

Solid Mechanics (plane stress/strain)

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



한양대학교
HANYANG UNIVERSITY



- **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.

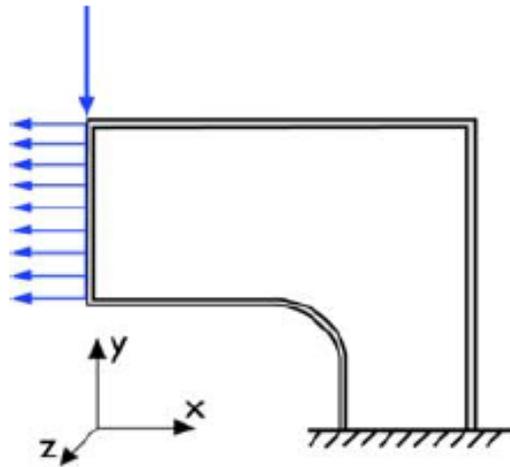


Figure 14-2: Plane stress models plates where the loads are only in the plane; it does not include any out-of-plane stress components.

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.

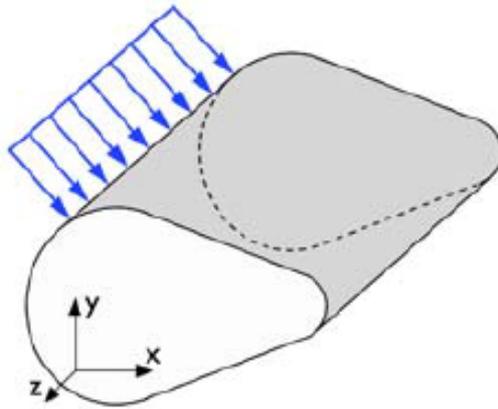


Figure 14-3: A geometry suitable for plane strain analysis.

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

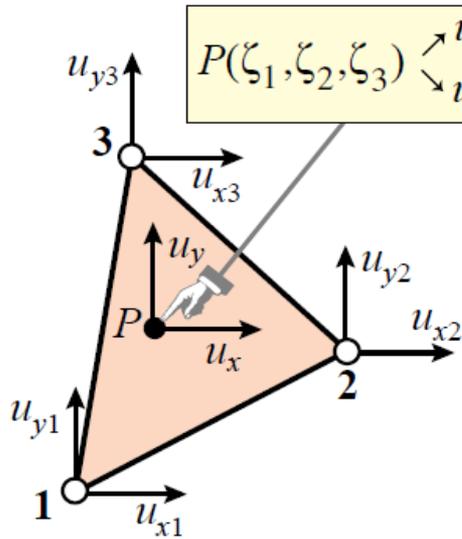
$$\left\{ \begin{array}{l} \varepsilon_x = \frac{1}{E} [\sigma_x - \nu(\sigma_y + \sigma_z)] \\ \varepsilon_y = \frac{1}{E} [\sigma_y - \nu(\sigma_z + \sigma_x)] \\ \varepsilon_z = \frac{1}{E} [\sigma_z - \nu(\sigma_x + \sigma_y)] \end{array} \right\} \xleftrightarrow{G = \frac{E}{2(1+\nu)}} \left\{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

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

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

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

TURNER TRIANGLE (1)



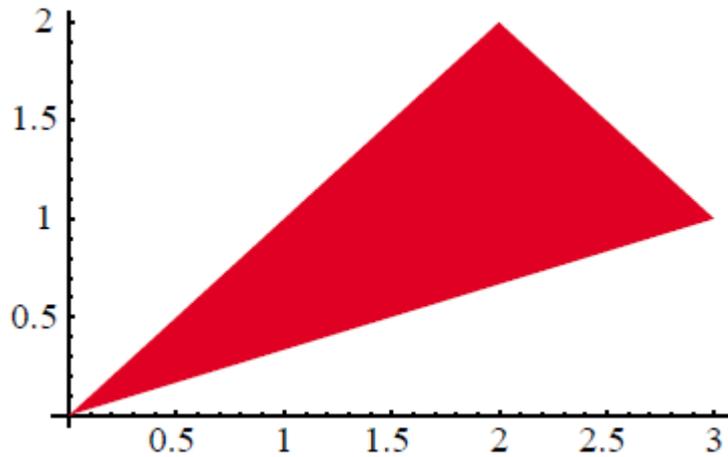
$$u_x = u_{x1}\zeta_1 + u_{x2}\zeta_2 + u_{x3}\zeta_3, \quad u_y = u_{y1}\zeta_1 + u_{y2}\zeta_2 + u_{y3}\zeta_3.$$

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

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

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

TURNER TRIANGLE (2)



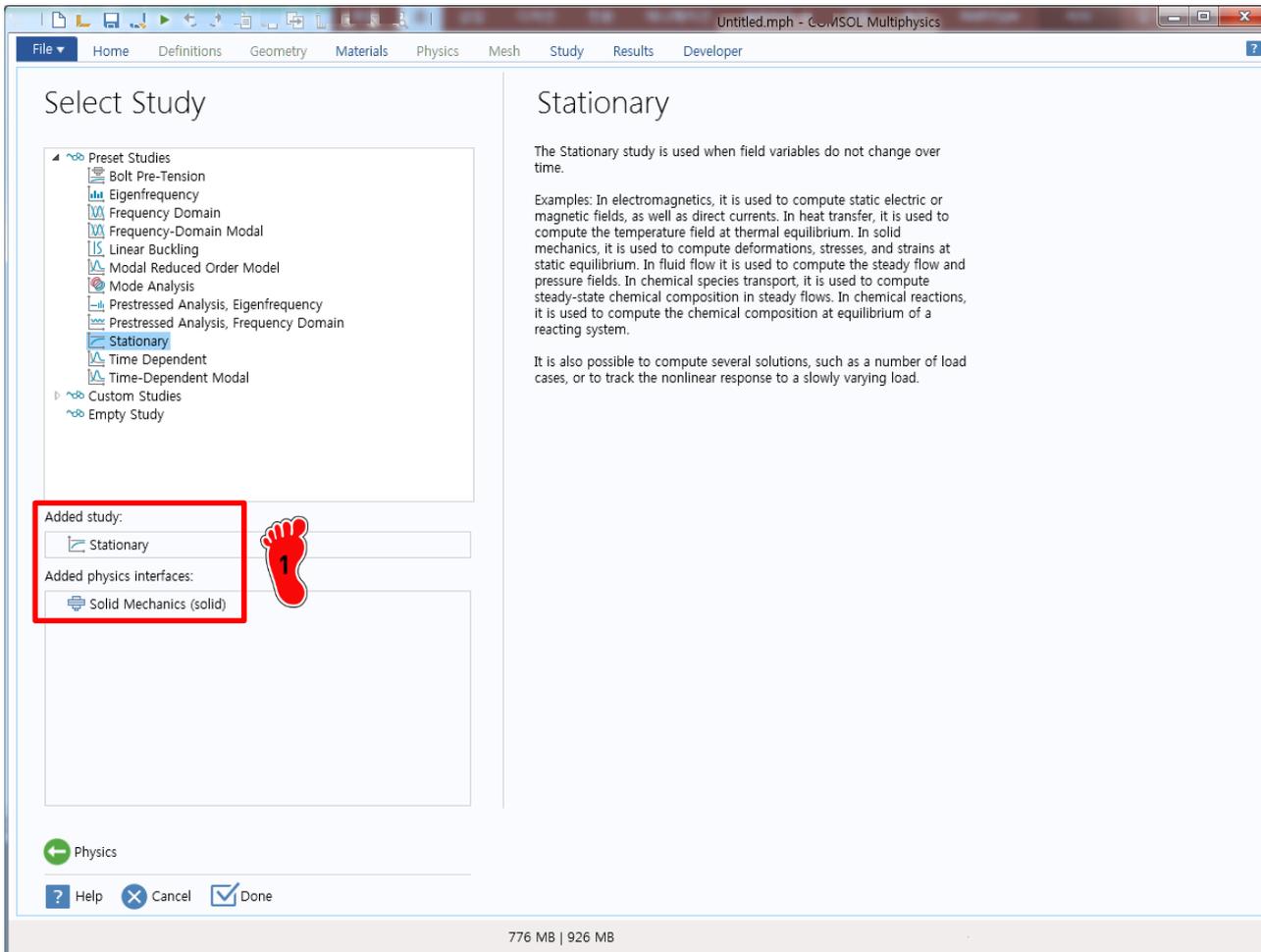
$$E_0 = 60, \nu = 0.25, h = 1$$

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

$$\mathbf{K}_{\text{e plane stress}} = \begin{bmatrix} 11 & 5 & -10 & -2 & -1 & -3 \\ 5 & 11 & 2 & 10 & -7 & -21 \\ -10 & 2 & 44 & -20 & -34 & 18 \\ -2 & 10 & -20 & 44 & 22 & -54 \\ -1 & -7 & -34 & 22 & 35 & 15 \\ 3 & -21 & 18 & -54 & -15 & 75 \end{bmatrix}$$

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

ENVIRONMENT SETTING



Dimension : 2D

Physics :

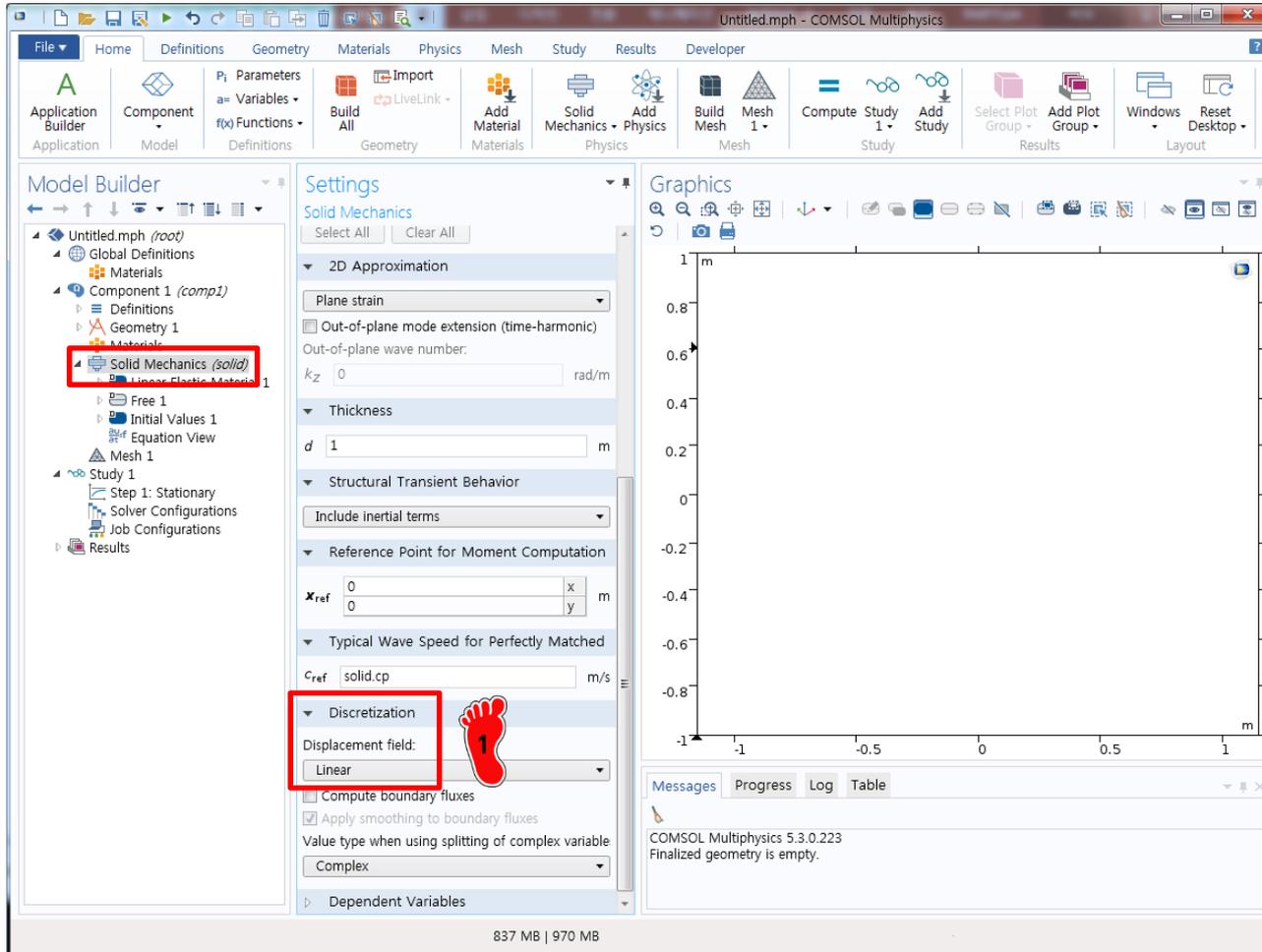
Structural Mechanics

→ Solid Mechanics

Study : Stationary

Done 클릭

DISCRETIZATION



Solid Mechanics 클릭

Discretization 메뉴에서
Displacement field를
Linear로 변경

GEOMETRY CREATION

Settings
Bézier Polygon

Build Selected Build All Objects

Label: Bézier Polygon 1

Type: Solid

Polygon Segments

Added segments

Segment 1 (linear)

Segment 2 (linear)

Add Linear Add Quadratic

Add Cubic Delete

Control points

	x	y	
1	2	2	m
2	3	1	m

Close Curve

Graphics

Messages Progress Log Table

COMSOL Multiphysics 5.3.0.223
Finalized geometry is empty.

832 MB | 965 MB



Bezier Polygon을 이용하여
삼각형 기하형상 생성

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

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

MATERIAL PROPERTY

The screenshot shows the COMSOL Multiphysics interface with the 'Settings' window for a 'Linear Elastic Material'. The 'Young's modulus' field is highlighted with a red box and a red footprint icon, indicating the input point. The 'Poisson's ratio' field is also highlighted with a red box and a red footprint icon. The 'Density' field is also highlighted with a red box. The 'Equation' section is expanded to show the 'Linear Elastic Material' settings. The 'Coordinate System Selection' is set to 'Global coordinate system'. The 'Solid model' is set to 'Isotropic'. The 'Specify' dropdown is set to 'Young's modulus and Poisson's ratio'. The 'Young's modulus' field is set to 'User defined' with a value of 60 Pa. The 'Poisson's ratio' field is set to 'User defined' with a value of 0.25. The 'Density' field is set to 'User defined' with a value of 0 kg/m³. The 'Geometric Nonlinearity' section is expanded to show 'Force linear strains' and 'Additive strain decomposition' options. The 'Energy Dissipation' section is also visible. The 'Graphics' window shows a 2D plot of a triangle with vertices at (0,0), (2,2), and (3,1). The 'Messages' window shows the text: 'COMSOL Multiphysics 5.3.0.223', 'Finalized geometry is empty.', and 'Finalized geometry has 1 domain, 3 boundaries, and 3 vertices.'



