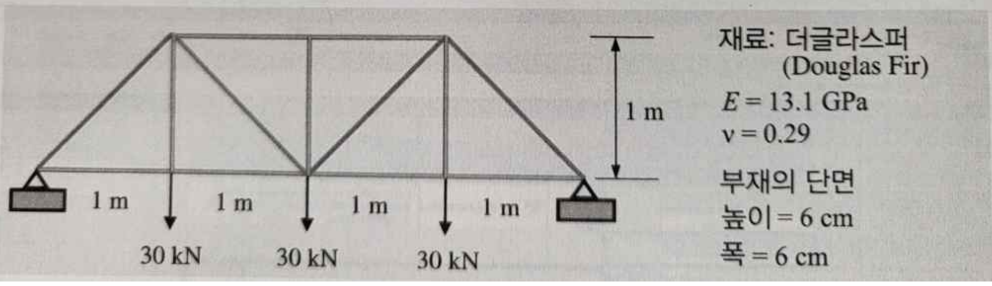
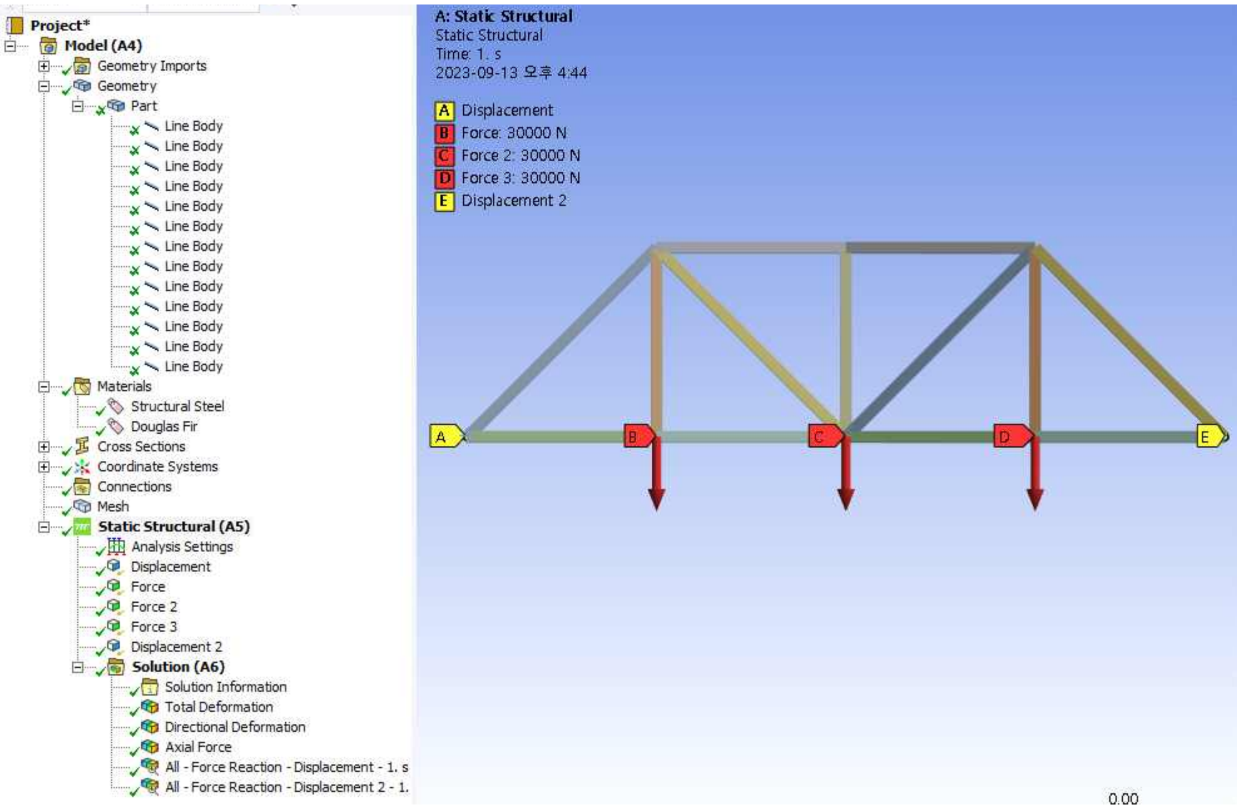
3. Truss and Beam Element (1D Element)

3-1. 2D Truss

문제 설명: 트러스 교량은 긴 거리에 걸쳐 중간 지지 없이도 무거운 하중을 지지하는 할 수 있다. 이러한 교량을 건설하는 것은 경제적이며 다양한 형태가 가능하다. 목재로 건축되어 병렬 교량 형태로 사용될 수 있는 평면 트러스를 고려하라. 아래의 주어진 하중조건에서 각 조인트에서 발생하는 처짐을 결정하라.





Go to Design Modeller

(1) Geometry

- Create Points: from a file (needs to be prepared in advance)

Details View

Details of Point1	
Point	Point1
Type	Construction Point
Definition	From Coordinates File
Coordinates File	D:₩cslee₩강의₩학부₩2학기-강의₩CAE기계설계해석2023₩실...₩Point_data_truss.txt
Coordinates Unit	Meter
Base Plane	XYPlane
Tolerance	Normal
Refresh	No
# Points generated	8

