Body Torsional Analysis

Computational Design Laboratory
Department of Automotive Engineering
Hanyang University, Seoul, Korea

OUTLINE

- **Lecture Goals**
  - ✓ Side frame모델(beam)의 유효전단강성과 passenger cabin모델(beam+shell)의 비틀림 강성을 계산하여 이론해와 비교한다.

- **Contents**
  - ✓ Body torsional stiffness analysis #1
    : Beam elements
  - ✓ Body torsional stiffness analysis #2
    : Shell + beam elements

Body Torsional Stiffness Analysis

Computational Design Laboratory
Department of Automotive Engineering
Hanyang University, Seoul, Korea

목차

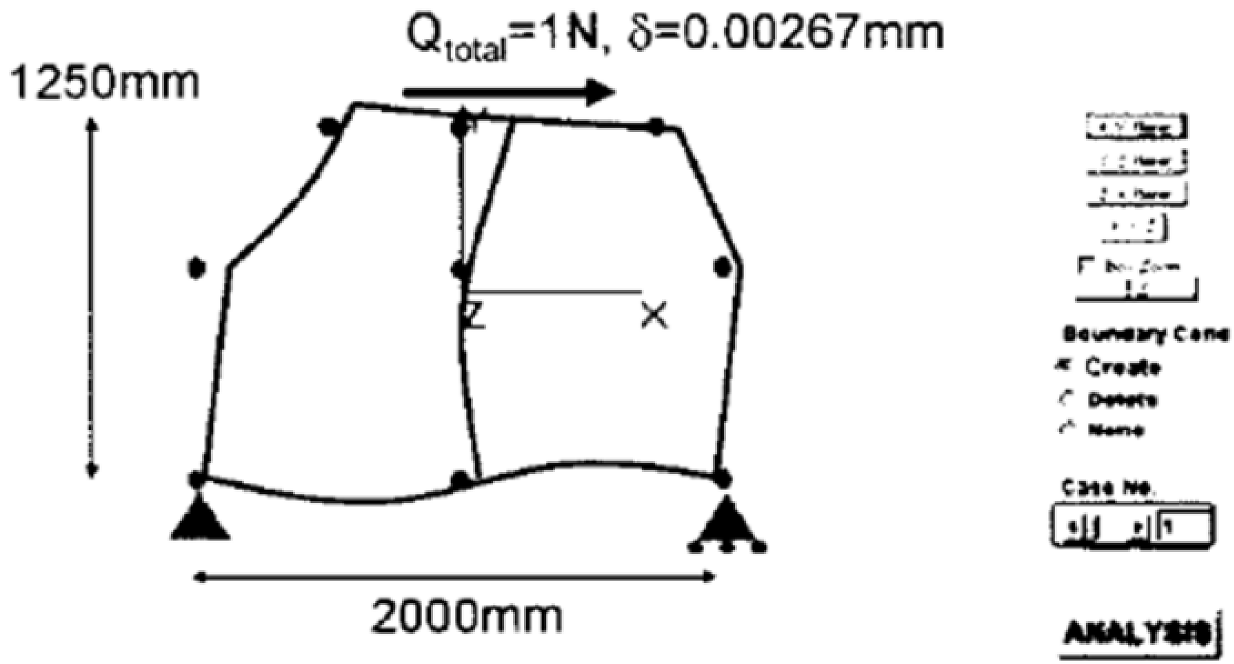
- 예제 문제
  - ➢ Effective shear stiffness analysis of side frame
  - ➢ Body torsional stiffness analysis
- 해석 프로세스
  - ➢ 기하형상 생성
  - ➢ 재료 물성 및 특성 입력
  - ➢ 요소망 생성
  - ➢ 구속조건 설정
  - ➢ 하중조건 설정
  - ➢ 해석케이스 정의 및 해석 실행
  - ➢ 후처리

EFFECTIVE SHEAR RIGIDITY (SIDE FRAME)

예제: TORSIONAL STIFFNESS (3)

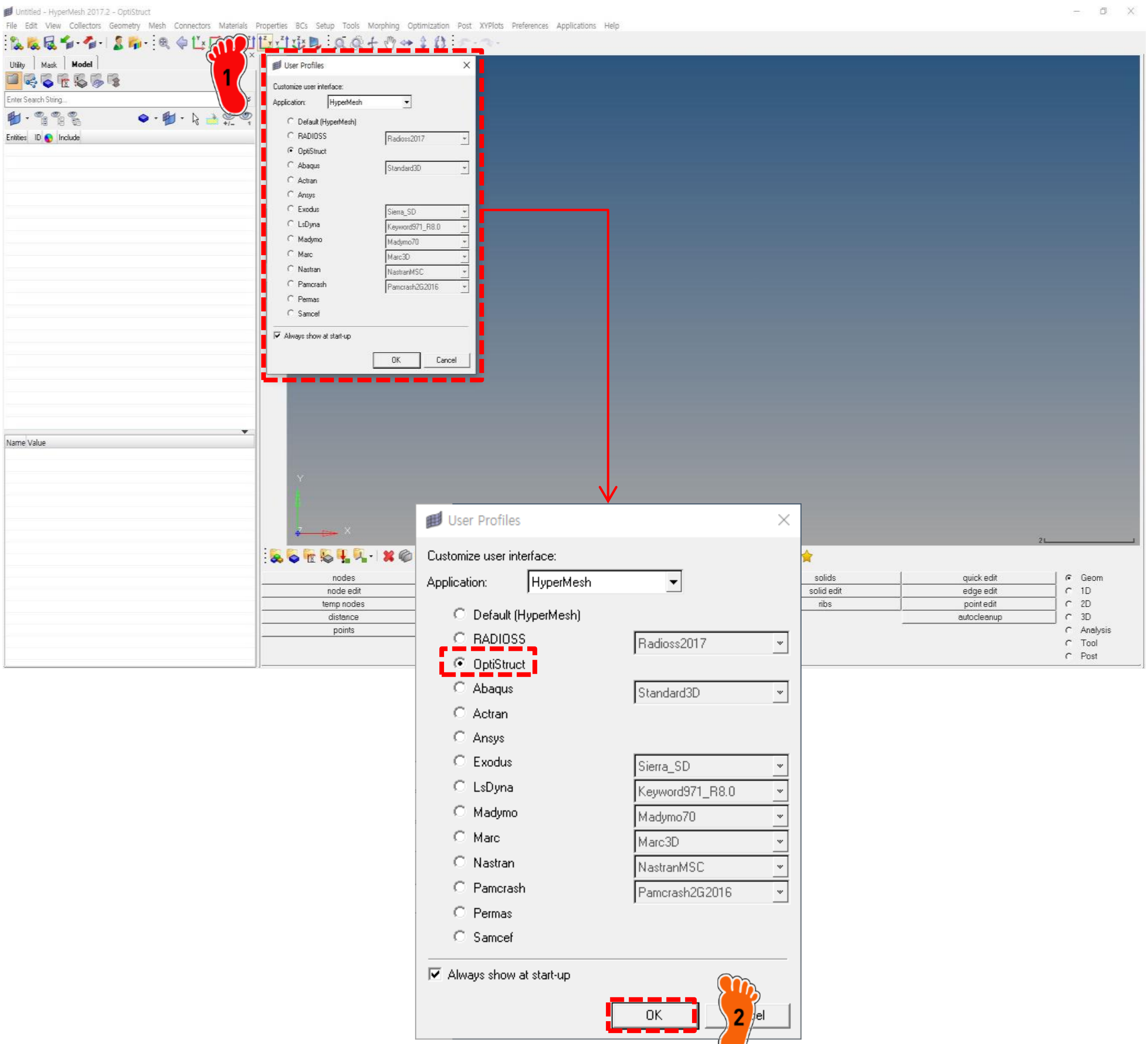
Side Frame Model 의 강성 계산

- FEA under shear loading



$$(Gt)_{Eff} = \left(\frac{Q}{\delta}\right)\frac{H}{L} = \left(\frac{1N}{.00267mm}\right)\frac{1250mm}{2000mm} = 234N/mm$$

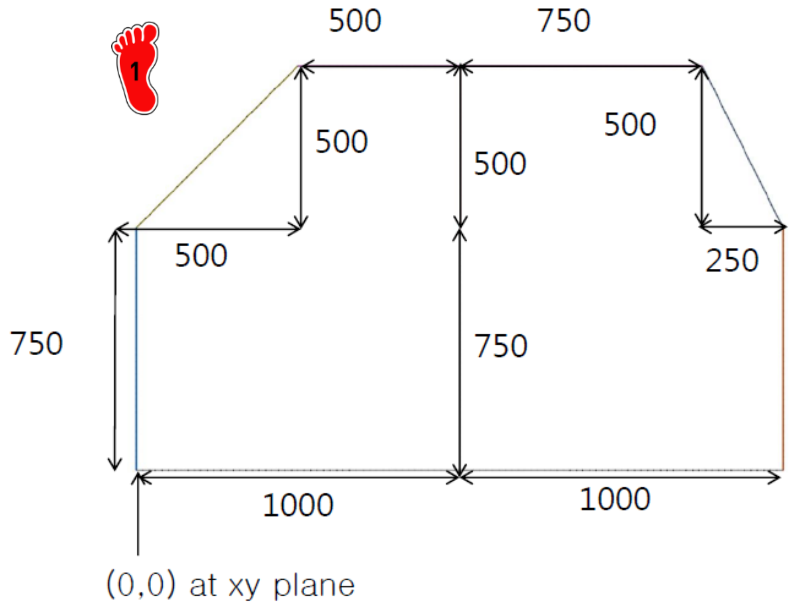
기하형상 생성 (1)

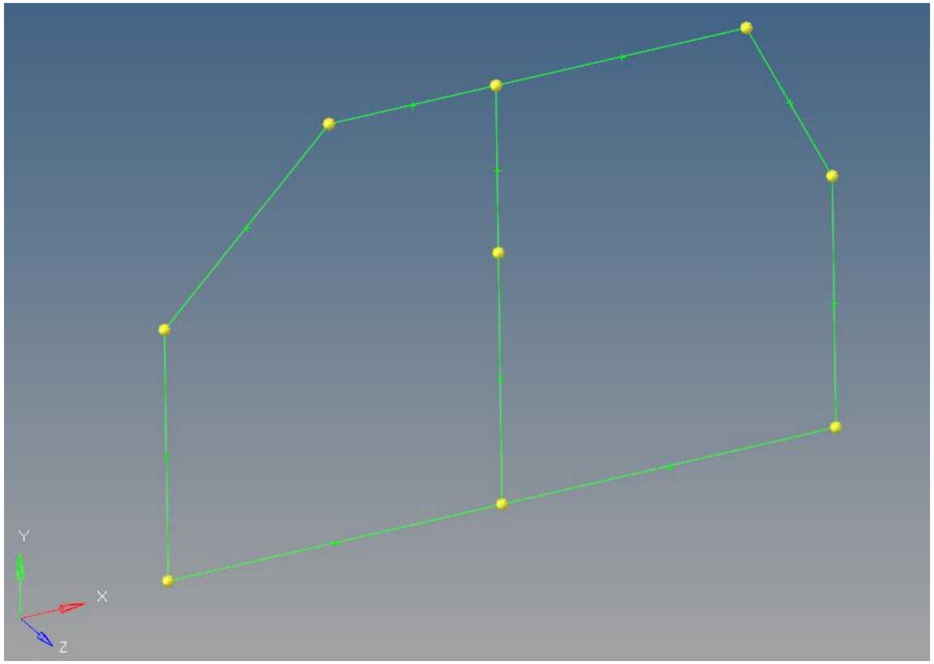


1 OptiStruct 선택

2 OK

기하형상 생성 (2)



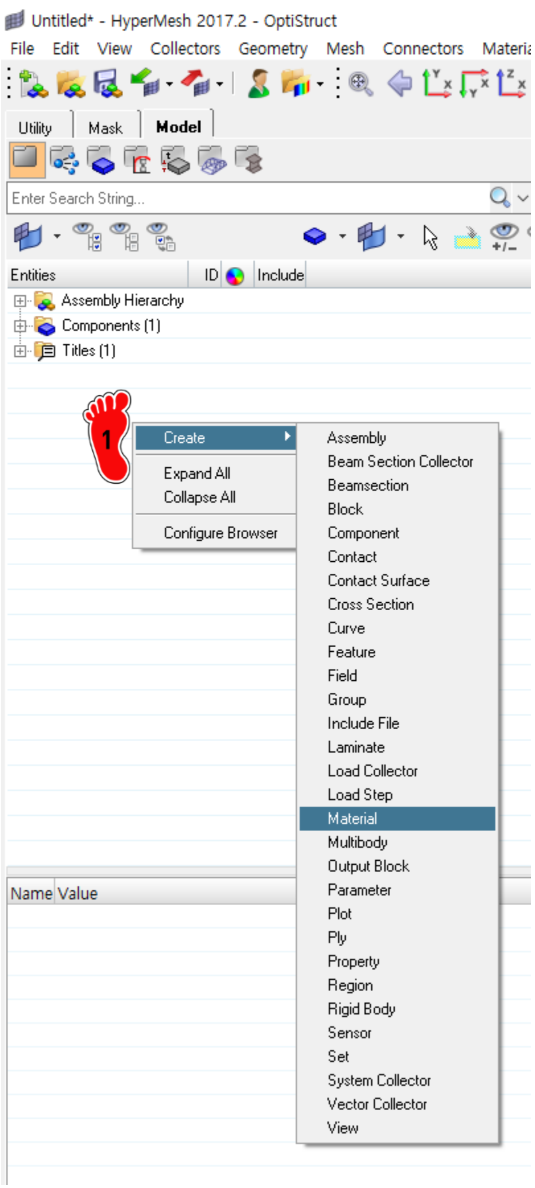


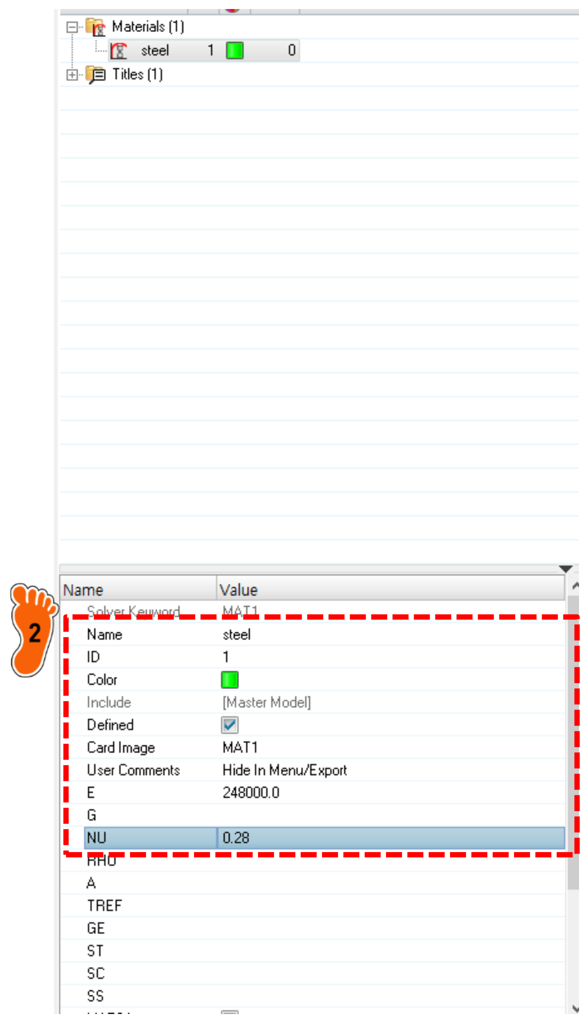
1 주어진 기하형상 생성

좌표(x,y,z)

(0,0,0)
(0,750,0)
(500,1250,0)
(1000,0,0)
(1000,750,0)
(1000,1250,0)
(1750,1250,0)
(2000,0,0)
(2000,750,0)

재료 물성 및 특성 입력 (1)

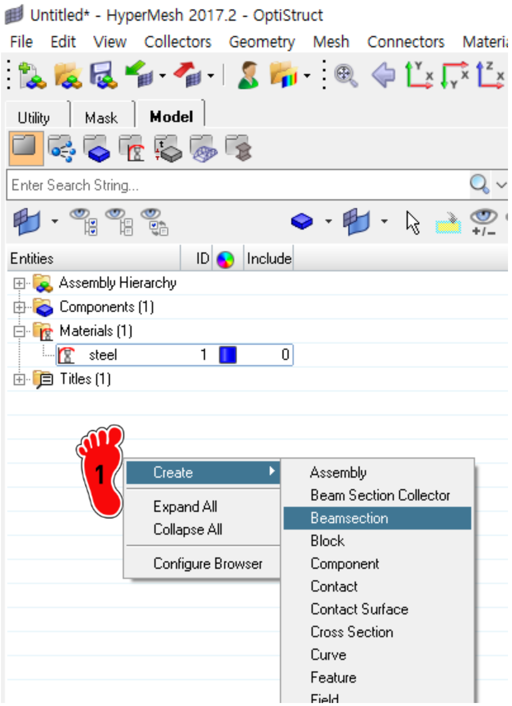


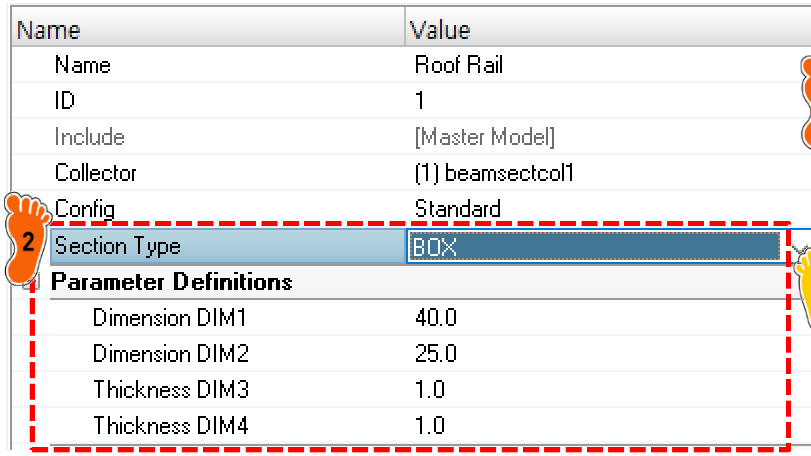


1 우클릭, Create > Material

2 Name > steel
탄성계수(E)
>248 Gpa (248000 N/mm^2)
푸아송비(NU) > 0.28
재료 생성

재료 물성 및 특성 입력 (2)

