

고체역학 기반 해석 프로세스

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



한양대학교
HANYANG UNIVERSITY

CDL Computational
Design
Lab

목차

- 메뉴 소개
- 조작법
- 해석 프로세스 (빔 요소)
 - Component-geometry 생성
 - Materials and properties
 - Component-mesh 생성
 - Load collectors-boundary conditions 설정
 - Load collectors-load 설정
 - Load steps 정의 및 해석
 - 후처리

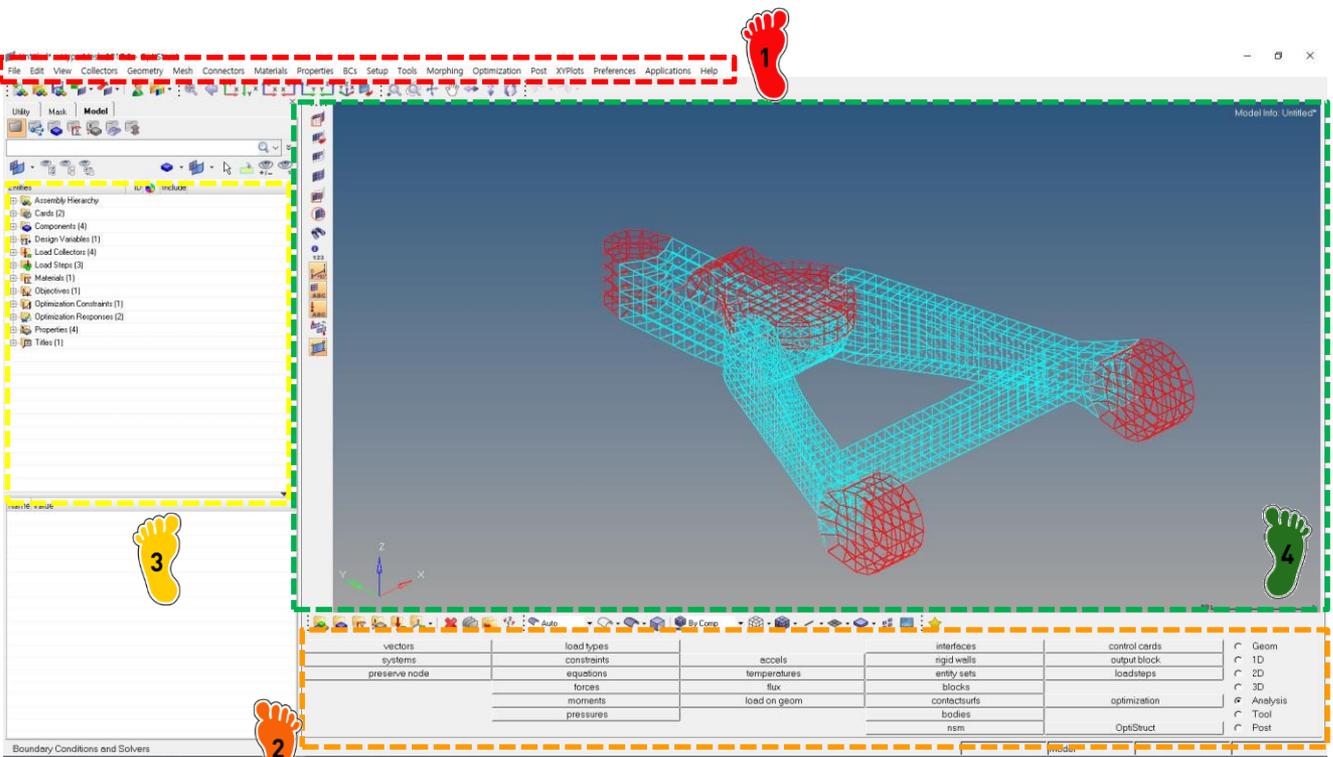
작업 메뉴

1 주 메뉴
→ 해석을 위한 모든 메뉴

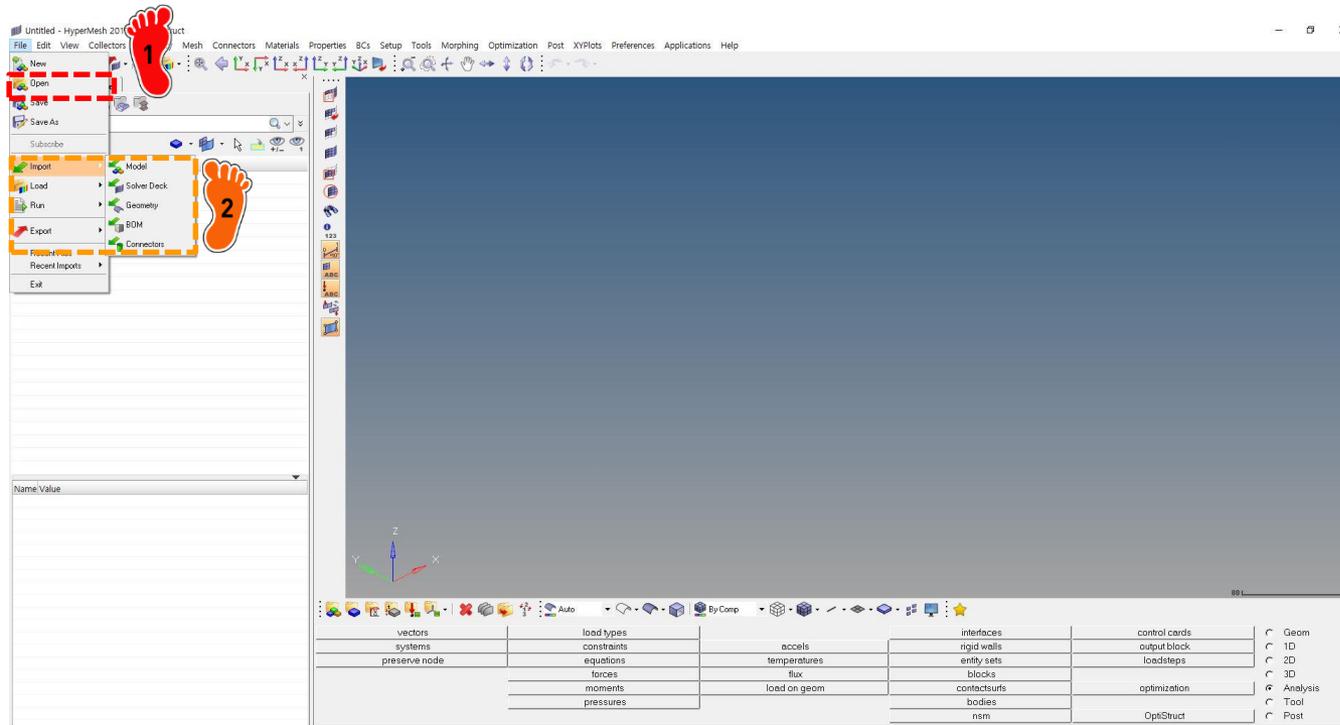
2 간편 메뉴

3 작업 트리
→ 해석 모델의 정보

4 작업 창
→ 해석 모델을 보여주는 GUI 창



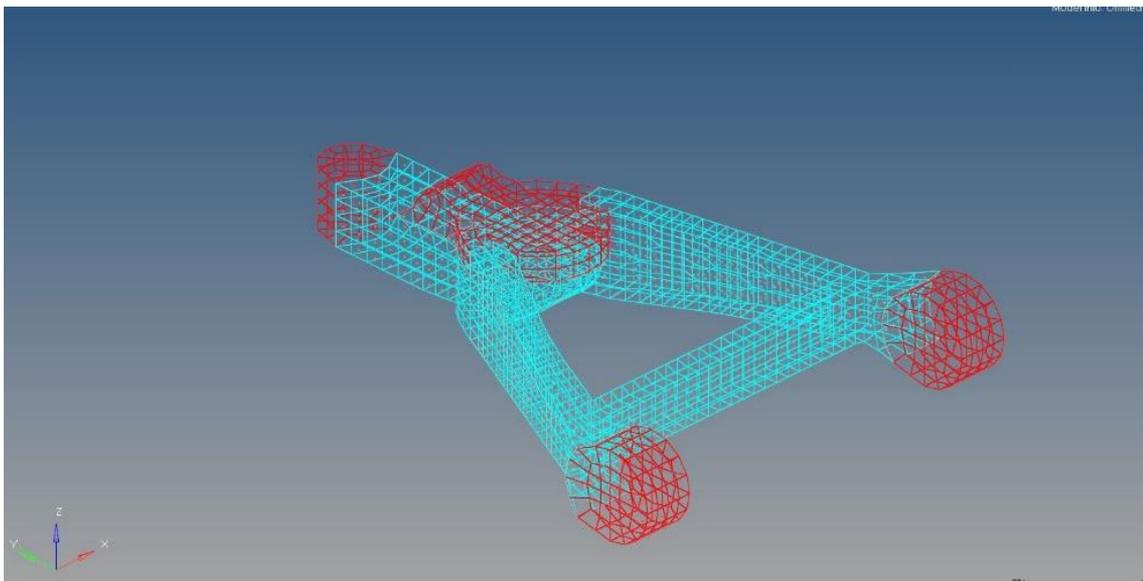
불러오기



1 HyperMesh 파일의 경우
'Open' 기능 이용

2 CAD 파일 및 외부 유한요소 모델(NASTRAN)의 경우
불러오기 기능 이용

작업 창 마우스 조작법



1 확대/축소



2 회전



3 이동



마우스의 휠을 가지고 조작

1 확대/축소: Ctrl + 마우스 휠
드래그

2 회전: Ctrl + 마우스 왼쪽클릭
후 드래그

3 이동: Ctrl + 마우스 오른쪽클릭
후 드래그

모델링 / 해석 순서

1. 형상 생성

nodes	lines	surfaces	solids	quick edit	Geom
node edit	line edit	surface edit	solid edit	edge edit	1D
temp nodes	length	defeature	ribs	point edit	2D
distance		midsurface		autocleanup	3D
points		dimensioning			Analysis
					Tool
					Post



2. 요소 생성

masses	bars	connectors	line mesh	edit element	Geom
joints	rods	spotweld	linear 1d	split	1D
markers	rigids	HyperBeam		replace	2D
	rbe3			detach	3D
	springs		vectors	order change	Analysis
	gaps		systems	config edit	Tool
				elem types	Post



3. 하중/경계 조건 입력

vectors	load types	accels	interfaces	control cards	Geom
systems	constraints	rigid wells	entity sets	output block	1D
preserve node	equations	temperatures	blocks	loadsteps	2D