Point_data_frame - Windows 메모장

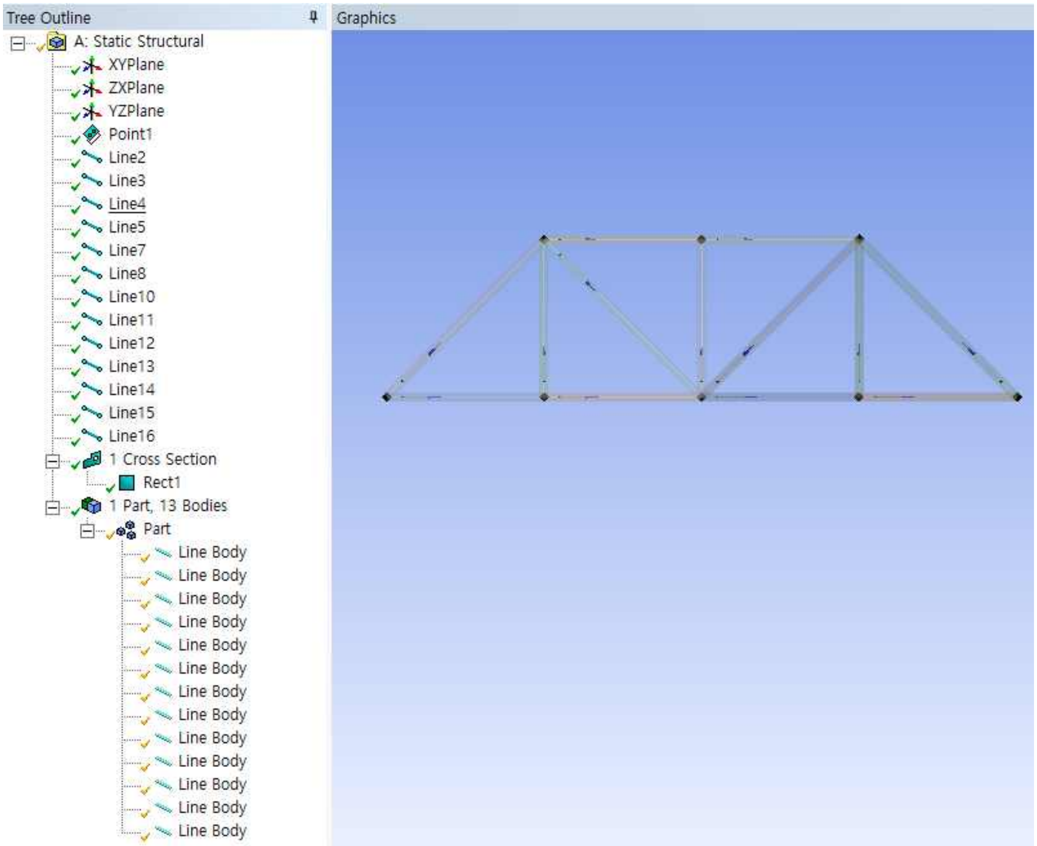
파일(F) 편집(E) 서식(O) 보기(V) 도

```
1 1 0 0 0
1 2 1 0 0
1 3 2 0 0
1 4 3 0 0
1 5 4 0 0
1 6 1 1 0
1 7 2 1 0
1 8 3 1 0
```

- Line generation
    - Concepts --> Line from Points, Draw a line connecting two nodes.
      "add frozen" option to separate line body each other
    - Click Generate Button after finishing line generation. Line Bodies are generated.
    - Repeat above 13 times to generate 13 line bodies

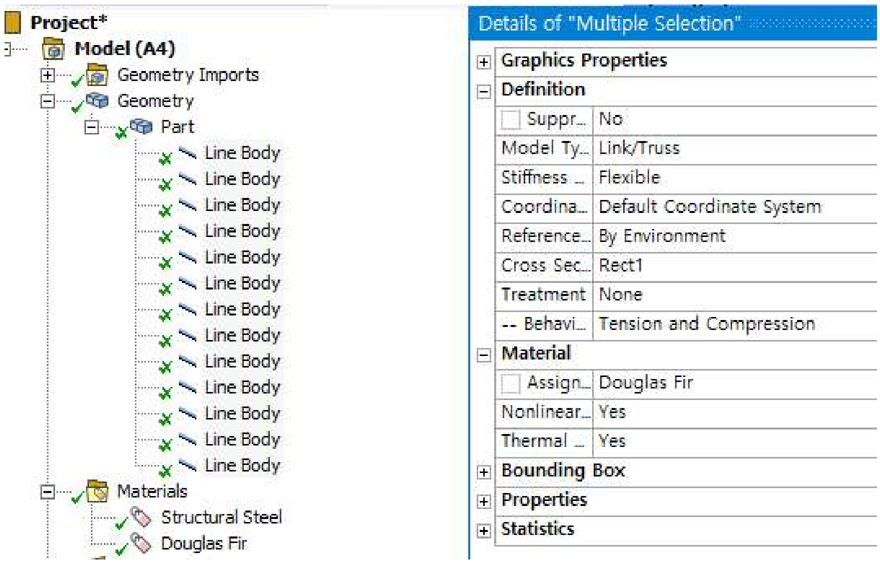
- Concepts --> Cross section, Rectangular 0.06 x 0.06 m
    - Select all line bodies and assign the cross section generated above

- Generate only one part
    - Select all line bodies, RMB --> Form a new part



Return to Work Bench
(2) Material
- General materials --> Create new material
- Douglas Fir (E=13.1GPa, v=0.29)
- Click Update Button to include the new material in Work Bench

Go to Static Structural
(3) Geometry
- Select all line bodies: Beam --> Link/Truss for all line bodies (pin connected between joints)
- Assign the new material (Douglas Fir)



(4) Mesh

- Size to 100mm or division to 10 (not important for Truss element)

(5) Boundary Condition

- Left end (pin support): displacement --> x=y=z=0
- Right end (roller support): displacement --> x=free, y=z=0
- Select all nodes (rigid body motion prevented): displacement --> x=y=free, z=0

(6) Load Condition

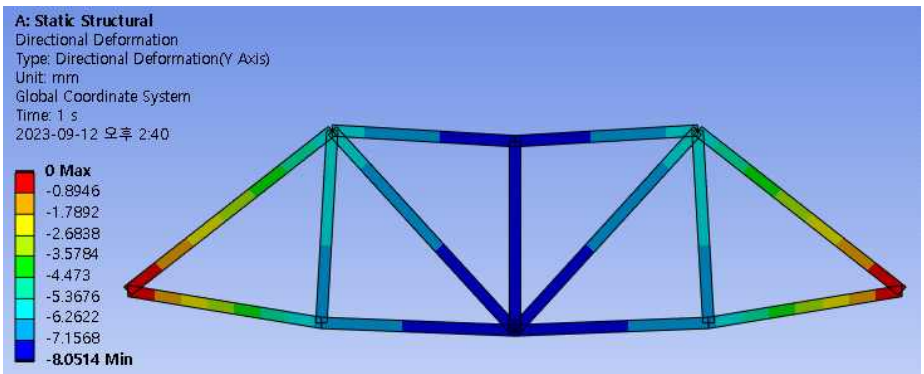
- 30,000N downward at three lower joints respectively

