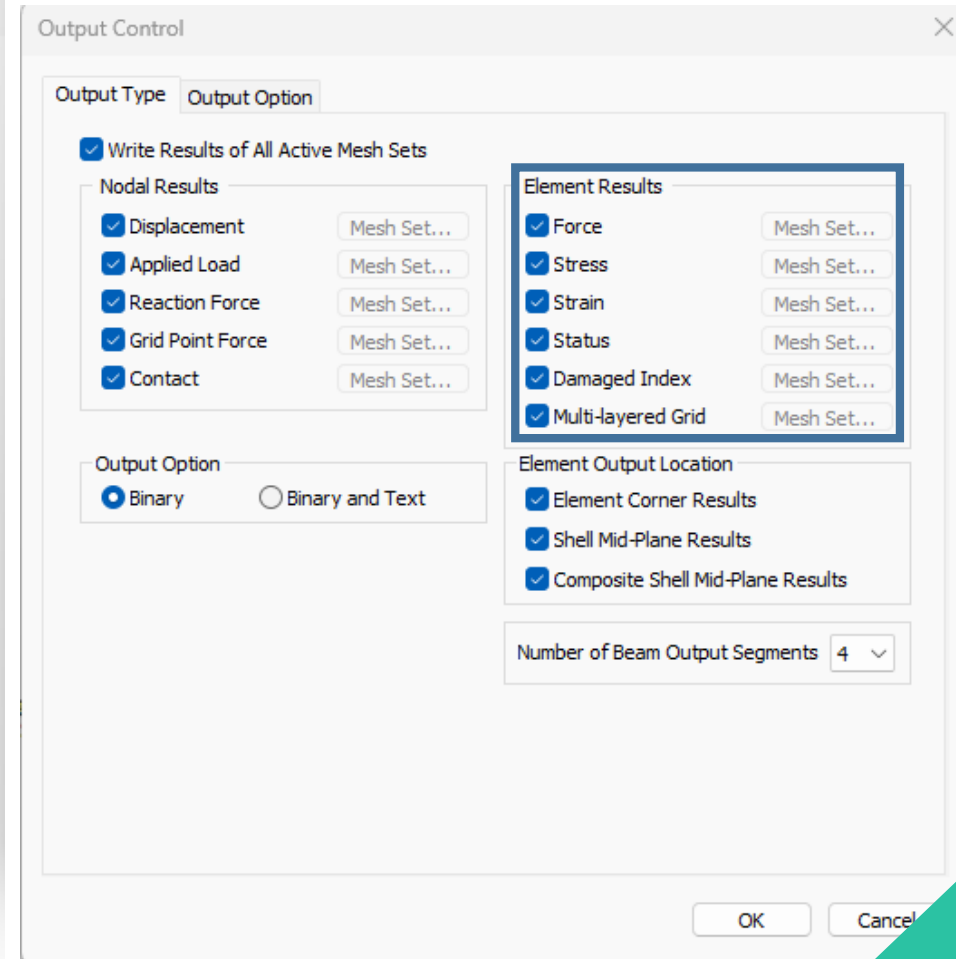
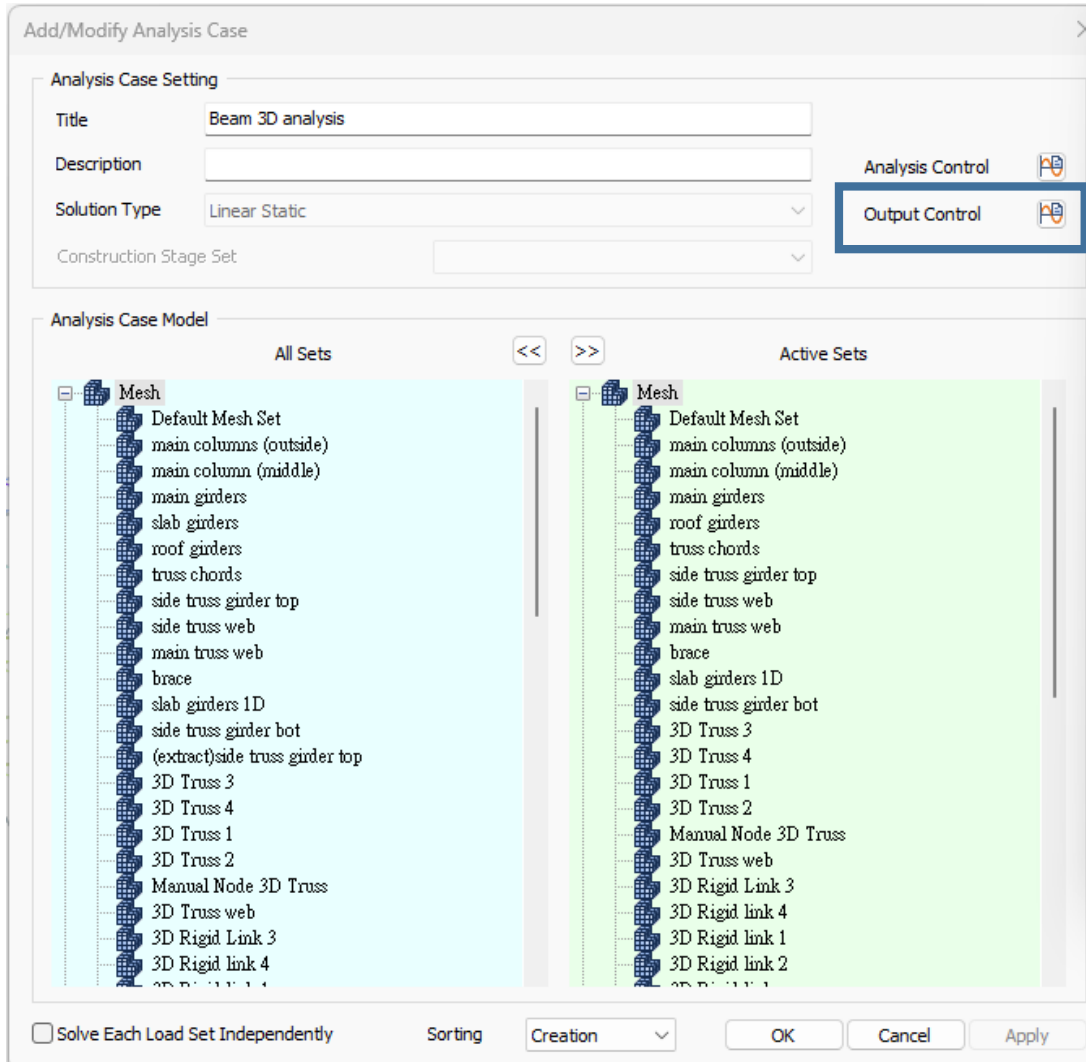
# ANALYSIS CASE



## NOTE:

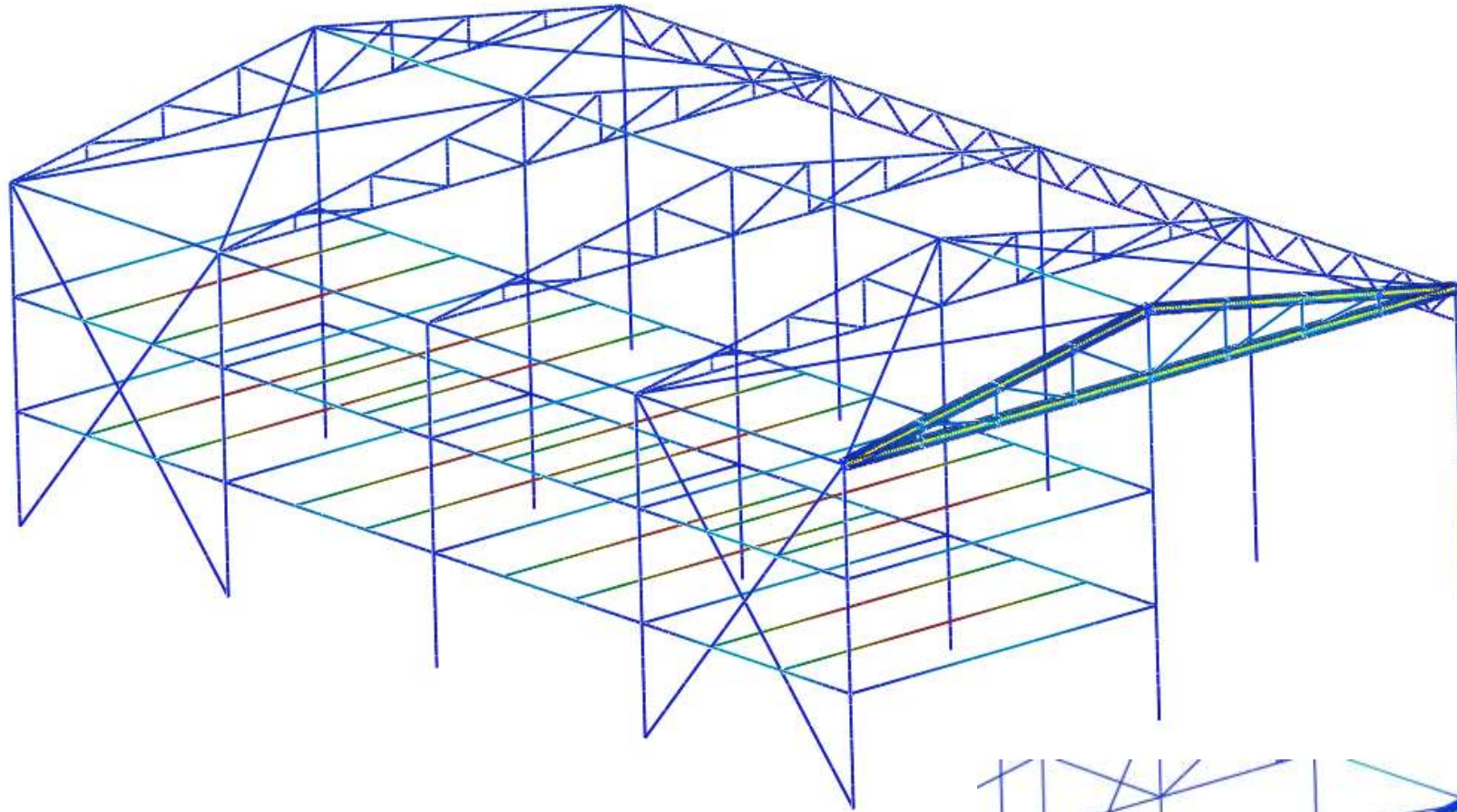
1. Include all the necessary meshes, constraints, and loads.

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

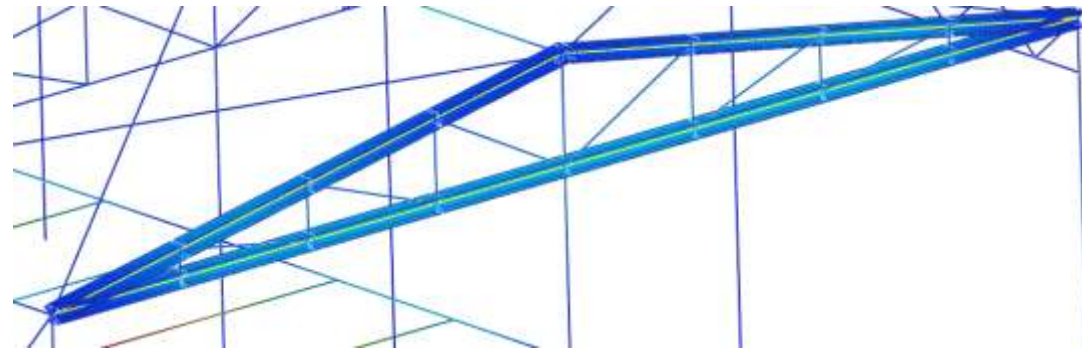
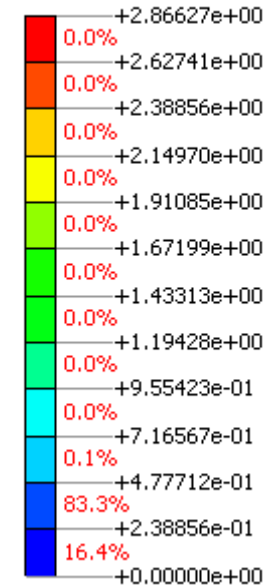




# RESULTS - DISPLACEMENT

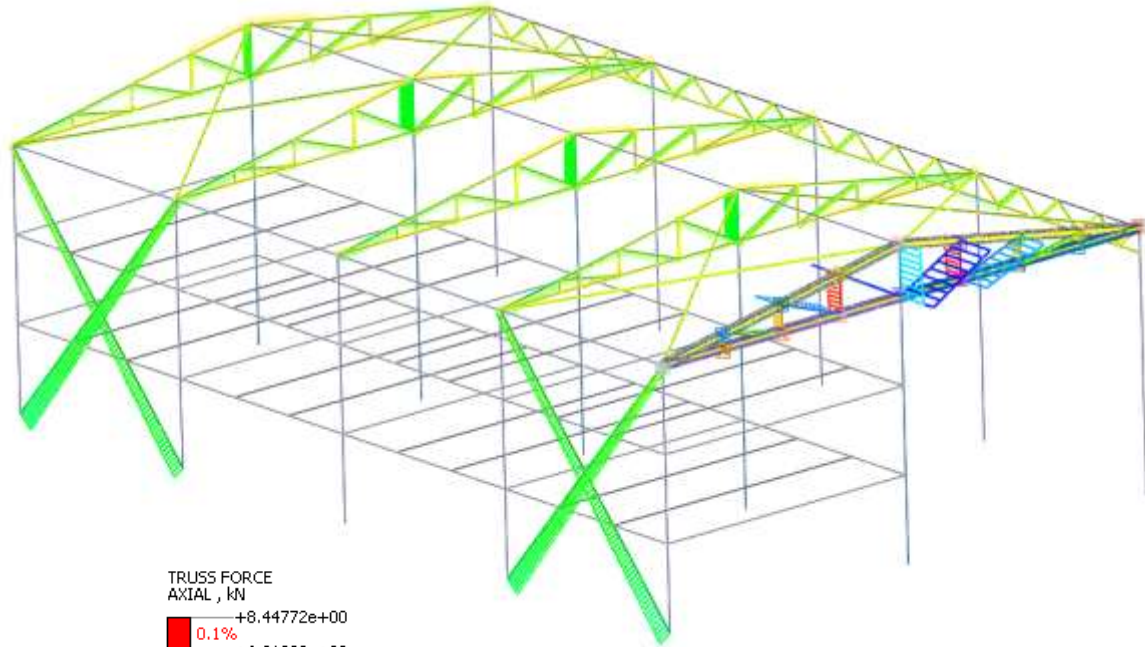


DISPLACEMENT  
TOTAL T, mm

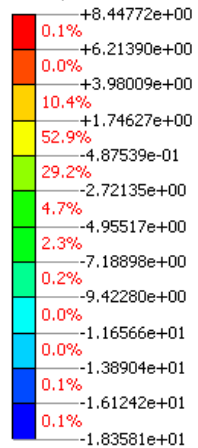




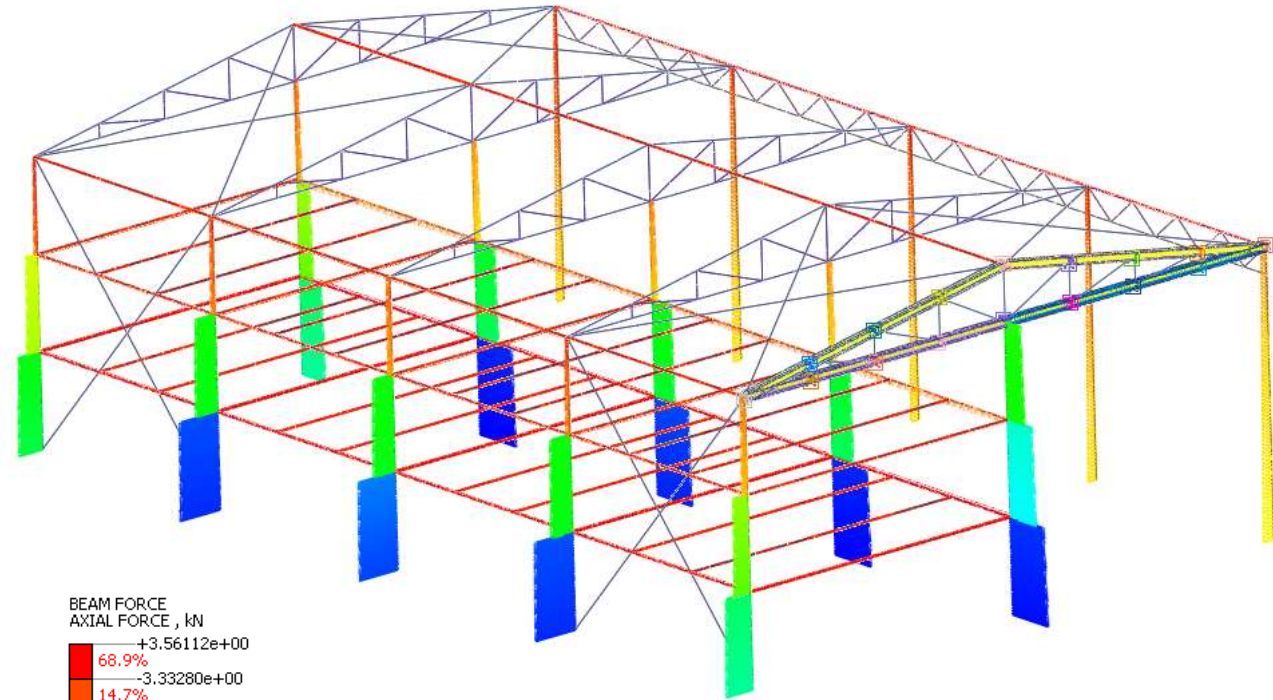
# RESULTS - 1D AXIAL FORCES



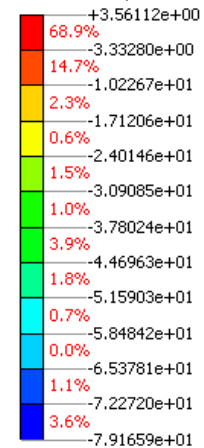
TRUSS FORCE  
AXIAL , kN



## TRUSS

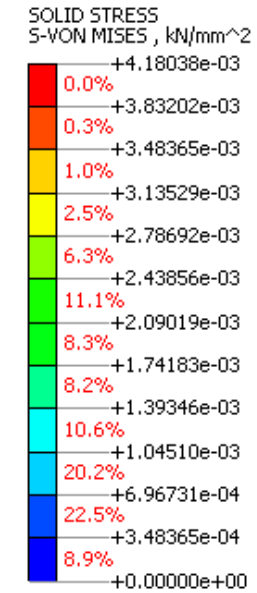
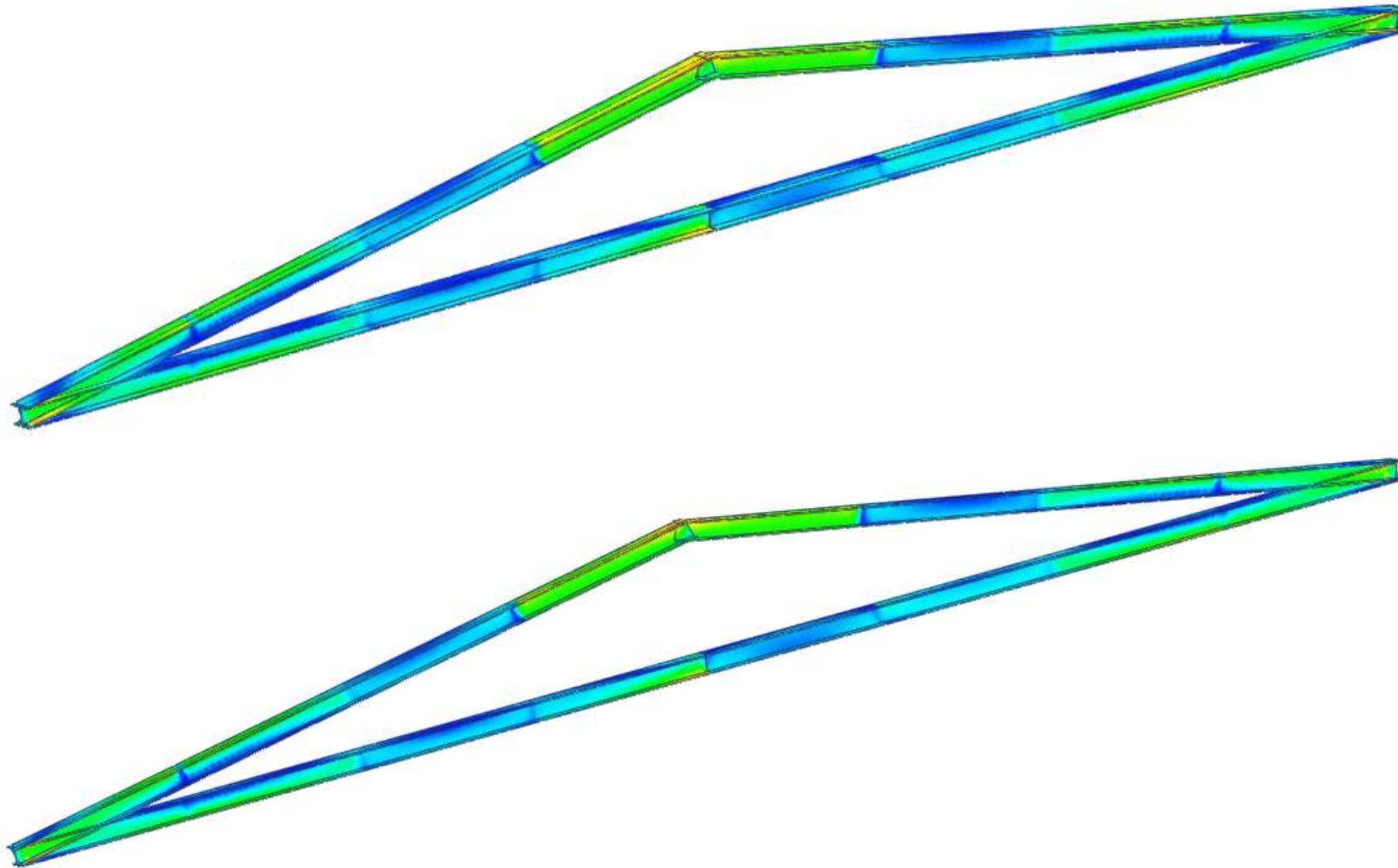