Linear Elastic Material 클릭

물성치 입력

$E: 60$

$\mu: 0.25$

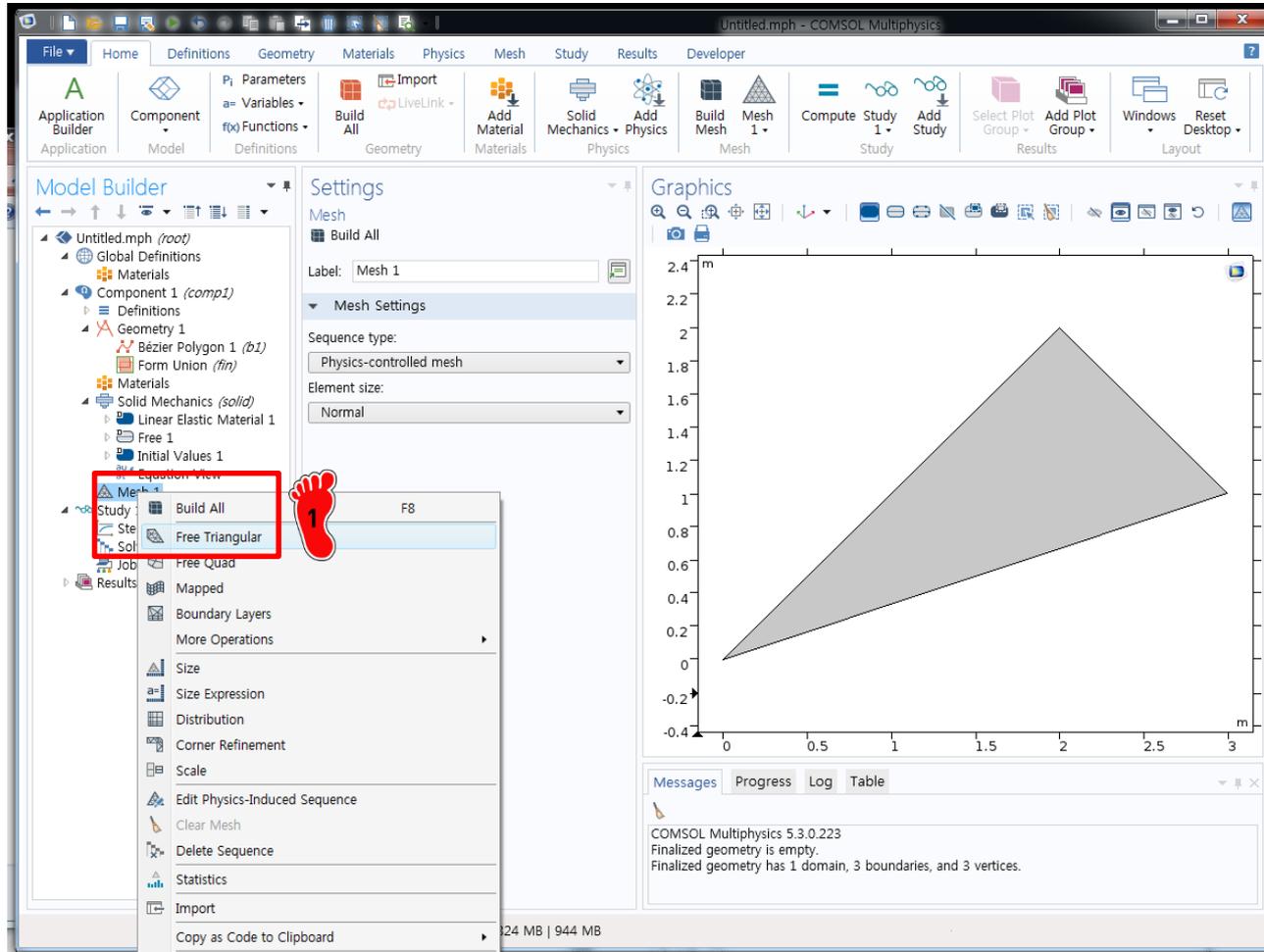
$\rho: 0$

MESH

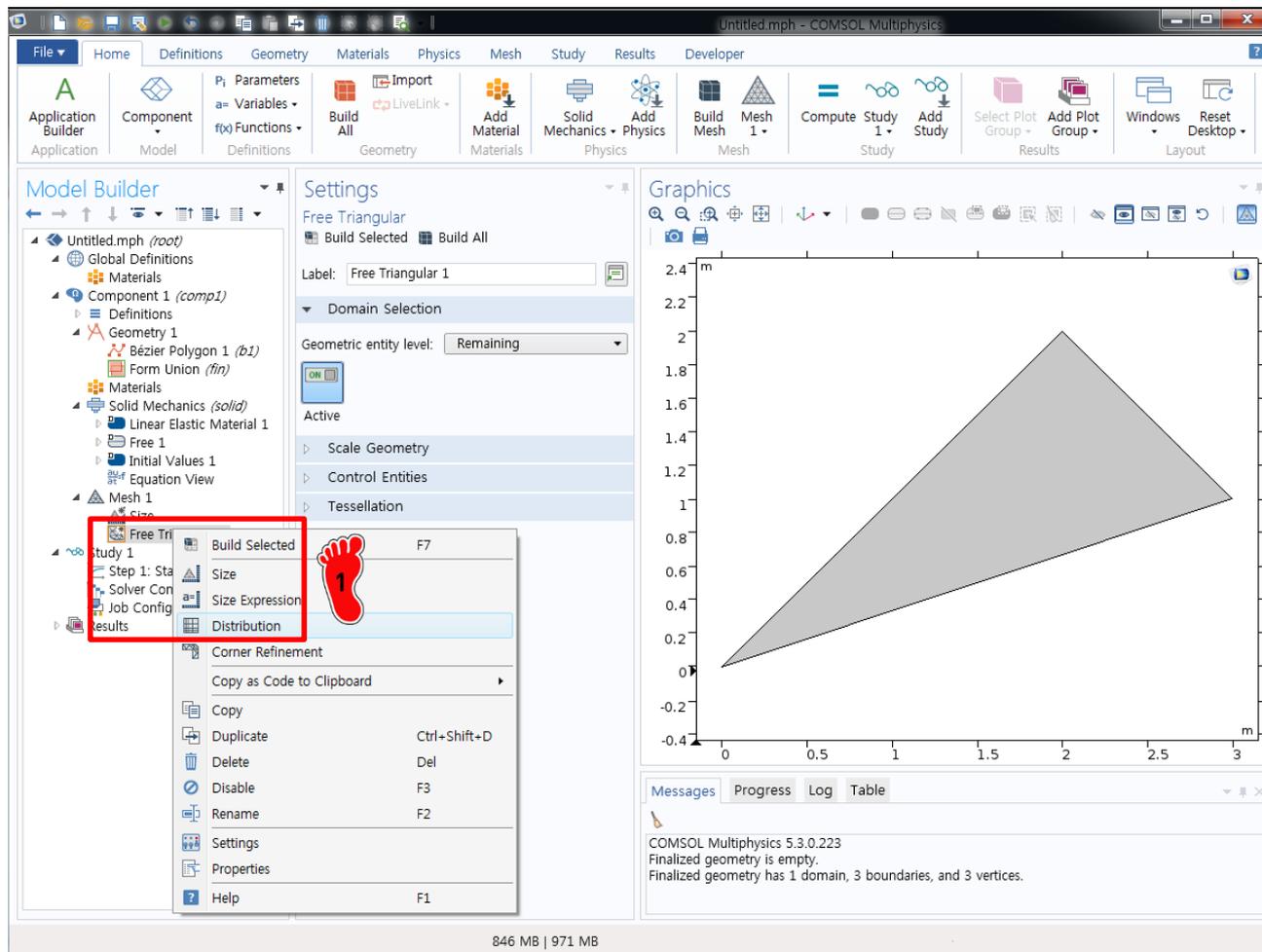


Mesh 마우스 우클릭

Free Triangular 클릭



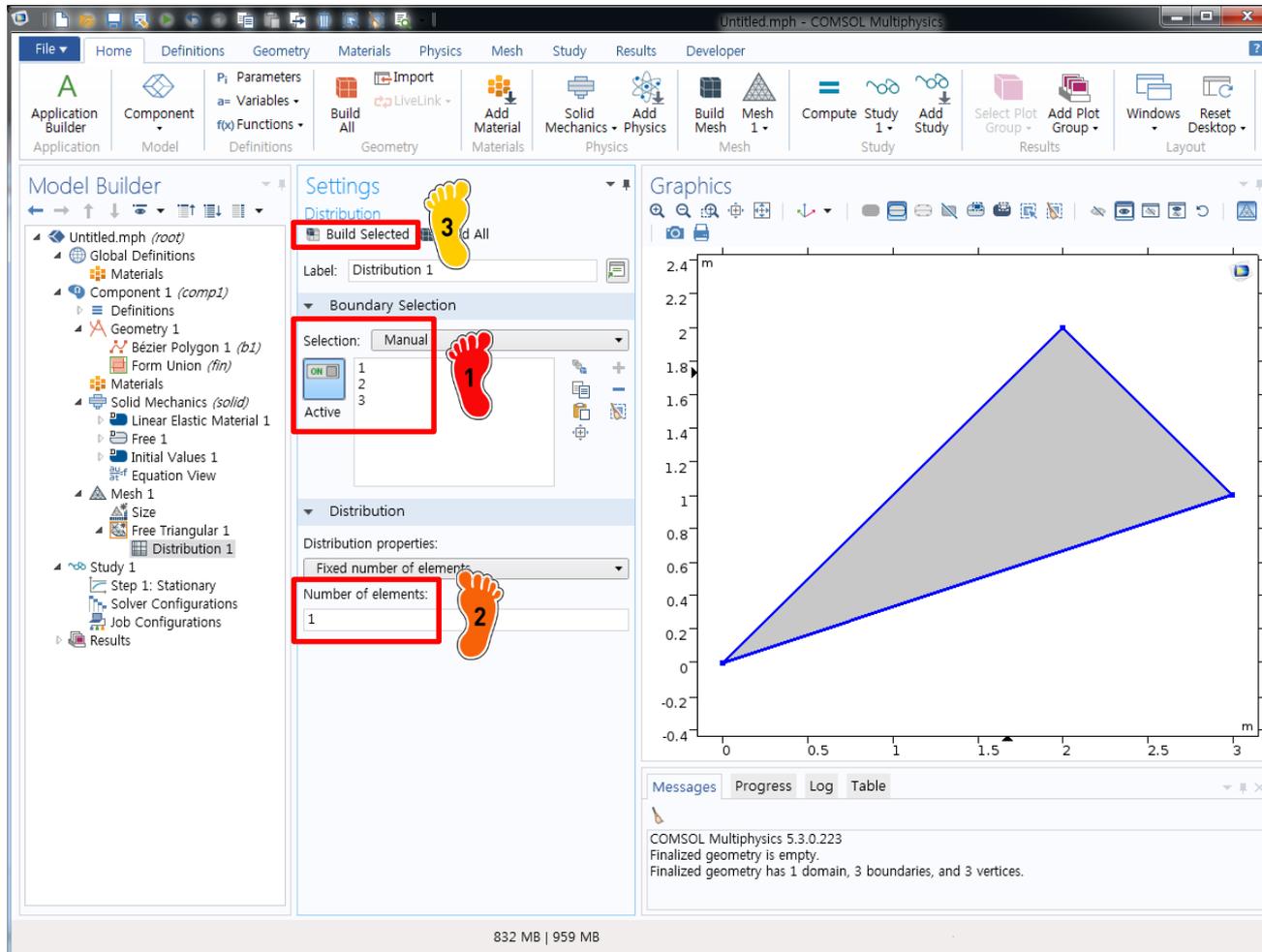
MESH



Free Triangular 마우스
우클릭

Distribution 클릭

MESH



1 3개의 boundary 선택

2 Number of elements를 1로 설정

3 1개의 삼각형 요소 생성

STIFFNESS MATRIX

The screenshot shows the COMSOL Multiphysics interface. The Model Builder on the left shows a study configuration for a linear elastic material. The Graphics window displays a triangular mesh. A context menu is open over the 'Assemble' button, with a red footprint icon pointing to it. The 'Assemble' button is highlighted in blue.

