Solid Mechanics (plate/shell, 3D)

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

CONTENTS

- Plate and Shell Model
  - Thick plate
  - Hood
- 3D Model

SHELLS AND PLATES

SHELLS AND PLATES

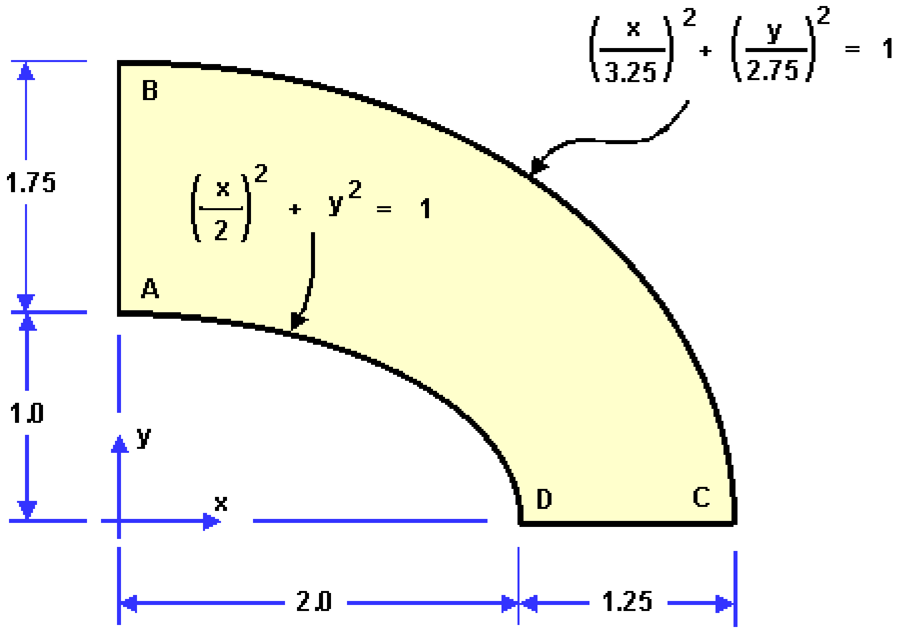
The Shell interface ( ) is intended for the structural analysis of thin-walled structures. The formulation used in the Shell interface is a Mindlin-Reissner type, which means that transverse shear deformations are accounted for, and it can therefore be used for rather thick shells. It is also possible to prescribe an offset in the direction normal to a selected surface. The Shell interface also includes other features such as damping, thermal expansion, and initial stresses and strains. The preset studies available are the same as for the Solid Mechanics interface.

The Plate interface ( ) is the 2D analogy to the 3D Shell interface. Plates are similar to shells but act in a single plane and usually only with out-of-plane loads.

NAFEMS BENCHMARK: LE10

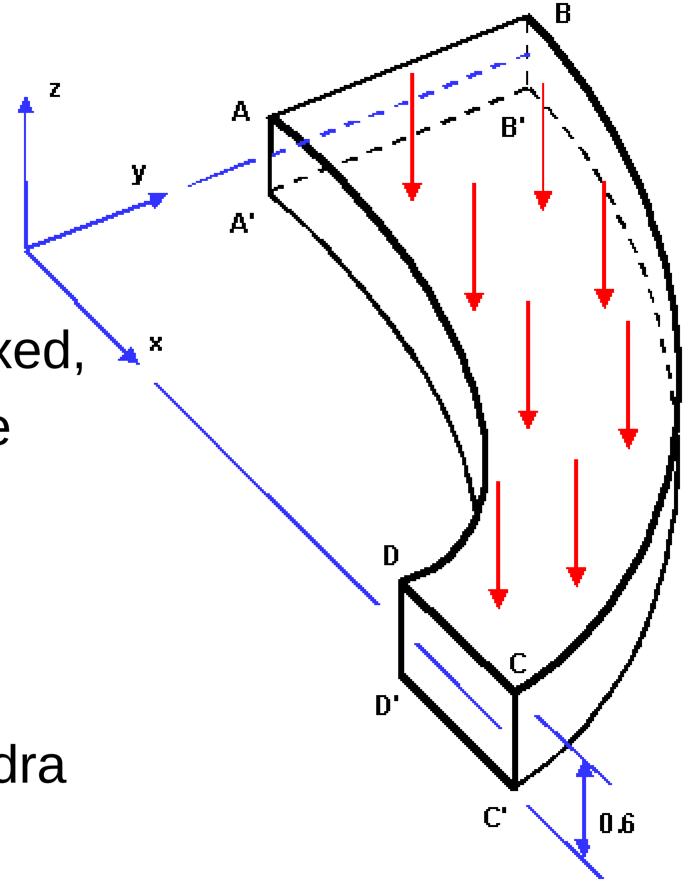
Thick Plate – Pressure

- Analysis Type: Linear elastic solid vs. plate
- Geometry: meters, thickness 0.6

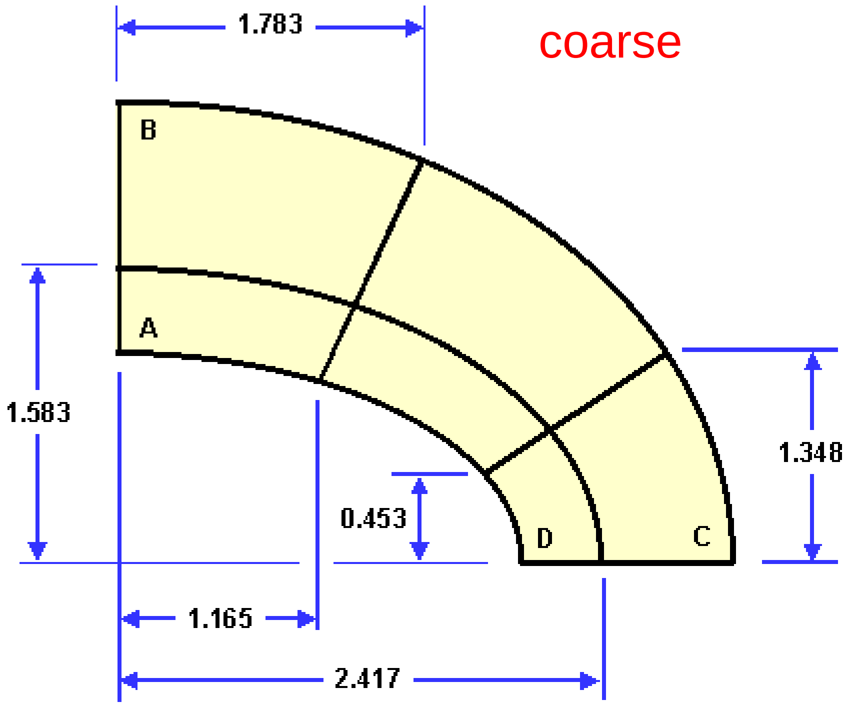


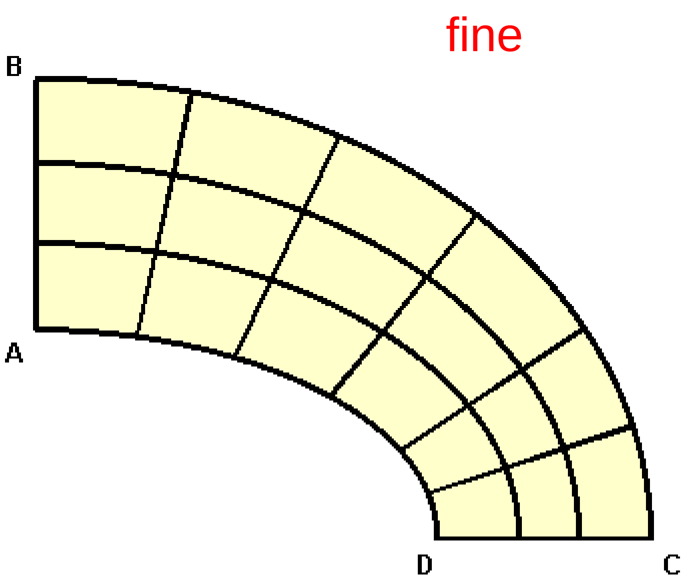
COMSOL Structural Mechanics Verification Examples: Thick Plate Stress Analysis

- Loading
  - Uniform normal pressure of 1 MPa on the upper surface of the plate
- Boundary conditions
  - Face DCD'C': zero y-displacement
  - Face ABA'B': zero x-displacement
  - Face BCB'C': x and y-displacement fixed, z-displacements fixed along mid-plane
- Material properties
  - Isotropic: E = 210 GPa, ν = 0.3
- Element types
  - Solid hexahedra, wedges and tetrahedra



- Meshes



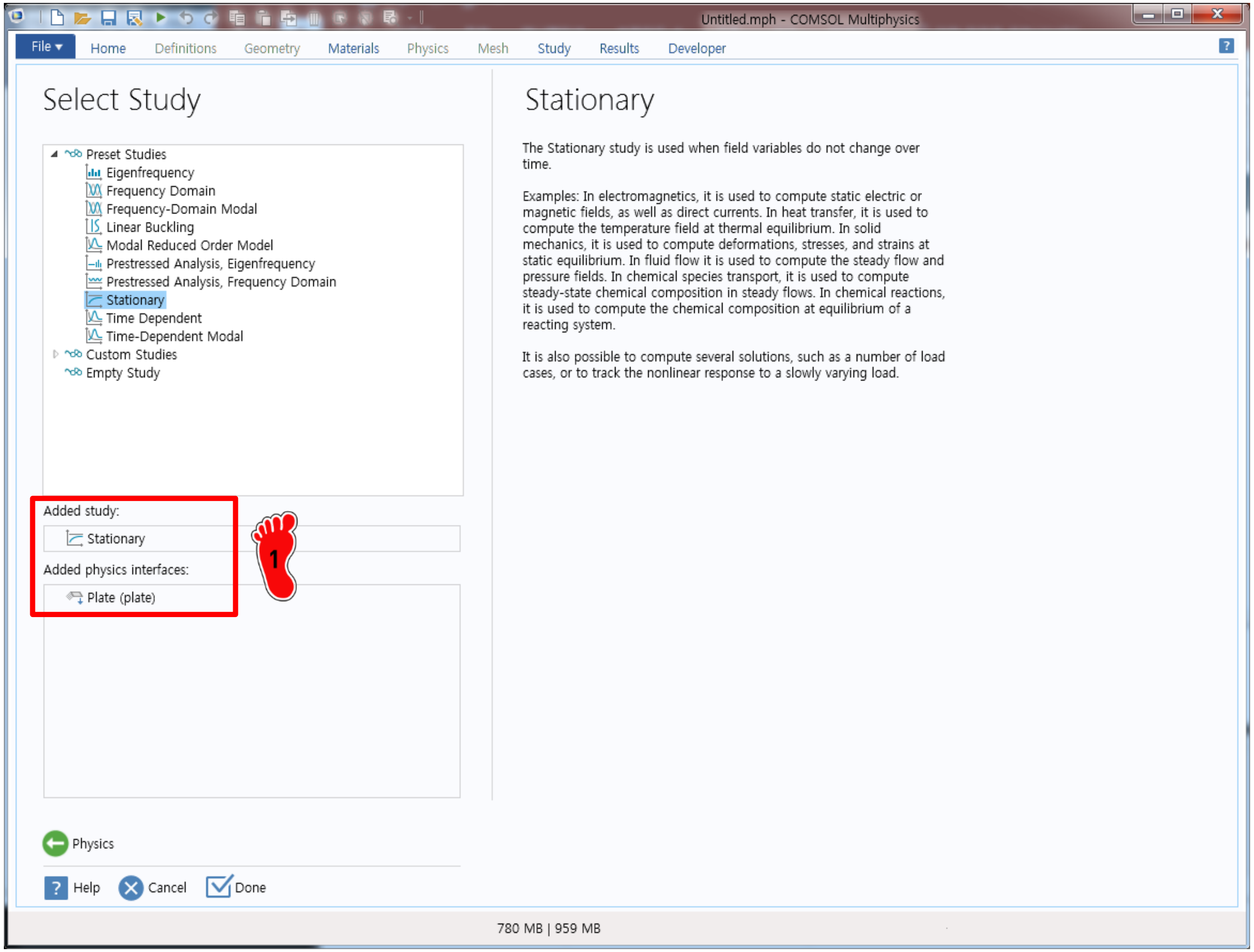


- Output
  - Direct stress (σ_{yy}) at point D: -5.38 MPa

LE10 COMPARISON

LE 10	order	integration points	element	coarse mesh s22	coarse mesh error	fine mesh s22	fine mesh error
Abaqus *	2	8	*C3D20R*	*-7.93E+06*	*47.40%*	*-5.53E+06*	*2.79%*
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
Abaqus *	2	27	*C3D20*	*-6.72E+06*	*24.91%*	*-5.64E+06*	*4.83%*
Ansys	2	*14*	SOLID95	-5.40E+06	0.46%	-5.61E+06	4.33%
CalculiX	2	27	C3D20	-5.20E+06	-3.32%	-5.50E+06	2.18%
Tahoe	2	27	hexahedron				

