

Solid Mechanics (beam)

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



CONTENTS

- **Beam model: 2D**
 - ✓ Textbook of “Solid Mechanics”
- **Beam model: 3D**
- **Assignment**

STRUCTURAL MECHANICS MODULE

PHYSICS	DEFAULT NAME	DEPENDENT VARIABLES	GEOMETRY LEVEL			
			POINTS	EDGES	BOUNDARIES	DOMAINS
STRUCTURAL MECHANICS						
Solid Mechanics	solid	$\mathbf{u}, (p)$	✓	✓	✓	✓
Shell	shell	$\mathbf{u}, a_x, a_y, a_z$	✓	✓	✓	
Plate	plate	$\mathbf{u}, a_x, a_y$	✓		✓	✓
Beam	beam	$\mathbf{u}, \theta$	✓	✓		
Truss	truss	$\mathbf{u}$	✓	✓		
Membrane	mem	$\mathbf{u}$	✓	✓	✓	
Thermal Stress	ts	$\mathbf{u}, (p), T$	✓	✓	✓	✓
Joule Heating and Thermal Expansion	tem	$\mathbf{u}, (p), T, V$	✓	✓	✓	✓
Piezoelectric Devices	pzd	$\mathbf{u}, V$	✓	✓	✓	✓
Fluid Flow						
Fluid-Structure Interaction	fsi	$\mathbf{u}_{\text{solid}}, \mathbf{u}_{\text{fluid}}, p$	✓	✓	✓	✓

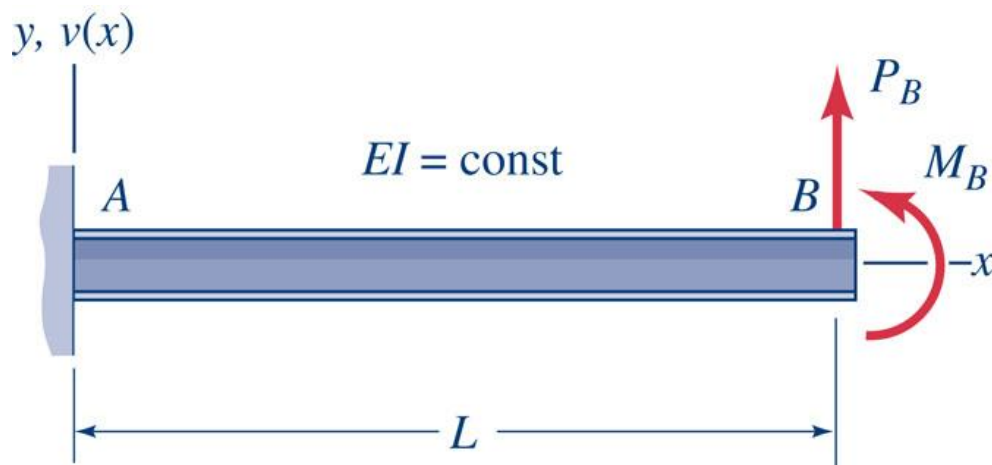
- Beam model: 2D
 - ✓ Textbook of “Solid Mechanics”
- Beam model: 3D
- Assignment

BEAMS

BEAMS

The Beam interface () is intended for the modeling of slender structures (beams) that can be fully described by cross-section properties, such as area and moments of inertia. The Beam interface defines stresses and strains using **Hermitian elements** and **Euler-Bernoulli theory**. Beam elements are used to model frame structures, both planar and three-dimensional. It is also suitable for modeling reinforcements of solid and shell structures. The Beam interface includes a library for rectangular, box, circular, pipe, H-profile, U-profile, and T-profile beam sections. Additional features include damping, thermal expansion, and initial stresses and strains. The preset studies for this physics interface are almost the same as for the Solid Mechanics interface, with two exceptions—it does not include the Linear Buckling or Prestressed study types.

EXAMPLE 7.1 (PP.471)



Analytical solution

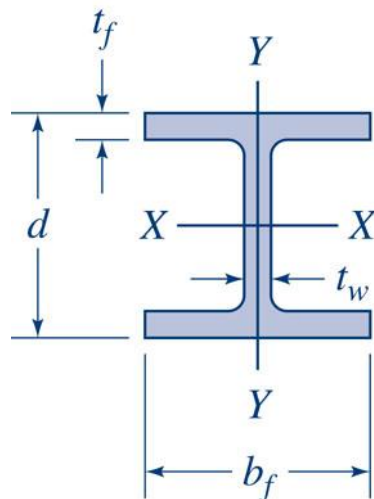
- Geometry
 - $L = 10$ ft
 - $P_B = 4.5$ kips
 - $M_B = 50$ kips-in
- Material Properties
 - A-36 steel
 - $E = 29 \times 10^3$ ksi
- Element Properties
 - Wide-flange beam
 - W10x30
 - $I = 170$ in⁴

kips: kilo pounds

ksi: kilo pounds per square inch

$$\begin{cases} \delta_B = \frac{1}{EI} \left[M_B \left(\frac{L^2}{2} \right) + P_B \left(\frac{L^3}{3} \right) \right] = 0.0730 + 0.5258 = 0.5988 \text{ in} \\ \theta_B = \frac{1}{EI} \left[M_B (L) + P_B \left(\frac{L^2}{2} \right) \right] = 0.00122 + 0.00657 = 0.00779 \text{ rad} \end{cases}$$

WIDE-FLANGE BEAM

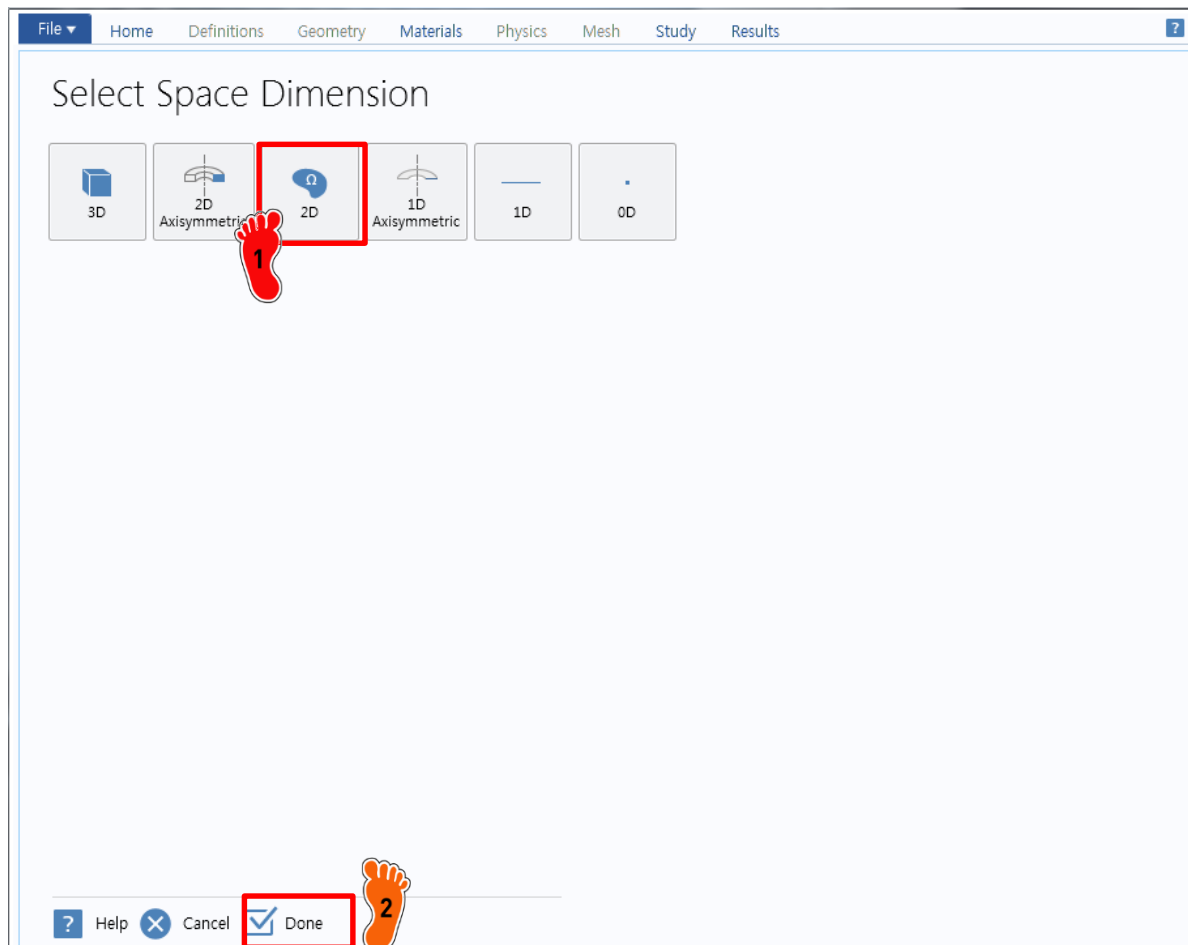


D.1. Properties of Steel Wide-Flange (W) Shapes (U.S. Customary Units)

Designation®	Area A in ²	Depth d in.	Flange		Web	Elastic Properties						Plastic Modulus Z_x in ³
			Width b_f in.	Thickness t_f in.	Thickness t_w in.	Axis $X-X$			Axis $Y-Y$			
						I_x in ⁴	S_x in ³	r_x in.	I_y in ⁴	S_y in ³	r_y in.	
W36×230	67.6	35.90	16.470	1.260	0.760	15000	837	14.9	940	114	3.73	943
×150	44.2	35.85	11.975	0.940	0.625	9040	504	14.3	270	45.1	2.47	581
W33×201	59.1	33.68	15.745	1.150	0.715	11500	684	14.0	749	95.2	3.56	772
×130	38.3	33.09	11.510	0.855	0.580	6710	406	13.2	218	37.9	2.39	467
W30×173	50.8	30.44	14.985	1.065	0.655	8200	539	12.7	598	79.8	3.43	605
×90	26.4	29.53	10.400	0.610	0.470	3620	245	11.7	115	22.1	2.09	283
W27×146	42.9	27.38	13.965	0.975	0.605	5630	411	11.4	443	63.5	3.21	461
×84	24.8	26.71	9.960	0.640	0.460	2850	213	10.7	106	21.2	2.07	244
W24×94	27.7	24.31	9.065	0.875	0.515	2700	222	9.87	109	24.0	1.98	254
×62	18.2	23.74	7.040	0.590	0.430	1550	131	9.23	34.5	9.80	1.38	153
W21×101	29.8	21.36	12.290	0.800	0.500	2420	227	9.02	248	40.3	2.89	253
×73	21.5	21.24	8.295	0.740	0.455	1600	151	8.64	70.6	17.0	1.81	172
×50	14.7	20.83	6.530	0.535	0.380	984	94.5	8.18	24.9	7.64	1.30	110
W18×130	38.2	19.25	11.160	1.200	0.670	2460	256	8.03	278	49.9	2.70	291
×76	22.3	18.21	11.035	0.680	0.425	1330	146	7.73	152	27.6	2.61	163
W18×60	17.6	18.24	7.555	0.695	0.415	984	108	7.47	50.1	13.3	1.69	123
×50	14.7	17.99	7.495	0.570	0.355	800	88.9	7.38	40.1	10.7	1.65	101
W16×100	29.4	16.97	10.425	0.985	0.585	1490	175	7.10	186	35.7	2.51	198
×67	19.7	16.33	10.235	0.665	0.395	954	117	6.96	119	23.2	2.46	130
×50	14.7	16.26	7.070	0.630	0.380	659	81.0	6.68	37.2	10.5	1.59	92.0
×40	11.8	16.01	6.995	0.505	0.305	518	64.7	6.63	28.9	8.25	1.57	72.9
W14×176	51.8	15.22	15.650	1.310	0.830	2140	281	6.43	838	107	4.02	320
×120	35.3	14.48	14.670	0.940	0.590	1380	190	6.24	495	65.7	3.74	212
×82	24.1	14.31	10.130	0.855	0.510	882	123	6.05	148	29.3	2.48	139
×53	15.6	13.92	8.060	0.660	0.370	541	77.8	5.89	57.7	14.3	1.92	87.1
×26	7.69	13.91	5.025	0.420	0.255	245	35.3	5.65	8.91	3.54	1.08	40.2
W12×152	44.7	13.71	12.480	1.400	0.870	1430	209	5.66	454	72.8	3.19	243
×96	28.2	12.71	12.160	0.900	0.550	833	131	5.44	270	44.4	3.09	147
×65	19.1	12.12	12.000	0.605	0.390	533	87.9	5.28	174	29.1	3.02	96.8
×50	14.7	12.19	8.080	0.640	0.370	394	64.7	5.18	56.3	13.9	1.96	72.4
×35	10.3	12.50	6.560	0.520	0.300	285	45.6	5.25	24.5	7.47	1.54	51.2
×22	6.48	12.31	4.030	0.425	0.260	156	25.4	4.91	4.66	2.31	0.847	29.3
W10×60	17.6	10.22	10.080	0.680	0.420	341	66.7	4.39	116	23.0	2.57	74.6
×45	13.3	10.10	8.020	0.620	0.350	248	49.1	4.32	53.4	13.3	2.01	54.9
×30	8.84	10.47	5.810	0.510	0.300	170	32.4	4.38	16.7	5.75	1.37	36.6
×12	3.54	9.87	3.960	0.210	0.190	53.8	10.9	3.90	2.18	1.10	0.785	12.6
W8×48	14.1	8.50	8.110	0.685	0.400	184	43.3	3.61	60.9	15.0	2.08	49.0
×40	11.7	8.25	8.070	0.560	0.360	146	35.5	3.53	49.1	12.2	2.04	39.8
×35	10.3	8.12	8.020	0.495	0.310	127	31.2	3.51	42.6	10.6	2.03	34.7
×21	6.16	8.28	5.270	0.400	0.250	75.3	18.2	3.49	9.77	3.71	1.26	20.4
×15	4.44	8.11	4.015	0.315	0.245	48.0	11.8	3.29	3.41	1.70	0.876	13.6
W6×25	7.34	6.38	6.080	0.455	0.320	53.4	16.7	2.70	17.1	5.61	1.52	18.9
×20	5.87	6.20	6.020	0.365	0.260	41.1	13.4	2.66	13.3	4.41	1.50	14.9
×16	4.74	6.28	4.030	0.405	0.260	32.1	10.2	2.60	4.43	2.20	0.966	11.7
×15	4.43	5.99	5.990	0.260	0.230	29.1	9.72	2.56	9.32	3.11	1.46	10.8

*W(nominal depth in inches) × (weight in pounds per foot)

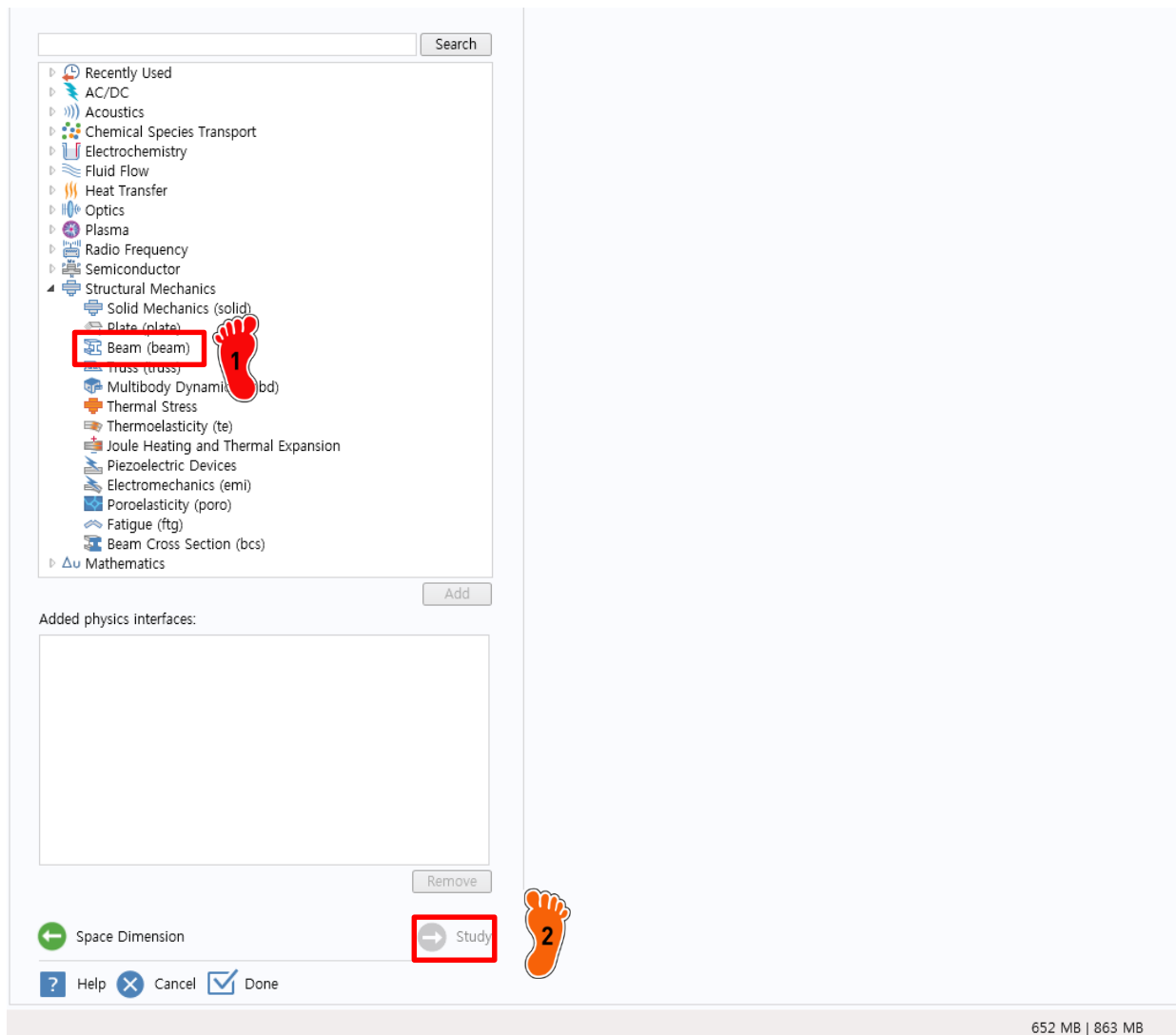
DIMENSION SELECTION



1 2D 선택

2 Done 클릭

PHYSICS SELECTION



1 Structural Mechanics 의
Beam 추가

2 Study 클릭

STUDY TYPE SELECTION

File ▾ Home Definitions Geometry Materials Physics Mesh Study Results

Select Study

Preset Studies

- Eigenfrequency
- Frequency Domain
- Frequency-Domain Modal
- Modal Body
- Stationary**
- Time Dependent
- Time-Dependent Modal

Custom Studies

- Empty Study

Added study:

Stationary

Added physics interfaces:

Beam (beam)

Physics

Help Cancel **Done**

Stationary

The Stationary study is used when field variables do not change over time.

Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to compute the temperature field at thermal equilibrium. In solid mechanics, it is used to compute deformations, stresses, and strains at static equilibrium. In fluid flow it is used to compute the steady flow and pressure fields. In chemical species transport, it is used to compute steady-state chemical composition in steady flows. In chemical reactions, it is used to compute the chemical composition at equilibrium of a reacting system.

It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.

1 Stationary 선택

2 Done 클릭

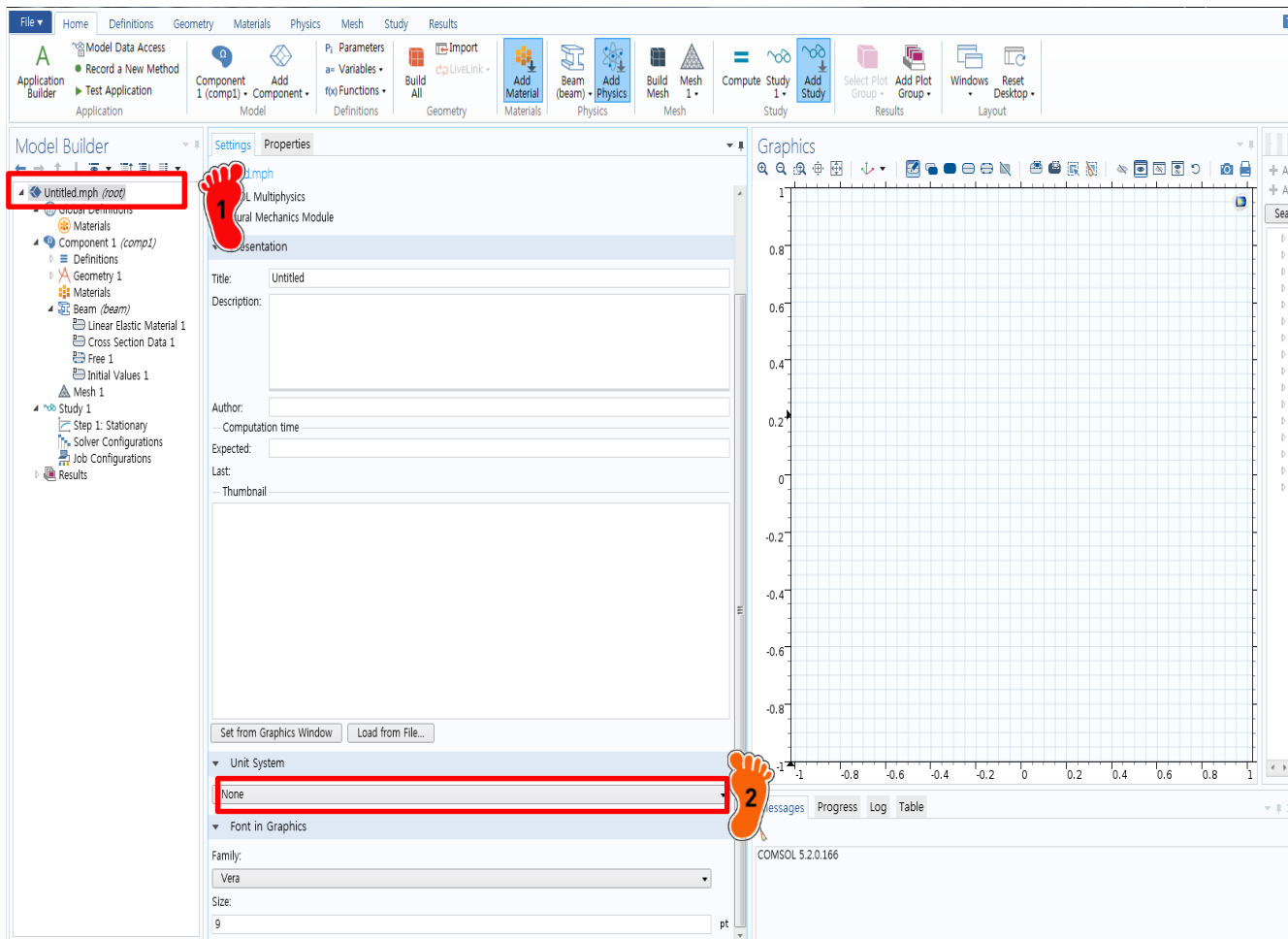
DIMENSIONLESS UNIT



Untitled.mph 클릭



Unit System 을 None 으로
변경

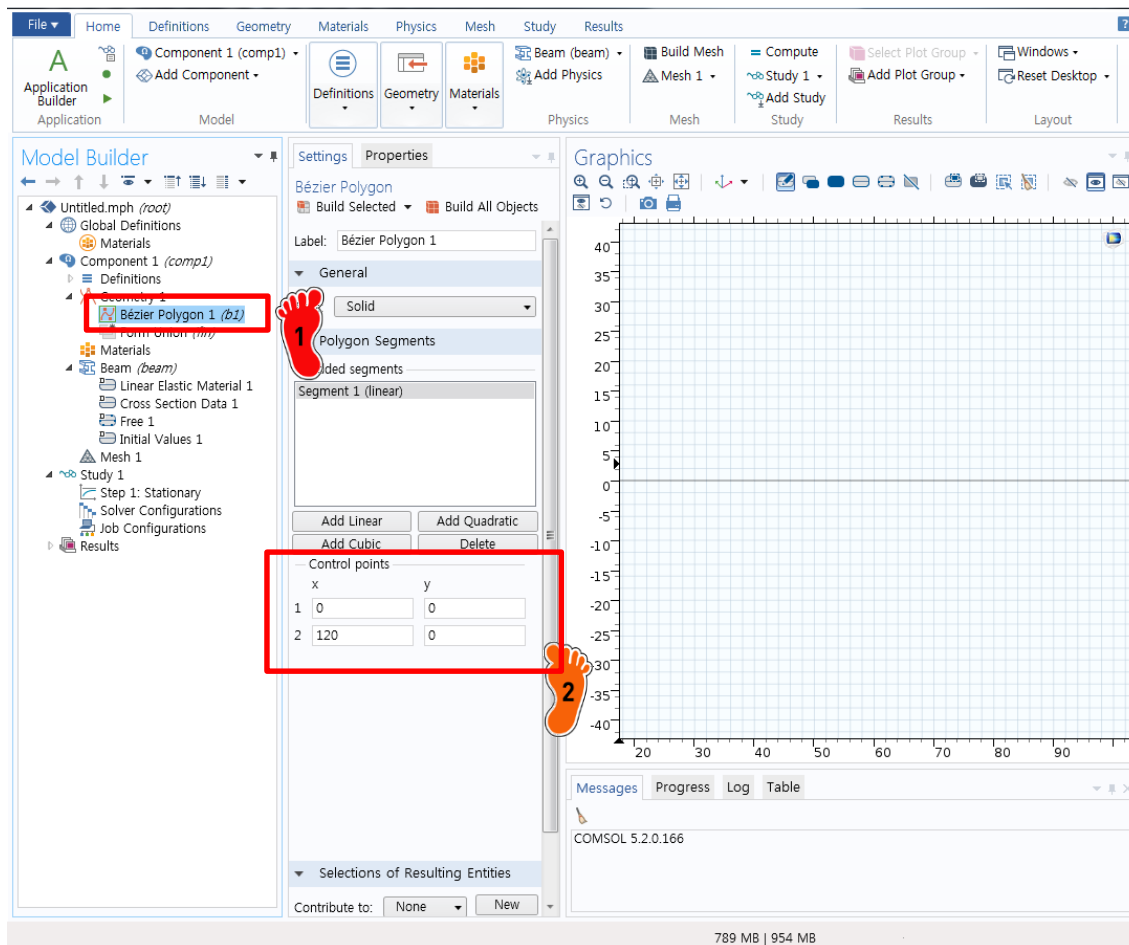


GEOMETRY CREATION

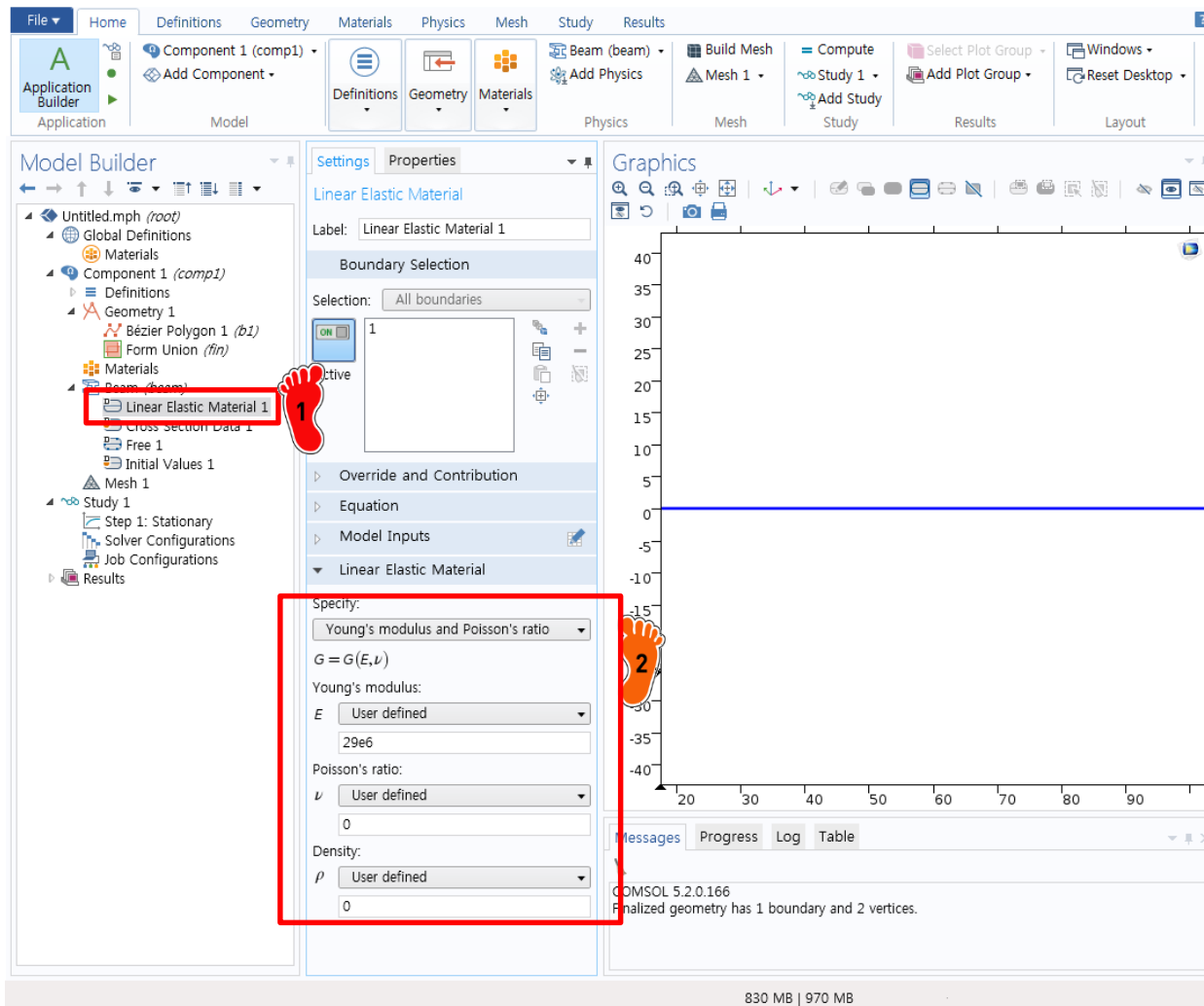
1 Bezier Polygon 생성

2 120 길이의 선분 생성

lb-inch 단위로 계산할 예정
따라서 $10 \text{ ft} = 120 \text{ in}$.



MATERIAL PROPERTY



Linear Elastic Material 클릭



E: 29e6
mu: 0
rho : 0

입력

lb-inch 단위
E = 29e3 ksi = 29e6 psi
= 29e6 lb/in.²

CROSS SECTION

The screenshot displays the COMSOL Multiphysics 5.2.0.166 interface. The Model Builder tree on the left shows the hierarchy: Untitled.mph (root) > Global Definitions > Materials > Component 1 (comp1) > Definitions > Geometry 1 > Bézier Polygon 1 (b1) > Form Union (fin) > Materials > Beam (beam) > Linear Elastic Material > Cross Section Data 1. The Cross Section Data 1 entry is highlighted with a red box and a red footprint icon labeled 1. The Properties window on the right shows the settings for Cross Section Data 1. The 'Basic Section Properties' section is highlighted with a red box and a red footprint icon labeled 2. The 'Area' is set to 8.84 and the 'Moment of inertia about local z-axis' is set to I_{zz} 170. The Graphics window on the right shows a plot of the cross-section data, with a blue line representing the cross-section. The status bar at the bottom indicates 827 MB | 981 MB.