1 해석 실행 후,
Assemble 탭에서 stiffness matrix를 선택 후 compute

Result → Derived Values에
서 System Matrix 클릭 후
Stiffness matrix 확인

Messages	Progress	Log	Table 1		
8.85 e-12	AUTC	8.5 e-1	850 e-3		
0.85					
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

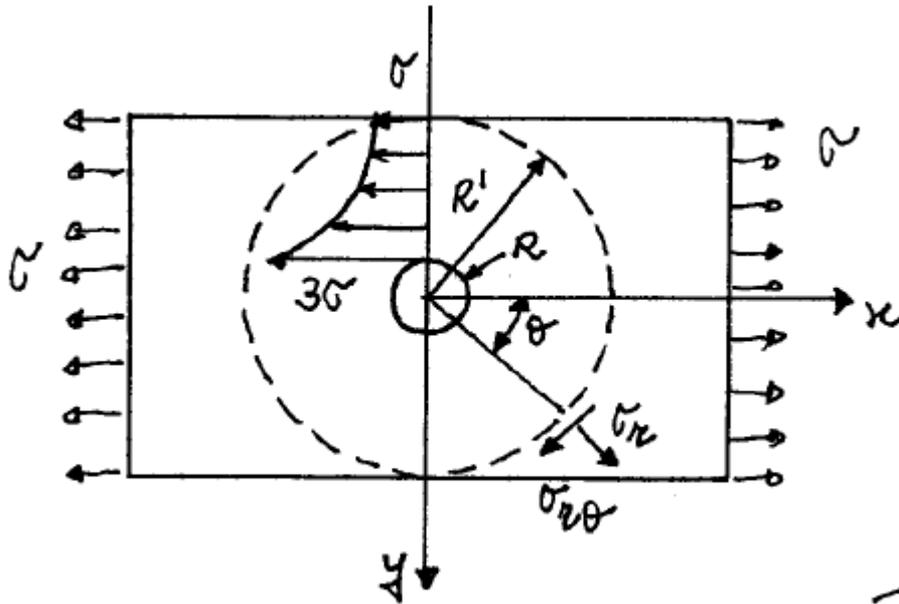
K_e plane strain =

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

- **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)

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

for $r = R$:

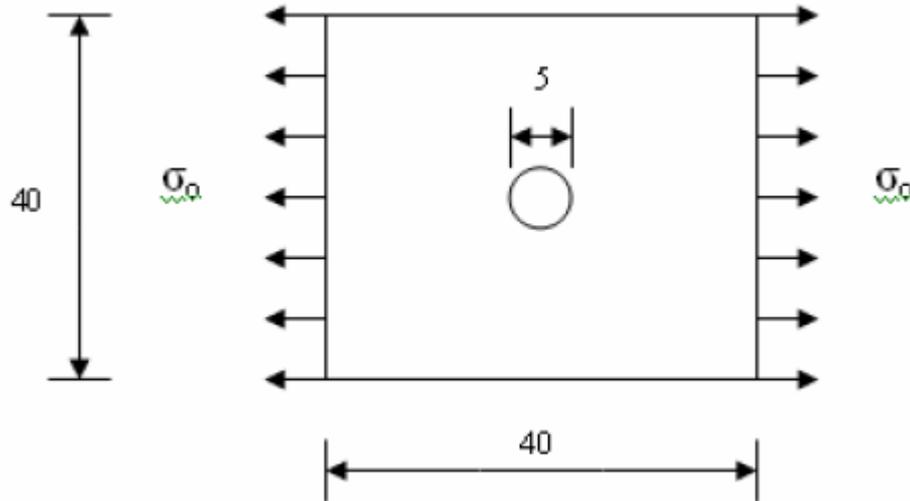
$$\left. \begin{array}{l} \sigma_r = 0 \\ \sigma_\theta = \sigma(1 - 2\cos 2\theta) \\ \sigma_{r\theta} = 0 \end{array} \right\} \rightarrow \begin{cases} \text{max: } \sigma_\theta = 3\sigma @ \theta = \frac{\pi}{2}, \frac{3\pi}{2} \\ \text{min: } \sigma_\theta = -\sigma @ \theta = 0, \pi \end{cases}$$

stress concentration factor = 3 independent of R

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

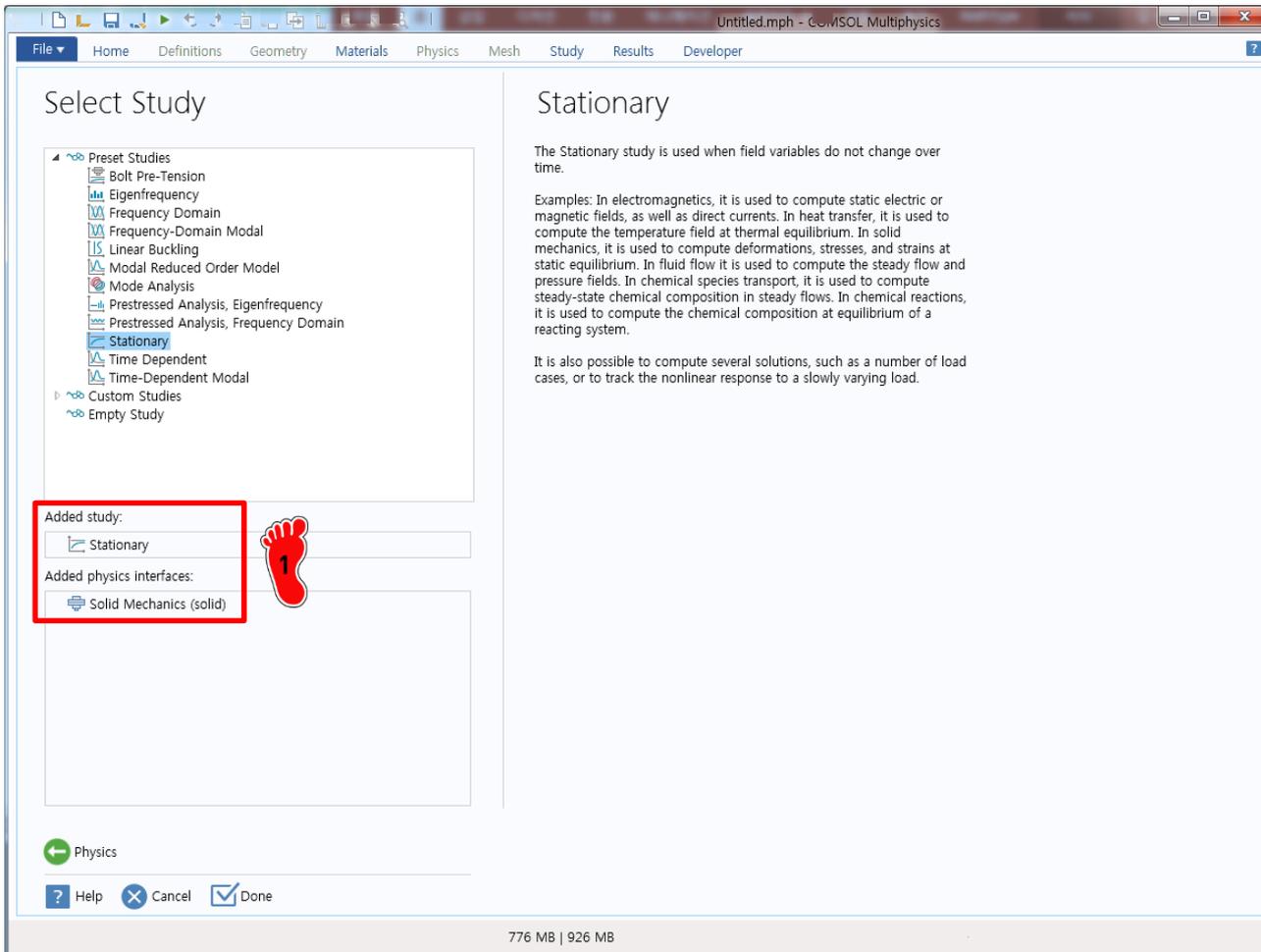
$$\text{solution: } \begin{cases} \sigma_r = \frac{\sigma}{2} \left(1 - \frac{R^2}{r^2} \right) + \frac{\sigma}{2} \left(1 + 3\frac{R^4}{r^4} - 4\frac{R^2}{r^2} \right) \cos 2\theta \\ \sigma_\theta = \frac{\sigma}{2} \left(1 + \frac{R^2}{r^2} \right) - \frac{\sigma}{2} \left(1 + 3\frac{R^4}{r^4} \right)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

KIRSCH'S PROBLEM: FEM



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

ENVIRONMENT SETTING



Dimension : 2D

Physics :

Structural Mechanics

→ Solid Mechanics

Study : Stationary

Done 클릭

PLANE STRESS

1 Solid Mechanics 클릭

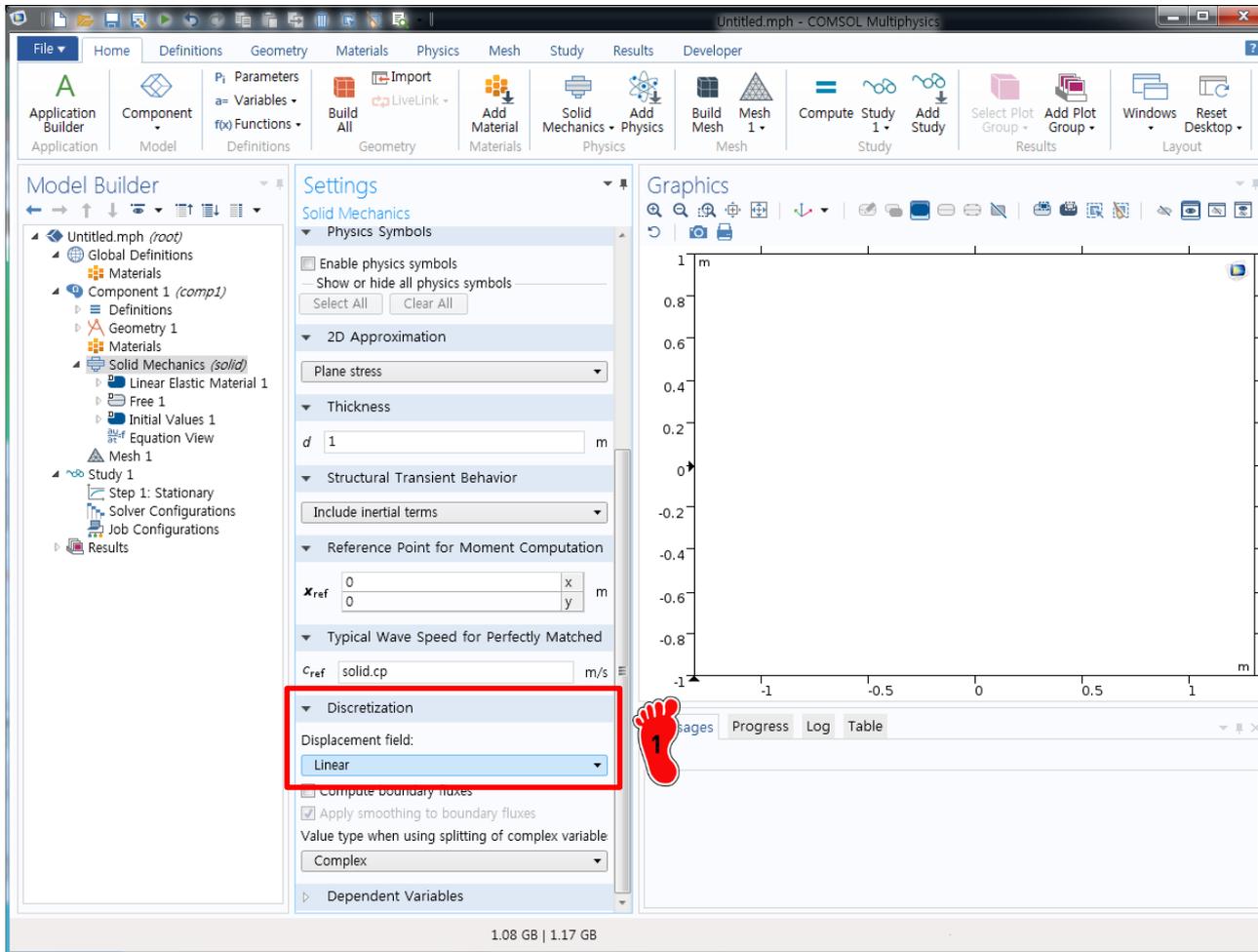
2 2D Approximation
Plane stress 로 변경

The screenshot displays the COMSOL Multiphysics software interface. The 'Model Builder' tree on the left shows the 'Solid Mechanics (solid)' node highlighted with a red box and a red footprint icon labeled '1'. The 'Settings' panel for 'Solid Mechanics' is visible, with the '2D Approximation' section expanded and 'Plane stress' selected in the dropdown menu, also highlighted with a red box and a red footprint icon labeled '2'. The 'Graphics' window shows a 2D coordinate system with axes ranging from -1 to 1 on both x and y axes. The status bar at the bottom indicates '1.1 GB | 1.21 GB'.

DISCRETIZATION



Discretization
Displacement field를
Linear로 변경



GEOMETRY CREATION

1 Square 생성

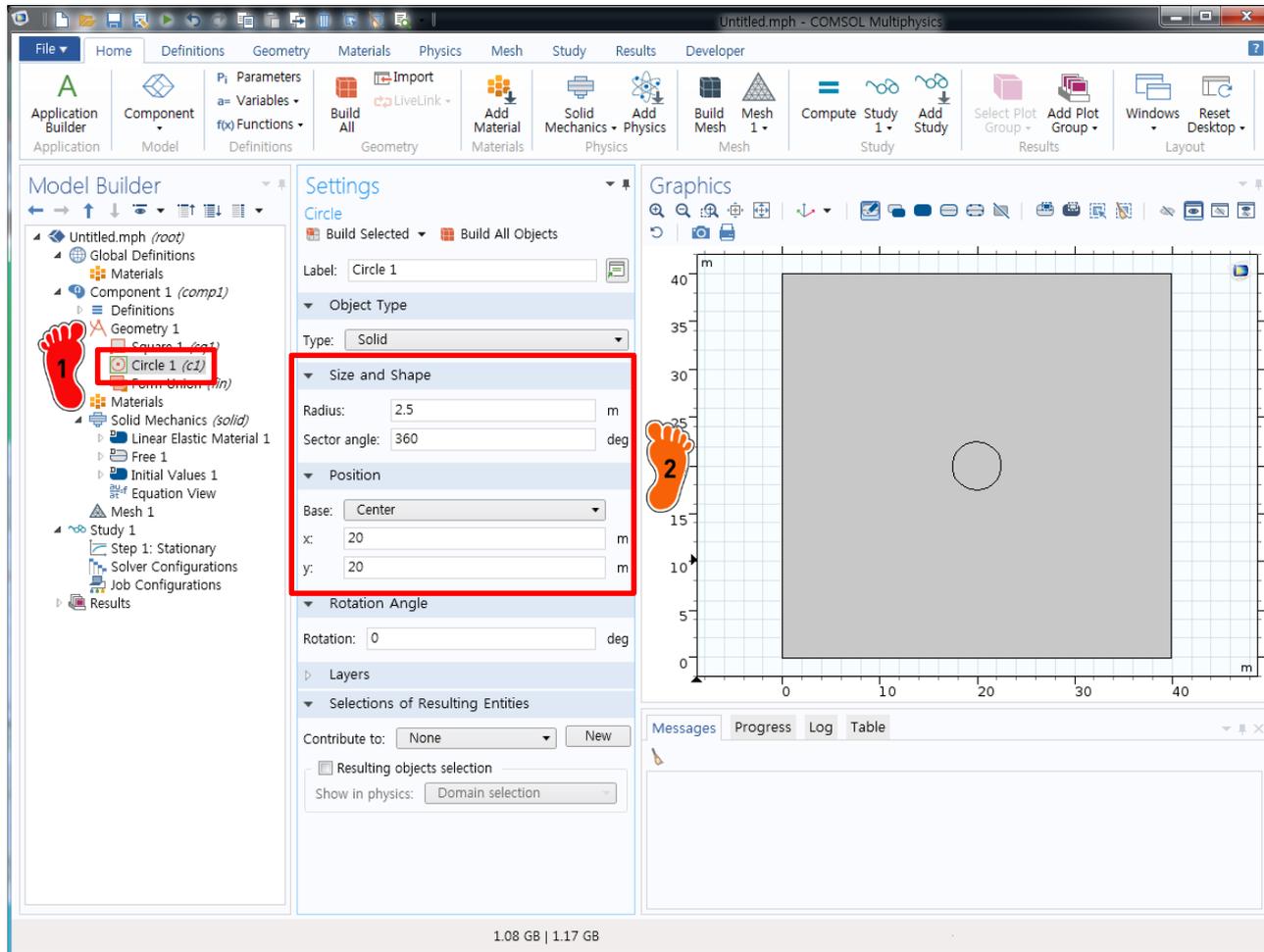
2 길이 40의 정사각형 생성

The screenshot displays the COMSOL Multiphysics software interface. The top menu bar includes File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer. The main workspace is divided into three panels: Model Builder, Settings, and Graphics.

