

PDE & Direct Stiffness Method by COMSOL

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



CONTENTS

- **PDE Examples**
- **Direct Stiffness Method: Truss**
- **Assignment**
- **Livelihood with MATLAB**

- **PDE Examples**
 - ✓ Laplace equation
 - ✓ Heat conduction equation

LAPLACE EQUATION

– PDE $\rightarrow$ algebraic difference equation

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} = 0 \leftarrow \begin{cases} \frac{\partial^2 T}{\partial x^2} = \frac{T_{i+1,j} - 2T_{i,j} + T_{i-1,j}}{\Delta x^2} \\ \frac{\partial^2 T}{\partial y^2} = \frac{T_{i,j+1} - 2T_{i,j} + T_{i,j-1}}{\Delta y^2} \end{cases}$$

$$\rightarrow \frac{T_{i+1,j} - 2T_{i,j} + T_{i-1,j}}{\Delta x^2} + \frac{T_{i,j+1} - 2T_{i,j} + T_{i,j-1}}{\Delta y^2} = 0$$

$$\xrightarrow{\Delta x = \Delta y} T_{i+1,j} + T_{i-1,j} + T_{i,j+1} + T_{i,j-1} - 4T_{i,j} = 0$$

apply boundary conditions (fixed/Dirichlet)

@(1,1): $T_{21} + \underbrace{T_{01}}_{75^\circ\text{C}} + T_{12} + \underbrace{T_{10}}_{0^\circ\text{C}} - 4T_{11} = 0 \rightarrow 4T_{11} - T_{21} - T_{12} = 75$

@(2,1): $T_{31} + T_{11} + T_{22} + \underbrace{T_{20}}_{0^\circ\text{C}} - 4T_{21} = 0 \rightarrow -T_{11} + 4T_{21} - T_{31} - T_{22} = 0$

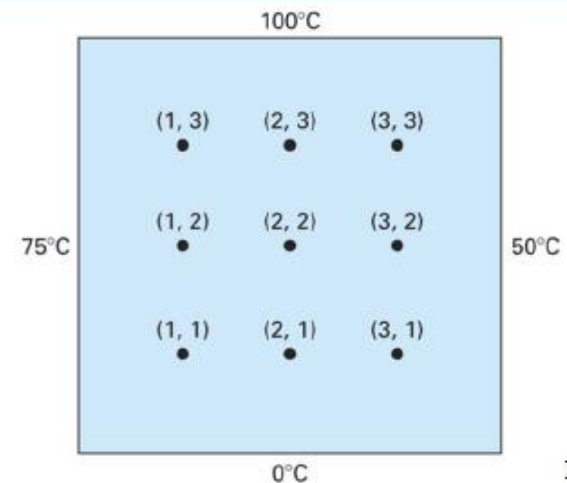
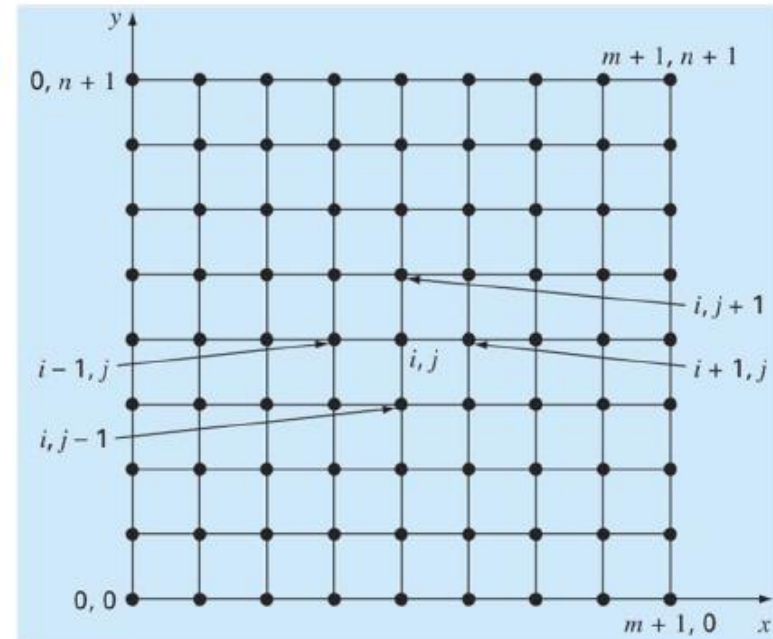
@(3,1): $\underbrace{T_{41}}_{50^\circ\text{C}} + T_{21} + T_{32} + \underbrace{T_{30}}_{0^\circ\text{C}} - 4T_{31} = 0 \rightarrow -T_{21} + 4T_{31} - T_{32} = 50$

⋮

9 equations

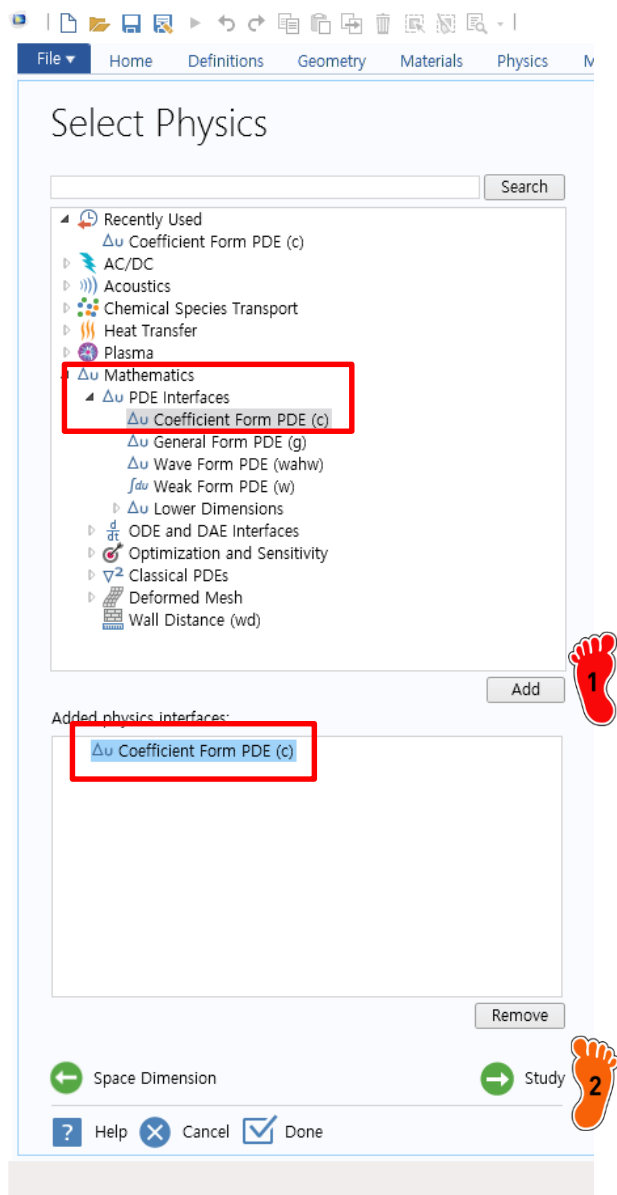
$$[K_{ij}] \{T_i\} = \{f_i\}$$

$[K_{ij}]$: coefficient matrix, $\{T_i\}$: solution vector, $\{f_i\}$: force vector



DE - 11

SETTING



1 Model Wizard → 2D
→ Mathematics - PDE
Interfaces – Coefficient
form PDE 선택
→ Add 클릭

2 Study 클릭

3 Stationary 선택 후
Done 클릭

GEOMETRY CREATION

1 Geometry 1 메뉴를 마우스 우클릭 → Rectangle 선택

2 Width: 4, Height: 4 입력 후 Build Selected 클릭

1.33 GB | 1.25 GB



Geometry 1 메뉴를 마우스
우클릭 → Rectangle 선택



Width: 4, Height: 4 입력
후 Build Selected 클릭

GEOMETRY CREATION

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Component 1 (comp1) Add Component

Definitions Geometry Materials

Physics Add Physics

Mesh Build Mesh Mesh 1 Compute Study 1 Add Study

Results Select Plot Group Add Plot Group Windows Reset Desktop

Layout

Model Builder

Untitled.mph (root)

Global Definitions

Materials

Component 1 (comp1)

Definitions

Geometry 1

Rectangle 1 (r1)

Point 1 (pt1)

Point 2 (pt2)

Point 3 (pt3)

Point 4 (pt4)

Point 5 (pt5)

Point 6 (pt6)

Point 7 (pt7)

Point 8 (pt8)

Point 9 (pt9)

Form Union (fin)

Materials

Coefficient Form PDE (c)

Coefficient Form PDE 1

Zero Flux 1

Initial Values 1

Mesh 1

Study 1

Step 1: Stationary

Results

Settings

Point

Build Selected

Label: Point 1

Point

x: 1 m

y: 1 m

Selections of Resulting Entities

Contribute to: None

Resulting objects selection

Show in physics: Point selection

Graphics Preview

4.5

4

3.5

3

2.5

2

1.5

1

0.5

0

-0.5

0 0.5 1 1.5 2 2.5 3 3.5 4

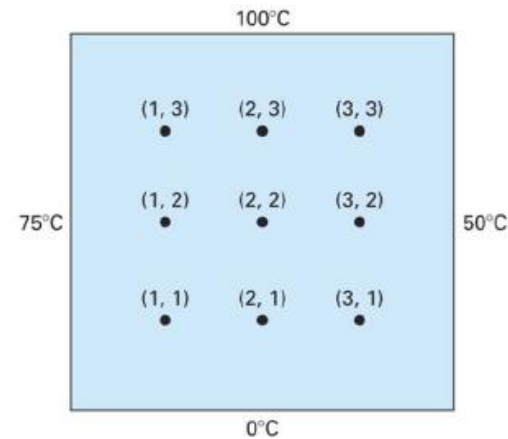
Messages Progress Log Table

Formed union of 1 solid object and 9 point objects.
Finalized geometry has 1 domain, 4 boundaries, and 13 vertices.
Number of degrees of freedom solved for: 1301 (plus 124 internal DOFs).
Solution time (Study 1): 1 s.
Saved file: 10_PDE_ex1.mph
Finalized geometry has 1 domain, 4 boundaries, and 4 vertices.

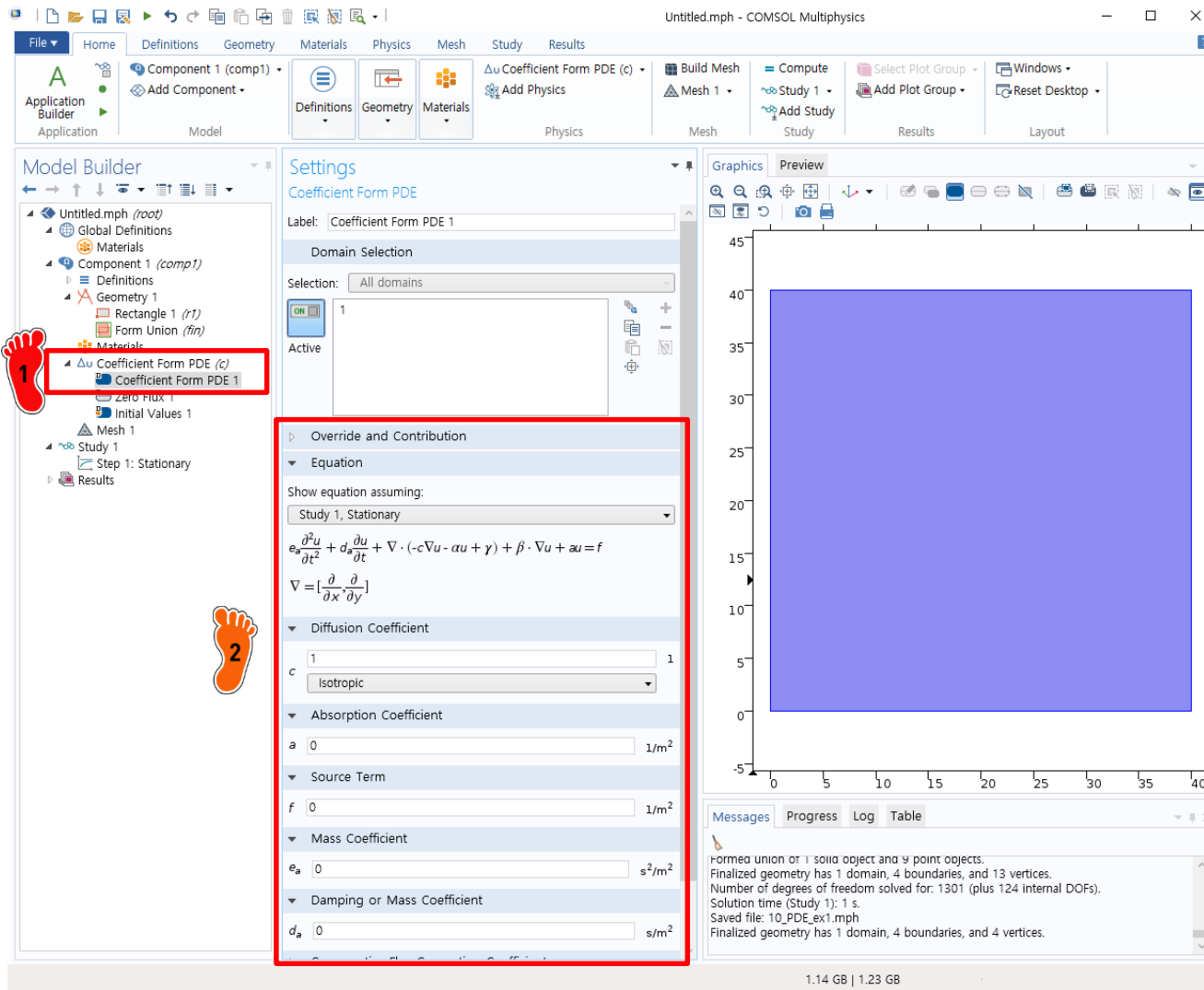
1.13 GB | 1.22 GB

1 Geometry 1 메뉴를 마우스 우클릭 → Point 선택

2 9개 점 좌표값 입력 (해석 후 결과 확인 점 표시)



COEFFICIENT INPUT



1 Coefficient Form PDE 1 선택

2 계수 값 입력: $c = 1$ (나머지 0)

$$\nabla = \left[\frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right]$$

$$\nabla^2 u = 0$$

$$\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} = 0$$

$$e \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot (-c \nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + a u = f \leftrightarrow \nabla^2 u = 0$$

BOUNDARY CONDITION

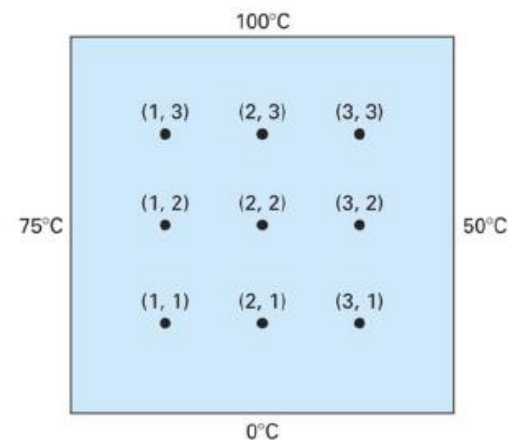
The screenshot shows the COMSOL Multiphysics interface with the following components:

- Model Builder:** Shows the hierarchy: Untitled.mph (root) > Global Definitions > Component 1 (comp1) > Definitions > Geometry 1 > Rectangle 1 (r1) > Point 1 (pt1) through Point 9 (pt9) > Materials > Coefficient Form PDE (c) > Coefficient Form PDE 1 > Zero Flux 1 > Initial Values 1 > Dirichlet Boundary Condition 1 through Dirichlet Boundary Condition 4.
- Settings:** The 'Dirichlet Boundary Condition' settings are shown. The 'Boundary Selection' is set to 'Manual'. The 'ON' button is highlighted with a red box and a red footprint icon (1). The 'Active' checkbox is checked. The 'Prescribed value of u' is set to 75, highlighted with a red box and a red footprint icon (2).
- Graphics:** A 2D plot of the domain is shown. The left boundary is highlighted with a red oval and a red footprint icon (1). The plot shows a square domain with a coordinate system from 0 to 4 on both axes.
- Messages:** The bottom panel shows the solution progress and log, indicating that the solution was solved for 1.301 seconds (plus 1.24 internal DOFs).

1 Coefficient Form PDE 메뉴
마우스 우클릭 → Dirichlet
Boundary Condition 클릭

Dirichlet Boundary Condition

2 4개 생성 및 경계 선택 후
경계값 입력



MESH CREATION

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Component 1 (comp1) Add Component

Definitions Geometry Materials

Physics

Mesh Build Mesh Mesh 1 Compute Study 1 Add Study Add Plot Group Select Plot Group Add Plot Group Windows Reset Desktop

Settings

Mesh

Label: Mesh 1

Mesh Settings

Sequence type: Physics-controlled mesh

Element size: Normal

Graphics Preview

Messages Progress Log Table

Number of degrees of freedom solved for: 1301 (plus 124 internal DOFs).
 Solution time (Study 1): 1 s.
 Saved file: 10_PDE_ex1.mph
 Finalized geometry has 1 domain, 4 boundaries, and 4 vertices.
 Formed union of 1 solid object and 9 point objects.
 Finalized geometry has 1 domain, 4 boundaries, and 13 vertices.
 Complete mesh consists of 620 domain elements and 60 boundary elements.

1.12 GB | 1.22 GB

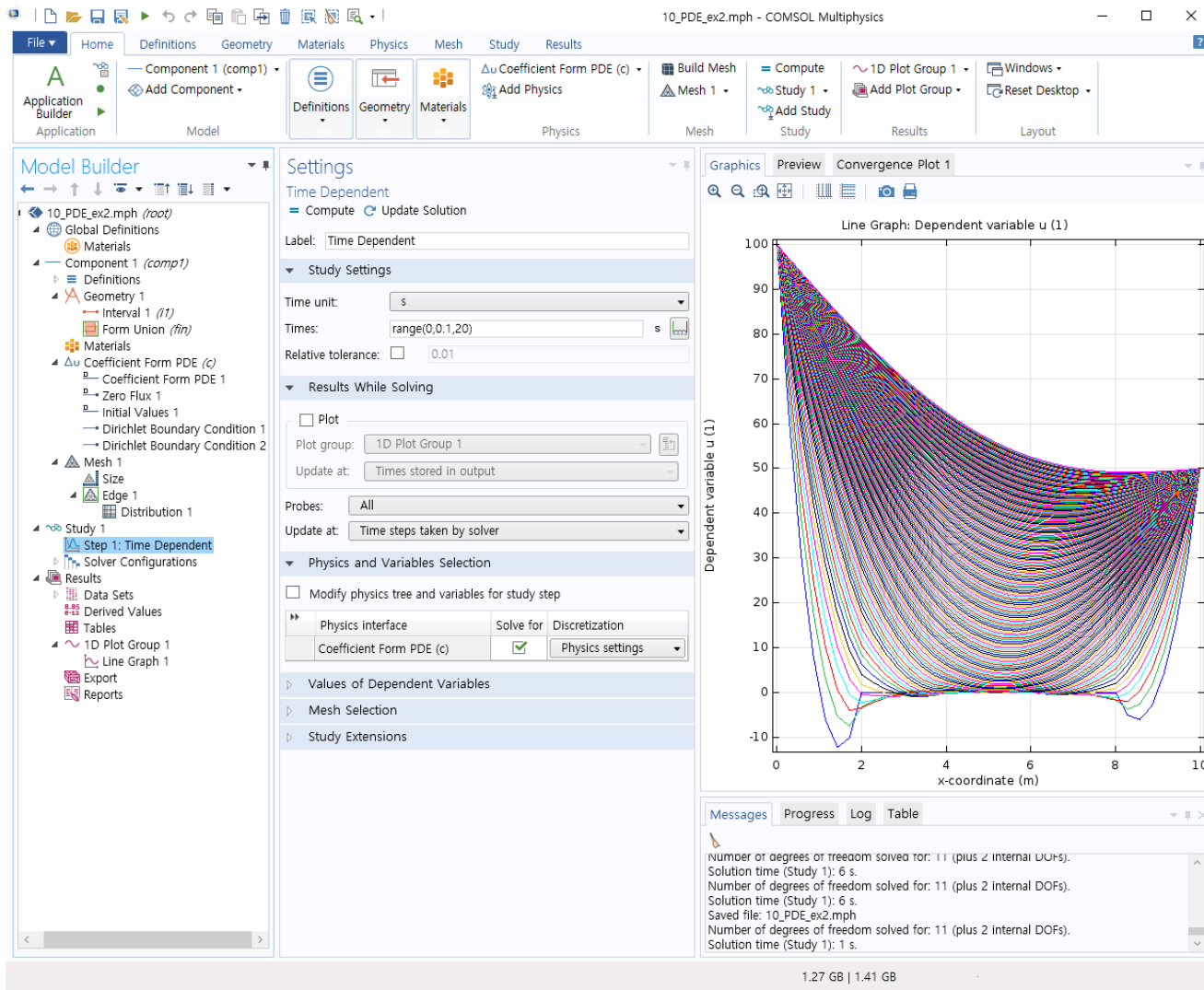


Mesh 1 메뉴 클릭



Build All 클릭

COMPUTE



Study 1 메뉴 선택



Compute 클릭

POST-PROCESSING

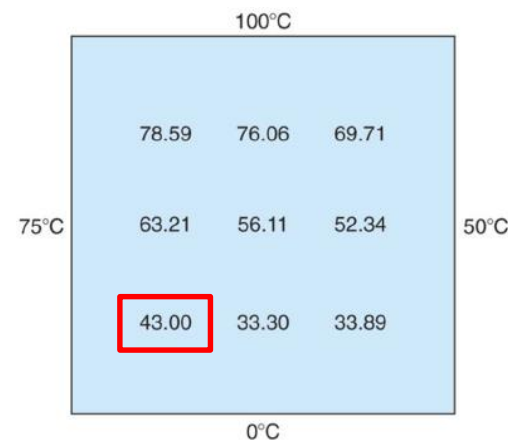
The screenshot shows the COMSOL Multiphysics interface for a 2D plot of a coefficient form PDE. The interface is divided into several panes:

- Model Builder:** Shows the hierarchy of the model. The 'Results' section is expanded, and 'Point Evaluation 1' is selected under 'Derived Values'.
- Settings:** The 'Point Evaluation' settings are displayed. The 'Evaluate' button is highlighted with a red box and a red footprint icon (1). The 'Data' section shows 'Study 1/Solution 1 (sol1)' as the data set. The 'Selection' section shows 'Manual' selection, and the 'Active' list contains 'Point 1 (pt1)', which is highlighted with a red box and a red footprint icon (3).
- Graphics:** A 2D plot of the domain is shown. A red box and a red footprint icon (2) highlight the selected point (pt1) on the plot.
- Messages:** The bottom pane shows the message 'Dependent variable u (1), Point: 3' with the value '42.598', which is highlighted with a red box and a red footprint icon (3).