	forces	flux	contactsurfs	optimization	3D
	moments	load on geom	bodies		Analysis
	pressures		nsm	OptStruct	Tool
					Post



3. 해석 정의 및 실행

vectors	load types	accels	interfaces	control cards	Geom
systems	constraints	rigid wells	entity sets	output block	1D
preserve node	equations	temperatures	blocks	loadsteps	2D
	forces	flux	contactsurfs	optimization	3D
	moments	load on geom	bodies		Analysis
	pressures		nsm	OptStruct	Tool
					Post

빔 해석 예제

외팔보 예제



빔 단면 정보
: W360 X 101

Name	Value
Name	beamsection1
ID	1
Include	[Master Model]
Collector	(1) beamsectcol1
Config	Standard
Section Type	I
Parameter Definitions	
Dimension DIM1	357.0
Dimension DIM2	255.0
Dimension DIM3	255.0
Thickness DIM4	10.5
Thickness DIM5	18.3
Thickness DIM6	18.3

기하형상

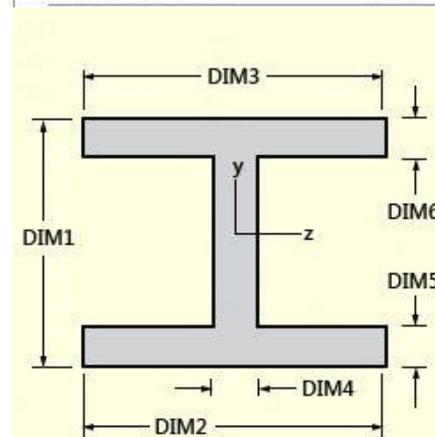
- L = 5000 mm
- 재료 : alloy steel
- E = 210 GPa
- ν = 0.28
- 굽힘 하중: 200 kN

이론 해

$$\sigma = \frac{My}{I} = 601.607 \text{ MPa}$$

$$y_{\max} = \frac{PL^3}{3EI} + \frac{PL}{GA_s} = -137.267 \text{ mm}$$

-133.744 mm



$A_s = kA$: 유효전단면적

해석 프로세스

1. 기하형상 생성

2. 재료 물성 및 특성 입력
3. 요소망 생성
4. 구속조건 설정
5. 하중조건 설정
6. 해석케이스 정의 및 해석 실행
7. 후처리

기하형상 생성 (1)



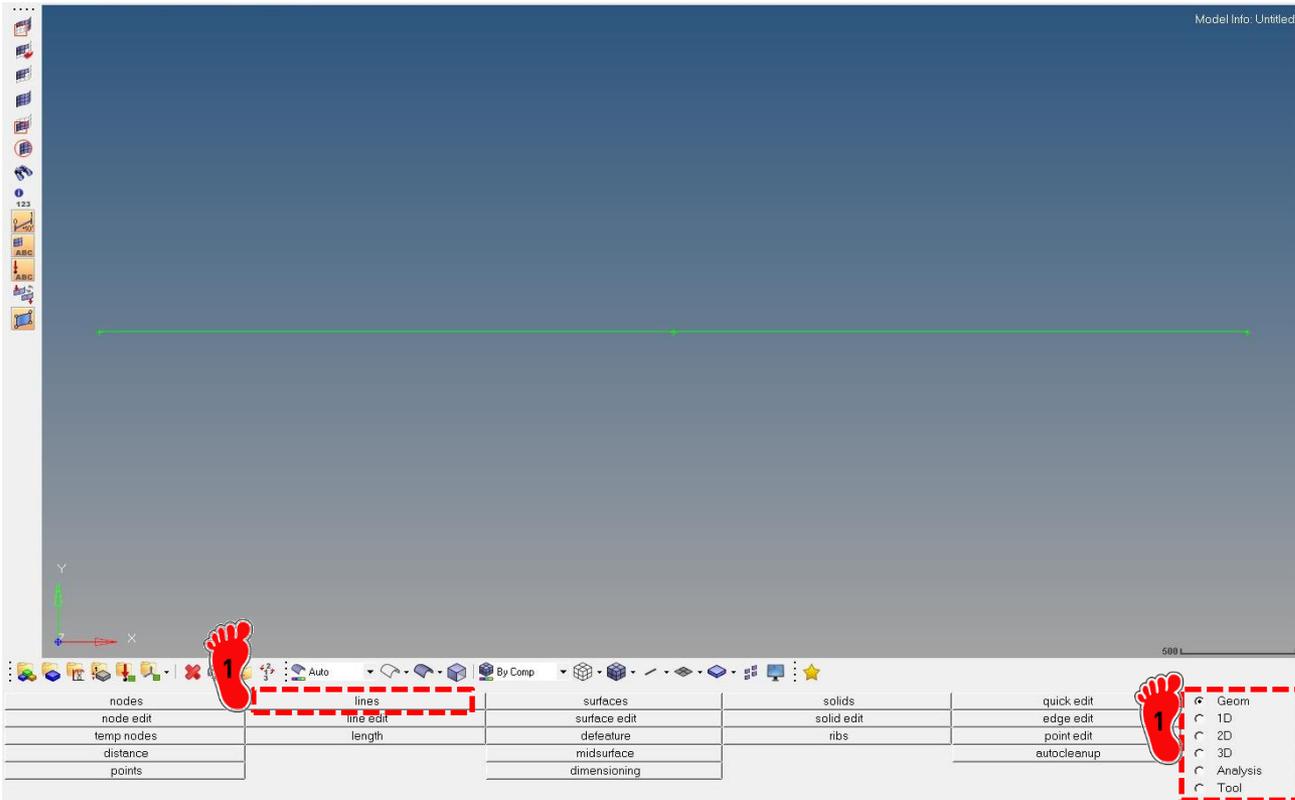
OptiStruct 선택, OK

Default (HyperMesh)
 RADIOS
 OptiStruct
 Abaqus
 Actran
 Ansys
 Exodus
 LiDyna
 Madymo
 Marc
 Nastran
 Panscrah
 Pemas
 Sancel

Application: HyperMesh
 Radios2017
 Standard3D
 Sierra_3D
 Keyword971_B3.0
 Madymo70
 Marc3D
 NastranHSC
 Panscrah2016