ENVIRONMENT SETTING

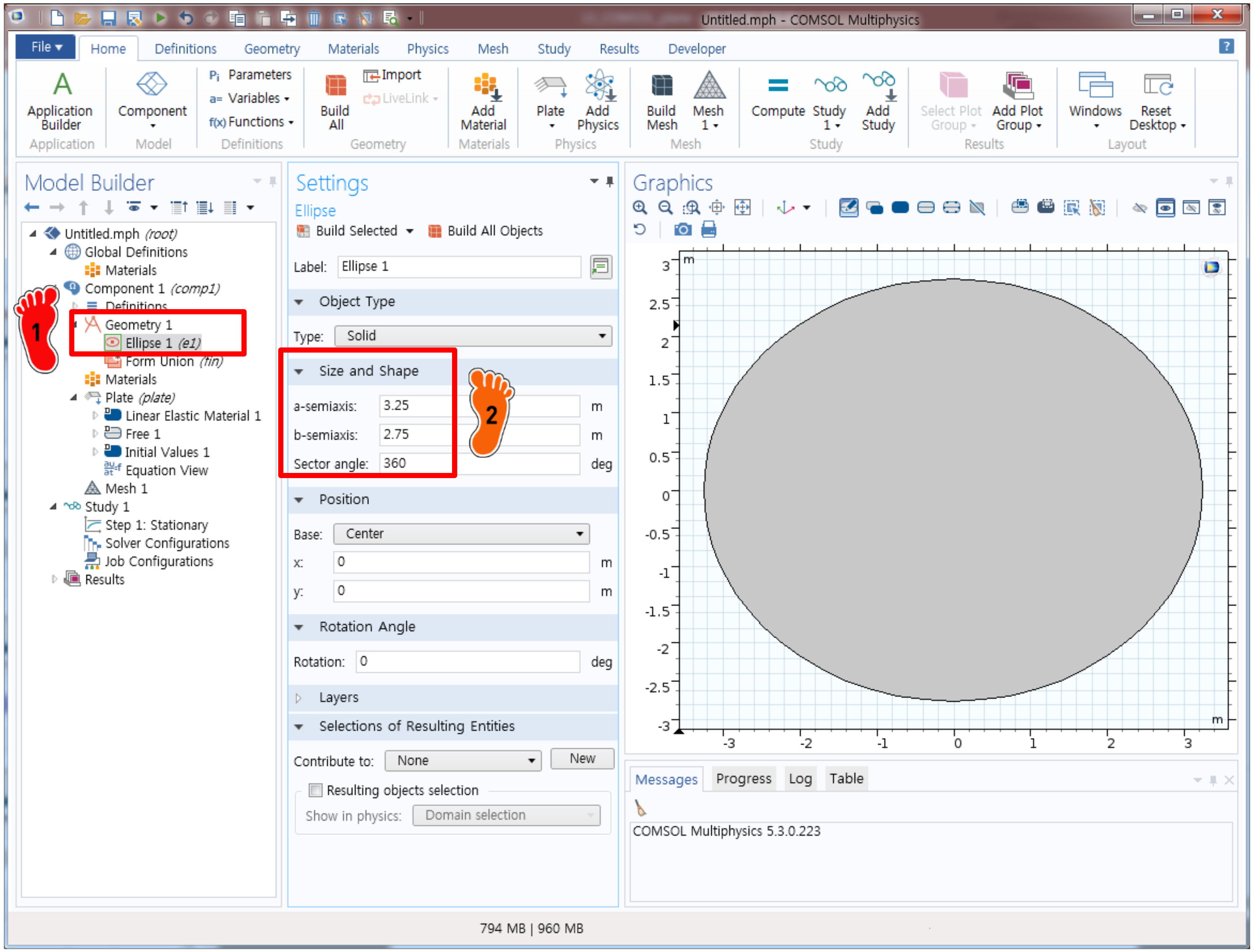


1

Dimension : 2D
Physics :
Structural Mechanics
→ Plate
Study : Stationary

Done 클릭

GEOMETRY CREATION

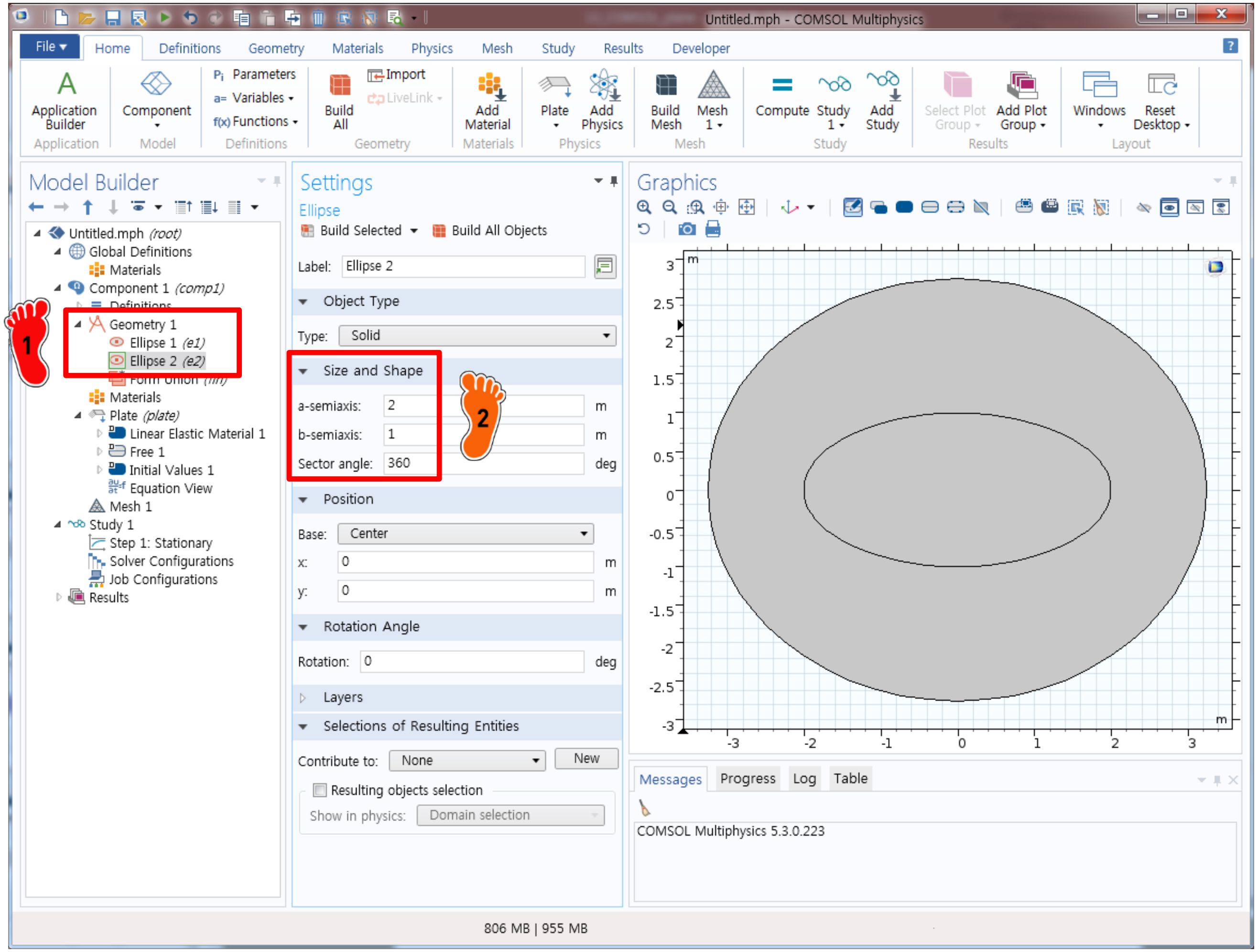


1 Ellipse 1 생성

2 a-semiaxis: 3.25
b-semiaxis: 2.75

입력 후 타원 생성

GEOMETRY CREATION

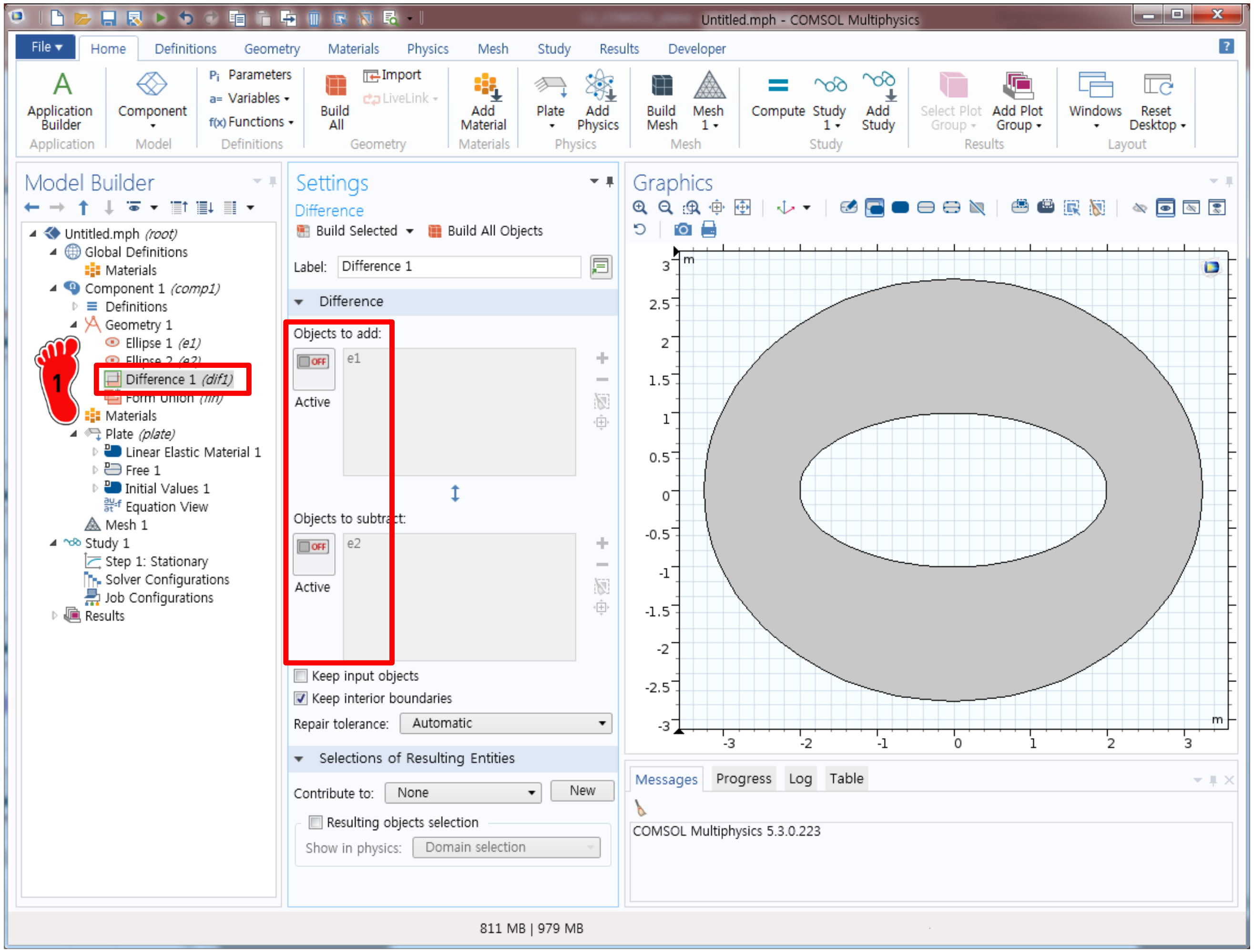


1. Ellipse 2 생성

2. a-semiaxis: 2
b-semiaxis: 1

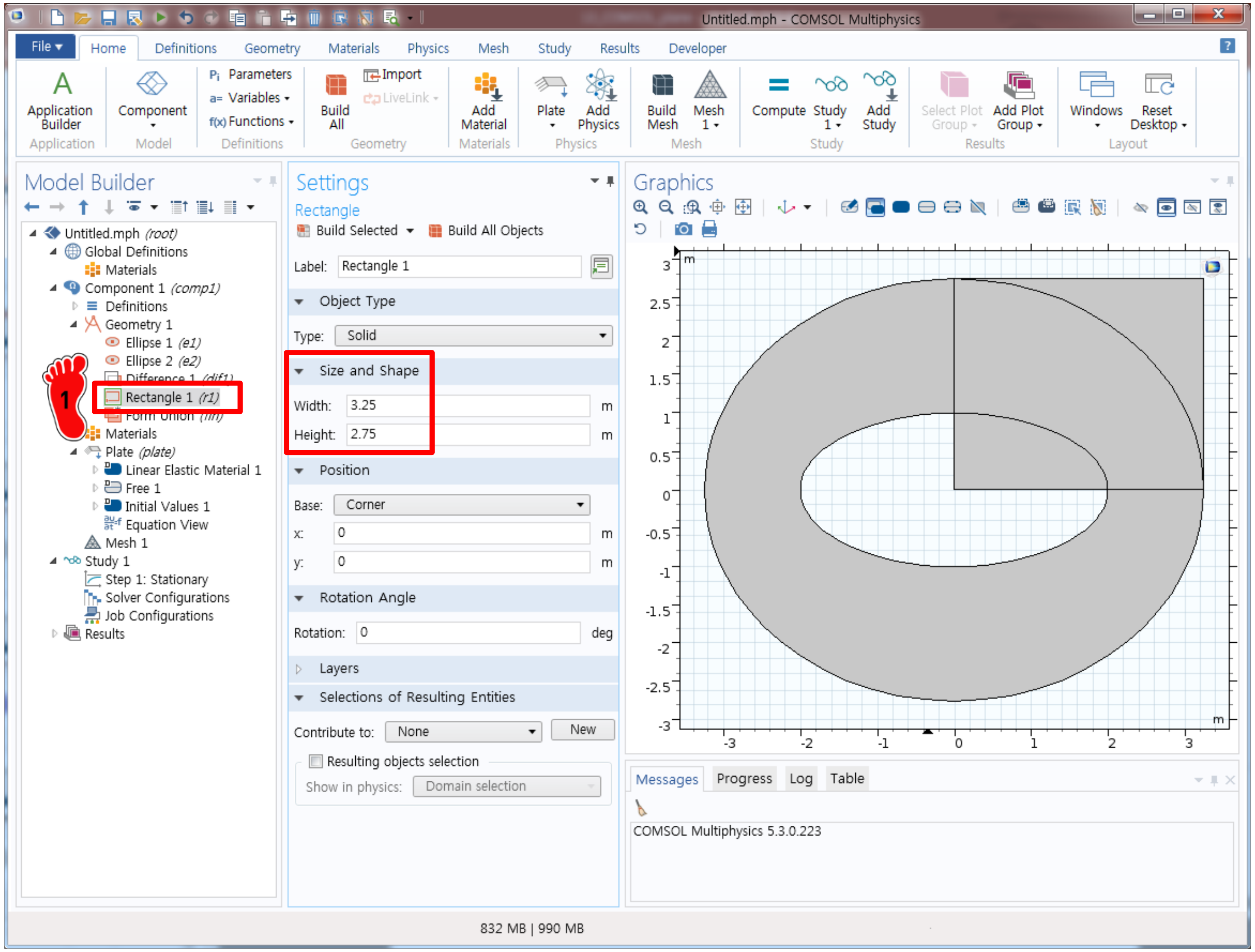
입력 후 타원 생성

GEOMETRY CREATION



1 Difference 기능을 이용해 구멍 생성

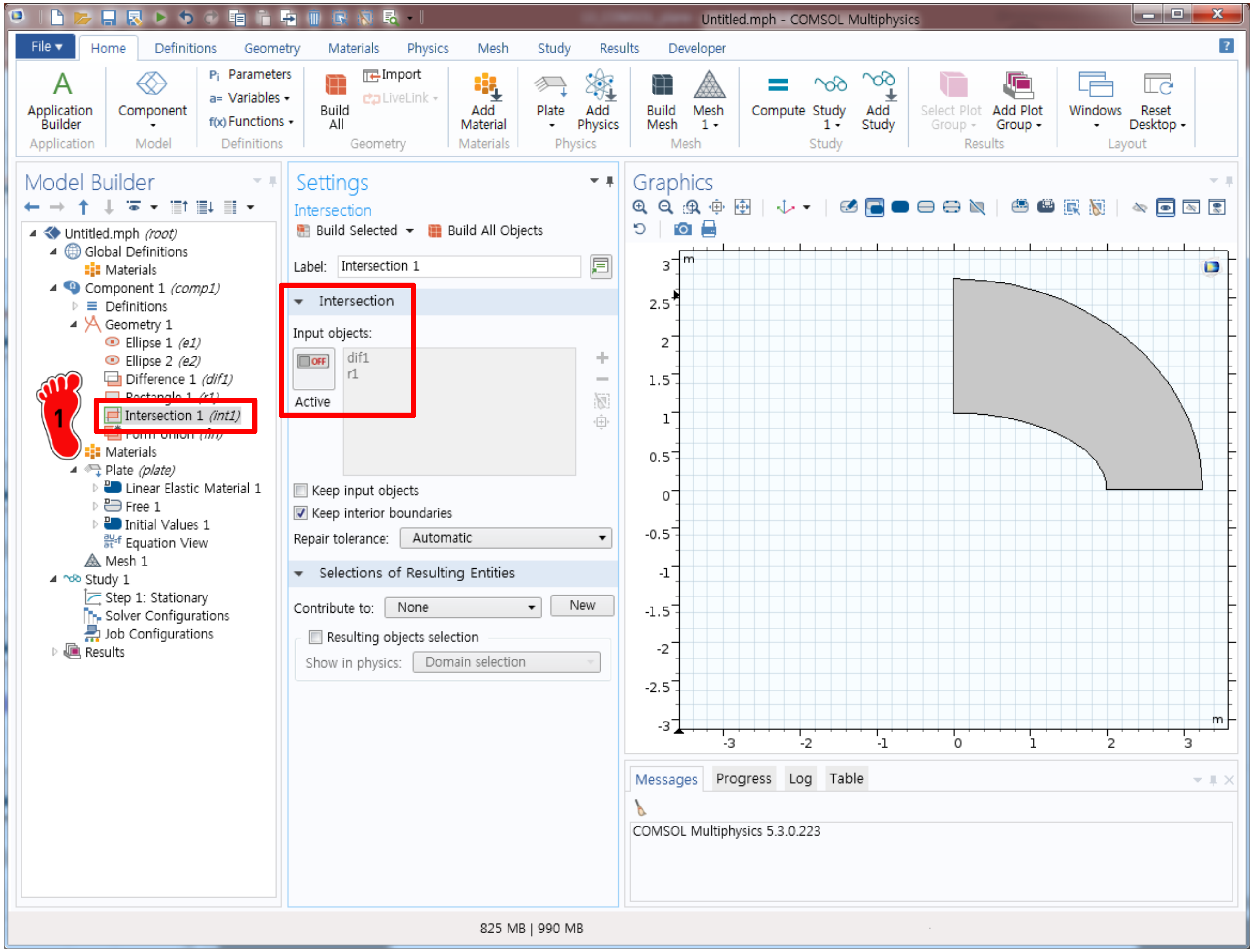
GEOMETRY CREATION



Rectangle 1 생성

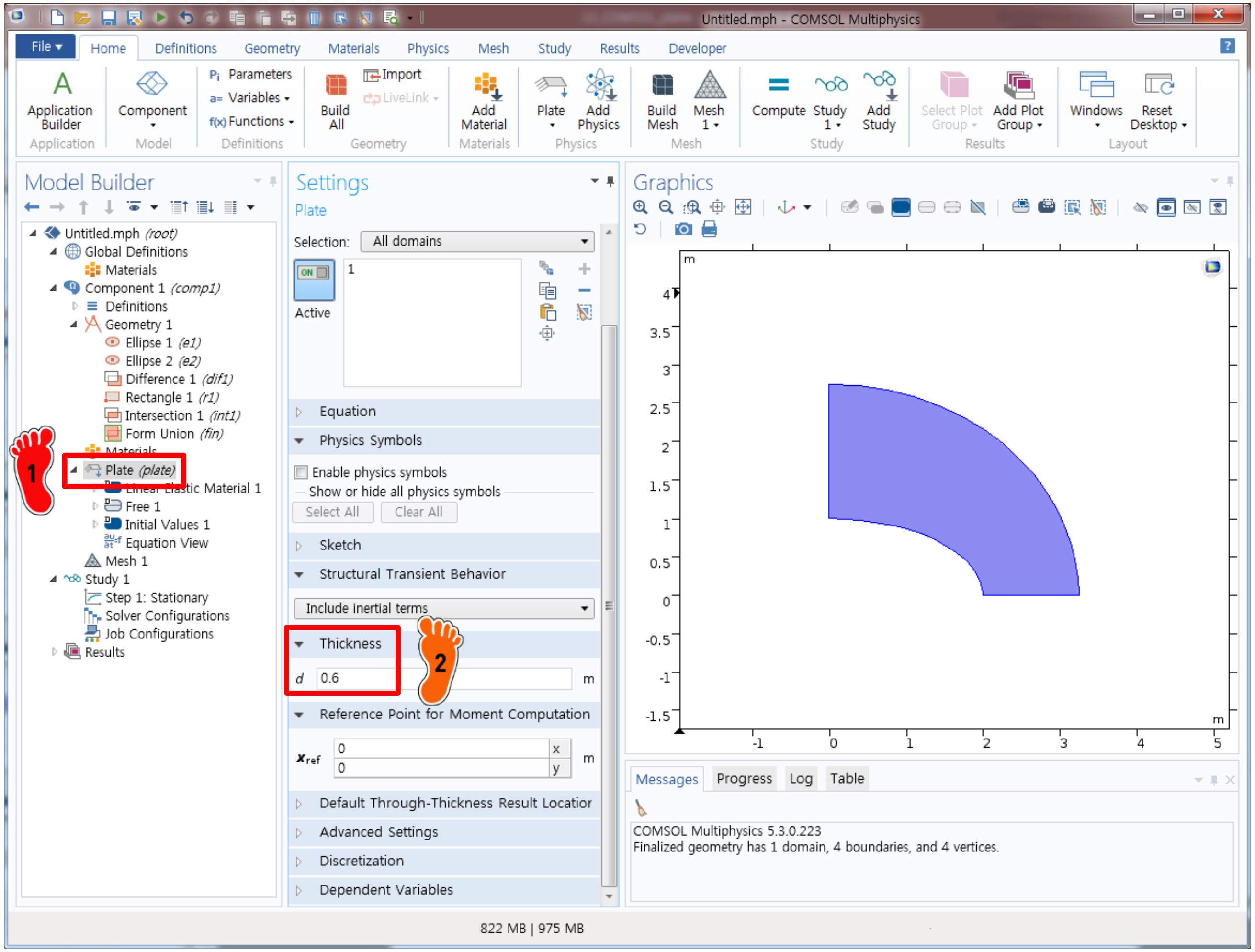
Width: 3.25
Height: 2.75

GEOMETRY CREATION



1 Intersection 기능을 사용하여 다음과 같은 형상 생성

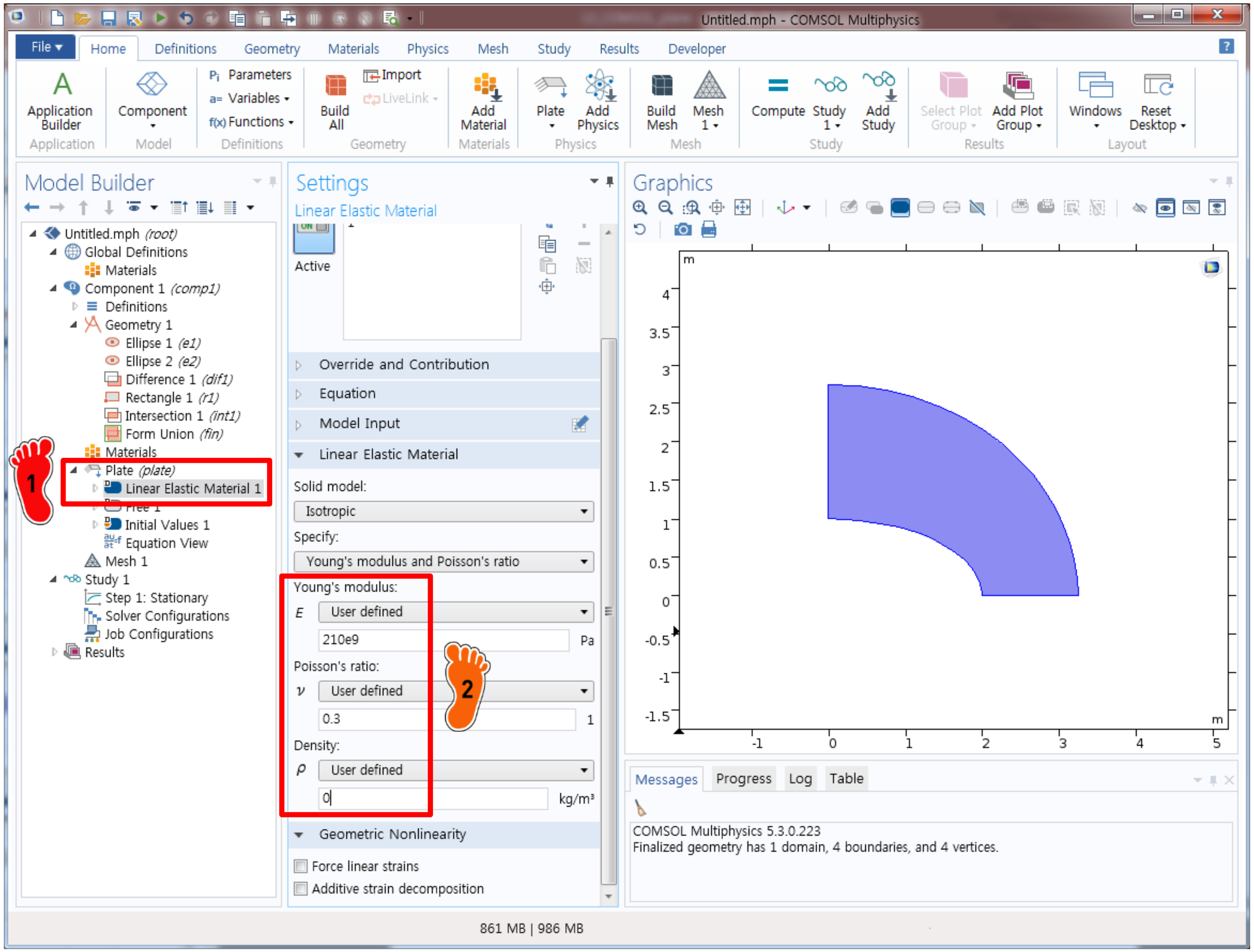
THICKNESS



1 Plate 클릭

2 Thickness
d: 0.6 입력

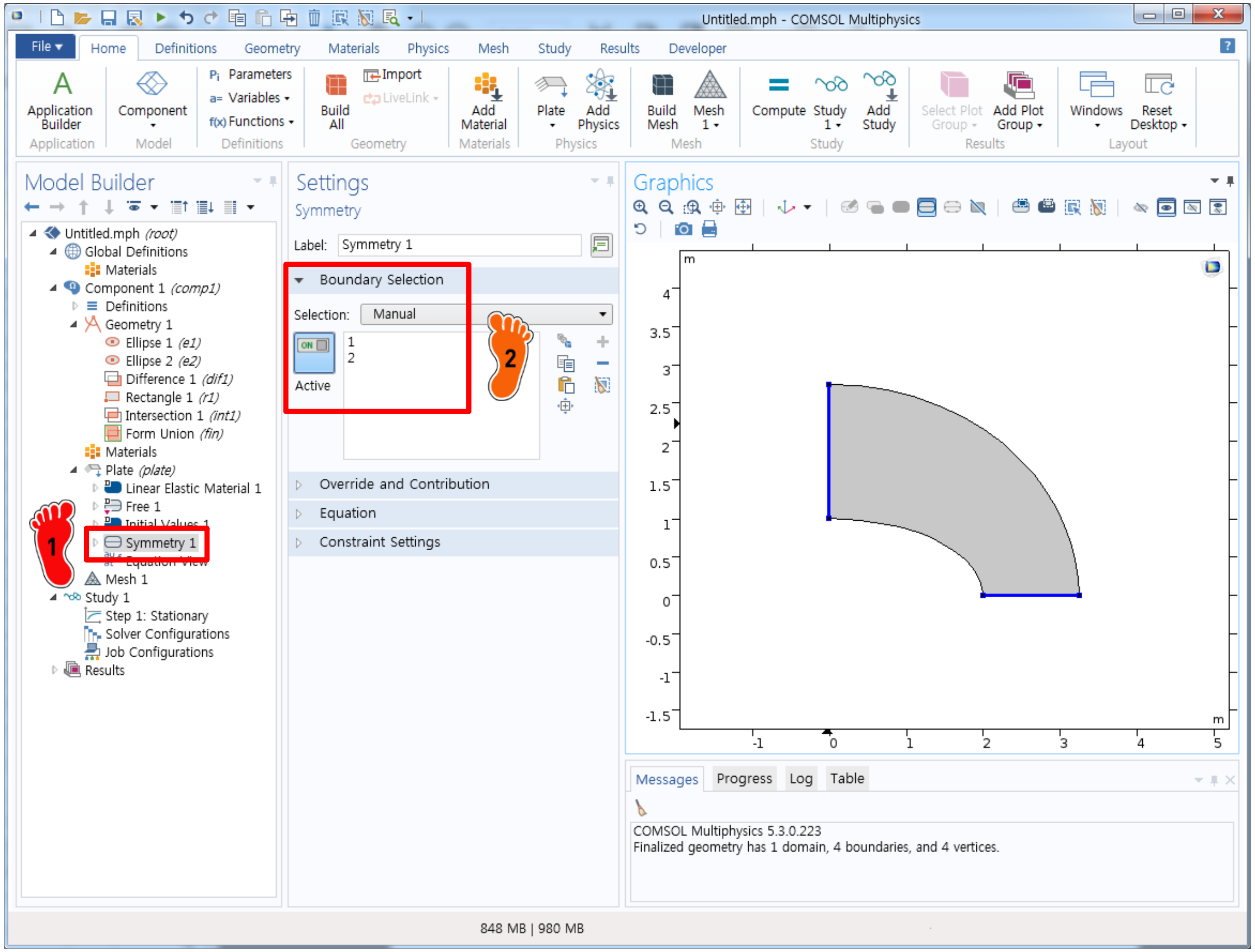
MATERIAL PROPERTY



1 Linear Elastic Material 클릭

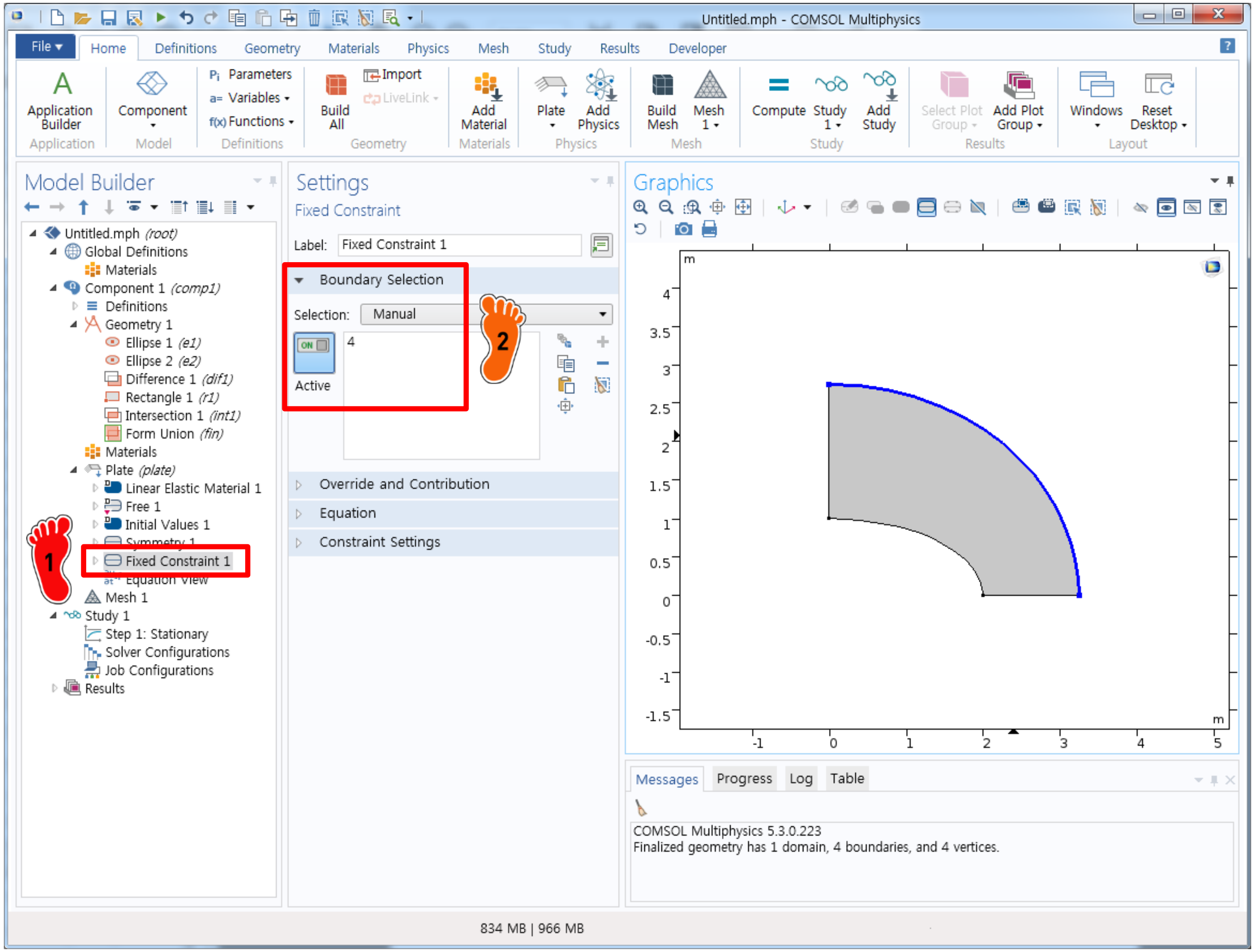
2 E: 210e9
mu: 0.3
rho : 0
입력

BOUNDARY CONDITION



1. Symmetry 경계조건 생성
2. 1, 2번 경계 선택

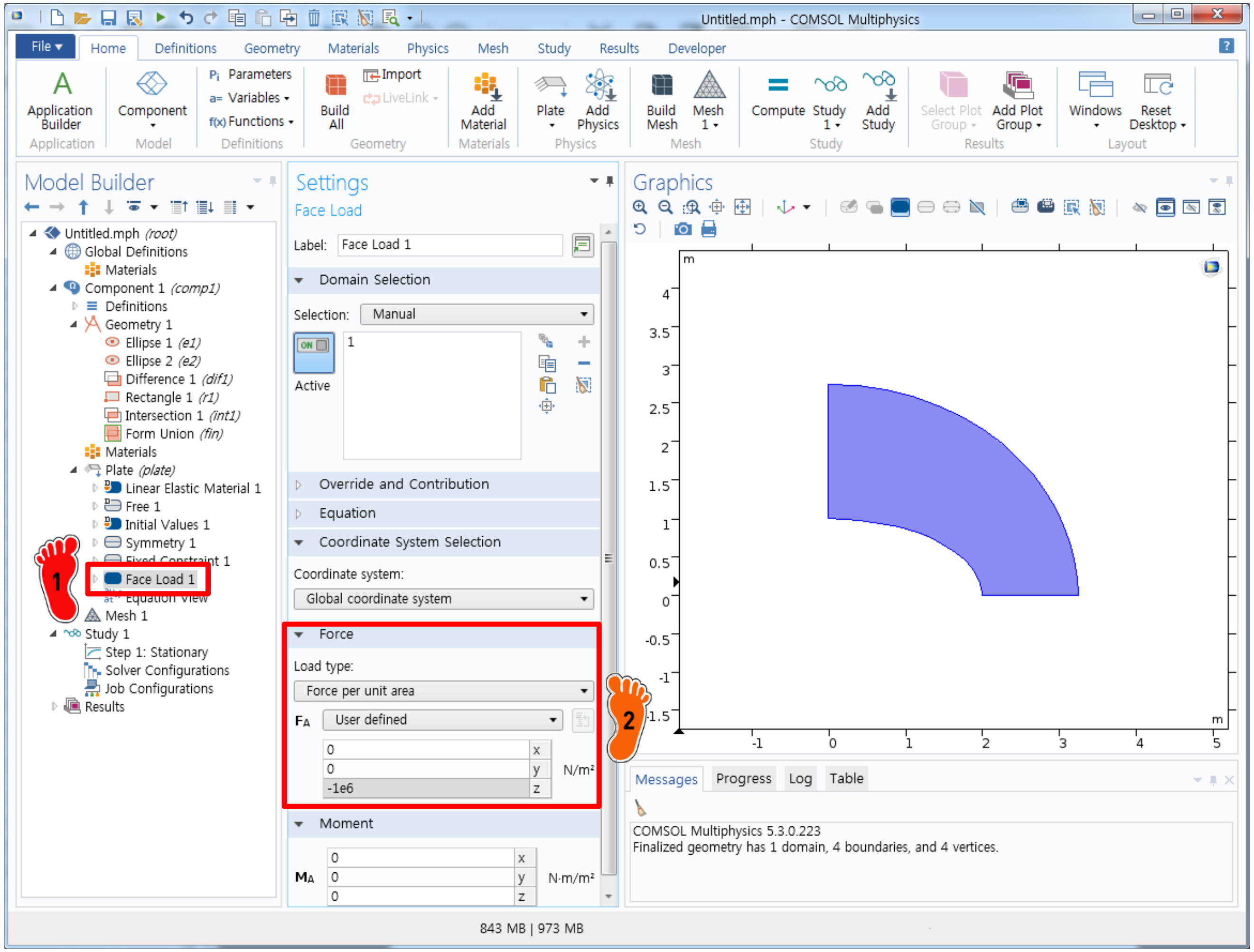
BOUNDARY CONDITION



1 Fixed 경계조건 생성

2 4번 경계 선택

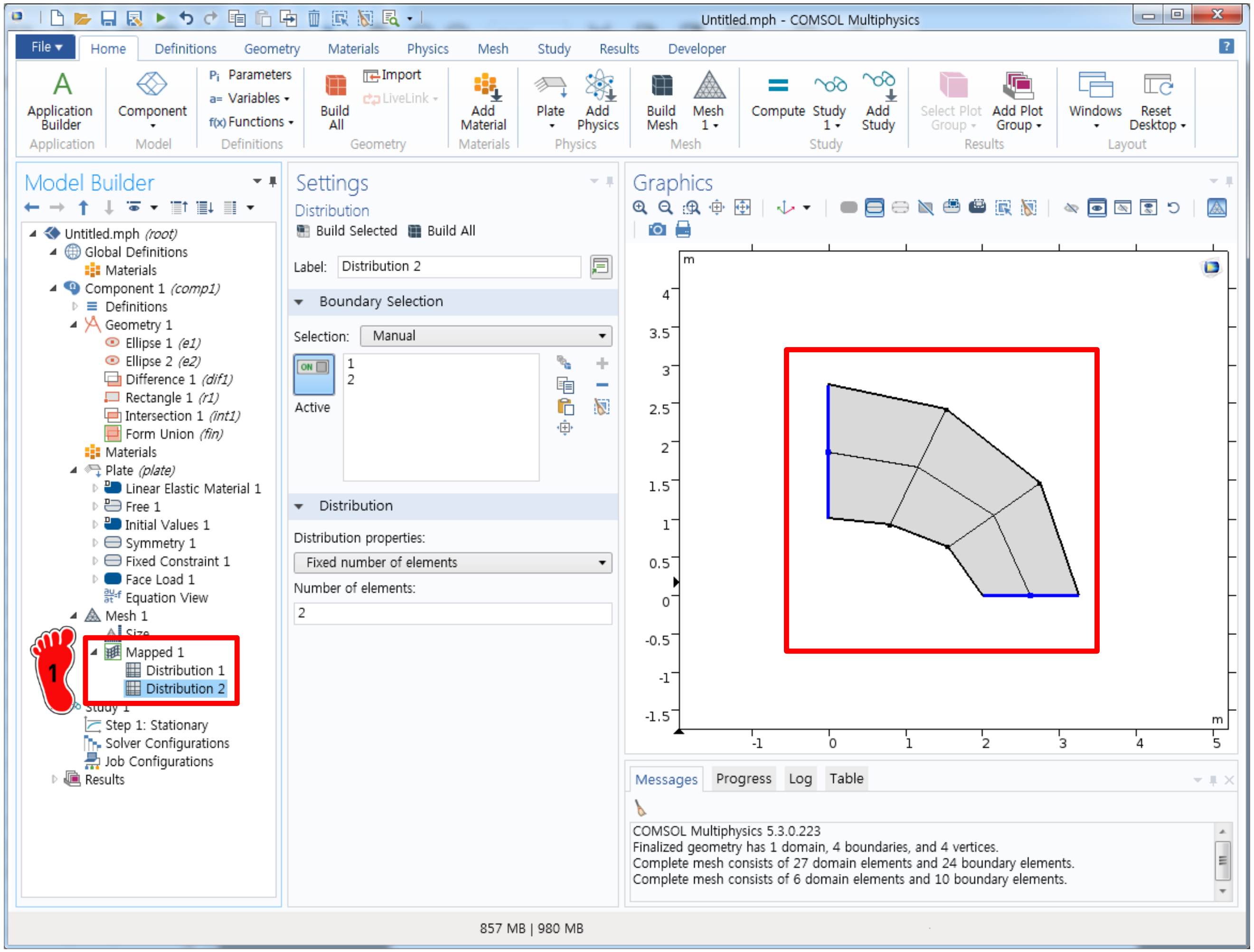
LOADING CONDITION



1 Face Load 생성

2 z 방향 -1e6 입력

MESH

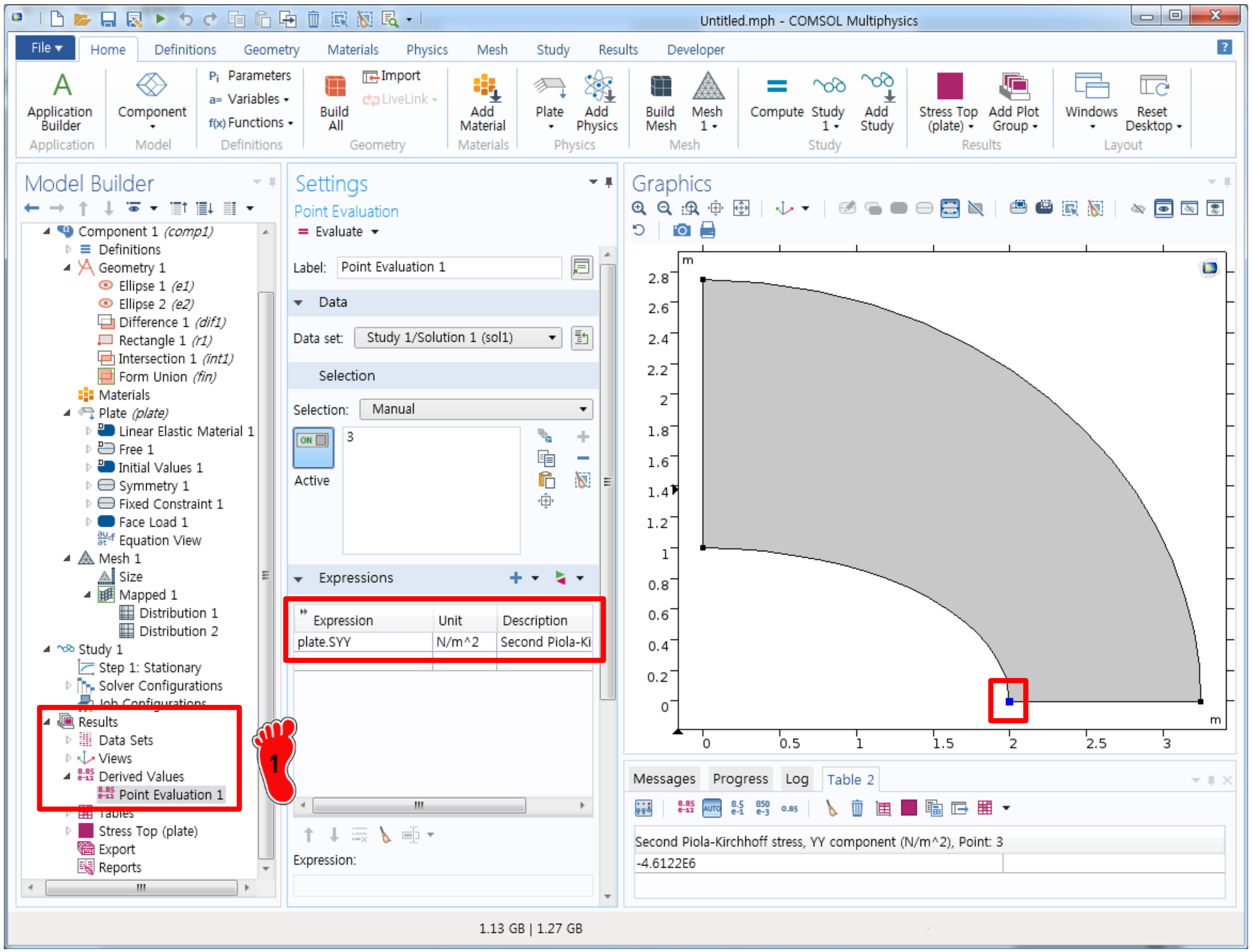


1 Mapped mesh의 Distribution 기능을 이용하여 요소 생성

Distribution 1
→ 3,4 bnd. elements: 3
Distribution 2
→ 1,2 bnd. elements: 2

Compute를 클릭하여 해석 수행

POST-PROCESSING



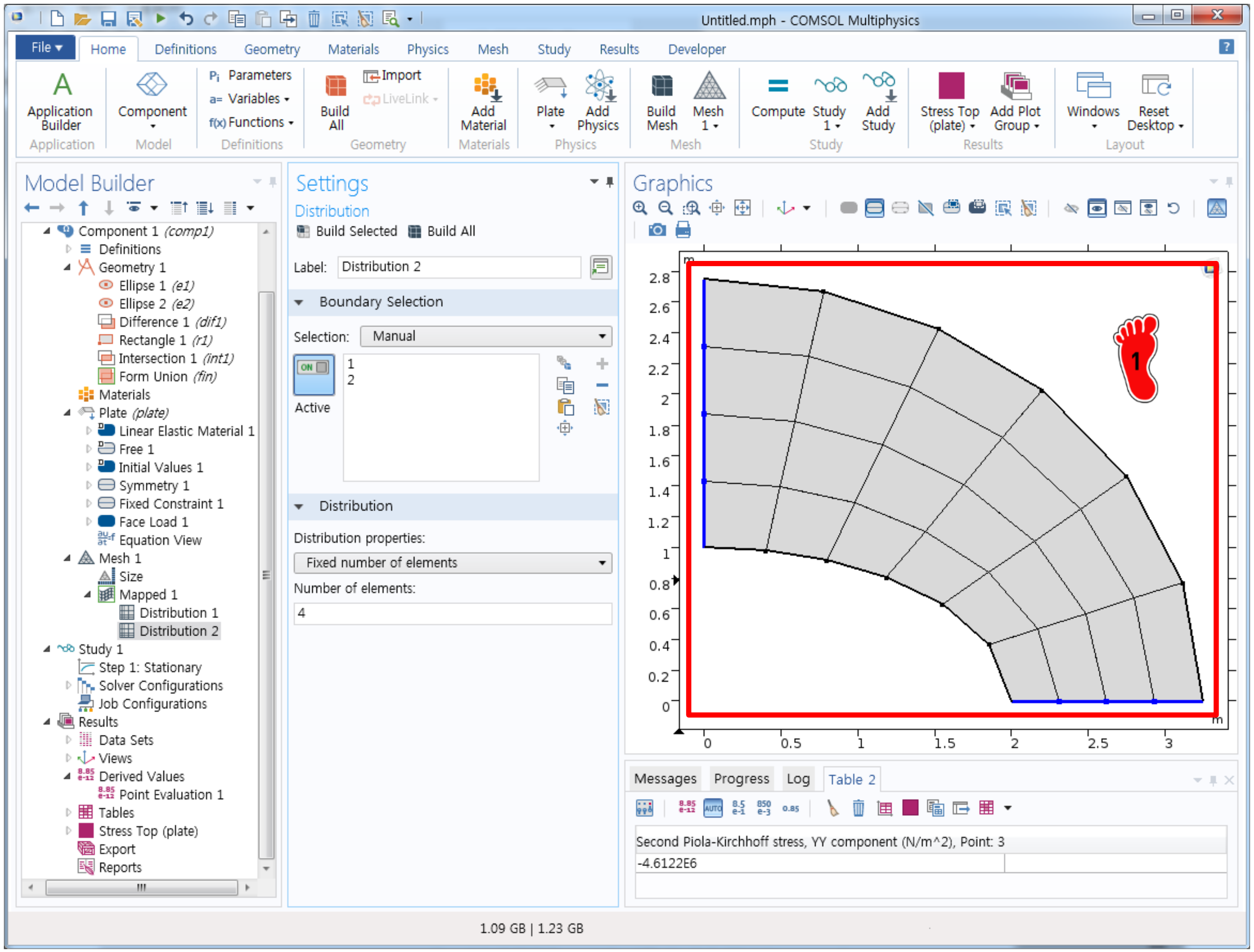
1 Derived Values 의 Point Evaluation 기능을 이용하여 3번 절점의 plate.Syy 응력을 계산