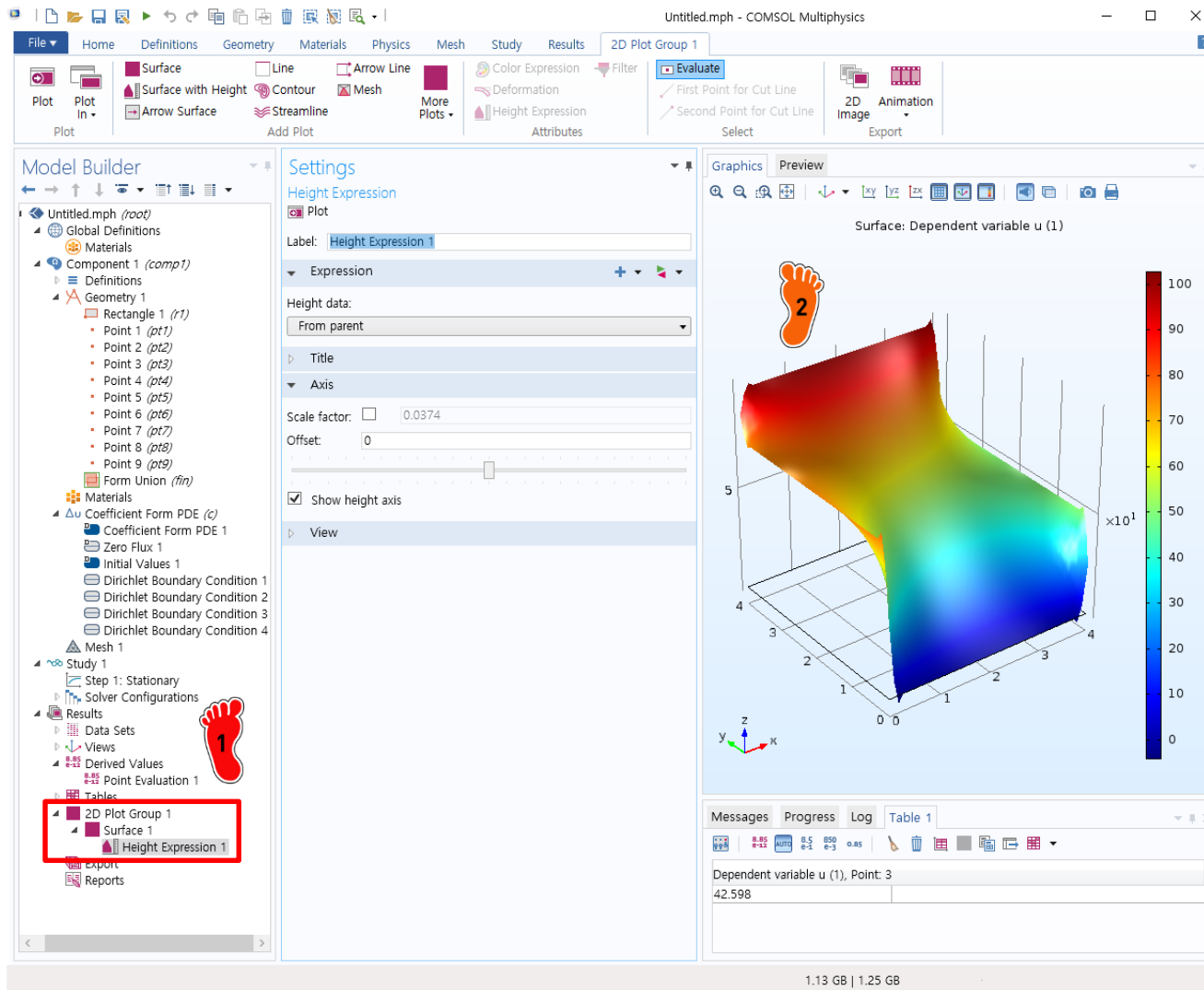
1 Results → Derived Values
우클릭 → Point Evaluation
선택

2 측정하고자 하는 Point 선택
후 Evaluate 클릭

3 결과값 확인

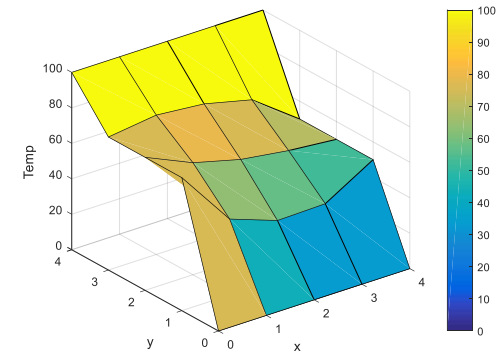


POST-PROCESSING



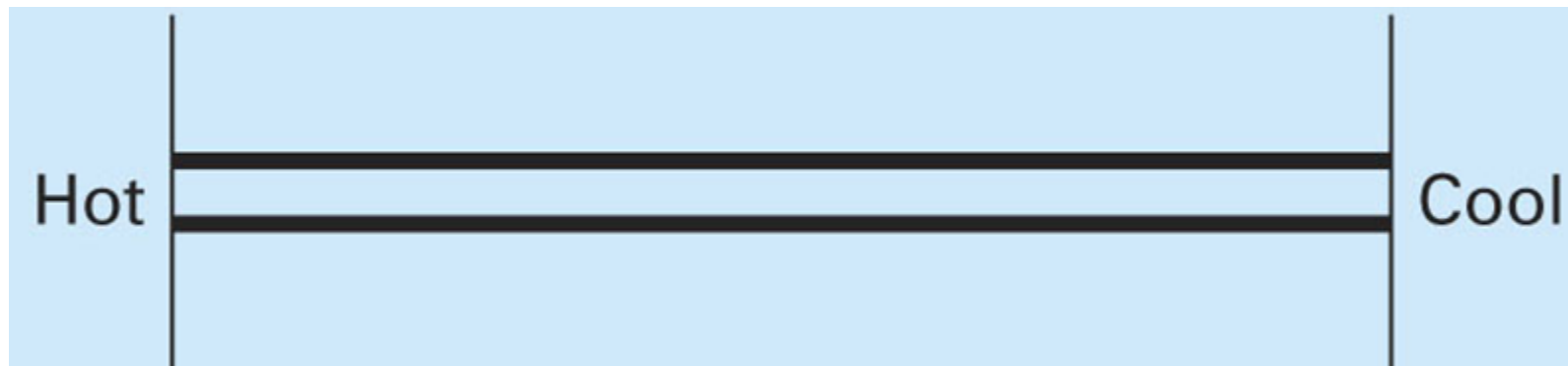
1 2D Plot Group 1 →
Surface 1 우클릭
→ Height Expression 선택

2 3차원 그래프 확인



- **PDE Examples**
 - ✓ Laplace equation
 - ✓ Heat conduction equation

HEAT CONDUCTION EQUATION



heat stored in the element over a unit time period $\Delta t = (In) - (Out)$

$$q(x)(\Delta y \Delta z) \Delta t - q(x + \Delta x)(\Delta y \Delta z) \Delta t = \rho C (\Delta x \Delta y \Delta z) \Delta T$$

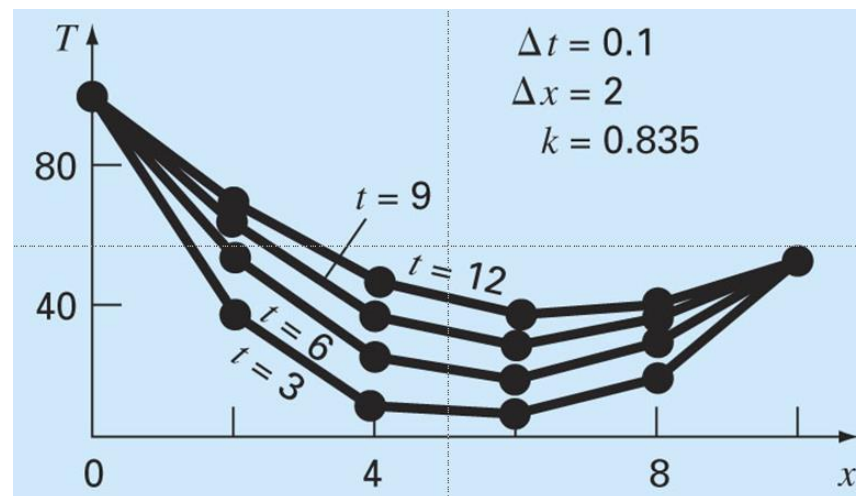
$$\frac{q(x) - q(x + \Delta x)}{\Delta x} = \rho C \frac{\Delta T}{\Delta t} \xrightarrow{\text{taking the limit}} -\frac{\partial q}{\partial x} = \rho C \frac{\partial T}{\partial t} \xrightarrow{q = -k\rho C \frac{\partial T}{\partial x}} k \frac{\partial^2 T}{\partial x^2} = \frac{\partial T}{\partial t}$$

$$\left\{ \begin{array}{l} k = 0.835 \text{ cal}/(\text{s} \cdot \text{cm} \cdot ^\circ\text{C}) \end{array} \right.$$

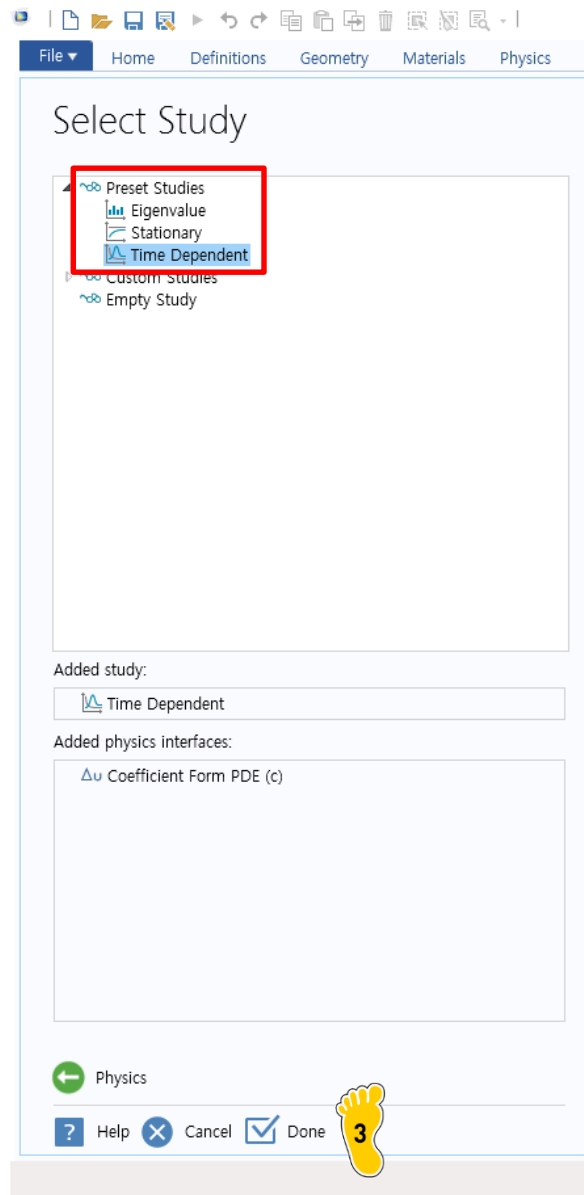
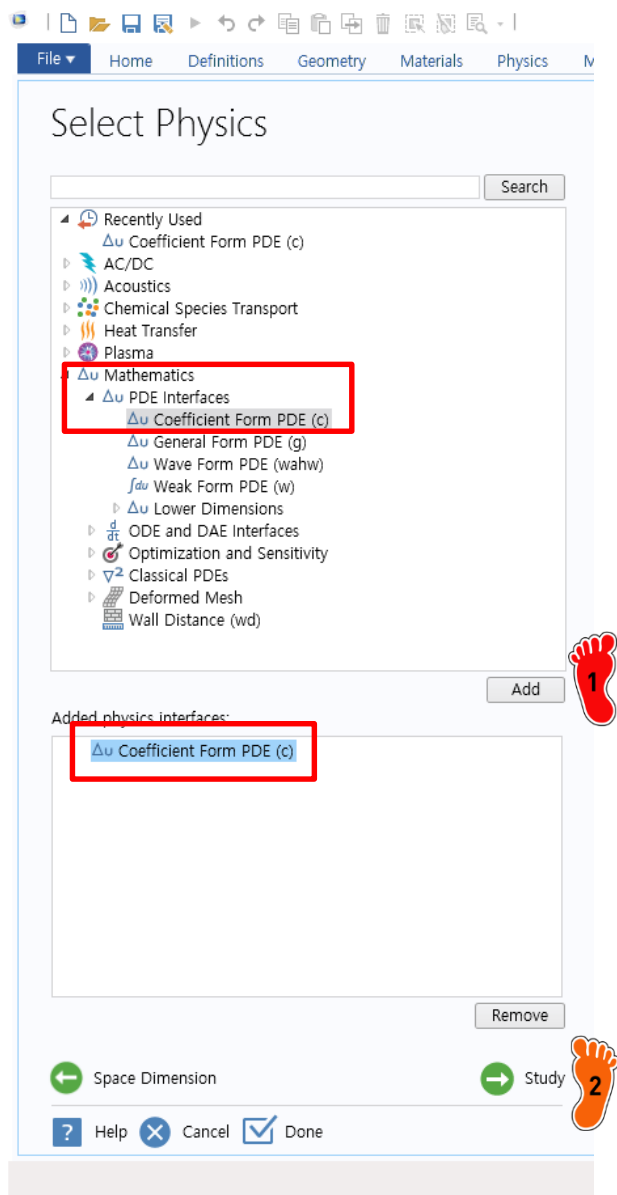
$$\left\{ \begin{array}{l} \Delta x = 2 \text{ cm} \end{array} \right.$$

$$\left\{ \begin{array}{l} \Delta t = 0.1 \text{ s} \end{array} \right.$$

$$\left\{ \begin{array}{l} \text{Boundary condition} \\ T(0, t) = 100^\circ\text{C} \\ T(10, t) = 50^\circ\text{C} \end{array} \right. \left\{ \begin{array}{l} \text{Initial condition} \\ T(i, t = 0) = 0^\circ\text{C} \end{array} \right.$$



SETTING



1 Model Wizard → 1D
→ Mathematics - PDE
Interfaces – Coefficient
form PDE 선택
→ Add 클릭

2 Study 클릭

3 Time Dependent 선택 후
Done 클릭

GEOMETRY CREATION

The screenshot displays the COMSOL Multiphysics interface for creating a geometry. The left sidebar shows the 'Model Builder' tree with 'Geometry 1' selected. The 'Settings' panel for 'Interval' is open, showing 'Number of intervals' set to 'One', 'Left endpoint' at '0', and 'Right endpoint' at '10'. The 'Build Selected' button is highlighted with a red box and a red footprint icon labeled '2'. The 'Interval' section is also highlighted with a red box and a red footprint icon labeled '2'. The 'Messages' window at the bottom shows the solution time and degrees of freedom for the mesh.

1. Geometry 1 우클릭 → Interval 선택

2. Left: 0, Right: 10 입력 후 Build Selected 클릭



Geometry 1 우클릭 → Interval 선택



Left: 0, Right: 10 입력 후 Build Selected 클릭

COEFFICIENT INPUT

1

2

1 Coefficient Form PDE 1 선택

2 계수 값 입력 (나머지 0)

$$c = 0.835$$

$$d_a = 1$$

$$c \nabla^2 u = d_a \frac{\partial u}{\partial t}$$

$$\updownarrow$$

$$k \frac{\partial^2 T}{\partial x^2} = \frac{\partial T}{\partial t}$$

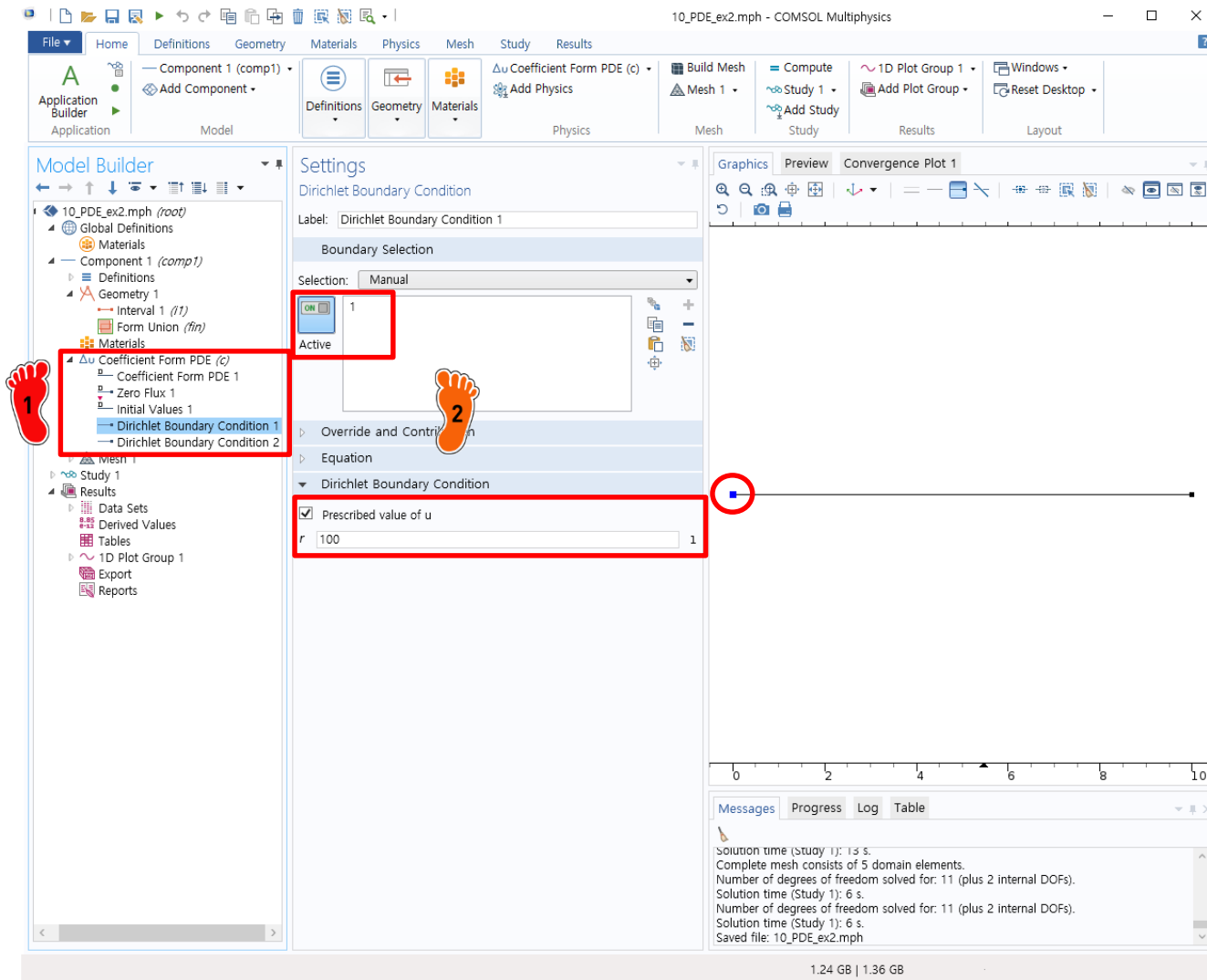
$$k = 0.835 \text{ cal}/(\text{s} \cdot \text{cm} \cdot ^\circ\text{C})$$

$$\Delta x = 2 \text{ cm}$$

$$\Delta t = 0.1 \text{ s}$$

$$e \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot (-c \nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + a u = f \leftrightarrow c \nabla^2 u = d_a \frac{\partial u}{\partial t}$$

BOUNDARY CONDITION



1 Coefficient Form PDE 우클릭
→ Dirichlet Boundary
Condition 클릭

2 2개 생성 및 경계 선택 후
경계값 입력
(Initial Values는 default 값
이 0이므로 별도 입력 X)

Boundary condition

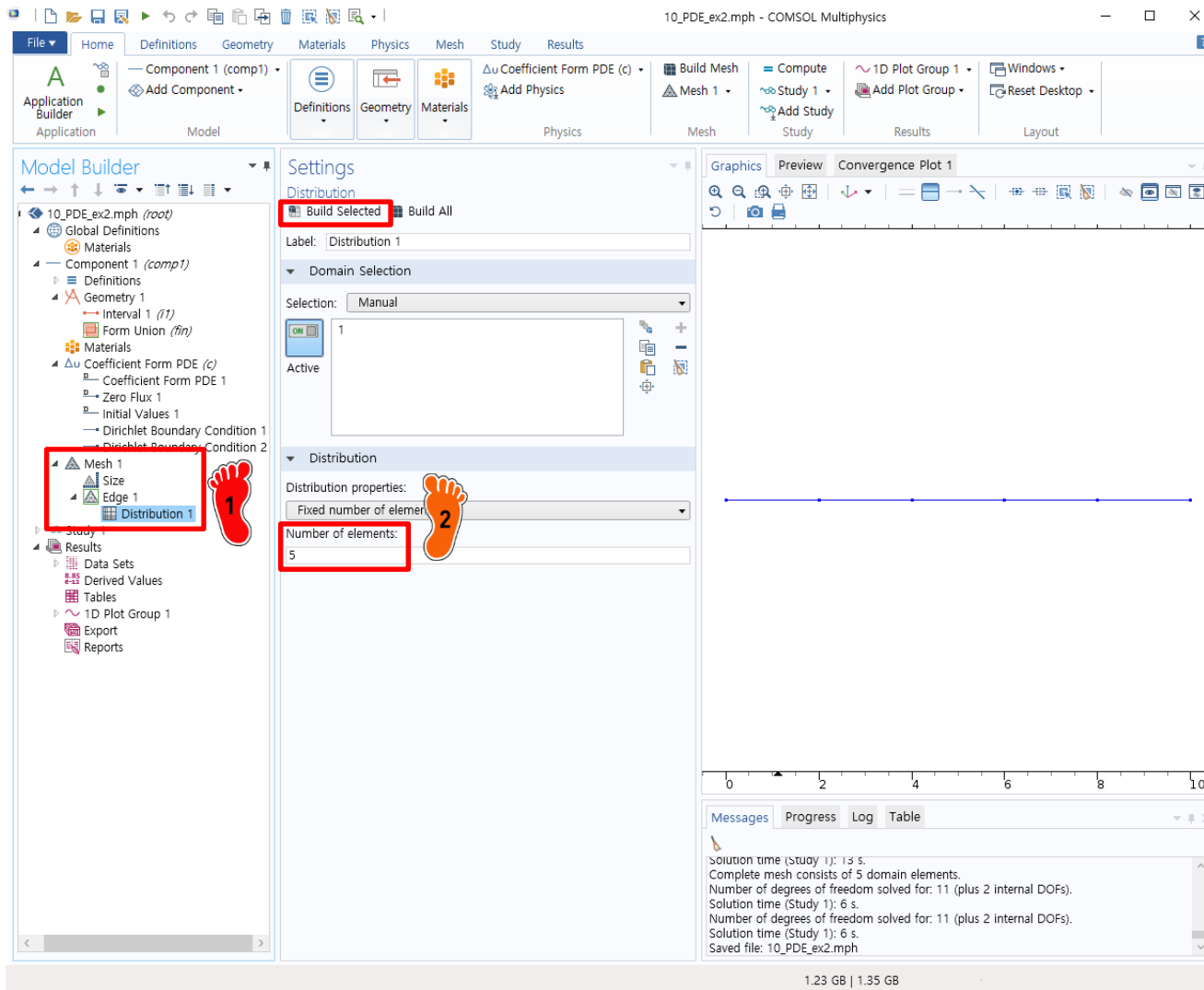
$$T(0, t) = 100^{\circ}\text{C}$$

$$T(10, t) = 50^{\circ}\text{C}$$

Initial condition

$$T(i, t = 0) = 0^{\circ}\text{C}$$

MESH CREATION



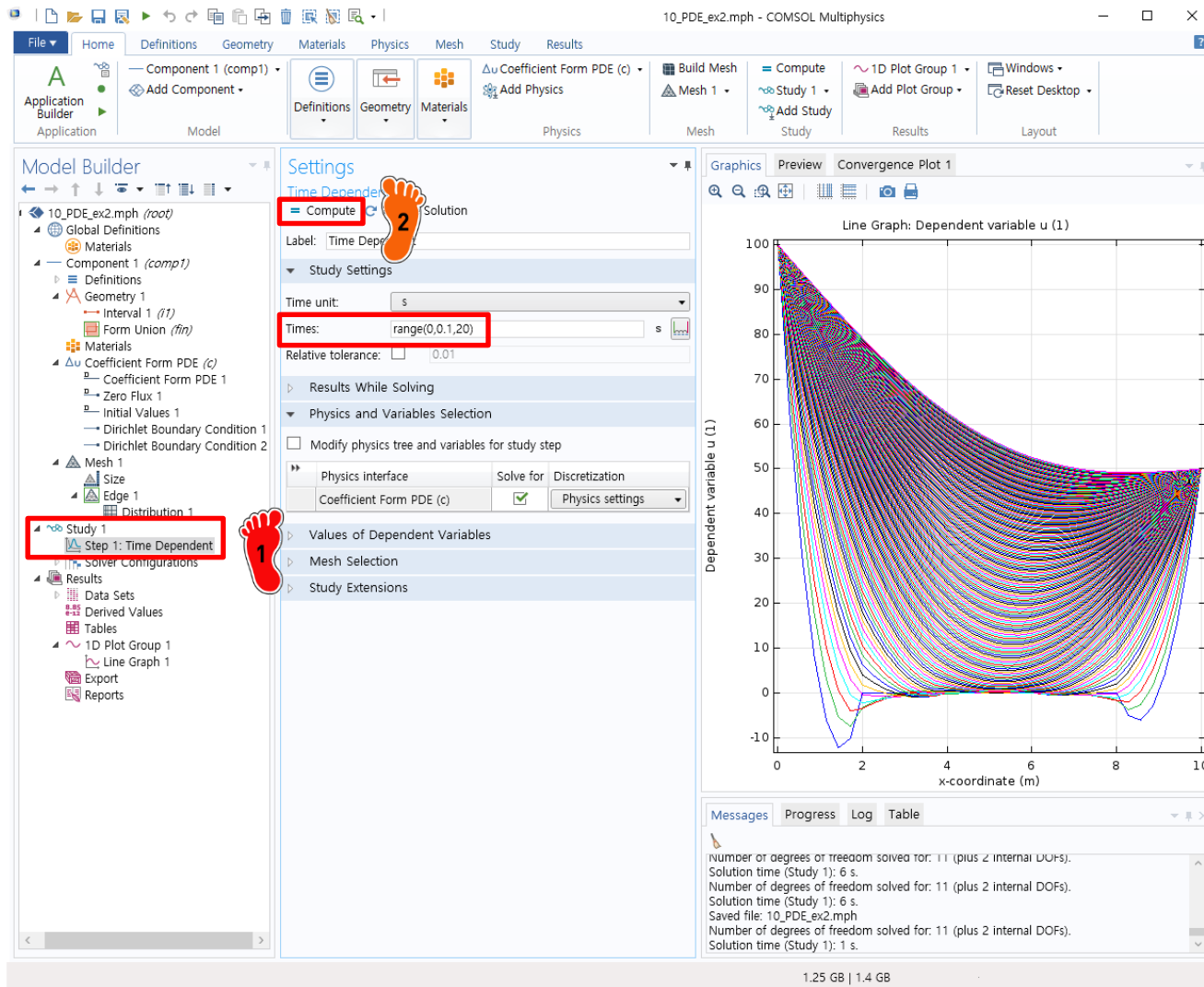
Mesh 1 메뉴 우클릭
→ Edge 선택
→ Edge 메뉴 우클릭
→ Distribution 선택



Number of elements에 5
입력 → Build Selected 클릭

$$\begin{cases} k = 0.835 \text{ cal}/(\text{s} \cdot \text{cm} \cdot ^\circ\text{C}) \\ \Delta x = 2 \text{ cm} \\ \Delta t = 0.1 \text{ s} \end{cases}$$