- Model Builder:** Shows a tree view of the model. Under 'Definitions', 'Geometry 1' is expanded, and 'Square 1 (sq1)' is highlighted with a red box and a red footprint icon labeled '1'.
- Settings:** Shows the configuration for 'Square 1'. The 'Size' section has 'Side length: 40 m' highlighted with a red box and an orange footprint icon labeled '2'. Other settings include 'Type: Solid', 'Base: Corner', and 'Rotation: 0 deg'.
- Graphics:** Shows a 2D plot of a square with side length 40 m on a grid. The axes are labeled from 0 to 40 m.

At the bottom of the window, the memory usage is shown as 1.08 GB | 1.16 GB.

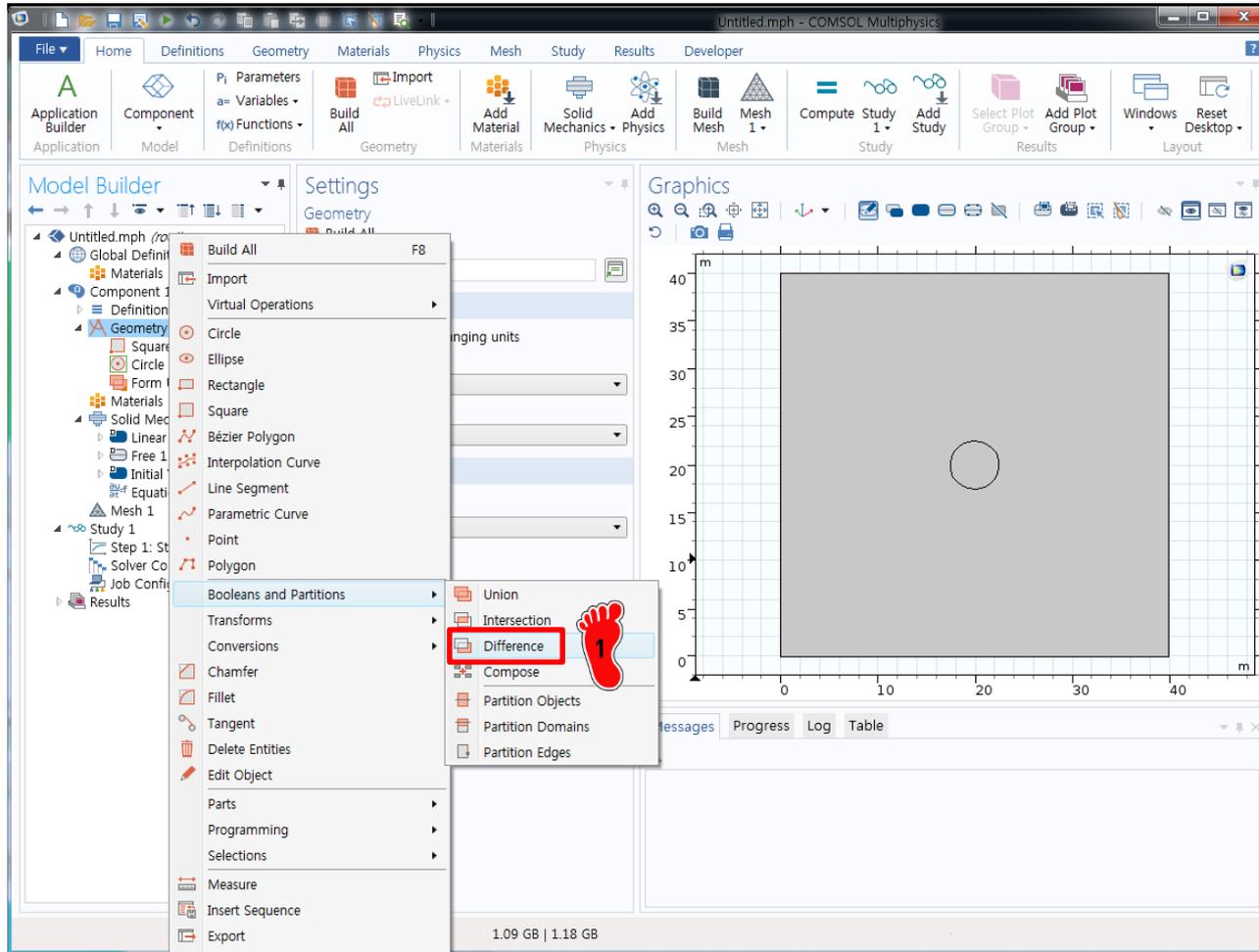
GEOMETRY CREATION



1 Circle 생성

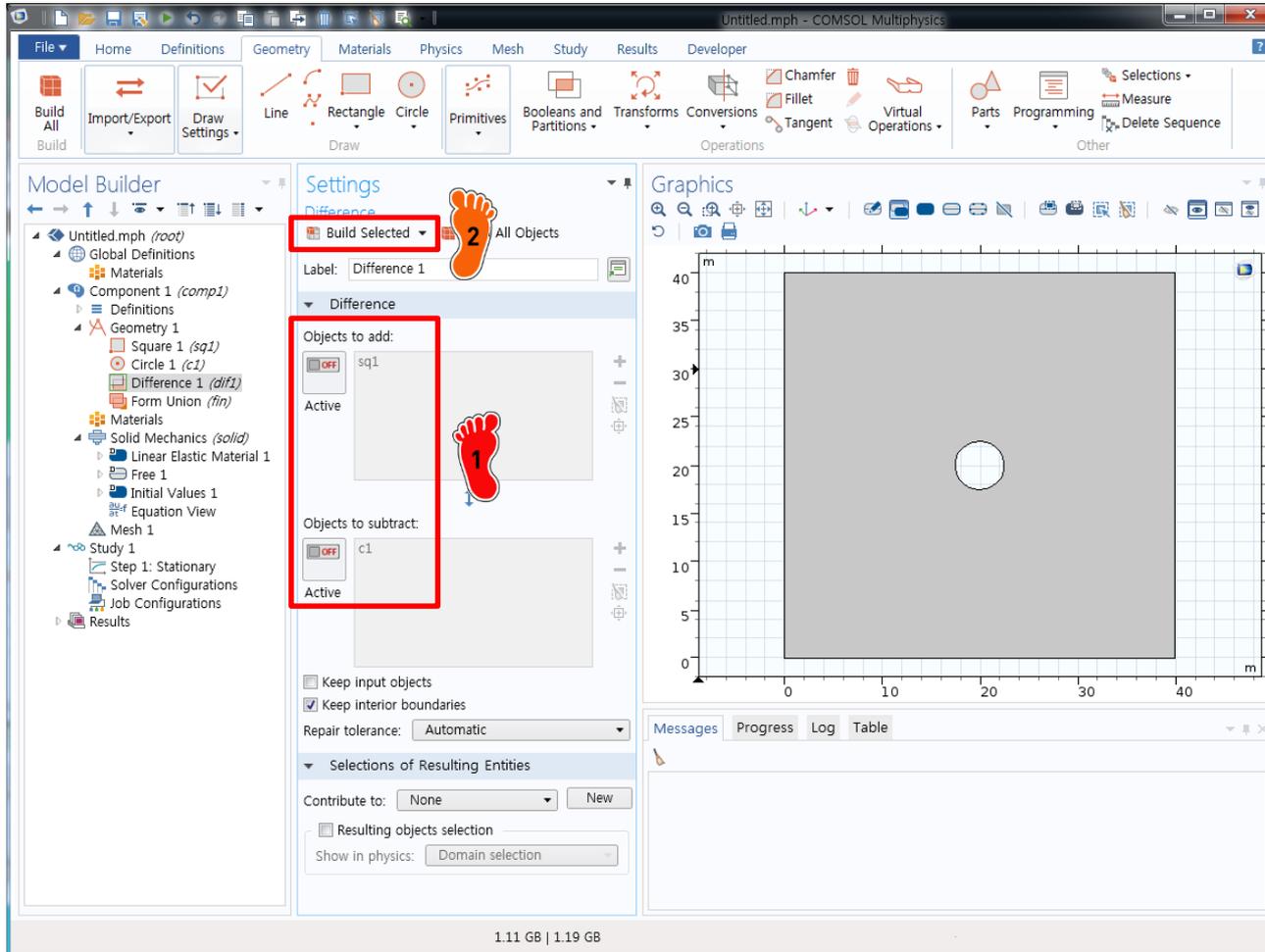
2 중심 (20,20)
반지름 2.5 원 생성

GEOMETRY CREATION



Geometry 오른쪽 클릭
 → Booleans and Partitions
 → Difference 클릭

GEOMETRY CREATION

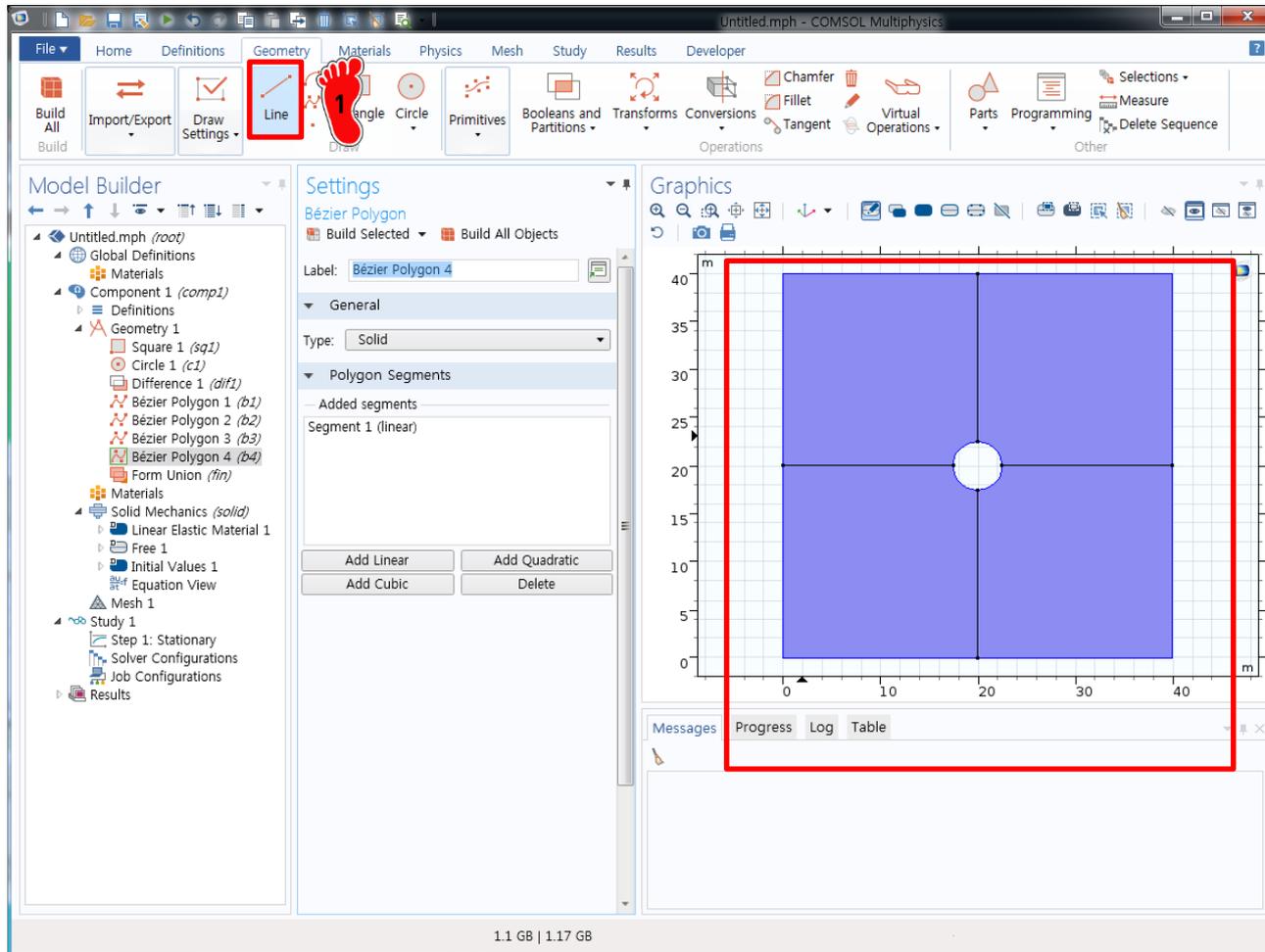


1 Objects to add
사각형 선택

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

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Build All Build Import/Export Draw Settings

Line Rectangle Circle Primitives

Booleans and Partitions Transforms Conversions

Chamfer Fillet Tangent Virtual Operations

Parts Programming Measure Delete Sequence

Model Builder

Settings
Linear Elastic Material

Graphics

40 m
35
30
25
20
15
10
5
0

0 10 20 30 40 m

Messages Progress Log Table

Formed union of 1 solid object and 4 curve objects.
Finalized geometry has 4 domains, 16 boundaries, and 12 vertices.