plate.SYY: Second Piola-Kirchhoff stress, yy component

결과값: -4.61 MPa

※ Ref : -5.38 MPa

MESH



Fine mesh로 변경

Distribution 1
→ 3,4 bnd. elements: 6
Distribution 2
→ 1,2 bnd. elements: 4

Compute를 클릭하여
해석 수행

결과값: -4.98 MPa

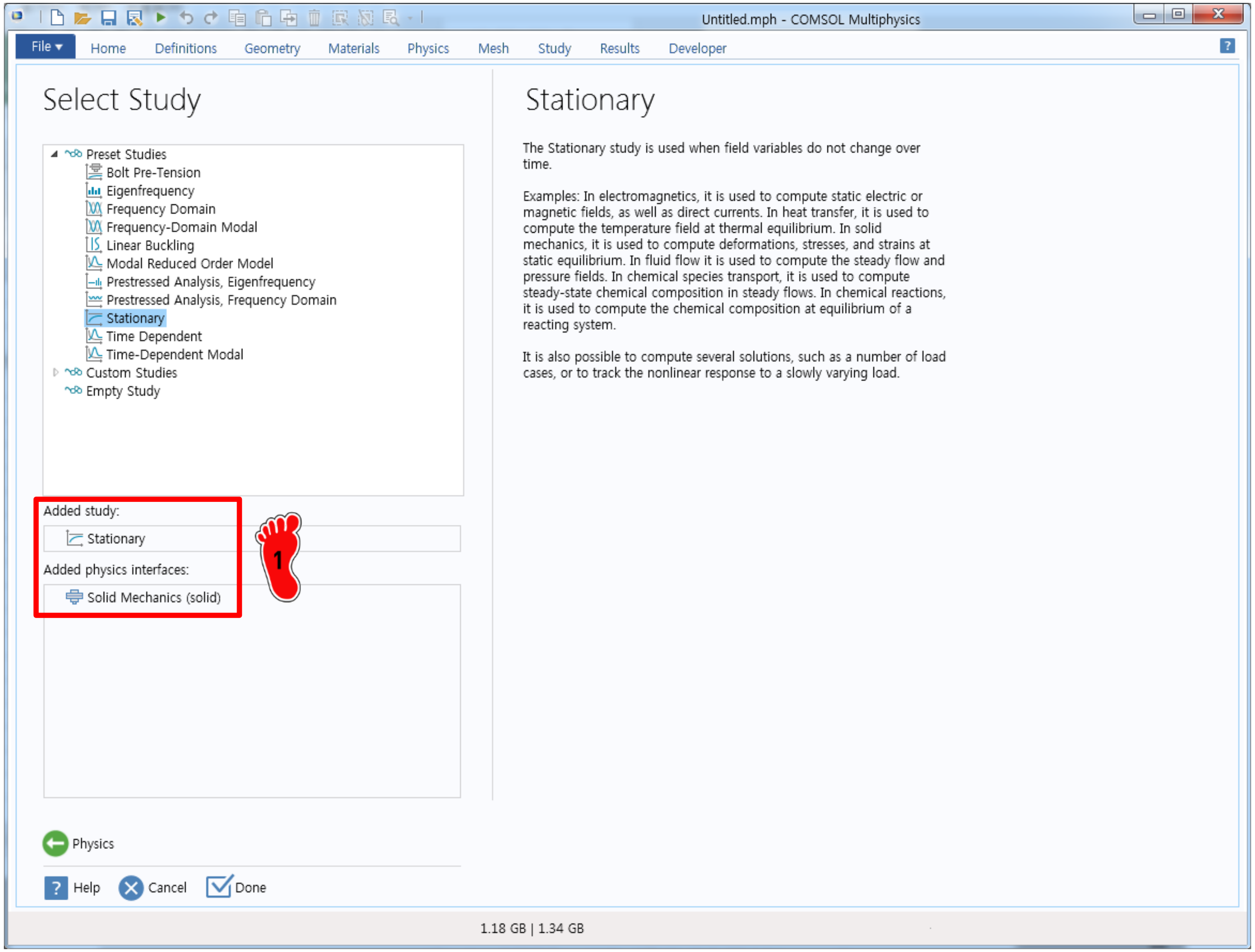
※ Ref : -5.38 MPa

SUMMARY

LE 10	order	integration points	element	coarse mesh		fine mesh	
				s22	error	s22	error
Abaqus *	2	8	*C3D20R*	*-7.93E+06*	*47.40%*	*-5.53E+06*	*2.79%*
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
Abaqus *	2	*27*	*C3D20*	*-6.72E+06*	*24.91%*	*-5.64E+06*	*4.83%*
Ansys	2	*14*	SOLID95	-5.40E+06	0.46%	-5.61E+06	4.33%
CalculiX	2	27	C3D20	-5.20E+06	-3.32%	-5.50E+06	2.18%
Tahoe	2	27	hexahedron				

COMSOL	2		plate	-4.61E+06	14.3 %	-4.98E+06	7.4 %
COMSOL	2		hexahedron				

ENVIRONMENT SETTING

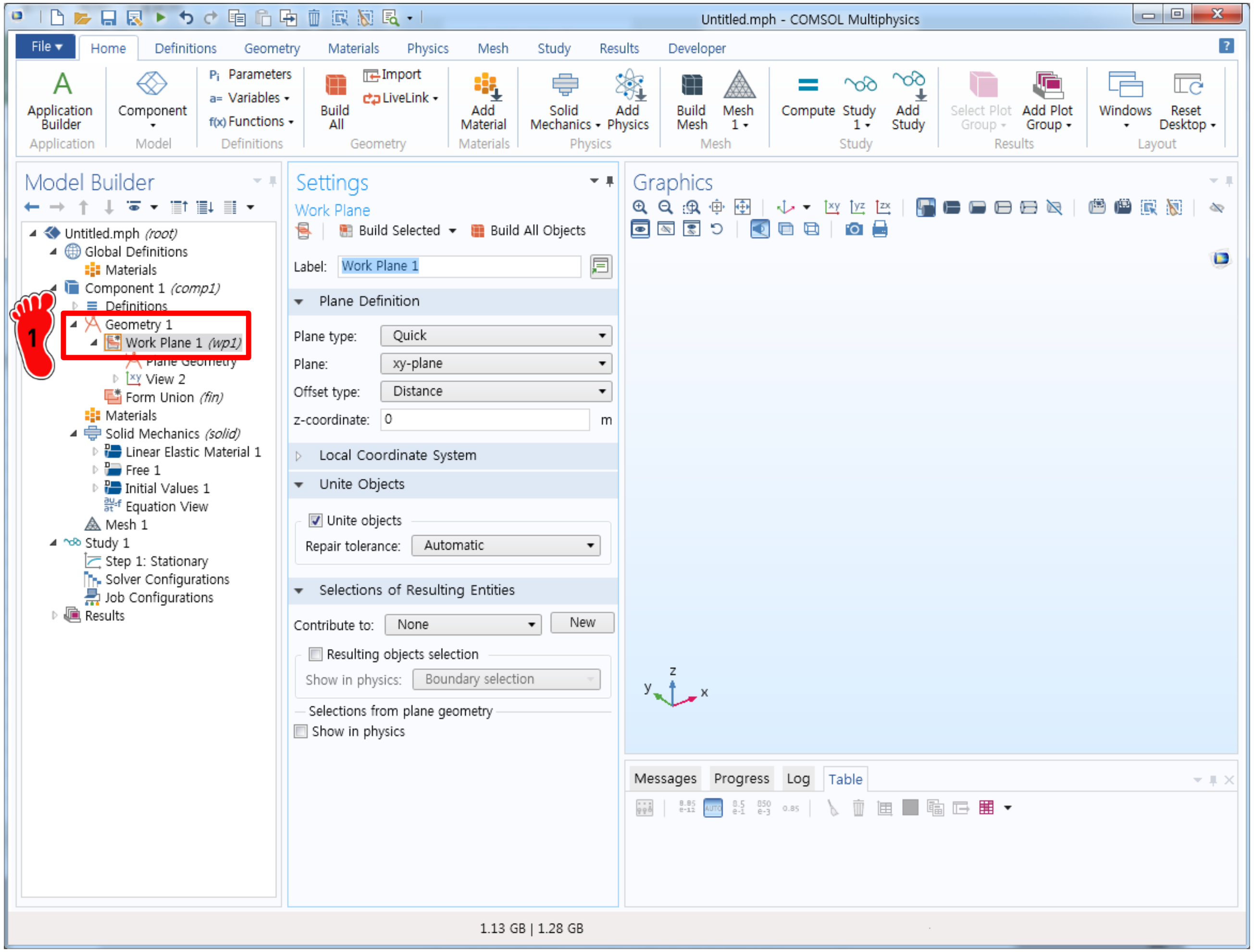


1

Dimension : 3D
Physics :
Structural Mechanics
→ Solid Mechanics
Study : Stationary