COMPUTE

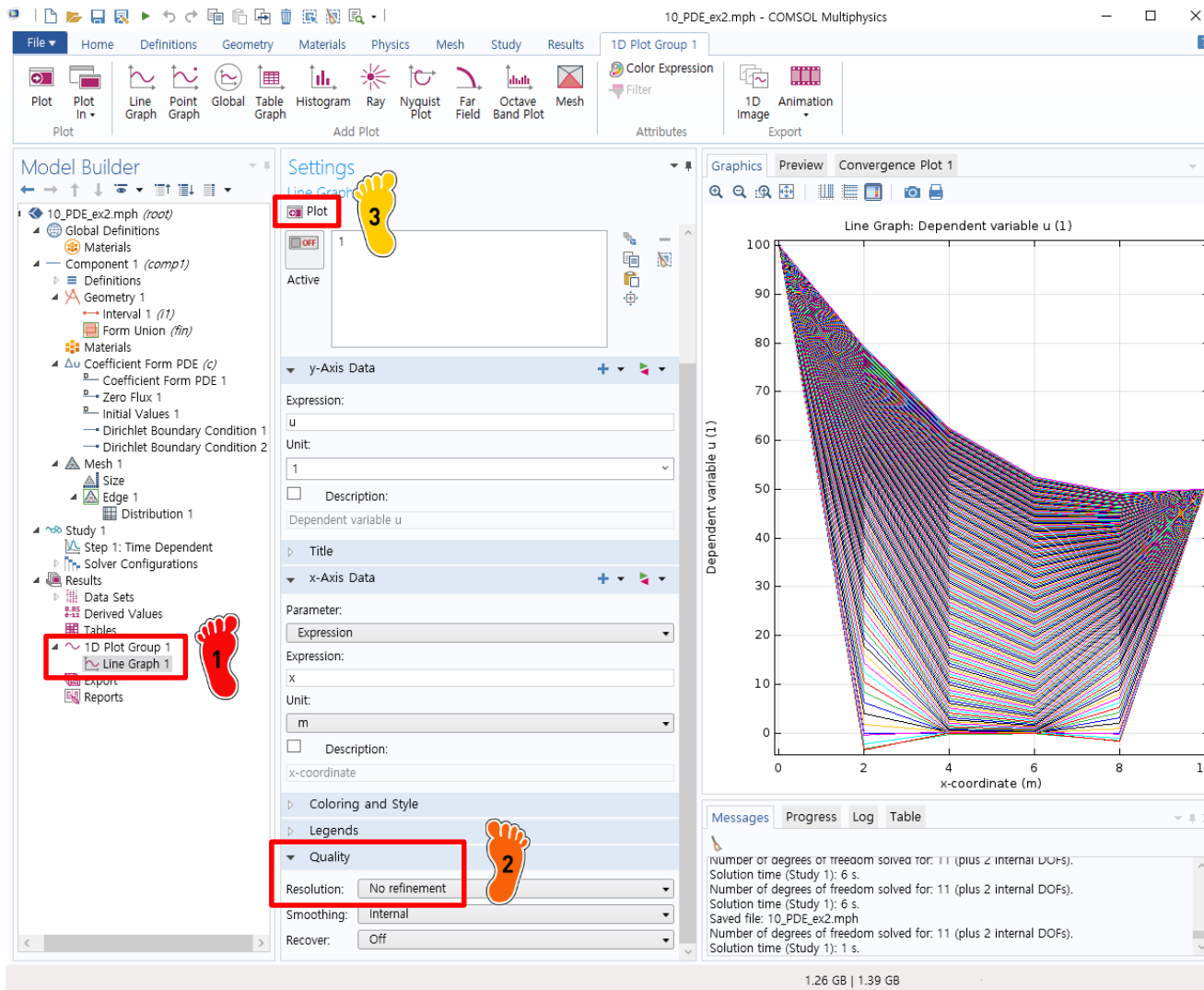


1 Study 1 → Step 1: Time Dependent 클릭

2 Times: range(0,0.1,20) 입력 후 Compute 클릭

$$\begin{cases} k = 0.835 \text{ cal}/(\text{s} \cdot \text{cm} \cdot ^\circ\text{C}) \\ \Delta x = 2 \text{ cm} \\ \Delta t = 0.1 \text{ s} \end{cases}$$

POST-PROCESSING



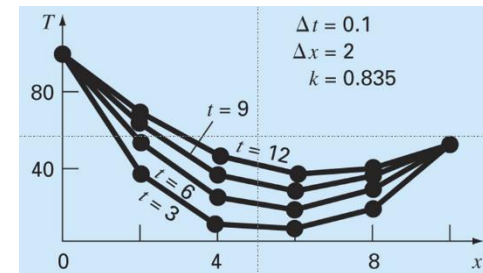
Line Graph 1 클릭



Quality → Resolution 에서
No refinement로 변경



Plot 클릭



POST-PROCESSING

10_PDE_ex2.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Component 1 (comp1) Add Component

Settings

Time Dependent

Compute

Study Settings

Label: Time Dependent

Time unit: s

Times: range(0,0.1,20)

Relative tolerance: 0.01

Results While Solving

☒ Plot

Plot group: 1D Plot Group 1

Update at: Times stored in output

Probes: All

Update at: Time steps taken by solver

Physics and Variables Selection

☐ Modify physics tree and variables for study step

Physics interface	Solve for	Discretization
Coefficient Form PDE (c)	☒	Physics settings

Values of Dependent Variables

Mesh Selection

Study Extensions

Graphics Preview Convergence Plot 1

Line Graph: Dependent variable u (1)

Dependent variable u (1)

x-coordinate (m)

Messages Progress Log Table

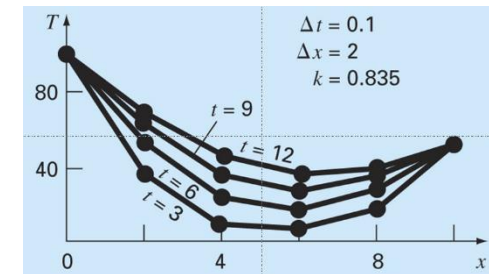
Number of degrees of freedom solved for: 11 (plus 2 internal DOFs).
 Solution time (Study 1): 6 s.
 Saved file: 10_PDE_ex2.mph
 Number of degrees of freedom solved for: 11 (plus 2 internal DOFs).
 Solution time (Study 1): 1 s.
 Number of degrees of freedom solved for: 11 (plus 2 internal DOFs).
 Solution time (Study 1): 6 s.

1.29 GB | 1.42 GB

1 Study 1 → Step 1: Time Dependent 클릭

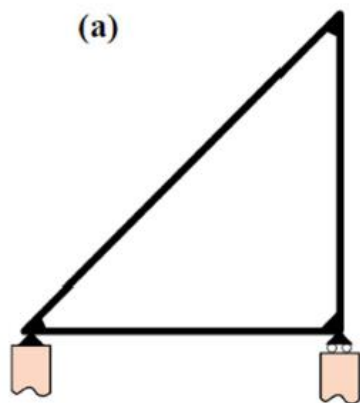
2 Results While Solving에서 Plot 선택

3 Compute 클릭
(시간에 따른 변화 확인)



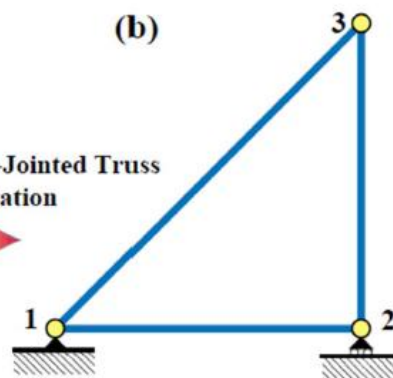
- **Direct Stiffness Method: Truss**
 - ✓ Geometry creation
 - ✓ **Material property**
 - ✓ **Cross section property**
 - ✓ **Boundary condition**
 - ✓ **Nodal force**
 - ✓ **Analysis**

EXAMPLE TRUSS STRUCTURE

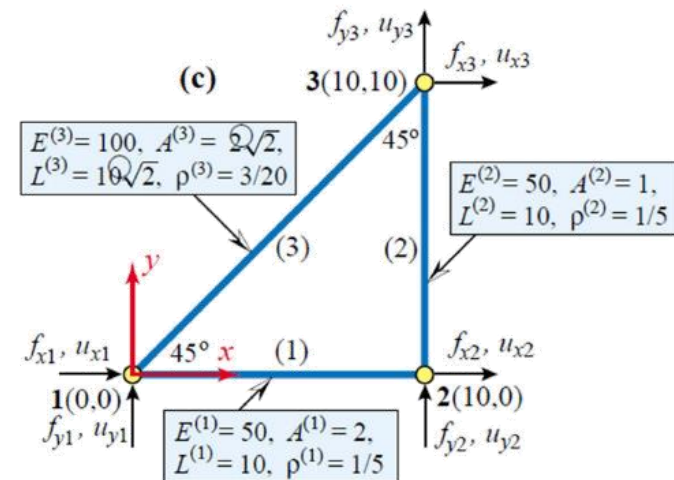


Physical structure

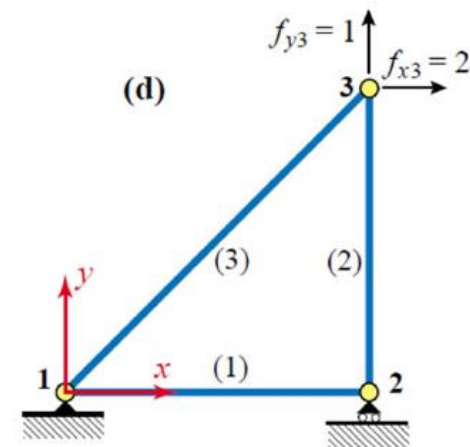
Idealization as Pin-Jointed Truss
and FEM Discretization



Idealization as a pin-jointed
bar assemblage



Geometric, material and fabrication properties

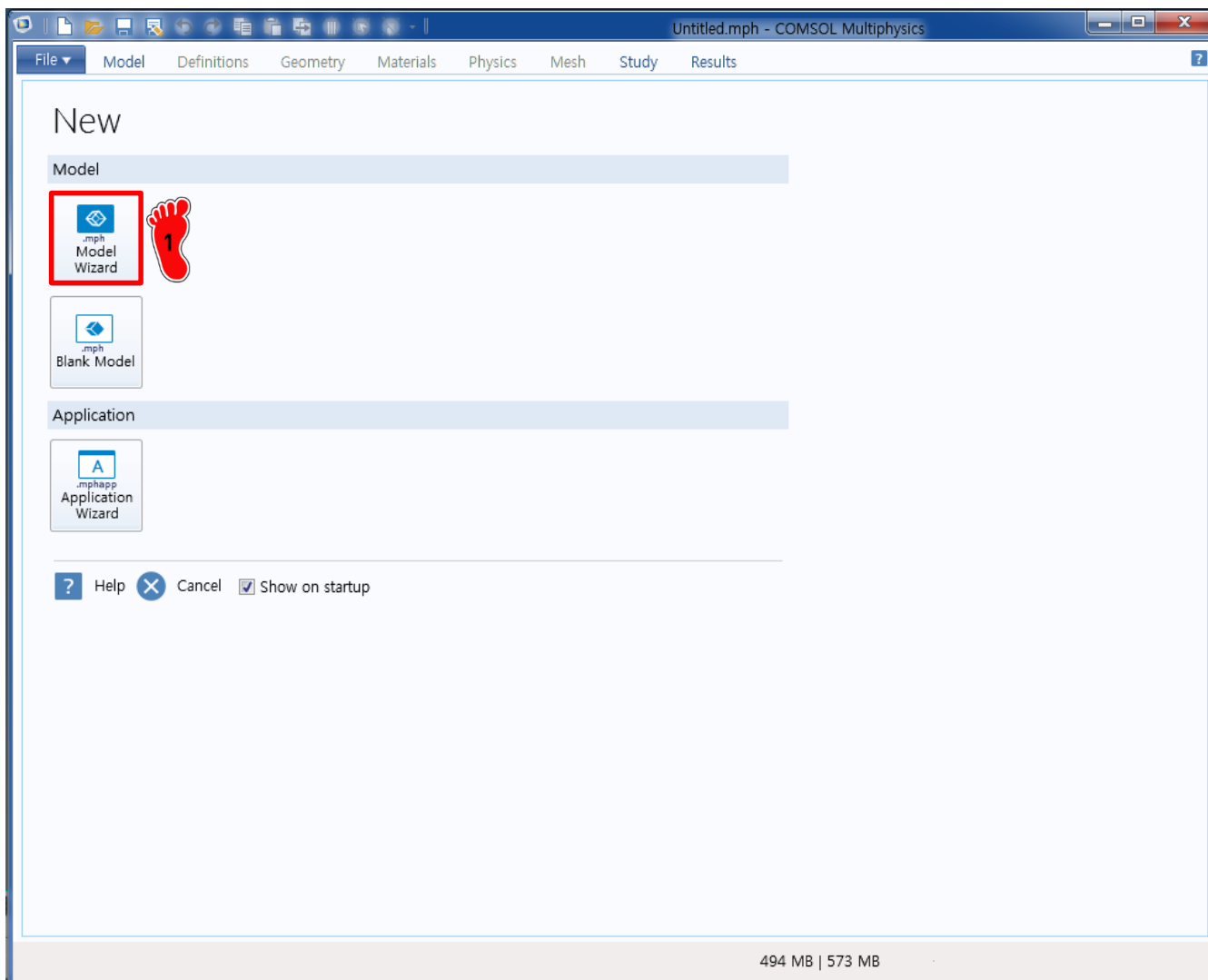


Support conditions and applied loads

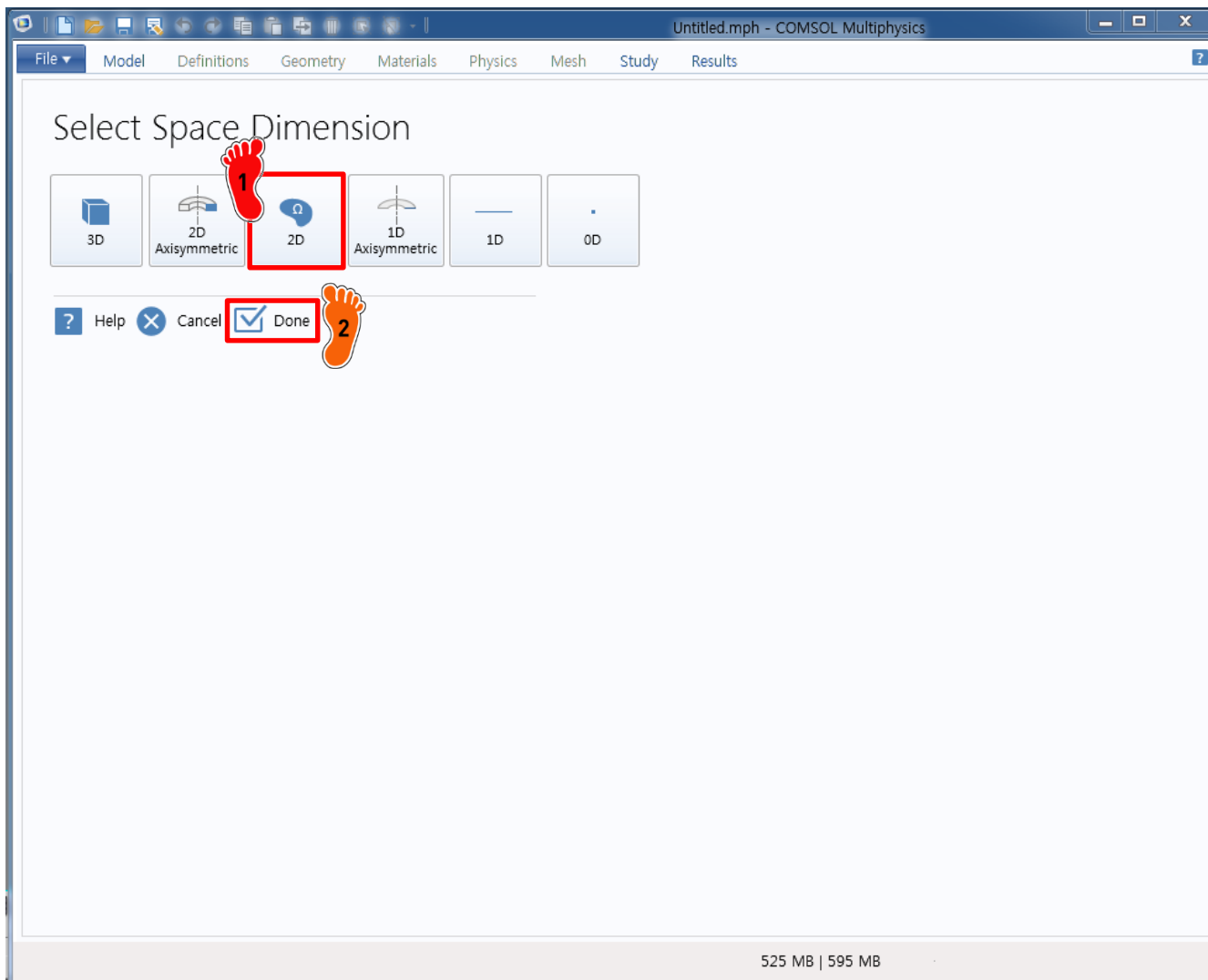
MODEL WIZARD



Model Wizard 선택



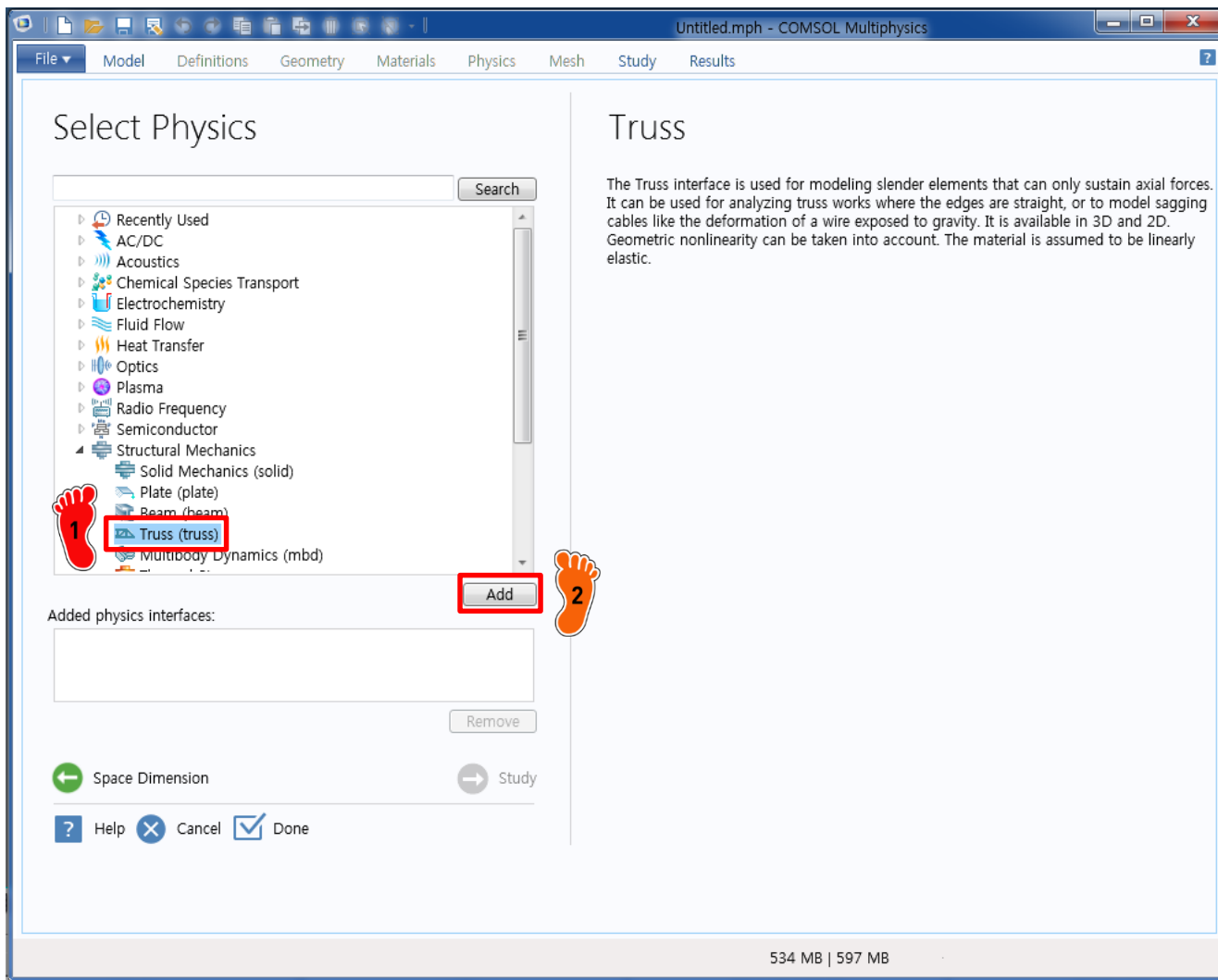
DIMENSION SELECTION



1 2D 선택

2 Done 클릭

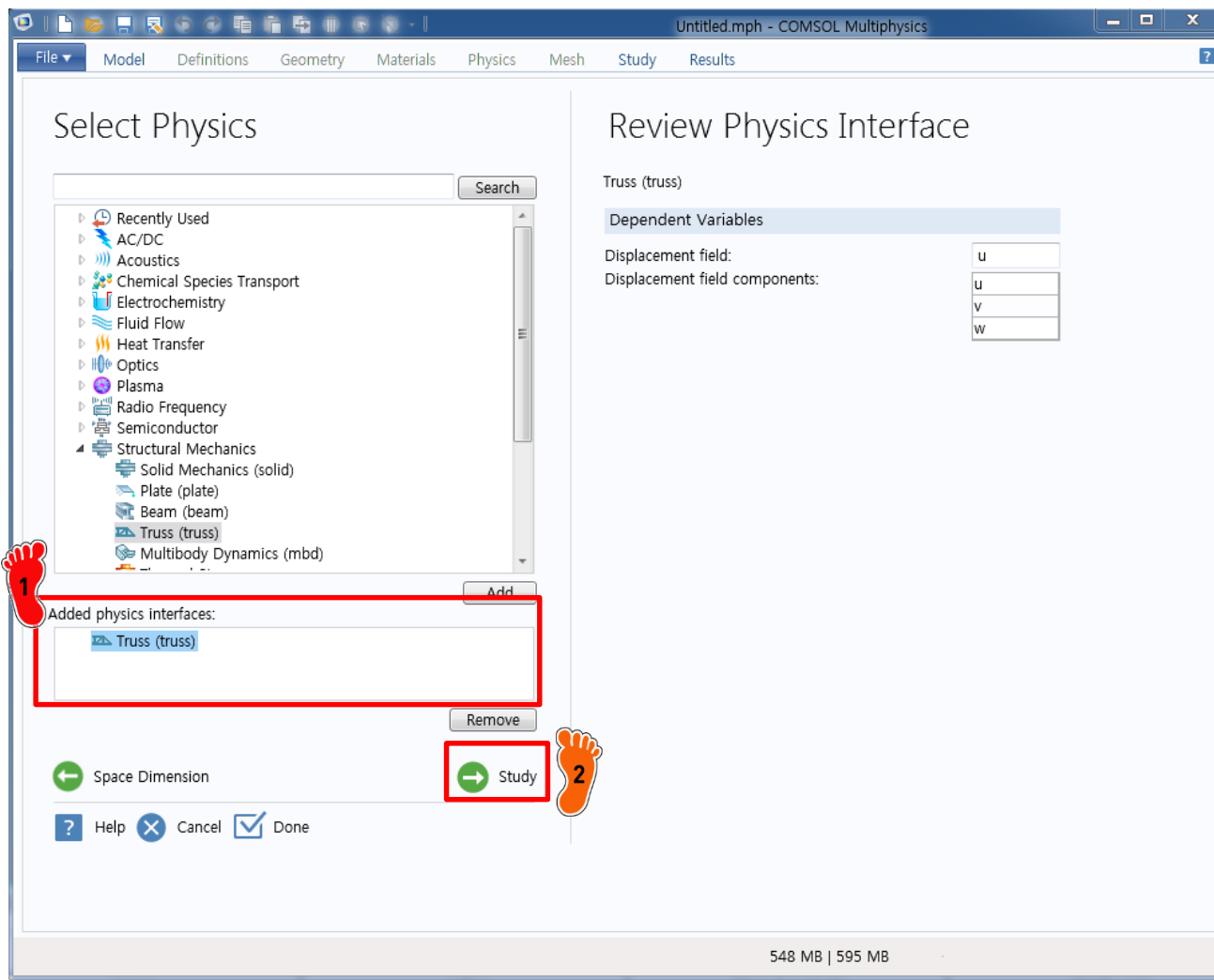
PHYSICS SELECTION



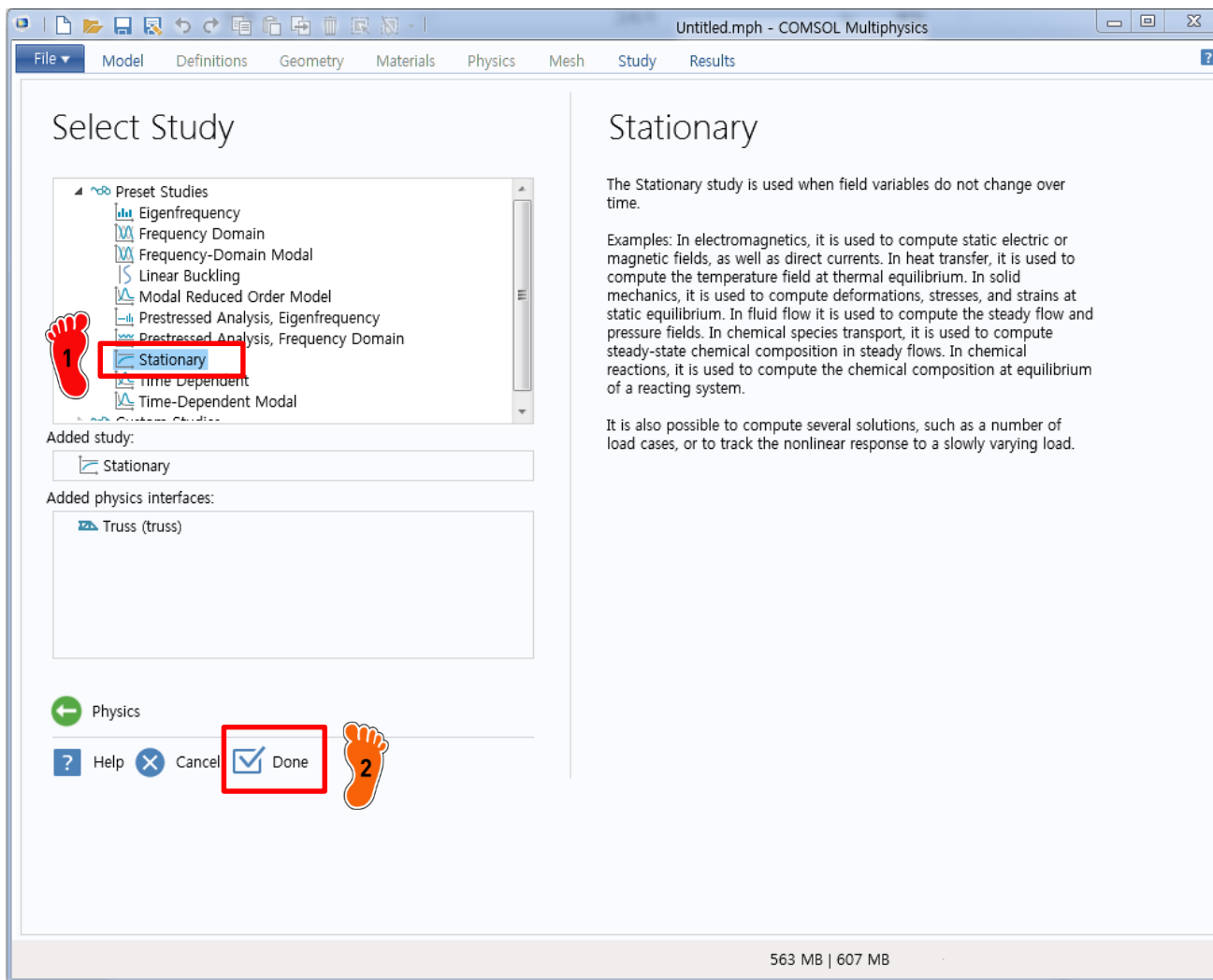
1 Structural Mechanics 모듈
의 Truss physics 선택