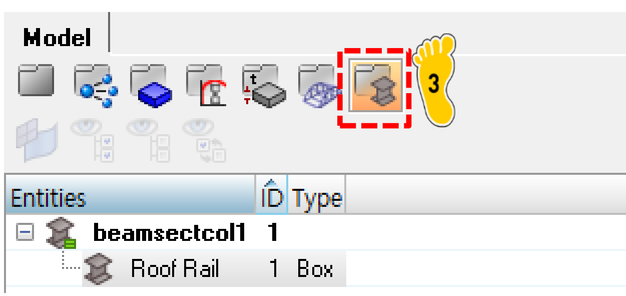


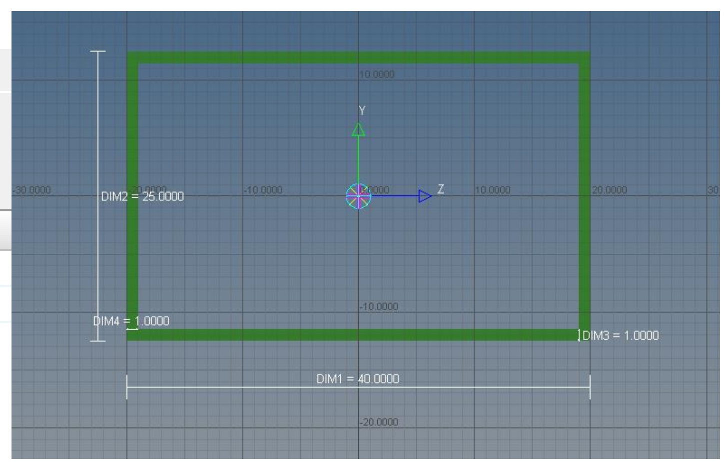


1 우클릭,
Create > Beamsection

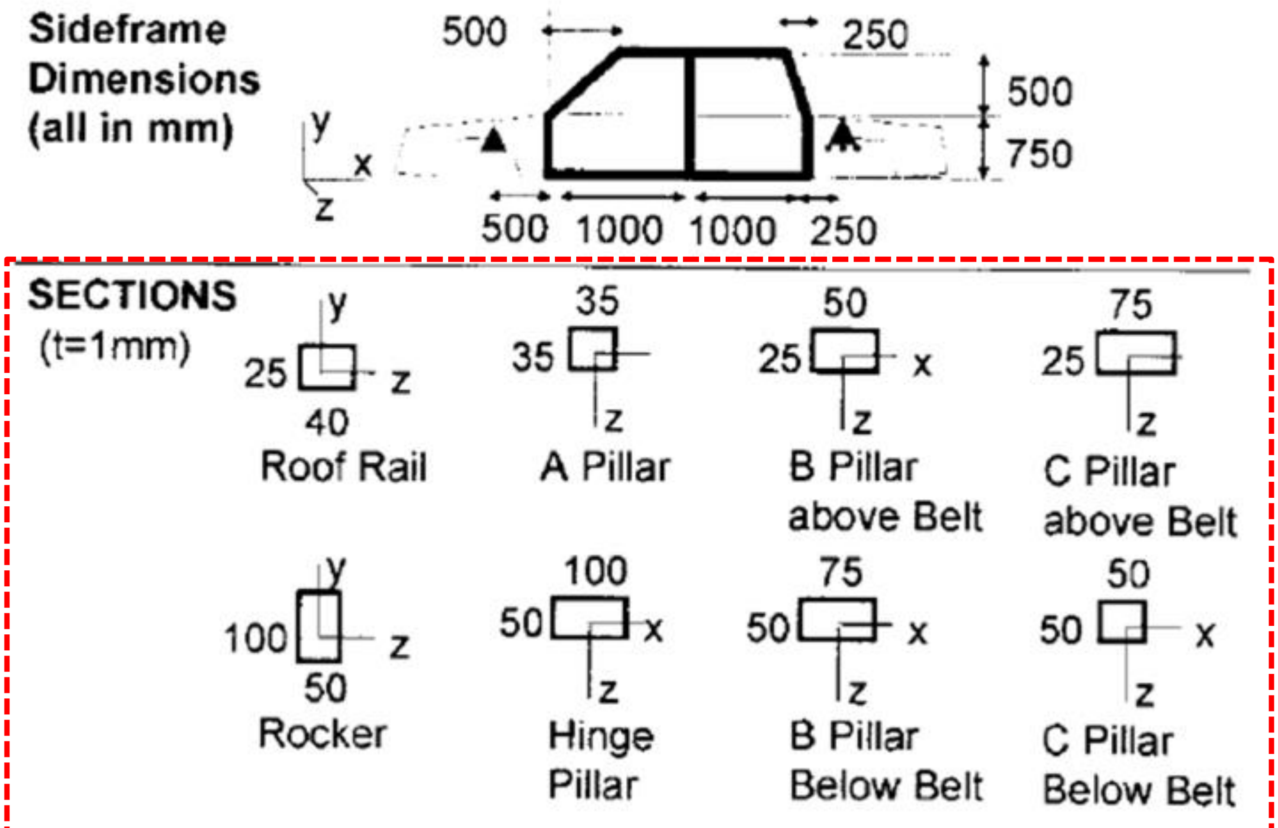
2 Section Type > BOX,
단면형상 입력

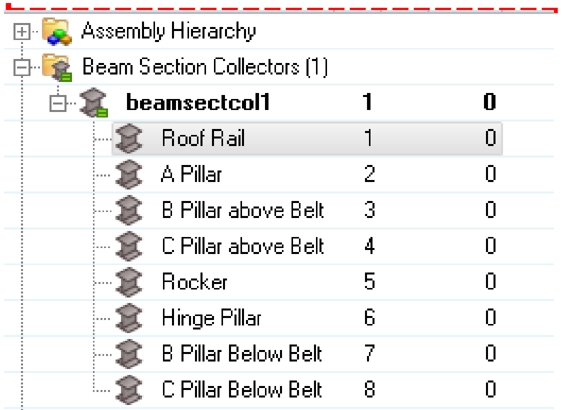
3 Hyperbeam view에서 단면
형상 확인 가능

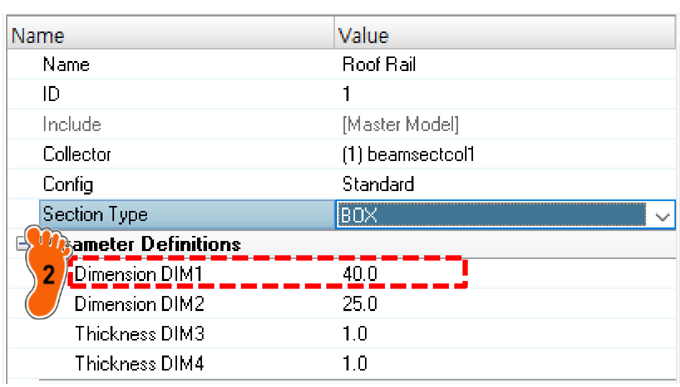




재료 물성 및 특성 입력 (3)



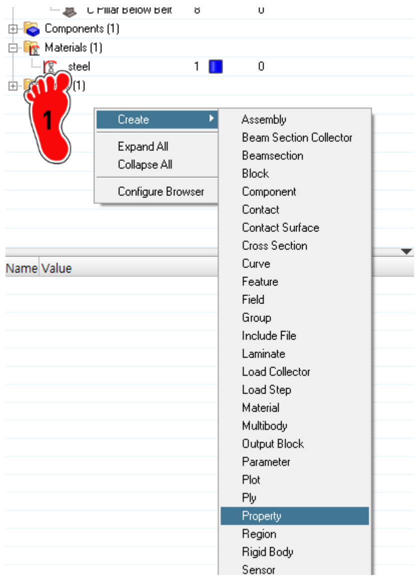


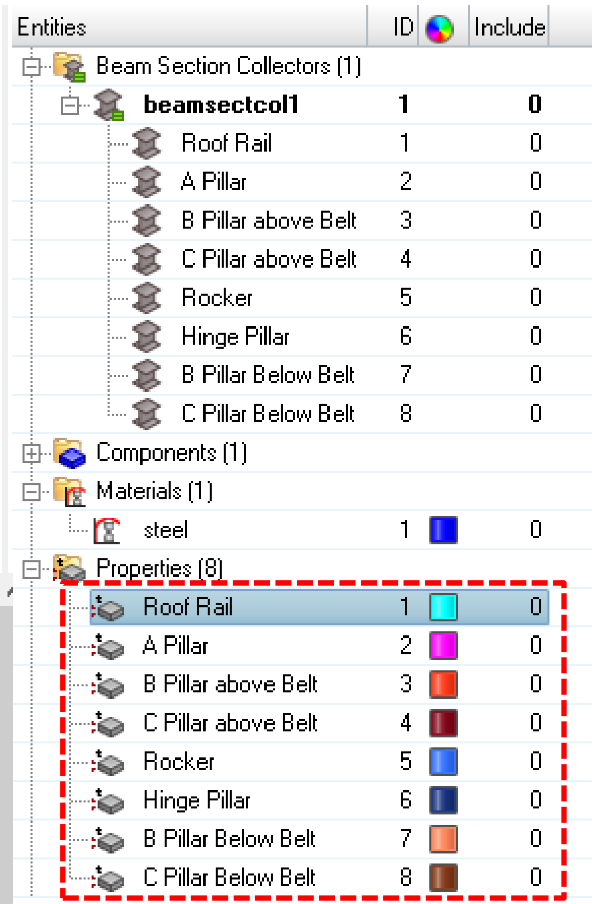


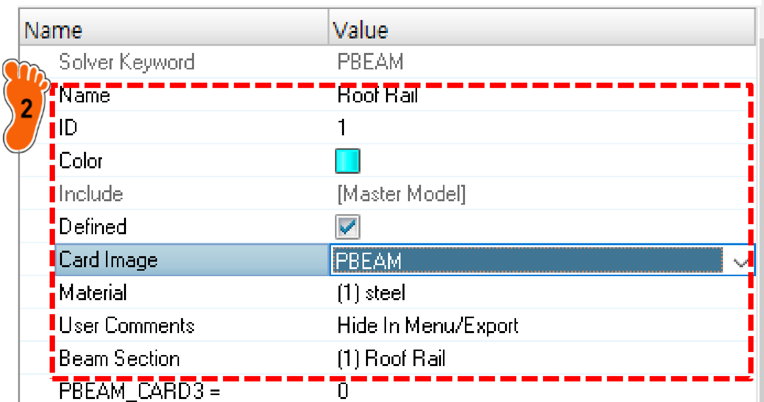
1 예제에서 주어진 단면형상 8개를 각각 생성

2 Dimension DIM1에 Z방향 길이 입력

재료 물성 및 특성 입력 (4)



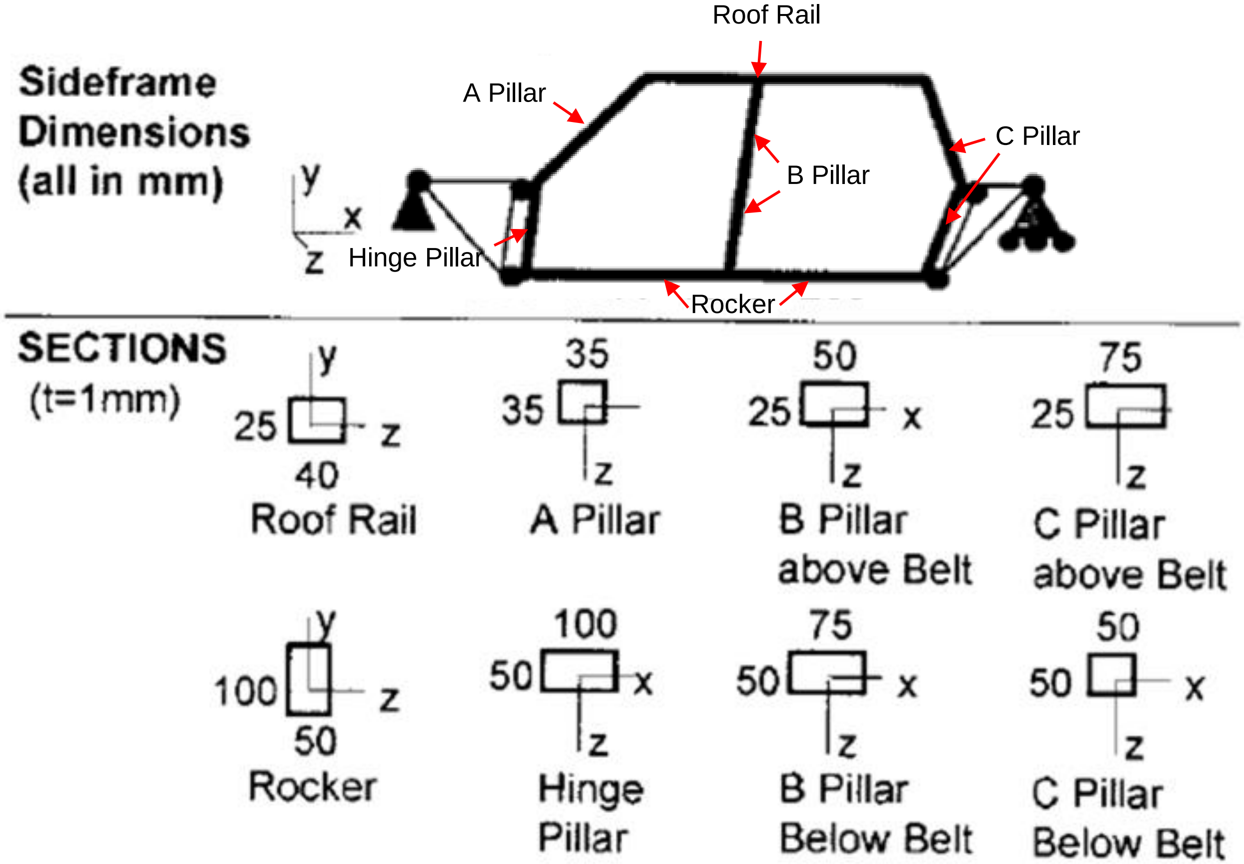


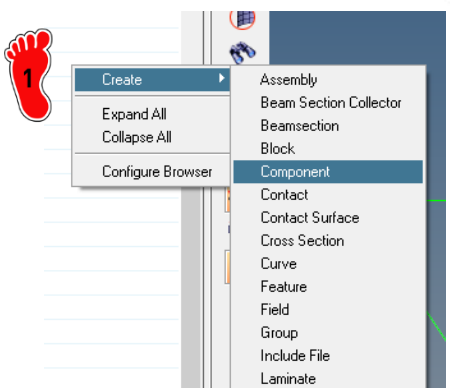


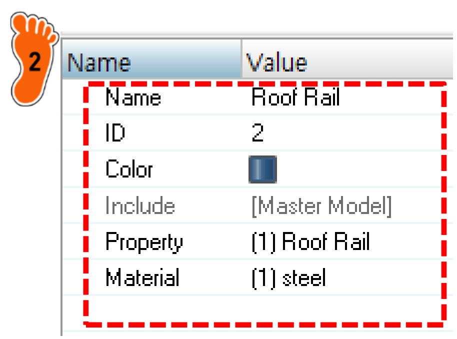
1. 우클릭,
Create > Property

2. Property 8개 각각 생성, 단면 형상과 각각 연결
Card Image > PBEAM
Material > steel
Beam Section > 단면형상 각각 선택
Name > 단면 형상과 동일하게

요소망 생성 (1)



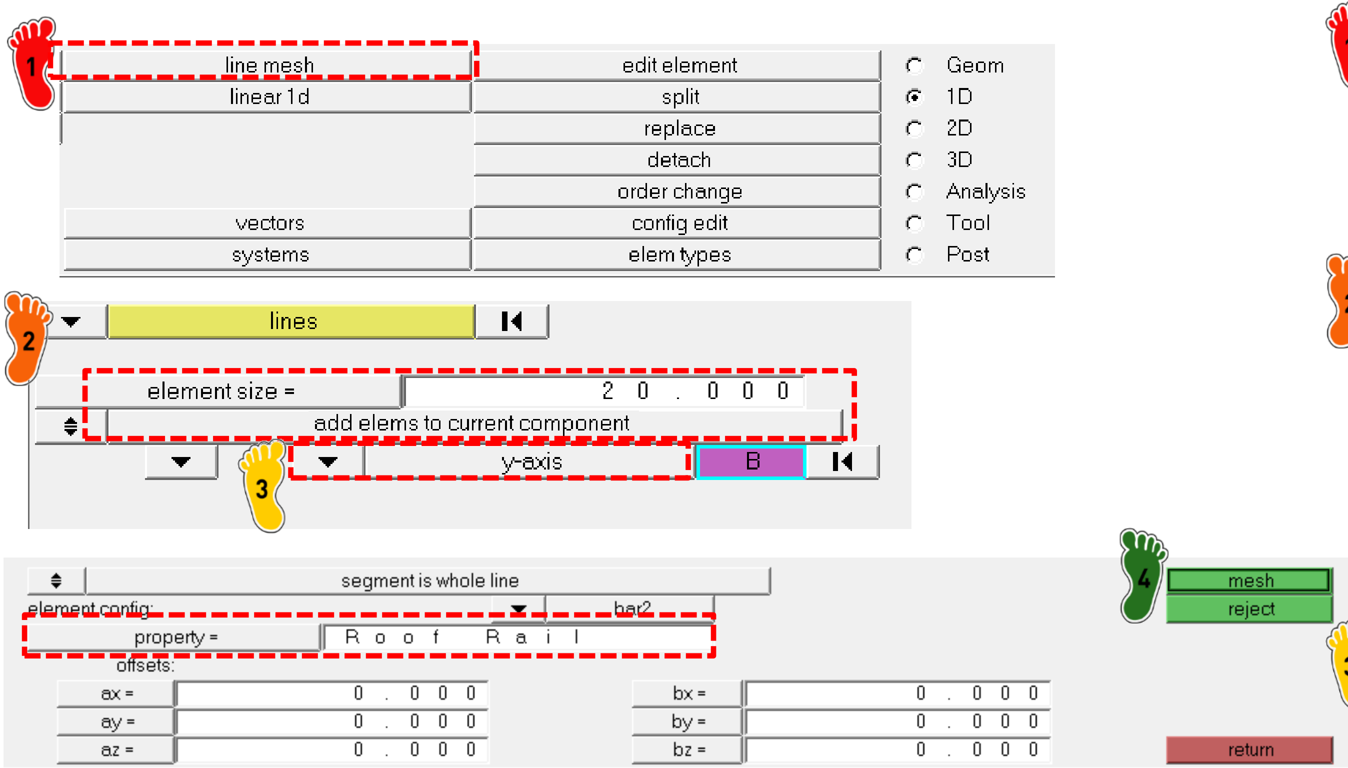


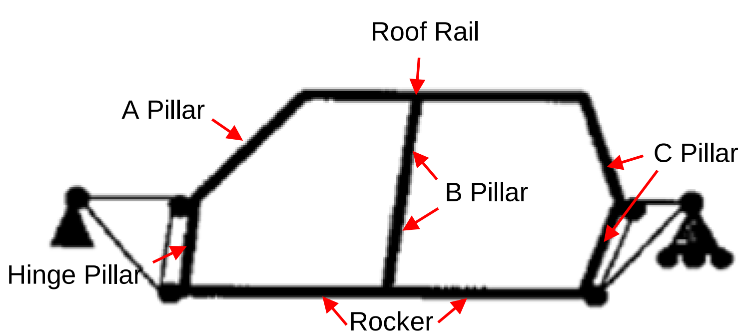


1 우클릭,
Create > Component

2 Name, Property
>요소망 생성할 단면형상의
이름과 동일하게

요소망 생성 (2)





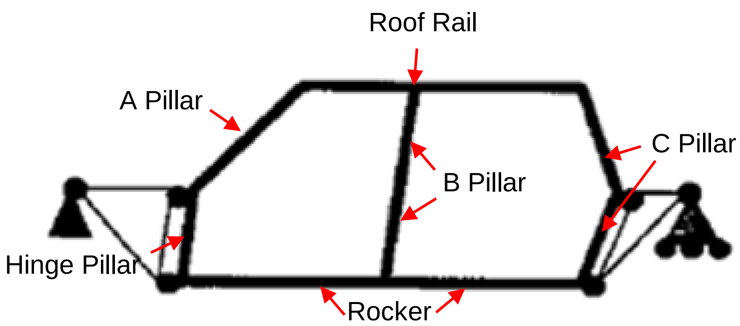
1. 1D > line mesh 클릭

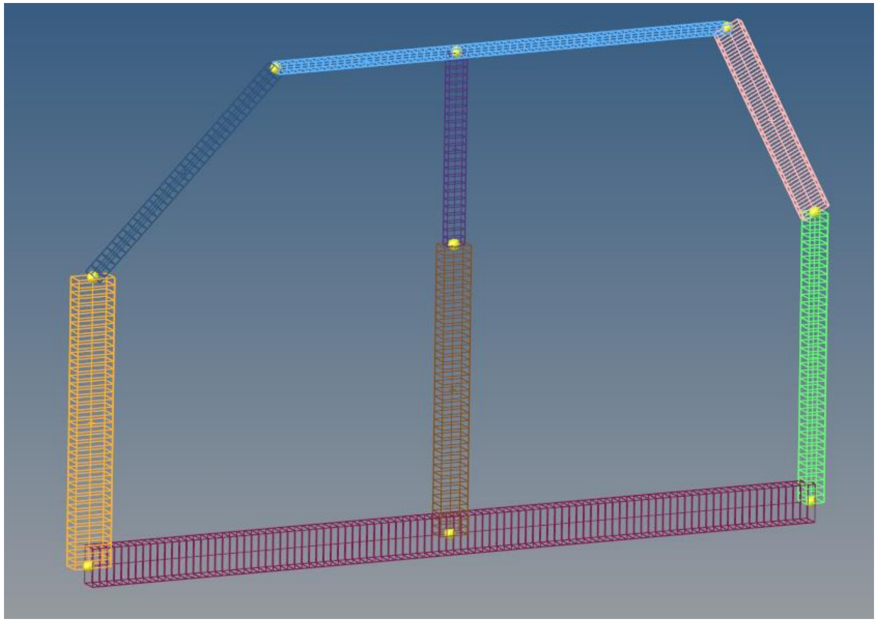
2. 요소망 생성
크기 > 20
Add elems to current component
Property > 단면형상 이름과 동일하게