1 Cross Section Data 클릭

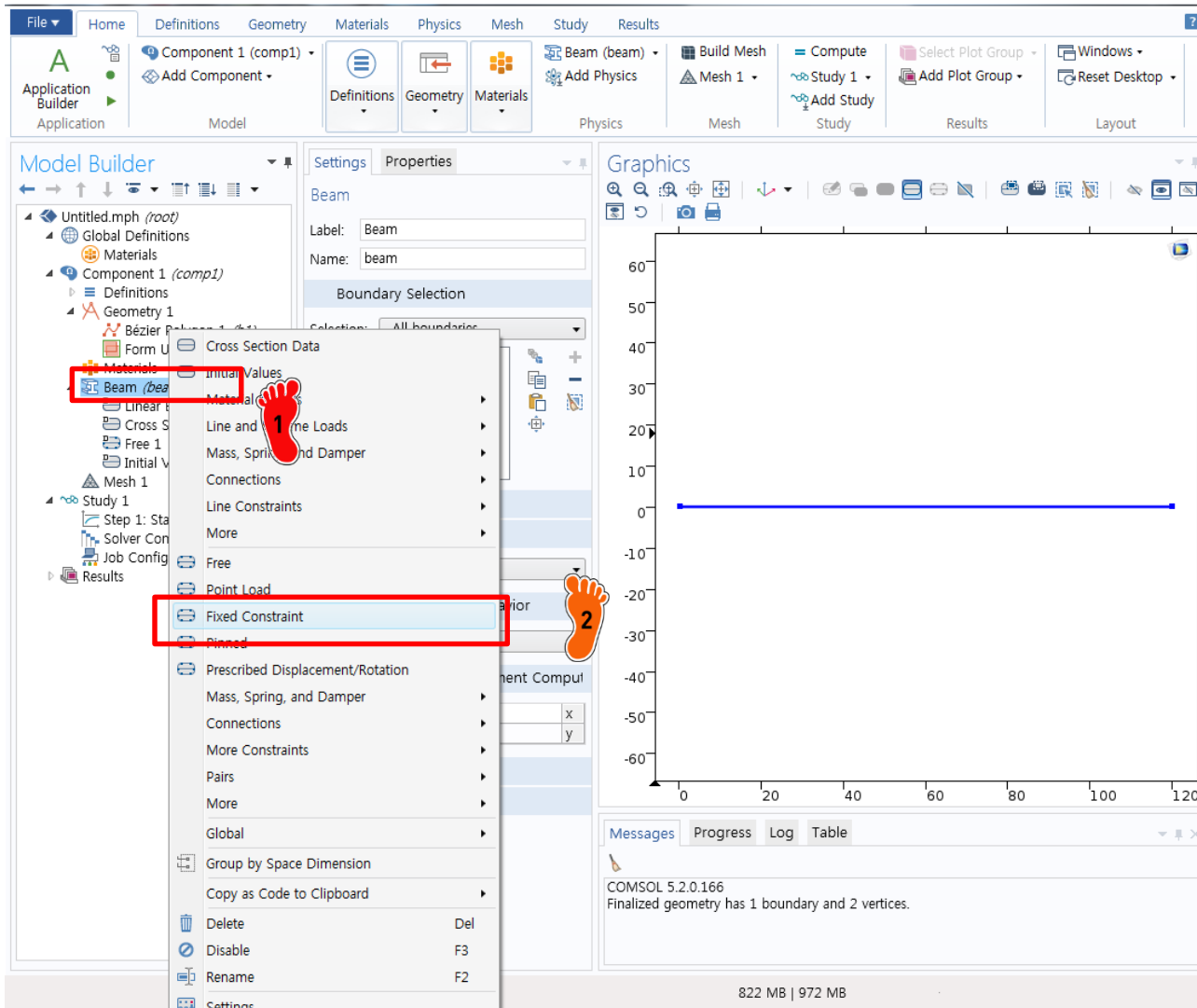
2 A: 8.84
I: 170

입력

BOUNDARY CONDITION

1 Beam 마우스 우클릭

2 Fixed Constraint 클릭



BOUNDARY CONDITION

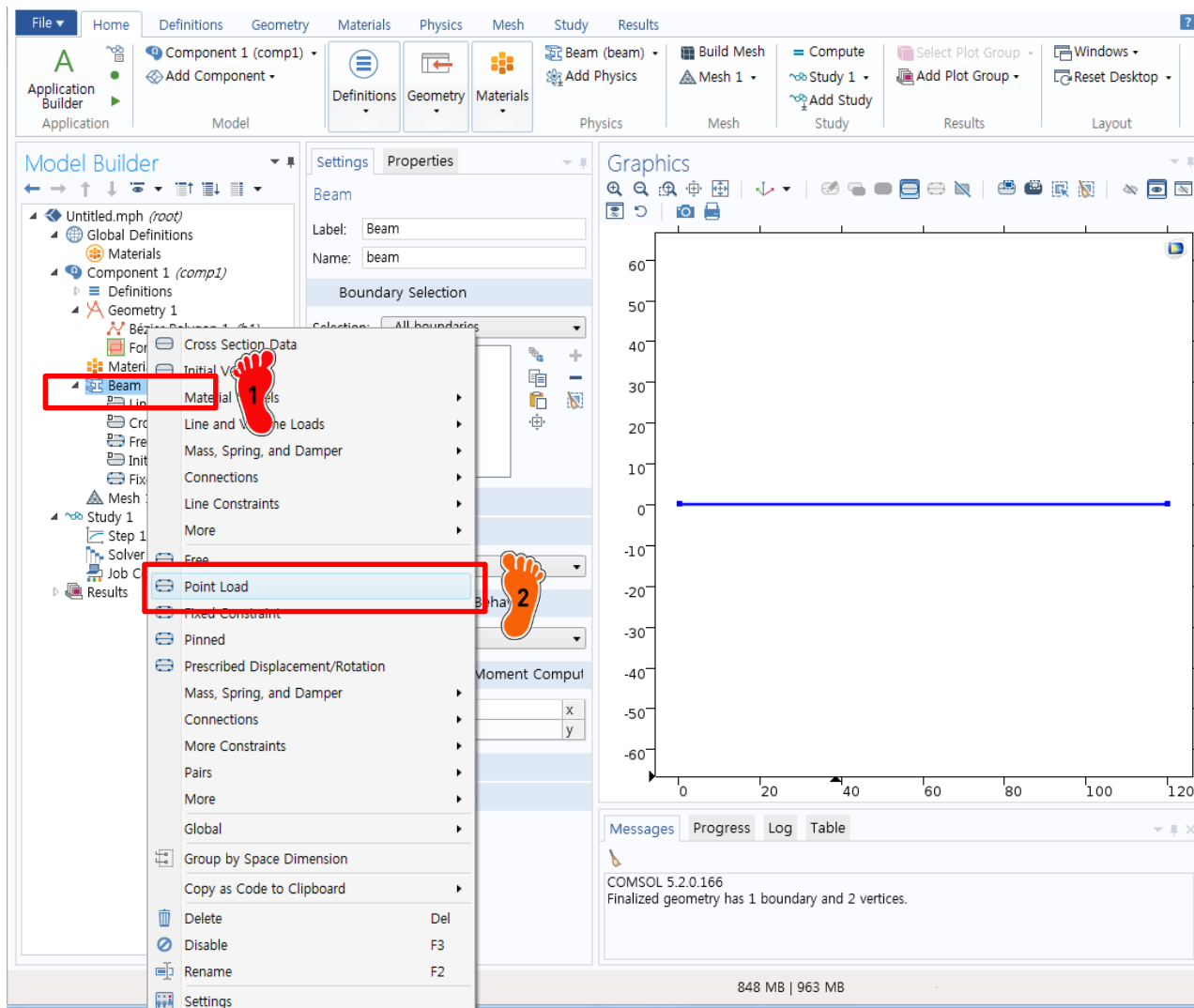


왼쪽 절점 선택

The screenshot displays the COMSOL Multiphysics software interface. The top ribbon includes tabs for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, and Results. The left pane, labeled 'Model Builder', shows a tree structure for 'Untitled.mph (root)' with folders for Global Definitions, Materials, Component 1 (comp1), Definitions, Geometry 1, Materials, Beam (beam), Mesh 1, and Study 1. Under 'Mesh 1', 'Fixed Constraint 1' is selected. The middle pane, labeled 'Settings', shows the 'Fixed Constraint' properties. The 'Label' is 'Fixed Constraint 1'. Under 'Point Selection', 'Selection' is set to 'Manual' and 'Active' is checked. The right pane, labeled 'Graphics', shows a 2D plot of a horizontal beam. A red footprint icon labeled '1' is placed at the left end of the beam, indicating the selected node. The bottom pane, labeled 'Messages', shows the text: 'COMSOL 5.2.0.166 Finalized geometry has 1 boundary and 2 vertices.'

842 MB | 966 MB

LOADING CONDITION



1 Beam 마우스 우클릭

2 Point Load 클릭

LOADING CONDITION

The screenshot displays the COMSOL Multiphysics interface with the following components:

- Top Toolbar:** Includes tabs for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, and Results. The Physics tab is active, showing options for Beam (beam), Build Mesh, Compute, Select Plot Group, and Windows.
- Model Builder:** Shows a tree view of the model structure. The 'Point Load 1' node is highlighted under the 'Physics' branch.
- Settings Panel:**
 - Point Load:** Label: Point Load 1.
 - Point Selection:** Selection: Manual. A point is selected on the beam.
 - Coordinate System Selection:** Global coordinate system.
 - Force:** F_p is set to 'User defined'. The y-component is set to $4.5e3$.
- Graphics Window:** Shows a 2D plot of the beam geometry. A red box highlights the point where the load is applied, and a red footprint icon with the number '1' indicates the selection point.
- Messages Panel:** Displays the message: 'COMSOL 5.2.0.166 Finalized geometry has 1 boundary and 2 vertices.'



오른쪽 절점 선택



y 방향에 $4.5e3$ 입력

lb-inch 단위

$$P_B = 4.5 \text{ kips} = 4.5e3 \text{ lb}$$



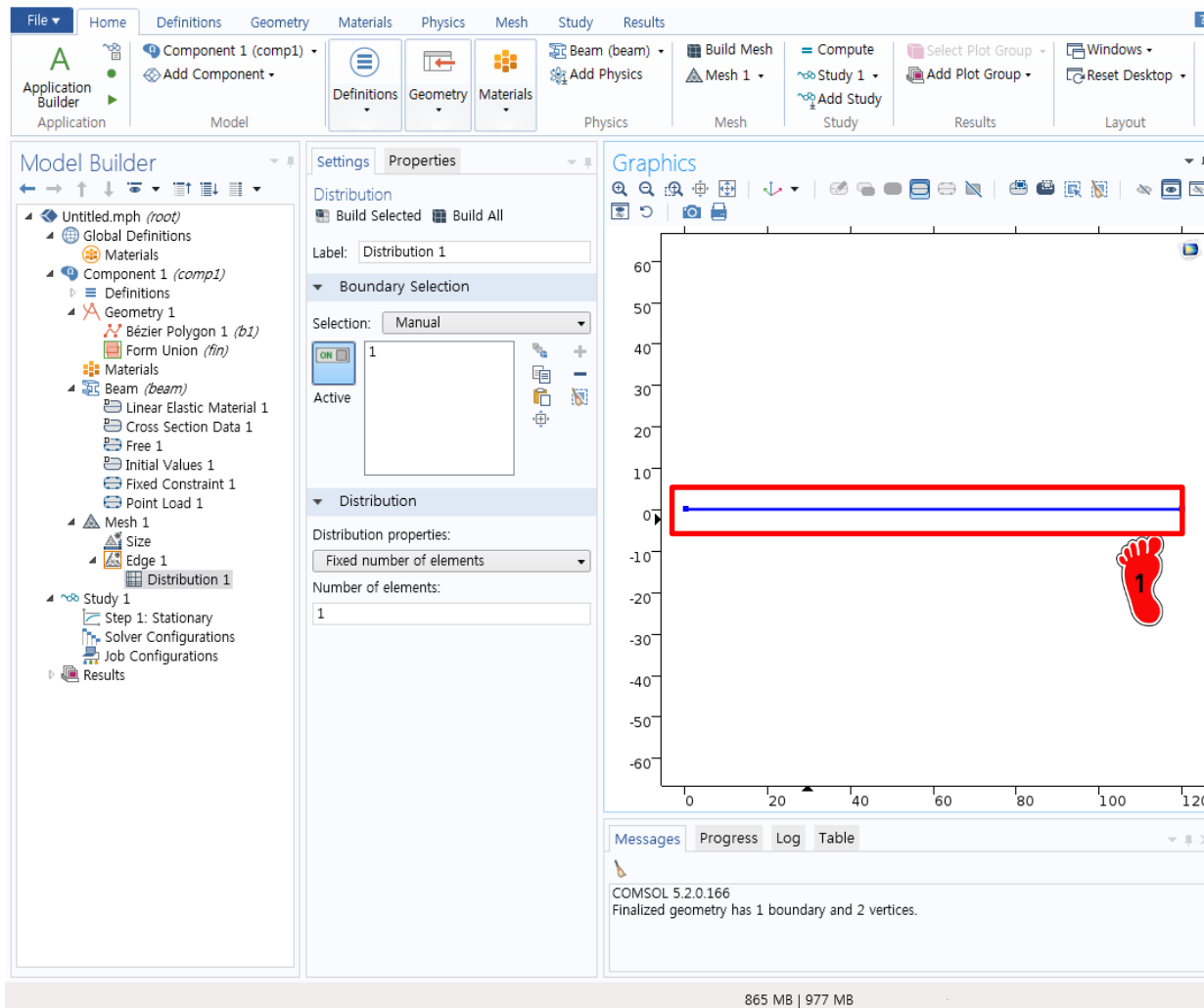
863 MB | 971 MB

MESH



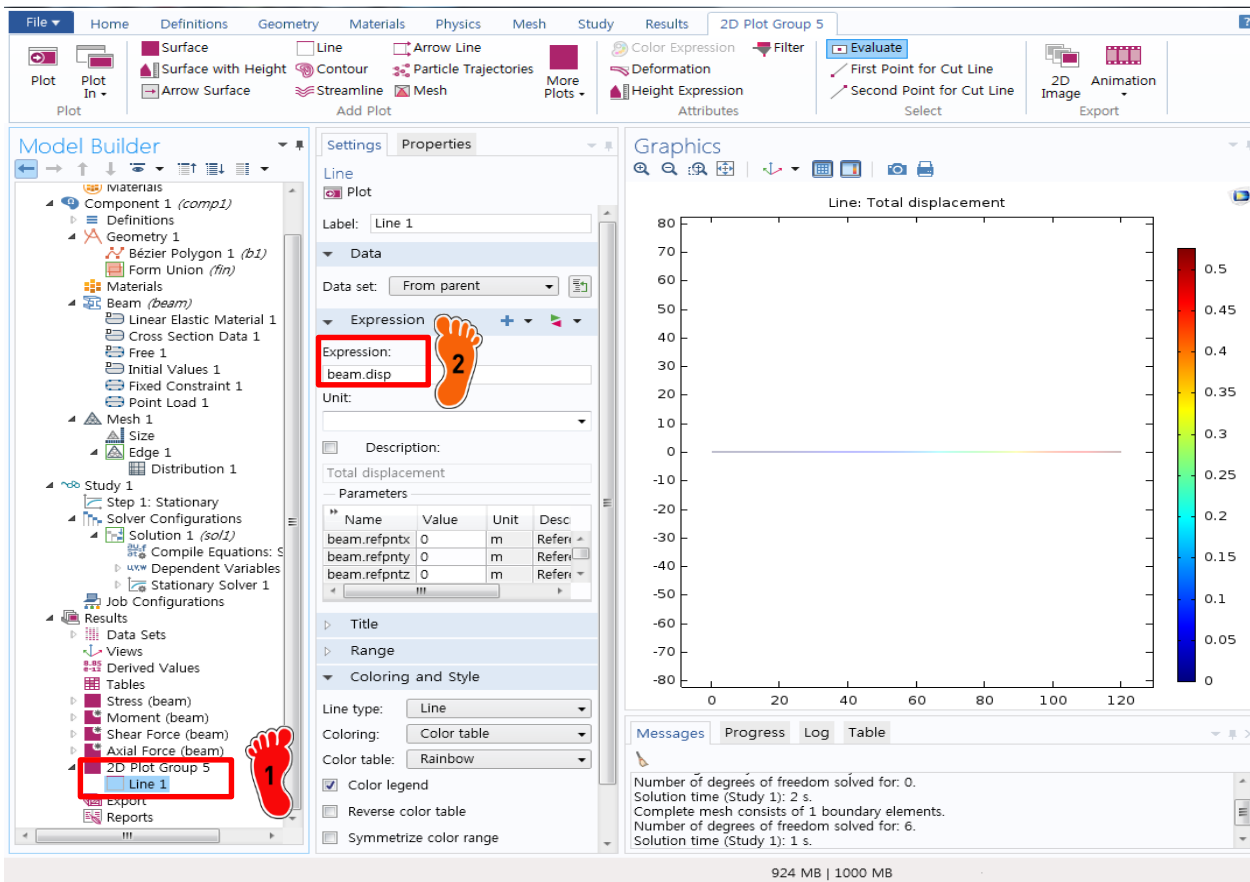
한 개의 요소가 되도록 메시 생성

그 후 compute 로 유한요소 해석 수행



865 MB | 977 MB

POST-PROCESSING



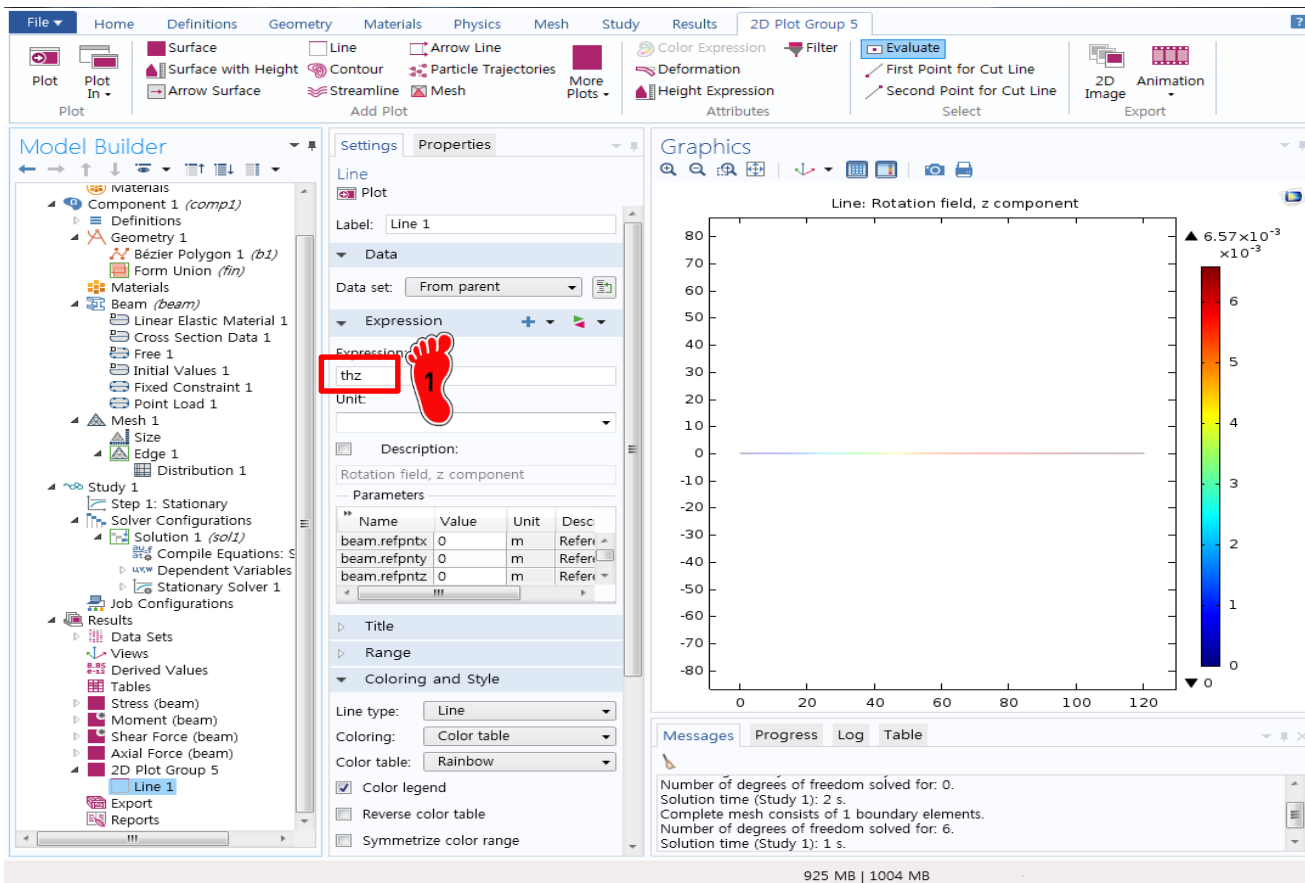
2D Plot Group 에서 Line
추가 후 beam.disp 를 확인

최대 변위는 0.5258

Analytical solution
0.5258

$$\left\{ \begin{aligned} \delta_B &= \frac{1}{EI} \left[M_B \left(\frac{L^2}{2} \right) + P_B \left(\frac{L^3}{3} \right) \right] = 0.0730 + \boxed{0.5258} = 0.5988 \text{ in} \\ \theta_B &= \frac{1}{EI} \left[M_B (L) + P_B \left(\frac{L^2}{2} \right) \right] = 0.00122 + 0.00657 = 0.00779 \text{ rad} \end{aligned} \right.$$

POST-PROCESSING



Expression 을 thz 로 변경

최대 각도는 6.57e-3

Analytical solution
6.57e-3

$$\left\{ \begin{aligned} \delta_B &= \frac{1}{EI} \left[M_B \left(\frac{L^2}{2} \right) + P_B \left(\frac{L^3}{3} \right) \right] = 0.0730 + 0.5258 = 0.5988 \text{ in} \\ \theta_B &= \frac{1}{EI} \left[M_B (L) + P_B \left(\frac{L^2}{2} \right) \right] = 0.00122 + \boxed{0.00657} = 0.00779 \text{ rad} \end{aligned} \right.$$