(7) Static Structural

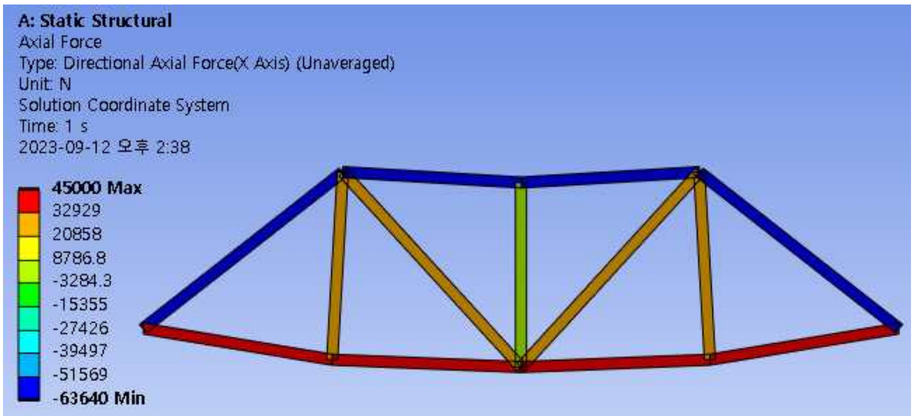
- Analysis Settings: Weak Springs --> Program Controlled (Rigid body motion prevented)

(8) Results

- Directional Deformation



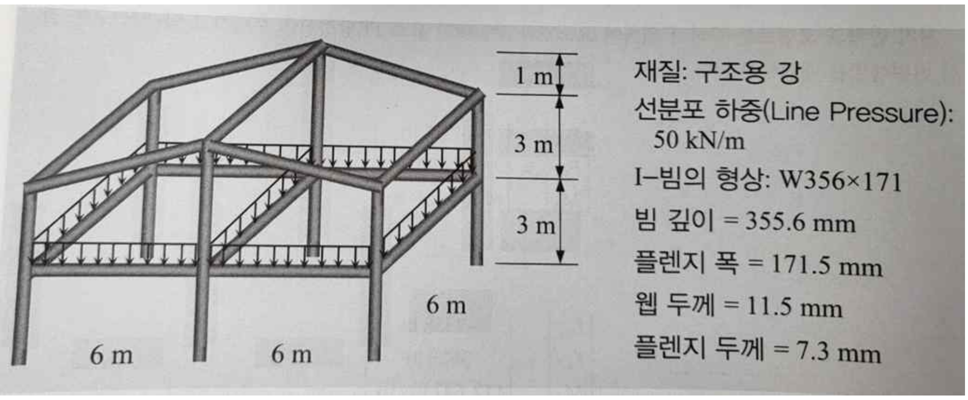
- Beam Results --> Axial Force (Others terms not activated in Link/Truss Mode)
    - Axial stress can be calculated: axial force / cross section area



(9) Further Studies

- Change the right support to pin support and compare the results
    - directional deformation (downward), axial force
- Select all line bodies and change to Truss/Link to Beam (welded joints)
    - Compare directional deformation with the results of Truss/Link
    - Beam Tool: check axial stress, bending stress, max./min. combined stress

3-2. 3D Beam

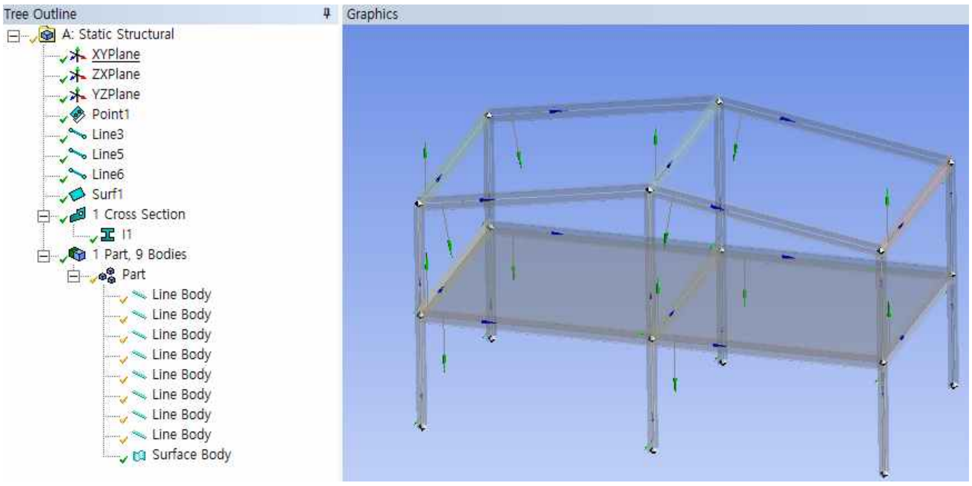


Feel the middle floor with a steel plate having 50 mm thickness

Go to Design Modeller

(1) Geometry

- Create Points: from a file (needs to be prepared)



- Concepts --> Line from Points connecting two nodes
    - "add frozen" option to separate line body into three groups
      (side beams, interfacing beams on top floor and on middle floor)
    - Click Generate Button after finishing line generation. Line bodies are generated.
- Concepts --> Cross section, I-beam

Details View

Details of I1	
Sketch	I1
Show Constraints?	No
Dimensions: 6	
W1	0.1715 m
W2	0.1715 m
W3	0.3556 m
t1	0.0073 m
t2	0.0073 m
t3	0.0115 m

