

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

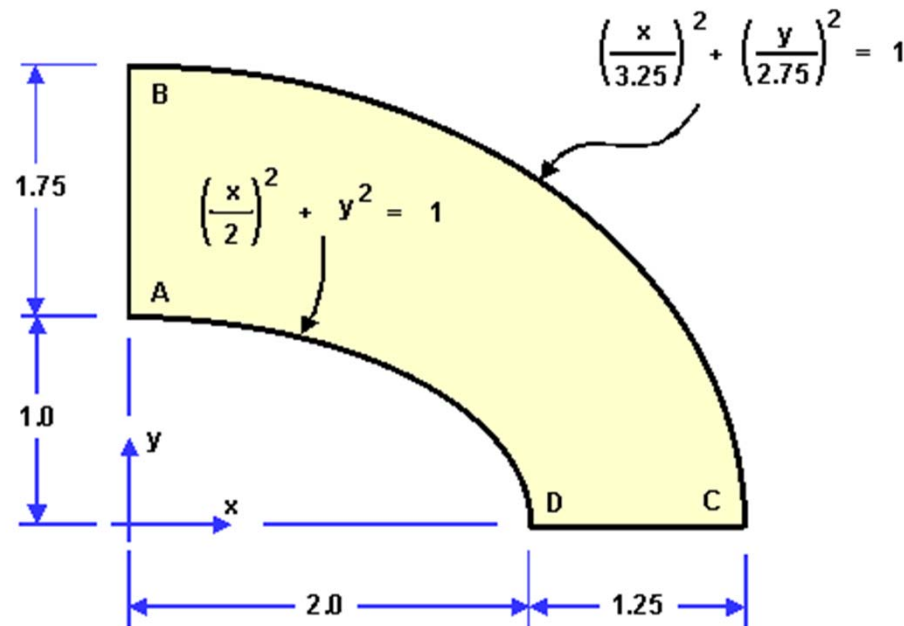
The Shell interface () is intended for the structural analysis of thin-walled structures. The formulation used in the Shell interface is a Mindlin-Reissner type, which means that transverse shear deformations are accounted for, and it can therefore be used for rather thick shells. It is also possible 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

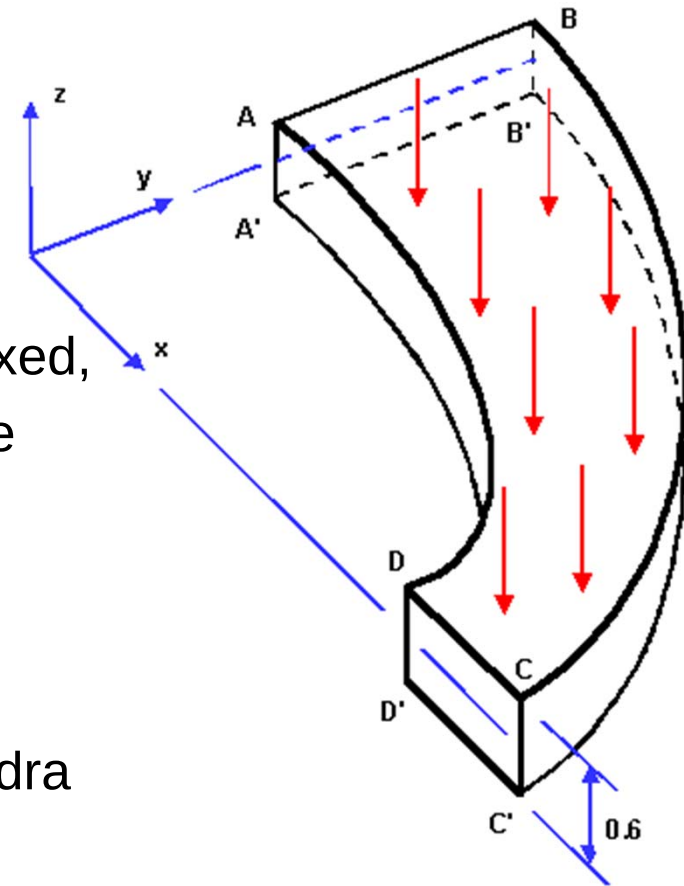
Thick Plate – Pressure

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

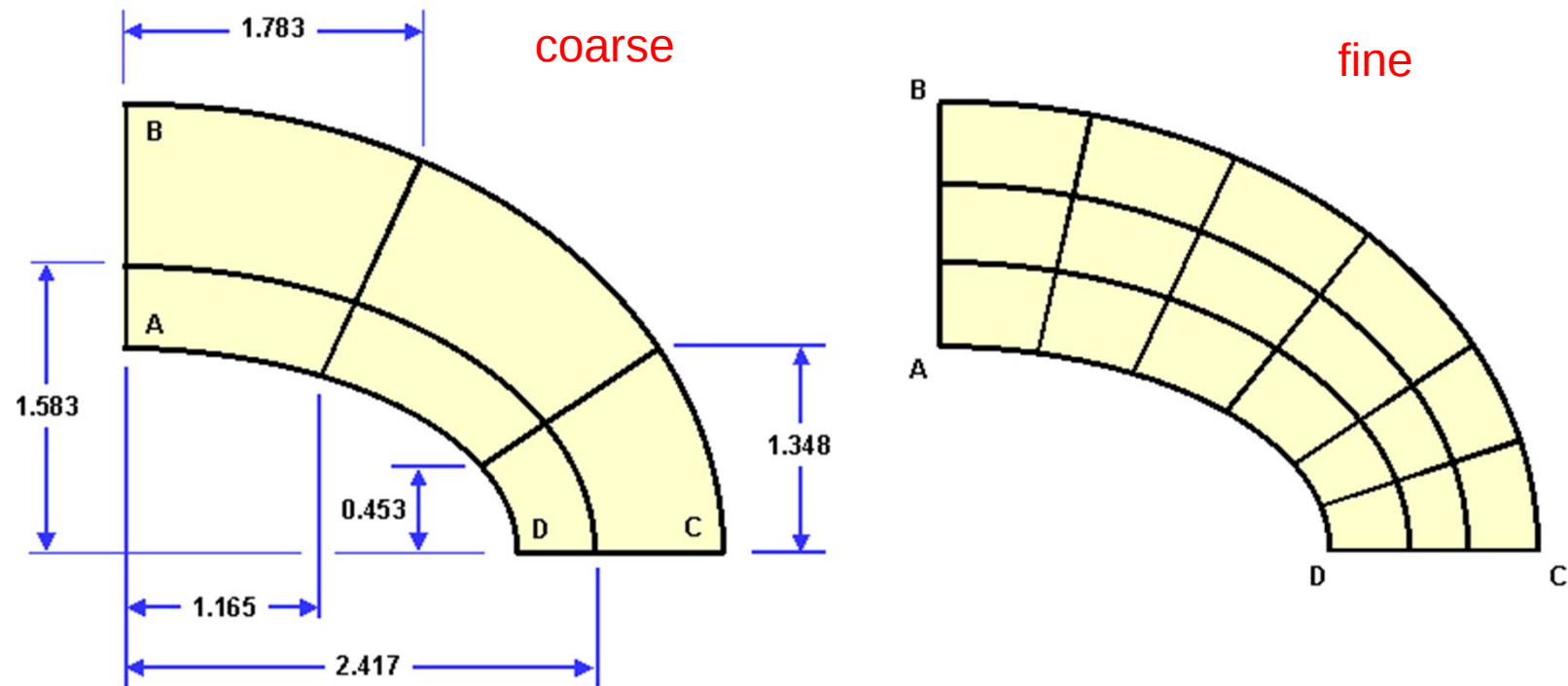


COMSOL Solid Mechanics Module Verification Models: Thick Plate Stress Analysis

- 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: $E = 210 \cdot 10^3$ MPa, $\nu = 0.3$
- Element types
 - Solid hexahedra, wedges and tetrahedra



- Meshes



- Output

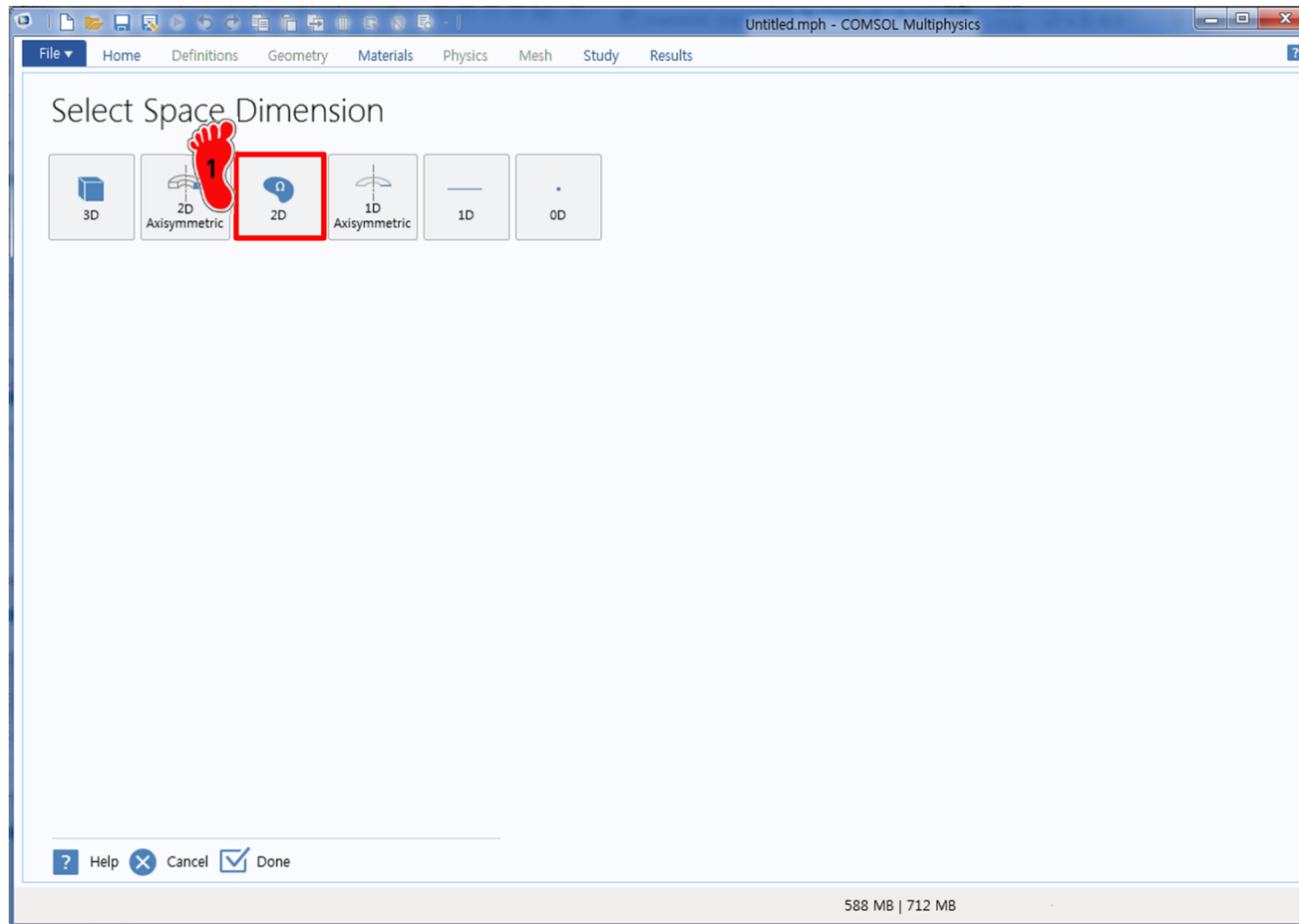
- Direct stress (σ_{yy}) at point D: -5.38 MPa

LE10 COMPARISON

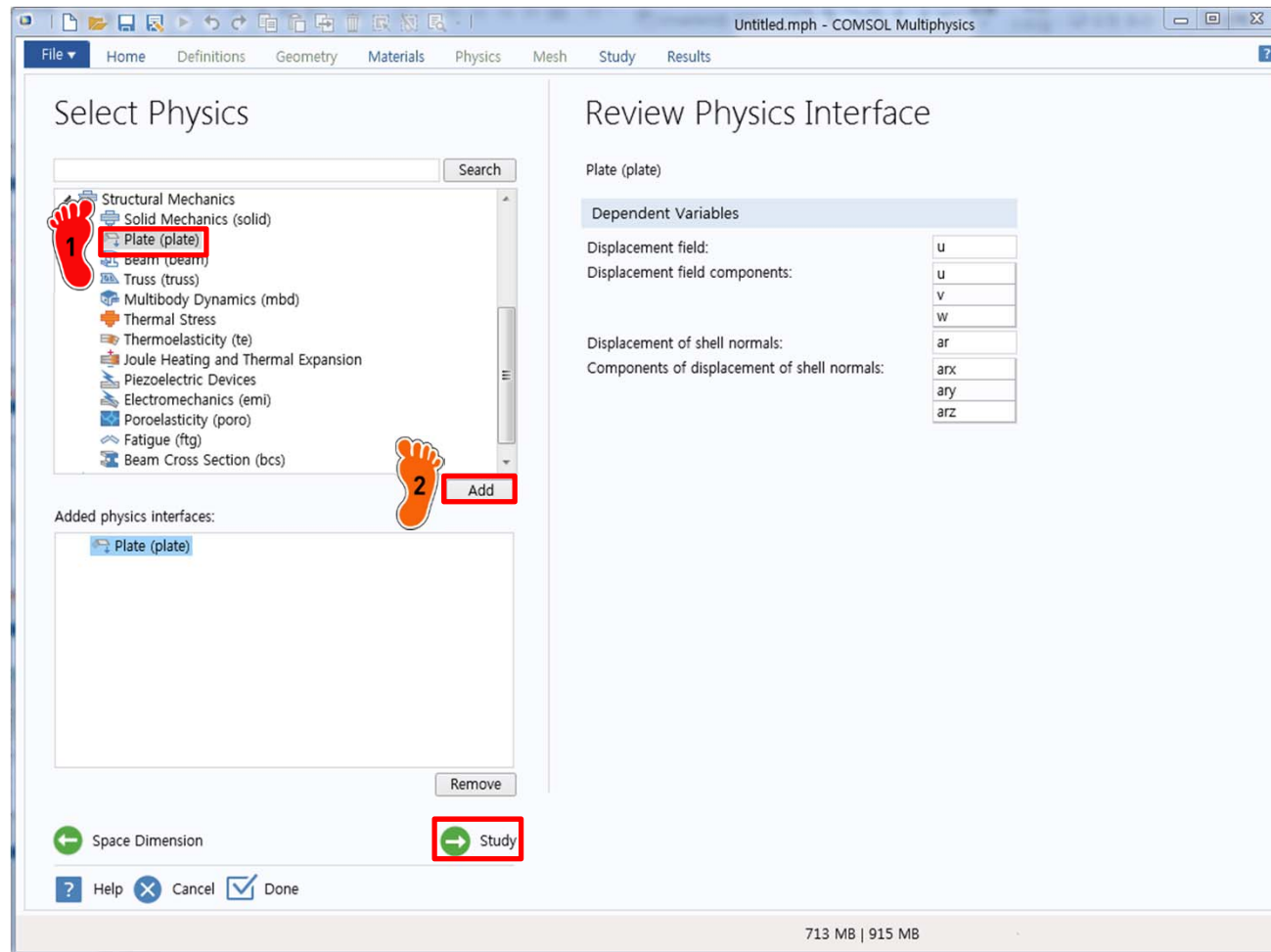
LE 10	order	integration points	element	coarse mesh		fine mesh	
				s22	error	s22	error
<i>Abaqus</i> *	2	8	C3D20R	-7.93E+06	47.40%	-5.53E+06	2.79%
Ansys	2	8	SOLID95	-5.36E+06	-0.32%	-5.61E+06	4.26%
CalculiX	2	8	C3D20R	-5.36E+06	-0.32%	-5.61E+06	4.26%
Tahoe	2	8	hexahedron				
<i>Abaqus</i> *	2	27	C3D20	-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				