Always show at start-up
 OK Cancel

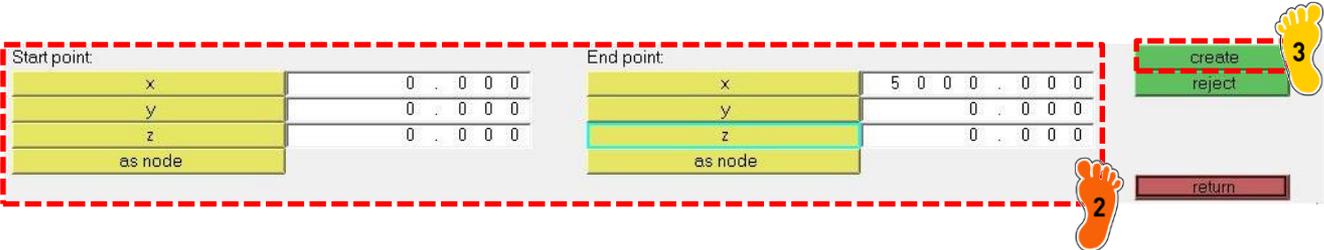
기하형상 생성 (2)



1 'Geom' > 'lines' 클릭

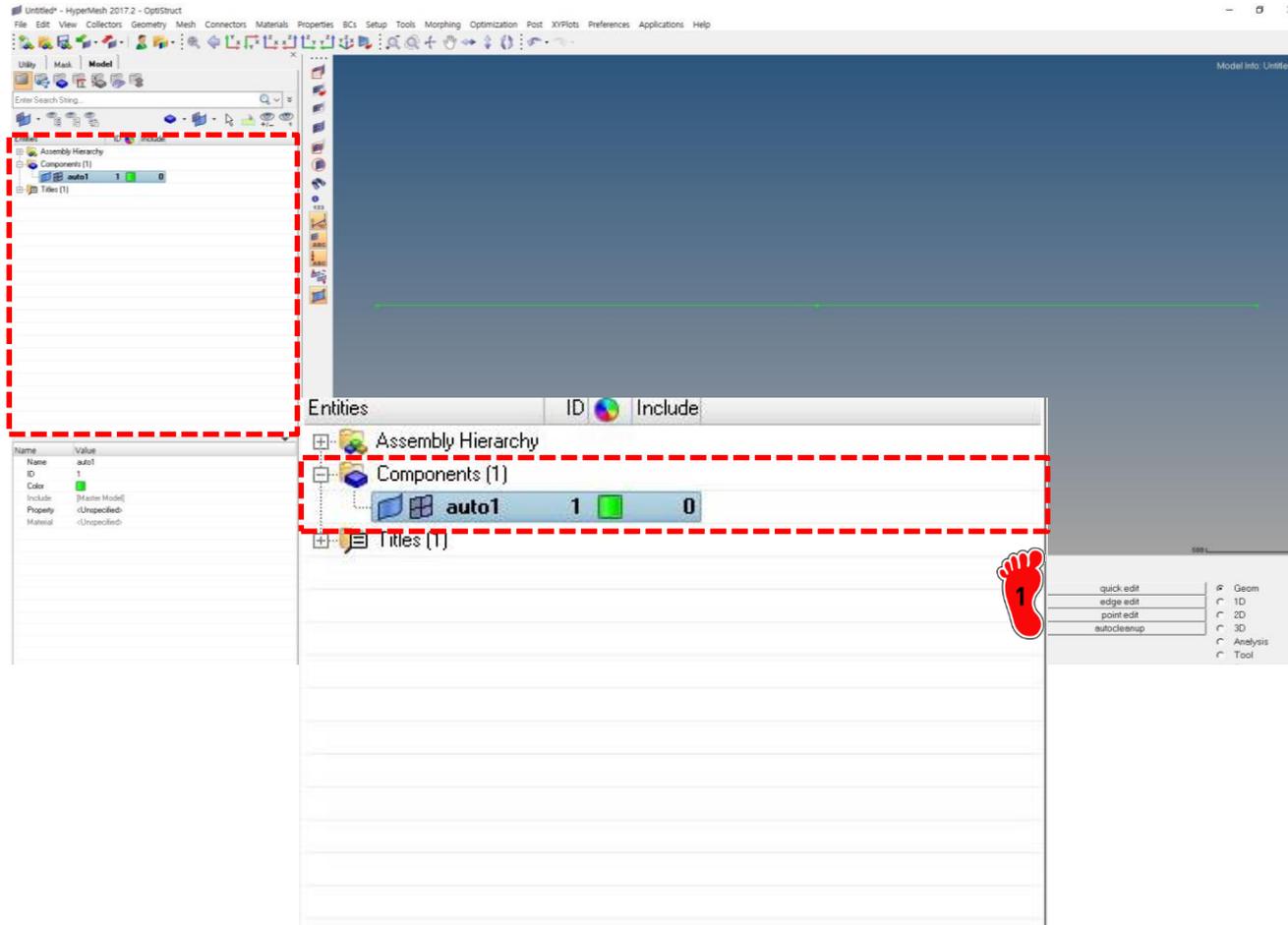
2 시작점 위치 (0,0,0), 끝점 위치 (5000,0,0) 입력
(길이 기본 단위는 mm)

3 생성



기하형상 생성 (3)

1 작업 트리의 Components 항목에서 기하형상 생성된 것을 확인



Quantity	Symbol	Dimension	SI-System (MKS)	System mm-t-s		System mm-kg-ms	
				Unit	Mult.	Unit	Mult.
Length	l	L	m	mm	10 ³	mm	10 ³
Mass	m	M	kg	t (tonne)	10 ³	kg	1
Time	t	T	s	s	1	ms	10 ³
Temperature	T	degrees	K	K	1	K	1
Work, Energy	W, E	ML ² T ⁻²	J=Nm=W · s	mJ	10 ³	J	1
Acceleration	a	LT ⁻²	m · s ⁻²	mm · s ⁻²	10 ³	mm · ms ⁻²	10 ⁻³
Area	A	L ²	m ²	mm ²	10 ⁶	mm ²	10 ⁶
Frequency	f	T ⁻¹	Hz=s ⁻¹	Hz=s ⁻¹	1	ms ⁻¹	10 ⁻³
Velocity	v	LT ⁻¹	m · s ⁻¹	mm*s ⁻¹	10 ³	mm · ms ⁻¹	1
Volume	V	L ³	m ³	mm ³	10 ⁹	mm ³	10 ⁹
Angular Acceleration	α	T ⁻²	rad · s ⁻² =s ⁻²	rad · s ⁻² =s ⁻²	1	rad · ms ⁻² =ms ⁻²	10 ⁻⁶
Angular Velocity	ω	T ⁻¹	rad · s ⁻¹ =s ⁻¹	rad · s ⁻¹ =s ⁻¹	1	rad · ms ⁻¹ =ms ⁻¹	10 ⁻³
Density	ρ	ML ⁻³	kg · m ⁻³	t · mm ⁻³	10 ⁻¹²	kg · mm ³	10 ⁻⁹
Pressure, Stress, Young's Modulus	p, σ, τ, E	ML ⁻¹ T ⁻²	Pa=N · m ⁻²	MPa=N · mm ⁻²	10 ⁶	GPa=kN · mm ²	10 ⁻⁹
Force	F	MLT ⁻²	N=kg · m · s ⁻²	N	1	kN	10 ⁻³
Moment	M	ML ² T ⁻²	N · m	N · mm	10 ³	kN · mm	1
Stiffness	c	MT ⁻²	N · m ⁻¹	N · mm ⁻¹	10 ⁻³	kN · mm ⁻¹	10 ⁻⁶

해석 프로세스

1. 기하형상 생성
- 2. 재료 물성 및 특성 입력**
3. 요소망 생성
4. 구속조건 설정
5. 하중조건 설정
6. 해석케이스 정의 및 해석 실행
7. 후처리

재료 물성 및 특성 입력 (1)

The screenshot shows the HyperMesh 2017.2 interface. The 'Materials' menu is open, and the 'Create' option is selected. The 'Create material' dialog box is displayed, showing the following fields and values:

Name	Value
Solver Keyword	MAT1
Name	material1
ID	1
Color	[Blue]
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	MAT1
User Comments	
E	210000.0
G	
NU	0.28
RHO	
A	
TREF	
GE	
ST	
SC	
SS	
MATS1	<input type="checkbox"/>
MATT1	<input type="checkbox"/>
MAT4	<input type="checkbox"/>
MAT5	<input type="checkbox"/>
MATFAT	<input type="checkbox"/>
MATF1	<input type="checkbox"/>
MATX...	<input type="checkbox"/>

The dialog box also includes a 'Close' button at the bottom right.