BEAM FORCE  
AXIAL FORCE , kN

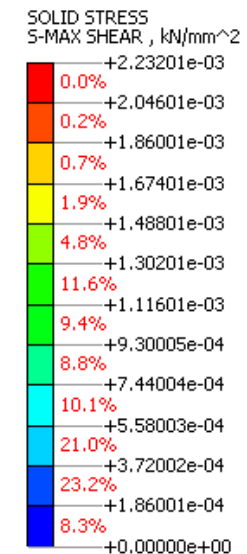


## BEAM

# RESULTS - 3D STRESS VON MISES



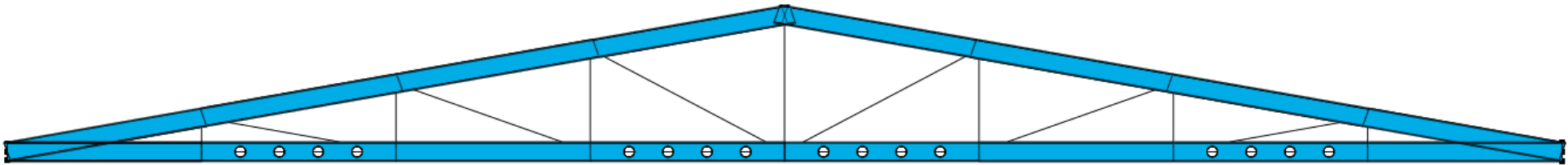
**STRESS**  
**Von Mises**



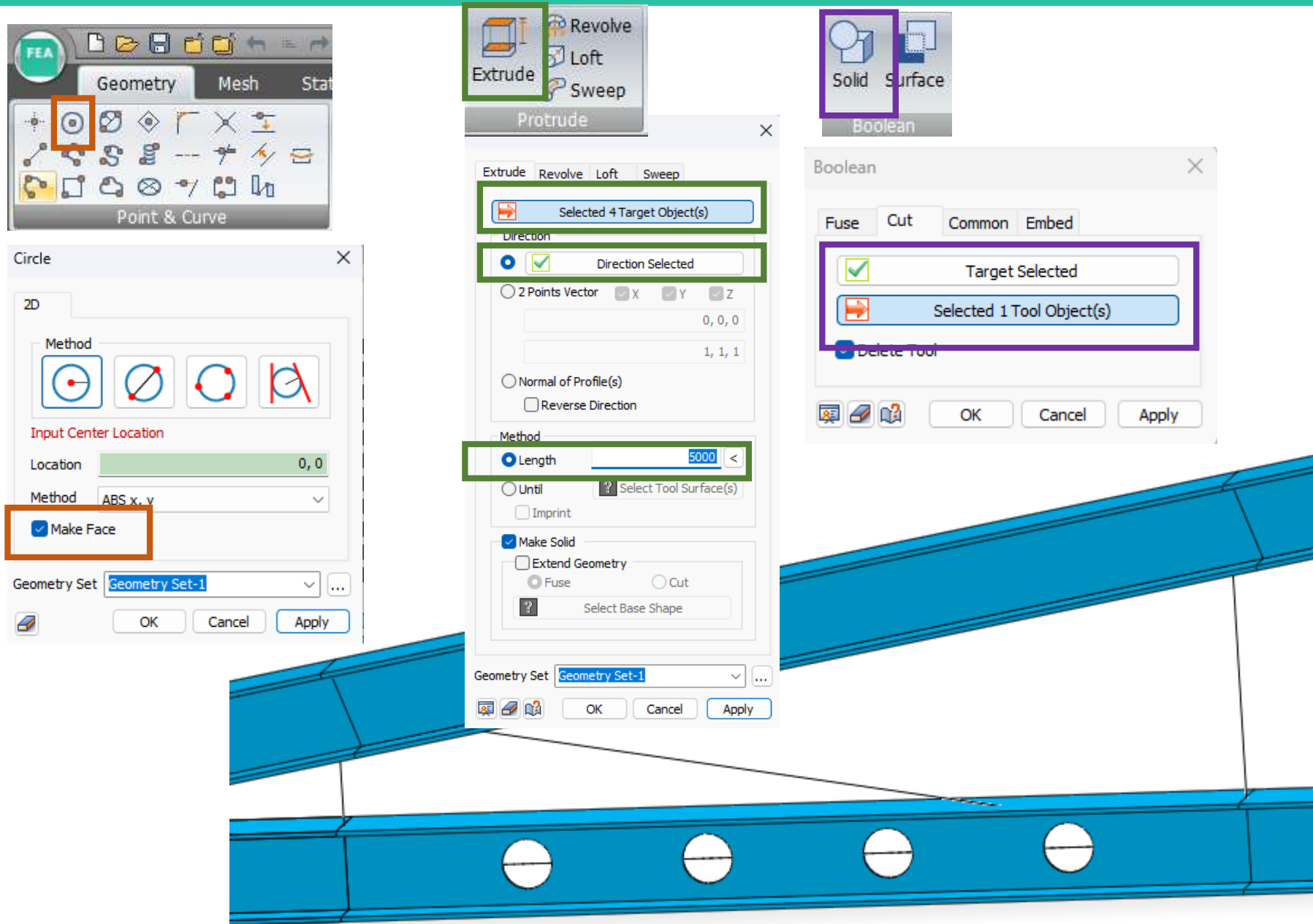
**STRESS**  
**Max Shear**

# Part 2

## Modified Frame Detailed Analysis



# GEOMETRY



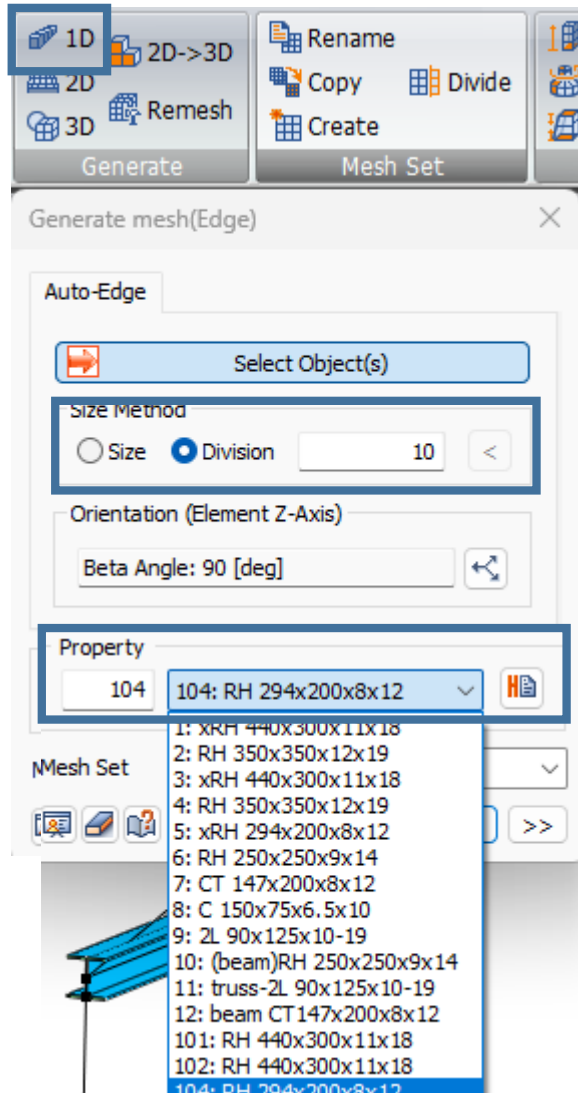
## MODIFYING BEAM ELEMENT

(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)



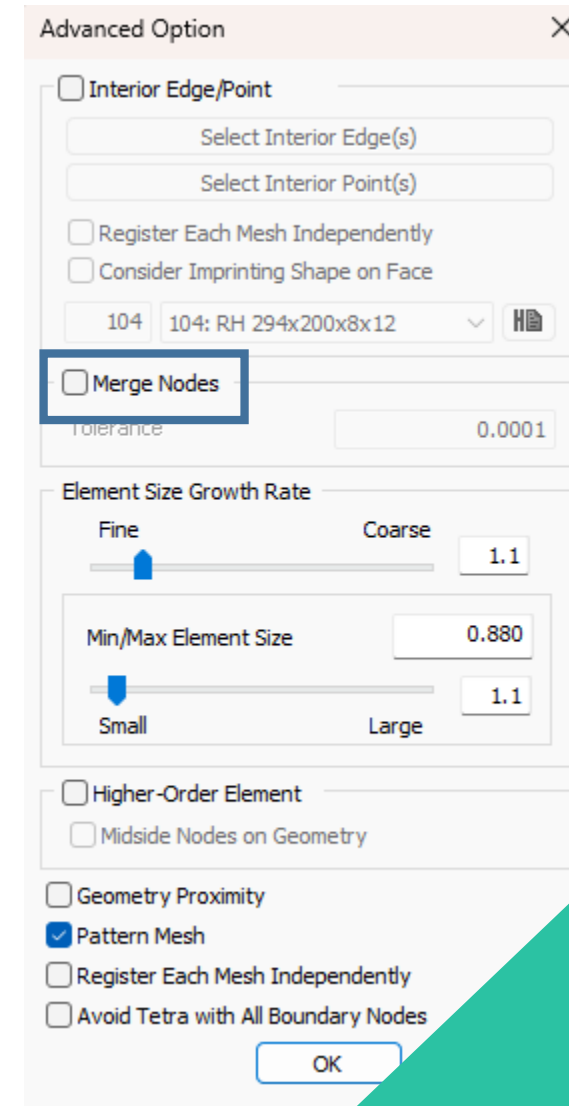
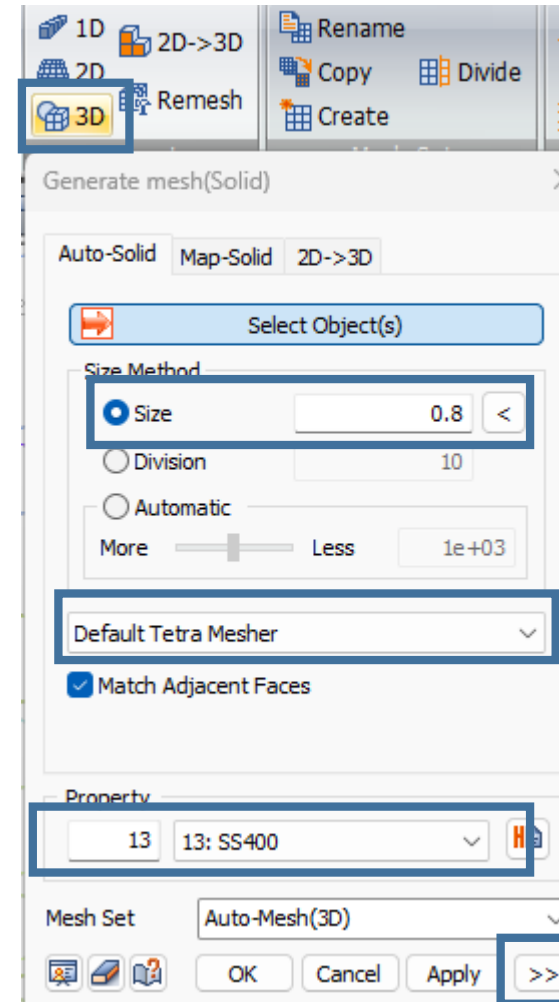
For **1D elements**, define the appropriate section property.

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

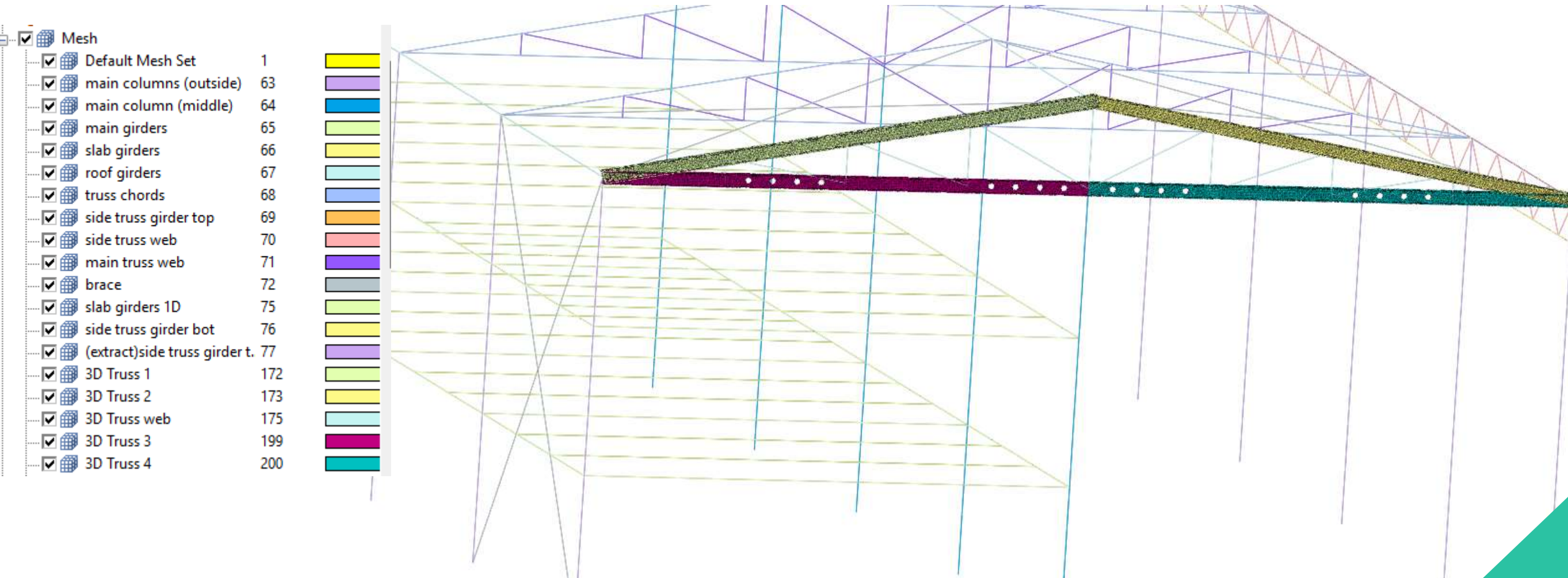
For **3D elements**, define the mesh size with the appropriate materials.

NOTE:

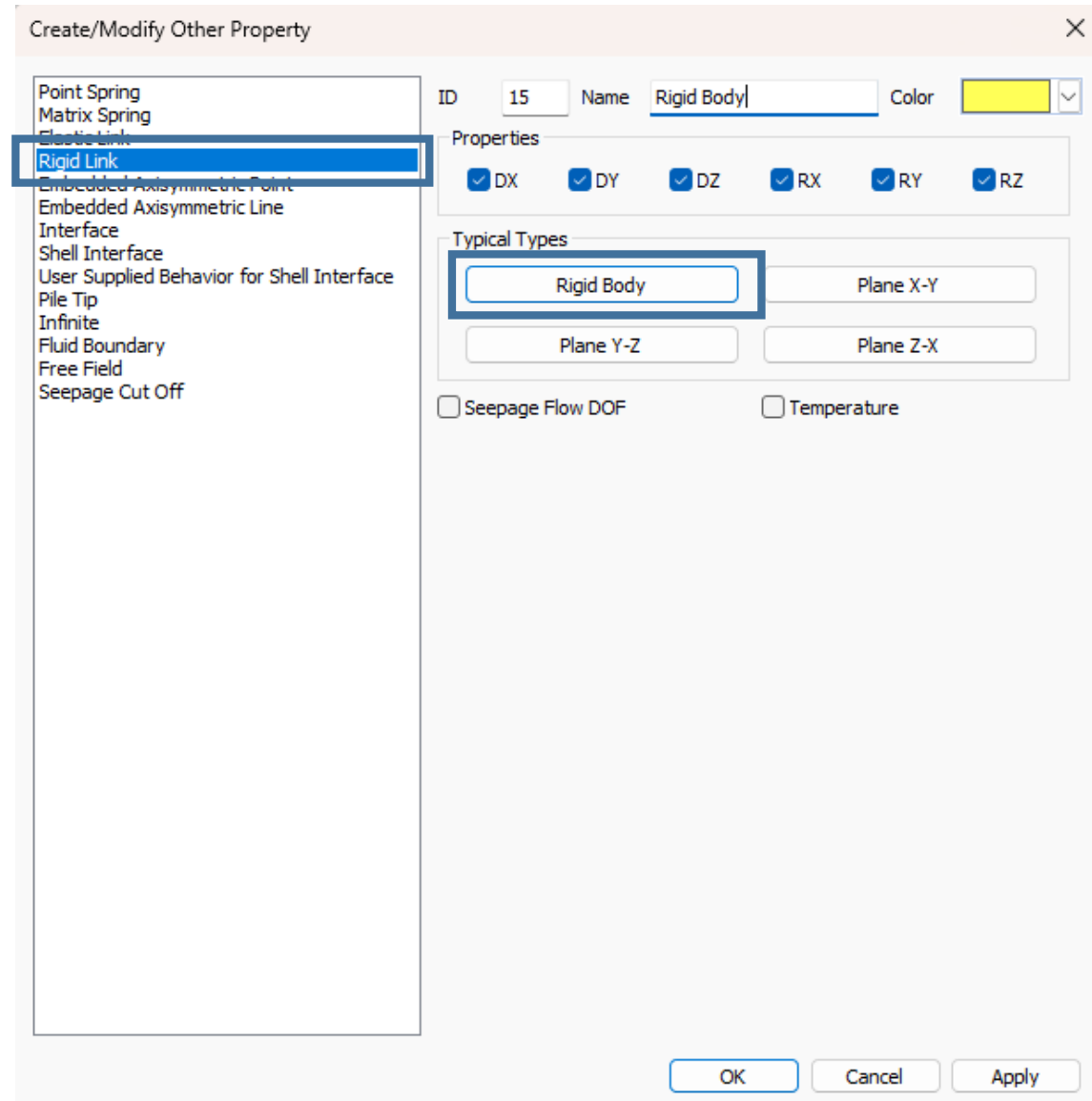
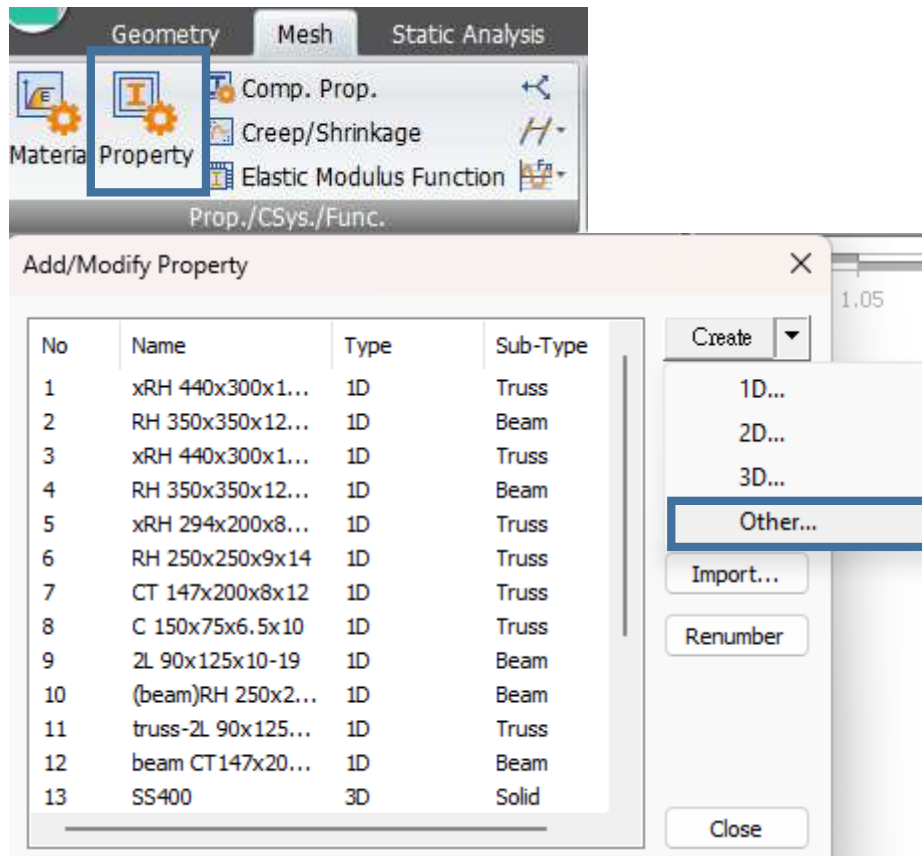
1. For future GEN export, use Tetra Mesher as mesh type
2. Deselect Merge nodes to create a segmented parts of the truss.



# MODEL



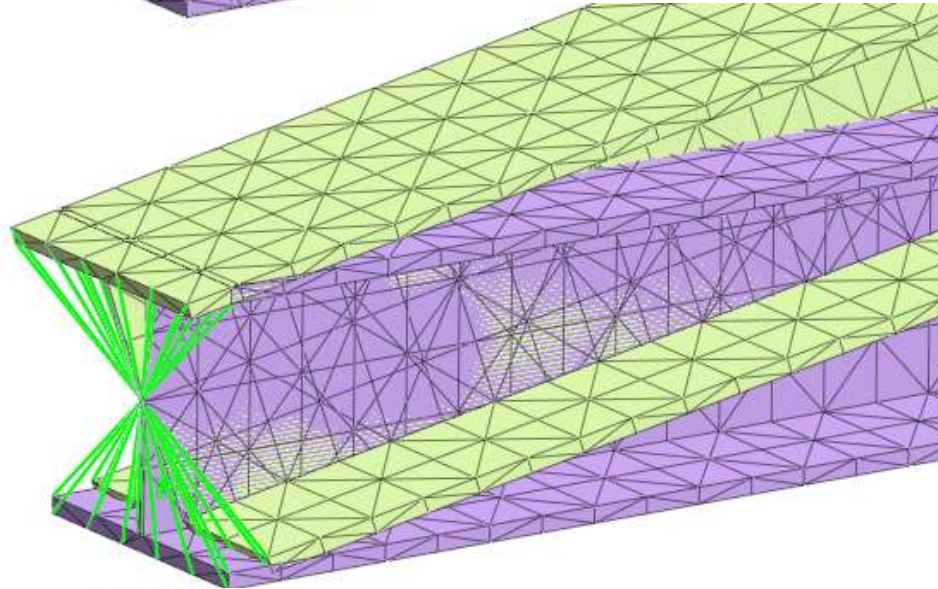
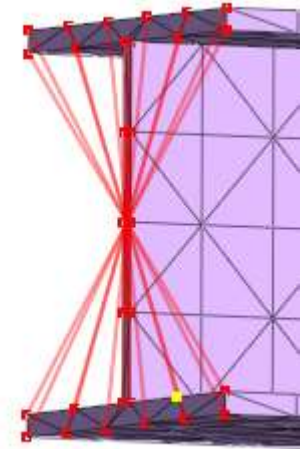
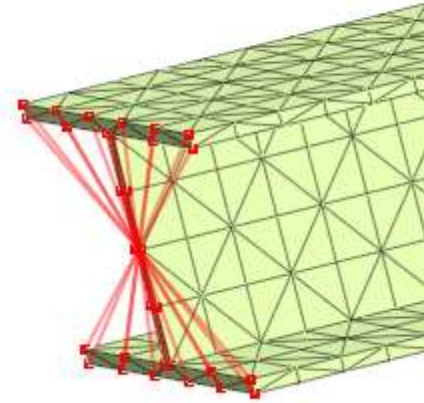
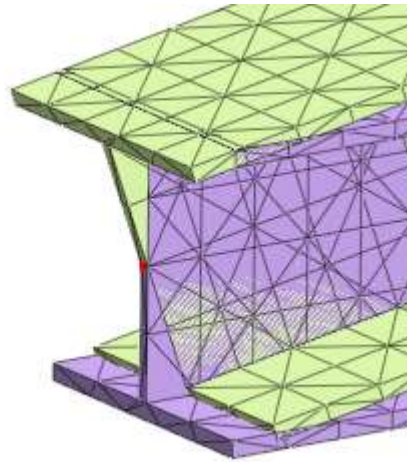
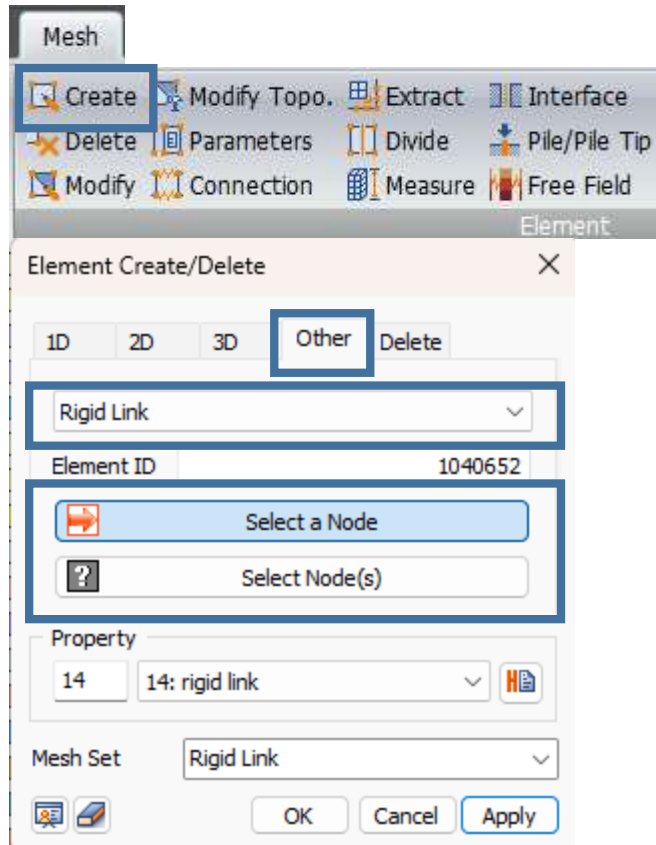
# RIGID LINKS



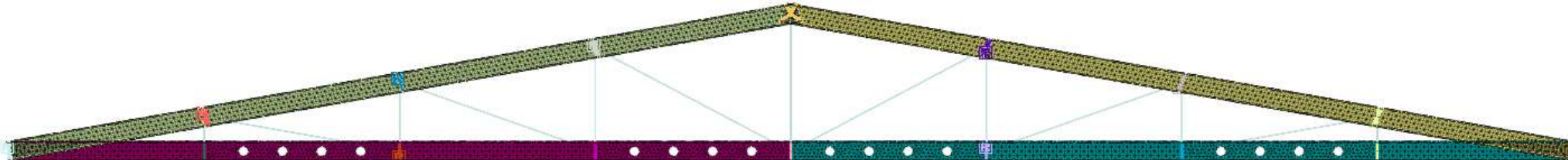
Define a rigid body in the property to create a rigid link.  
NOTE: No material definition needed for rigid body.



