Solid Mechanics (plane stress/strain)

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

한양대학교 HANYANG UNIVERSITY
CDL Computational Design Lab

- **Stiffness matrix of Turner triangle**
- **2D plane stress model**
  - ✓ **Kirsh's problem**
- **Quarter model**

PLANE STRESS

PLANE STRESS

The plane stress variant of the 2D interface is useful for analyzing thin in-plane loaded plates. For a state of plane stress, the out-of-plane components of the stress tensor are zero.

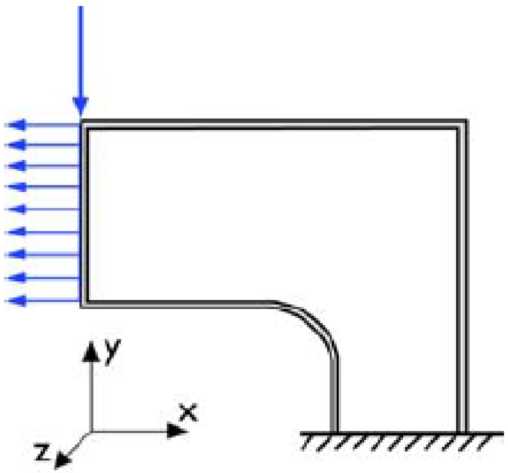


Figure 14-2: Plane stress models plates where the loads are only in the plane; it does not include any out-of-plane stress components.

The 2D interface for plane stress allows loads in the x and y directions, and it assumes that these are constant throughout the material's thickness, which can vary with x and y. The plane stress condition prevails in a thin flat plate in the xy-plane loaded only in its own plane and without any z direction restraint.

PLANE STRAIN

PLANE STRAIN

The plane strain variant of the 2D interface that assumes that all out-of-plane strain components of the total strain ε_z, ε_{yz}, and ε_{xz} are zero.

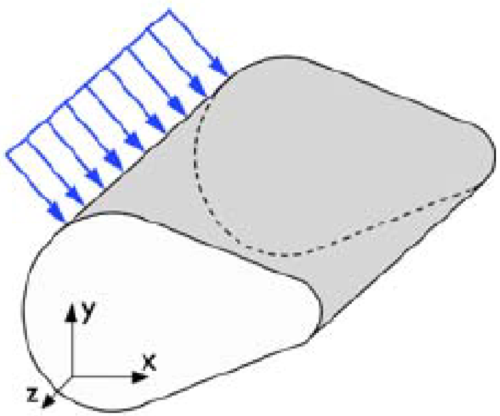


Figure 14-3: A geometry suitable for plane strain analysis.

Loads in the x and y directions are allowed. The loads are assumed to be constant throughout the thickness of the material, but the thickness can vary with x and y. The plane strain condition prevails in geometries, whose extent is large in the z direction compared to in the x and y directions, or when the z displacement is in some way restricted. One example is a long tunnel along the z-axis where it is sufficient to study a unit-depth slice in the xy-plane.

RELATION OF STRESS/STRAIN

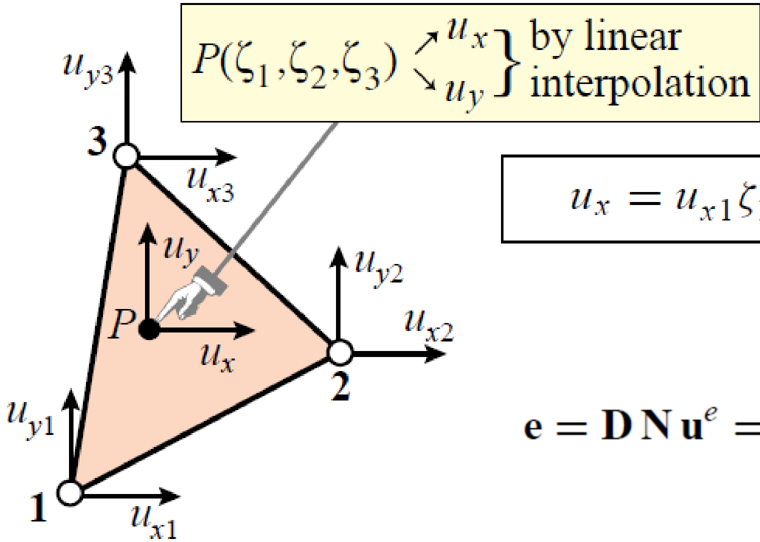
$$\begin{cases} \varepsilon_x = \dfrac{1}{E}\left[\sigma_x - \nu\left(\sigma_y + \sigma_z\right)\right] \\ \varepsilon_y = \dfrac{1}{E}\left[\sigma_y - \nu\left(\sigma_z + \sigma_x\right)\right] \\ \varepsilon_z = \dfrac{1}{E}\left[\sigma_z - \nu\left(\sigma_x + \sigma_y\right)\right] \end{cases} \xleftrightarrow{G=\frac{E}{2(1+\nu)}} \begin{cases} \gamma_{xy} = \dfrac{1}{G}\tau_{xy} \\ \gamma_{yz} = \dfrac{1}{G}\tau_{yz} \\ \gamma_{zx} = \dfrac{1}{G}\tau_{zx} \end{cases}$$

$$\text{plane stress}\left(\sigma_{zz} = \sigma_{xz} = \sigma_{yz} = 0\right): \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \dfrac{1-\nu}{2} \end{bmatrix}\begin{bmatrix} e_{xx} \\ e_{yy} \\ 2e_{xy} \end{bmatrix}$$

$$\text{plane strain}\left(e_{zz} = e_{xz} = e_{yz} = 0\right): \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \frac{E}{(1+\nu)(2-\nu)}\begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \dfrac{1}{2}(1-2\nu) \end{bmatrix}\begin{bmatrix} e_{xx} \\ e_{yy} \\ 2e_{xy} \end{bmatrix}$$

- **Stiffness matrix of Turner triangle**
- **2D plane stress model**
  - ✓ **Kirsh’s problem**
- **Quarter model**

TURNER TRIANGLE (1)



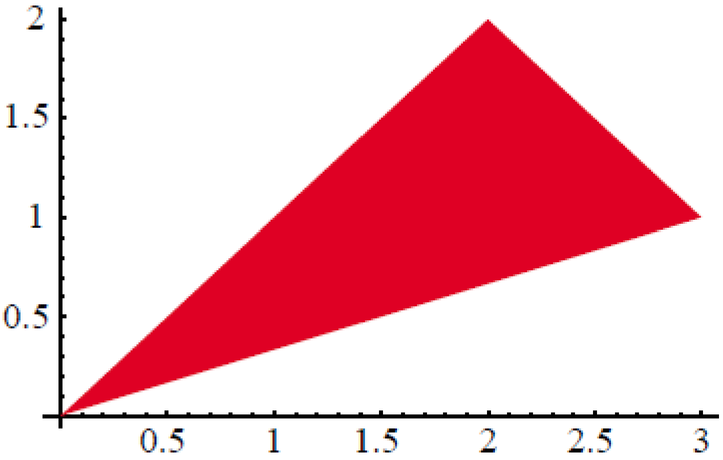
$$\boxed{u_x = u_{x1}\zeta_1 + u_{x2}\zeta_2 + u_{x3}\zeta_3, \qquad u_y = u_{y1}\zeta_1 + u_{y2}\zeta_2 + u_{y3}\zeta_3.}$$

$$\mathbf{e} = \mathbf{D}\,\mathbf{N}\,\mathbf{u}^e = \frac{1}{2A}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix}\begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \mathbf{B}\,\mathbf{u}^e$$

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} \\ E_{12} & E_{22} & E_{23} \\ E_{13} & E_{23} & E_{33} \end{bmatrix}\begin{bmatrix} e_{xx} \\ e_{yy} \\ 2e_{xy} \end{bmatrix} = \mathbf{E}\,\mathbf{e}$$

$$\boxed{\mathbf{K}^e = A\,h\,\mathbf{B}^T\,\mathbf{E}\,\mathbf{B} = \frac{h}{4A}\begin{bmatrix} y_{23} & 0 & x_{32} \\ 0 & x_{32} & y_{23} \\ y_{31} & 0 & x_{13} \\ 0 & x_{13} & y_{31} \\ y_{12} & 0 & x_{21} \\ 0 & x_{21} & y_{12} \end{bmatrix}\begin{bmatrix} E_{11} & E_{12} & E_{13} \\ E_{12} & E_{22} & E_{23} \\ E_{13} & E_{23} & E_{33} \end{bmatrix}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix}.}$$

TURNER TRIANGLE (2)

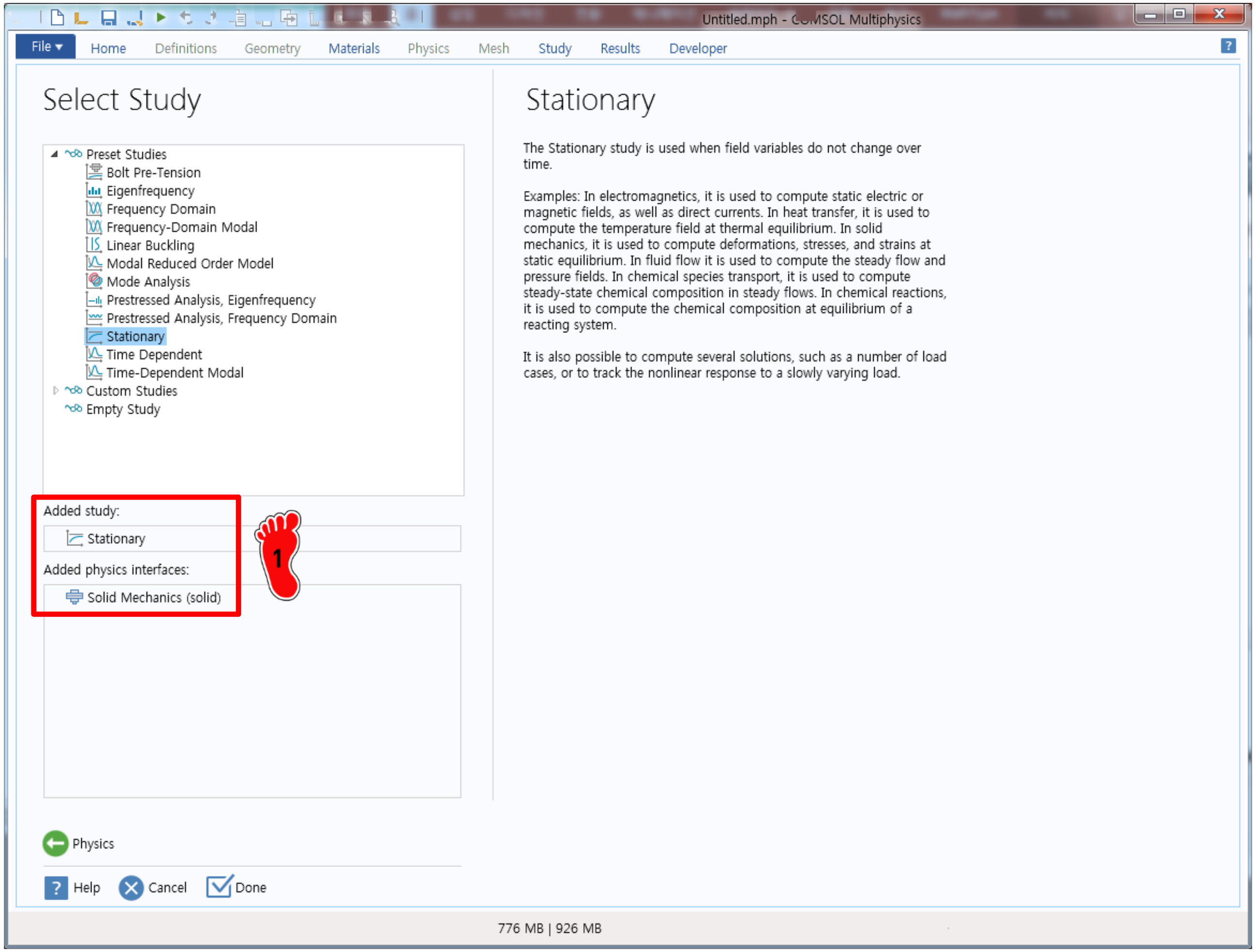


$$E_0 = 60\ ,\ \nu = 0.25\ ,\ h = 1$$

$$\mathbf{E}_{\text{plane stress}} = \begin{bmatrix} 64 & 16 & 0 \\ 16 & 64 & 0 \\ 0 & 0 & 24 \end{bmatrix} \qquad \mathbf{E}_{\text{plane strain}} = \begin{bmatrix} 72 & 24 & 0 \\ 24 & 72 & 0 \\ 0 & 0 & 24 \end{bmatrix}$$

$$\mathbf{Ke}_{\text{plane stress}} = \begin{bmatrix} 11 & 5 & -10 & -2 & -1 & -3 \\ 5 & 11 & 2 & 10 & -7 & -21 \\ -10 & 2 & 44 & -20 & -34 & 18 \\ -2 & 10 & -20 & 44 & 22 & -54 \\ -1 & -7 & -34 & 22 & 35 & 15 \\ 3 & -21 & 18 & -54 & -15 & 75 \end{bmatrix} \qquad \mathbf{Ke}_{\text{plane strain}} = \begin{bmatrix} 12 & 6 & -12 & 0 & 0 & -6 \\ 6 & 12 & 0 & 12 & -6 & -24 \\ -12 & 0 & 48 & -24 & -36 & 24 \\ 0 & 12 & -24 & 48 & 24 & -60 \\ 0 & -6 & -36 & 24 & 36 & -18 \\ -6 & -24 & 24 & -60 & -18 & 84 \end{bmatrix}$$