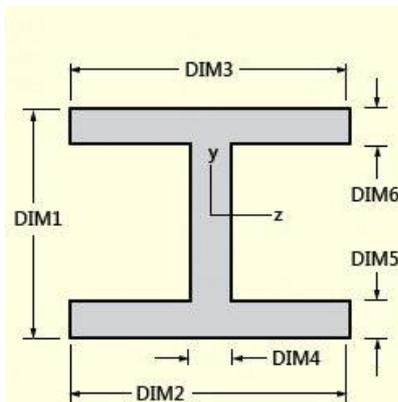
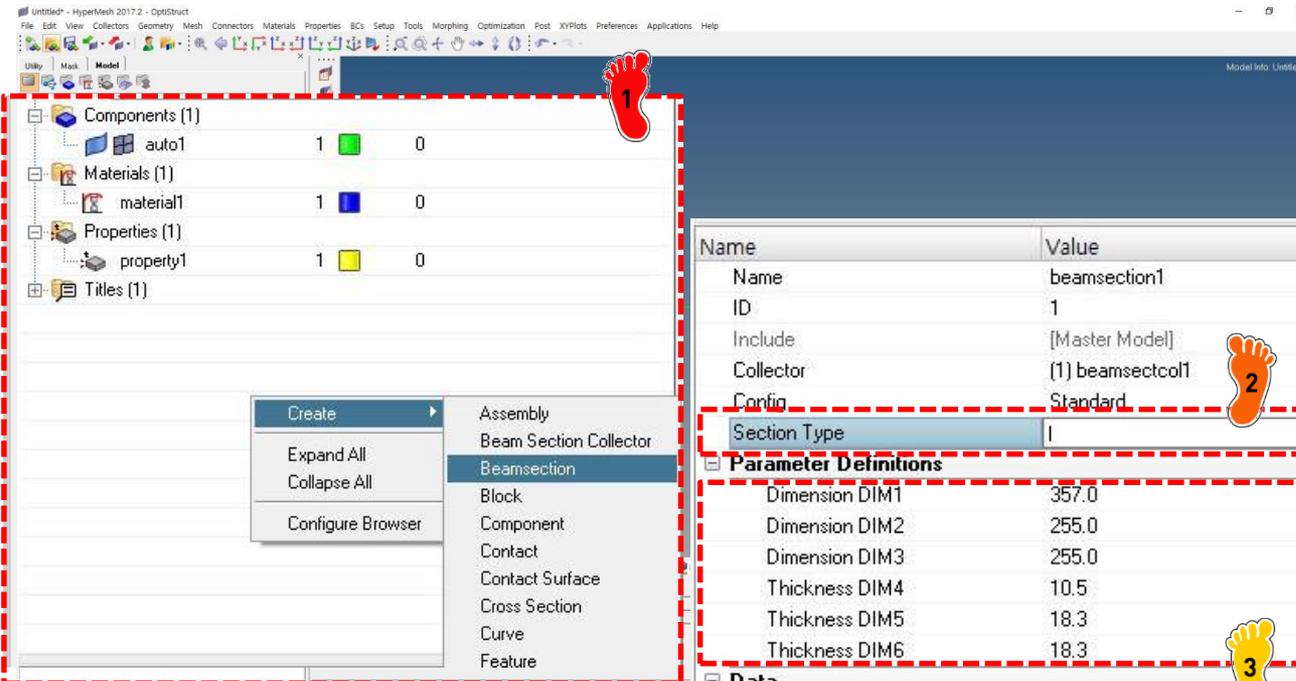
1 주 메뉴 창에서 'Materials' 탭 메뉴 클릭, 'Create' 클릭

2 재료 물성치 입력 화면

3 'Card Image' MAT1 선택

4 현재 예제는 Alloy Steel 과 동일한 물성치를 가지므로 해당 값 입력

재료 물성 및 특성 입력 (2)



1 작업 트리 창에서 마우스 우클릭, 'Create' > 'Beamsection' 클릭

2 'Section Type' 항목에서 'I' 선택

3 앞서 주어진 단면 형상 치수들 입력 ('Section Type'에 마우스를 두면 형상 확인 가능)

재료 물성 및 특성 입력 (3)

1 주 메뉴 창에서 'Properties' 탭 메뉴 클릭, 'Create' 클릭

2 'Card Image' 항목에서 'PBARL' 선택 ('PBAR'은 축 방향 응력 해석 결과를 포함하지 않음)

3 'Material' 항목에 '<Unspecified>' 클릭하여 생성한 재료 적용

'Beam Section' 항목에 '<Unspecified>' 클릭하여 생성한 단면 형상 적용

Name	Value
Solver Keyword	PBARL
Name	property1
ID	1
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	PBARL
Material	<Unspecified>
User Comments	Hide In Menu/Export
Beam Section	<Unspecified>
GROUP	
CStype	I
No of rows	6
Data: DIMs	
NSM	

Name	Value
Solver Keyword	PBARL
Name	property1
ID	1
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	PBARL
Material	(1) material1
User Comments	Hide In Menu/Export
Beam Section	Beamsection
GROUP	
CStype	I
No of rows	6
Data: DIMs	
NSM	

Name	ID
beamsection1	1

재료 물성 및 특성 입력 (4)

The screenshot shows the HyperMesh 2017.2 interface. The 'Assembly Hierarchy' tree on the left is highlighted with a red dashed box and a red footprint icon labeled '1'. The tree structure is as follows:

- beamsection1 (ID: 1, Value: 0)
- Components (1)
 - auto1 (ID: 1, Value: 0)
- Materials (1)
 - material1 (ID: 1, Value: 0)
- Properties (1)
 - property1 (ID: 1, Value: 0)
- Titles (1)

The 'Select Property' dialog box is open, also highlighted with a red dashed box and a red footprint icon labeled '2'. It displays a table with the following data:

Name	ID	Color	Card Image	Defined
property1	1		PBARL	<input checked="" type="checkbox"/>

Below the dialog box, the 'Property' card is visible in the main window, showing the following details:

Name	Value
Name	auto1
ID	1
Color	
Include	[Master Model]
Property	Property
Material	(1) material1

1 작업 트리 창에서
'Components' > 생성한
기하형상 선택

2 'Property' 항목에서 생성한
'Property1' 적용

TYPES OF 1D ELEMENTS

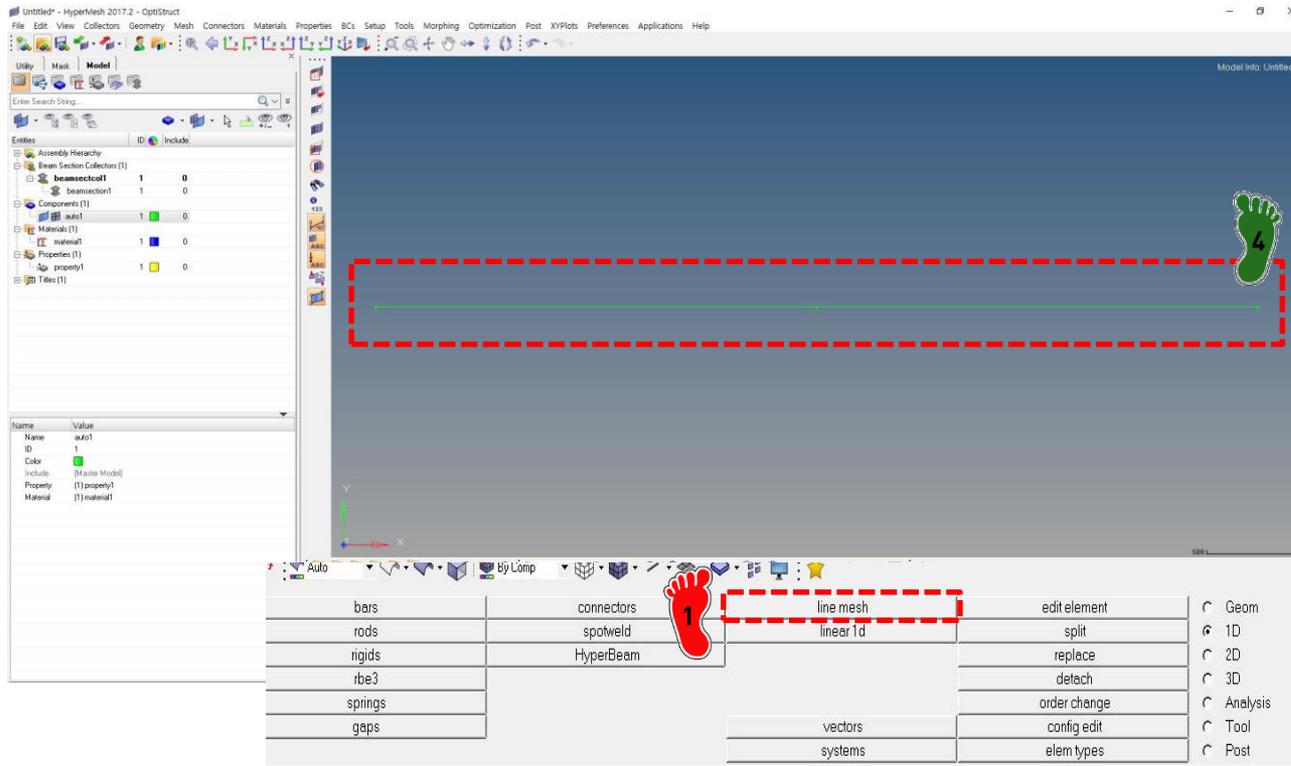
Types Of 1D Elements:

Rod	Bar	Beam	Pipe	Axisymmetric shell
Tension/ compression (and Torque for some software) U_x, R_x (1,4)	All 6 dofs $U_x, U_y, U_z, R_x, R_y, R_z$ (123456) Applicable for symmetric c/s	Same as bar but also support unsymmetric sections i.e. shear center and warpage	Same as beam. Except it has internal non zero diameter	U_x, U_z, R_y (1,3,5) Z- axis of symmetry, X as radial axis. For objects symmetric about the axis of rotation and subjected to the axisymmetric boundary conditions.
Tension compression members (truss), Shafts subjected to Torque, Connection elements	Shaft subjected to multiaxial loading, bolted, welded joints, connection elements	Same as bar + for unsymmetric c/s	Piping systems, Structural analysis	Thin shell pressure vessels, cylindrical, conical objects etc.

해석 프로세스

1. 기하형상 생성
2. 재료 물성 및 특성 입력
- 3. 요소망 생성**
4. 구속조건 설정
5. 하중조건 설정
6. 해석케이스 정의 및 해석 실행
7. 후처리

요소망 생성 (1)

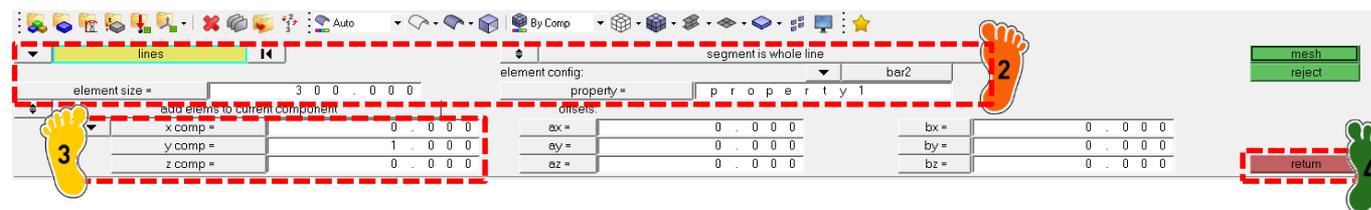


1 '1D' > 'line mesh' 클릭

2 'lines' 선택, 'bar2' 확인,
'element size' 300mm 설정,
Property 설정

3 Components (0,1,0) 입력
(Beam section의 y축 설정)

4 'mesh' 클릭 후 'return'



요소망 생성 (2)

The screenshot shows a CAD software interface with a 3D model area at the top and a command palette at the bottom. A red dashed box highlights the 'numbers' menu and the '1D Detailed Element Representation' tool. A yellow footprint icon labeled '3' is on the 3D model, and a red footprint icon labeled '1' is on the 'numbers' menu. A blue footprint icon labeled '5' is on the '1D Detailed Element Representation' tool. An orange footprint icon labeled '2' is on the 'elems' list, and a green footprint icon labeled '4' is on the 'on/off/all off' buttons.

translate	check elems	numbers	Geom
rotate	edges	renumber	1D
scale	faces	count	2D
reflect	features	mass calc	3D
project	normals	tags	Analysis
position	dependency	HyperMorph	Tool
permute	penetrate		Post

elems

display

on
off
all off

return

1 'Tool' > 'numbers' 클릭

2 'element' 선택

3 요소망 선택

4 'on' 클릭하여 요소망 번호 확인

5 bar 단면 형상 표시 기능

해석 프로세스

1. 기하형상 생성
2. 재료 물성 및 특성 입력
3. 요소망 생성
- 4. 구속조건 설정**
5. 하중조건 설정
6. 해석케이스 정의 및 해석 실행
7. 후처리

구속조건 설정

1 주 메뉴 창에서 'BCs' > 'Create' > 'Constraints' 클릭

2 원점에 있는 절점 선택

3 모든 자유도 구속

4 'create' 클릭 후 'return'

5 구속조건 이름 변경

해석 프로세스

1. 기하형상 생성
2. 재료 물성 및 특성 입력
3. 요소망 생성
4. 구속조건 설정
- 5. 하중조건 설정**
6. 해석케이스 정의 및 해석 실행
7. 후처리

하중조건 설정

1 'Load Collectors' 우클릭, 'Create'.

2 주 메뉴 창에서 'BCs' > 'Create' > 'Forces' 클릭

3 하중을 적용할 절점 선택

4 하중 값 -200kN 입력, 'create' 클릭, 'return'

해석 프로세스

1. 기하형상 생성
2. 재료 물성 및 특성 입력
3. 요소망 생성
4. 구속조건 설정
5. 하중조건 설정

6. 해석케이스 정의 및 해석 실행

7. 후처리

해석 케이스 정의 및 해석 실행 (1)

1

2

Name	Value
Solver Keyword	SUBCASE
Name	loadstep1
ID	1
Include	[Master Model]
User Comments	Hide in Menu/Export
Subcase Definition	
Analysis type	Linear Static
SPC	(1) spc
LOAD	(2) force
SUPPORT1	<Unspecified>
PRETENSION	<Unspecified>
MPC	<Unspecified>
DEFORM	<Unspecified>
STATSUB (PRELOAD)	<Unspecified>
STATSUB (PRETENS)	<Unspecified>
SUBCASE OPTIONS	
LABEL	<input type="checkbox"/>
SUBTITLE	<input type="checkbox"/>
ANALYSIS	<input checked="" type="checkbox"/>
TYPE	STATICS

1 작업 트리 창 우클릭, 'Create' > 'Load Step' 클릭

2 'Analysis type', 'SPC', 'LOAD' 항목 설정

해석 케이스 정의 및 해석 실행 (2)

1 'Analysis' > 'OptiStruct' 클릭

2 해석파일(.fem)저장 경로 설정, 'run options' 항목 'analysis' 선택

3 'OptiStruct' 클릭

4 해석 완료 확인 후 'Results' 클릭

Solver: optistruct_2017.2_win64.exe
 Input file: bar1.fem Job completed

Run command: .../hwsolver.tcl -solver OS -screen .../bar1.fem -analysis -optskip

Message log:
 Messages for the job:
 ANALYSIS COMPLETED.

Iteration	Subcase	Variable	Grid/Elem ID	Value
0	1	MaxDisp	11_Y	-137.368

Run summary:
 ** Contains trade secrets of Altair Engineering, Inc. **
 ** Decompilation or disassembly of this software strictly prohibited. **

The amount of memory allocated for the run is 800 MB.
 This run will use in-core processing in the solver.

ANALYSIS COMPLETED.