1.09 GB | 1.17 GB

Model Builder

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Square 1 (sq1)
 - Circle 1 (c1)
 - Difference 1 (dif1)
 - Bézier Polygon 1 (b1)
 - Bézier Polygon 2 (b2)
 - Bézier Polygon 3 (b3)
 - Bézier Polygon 4 (b4)
 - Form Union (fin)
 - Materials
 - Linear Elastic Material 1**
 - Free 1
 - Initial Values 1
 - Equation View
 - Mesh 1
 - Study 1
 - Step 1: Stationary
 - Solver Configurations
 - Job Configurations
 - Results

Settings

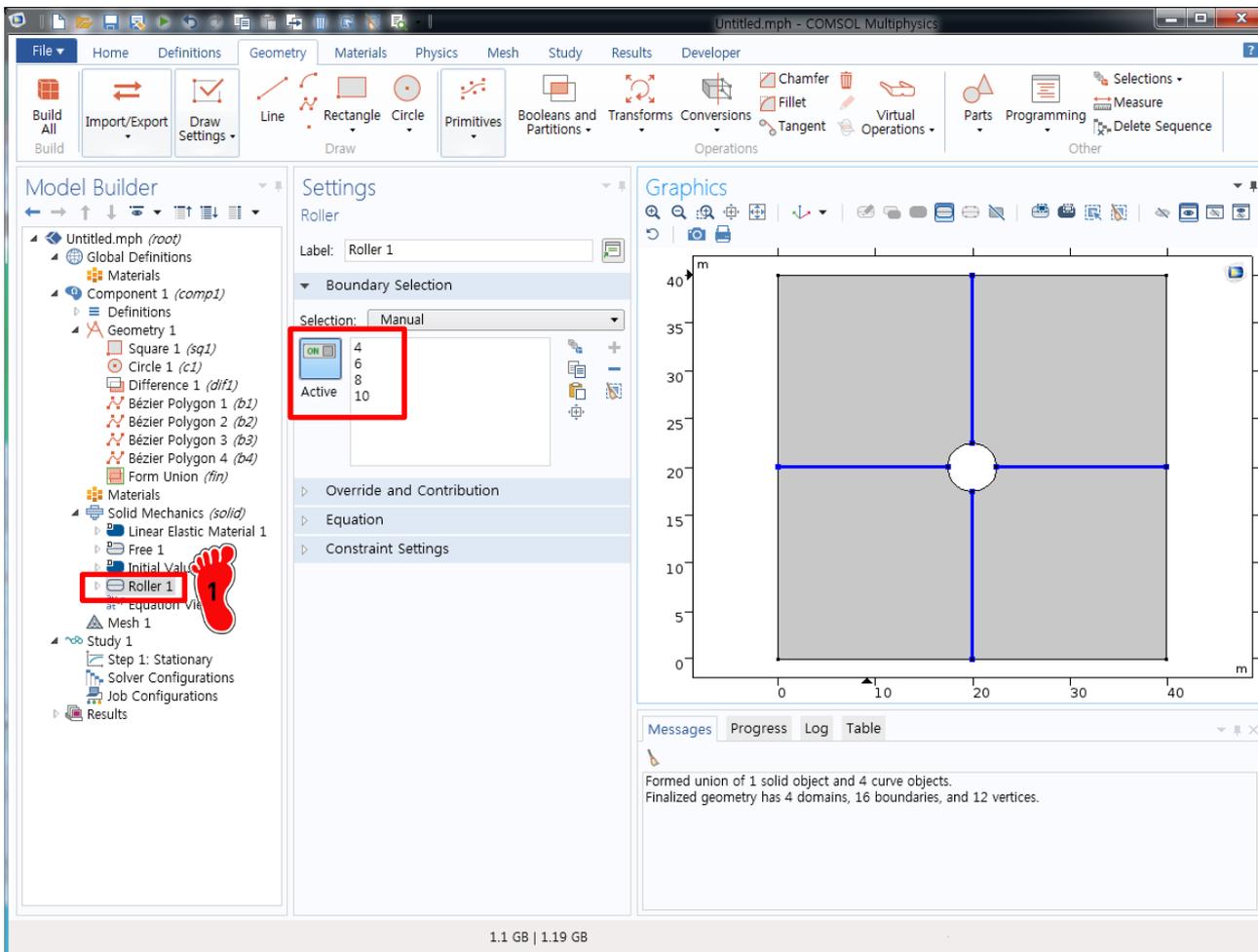
Linear Elastic Material

- Override and Contribution
- Equation
- Model Input
- Coordinate System Selection
- Coordinate system: Global coordinate system
- Linear Elastic Material
 - Nearly incompressible material
 - Solid model: Isotropic
 - Specify: Young's modulus and Poisson's ratio
 - Young's modulus:
 - E: User defined
 - 200e9 Pa
 - Poisson's ratio:
 - ν : User defined
 - 0.3
 - Density:
 - ρ : User defined
 - 0 kg/m³
- Geometric Nonlinearity

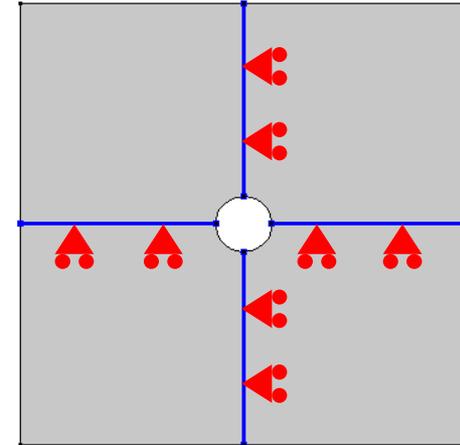
1 Linear Elastic Material 클릭

2 E: 200e9
mu: 0.3
rho : 0

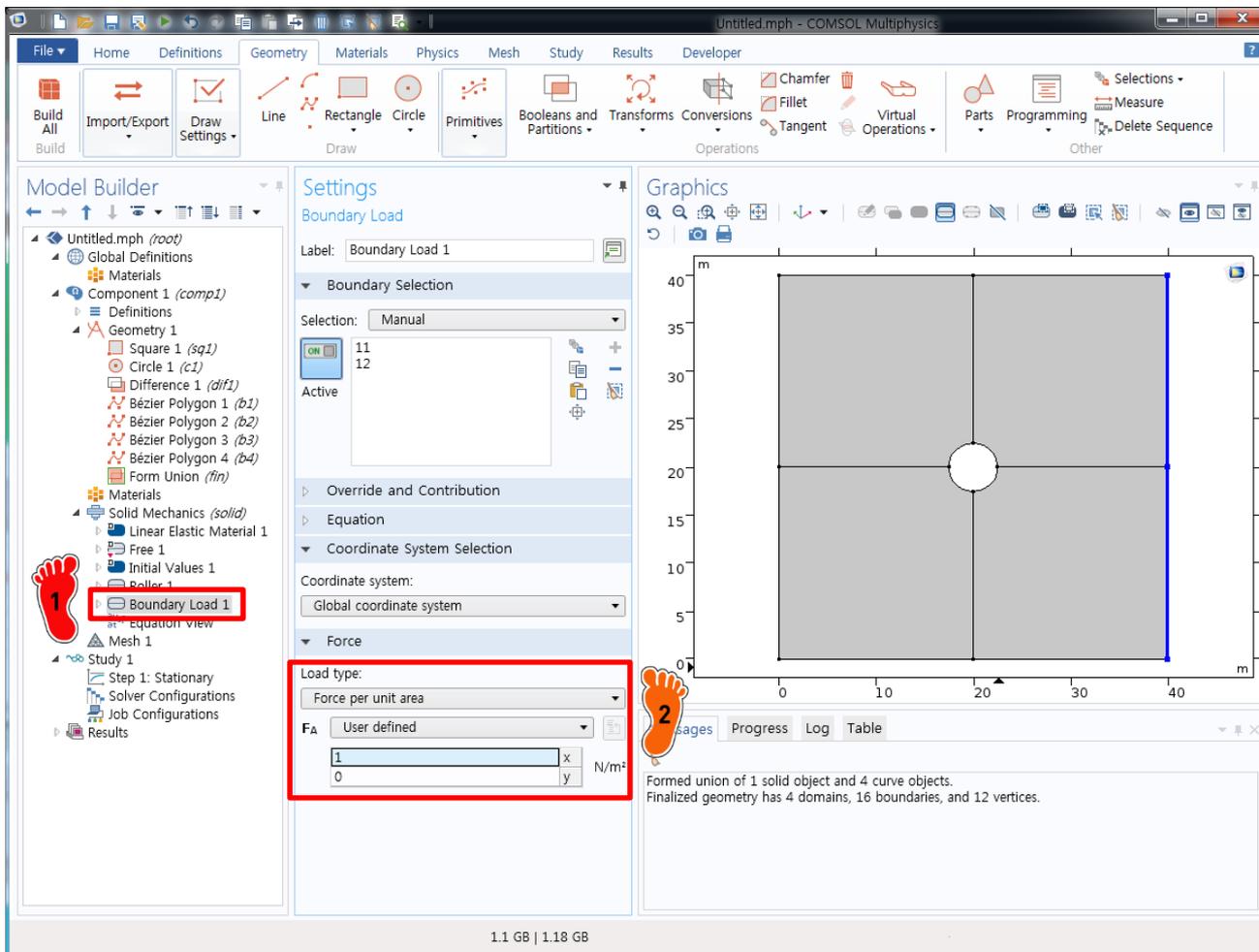
BOUNDARY CONDITION



1 사각형 내부 4개 변을
Roller 조건으로 입력

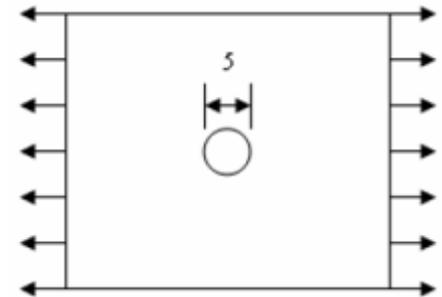


LOADING CONDITION



1 Boundary Load 생성

2 오른쪽 두 변에 x 방향
1 입력 (단위는 N/m²임)



LOADING CONDITION

Model Builder: Boundary Load 2

Settings: Boundary Load

Label: Boundary Load 2

Boundary Selection: Manual

Selection: 1, 3

Active: ON

Override and Contribution: []

Equation: []

Coordinate System Selection: Global coordinate system

Force

Load type: Force per unit area

FA: User defined

-1	x	N/m ²
0	y	

Graphics: 40m, 35, 30, 25, 20, 15, 10, 5, 0, 0, 10, 20, 30, 40, m

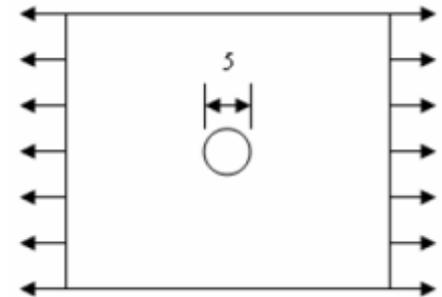
Pages: Progress Log Table

Formed union of 1 solid object and 4 curve objects.
Finalized geometry has 4 domains, 16 boundaries, and 12 vertices.

1.1 GB | 1.18 GB

1 Boundary Load 생성

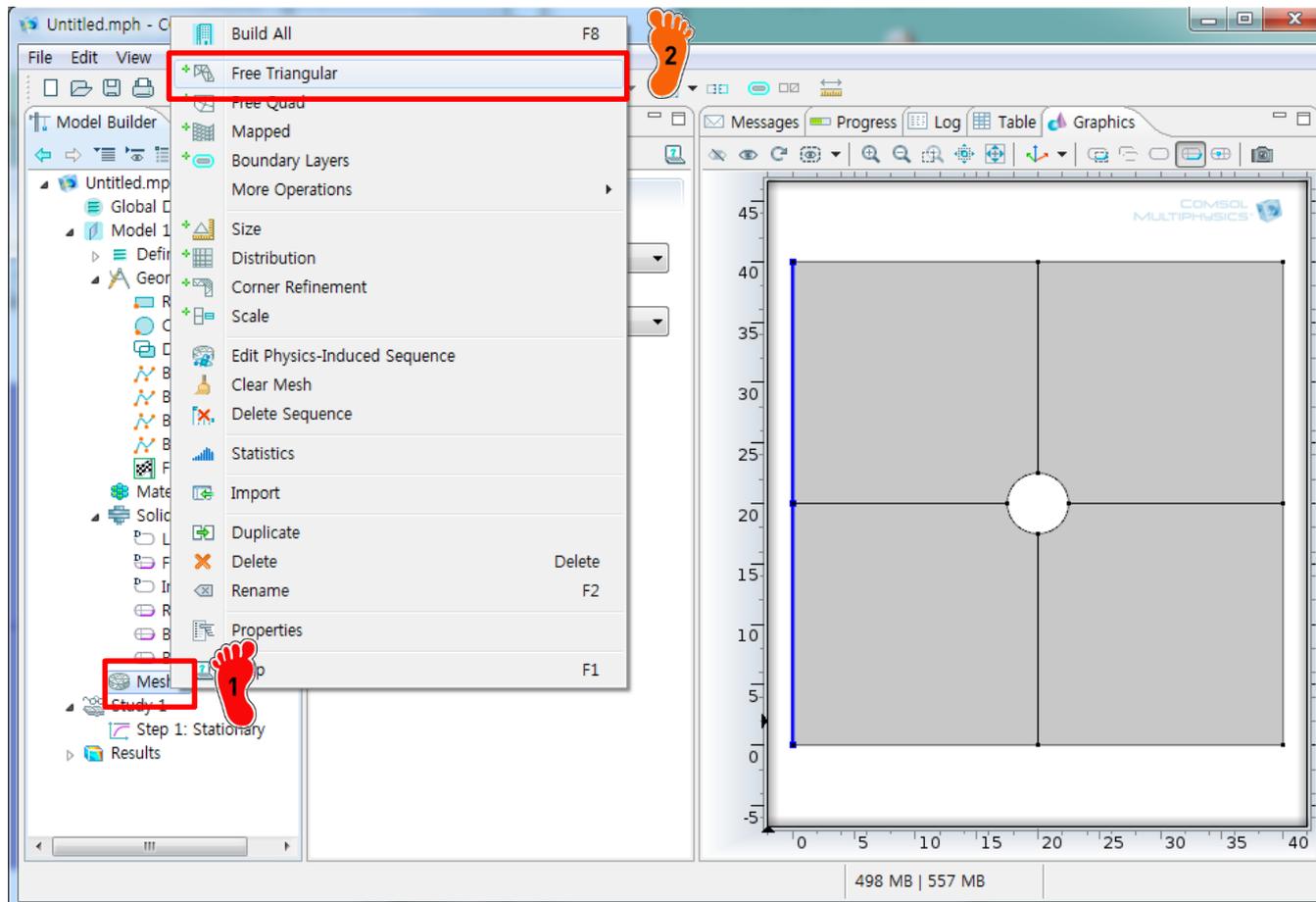
2 오른쪽 두 변에 x 방향
-1 입력 (단위는 N/m²임)



MESH

1 Mesh 마우스 우클릭

2 Free Triangular 클릭



MESH



기본 Mesh 생성(삼각형)

The screenshot shows the COMSOL Multiphysics interface for a meshing task. The top toolbar includes various modeling tools like 'Build All', 'Import/Export', 'Draw Settings', 'Line', 'Rectangle', 'Circle', 'Primitives', 'Booleans and Partitions', 'Transforms', 'Conversions', 'Chamfer', 'Fillet', 'Tangent', 'Virtual Operations', 'Parts', 'Programming', 'Measure', and 'Delete Sequence'. The 'Model Builder' tree on the left shows a hierarchy starting with 'Untitled.mph (root)', followed by 'Global Definitions', 'Materials', 'Component 1 (comp1)', 'Definitions', 'Geometry 1' (containing Square 1, Circle 1, and a Difference of Béziers), 'Materials', 'Solid Mechanics (solid)', and 'Mesh 1' (highlighted with a red box and a red footprint icon). The 'Settings' panel for 'Mesh 1' shows 'Build All' highlighted with a red box, and 'Mesh Settings' with 'Sequence type' set to 'Physics-controlled mesh' and 'Element size' set to 'Normal'. The 'Graphics' window displays a 2D mesh of a square domain with a circular hole, with axes ranging from 0 to 40 meters. The 'Messages' window at the bottom provides details: 'Formed union of 1 solid object and 4 curve objects. Finalized geometry has 4 domains, 16 boundaries, and 12 vertices. Complete mesh consists of 952 domain elements and 124 boundary elements.'