DIMENSION SELECTION

1 2D 선택



PHYSICS SELECTION

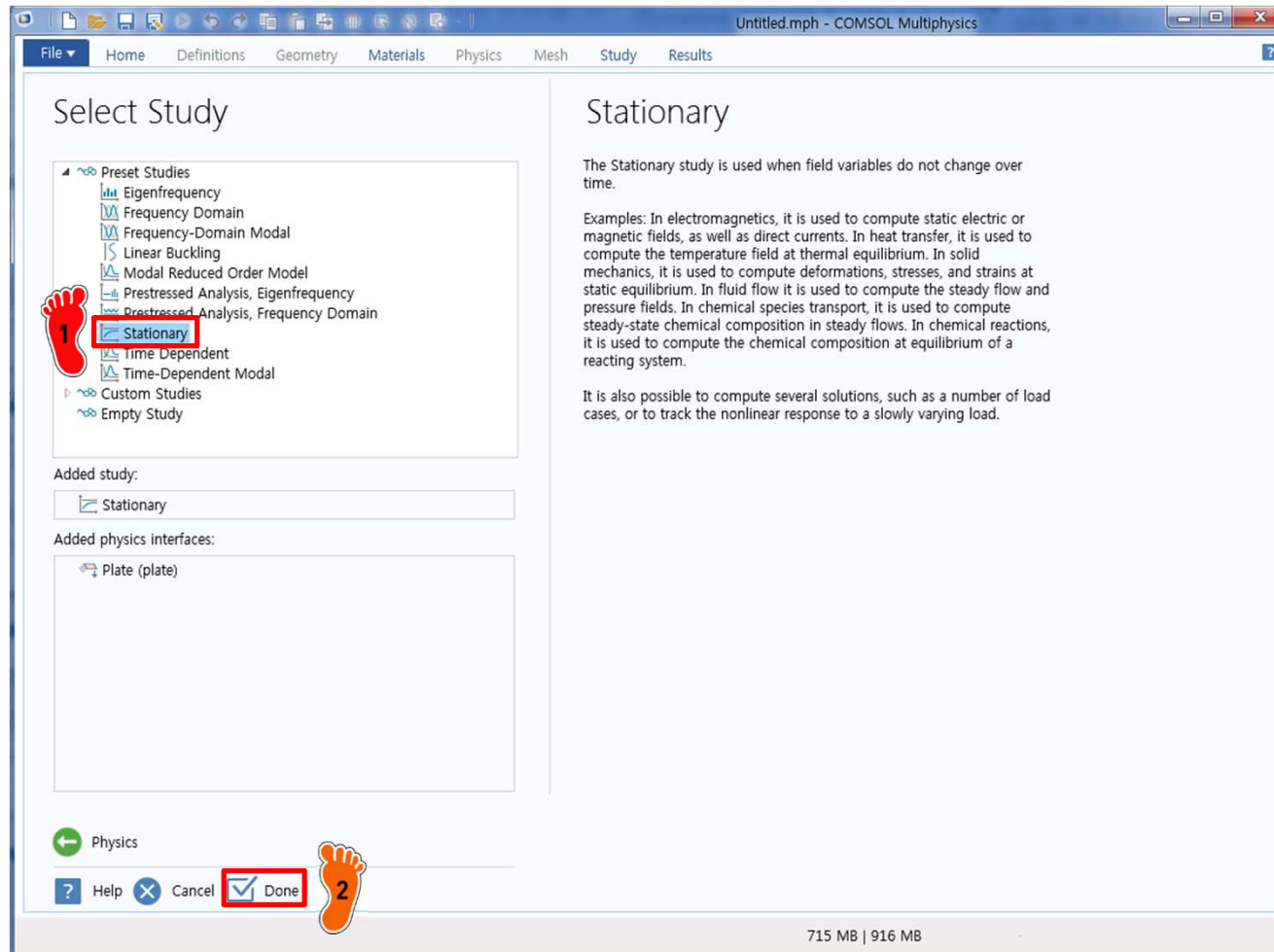


1 Structural Mechanics 의 Plate 선택

2 Add 클릭

3 Study 클릭

STUDY TYPE SELECTION



1 Stationary 선택

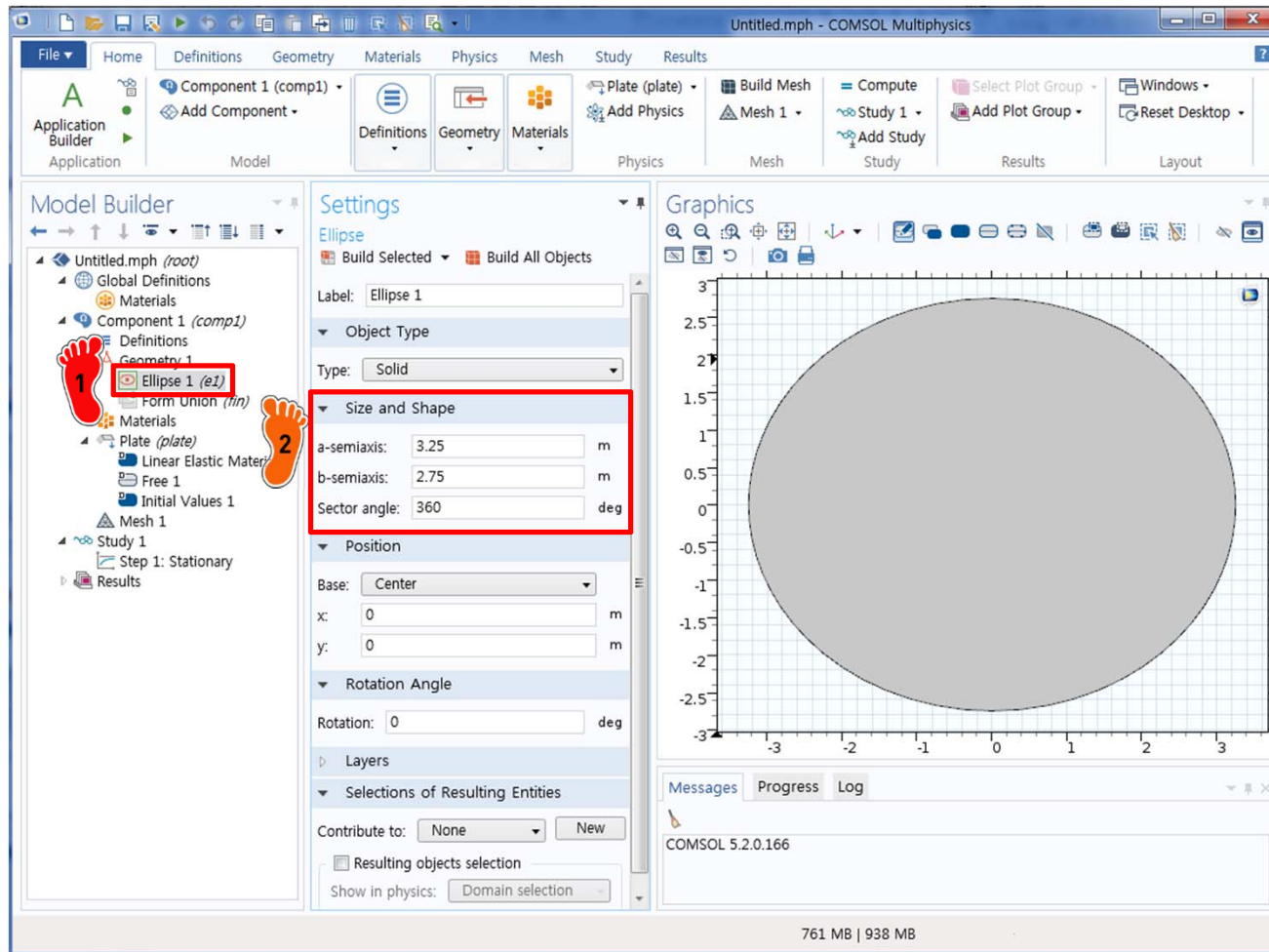
2 Done 클릭

GEOMETRY CREATION

1 Ellipse 1 생성

2 a-semiaxis: 3.25
b-semiaxis: 2.75

입력 후 타원 생성

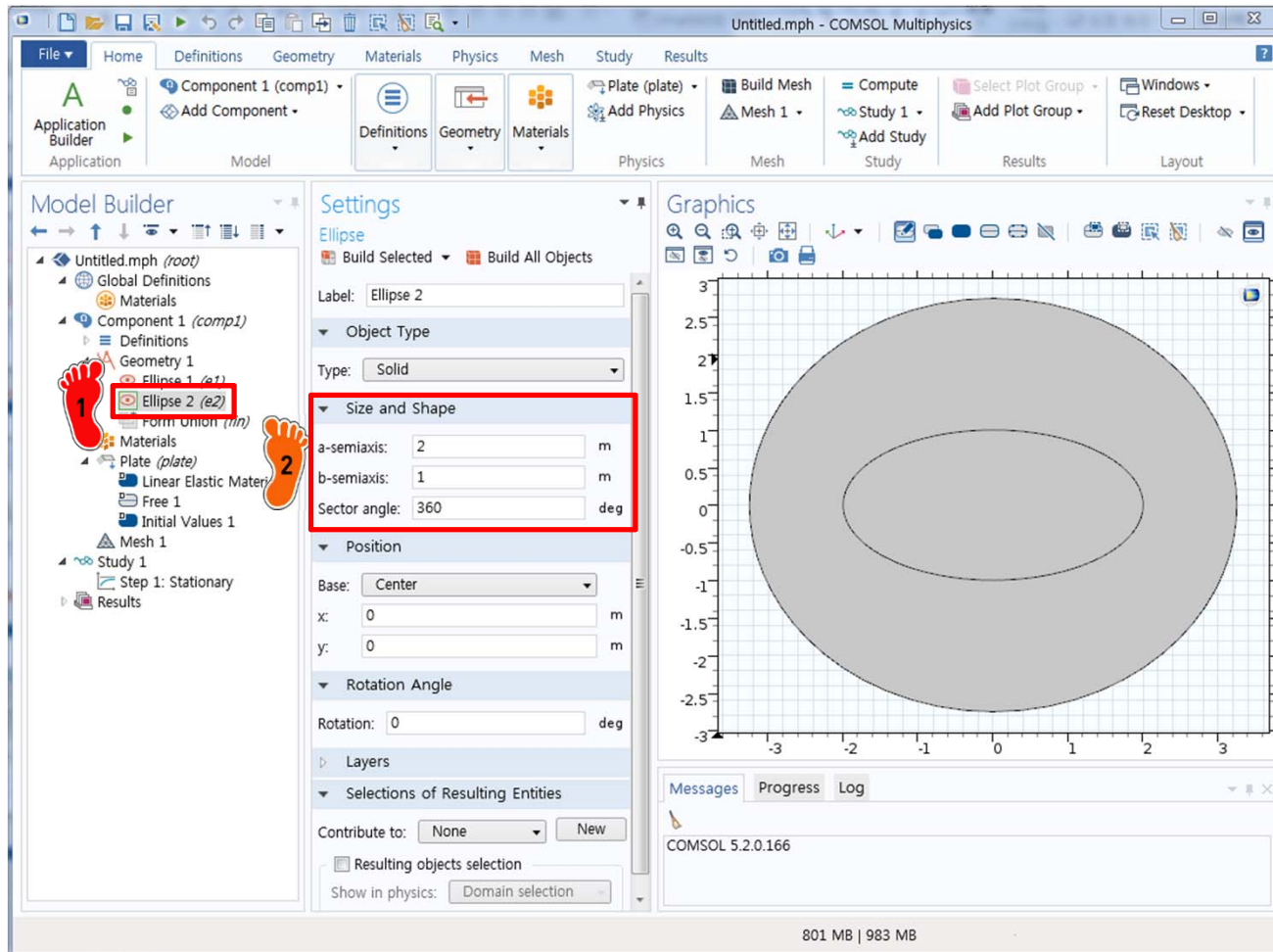


GEOMETRY CREATION

1 Ellipse 2 생성

2 a-semiaxis: 2
b-semiaxis: 1

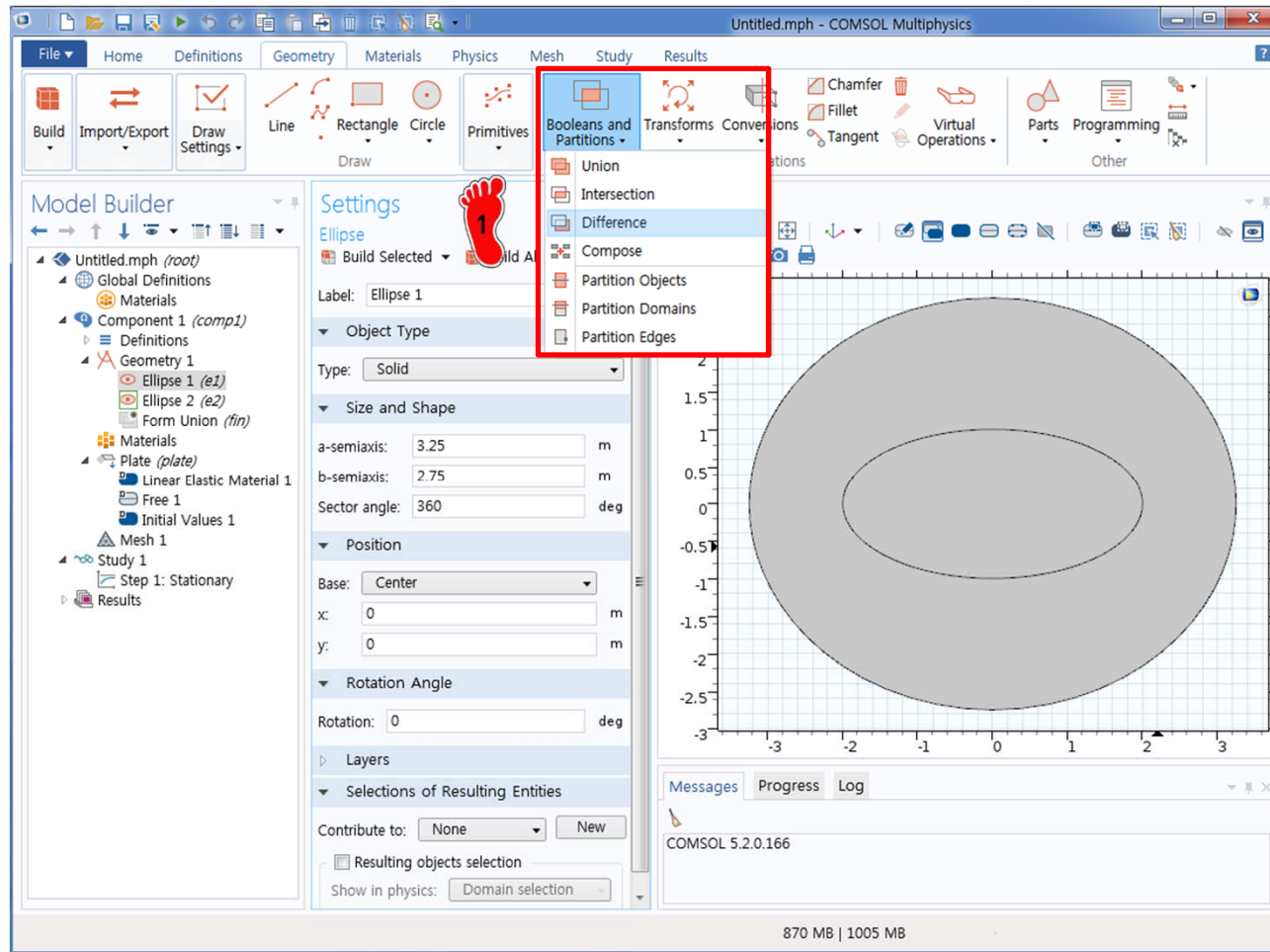
입력 후 타원 생성



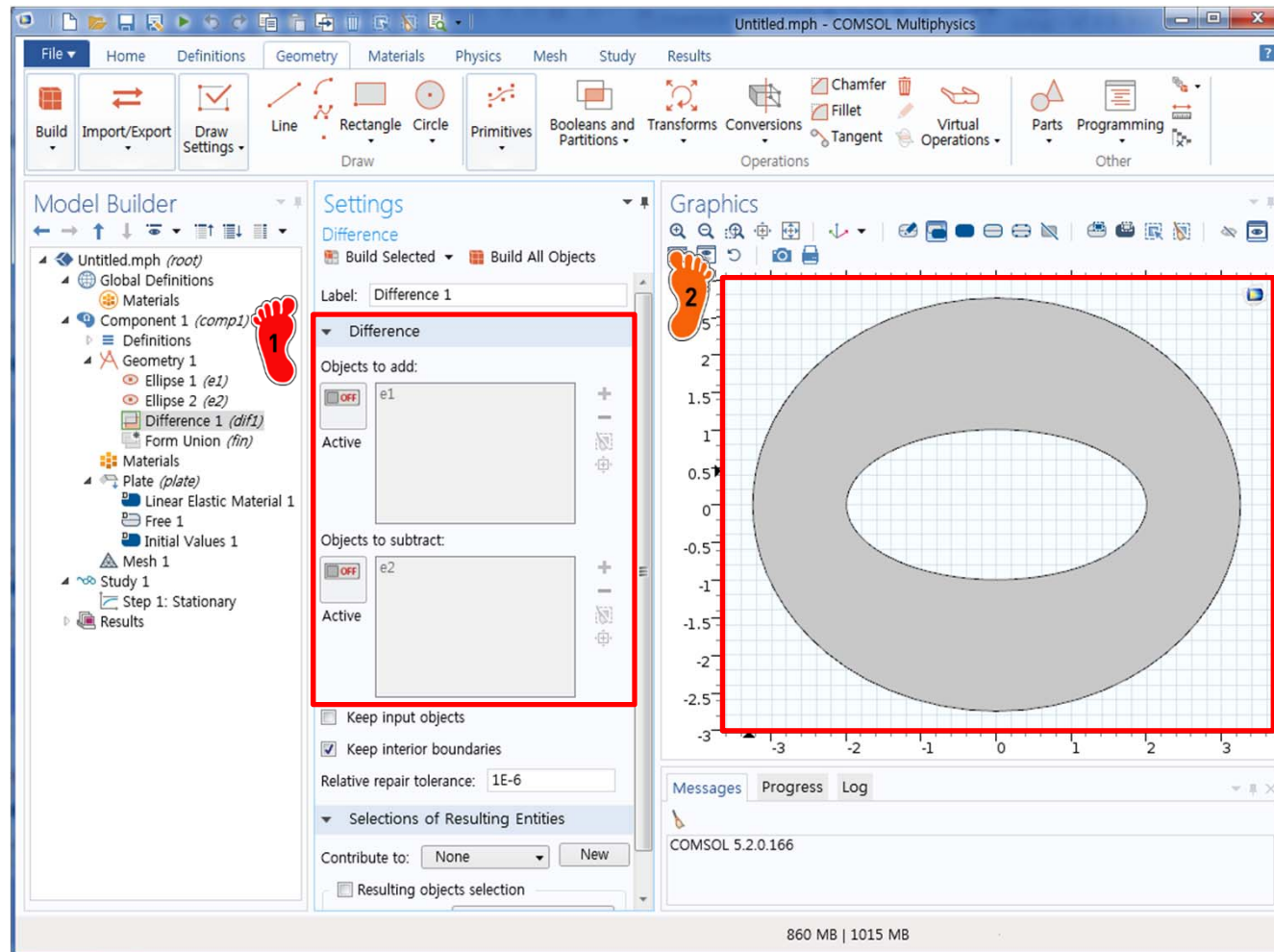
GEOMETRY CREATION



Booleans and Partitions에서 Difference를 선택



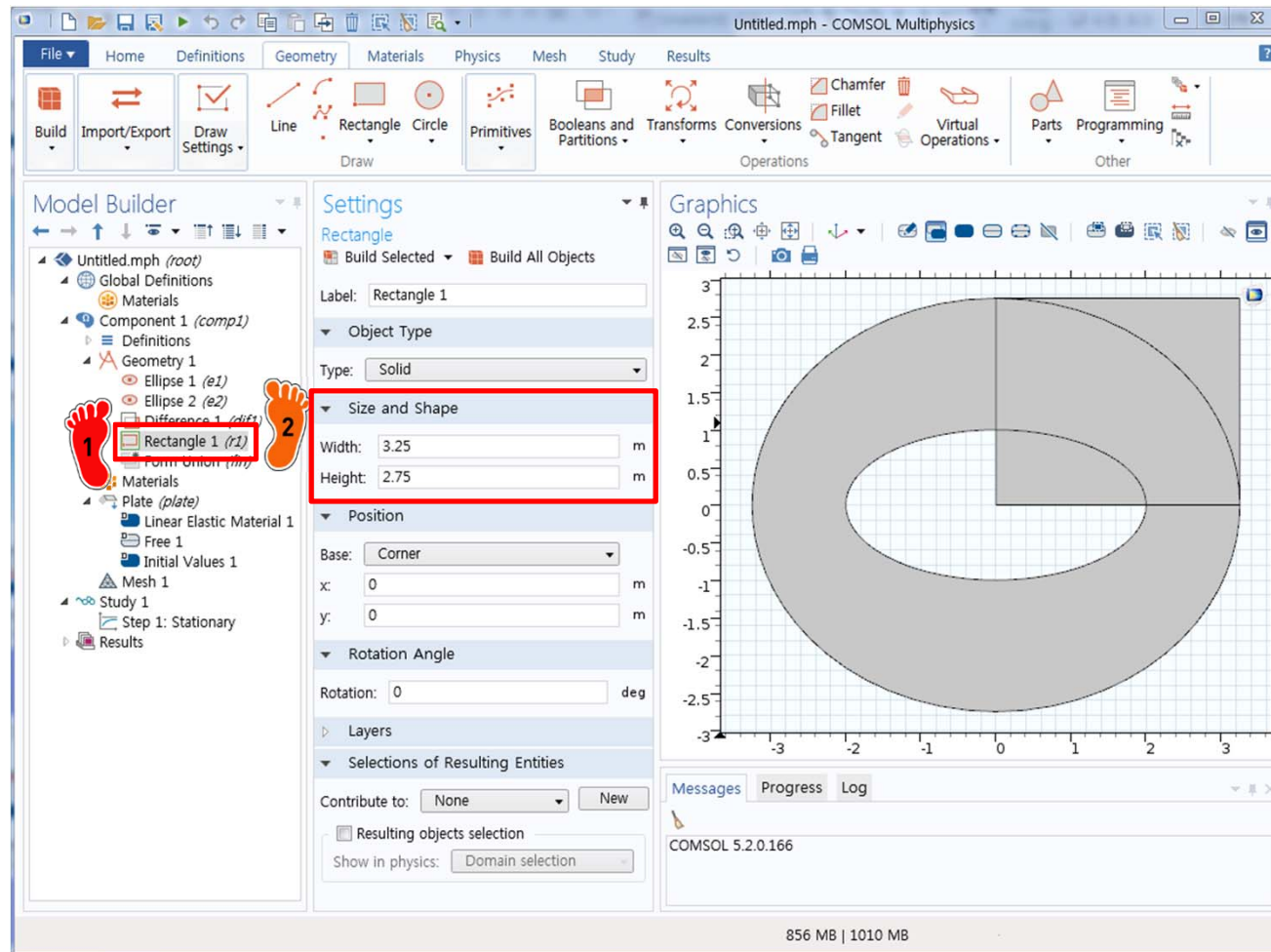
GEOMETRY CREATION



1 Objects to add에서
바깥 타원 선택
Objects to subtract에서
안쪽 타원 선택

2 타원 구멍 생성

GEOMETRY CREATION

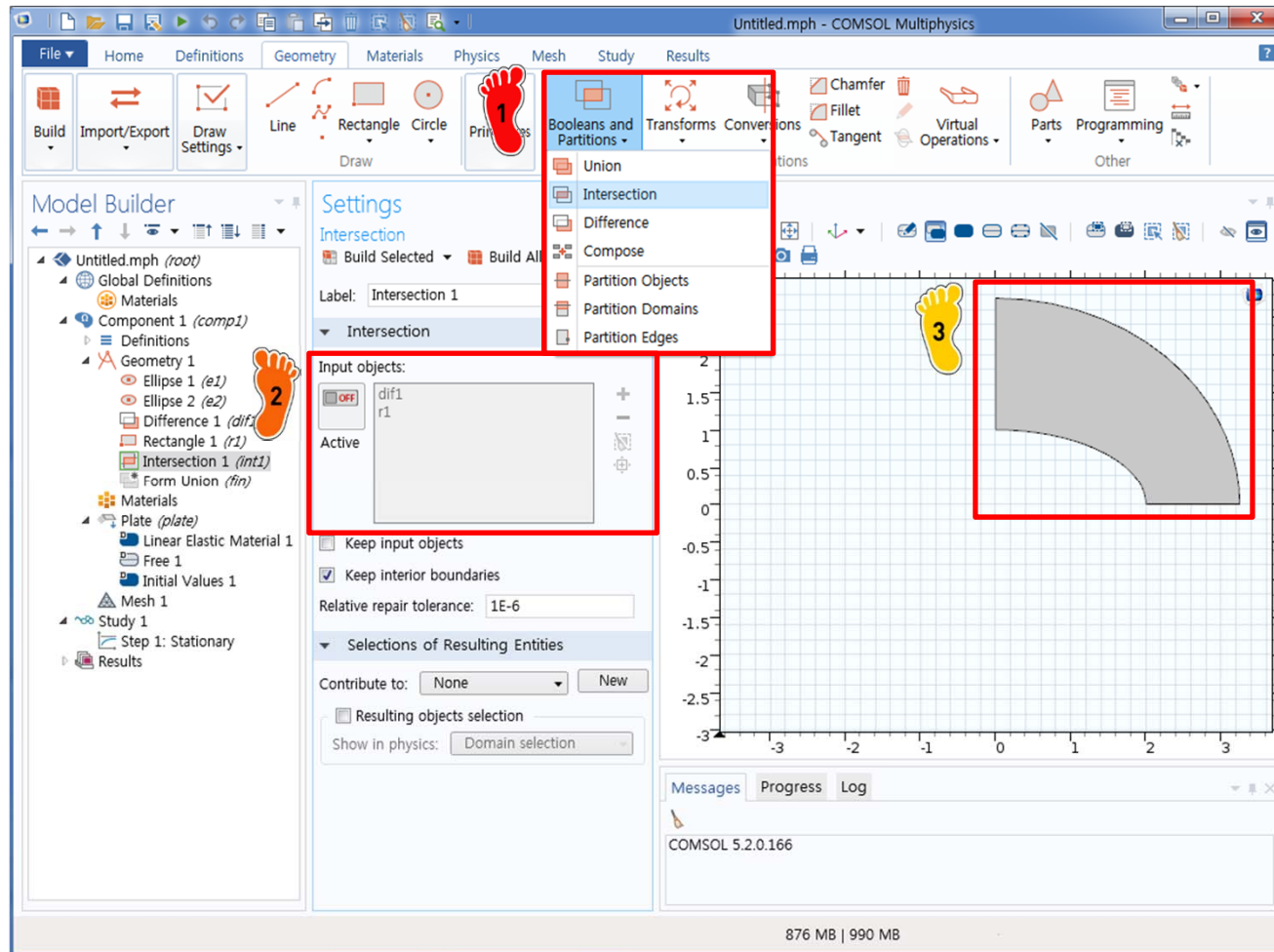


1 Rectangle 1 생성

2 Width: 3.25
Height: 2.75

입력 후 사각형 생성

GEOMETRY CREATION

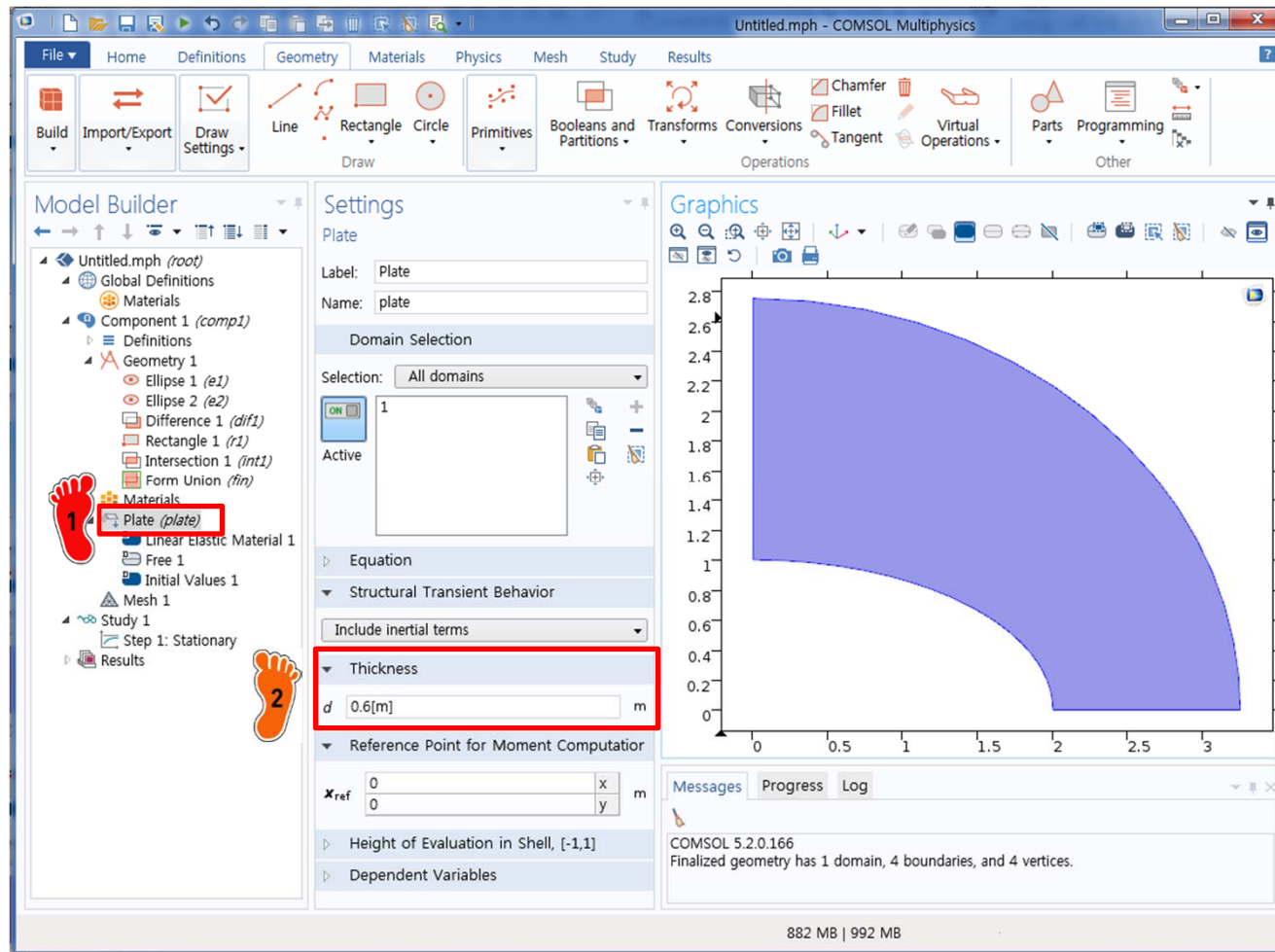


1 Booleans and Partitions에서 Intersection 선택

2 Input object에 두 형상 선택

3 다음과 같은 형상 생성

THICKNESS



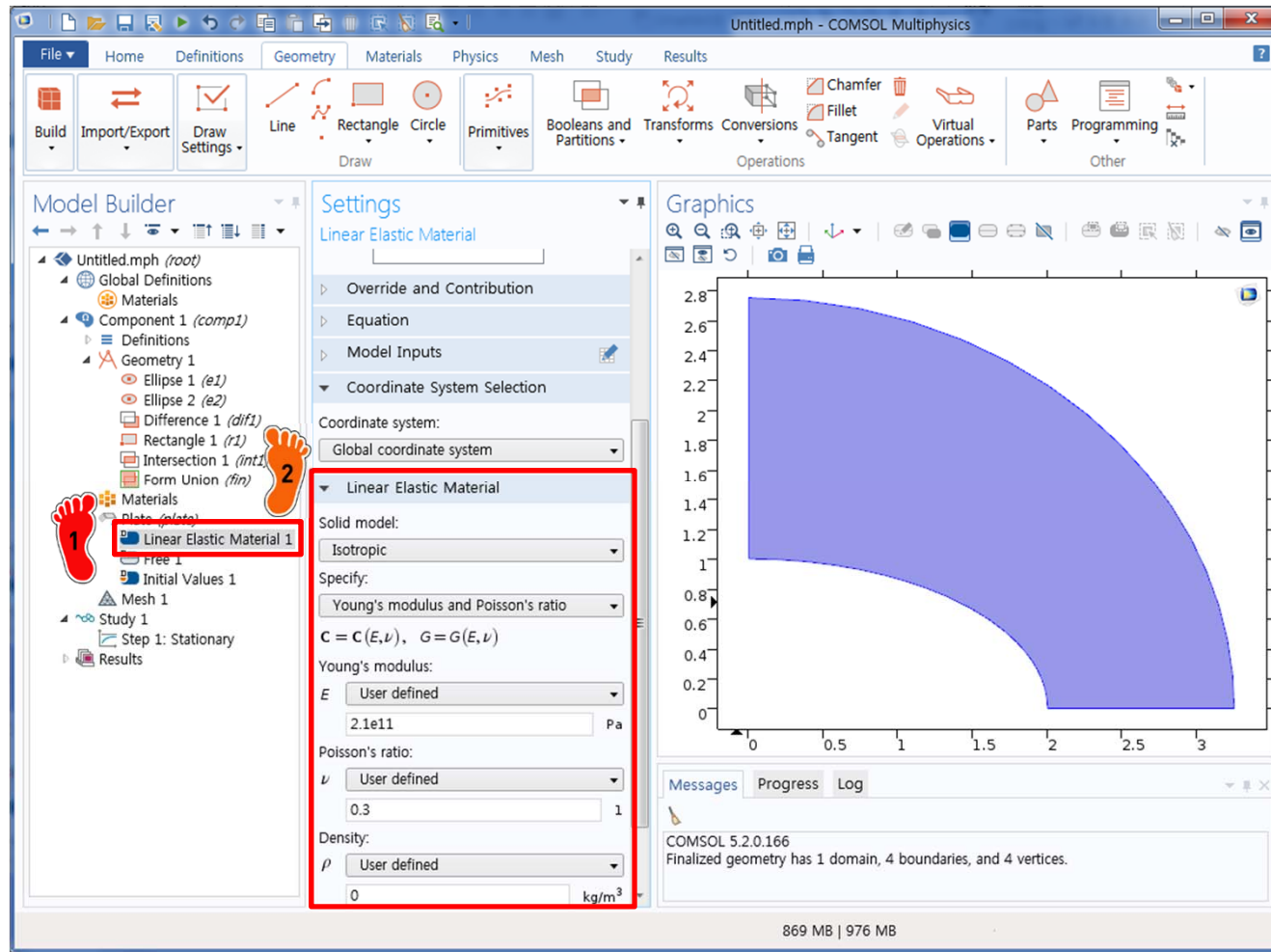
1 Plate 클릭

2 두께 d: 0.6

입력

요소 차수는
quadratic(default)

MATERIAL PROPERTY



1 Linear Elastic Material 1
클릭

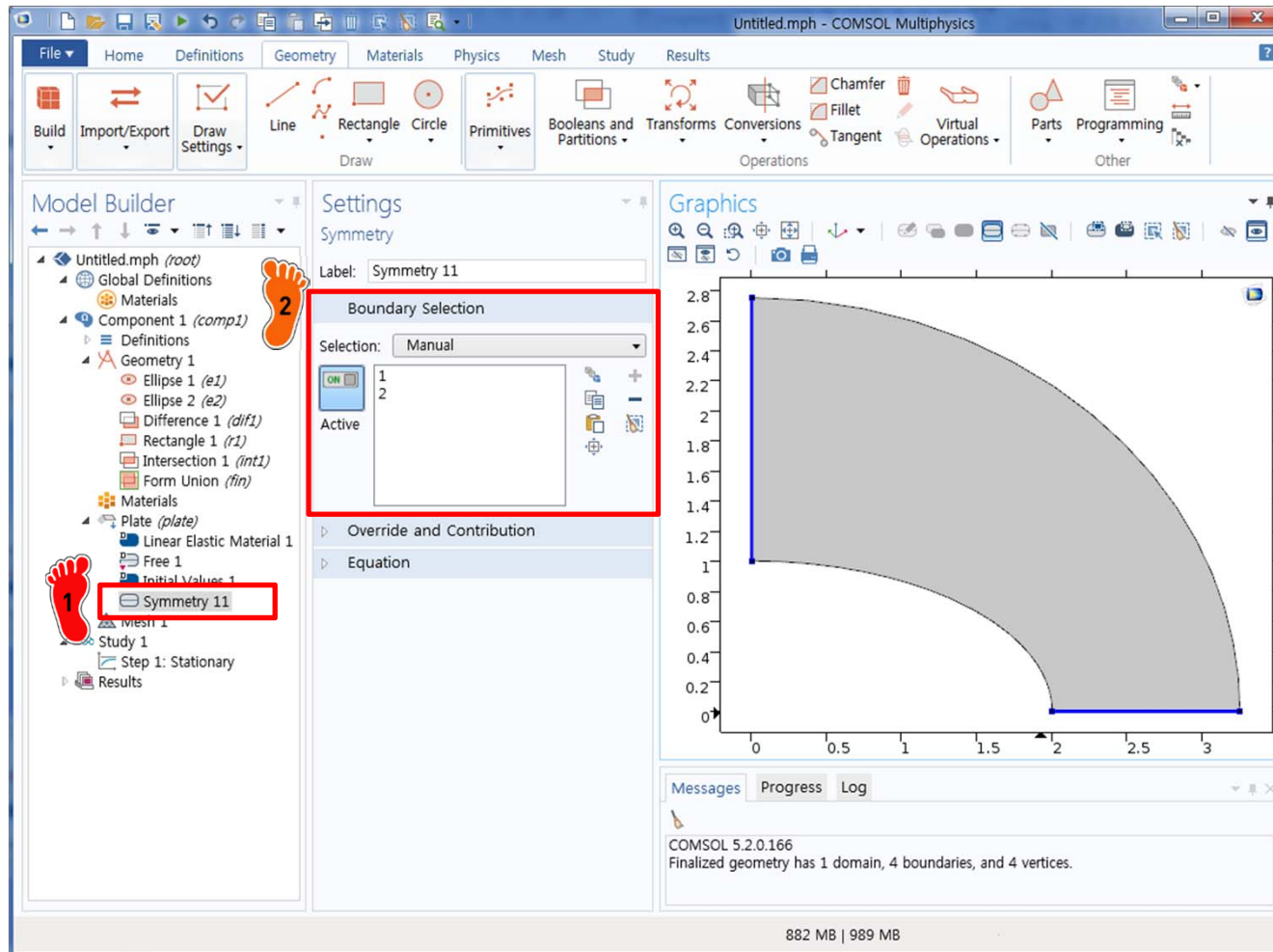
2 E: 2.1e11
mu: 0.3
rho : 0

입력

BOUNDARY CONDITION

1 Symmetry 경계조건 생성

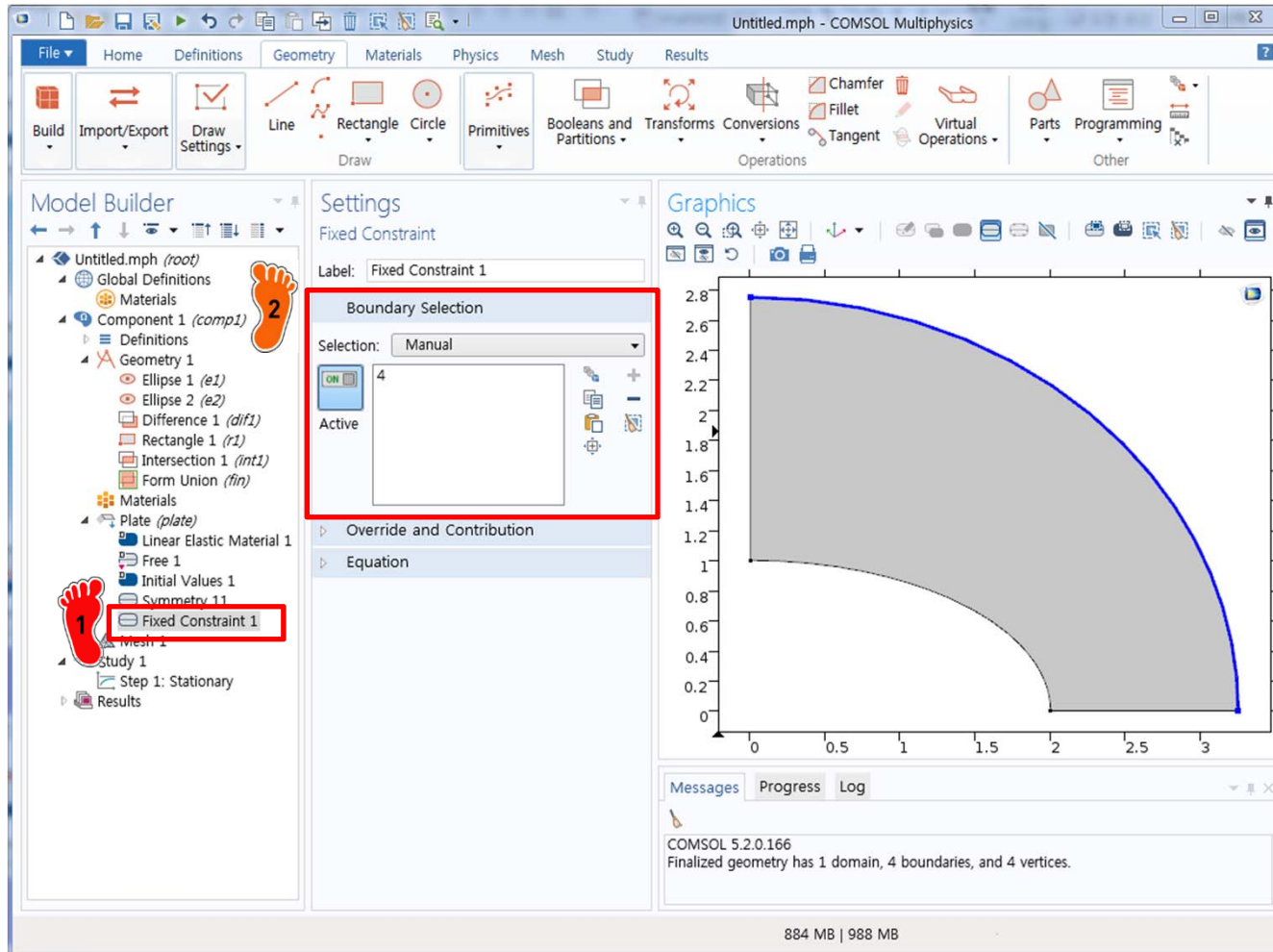
2 1, 2번 경계 선택



BOUNDARY CONDITION

1 Fixed 경계조건 생성

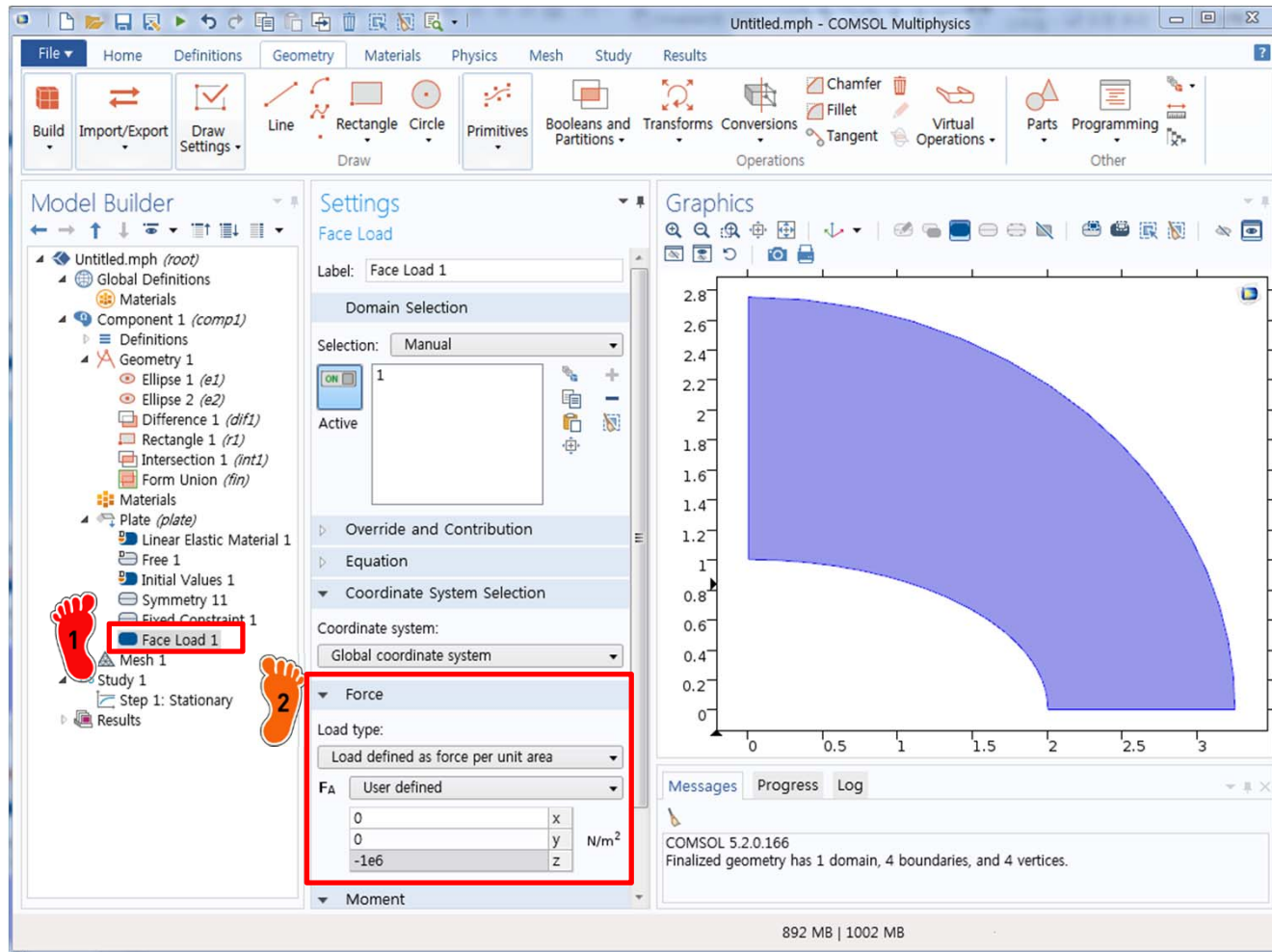
2 4번 경계 선택



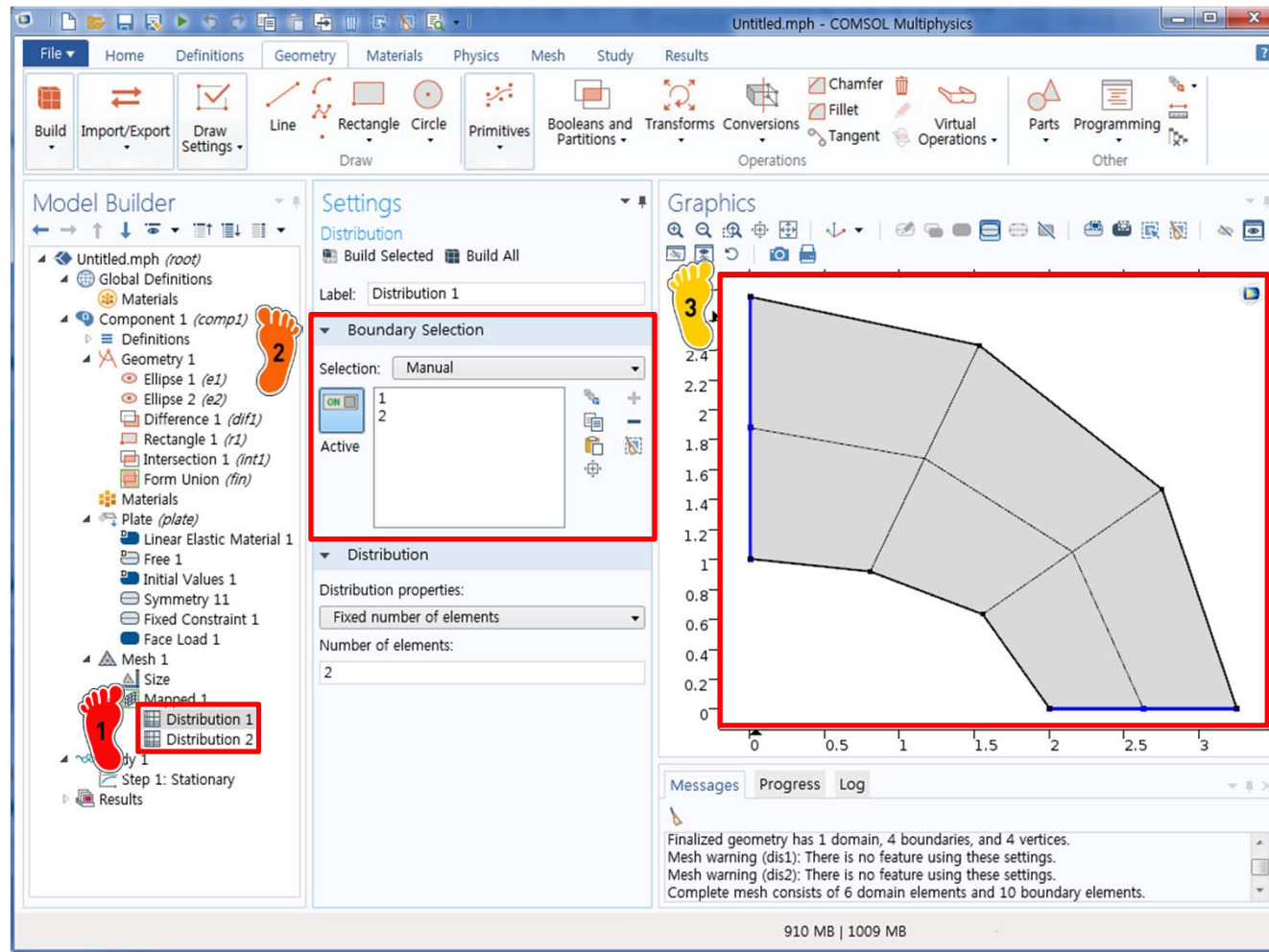
LOADING CONDITION

1 Face Load 생성

2 z 방향 $-1e6$ 입력



MESH



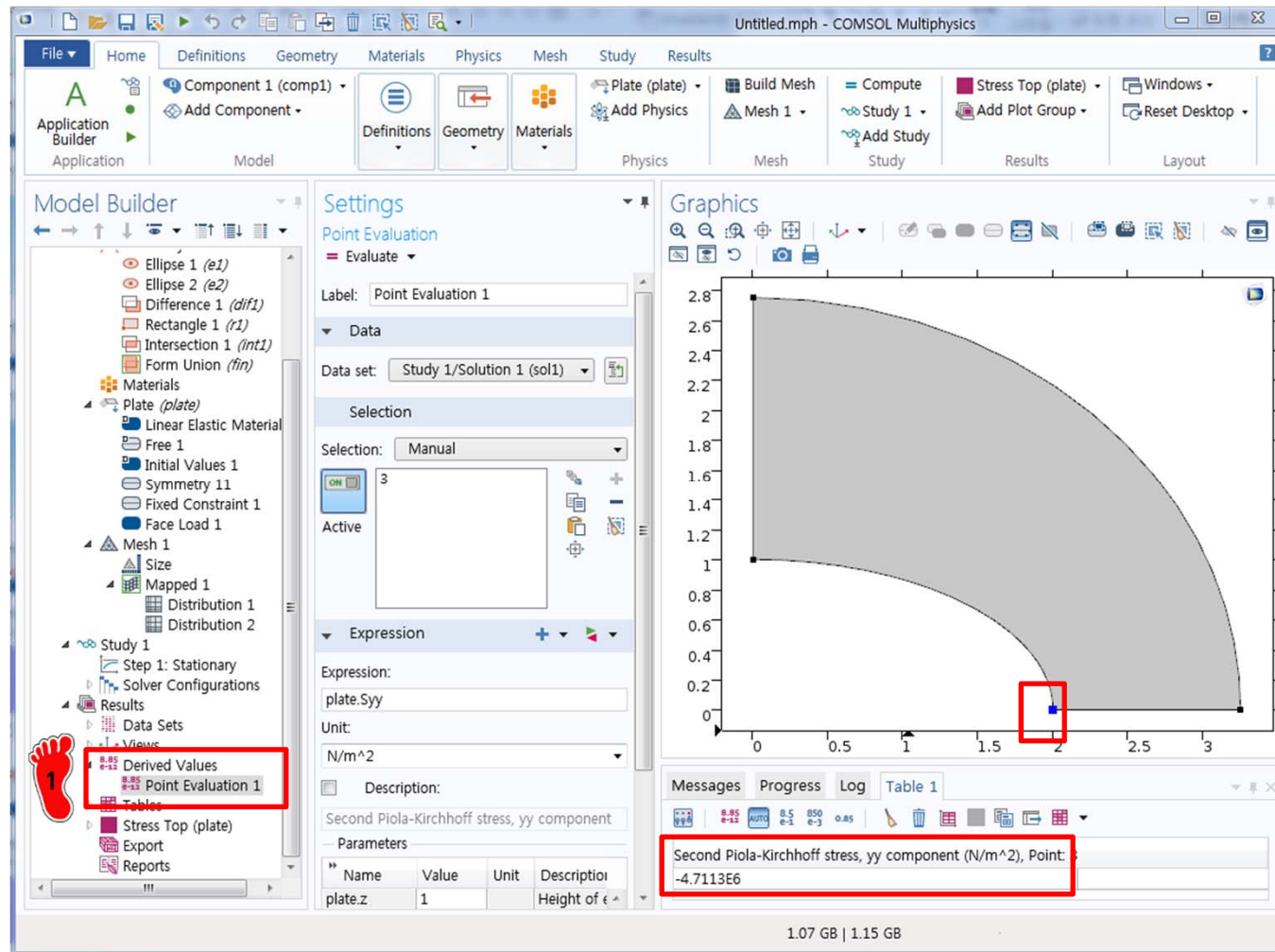
1 Mesh1에서 Mapped를 생성하고 Distribution1, 2 생성