ENVIRONMENT SETTING

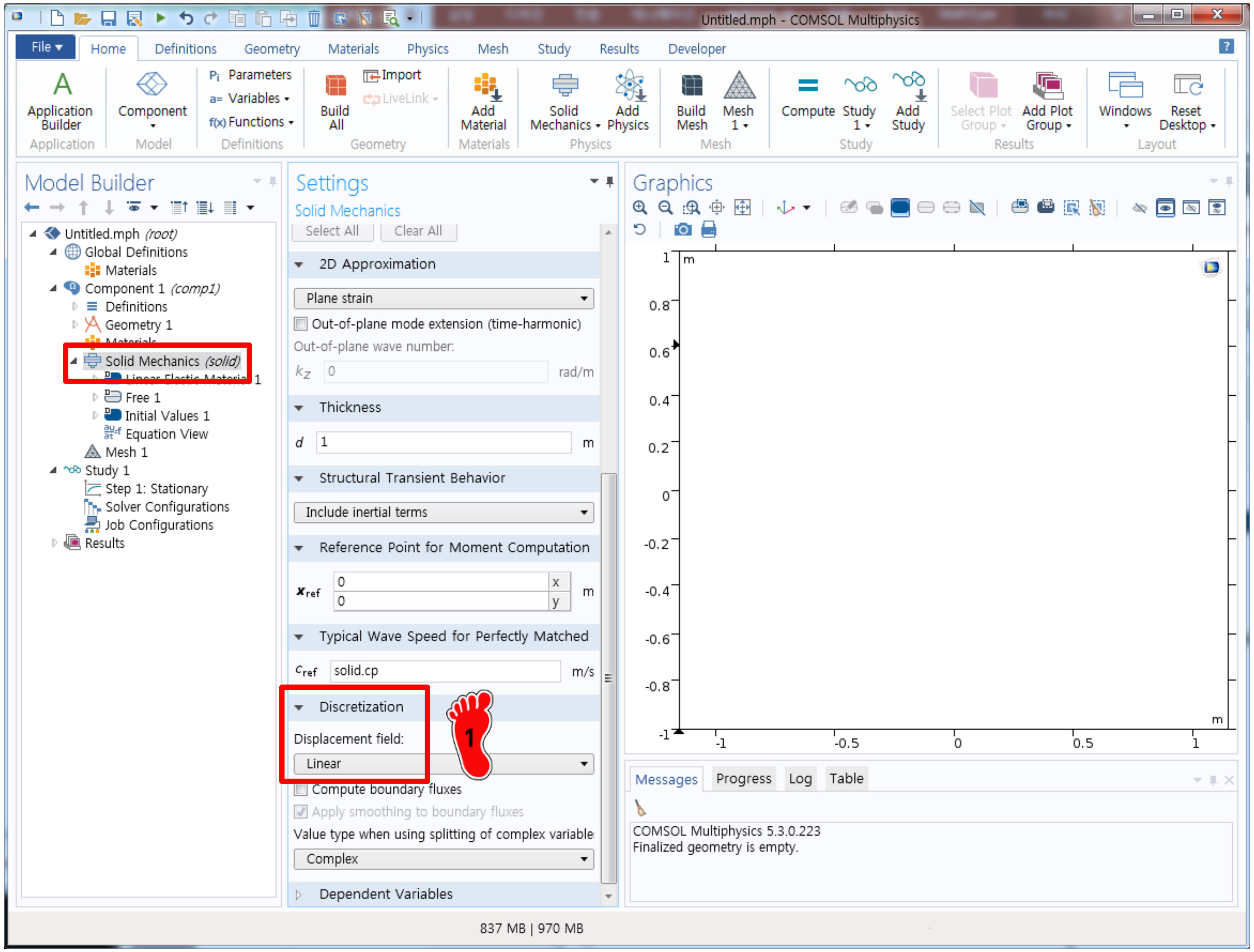


1

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

Done 클릭

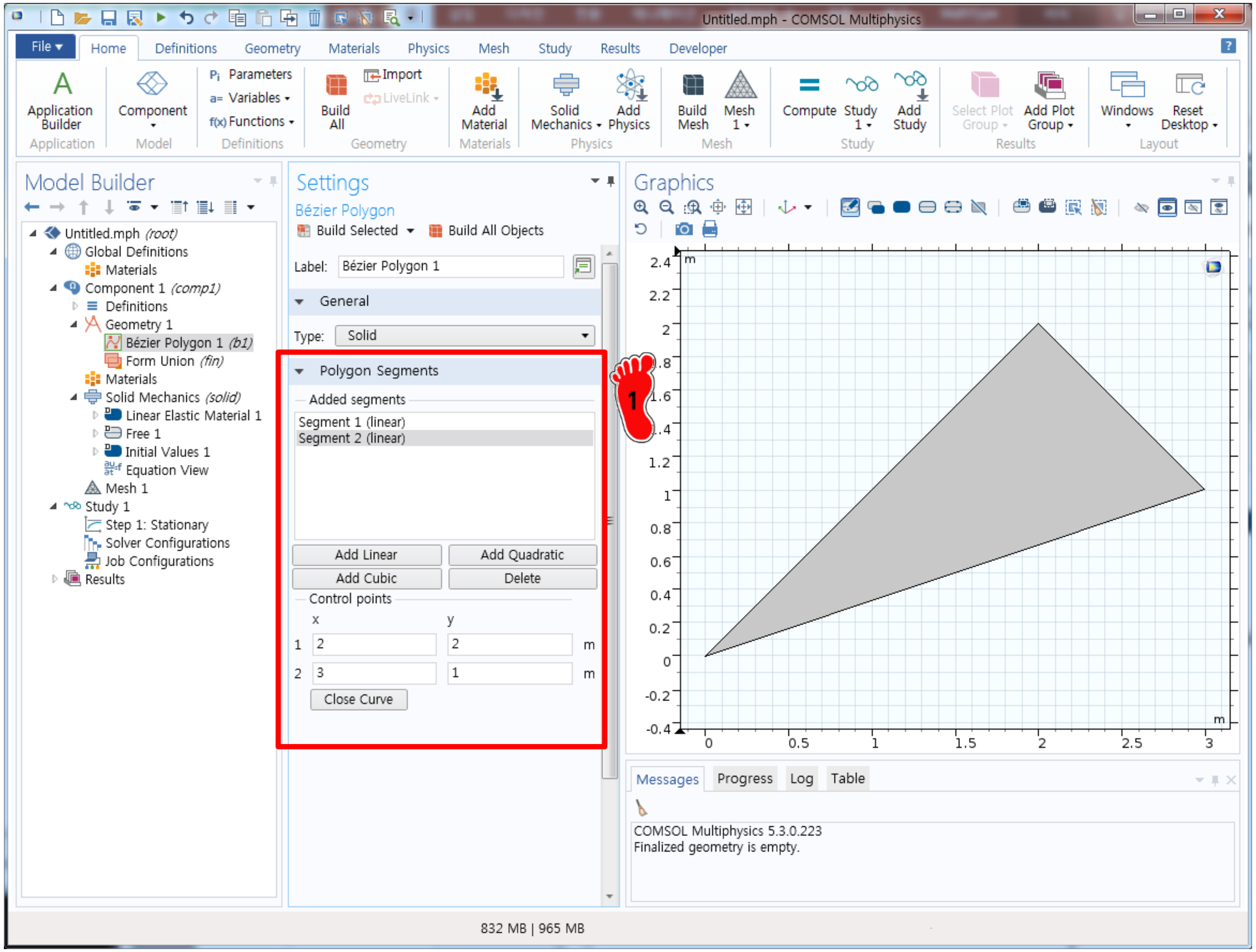
DISCRETIZATION



1 Solid Mechanics 클릭

Discretization 메뉴에서 Displacement field를 Linear로 변경

GEOMETRY CREATION

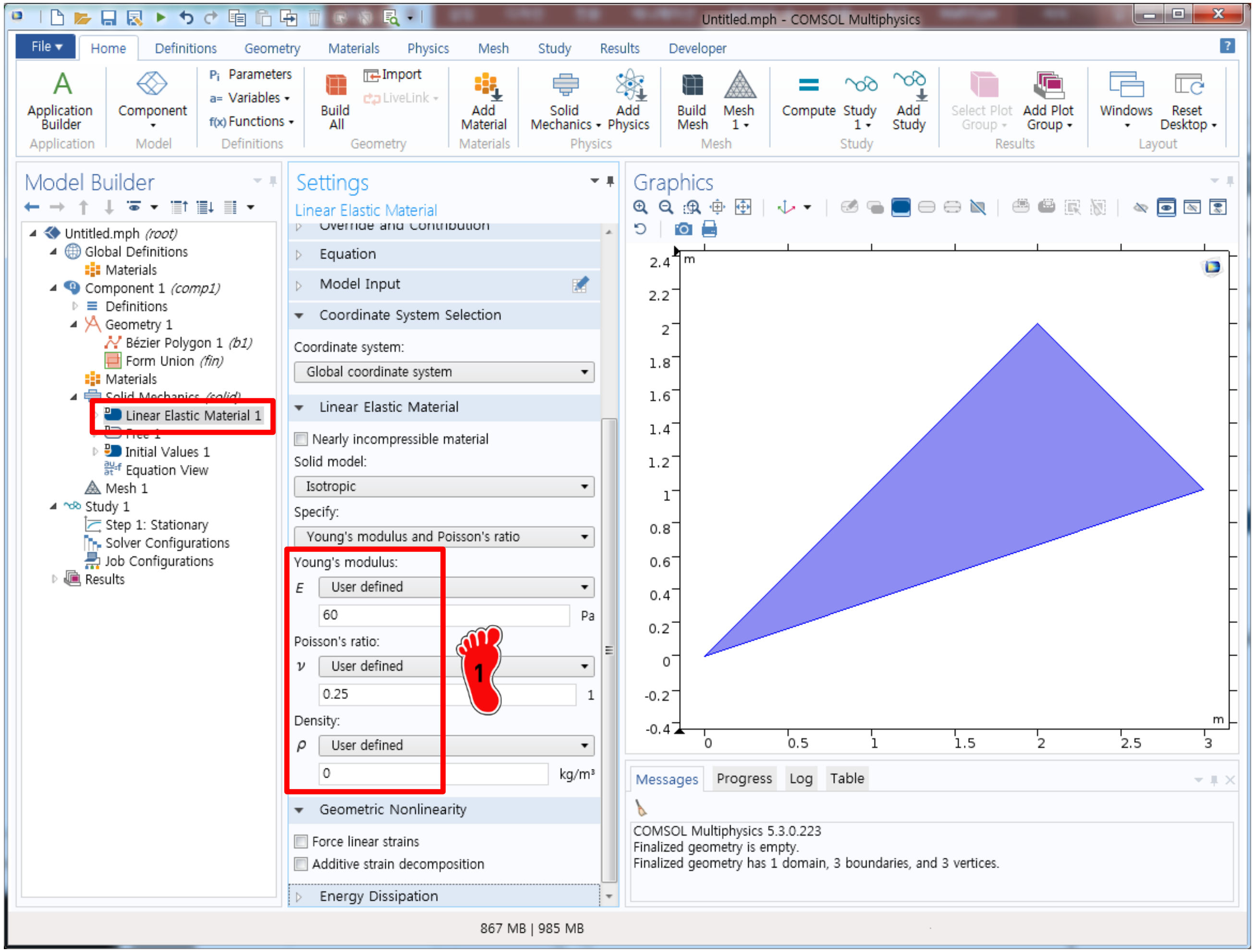


1 Bezier Polygon을 이용하여 삼각형 기하형상 생성

Seg. 1 : (0,0),(2,2)

Seg. 2 : (2,2),(3,1)

MATERIAL PROPERTY



1 Linear Elastic Material 클릭

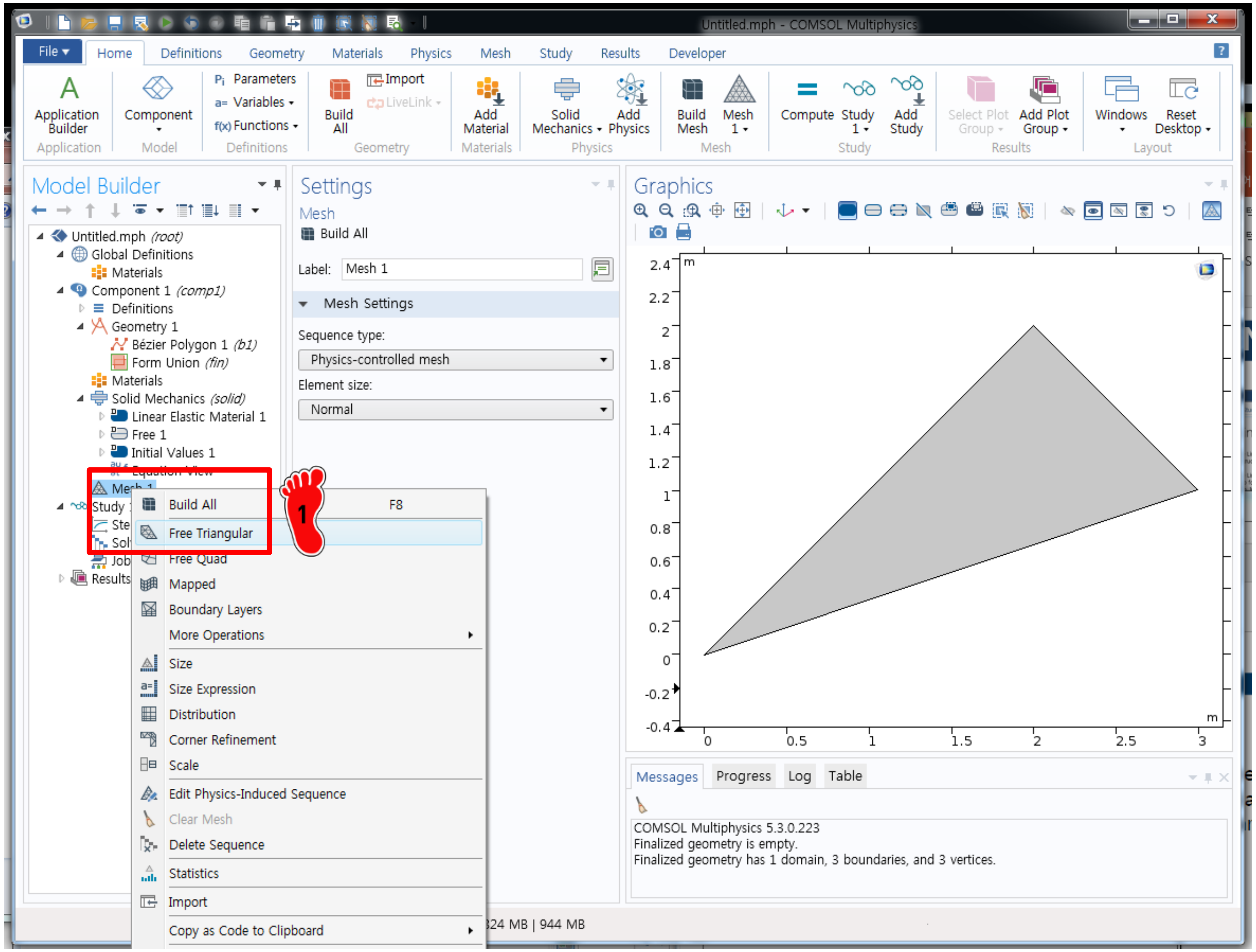
물성치 입력

E: 60

mu: 0.25

rho : 0

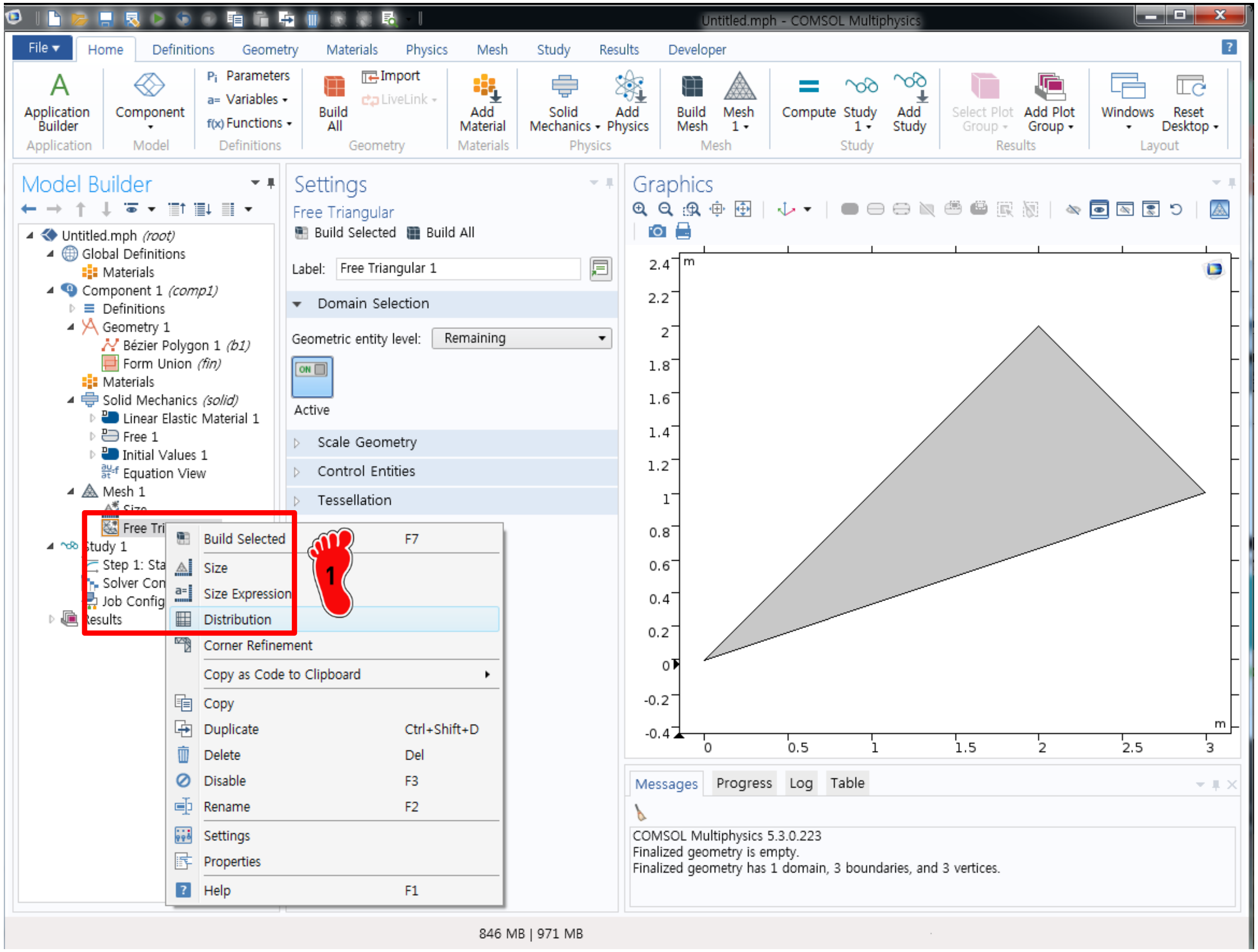
MESH



1 Mesh 마우스 우클릭

Free Triangular 클릭

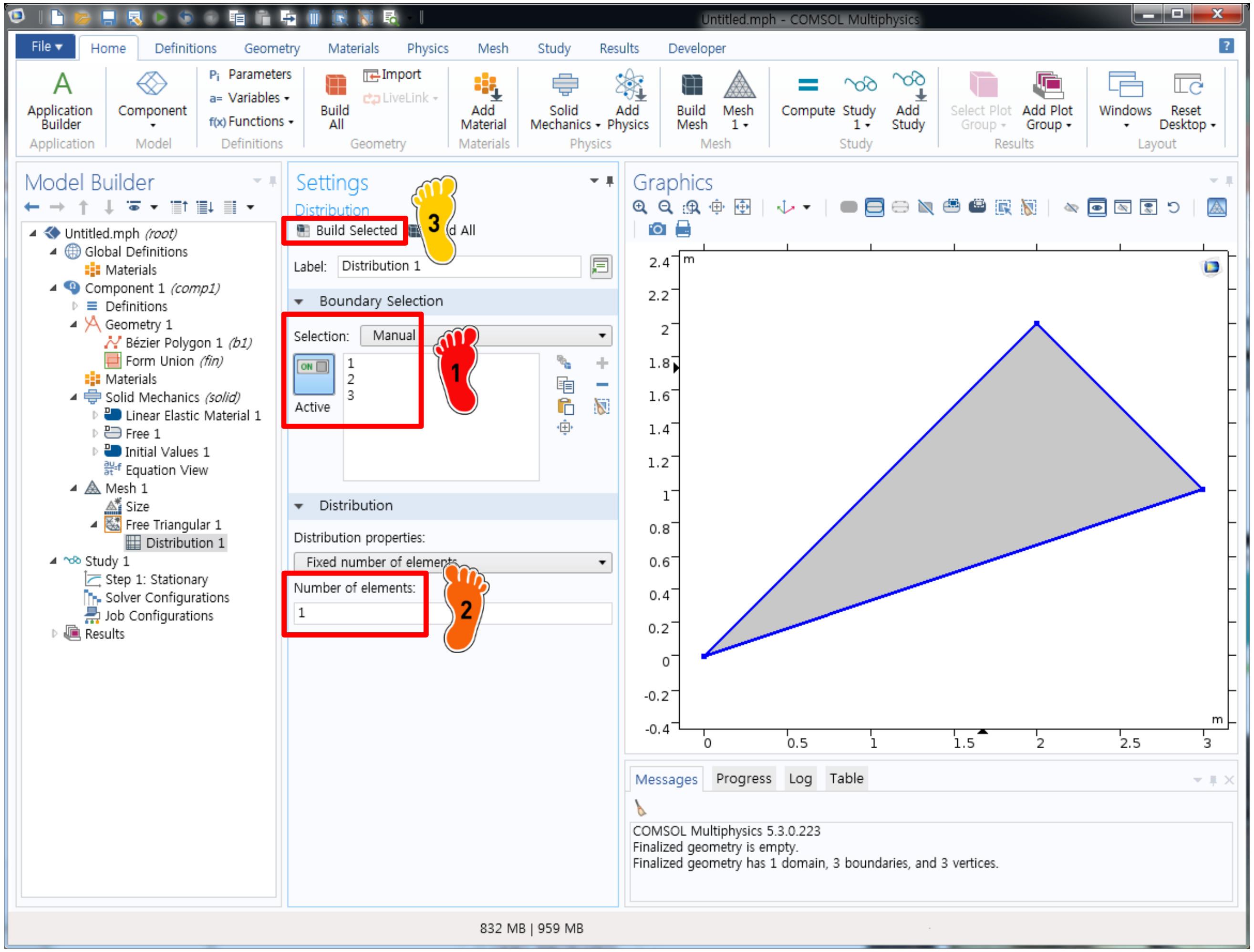
MESH



1 Free Triangular 마우스 우클릭

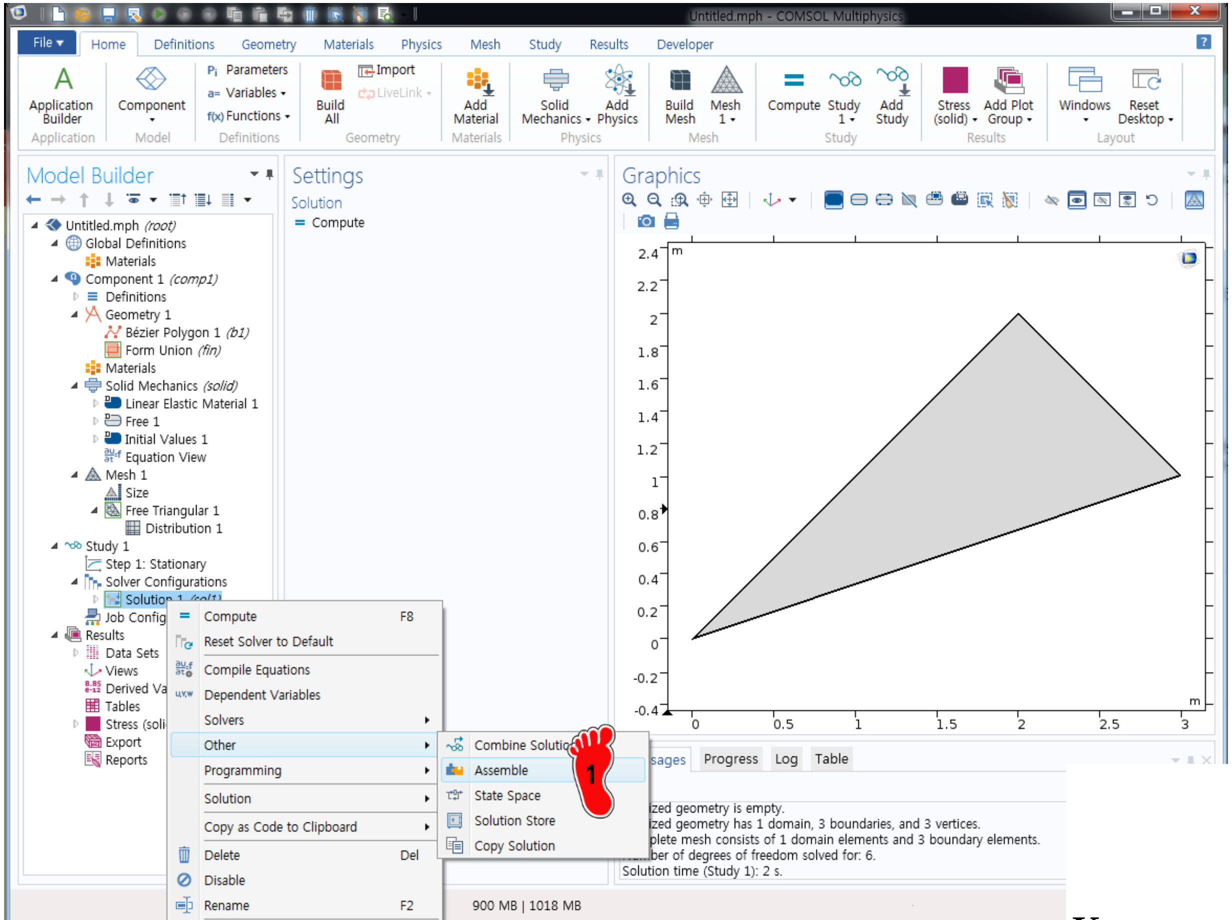
Distribution 클릭

MESH



1. 3개의 boundary 선택
2. Number of elements를 1로 설정
3. 1개의 삼각형 요소 생성

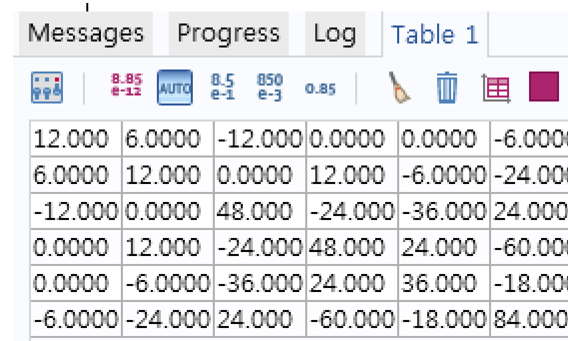
STIFFNESS MATRIX



1

해석 실행 후,
Assemble 탭에서 stiffness matrix를 선택 후 compute

Result → Derived Values에서 System Matrix 클릭 후 Stiffness matrix 확인



12.000	6.0000	-12.000	0.0000	0.0000	-6.0000
6.0000	12.000	0.0000	12.000	-6.0000	-24.000
-12.000	0.0000	48.000	-24.000	-36.000	24.000
0.0000	12.000	-24.000	48.000	24.000	-60.000
0.0000	-6.0000	-36.000	24.000	36.000	-18.000
-6.0000	-24.000	24.000	-60.000	-18.000	84.000

$$\mathbf{Ke}_{\text{plane strain}} = \begin{bmatrix} 12 & 6 & -12 & 0 & 0 & -6 \\ 6 & 12 & 0 & 12 & -6 & -24 \\ -12 & 0 & 48 & -24 & -36 & 24 \\ 0 & 12 & -24 & 48 & 24 & -60 \\ 0 & -6 & -36 & 24 & 36 & -18 \\ -6 & -24 & 24 & -60 & -18 & 84 \end{bmatrix}$$