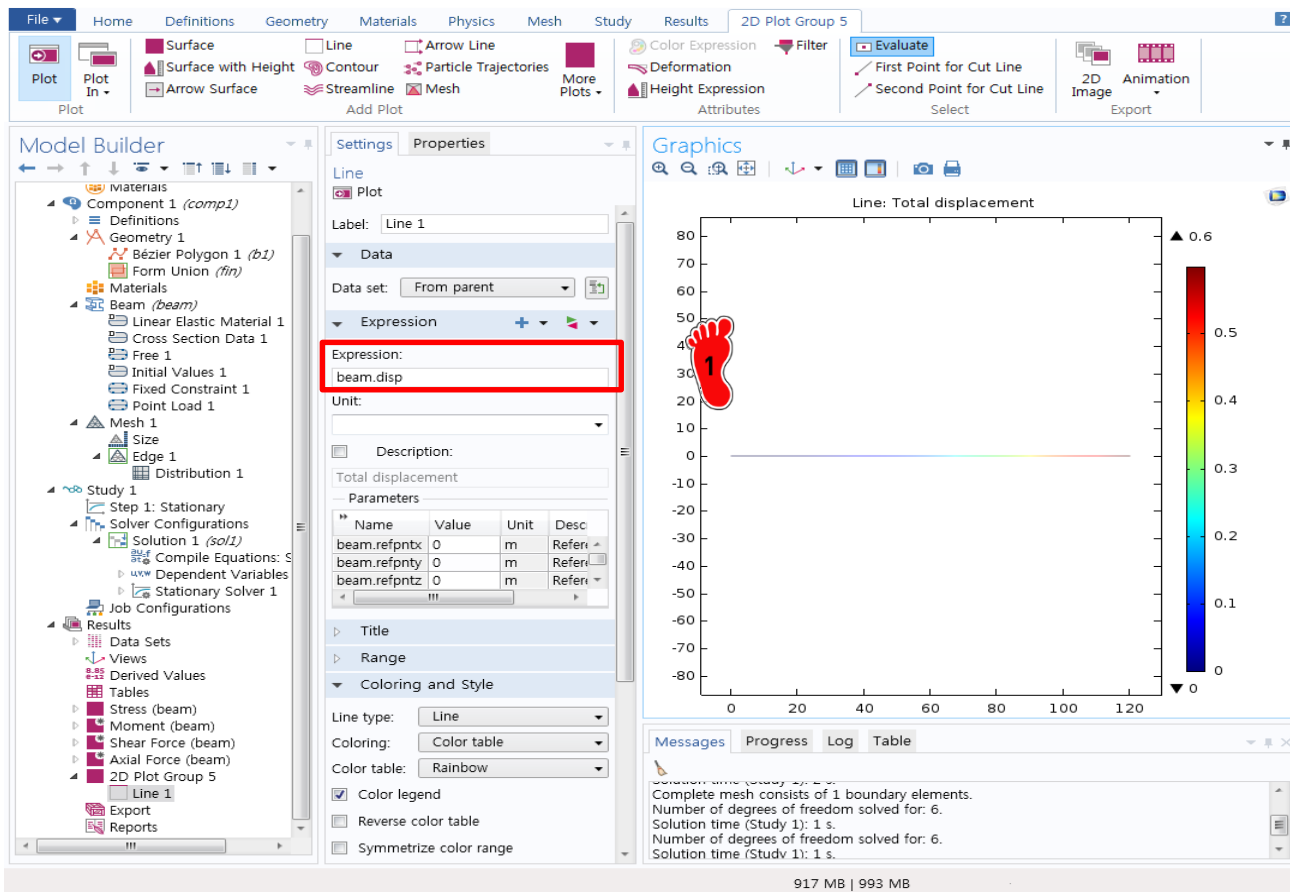
LOADING CONDITION

The screenshot displays the ANSYS Workbench environment. The 'Model Builder' tree on the left shows the hierarchy: Untitled.mph (root) > Global Definitions > Materials > Component 1 (comp1) > Definitions > Geometry 1 > Bézier Polygon 1 (b1) > Form Union (fin) > Materials > Beam (beam) > Linear Elastic Material 1 > Cross Section Data 1 > Free 1 > Initial Values 1 > Fixed Constraint 1 > Point Load 1. The 'Point Load 1' is highlighted in blue. The 'Properties' panel on the right shows the 'Point Load' settings. The 'Point Selection' is set to 'Manual' with a value of 2. The 'Coordinate system' is 'Global coordinate system'. The 'Force' section is expanded, showing 'F_p' set to 'User defined' with a value of 4.5e3 in the y-direction. The 'Moment' section is also expanded, showing 'M_{1z}' set to 50e3. The 'Graphics' window on the right shows a 2D plot of the beam with a point load applied at the right end. The 'Messages' window at the bottom shows the following text: 'Complete mesh consists of 1 boundary elements. Number of degrees of freedom solved for: 6. Solution time (Study 1): 1 s. Number of degrees of freedom solved for: 6. Solution time (Study 1): 1 s.'

Point load 설정에서 모멘트 추가

50e3 입력

POST-PROCESSING



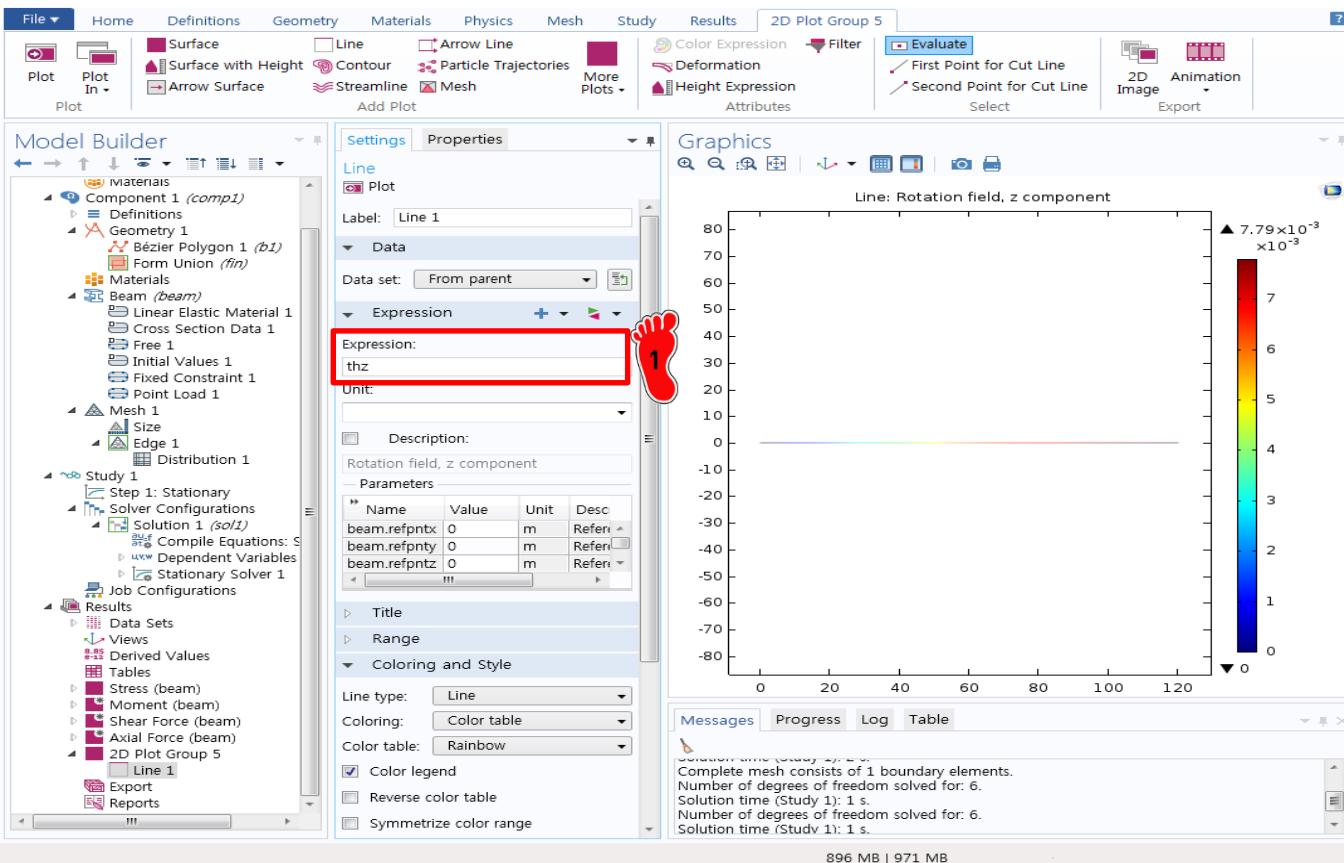
beam.disp 를 확인

최대 변위는 0.599

Analytical solution
0.599

$$\left\{ \begin{aligned} \delta_B &= \frac{1}{EI} \left[M_B \left(\frac{L^2}{2} \right) + P_B \left(\frac{L^3}{3} \right) \right] = 0.0730 + 0.5258 = \boxed{0.599} \text{ in} \\ \theta_B &= \frac{1}{EI} \left[M_B (L) + P_B \left(\frac{L^2}{2} \right) \right] = 0.00122 + 0.00657 = 0.00779 \text{ rad} \end{aligned} \right.$$

POST-PROCESSING



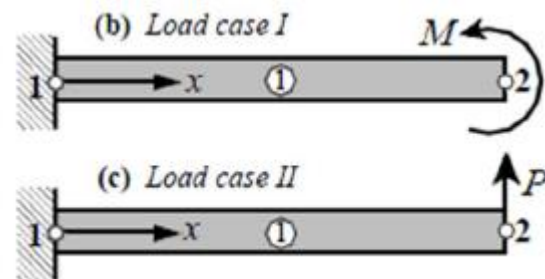
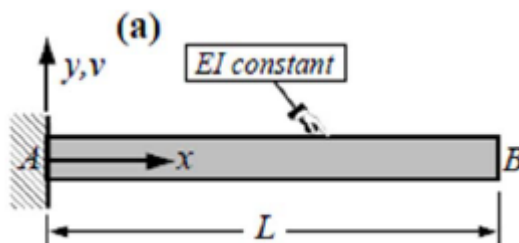
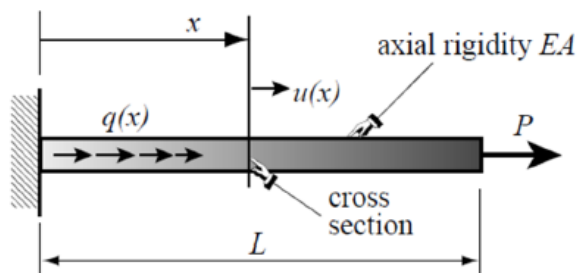
Expression 을 thz 로 변경

최대 각도는 7.79e-3 rad

Analytical solution
7.79e-3 rad

$$\left\{ \begin{aligned} \delta_B &= \frac{1}{EI} \left[M_B \left(\frac{L^2}{2} \right) + P_B \left(\frac{L^3}{3} \right) \right] = 0.0730 + 0.5258 = 0.599 \text{ in} \\ \theta_B &= \frac{1}{EI} \left[M_B (L) + P_B \left(\frac{L^2}{2} \right) \right] = 0.00122 + 0.00657 = 0.00779 \text{ rad} \end{aligned} \right.$$

SYSTEM EQUATION BY HAND



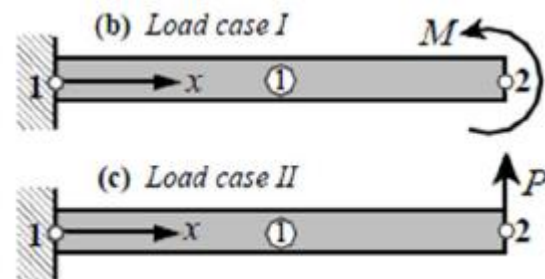
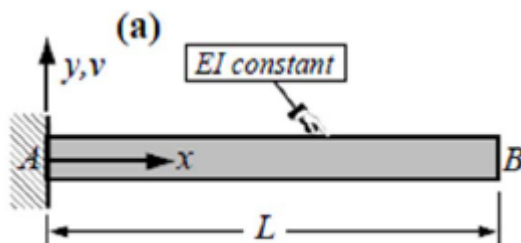
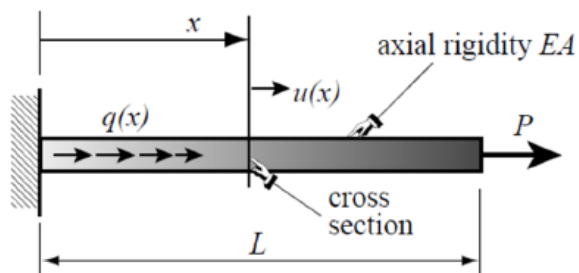
$$\frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 0 \\ P \end{bmatrix}$$

$$\rightarrow u_2 = \frac{PL}{EA}$$

$$\frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ M \end{bmatrix} \rightarrow \begin{bmatrix} v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} \frac{ML^2}{2EI} \\ \frac{ML}{EI} \end{bmatrix}$$

$$\frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ P \\ 0 \end{bmatrix} \rightarrow \begin{bmatrix} v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} \frac{PL^3}{3EI} \\ \frac{PL^2}{2EI} \end{bmatrix}$$

SYSTEM EQUATION BY HAND



$$\begin{bmatrix}
 1.64e^8 & 0 & -2.05e^6 & 8.22e^7 & 0 & 2.05e^6 \\
 0 & 2.14e^6 & 0 & 0 & -2.14e^6 & 0 \\
 -2.05e^6 & 0 & 3.42e^4 & -2.05e^6 & 0 & -3.42e^4 \\
 8.22e^7 & 0 & -2.05e^6 & 1.64e^8 & 0 & 2.05e^6 \\
 0 & -2.14e^6 & 0 & 0 & 2.14e^6 & 0 \\
 2.05e^6 & 0 & -3.42e^4 & 2.05e^6 & 0 & 3.42e^4
 \end{bmatrix}
 \begin{bmatrix}
 \theta_2 \\
 u_2 \\
 v_2 \\
 \theta_1 \\
 u_1 \\
 v_1
 \end{bmatrix}
 =
 \begin{bmatrix}
 M \\
 P_x \\
 P_y \\
 0 \\
 0 \\
 0
 \end{bmatrix}$$

ASSEMBLE OPTION

The screenshot shows the ANSYS Workbench interface. The 'Model Builder' tree on the left lists the components of the model, including 'Component 1 (comp1)', 'Geometry 1', 'Materials', 'Beam (beam)', 'Mesh 1', and 'Study 1'. The 'Solution' menu is open, and the 'Assemble' option is highlighted. The 'Graphics' window on the right displays a line plot of the 'Rotation field, z component'.

1 Solution

2 Assemble

Line: Rotation field, z component

Y-axis: -160 to 160

X-axis: 0 to 100

Color scale: 0 to 7.79×10^{-3}

Messages:

```

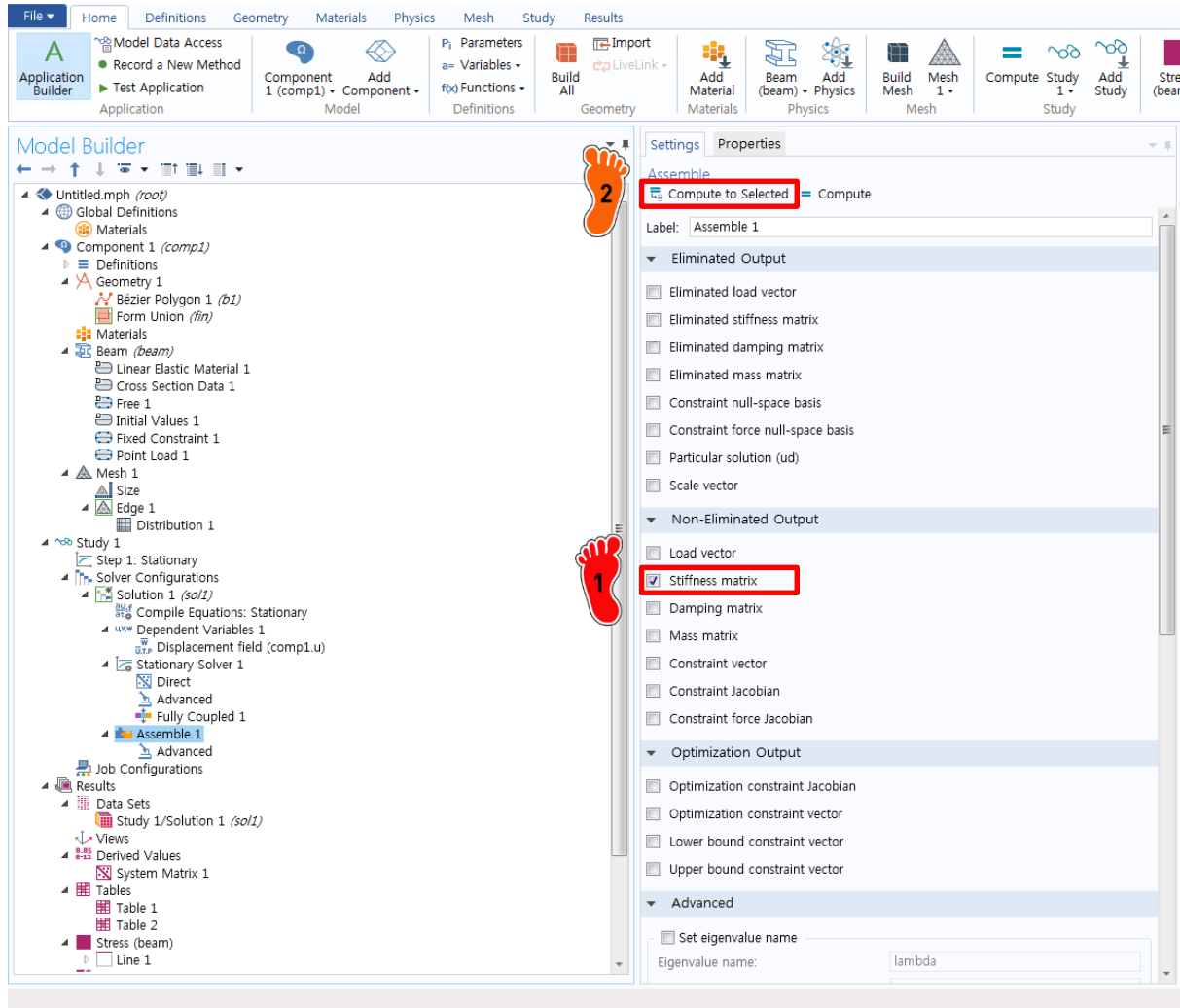
Solution time (Study 1): 1 s.
Complete mesh consists of 1 boundary elements.
Number of degrees of freedom solved for: 6.
Solution time (Study 1): 1 s.
Number of degrees of freedom solved for: 6.
Solution time (Study 1): 1 s.
  