2 Distribution1에서 1, 2번 경계 선택
Number of elements에 2 입력

Distribution2에서 3, 4번 경계 선택
Number of elements에 3 입력

다음과 같은 격자 생성
그 후, 해석실행

POST-PROCESSING



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

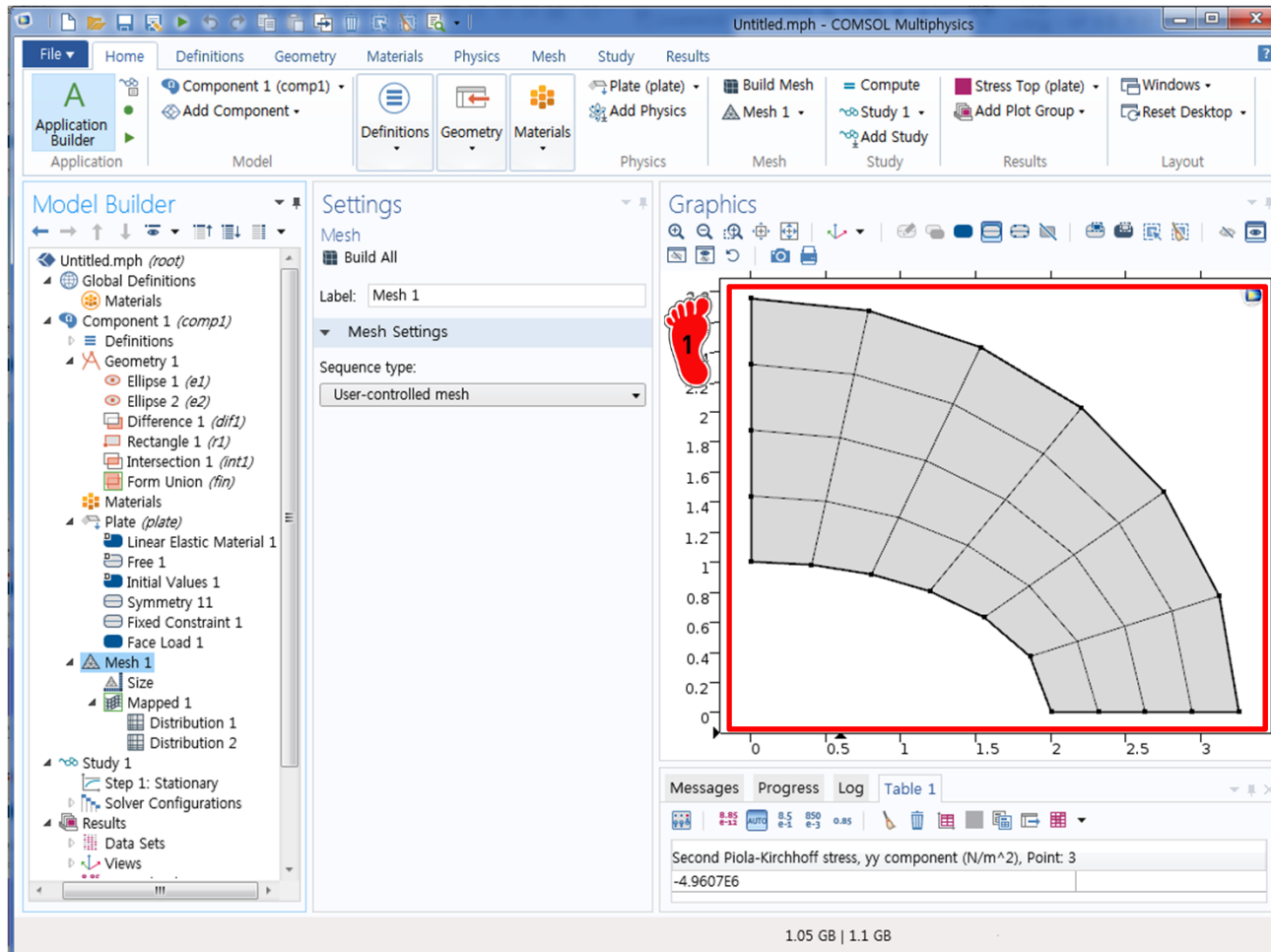
plate.Syy: Second Piola-Kirchhoff stress, yy component

결과값: -4.7113 MPa

MESH

1 Fine 메시로 변경

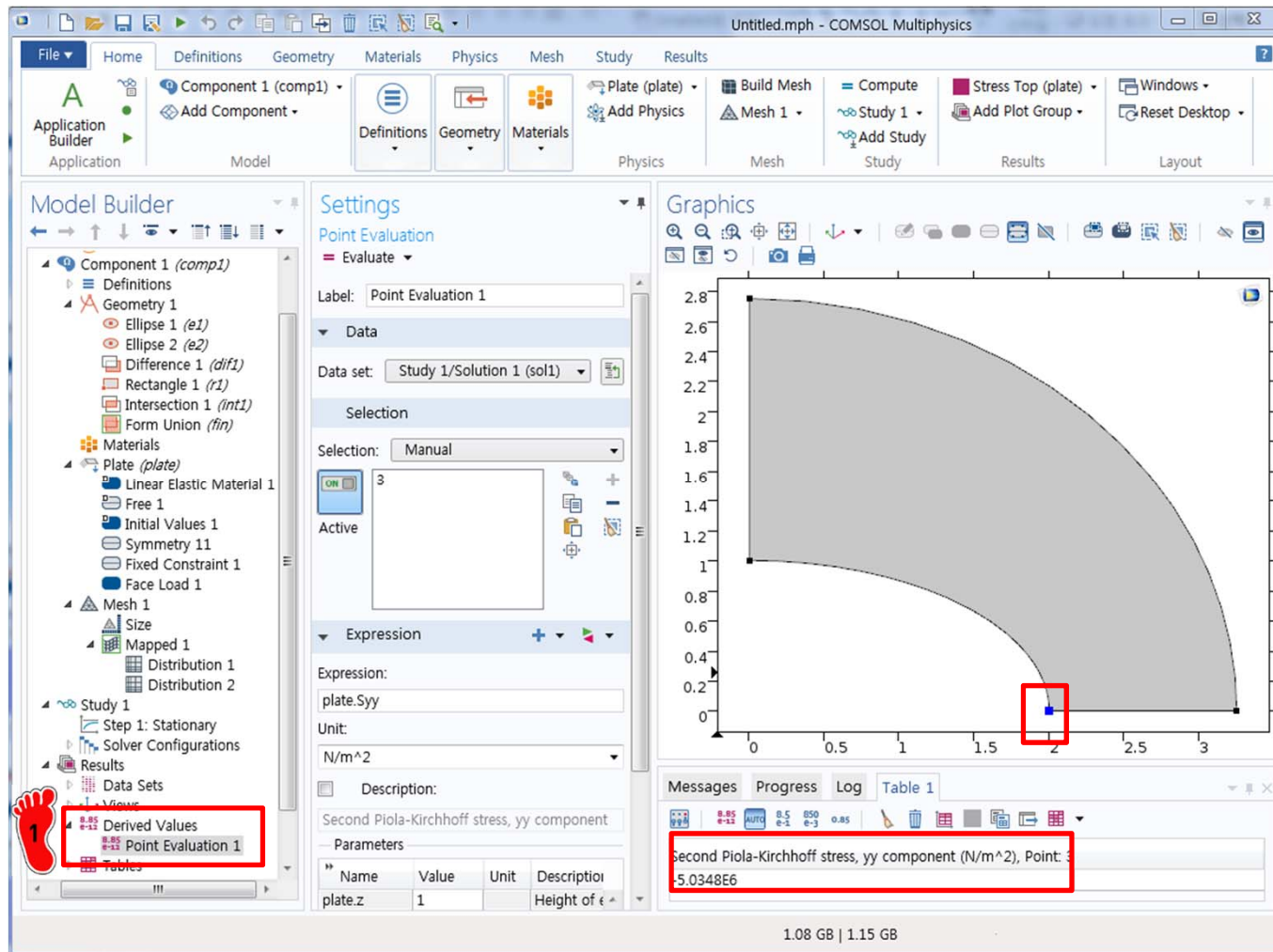
그 후, 해석실행



POST-PROCESSING

1 Point Evaluation 1 계산

결과값: -5.0348 MPa



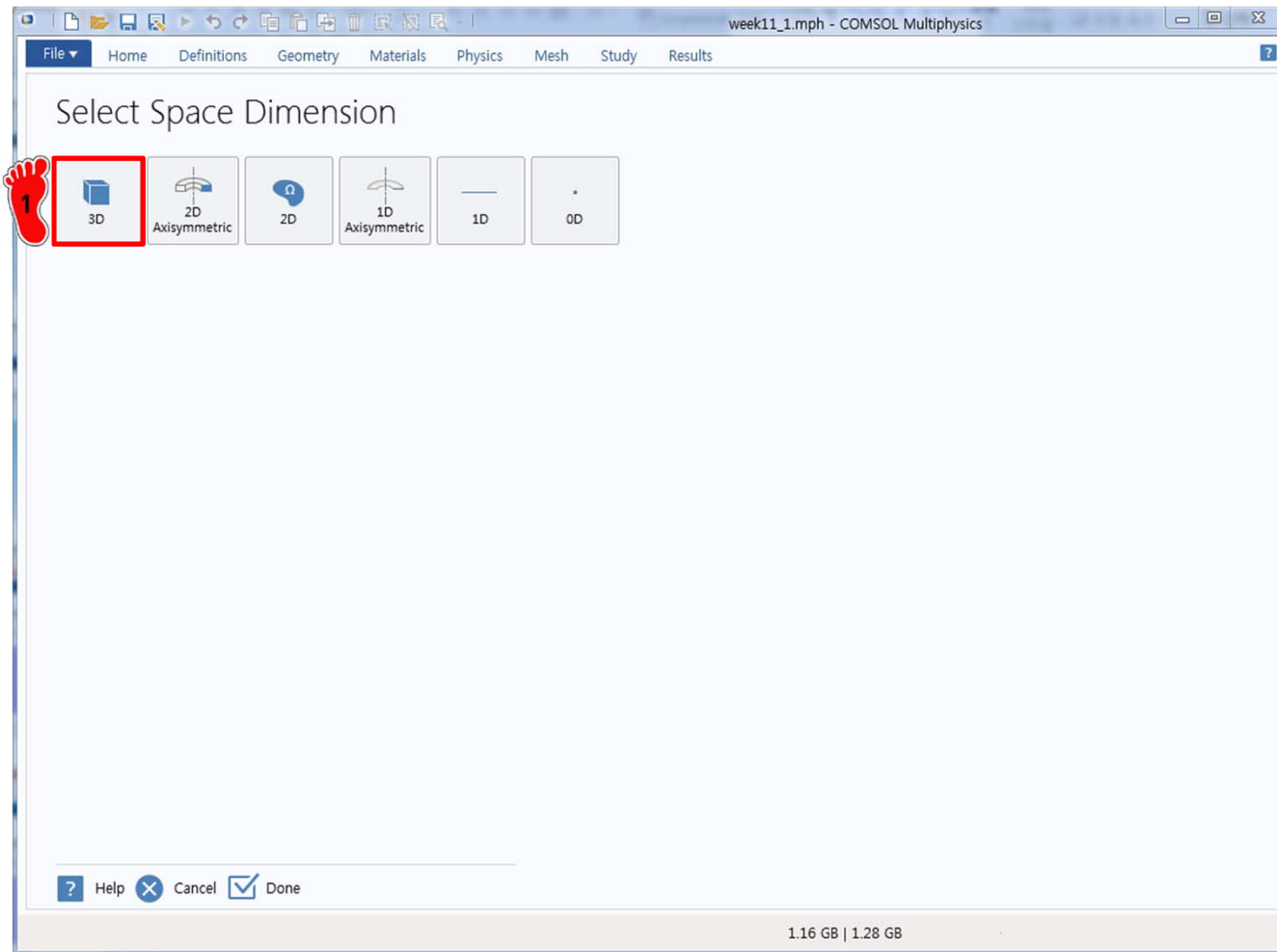
SUMMARY

LE 10	order	integration points	element	coarse mesh		fine mesh	
				s22	error	s22	error
<i>Abaqus</i> *	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				
<i>Abaqus</i> *	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				

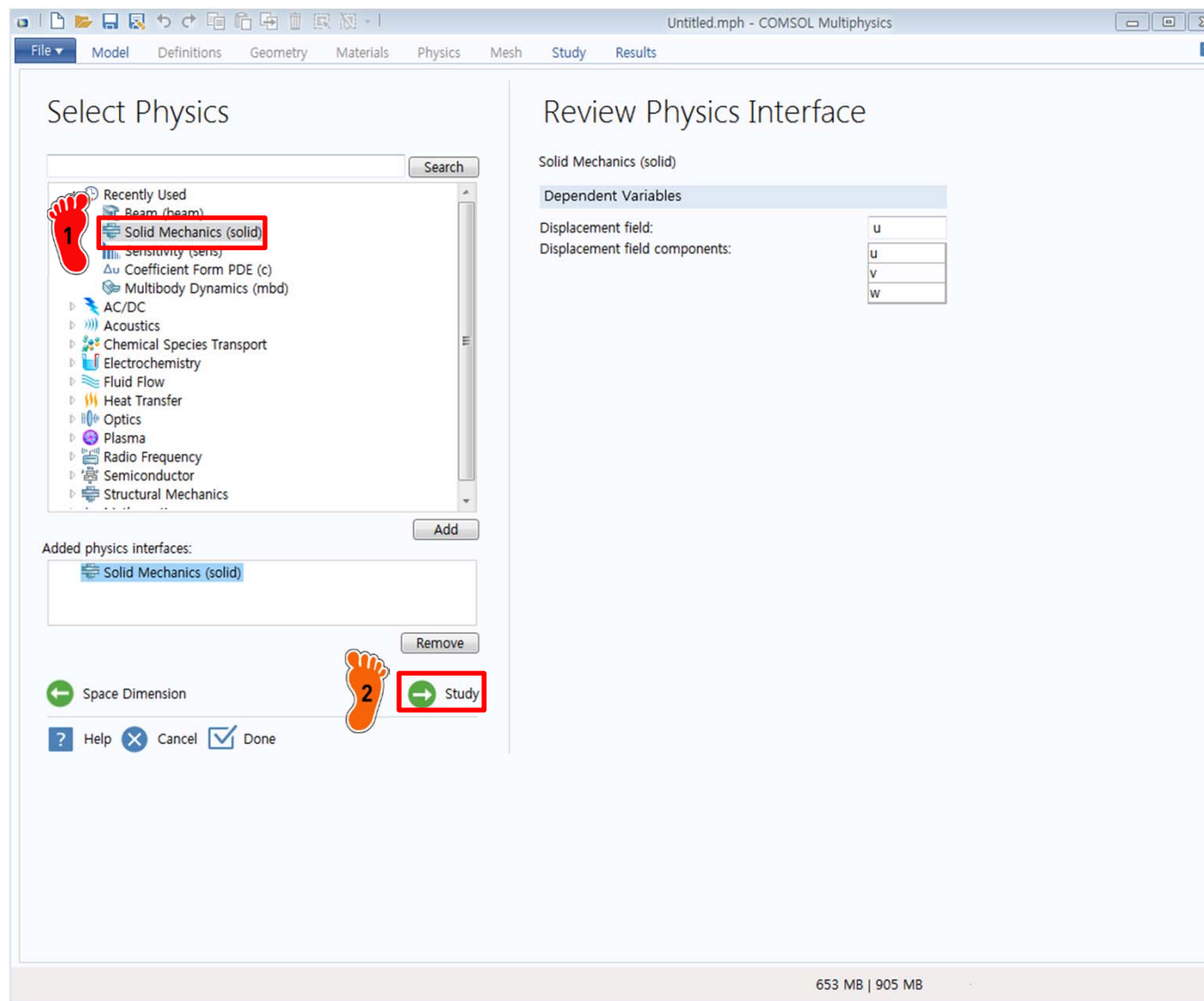
COMSOL	2		plate	-4.71E+06	12.45%	-5.03E+06	6.51%
COMSOL	2		hexahedron				