- **Stiffness matrix of Turner triangle**

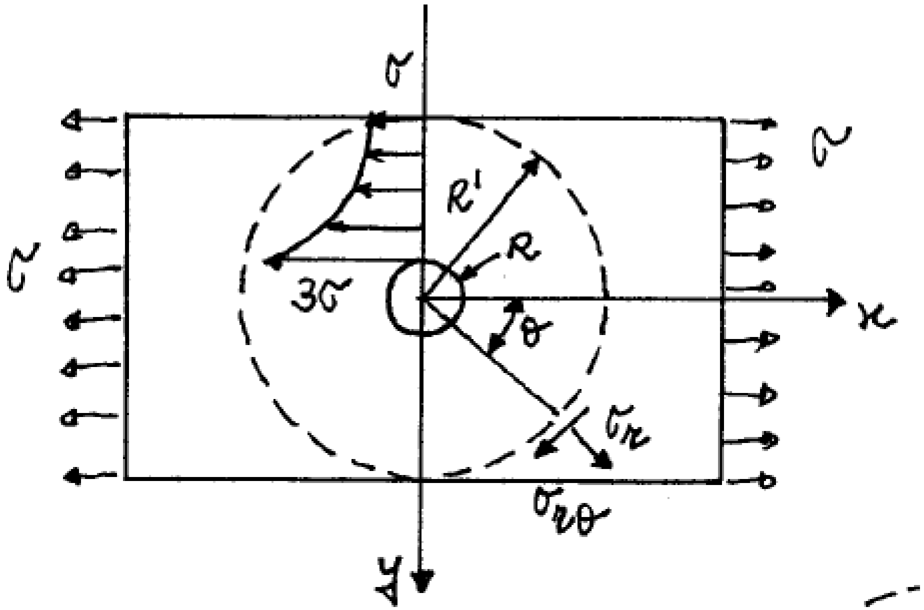
- **2D plane stress model**

  ✓ **Kirsh's problem**

- **Quarter model**

KIRSCH'S PROBLEM: THEORY

Infinite plate containing a circular hole (Kirsh, G. (1898), V.D.I., 42, 797-807)



Consider portion of plate within concentric circle of radius $R' \gg R$ so that stress field is not perturbed by hole (Saint-Venant's Principle)

$$\text{stress field at } r = R' \left(\text{Mohr's cirle}\right): \begin{cases} \sigma_r = \dfrac{\sigma}{2}\left(1+\cos 2\theta\right) \\ \sigma_{r\theta} = -\dfrac{\sigma}{2}\sin 2\theta \end{cases}$$

$$\text{solution: } \begin{cases} \sigma_r = \dfrac{\sigma}{2}\left(1-\dfrac{R^2}{r^2}\right)+\dfrac{\sigma}{2}\left(1+3\dfrac{R^4}{r^4}-4\dfrac{R^2}{r^2}\right)\cos 2\theta \\ \sigma_\theta = \dfrac{\sigma}{2}\left(1+\dfrac{R^2}{r^2}\right)-\dfrac{\sigma}{2}\left(1+3\dfrac{R^4}{r^4}\right)\cos 2\theta \\ \sigma_{r\theta} = -\dfrac{\sigma}{2}\left(1-3\dfrac{R^4}{r^4}+2\dfrac{R^2}{r^2}\right)\sin 2\theta \end{cases}$$

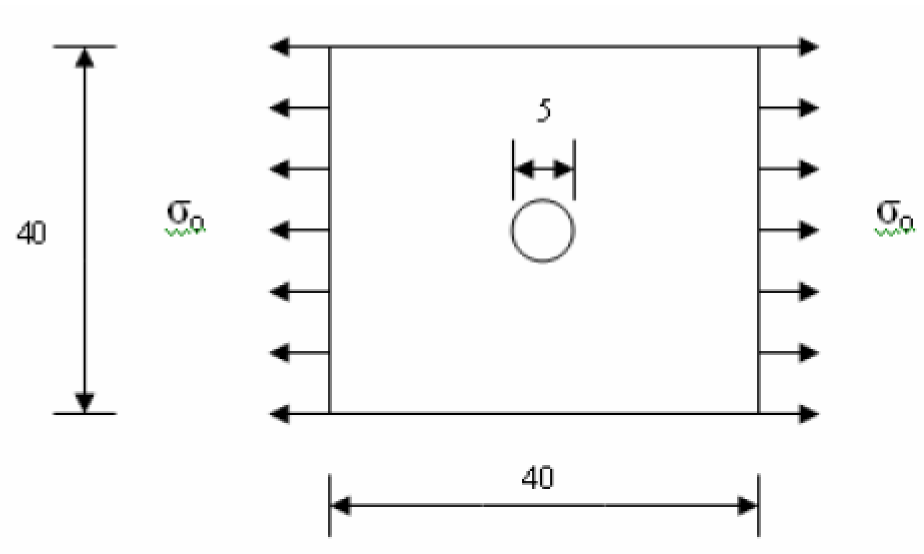
for $r = R$:

$$\left.\begin{aligned} \sigma_r &= 0 \\ \sigma_\theta &= \sigma\left(1-2\cos 2\theta\right) \\ \sigma_{r\theta} &= 0 \end{aligned}\right\} \rightarrow \begin{cases} \text{max: } \sigma_\theta = 3\sigma \ @\ \theta = \dfrac{\pi}{2}, \dfrac{3\pi}{2} \\ \text{min: } \sigma_\theta = -\sigma \ @\ \theta = 0, \pi \end{cases}$$

stress concentration factor = 3 independent of R

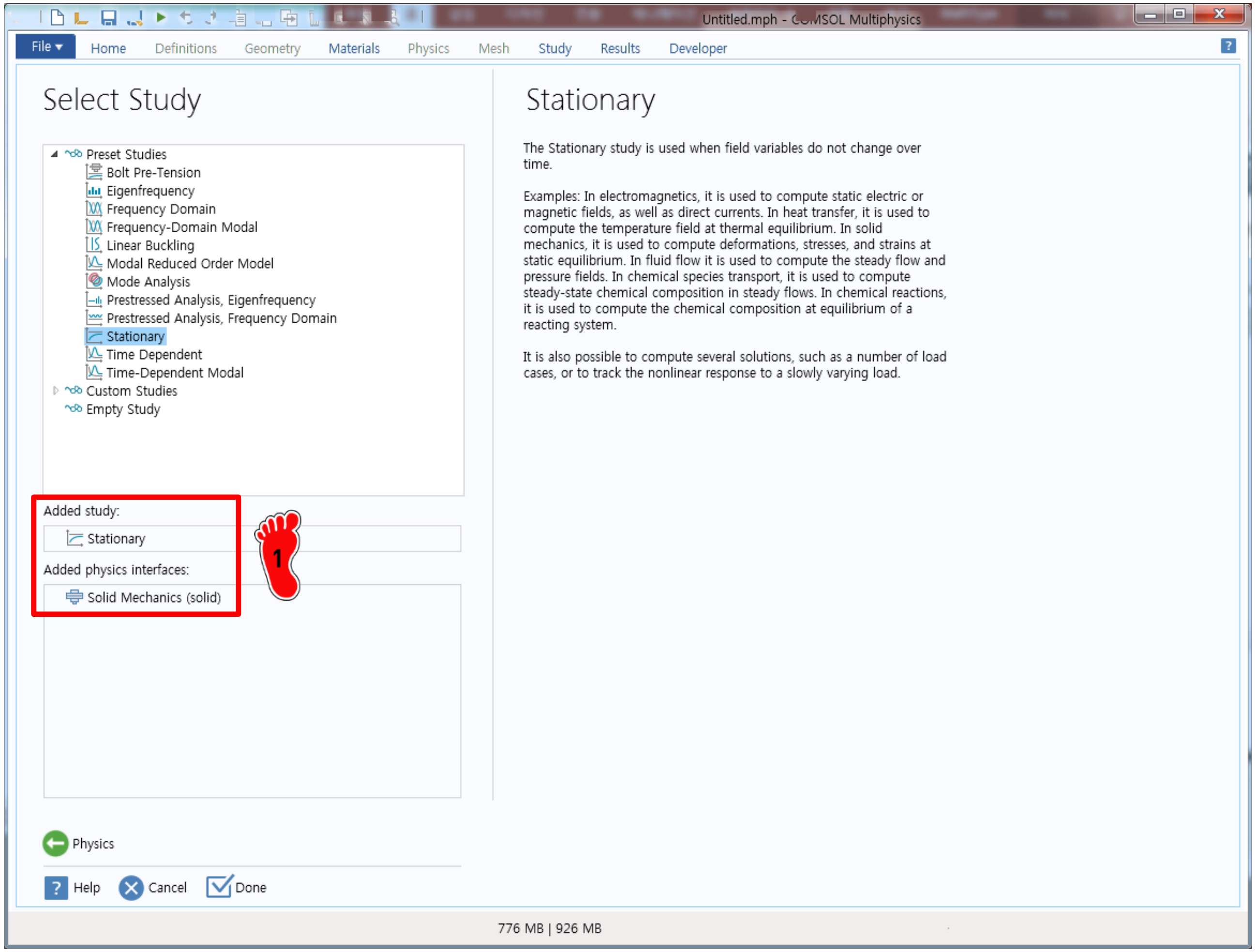
solution applicable to finite plates with width > $4R$

KIRSCH'S PROBLEM: FEM



- 2D approximation
  - Plane stress
  - Plane stain
- Material Properties
  - $E = 200 \times 10^9$
  - $\nu = 0.3$
- Element Properties
  - Thickness = 1?
- Loads: $\sigma_0 = 1$
- BCs: none

ENVIRONMENT SETTING



1

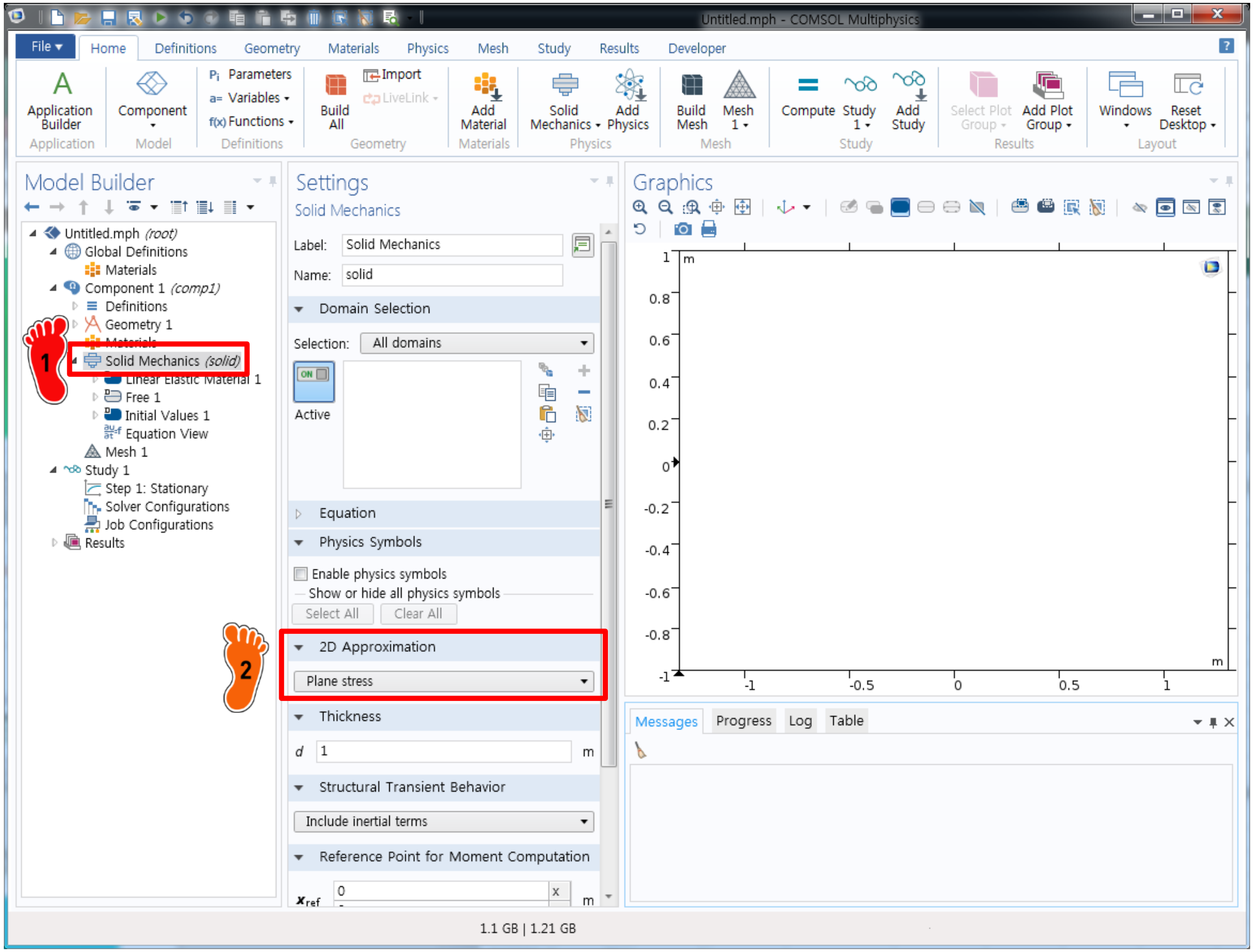
Dimension : 2D
Physics :
Structural Mechanics
→ Solid Mechanics
Study : Stationary