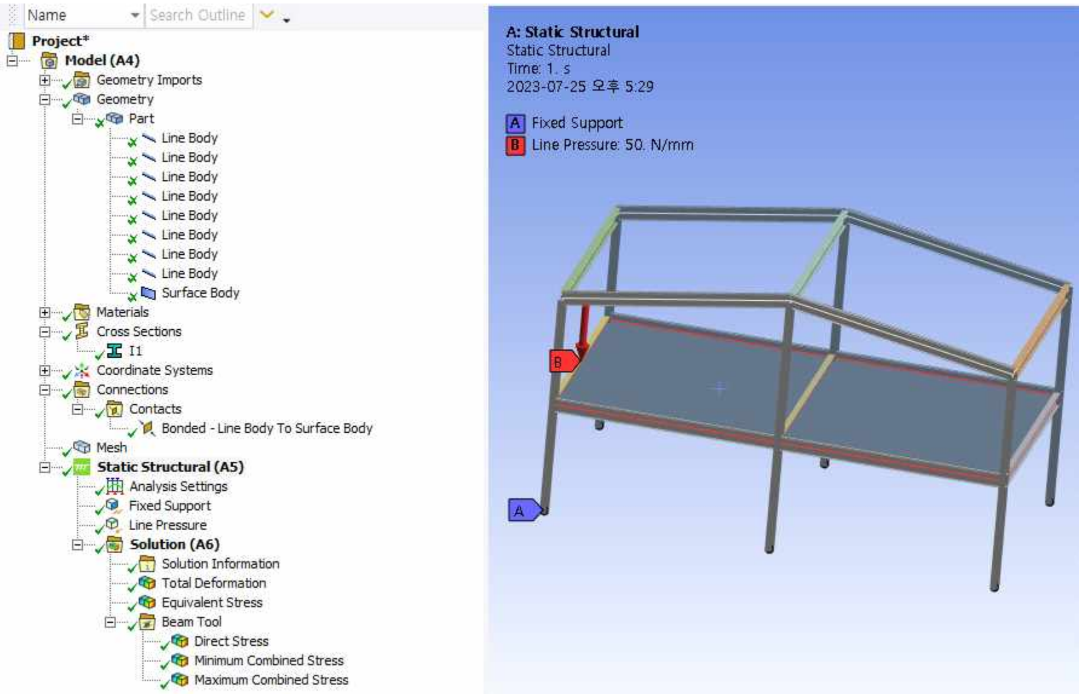
Details View

Details of Surface Body	
Body	Surface Body
Thickness Mode	Inherited
Thickness (>=0)	0.05 m
Surface Area	72 m²
Faces	1
Edges	6
Vertices	6
Fluid/Solid	Solid
Shared Topology Method	Default
Geometry Type	DesignModeler

- Need to change beam orientation in each line body if not correct
    - Select line body
    - RMB, select unaligned line edges and give rotation angle appropriately
- Concepts --> Surface from edges

- Select 6 edges in the middle floor (set Thickness = 0.05 m)
- Form a new part by selecting all line bodies and a surface body

Go to Static Structural



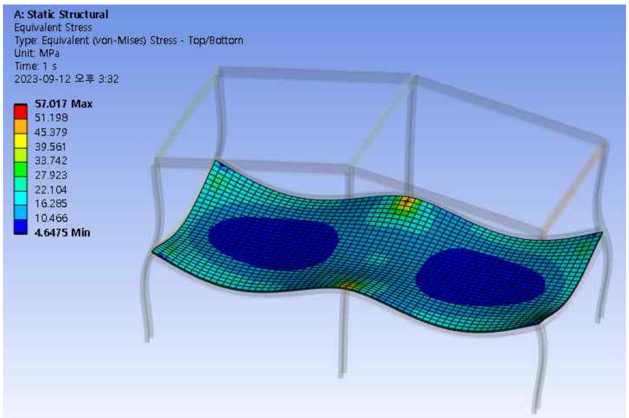
(2) Mesh
- Size: Division 10 (not important)

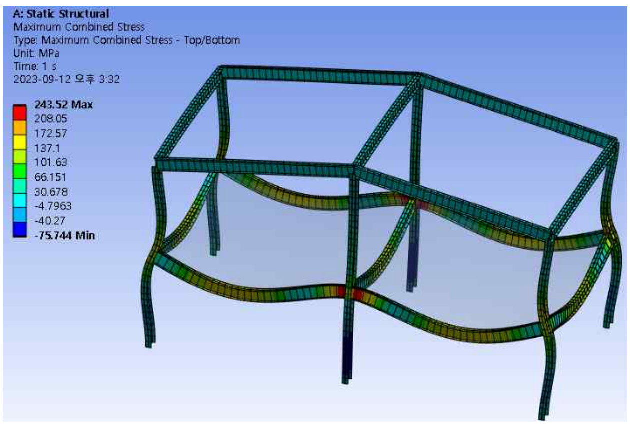
(3) Contact
- Bonded: Line body at the middle of the surface(Contact) to Surface body(Target)

(4) Boundary Condition
- 6 vertex at bottom: Fixed Support

(5) Load Condition
-Line Pressure: 50KN/m (50 N/mm) on all I-beam edges in the middle floor

(5) Results
- Deformation, Equivalent Stress
- Beam Tool “Max. Combined Stress





<3D 빔 문제의 몇가지 이슈 정리>

1. Plate(Shell 요소)의 집중응력

메쉬수를 증가시키면 응력이 계속 커진다. 이것은 종류가 다른 두 요소 접합부의 Singularity 현상으로 해석상에는 피할 수 없으나 실제 재료에서는 일어나지 않는다. 그 이유는,

- 실제로 접합부위는 적절하게 라운딩 되어 있어서 샤프코너와 같은 심각한 응력집중은 안일어난다
- 응력집중부의 국부 항복으로 인해 계속 응력이 증가할 수 없음

2. 빔의 최대응력

요소수를 늘이면 응력이 조금씩 증가한다. 이것은 Shear force가 증가하기 때문이다.(굽힘모멘트는 거의 안변함) 오일러 빔 말고 티문센코 빔의 경우, 전단변형이 굽힘응력을 올리는 효과가 있다. 앤시스의 빔요소는 티문센코 빔을 채택하고 있다. 한편 변위(deformation)와 축응력(direct stress)은 거의 안바뀐다.

3. 수렴 판단 기준

응력의 경우는 요소 크기를 반으로 줄였을 때, 응력변화가 5% 이내이면 수렴으로 보고(exact solution과 10% 이내 차이 예상), 변위의 경우는 1% 이내 변화이면 수렴으로 보면 된다. 이 문제에서는 요소 크기 50mm가 적정 크기로 판단된다.