```

919 MB | 1000 MB

1 Solution 마우스 우클릭

2 Assemble 클릭

ASSEMBLE OPTION



1 Stiffness matrix 체크박스에 체크

2 Compute to Selected 클릭

COMPARISON

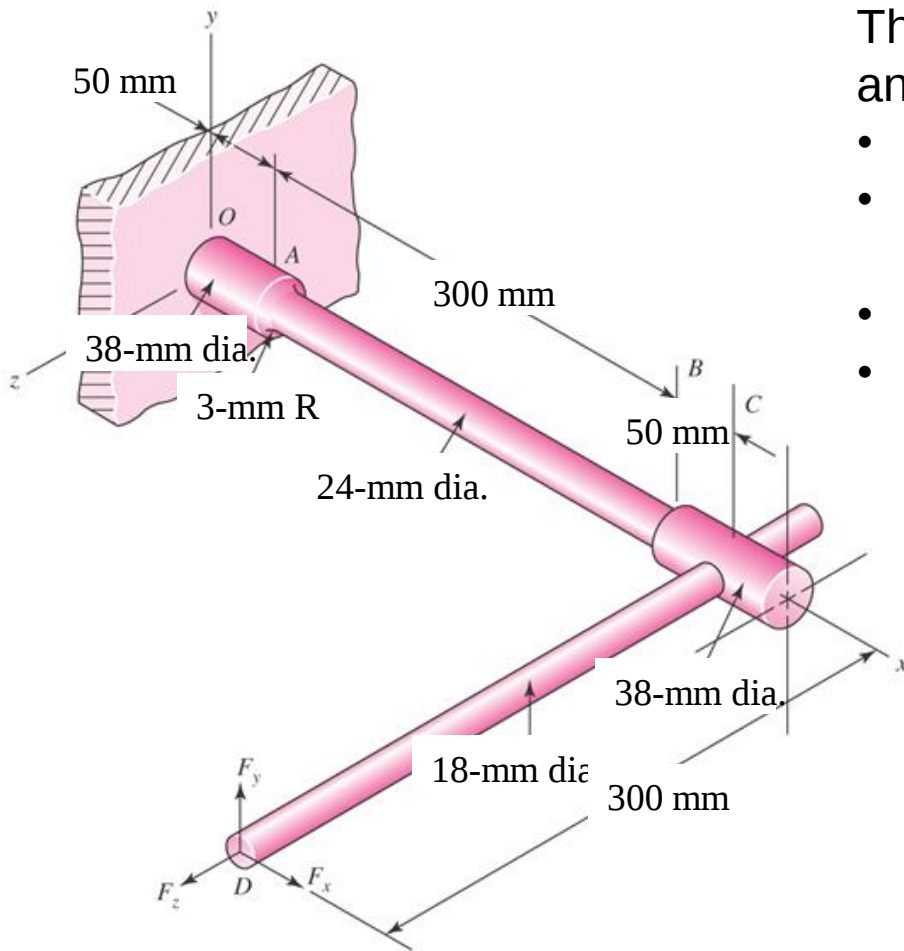
$$\mathbf{K}_{by\ hand} = \begin{bmatrix} 1.64e^8 & 0 & -2.05e^6 & 8.22e^7 & 0 & 2.05e^6 \\ 0 & 2.14e^6 & 0 & 0 & -2.14e^6 & 0 \\ -2.05e^6 & 0 & 3.42e^4 & -2.05e^6 & 0 & -3.42e^4 \\ 8.22e^7 & 0 & -2.05e^6 & 1.64e^8 & 0 & 2.05e^6 \\ 0 & -2.14e^6 & 0 & 0 & 2.14e^6 & 0 \\ 2.05e^6 & 0 & -3.42e^4 & 2.05e^6 & 0 & 3.42e^4 \end{bmatrix}$$

1.6433E8	0.0000	-2.0542E6	8.2167E7	0.0000	2.0542E6
0.0000	2.1363E6	0.0000	0.0000	-2.1363E6	0.0000
-2.0542E6	0.0000	34236	-2.0542E6	0.0000	-34236
8.2167E7	0.0000	-2.0542E6	1.6433E8	0.0000	2.0542E6
0.0000	-2.1363E6	0.0000	0.0000	2.1363E6	0.0000
2.0542E6	0.0000	-34236	2.0542E6	0.0000	34236

Stiffness matrix by COMSOL

- **Beam model: 2D**
 - ✓ Textbook of “Solid Mechanics”
- **Beam model: 3D**
- **Assignment**

CANTILEVERED HANDLE



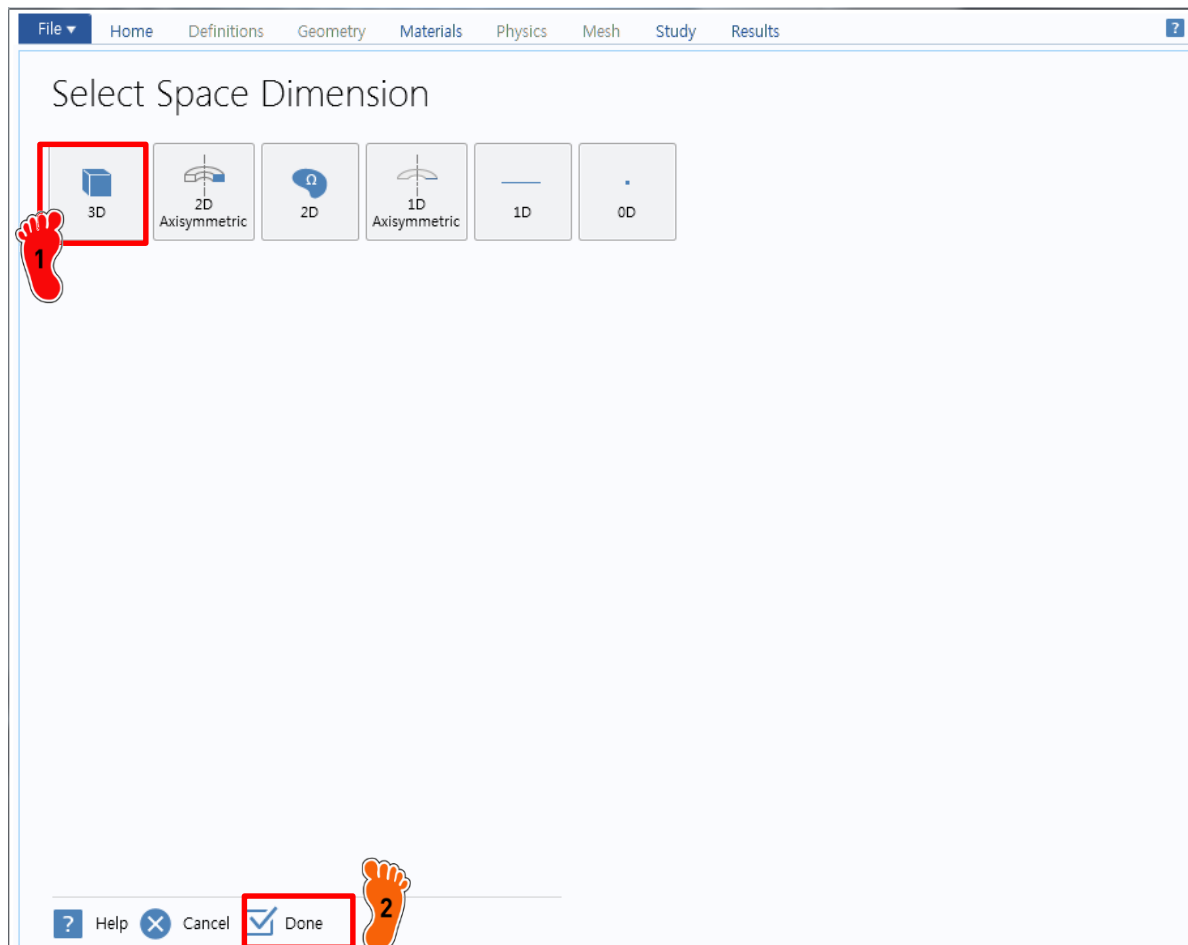
- The cantilevered bar is made from mild steel and is statically loaded. **At cross section A,**
- Determine the location of the critical stress.
 - Determine the principal stresses and the maximum shear stress.
 - Determine the angle of twist in bar OC.
 - Determine the vertical deflection at the tip.

$$\begin{aligned}
 E &= 207 \text{ GPa} & \sigma_1 &= 298.8 \text{ MPa} \\
 G &= 79.3 \text{ GPa} & \sigma_2 &= -40.9 \text{ MPa} \\
 F_{y@D} &= 1000 \text{ N} & \tau_{\max} &= 169.8 \text{ MPa} \\
 & & \theta_C &= 0.0367 \text{ rad} \\
 & & y_D &= 24.005 \text{ mm}
 \end{aligned}$$

The shear and normal stresses can be determined as functions of x and y directly from these functions, as well as such parameters as the principal stress. Since σ_y is zero everywhere, the principal stress is

$$\sigma_{p1} = \frac{\sigma_x}{2} + \sqrt{\left(\frac{\sigma_x}{2}\right)^2 + \tau_{xy}^2}$$

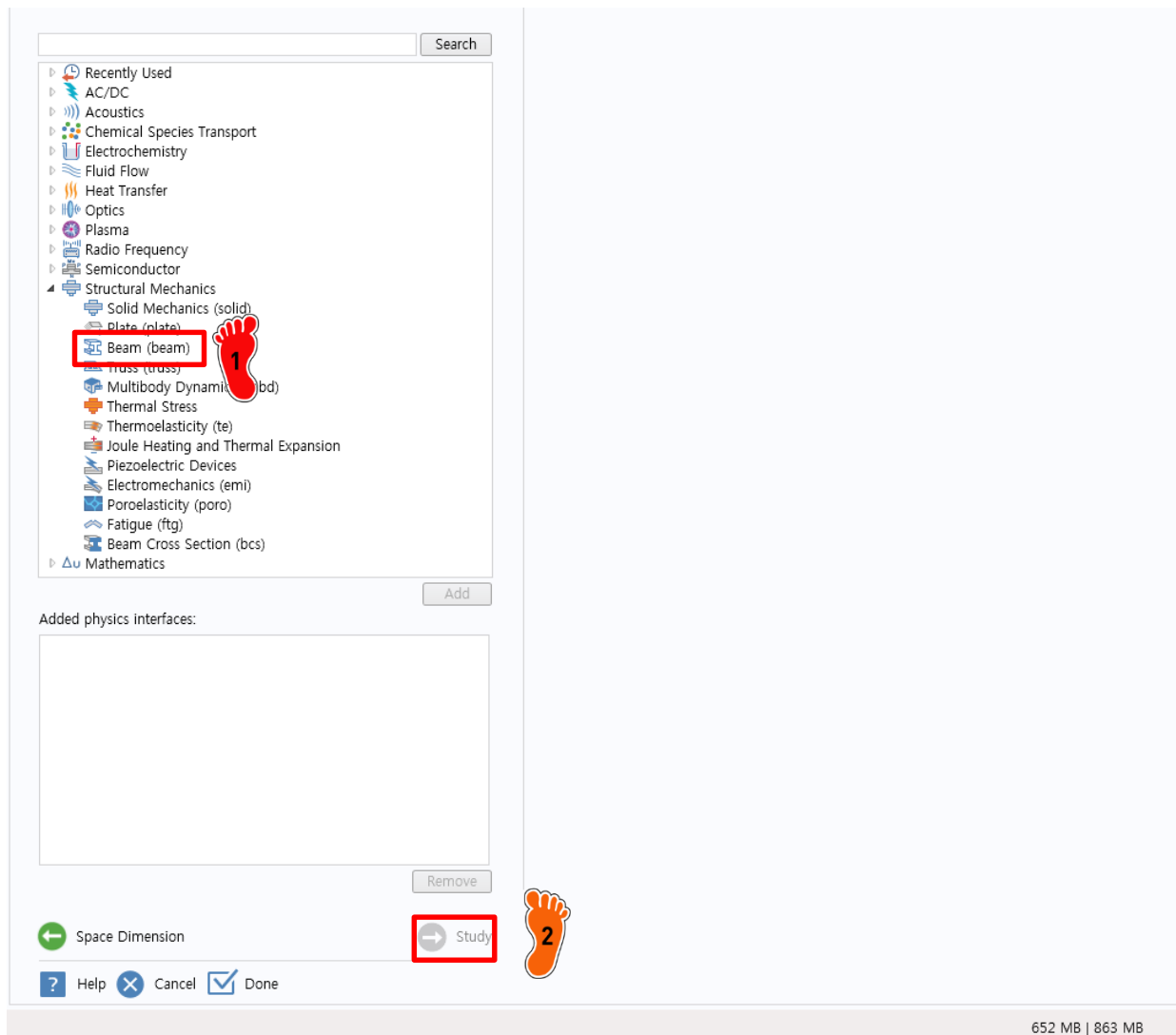
DIMENSION SELECTION



1 3D 선택

2 Done 클릭

PHYSICS SELECTION



1 Structural Mechanics 의
Beam 추가

2 Study 클릭

STUDY TYPE SELECTION

File ▾ Home Definitions Geometry Materials Physics Mesh Study Results

Select Study

Preset Studies

- Eigenfrequency
- Frequency Domain
- Frequency-Domain Modal
- Modal Body
- Modal Body and Fluid Model
- Stationary**
- Time Dependent
- Time-Dependent Modal

 Custom Studies

- Empty Study

Added study:

Stationary

Added physics interfaces:

Beam (beam)

Physics

Help Cancel **Done**

Stationary

The Stationary study is used when field variables do not change over time.

Examples: In electromagnetics, it is used to compute static electric or magnetic fields, as well as direct currents. In heat transfer, it is used to compute the temperature field at thermal equilibrium. In solid mechanics, it is used to compute deformations, stresses, and strains at static equilibrium. In fluid flow it is used to compute the steady flow and pressure fields. In chemical species transport, it is used to compute steady-state chemical composition in steady flows. In chemical reactions, it is used to compute the chemical composition at equilibrium of a reacting system.

It is also possible to compute several solutions, such as a number of load cases, or to track the nonlinear response to a slowly varying load.

1 Stationary 선택

2 Done 클릭

GEOMETRY CREATION

1 Bezier Polygon 생성

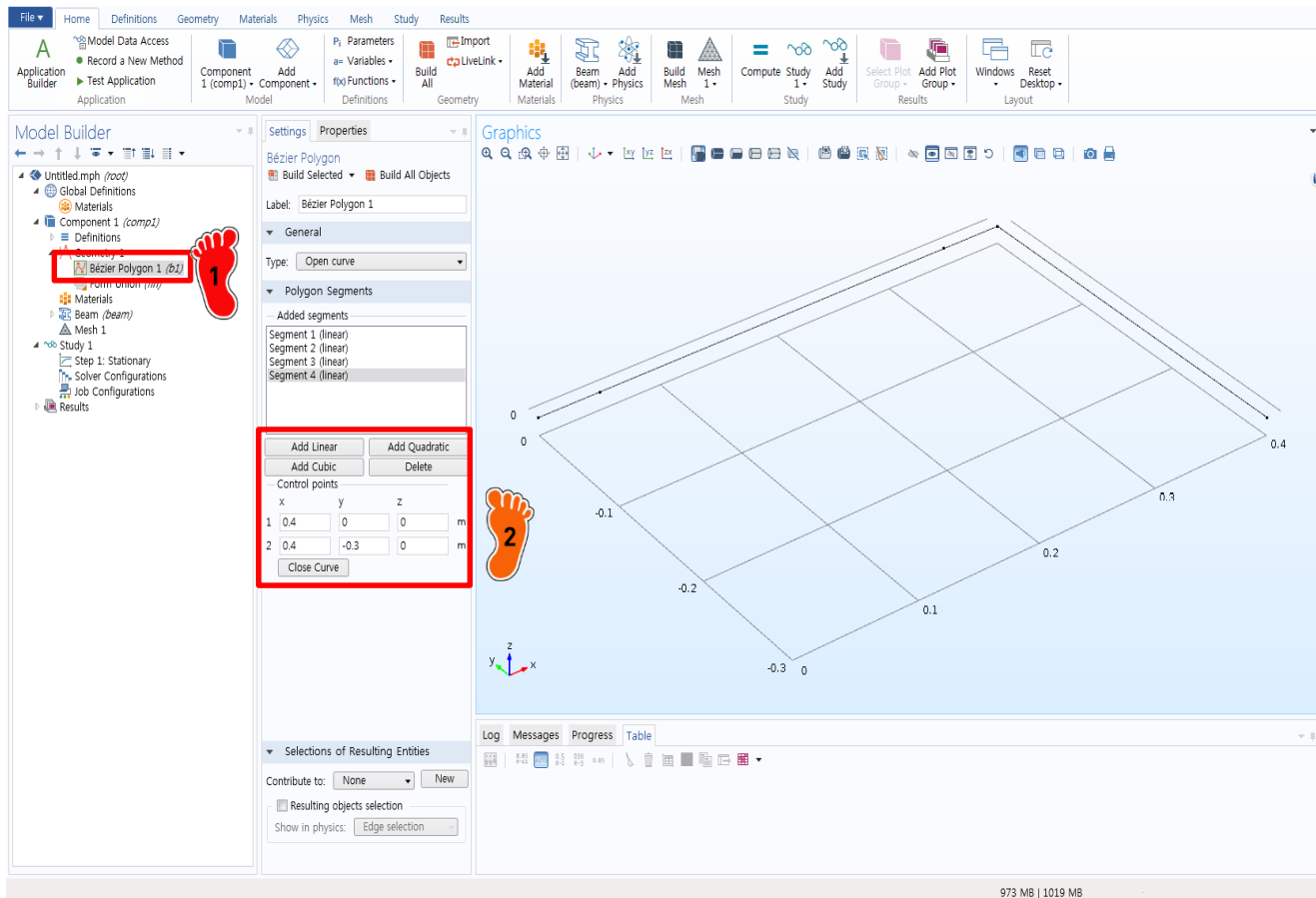
2 Segment1
(0,0,0)~(0.05,0,0)

Segment2
(0.05,0,0)~(0.35,0,0)

Segment3
(0.35,0,0)~(0.4,0,0)

Segment4
(0.4,0,0)~(0.4,-0.3,0)