Done 클릭

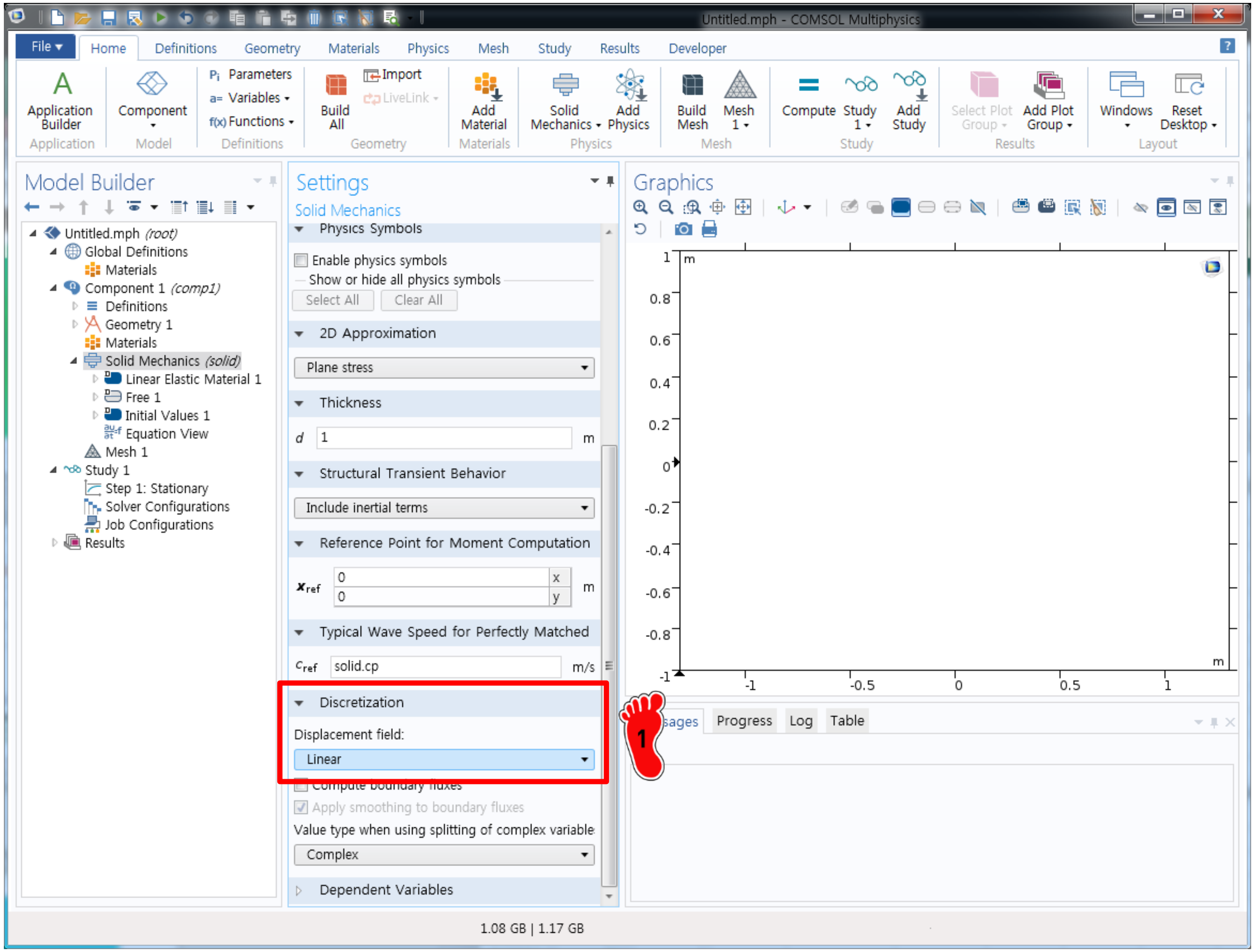
PLANE STRESS

1 Solid Mechanics 클릭

2 2D Approximation
Plane stress 로 변경

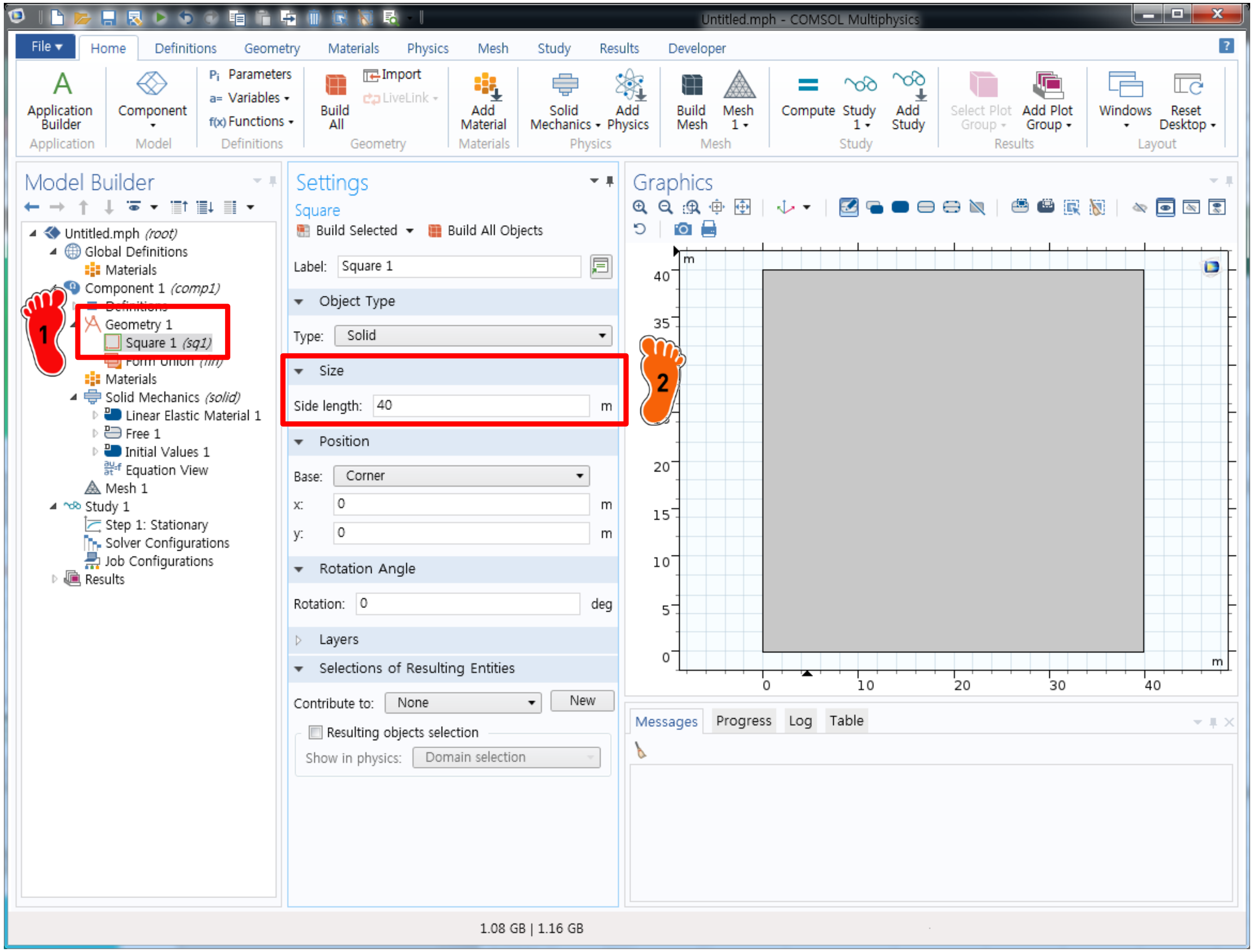


DISCRETIZATION



1 Discretization Displacement field를 Linear로 변경

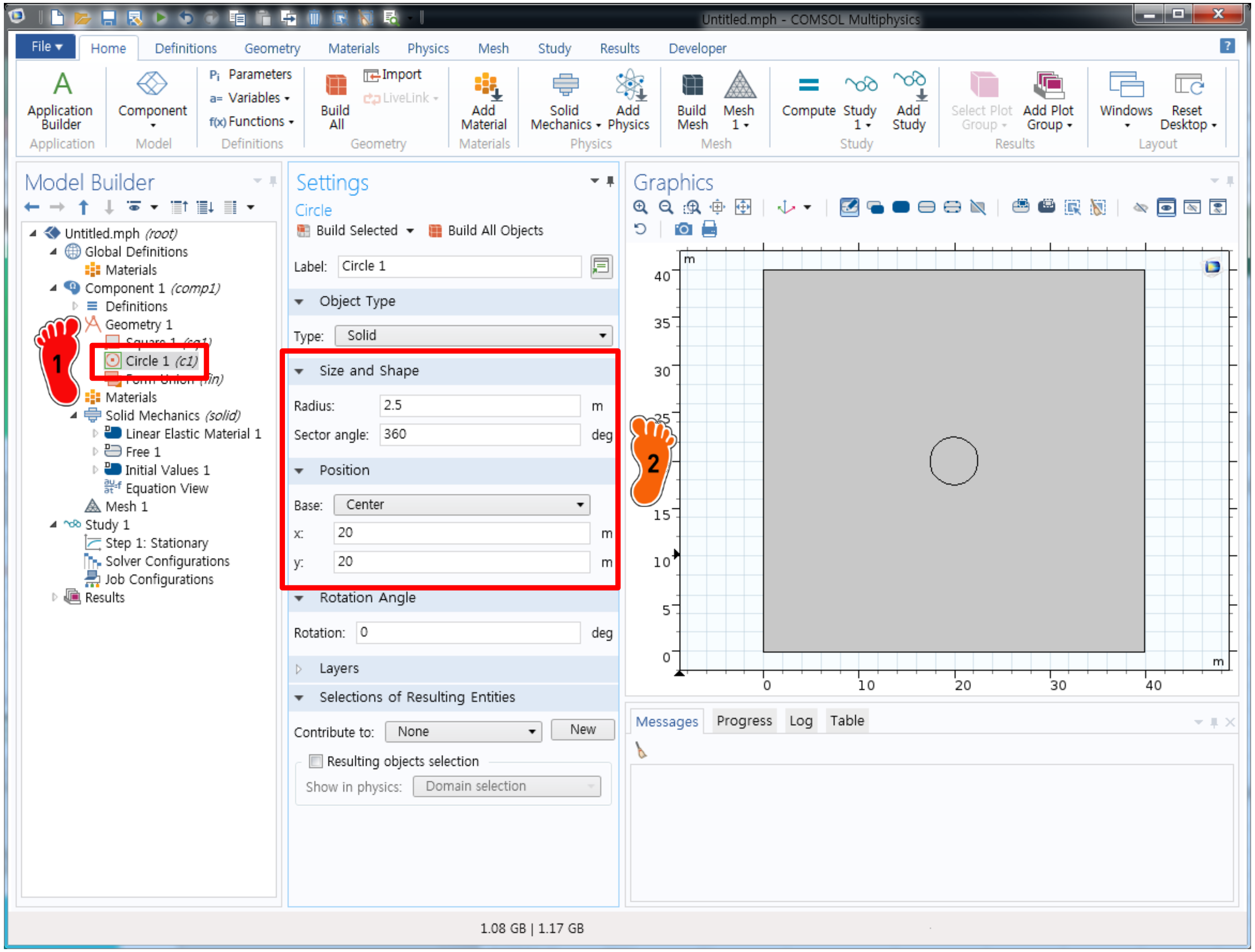
GEOMETRY CREATION



1 Square 생성

2 길이 40의 정사각형 생성

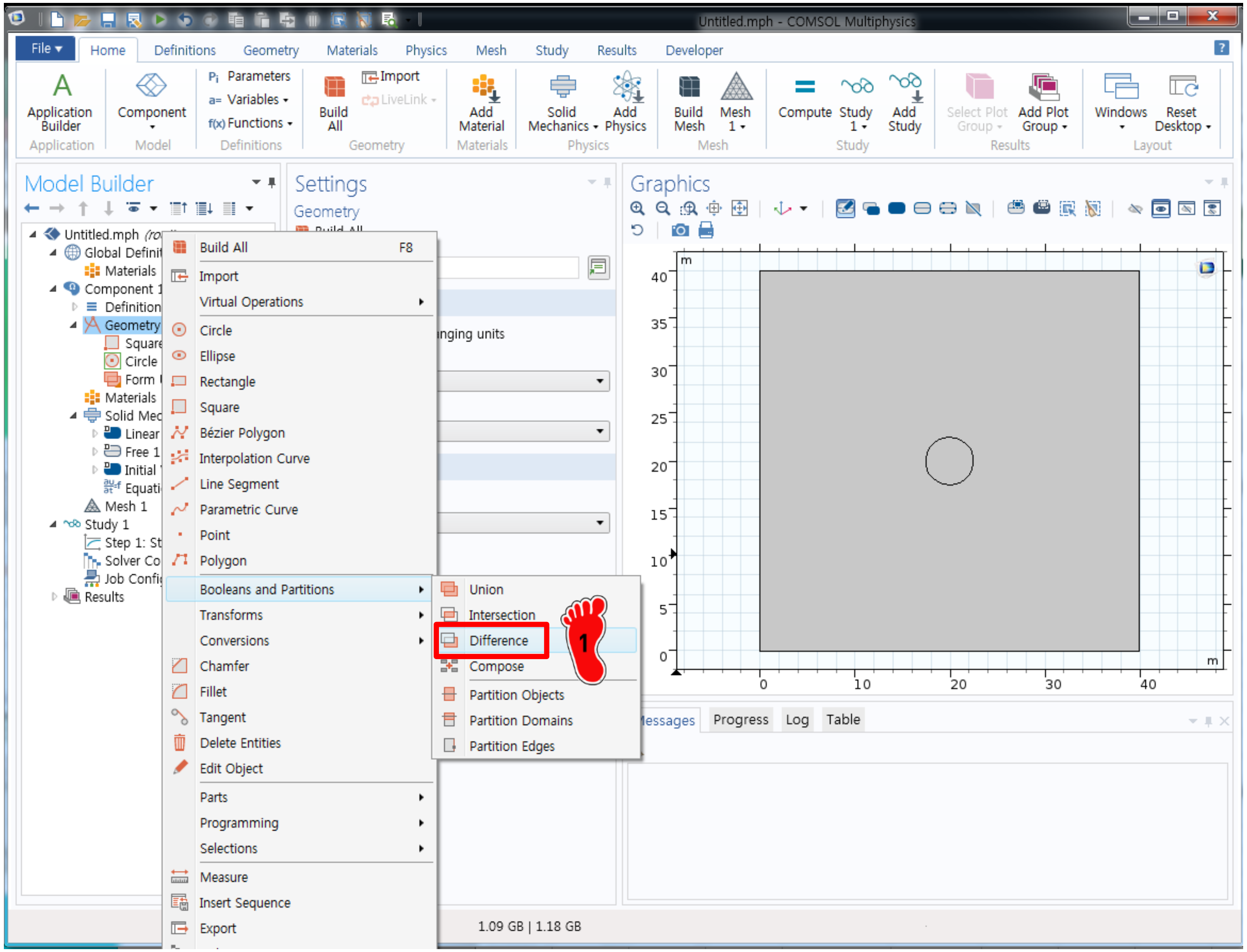
GEOMETRY CREATION



1 Circle 생성

2 중심 (20,20)
반지름 2.5 원 생성

GEOMETRY CREATION



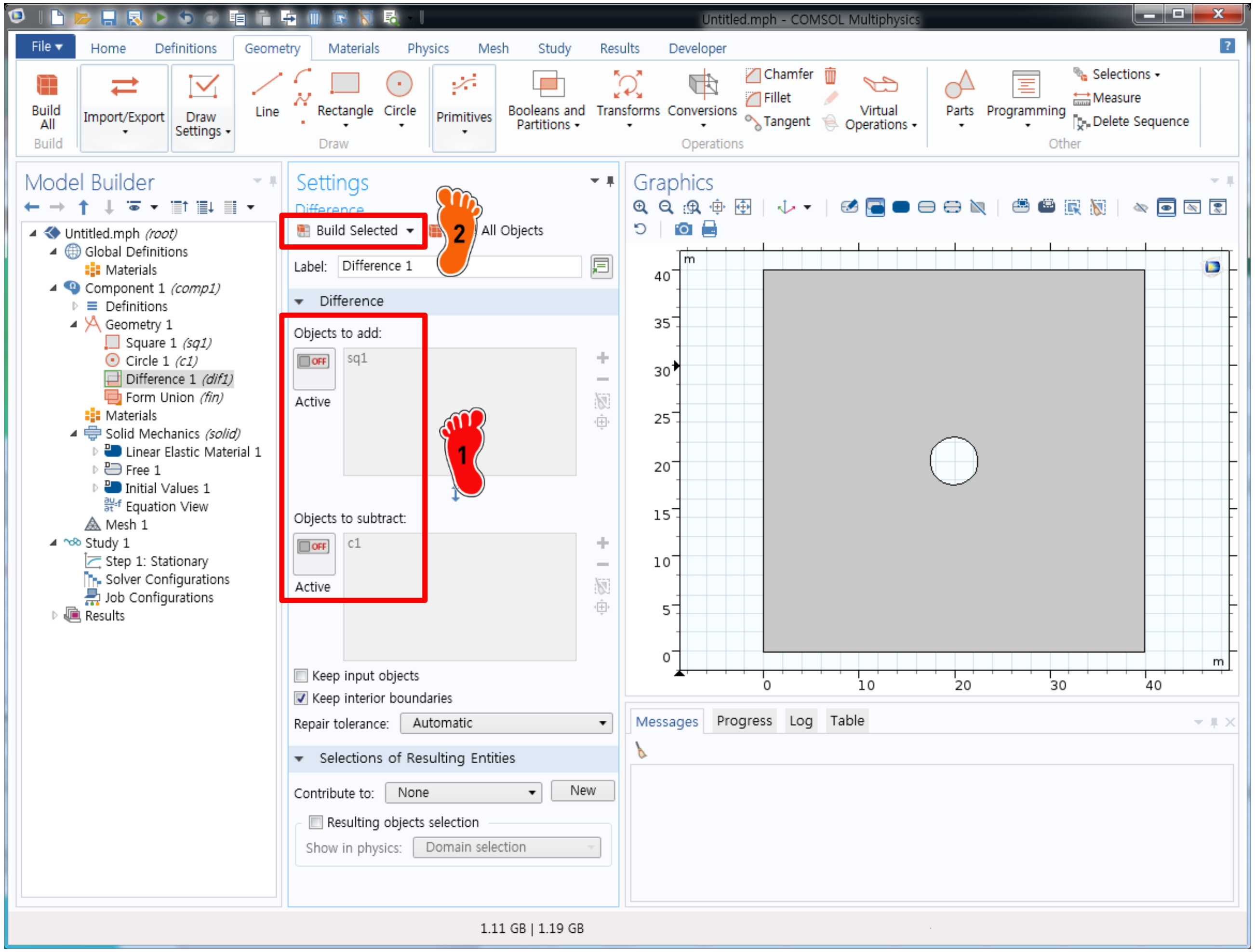
1

Geometry 오른쪽 클릭

→ Booleans and Partitions

→ Difference 클릭

GEOMETRY CREATION

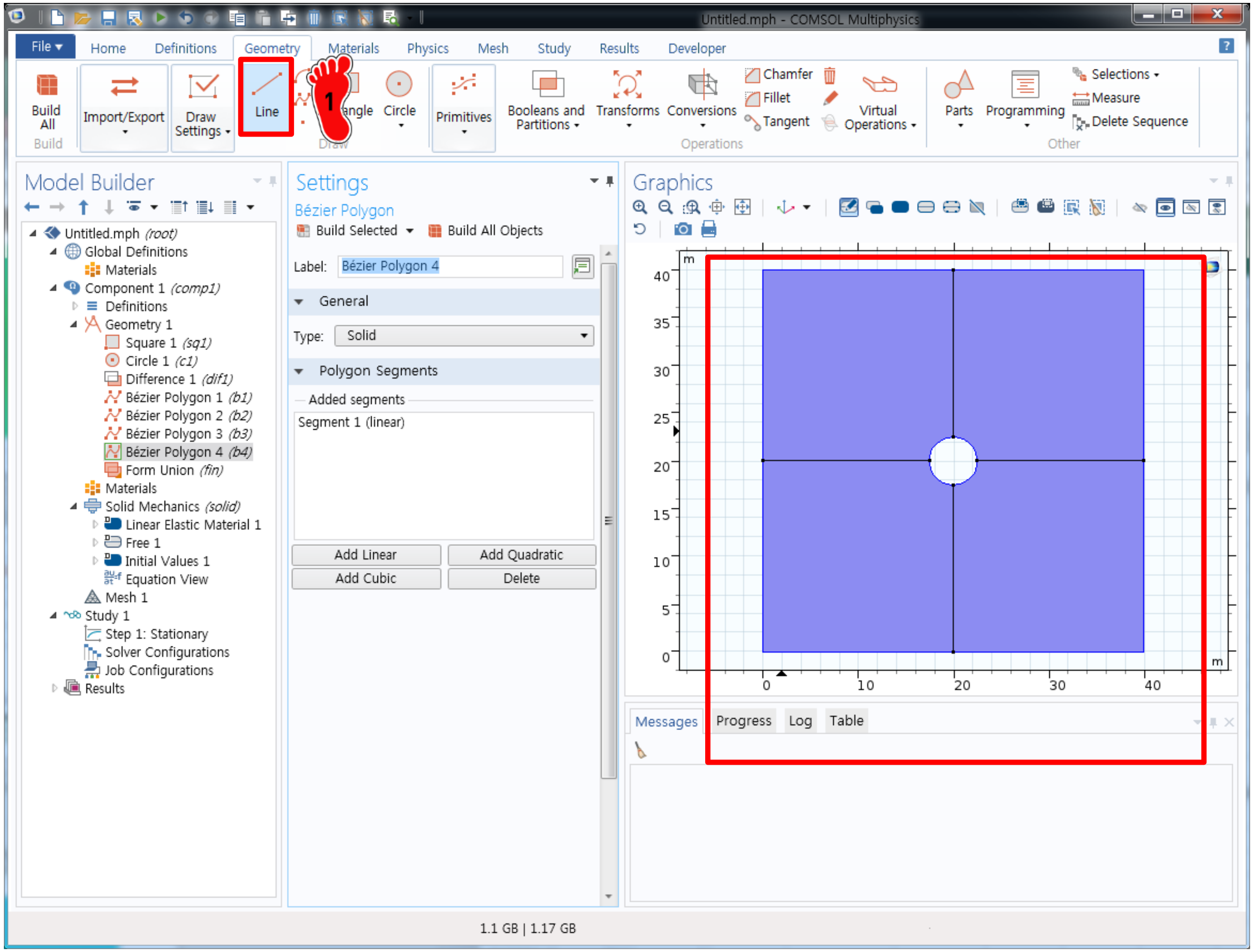


1 Objects to add
사각형 선택

Objects to subtract
원 선택(Active ON)

2 Build Selected 클릭

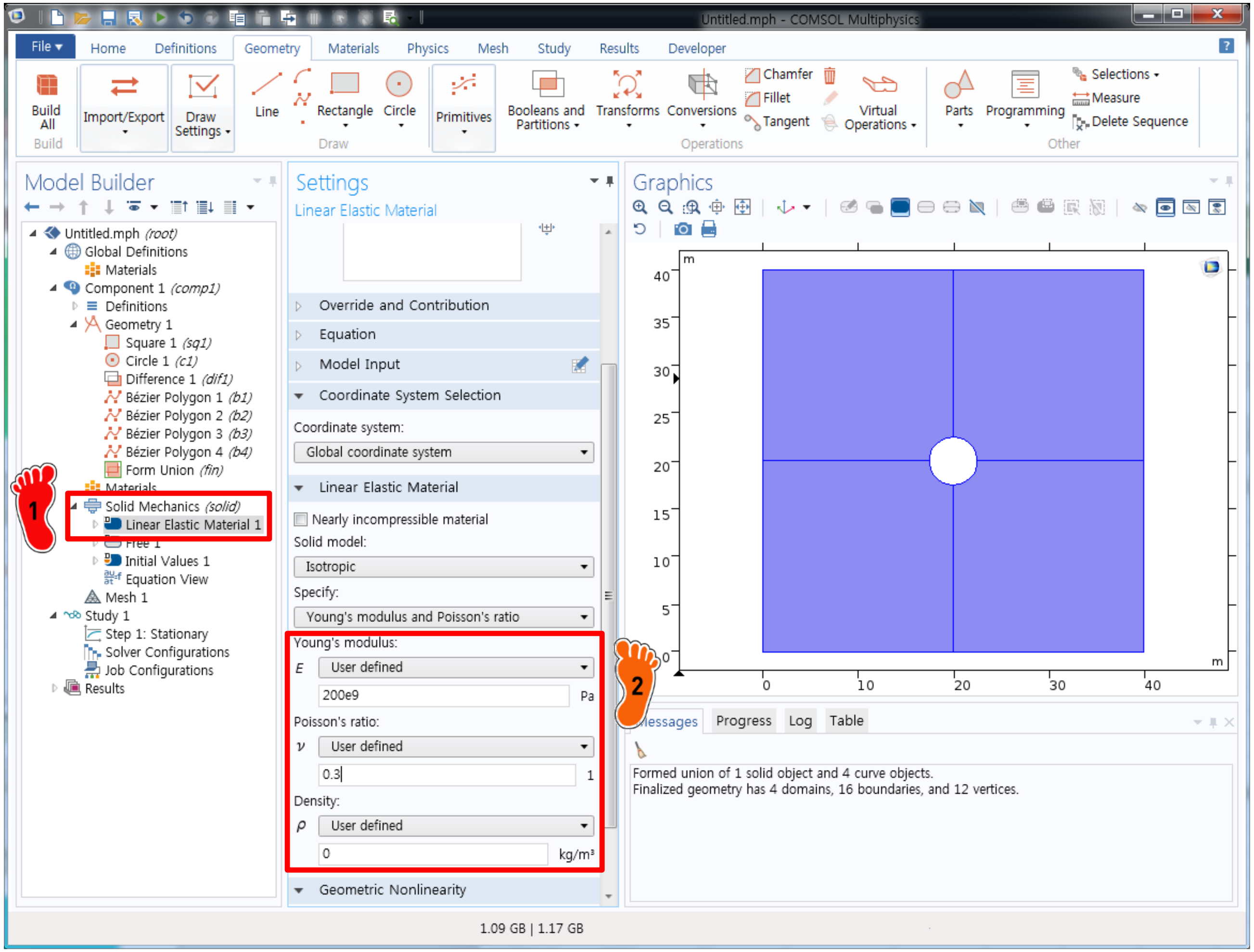
GEOMETRY CREATION



1 위쪽 Geometry tab에서 line 메뉴를 이용

왼쪽 마우스 클릭으로 선분 4개 생성(오른쪽 클릭: 완료)