# RIGID LINKS



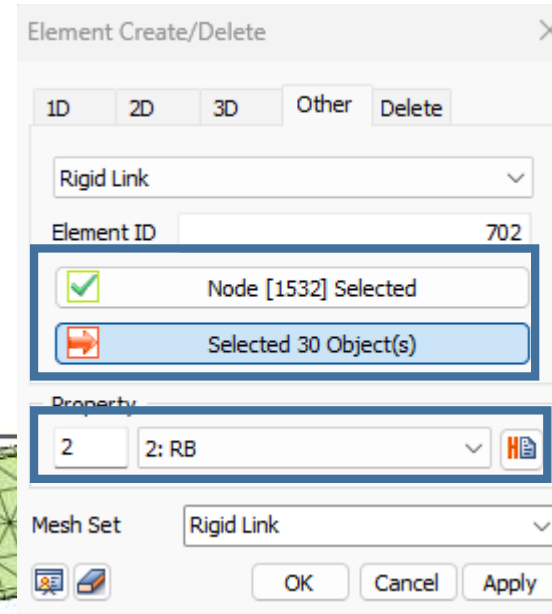
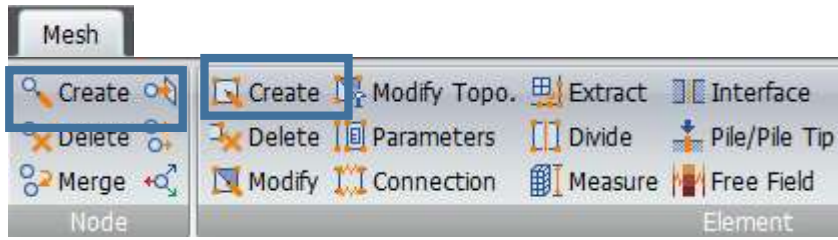
NOTE: Define rigid link to all joints





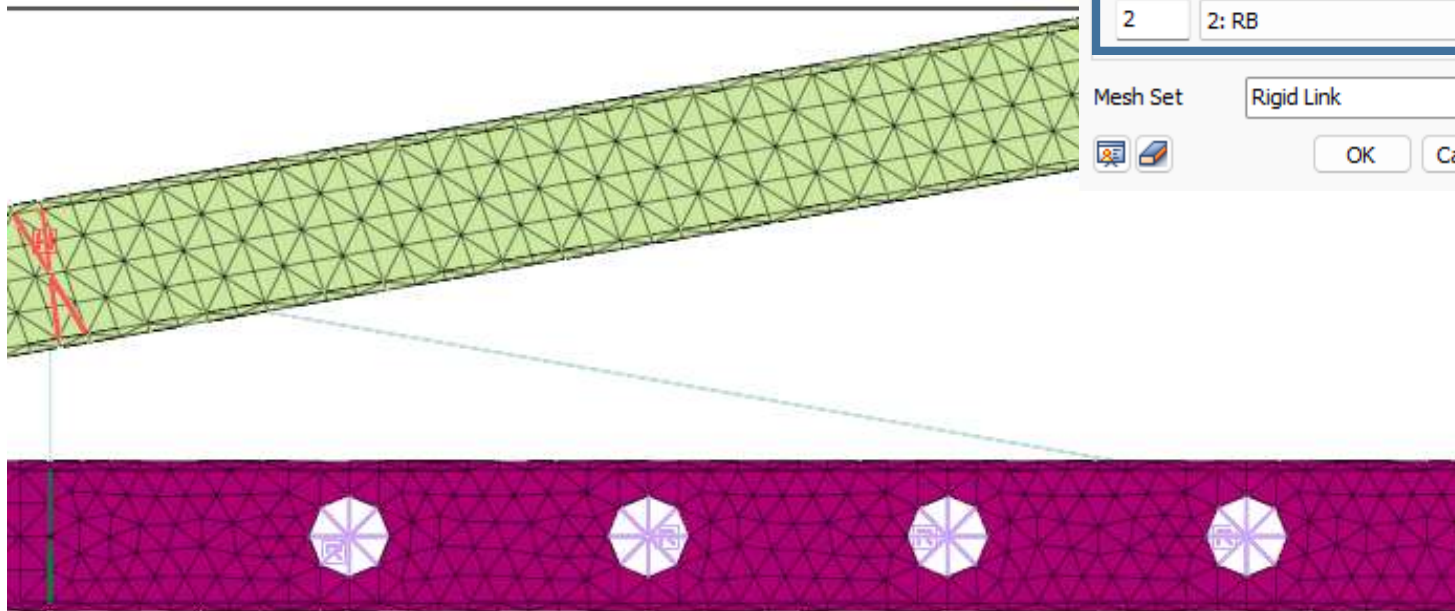
# RIGID LINK

## RIGID LINK-CREATE LINKS FOR HOLES



## STEPS:

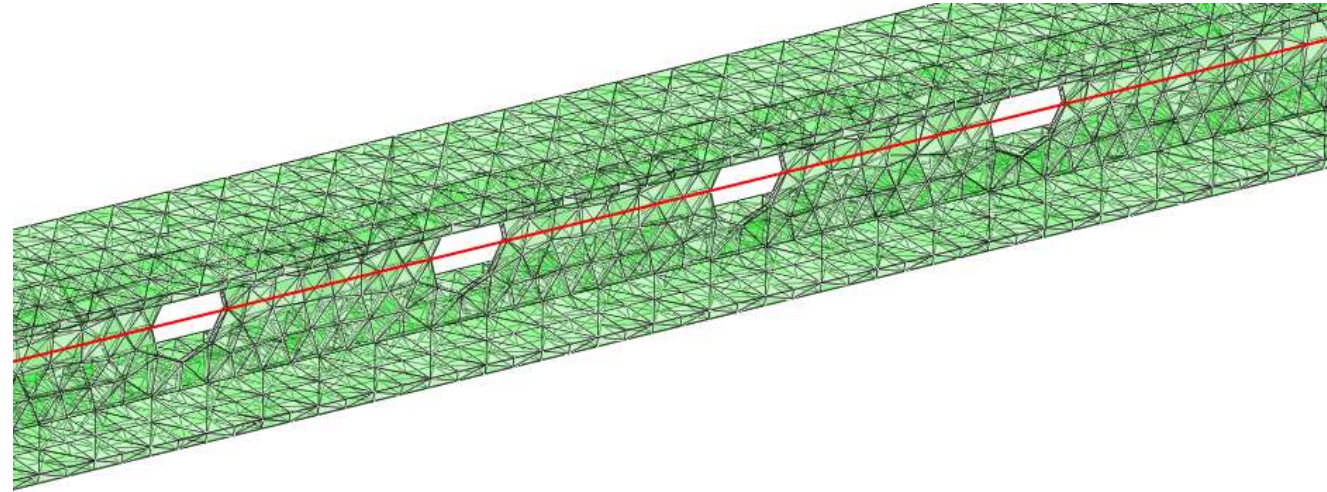
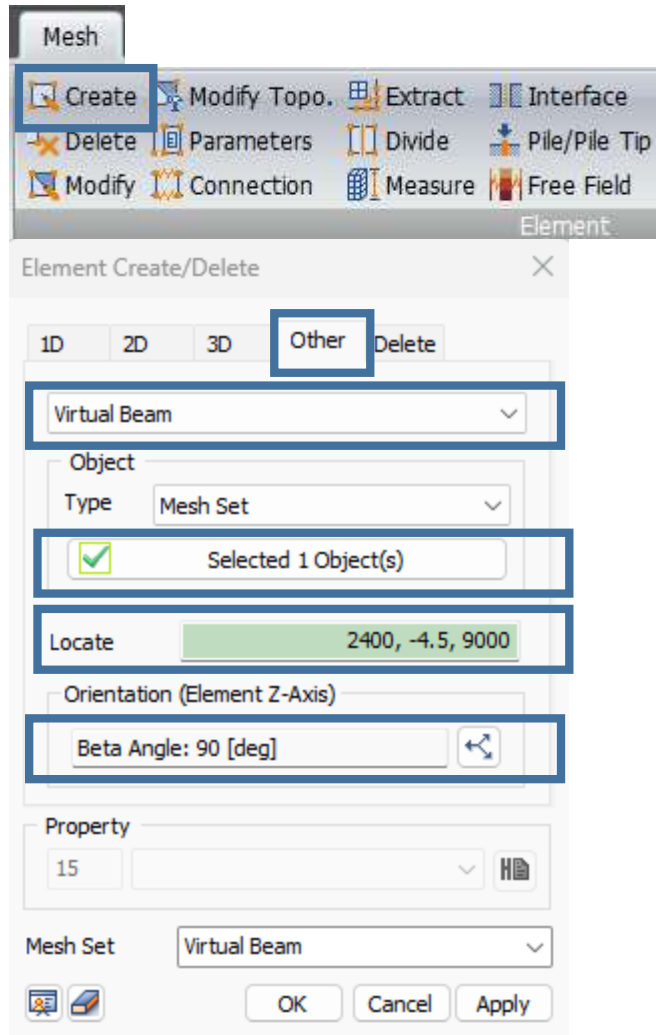
1. Define the master node in the center of the holes.
2. Link the edge of the hole to the master node.



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



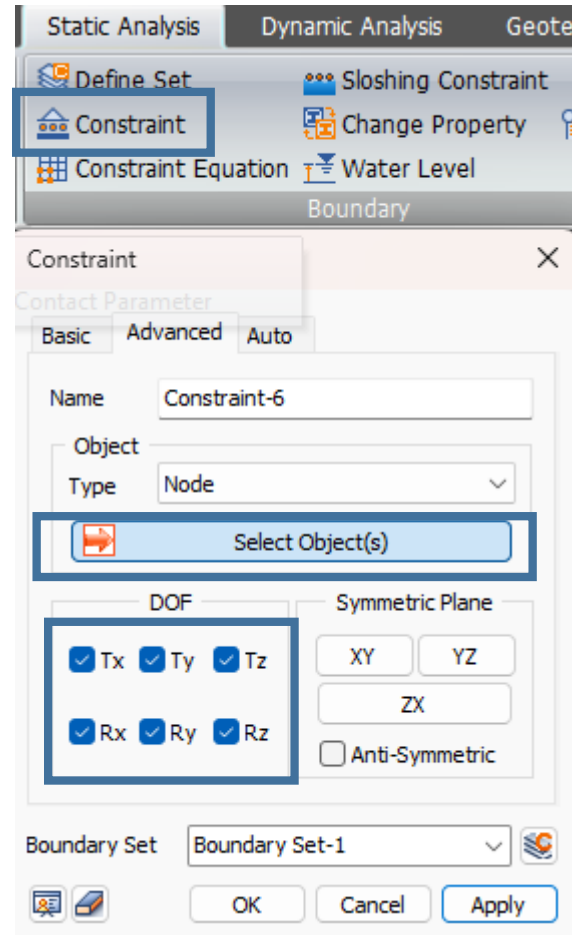
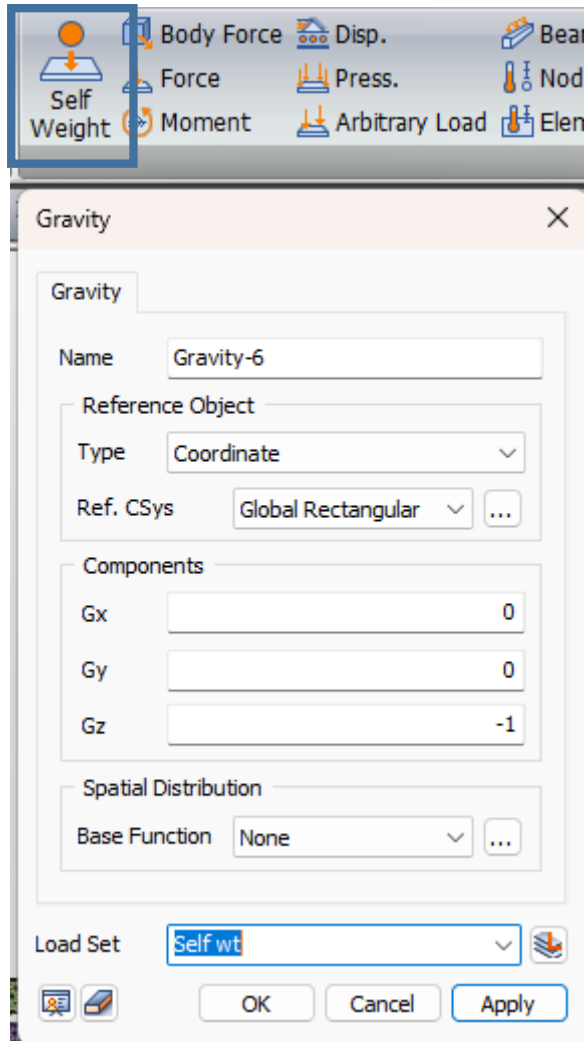
# VIRTUAL BEAM



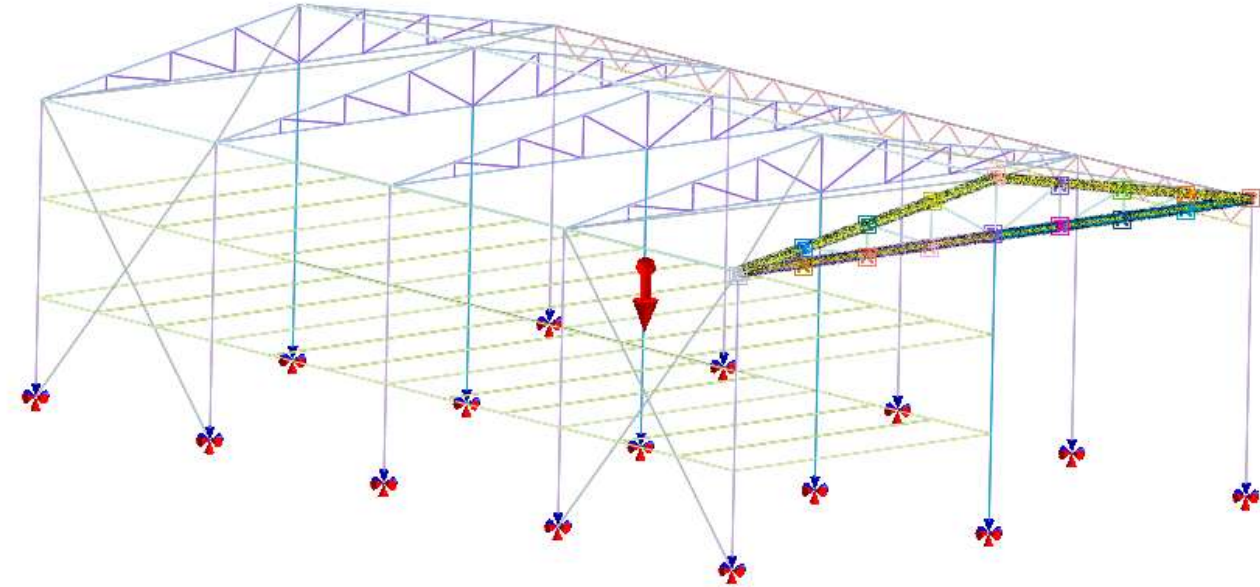
## STEPS:

1. Select mesh set.
2. Create a line (end-to-end) along the geometric center of the mesh.
3. Choose the proper orientation.

# LOADS and CONSTRAINTS



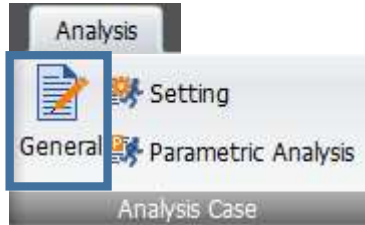
NOTE: Set constraint on the column to be fixed



NOTE:

1. Initially, set to simplest load case (self weight)
2. Set constraint on the column base to be fixed (DOF: Txyz, Rxyz)

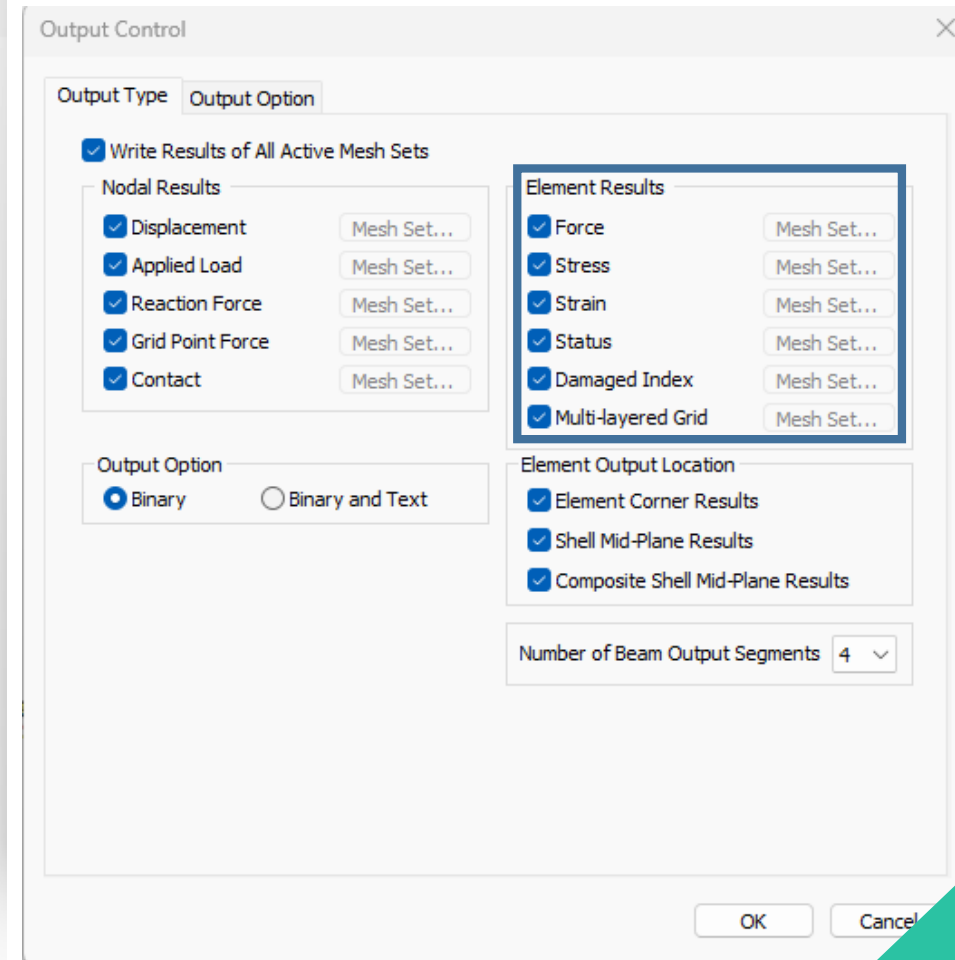
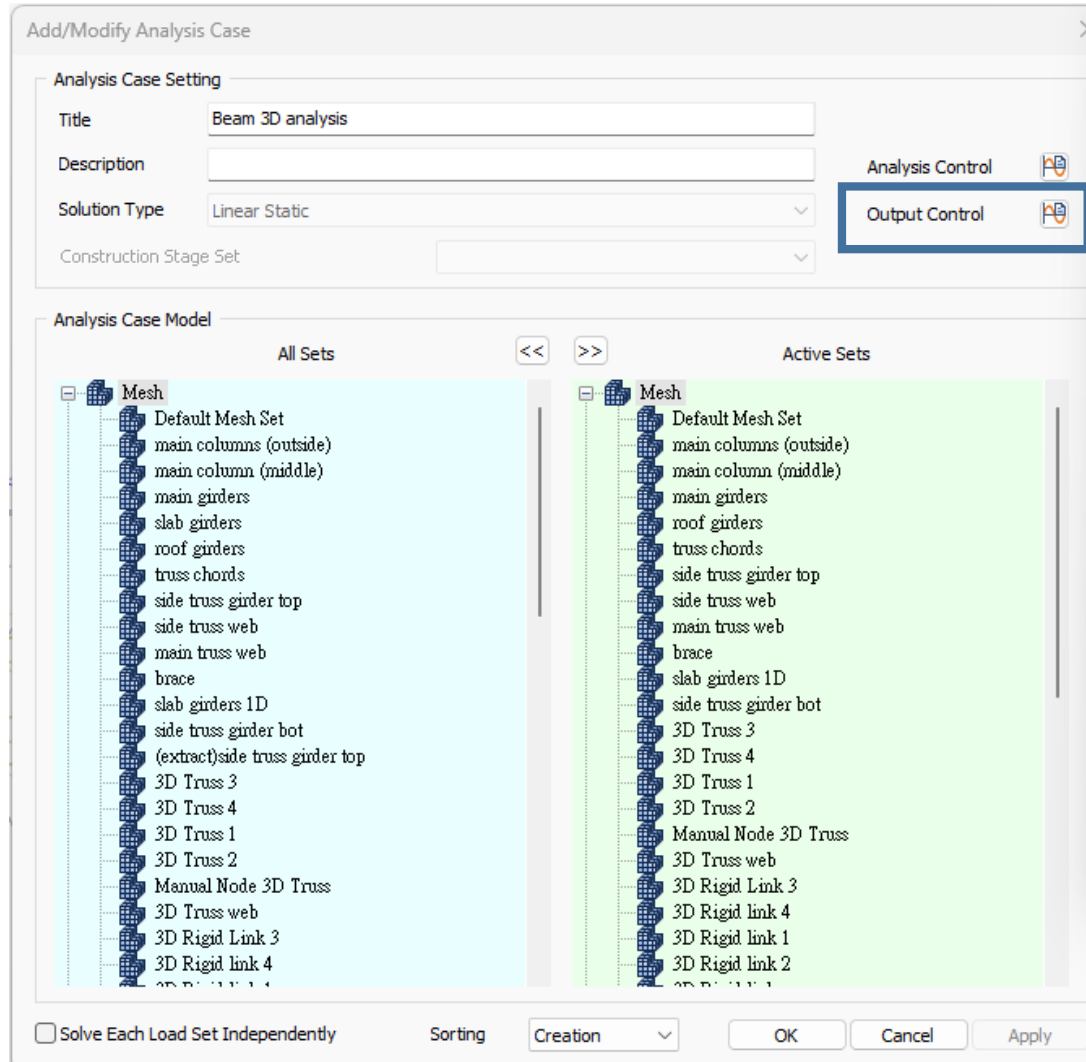
# ANALYSIS CASE