기하형상 생성



MATERIAL PROPERTY



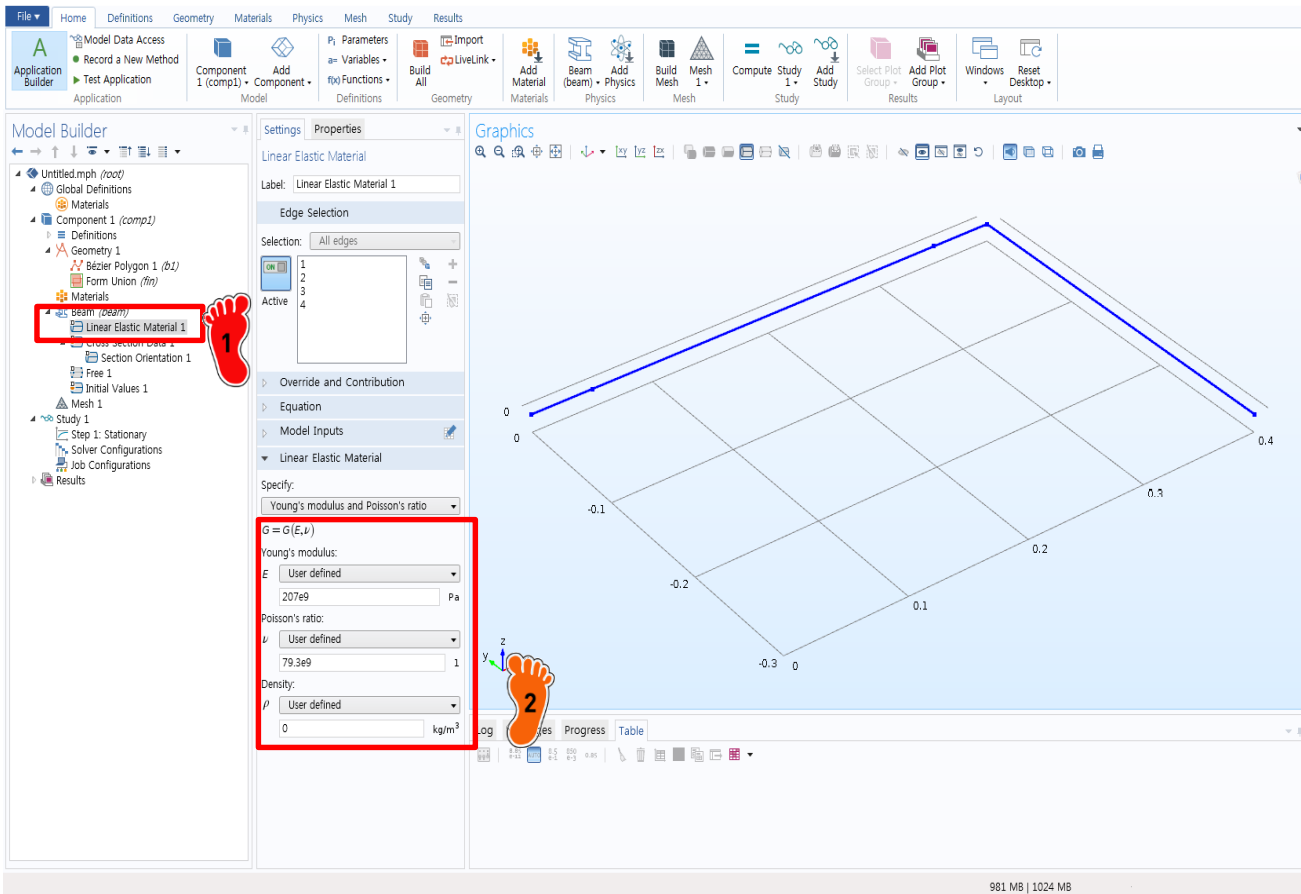
Linear Elastic Material 클릭



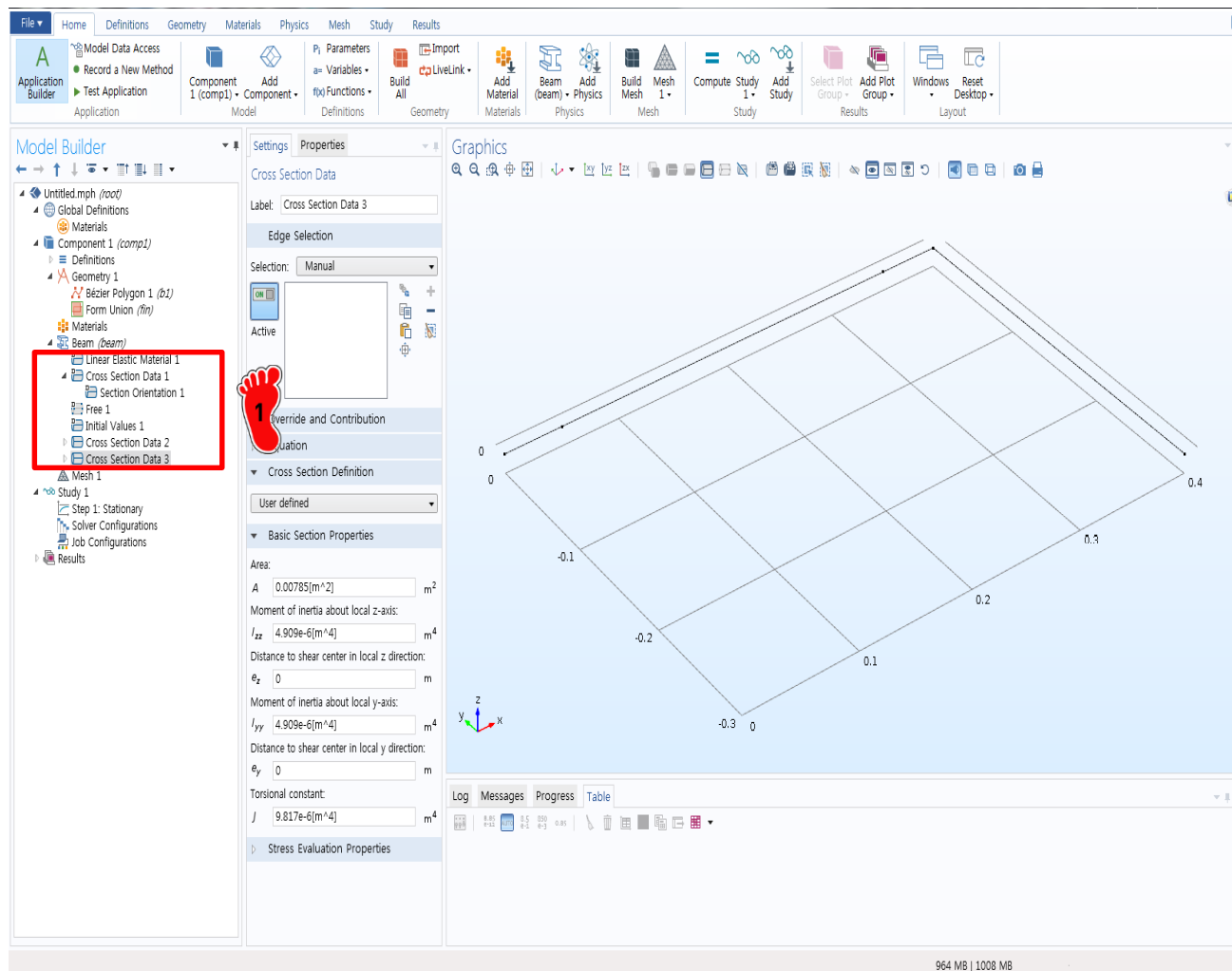
Specify 를
Young's modulus and
shear modulus 로 변경 후