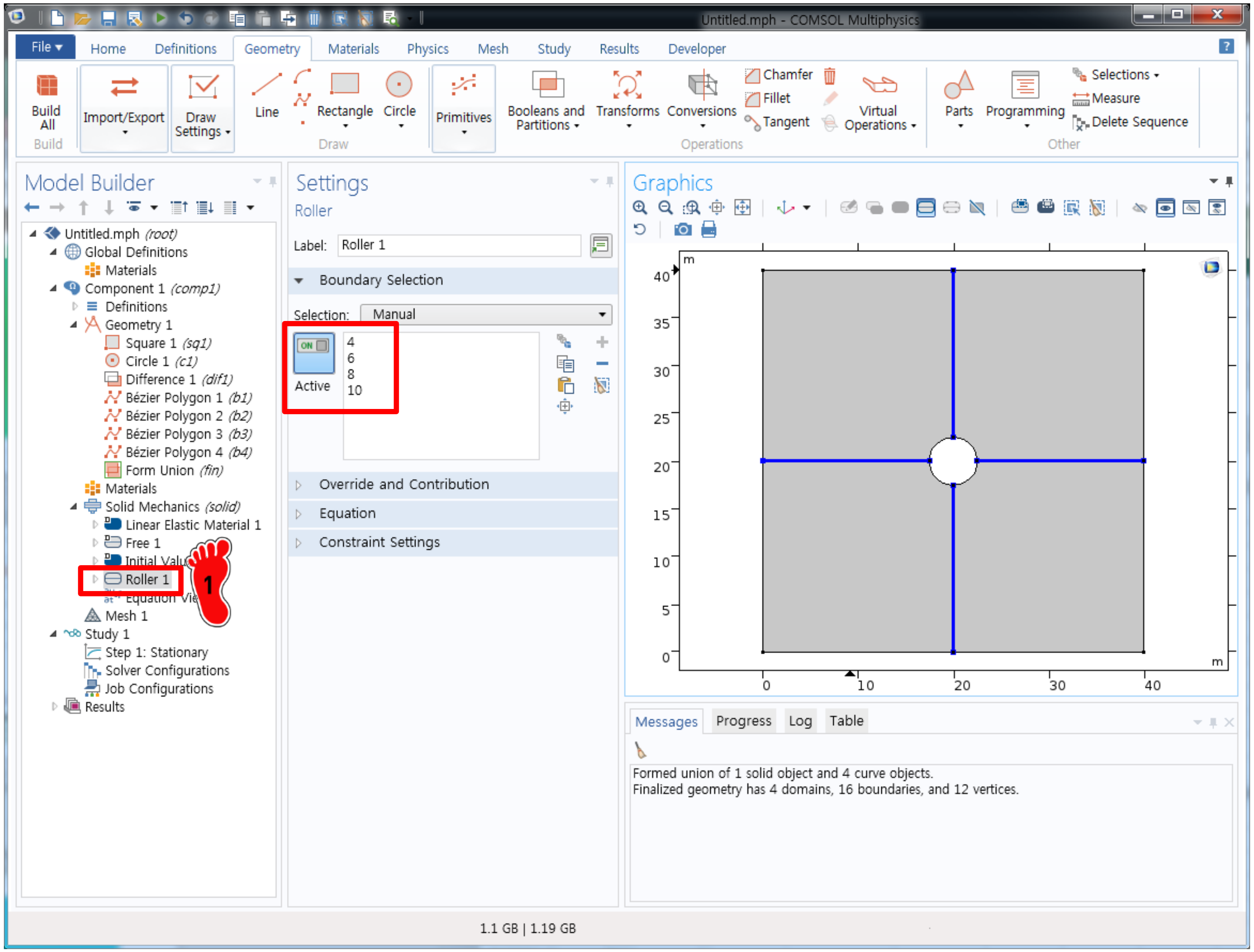
MATERIAL PROPERTY



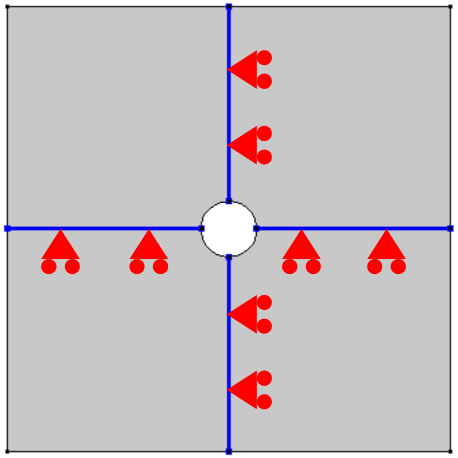
1 Linear Elastic Material 클릭

2 E: 200e9
mu: 0.3
rho : 0

BOUNDARY CONDITION



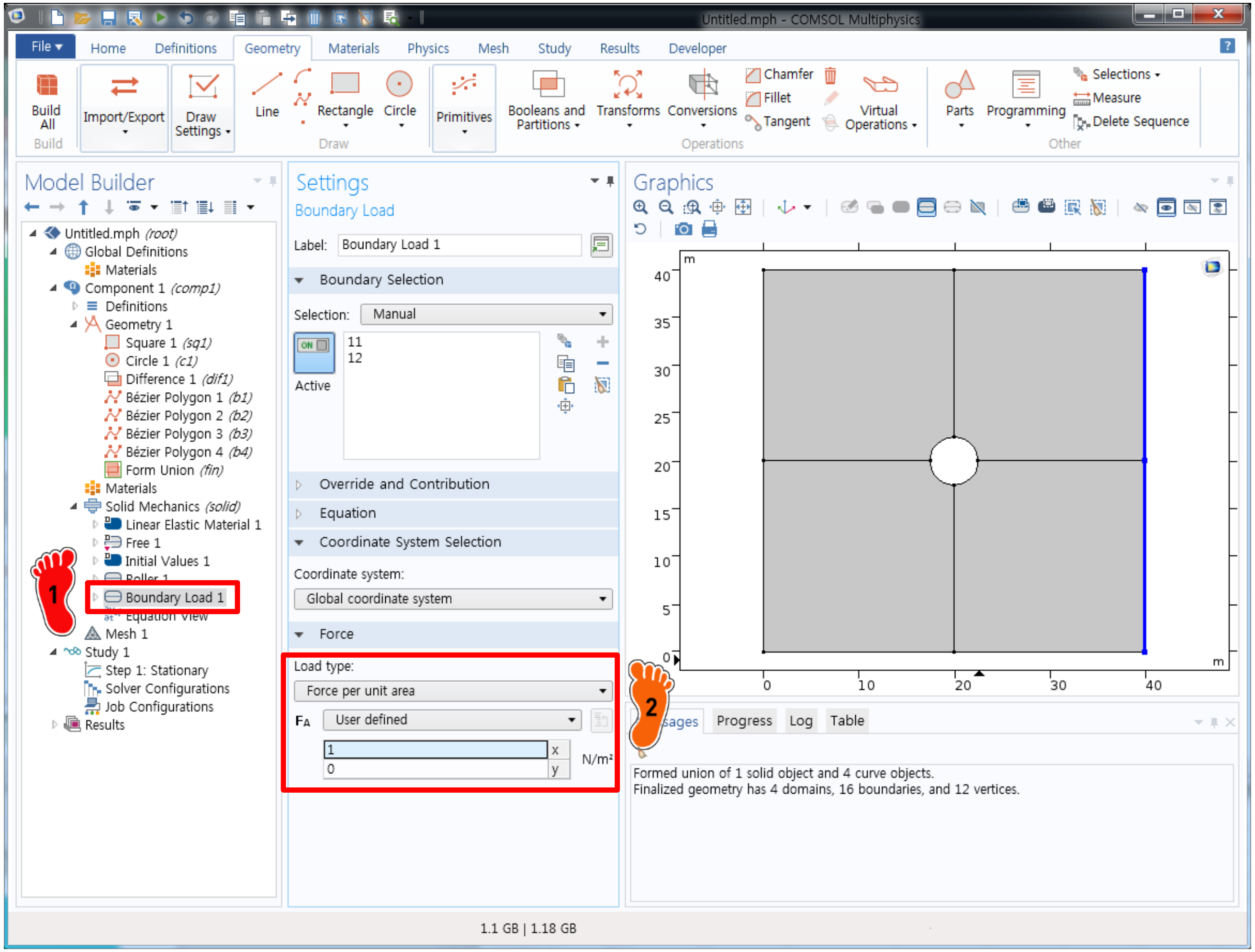
1 사각형 내부 4개 변을 Roller 조건으로 입력

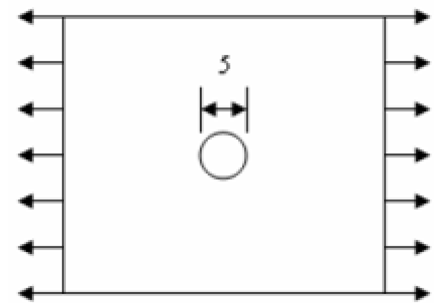


LOADING CONDITION

1. Boundary Load 생성

2. 오른쪽 두 변에 x 방향 1 입력 (단위는 N/m^2임)

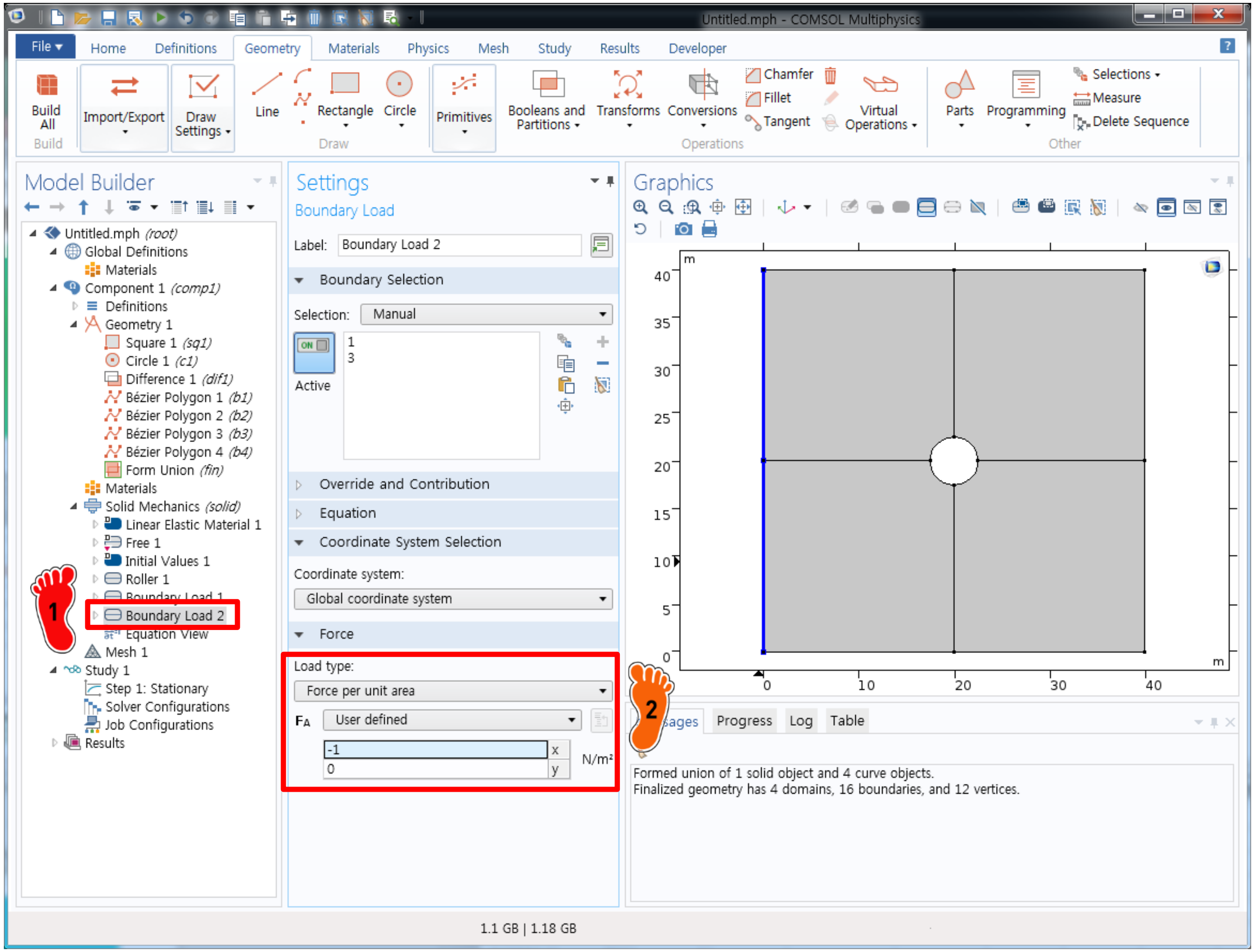


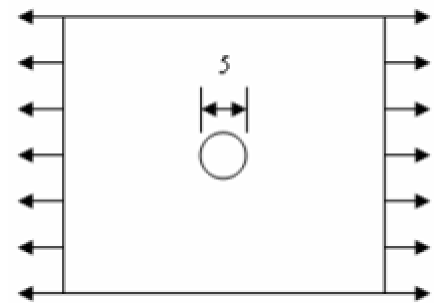


LOADING CONDITION

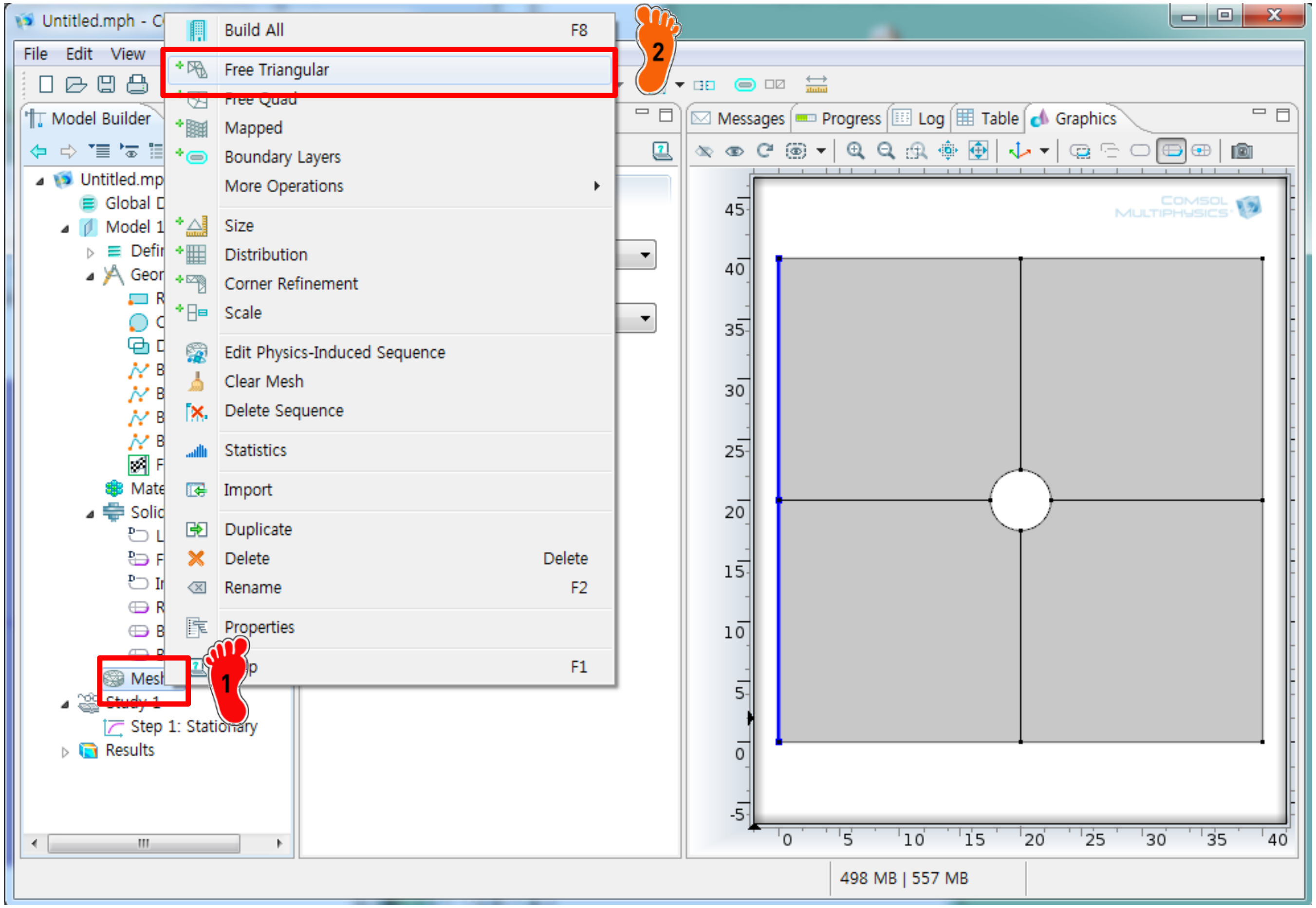
1 Boundary Load 생성

2 오른쪽 두 변에 x 방향 -1 입력 (단위는 N/m²임)





MESH

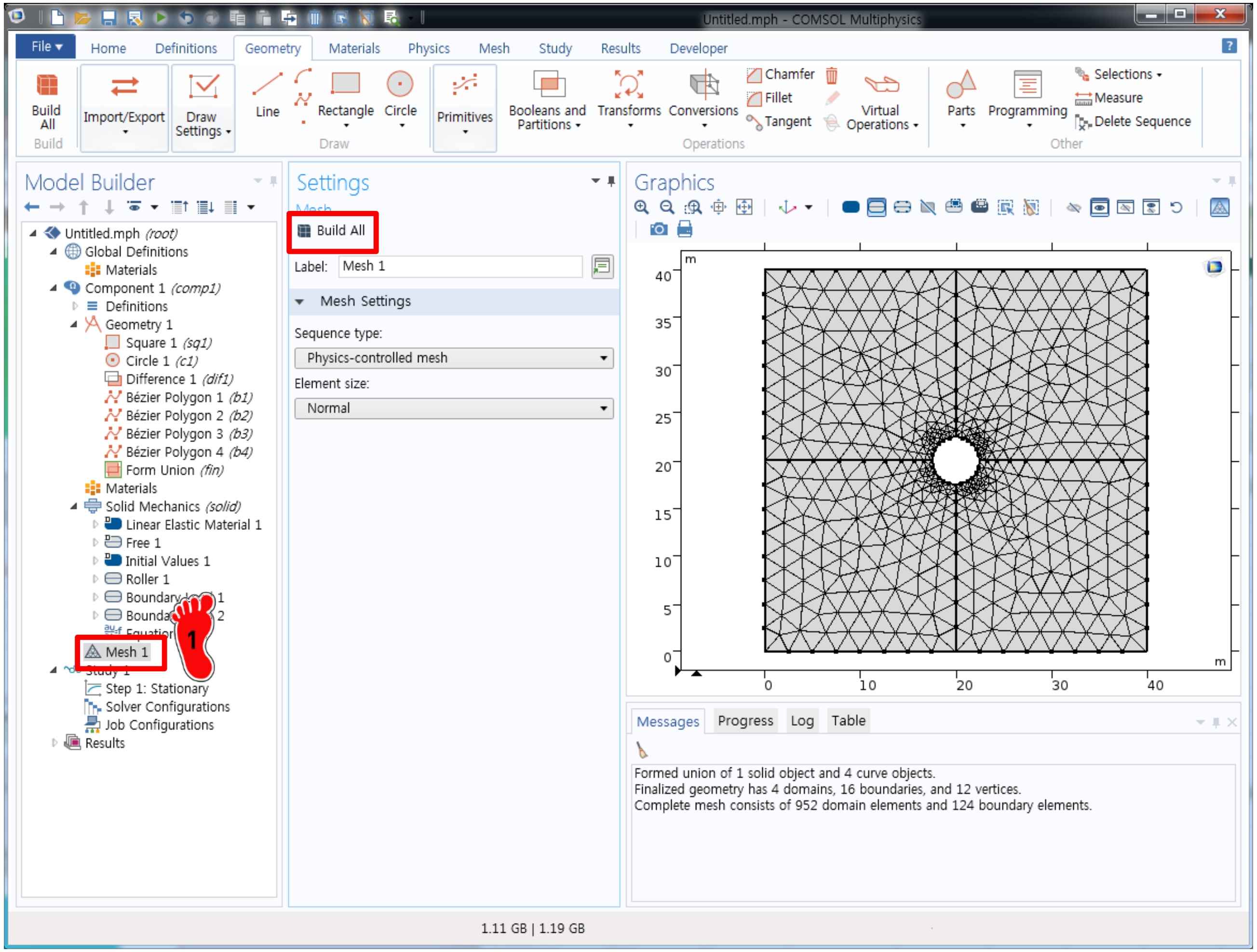


1 Mesh 마우스 우클릭

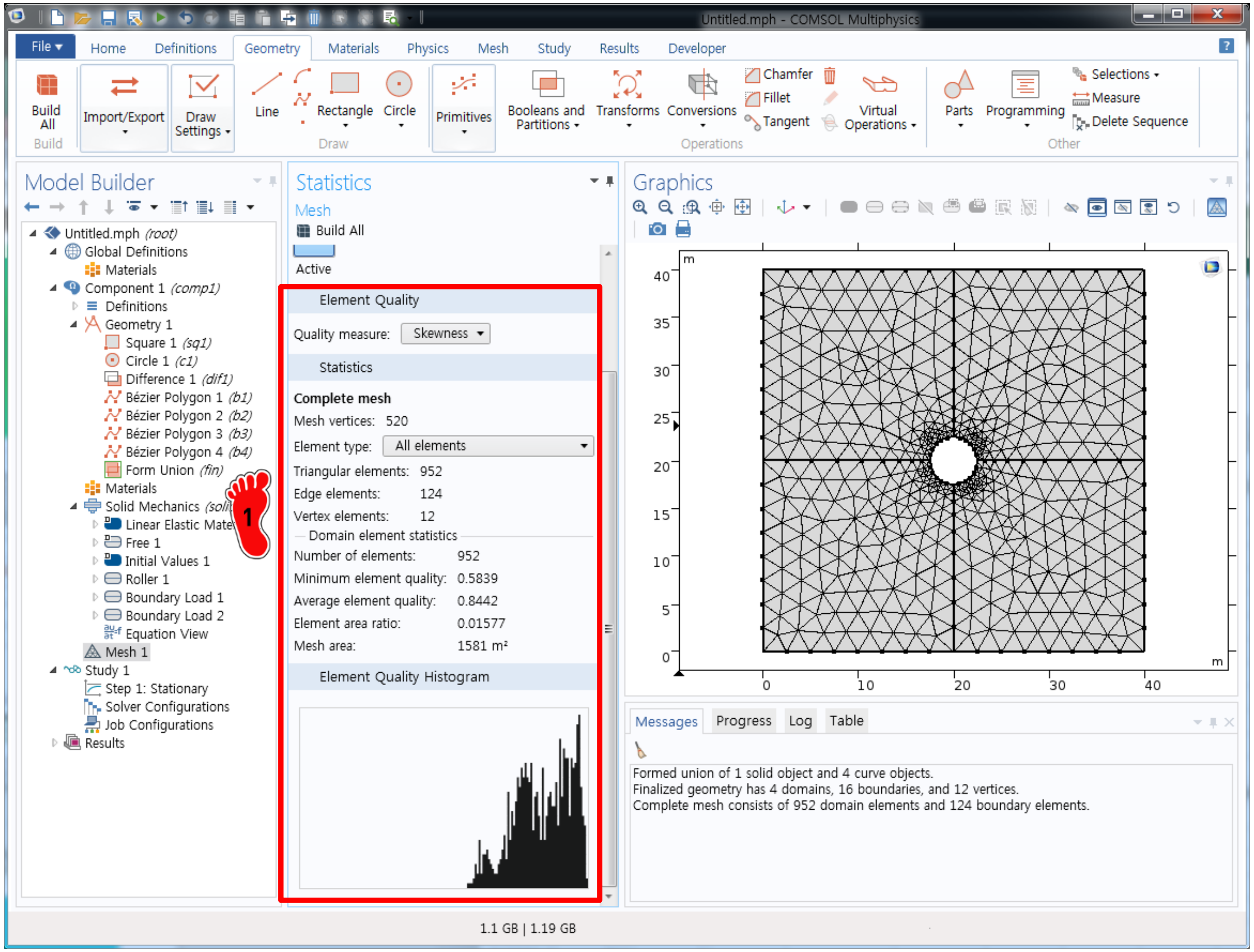
2 Free Triangular 클릭

MESH

1 기본 Mesh 생성(삼각형)