1.11 GB | 1.19 GB

MESH

Mesh 우클릭
→ Statistics 클릭

요소 개수, DOF, Quality 분포 정보 등을 보여줌

1

Element Quality

Quality measure: Skewness

Statistics

Complete mesh

Mesh vertices: 520

Element type: All elements

Triangular elements: 952

Edge elements: 124

Vertex elements: 12

Domain element statistics

Number of elements: 952

Minimum element quality: 0.5839

Average element quality: 0.8442

Element area ratio: 0.01577

Mesh area: 1581 m²

Element Quality Histogram

1.1 GB | 1.19 GB

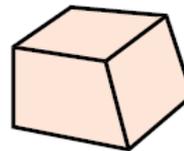
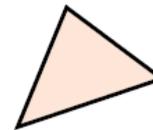
Messages Progress Log Table

Formed union of 1 solid object and 4 curve objects.
Finalized geometry has 4 domains, 16 boundaries, and 12 vertices.
Complete mesh consists of 952 domain elements and 124 boundary elements.

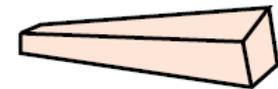
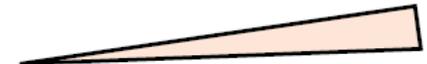
2D/3D BAD ASPECT RATIO ELEMENT

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

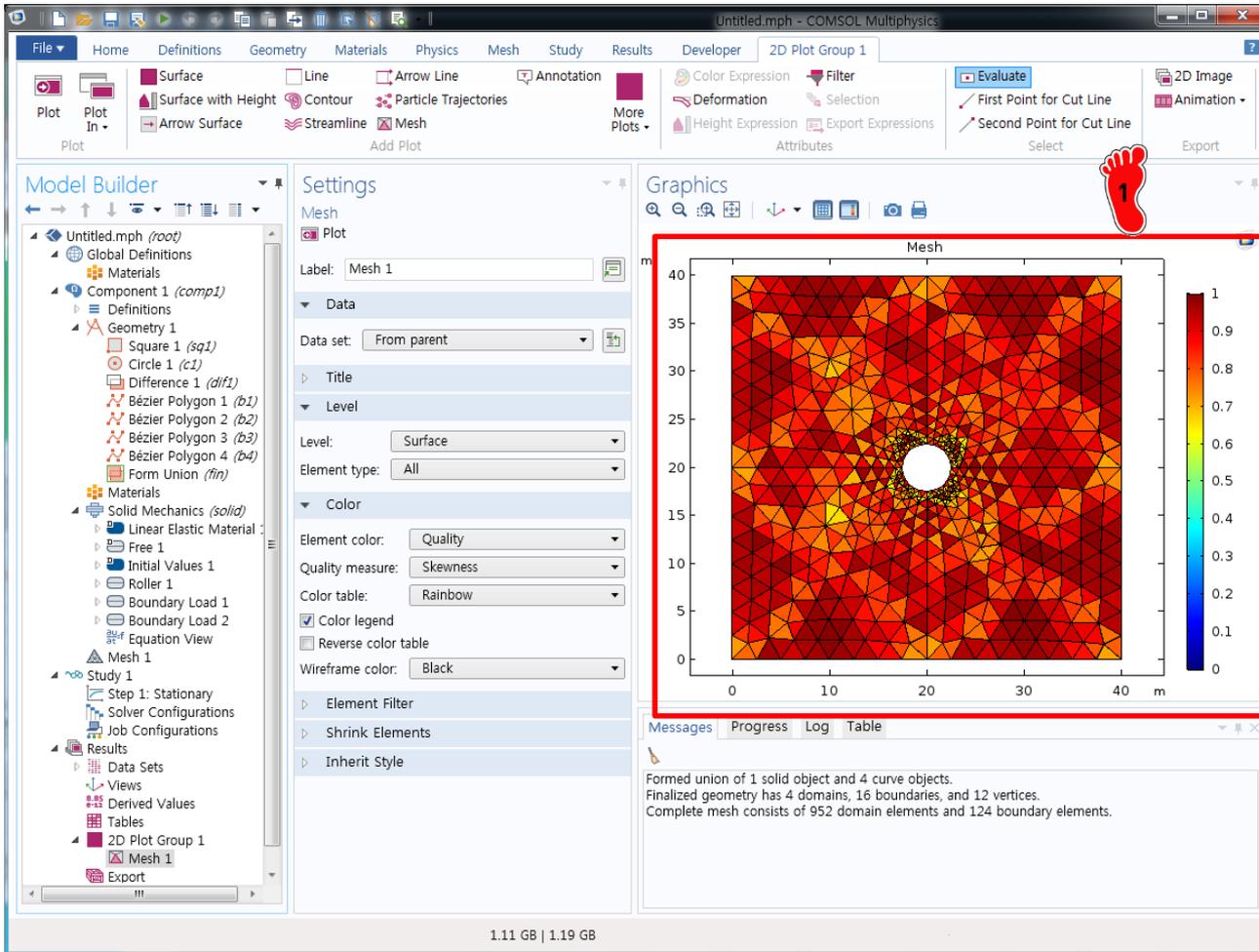
Good



Bad



MESH

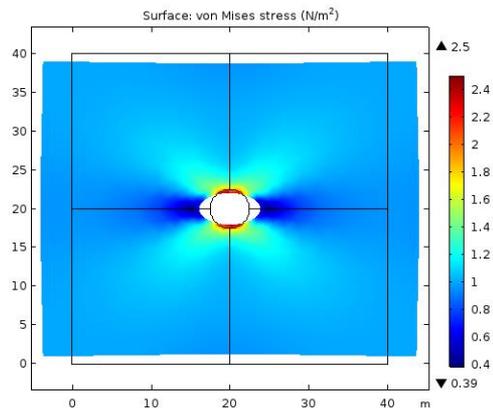


Mesh 우클릭 → Plot

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

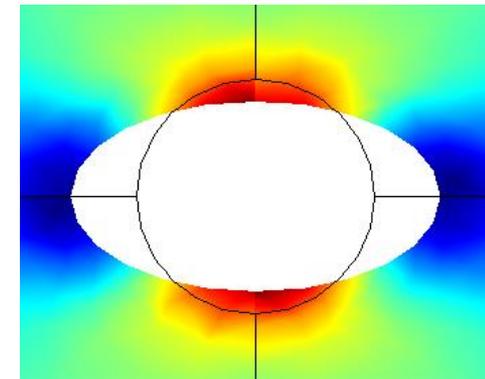
최대 수직 응력은 원에 접하는 부분에서 약 2.59 Pa

Expression:
solid.sx

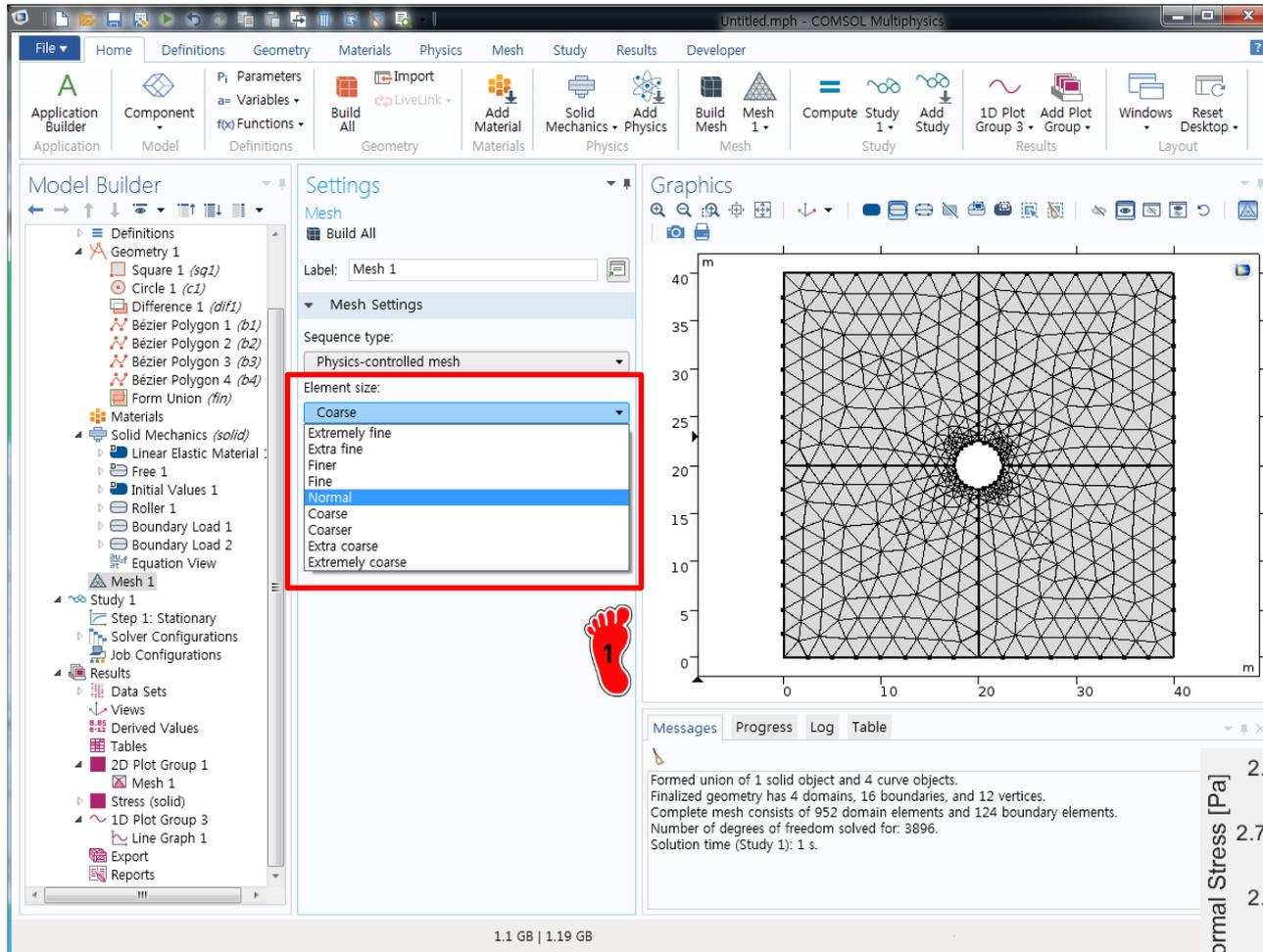
1 Results → Stress(solid) 클릭
Color Legend에서 Show maximum and minimum values 체크