2 Add 클릭

PHYSICS SELECTION



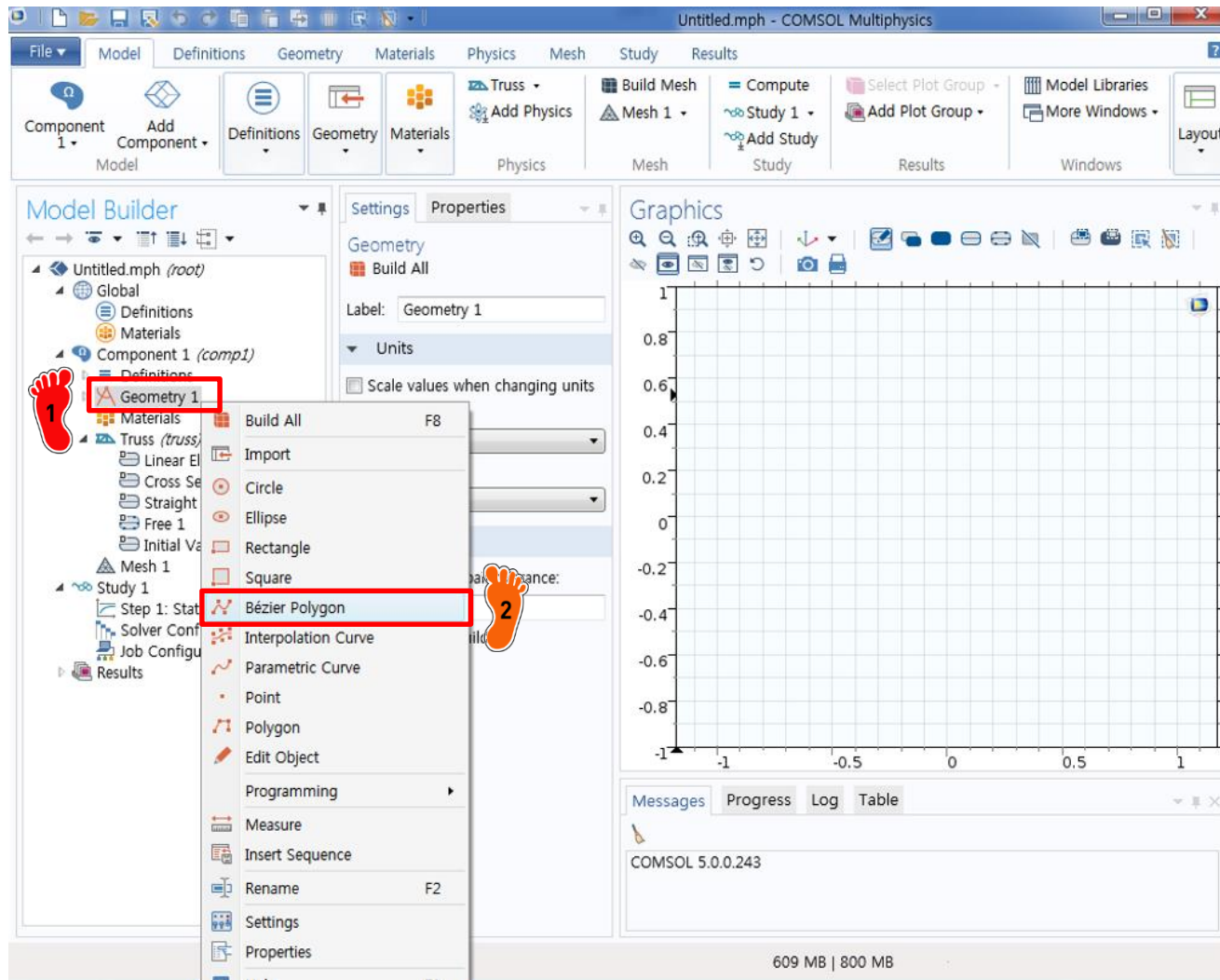
STUDY TYPE SELECTION



1 Preset Studies에서
Stationary 선택

2 Done 클릭

GEOMETRY CREATION



Geometry 1 마우스 우클릭

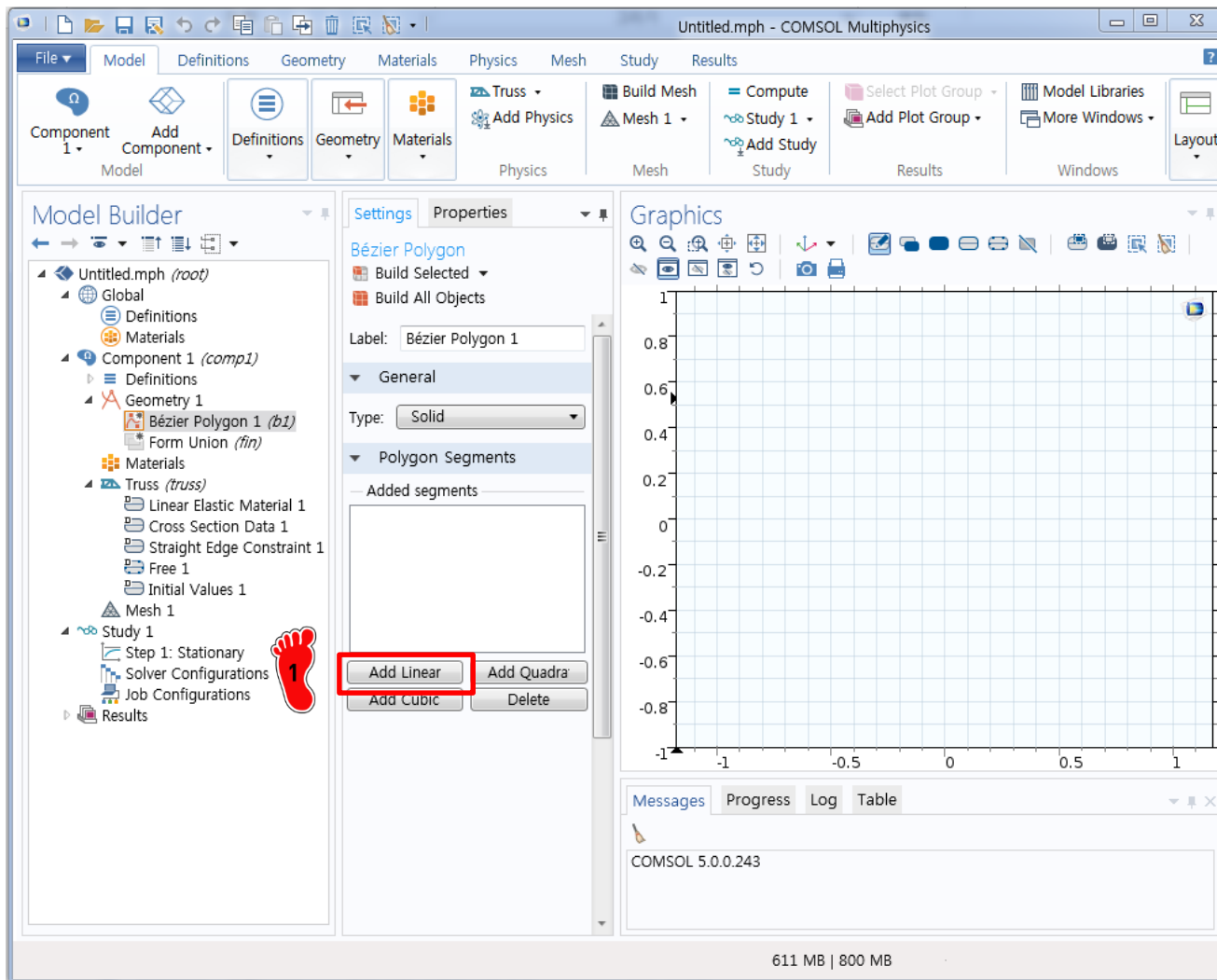


Bezier Polygon 클릭

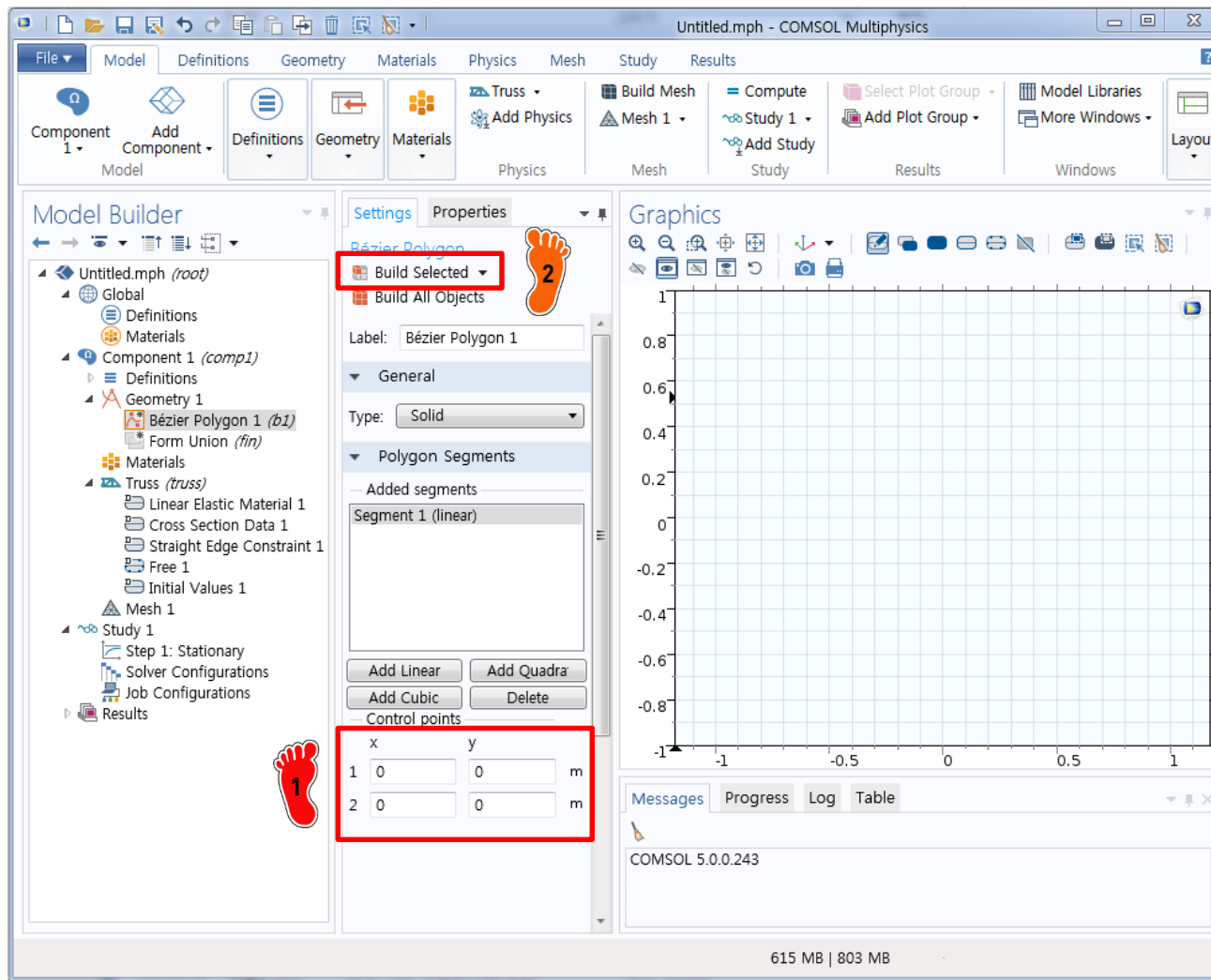
GEOMETRY CREATION



Add Linear 클릭



GEOMETRY CREATION



1 Control points 시작점 (0,0)
끝나는 점 (10,0) 을 입력

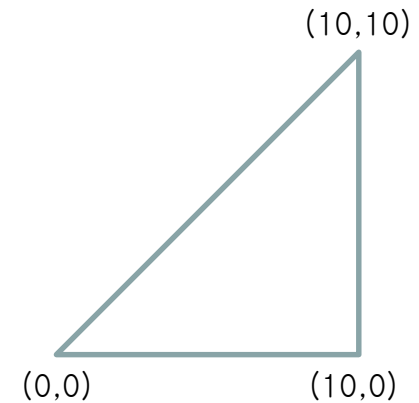
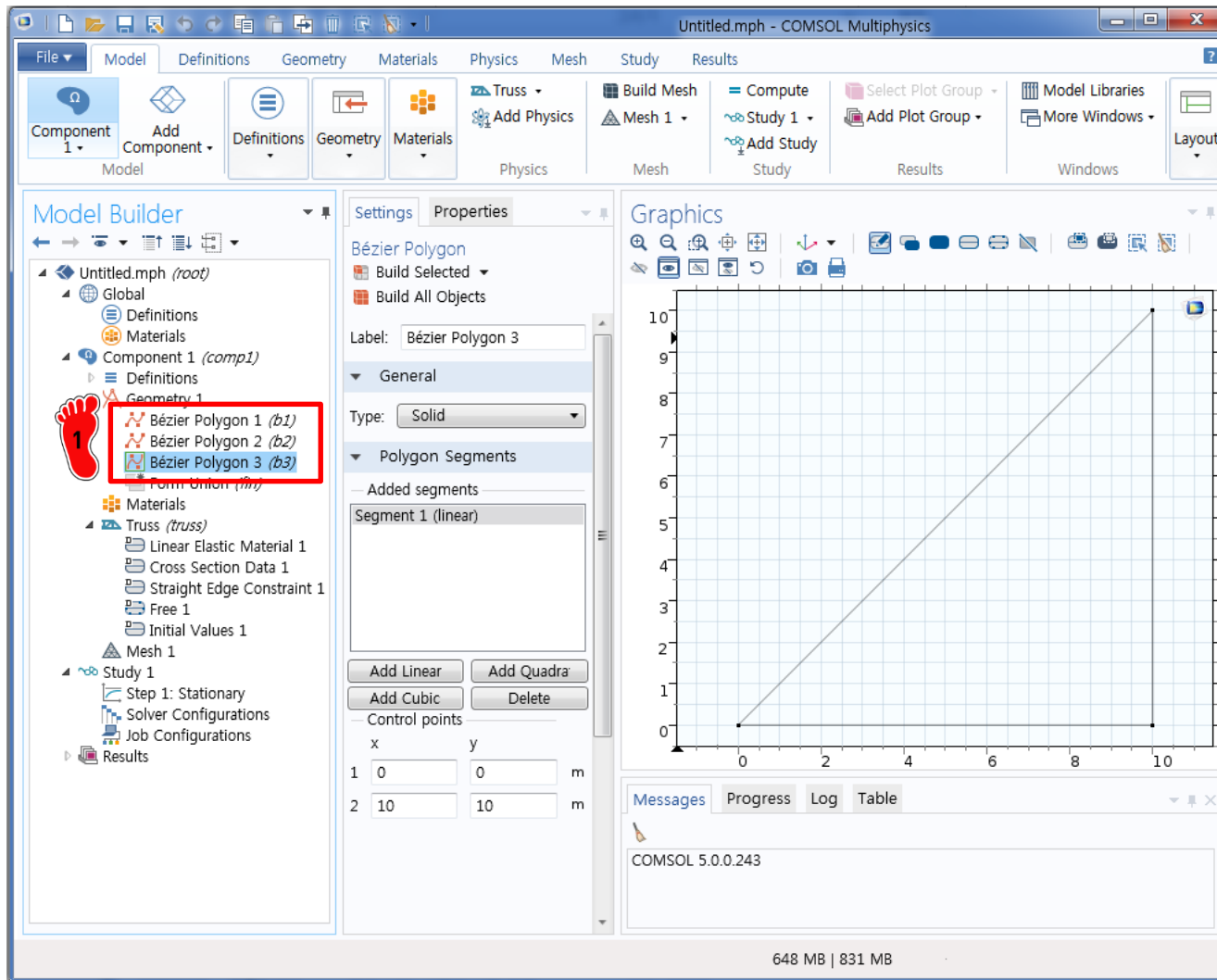
옆의 숫자 1 은 시작점 좌표,
2는 끝나는 점을 의미