NOTE:

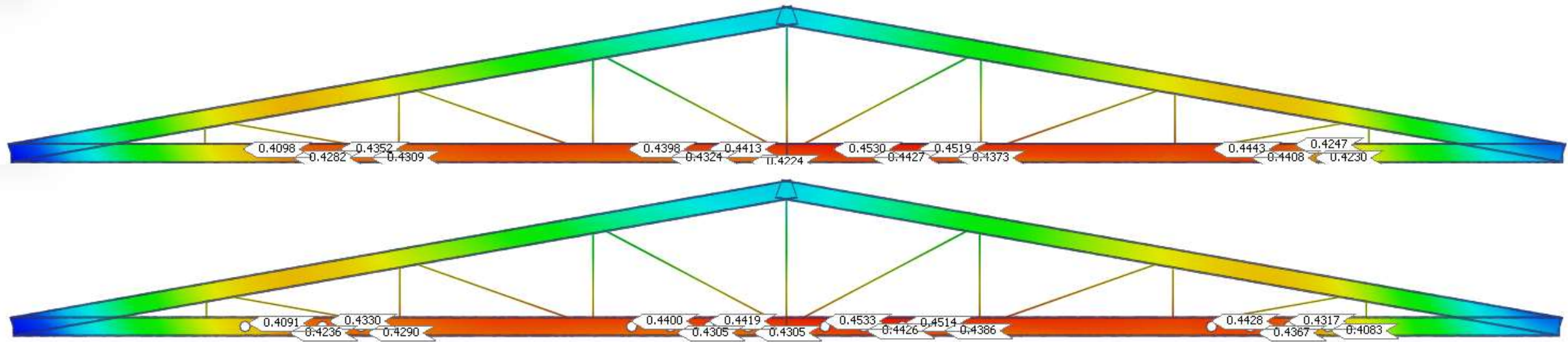
1. Include all the necessary meshes, constraints, and loads.

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



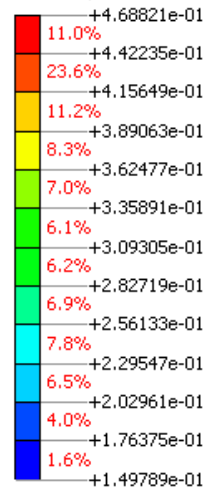


# RESULT COMPARISON - DISPLACEMENT



## STANDARD

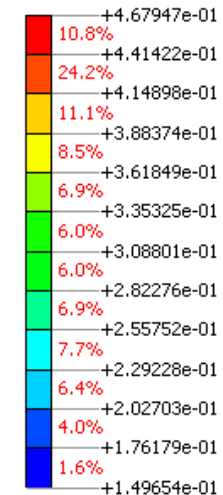
DISPLACEMENT  
TOTAL T, mm



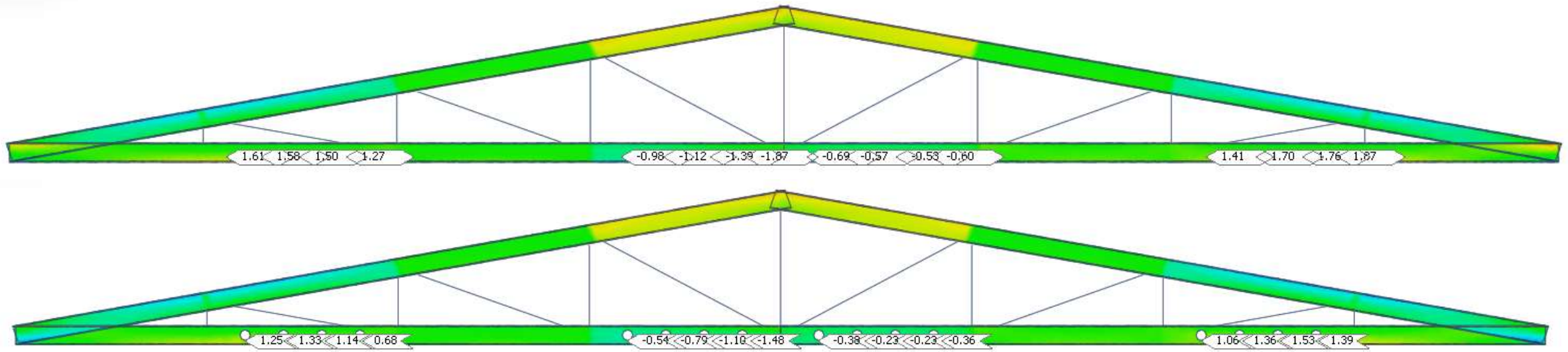
Overall, there is a small difference between the displacement after modifying the beams.

## MODIFIED

DISPLACEMENT  
TOTAL T, mm

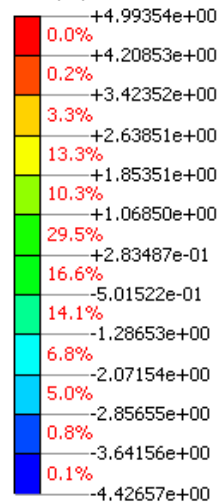


# RESULT COMPARISON - STRESS VON MISES



## STANDARD

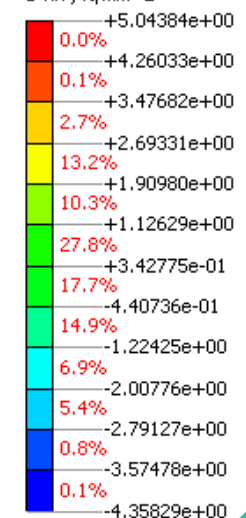
SOLID STRESS  
S-XX, N/mm<sup>2</sup>



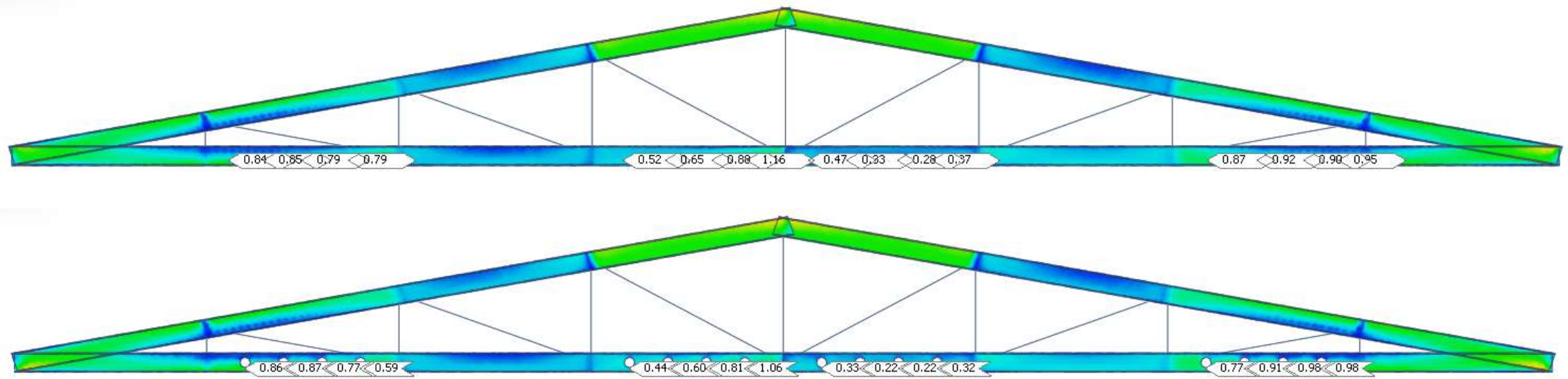
The modification created an S-XX increase of 13% - 60% on the hole areas.

## MODIFIED

SOLID STRESS  
S-XX, N/mm<sup>2</sup>

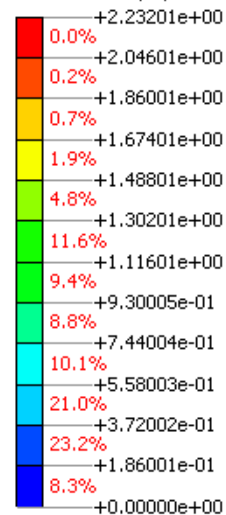


# RESULT COMPARISON - STRESS MAX SHEAR



## STANDARD

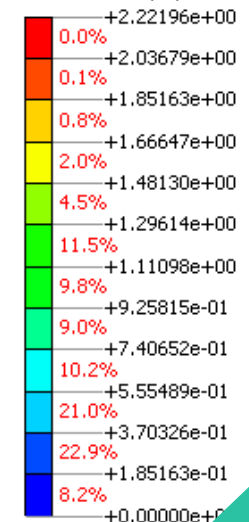
SOLID STRESS  
S-MAX SHEAR, N/mm<sup>2</sup>



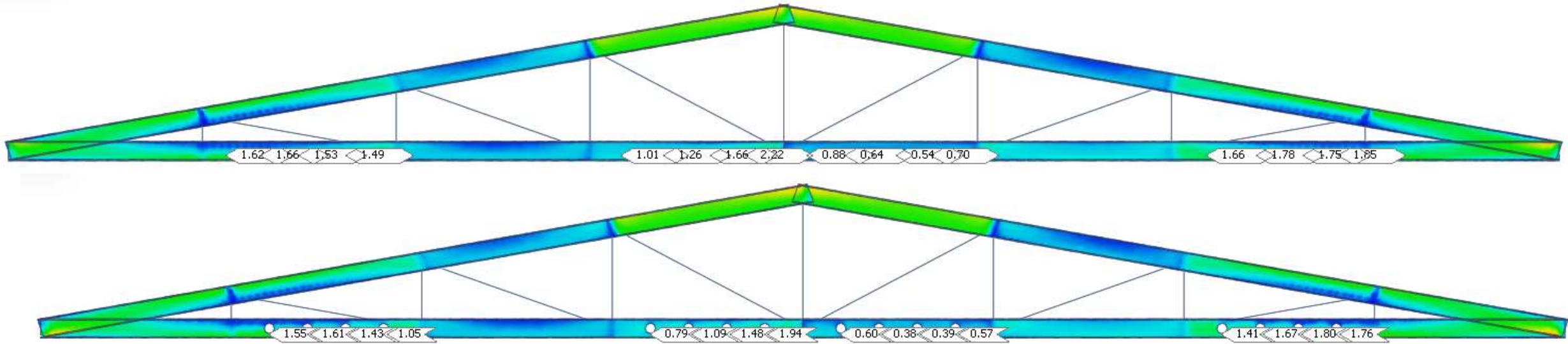
The modification created a max shear decrease in the midspan by 17%.

## MODIFIED

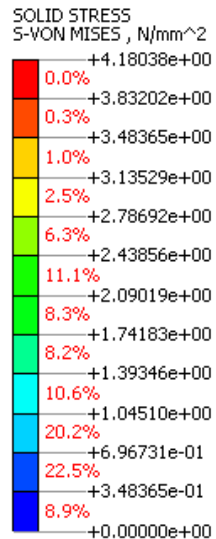
SOLID STRESS  
S-MAX SHEAR, N/mm<sup>2</sup>



# RESULT COMPARISON - STRESS VON MISES

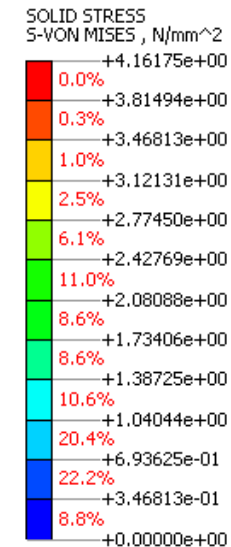


## STANDARD



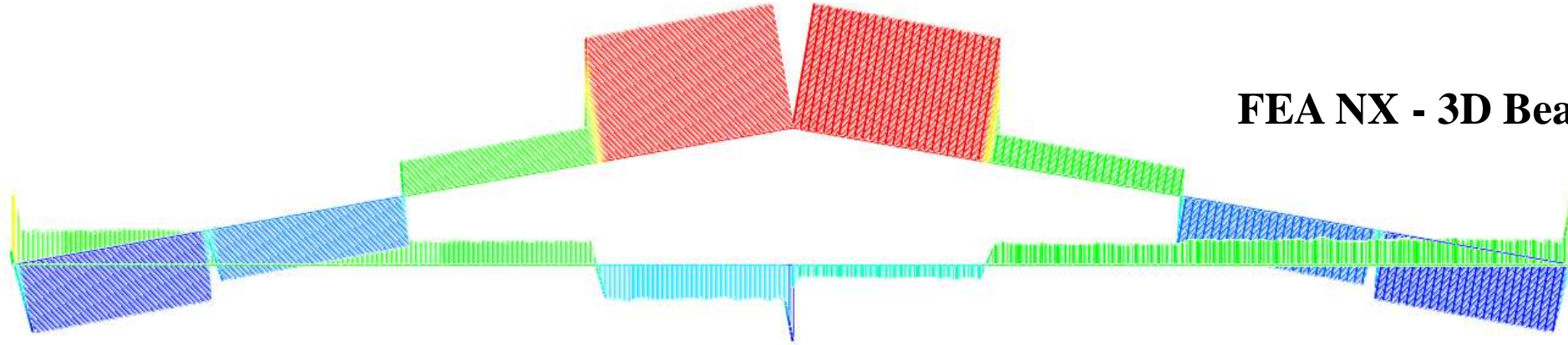
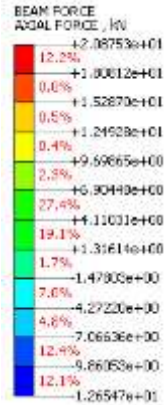
The modification created a stress von mises decrease in the midspan by 22.2%.

## MODIFIED

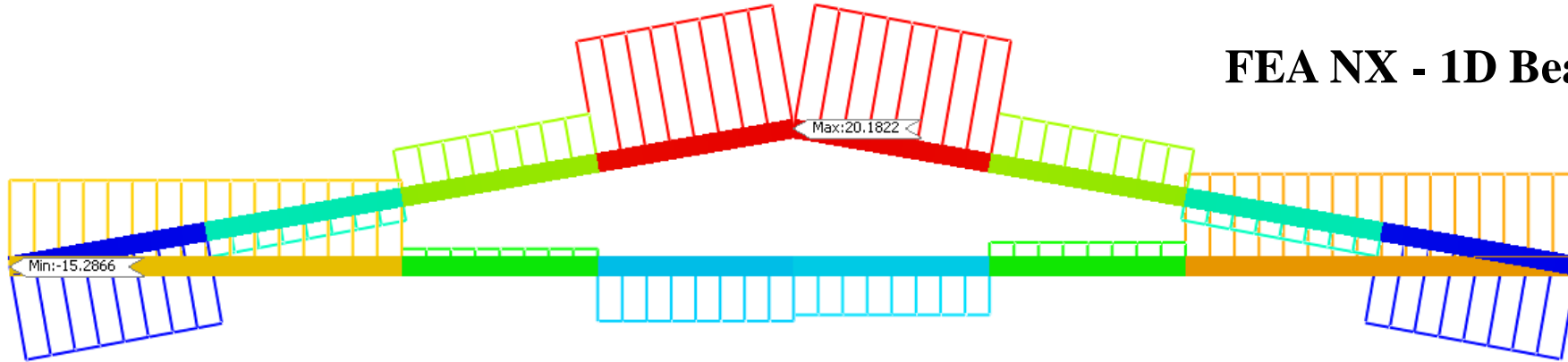
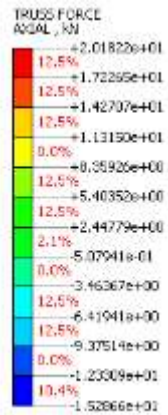




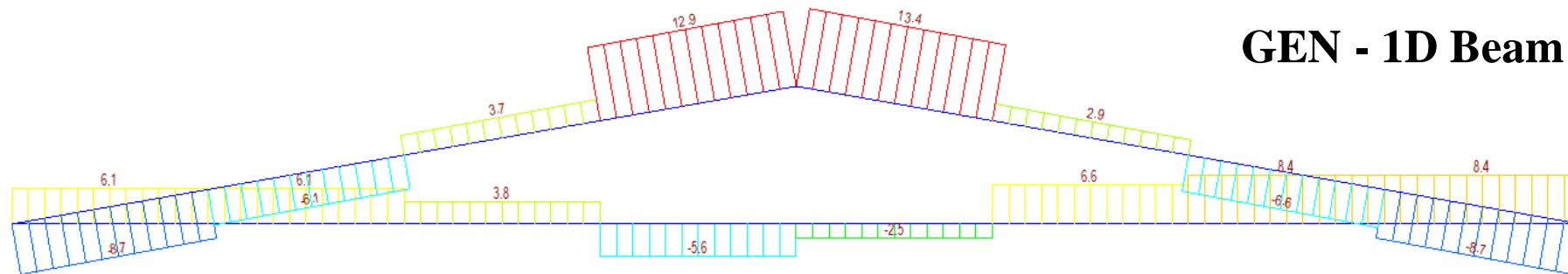
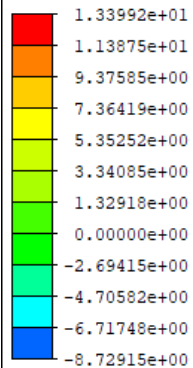
# RESULT COMPARISON - AXIAL X ( $F_x$ )



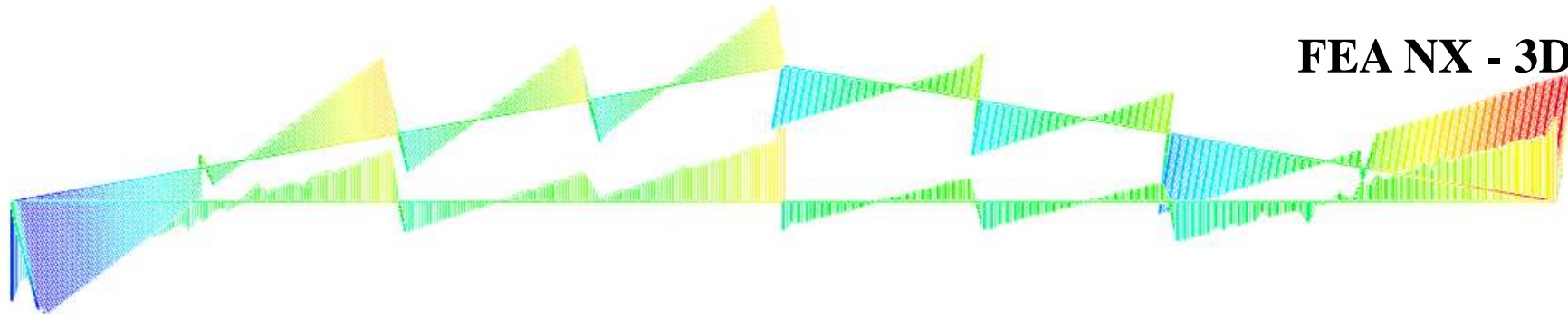
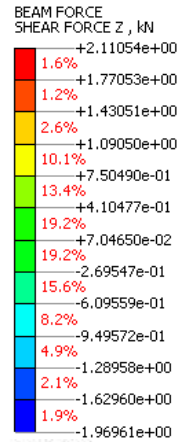
FEA



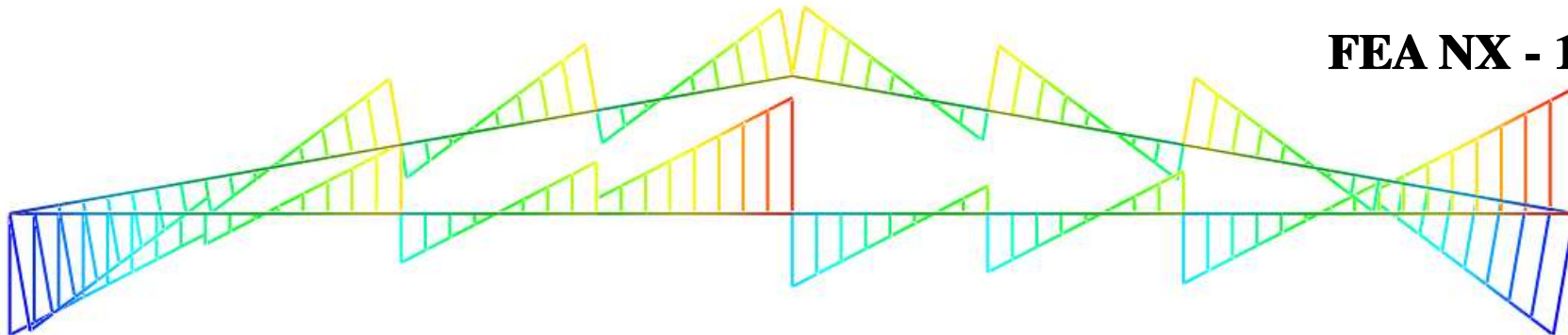
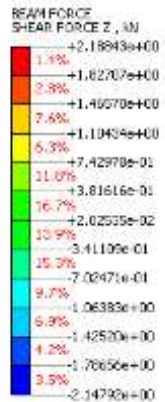
FEA