DIMENSION SELECTION

1 3D 선택



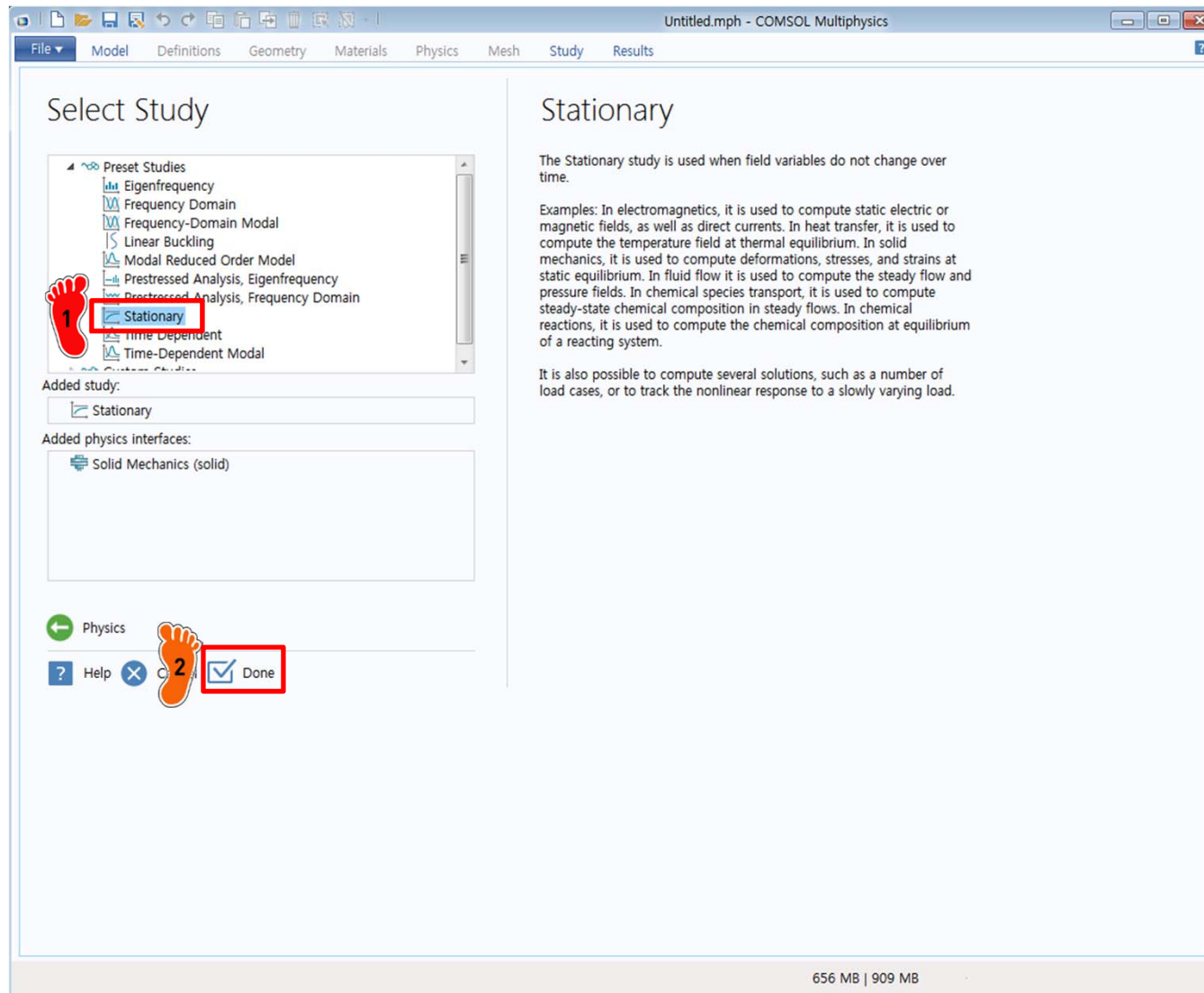
PHYSICS SELECTION



1 Structural Mechanics 의
Solid Mechanics 선택

2 Study 클릭

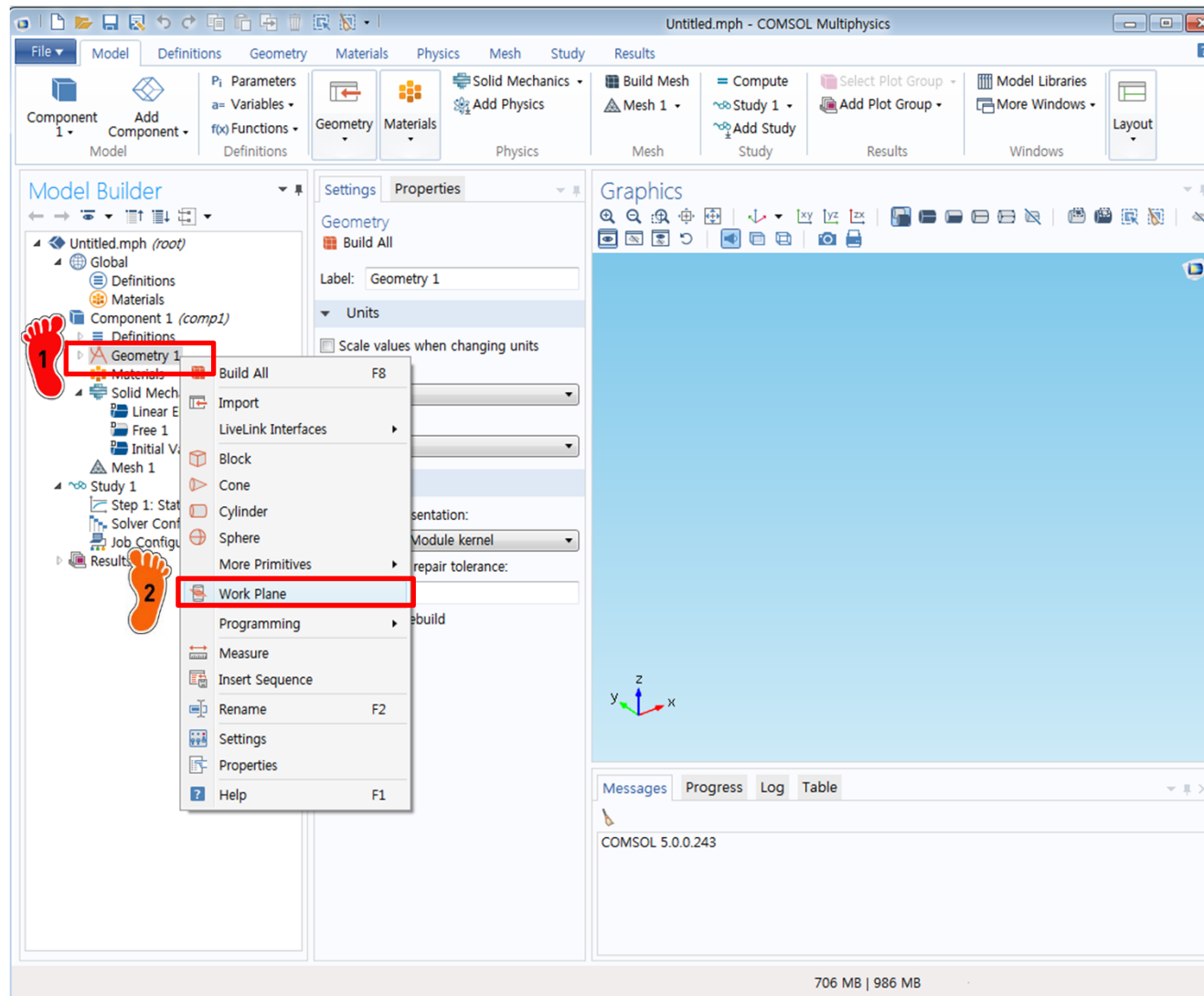
STUDY TYPE SELECTION



1 Stationary 선택

2 Done 클릭

GEOMETRY CREATION

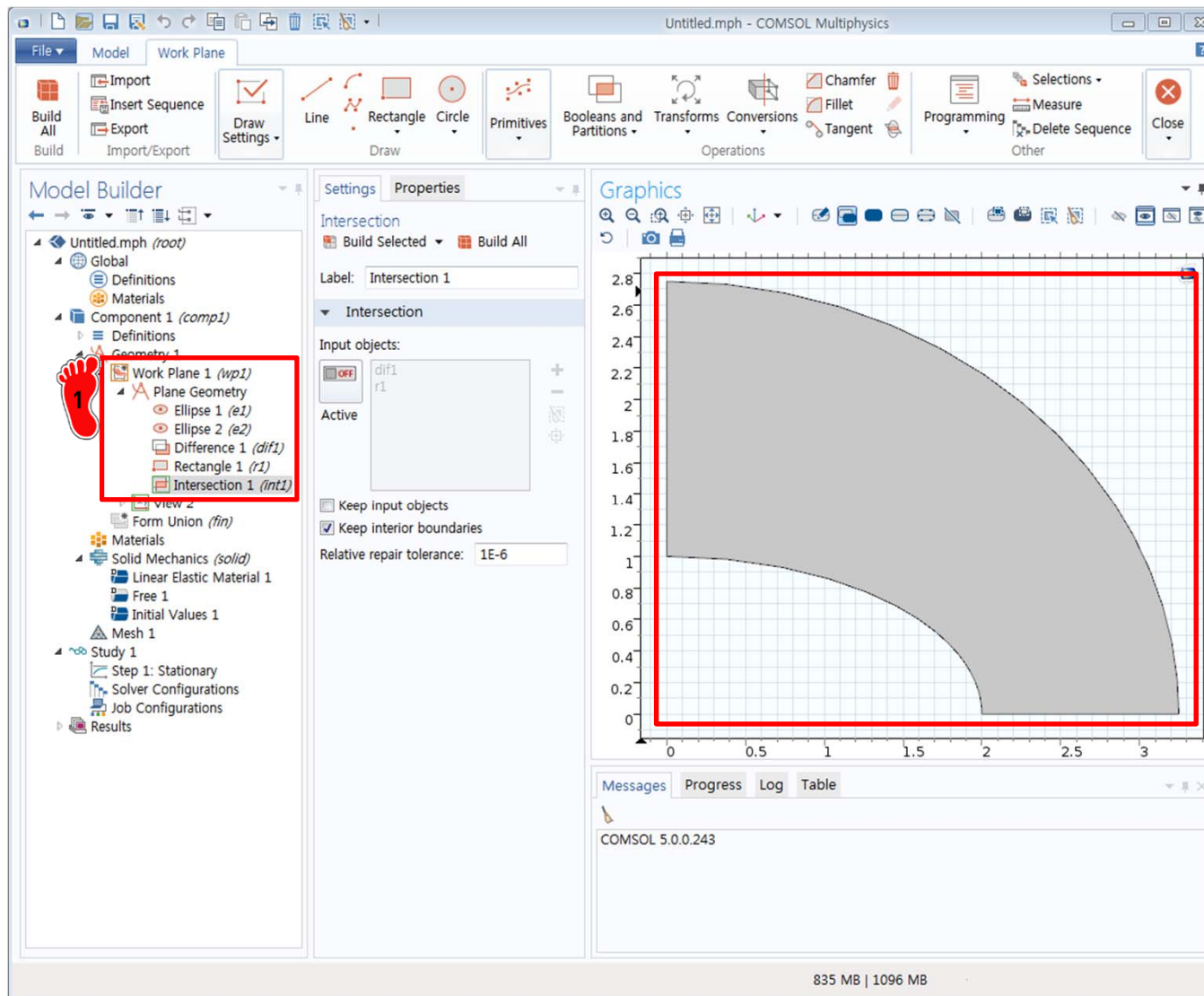


1 Geometry 1 마우스 우클릭

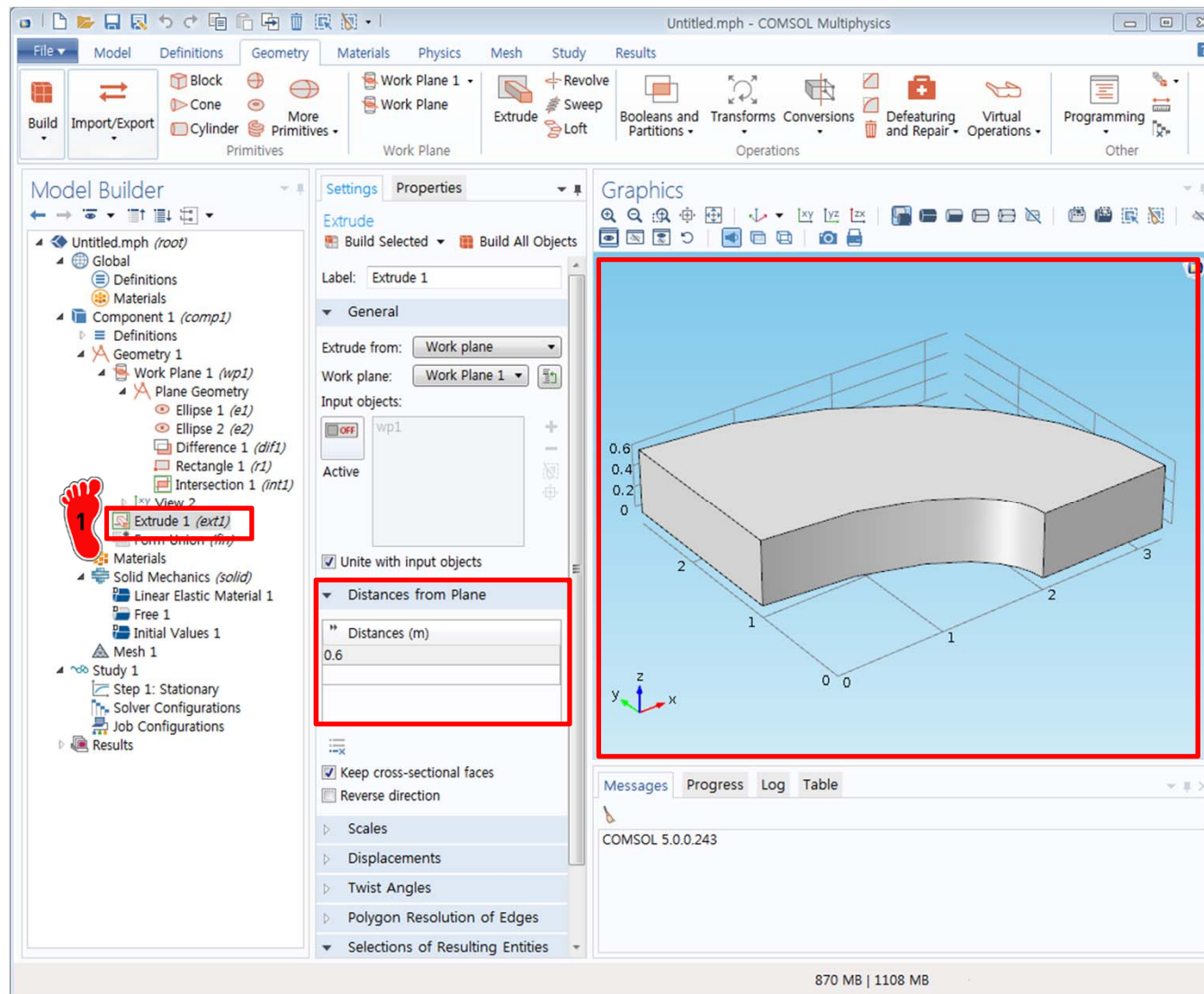
2 Work Plane 클릭

GEOMETRY CREATION

1 Plane Geometry 에 2차원
형상 생성

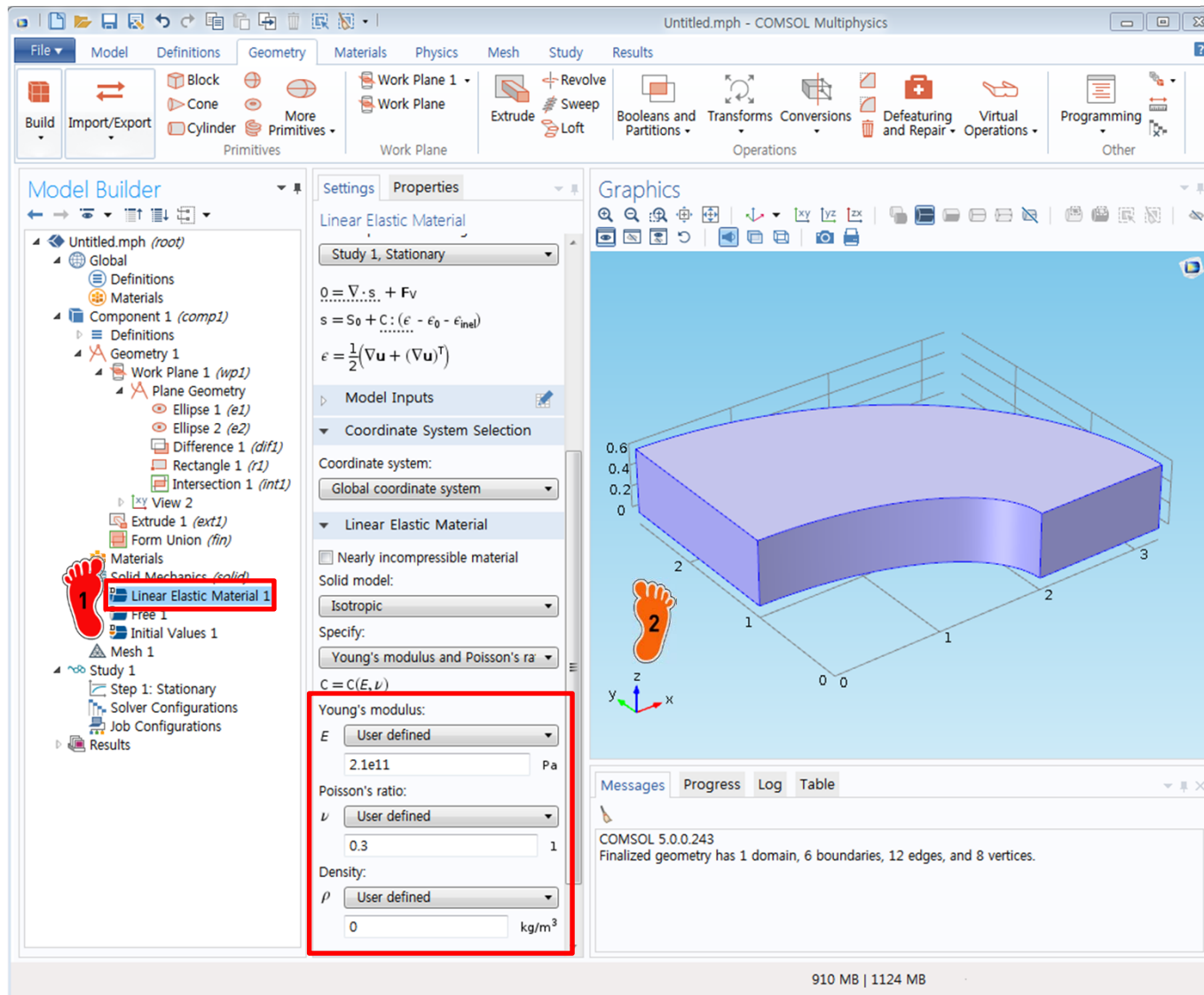


GEOMETRY CREATION

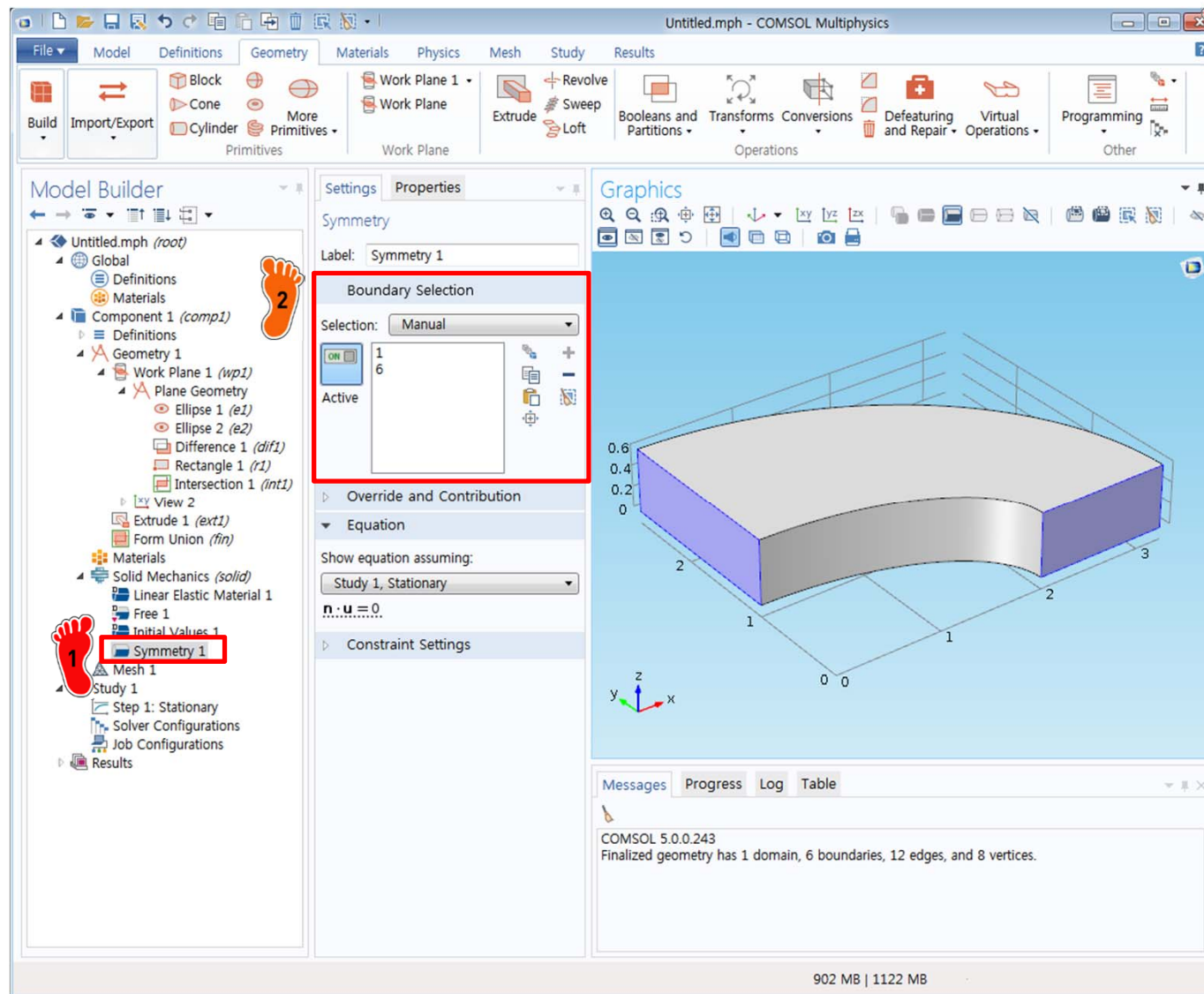


Extrude 메뉴를 이용하여 z 방향으로 0.6 m 두께를 가지는 형상 생성

MATERIAL PROPERTY



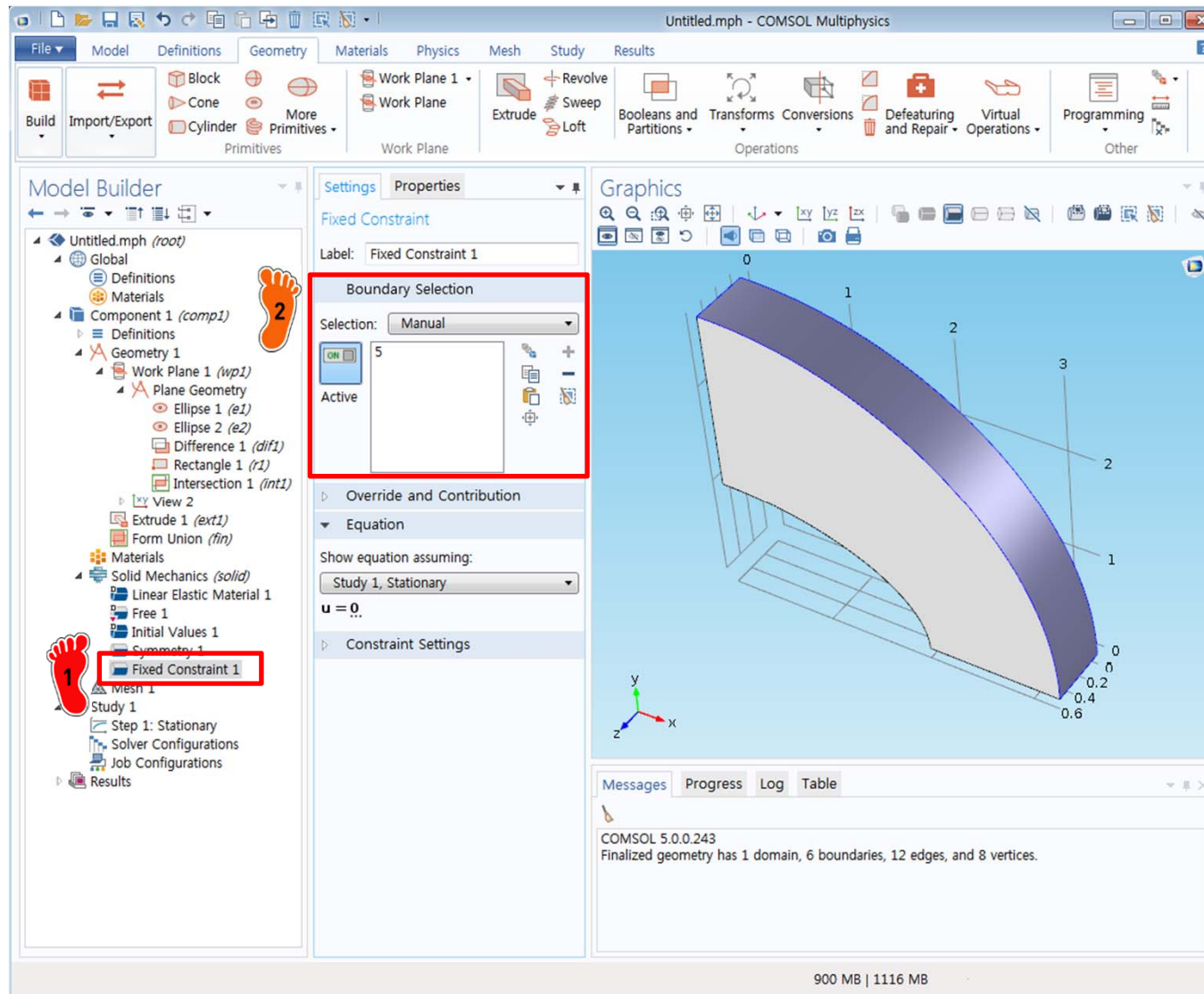
BOUNDARY CONDITION



1 Symmetry 경계조건 생성

2 1, 6번 경계 선택

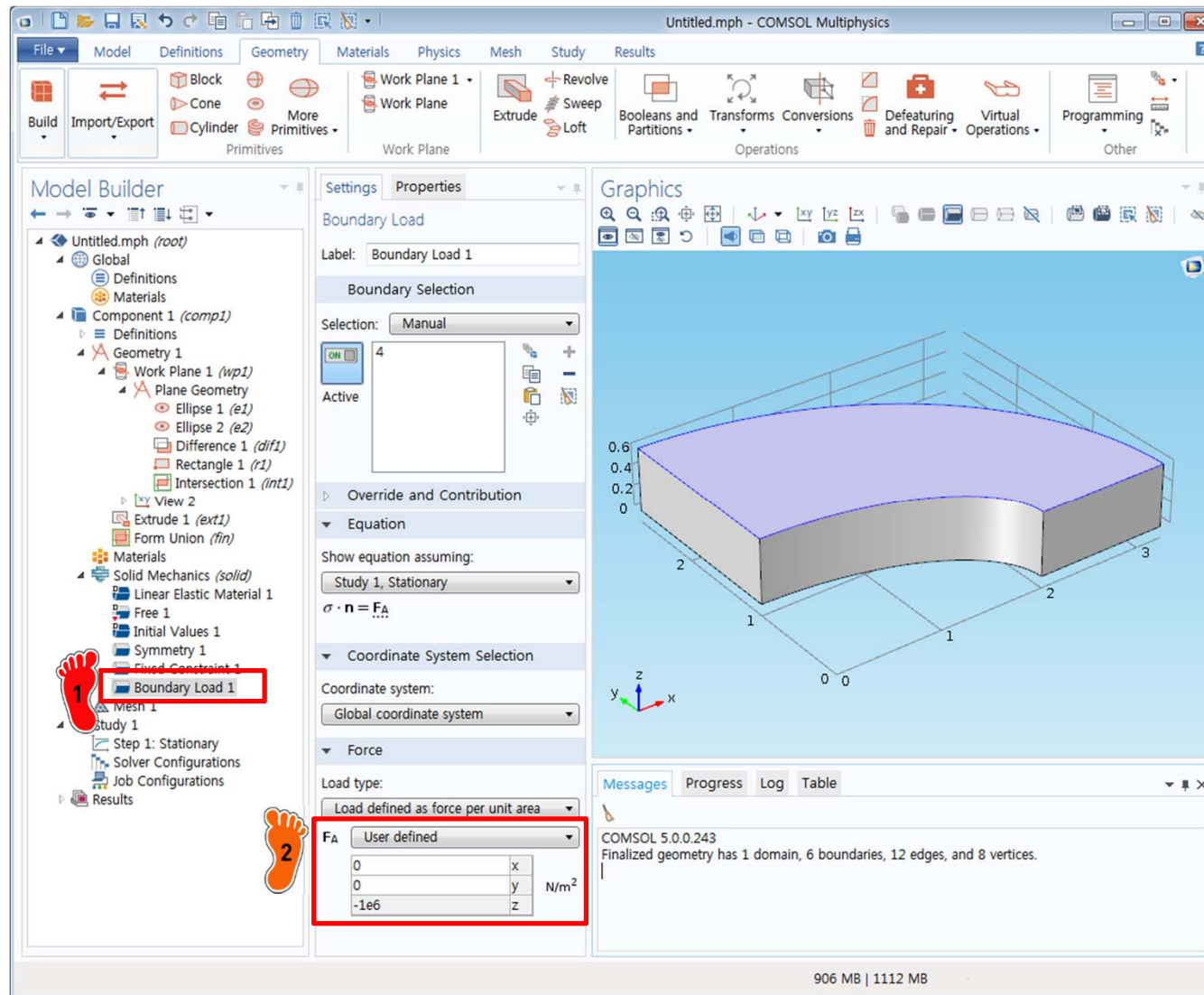
BOUNDARY CONDITION



1 Fixed 경계조건 생성

2 5번 경계 선택

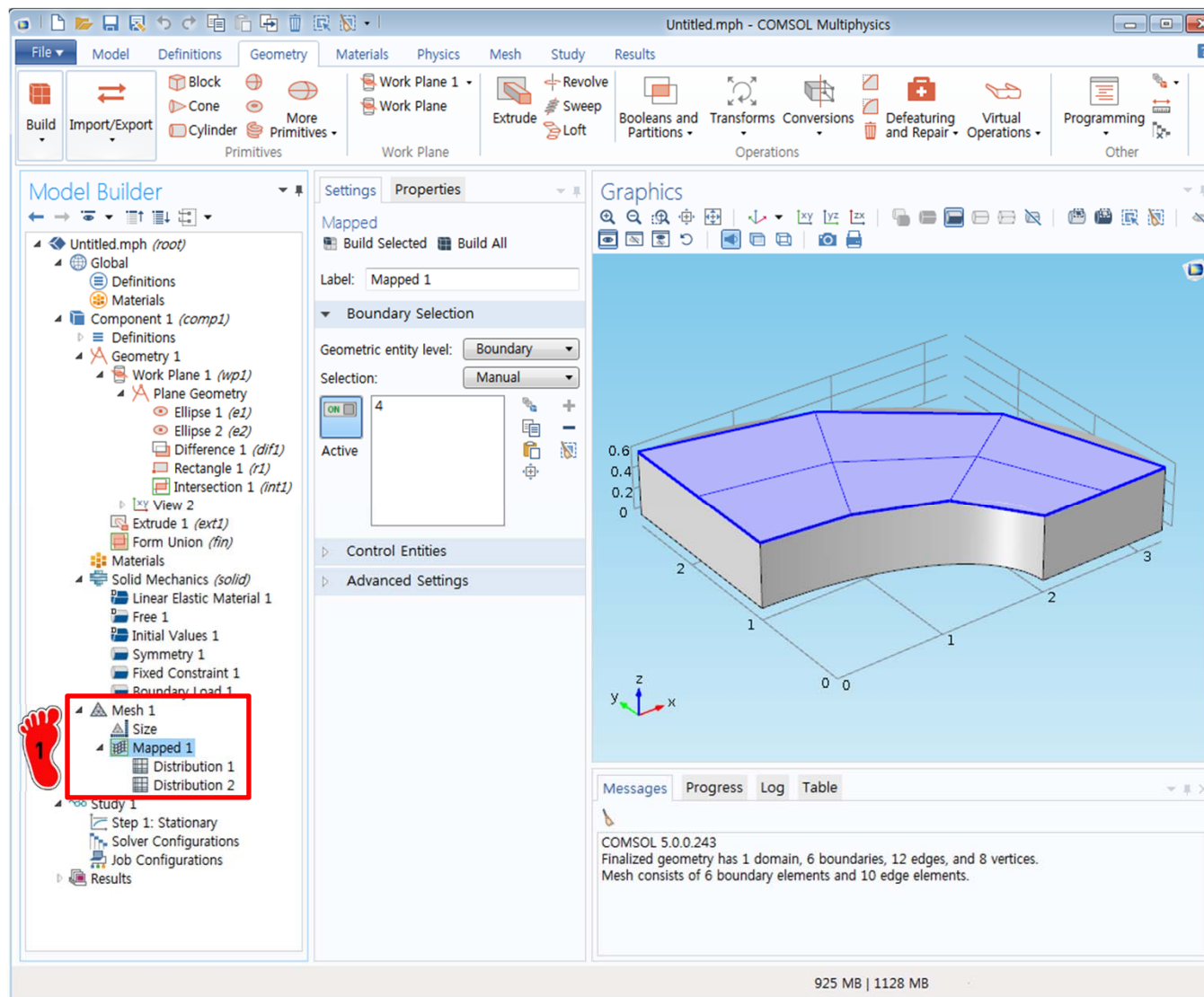
LOADING CONDITION



1 Boundary Load 생성

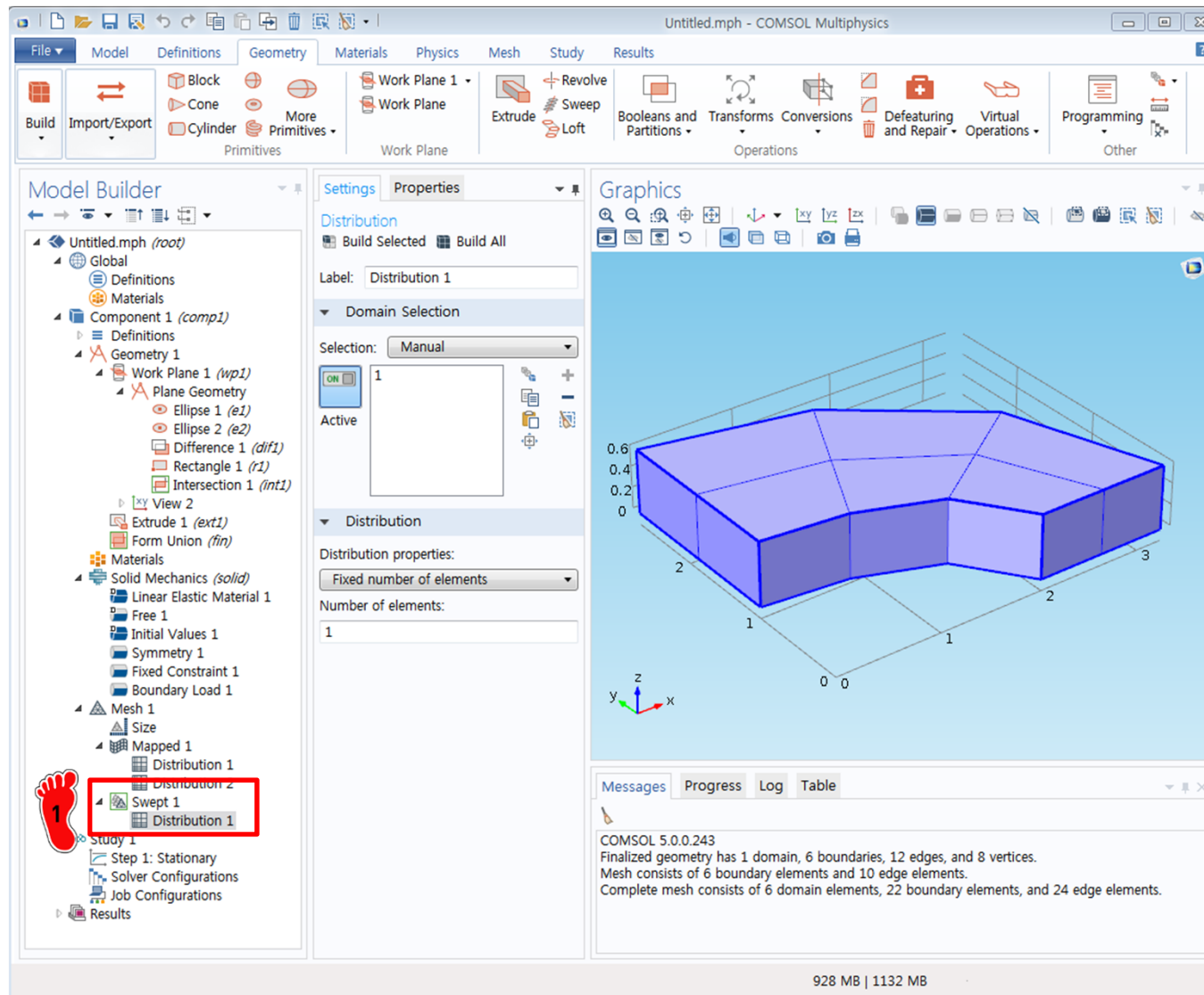
2 z 방향 -1e6 입력

MESH



1 Mapped mesh의
Distribution 기능을 이용하여 4번 평면에 요소 생성

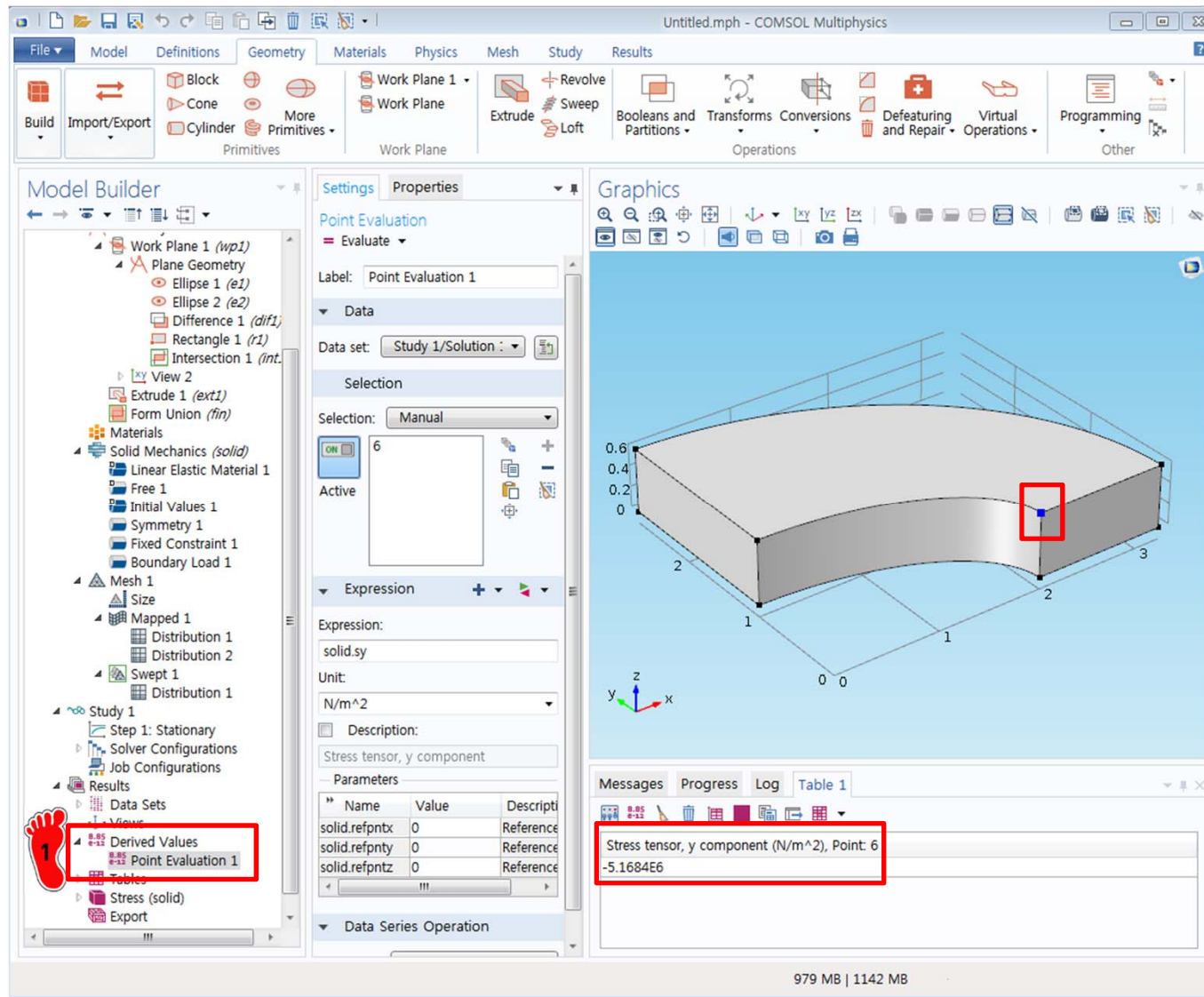
MESH



1 Swept 메뉴를 이용하여 전체 영역에 한 개의 layer 를 가지는 요소 생성

그 후 해석 실행

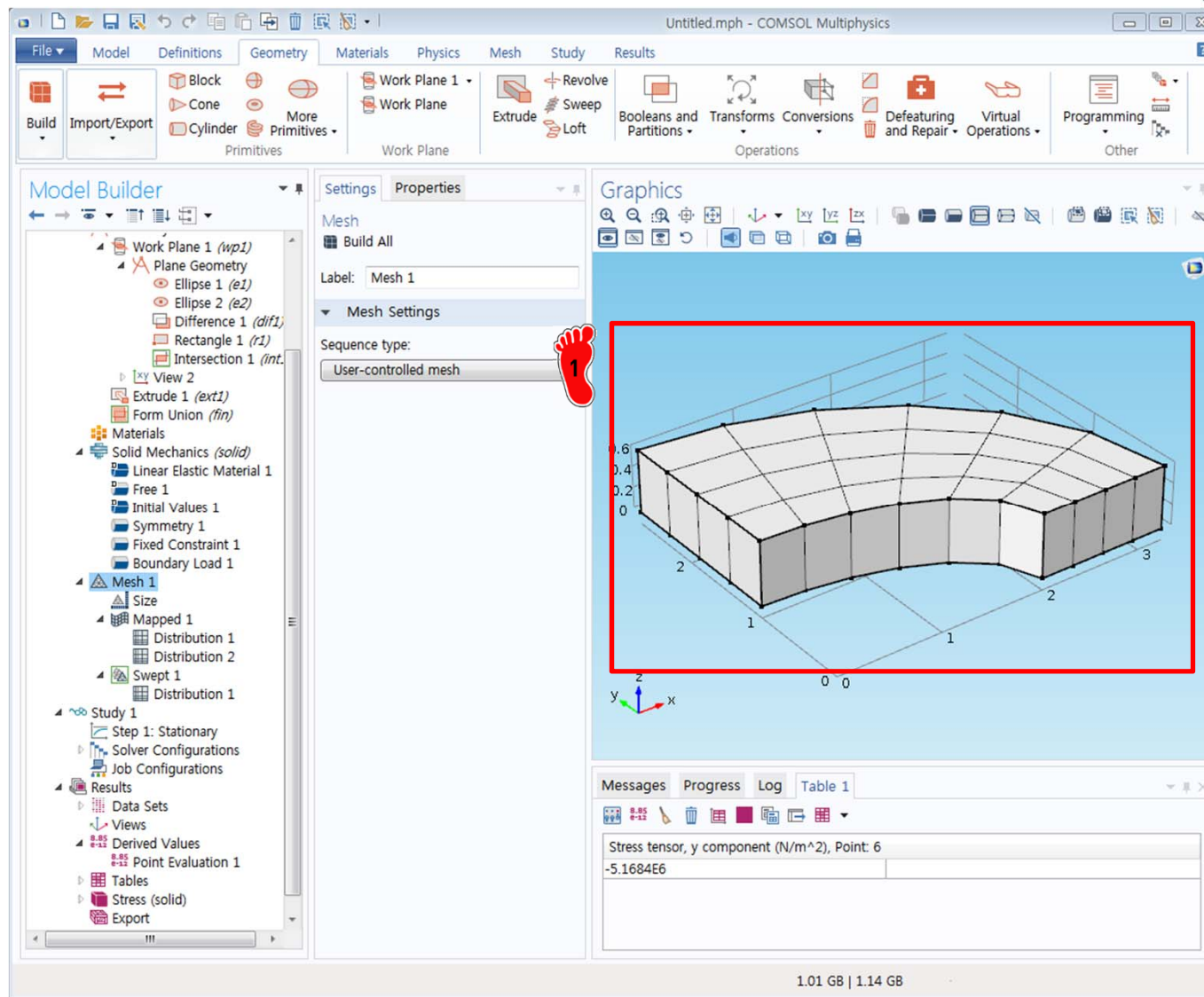
POST-PROCESSING



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

결과값: -5.1684 MPa

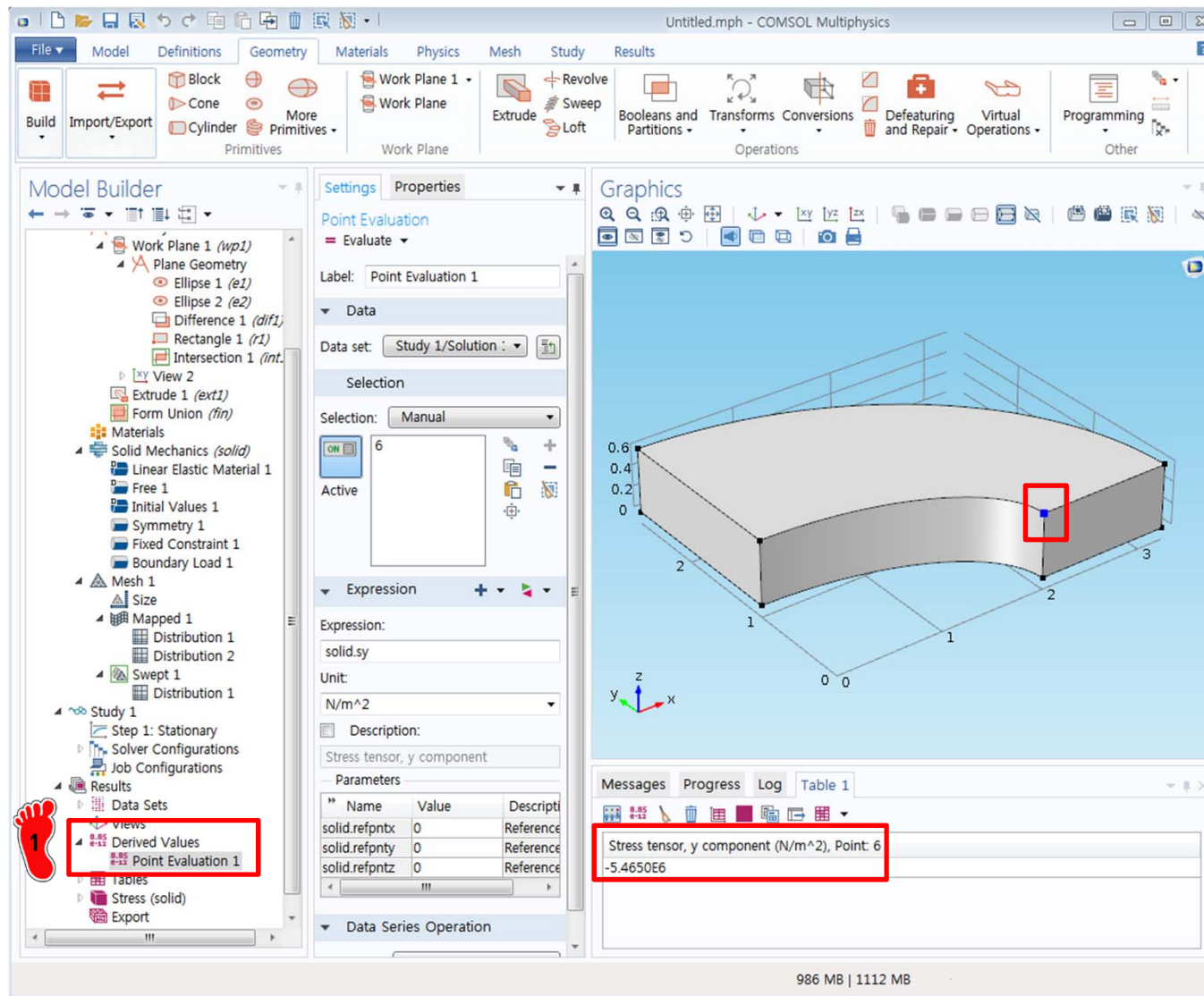
MESH



1 Fine 메시로 변경

그 후, 해석실행

POST-PROCESSING



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

결과값: -5.4650 MPa

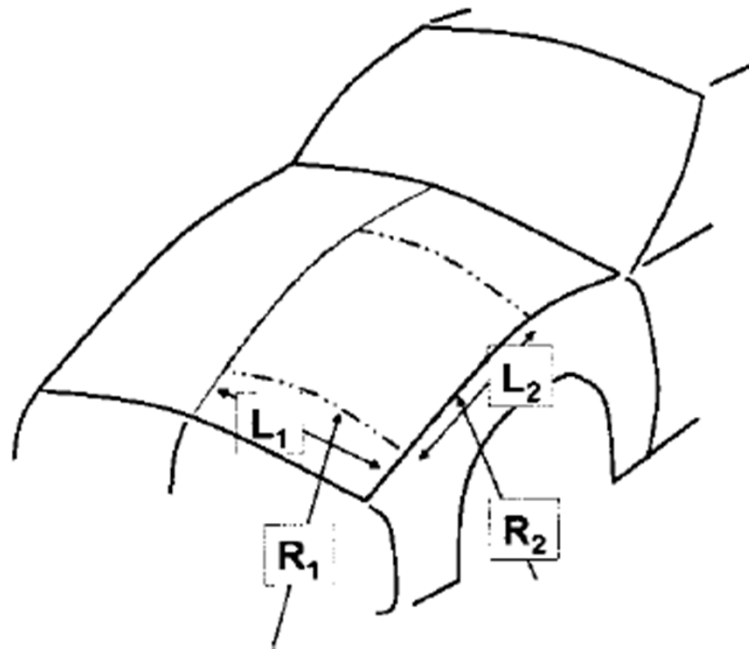
SUMMARY

LE 10	order	integration points	element	coarse mesh		fine mesh	
				s22	error	s22	error
<i>Abaqus</i> *	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				
<i>Abaqus</i> *	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	-4.71E+06	12.45%	-5.03E+06	6.51%
COMSOL	2		hexahedron	-5.17E+06	3.90%	-5.47E+06	1.67%

HOOD PANEL ANALYSIS

- Simply supported boundary conditions
- Dynamic yield stress: $s_{yd} = 298 \text{ N/mm}^2$
- Panel stiffness: **load(1N)/deflection**
- Oil-can load: **buckling load**



$L_1 = 660 \text{ mm}$
 $L_2 = 762 \text{ mm}$
 $R_1 = 5715 \text{ mm}$
 $R_2 = 15750 \text{ mm}$
 $t = 0.71 \text{ mm}$

$$K = \frac{B}{R} \frac{Et^2}{\sqrt{1-\mu^2}} = 28.39 \text{ N/mm}$$

where

R Spherical radius

B Constant (2.309)

E Young's Modulus

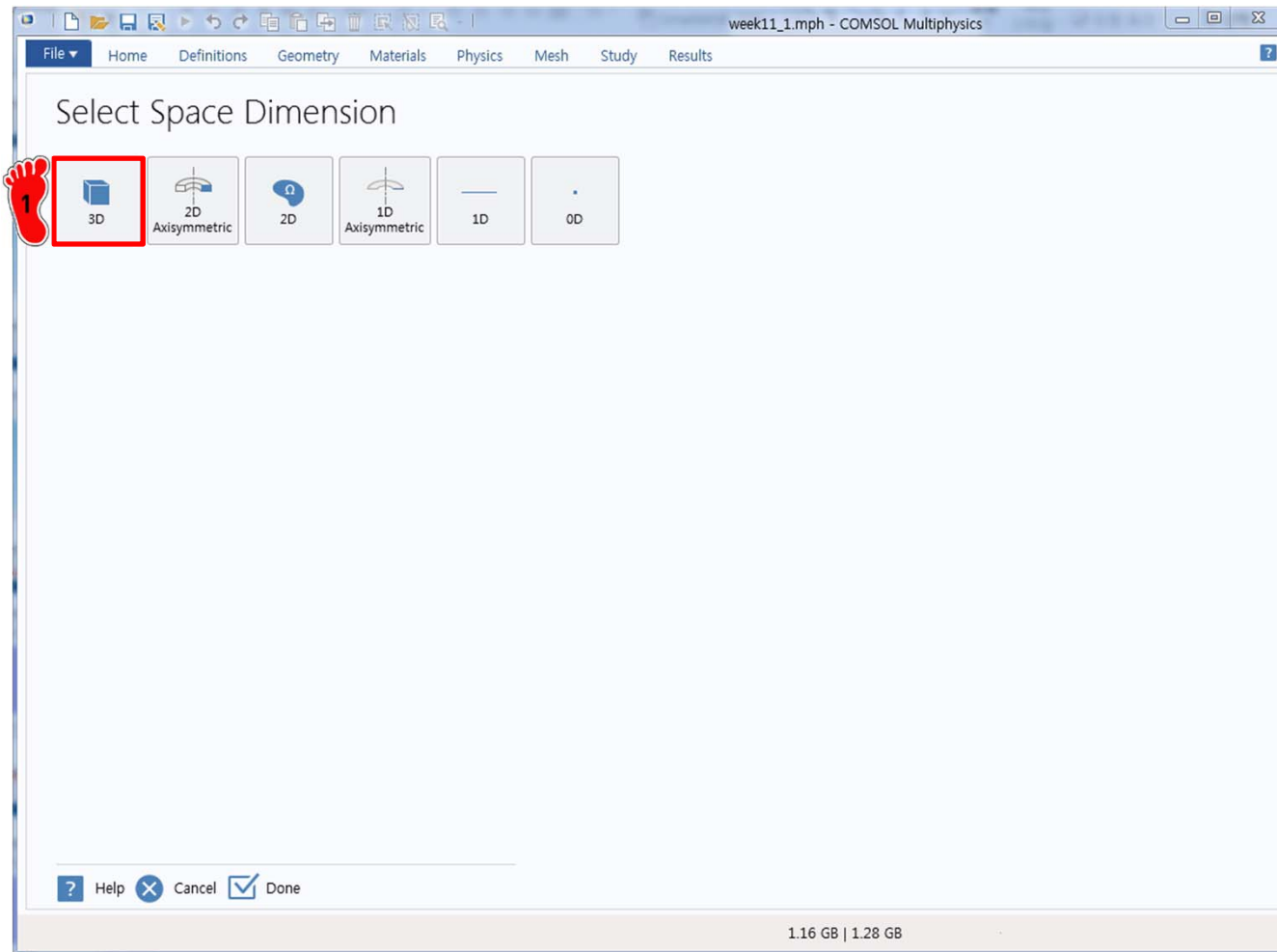
μ Poisson's ratio

t Panel thickness

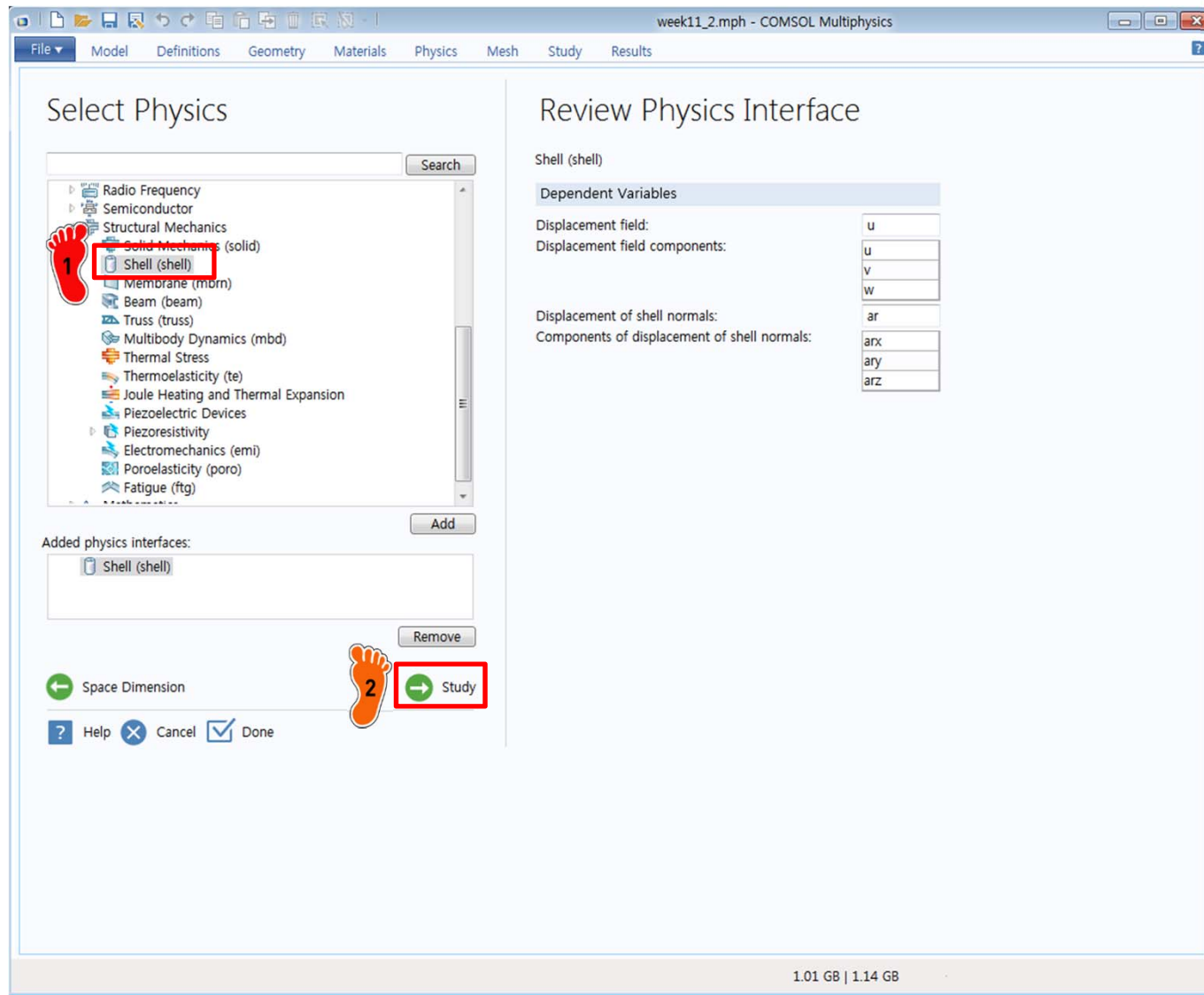
$$P_{CR} = \frac{CR_{CR} \pi^2 Et^4}{L_1 L_2 (1 - \mu^2)} = 50 \text{ N}$$

DIMENSION SELECTION

1 3D 선택



PHYSICS SELECTION



STUDY TYPE SELECTION

week11_2.mph - COMSOL Multiphysics

File Model Definitions Geometry Materials Physics Mesh Study Results

Select Study

1 Stationary 선택

2 Done 클릭

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.