2 Build Selected 클릭

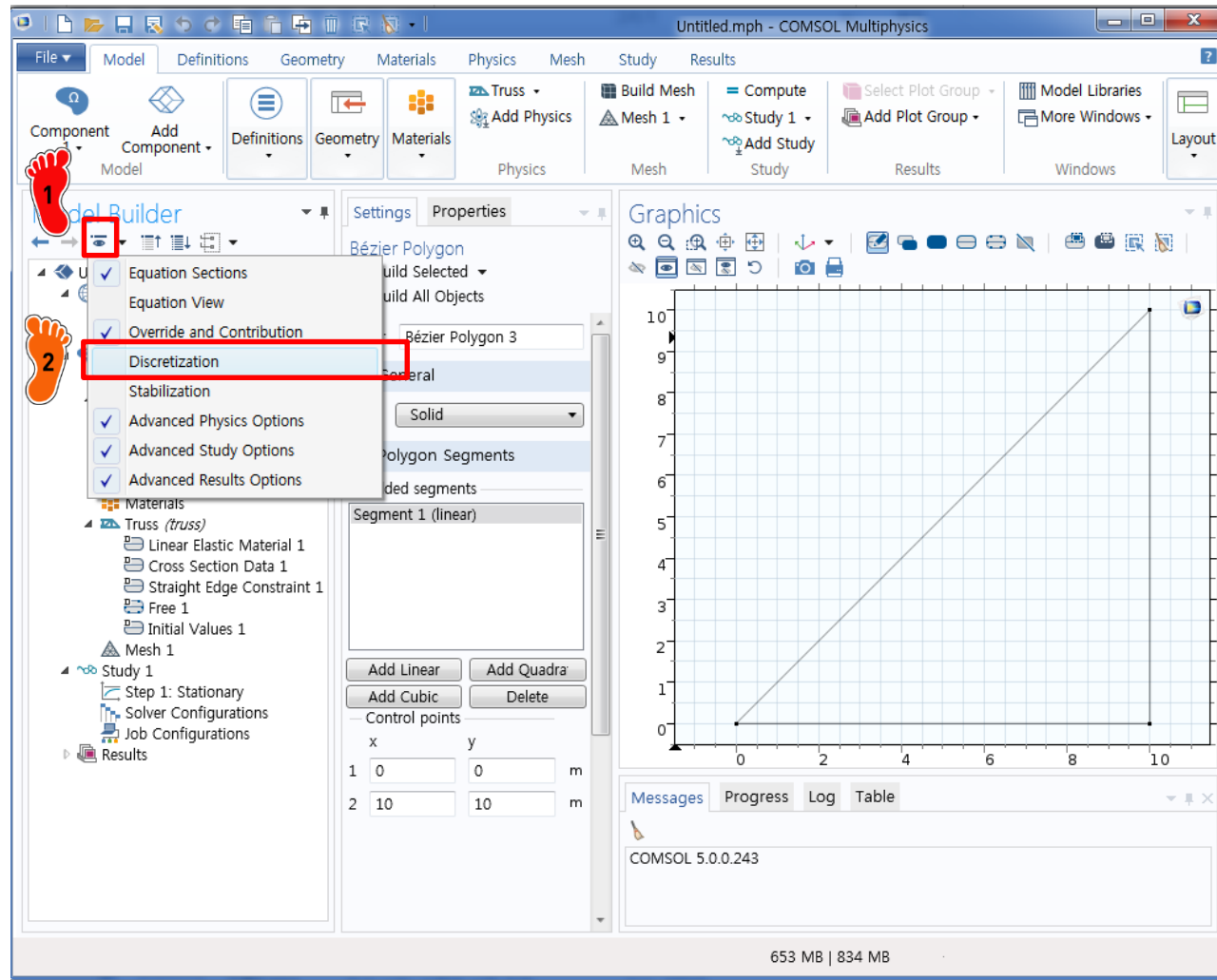
GEOMETRY CREATION



같은 방식으로 총 3개의 직선을 생성



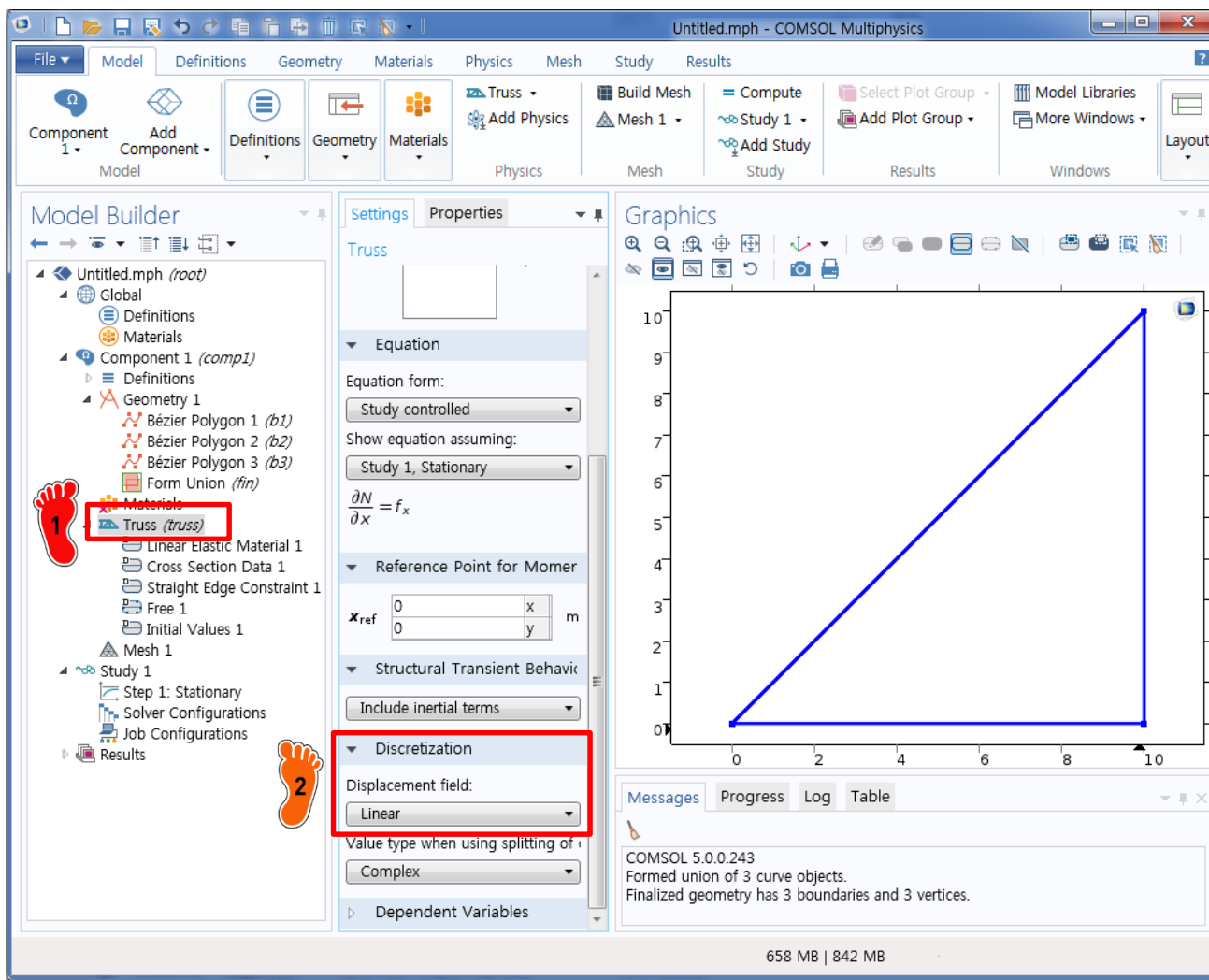
ELEMENT ORDER CHANGING



1 Model Builder 밑에 있는
눈 모양 메뉴 클릭

2 Discretization 클릭

ELEMENT ORDER CHANGING



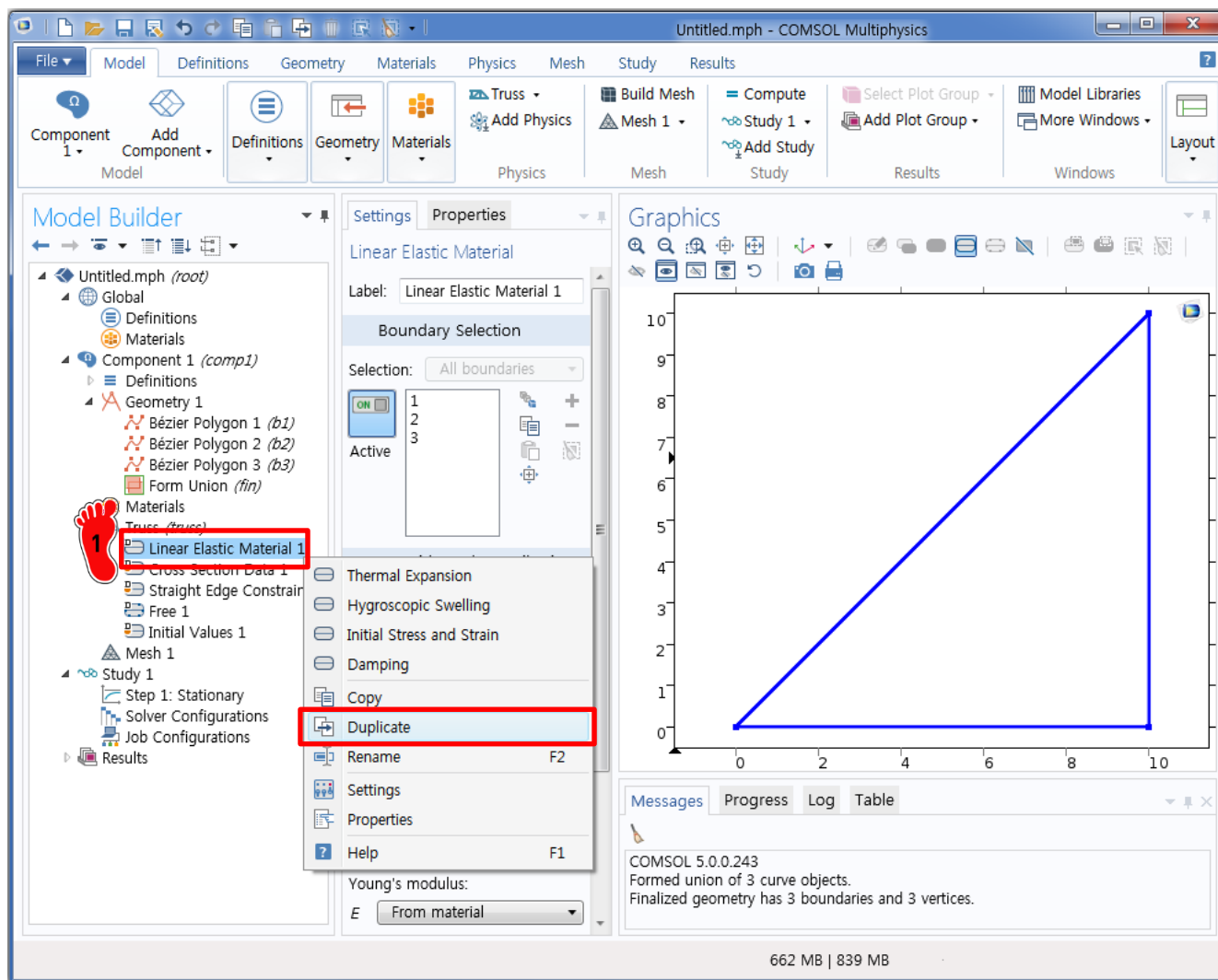
Truss physics 클릭



Displacement field 를
Linear 로 변경

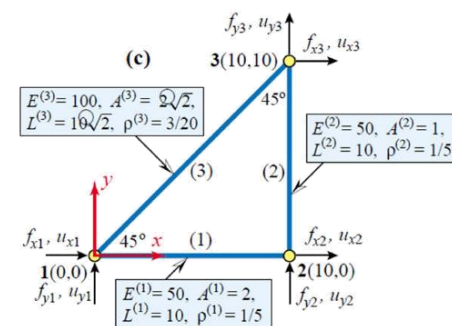
- **Direct Stiffness Method: Truss**
 - ✓ **Geometry creation**
 - ✓ **Material property**
 - ✓ **Cross section property**
 - ✓ **Boundary condition**
 - ✓ **Nodal force**
 - ✓ **Analysis**

MATERIAL PROPERTY



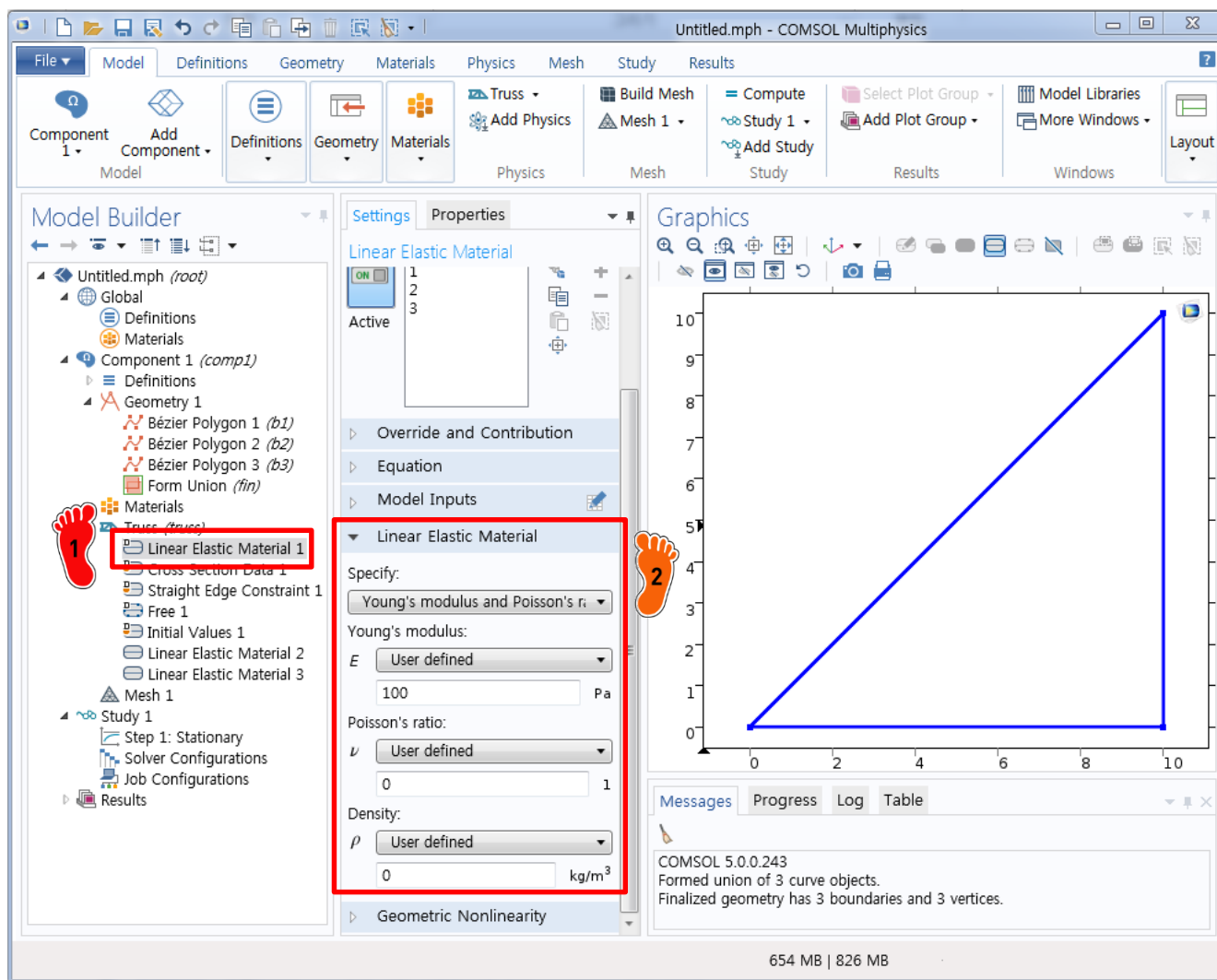
1 Linear Elastic Material 마우스 우클릭

2 Duplicate 를 이용하여 Linear Elastic Material 메뉴 2 개 더 생성



Geometric, material and fabrication properties

MATERIAL PROPERTY

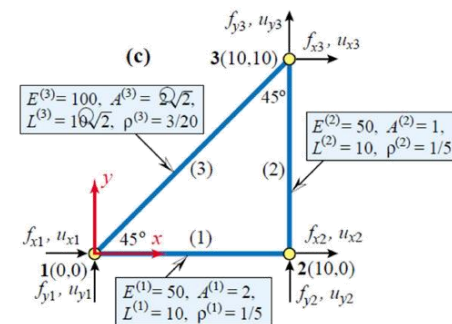


1 첫 번째 Linear Elastic Material 클릭

2 From material을 User defined로 변경 후

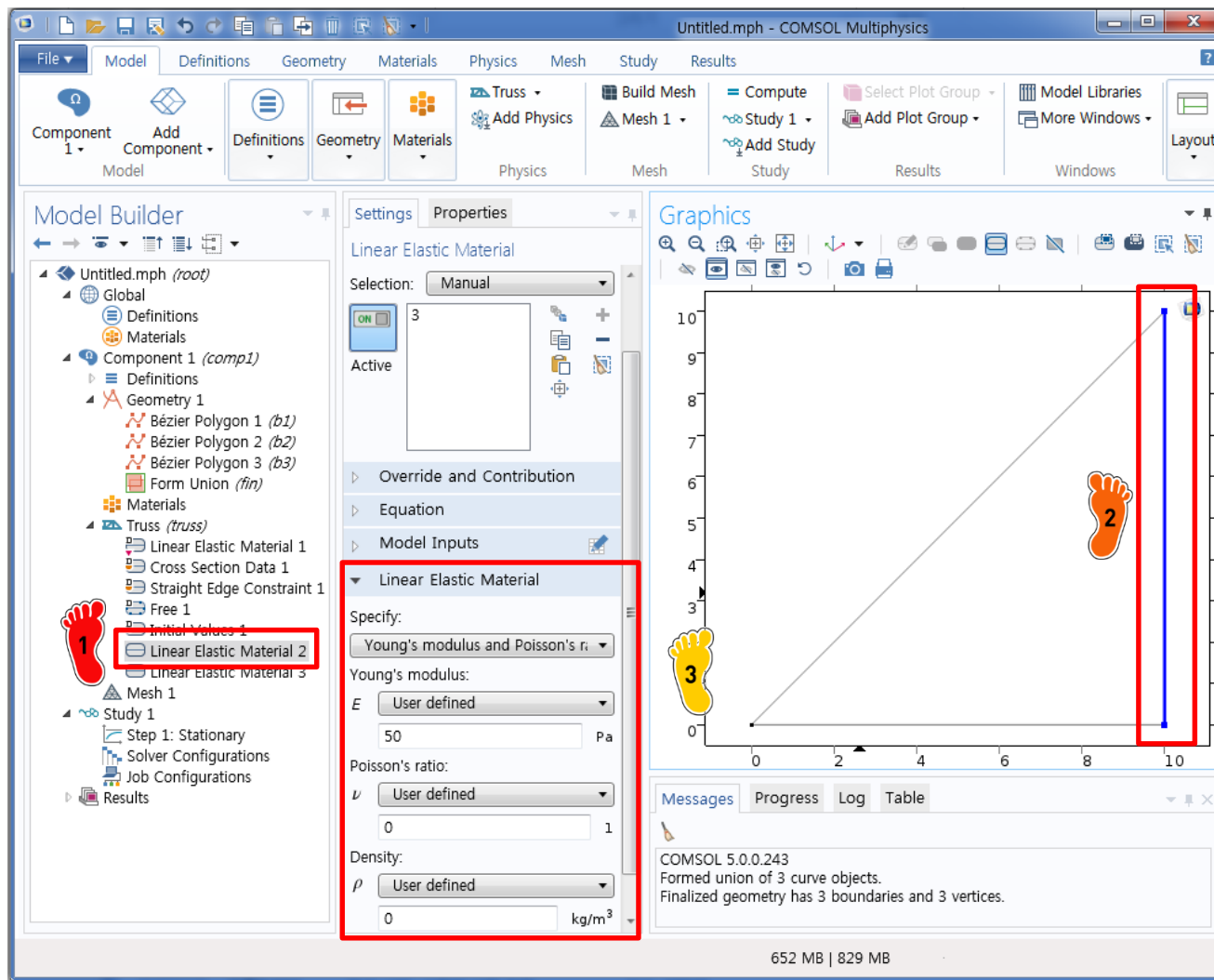
Young's modulus 100 입력

푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

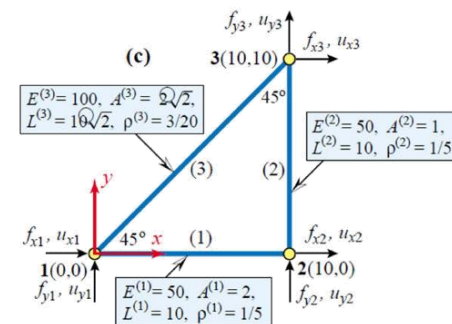
MATERIAL PROPERTY



1 두 번째 Linear Elastic Material 클릭

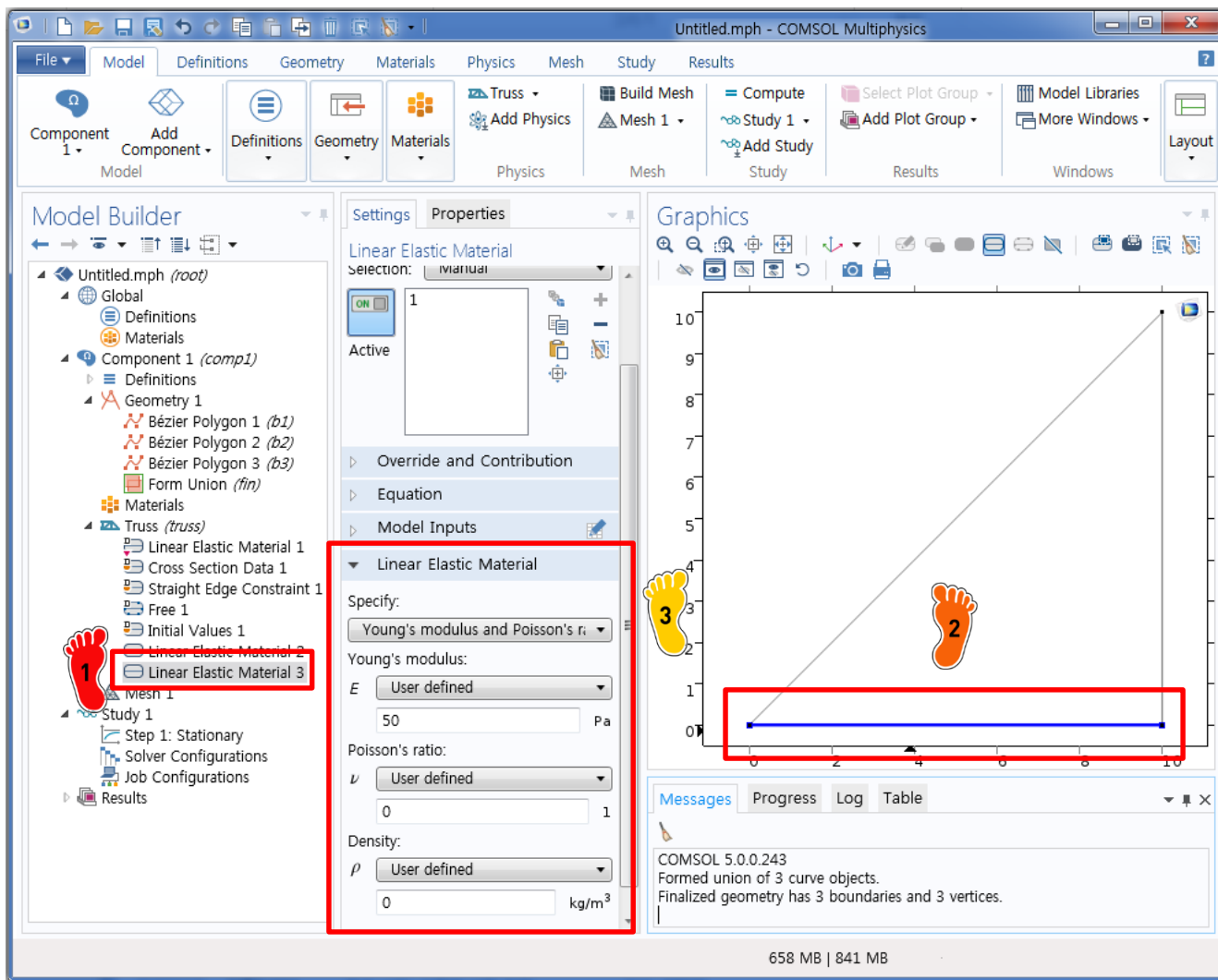
2 제일 오른쪽 부재 선택

3 Young's modulus 50 입력
푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

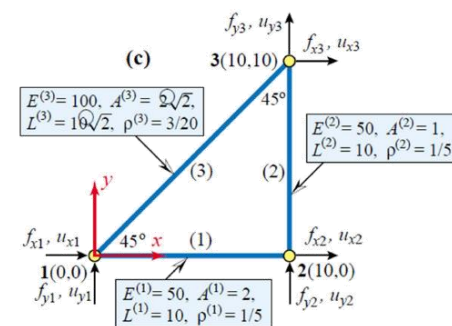
MATERIAL PROPERTY



1 마지막 Linear Elastic Material 클릭

2 제일 아래 부재 선택

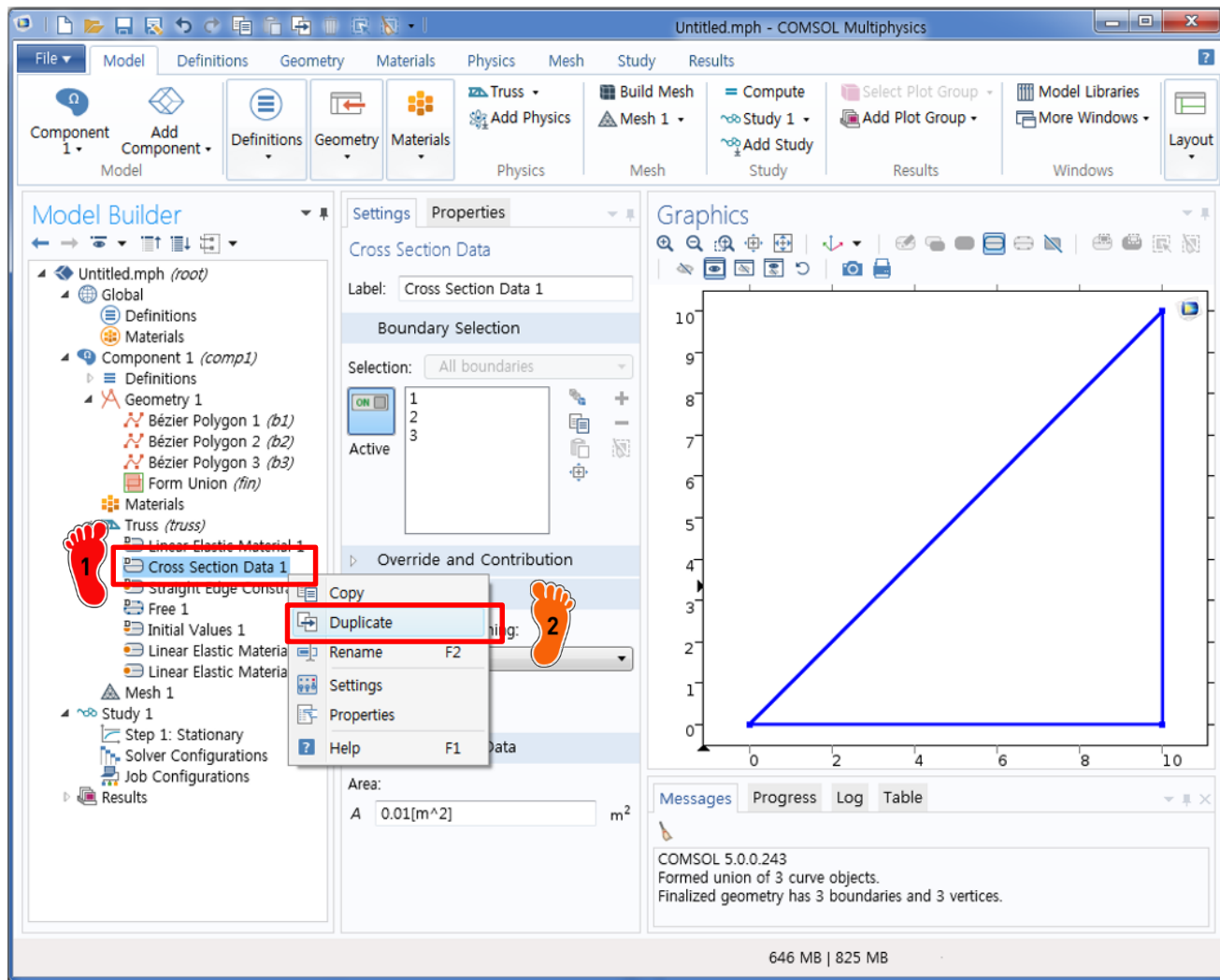
3 Young's modulus 50 입력
푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

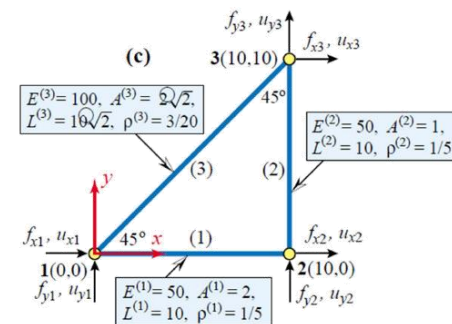
- **Direct Stiffness Method: Truss**
 - ✓ **Geometry creation**
 - ✓ **Material property**
 - ✓ **Cross section property**
 - ✓ **Boundary condition**
 - ✓ **Nodal force**
 - ✓ **Analysis**

SECTION PROPERTY



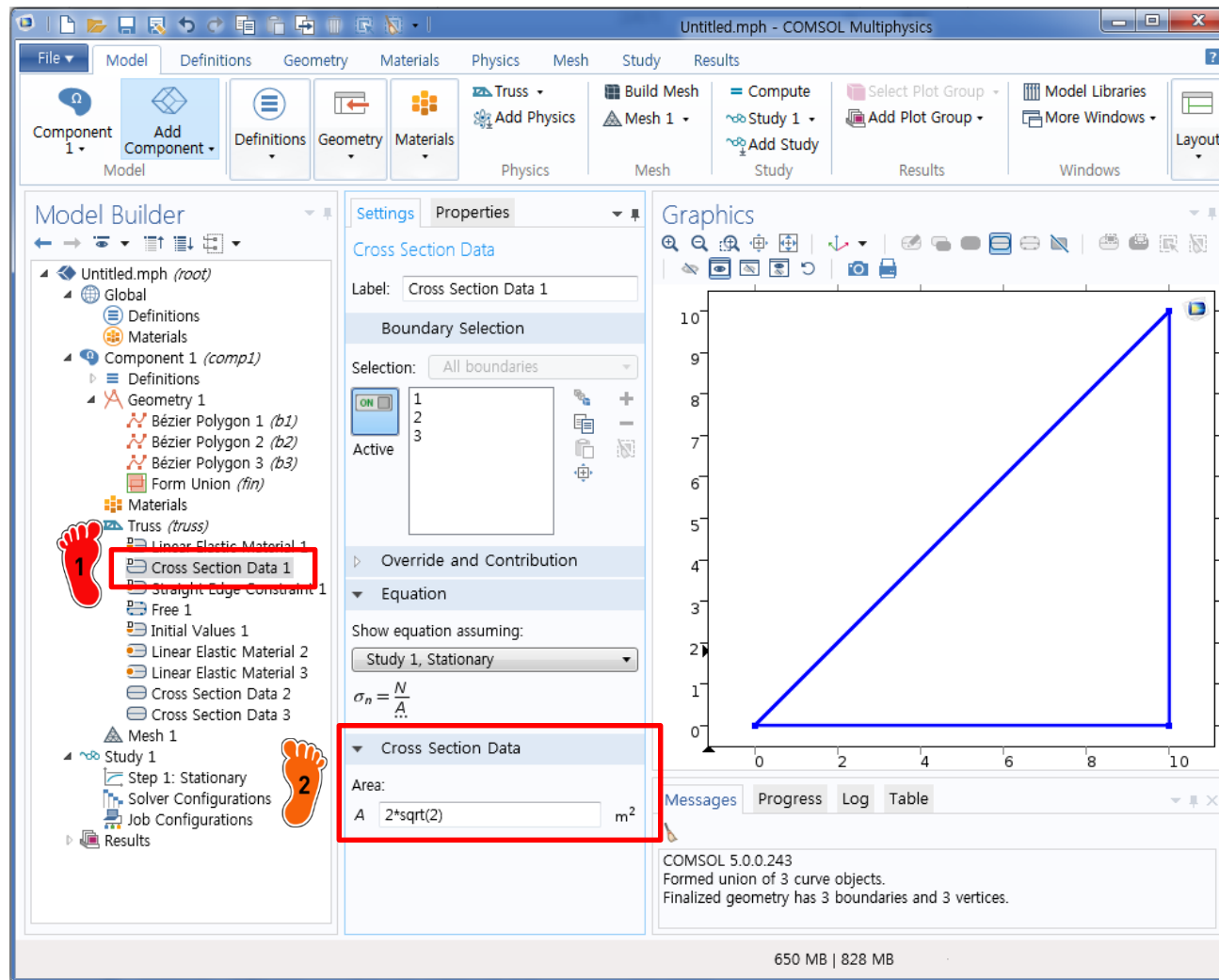
1 Cross Section Data 마우스 우클릭

2 Duplicate 를 이용하여 Cross Section Data 메뉴를 2 개 더 생성



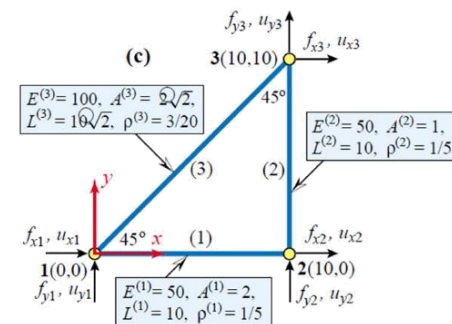
Geometric, material and fabrication properties