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

최대 수직 응력은 원에 접하는 부분에서 약 2.59 Pa



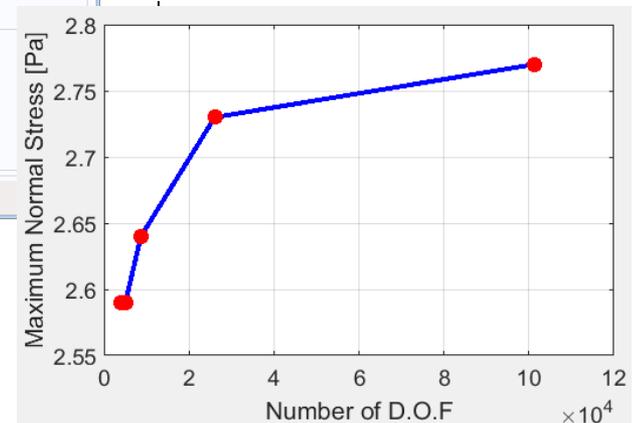
MESH REFINEMENT



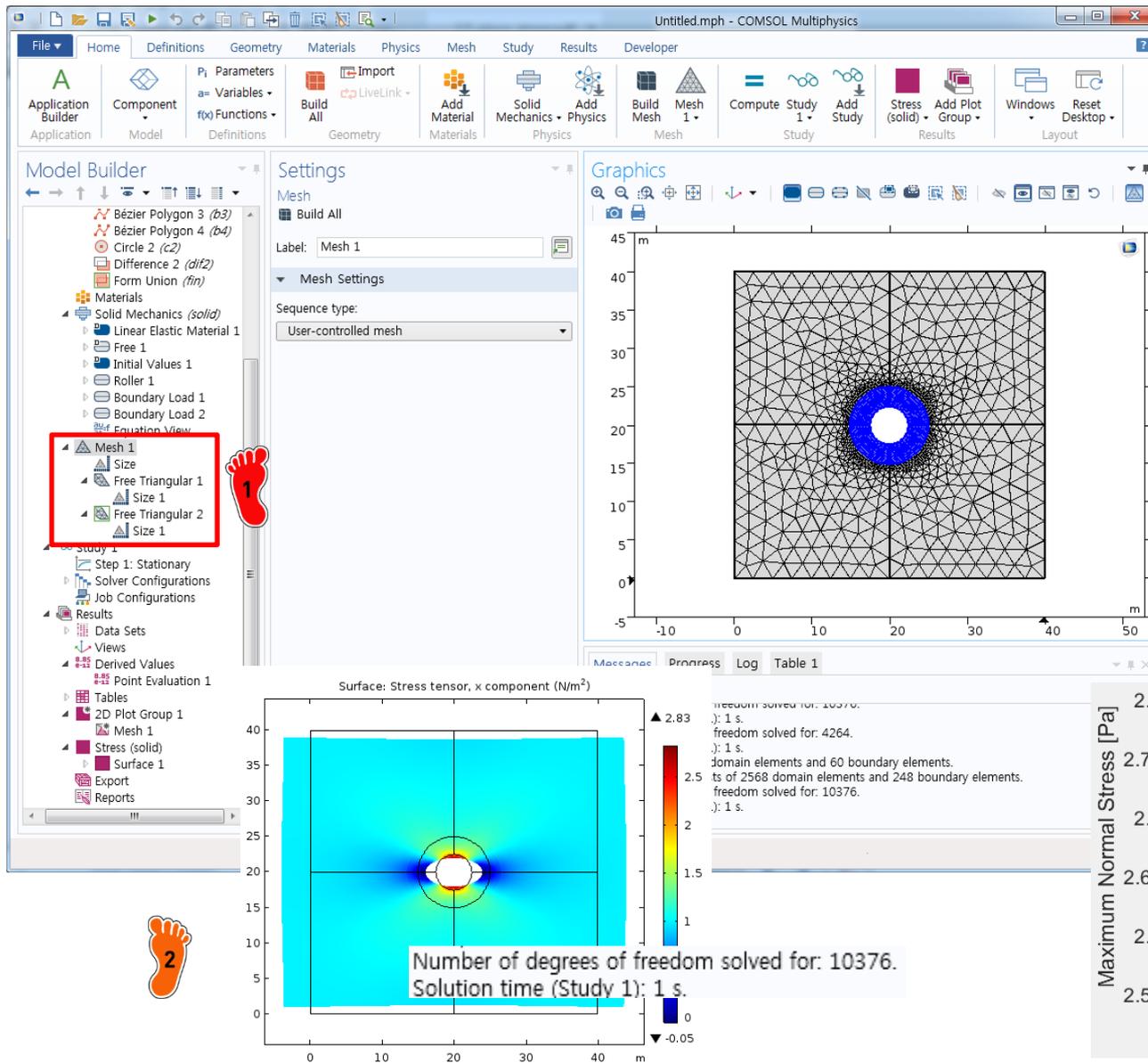
1 Mesh에서 Element Size
클릭 → Fine으로 변경 후
Build All → Compute

2 위와 같은 방식으로 요소 수
를 증가시켜 응력 결과값을
확인 출력한 그래프

Analytical solution 인
3*sigma 대비 작은
값으로 수렴해 감

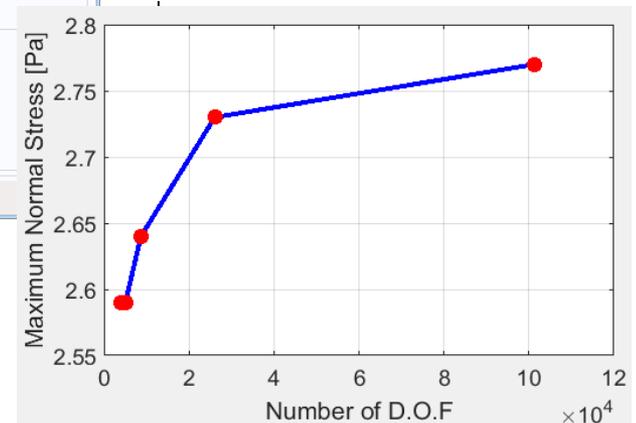


MESH REFINEMENT



1 Geometry를 분할하여 응력이 높게 나타나는 곳만 fine mesh 적용

2 자유도 수 약 10,000개 정도로 더 정확한 결과 계산



QUADRILATERAL ELEMENT



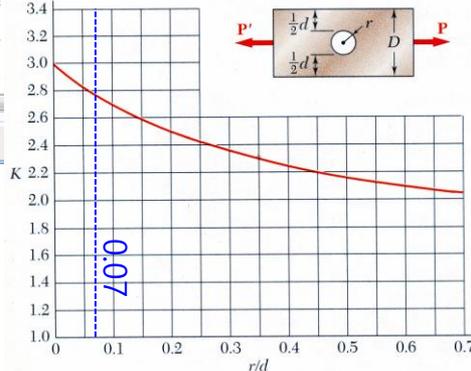
Mesh 우클릭
→ Free Quad 생성(사각형)
→ Size 생성 후 Build



Fine 으로 변경 후 해석

위와 같은 방식으로 요소 수를 증가시켜 응력 결과값을 출력한 그래프

삼각형보다 사각형 메시가 DOF수에 따라 변화 적음



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

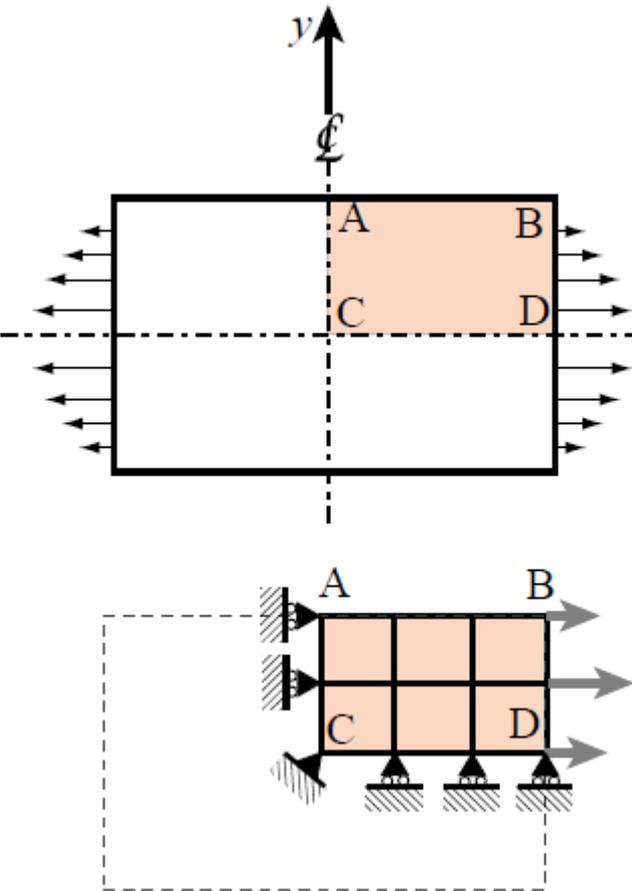
$$\sigma_{ave} = \frac{P}{A} = \frac{P}{W - 2R} = \frac{40}{40 - 5} = 1.14$$

$$\sigma_{max} = K \sigma_{ave} = 2.8 \times 1.14 = 3.2 \text{ Pa}$$

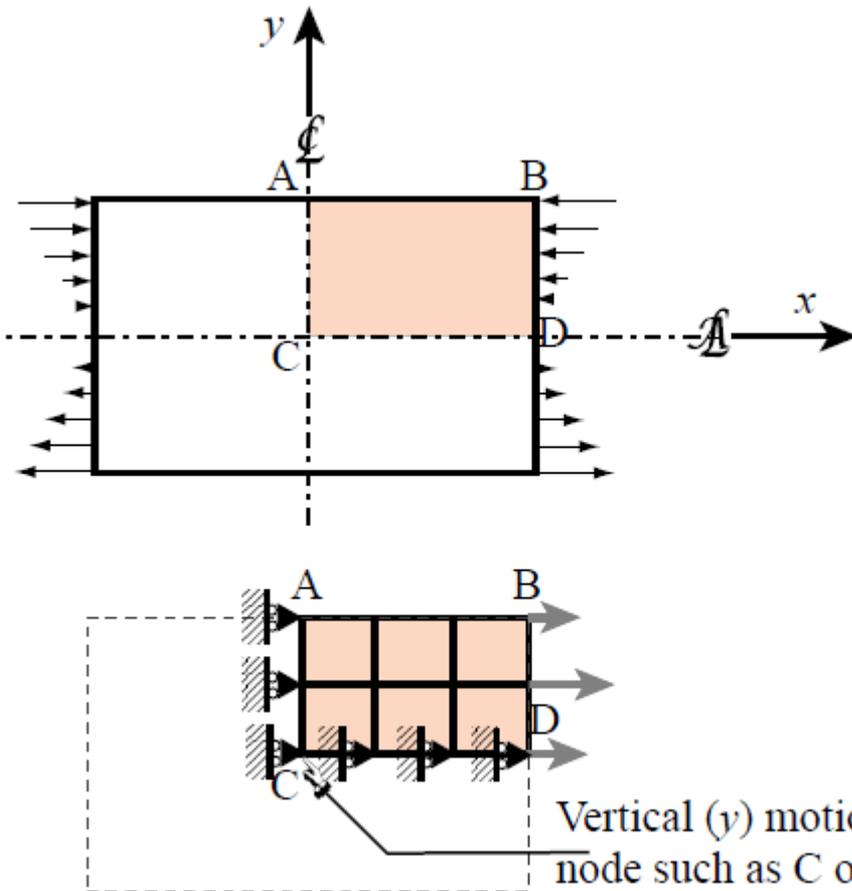
- **Stiffness matrix of Turner triangle**
- **2D plane stress model**
 - ✓ Kirsh's problem
- **Quarter model**

SYMMETRY CONDITION

Symmetry condition



AntiSymmetry condition

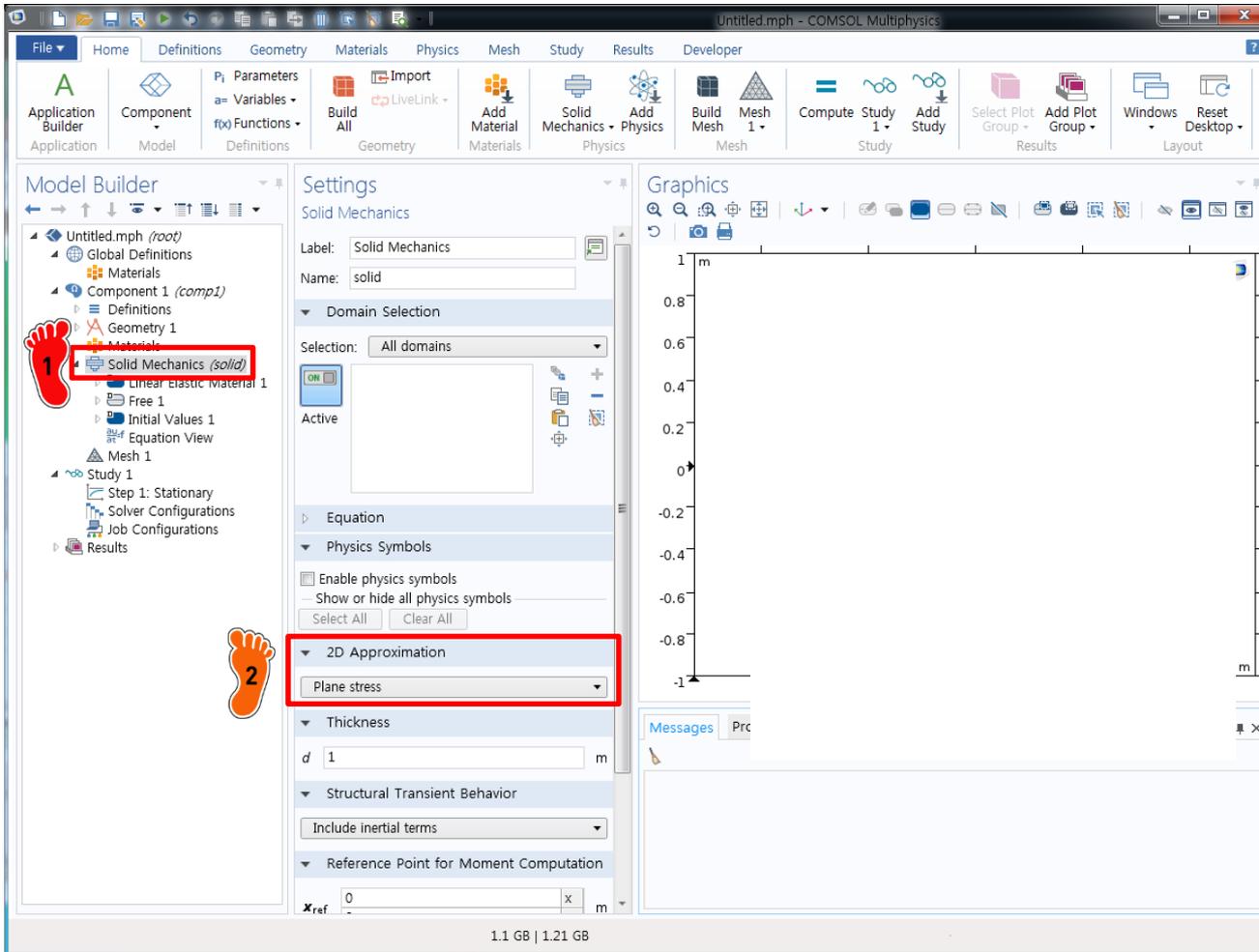


Vertical (y) motion of one node such as C or D may be constrained to suppress y -RBM