- 100mm, 최대굽힘응력 268MPa --> 50mm, 278MPa --> 25mm, 281MPa
- 25mm부터는 학생버전 해석 불가, 학생버전에서는 요수수 + 절점수가 128K를 넘으면 해석이 안된다

4. 요소의 차수

Program controlled, Linear, Quadratic 선택 가능함.

이 문제에서는 Program controlled 인데, Linear로 처리 되고 있음을 알 수 있다.(옵션을 Linear로 바꾸어도 요소수가 같음) Quadratic으로 차수를 높이면 절점수는 2배 이상 증가하며 같은 요소 크기임에도 더 빨리 수렴응력값에 다다른다. 정확도를 높이려면 차수를 높이는 것과 요수수를 증가시키는 방법이 있으며 일장일단이 있다. 일반적인 문제에서는 디폴트 값인 Program controlled로 놓고 신경쓰지 말 것,

5. 경계조건

Fixed --> Displacement와 Rotation으로 분리 처리 가능. 이렇게 처리하면 다양한 경계조건을 처리 할 수 있다. 빔요소의 경우는, 점점에서 자유도가 6개임.

English trnaslation of the previous page
<Summary of some issues in the 3D beam problem>

1. Stress Concentration in the Plate (Shell Element)

If the number of elements increases, the stress will continue to increase. This is a phenomenon of singularity at the junction of two different types of elements, which is unavoidable in theory and numerical analysis but does not occur in real materials. The reasons are as follows.

- In fact, the joint is properly rounded, so the severe stress concentration like a sharp corner does not occur
- Due to the local yielding of the stress concentration part, the stress cannot continue to increase

2. Highest Stress in Beam Element

When the number of mesh is increased, the stress also increases little by little. This is due to an increase in shear force. In the case of the Timunsenko beam instead of the Euler beam, the shear deformation has the effect of increasing the bending stress. ANSYS beam element employs a Timunshenko beam. On the other hand, deformation and direct stress rarely change with increase of the number of elements.

3. Convergence Criteria

When the element size is reduced by half, if the stress change is within 5%, it is considered convergence (expected to differ within 10% from the exact solution), and in the case of displacement, it is considered convergence if the change is within 1%. In this problem, the element size of 50 mm is considered to be the appropriate size.

- 100mm, maximum bending stress 268MPa --> 50mm, 278MPa --> 25mm, 281MPa
- From 25mm, the student version can not run. The student version allows the number of element + the number of nodes up to 128,000.

4. Order of Elements

Program controlled, linear, and quadratic selectable.

In this problem, it is program controlled, and you can see that it is handled by linear. (Even if you change the option to linear, the number of elements is the same) If the order is increased by quadratic, the node number is more than doubled, and the convergent stress value is reached faster even though it is the same element size. To improve accuracy, there are two ways: increase the number of degrees and or increase the number of elements. For general problems, leave it to program controlled, which is the default.

5. Boundary Conditions

Fixed --> can be separated by displacement and rotation BC. In this way, various boundary conditions can be handled. In the case of beam elements, there are 6 degrees of freedom at each node.

3-3. Home Works: Gantry Crane

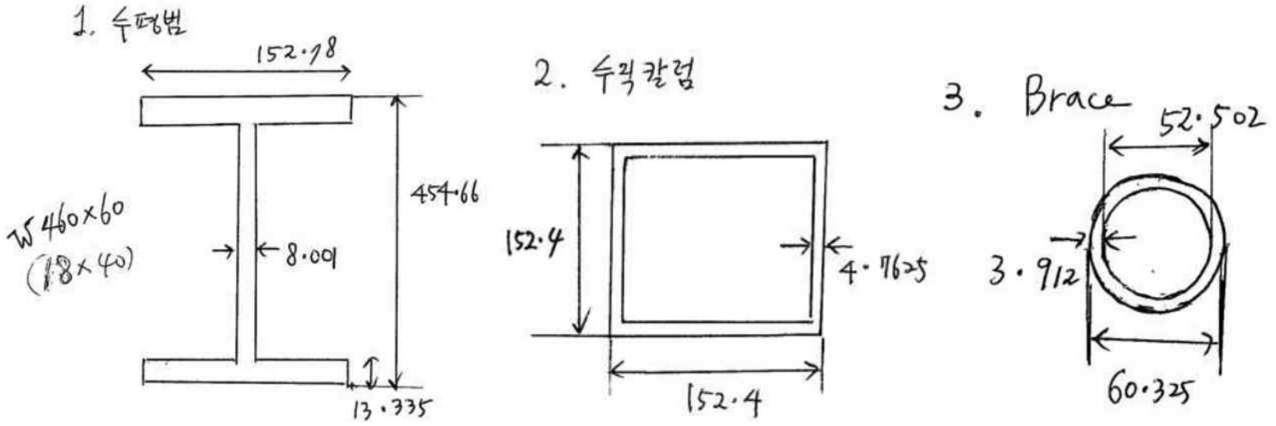
(Use SI unit by conversion)

A gantry crane is designed that must be able to lift 10 tons(use 100KN) as it must lift compressors, motors, heat exchangers, and controls. This load should be placed at the center of one of the main 12-ft-long beams(use 1ft=0.3m) as shown below by the hoisting device location. Weight of the structure is ignored in the analysis. Assume you are using ASTM A36 structural steel(SS400). The crane must be 12 feet long, 8 feet wide, and 15 feet high. The beams should all be the same size, the columns all the same size, and the bracing all the same size. Their cross sections are selected from Appedix F(4th ed.) and shown below. You must verify that the structure is safe by checking the beam's bending strength and allowable deflection. A required safety factor against material yielding of the beam is 3. Verify that the beam deflection is less than L/360 (12/360ft=10mm, downward deflection of the beam center with respect to the ground), where L is the span of the beam. Check yielding and Euler buckling of the long columns. A required factor of safety is 3 against yielding of the column and 5 against buckling of the column.(Ignore local buckling of the horizontal beam) Assume the column-to-beam joints to be rigid while the bracing (a total of eight braces) is pinned to the column and beam at each of the four corners. Use appropriate boundary conditions for the four supports of the gantry crane.