3. 요소망 생성시 단면 형상이 원하는 좌표축에 놓이도록 주의(x축과 평행한 빔은 y-aixs로 선택, y축과 평행한 빔은 x-aixs로 선택)

4. Mesh,
Return

요소망 생성 (3)





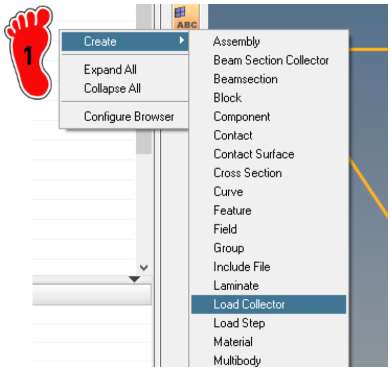
1 앞의 과정(1,2) 반복해서 8개 빔 요소망 생성

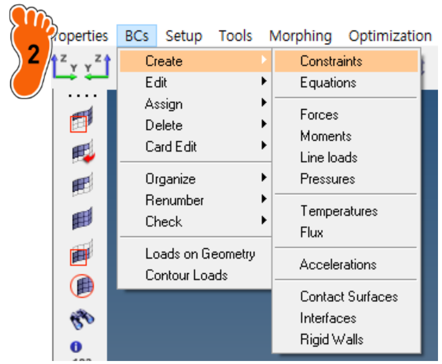
그림과 비슷한 형상이 나와야 함

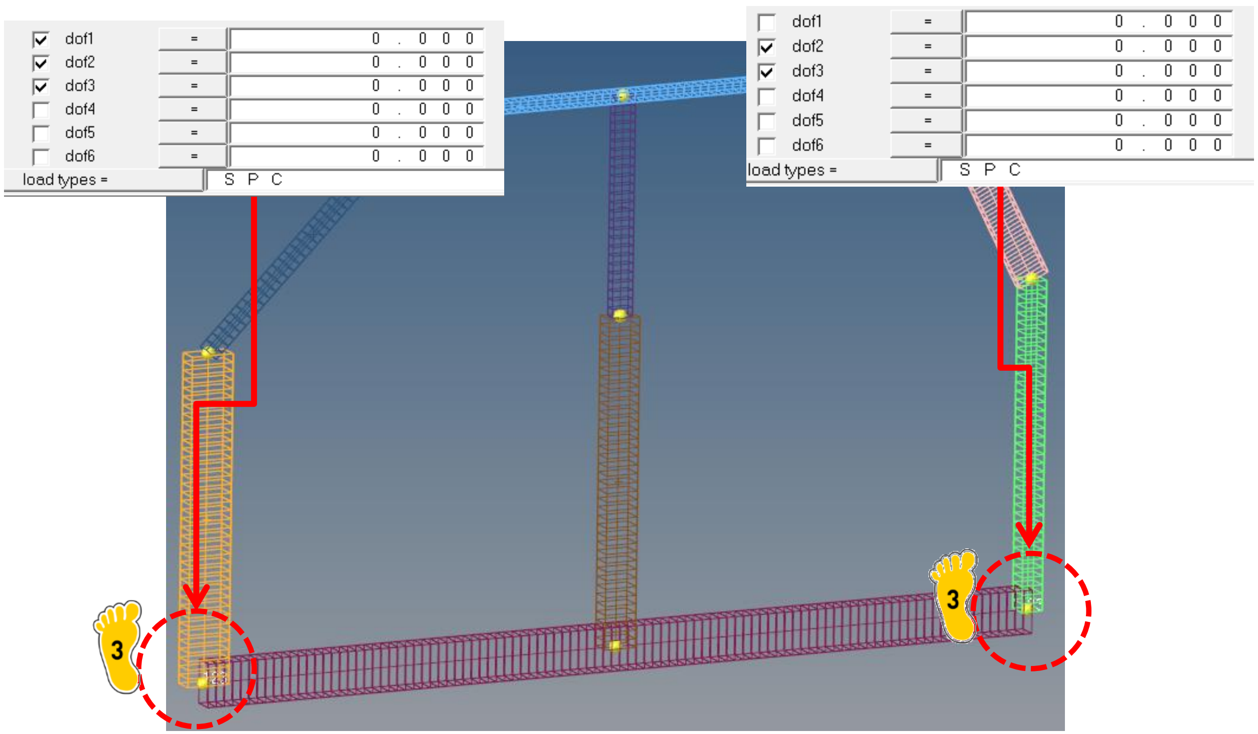
* Tool -> eadges -> tolerance -> equivalence (모서리 병합)

* Geom -> temp nodes -> clear all (Nodes 제거)

구속조건 및 하중조건 설정 (1)





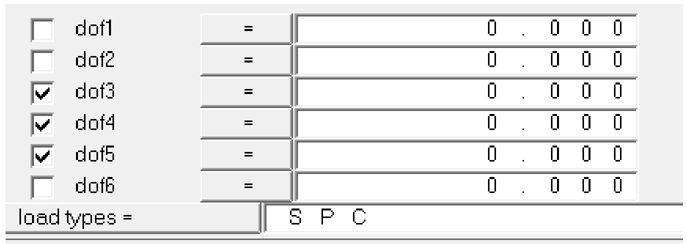


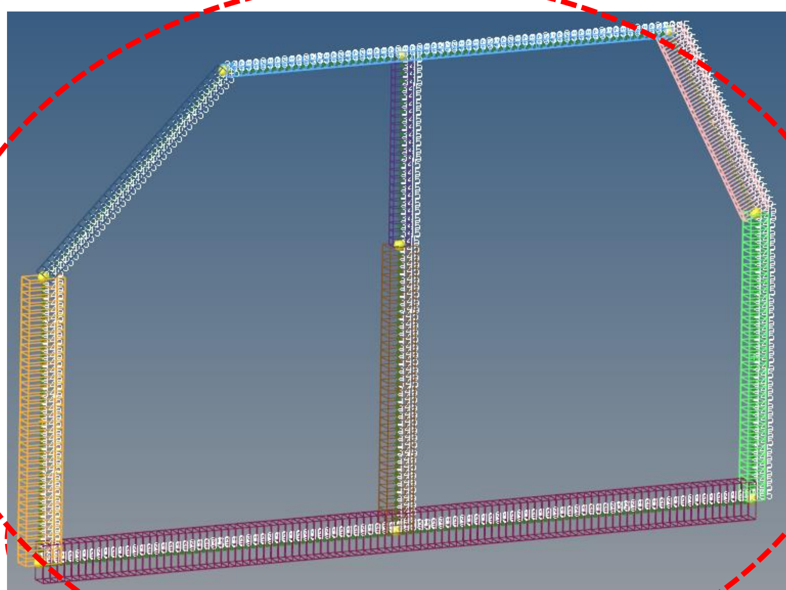
1 Create > Load Collector
Name > spc

2 BCs > Create
> Constraints

3 좌측 dof1,2,3 check
Create
우측 dof2,3 check
Create

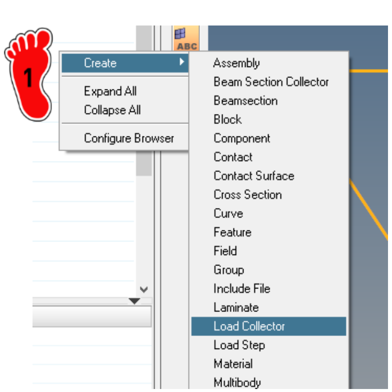
구속조건 및 하중조건 설정 (2)

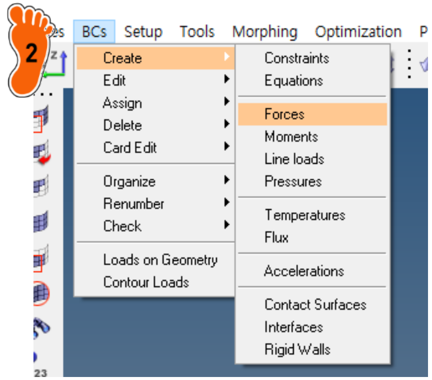


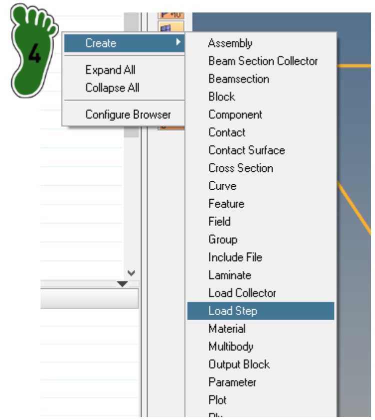


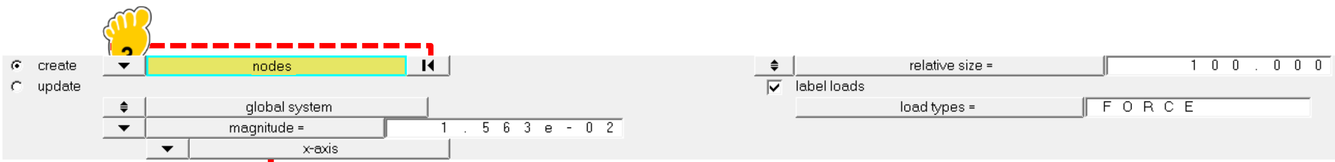
1 나머지는 평면 거동을 위해 z방향 병진운동, x,y 방향 회전운동 구속
(dof3,4,5)

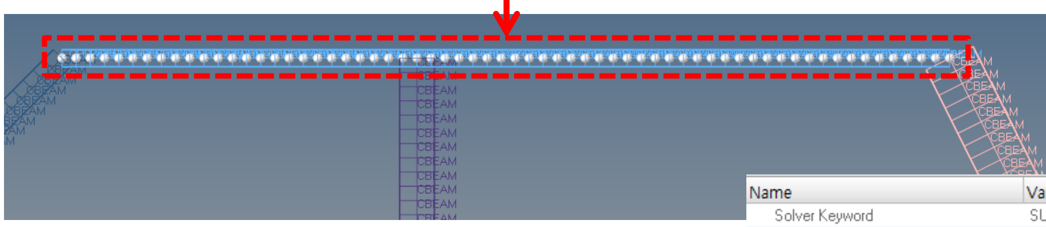
구속조건 및 하중조건 설정 (3)

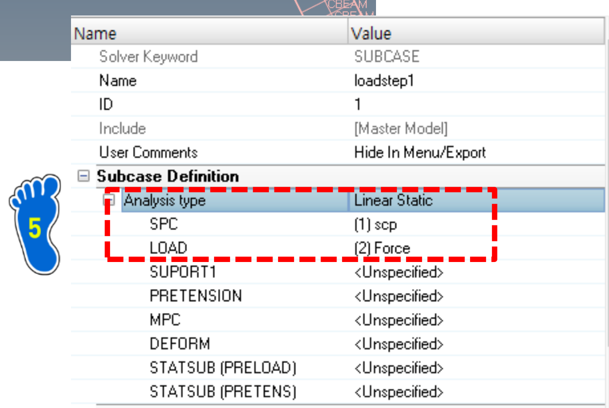






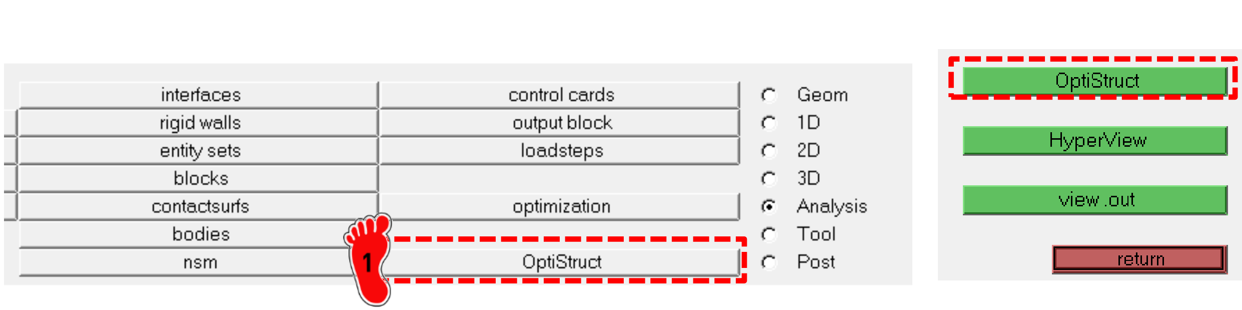


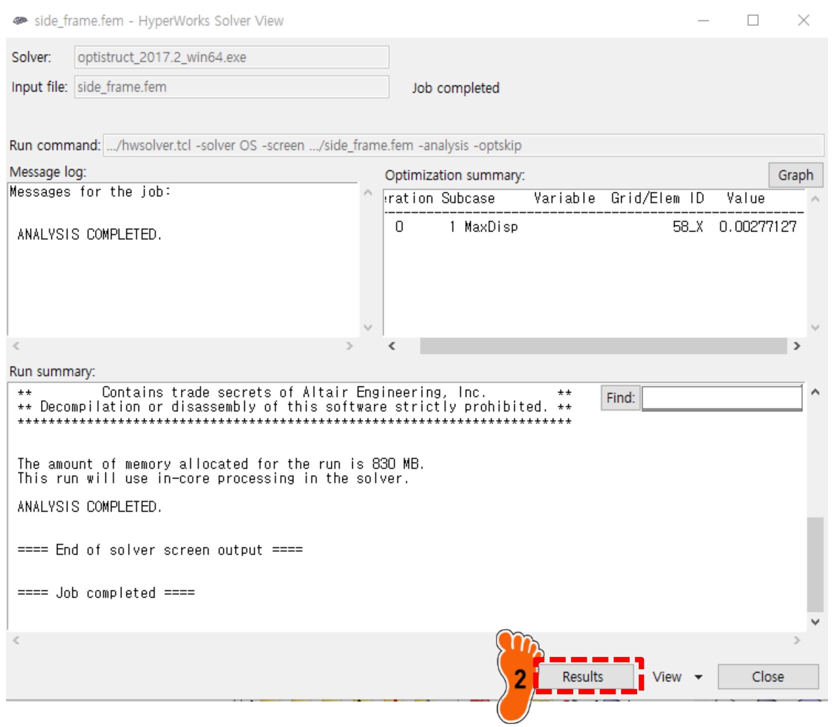




1. Create > Load Collector
   Name > Force
2. BCs > Create
   > Forces
3. 상단에 X방향으로 0.0156N
   Create (1N/64개 노드)
4. Create > Load Step
5. Analysis type
   >Linear Static
   SPC > spc
   LOAD > Force