Added study:
Stationary

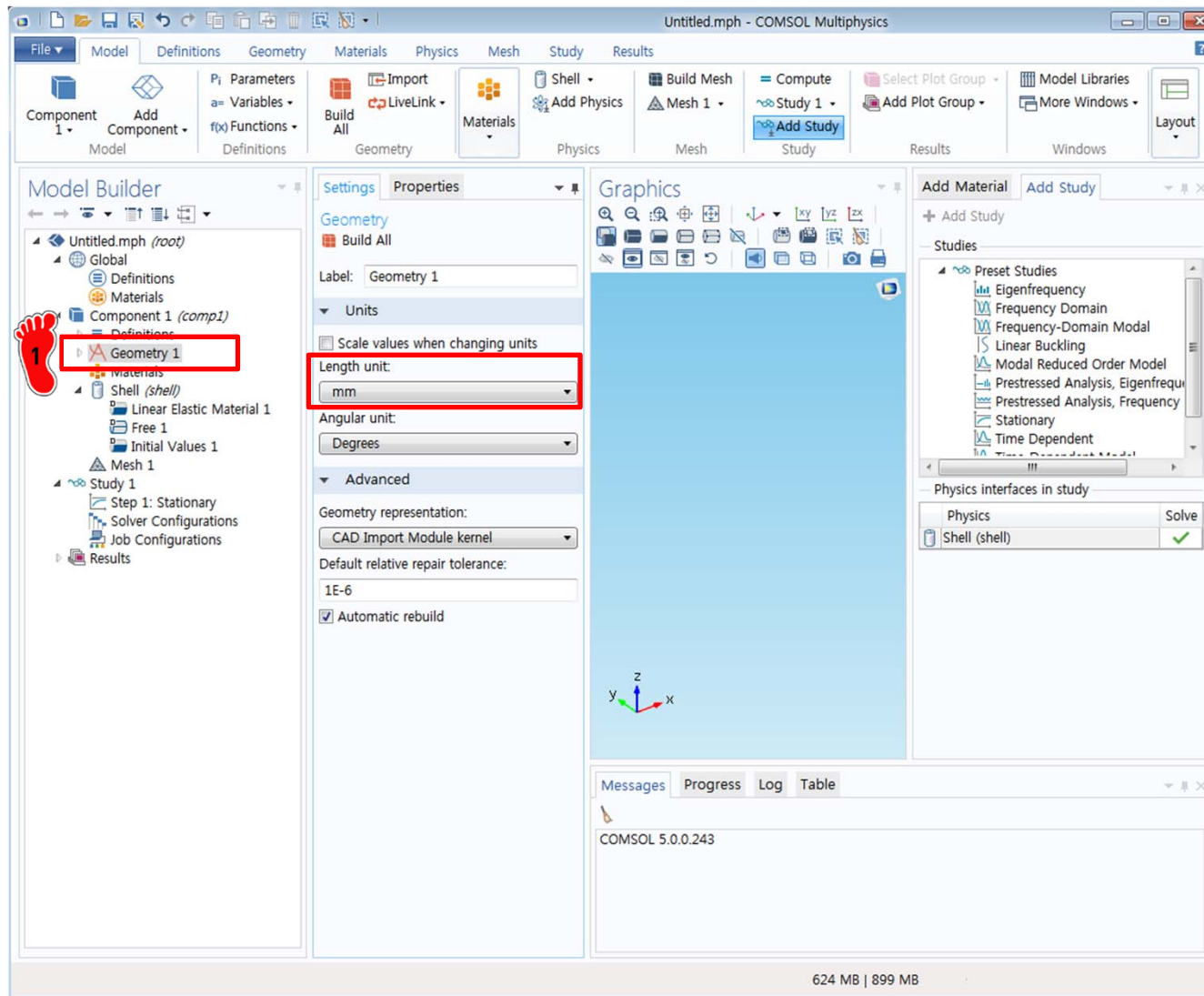
Added physics interfaces:
Shell (shell)

Physics

Help Done

1.01 GB | 1.14 GB

UNIT SETTING

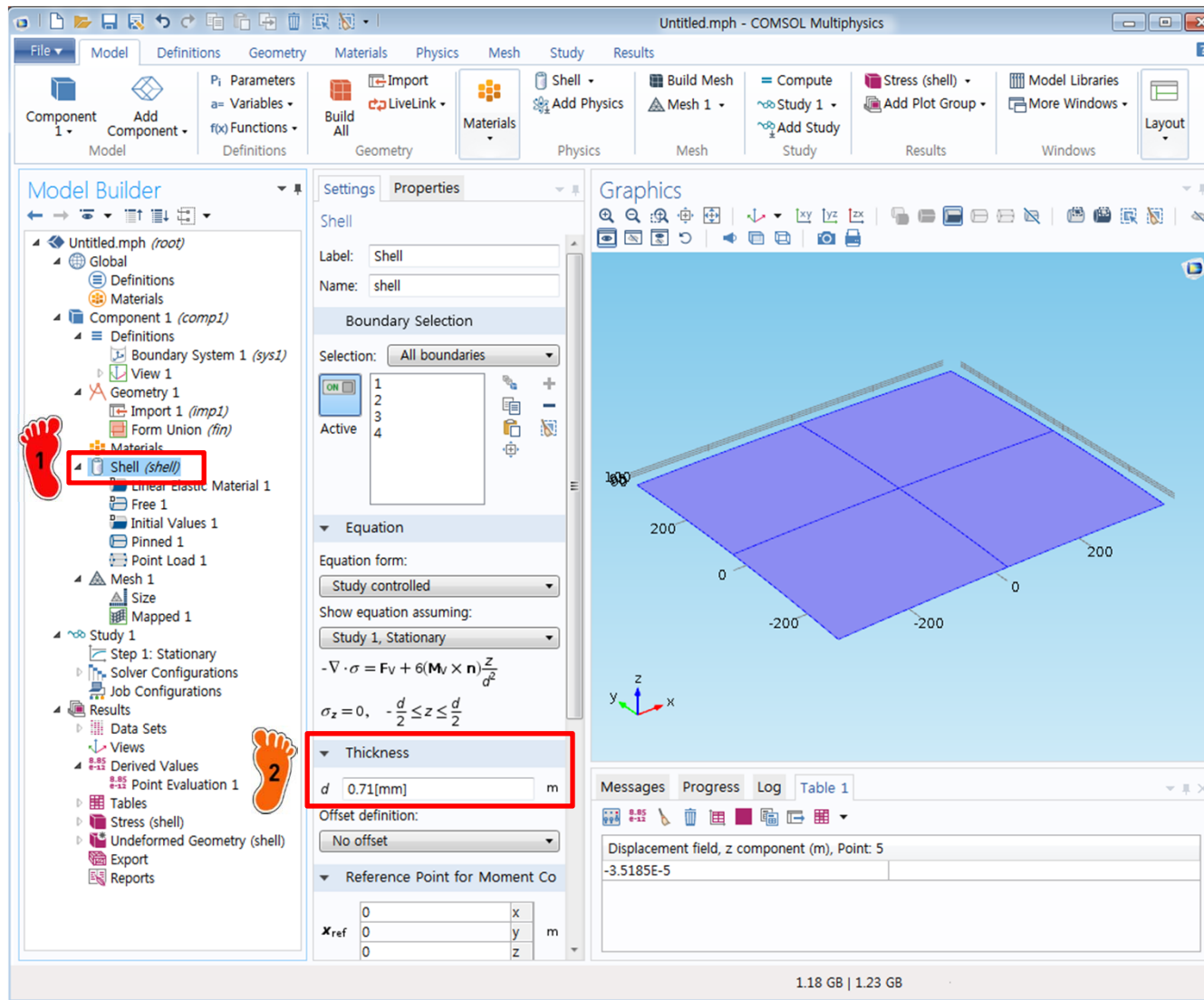


1 Geometry 1 에서 단위를 mm로 설정



옵션 변경 후 카드 파일 삽입

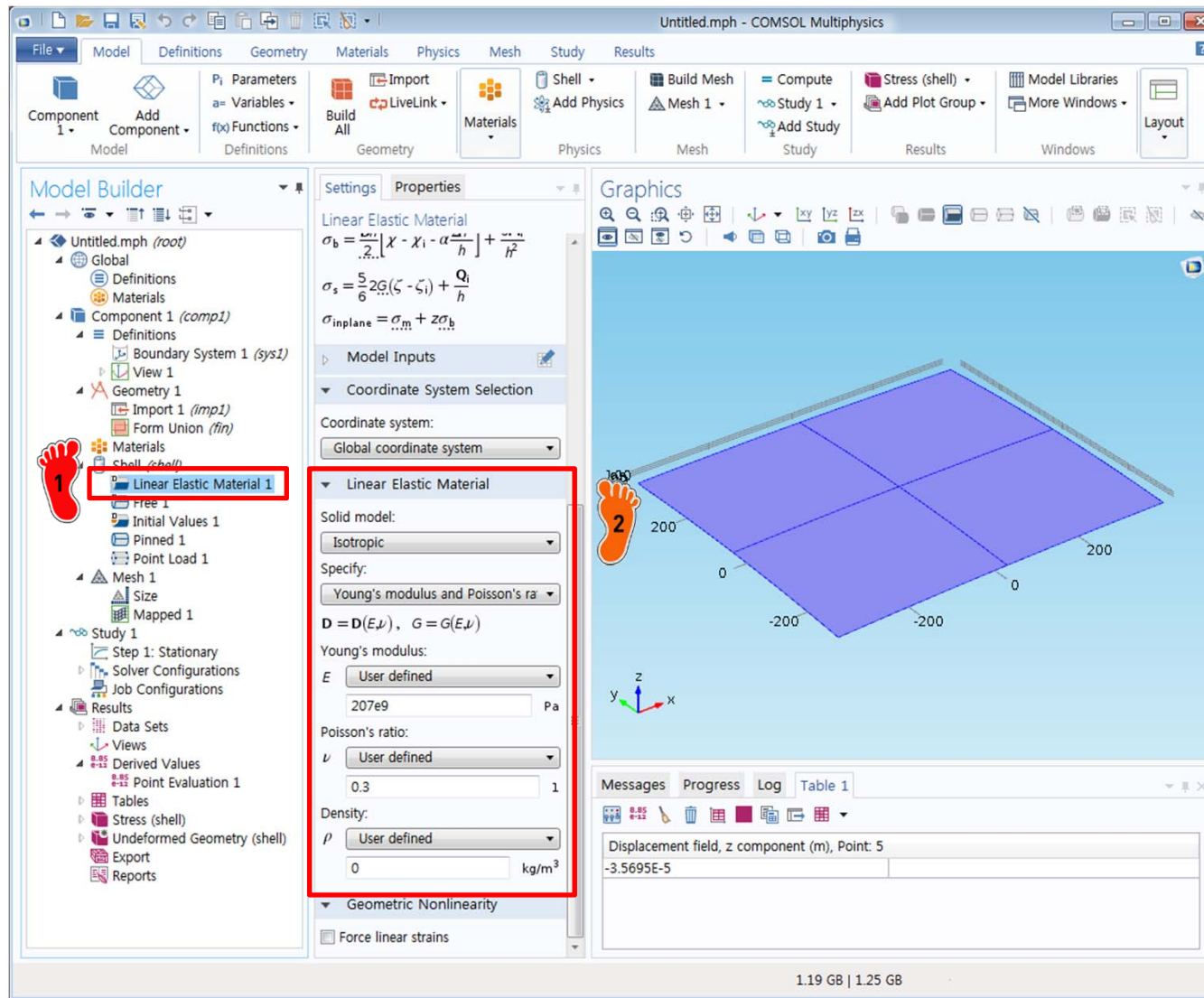
MATERIAL PROPERTY



1 Shell클릭

2 Thickness 0.71 [mm] 입력

MATERIAL PROPERTY

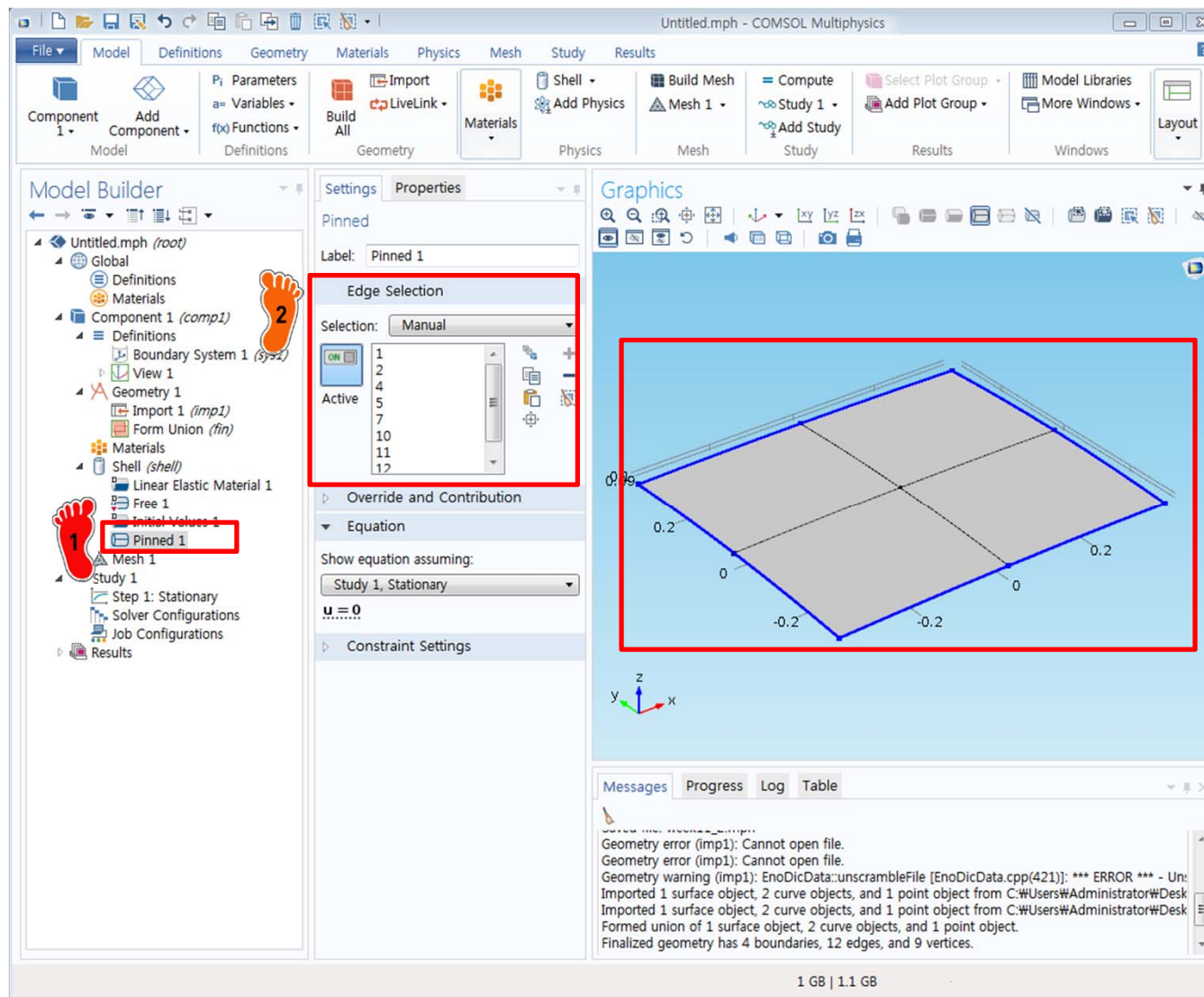


1 Linear Elastic Material 클릭

2 E: 207e9
mu: 0.3
rho : 0

입력

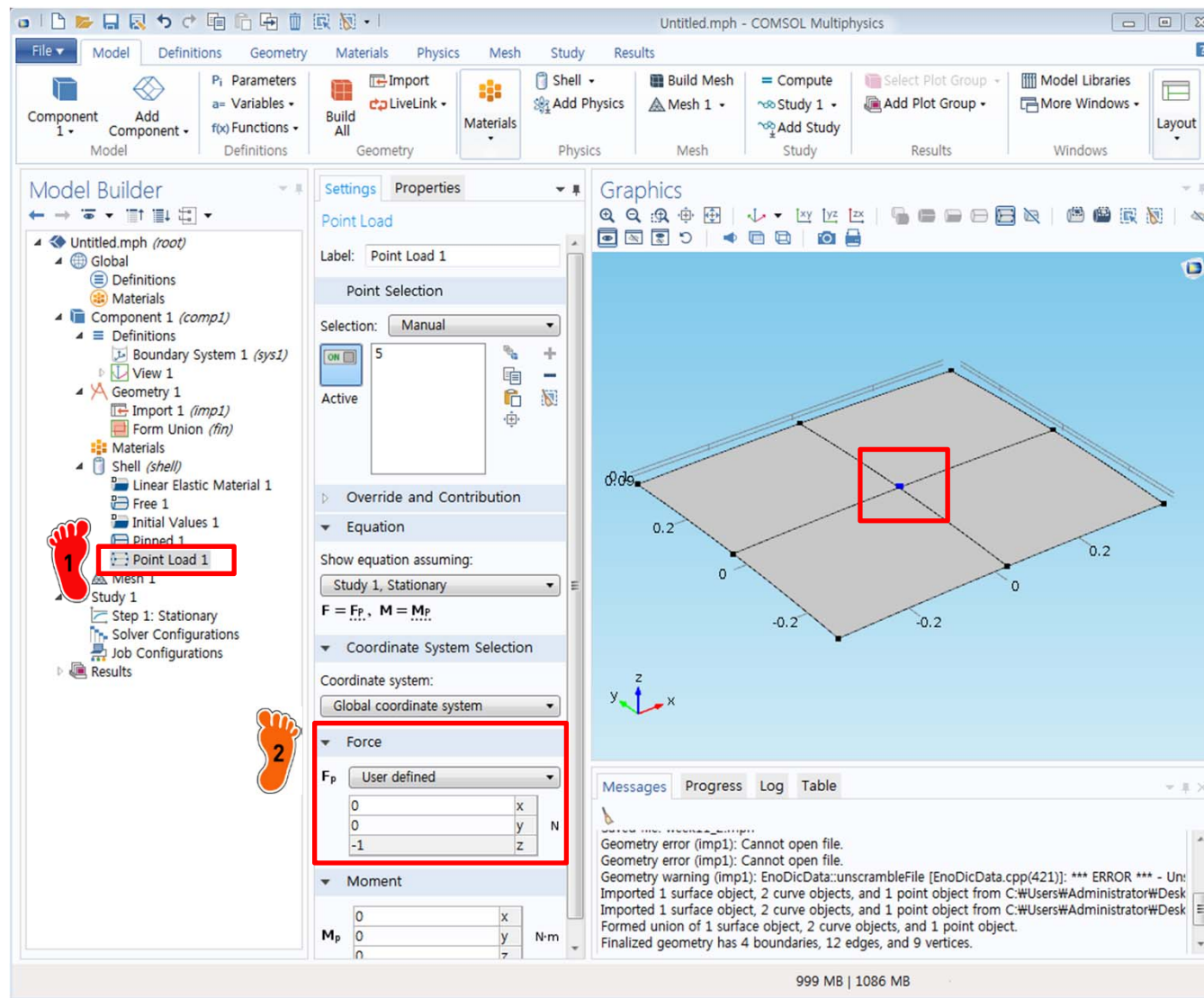
BOUNDARY CONDITION



1 Pinned경계조건 생성

2 최외각 경계 선택

LOADING CONDITION



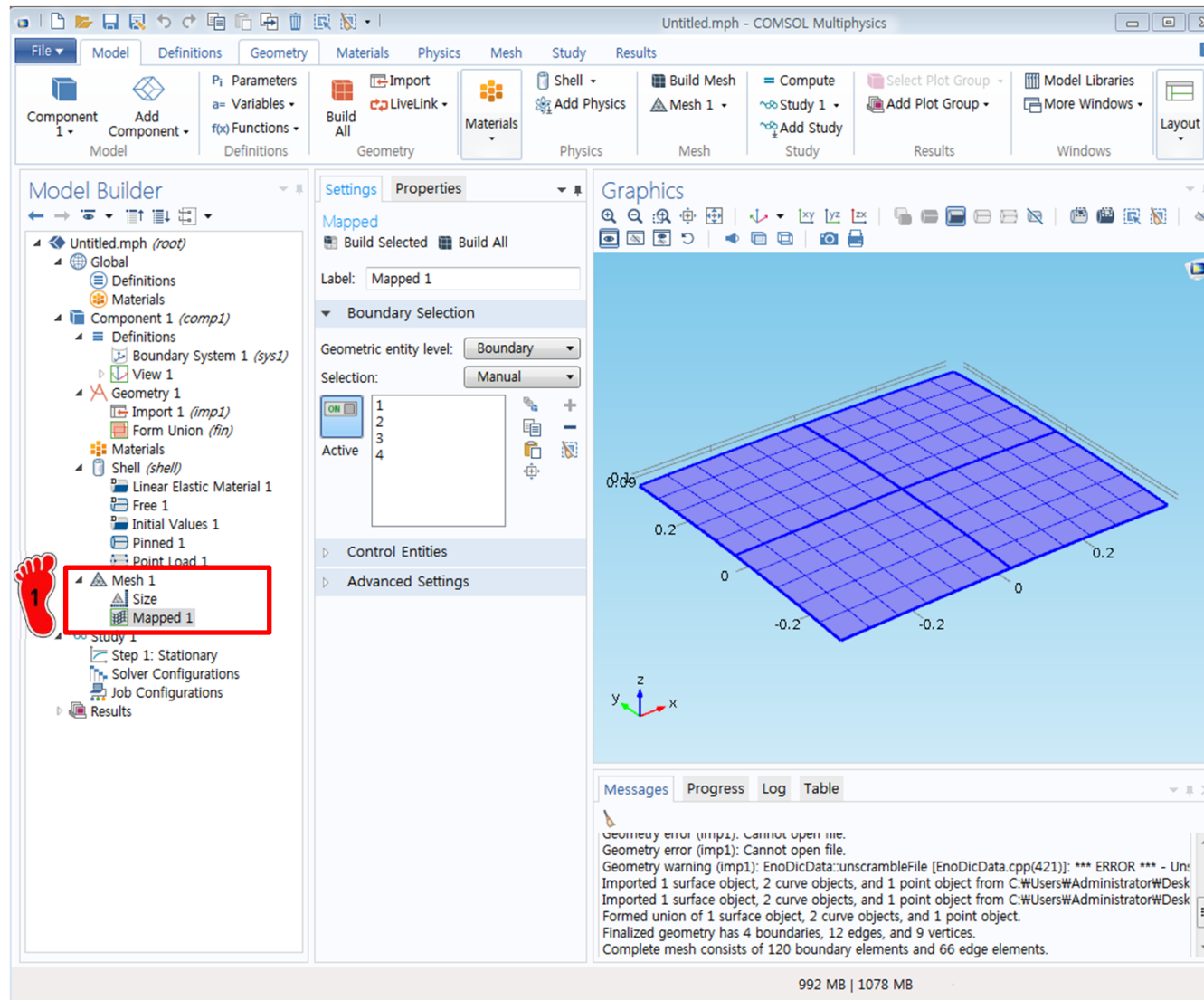
1 Point Load 메뉴 생성

2 5번 절점에 z 방향 -1 N 하중 입력

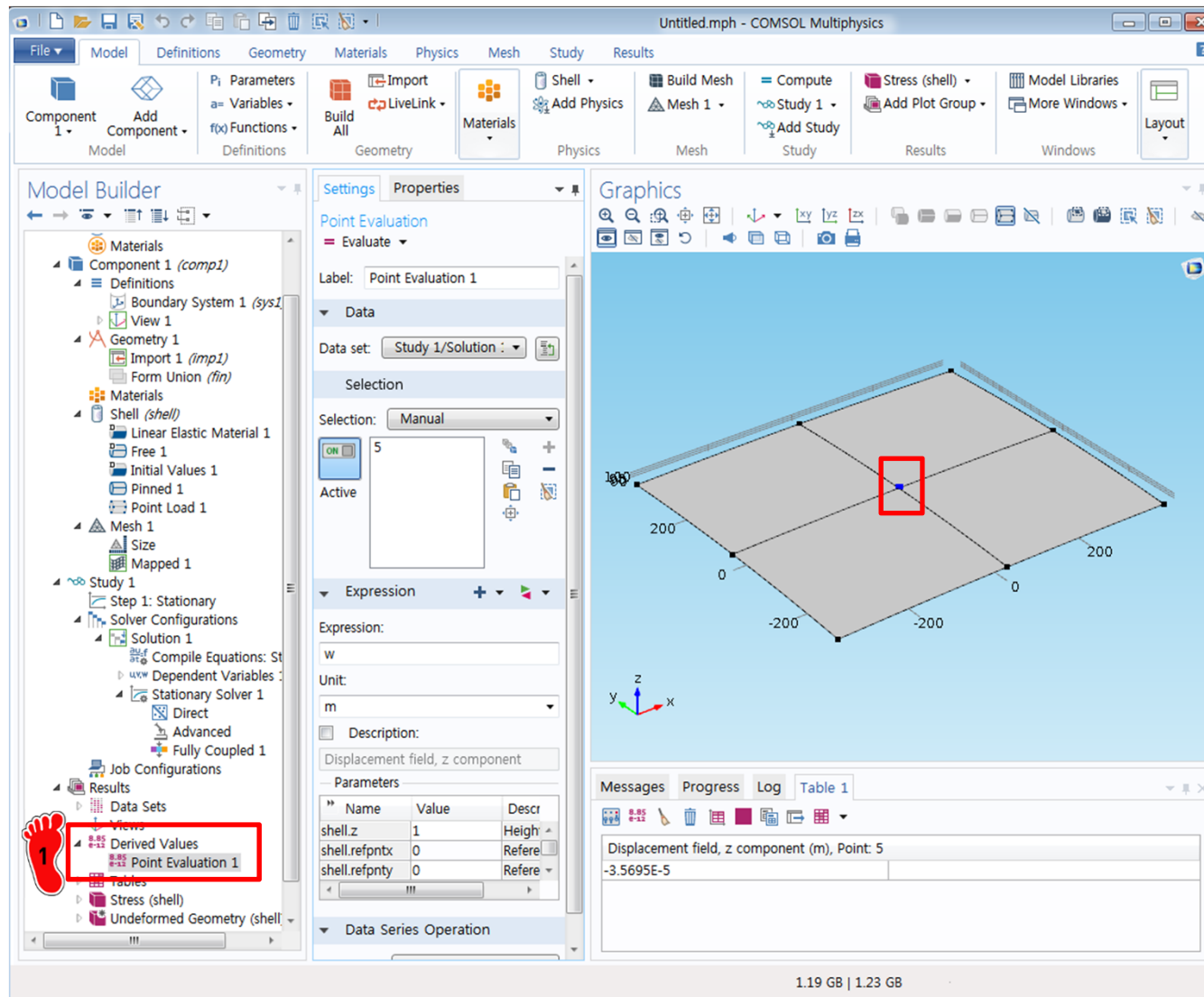
MESH

1 Mapped mesh 기능을 이용하여 요소 생성

그 후, 해석실행



POST-PROCESSING



1 Derived Values 의 Point Evaluation 기능을 이용하여 5번 절점의 z 방향 변위를 계산

결과값: -3.5695×10^{-2} mm

Analytic solution

$$K = \frac{B}{R} \frac{Et^2}{\sqrt{1-\mu^2}} = 28.39 \text{ N/mm}$$

COMSOL

$$K = \frac{P}{\delta} = \frac{1 \text{ N}}{3.5695 \times 10^{-2} \text{ mm}} = 28.02 \text{ N/mm}$$

DIMENSION SELECTION

1 상단 메뉴에서 Add Study 선택

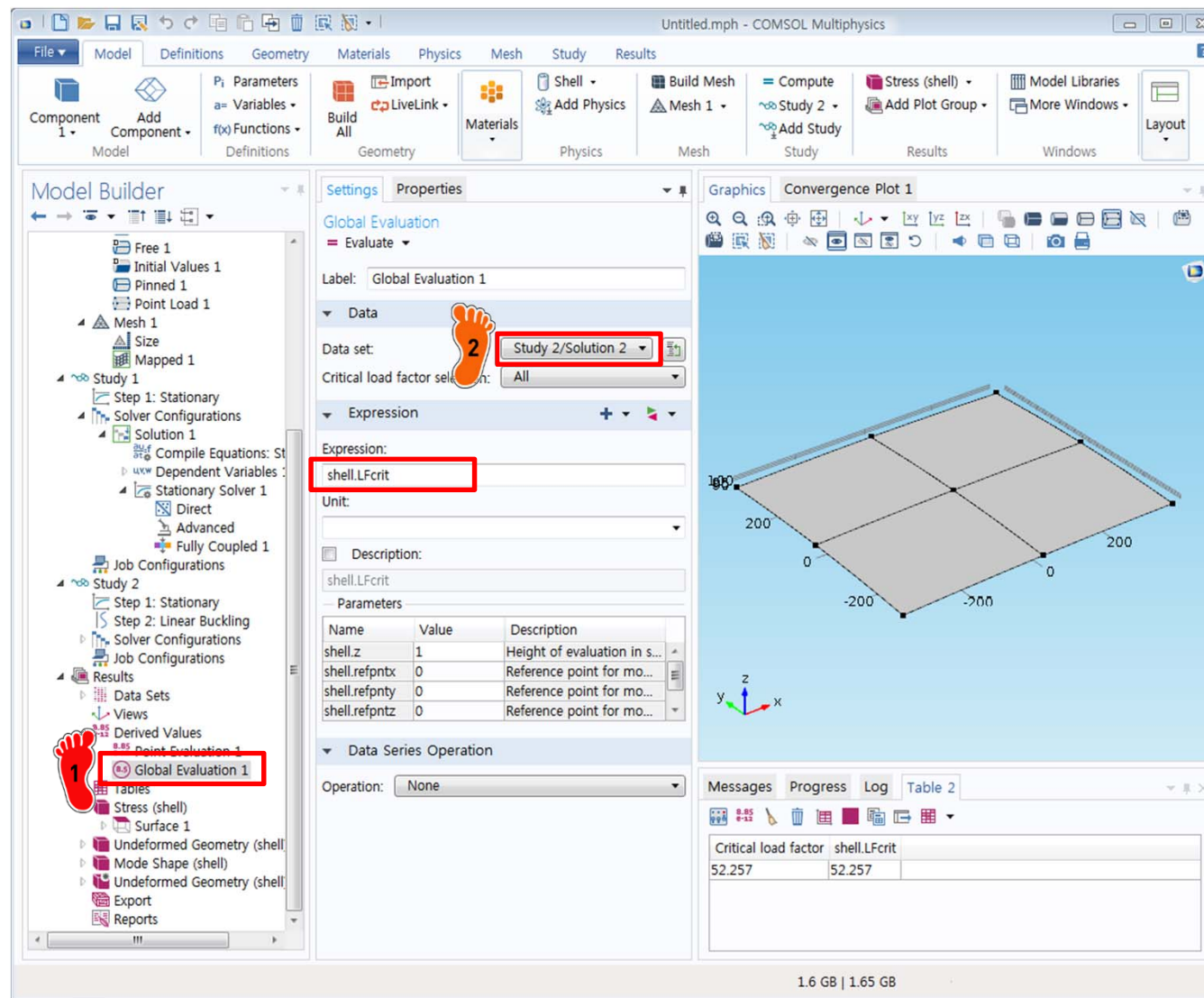
2 Linear Buckling 추가

Study 2 생성 확인 후 해석 실행

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

- Top Toolbar:** Contains icons for 'Compute', 'Add Study', 'Update Solution', 'Get Initial Value', 'Show Default Solver', 'Study Steps', 'Parametric Sweep', 'Function Sweep', 'Material Sweep', 'Optimization', 'Create Solution Copy', 'Operations', 'Statistics', 'Evaluate', 'Clear Solutions', and 'Clear All Solutions'.
- Model Builder:** Shows a tree structure with 'Component 1 (comp1)' containing 'Definitions', 'Geometry 1', 'Materials', 'Mesh 1', 'Study 1', and 'Study 2'. 'Study 2' is highlighted with a red box and a foot icon.
- Settings Panel:** Shows 'Stationary' settings with 'Include geometric nonlinearity' checked and 'Physics interface' set to 'Shell (shell)'.
- Add Study Dialog:** A red box highlights the 'Add Study' dialog, showing 'Linear Buckling' selected under 'Preset Studies'.
- Graphics:** Displays a 3D mesh of a square plate with a coordinate system (x, y, z).
- Messages:** Shows the displacement field, z component (m), with a value of -3.5695E-5.