SECTION PROPERTY



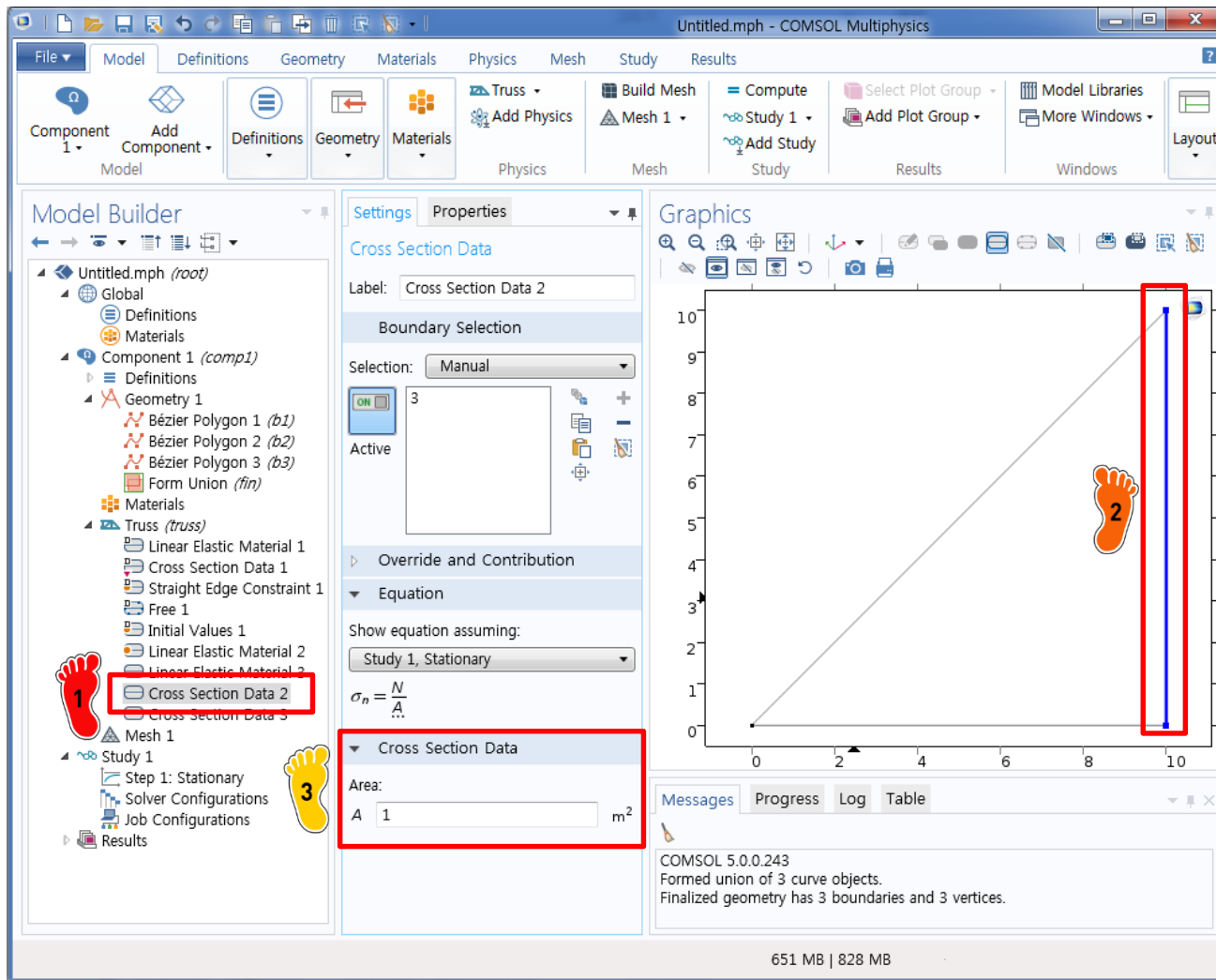
1 첫 번째 Cross Section Data 클릭

2 Area 에 $2\sqrt{2}$ 입력



Geometric, material and fabrication properties

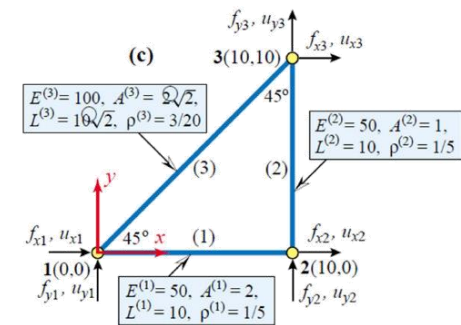
SECTION PROPERTY



1 두 번째 Cross Section Data 클릭

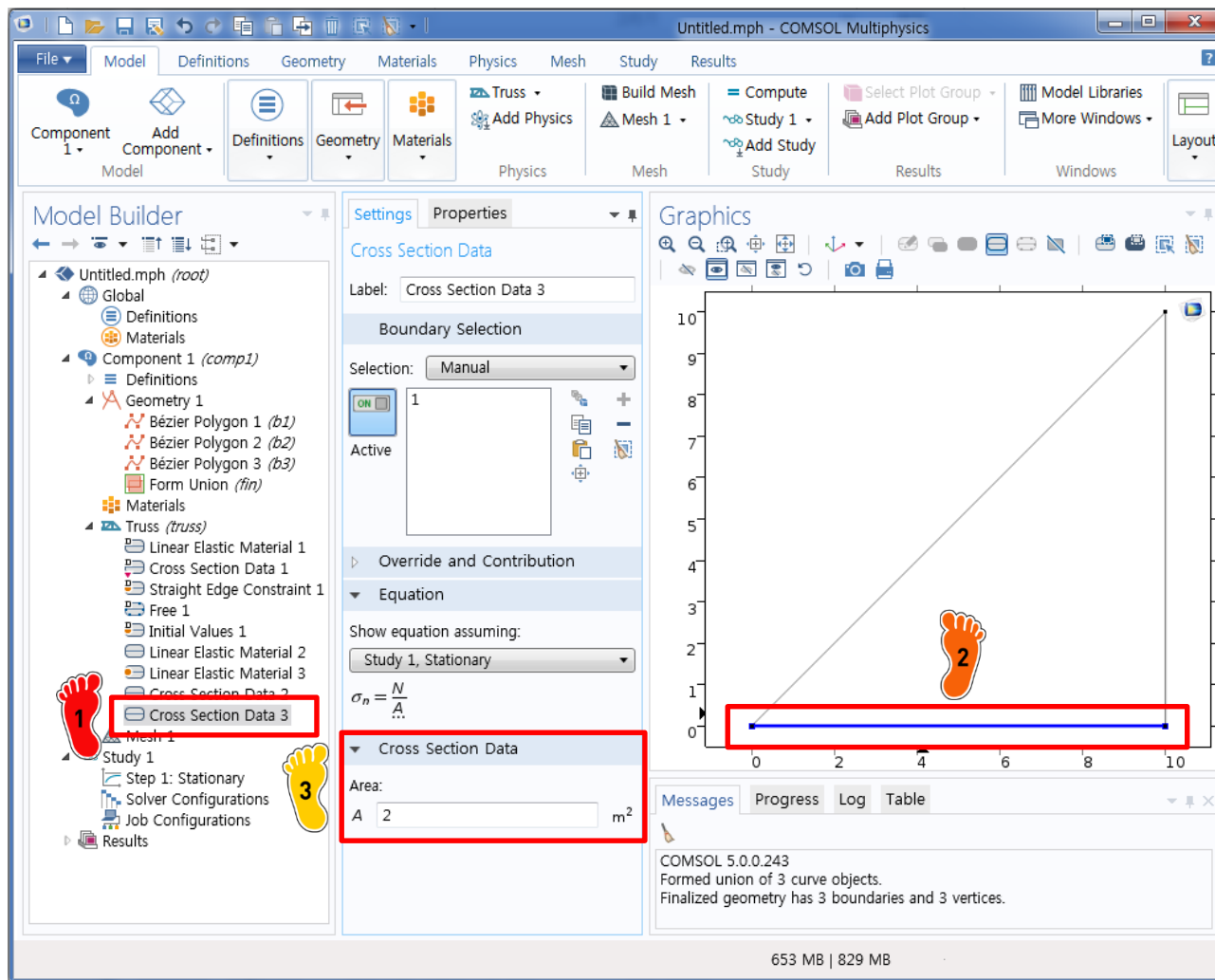
2 제일 오른쪽 부재 선택

3 Area 에 1 입력



Geometric, material and fabrication properties

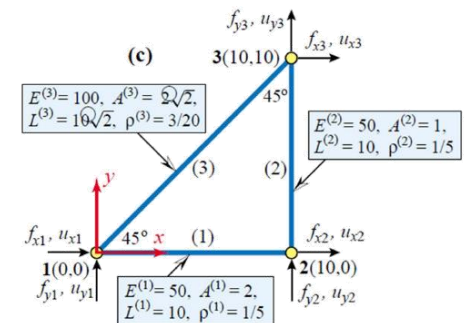
SECTION PROPERTY



1 마지막 Cross Section Data 클릭

2 제일 아래 부재 선택

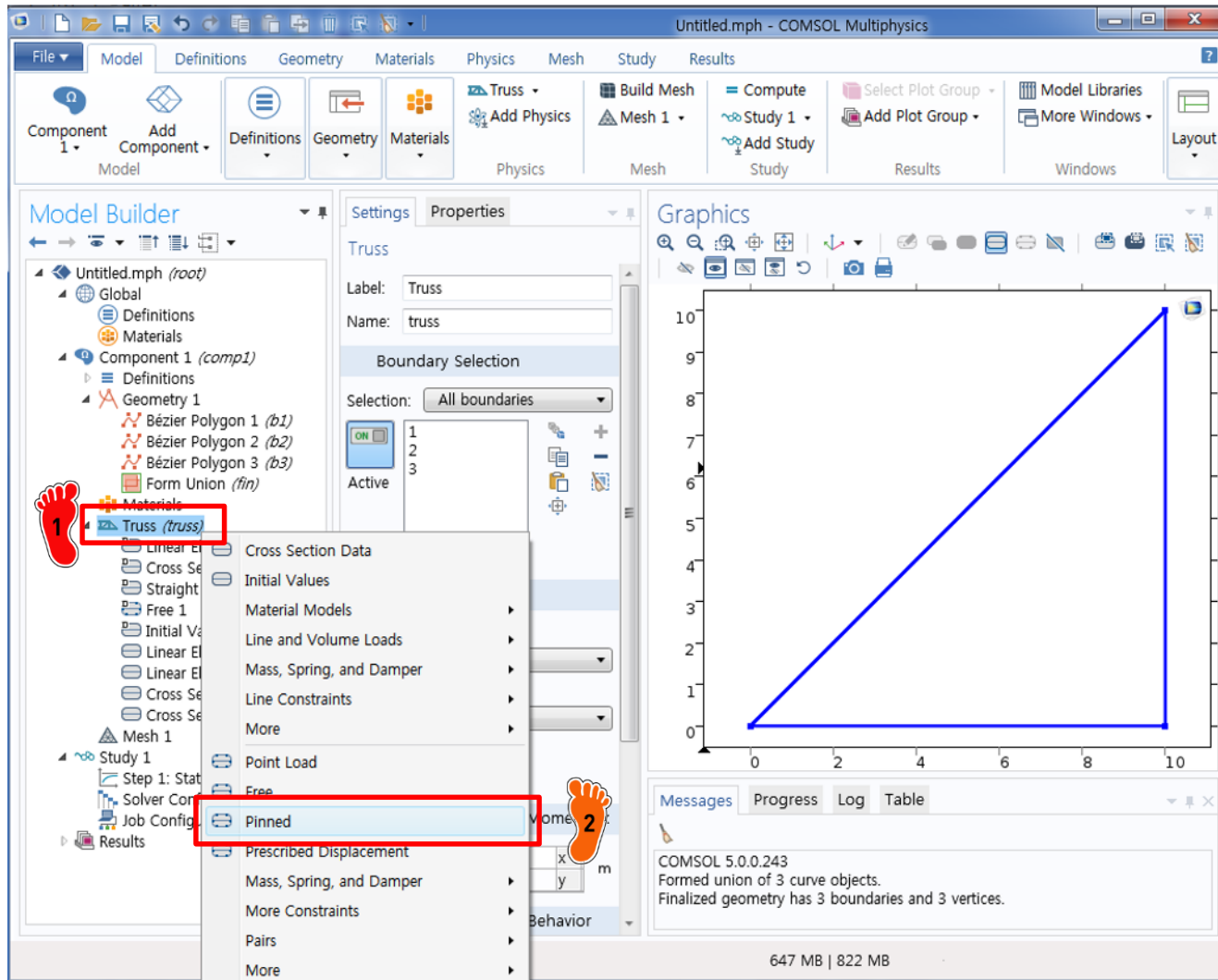
3 Area 에 2 입력



Geometric, material and fabrication properties

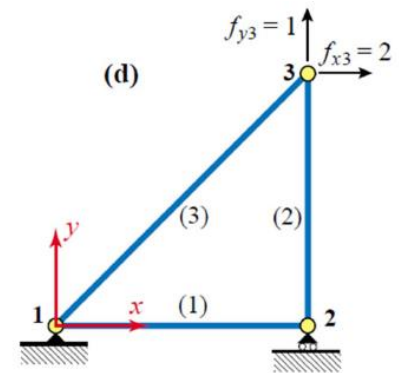
- **Direct Stiffness Method: Truss**
 - ✓ **Geometry creation**
 - ✓ **Material property**
 - ✓ **Cross section property**
 - ✓ **Boundary condition**
 - ✓ **Nodal force**
 - ✓ **Analysis**