해석 케이스 정의 및 해석 실행

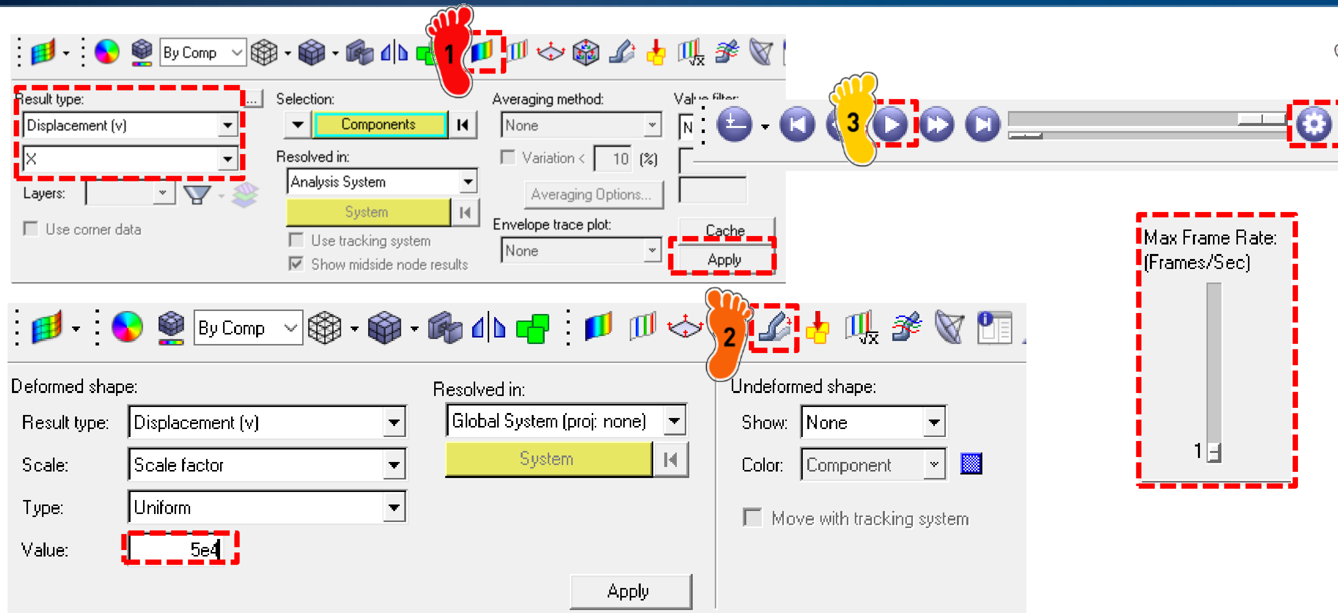




1. Analysis > OptiStruct 클릭
OptiStruct

2. Results 클릭

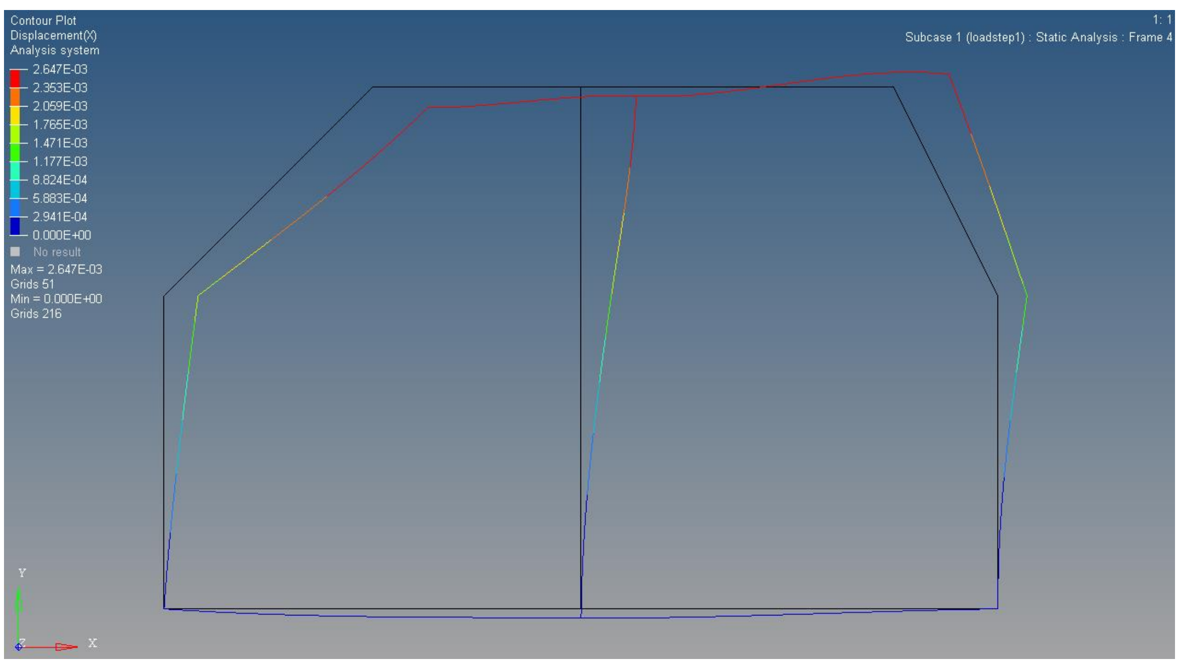
후처리



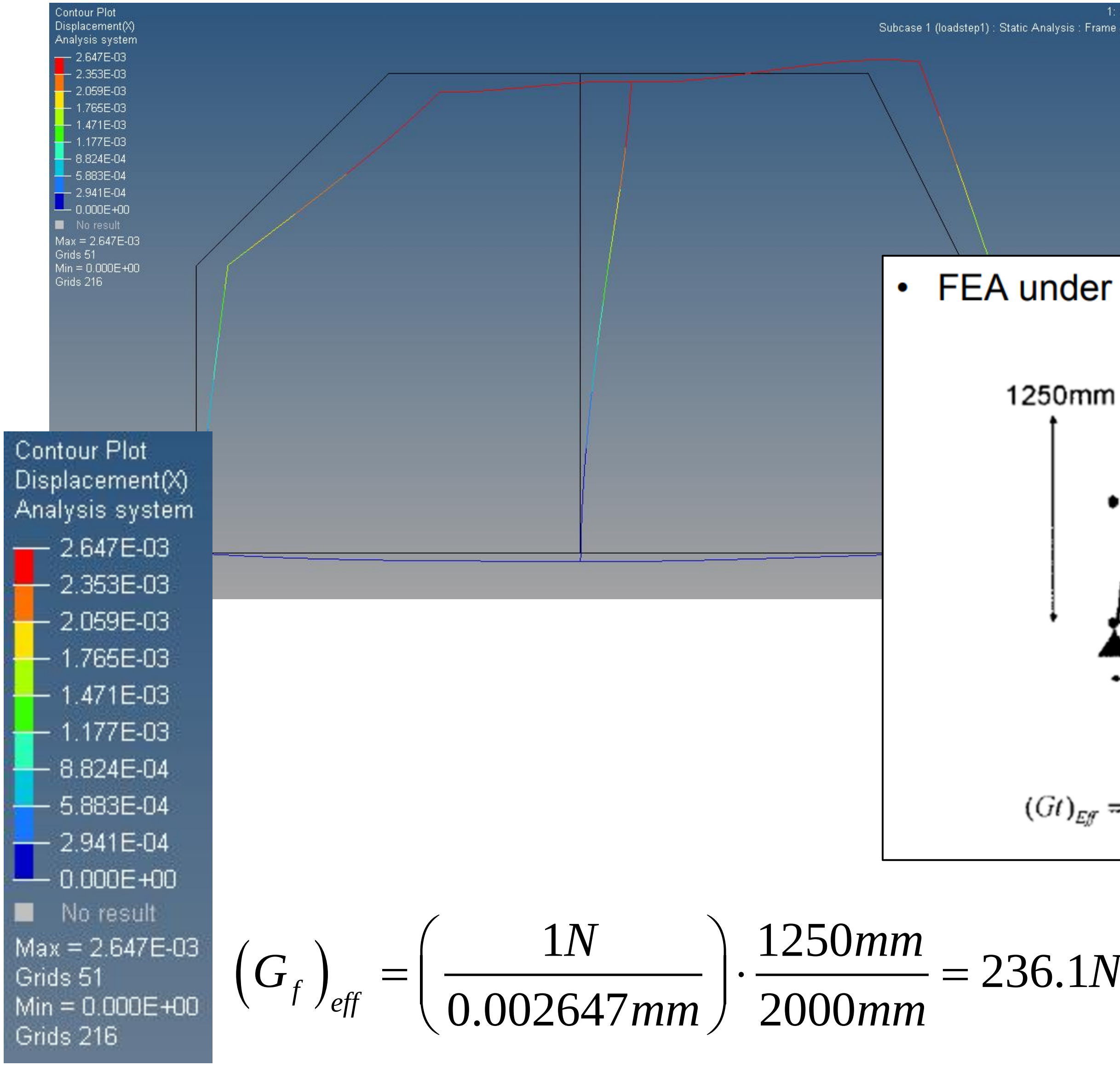
1 Contour 클릭
Displacement, X
Apply

2 Deformed 클릭
Value > 5e4
Apply

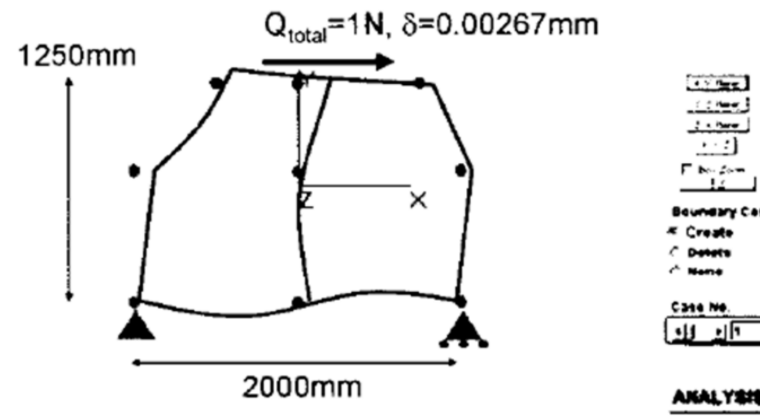
3 Animation Control 클릭
Max Frame Rate > 1
Start animation



후처리



- FEA under shear loading



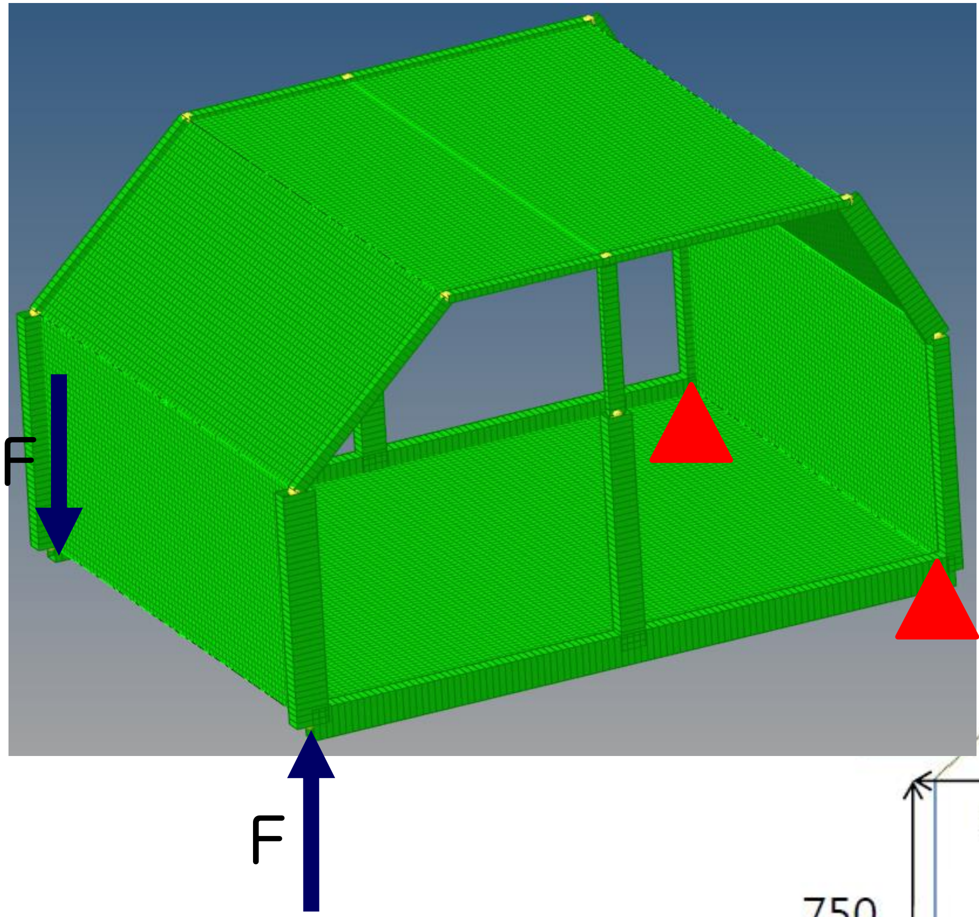
$$(Gt)_{Eff} = \left(\frac{Q}{\delta}\right)\frac{H}{L} = \left(\frac{1N}{.00267mm}\right)\frac{1250mm}{2000mm} = 234N/mm$$

$$\left(G_f\right)_{eff} = \left(\frac{1N}{0.002647mm}\right)\cdot\frac{1250mm}{2000mm} = 236.1N/mm$$

TORSIONAL STIFFNESS (SHELL + BEAM ELEMENTS)

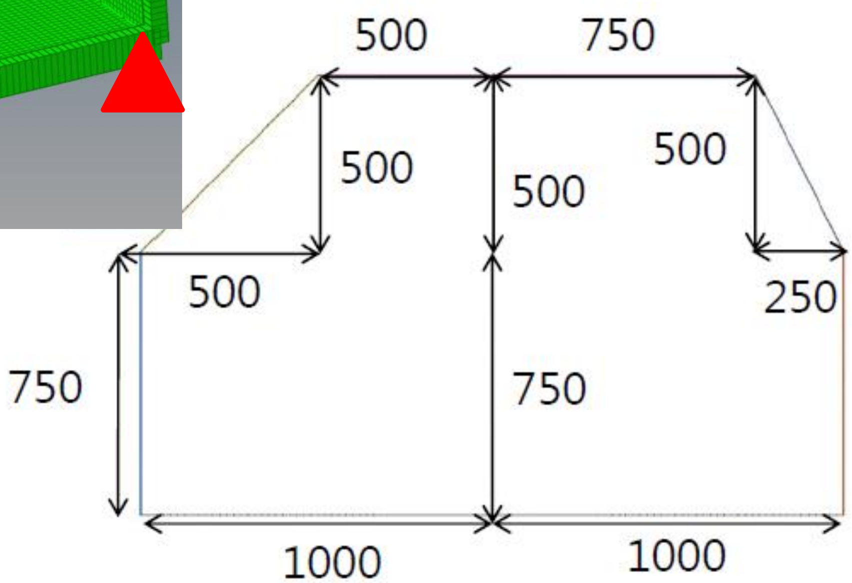
예제: TORSIONAL STIFFNESS (1)

다음 주어진 기하형상의 비틀림 강성을 구하시오. (프레임은 앞 예제의 모델 이용)



Material
- E = 204.8 GPa
- ν = 0.28
- G = 80 Gpa

Force
- T = 7730000 Nmm
- F = 7730000/1560
  = 4955.13 N



예제: TORSIONAL STIFFNESS (2)

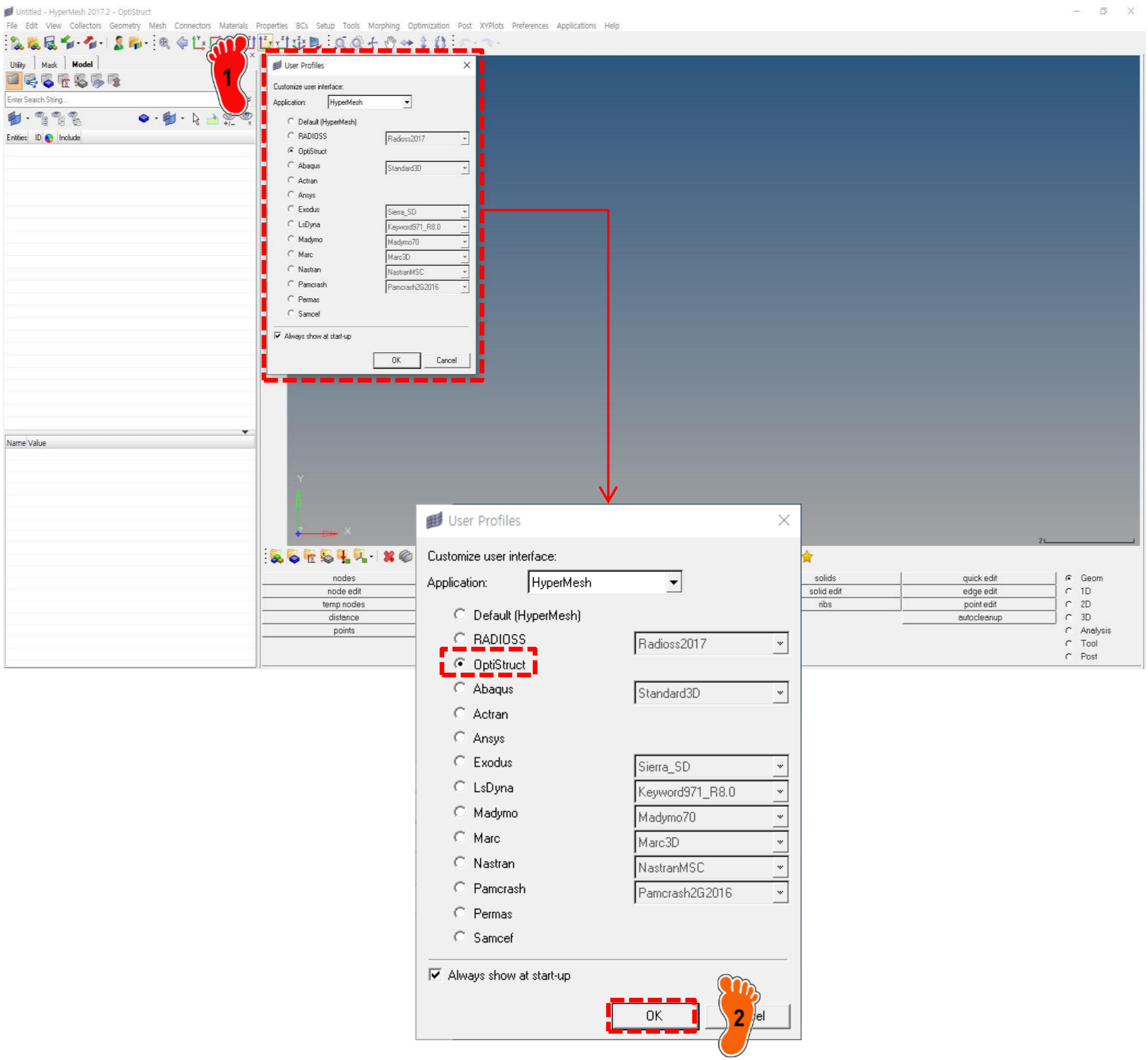
다음 주어진 기하형상의 비틀림 강성을 구하시오

$$T = 7,730,000 \text{ Nmm}, \ q = 2678\text{N} / 1250\text{mm} = 2.1414\text{N/mm}$$

Panel	Area of panel (mm^2)	Effective shear rigidity $(Gt)_{EFF}$ (N/mm)	Area of surface / $(Gt)_{EFF}$ (mm^3/N)
dash	1170000	80000	14.6
windshield	1103087	80000	13.8
roof	1950000	80000	24.4
back light	872067	80000	10.9
seat back	1170000	80000	14.6
floor	3120000	80000	29.0
side frame-left	2312500	234	9882.5
side frame-right	2312500	234	9882.5

$$K = \frac{1}{\left(\dfrac{q}{T}\right)^2 \displaystyle\sum_{\text{ALL SURFACE}} \left[\dfrac{\text{area of surface}}{(Gt)_{EFF}}\right]} = 6.55 \times 10^8 \ \text{Nmm/rad}$$

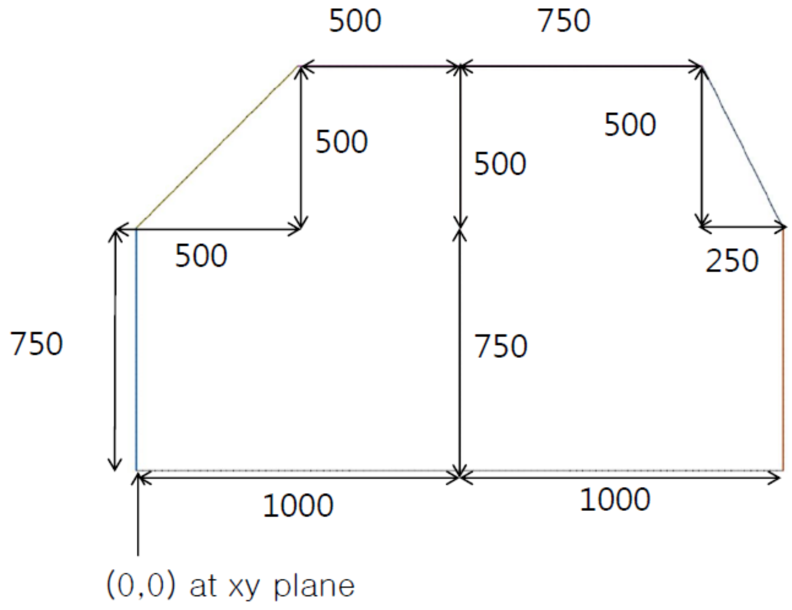
기하형상 생성 (1)

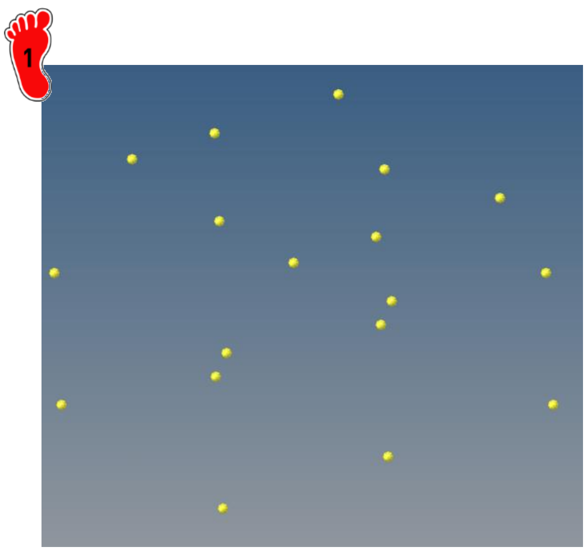


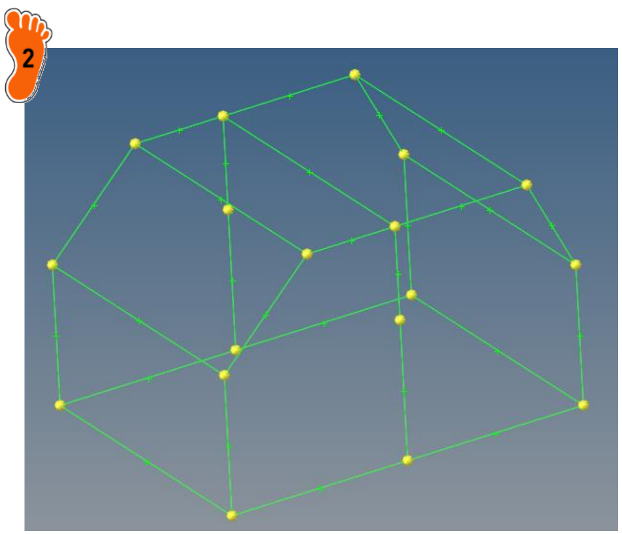
1 OptiStruct 선택

2 OK

기하형상 생성 (2)