POST-PROCESSING



1 Global Evaluation 생성

2 Data set을
Study 2/Solution 2로 변경,
shell.LFcrit값 확인

shell.LFcrit: Critical load
factor from linear buckling
analysis

결과값: 52.257 N

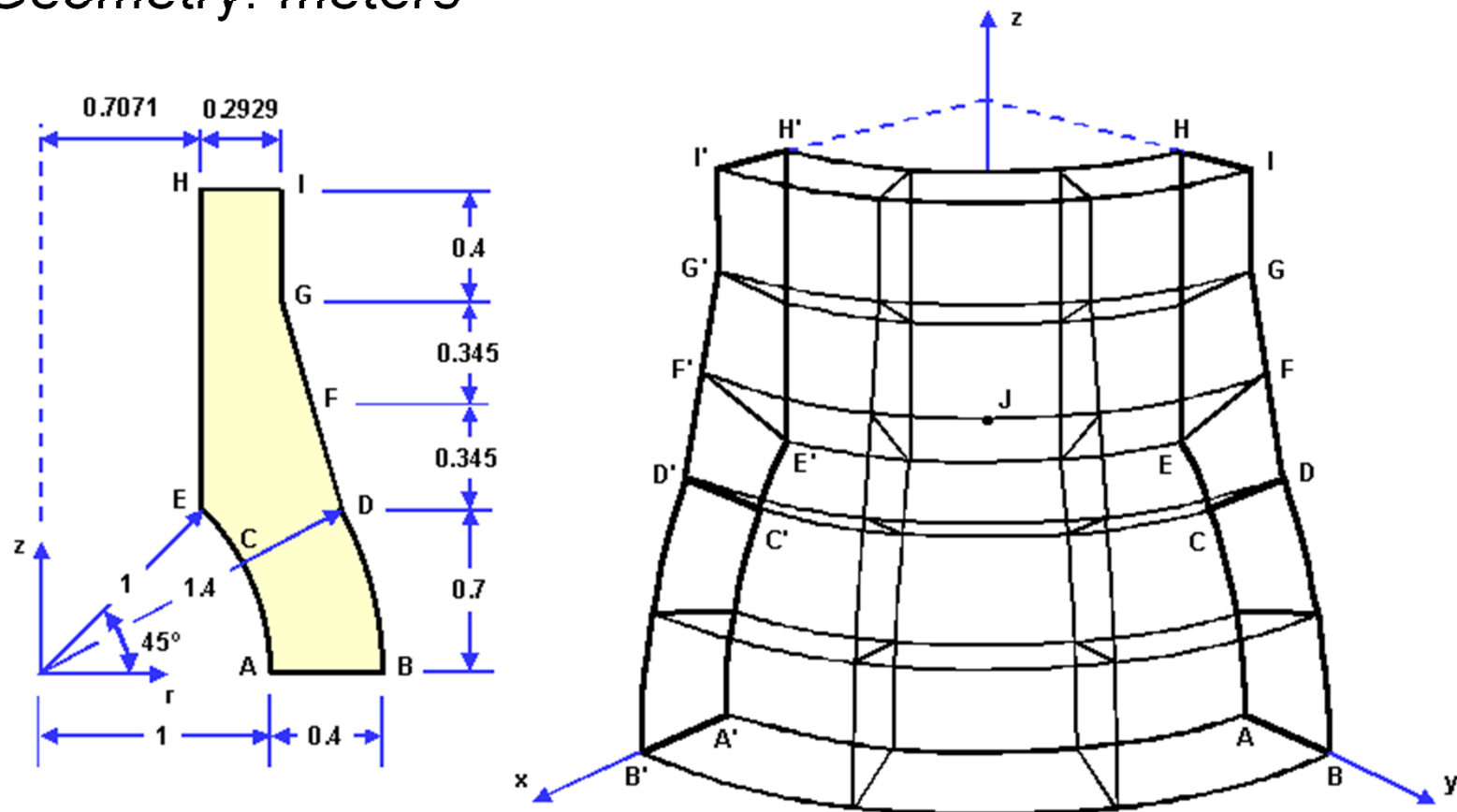
Analytic solution

$$P_{CR} = \frac{CR_{CR} \pi^2 E t^4}{L_1 L_2 (1 - \mu^2)} = 50 \text{ N}$$

NAFEMS BENCHMARK: LE11

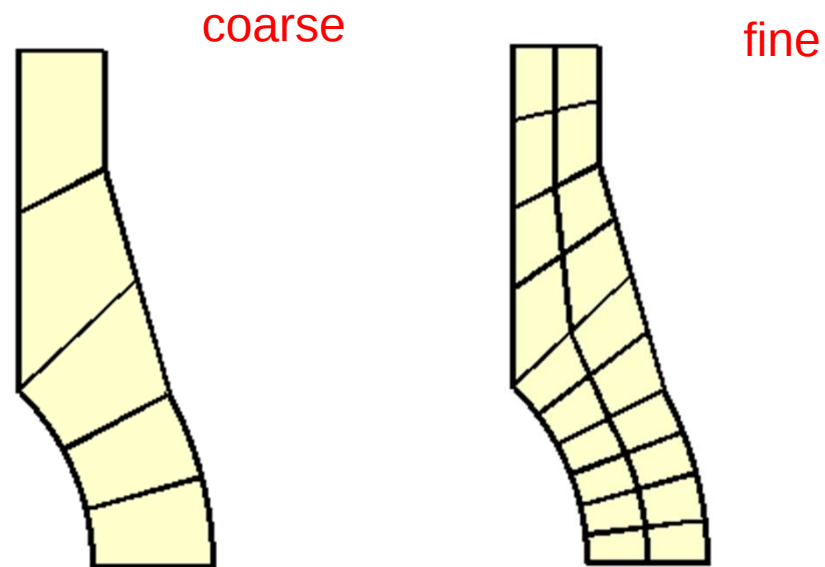
Solid Cylinder/Taper/Sphere – Temperature

- Analysis Type: Linear elastic solid
- Geometry: meters



- Loading
 - Linear temperature gradient in the radial and axial direction
 - $T (^{\circ}\text{C}) = (x^2 + y^2)^{1/2} + z$
- Boundary conditions
 - Symmetry on xz-plane
 - Symmetry on yz-plane
 - Face on xy-plane: zero z-displacement
 - Face HH'I': zero z-displacement
- Material properties
 - Isotropic: $E = 210 * 10^3 \text{ MPa}$, $\nu = 0.3$
 - $\alpha = 2.3 * 10^{-4} / ^{\circ}\text{C}$
- Element types
 - Solid hexahedra, wedges and tetrahedra

- Meshes

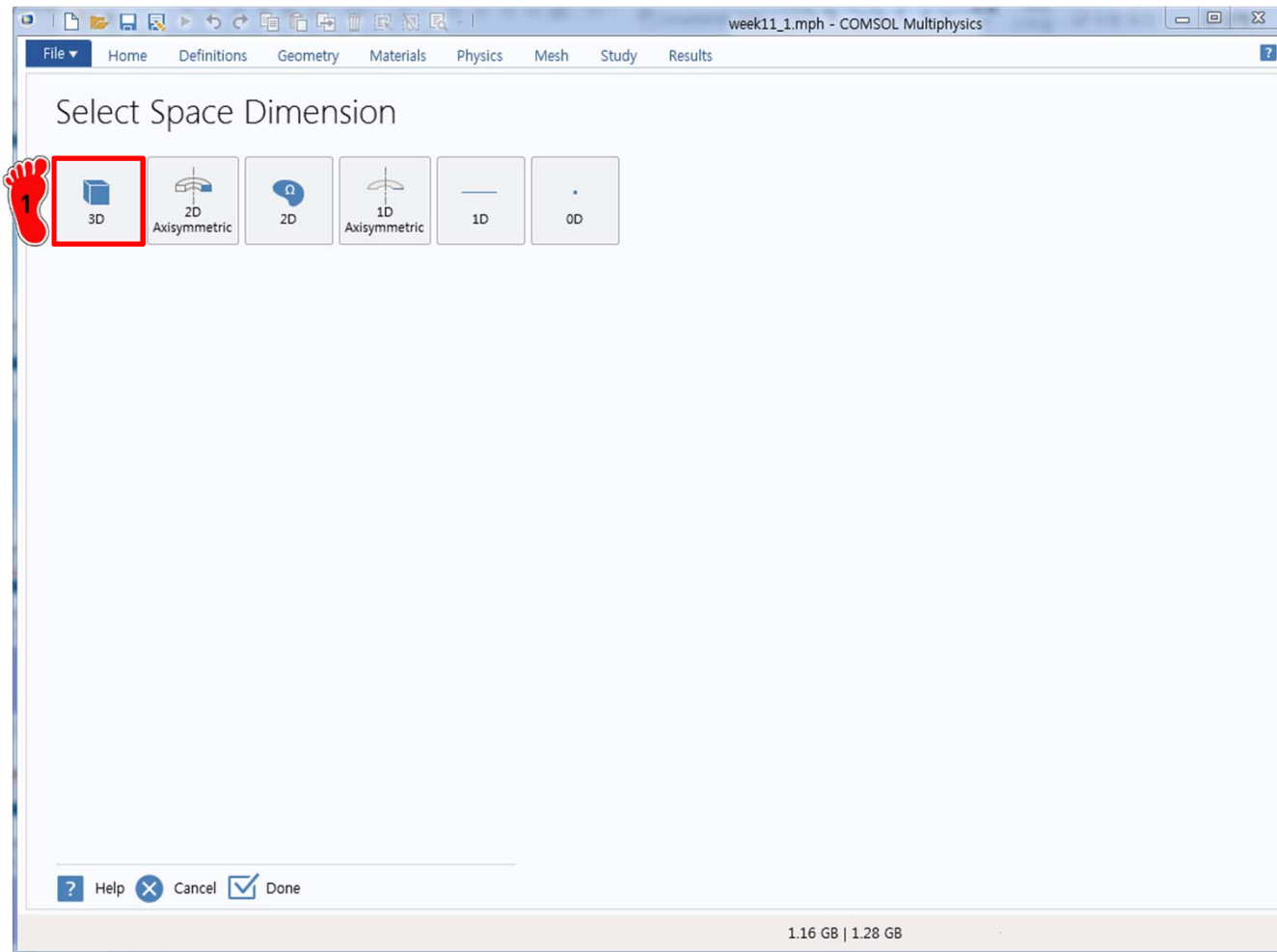


- Output

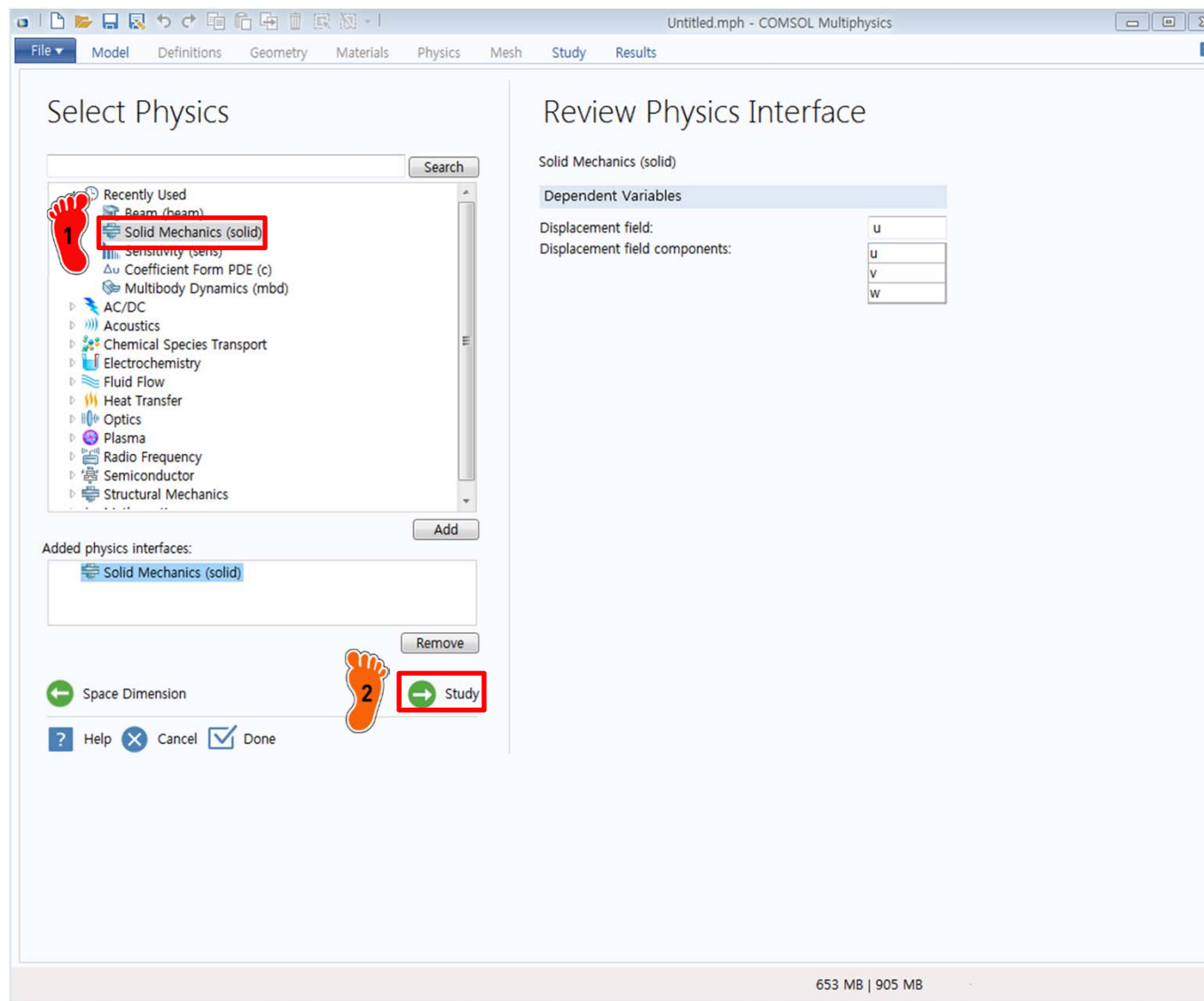
- Direct stress (σ_{zz}) at point A: -105 MPa

DIMENSION SELECTION

1 3D 선택



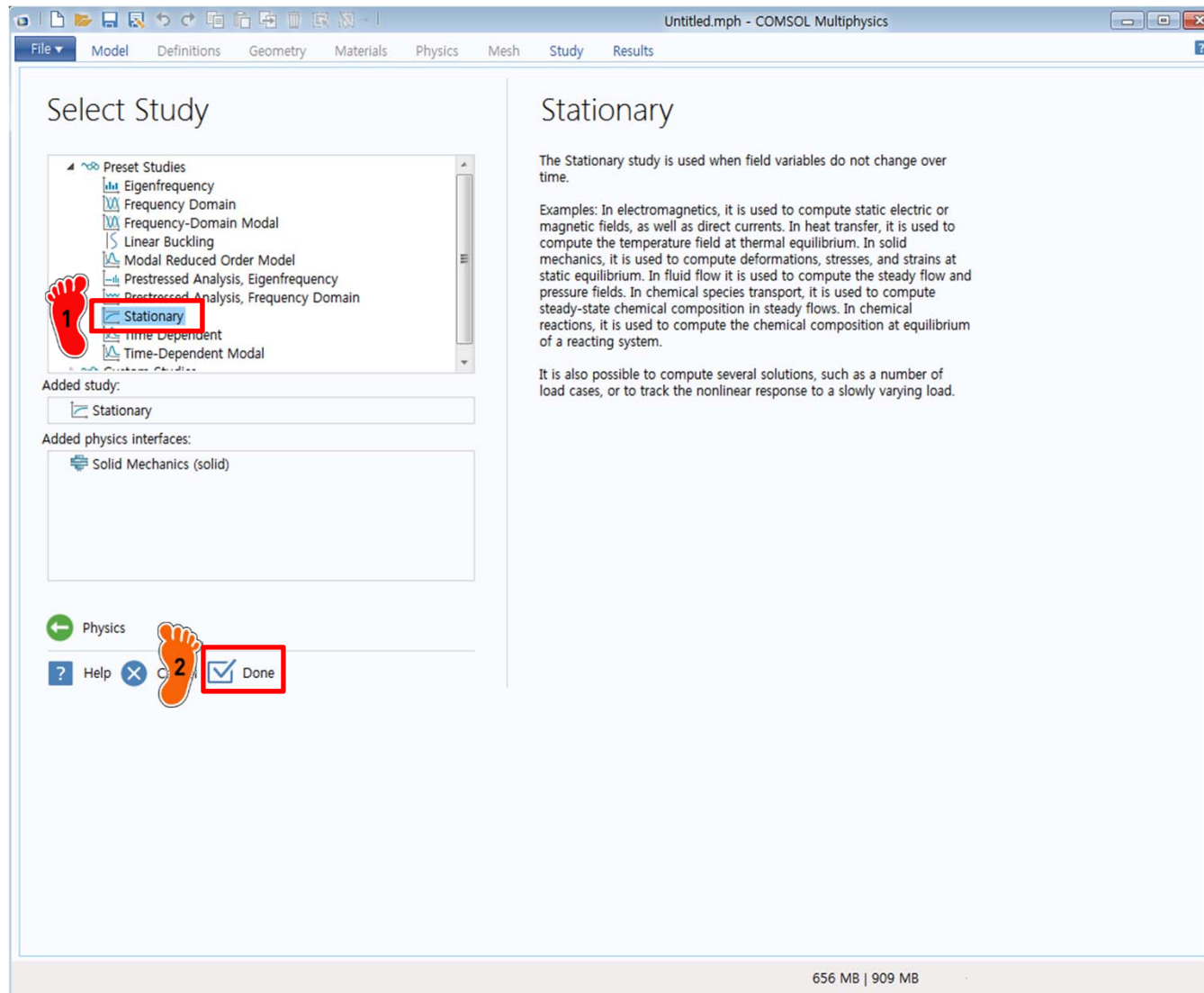
PHYSICS SELECTION



1 Structural Mechanics 의
Solid Mechanics 선택

2 Study 클릭

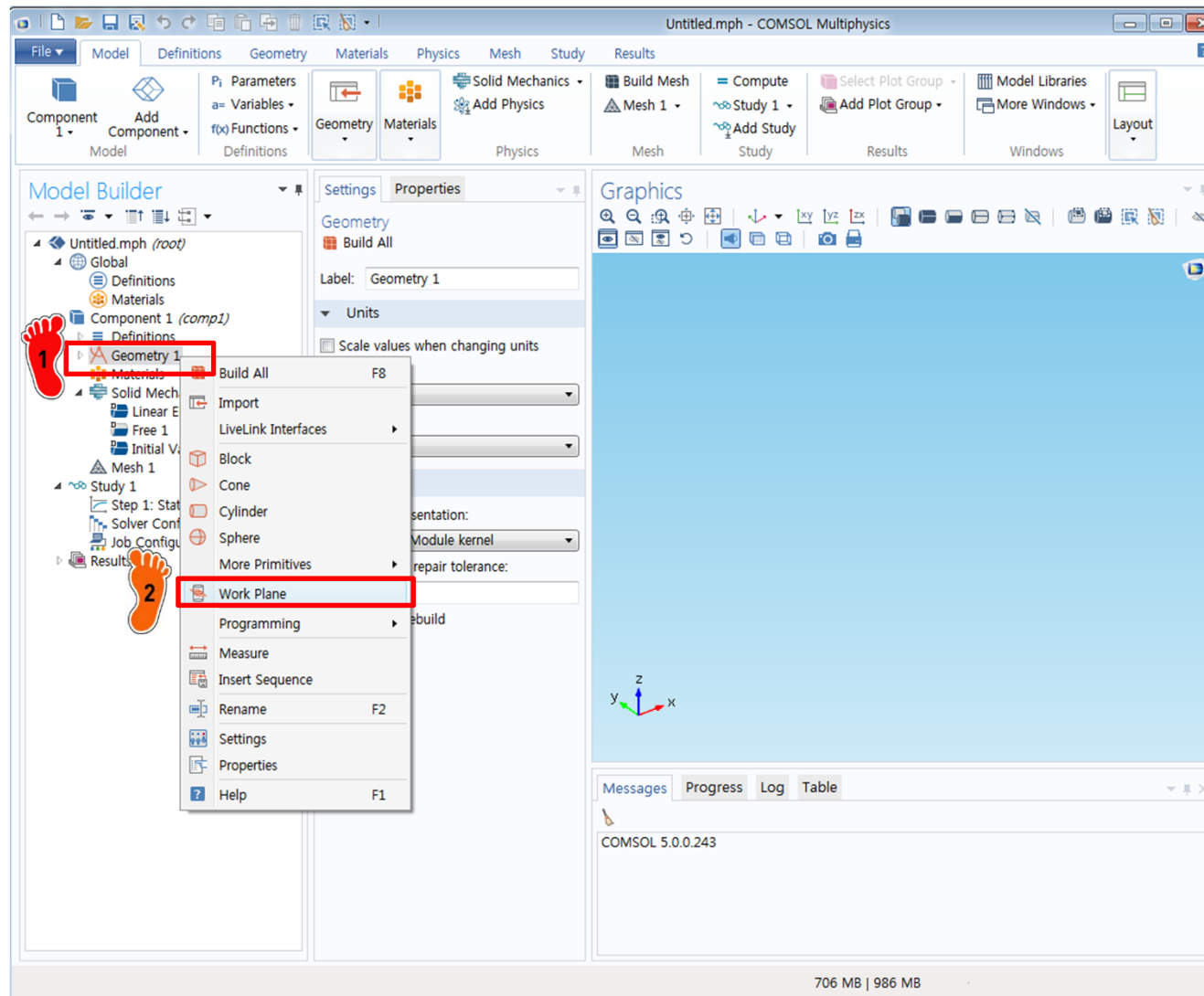
STUDY TYPE SELECTION



1 Stationary 선택

2 Done 클릭

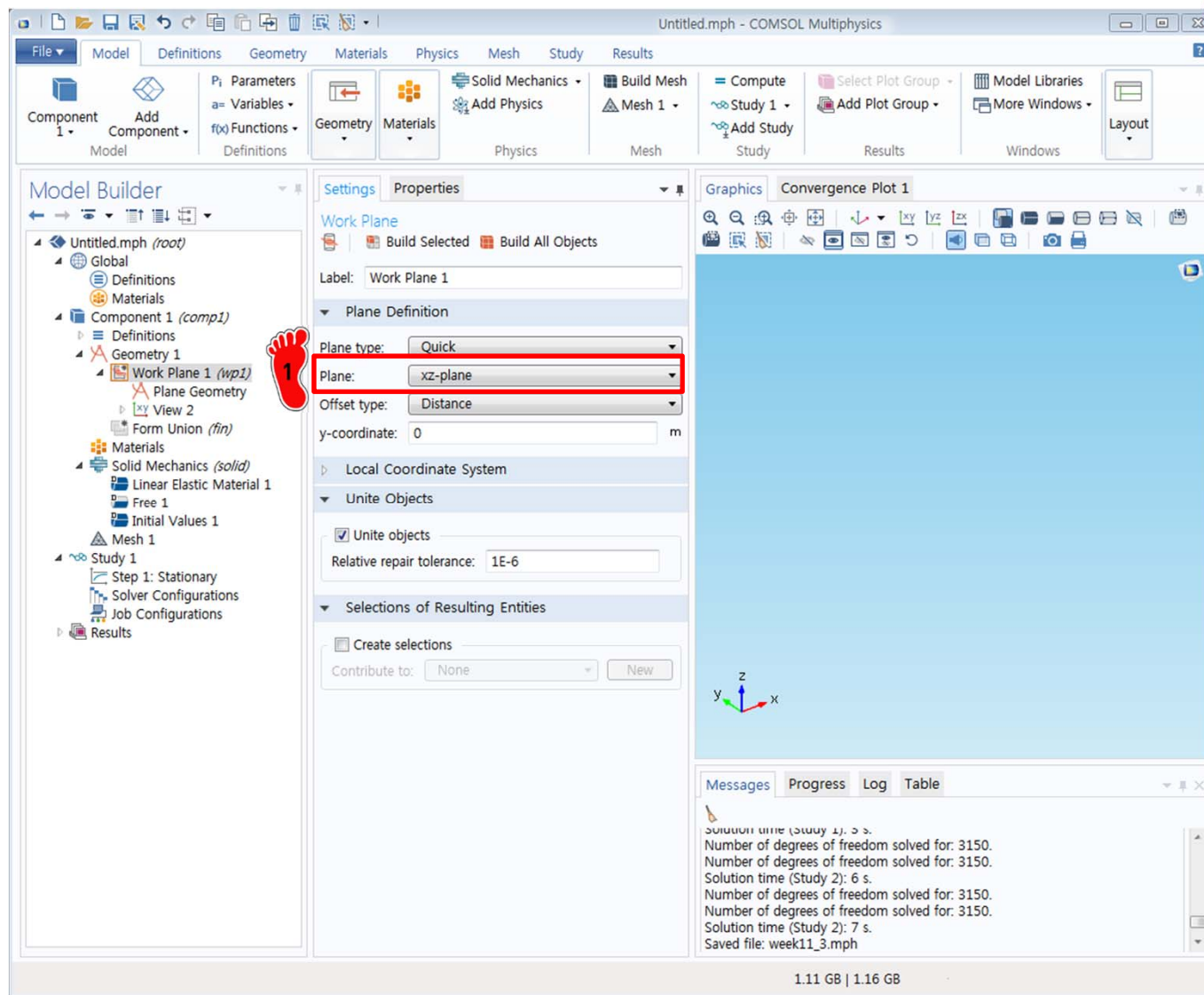
GEOMETRY CREATION



1 Geometry 1 마우스 우클릭

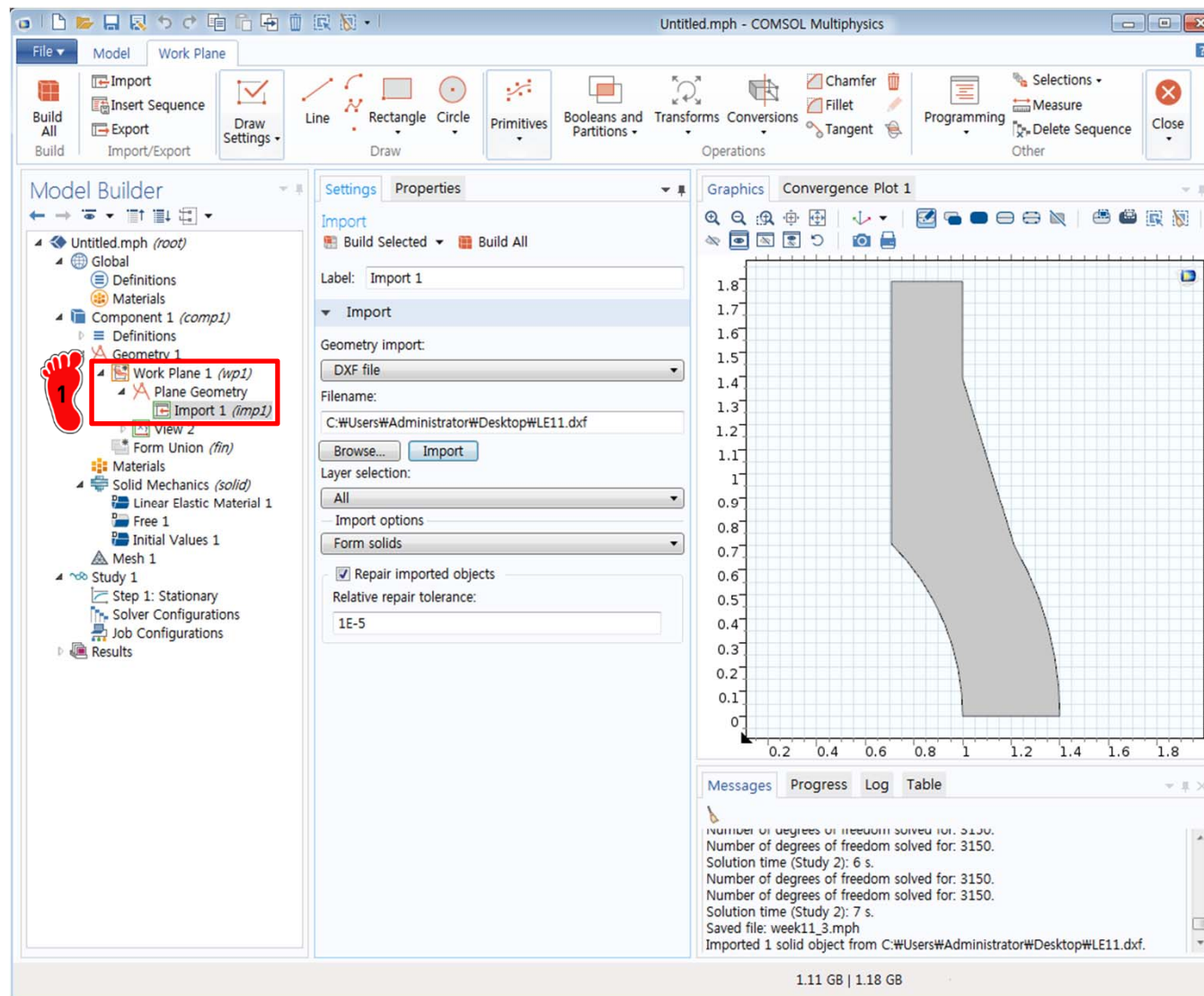
2 Work Plane 클릭

GEOMETRY CREATION



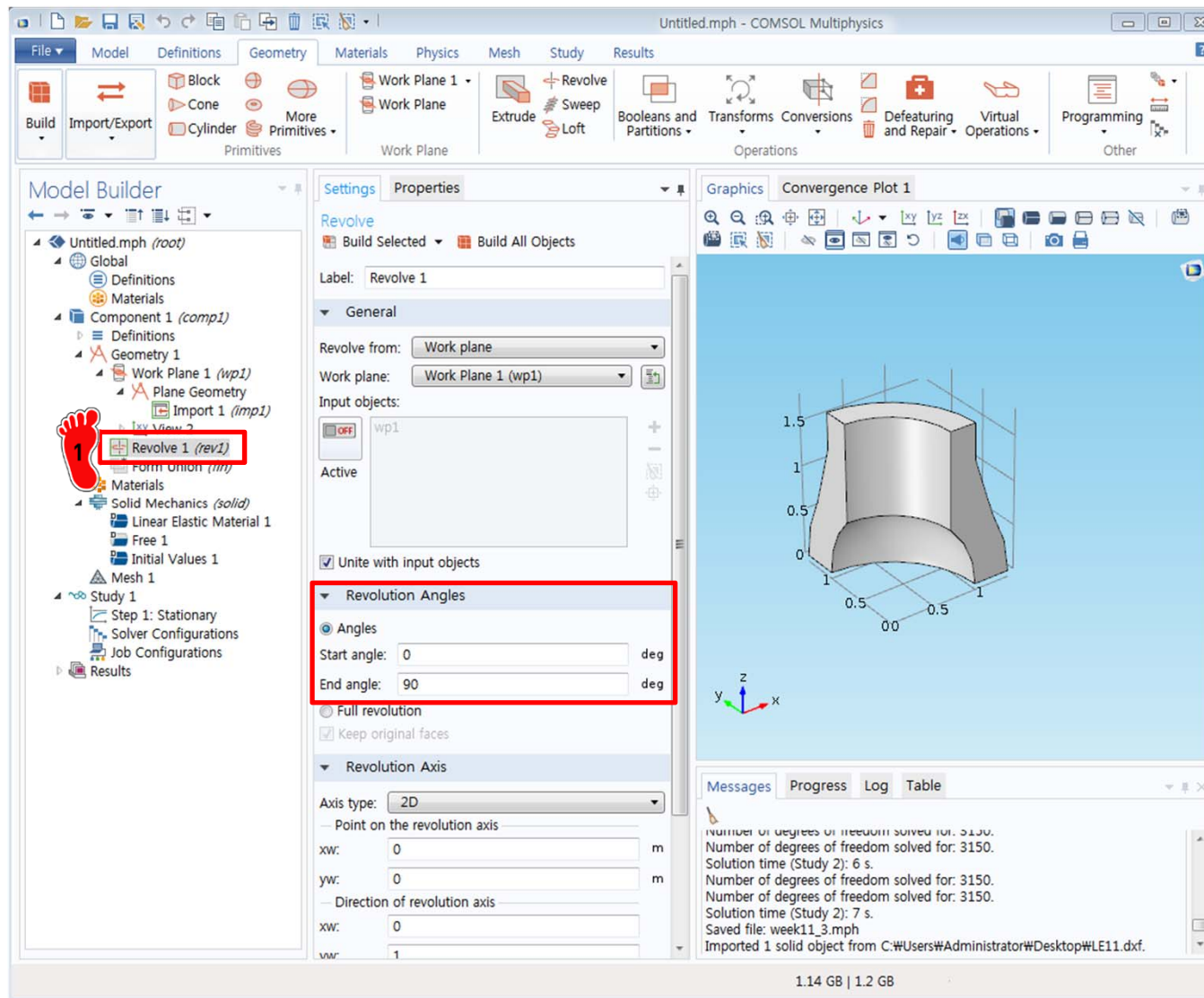
1 work plane 방향을 xz-plane 으로 변경

GEOMETRY CREATION



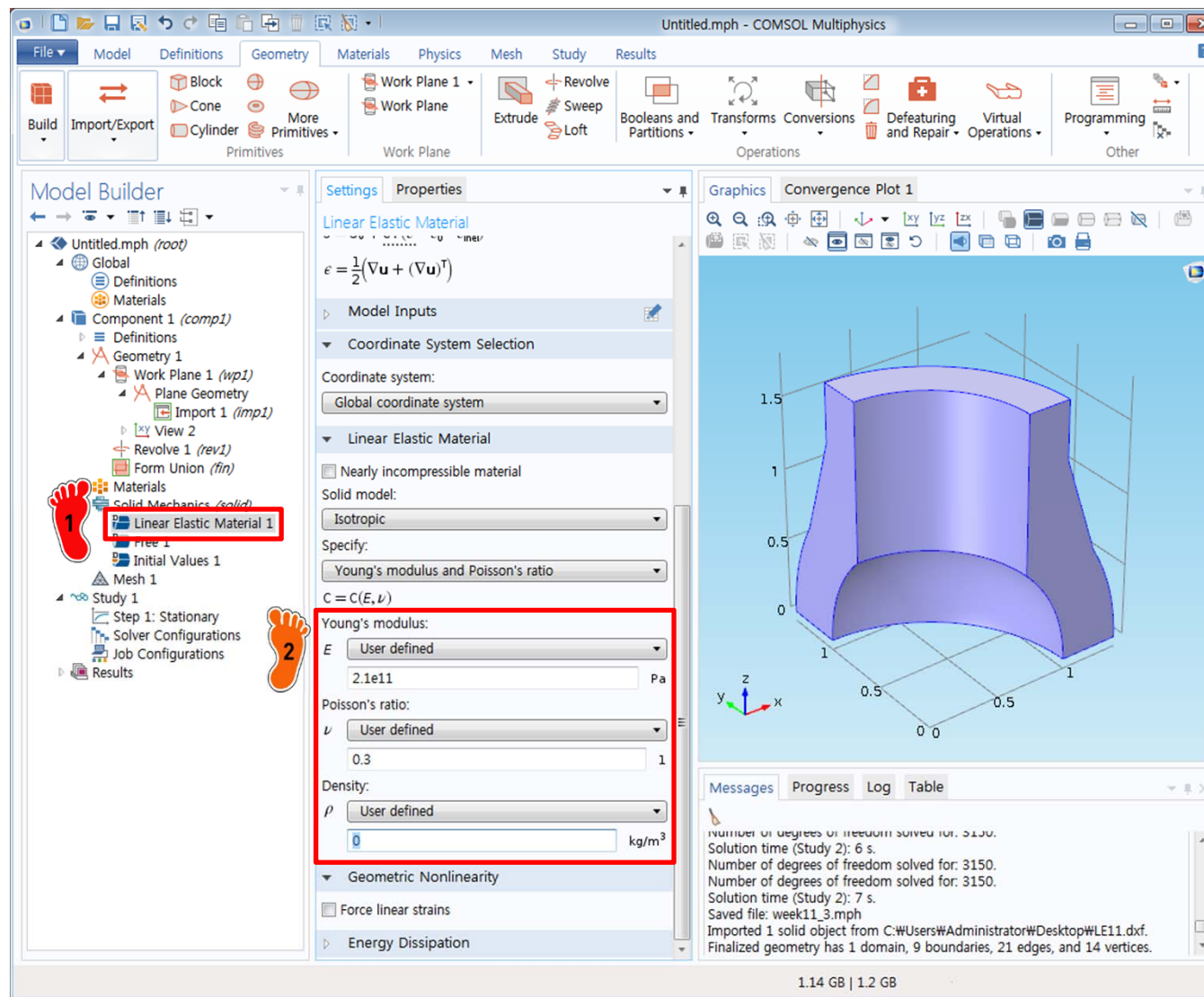
Plane Geometry에서
Import 메뉴를 이용하여 주
어진 캐드 기하형상을 삽입