MESH

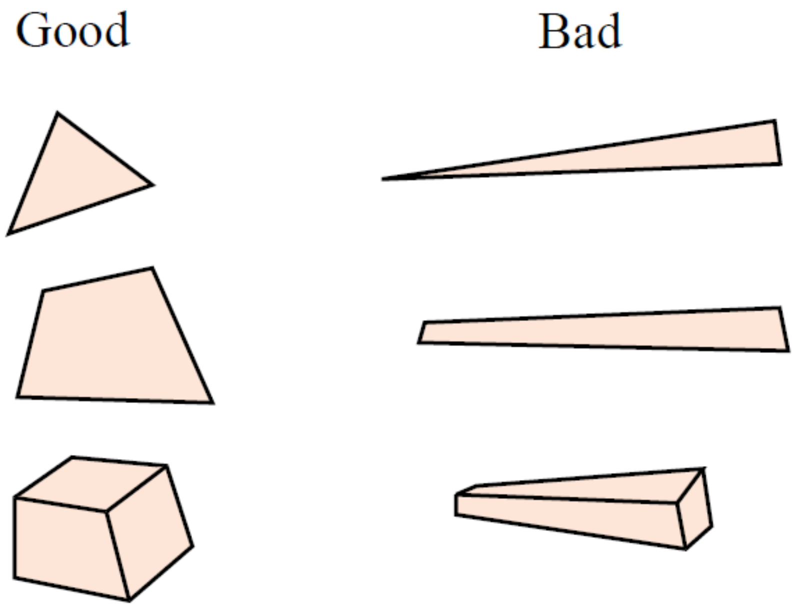


1 Mesh 우클릭
→ Statistics 클릭

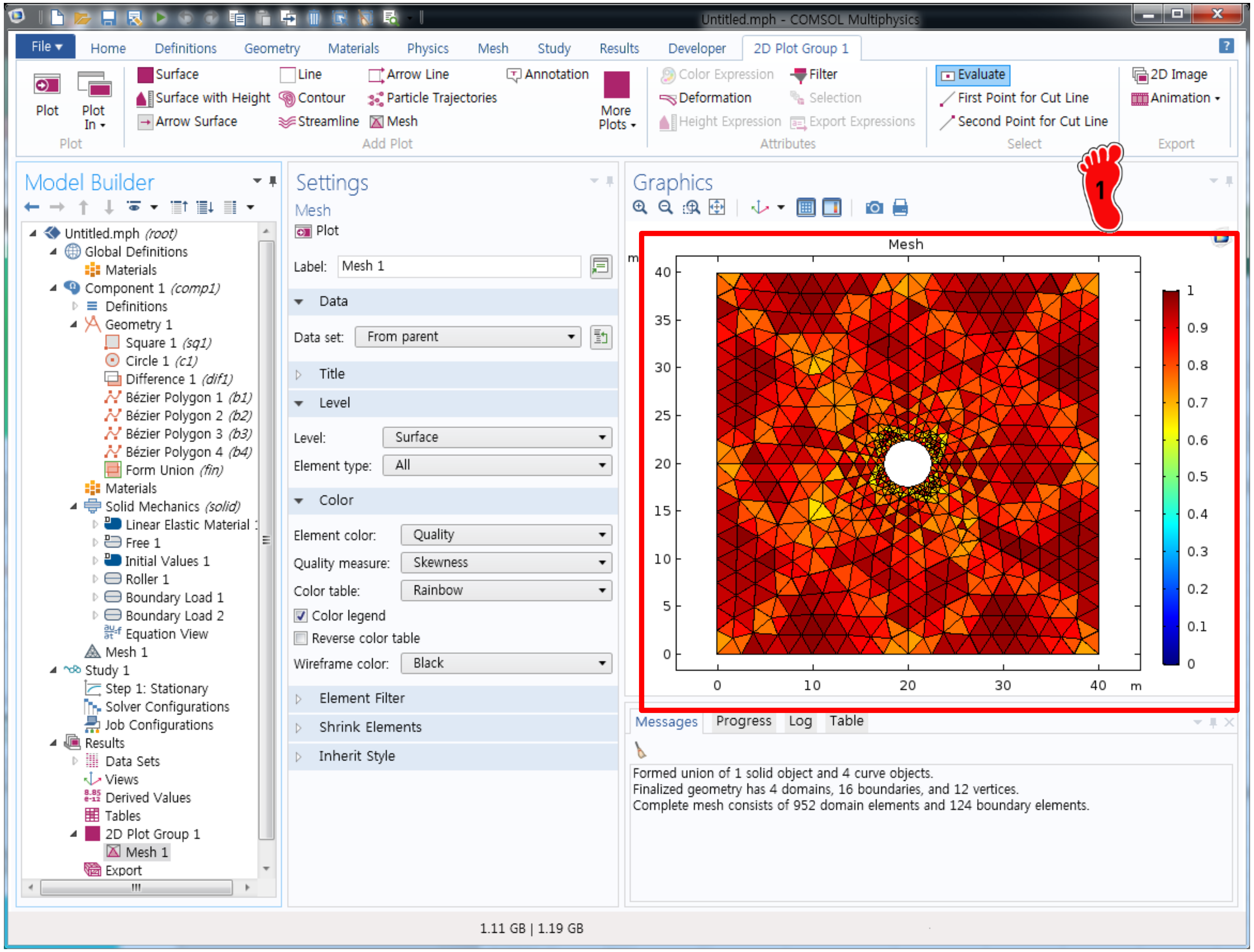
요소 개수, DOF, Quality
분포 정보 등을 보여줌

2D/3D BAD ASPECT RATIO ELEMENT

- “thin” structures modeled as continuous bodies
  - Elongated or “skinny” element
- Aspect ratio
  - Ratio between its largest and smallest dimension
  - > 3: caution
  - > 10: alarm



MESH

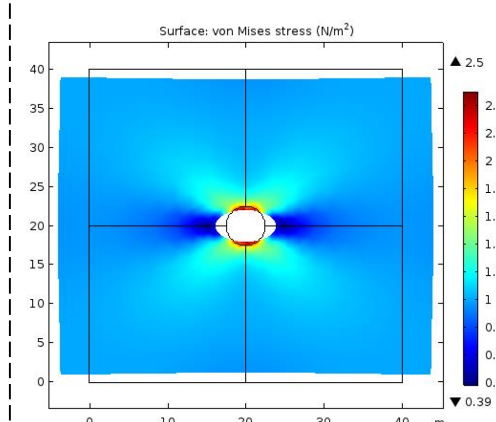


1 Mesh 우클릭 → Plot

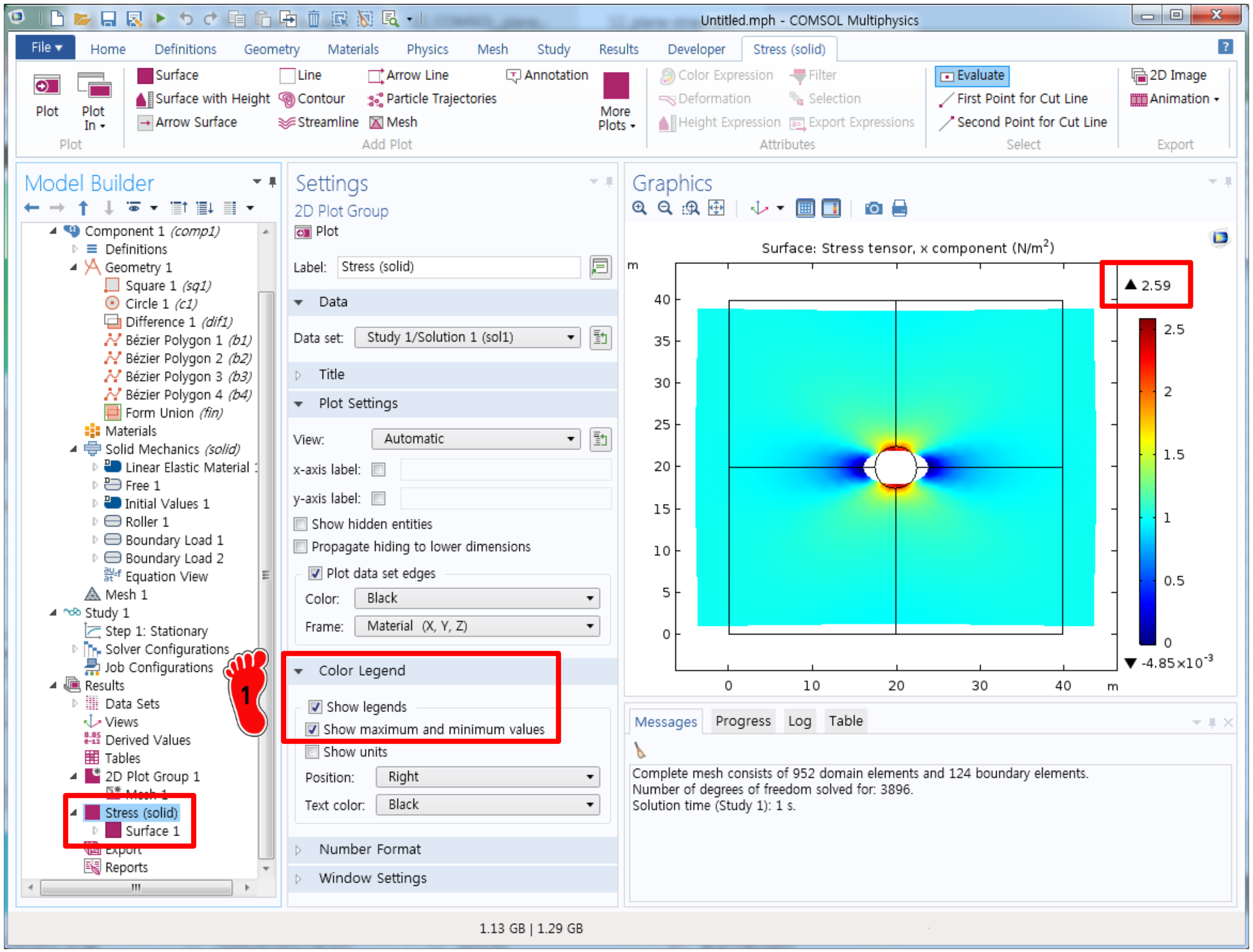
빨간색에 가까운 요소는
정삼각형에 가까운 요소
(Quality가 1에 가까움)

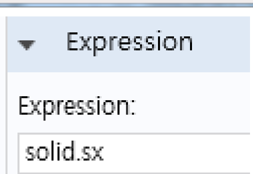
파란색에 가까운 요소는
정삼각형과 먼 요소
(Quality가 1에서 멀어짐)

compute를 클릭하여
해석 수행



POST-PROCESSING

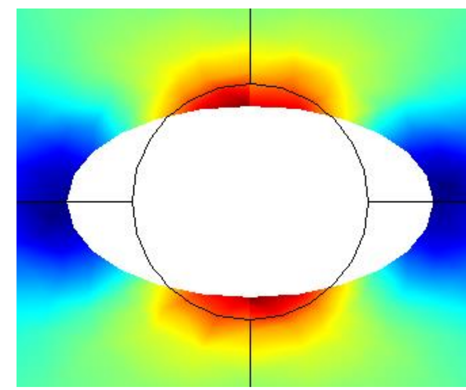




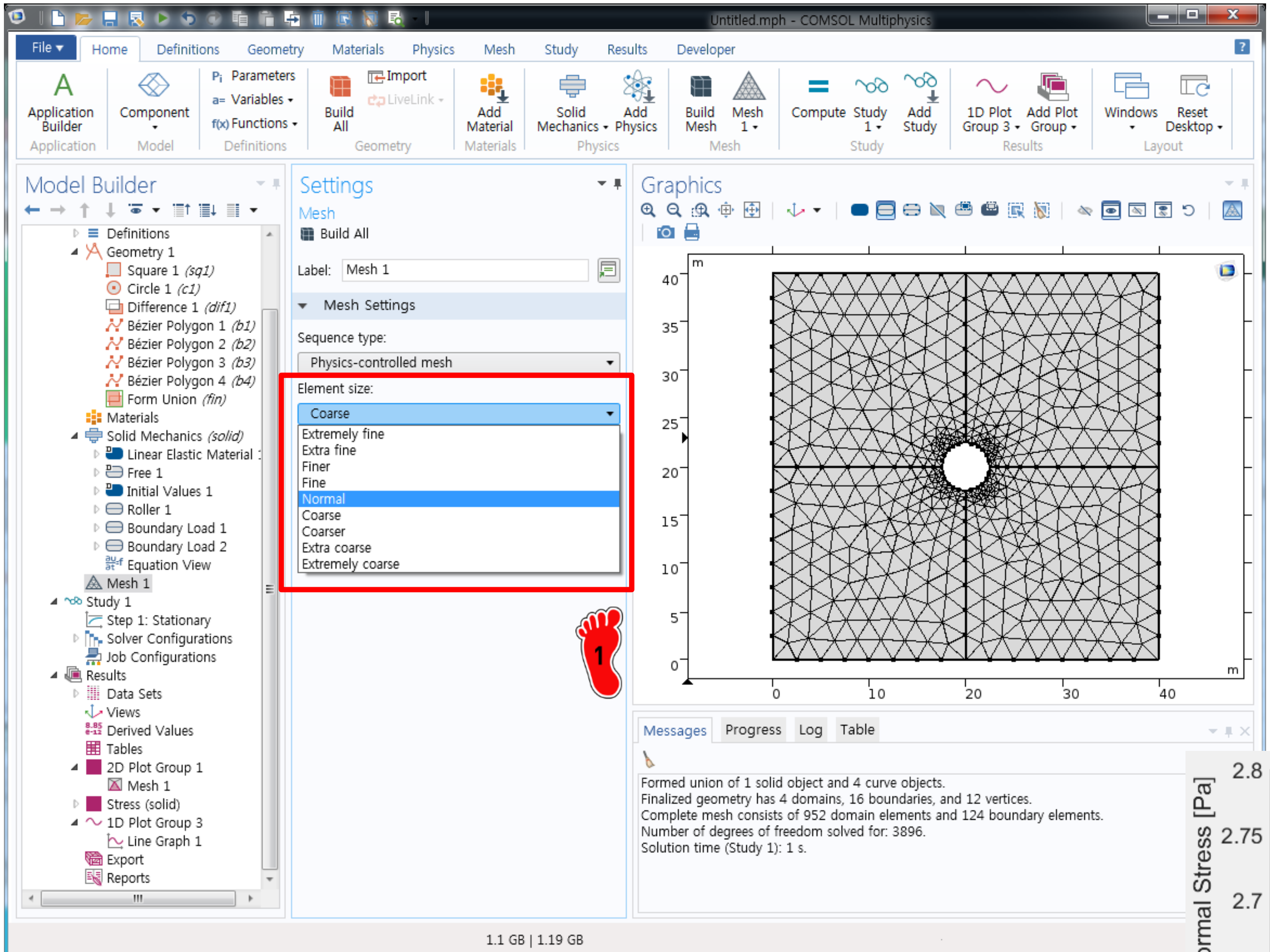
1 Results → Stress(solid) 클릭 Color Legend에서 Show maximum and minimum values 체크