BOUNDARY CONDITION



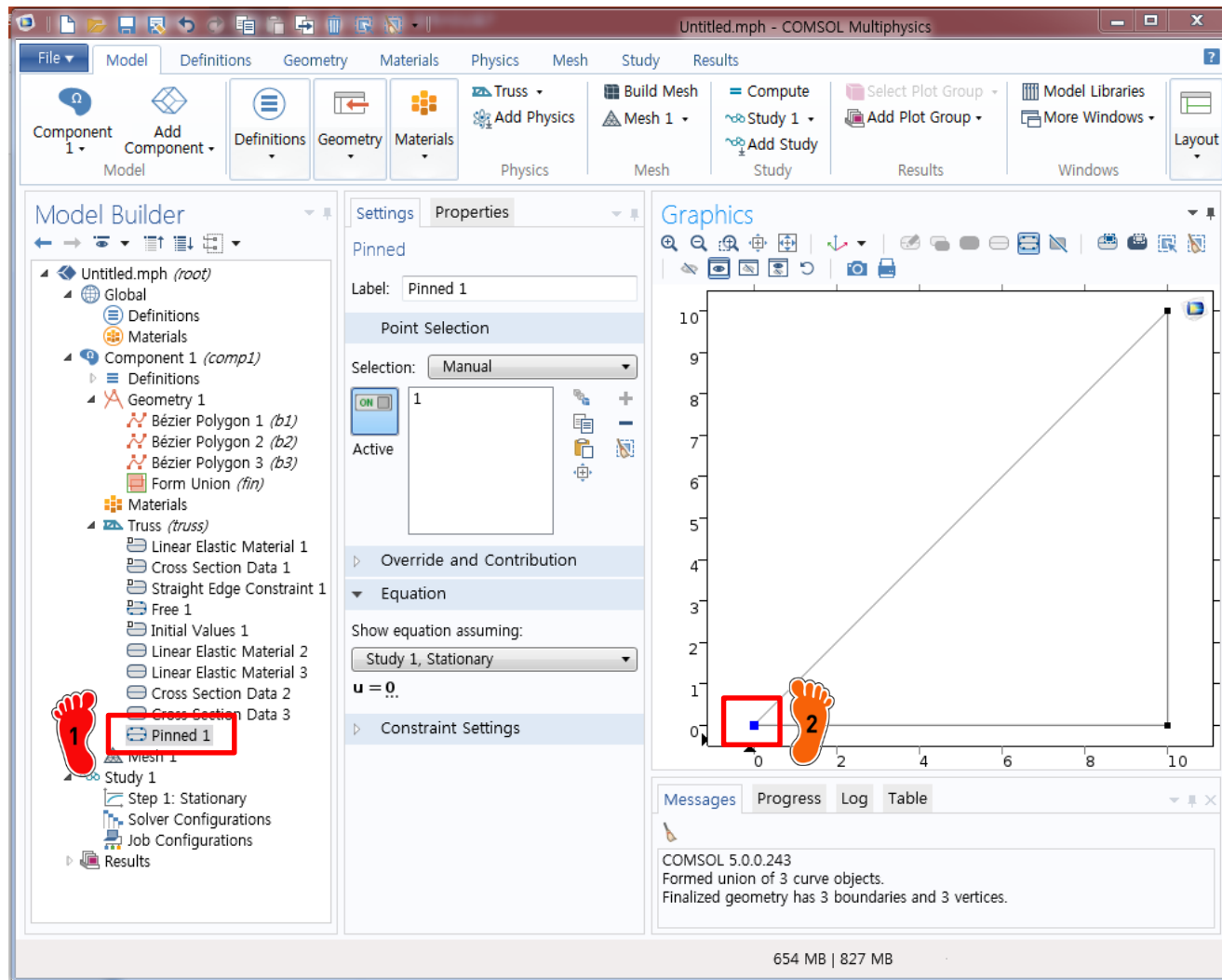
1 Truss 마우스 우클릭

2 Pinned 클릭



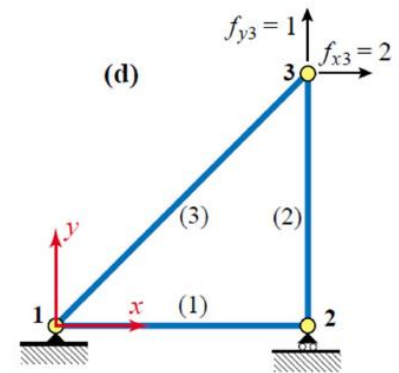
Support conditions and applied loads

BOUNDARY CONDITION



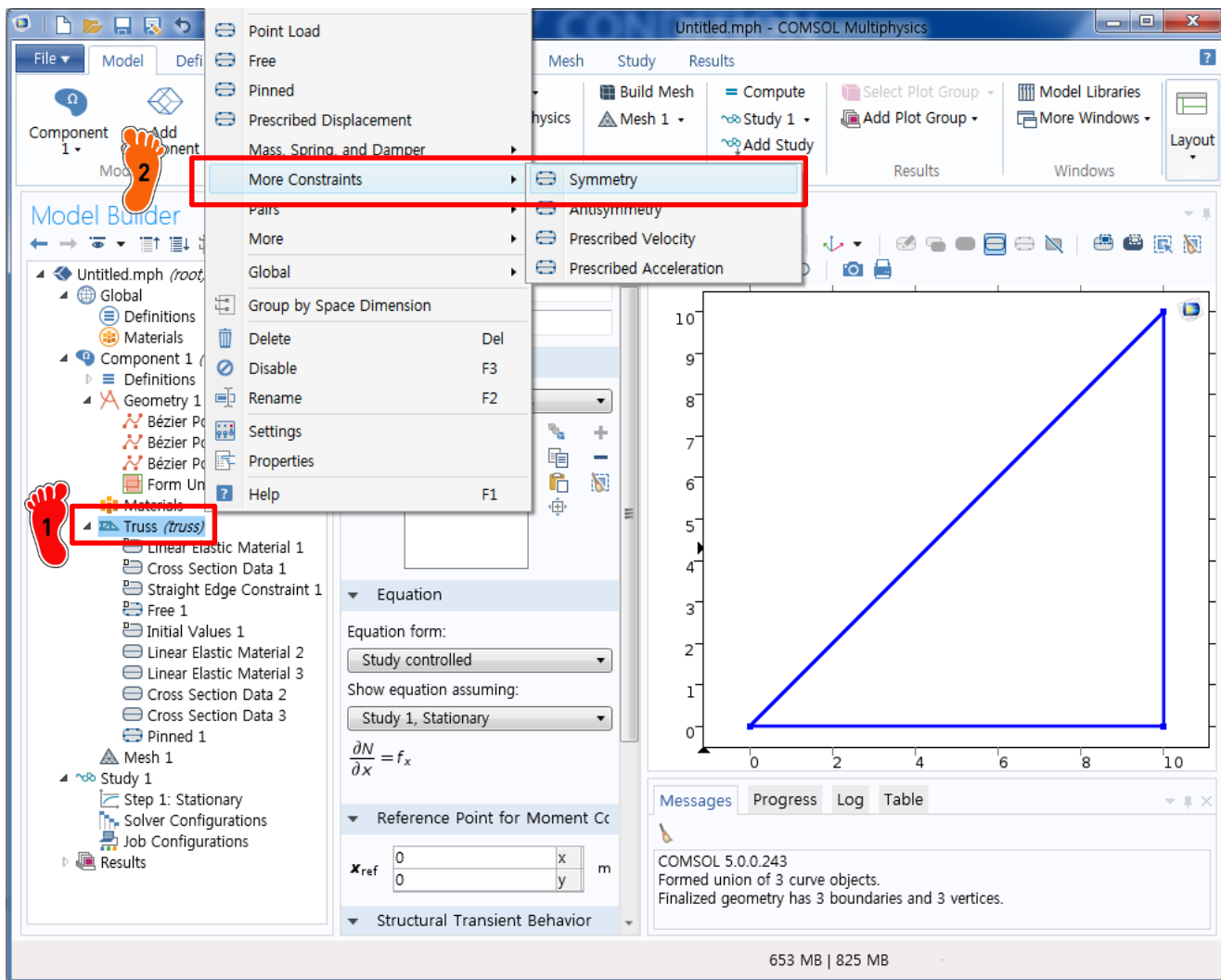
1 Pinned 클릭

2 왼쪽 아래 절점 선택



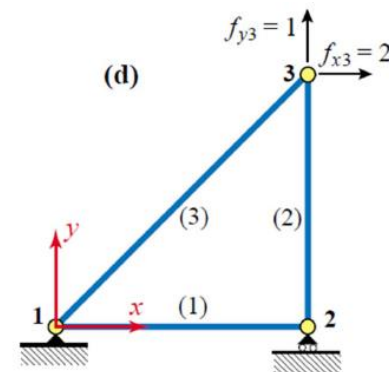
Support conditions and applied loads

BOUNDARY CONDITION



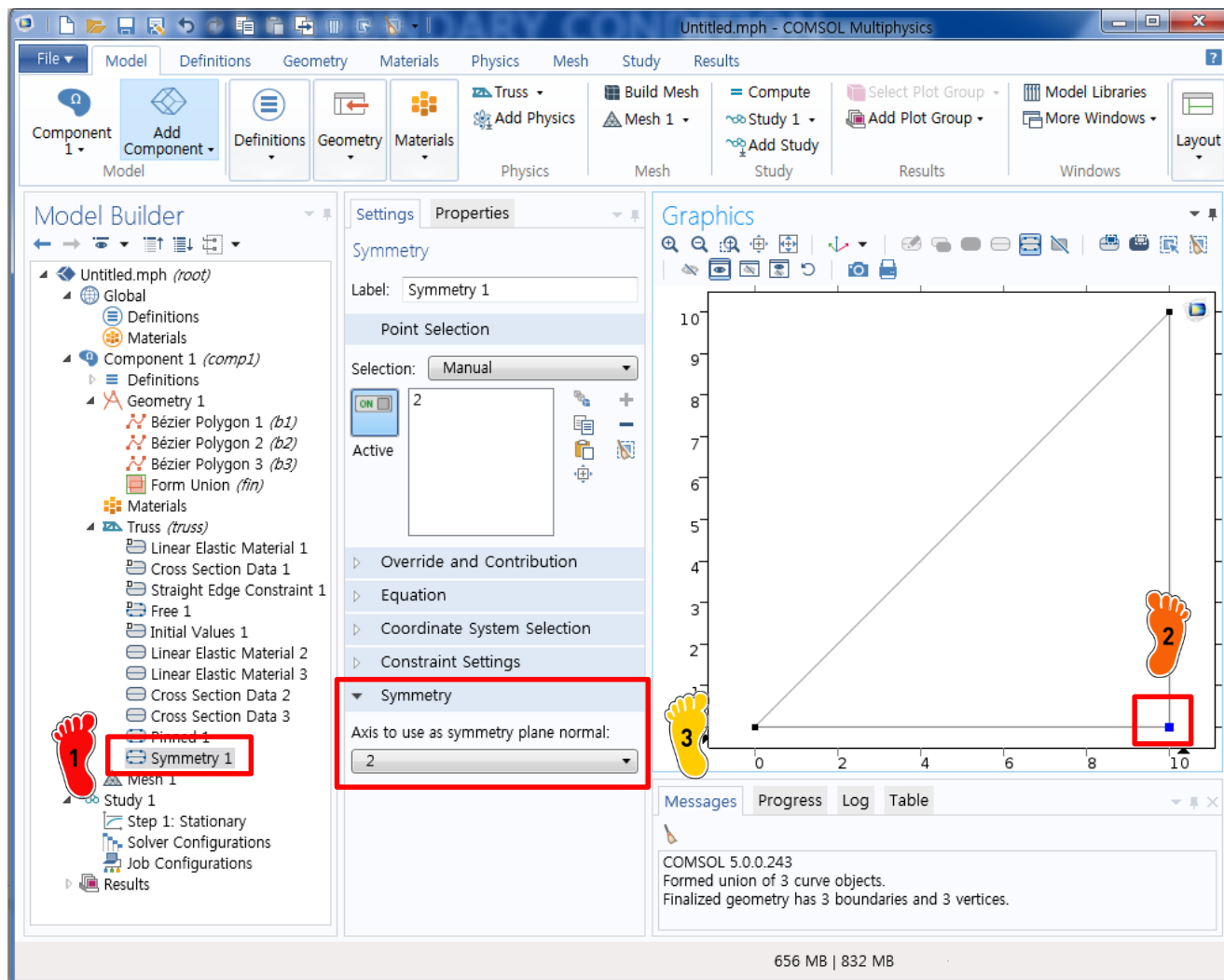
1 Truss 마우스 우클릭

2 More Constraints에서 Symmetry 클릭



Support conditions and applied loads

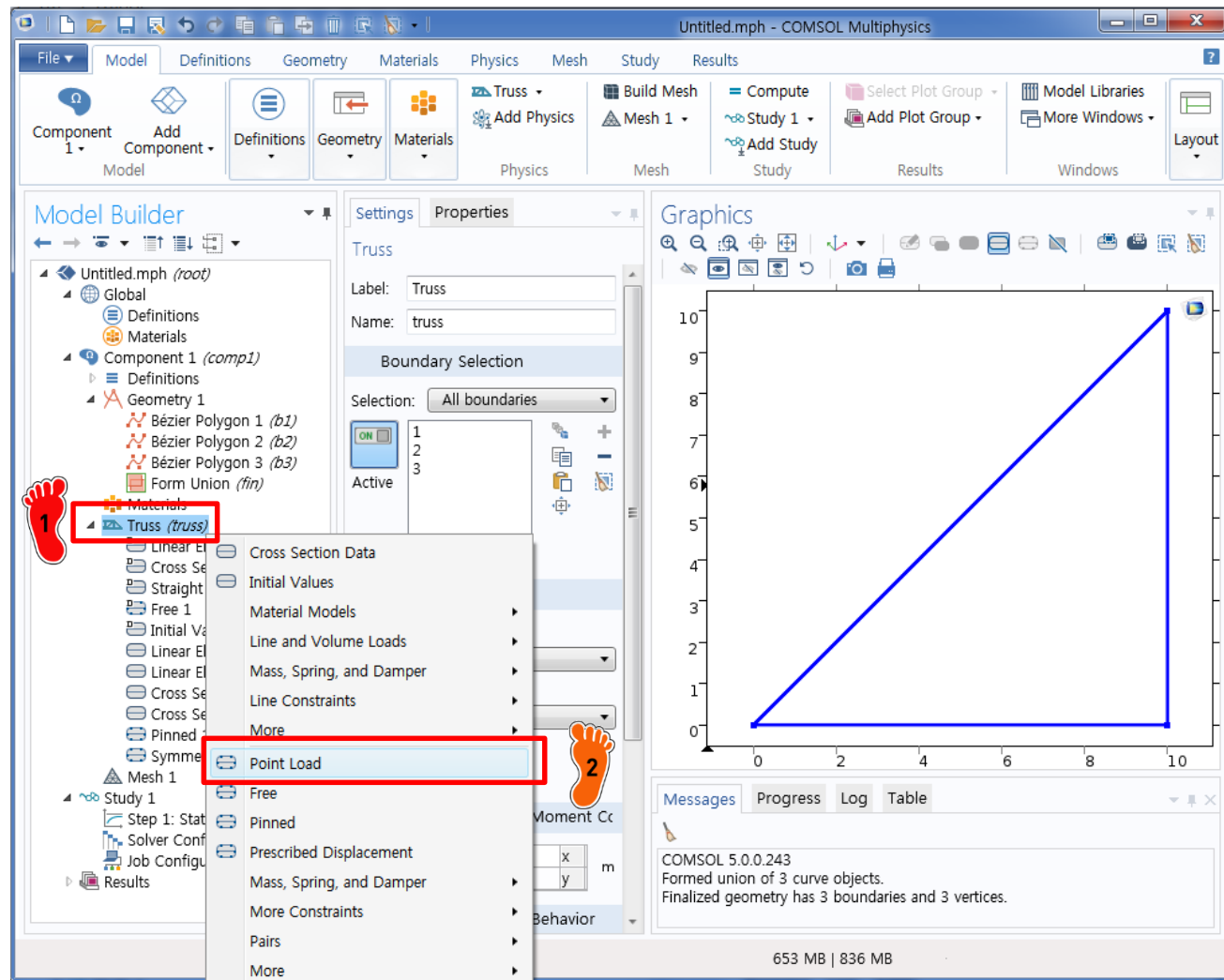
BOUNDARY CONDITION



Support conditions and applied loads

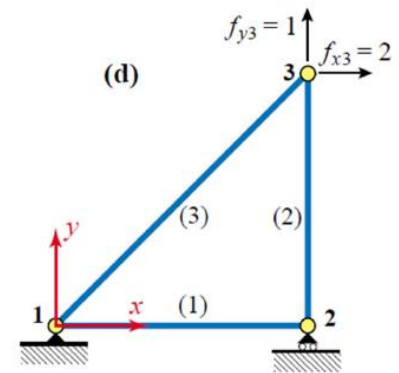
- **Direct Stiffness Method: Truss**
 - ✓ **Geometry creation**
 - ✓ **Material property**
 - ✓ **Cross section property**
 - ✓ **Boundary condition**
 - ✓ **Nodal force**
 - ✓ **Analysis**

POINT LOAD



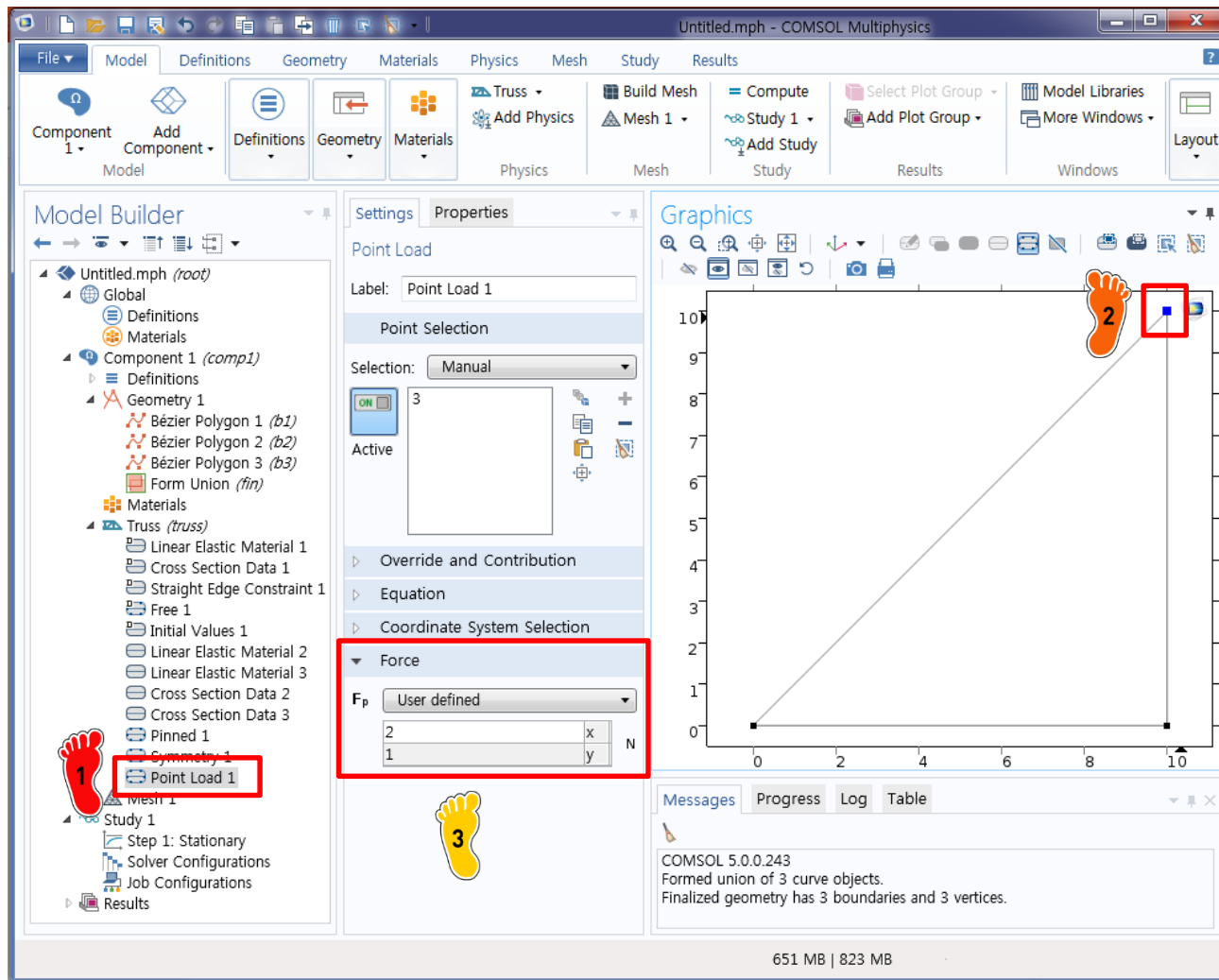
1 Truss 마우스 우클릭

2 Point Load 클릭

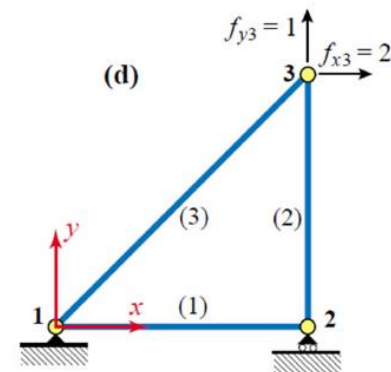


Support conditions and applied loads

POINT LOAD



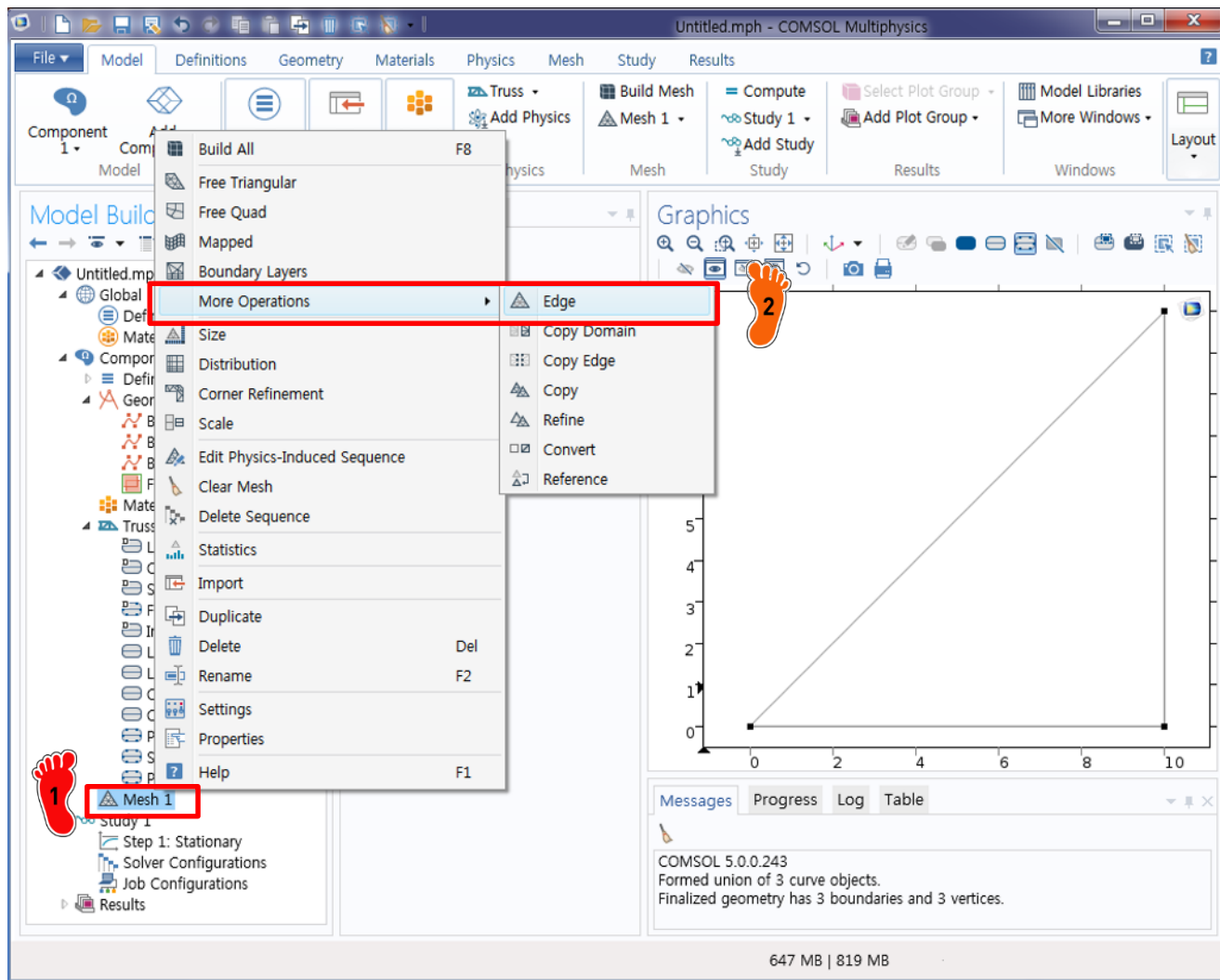
- 1 Point Load 1 클릭
- 2 오른쪽 위 절점 선택
- 3 x 방향 2, y 방향 1 입력



Support conditions and applied loads

- **Direct Stiffness Method: Truss**
 - ✓ **Geometry creation**
 - ✓ **Material property**
 - ✓ **Cross section property**
 - ✓ **Boundary condition**
 - ✓ **Nodal force**
 - ✓ **Analysis**

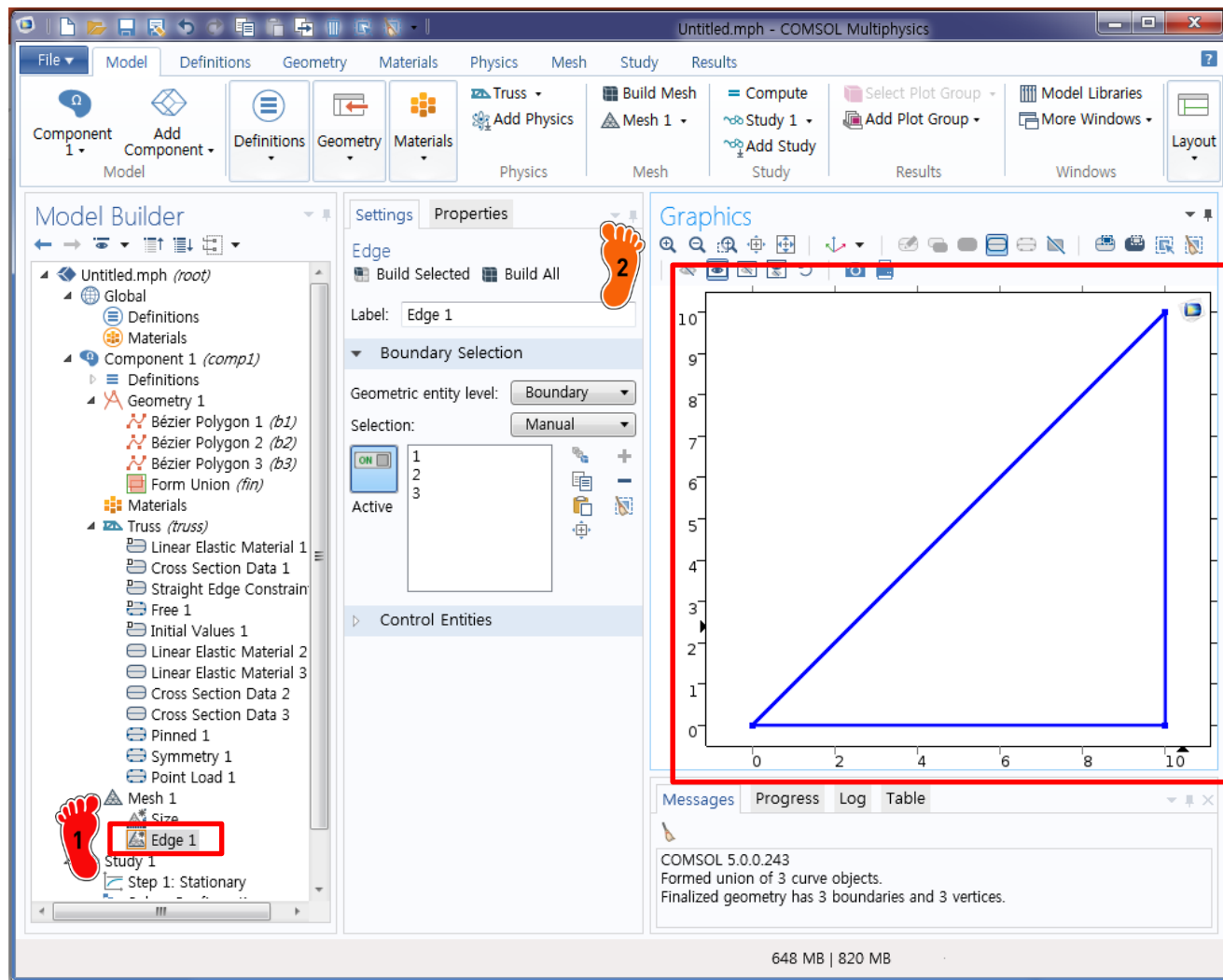
MESH CREATION



1 Mesh 1 마우스 우클릭

2 More Operations에서
Edge 클릭

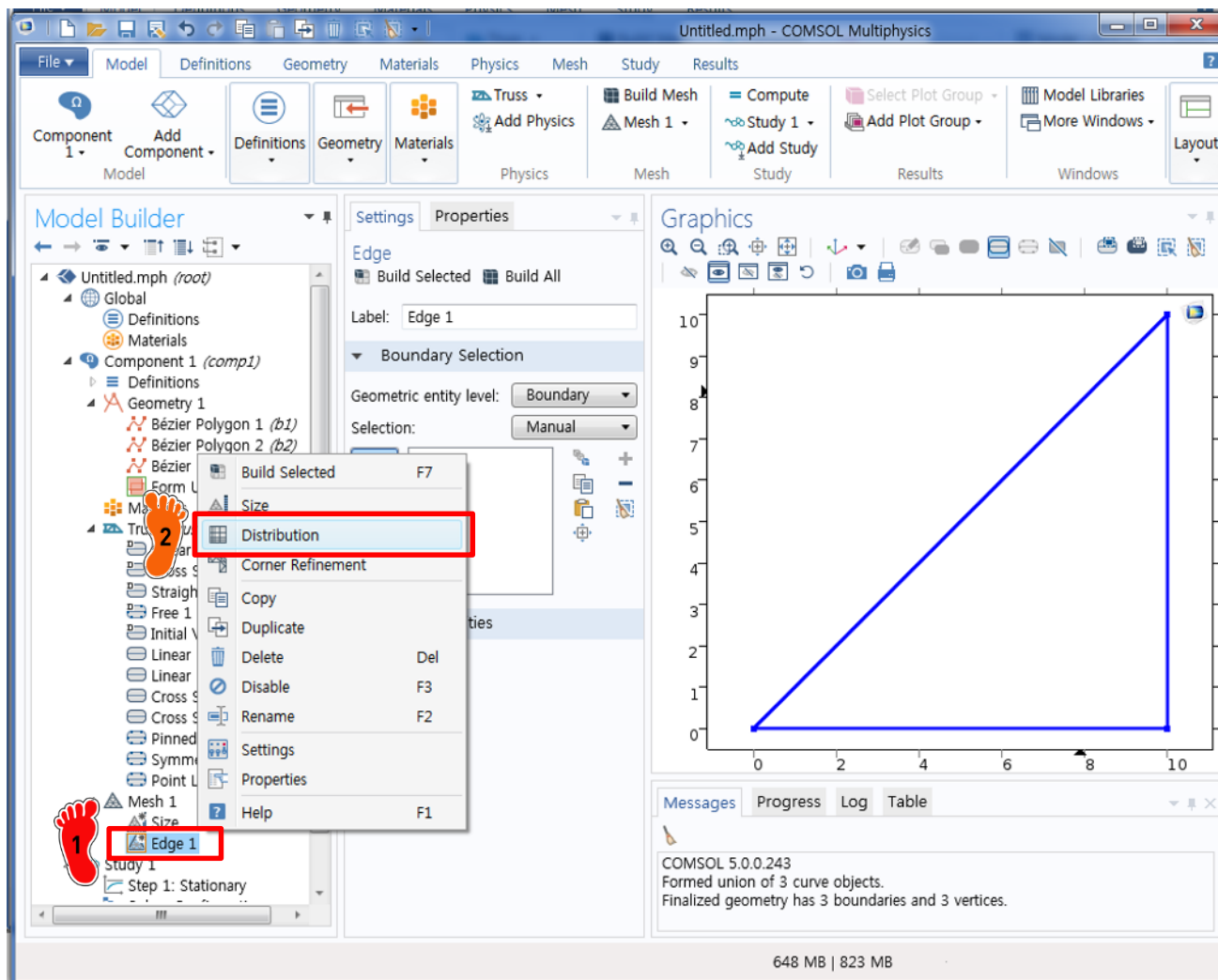
MESH CREATION



1 Edge 1 클릭

2 부재 세 개 선택

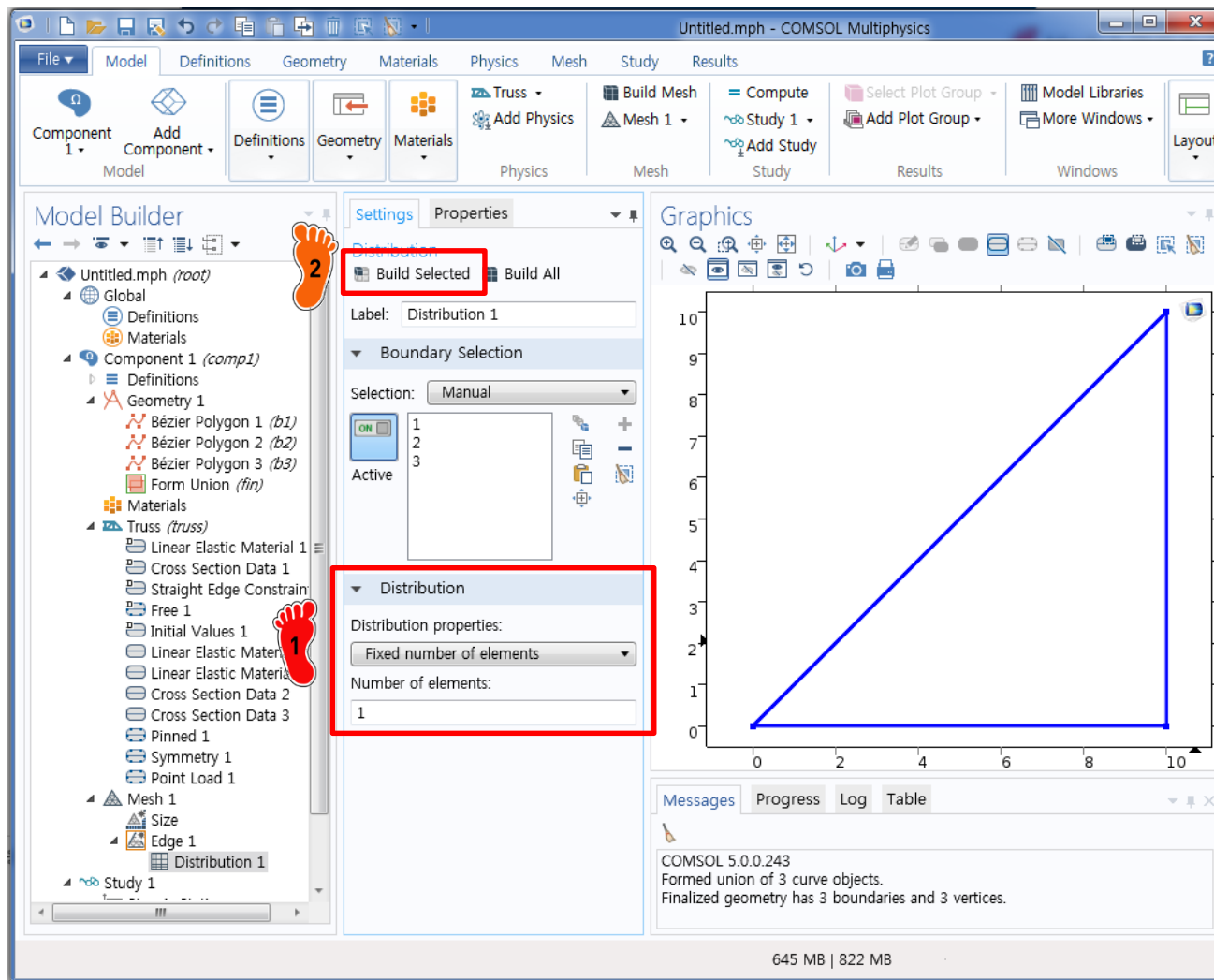
MESH CREATION



1 Edge 1 우클릭

2 Distribution 클릭

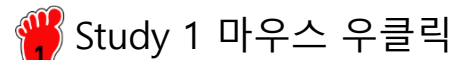
MESH CREATION



Number of elements 1
입력



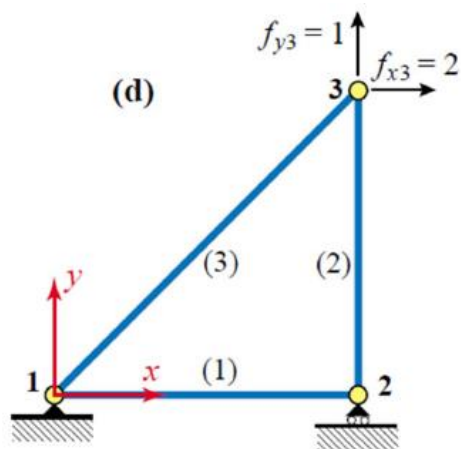
Build Selected 클릭



2 Compute 클릭

- **Stiffness matrix**
 - ✓ **Global stiffness matrix by hand**
 - ✓ **Global stiffness matrix by COMSOL**