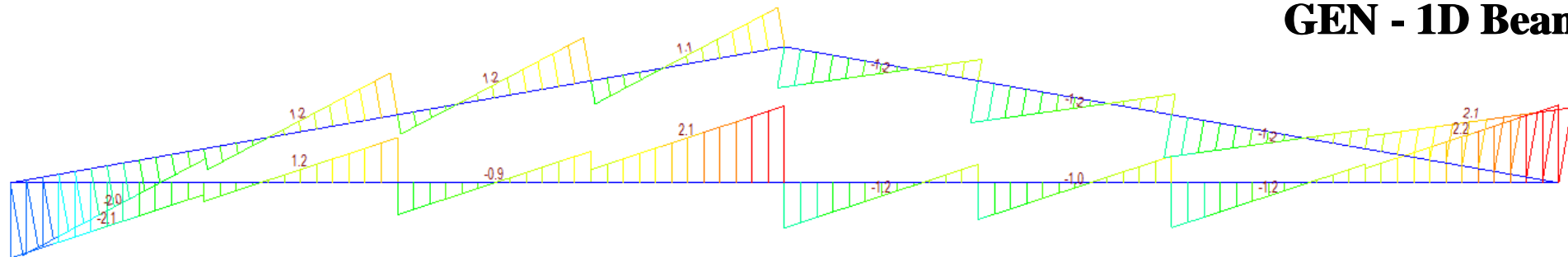
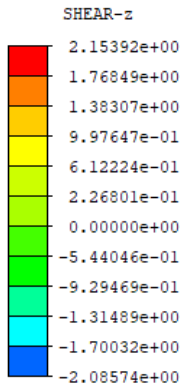
# RESULT COMPARISON - AXIAL Z (Fz)



FEA



FEA

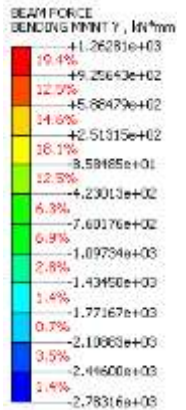


# RESULT COMPARISON - MOMENT Y (My)



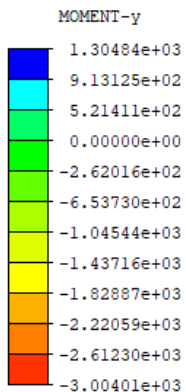
**FEA NX - 3D Beam**

FEA



**FEA NX - 1D Beam**

FEA

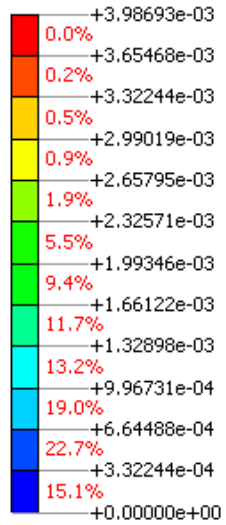


**GEN - 1D Beam**



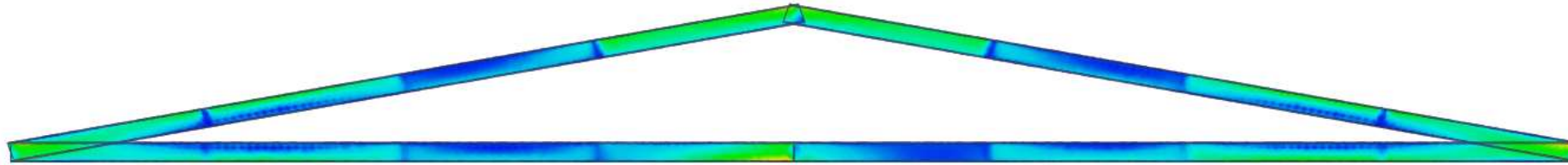
# RESULT COMPARISON - STRESS VON MISES

SOLID STRESS  
S-VON MISES , kN/mm<sup>2</sup>

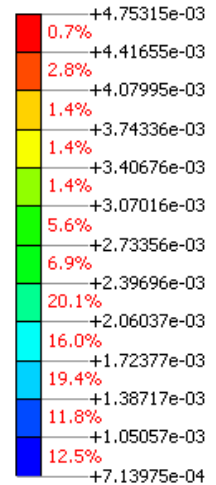


FEA NX - 3D Beam

FEA

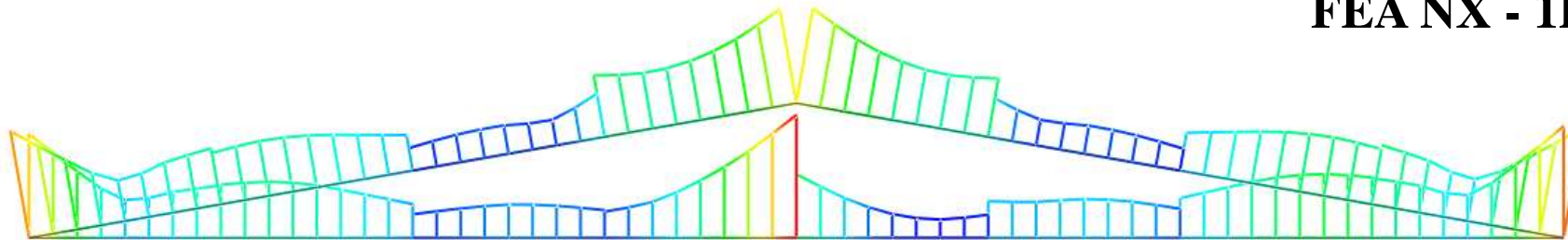


BEAM STRESS  
S-VON MISES , kN/mm<sup>2</sup>



FEA NX - 1D Beam

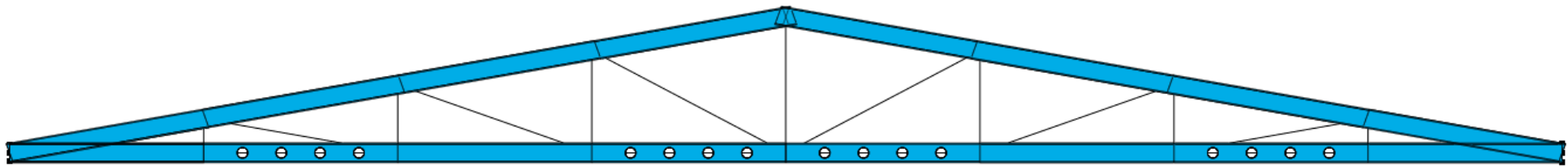
FEA



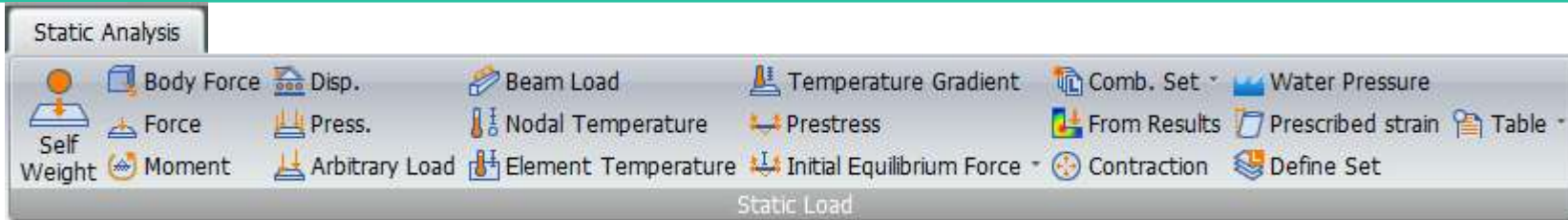


# Part 3

## 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

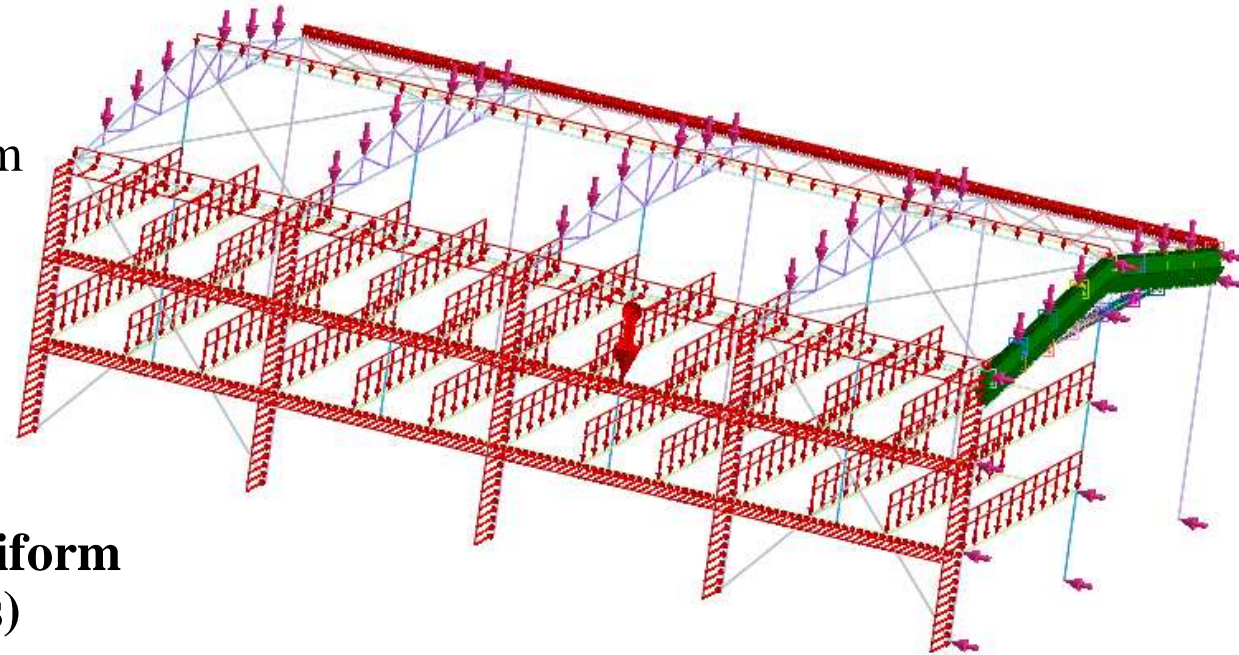
## Wind Load

$W_x = 0.88$  tonf/m

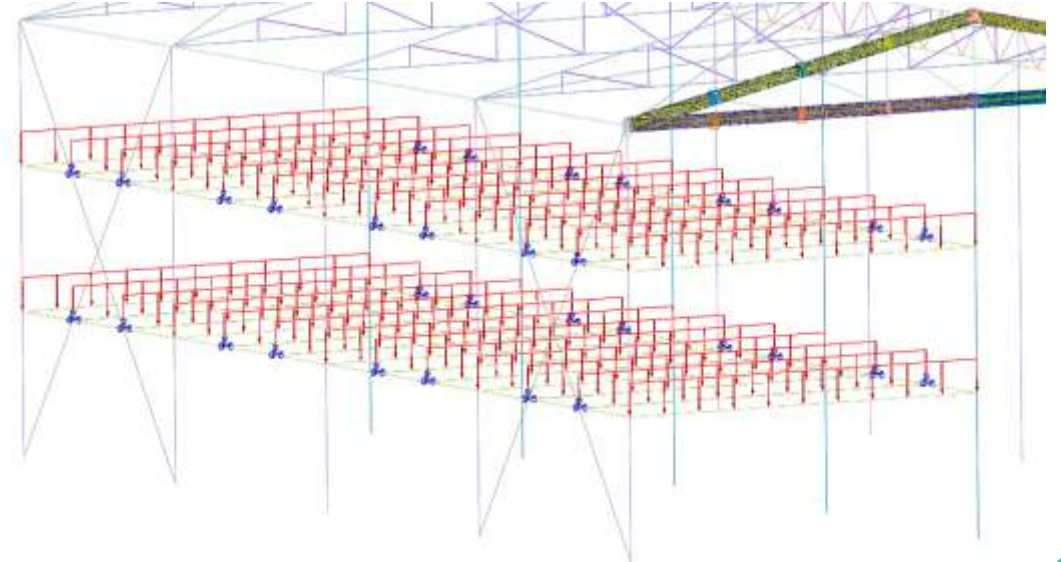
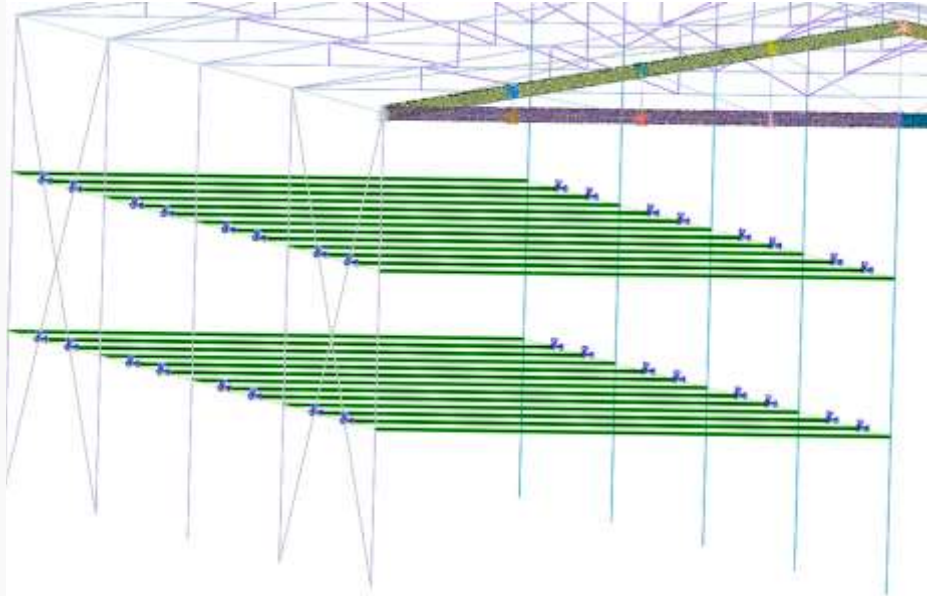
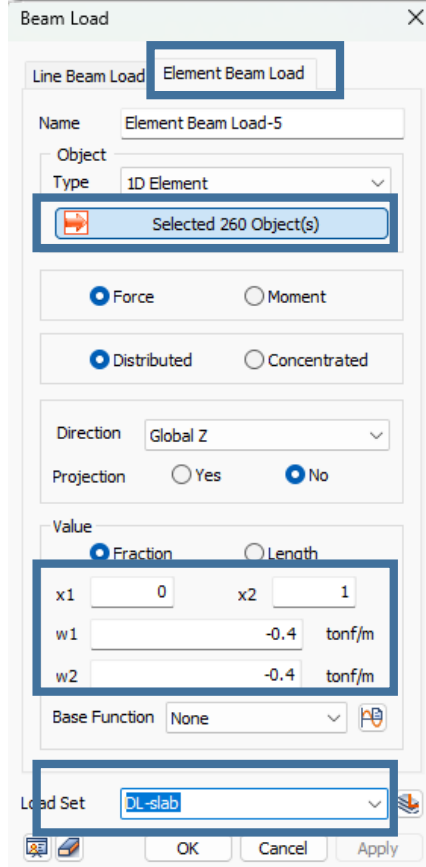
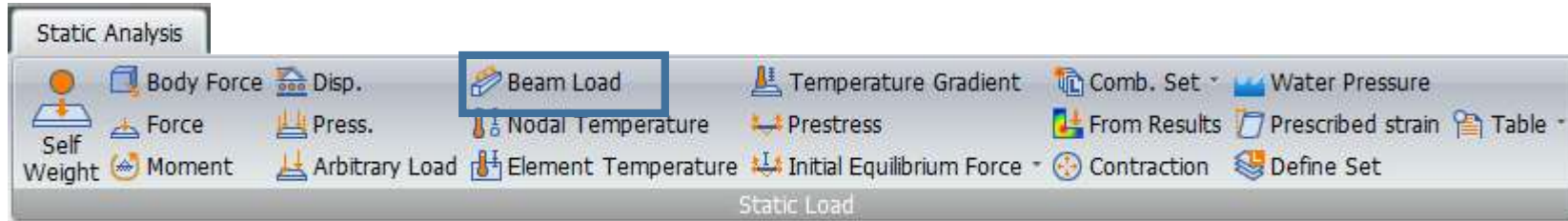
$W_y = 0.92$  tonf

## Additional Load (Uniform Load front Truss)

Uniform load = 2 tonf/m

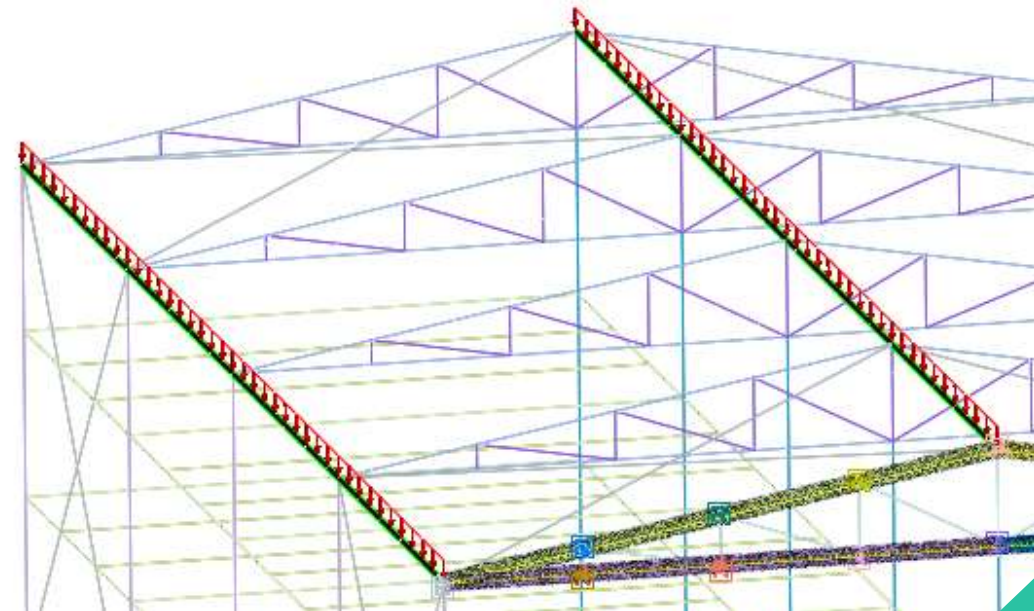
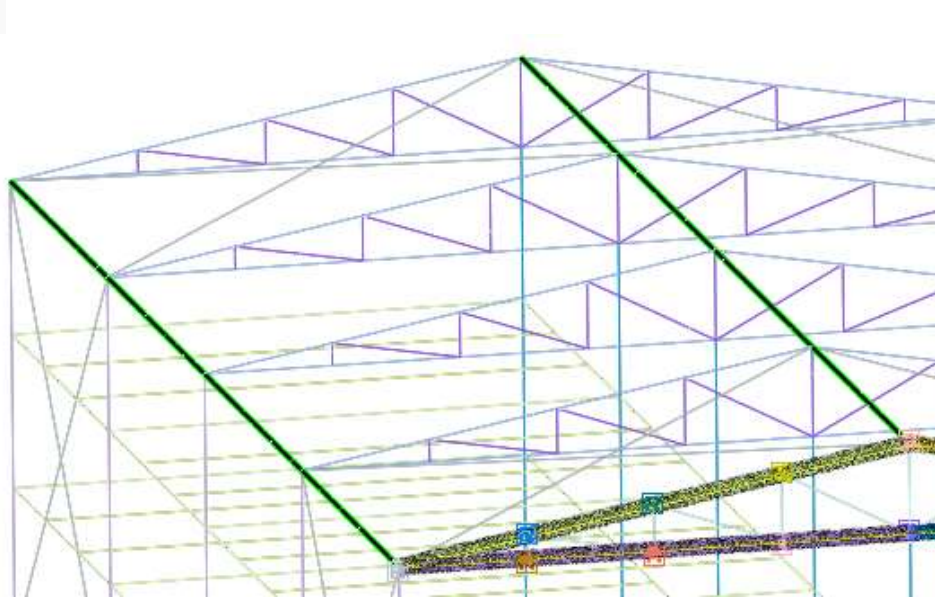
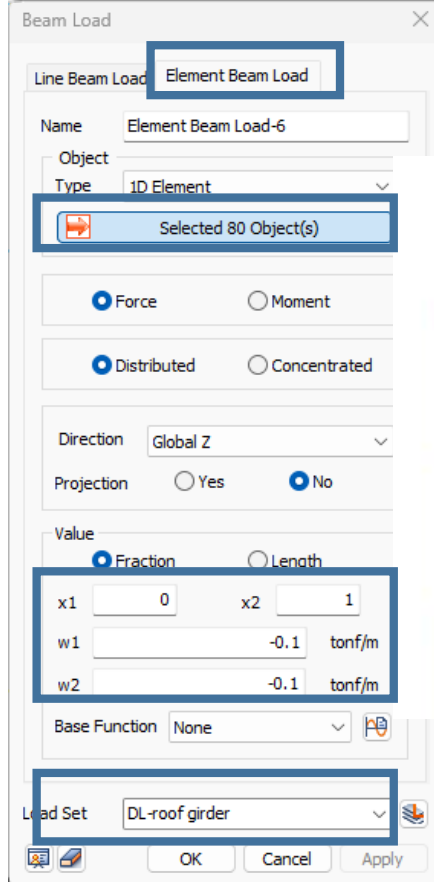
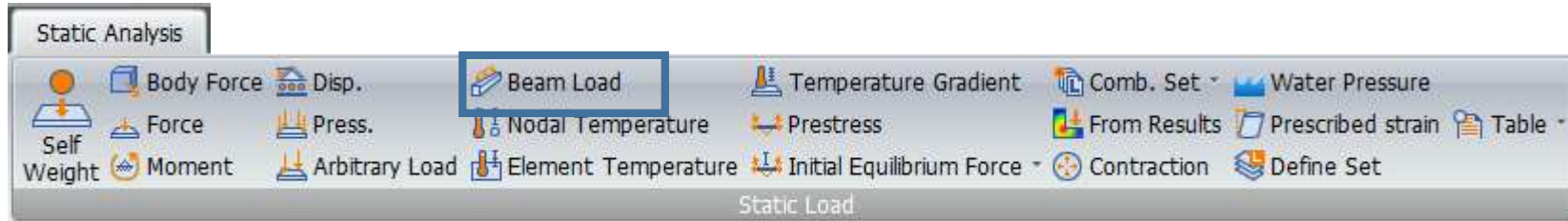