2 Surface 클릭후 Expression을 "solid.sx"(normal stress)로 변경 후 plot

최대 수직 응력은 원에 접하는 부분에서 약 2.59 Pa



MESH REFINEMENT



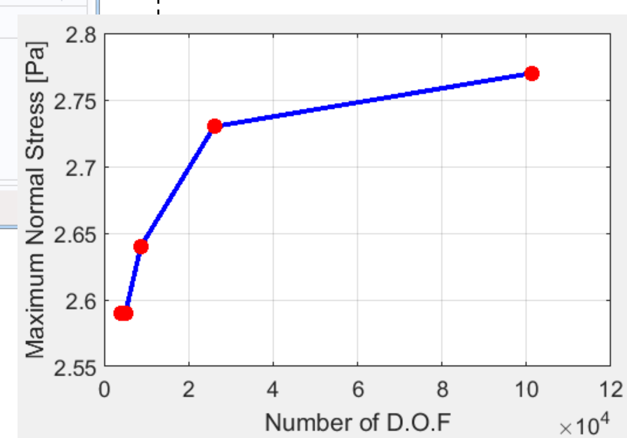
1

Mesh에서 Element Size 클릭 → Fine으로 변경 후 Build All → Compute

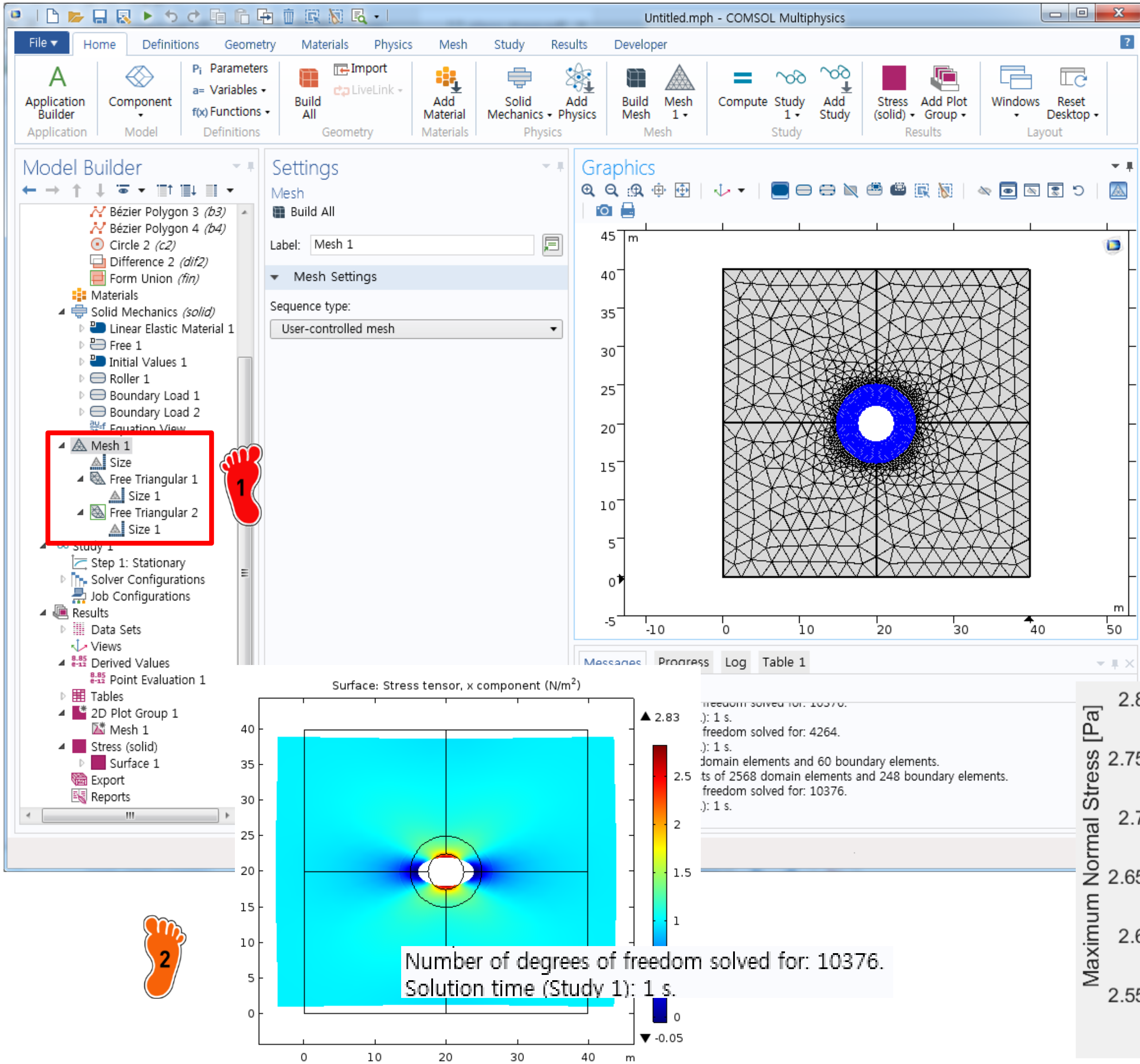
2

위와 같은 방식으로 요소 수를 증가시켜 응력 결과값을 확인 출력한 그래프

Analytical solution 인 3*sigma 대비 작은 값으로 수렴해 감

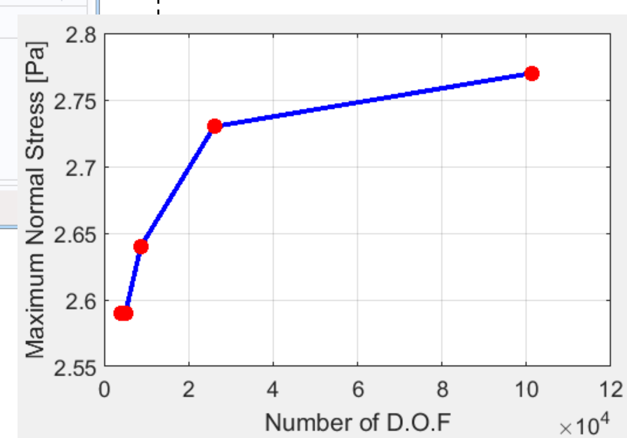


MESH REFINEMENT

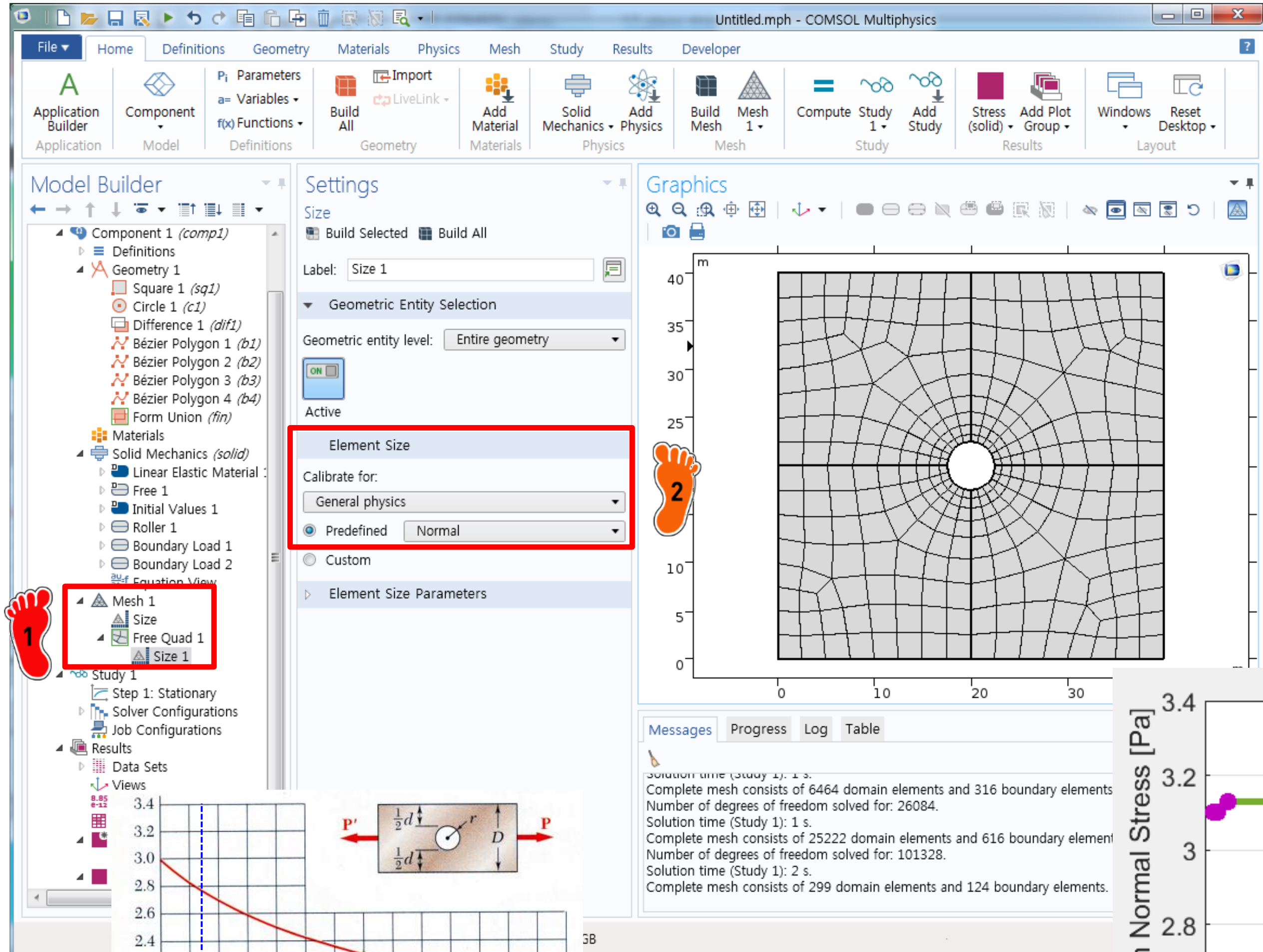


1 Geometry를 분할하여 응력이 높게 나타나는 곳만 fine mesh 적용

2 자유도 수 약 10,000개 정도로 더 정확한 결과 계산



QUADRILATERAL ELEMENT

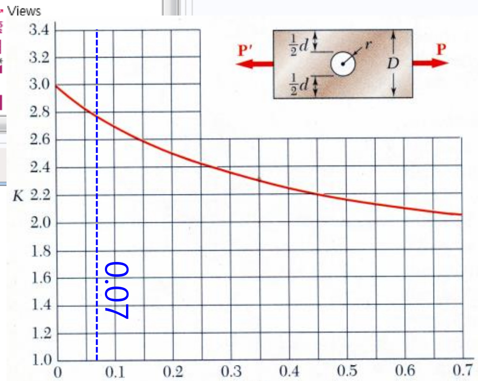


1. Mesh 우클릭
→ Free Quad 생성(사각형)
→ Size 생성 후 Build

2. Fine 으로 변경 후 해석

위와 같은 방식으로 요소 수를 증가시켜 응력 결과값을 출력한 그래프

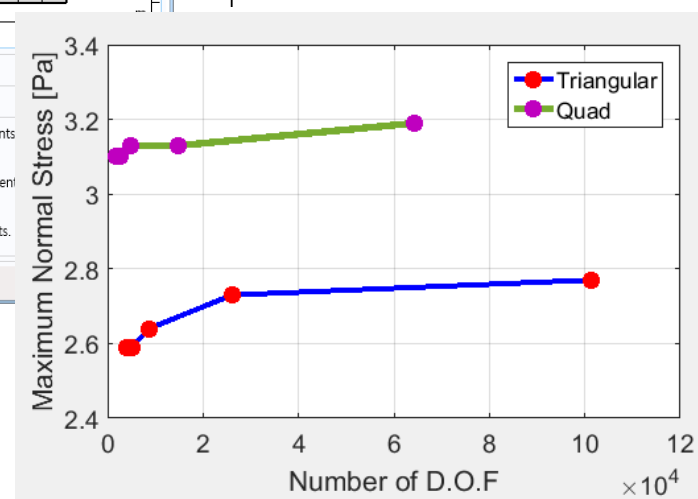
삼각형보다 사각형 메시가 DOF수에 따라 변화 적음



$$\frac{r}{d}=\frac{2.5}{35}=0.07,\ K\approx 2.8$$

$$\sigma_{ave}=\frac{P}{A}=\frac{P}{W-2R}=\frac{40}{40-5}=1.14$$

$$\sigma_{max}=K\sigma_{ave}=2.8\times1.14=3.2\ \text{Pa}$$



- **Stiffness matrix of Turner triangle**

- **2D plane stress model**

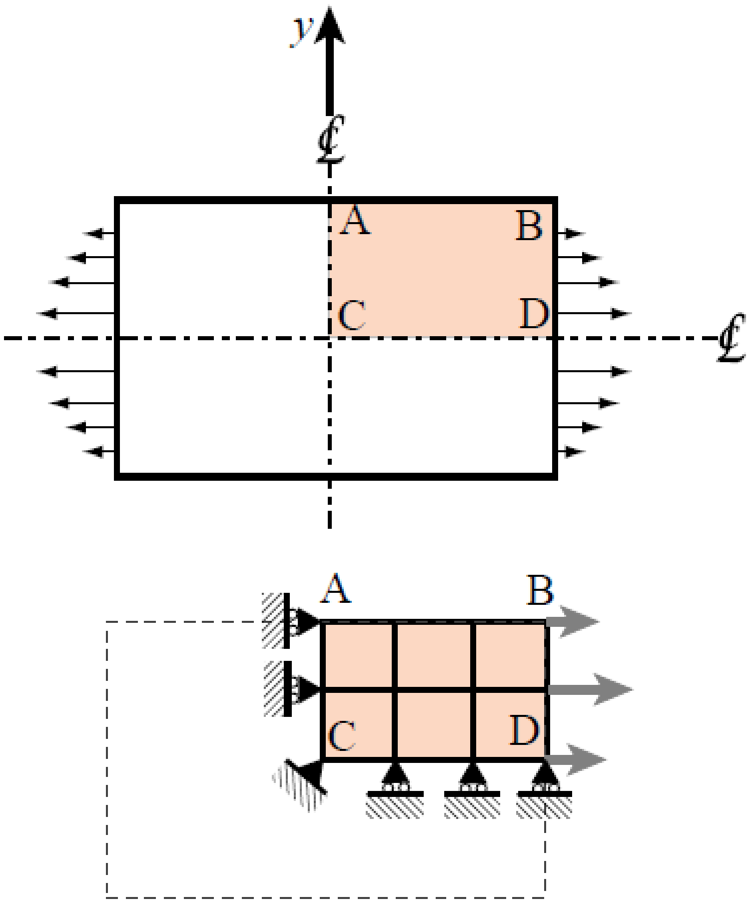
  ✓ **Kirsh's problem**

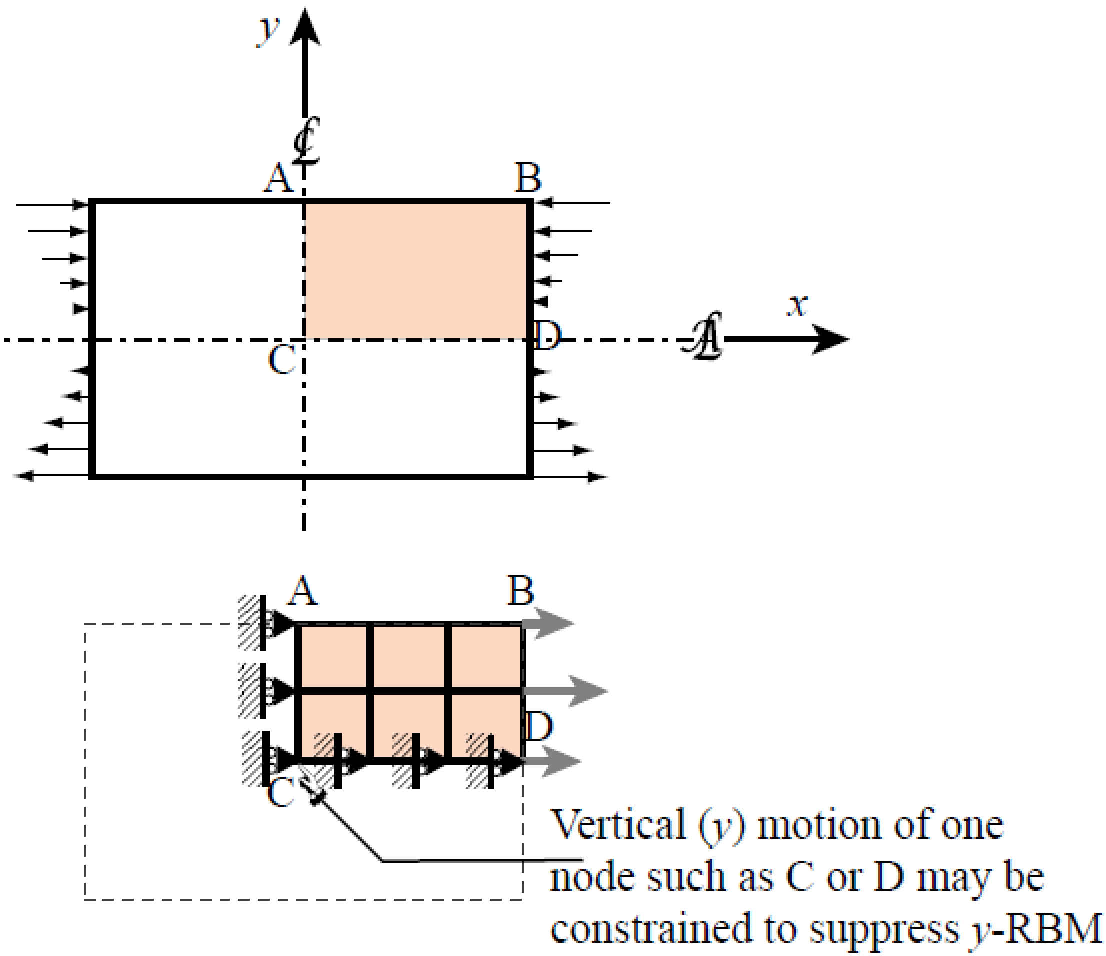
- **<u>Quarter model</u>**

SYMMETRY CONDITION

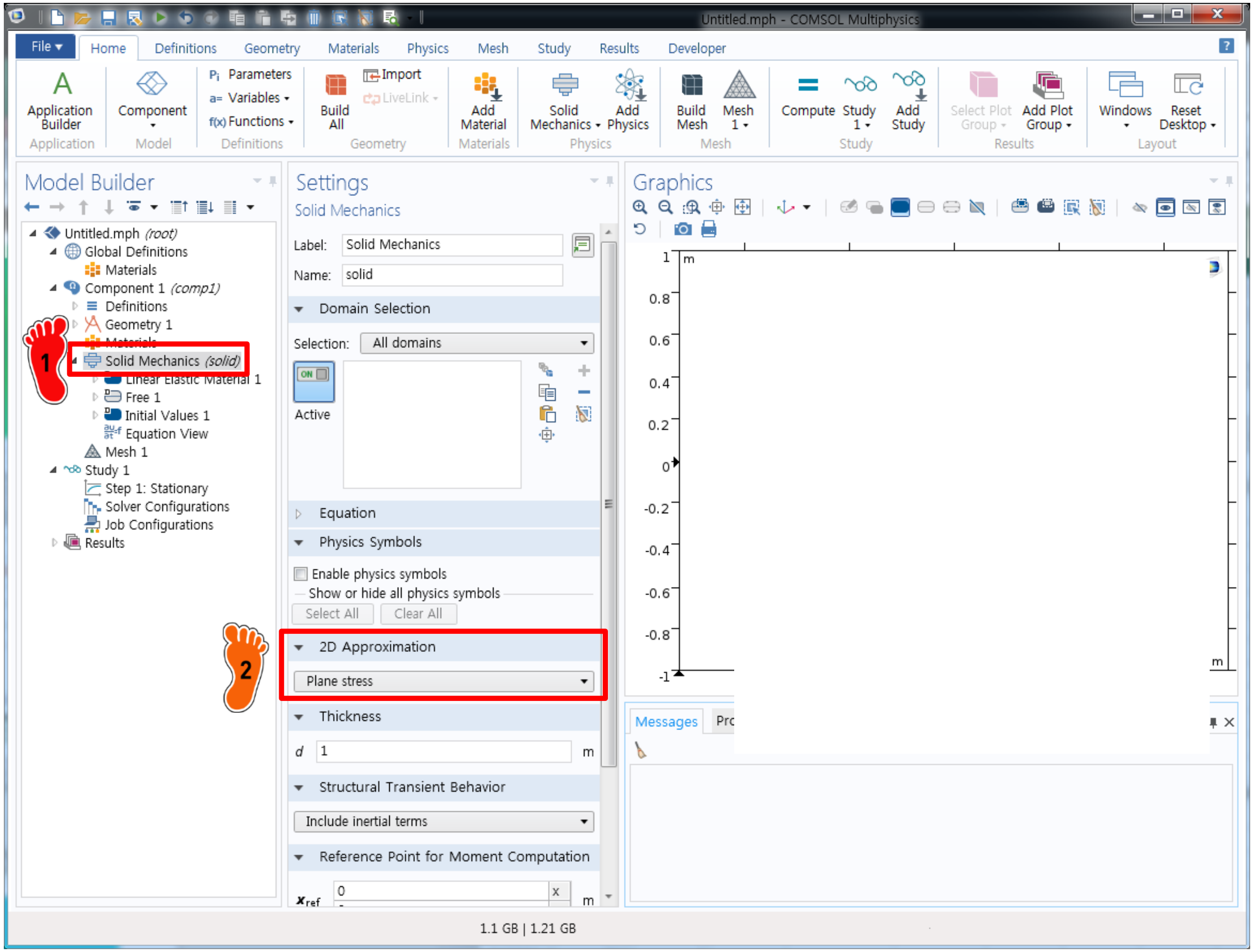
Symmetry condition

AntiSymmetry condition





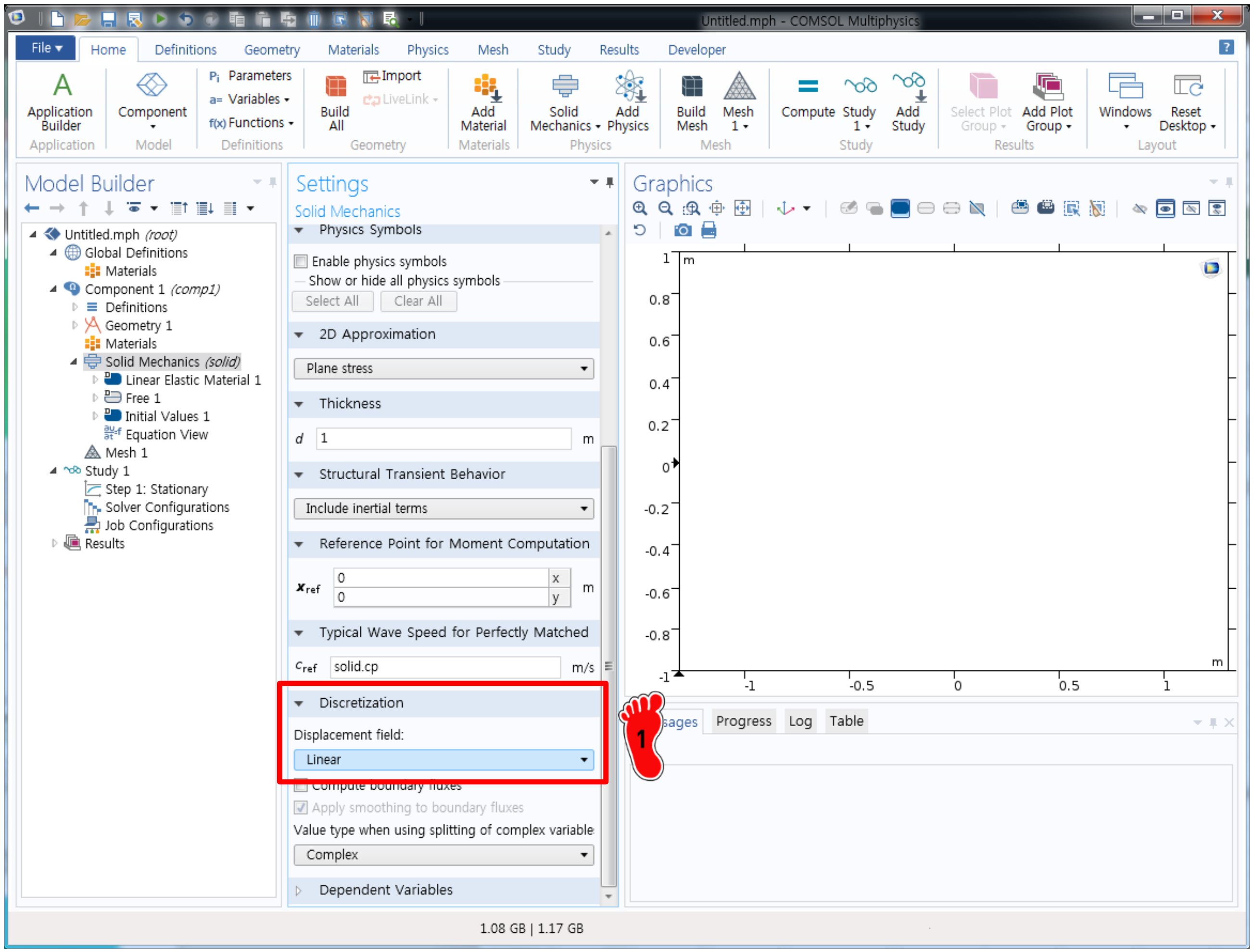
PLANE STRESS



Solid Mechanics 클릭

2D Approximation
Plane stress 로 변경

DISCRETIZATION



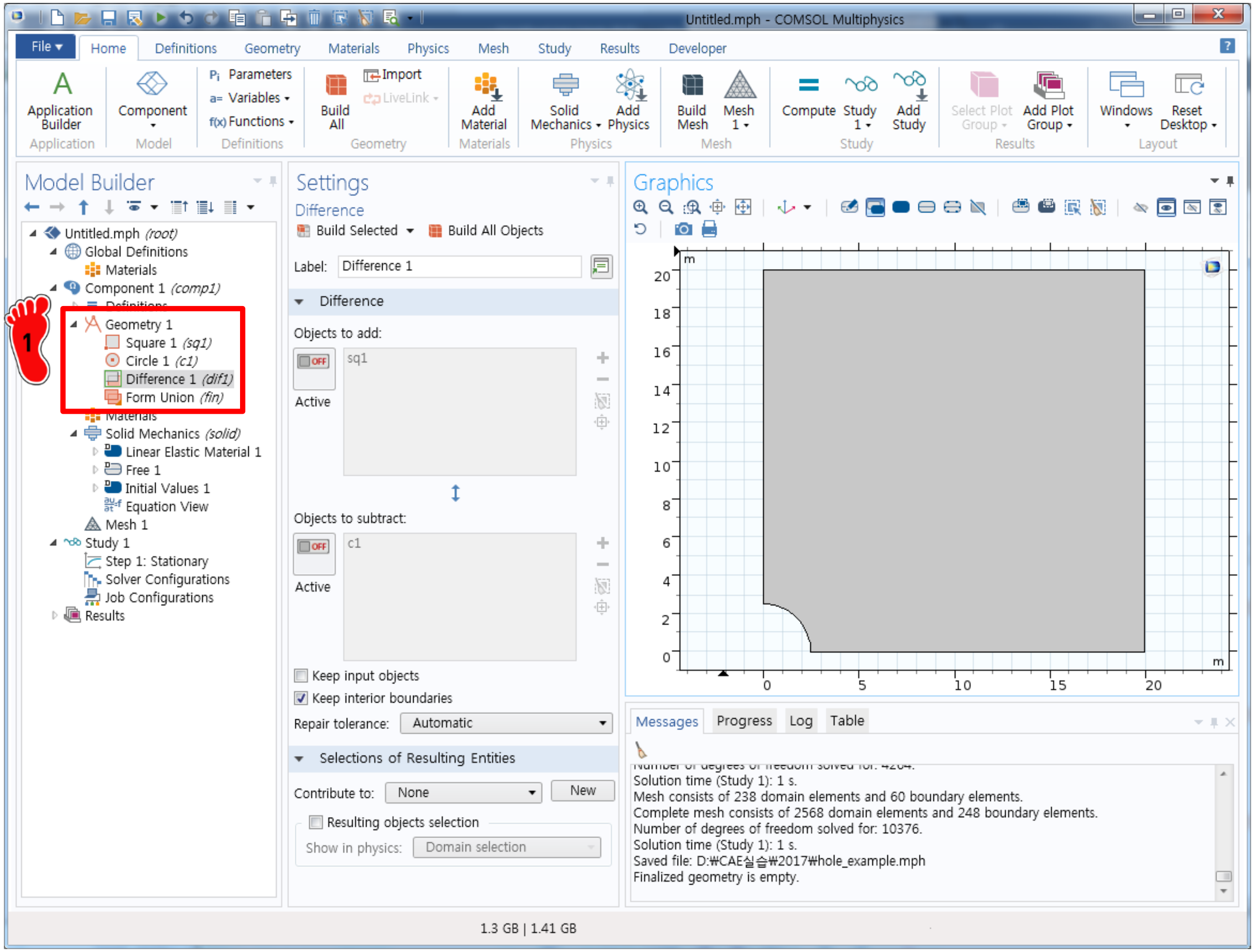
1

Discretization

Displacement field를

Linear로 변경

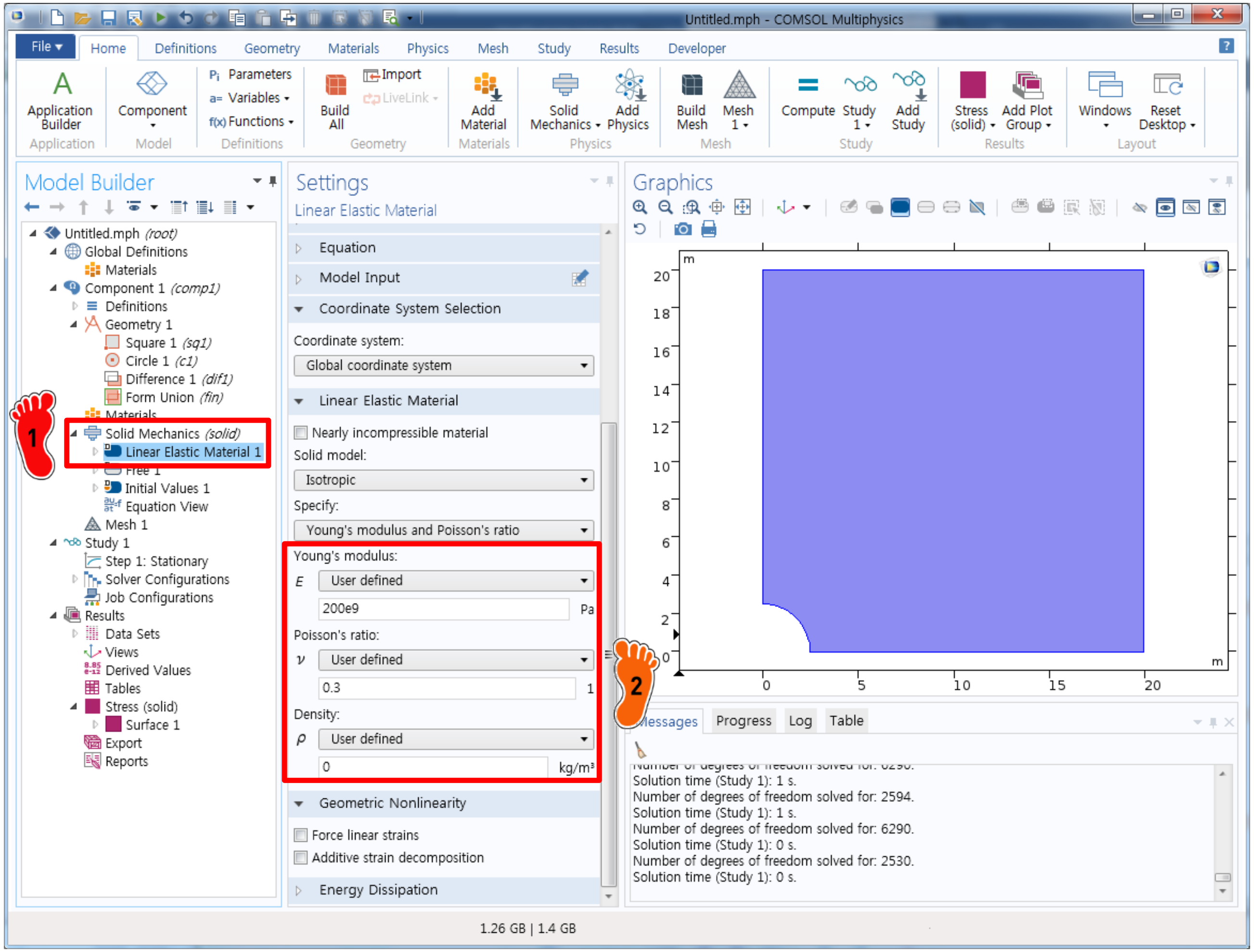
GEOMETRY CREATION



1 ¼ 모델 기하형상 생성

길이 20의 정사각형에서
반경 2.5인 원을 뺌

MATERIAL PROPERTY