1. Generate an ANSYS beam model and get safety factors and deflections
2. Compare simulation results with appropriate theoretical results (stresses and downward deflections of the horizontal beam, critical buckling load of the columns)
3. Discuss on boundary conditions of the four supports
4. What do you think the function of the braces are?
5. How many number of elements is required to accurately solve this problem?

Guideline for Solution:

1. Selected cross sections of columns, beams and bracings are shown below (Apenndix F)



2. Do not use values of area(A) and moment of inertia(I) appearing in the appendix F. You need to calculate them from dimensions of the cross section given above. We will not consider roundings on the cross section which makes the problem too complex.

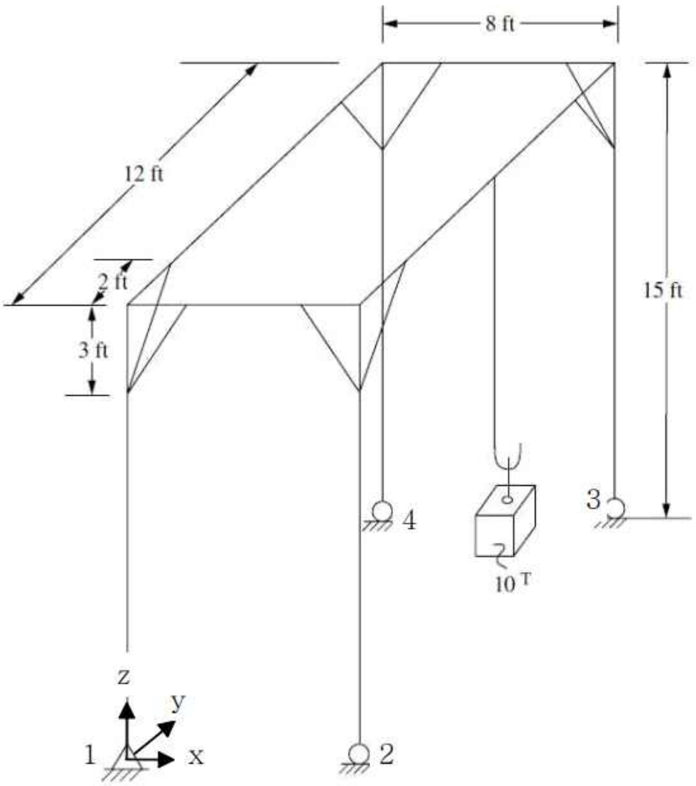


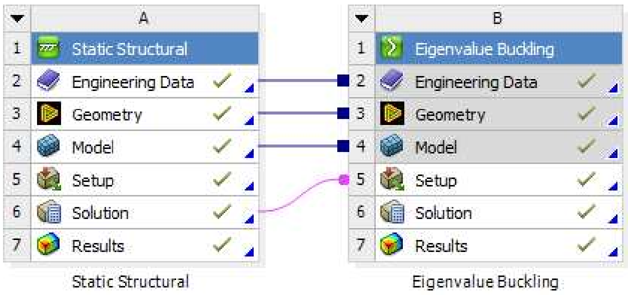
Figure P5–70

Problem Given

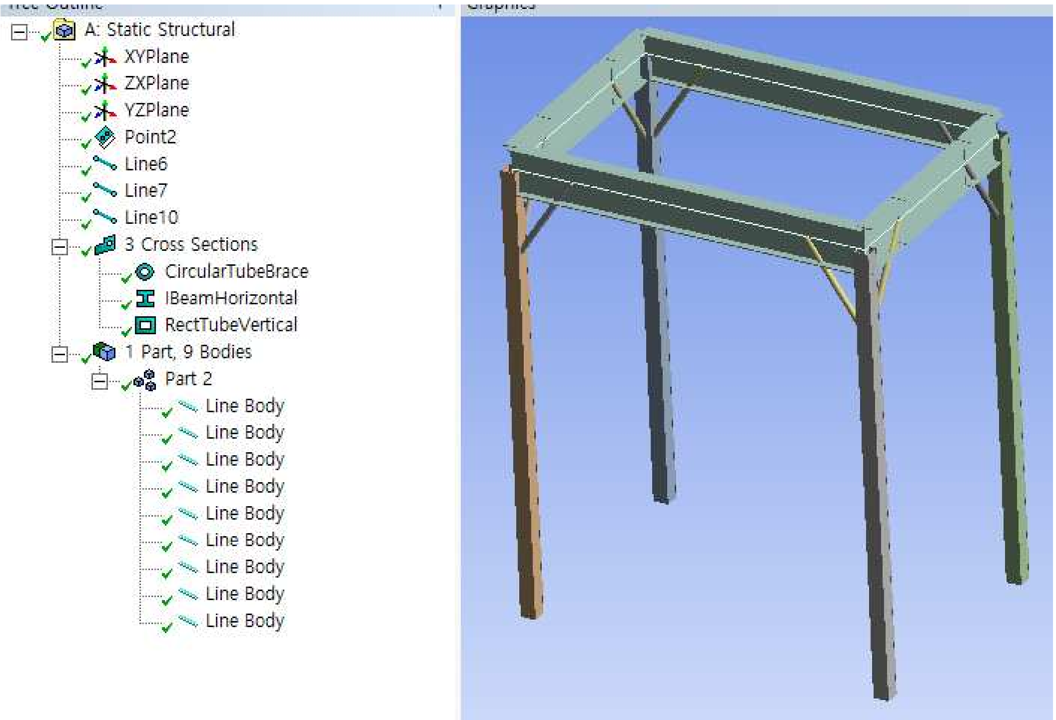
Illustration of A Gantry Crane

Hint:

Work Bench



Go to Design Modeller



(1) Generate nodal points

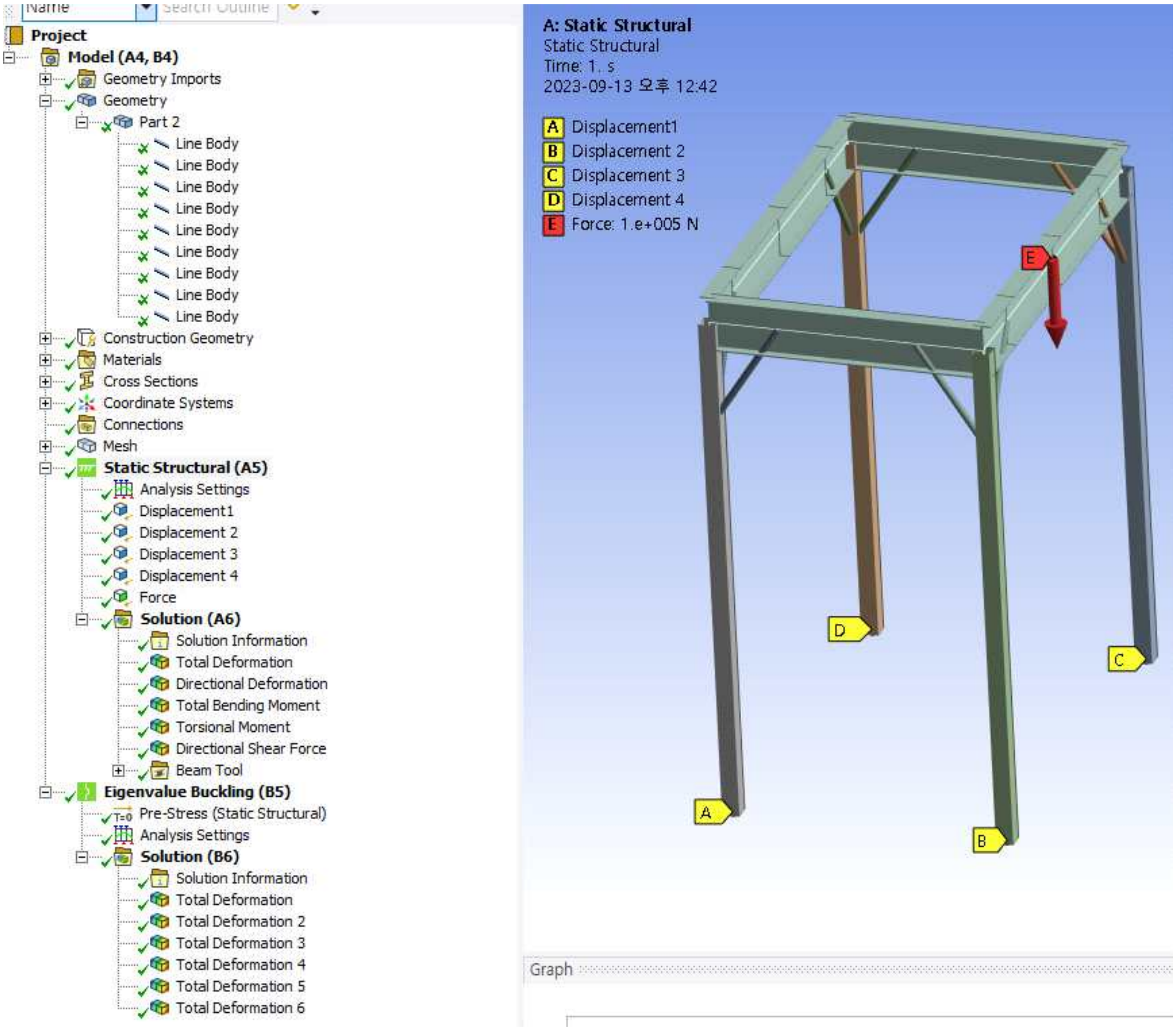
You had better use a pre-made coordinate file as below (22 points required)

- 1 1 0 0 0
- 1 2 2.4 0 0
- 1 3 0 3.6 0
- 1 4 2.4 3.6 0
- ....

(2) Generate lines by connecting nodes by using “Add Frozen” option

- Line6: 4 Horizontal Beam, Line7: 4 Columns, Line10: 8 Braces
- Generate three different cross sections(CS) and assign an appropriate CS to each line body
- Be careful on I-beam orientations
- Form a new part to assemble all the line elements

Go to Static Structural



(1) Boundary Conditions

- displacement 1, 2, 3, 4 --> x=free, y=z=0 (reflecting rail boundary)

Details of "Displacement1"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	0. mm (ramped)
Z Component	0. mm (ramped)
Suppressed	No

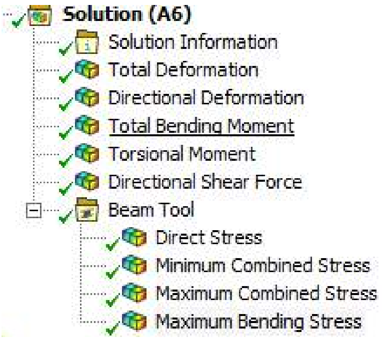
(2) Load Condition

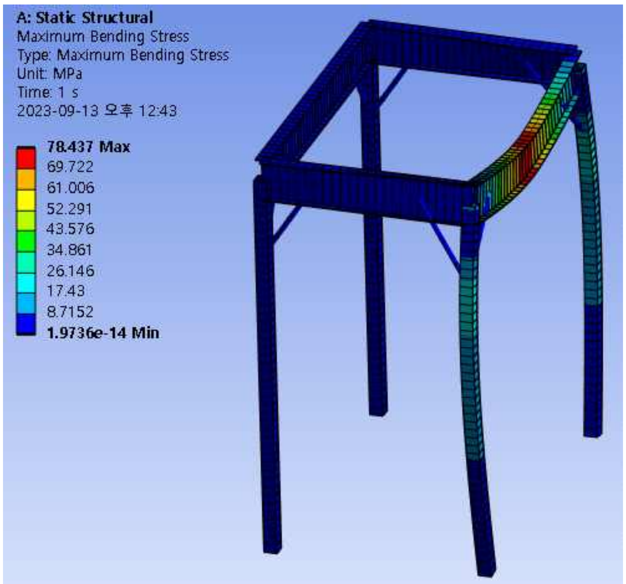
- Force: 100,000N (z-direction)