E: 207e9
G: 79.3e9
rho : 0

입력

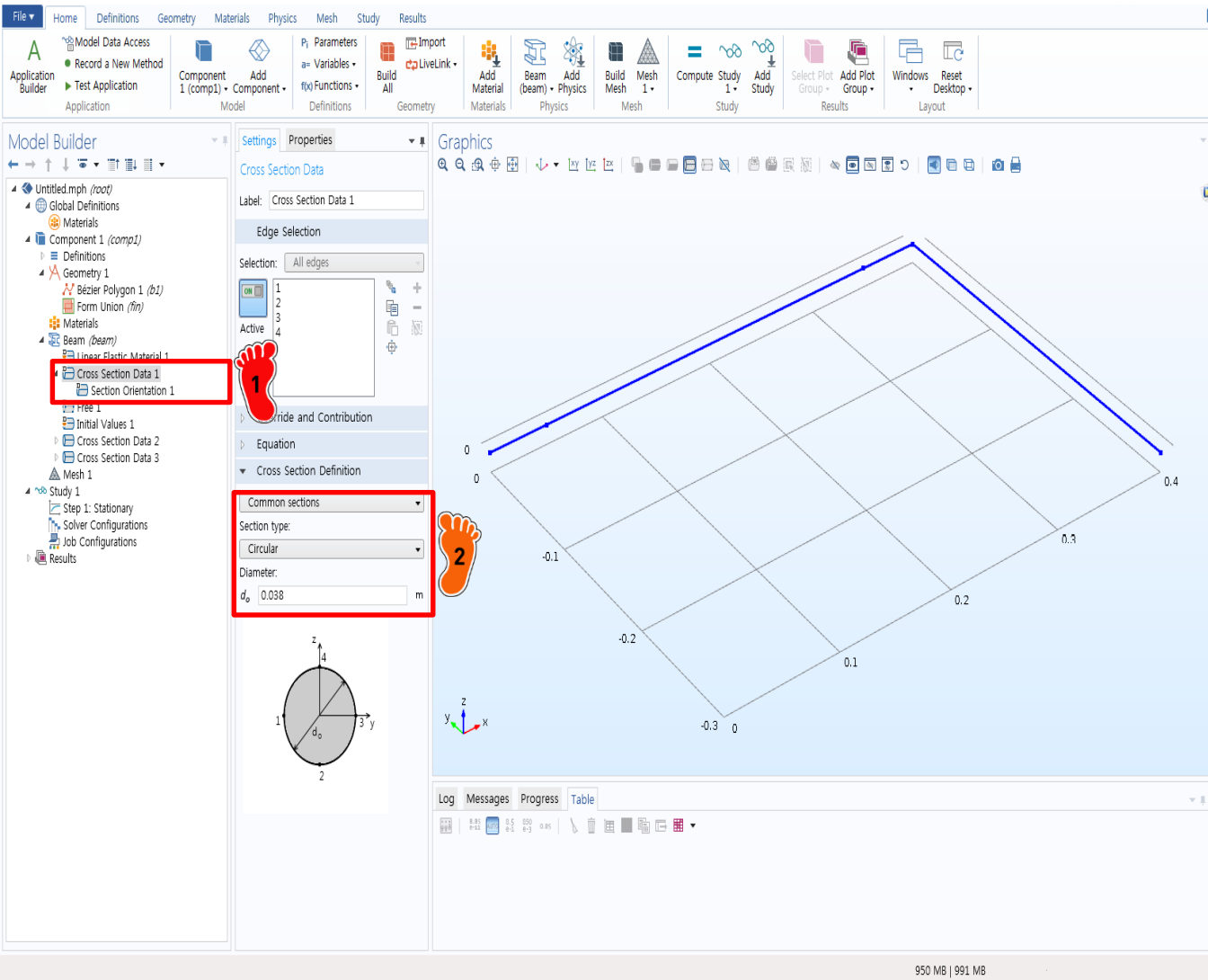


CROSS SECTION



1 Cross Section Data 를
Duplicate 로 2 개 더 복사

CROSS SECTION



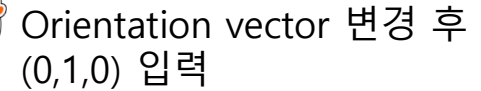
Cross Section Data 1 클릭



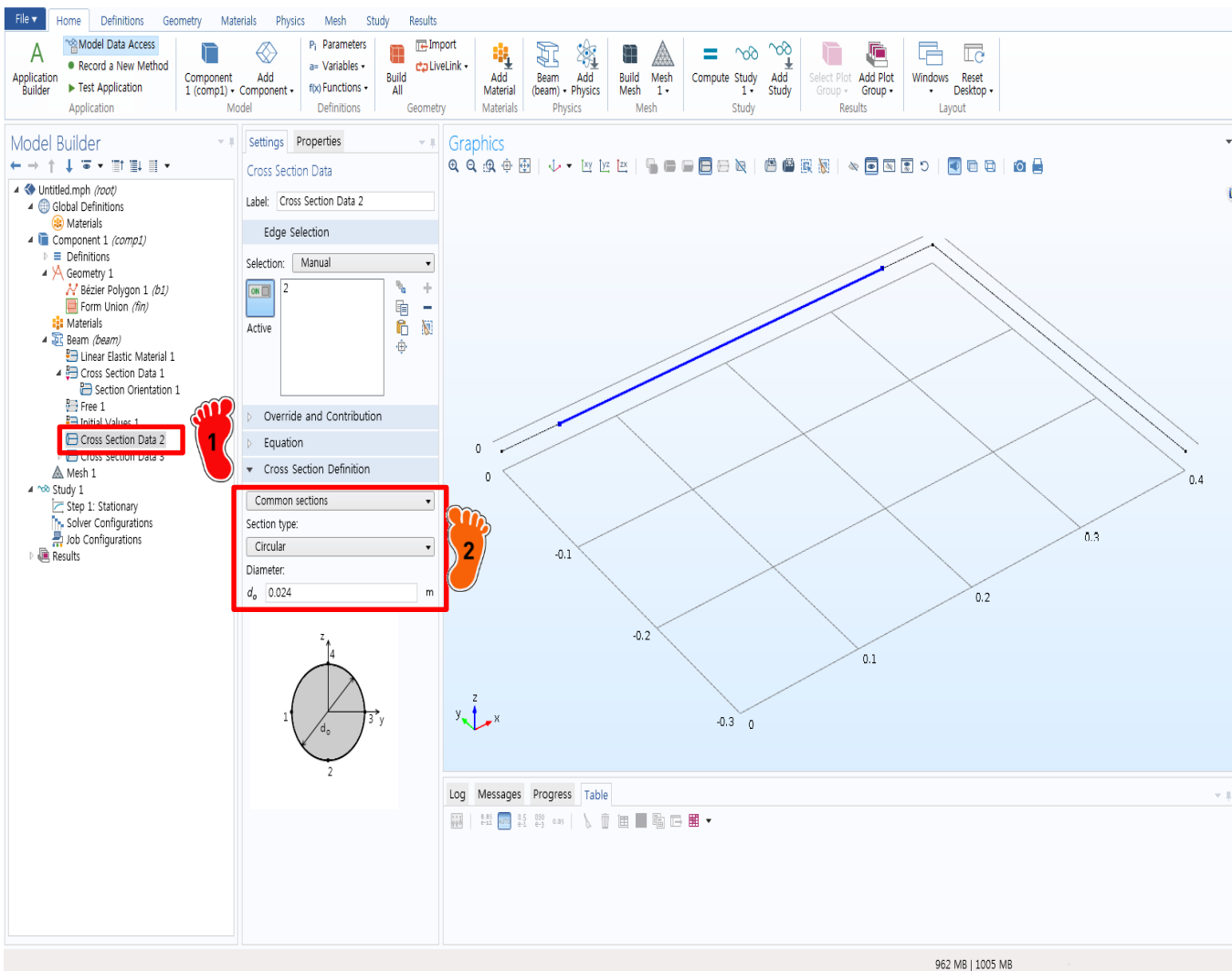
Common sections 변경

Circular 변경

지름 0.038 입력



CROSS SECTION



1 Cross Section Data 2 클릭

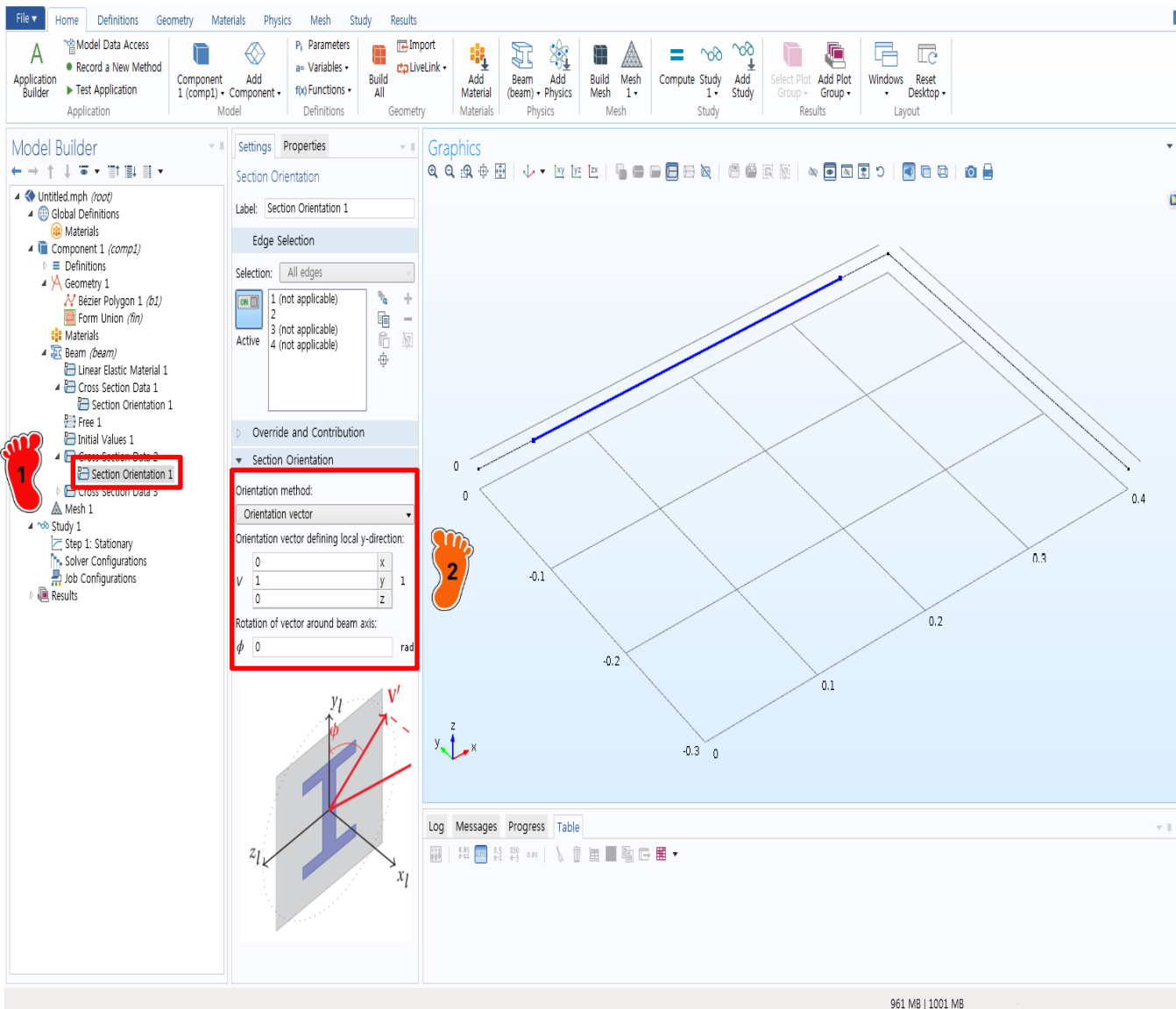
2 AB 부재 선택 후

Common sections 변경

Circular 변경

지름 0.024 입력

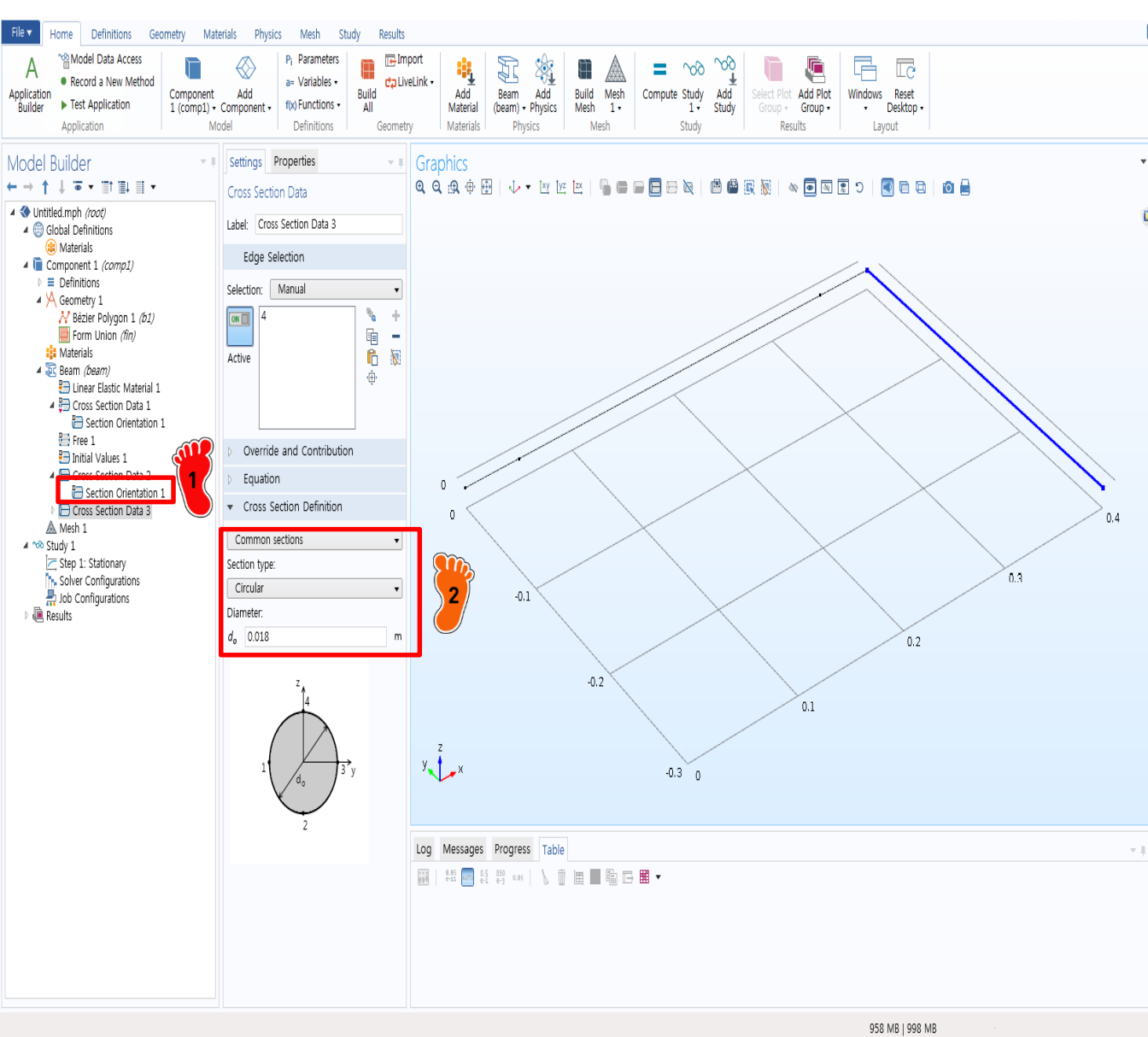
CROSS SECTION



1 Cross Section Data 2 하위 메뉴인 Section Orientation 1 클릭

2 Orientation vector 변경 후 (0,1,0) 입력

CROSS SECTION



1 Cross Section Data 3 클릭

2 CD 부재 선택 후

Common sections 변경

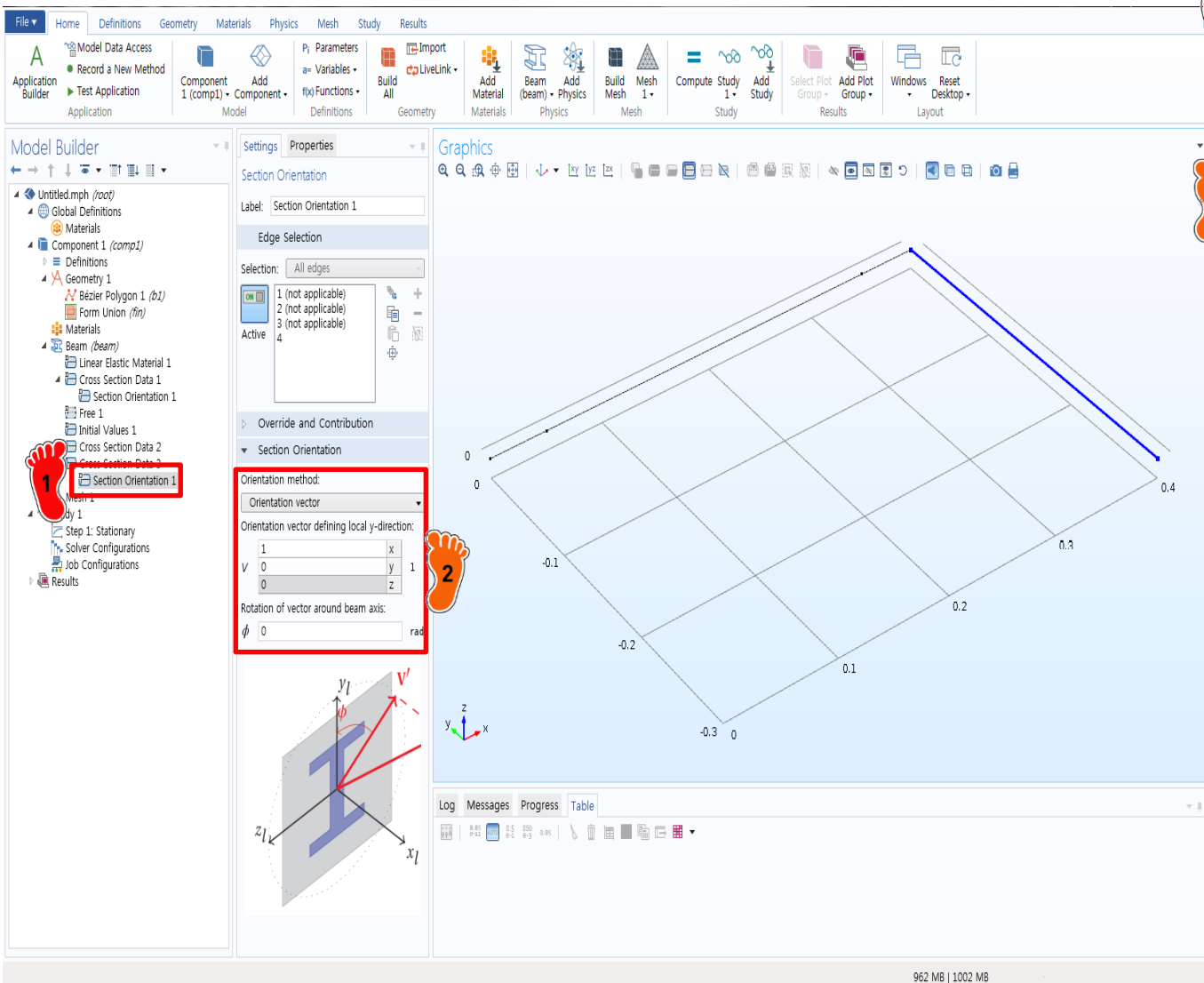
Circular 변경

지름 0.018 입력

CROSS SECTION

1 Cross Section Data 3 하위 메뉴인 Section Orientation 1 클릭

2 Orientation vector 변경 후 (1,0,0) 입력

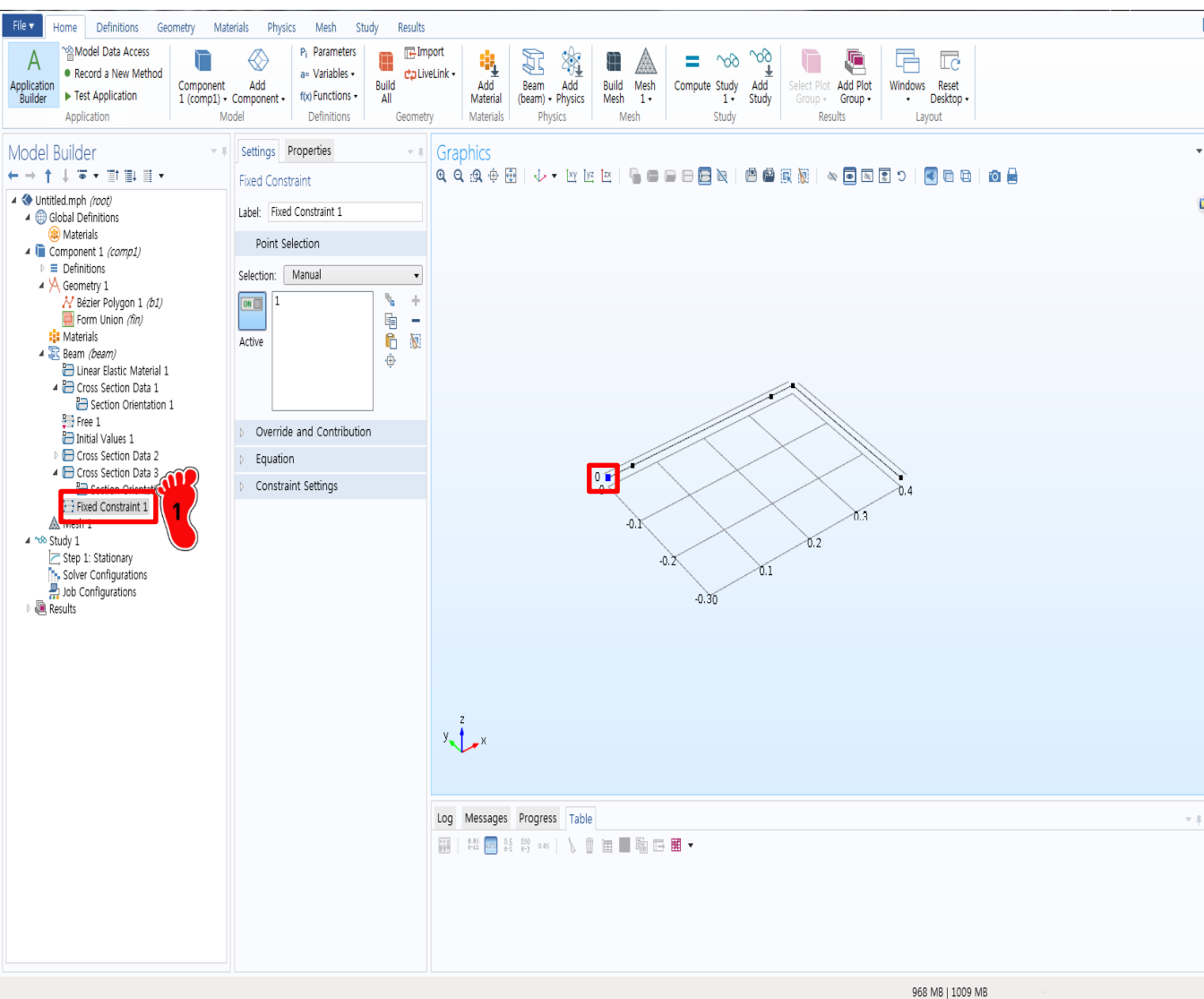


962 MB | 1002 MB

BOUNDARY CONDITION



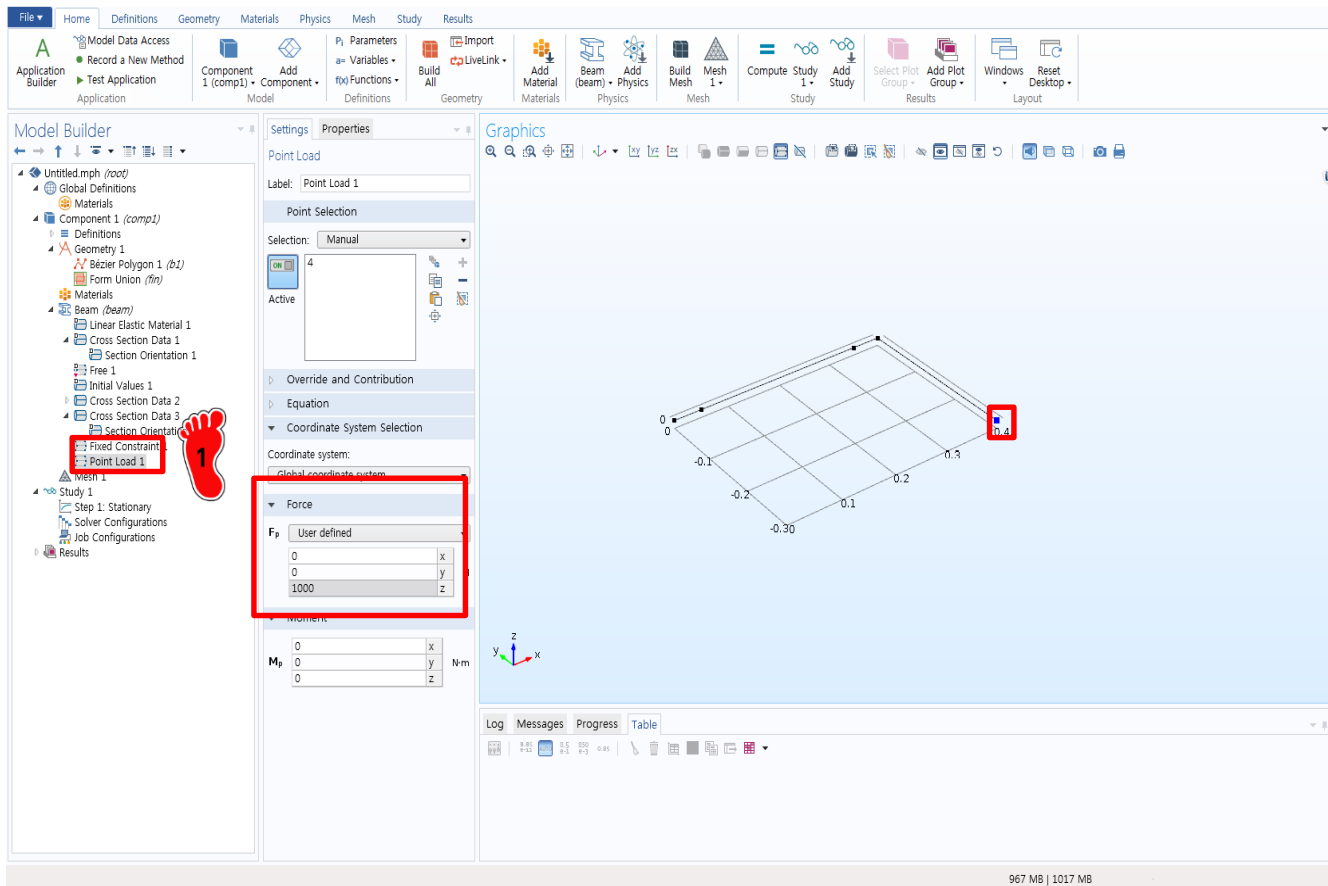
Fixed Constraint 를 그림에
표기된 절점에 입력



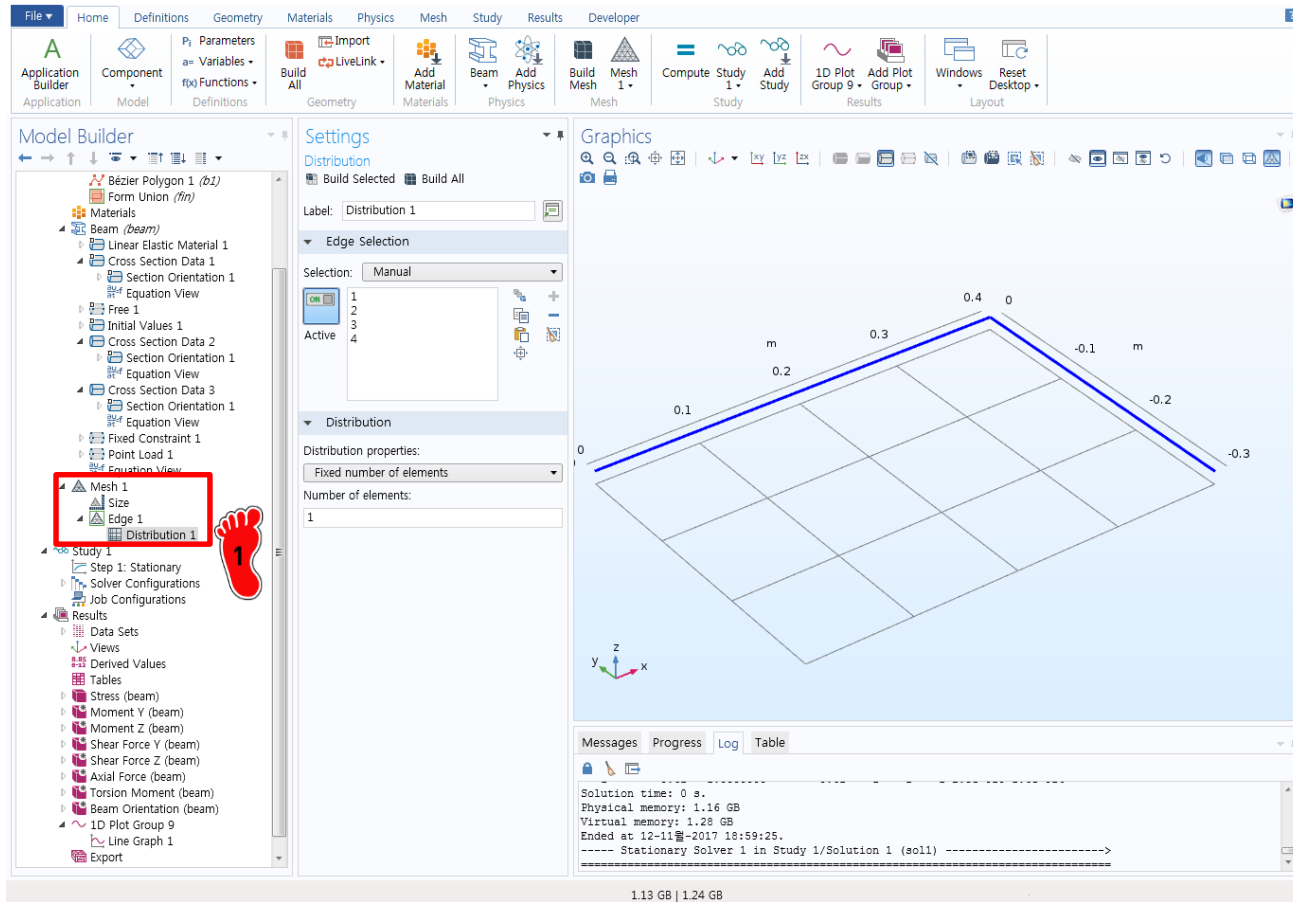
LOADING CONDITION



Point Load 조건을 그림에
표기된 절점에 z 방향으로
1000 N 입력



MESH



Distribution 1 를 갖는 메시 생성

확인 후 유한요소 해석 실행

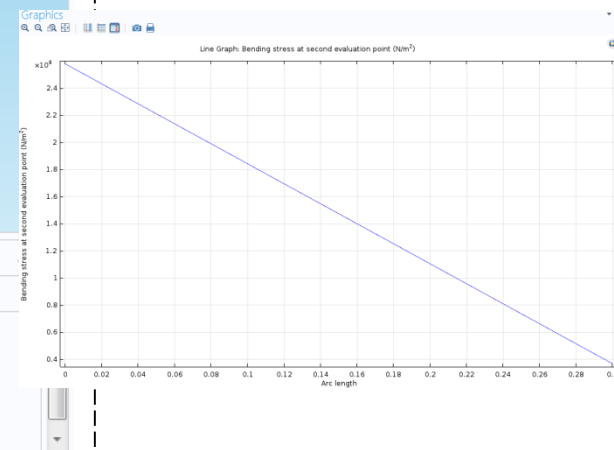
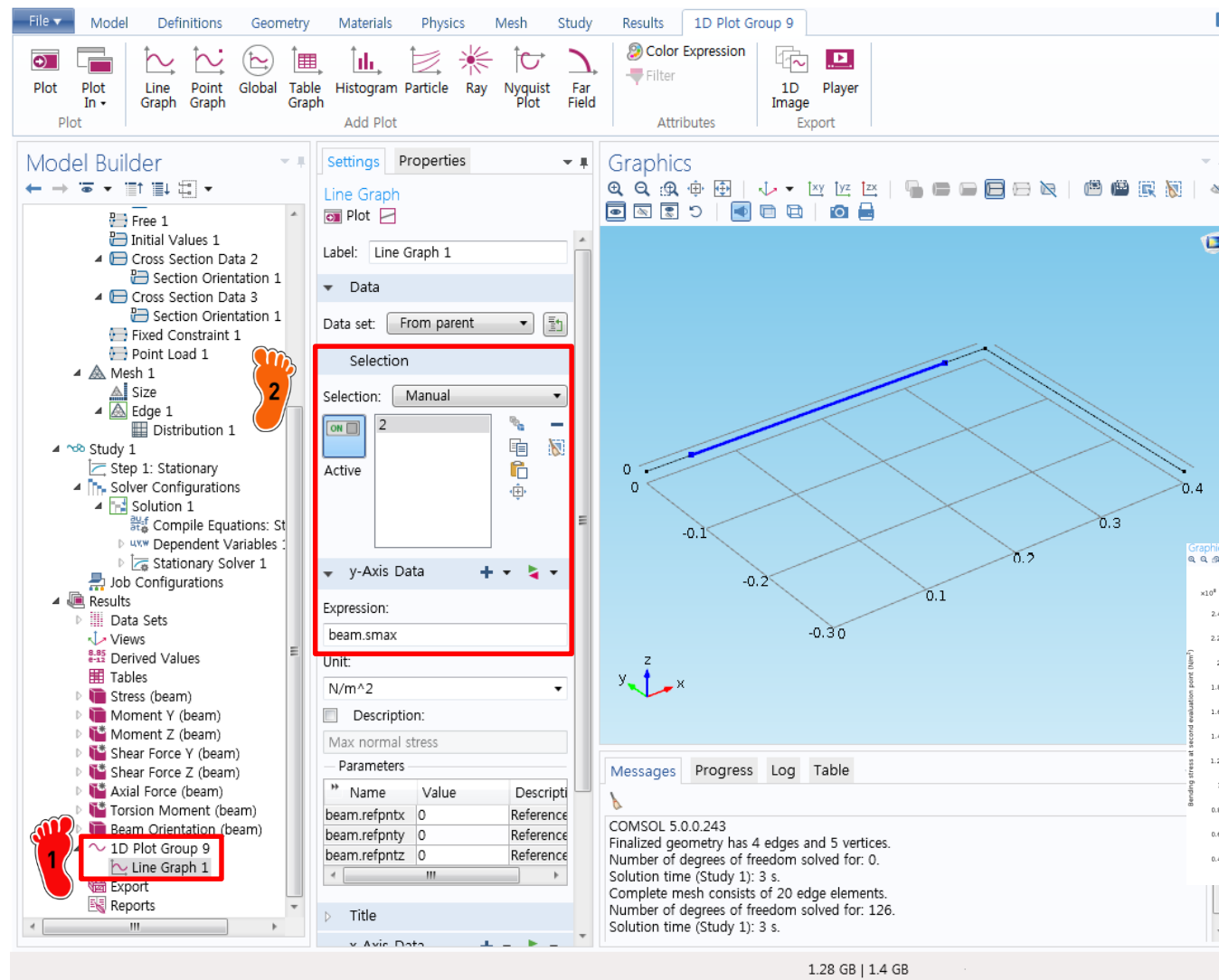
POST-PROCESSING

1 1D Plot Group 에서 Line 추가

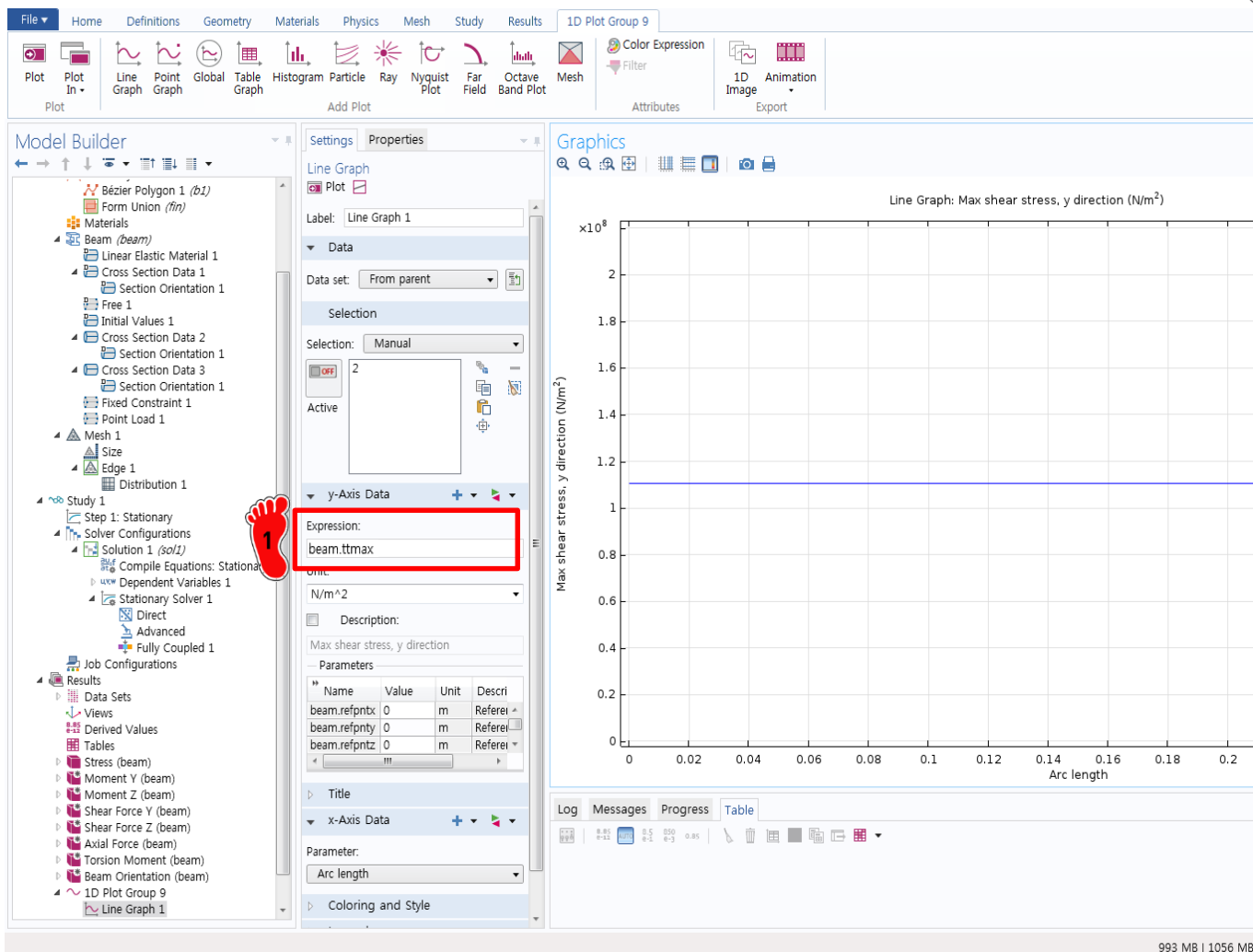
2 2번 부재 선택 후 beam.smax 입력

Plot 클릭

A 지점에서 $\sigma_x = 258 \text{ MPa}$