==== End of solver screen output ====

==== Job completed ====

- control cards
- output block
- loadsteps
- optimization
- OptiStruct
- Geom
- 1D
- 2D
- 3D
- Analysis
- Tool
- Post

input file: C:\Users\cdl\Desktop\HW\day1\bar1.fem

export options: all

run options: analysis

memory options: memory default

opt - o p t s k i p

OptiStruct

HyperView

view out

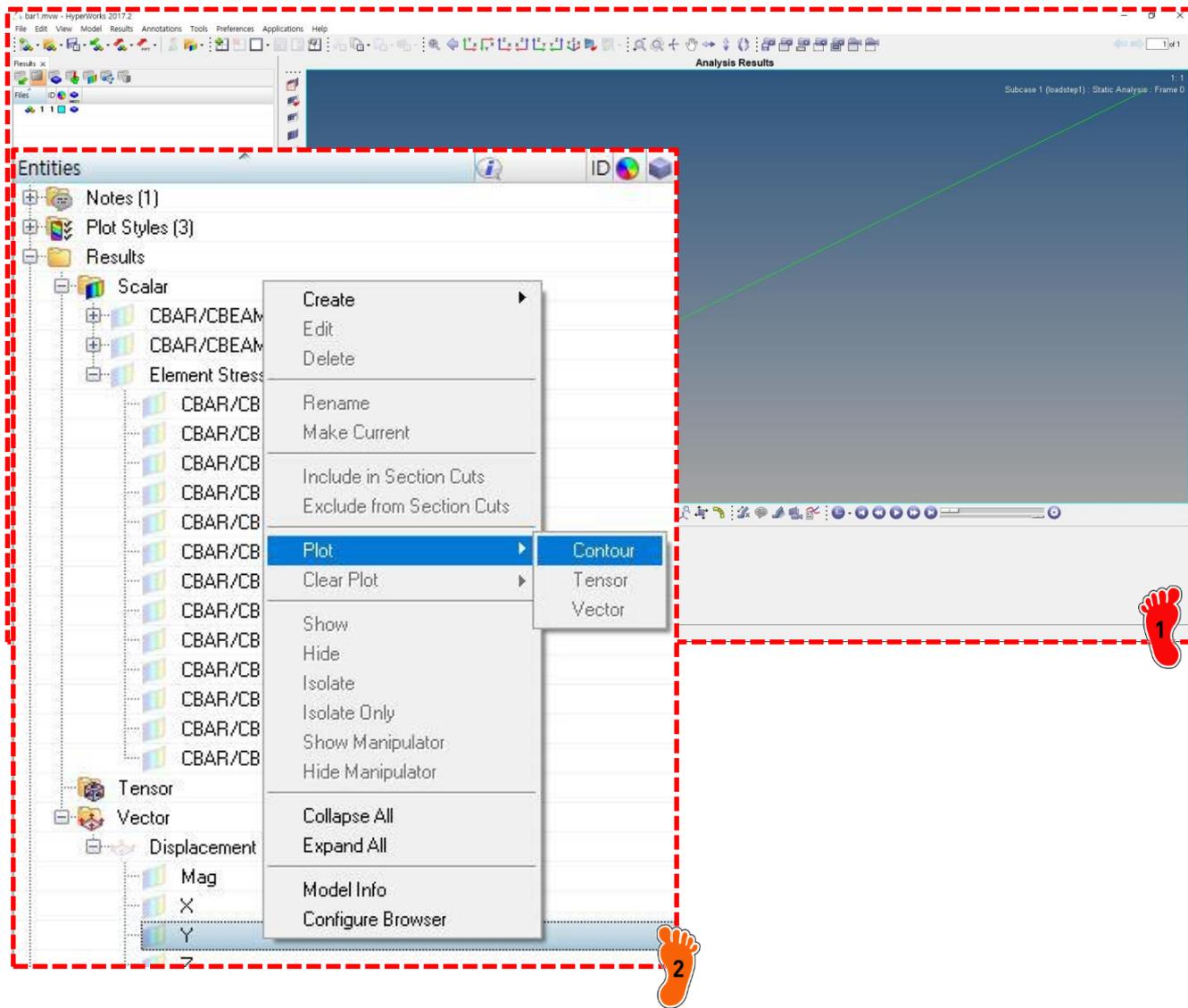
return

해석 프로세스

1. 기하형상 생성
2. 재료 물성 및 특성 입력
3. 요소망 생성
4. 구속조건 설정
5. 하중조건 설정
6. 해석케이스 정의 및 해석 실행

7. 후처리

후처리 (1)



1 .mvw 파일 자동저장 후 결과 창이 열림

변위, 응력 등의 결과값 확인 가능

2 원하는 결과값 우클릭 후 'Plot' > 'Contour' 클릭

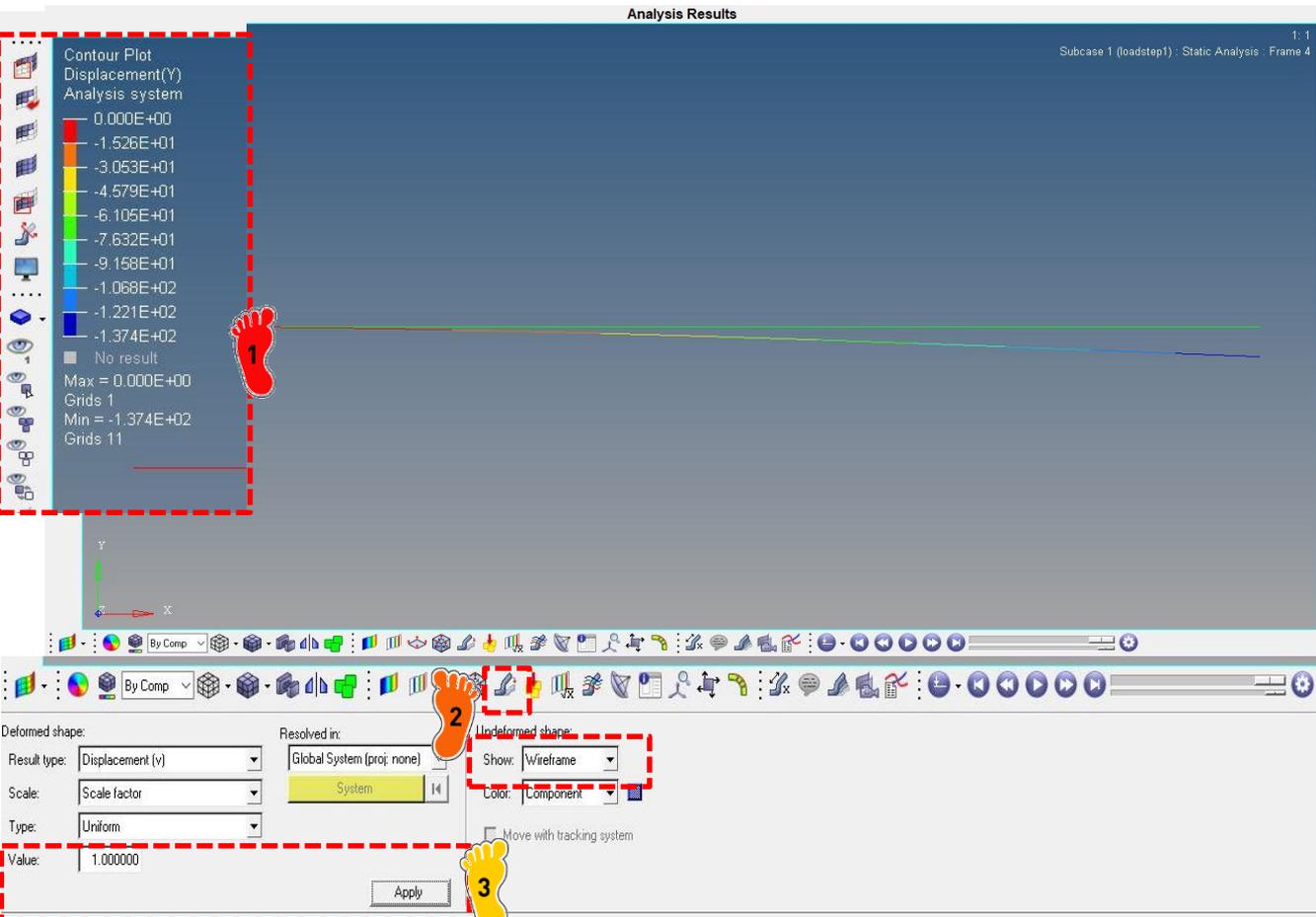
후처리 (2)