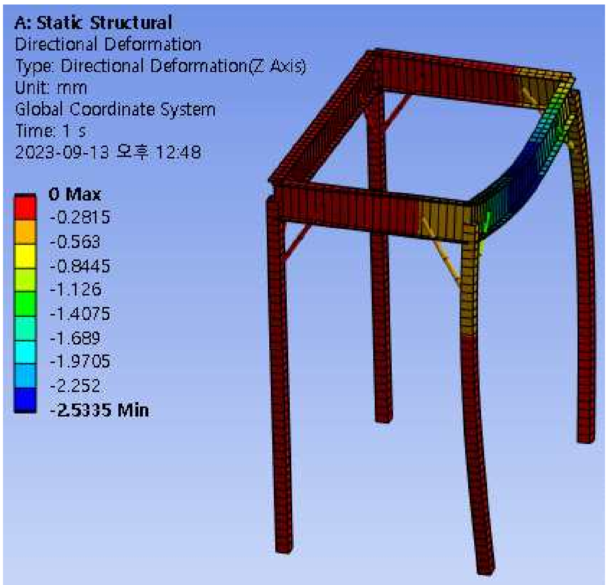
(3) Static Structural

- Analysis Settings: Weak Springs --> Program Controlled
  (Rigid body motion prevented)

(4) Results: Von-Mises Equivalent Stress

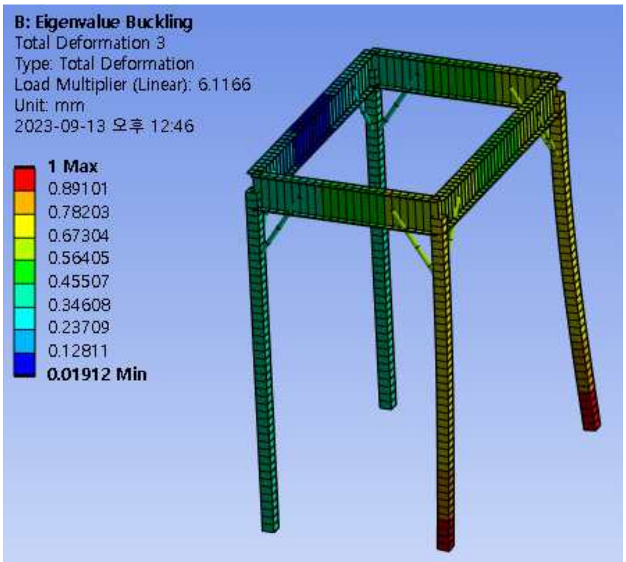






Go to Eigenvalue Buckling

- Load Multiplier and Mode Shape

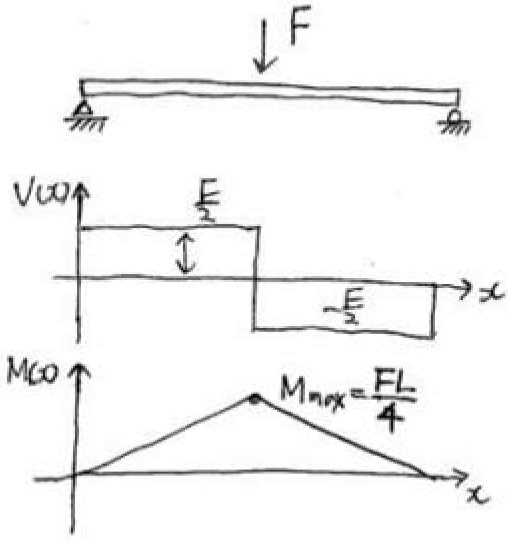


Tabular Data

	Mode	Load Multiplier
1	1.	-4.2234
2	2.	4.1996
3	3.	6.1166
4	4.	6.2674
5	5.	6.4283
6	6.	8.369

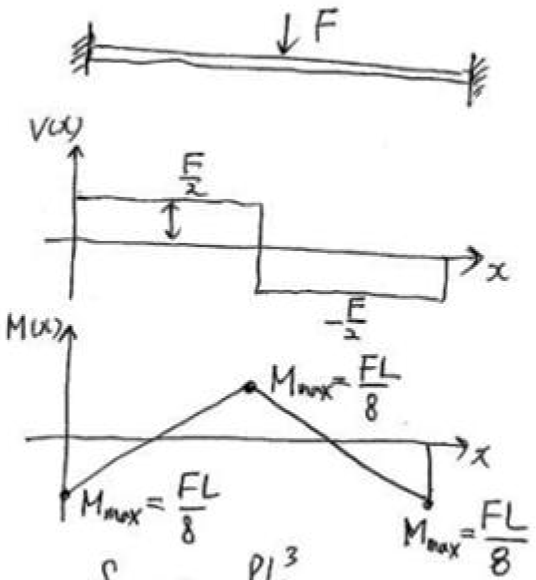
<참고 자료> 빔의 모멘트선도와 굽힘응력과 처짐식 (단순지지, 양단고정)

단순지지보



$$\delta_{max} = \frac{PL^3}{48EI}$$

양단고정보



$$\delta_{max} = \frac{PL^3}{192EI}$$

분포하중인 경우 (ω N/m)

단순지지보 ;

$$M_{max} = \frac{\omega L^2}{8}$$

$$\delta_{max} = \frac{5\omega L^4}{384}$$

양단고정보 ;

$$M_{max} = \frac{\omega L^2}{12} \text{ at both ends}$$

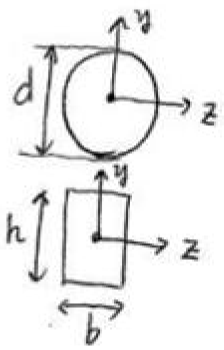
$$= \frac{\omega L^2}{24} \text{ at the middle}$$

$$\delta_{max} = \frac{\omega L^4}{384}$$

최대응력, $\sigma_b = \frac{Mc}{I}$

$I = \frac{\pi d^4}{64}$ for 원형단면, $c = \frac{d}{2}$

$I = \frac{bh^3}{12}$ for 직사각단면, $c = \frac{h}{2}$



$\sigma_b = \frac{32M}{\pi d^3}$ (원형단면)