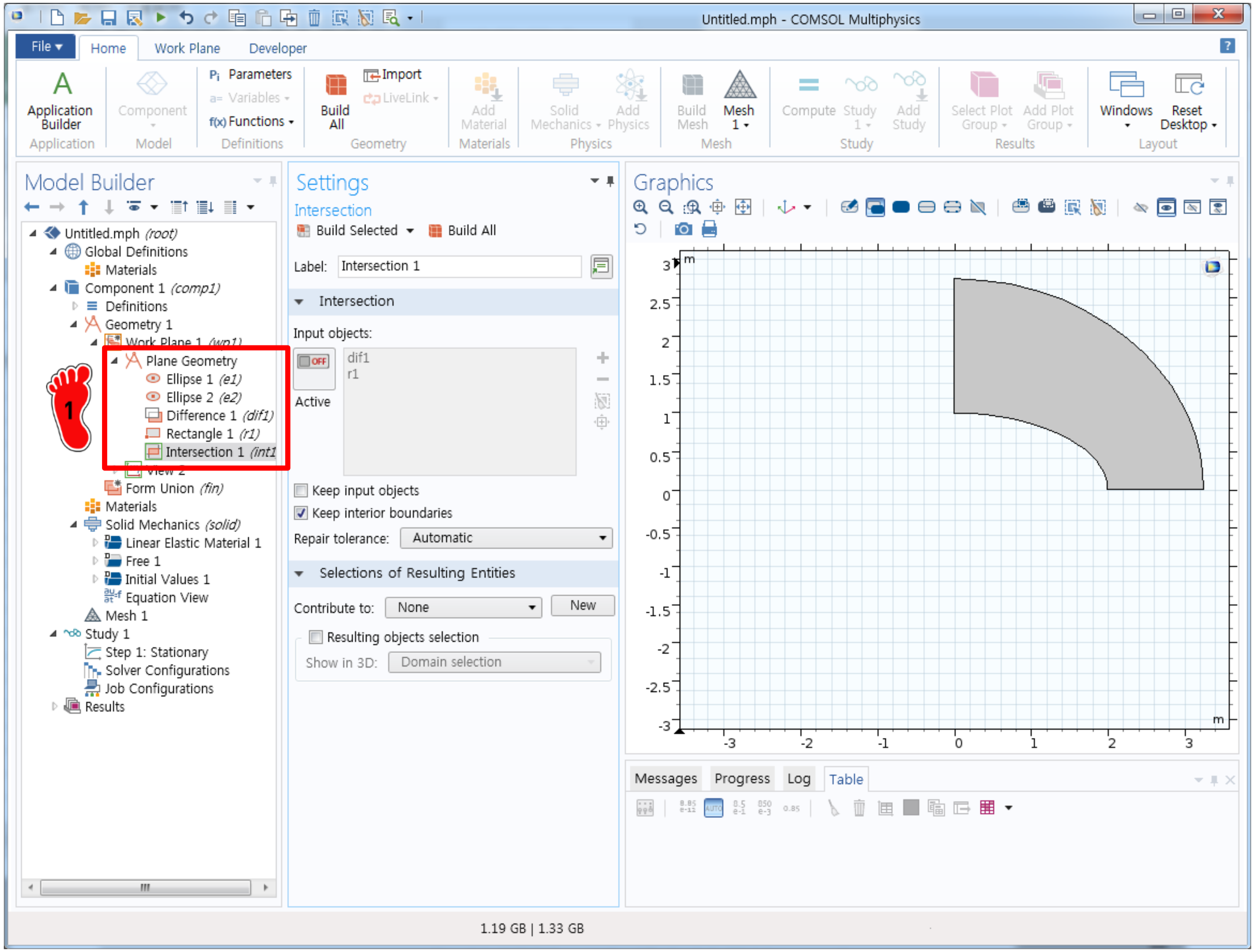
Done 클릭

GEOMETRY CREATION



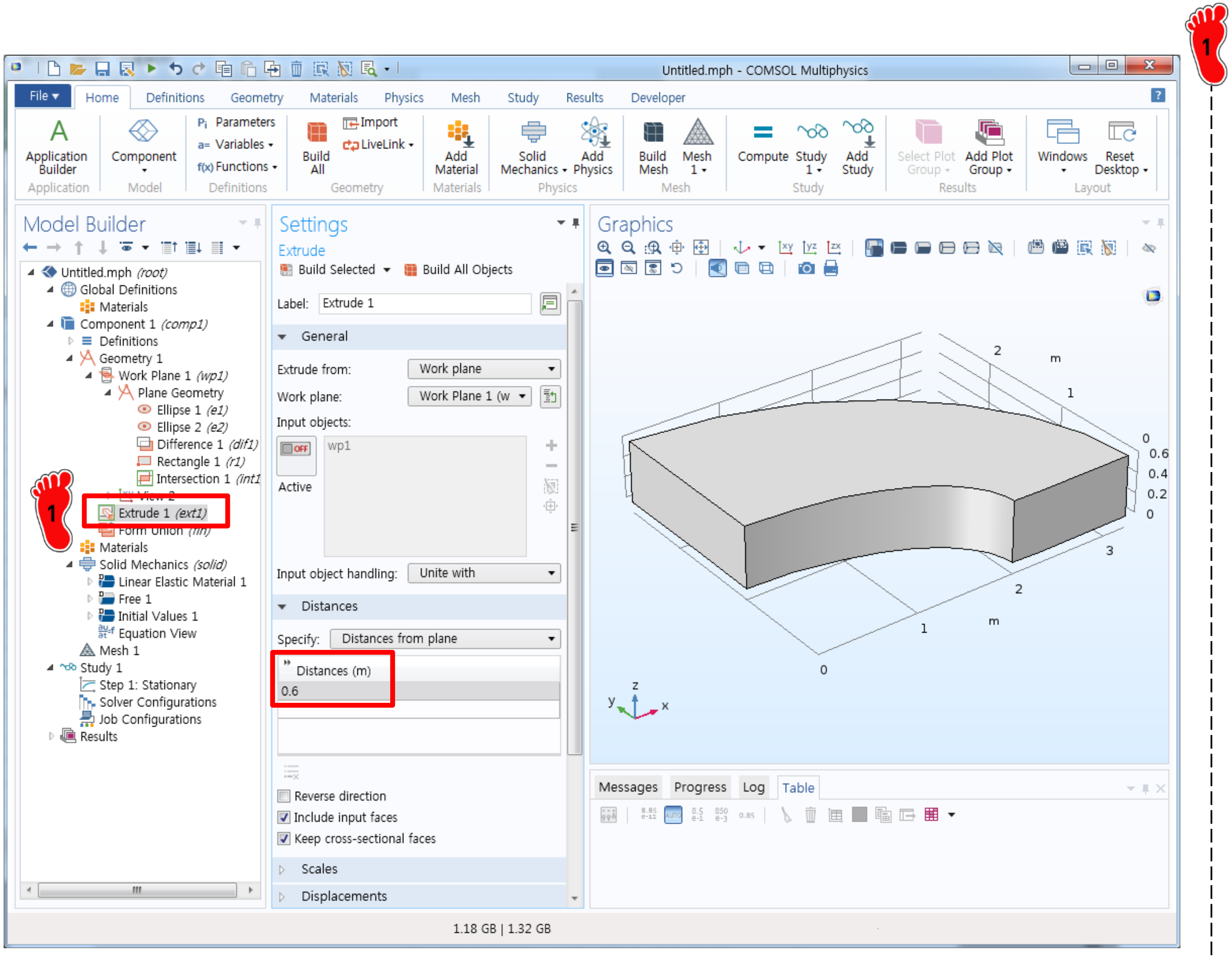
1 Geometry 1 마우스 우클릭
→ Work Plane 클릭

GEOMETRY CREATION



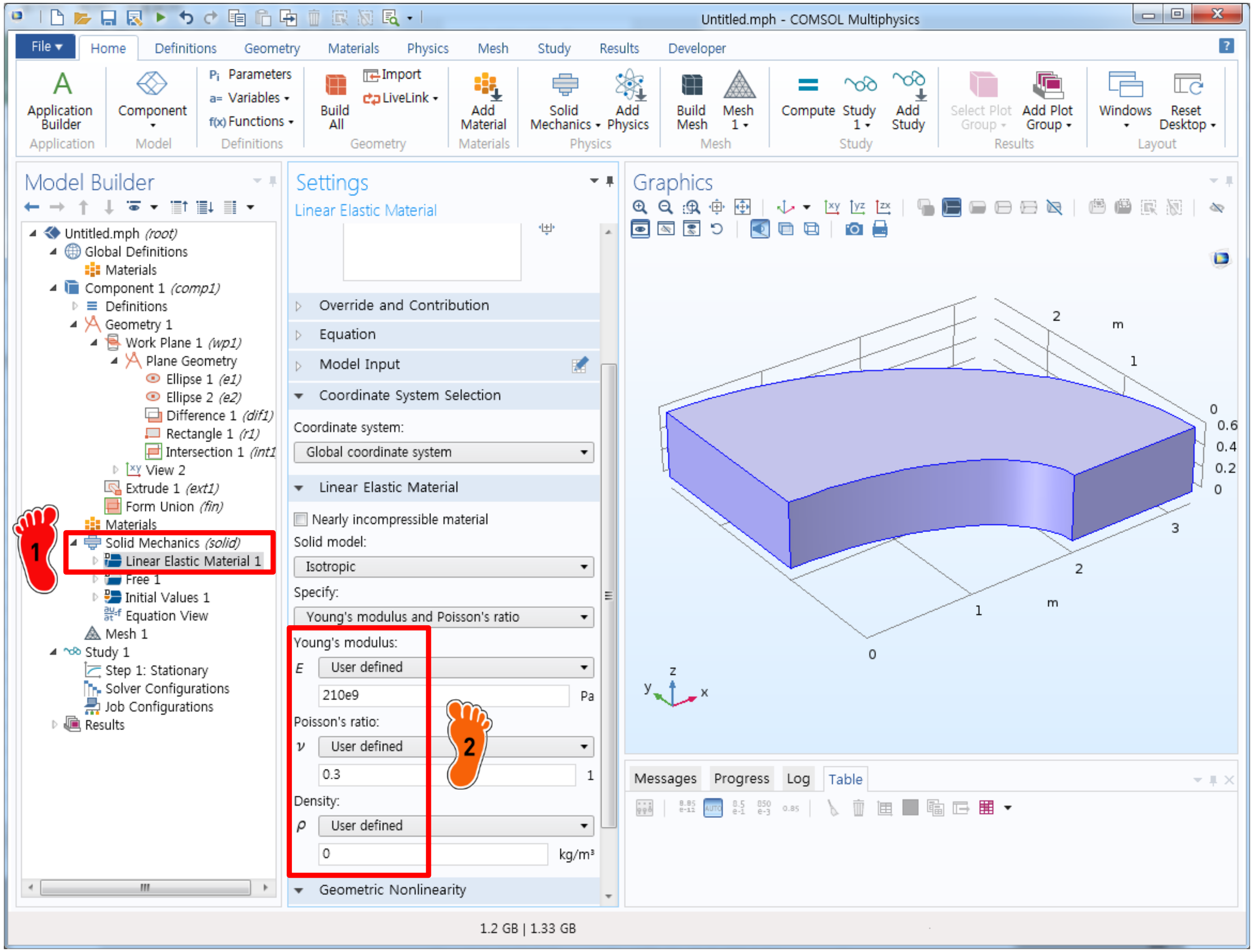
1 Plane Geometry 에 2차원 형상 생성

GEOMETRY CREATION



1 Extrude 메뉴를 이용하여 Z 방향으로 0.6 m 두께를 가지는 형상 생성

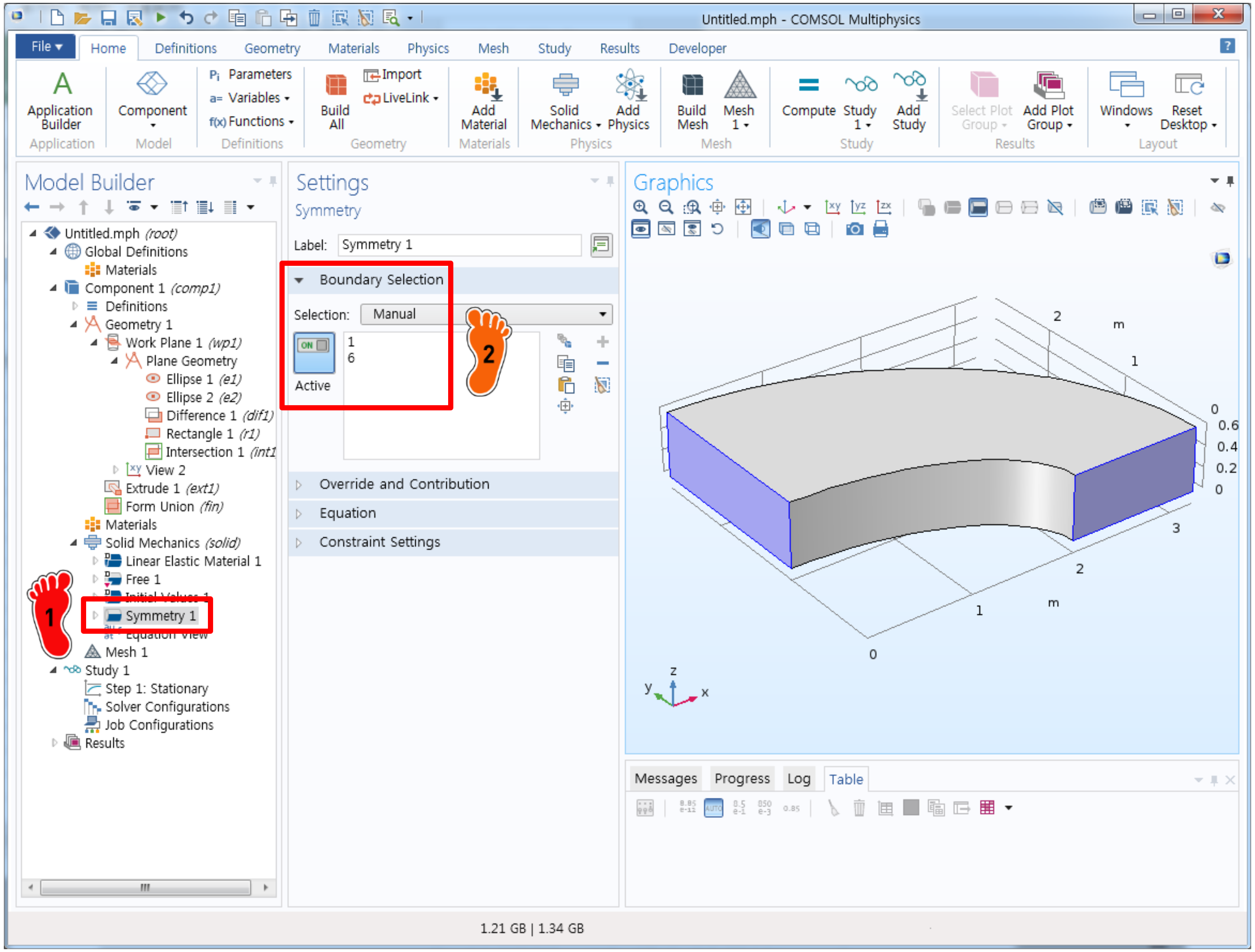
MATERIAL PROPERTY



1 Linear Elastic Material 클릭

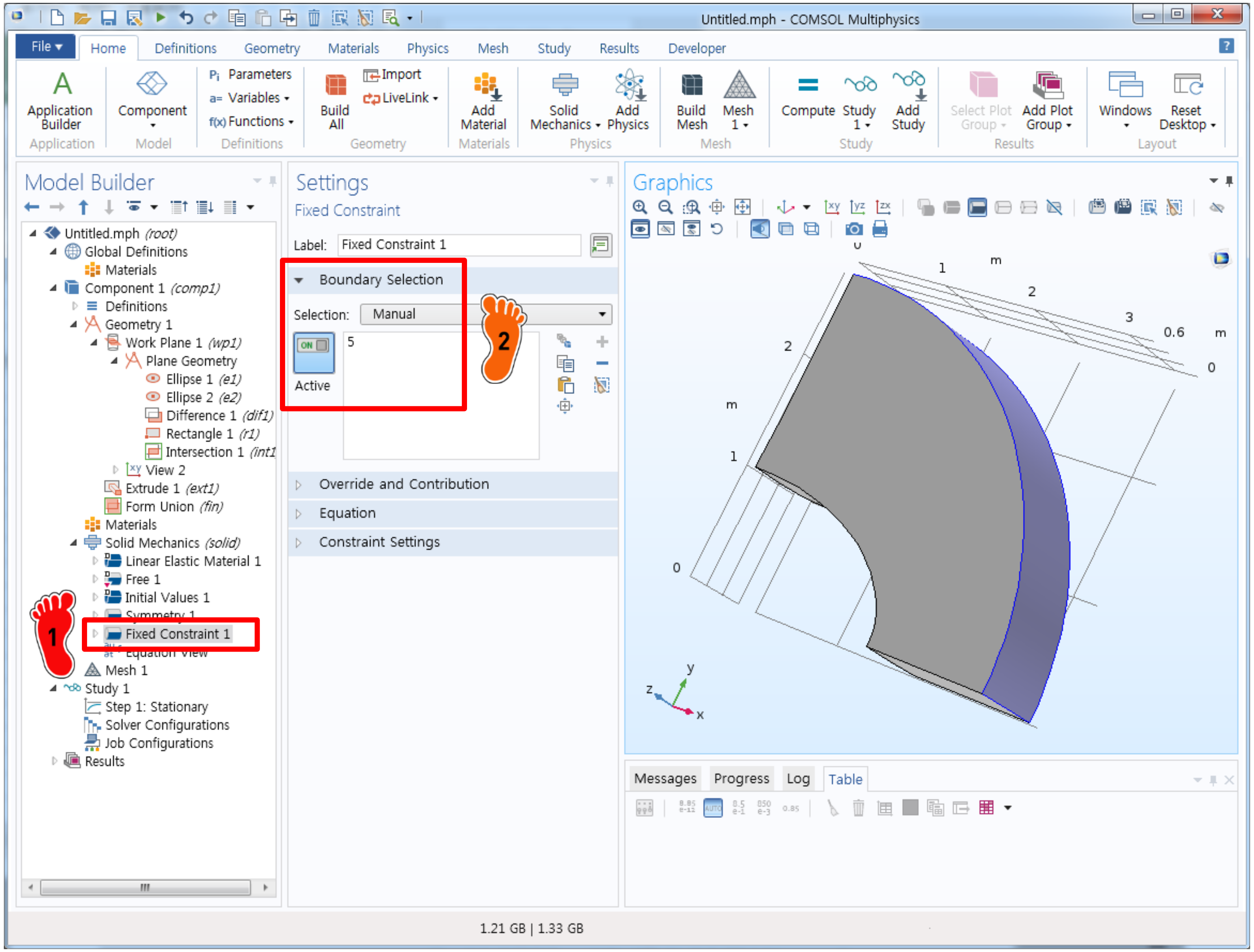
2 E: 210e9
mu: 0.3
rho : 0 입력

BOUNDARY CONDITION



1. Symmetry 경계조건 생성
2. 1, 6번 경계 선택

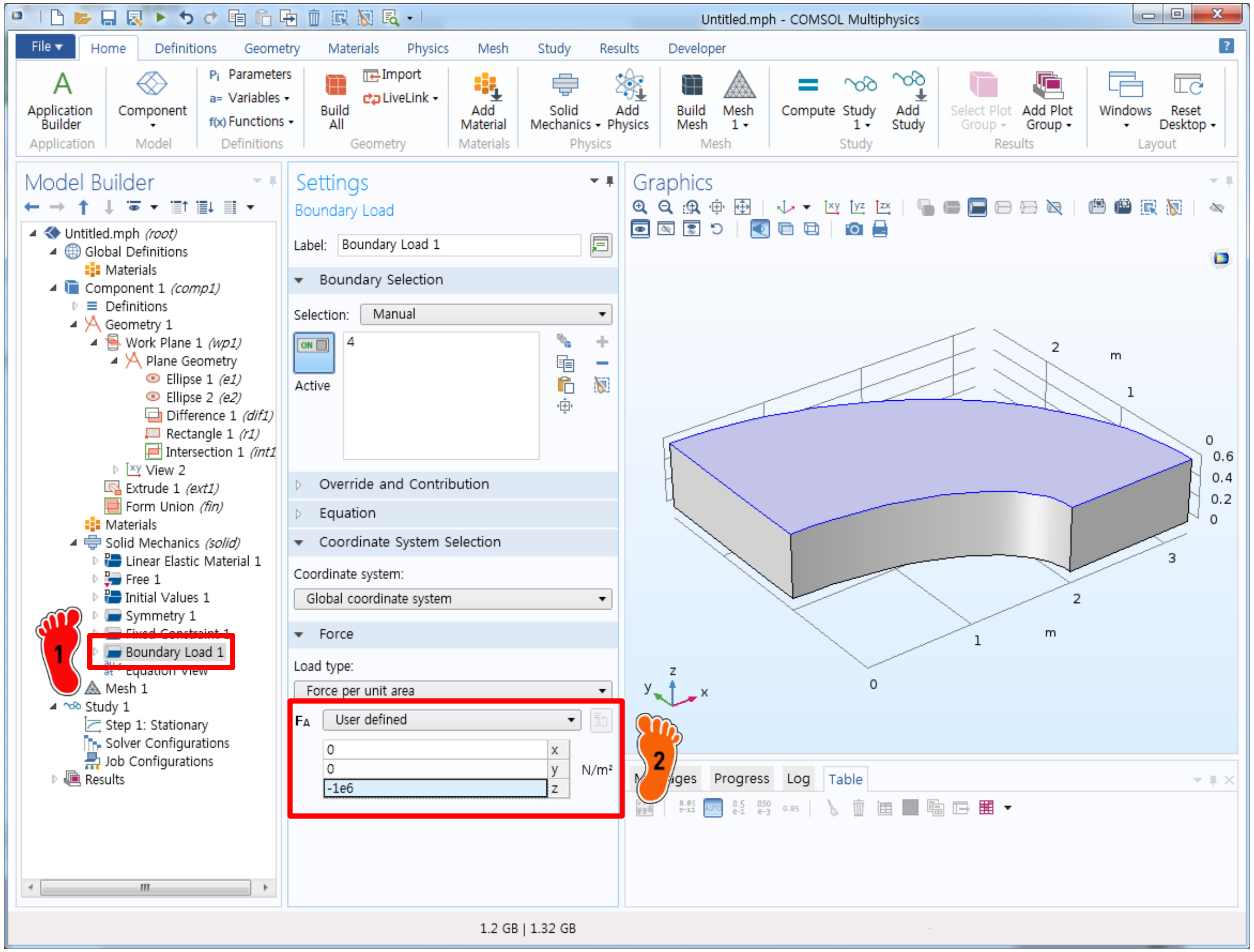
BOUNDARY CONDITION



1 Fixed 경계조건 생성
(왼쪽버튼 누른 채로 회전)

2 5번 경계 선택

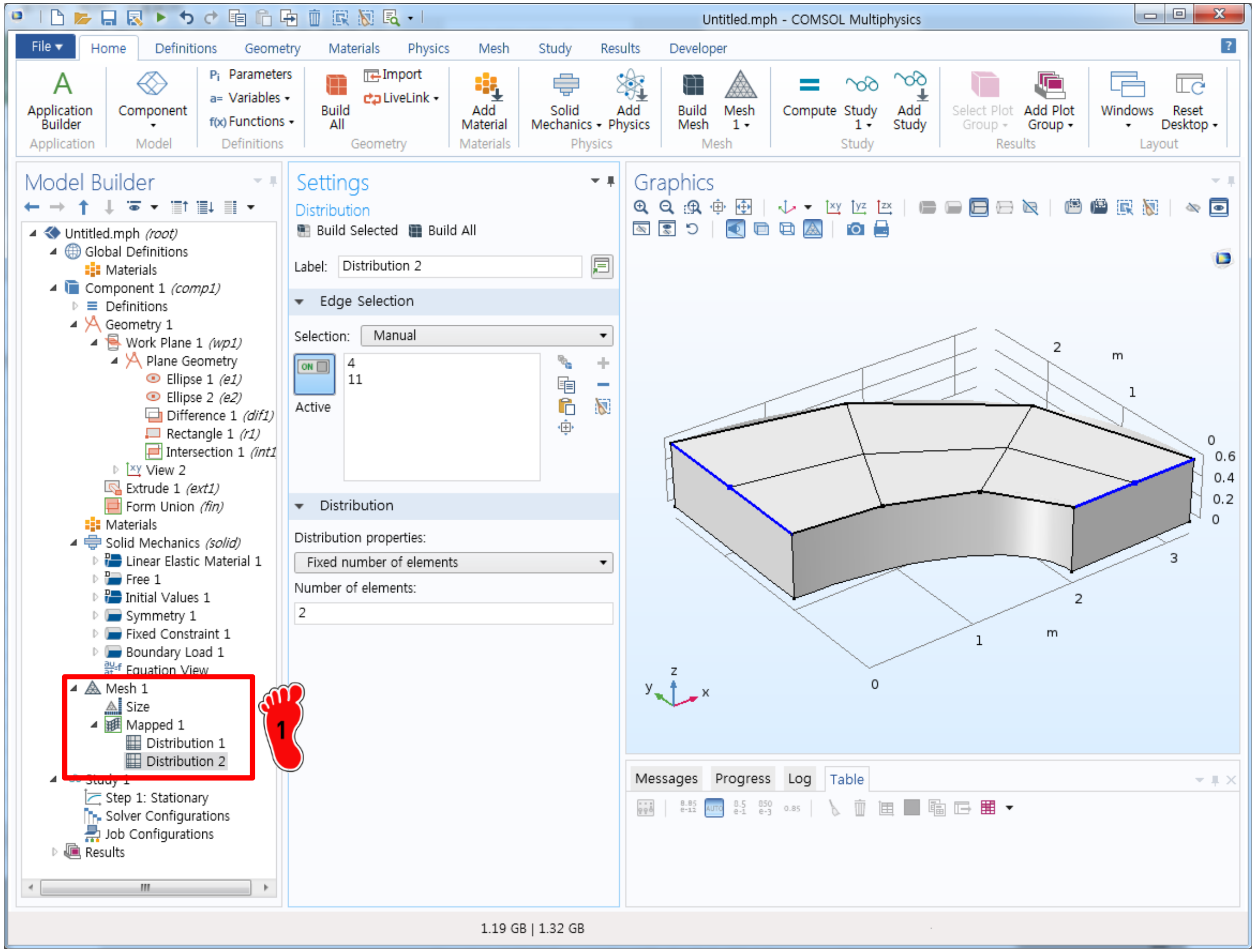
LOADING CONDITION



1 Boundary Load 생성

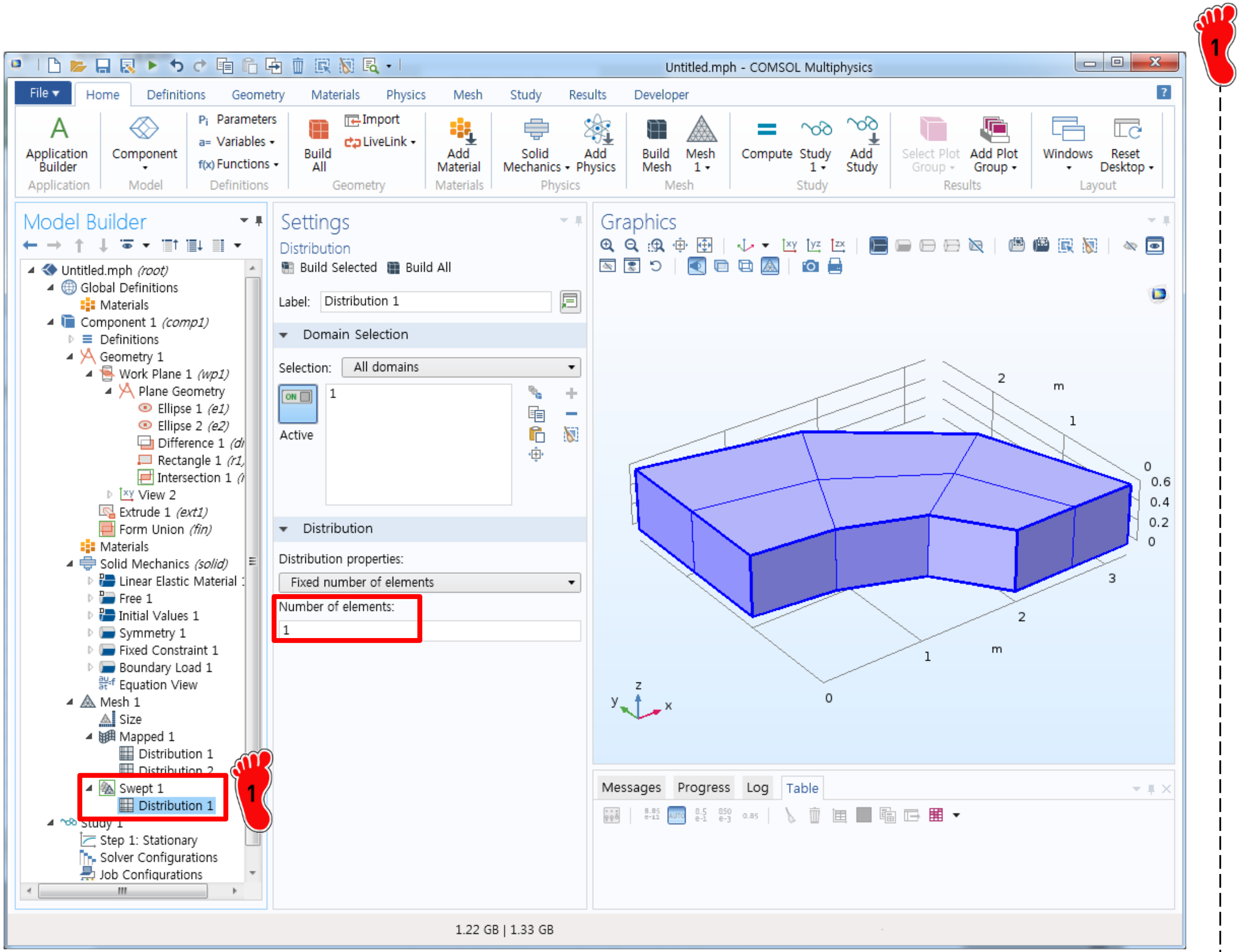
2 z 방향 -1e6 입력

MESH



1 Mapped mesh의 Distribution 기능을 이용 4번 평면에 요소 생성

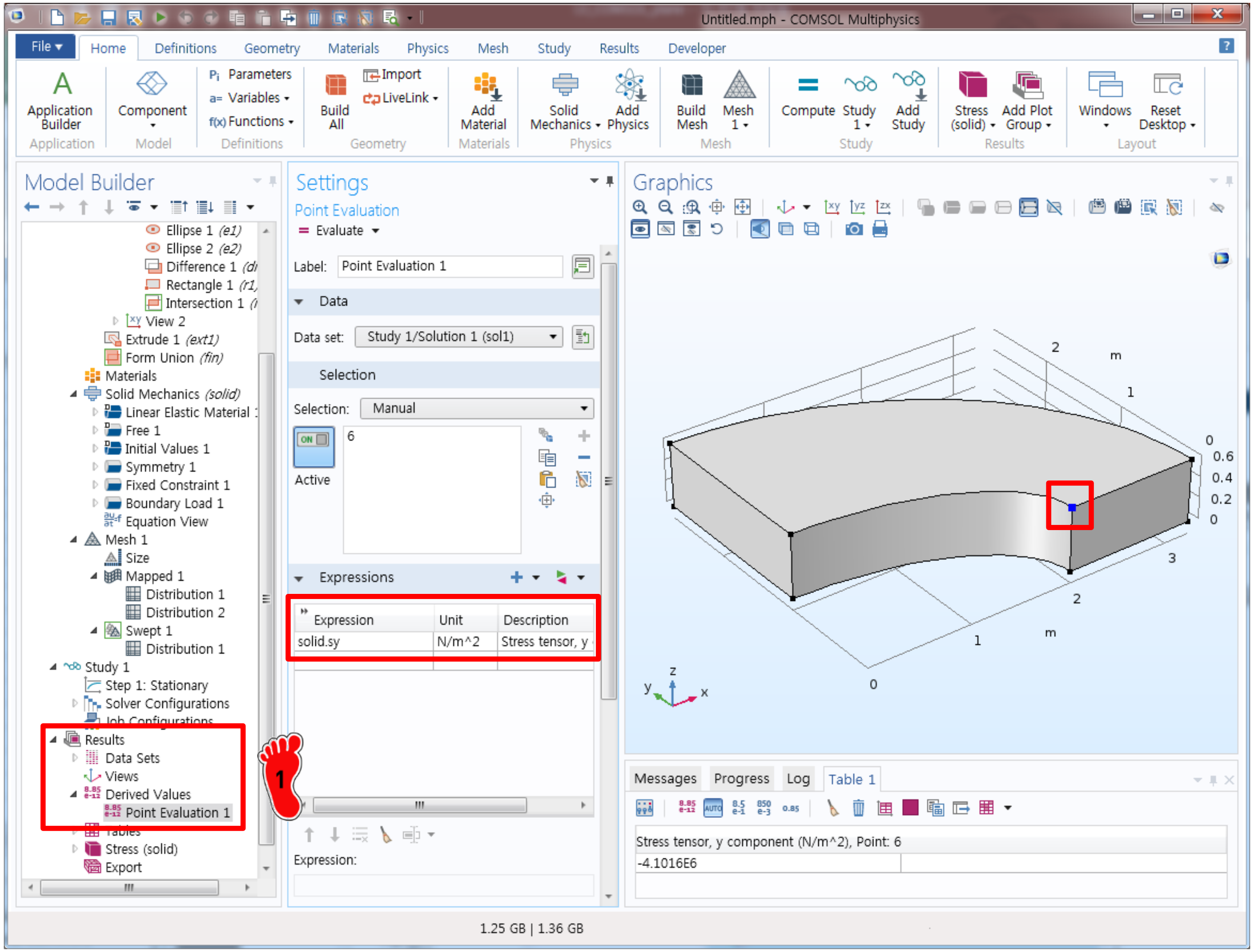
MESH



Swept 메뉴를 이용하여 전체 영역에 한 개의 layer를 가지는 요소 생성

Compute를 클릭하여 해석 수행

POST-PROCESSING

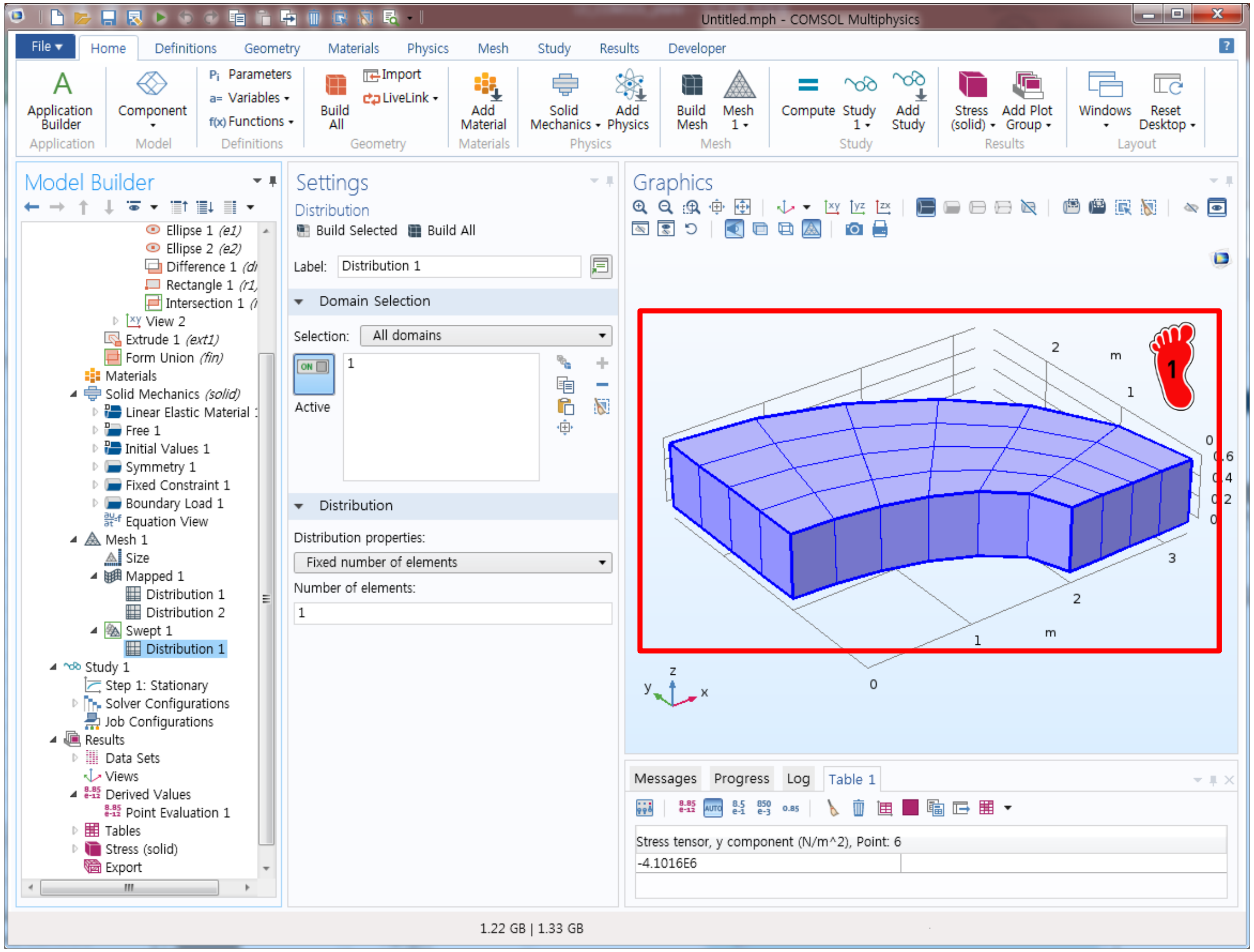


1 Derived Values 의 Point Evaluation 기능을 이용하여 6번 절점의 solid.sy 응력을 계산

결과값: -4.10 MPa

※ Ref : -5.38 MPa

MESH



Fine mesh로 변경

Compute를 클릭하여 해석 수행

결과값: -5.13 MPa

※ Ref : -5.38 MPa

SUMMARY

LE 10	order	integration points	element	coarse mesh s22	coarse mesh error	fine mesh s22	fine mesh error
Abaqus *	2	8	*C3D20R*	*-7.93E+06*	*47.40%*	*-5.53E+06*	*2.79%*
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
Abaqus *	2	*27*	*C3D20*	*-6.72E+06*	*24.91%*	*-5.64E+06*	*4.83%*
Ansys	2	*14*	SOLID95	-5.40E+06	0.46%	-5.61E+06	4.33%
CalculiX	2	27	C3D20	-5.20E+06	-3.32%	-5.50E+06	2.18%
Tahoe	2	27	hexahedron				

COMSOL	2		plate	-4.61E+06	14.3 %	-4.98E+06	7.4 %
COMSOL	2		hexahedron	-4.10E+06	23.8 %	-5.13E+06	4.6 %

HOOD PANEL ANALYSIS

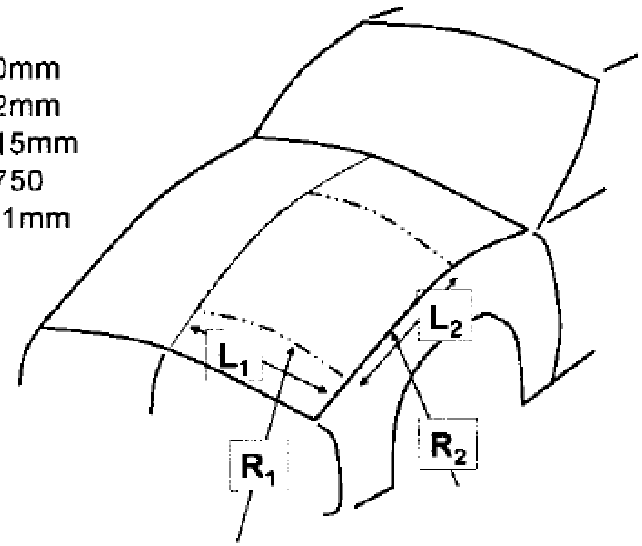
– Panel stiffness: load(1N)/deflection

$$K = \frac{CEt^2}{R\sqrt{1-\nu^2}} \quad \text{where curvature} \quad \frac{1}{R} = \frac{\left(\frac{L_1^{\,2}}{R_1}\right) + \left(\frac{L_2^{\,2}}{R_2}\right)}{2L_1L_2}$$

$$\begin{cases} C: \ \text{constant } (2.309) \\ t: \ \text{panel thickness} \\ R: \ \text{spherical radius} \\ L_1, L_2: \ \text{rectangluar panel dimensions} \\ R_1, R_2: \ \text{panel radii of curvature in orthogonal directions} \end{cases}$$

Example: Hood

L_1=	660mm
L_2=	762mm
R_1=	5715mm
R_2=	15750
t=	0.71mm



– Oil-can load: buckling load

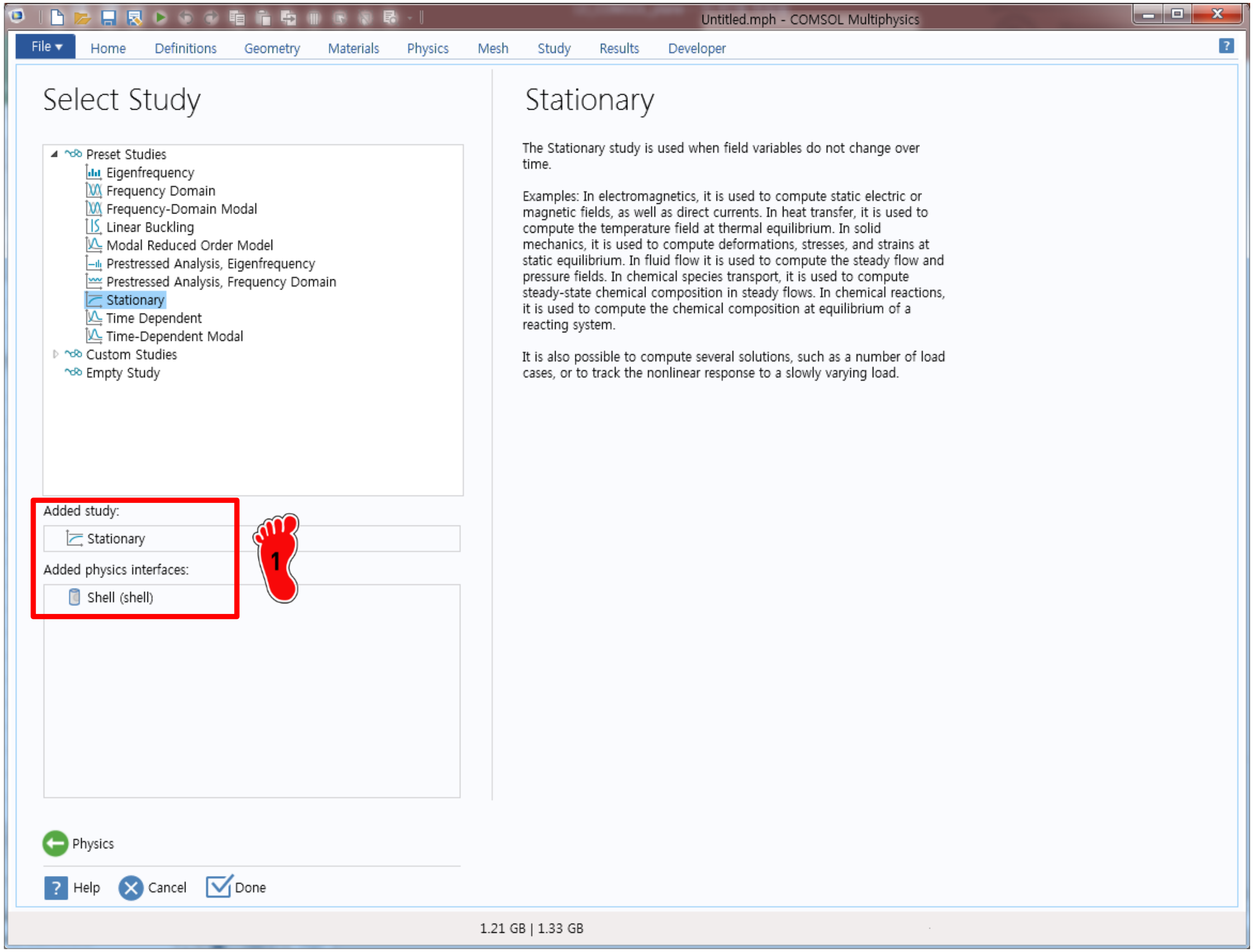
$$P_{cr} = \frac{CR_{cr}\pi^2Et^4}{L_1L_2\left(1-\nu^2\right)} \quad \text{where} \quad \begin{cases} R_{cr} = 45.929 - 34.183\lambda + 6.397\lambda^2 \\ \lambda = 0.5\sqrt{\dfrac{L_1L_2}{t}\sqrt{\dfrac{12\left(1-\nu^2\right)}{R_1R_2}}} \\ C = 0.645 - 0.775L_1L_2 \end{cases}$$

$$K = 28.81 \text{ N/mm}$$

$$P_{cr} = 50.53 \text{ N}$$

valid over the range $\frac{R_1}{L_1}$ and $\frac{R_2}{L_2} > 2, \ \frac{1}{3} < \frac{L_2}{L_1} < 3, \ L_1L_2 < 0.774m^2$