PLANE STRESS

1 Solid Mechanics 클릭

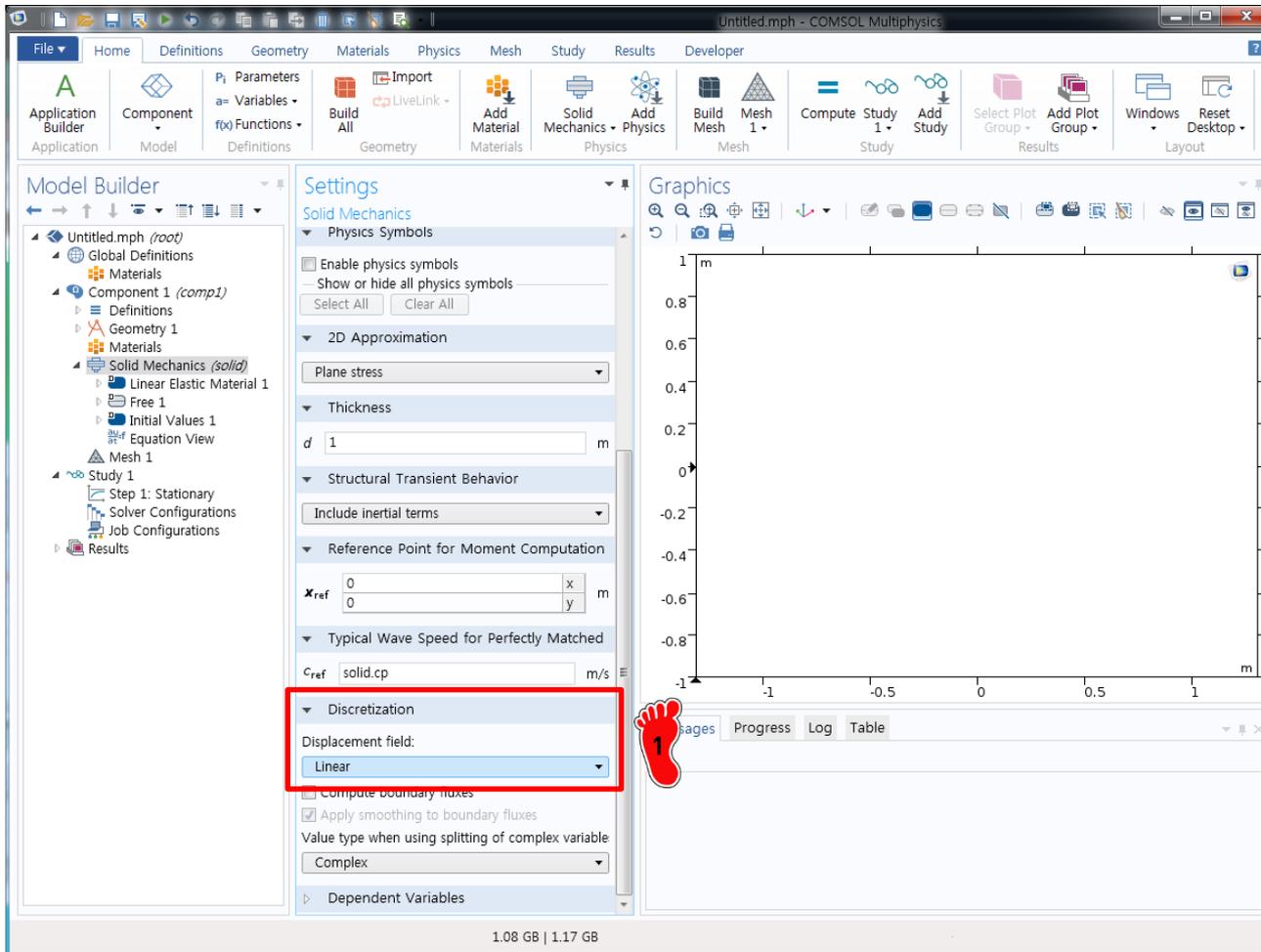
2 2D Approximation
Plane stress 로 변경



DISCRETIZATION



Discretization
Displacement field를
Linear로 변경



GEOMETRY CREATION



1/4 모델 기하형상 생성

길이 20의 정사각형에서
반경 2.5인 원을 뺀

The screenshot displays the COMSOL Multiphysics interface for creating a geometry. The Model Builder tree on the left shows the hierarchy: Untitled.mph (root) > Global Definitions > Materials > Component 1 (comp1) > Definitions > Geometry 1. Under Geometry 1, the following objects are listed: Square 1 (sq1), Circle 1 (c1), Difference 1 (dif1), and Form Union (fin). The Settings panel for 'Difference 1' shows 'Objects to add' as 'sq1' and 'Objects to subtract' as 'c1'. The Graphics window shows a 2D plot of a square with a quarter-circle cutout at the bottom-left corner. The Messages window at the bottom shows the following text:

```

number of degrees of freedom solved for: 4204.
Solution time (Study 1): 1 s.
Mesh consists of 238 domain elements and 60 boundary elements.
Complete mesh consists of 2568 domain elements and 248 boundary elements.
Number of degrees of freedom solved for: 10376.
Solution time (Study 1): 1 s.
Saved file: D:\#CAE실습\#2017\#hole_example.mph
Finalized geometry is empty.
  
```

1.3 GB | 1.41 GB

MATERIAL PROPERTY

Model Builder

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Square 1 (sq1)
 - Circle 1 (c1)
 - Difference 1 (dif1)
 - Form Union (fin)
 - Materials
 - Solid Mechanics (solid)
 - Linear Elastic Material 1**
 - Free 1
 - Initial Values 1
 - Equation View
 - Mesh 1
 - Study 1
 - Step 1: Stationary
 - Solver Configurations
 - Job Configurations
 - Results
 - Data Sets
 - Views
 - Derived Values
 - Tables
 - Stress (solid)
 - Surface 1
 - Export
 - Reports

Settings

Linear Elastic Material

- Equation
- Model Input
- Coordinate System Selection
- Coordinate system: Global coordinate system
- Linear Elastic Material
 - Nearly incompressible material
 - Solid model: Isotropic
 - Specify: Young's modulus and Poisson's ratio
 - Young's modulus:
 - E: User defined
 - 200e9 Pa
 - Poisson's ratio:
 - ν : User defined
 - 0.3
 - Density:
 - ρ : User defined
 - 0 kg/m³
- Geometric Nonlinearity
 - Force linear strains
 - Additive strain decomposition
- Energy Dissipation

Graphics

20 m

18

16

14

12

10

8

6

4

2

0

0 5 10 15 20 m

Messages Progress Log Table

Number of degrees of freedom solved for: 6250.
 Solution time (Study 1): 1 s.
 Number of degrees of freedom solved for: 2594.
 Solution time (Study 1): 1 s.
 Number of degrees of freedom solved for: 6290.
 Solution time (Study 1): 0 s.
 Number of degrees of freedom solved for: 2530.
 Solution time (Study 1): 0 s.

1.26 GB | 1.4 GB

1 Linear Elastic Material 클릭

2 E: 200e9
mu: 0.3
rho : 0

BOUNDARY CONDITION



사각형 왼쪽과 아래 변에
Roller 조건으로 입력

The screenshot displays the COMSOL Multiphysics software interface. The main window is titled "Untitled.mph - COMSOL Multiphysics". The interface is divided into several panels:

- Model Builder:** Shows the hierarchical structure of the model. Under "Geometry 1", "Square 1 (sq1)" is selected. Under "Solid Mechanics (solid)", "Roller 1" is highlighted with a red box and a red footprint icon with the number 1.
- Settings:** The "Roller" settings panel is active. The "Label" is "Roller 1". The "Boundary Selection" section is expanded, showing "Selection: Manual" and a list of boundaries with checkboxes. The checkboxes for boundaries 1 and 3 are checked, and the "Active" checkbox is also checked. This section is highlighted with a red box.
- Graphics:** A 2D plot of a square geometry with a rounded bottom-left corner. The x and y axes range from 0 to 20 meters. The left and bottom edges of the square are highlighted in blue, indicating they are selected for the boundary condition.
- Messages:** A log window at the bottom showing solver progress and statistics, such as "Number of degrees of freedom solved for: 6290." and "Solution time (Study 1): 1 s."

At the bottom of the interface, the memory usage is shown as "1.28 GB | 1.41 GB".

LOADING CONDITION

1

2

1.29 GB | 1.4 GB

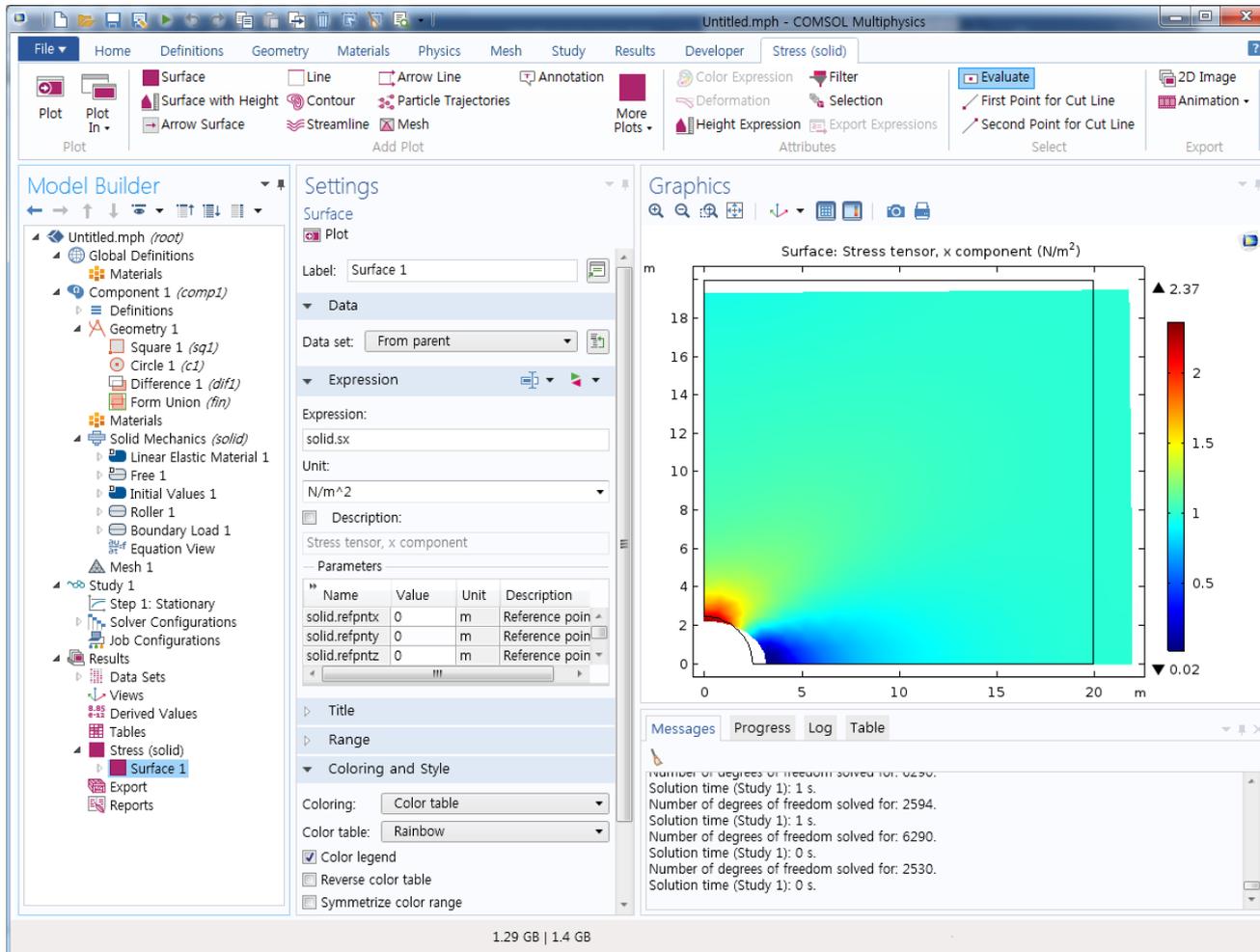
1 Boundary Load 생성

2 오른쪽 변에 x 방향
1 입력 (단위는 N/m²임)

RESULT

Mesh 생성 후 해석
Full model결과와 비교

응력, 자유도 수 확인

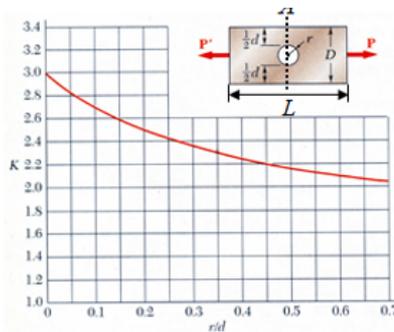
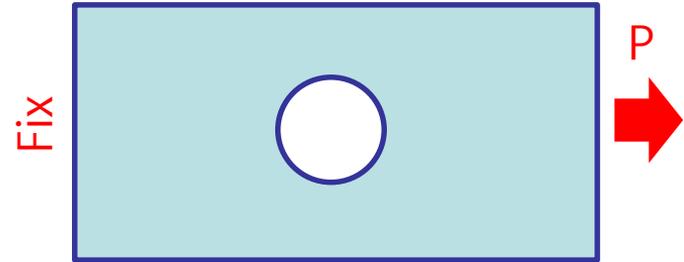
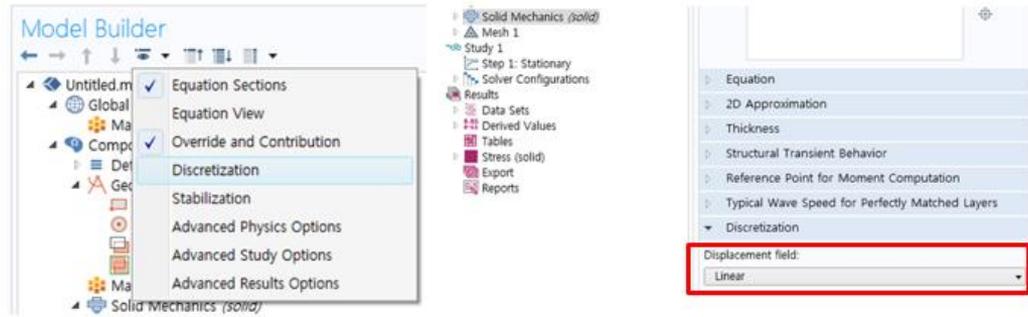


ASSIGNMENT

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.)

2016년 기말고사 2번 문제(40점)

※ Full Model 해석조건



$$E = 200 \text{ GPa}, \nu = 0.33$$

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

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

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 $r = 0.5 \text{ m}$. 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 $r = 0.5 \text{ m}$. Compare the quarter model with full model. (10 pts)