# DEAD LOAD - SLAB



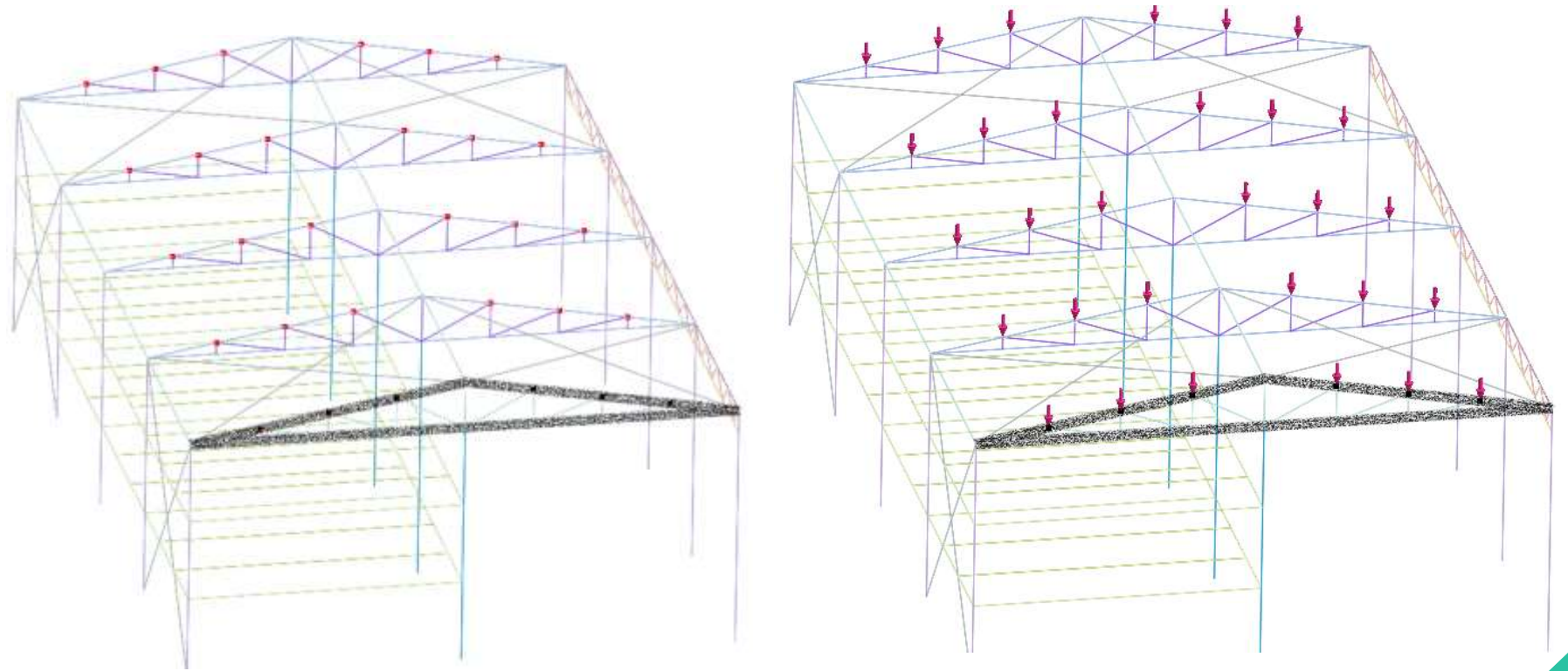
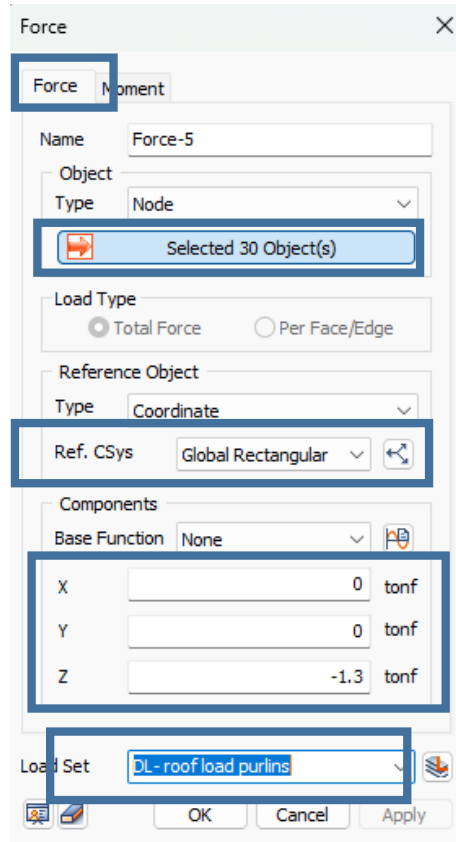
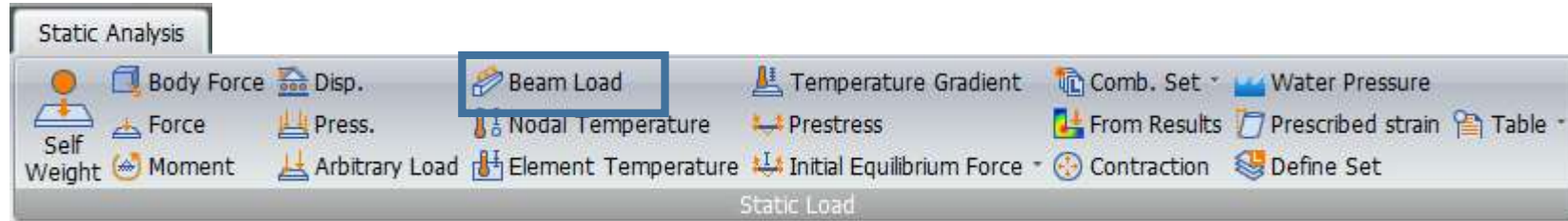


# DEAD LOAD - ROOF GIRDER

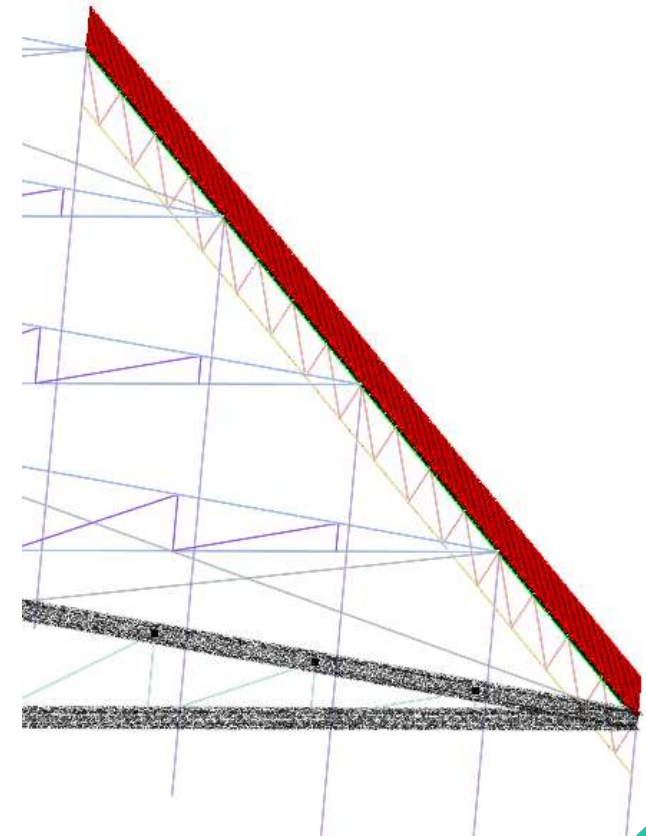
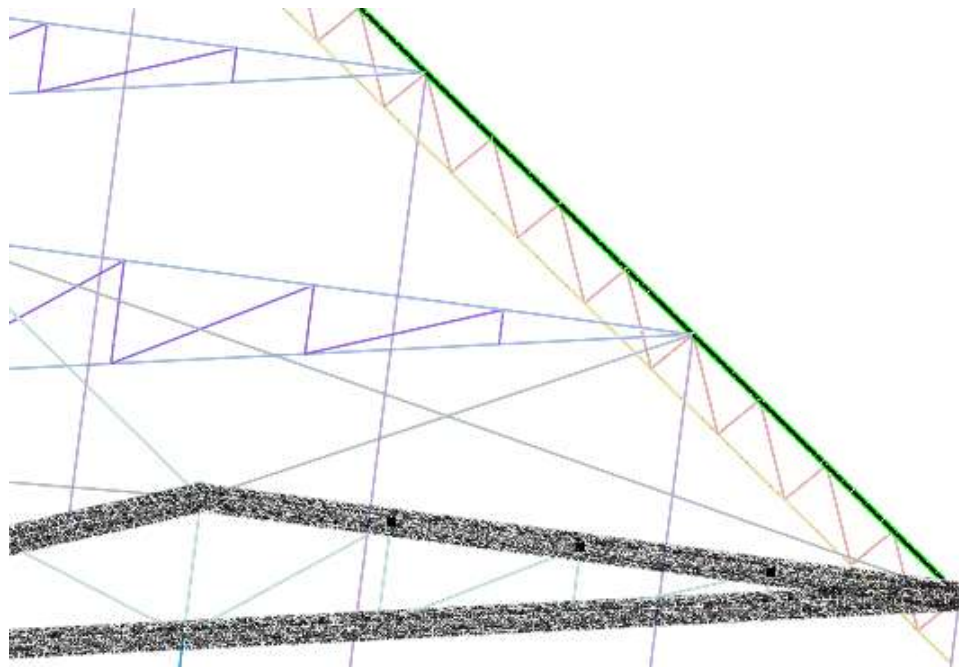
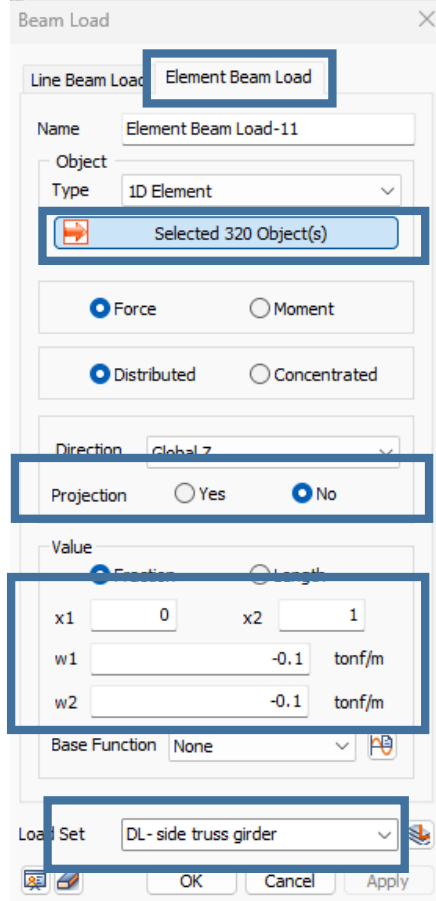
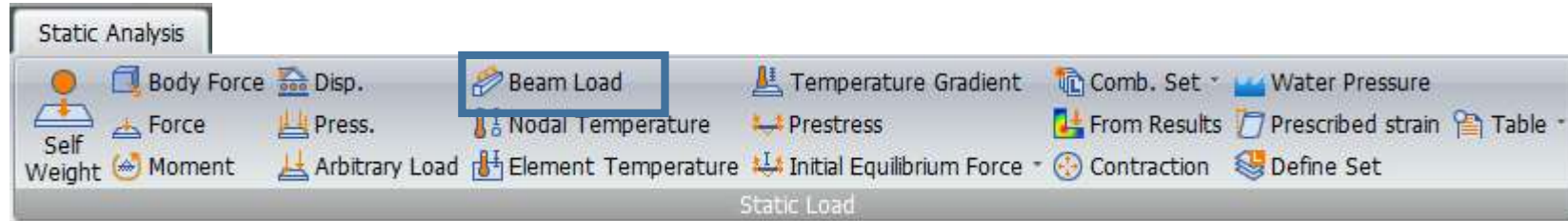




# DEAD LOAD - PURLINS

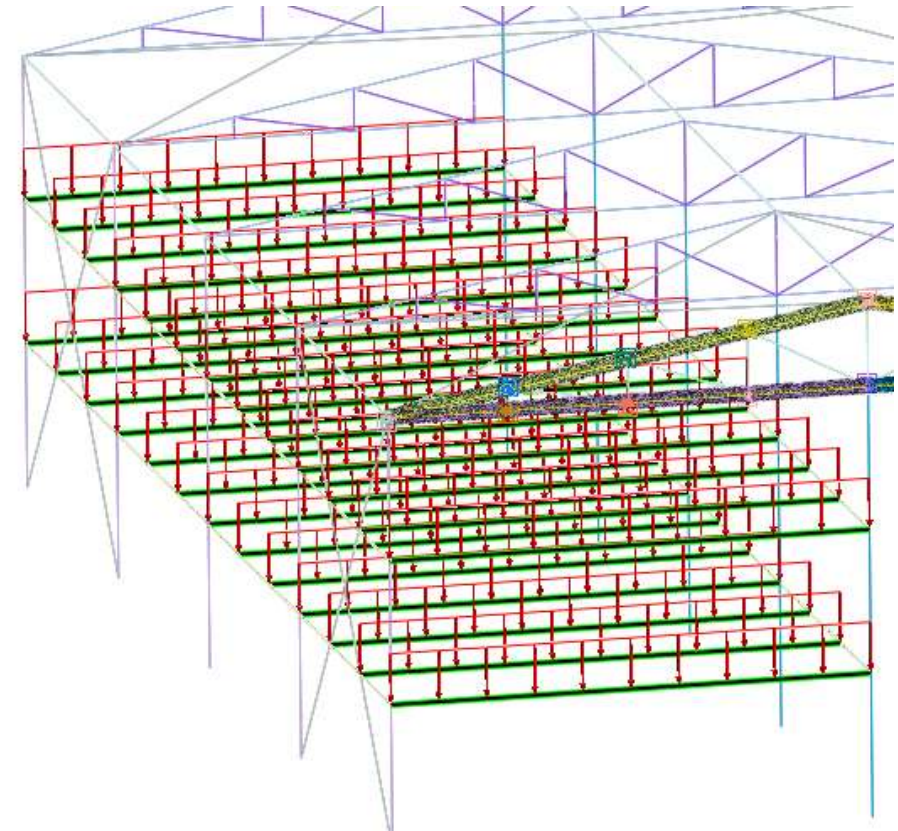
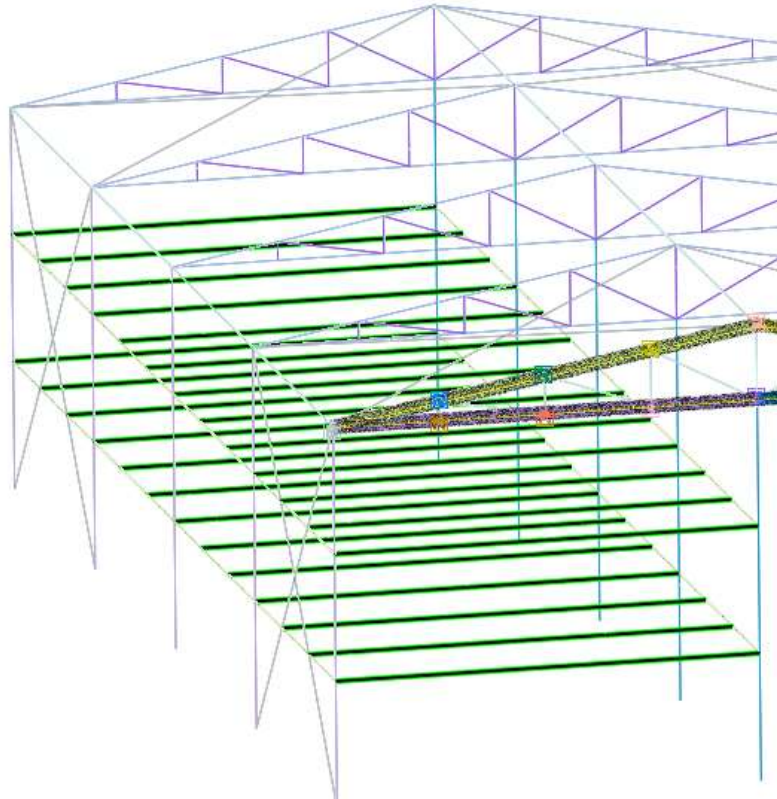
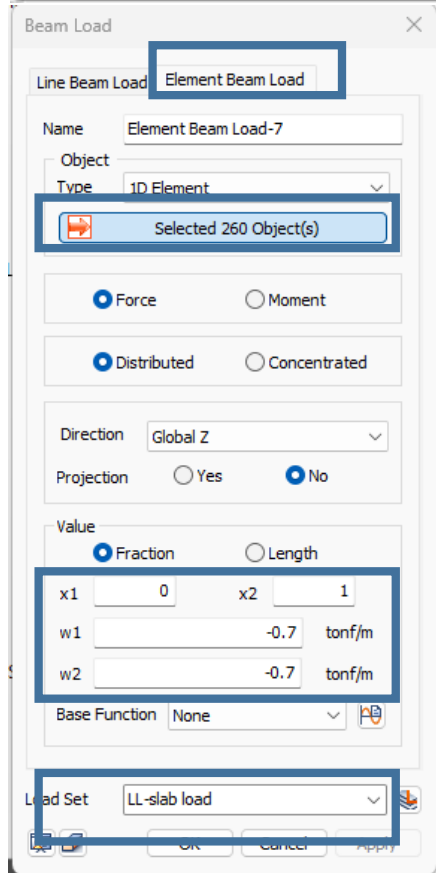
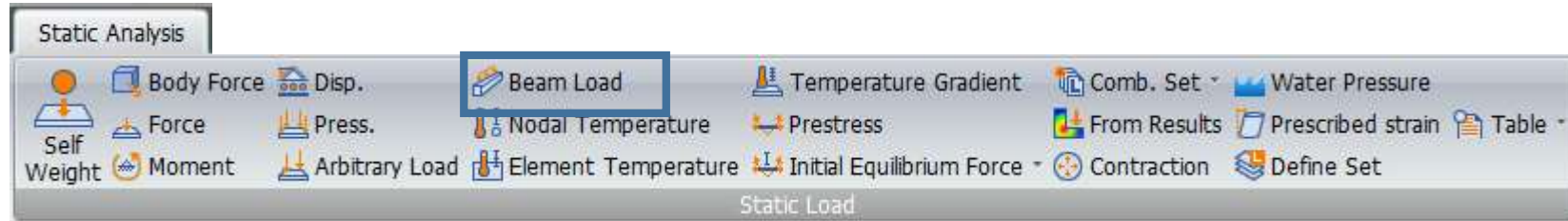


# DEAD LOAD - SIDE TRUSS

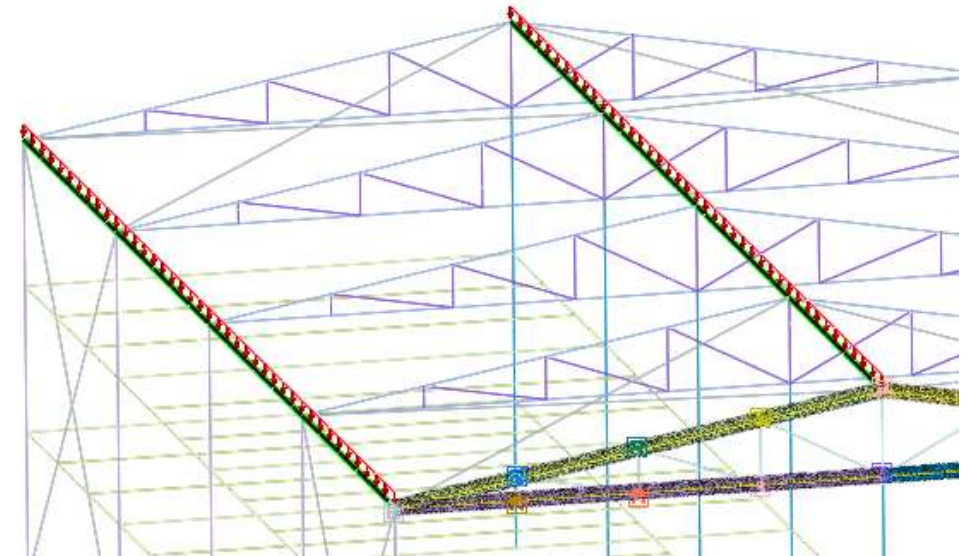
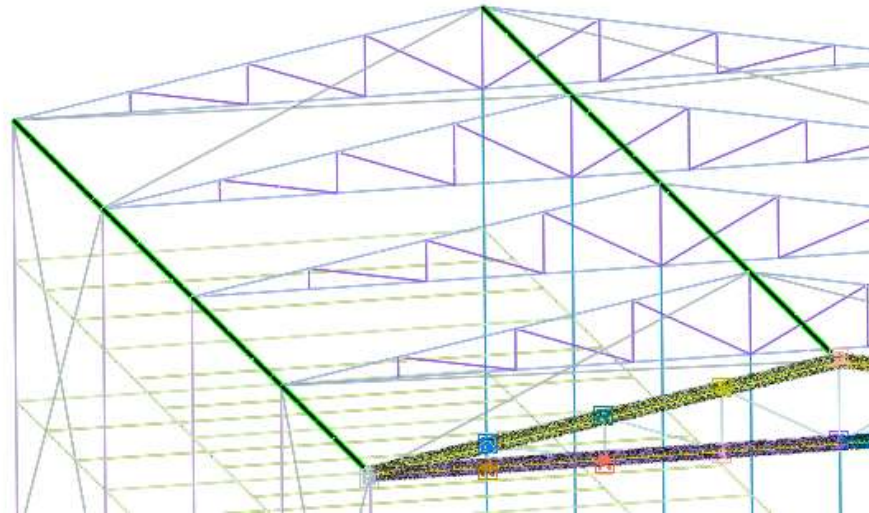
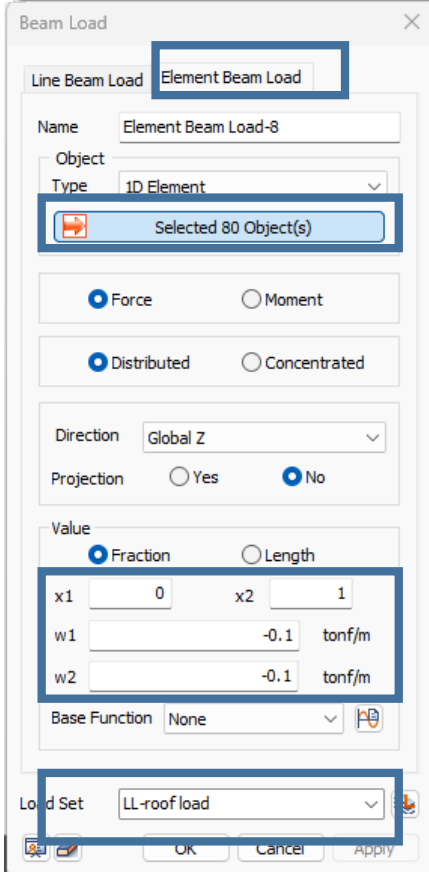
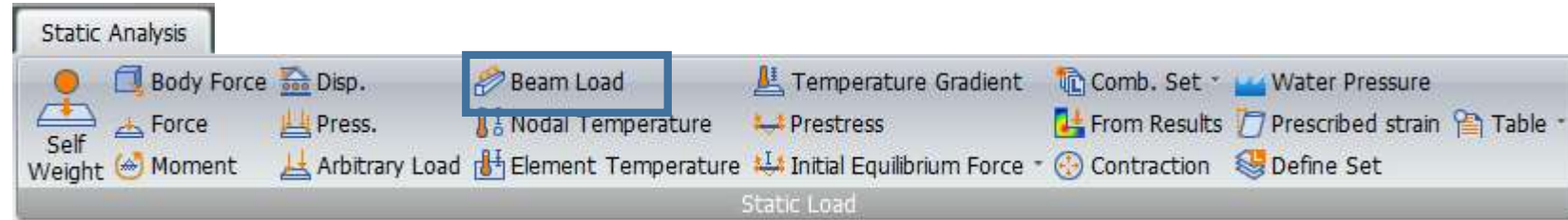