1 Y방향 최대 변위 결과값은 -137.4 mm 확인

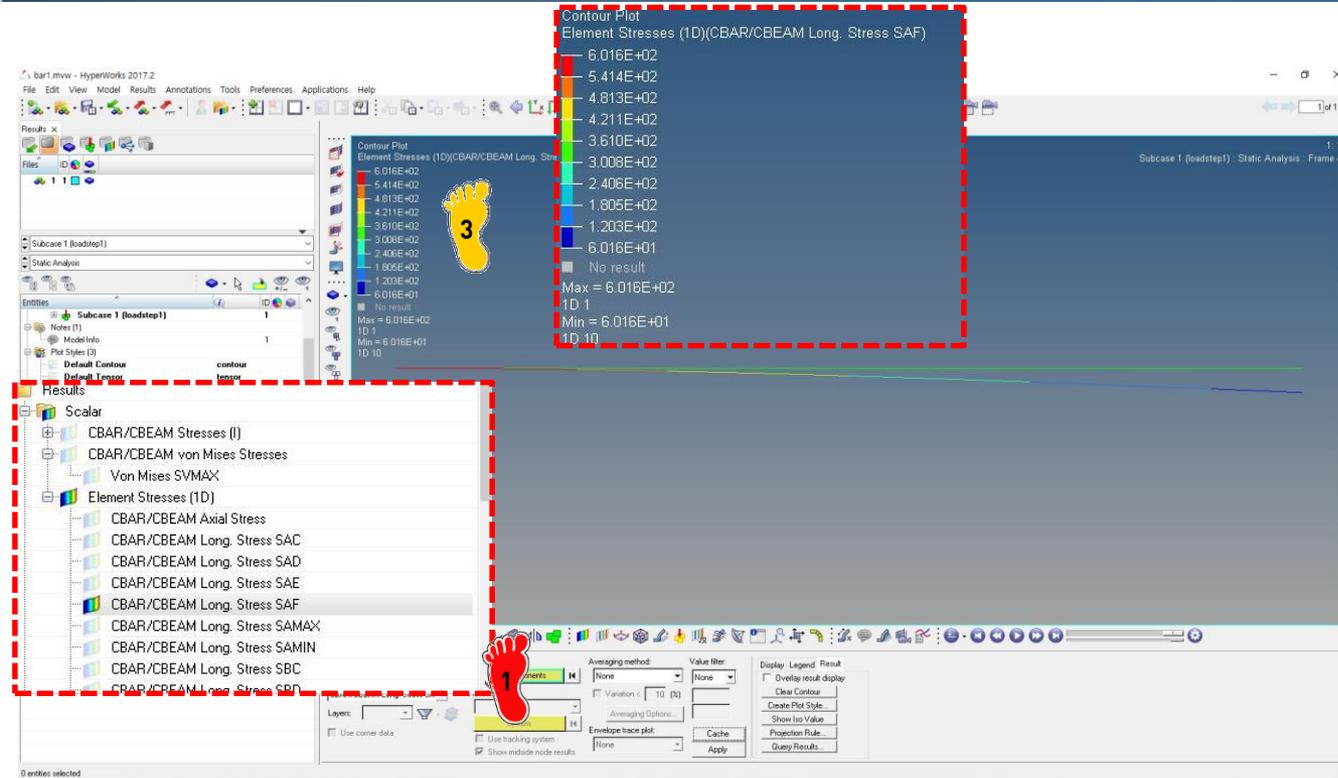
이론값과의 오차는 0.002%

2 'Deformation' 클릭, 'Wireframe' 선택

3 Value값 조절하여 시각적 효과 적용



후처리 (3)



1 바요소 F지점 축 방향 응력 결과 우클릭, 'Plot' > 'Contour'

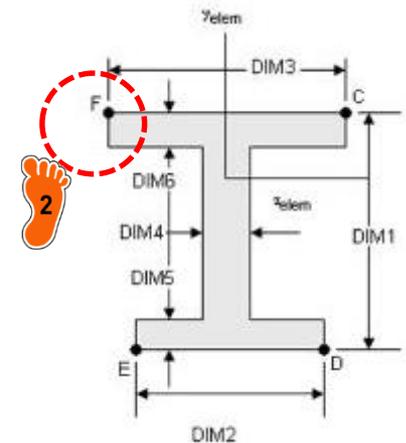
2 F 지점의 의미는 단면 형상을 입력할 때 나오는 F 를 의미함

3 결과값은 601.6 MPa 이론값과의 오차는 0.00% 확인

- Stress SAC - The longitudinal (extensional) stress at end A for point C.
- Stress SAD - The longitudinal (extensional) stress at end A for point D.
- Stress SAE - The longitudinal (extensional) stress at end A for point E.
- Stress SAF - The longitudinal (extensional) stress at end A for point F.
- Stress SBC - The longitudinal (extensional) stress at end B for point C.
- Stress SBD - The longitudinal (extensional) stress at end B for point D.
- Stress SBE - The longitudinal (extensional) stress at end B for point E.
- Stress SBF - The longitudinal (extensional) stress at end B for point F.

- Stress SAMIN - Minimum value of longitudinal stress over points C, D, E and F at end A.
- Stress SAMAX - Maximum value of longitudinal stress over points C, D, E and F at end A.

- Stress SBMIN - Minimum value of longitudinal stress over points C, D, E and F at end B.
- Stress SBMAX - Maximum value of longitudinal stress over points C, D, E and F at end B.



Type = I

(단면 형상 정보: PBEAML Help 참고)

참고: 단면 형상 정보 (1)

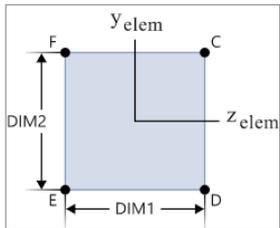


Figure 1. TYPE = BAR

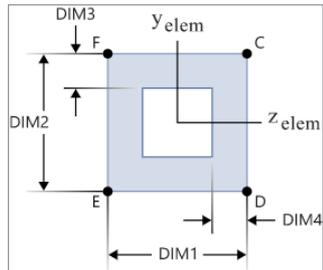


Figure 2. TYPE = BOX

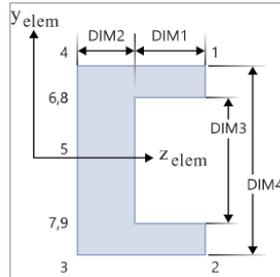


Figure 5. TYPE = CHAN1

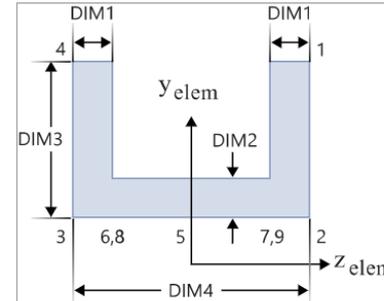


Figure 6. TYPE = CHAN2

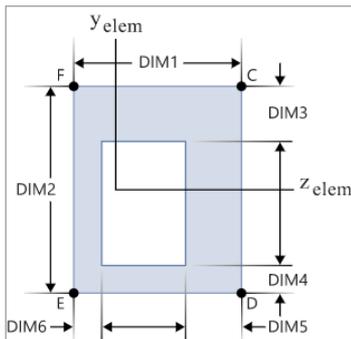


Figure 3. TYPE = BOX1

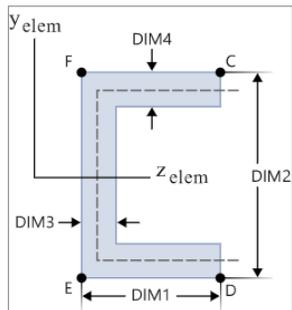


Figure 4. TYPE = CHAN

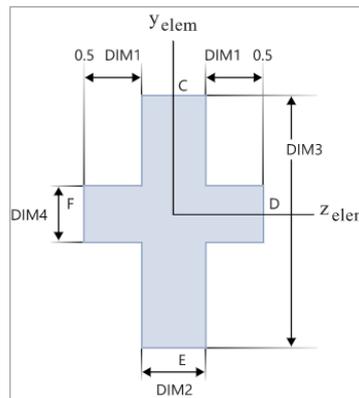


Figure 7. TYPE = CROSS

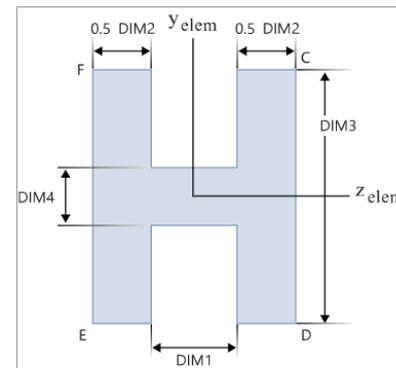


Figure 8. TYPE = H

참고: 단면 형상 정보 (2)