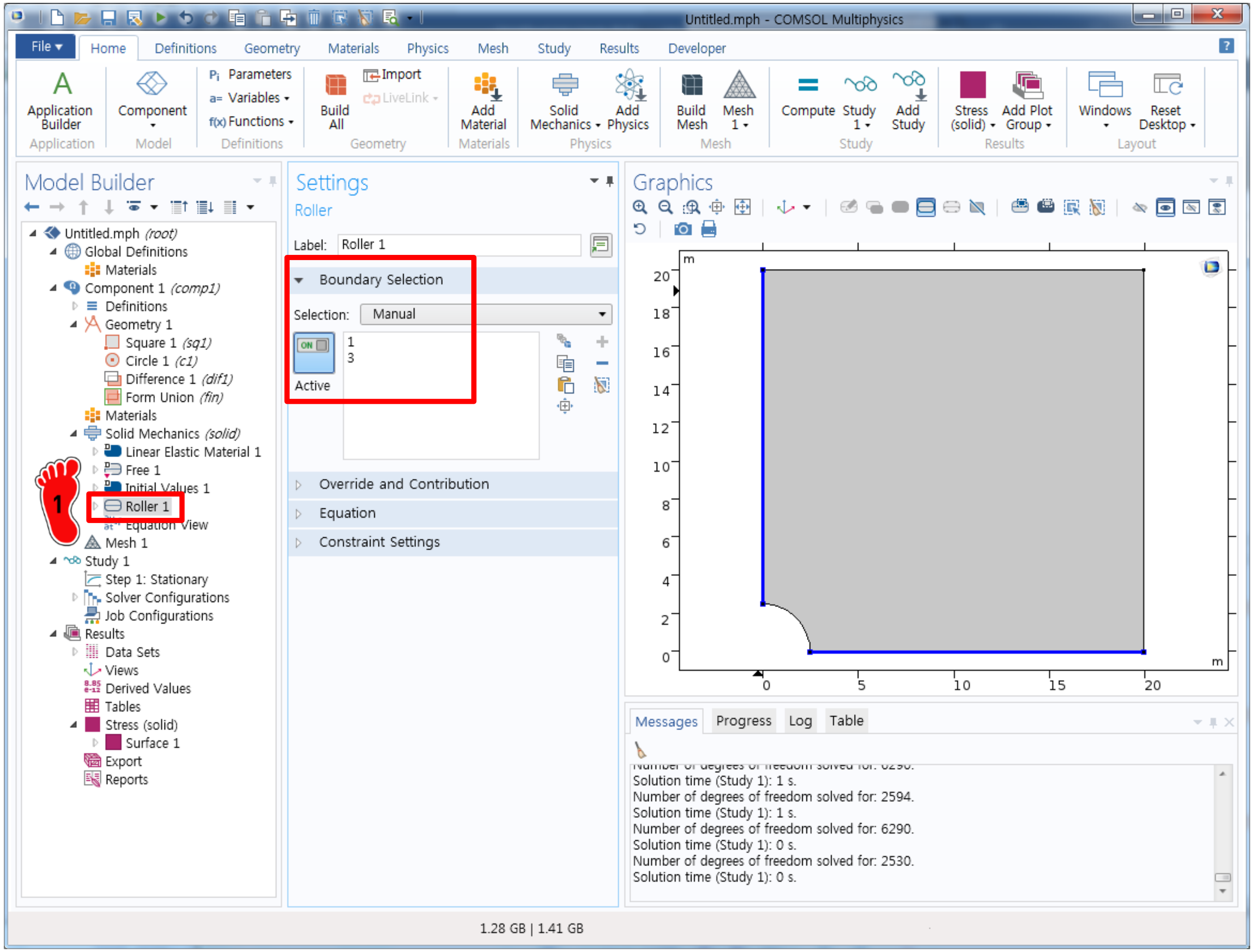
1 Linear Elastic Material 클릭

2 E: 200e9

mu: 0.3

rho : 0

BOUNDARY CONDITION

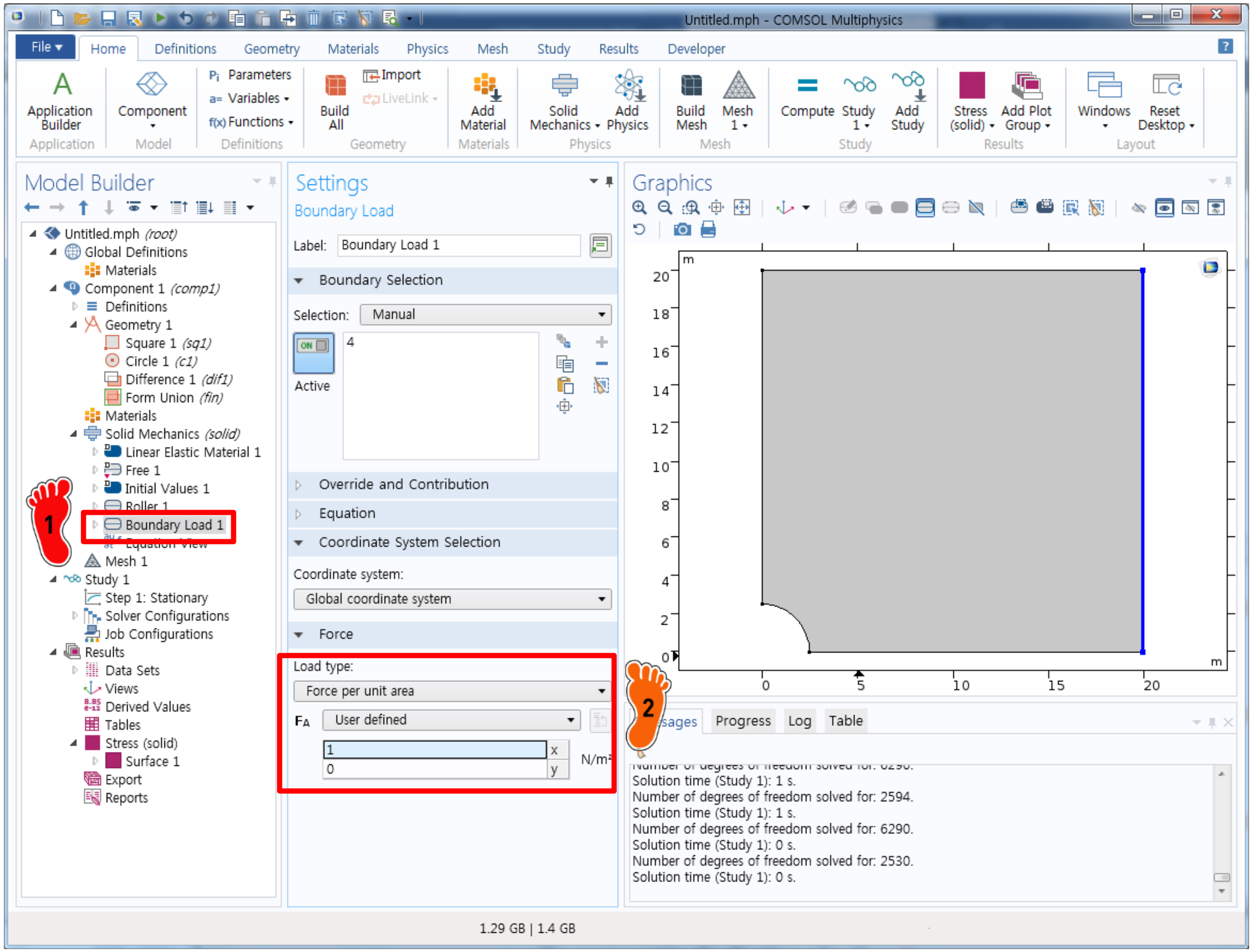


1 사각형 왼쪽과 아래 변에 Roller 조건으로 입력

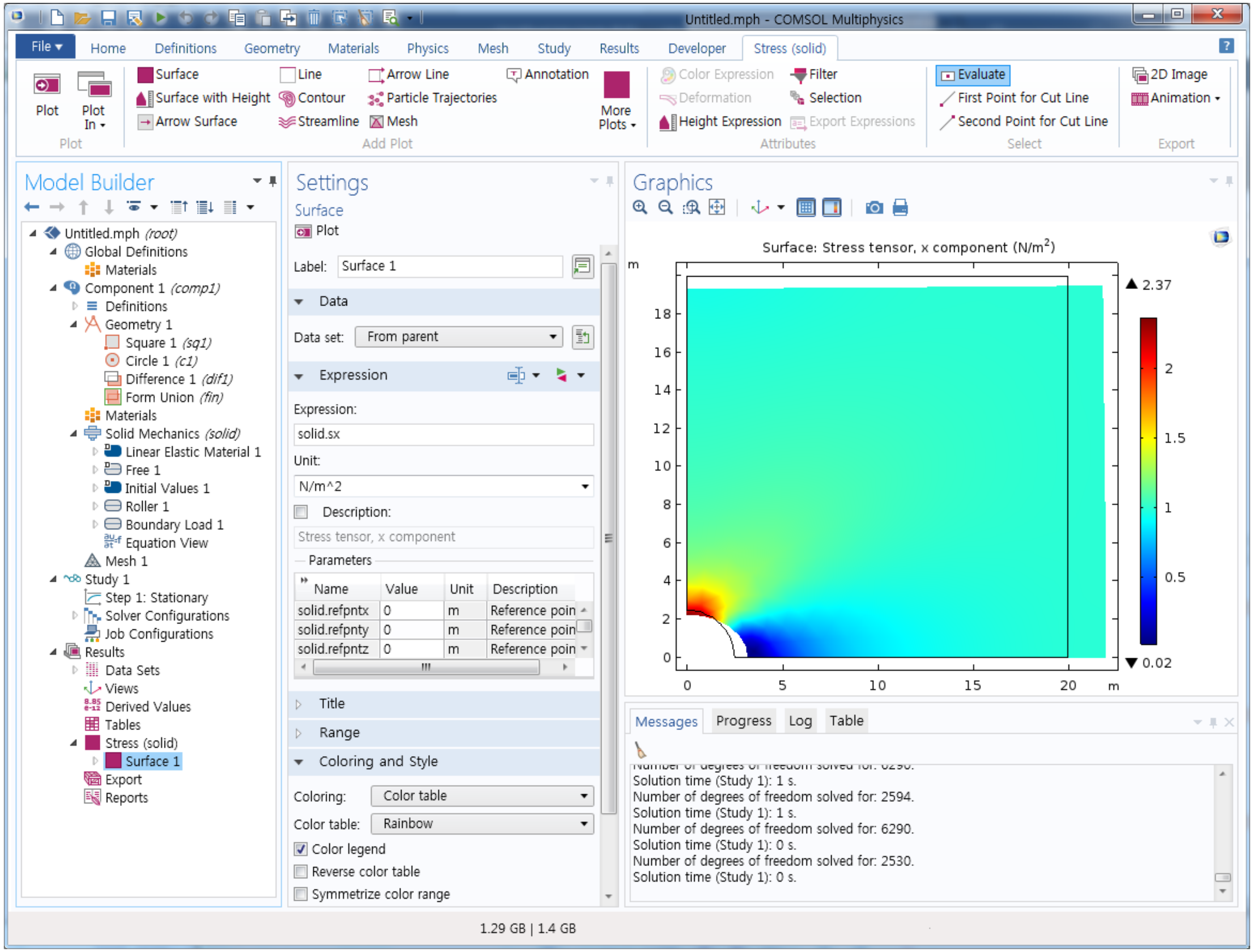
LOADING CONDITION

1 Boundary Load 생성

2 오른쪽 변에 x 방향
1 입력 (단위는 N/m²임)



RESULT



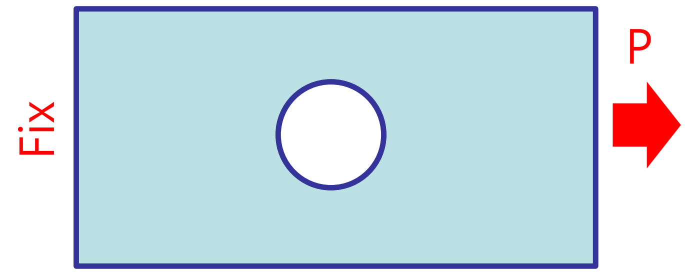
Mesh 생성 후 해석
Full model결과와 비교

응력, 자유도 수 확인

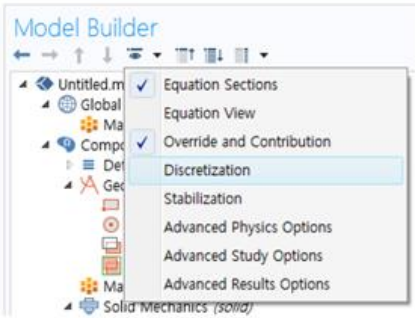
ASSIGNMENT

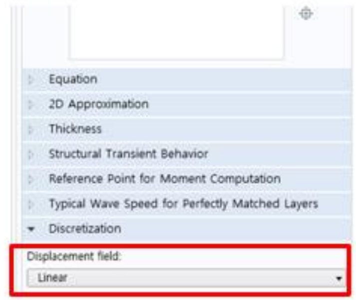
2016년 기말고사 2번 문제(40점)

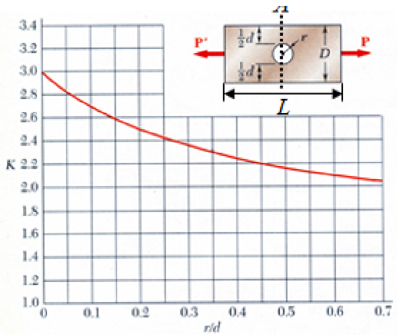
※ Full Model 해석조건



2. [Stress concentration] The following flat bar with hole has a thickness of 1 m. (left side is fixed. Set the element type following discretization option.)







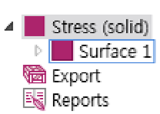
$$E = 200\,GPa,\ \nu = 0.33$$

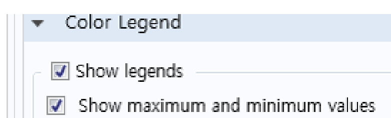
$$L = 5\,m,\ D = 2\,m,\ P = 200\,N$$

$$\sigma_{ave} = \frac{P}{A},\ K = \frac{\sigma_{max}}{\sigma_{ave}}$$

r [m]	0.5	0.4	0.3	0.2
r/d	0.5	0.333	0.214	0.125
K	2.18	2.30	2.46	2.62

(1) Compute the maximum normal stress(`solid.sx`) with r = 0.5 m. Check the stress by mesh dependency applying free triangular and quad elements(linear). Plot the graph as d.o.f vs stress changing mesh size with two cases.(mesh option : normal ~ extremely fine) (15 pts)





(2) Compute the value of K changing the radius(0.2 ~ 0.5 m). Compare the value of K on table with computing result from FEM. (mesh option : quad & extremely fine) (15 pts)

(3) Construct the quarter model and check the maximum stress with r = 0.5 m. Compare the quarter model with full model. (10 pts)