ENVIRONMENT SETTING

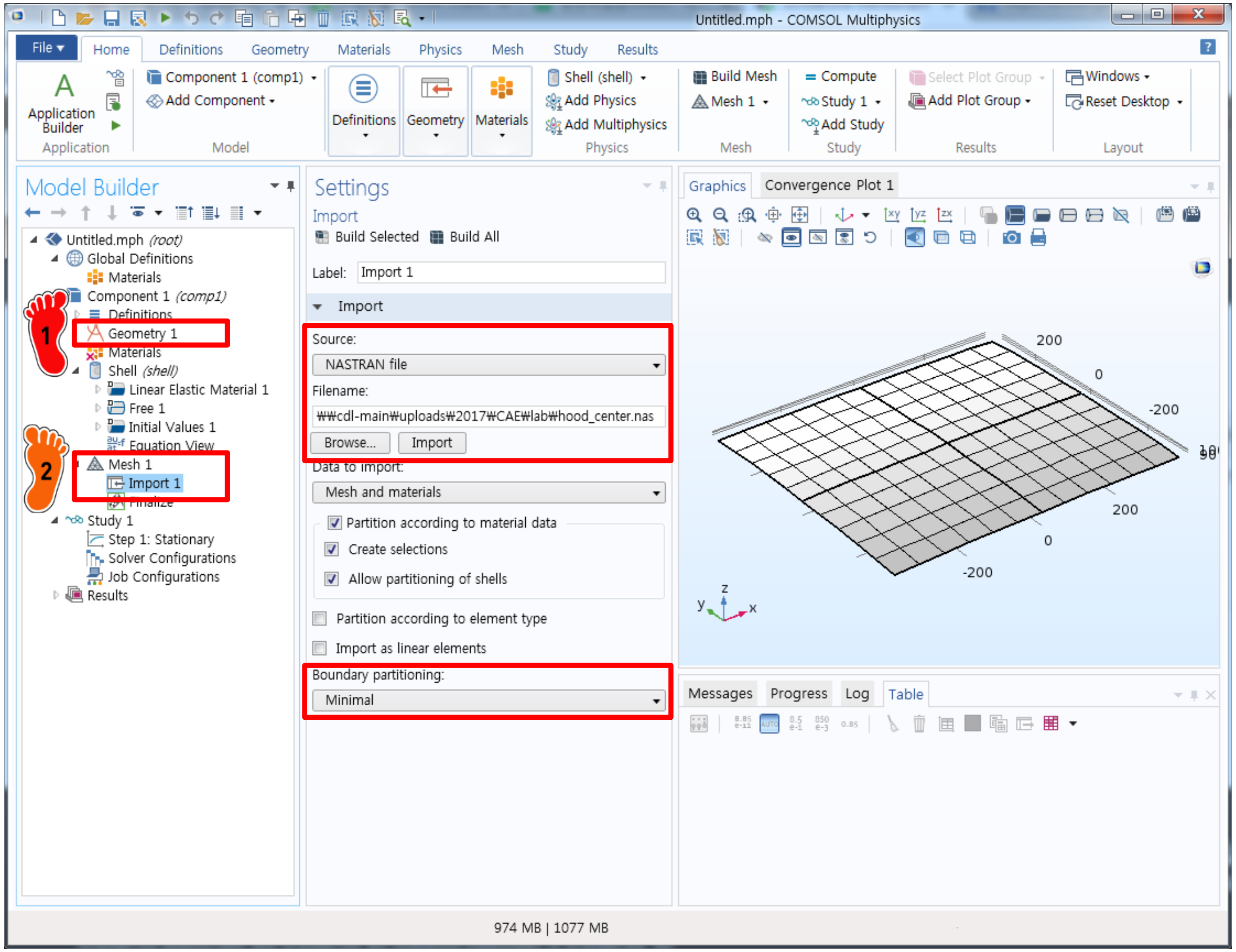


1

Dimension : 3D
Physics :
Structural Mechanics
→ Shell
Study : Stationary

Done 클릭

GEOMETRY IMPORT



1 Geometry에서 단위를 mm로 설정

2 Mesh 우클릭 → Import

Settings 화면에서

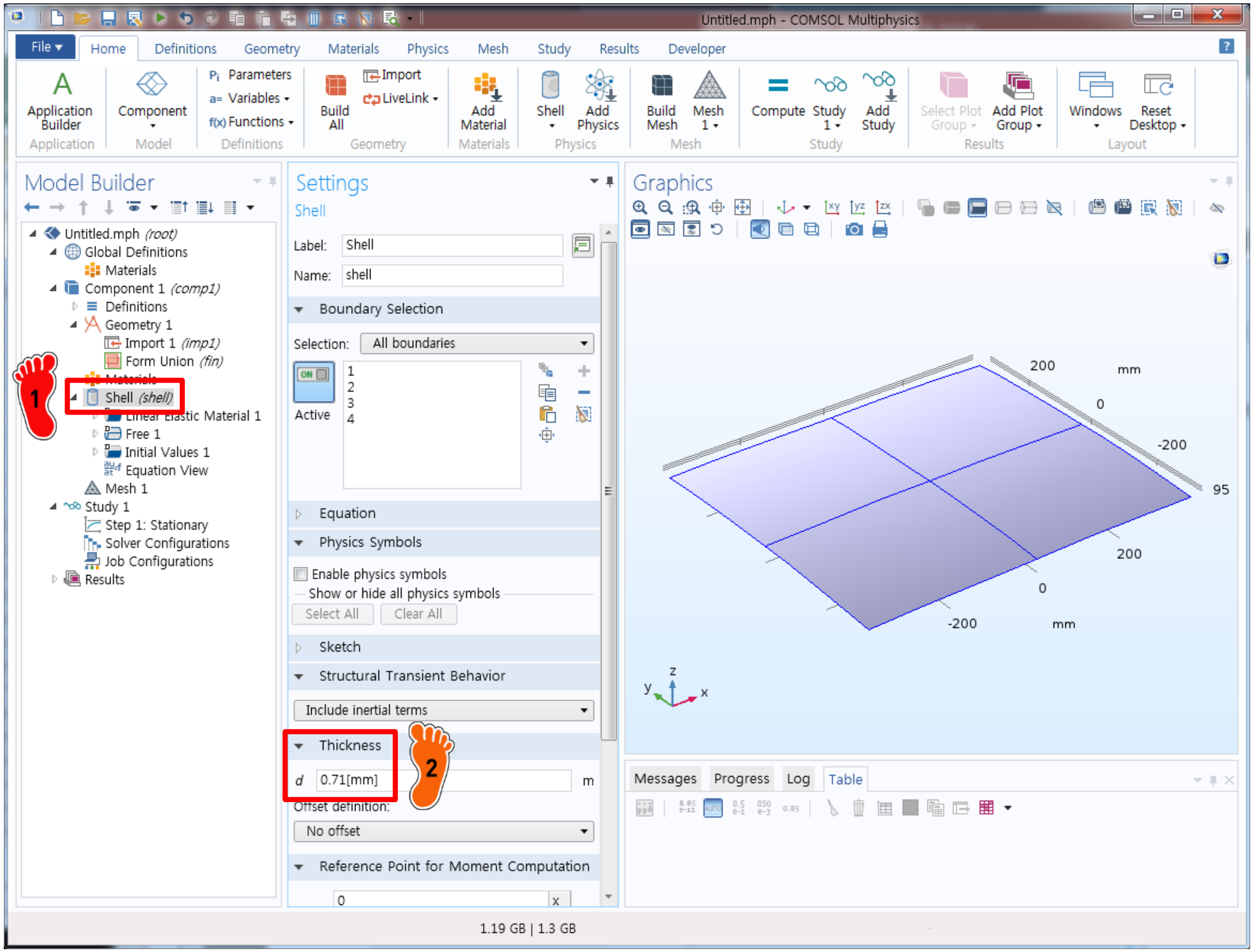
1) Source : NASTRAN file

2) Browse 버튼 클릭해서 hood_center.nas 파일 선택

3) Boundary partitioning : Minimal

4) Import 클릭

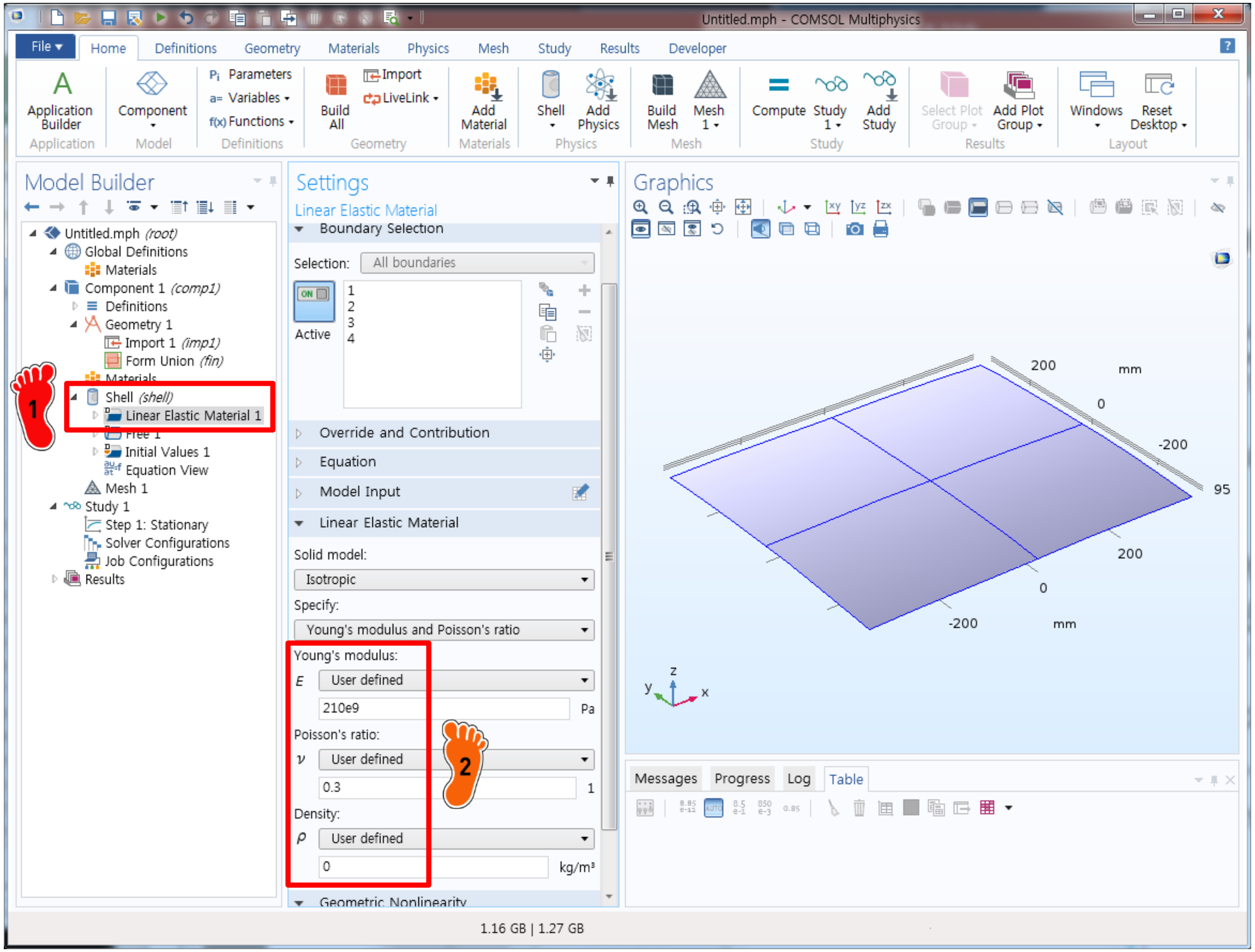
MATERIAL PROPERTY



1 Shell 클릭

2 Thickness d: 0.71[mm] 입력

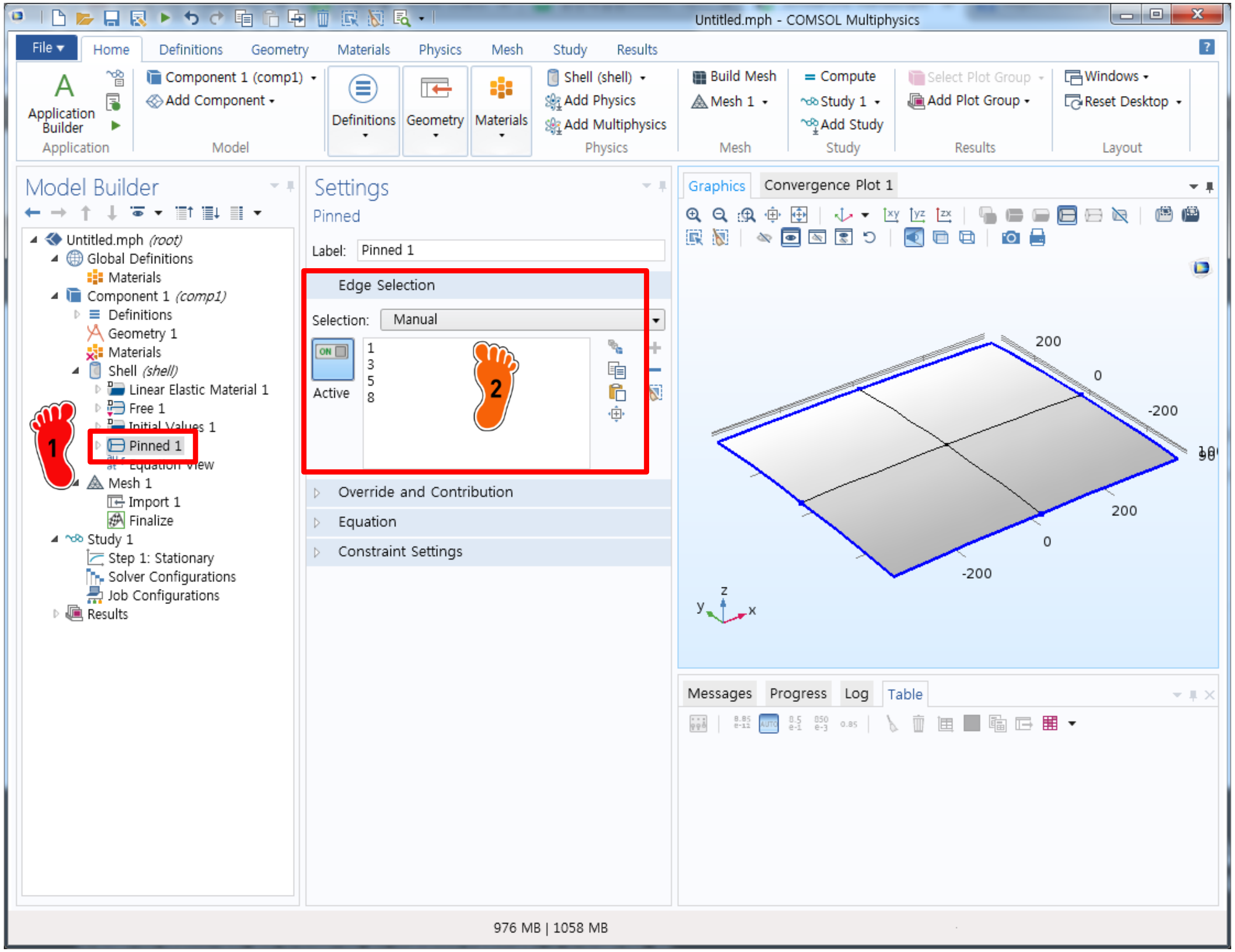
MATERIAL PROPERTY



1 Linear Elastic Material 클릭

2 E: 210e9
mu: 0.3
rho : 0 입력

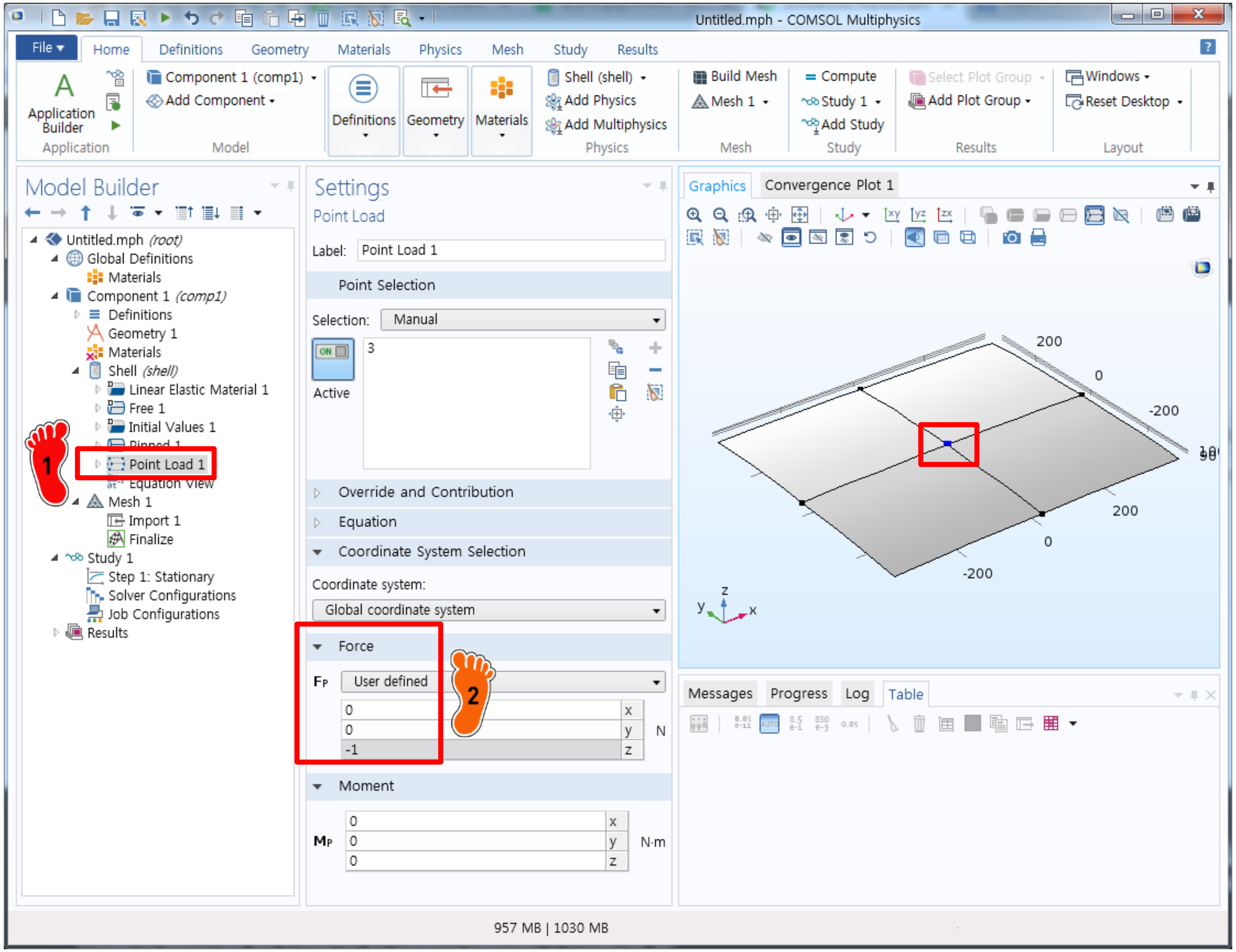
BOUNDARY CONDITION



1 Pinned경계조건 생성

2 최외각 경계 선택

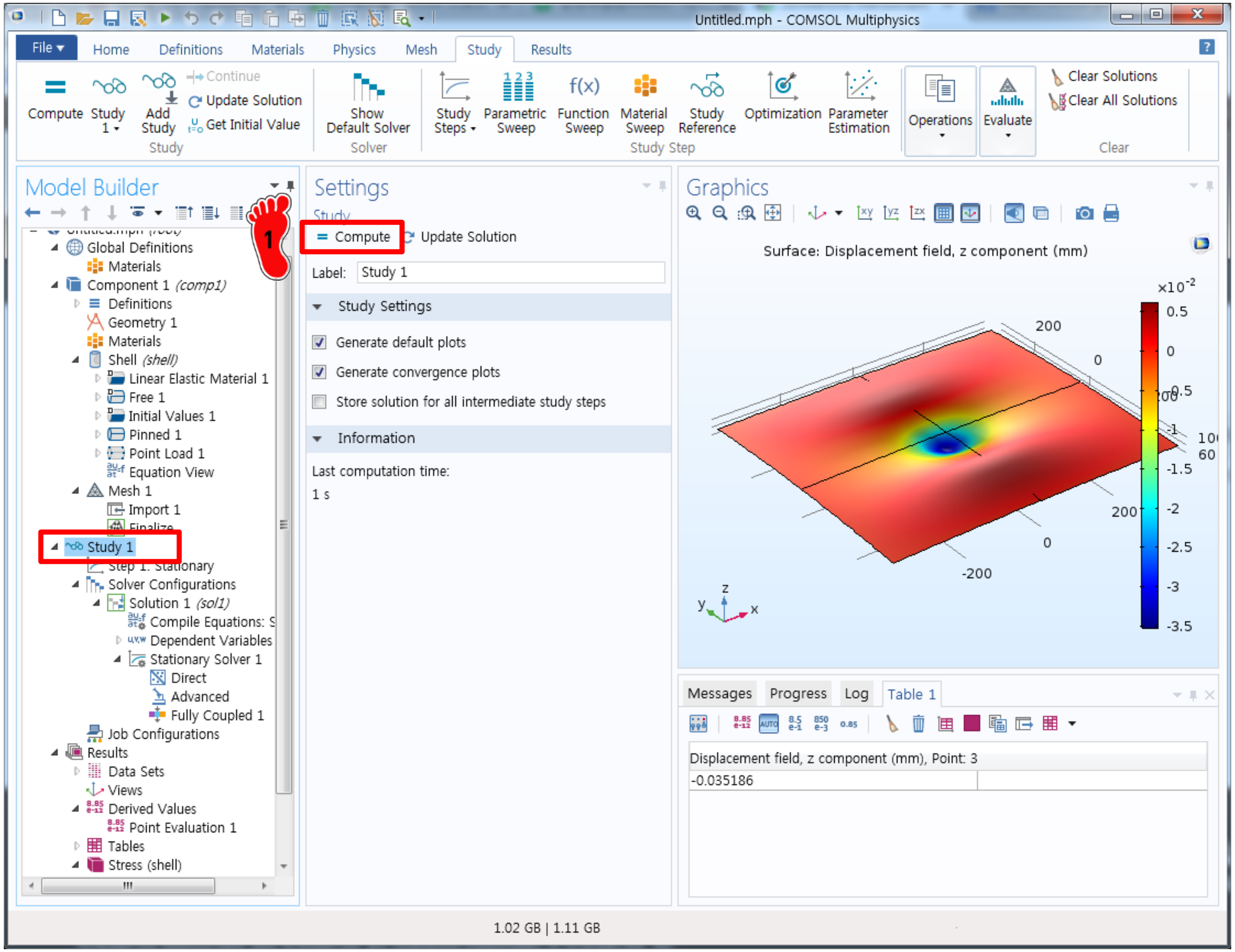
LOADING CONDITION



1 Point Load 메뉴 생성

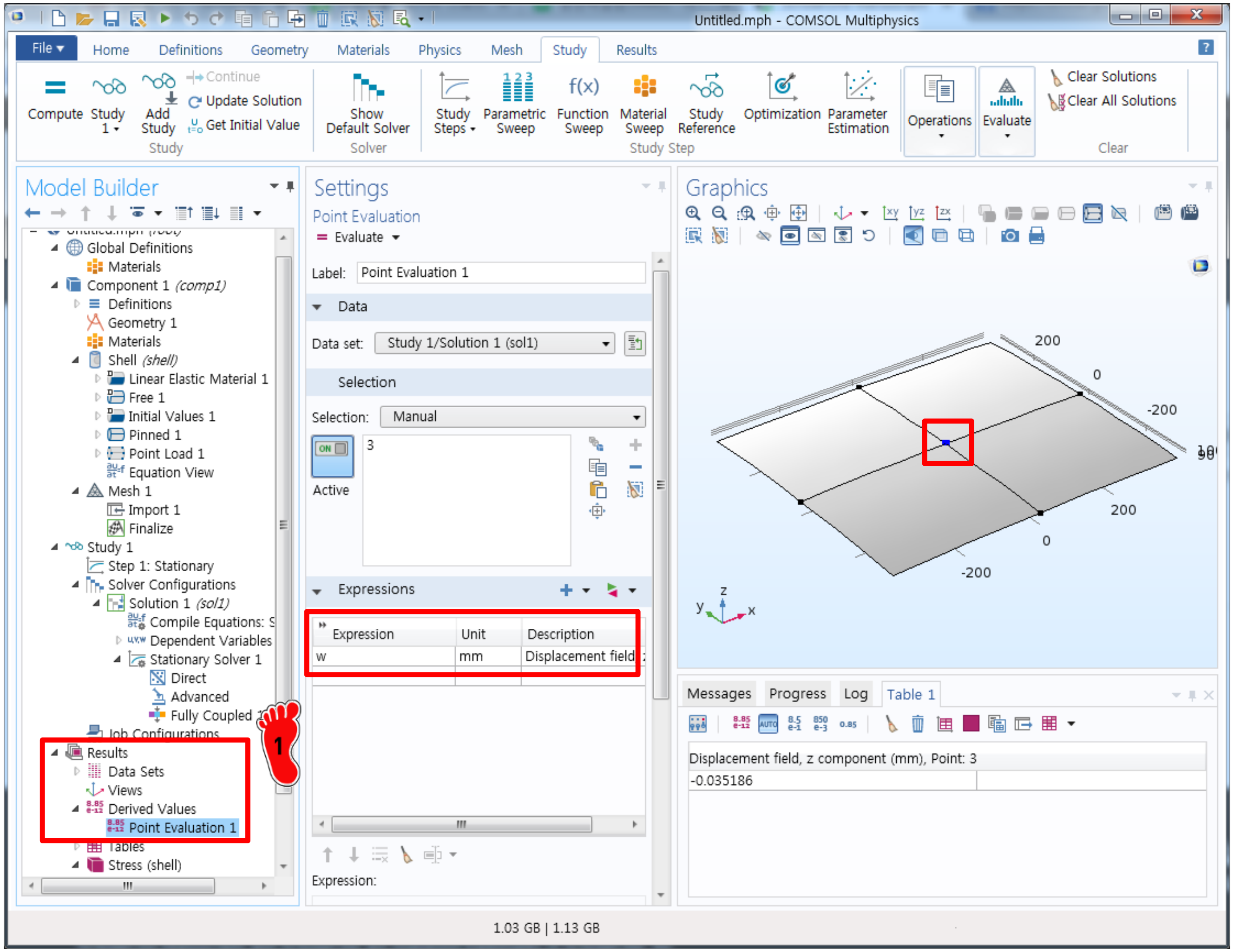
2 가운데 점에 z 방향 -1 N 하중 입력

MESH



Study → Compute 클릭
해석 수행

POST-PROCESSING



Derived Values 의 Point Evaluation 기능을 이용하여 가운데 점의 z 방향 변위를 계산

결과값: -3.5186e-2 mm

Analytic solution

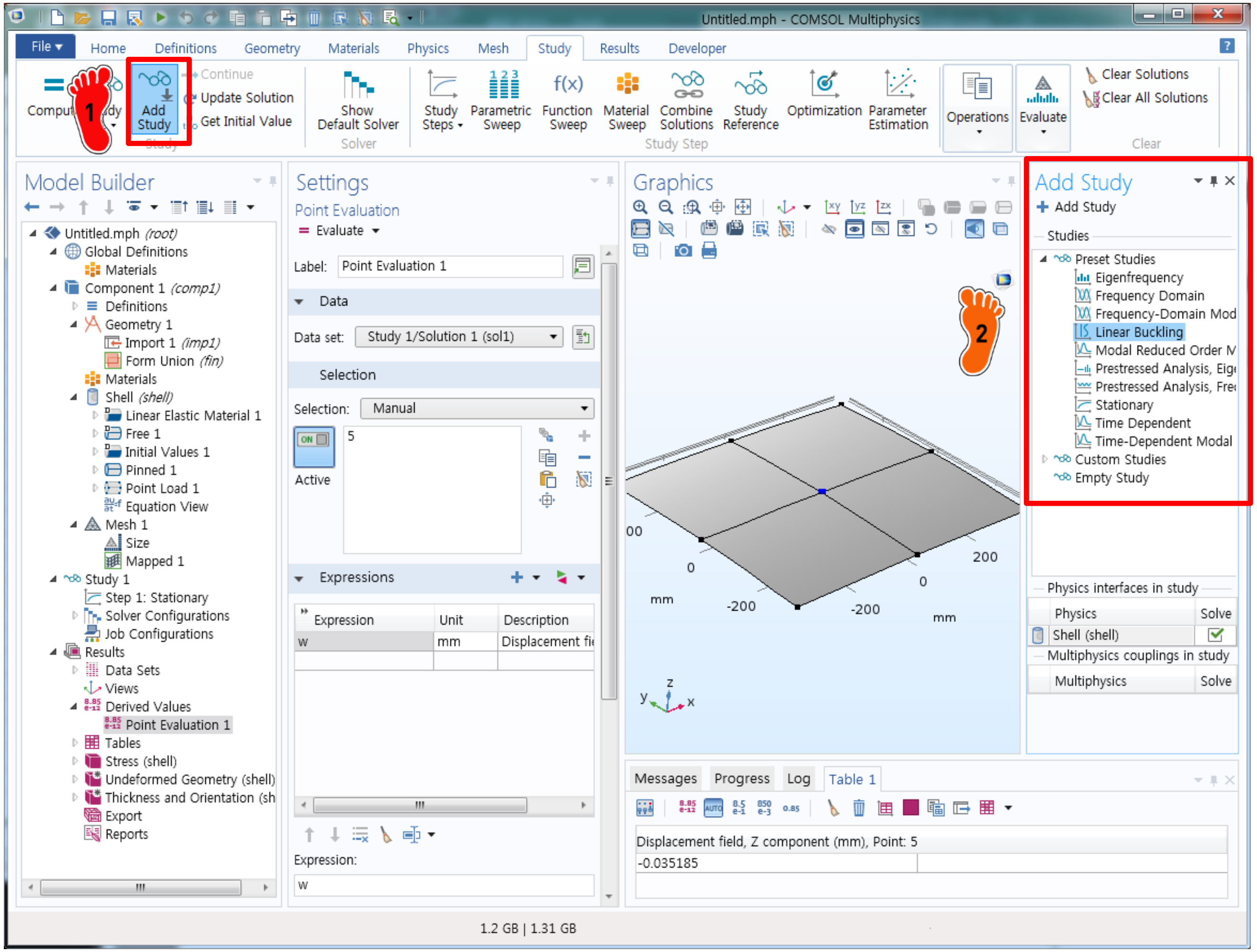
$$K = \frac{B}{R}\frac{Et^2}{\sqrt{1-\mu^2}} = 28.81\ \mathrm{N/mm}$$