1 주어진 기하형상 생성

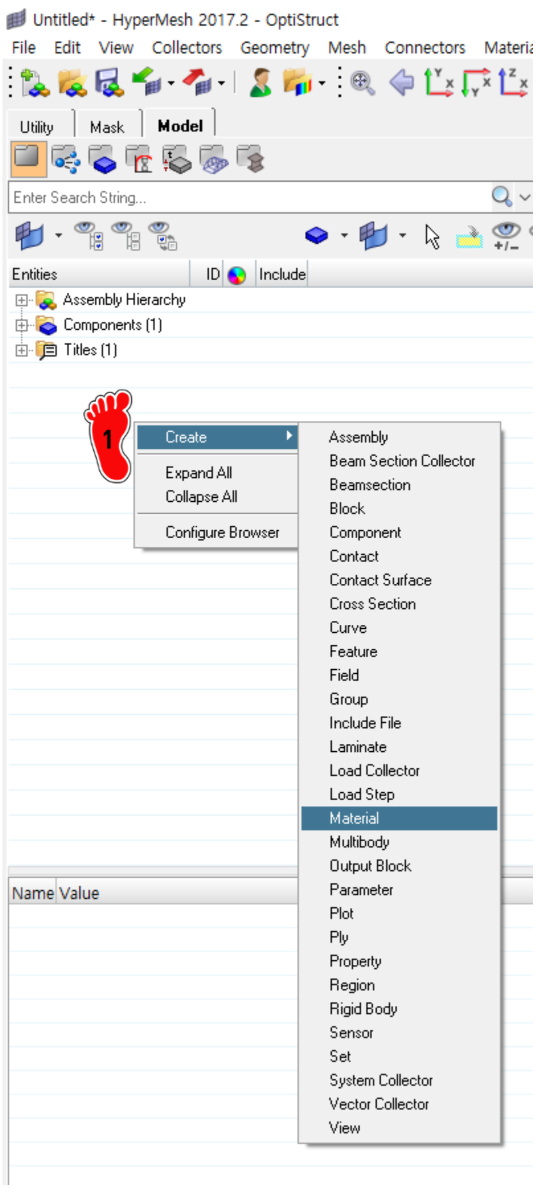
점 좌표

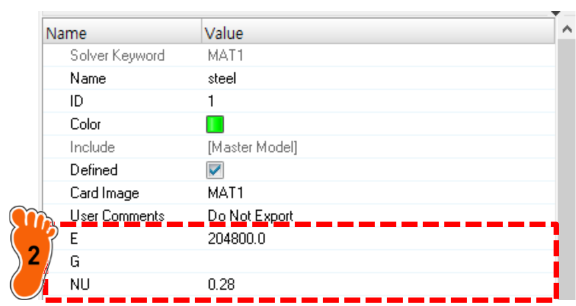
(0,0,0)
(0,750,0)
(500,1250,0)
(1000,0,0)
(1000,750,0)
(1000,1250,0)
(1750,1250,0)
(2000,0,0)
(2000,750,0)

반대편 sideframe을 모델링 하기 위해,
Z축방향으로 1560mm 평행 이동된 점 9개 추가로 생성

2 생성한 점들을 연결하여 그림과 같이 선 생성

재료 물성 및 특성 입력 (1)

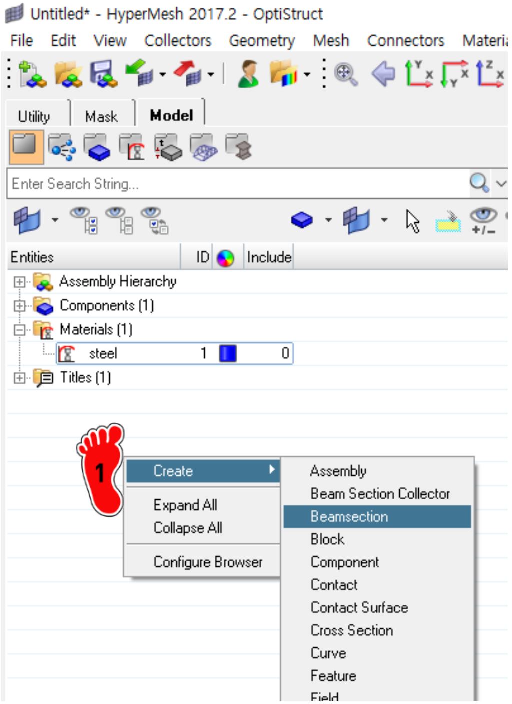


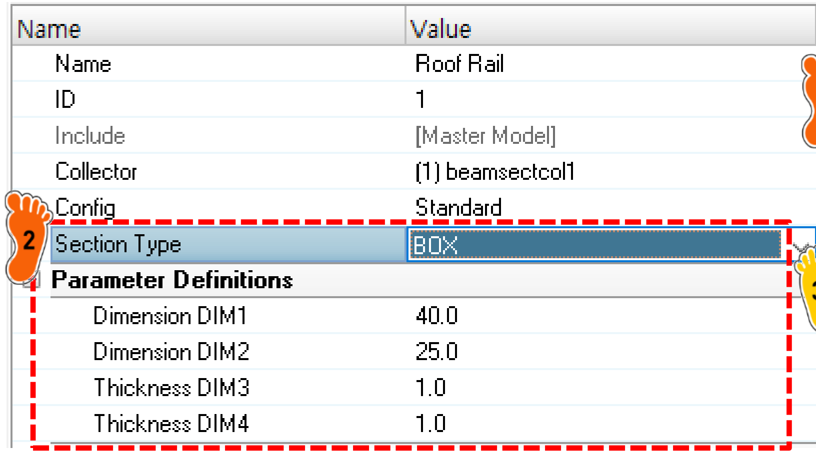


1 우클릭, Create > Material

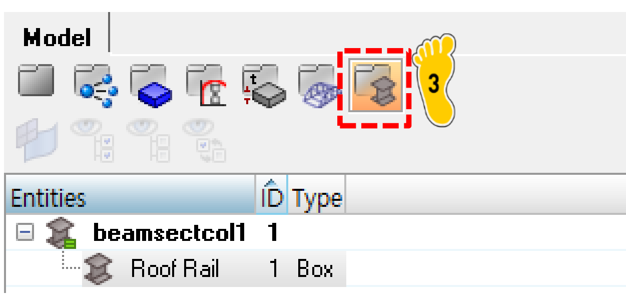
2 Name > steel
탄성계수(E)
>204.8Gpa (204800 N/mm^2)
푸아송비(NU) > 0.28
재료 생성

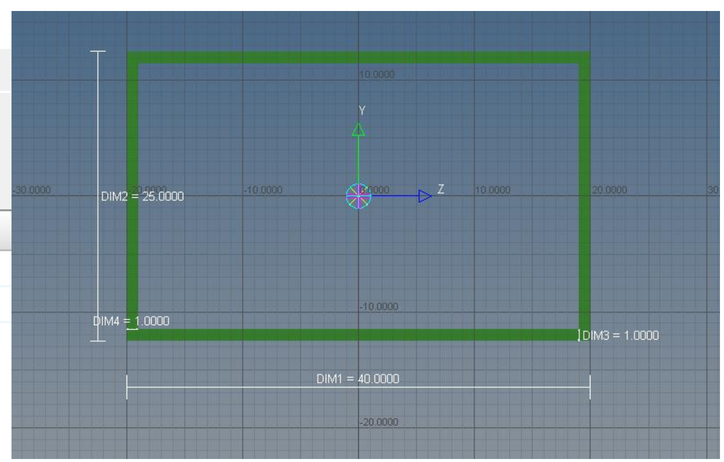
재료 물성 및 특성 입력 (2)





Name	Value
Name	Roof Rail
ID	1
Include	[Master Model]
Collector	(1) beamsectcol1
Config	Standard
Section Type	BOX
Parameter Definitions	
Dimension DIM1	40.0
Dimension DIM2	25.0
Thickness DIM3	1.0
Thickness DIM4	1.0



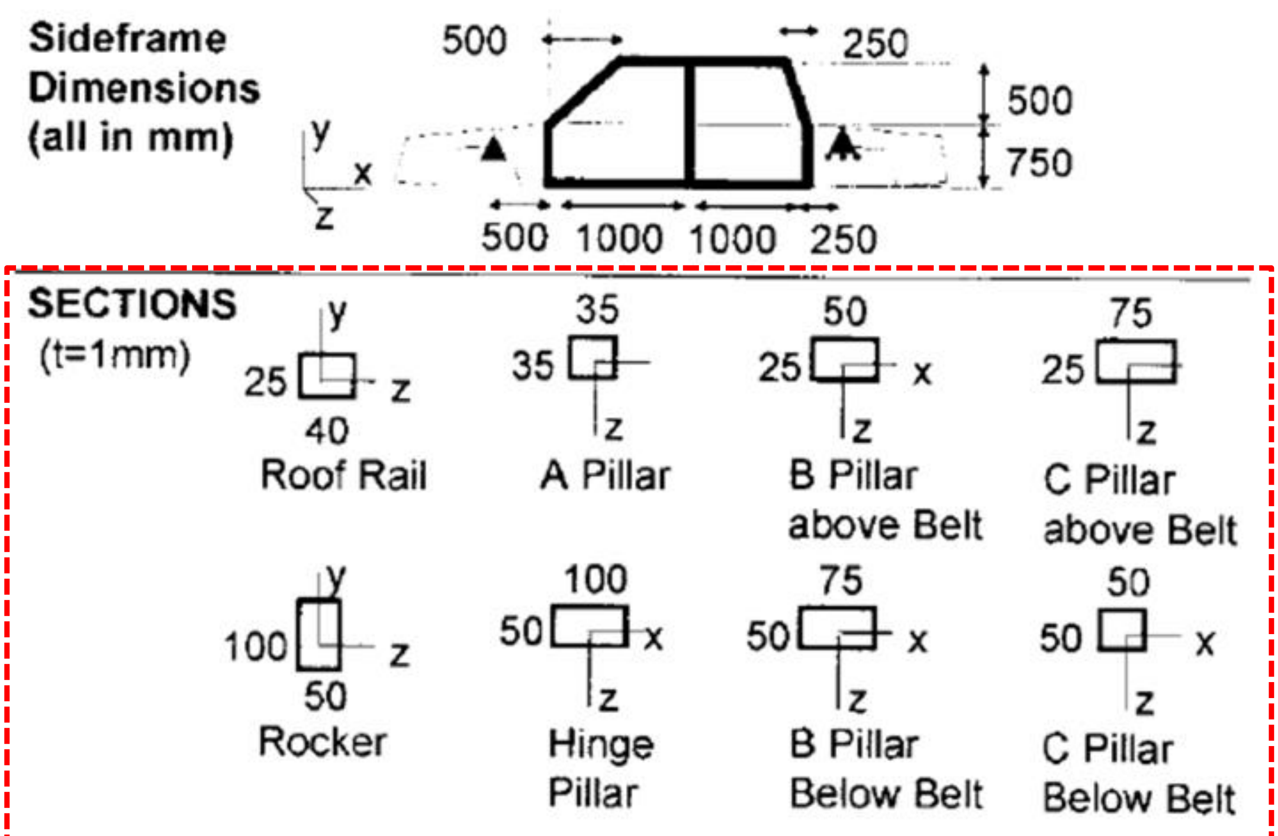


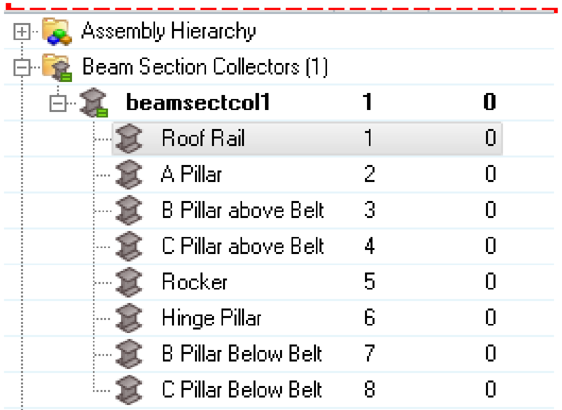
1 우클릭,
Create > Beamsection

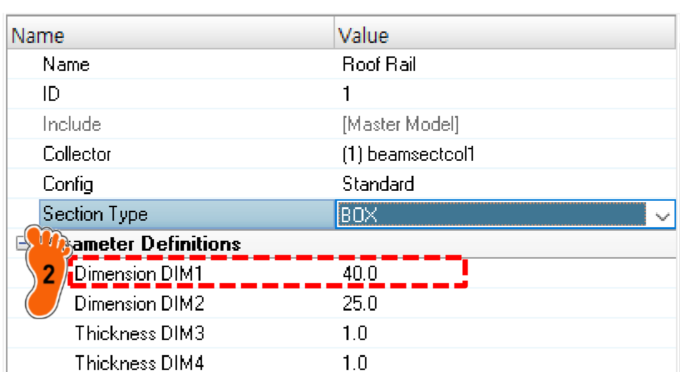
2 Section Type > BOX,
단면형상 입력

3 Hyperbeam view에서 단면
형상 확인 가능

재료 물성 및 특성 입력 (3)



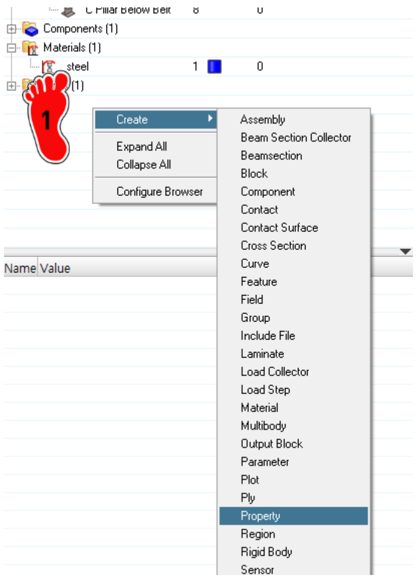


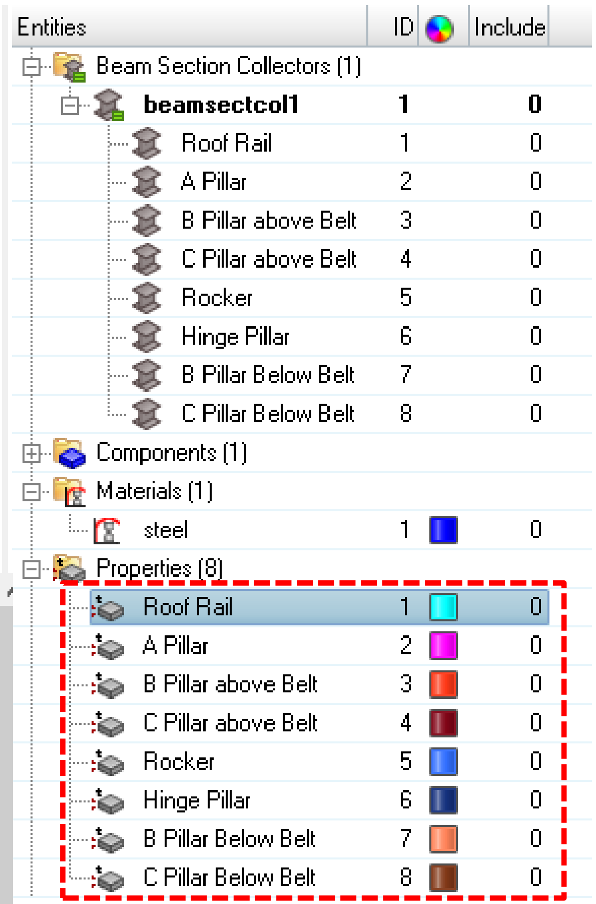


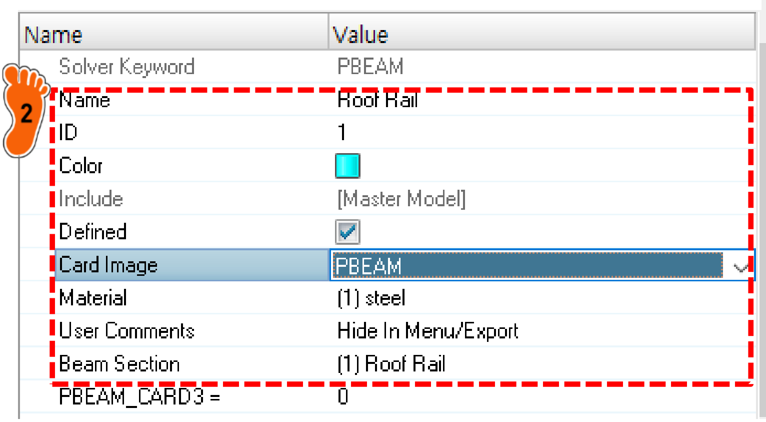
1 예제에서 주어진 단면형상 8개를 각각 생성

2 Dimension DIM1에 Z방향 길이 입력

재료 물성 및 특성 입력 (4)



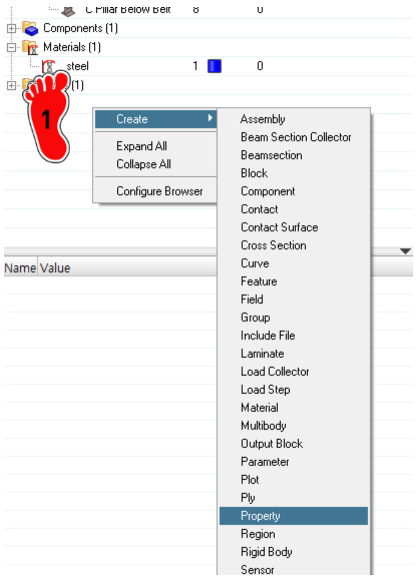


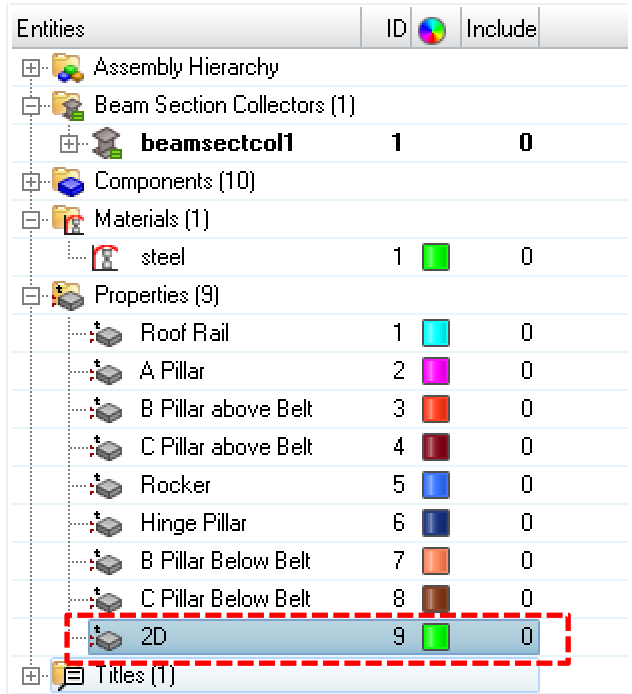


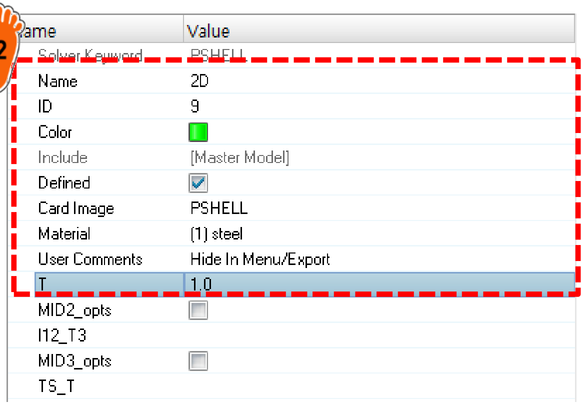
1. 우클릭,
Create > Property

2. Property 8개 각각 생성, 단면 형상과 각각 연결
Card Image > PBEAM
Material > steel
Beam Section > 단면형상 각각 선택
Name > 단면 형상과 동일하게

재료 물성 및 특성 입력 (5)