POST-PROCESSING

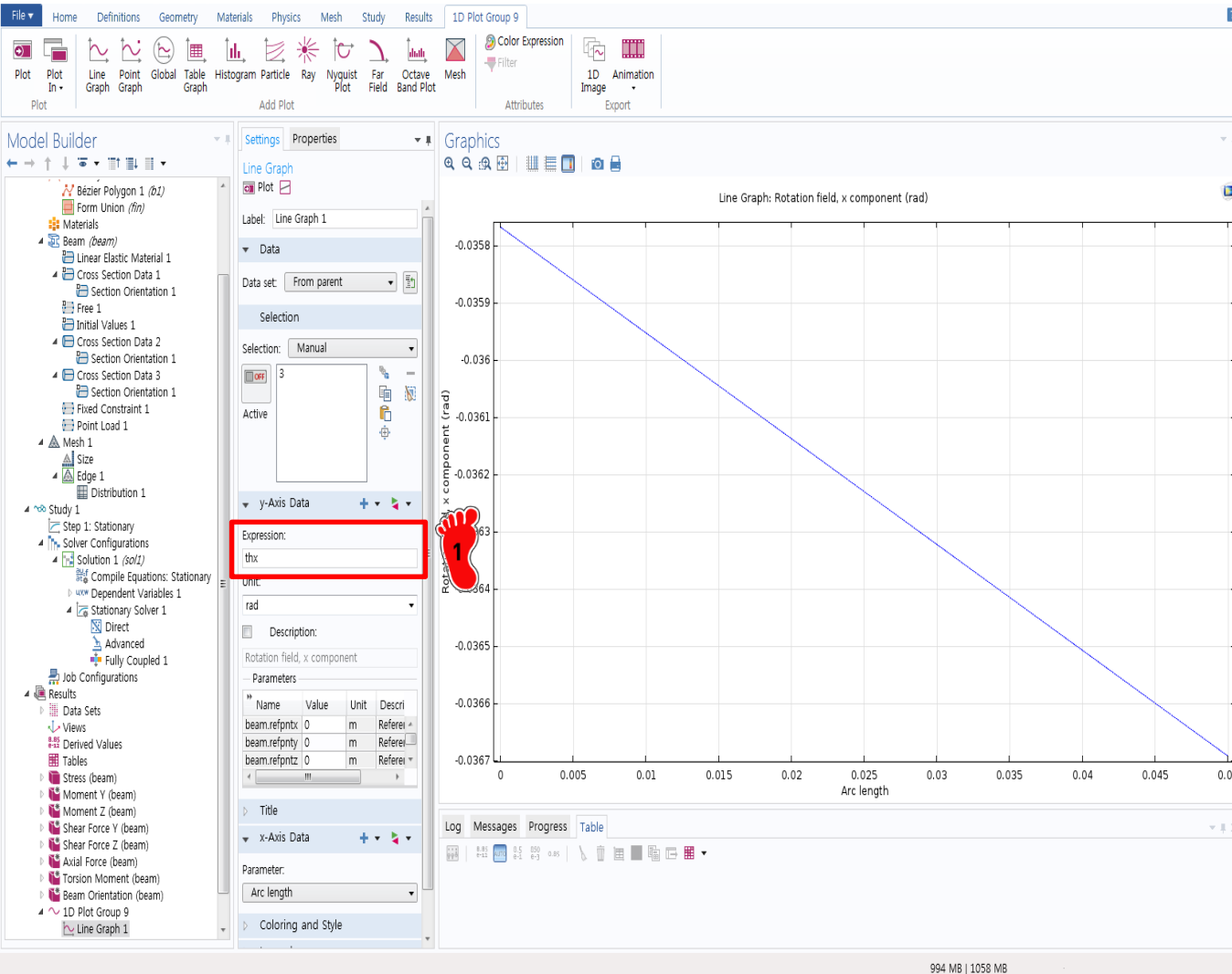


POST-PROCESSING



3번 부재의 회전 각도는 thx Expression 으로 확인

0.0367 rad 회전



POST-PROCESSING

The screenshot displays the ANSYS Workbench interface for post-processing. The Model Builder on the left shows the hierarchy: Model > Beam (beam) > Derived Values > Point Evaluation 1. The Properties panel for Point Evaluation 1 shows the Selection method set to 'Manual' and the Expression set to 'w'. The Graphics window shows a 3D plot of the displacement field. The Messages window at the bottom shows the value '0.024006' for the displacement at point 4.

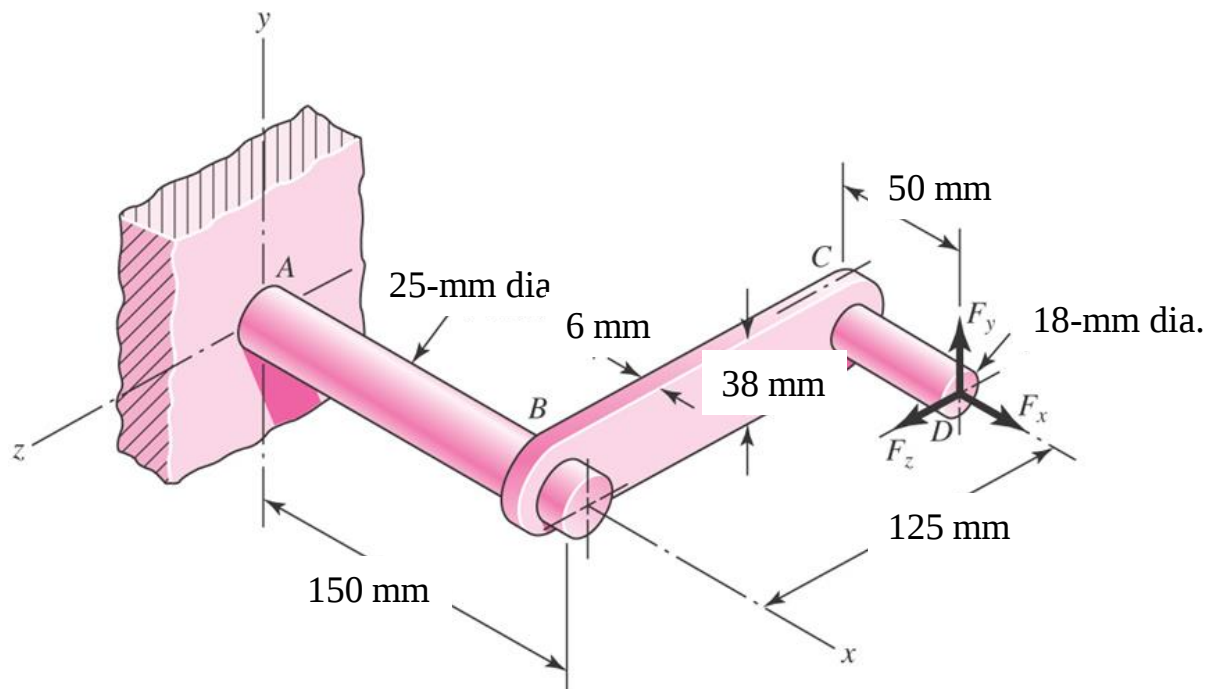
1 Derived Values에서
Point Evaluation 생성

2 4번 Point 선택 후
w 입력

z 방향 변위는 24.006 mm

	Anal.	FEM
σ_1 (MPa)	298.8	298.9
σ_2 (MPa)	-40.9	-40.9
$\tau_{\max}$ (MPa)	169.8	169.9
θ (rad)	0.0367	0.0367
y_D (mm)	24.005	24.006

ASSIGNMENT



Loads

- $F_y = -800 \text{ N}$
- $F_x = F_z = 0 \text{ N}$

[Obtain both analytical and numerical solutions]

The cantilevered handle is made from mild steel that has been welded at the joints.

- Determine the location of the critical stress.
- Determine the principal stresses and the maximum shear stress.
- Determine the vertical deflection at the tip. (Use superposition for the analytical solution.)

[solutions]

$$\sigma_1 = 113.7 \text{ MPa}, \sigma_2 = -9.4 \text{ MPa}, \tau_{\max} = 61.6 \text{ MPa}, y_D = -2.55 \text{ mm}$$