# LIVE LOAD - SLAB

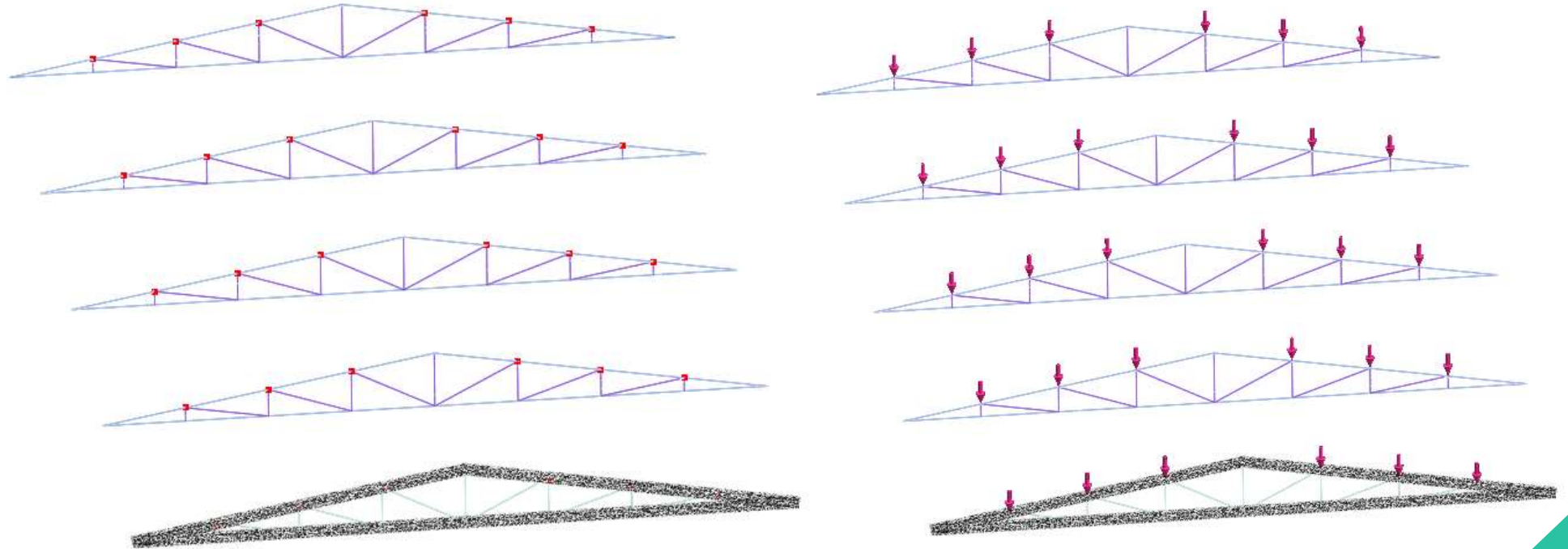
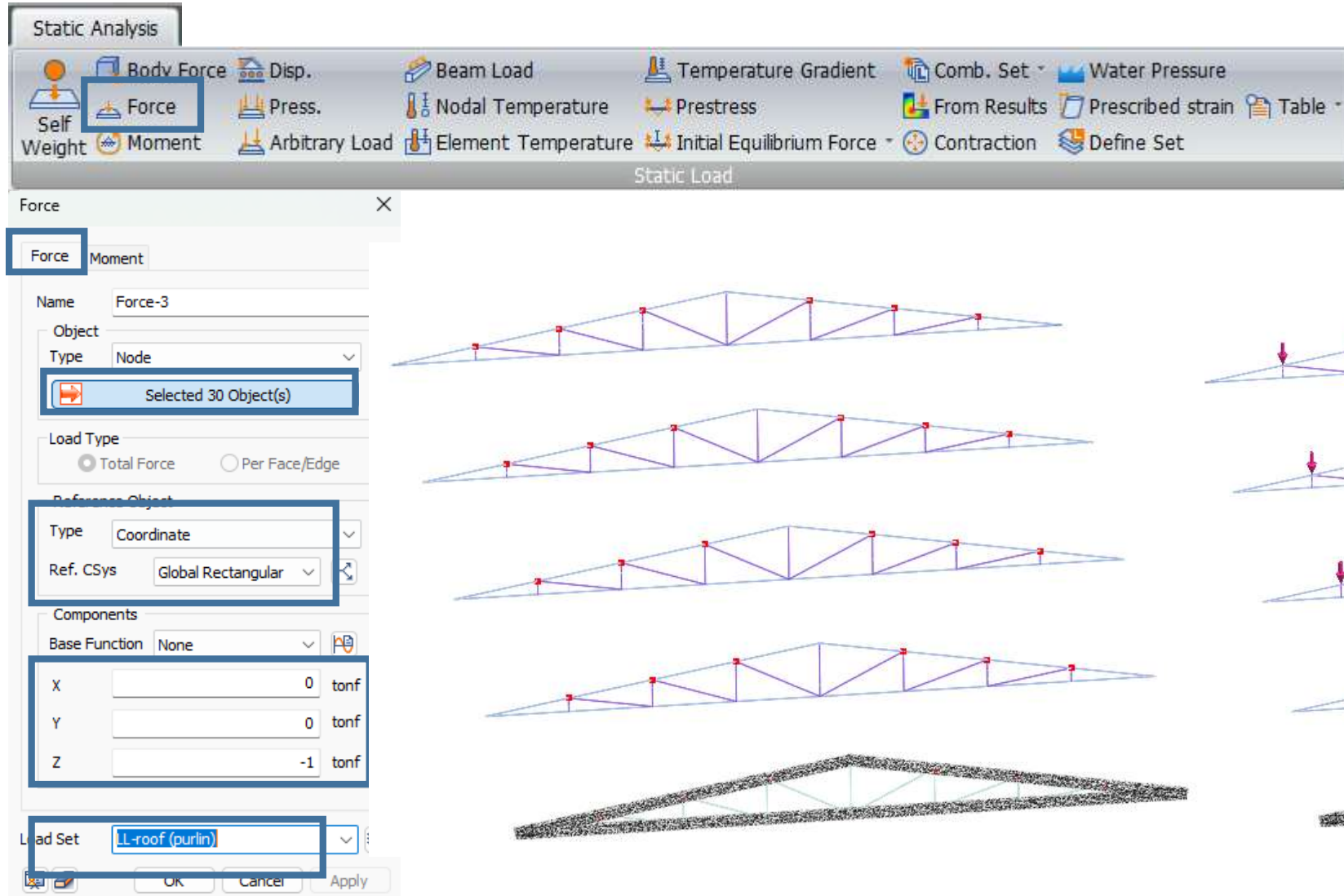


# LIVE LOAD - ROOF LOAD

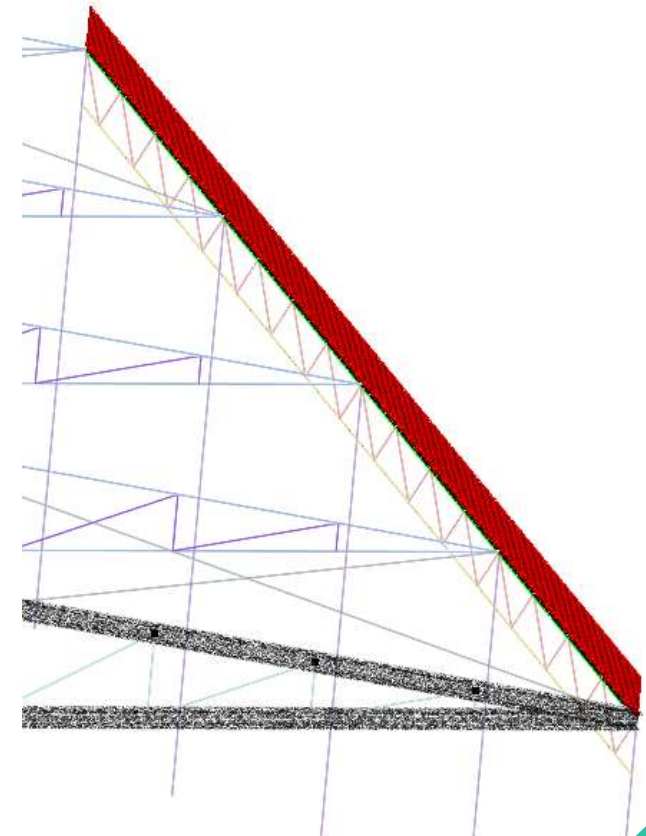
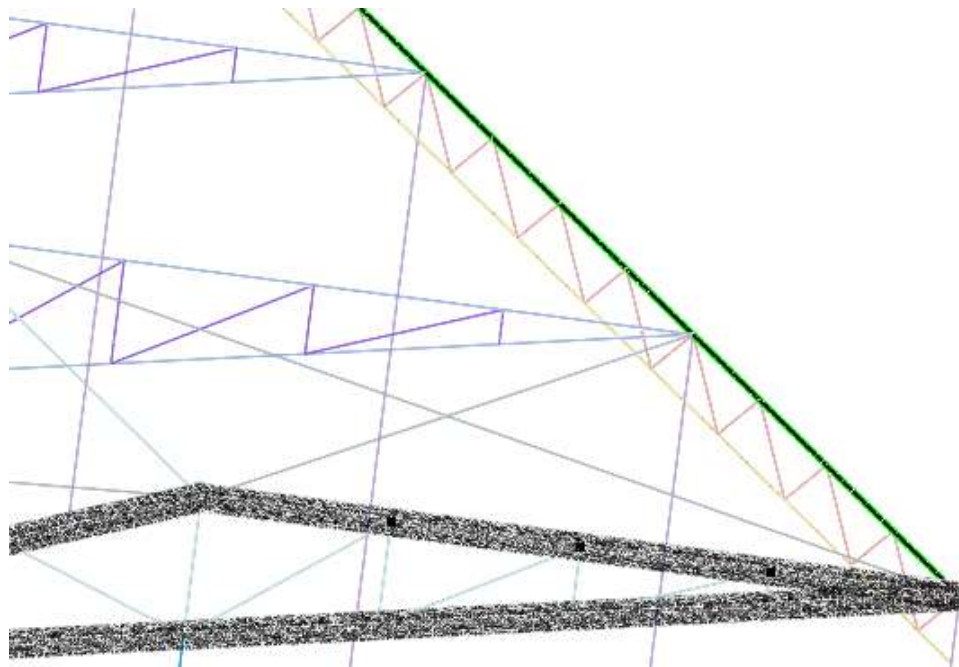
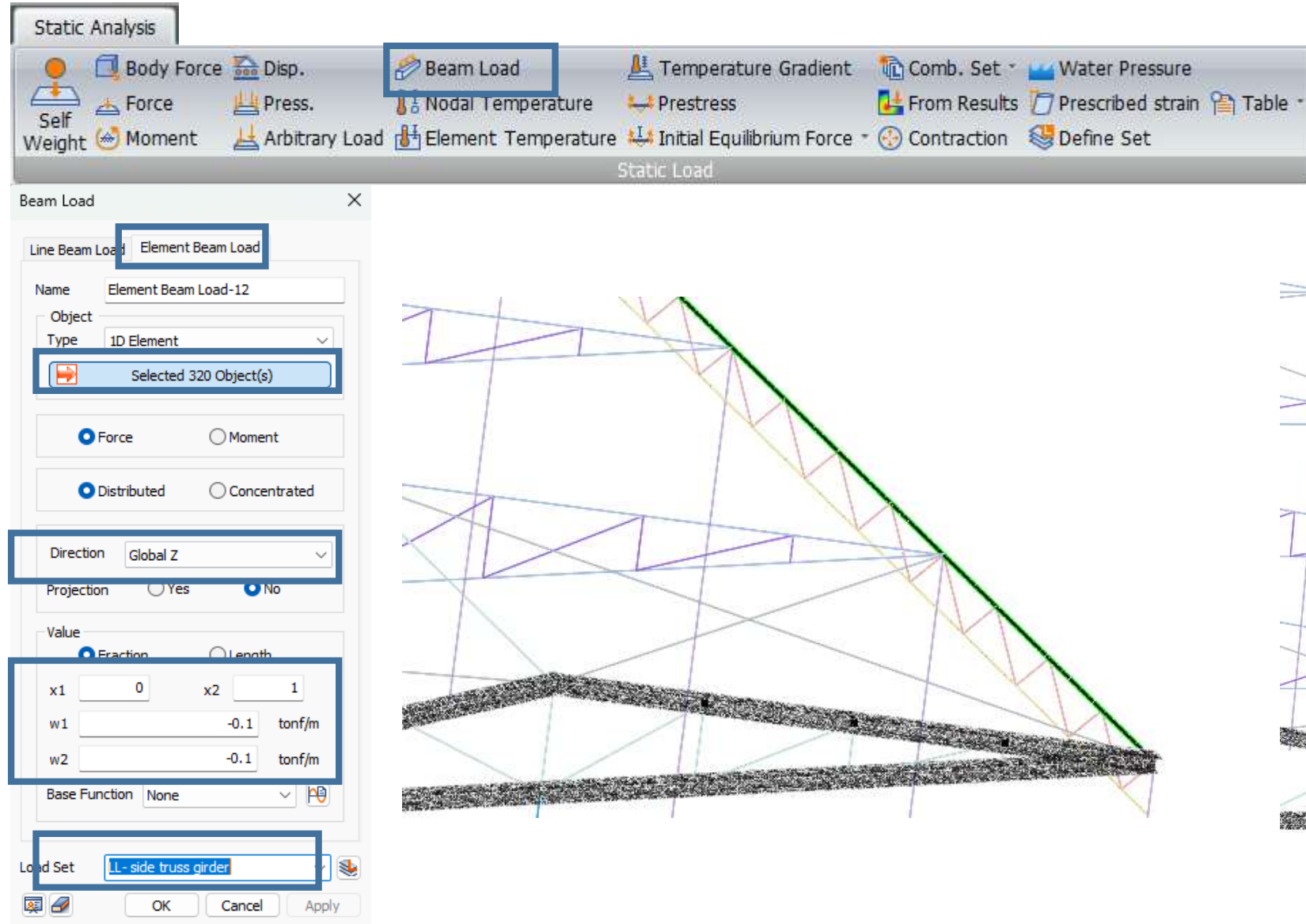




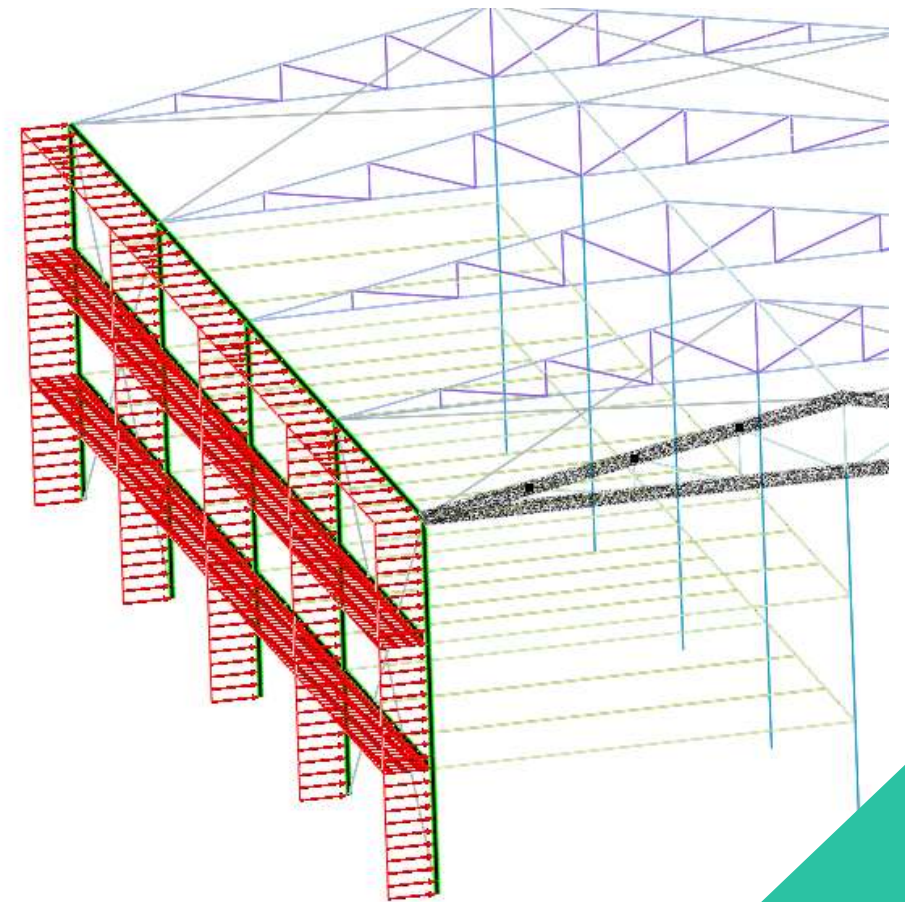
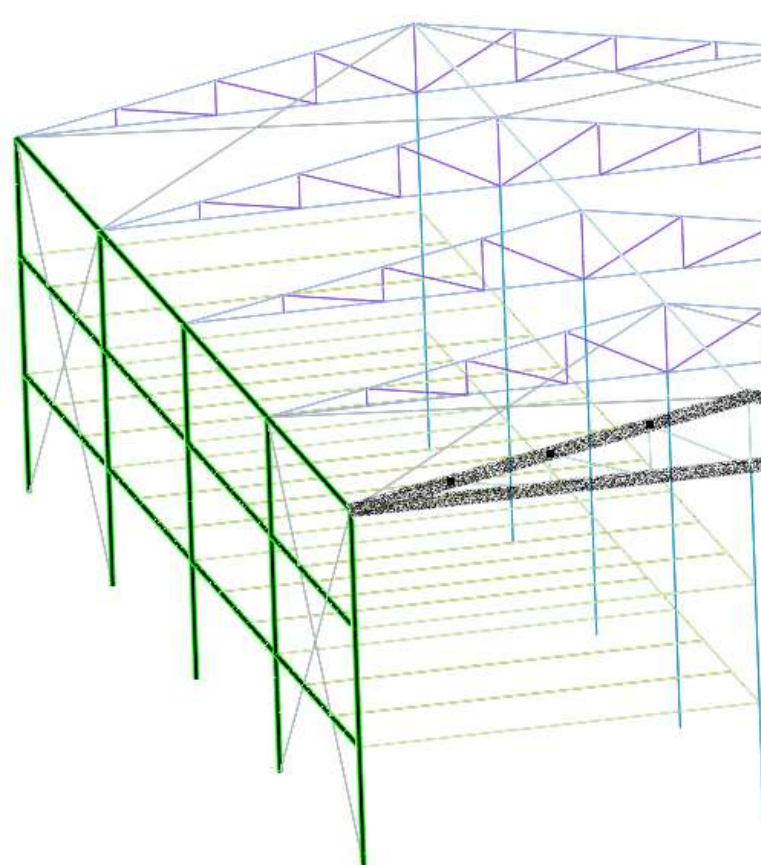
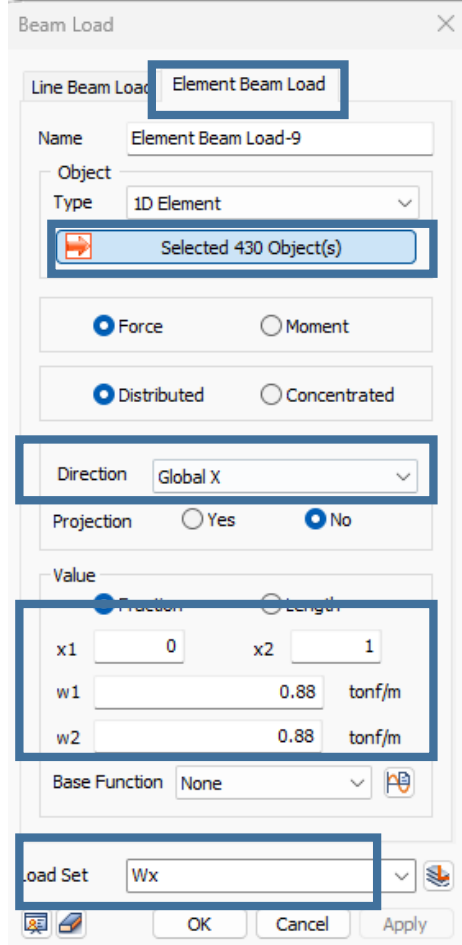
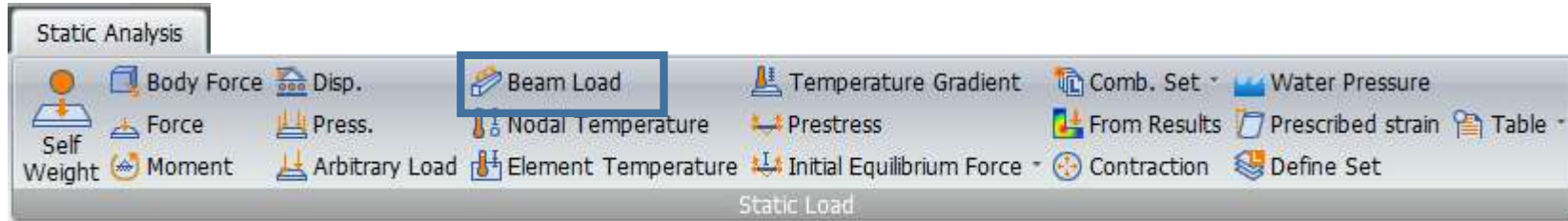
# LIVE LOAD - PURLINS