COMSOL

$$K = \frac{P}{\delta} = \frac{1\ \mathrm{N}}{3.5186e-2\ \mathrm{mm}}$$

$$= 28.4\ \mathrm{N/mm}$$

LINEAR BUCKLING

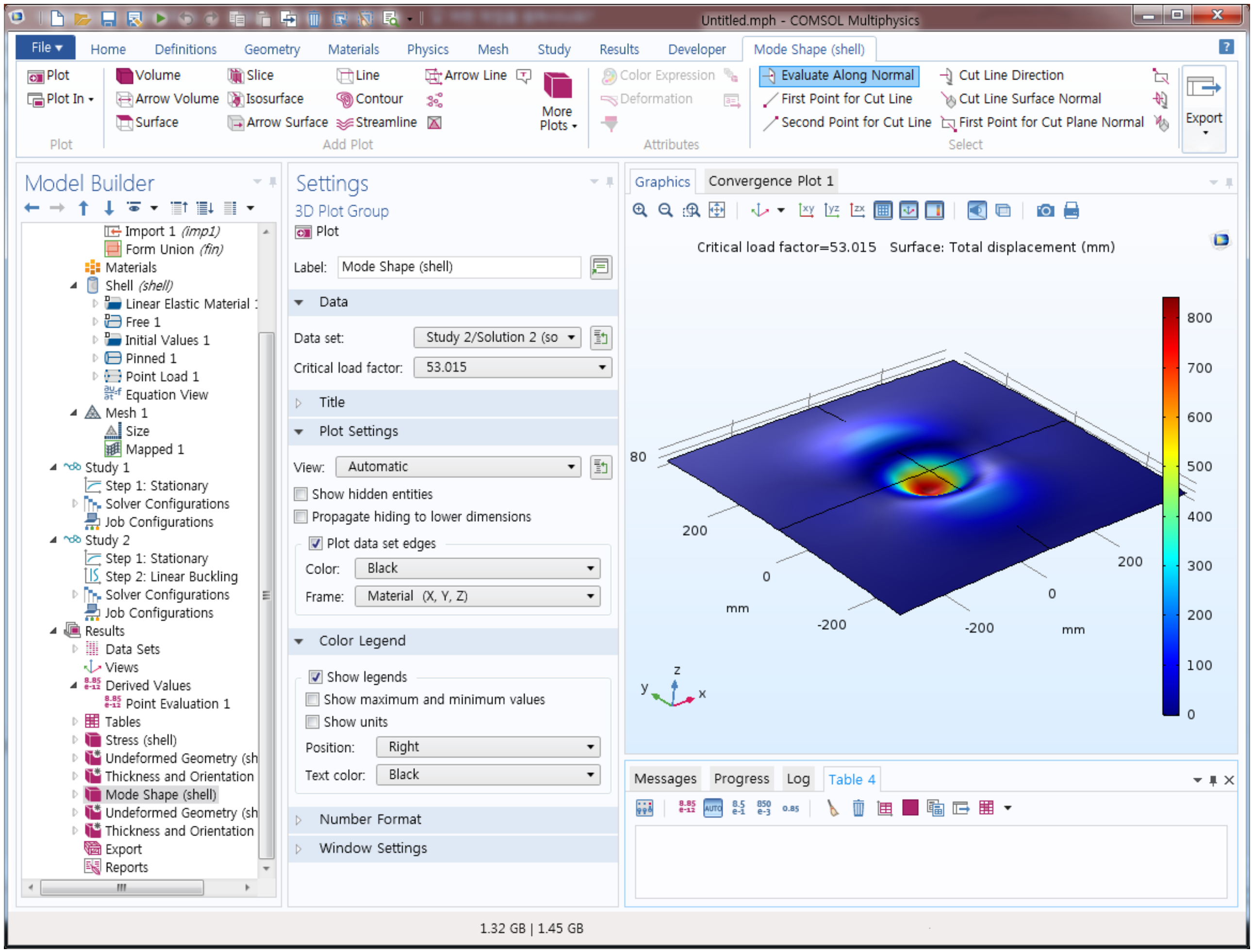


1 Toolbar에서 Study → Add Study 클릭

2 Linear Buckling 추가

Compute를 클릭하여 해석 수행

POST-PROCESSING



Critical load factor : 53.015

결과값: 53.015 N

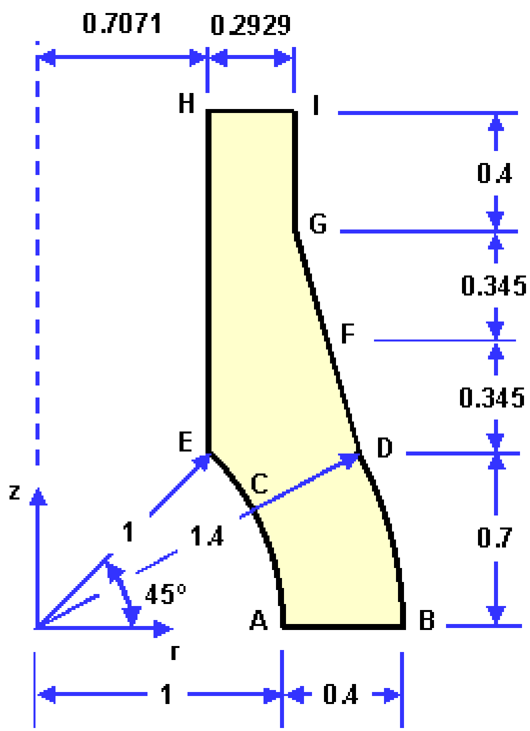
Analytic solution

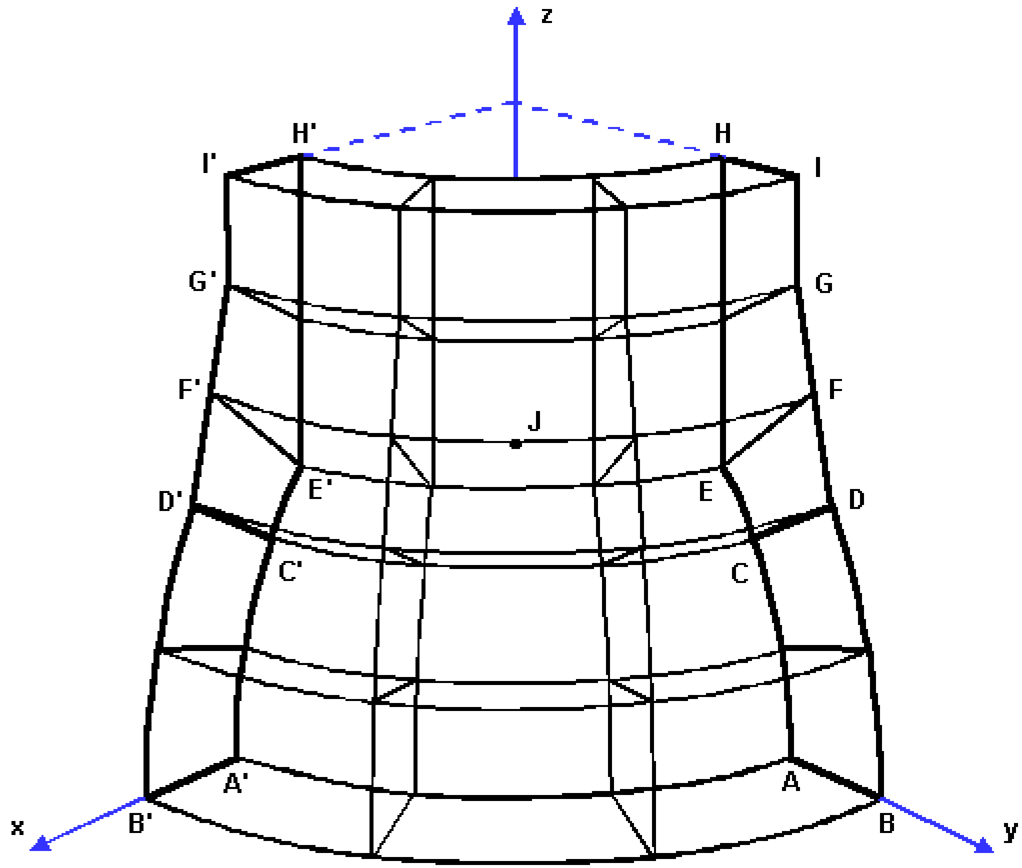
$$P_{cr} = \frac{CR_{cr}\pi^2 E t^4}{L_1 L_2 \left(1-\mu^2\right)} = 50.53 \text{ N}$$

NAFEMS BENCHMARK: LE11

Solid Cylinder/Taper/Sphere – Temperature

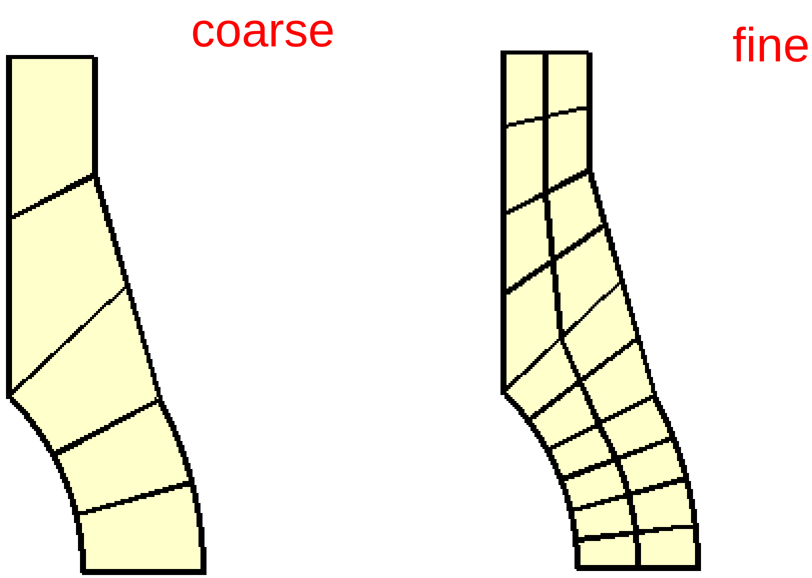
- Analysis Type: Linear elastic solid
- Geometry: meters





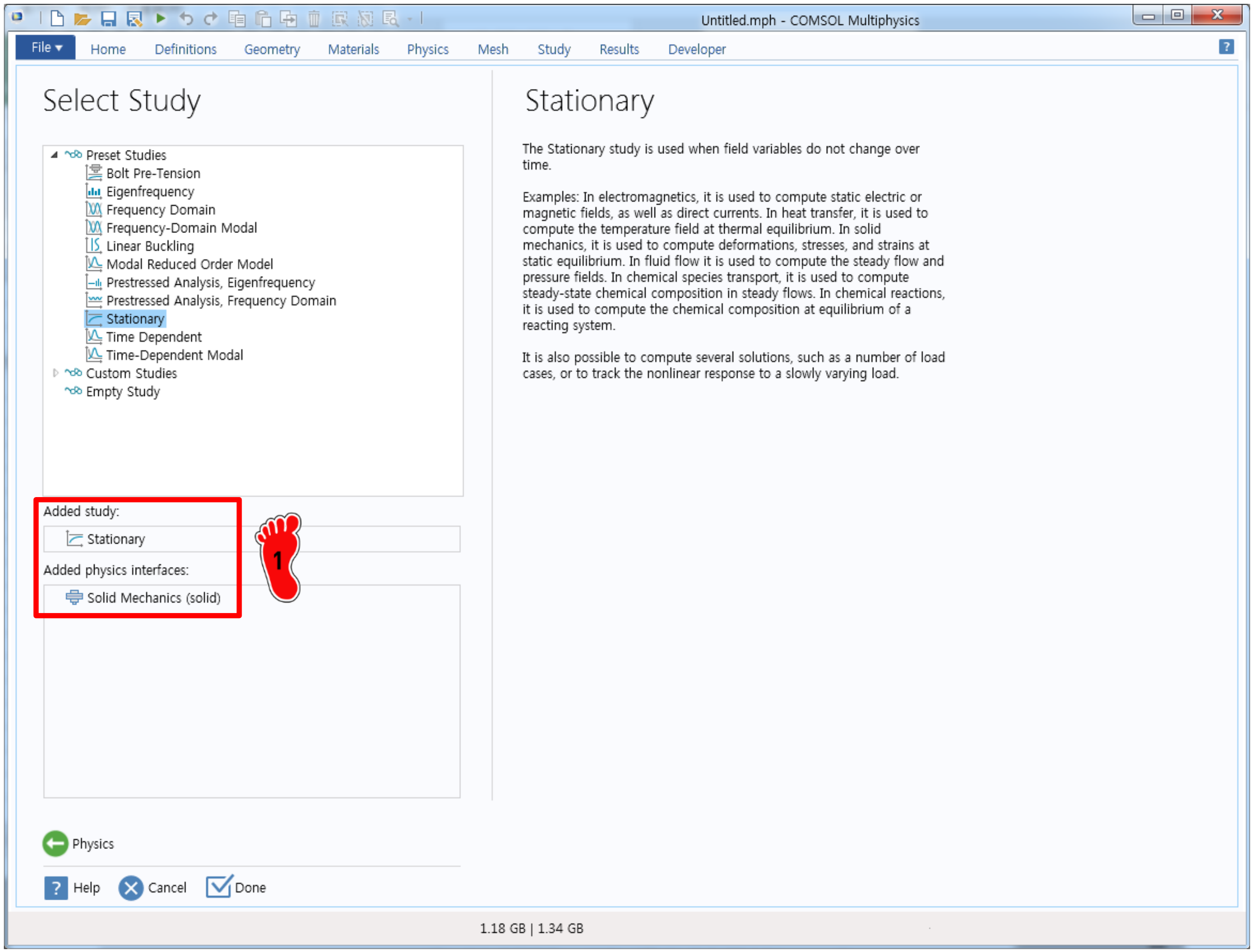
- Loading
  - Linear temperature gradient in the radial and axial direction
  - T (°C) = $(x^2 + y^2)^{1/2} + z$
- Boundary conditions
  - Symmetry on xz-plane
  - Symmetry on yz-plane
  - Face on xy-plane: zero z-displacement
  - Face HIH'I': zero z-displacement
- Material properties
  - Isotropic: E = 210 GPa, ν = 0.3
  - $\alpha = 2.3 * 10^{-4}$ /°C
- Element types
  - Solid hexahedra, wedges and tetrahedra

- Meshes



- Output
  - Direct stress (σ_{zz}) at point A: -105 MPa

ENVIRONMENT SETTING

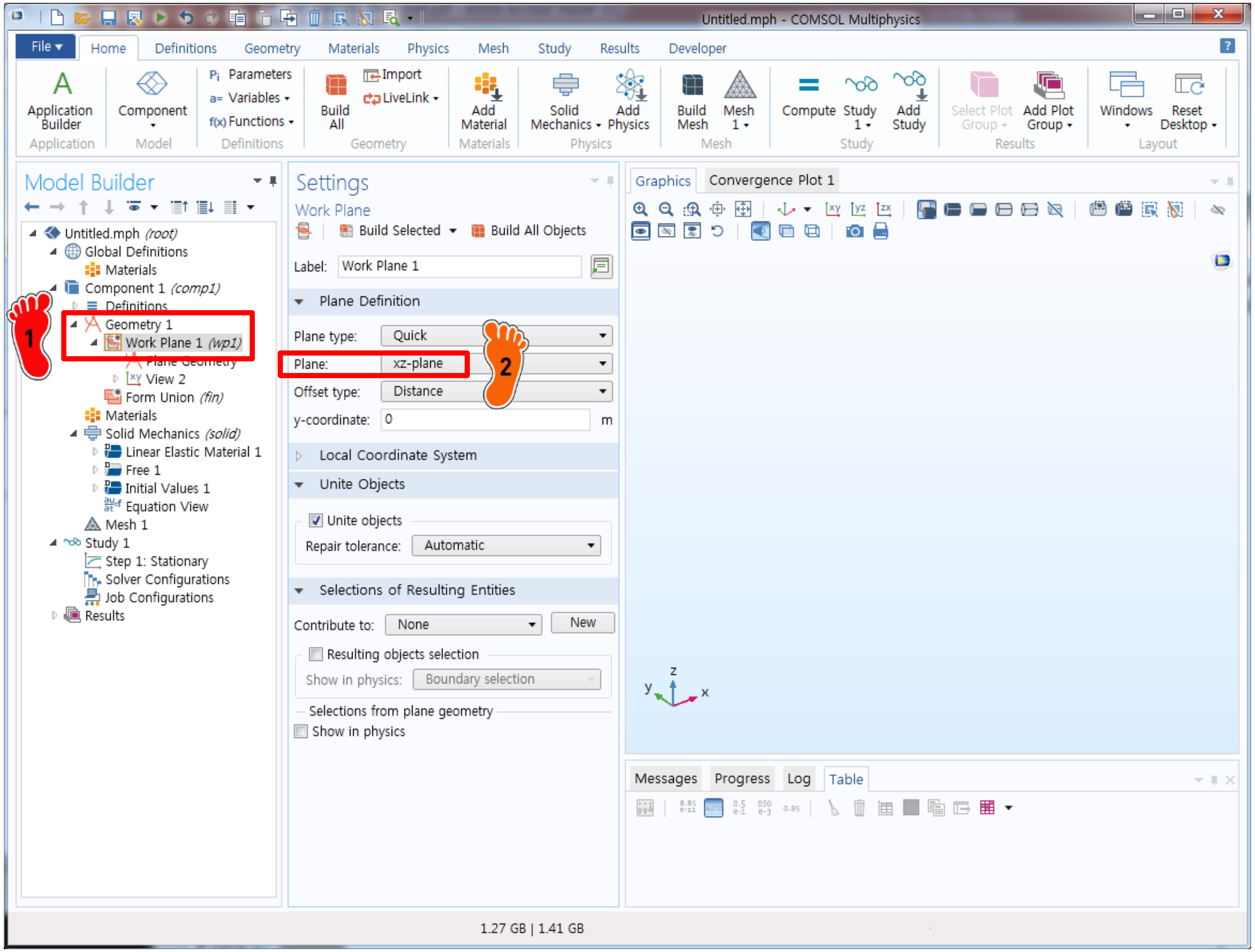


1

Dimension : 3D
Physics :
Structural Mechanics
→ Solid Mechanics
Study : Stationary