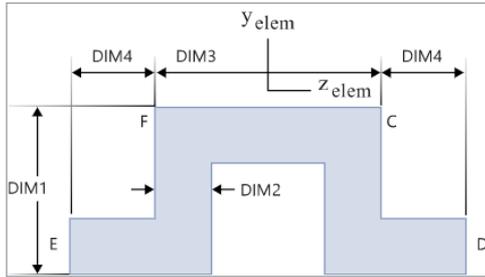


Figure 9. TYPE = HAT

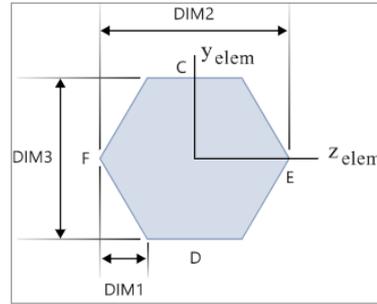


Figure 10. TYPE = HEXA

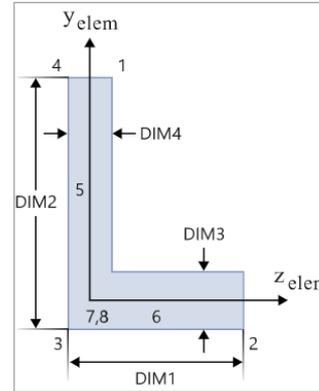


Figure 13. TYPE = L

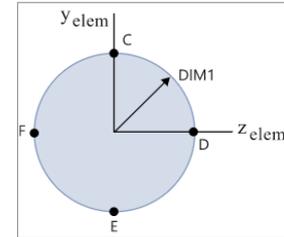


Figure 14. TYPE = ROD

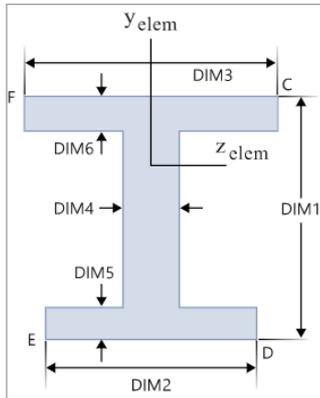


Figure 11. TYPE = I

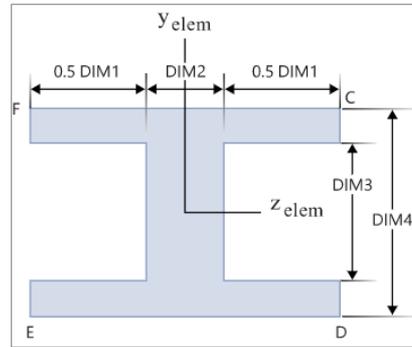


Figure 12. TYPE = I1

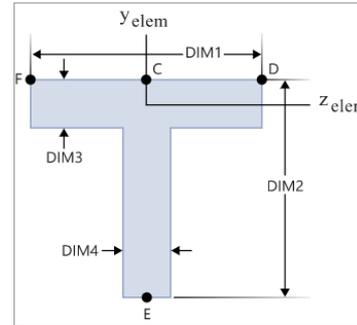


Figure 15. TYPE = T

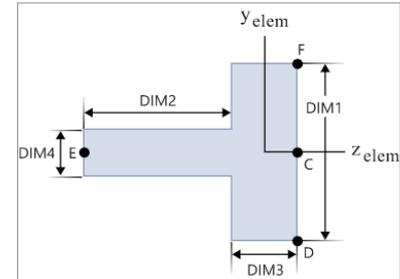


Figure 16. TYPE = T1

참고: 단면 형상 정보 (3)

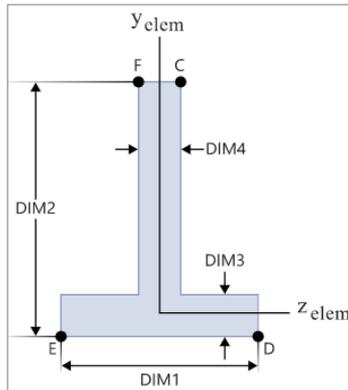


Figure 17. TYPE = T2

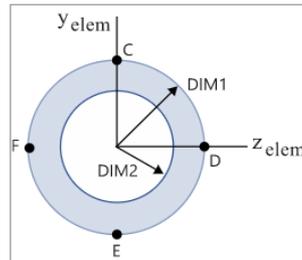


Figure 18. TYPE = TUBE

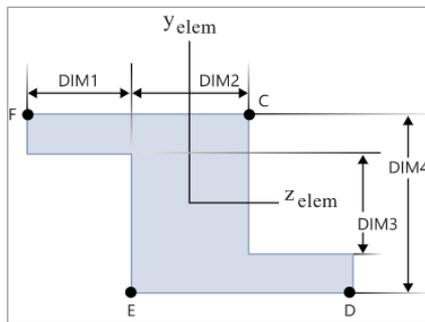


Figure 19. TYPE = Z

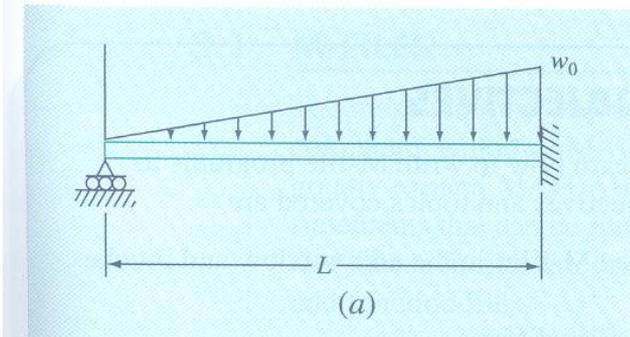
연습문제

2.21 Figure P2.21a shows a uniform beam subject to a linearly increasing distributed load. As depicted in Fig. P2.21b, deflection y (m) can be computed with

$$y = \frac{w_0}{120EI}(-x^5 + 2L^2x^3 - L^4x)$$

where E = the modulus of elasticity and I = the moment of inertia (m^4). Employ this equation and calculus to generate MATLAB plots of the following quantities versus distance along the beam:

- (a) displacement (y),
- (b) slope [$\theta(x) = dy/dx$],



- (c) moment [$M(x) = EId^2y/dx^2$],
- (d) shear [$V(x) = EId^3y/dx^3$], and
- (e) loading [$w(x) = -EId^4y/dx^4$].

Use the following parameters for your computation: $L = 600$ cm, $E = 50,000$ kN/cm², $I = 30,000$ cm⁴, $w_0 = 2.5$ kN/cm, and $\Delta x = 10$ cm. Employ the subplot

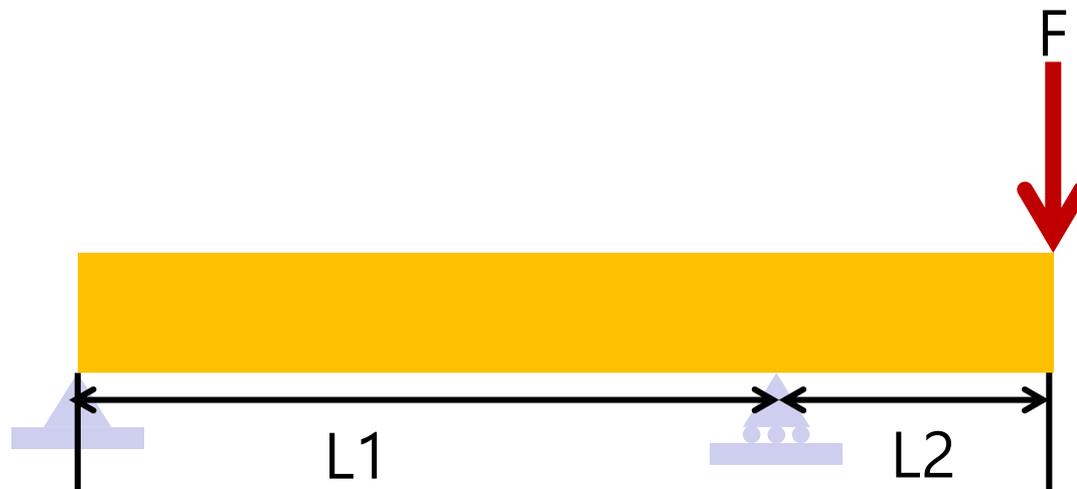
$$\begin{aligned} L &= 6000 \text{ mm} \\ E &= 500,000 \text{ MPa} \\ I &= 300,000,000 \text{ mm}^4 \\ w_0 &= 2500 \text{ N/mm} \end{aligned}$$

Beam section: ROD (Radius: 139.8 mm)

$$y_{\max} = y\left(\frac{L}{\sqrt{5}} = 268.328\right) = 0.515 \text{ cm}$$

숙제

이론 해와 유한요소 해를 비교하고 오차를 분석하시오



빔 단면 정보
: W360 X 101

기하형상

- $L1 = 4500$ mm
- $L2 = 1200$ mm

재료 : alloy steel

- $E = 210$ GPa
- $\nu = 0.28$

굽힘 하중: 200kN

이론 해

$$y_{\max} = ?$$

과제제출 <ftp://cdl.hanyang.ac.kr>→cdl/cdl
→ VehicleStructure→lab