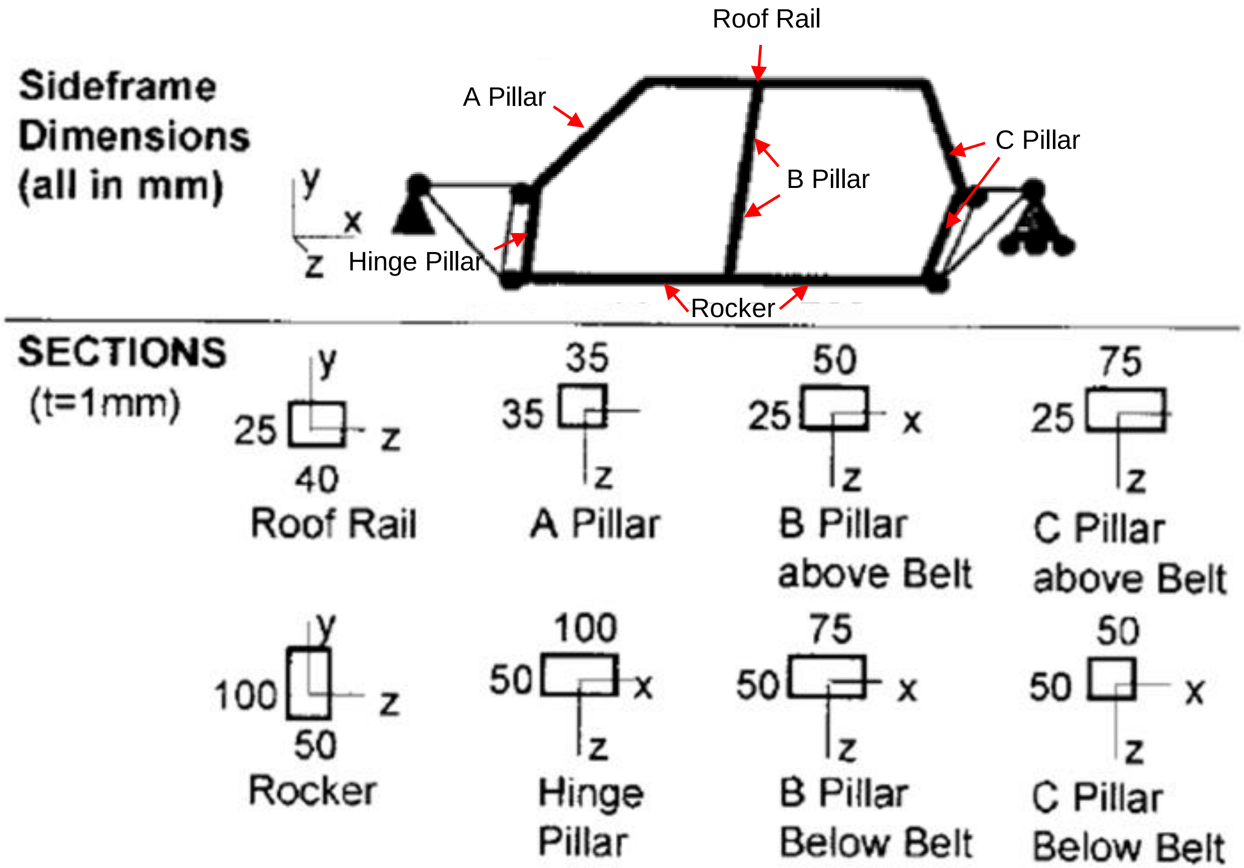




1 우클릭,
Create > Property

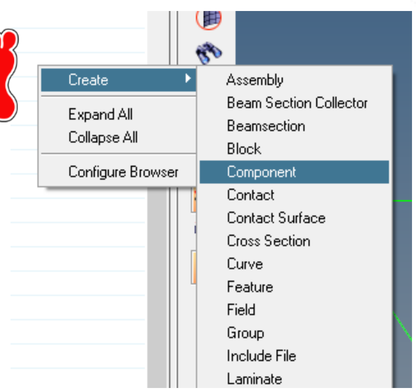
2 Card Image > PSHELL
Material > steel
T > 1mm
Name > 2D

요소망 생성 (1)



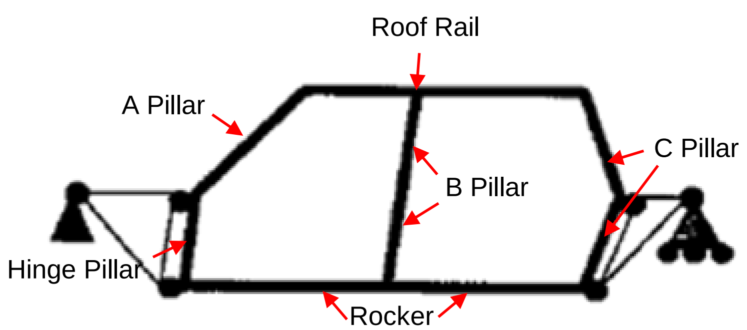
1 우클릭, Create > Component

2 Name, Property >요소망 생성할 단면형상의 이름과 동일하게



Name	Value
Name	Roof Rail
ID	2
Color	
Include	[Master Model]
Property	(1) Roof Rail
Material	(1) steel

요소망 생성 (2)



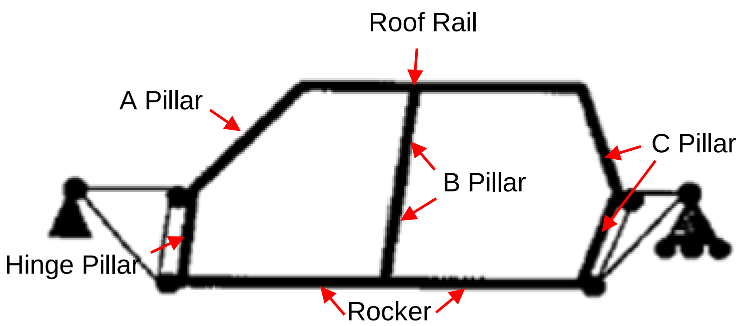
1 1D > line mesh 클릭

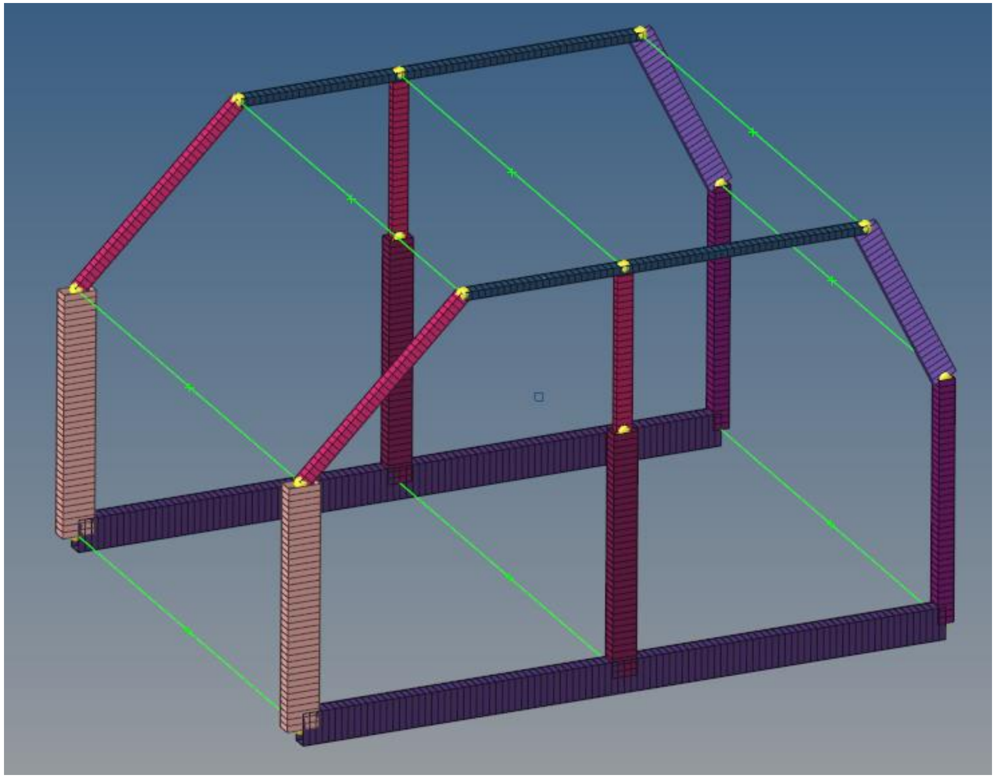
2 요소망 생성
크기 > 20
Add elems to current component
Property > 단면형상 이름과 동일하게

3 요소망 생성시 단면 형상이 원하는 좌표축에 놓이도록 주의(x축과 평행한 빔은 y-aixs로 선택, y축과 평행한 빔은 x-aixs로 선택)

4 Mesh,
Return

요소망 생성 (3)

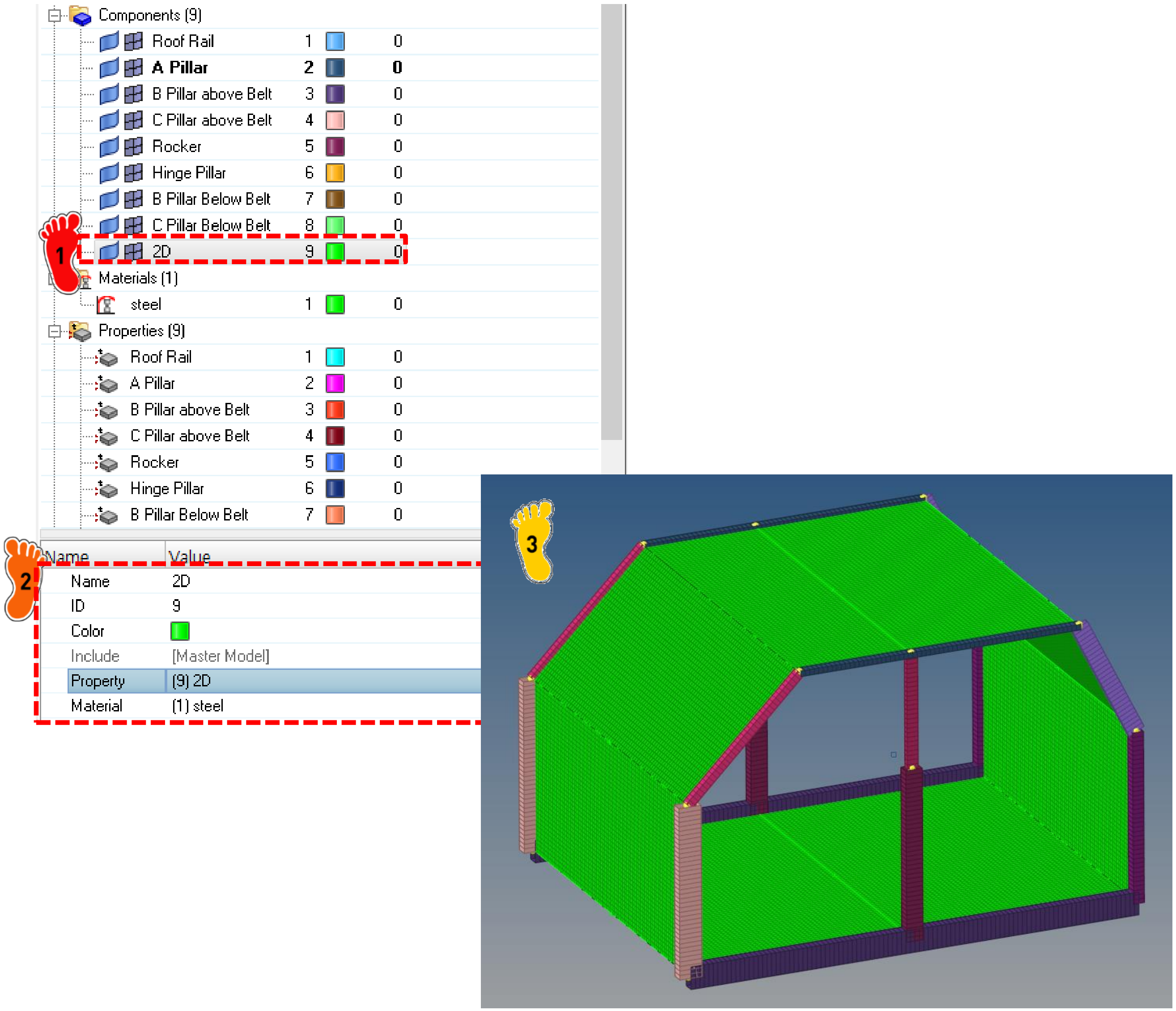




1 앞의 과정(1,2) 반복해서 8개 빔 요소망 생성

그림과 비슷한 형상이 나와야 함

요소망 생성 (4)



1 우클릭,
Create > Component

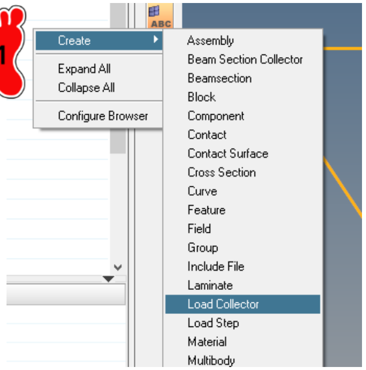
2 Name > 2D
Property > 2D
Material > steel

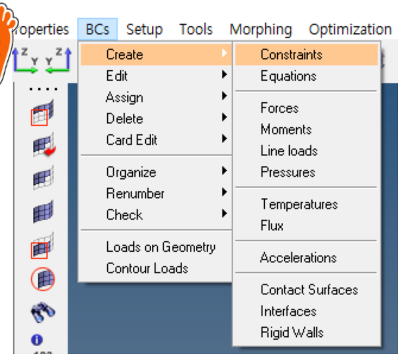
3 2D > planes
20 mm 인 크기로 평면요소 생성

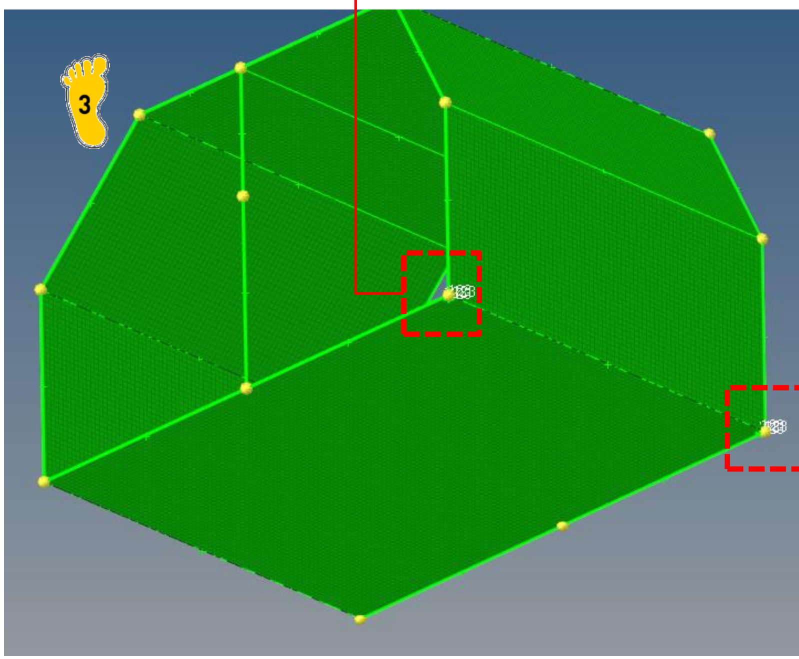
* Tool -> eadges -> tolerance -> equivalence (모서리 병합)

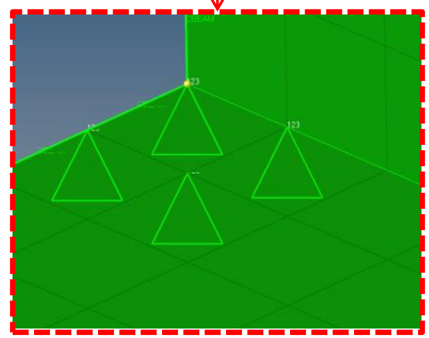
* Geom -> temp nodes -> clear all (Nodes 제거)

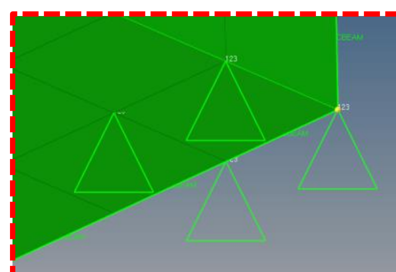
구속조건 및 하중조건 설정 (1)

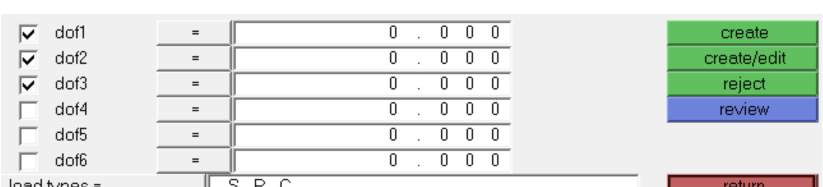










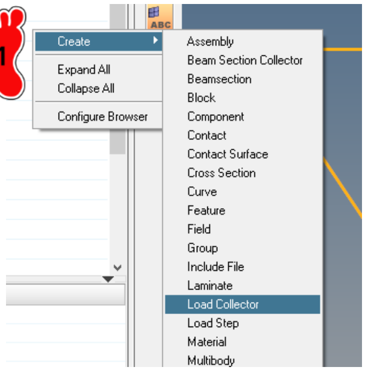


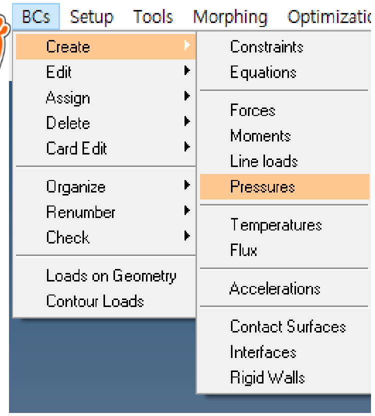
1. Create > Load Collector
Name > spc

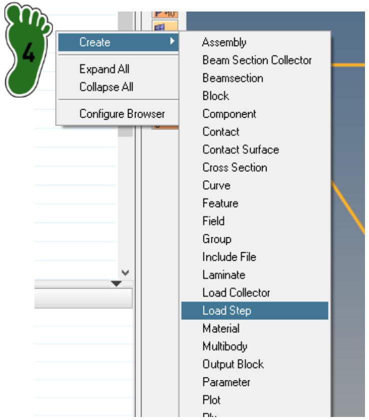
2. BCs > Create
> Constraints

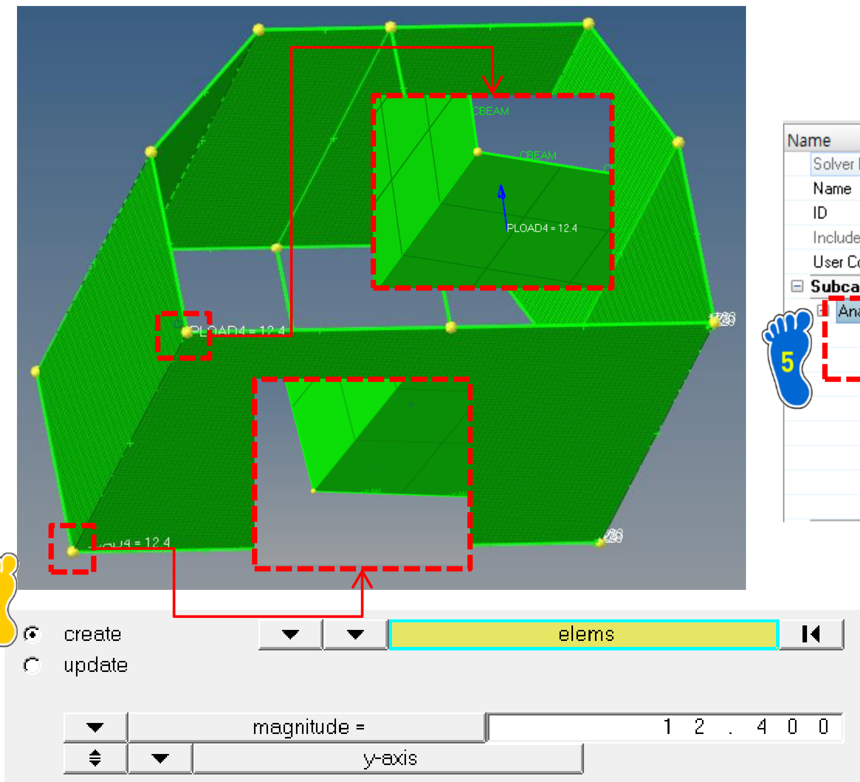
3. 뒤쪽 양 끝 요소의 절점 4개를 핀 구속조건으로 구속

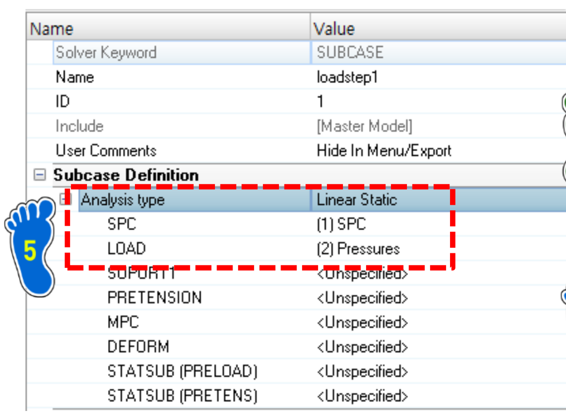
구속조건 및 하중조건 설정 (2)