Done 클릭

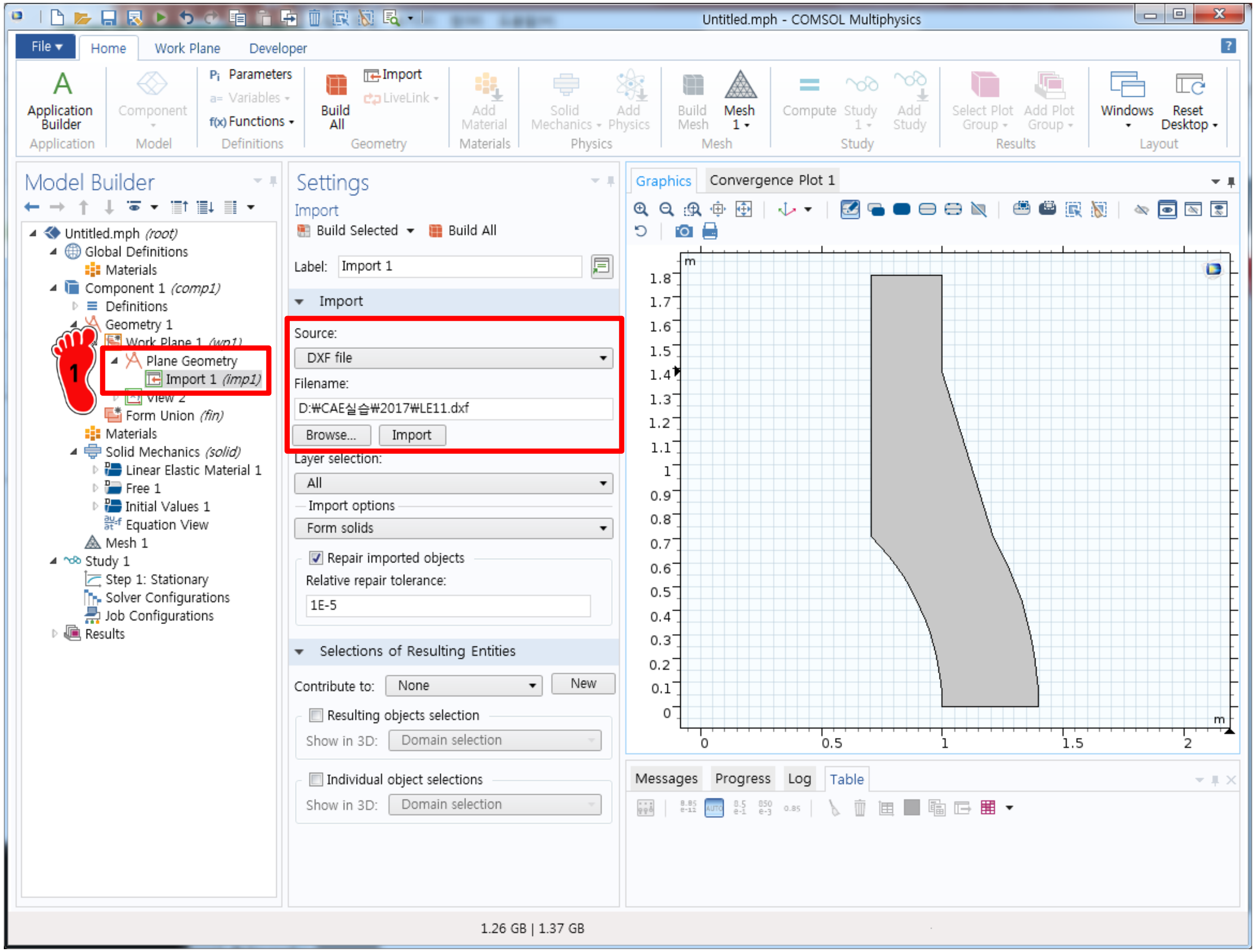
GEOMETRY CREATION



1 Geometry에서 Work Plane 생성

2 Work Plane에서 Plane을 xz-plane으로 변경

GEOMETRY CREATION



1 Plane Geometry 우클릭
→ Import

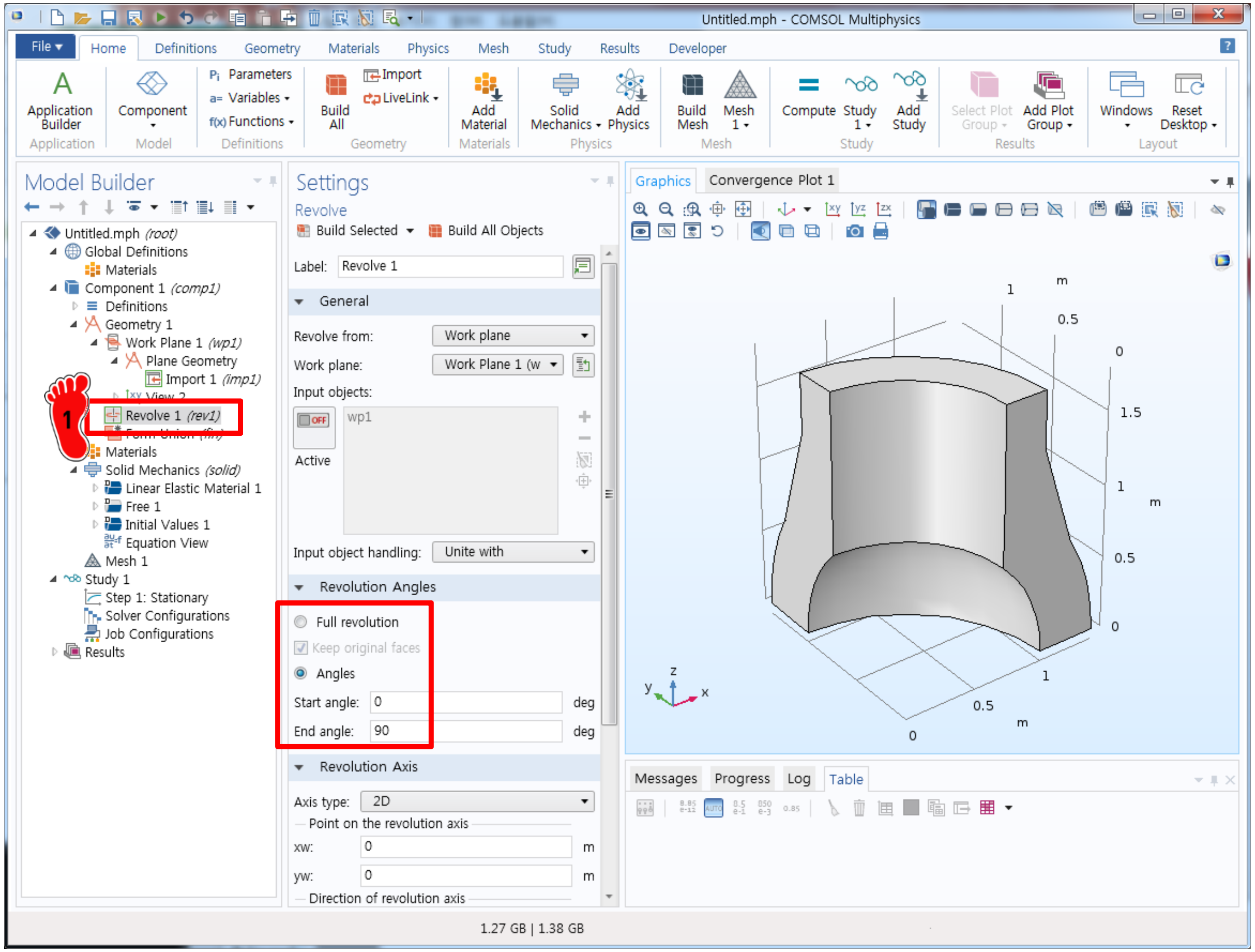
Settings 화면에서

1) Source : DXF file

2) Browse 버튼 클릭해서
LE11.dxf 파일 선택

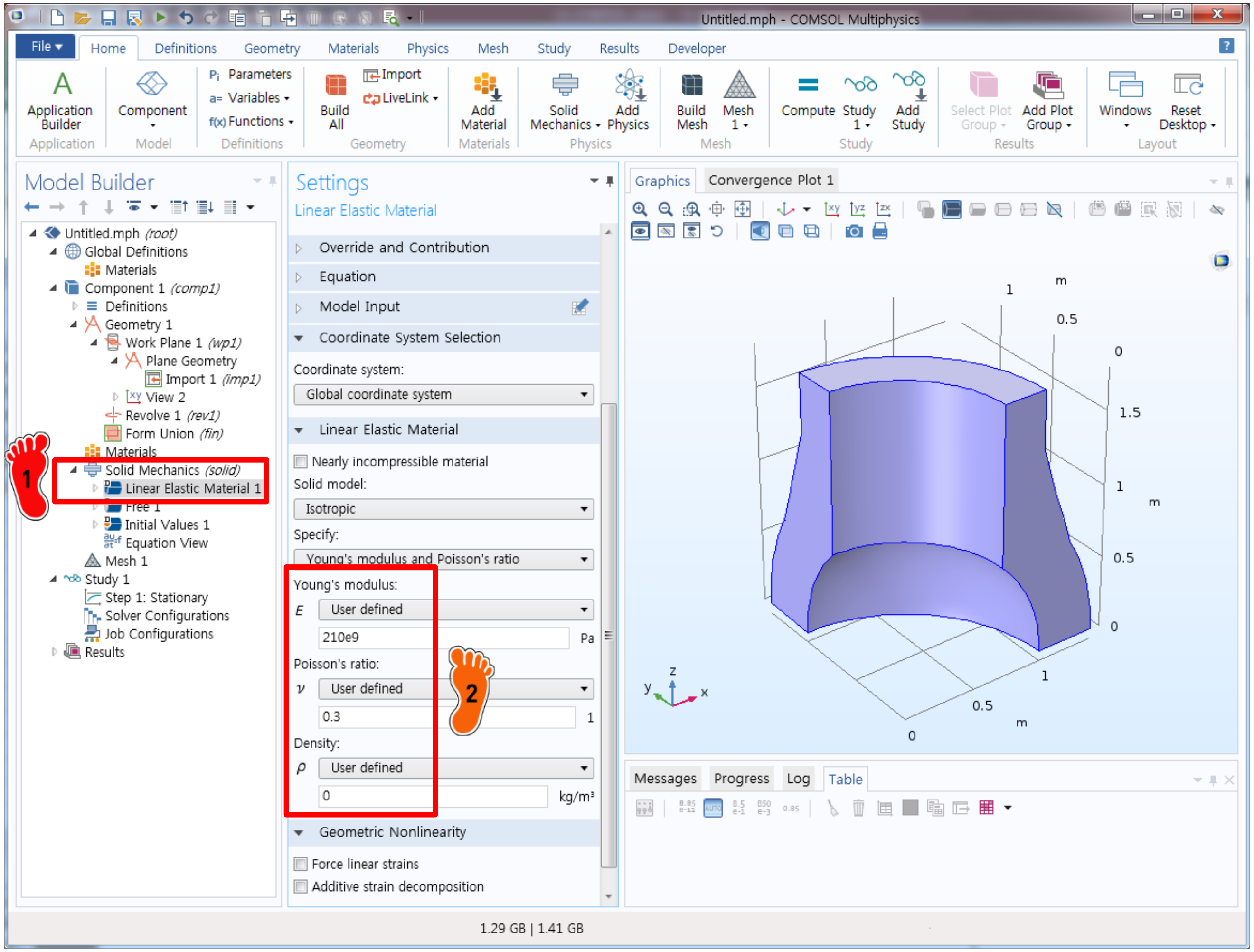
3) Import 클릭

GEOMETRY CREATION



1 Revolve 메뉴를 이용하여 90도 회전시킨 형상을 생성

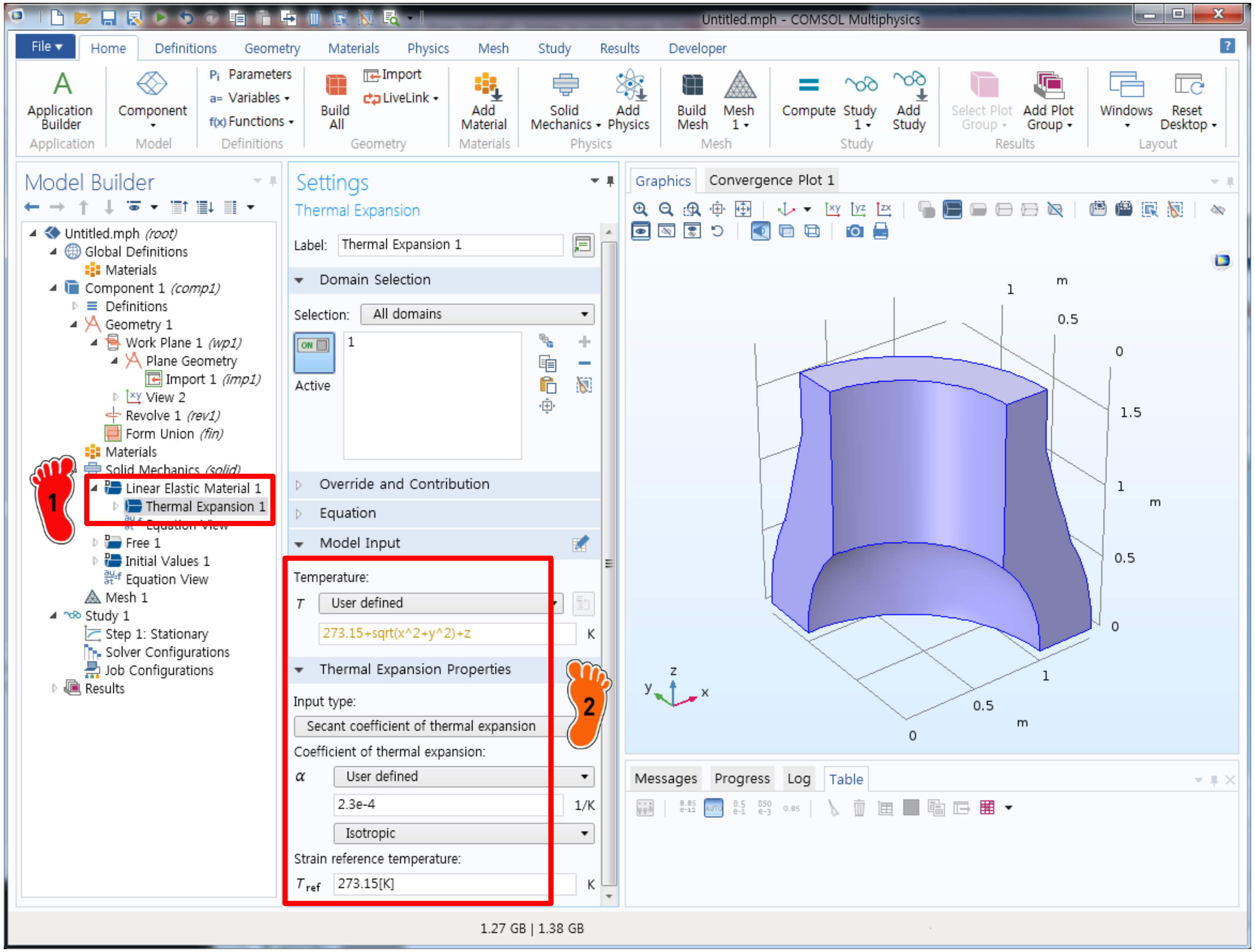
MATERIAL PROPERTY



1 Linear Elastic Material 클릭

2 E: 210e9
mu: 0.3
rho : 0 입력

MATERIAL PROPERTY



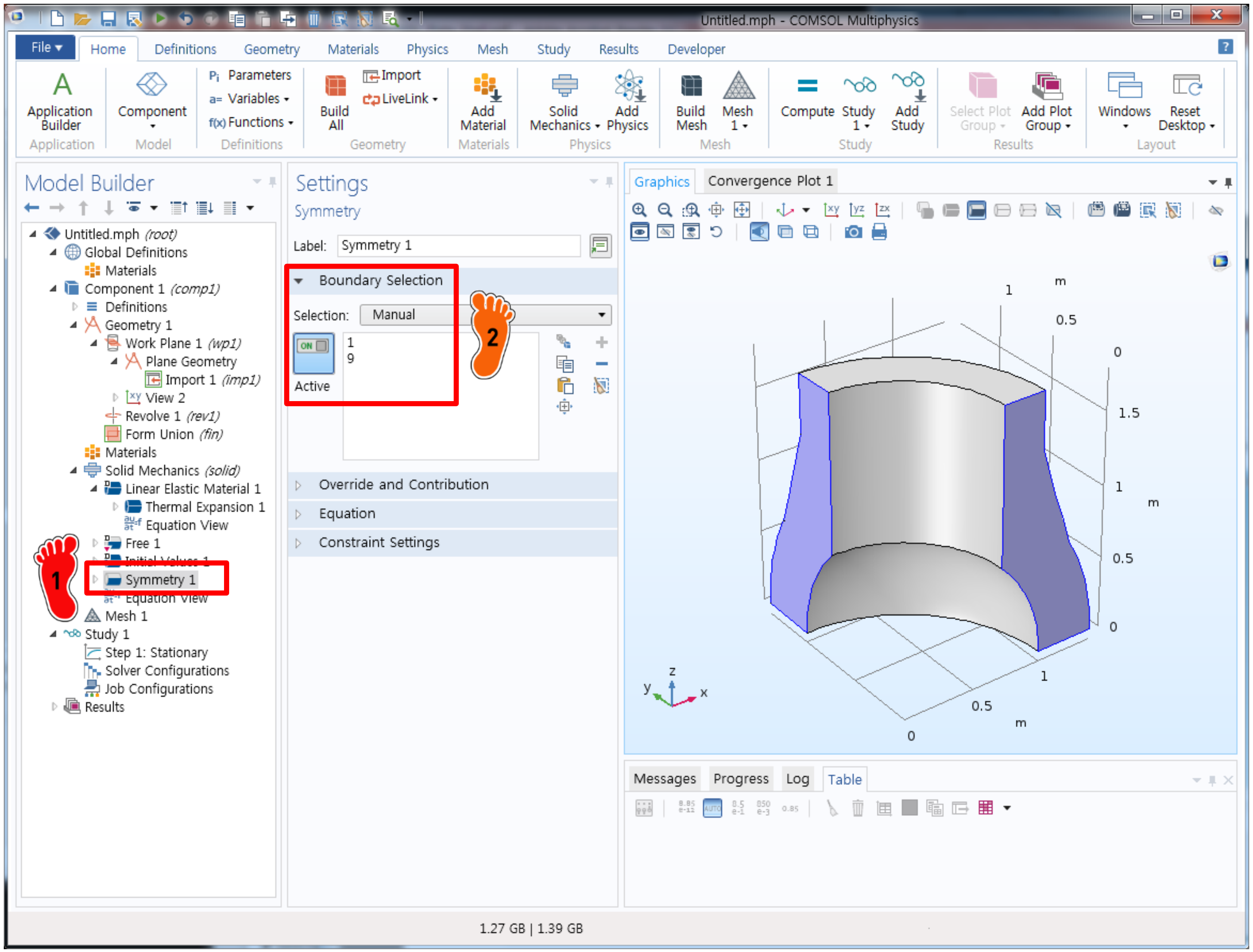
1 Linear Elastic Material 하위 메뉴로 Thermal Expansion 추가

2 ① Temperature: 273.15+sqrt(x^2+y^2)+z

② α: 2.3e-4

③ Strain reference Temperature: 273.15

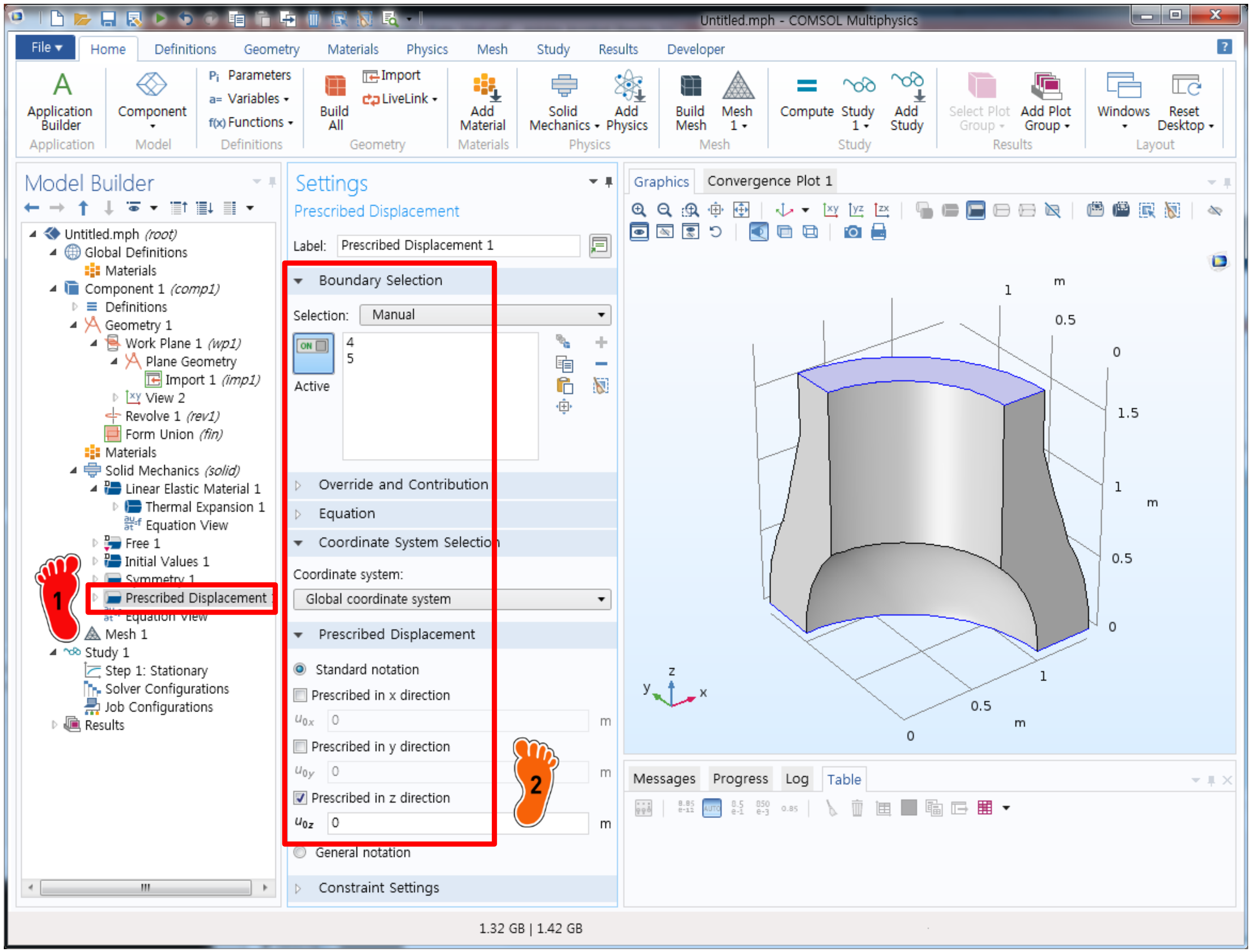
BOUNDARY CONDITION



1 Symmetry 경계조건 생성

2 1, 9번 경계 선택

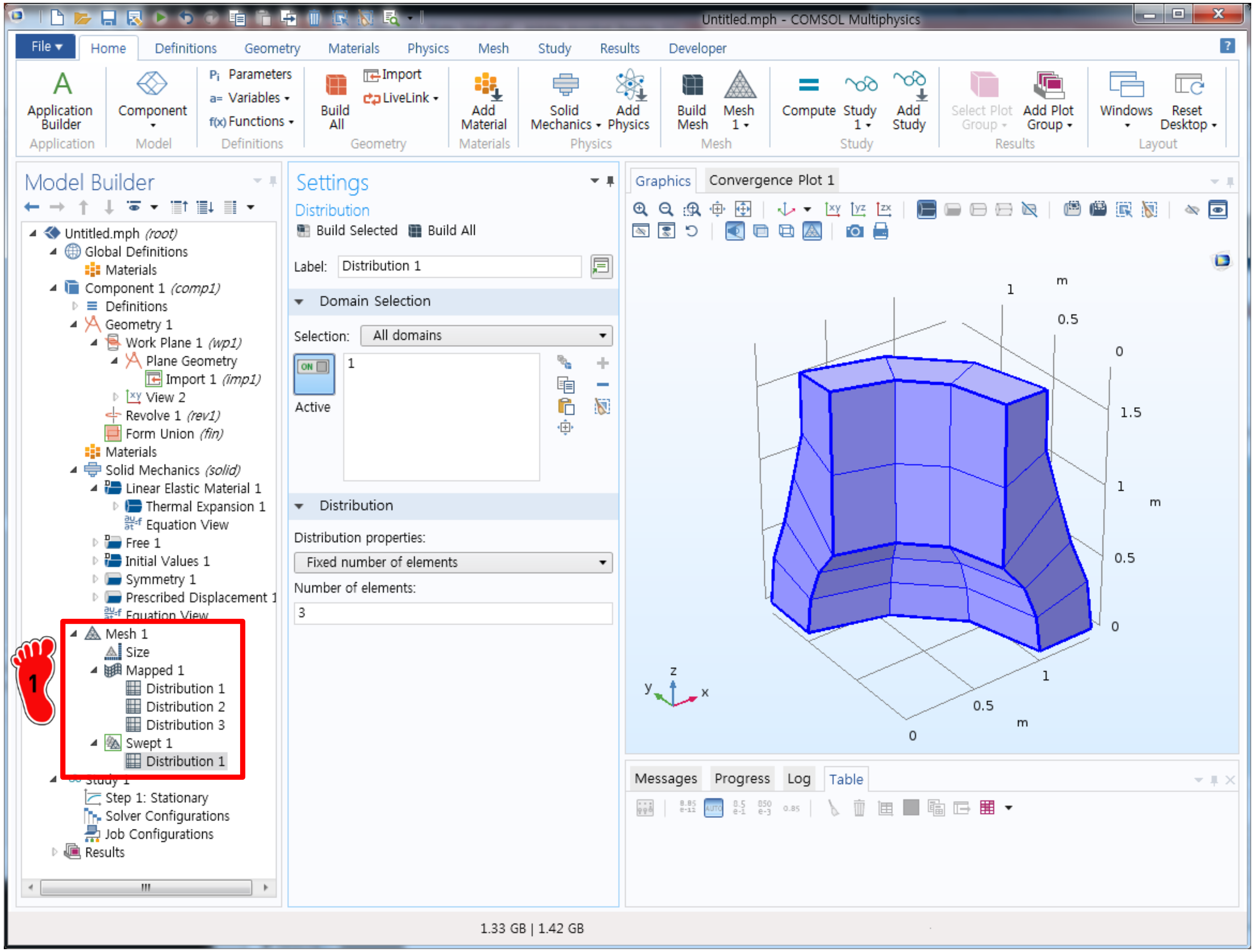
BOUNDARY CONDITION



1 Prescribed Displacement 경계조건 생성

2 4(위쪽), 5(아래쪽)번 경계 선택 후 z방향 변위에 0입력

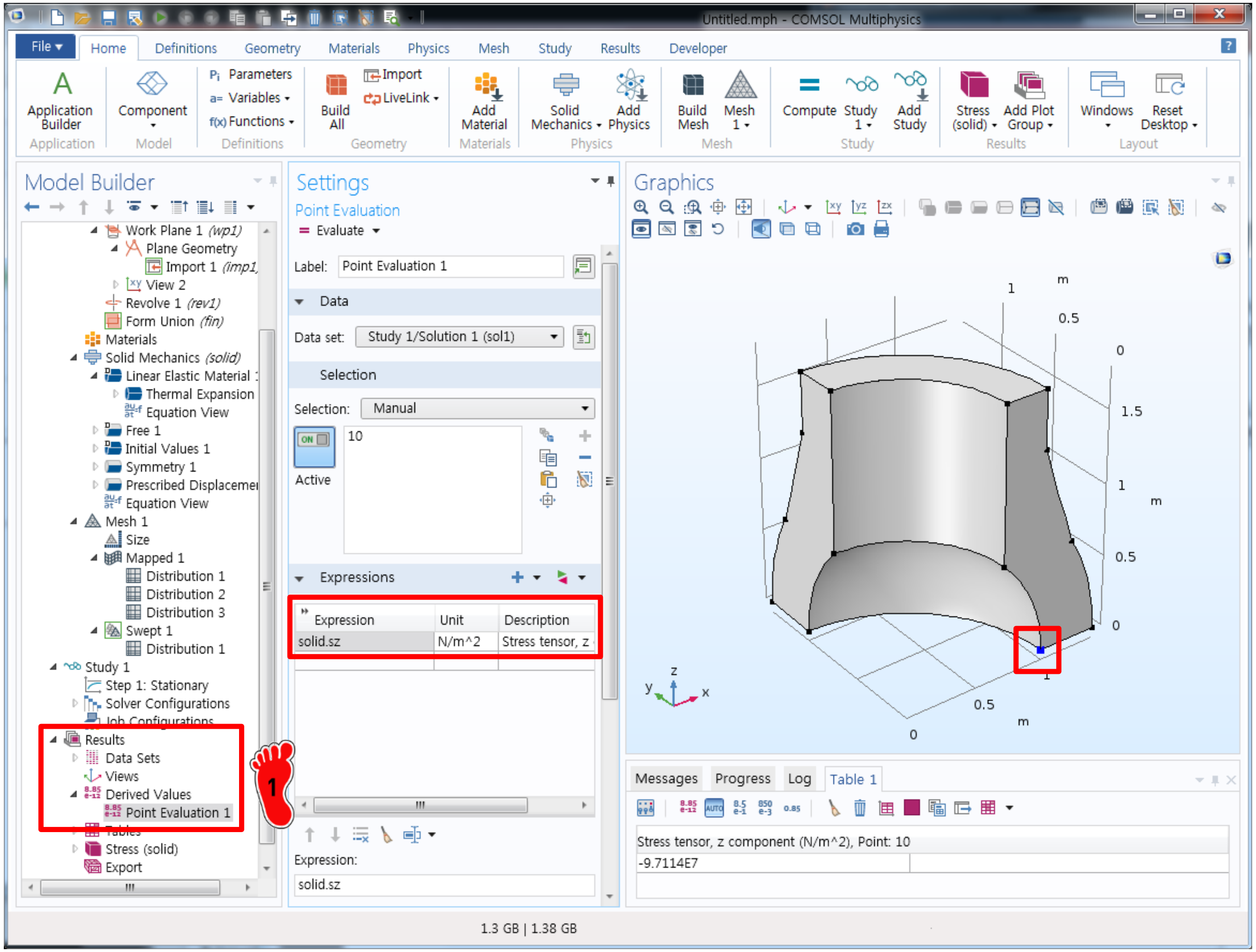
MESH



1 Mapped mesh 와 Swept 기능을 이용하여 요소 생성

Compute를 클릭하여 해석 수행

POST-PROCESSING

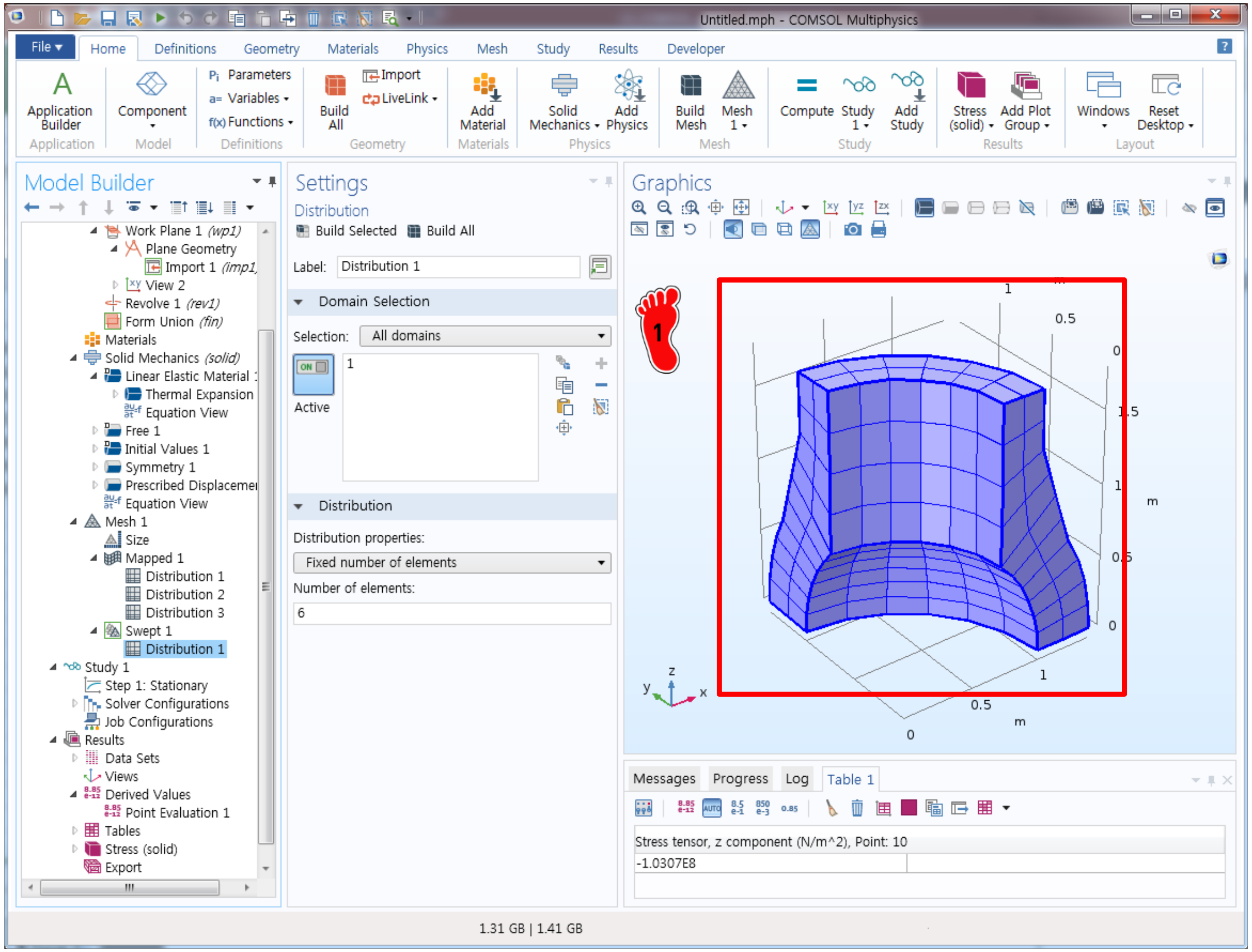


1 Derived Values 의 Point Evaluation 기능을 이용하여 10번 절점의 solid.sz 응력을 계산

결과값: -97.1 MPa

※ Ref : -105 MPa

MESH

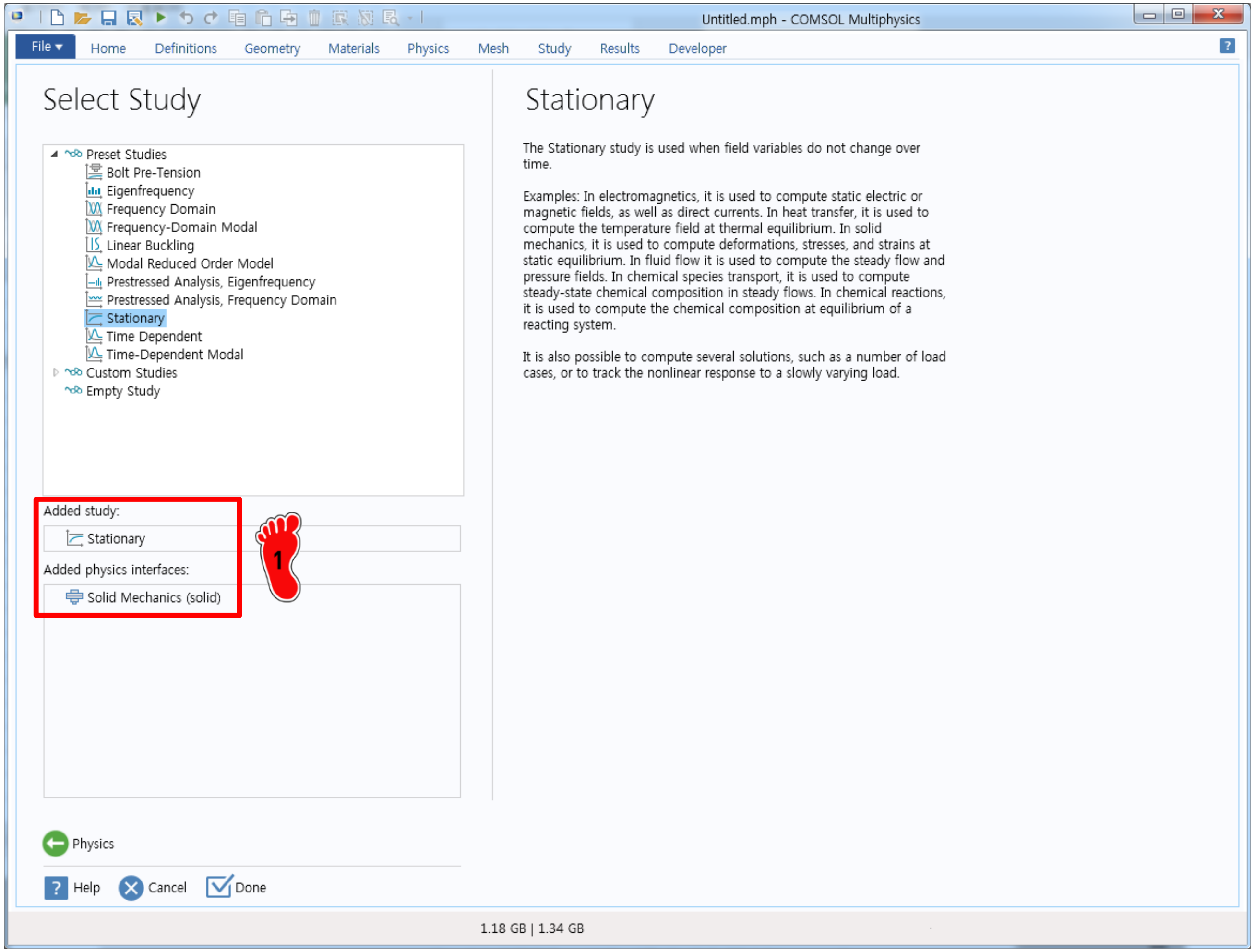


Fine mesh로 변경 후 해석 실행

결과값: -103.1 MPa

※ Ref : -105 MPa

ENVIRONMENT SETTING

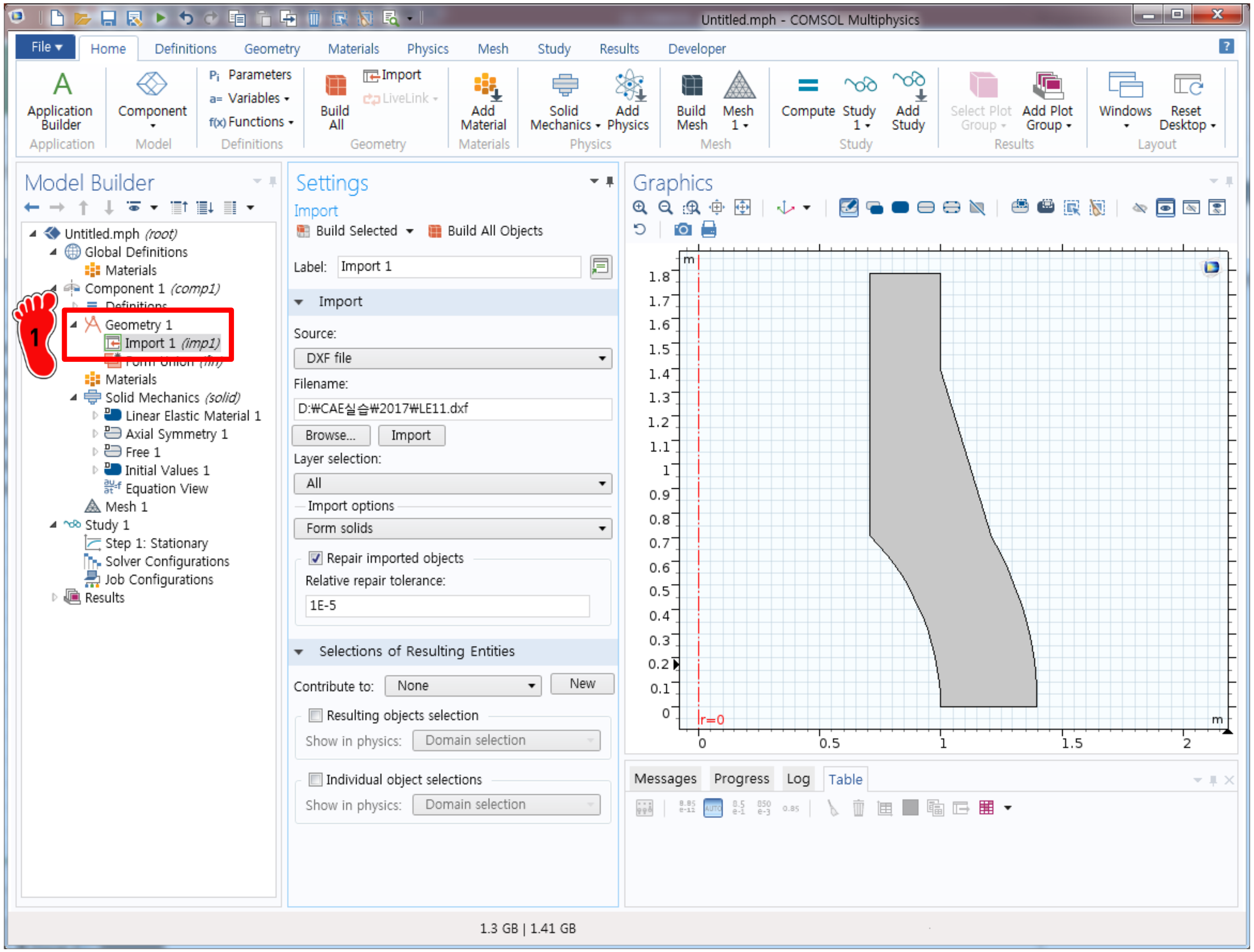


1

Dimension : 2D Axissym.
Physics :
Structural Mechanics
→ Solid Mechanics
Study : Stationary