GEOMETRY CREATION

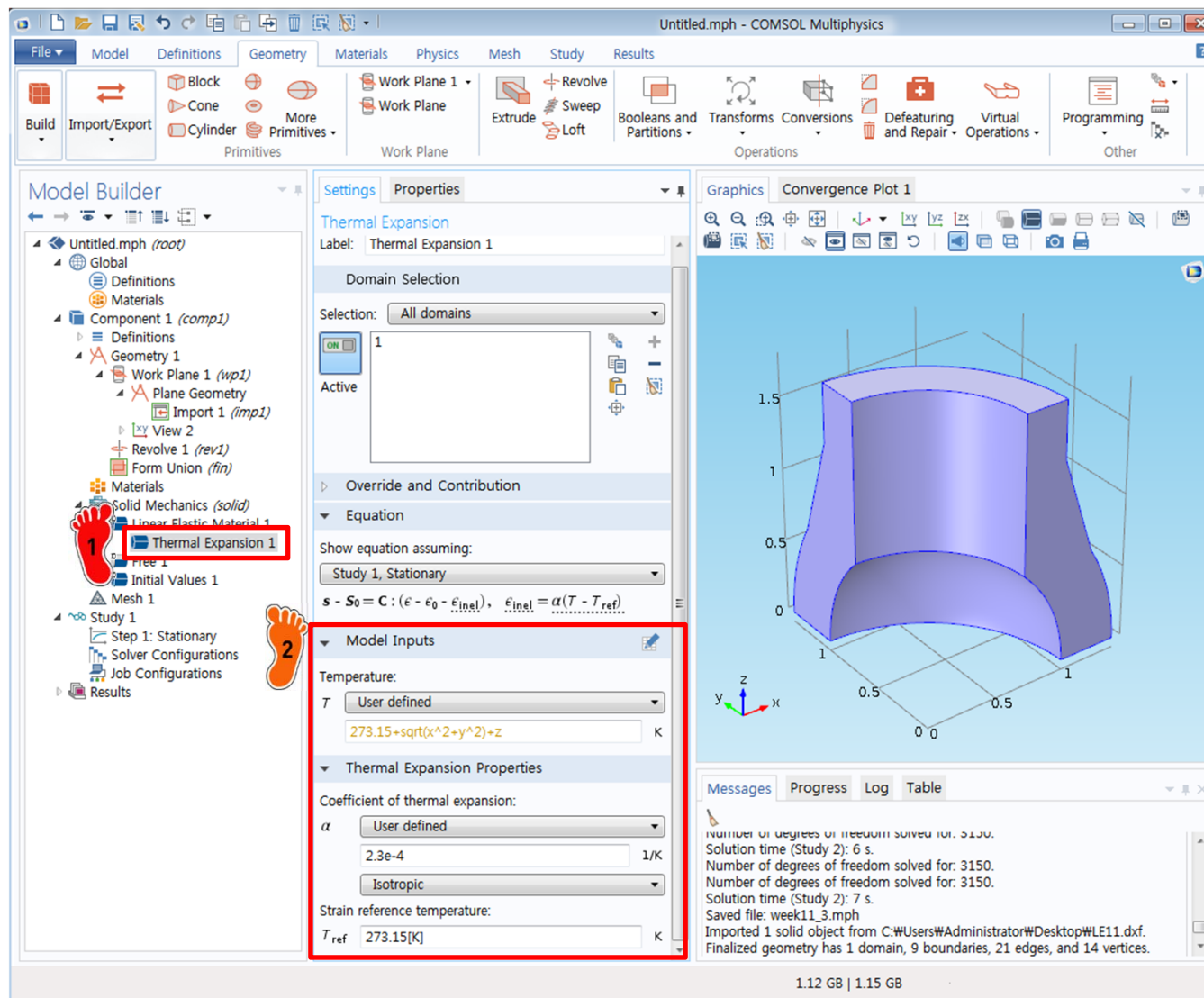


1 Revolve 메뉴를 이용하여
90도 회전시킨 형상을 생성

MATERIAL PROPERTY



MATERIAL PROPERTY

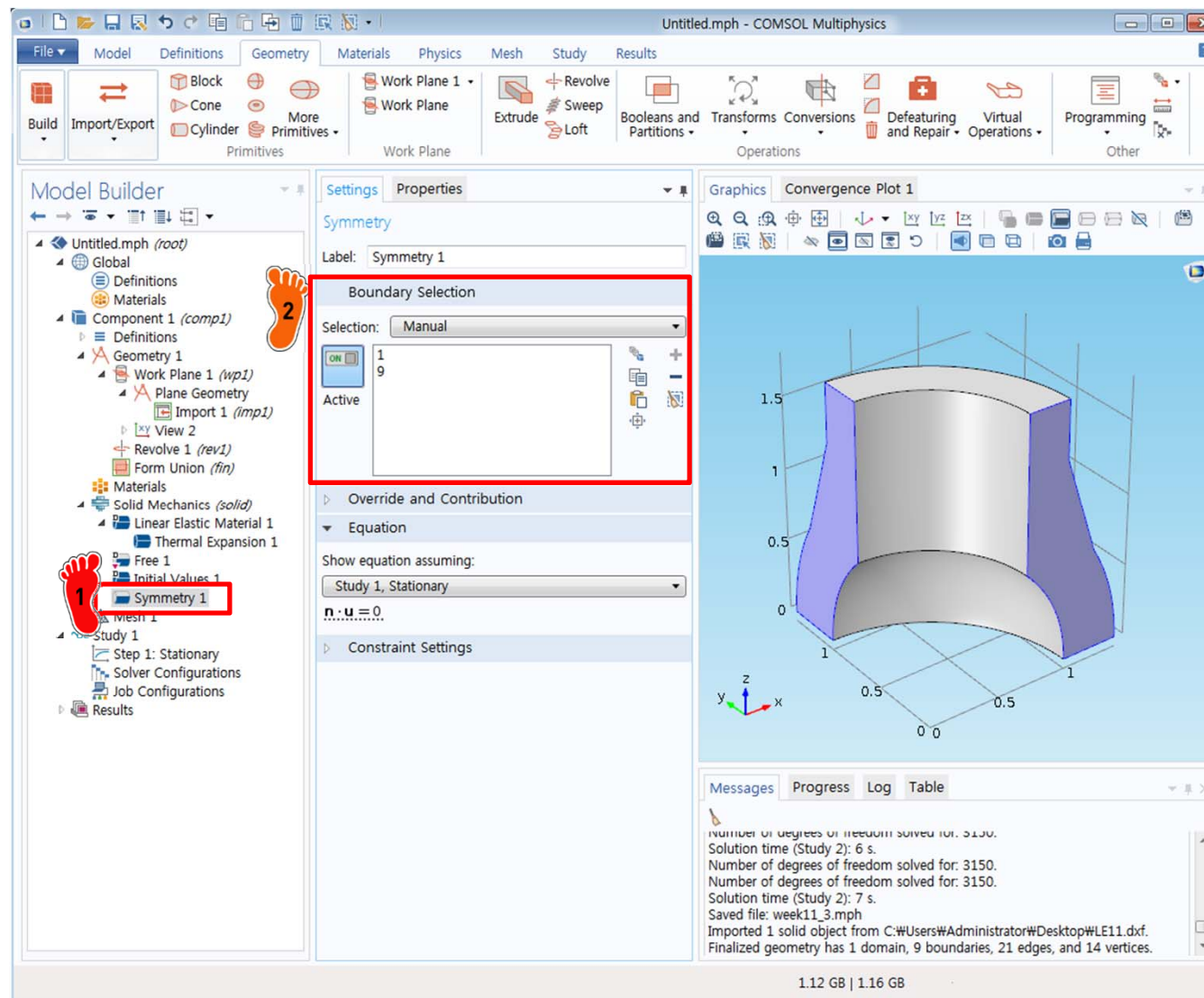


1 Linear Elastic Material 하위 메뉴로 Thermal Expansion 추가

2 Temperature:
 $273.15 + \sqrt{x^2 + y^2} + z$
 $\alpha: 2.3e-4$
 Strain reference temperature: 273.15

입력

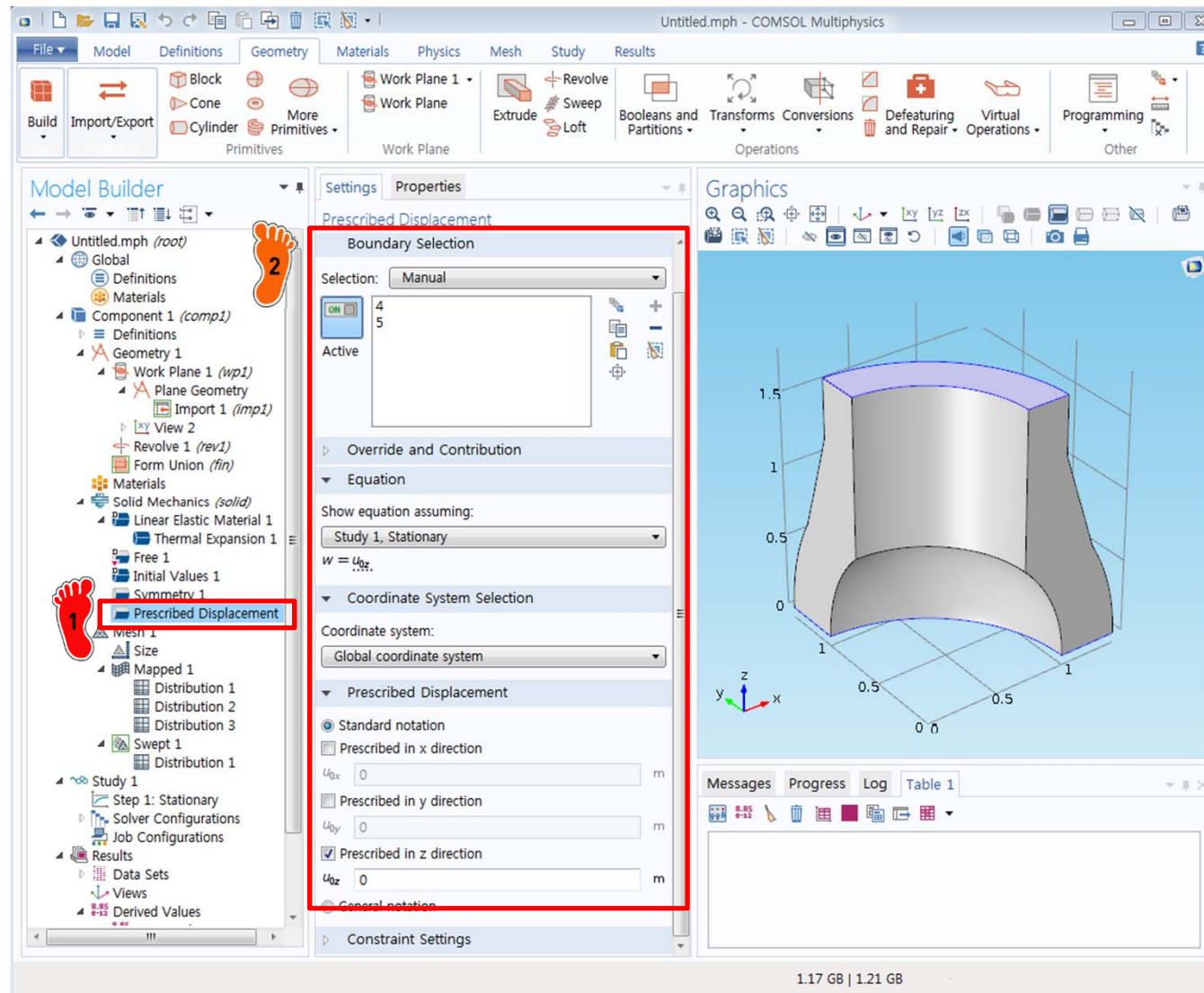
BOUNDARY CONDITION



1 Symmetry 경계조건 생성

2 1, 9번 경계 선택

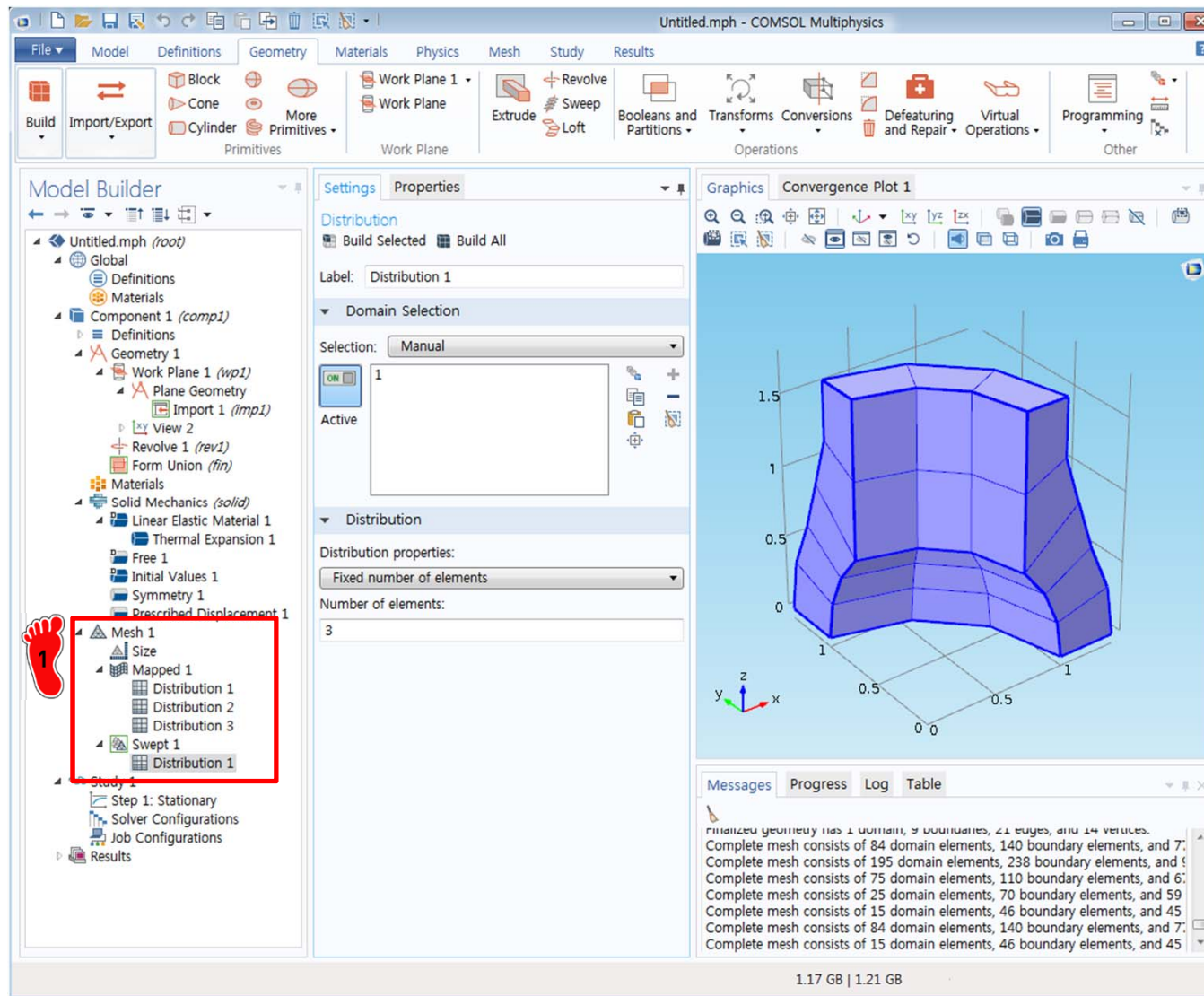
BOUNDARY CONDITION



1 Prescribed Displacement
경계조건 생성

2 4, 5번 경계 선택 후 z 방향
변위에 0 입력

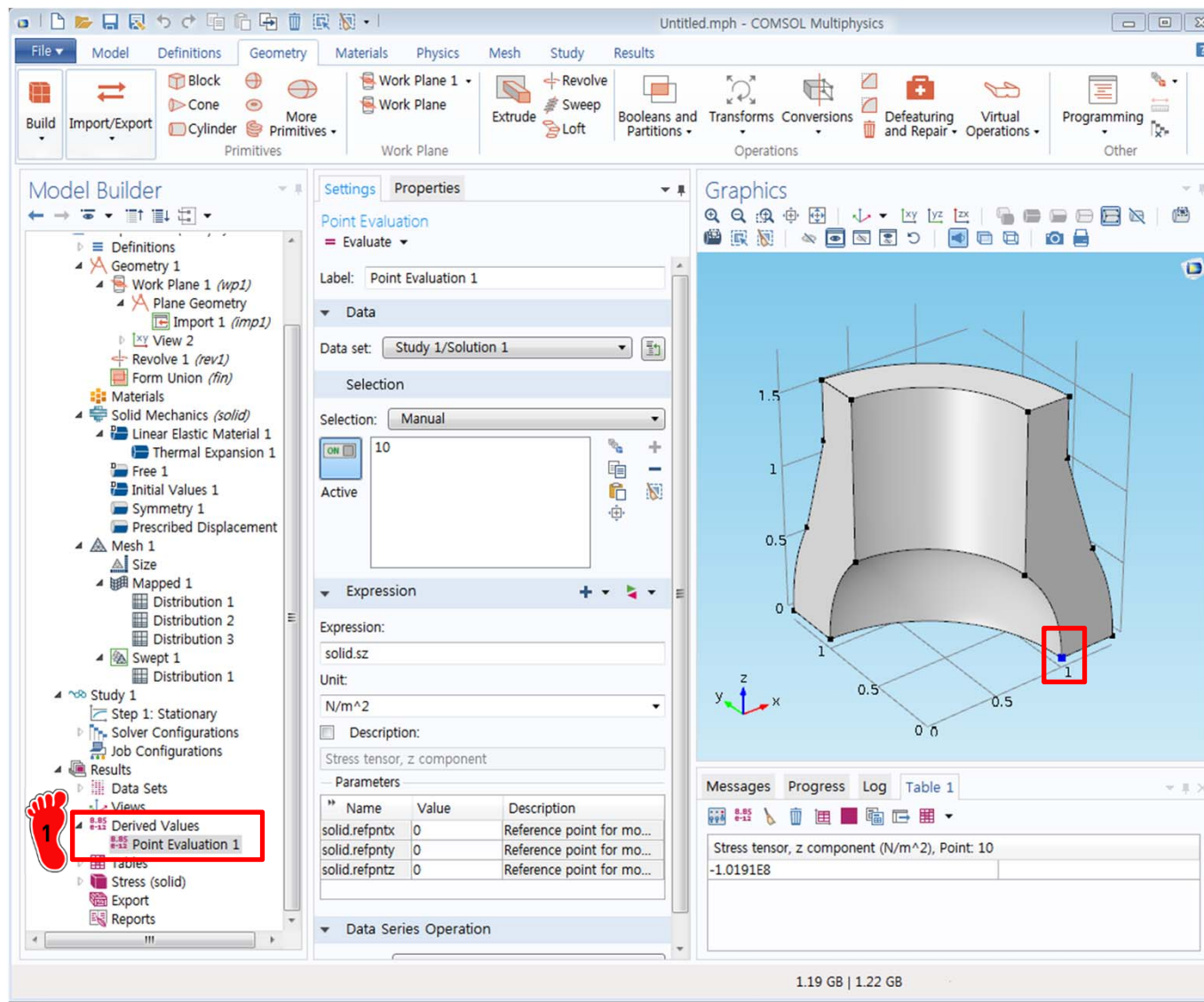
MESH



1 Mapped mesh 와 Swept 기능을 이용하여 요소 생성

그 후, 해석실행

POST-PROCESSING



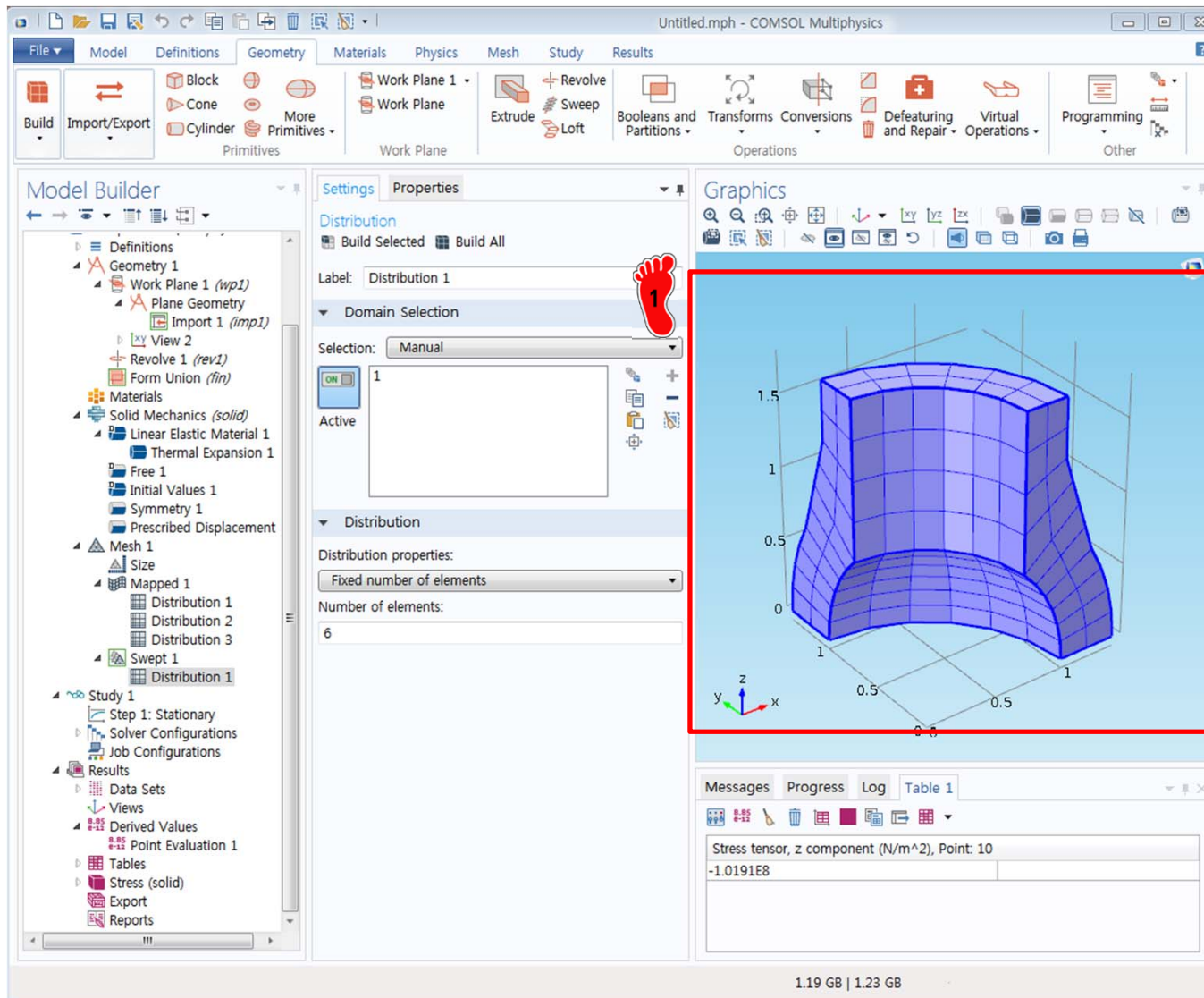
Derived Values 의 Point Evaluation 기능을 이용하여 10번 절점의 solid.sz 응력을 계산

결과값: -101.91 MPa

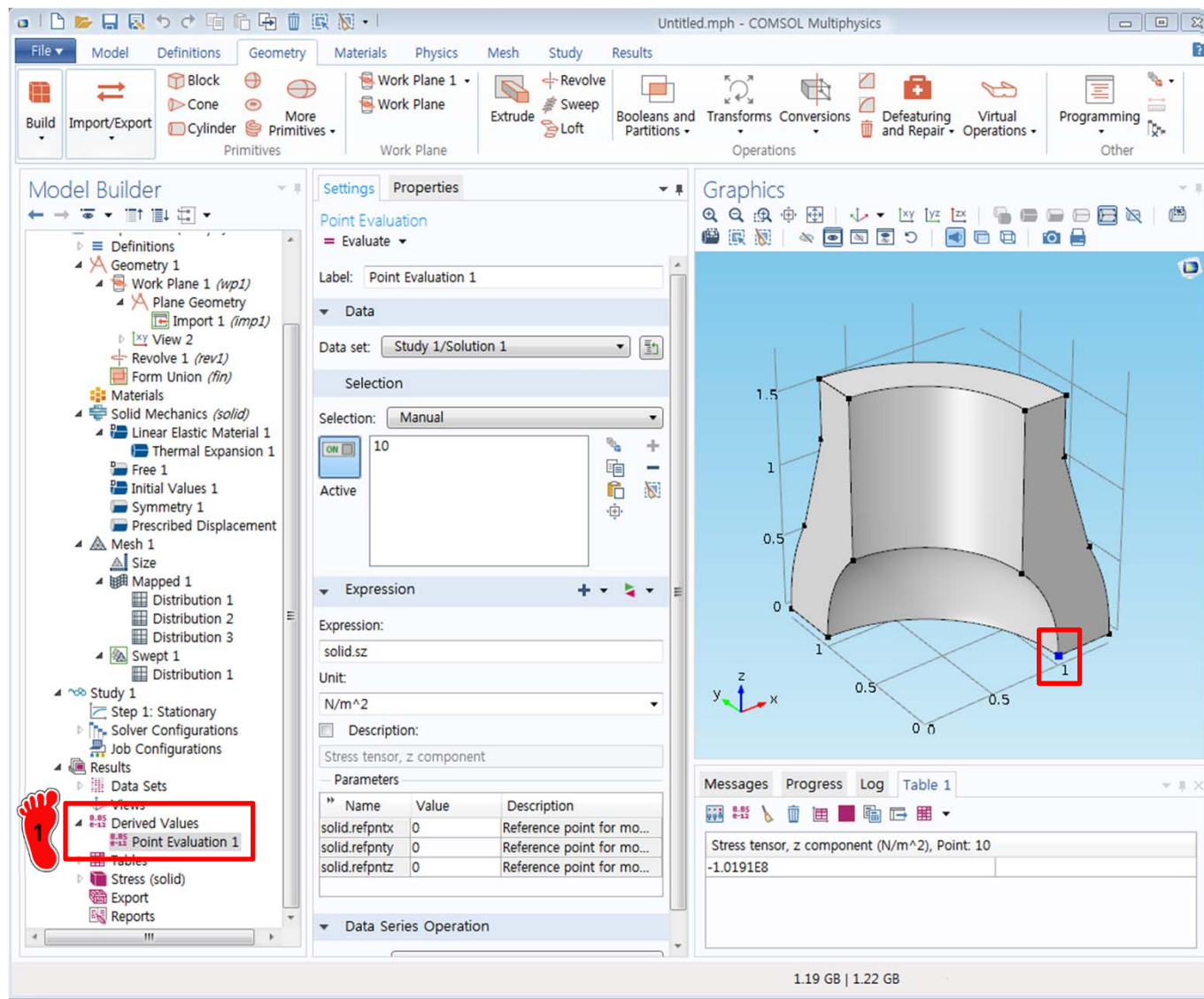
MESH



Fine mesh로 변경 후 해석
실행



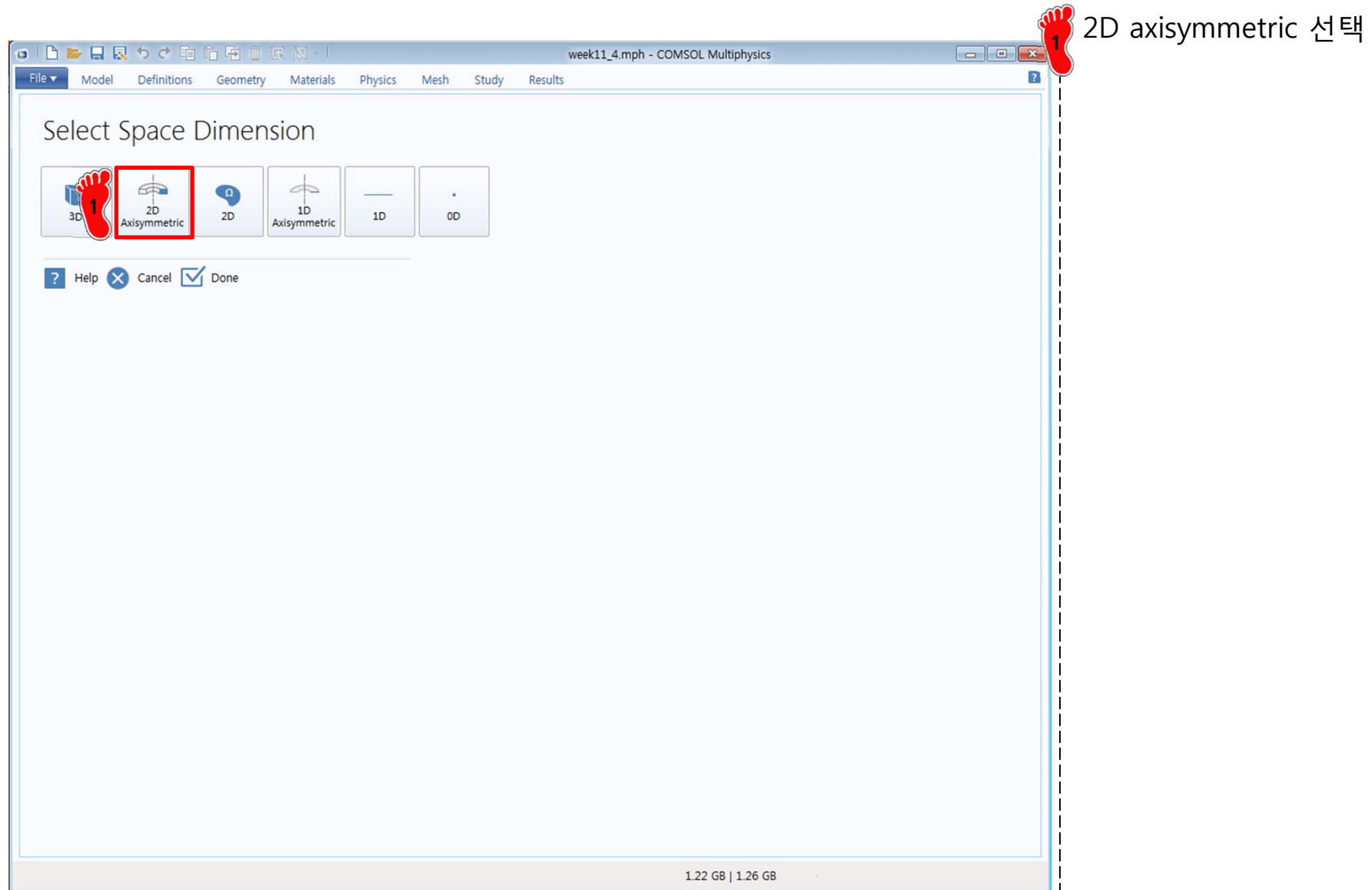
POST-PROCESSING



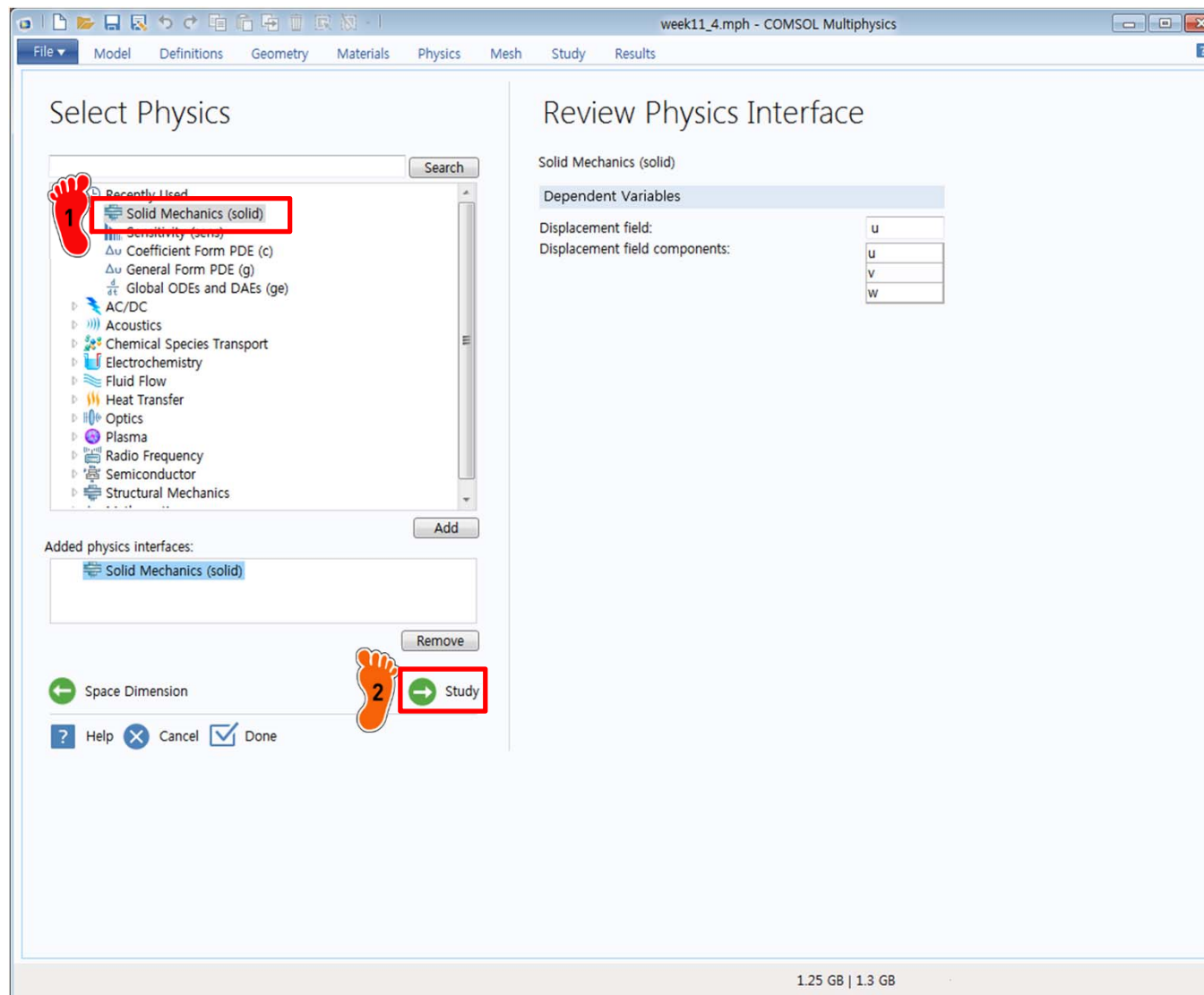
1 Derived Values 의 Point Evaluation 기능을 이용하여 10번 절점의 solid.sz 응력을 계산

결과값: -104.24 MPa

DIMENSION SELECTION



PHYSICS SELECTION



1 Structural Mechanics 의
Solid Mechanics 선택

2 Study 클릭

STUDY TYPE SELECTION

week11_4.mph - COMSOL Multiphysics

File Model Definitions Geometry Materials Physics Mesh Study Results

Select Study

Preset Studies

- Eigenfrequency
- Frequency Domain
- Frequency-Domain Modal
- Linear Buckling
- Modal Reduced Order Model
- Prestressed Analysis, Eigenfrequency
- Prestressed Analysis, Frequency Domain
- Stationary**
- Time Dependent
- Time-Dependent Modal

Added study:
Stationary

Added physics interfaces:
Solid Mechanics (solid)

Physics

Help 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.