1. Create > Load Collector
Name > Pressures

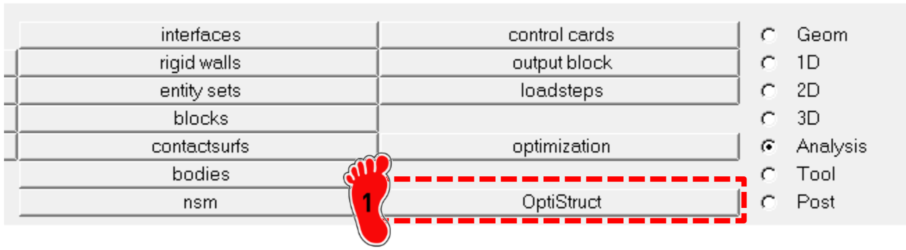
2. BCs > Create
> Pressures

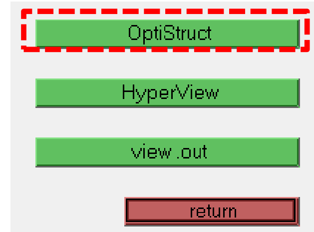
3. 하중/면적
=(4955/400)=12.4 N/mm^2
인 압력하중으로 비틀림 하중을 적용
(양쪽 부호 다르게)

4. Create > Load Step

5. Analysis type
>Linear Static
SPC > spc
LOAD > Pressures

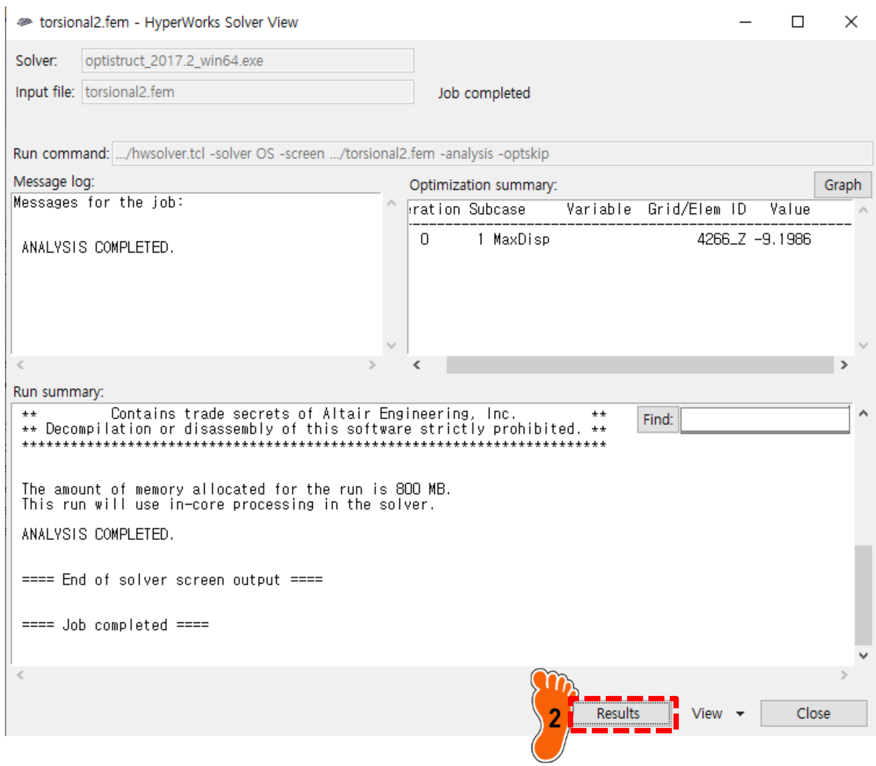
해석 케이스 정의 및 해석 실행



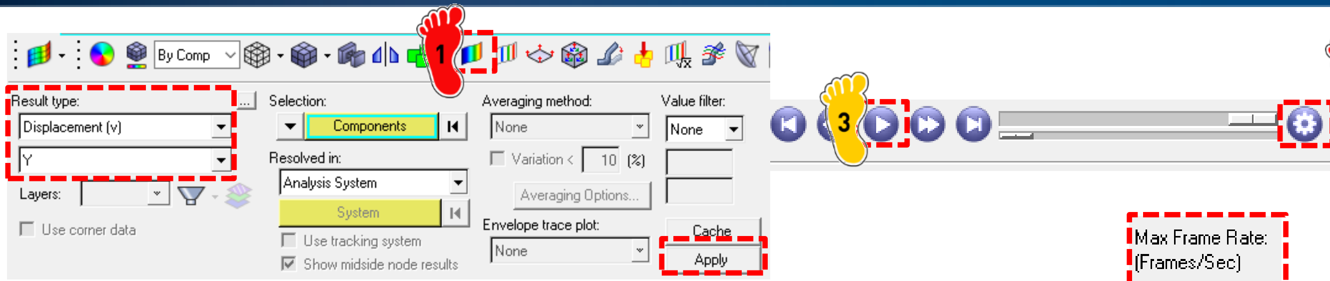


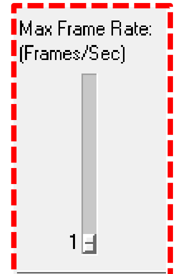
1 Analysis > OptiStruct 클릭
OptiStruct

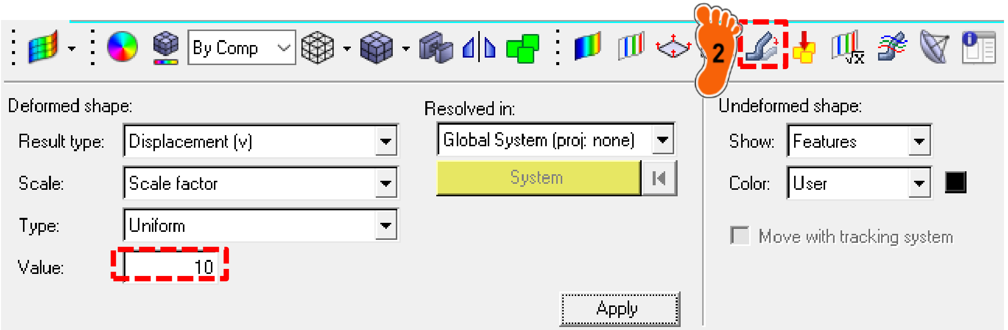
2 Results 클릭

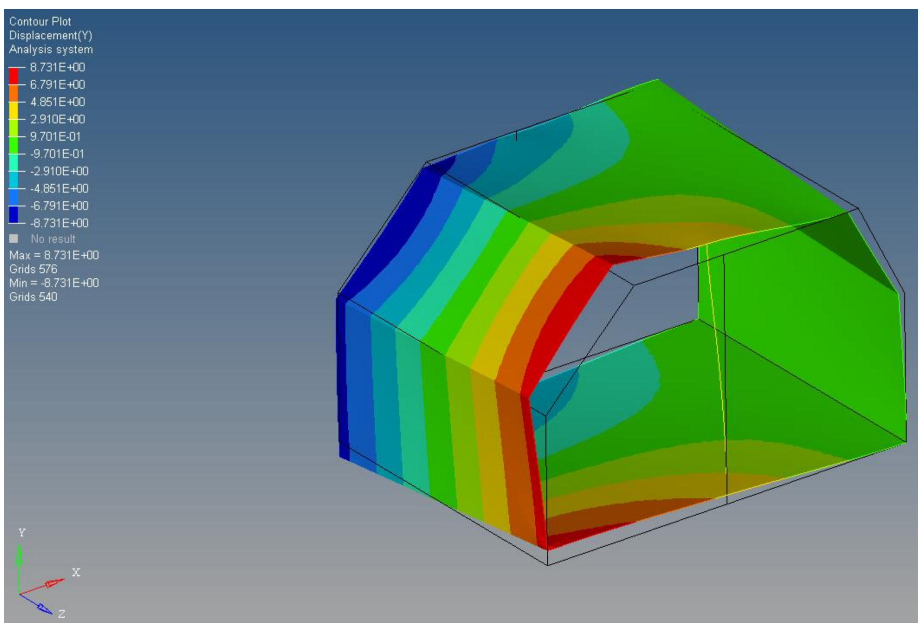


후처리







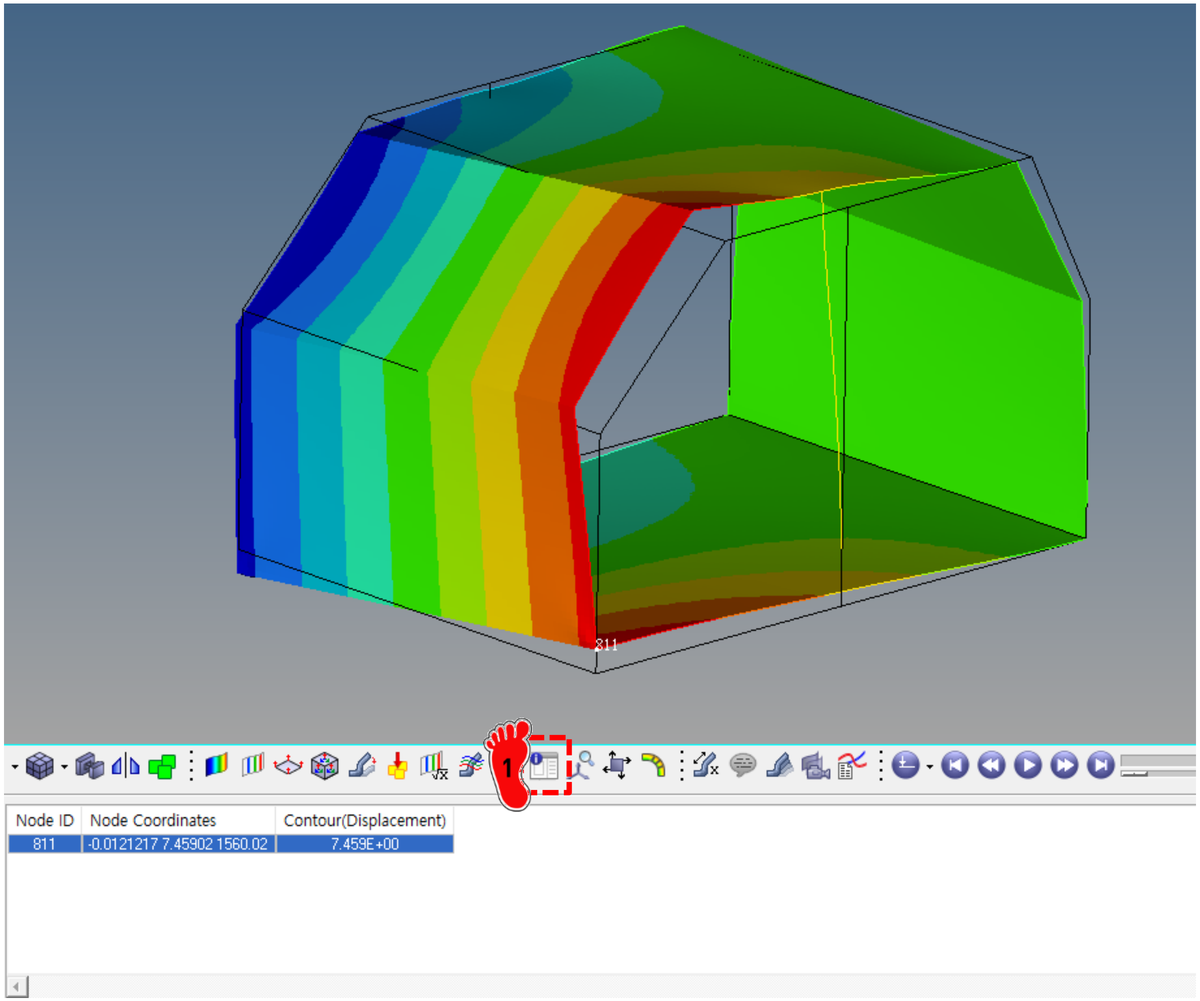


1 Contour 클릭
Displacement, Y
Apply

2 Deformed 클릭
Value > 10
Apply

3 Animation Control 클릭
Max Frame Rate > 1
Start animation

후처리

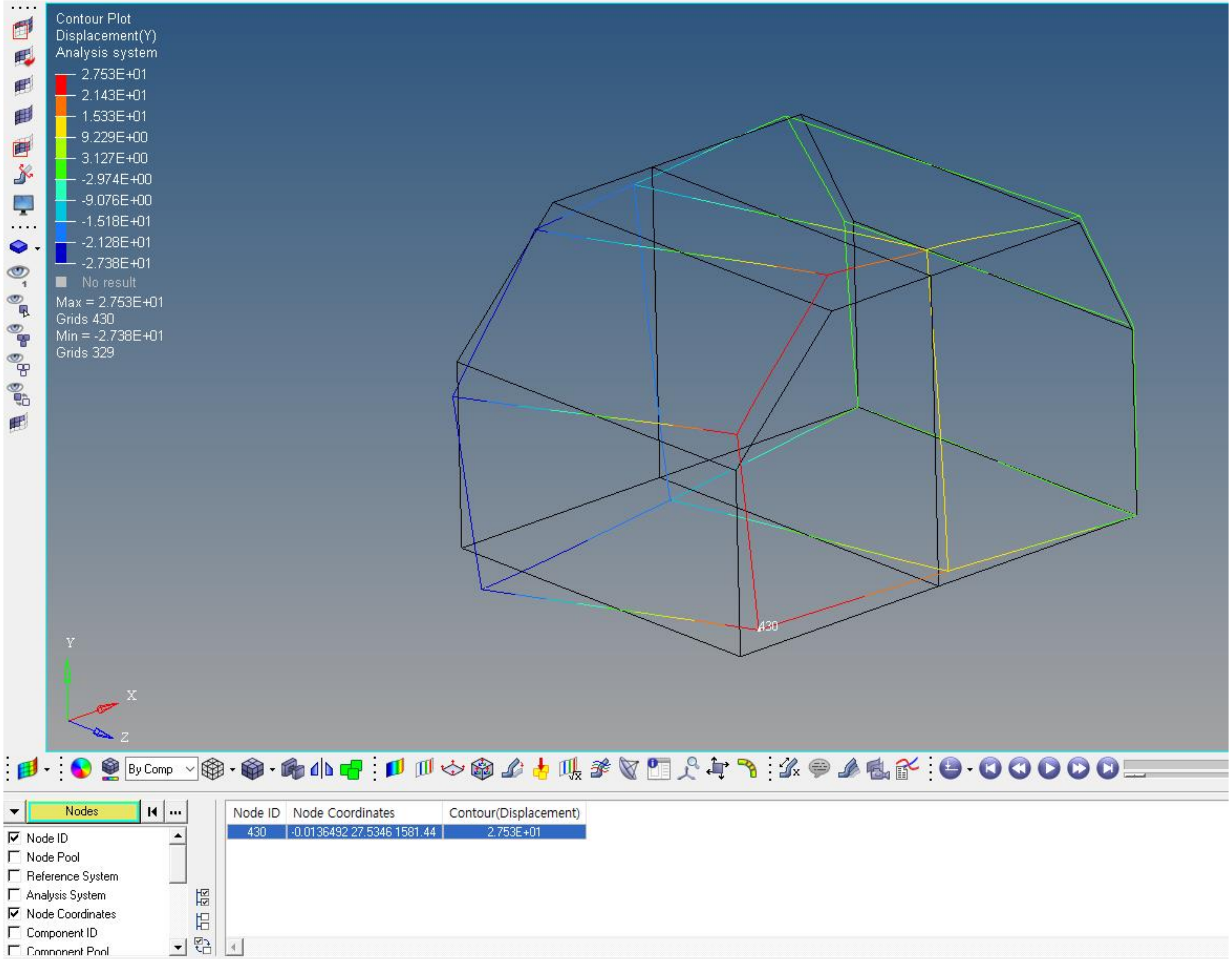


1. Query 클릭
2. 앞쪽 끝 노드 클릭
3.

$$K = \frac{T}{\theta} = \frac{Fw}{(2\delta/w)}$$
$$= \frac{7730000}{(2 \times 7.459/1560)}$$
$$= 8.08e8\ Nmm/rad$$

(이론: $6.55e8\ Nmm/rad$)

숙제



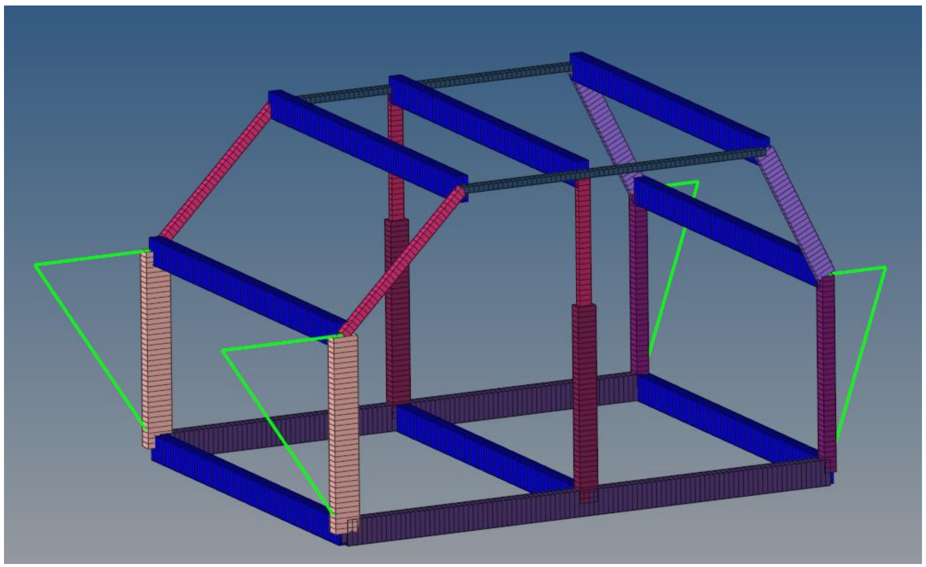
1차원 Beam으로 모델링 후 비틀림 강성을 구하고 앞선 예제와 비교
(Side frame 외 보강 빔(z축)의 단면 형상은 rocker로 가정하고 해석 수행)

ADDITIONAL EXAMPLE #1 (MODAL ANALYSIS)

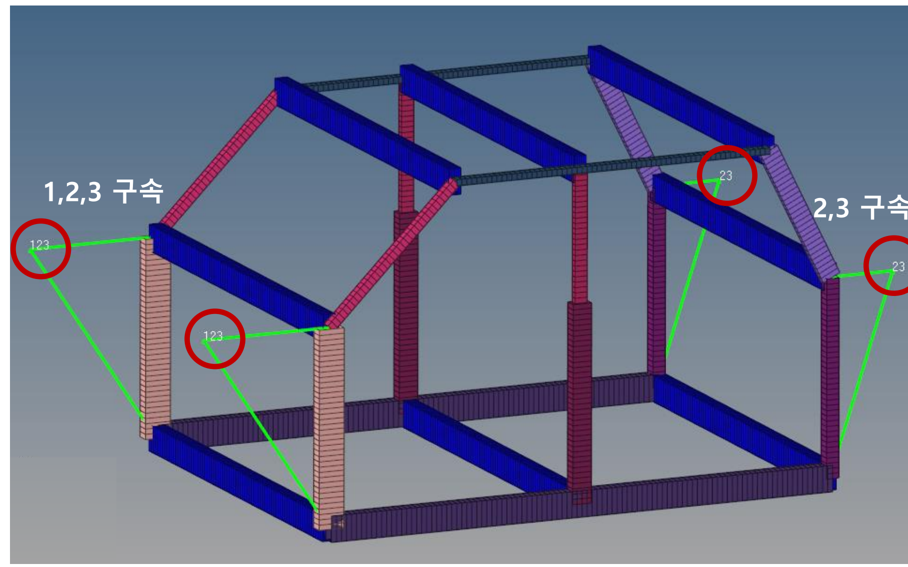
MODAL ANALYSIS

- 1차원 beam으로 모델링 후 modal analysis 수행 (자유진동 vs. 끝단 구속)