# LIVE LOAD - SIDE TRUSS

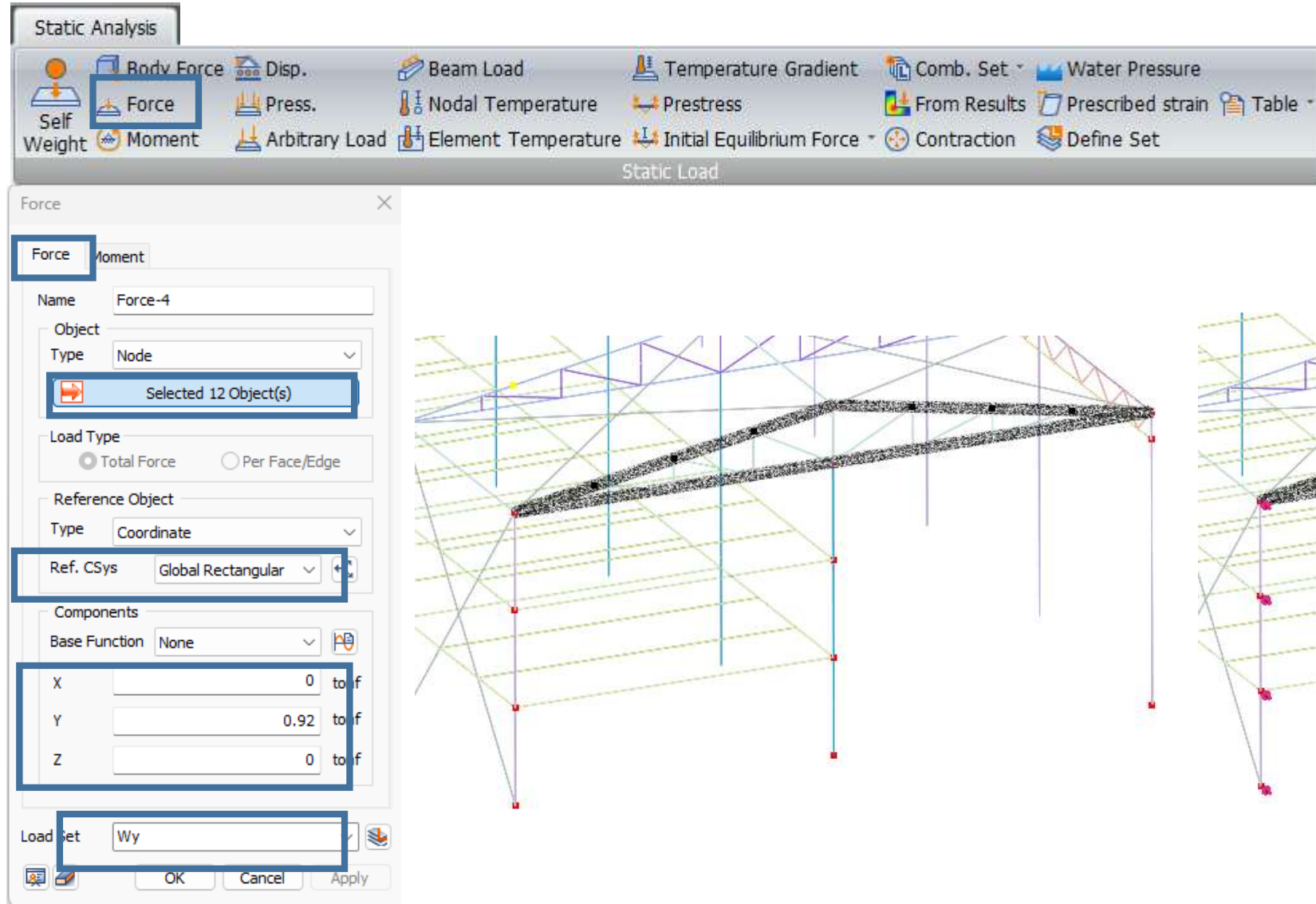


# WIND LOAD $W_x$



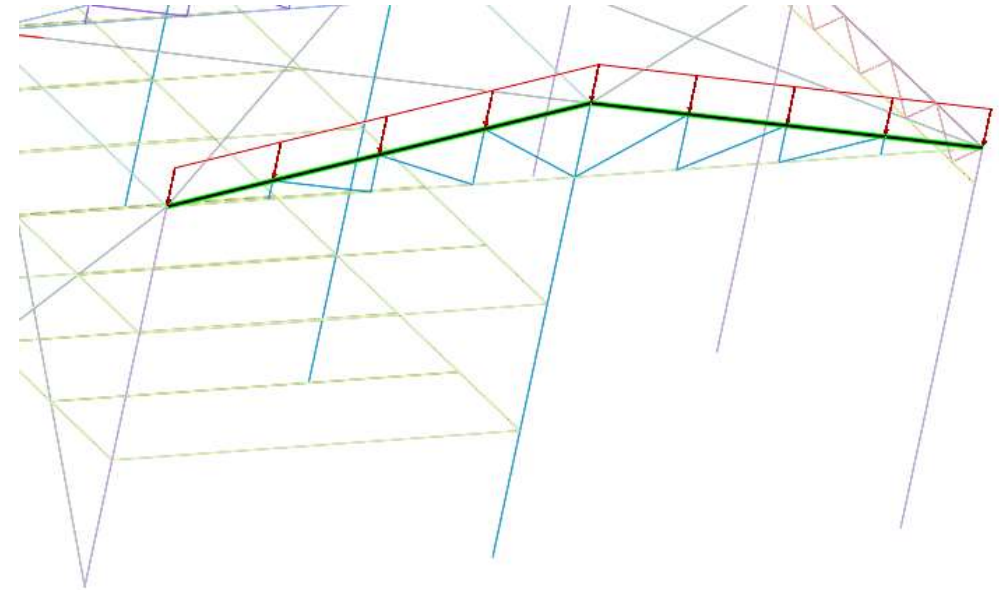
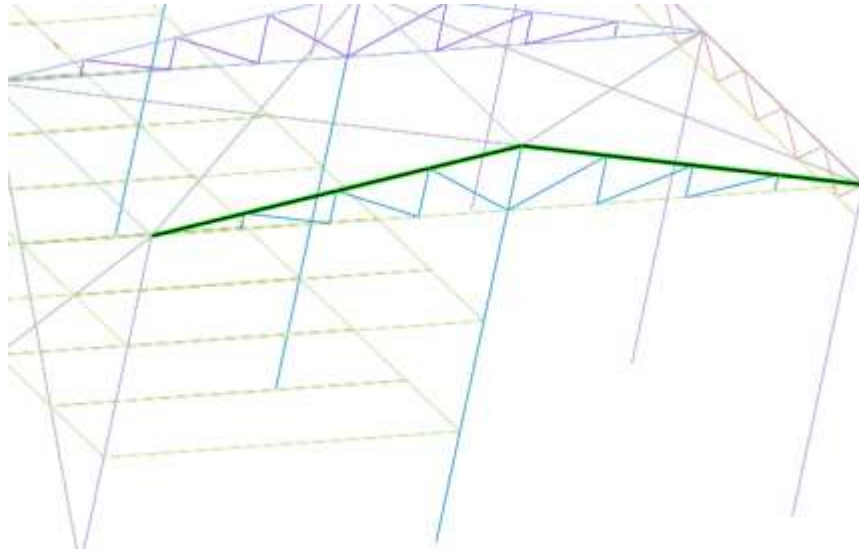
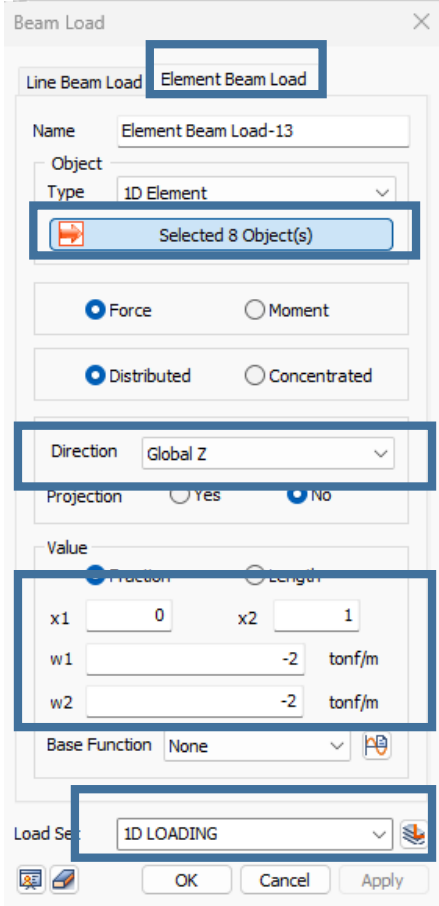
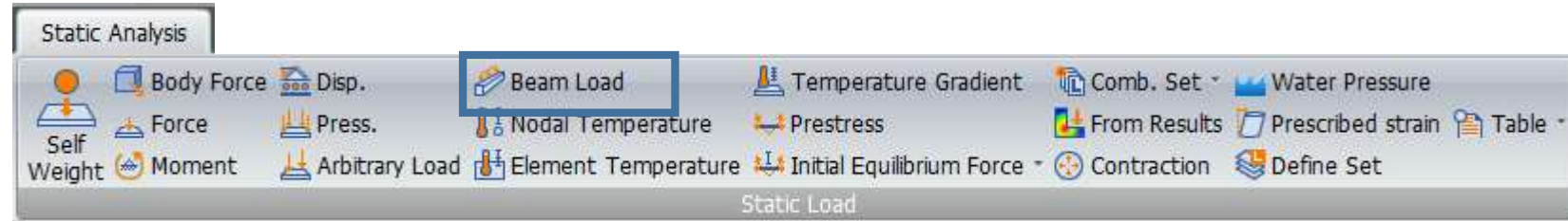


# WIND LOAD $W_y$

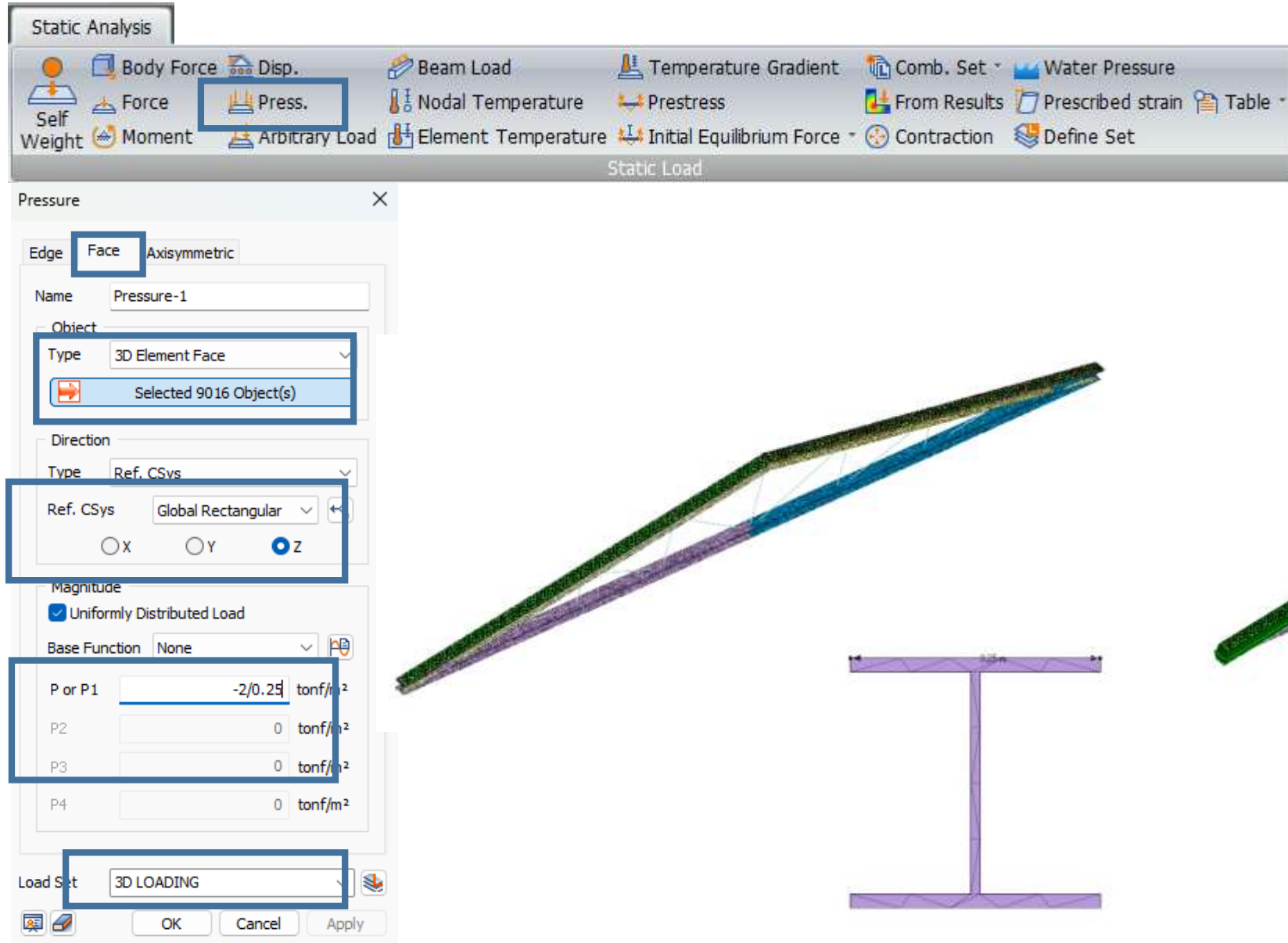




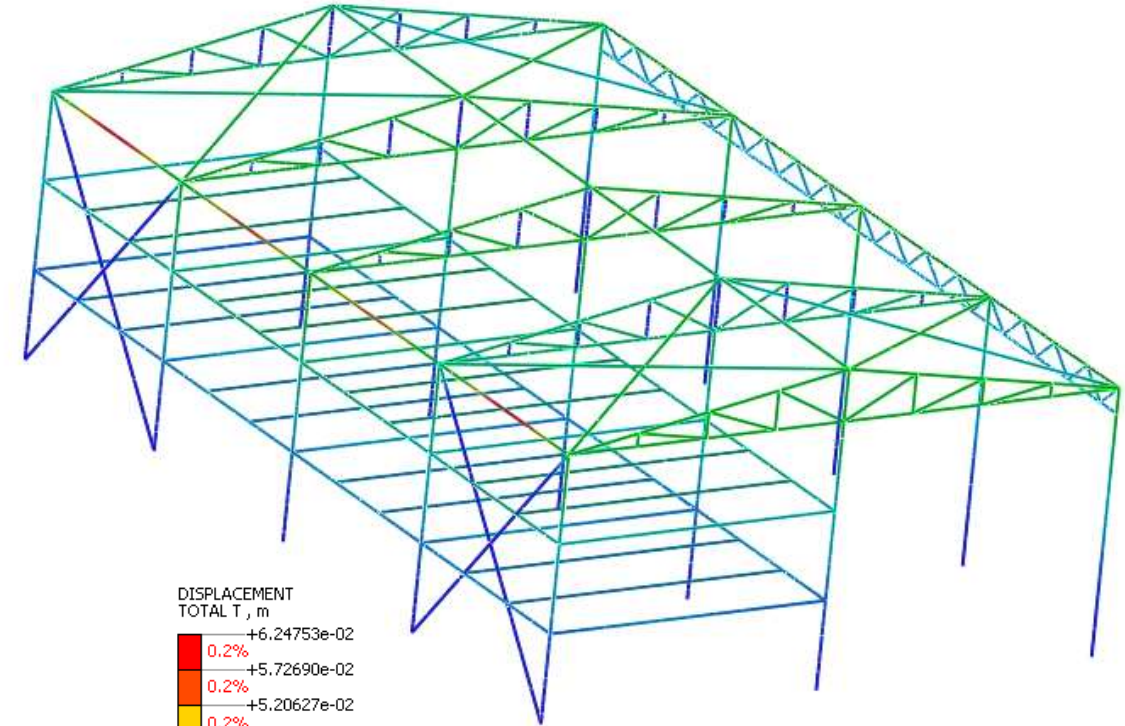
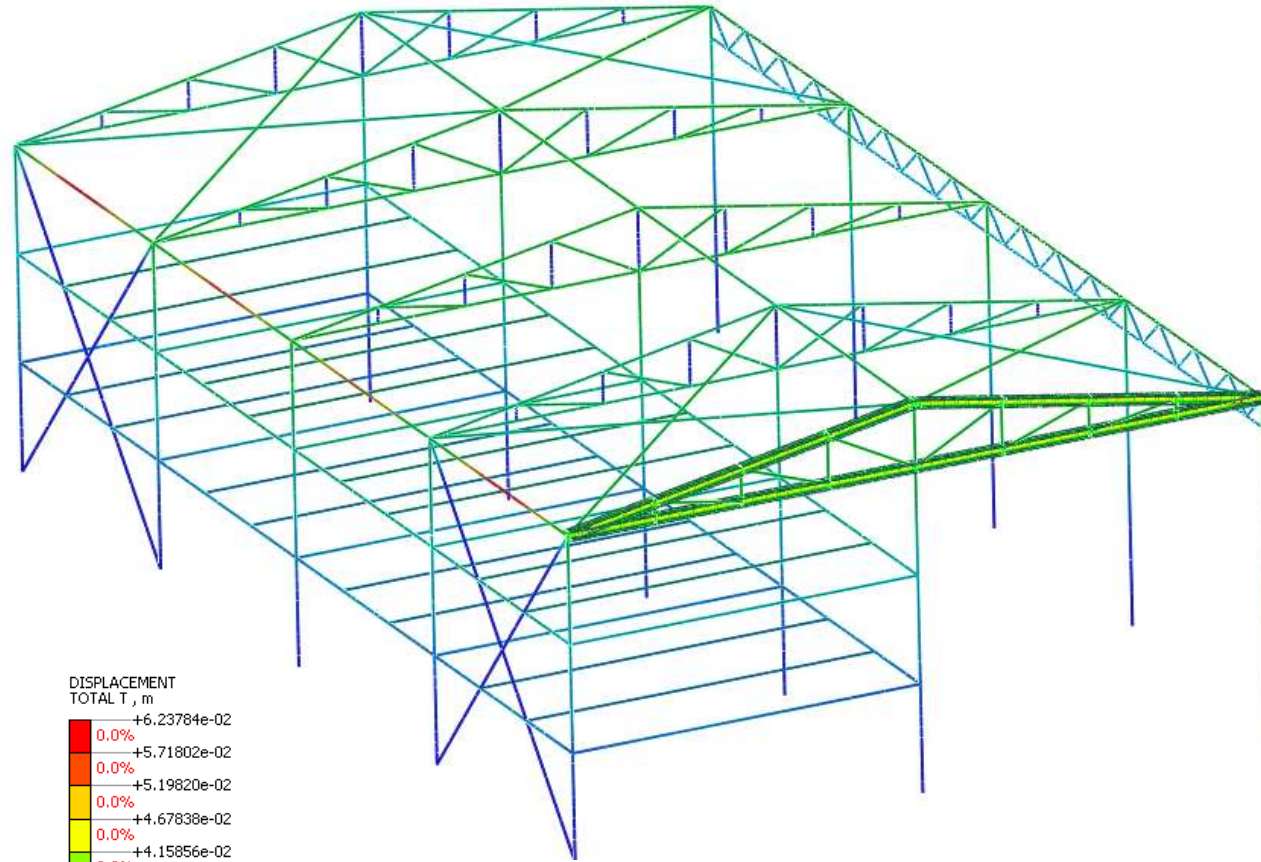
# 1D LOADING



# 3D LOADING



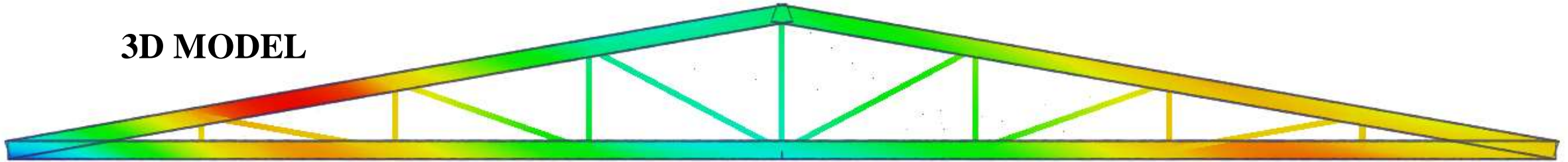
# RESULT COMPARISON



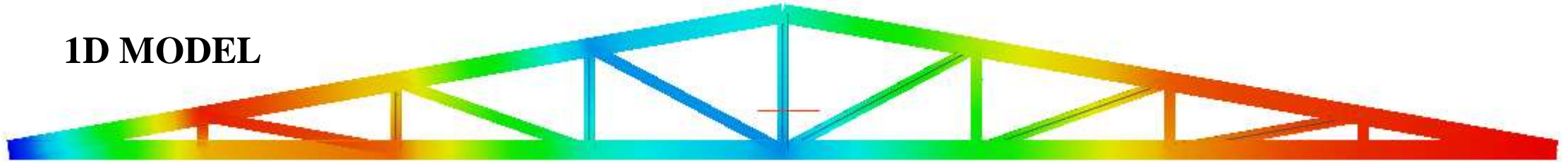


# RESULT COMPARISON

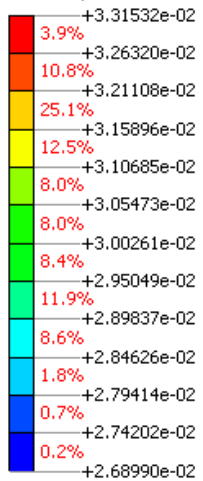
3D MODEL



1D MODEL



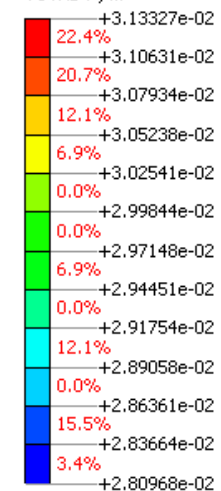
DISPLACEMENT  
TOTAL T, m



3D MODEL

DISPLACEMENT

DISPLACEMENT  
TOTAL T, m

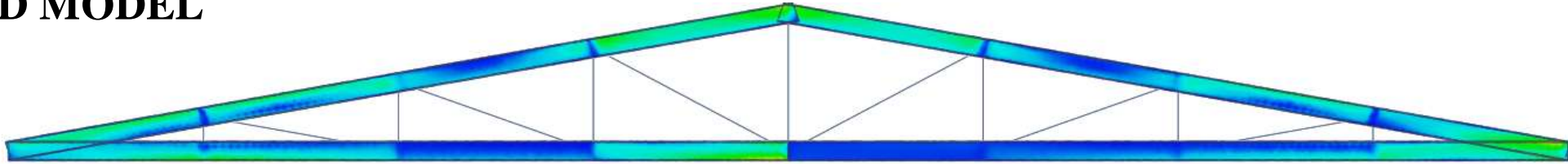


1D MODEL

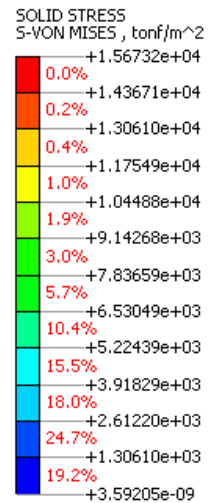
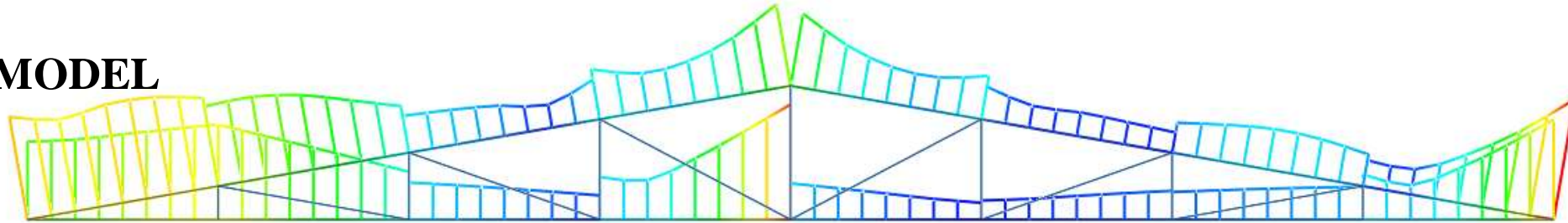


# RESULT COMPARISON

**3D MODEL**

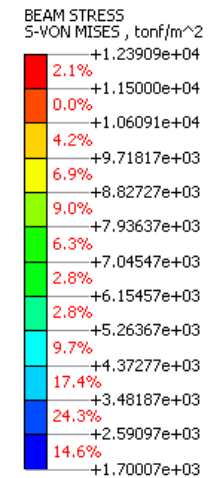


**1D MODEL**



**3D MODEL**

## STRESS VON MISES



**1D MODEL**