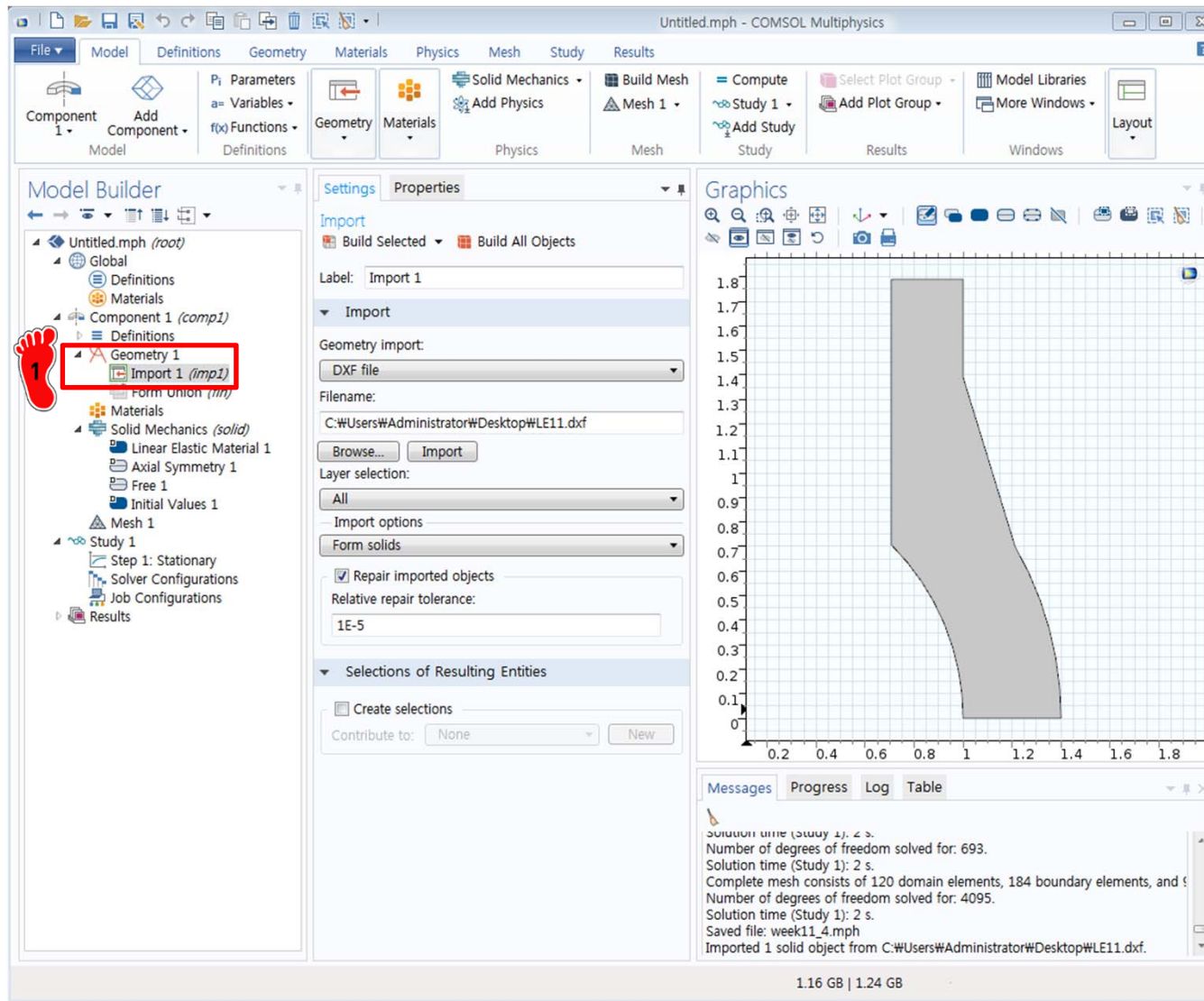
1. Stationary 선택

2. Done 클릭

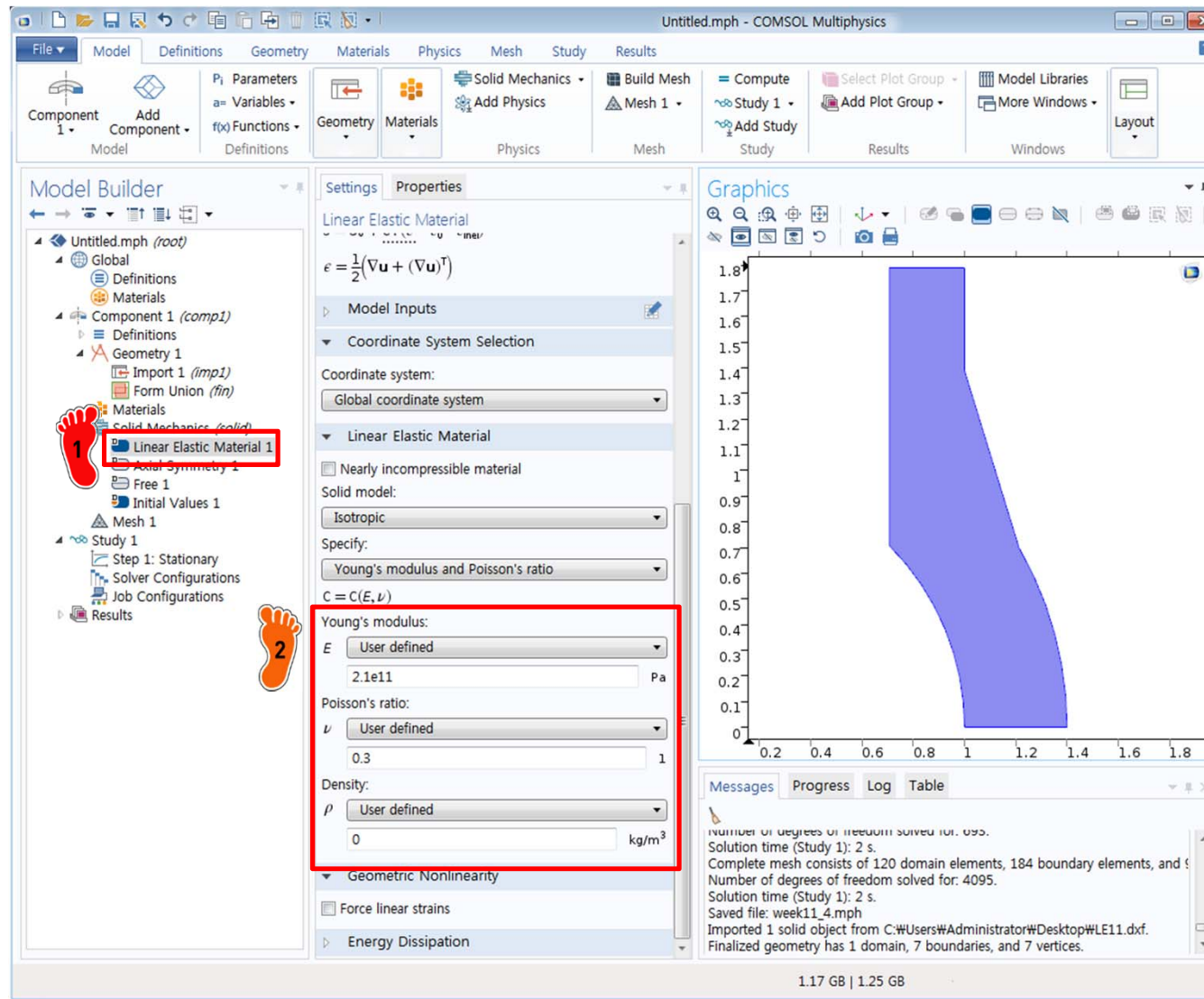
1.18 GB | 1.22 GB

GEOMETRY CREATION

1 Import 메뉴를 이용하여 주어진 캐드 기하형상을 삽입



MATERIAL PROPERTY

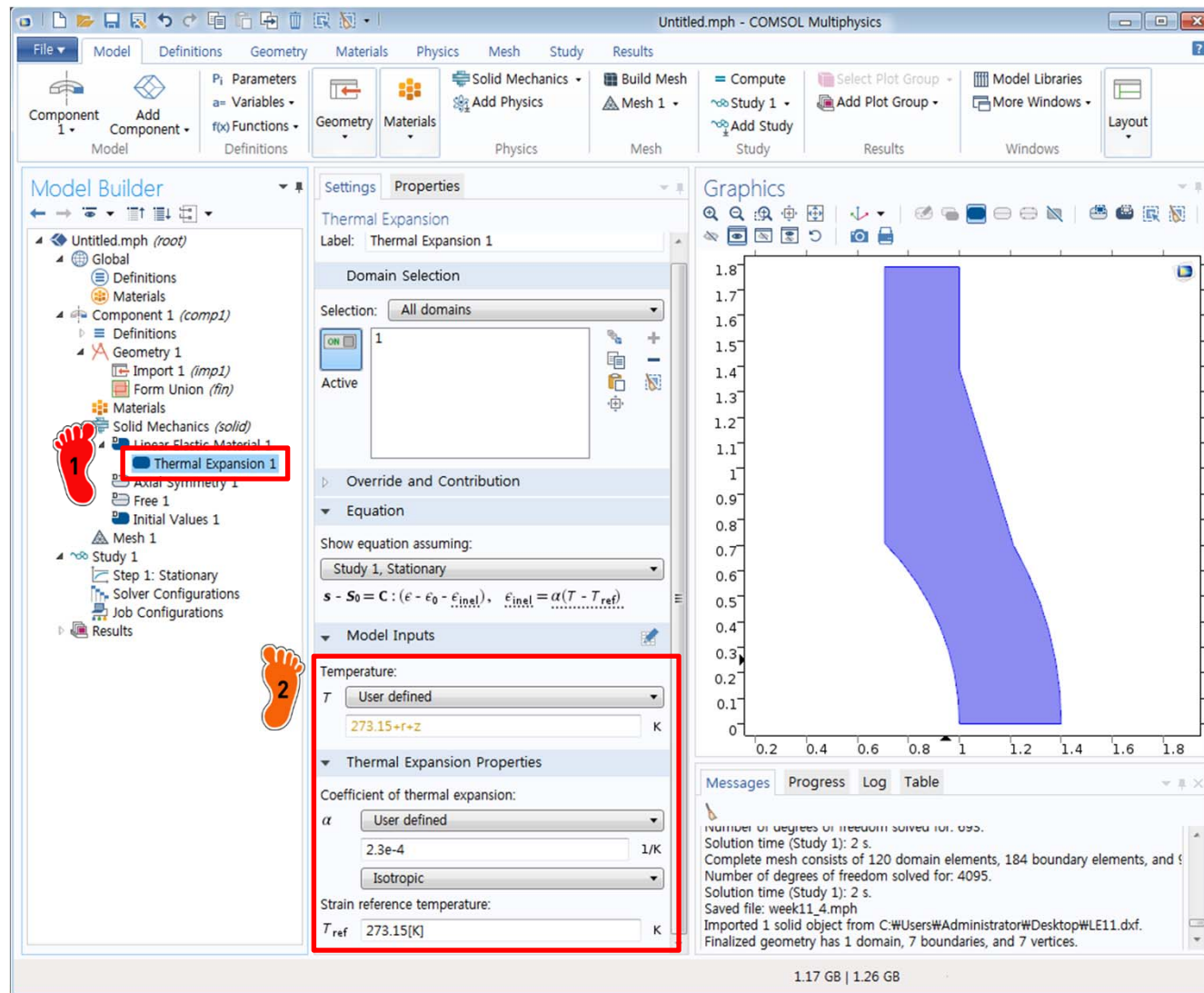


1 Linear Elastic Material 클릭

2 E: 2.1e11
mu: 0.3
rho : 0

입력

MATERIAL PROPERTY

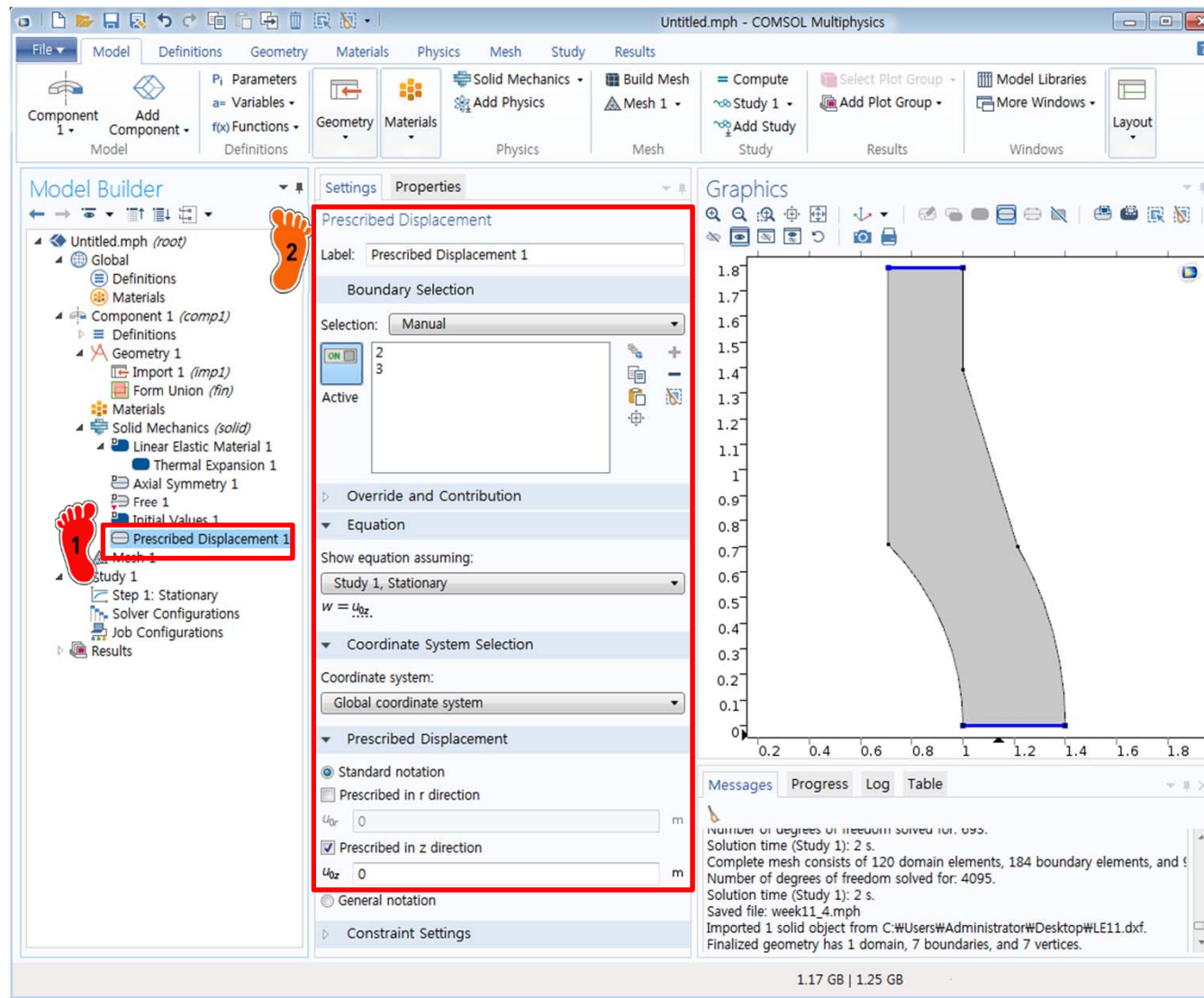


1 Linear Elastic Material 하위 메뉴로 Thermal Expansion 추가

2 Temperature:
 $273.15 + \sqrt{x^2 + y^2} + z$
 $\alpha: 2.3e-4$
 Strain reference temperature: 273.15

입력

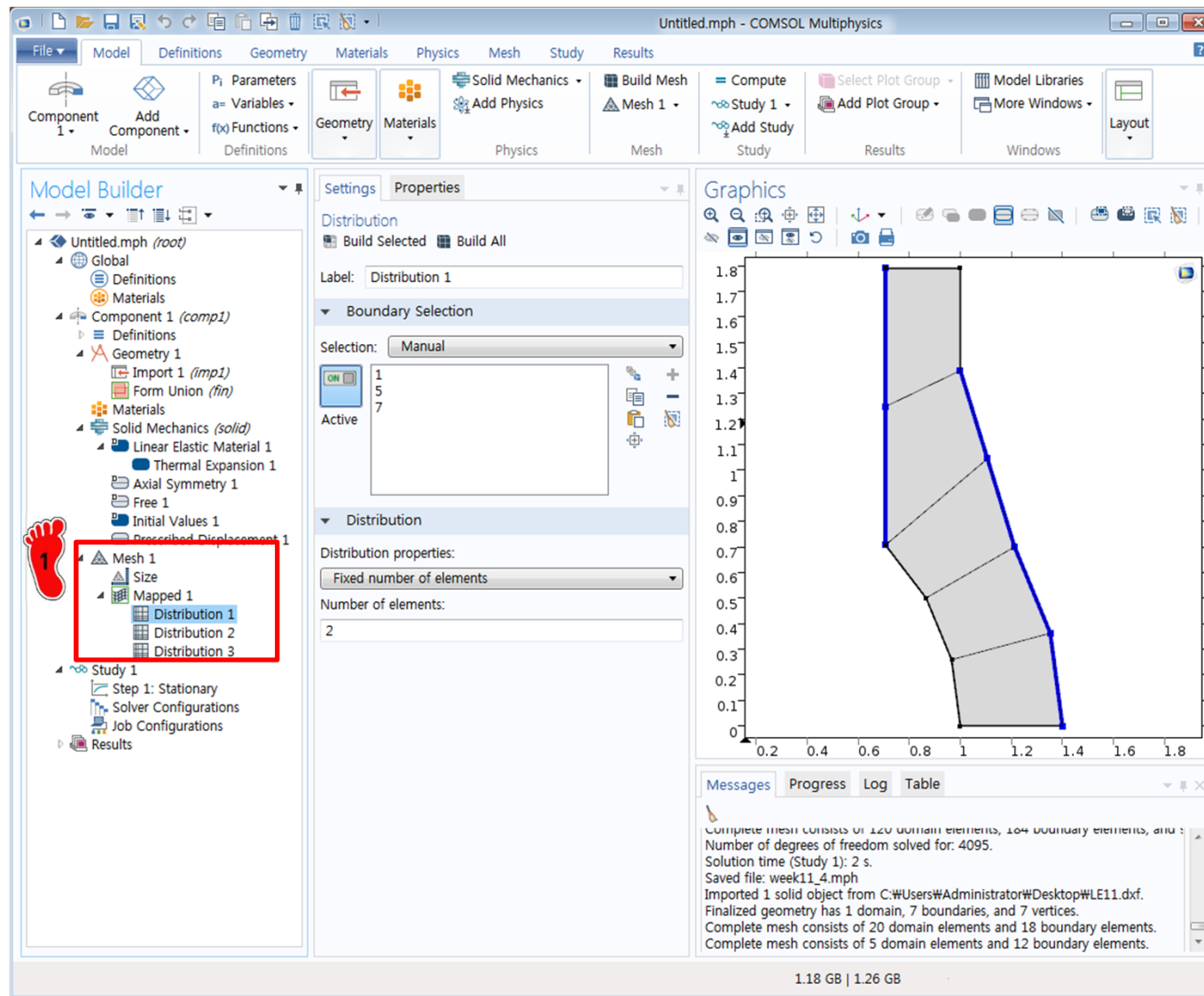
BOUNDARY CONDITION



1 Prescribed Displacement
경계조건 생성

2, 3번 경계 선택 후 z 방향
변위에 0 입력

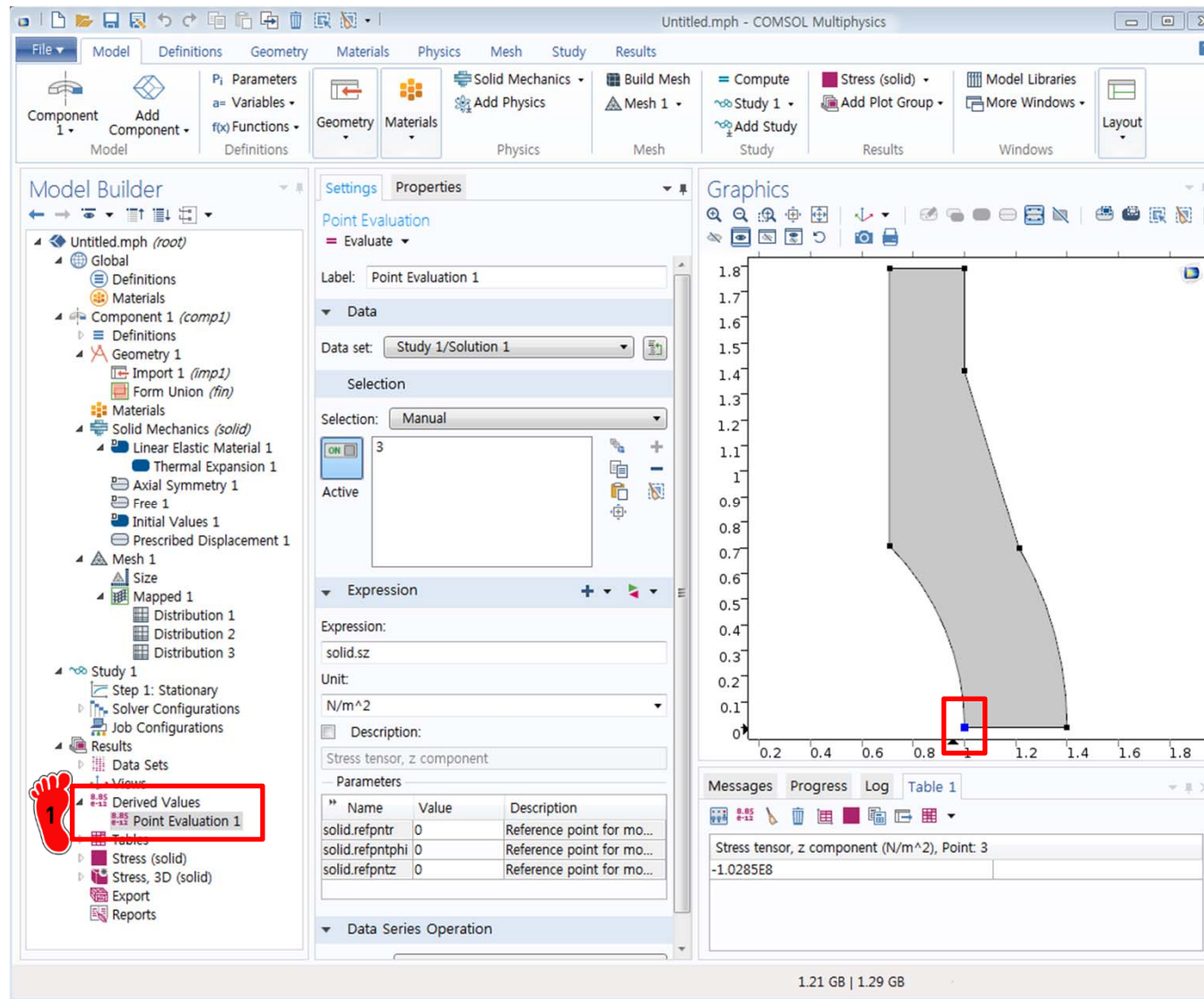
MESH



1 Mapped mesh 기능을 이용
하여 요소 생성

그 후, 해석실행

POST-PROCESSING



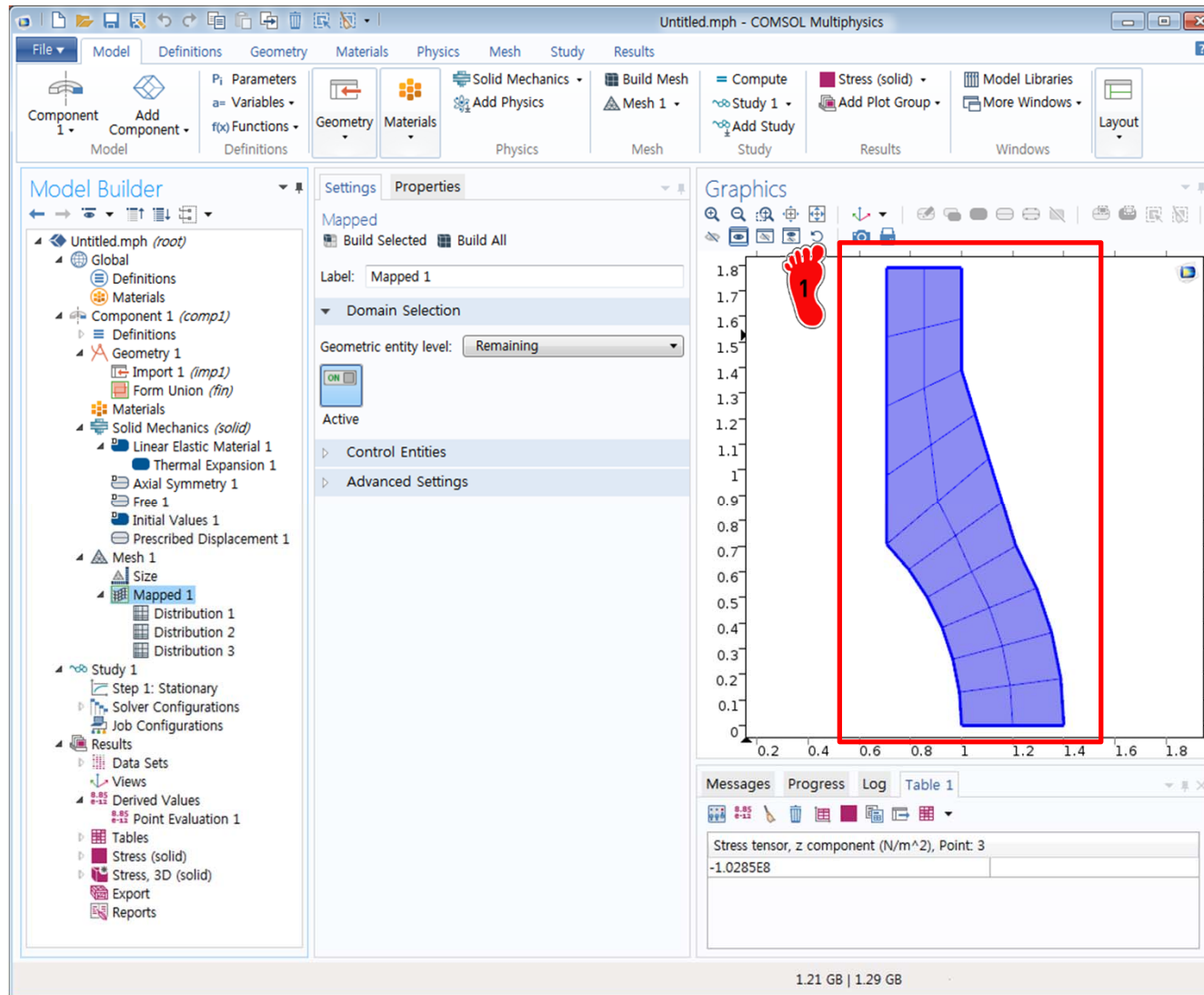
1 Derived Values 의 Point Evaluation 기능을 이용하여 3번 절점의 solid.sz 응력을 계산

결과값: -102.85 MPa

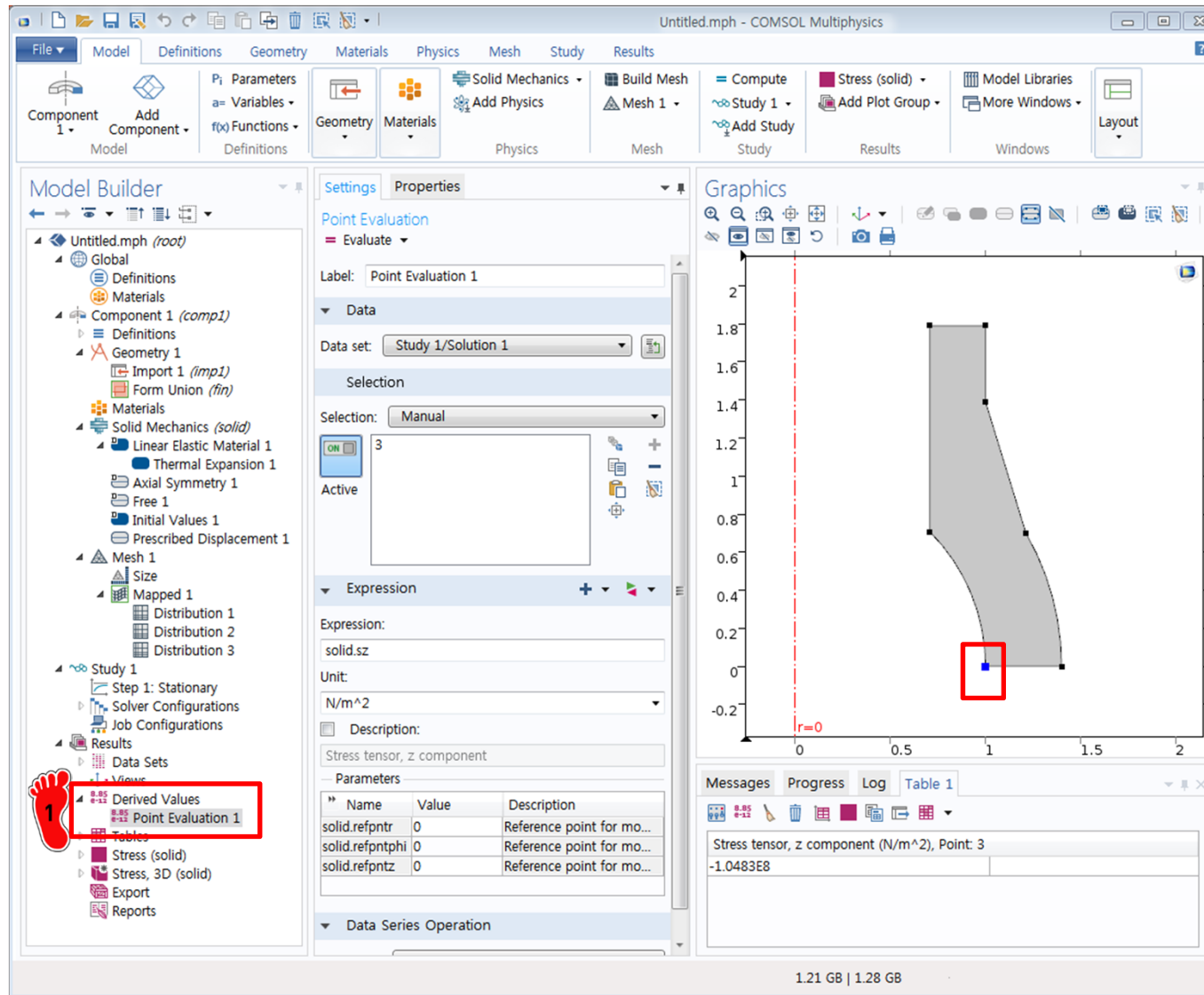
MESH



Fine mesh로 변경 후 해석
실행



POST-PROCESSING



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

결과값: -104.83 MPa

SUMMARY

Software	Order	Element	Coarse mesh		Fine mesh	
			Result	Error	Result	Error
COMSOL	2	axisymmetry	-102.85	2.05%	-104.83	0.16%
COMSOL	2	hexahedron	-101.91	2.94%	-104.24	0.72%

HOMEWORK: SOLID MECHANICS

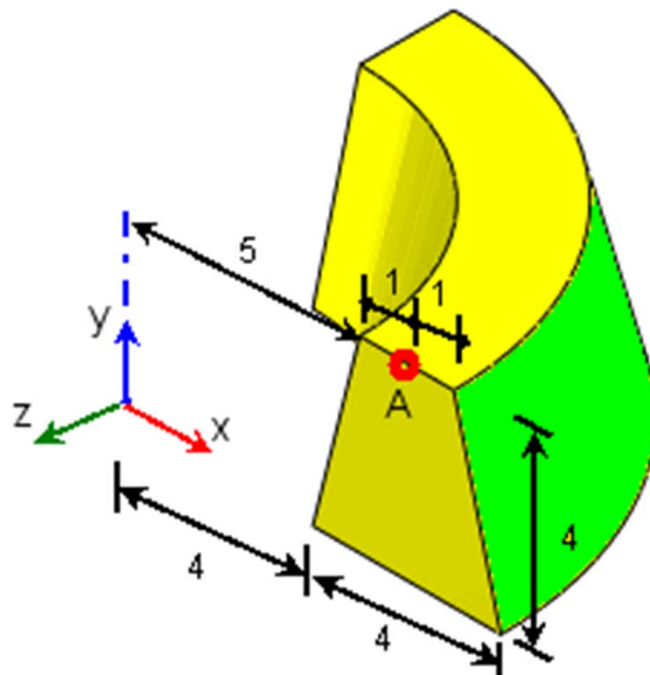
Tapered Thick Shell under Pressure and Gravity Loads

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/s² 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/m³

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