자유진동 가정

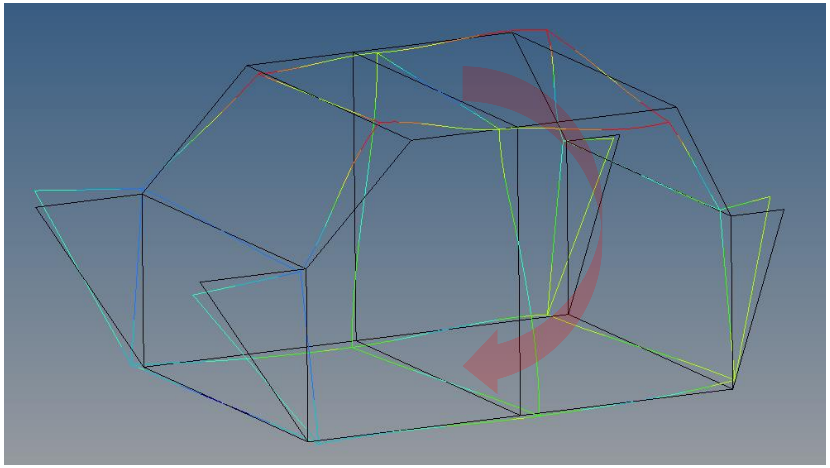


Front & Rear 구속

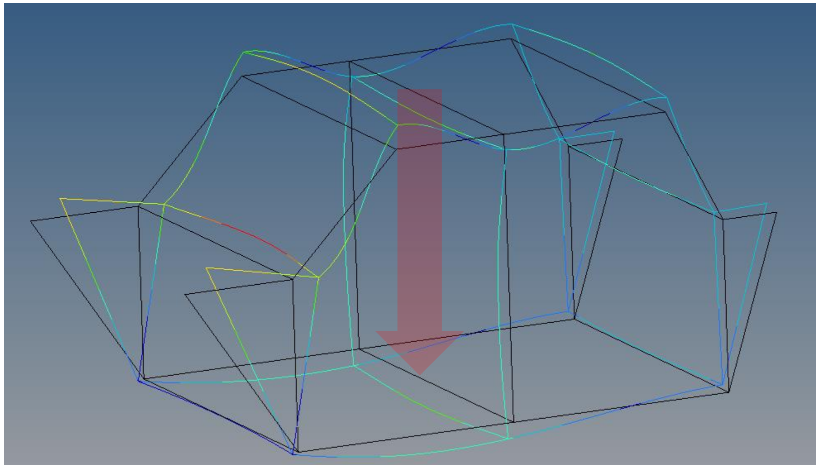


MODAL ANALYSIS

- 자유진동 가정

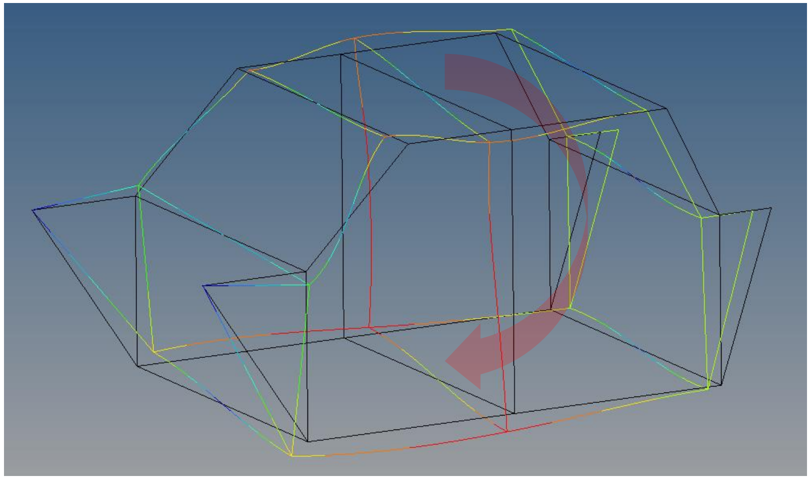


Mode 6(68.89 Hz): Torsion

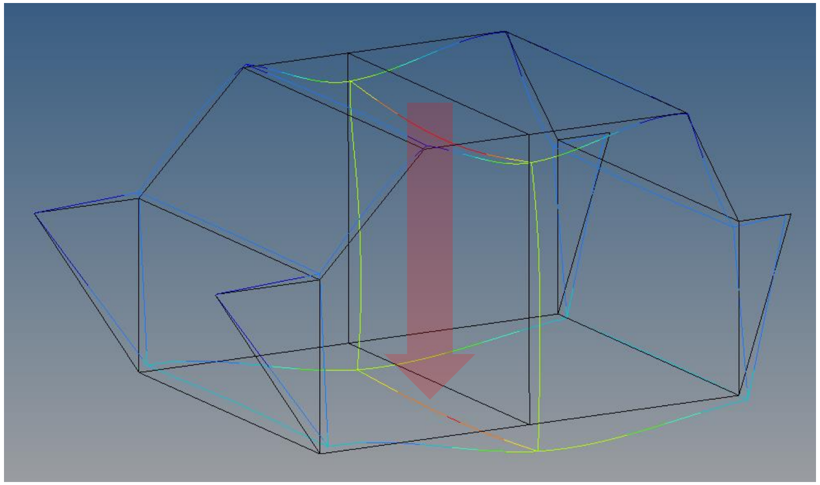


Mode 7(70.04 Hz): Bending

- Front & Rear 구속



Mode 5(54.77 Hz): Torsion

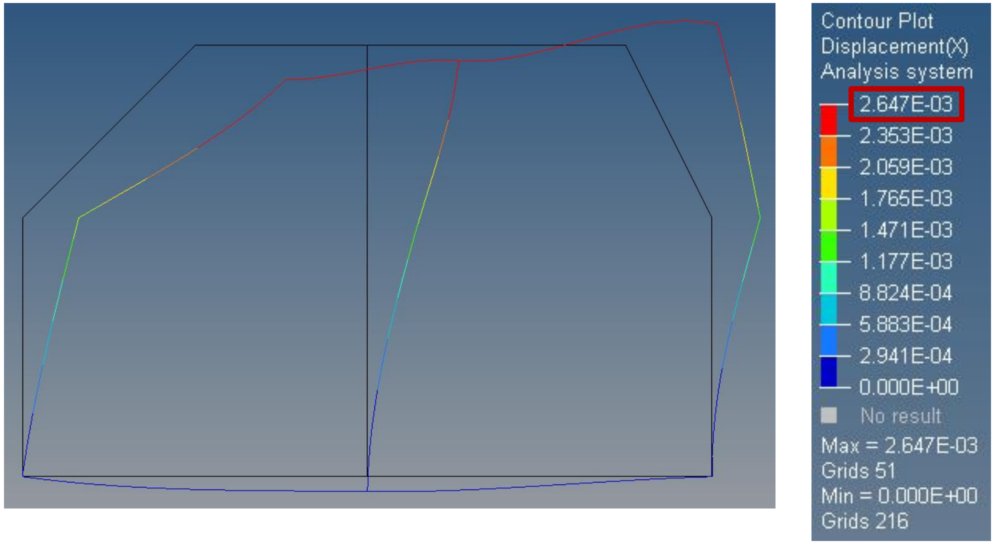


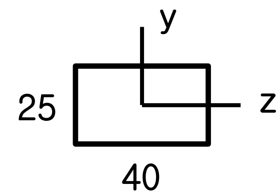
Mode 8(75.92 Hz): Bending

ADDITIONAL EXAMPLE #2
(EFFECT OF BEAM SECTION)

EFFECT OF BEAM SECTION

- Side frame 모델 비교(effective shear rigidity): Rroof rail 변경



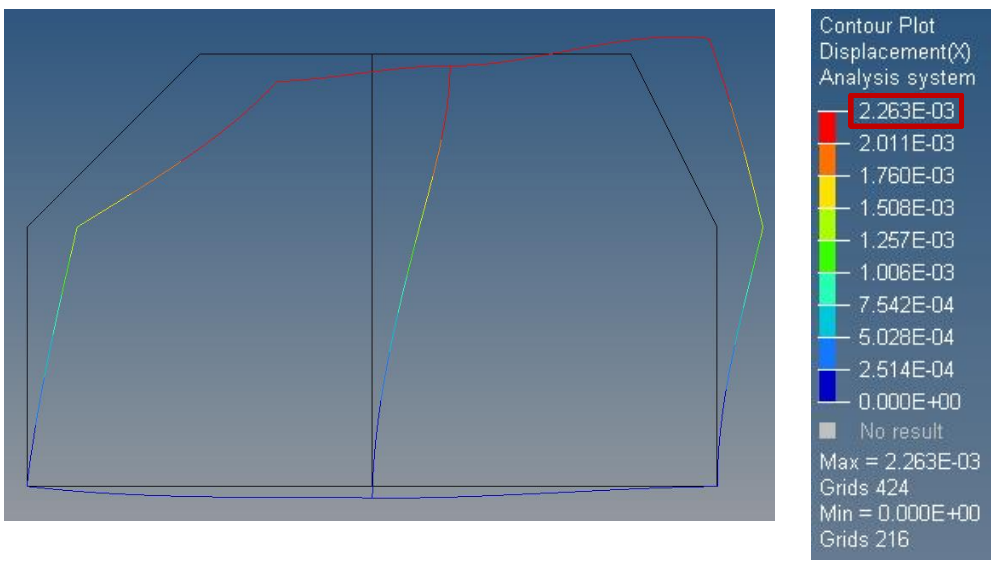


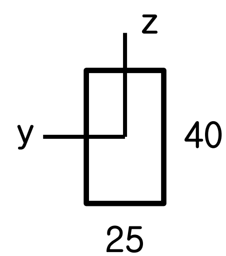
Roof Rail
(y-axis)

Original axis

$$\left(G_f\right)_{eff} = \left(\frac{1N}{0.002647mm}\right) \cdot \frac{1250mm}{2000mm} = 236.1N/mm$$

$$\delta = 0.002647mm$$





Roof Rail
(z-axis)

Rotate axis

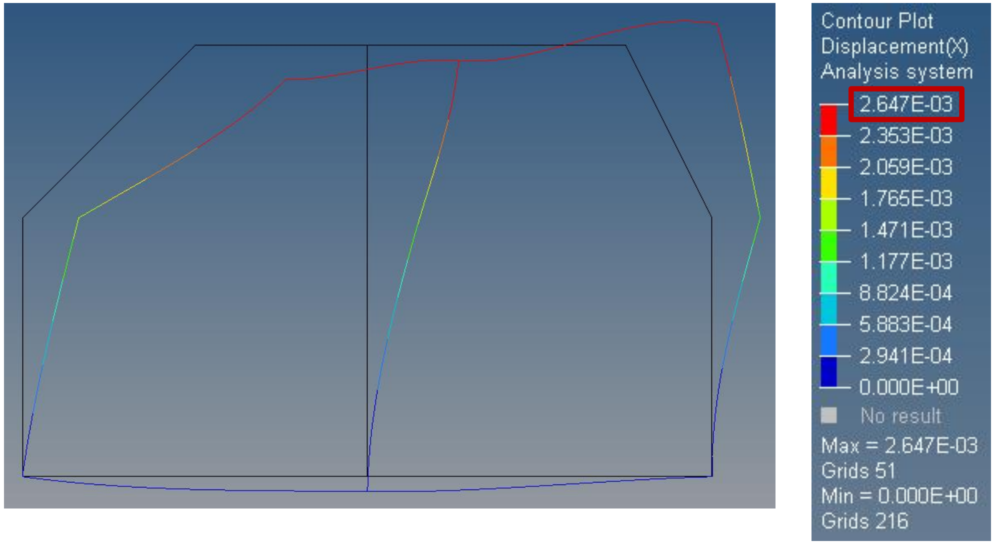
Difference: 16.9%

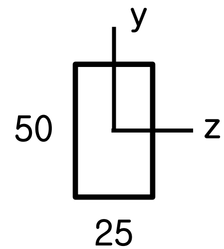
$$\left(G_f\right)_{eff} = \left(\frac{1N}{0.002263mm}\right) \cdot \frac{1250mm}{2000mm} = 276.2N/mm$$

$$\delta = 0.002263mm$$

EFFECT OF BEAM SECTION

- Side frame 모델 비교(effective shear rigidity): B Pillar above belt 변경



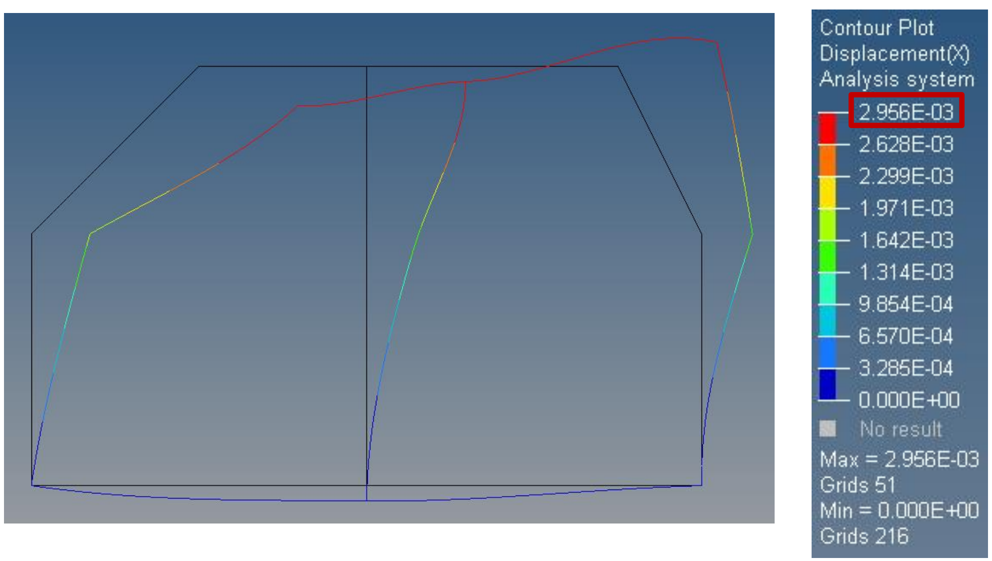


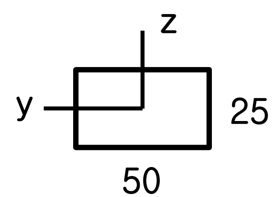
B Pillar above belt
(y-axis)

Original axis

$$\left(G_f\right)_{eff}=\left(\frac{1N}{0.002647mm}\right)\cdot\frac{1250mm}{2000mm}=236.1N/mm$$

$$\delta=0.002647mm$$





B Pillar above belt
(z-axis)

Rotate axis

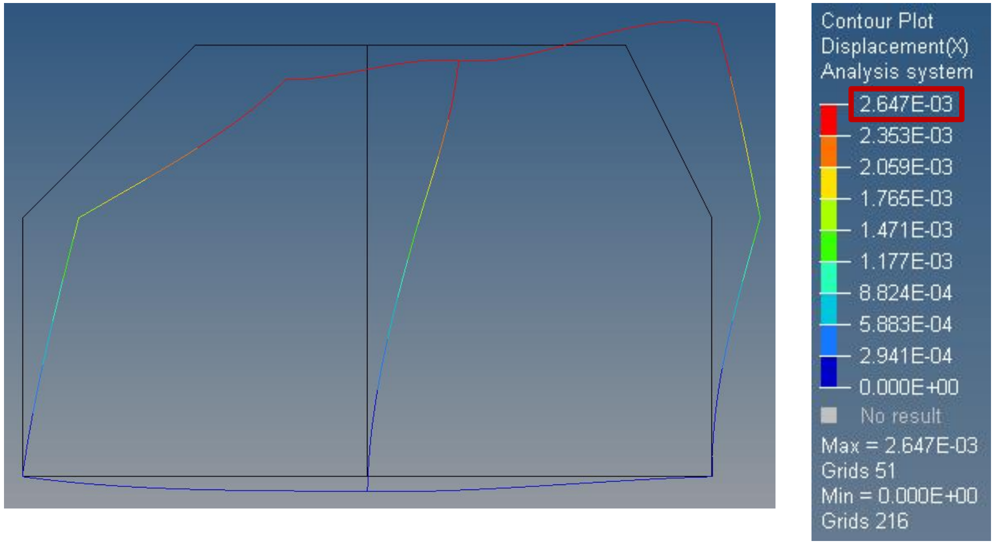
Difference: 10.4%

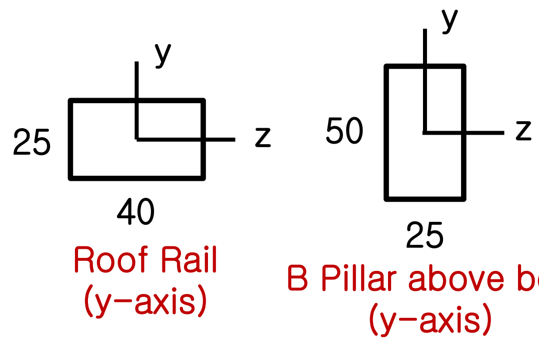
$$\left(G_f\right)_{eff}=\left(\frac{1N}{0.002956mm}\right)\cdot\frac{1250mm}{2000mm}=211.4N/mm$$

$$\delta=0.002956mm$$

EFFECT OF BEAM SECTION

- Side frame 모델 비교(effective shear rigidity): Roof Rail 및 B Pillar above belt 변경



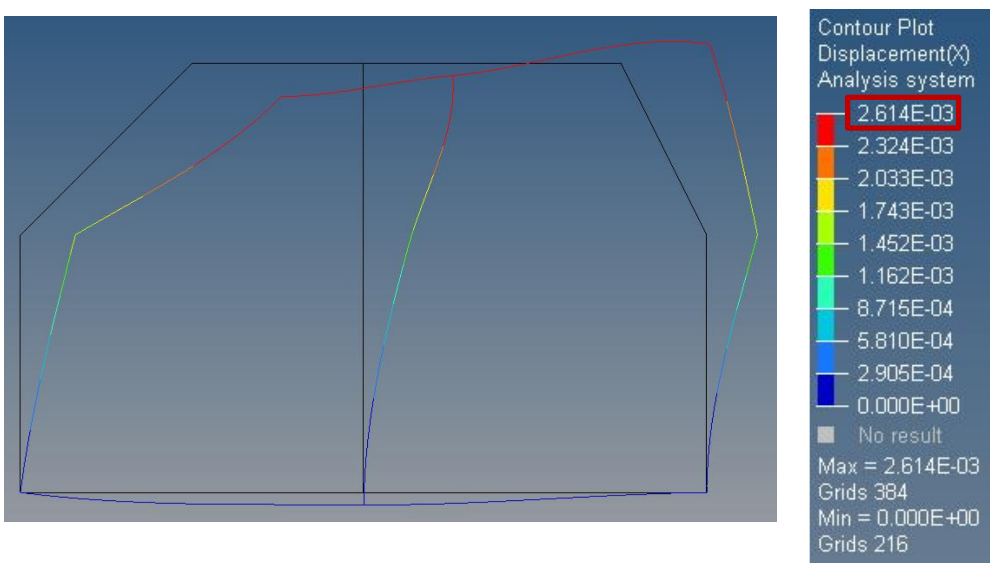


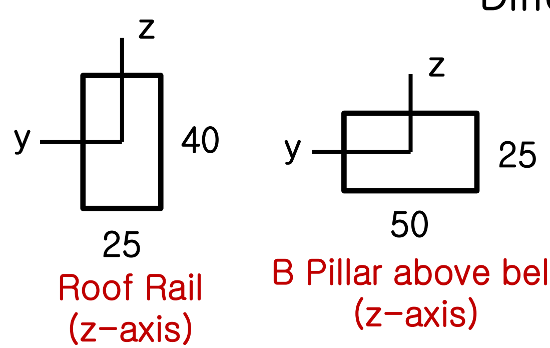
Original axis

$$\left(G_f\right)_{eff} = \left(\frac{1N}{0.002647mm}\right)\cdot\frac{1250mm}{2000mm} = 236.1N/mm$$

$$\delta = 0.002647mm$$

Difference: 1.2%





Rotate axis

$$\left(G_f\right)_{eff} = \left(\frac{1N}{0.002614mm}\right)\cdot\frac{1250mm}{2000mm} = 239.1N/mm$$

$$\delta = 0.002614mm$$