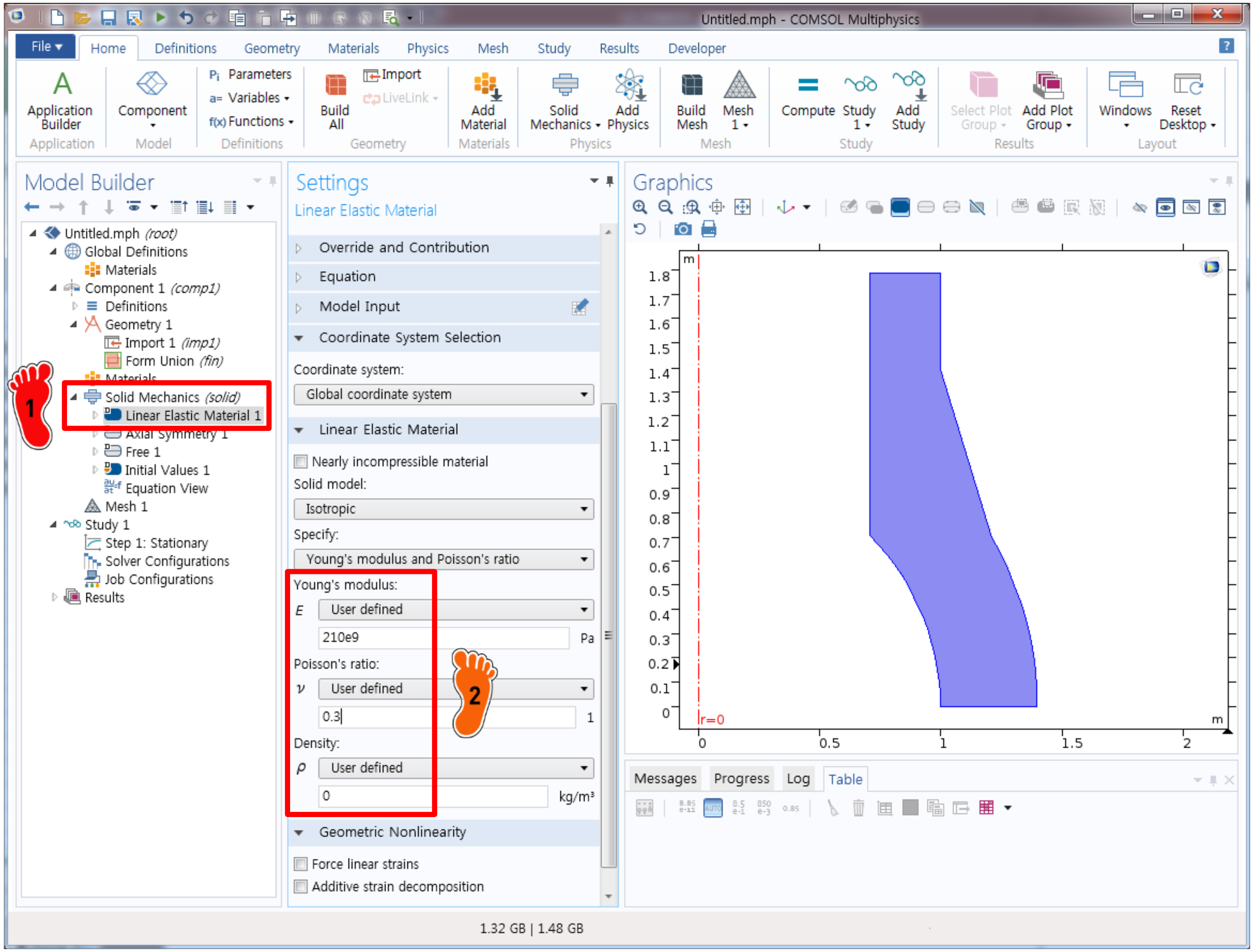
Done 클릭

GEOMETRY CREATION



1 Import 메뉴를 이용하여 주어진 CAD형상을 삽입

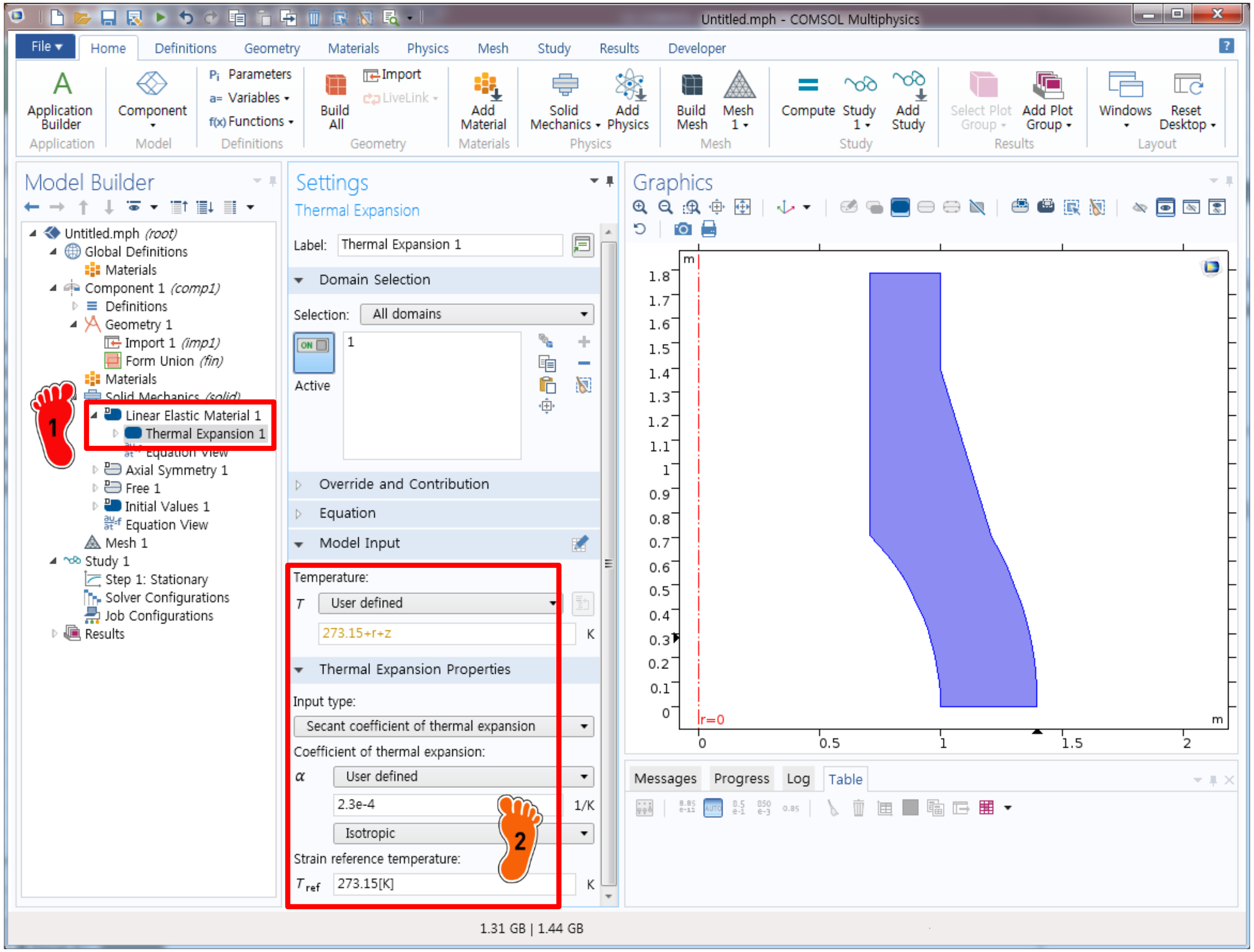
MATERIAL PROPERTY



1 Linear Elastic Material 클릭

2 E: 210e9
mu: 0.3
rho : 0 입력

MATERIAL PROPERTY



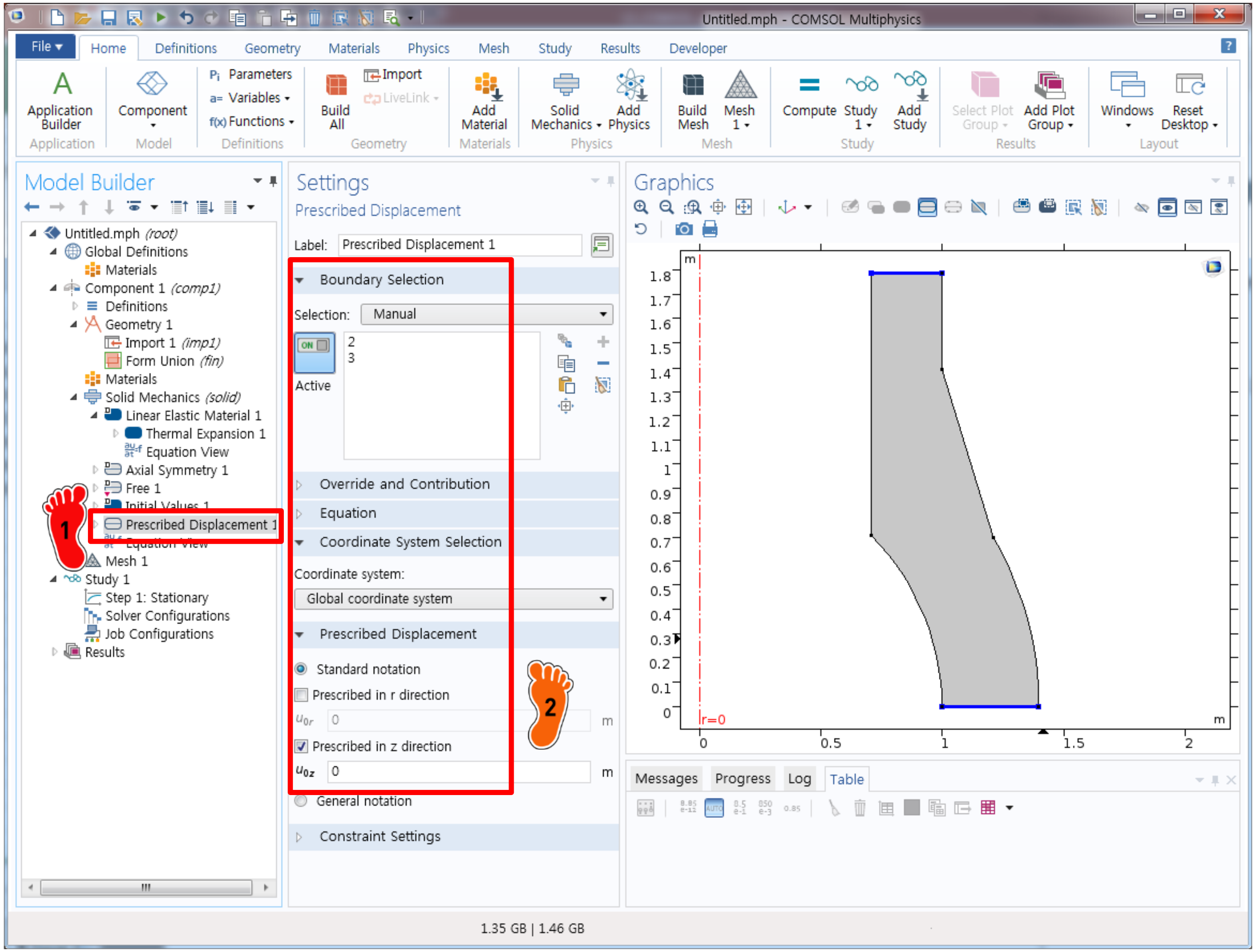
1 Linear Elastic Material 하위 메뉴로 Thermal Expansion 추가

2 ① Temperature: 273.15+r+z

② α: 2.3e-4

③ Strain reference Temperature: 273.15

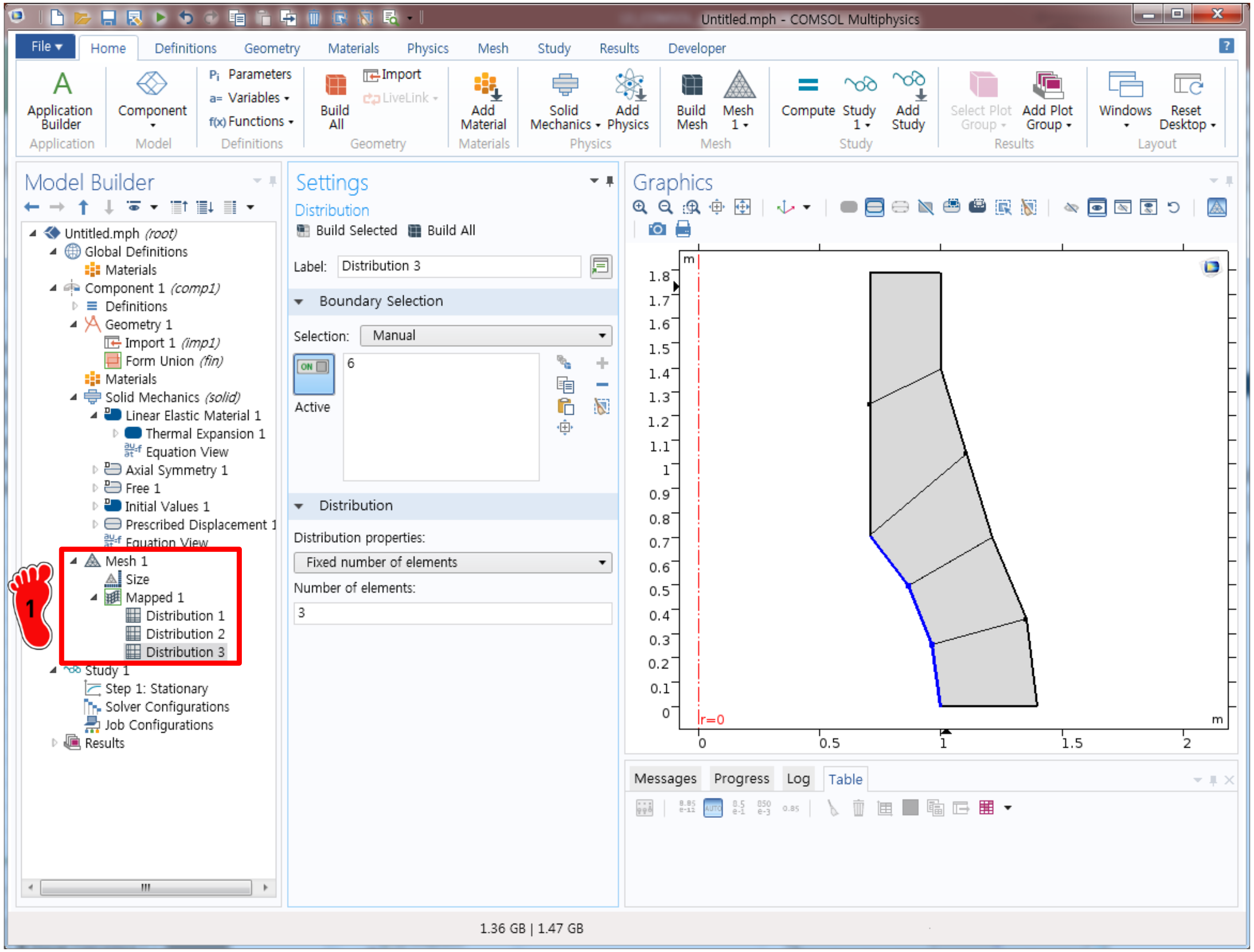
BOUNDARY CONDITION



1 Prescribed Displacement 경계조건 생성

2 2, 3번 경계 선택 후 z 방향 변위에 0 입력

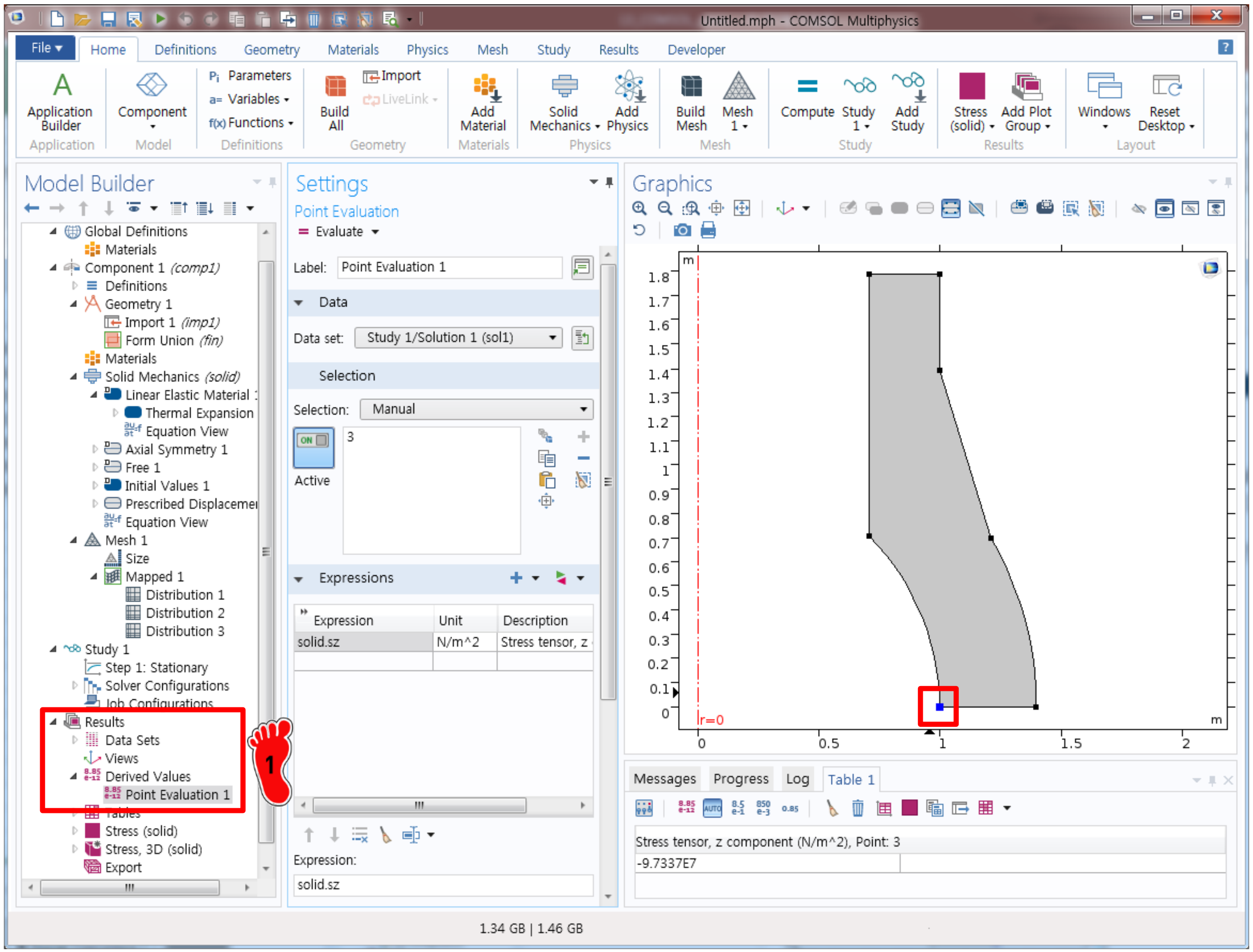
MESH



Mapped mesh 기능을 이용하여 요소 생성

Compute를 클릭하여 해석 수행

POST-PROCESSING

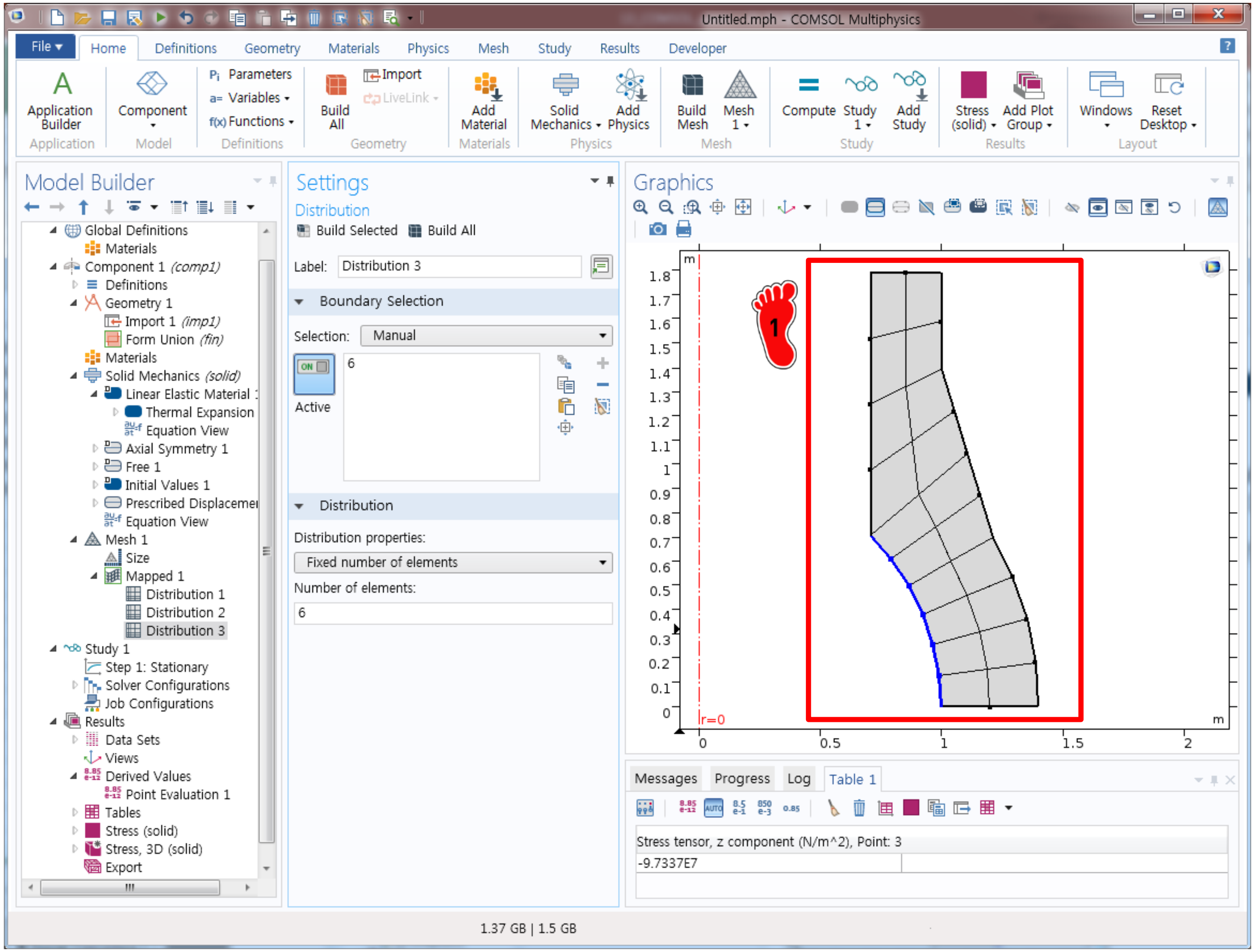


1 Derived Values 의 Point Evaluation 기능을 이용하여 3번 절점의 solid.sz 응력을 계산

결과값: -97.3 MPa

※ Ref : -105 MPa

MESH



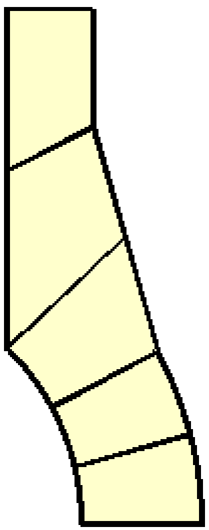
Fine mesh로 변경 후 해석 실행

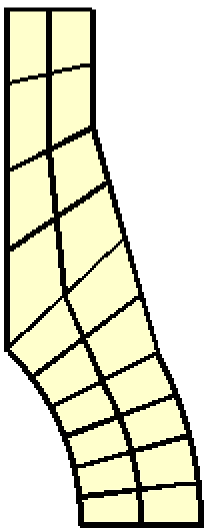
결과값: -103.7 MPa

※ Ref : -105 MPa

SUMMARY

- Meshes





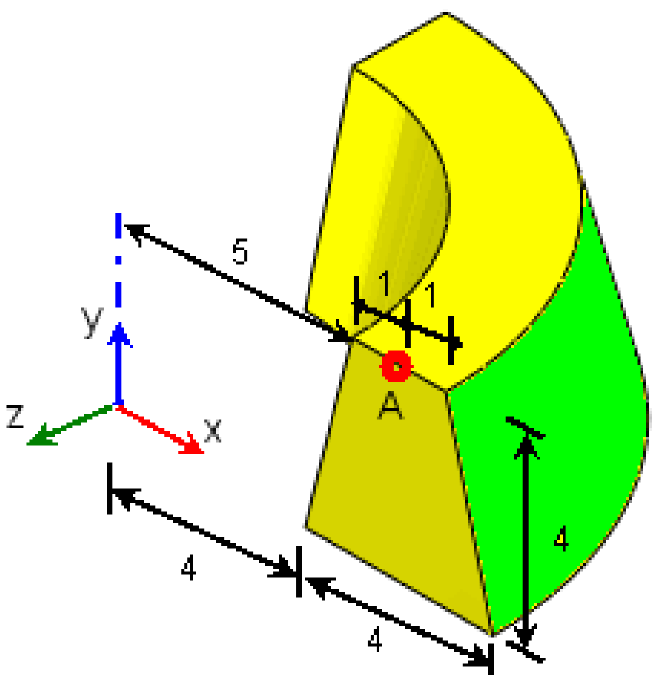
– Direct stress (σ_{zz}) at point A: -105 MPa

Software	Order	Element	Coarse mesh		Fine mesh	
			Result	Error	Result	Error
COMSOL	2	axisymmetry	-97.3	7.33 %	-103.7	1.24 %
COMSOL	2	hexahedron	-97.1	7.52 %	-103.1	1.81 %

HOMEWORK: SOLID MECHANICS

Tapered Thick Shell under Pressure and Gravity Loads

Determine the hoop stress at point A of a tapered thick shell subjected to the following loading conditions:
Study A. Uniform normal pressure of 100 MPa on outer face highlighted in green.
Study B. Uniform acceleration - 9.81 m/s^2 in global y-direction.
The bottom face on the xz plane is constrained in the vertical y-direction.



Modulus of elasticity (E) = 210 GPa
Poisson's ratio (ν) = 0.3
Density(ρ): 7000 kg/m^3

Von Mises stress at point A

Study A : 211.6 MPa , Study B : 0.031 MPa

Study case 1 : 3D Solid Model
Study case 2 : 2D Axisymmetric Model