ASSEMBLE OPTION



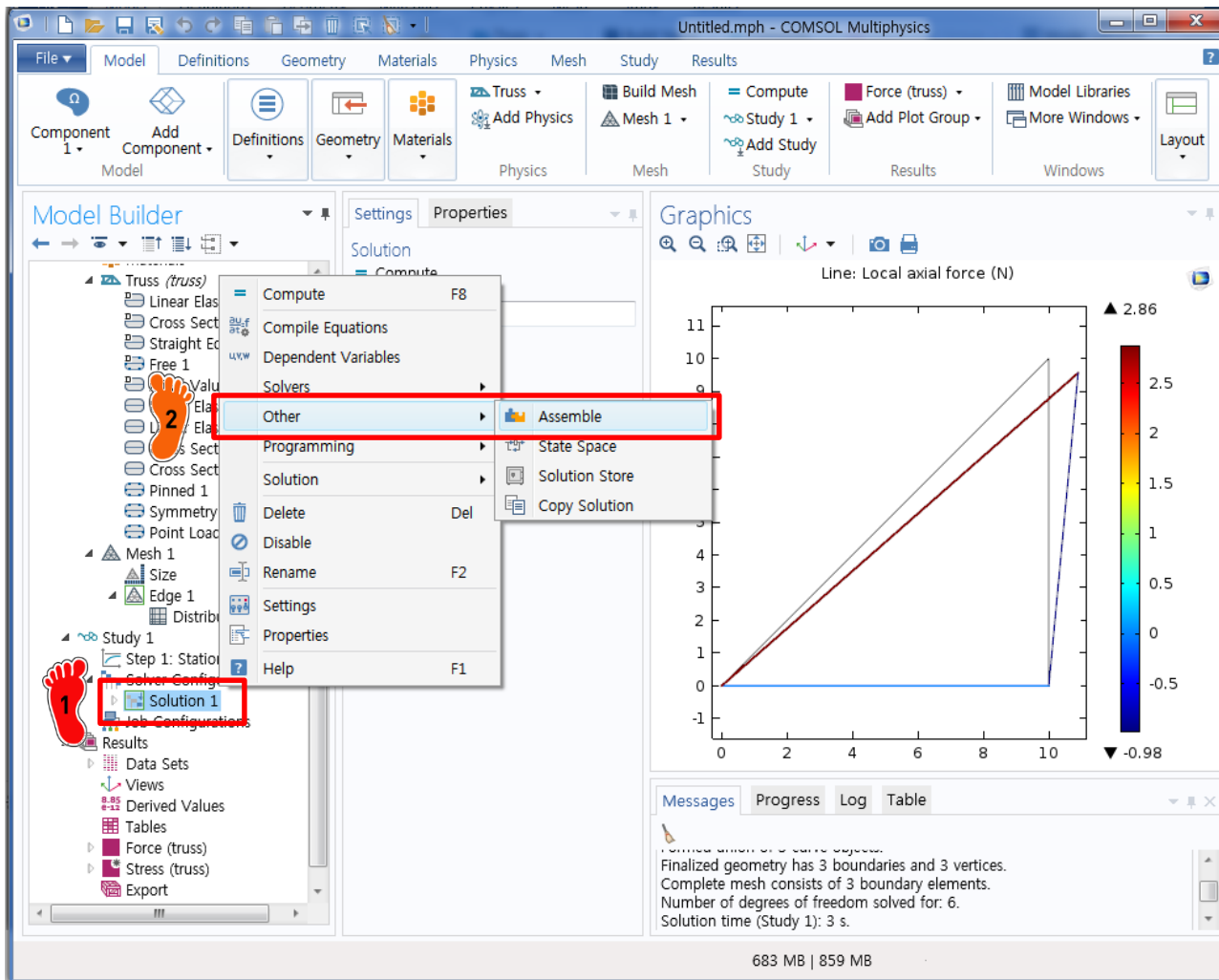
Support conditions and applied loads

$$\mathbf{f} = \mathbf{f}^{(1)} + \mathbf{f}^{(2)} + \mathbf{f}^{(3)} = \left(\mathbf{K}^{(1)} + \mathbf{K}^{(2)} + \mathbf{K}^{(3)} \right) \mathbf{u} = \mathbf{K} \mathbf{u}$$

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} 20 & 10 & -10 & 0 & -10 & -10 \\ 10 & 10 & 0 & 0 & -10 & -10 \\ -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 5 & 0 & -5 \\ -10 & -10 & 0 & 0 & 10 & 10 \\ -10 & -10 & 0 & -5 & 10 & 15 \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}$$

- **Stiffness matrix confirm**
 - ✓ **Global stiffness matrix by hand**
 - ✓ **Global stiffness matrix by COMSOL**

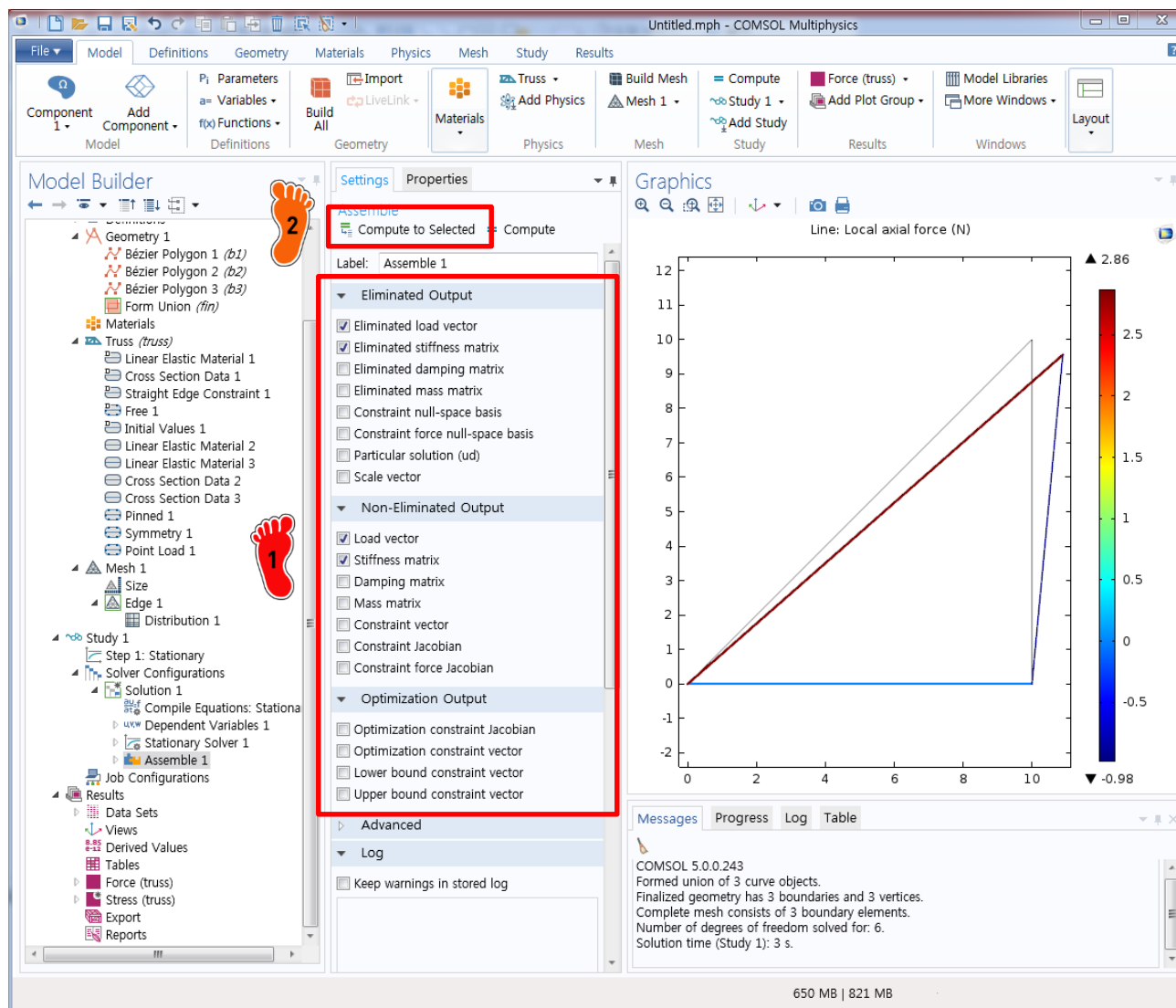
ASSEMBLE OPTION



1 Solver 1 마우스 우클릭

2 Other에서 Assemble 클릭

ASSEMBLE OPTION



Eliminated load vector
Eliminated stiffness matrix

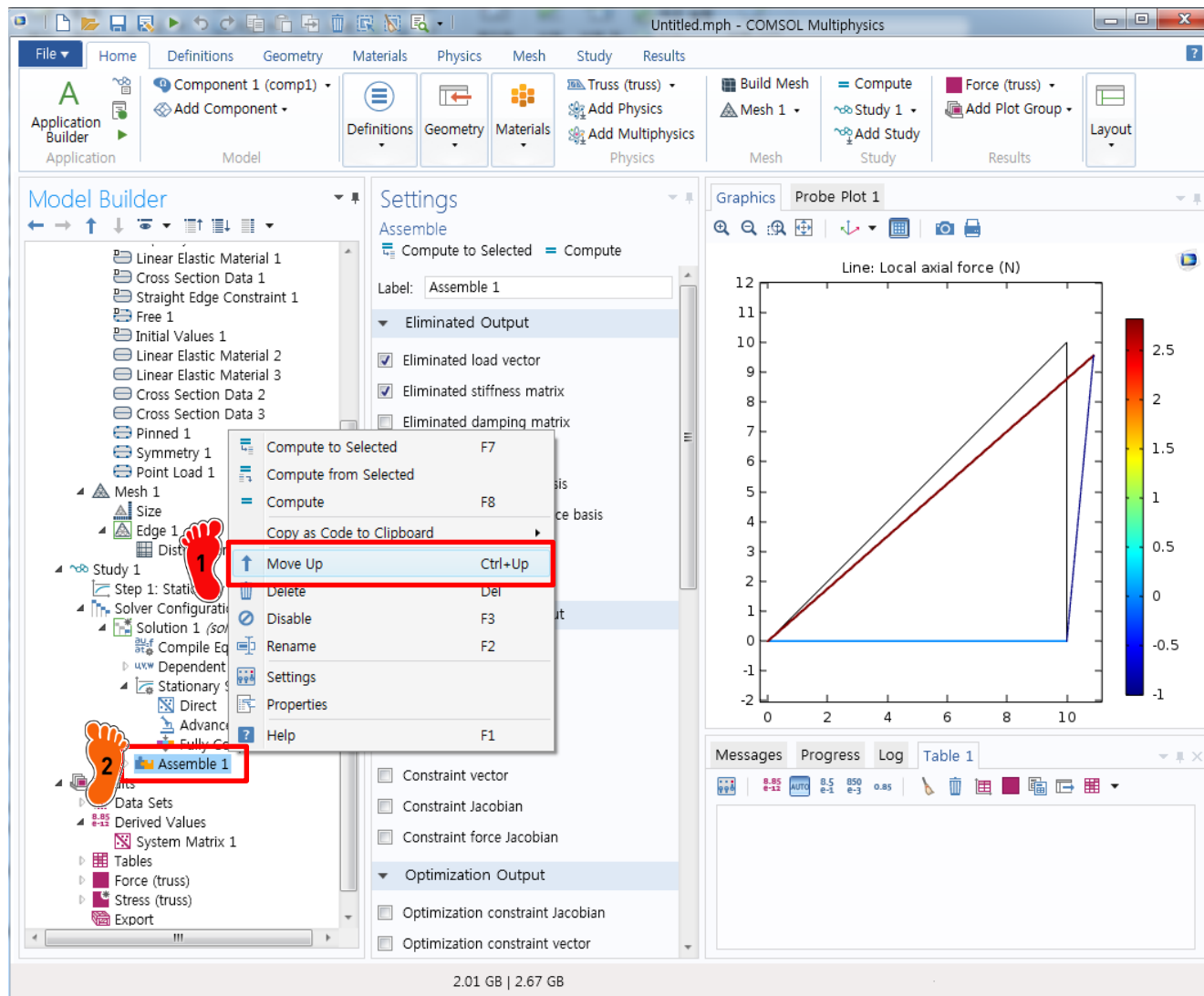
Load vector
Stiffness matrix

체크박스에 체크



Compute to Selected 클릭

ASSEMBLE OPTION



Assemble 1 마우스 우클릭

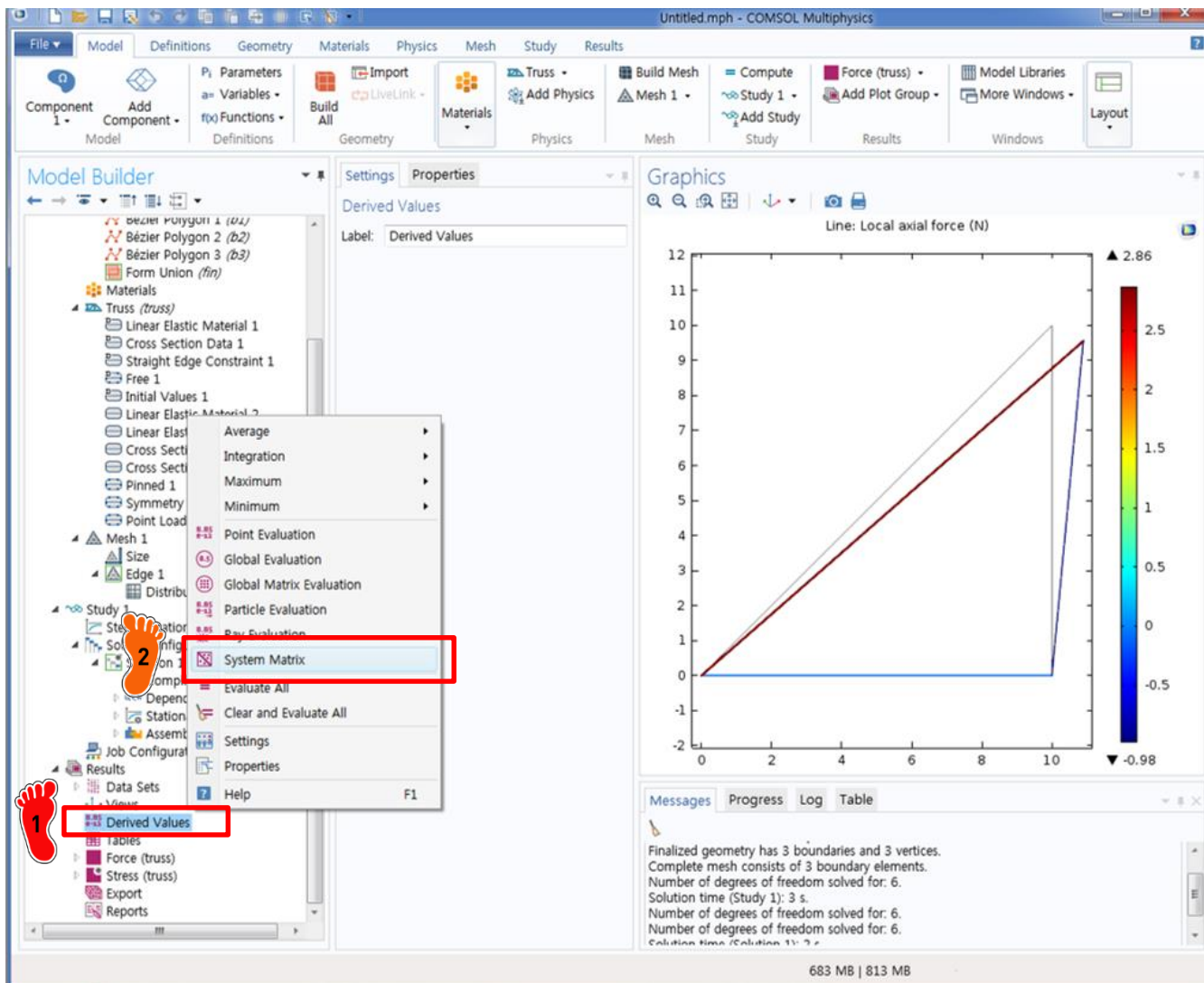


Move up을 클릭하여
Assemble 노드가
Solver 노드 위에 위치하도록 이동

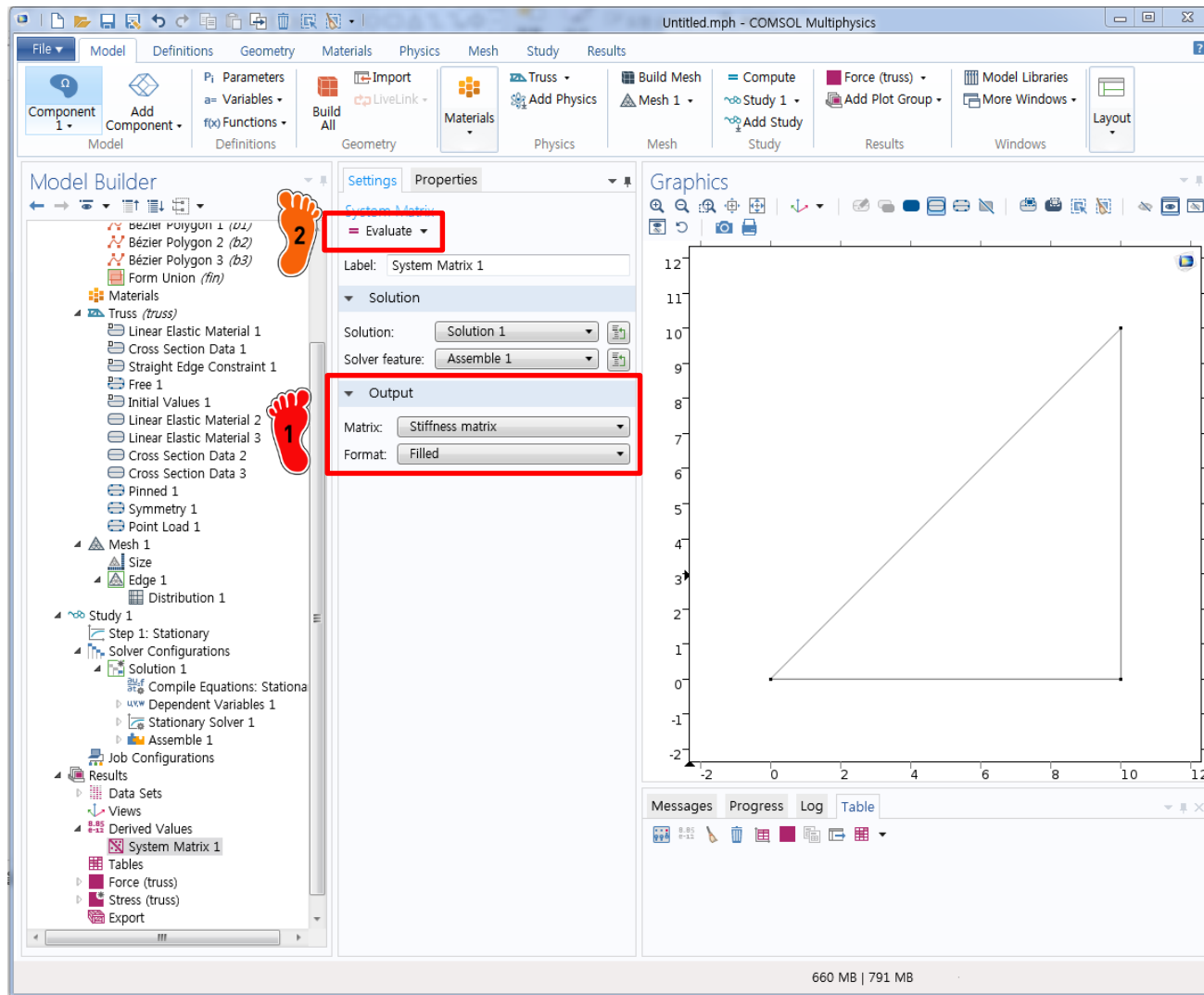
SYSTEM MATRIX

1 Derived Values 마우스 우클릭

2 System Matrix 클릭



SYSTEM MATRIX

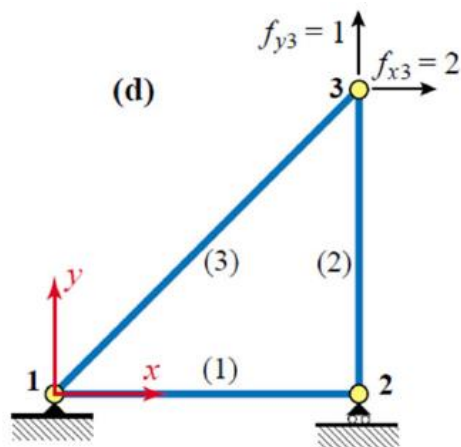


1 Matrix : Stiffness matrix
Format : Filled
선택

2 Evaluate 클릭

Load vector
Eliminated stiffness matrix
Eliminated load vector
메뉴 모두 Matrix 선택에서
조절 가능

COMPARISON



Support conditions and applied loads

$$\mathbf{f} = \mathbf{f}^{(1)} + \mathbf{f}^{(2)} + \mathbf{f}^{(3)} = \underbrace{(\mathbf{K}^{(1)} + \mathbf{K}^{(2)} + \mathbf{K}^{(3)})}_{\Downarrow} \mathbf{u} = \mathbf{K} \mathbf{u}$$

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} 20 & 10 & -10 & 0 & -10 & -10 \\ 10 & 10 & 0 & 0 & -10 & -10 \\ -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 5 & 0 & -5 \\ -10 & -10 & 0 & 0 & 10 & 10 \\ -10 & -10 & 0 & -5 & 10 & 15 \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}$$

COMSOL Result

Load Vector

Stiffness matrix

0	20.0000	10.0000	-10.0000	0.0000	-10.0000	-10.0000
0	10.0000	10.0000	0.0000	0.0000	-10.0000	-10.0000
0	-10.0000	0.0000	10.0000	0.0000	0.0000	0.0000
0	0.0000	0.0000	0.0000	5.0000	0.0000	-5.0000
2	-10.0000	-10.0000	0.0000	0.0000	10.0000	10.0000
1	-10.0000	-10.0000	0.0000	-5.0000	10.0000	15.0000

LOCAL STIFFNESS MATRIX

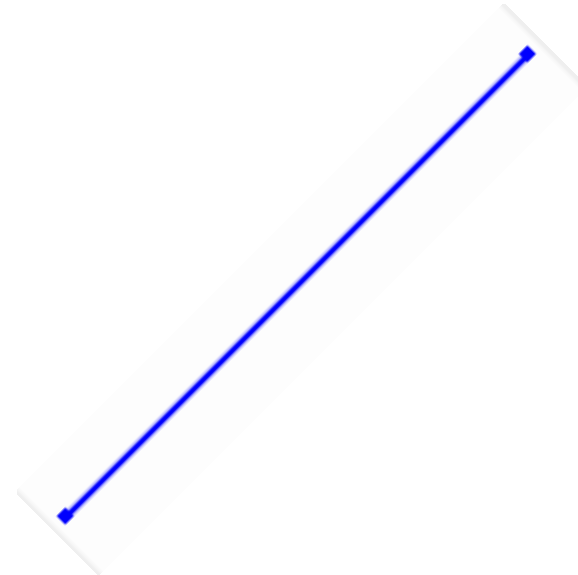
Stiffness matrix by Hand

$$\begin{bmatrix} f_{x1}^{(1)} \\ f_{y1}^{(1)} \\ f_{x2}^{(1)} \\ f_{y2}^{(1)} \end{bmatrix} = 10 \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_{x1}^{(1)} \\ u_{y1}^{(1)} \\ u_{x2}^{(1)} \\ u_{y2}^{(1)} \end{bmatrix}$$

$$\begin{bmatrix} f_{x2}^{(2)} \\ f_{y2}^{(2)} \\ f_{x3}^{(2)} \\ f_{y3}^{(2)} \end{bmatrix} = 5 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_{x2}^{(2)} \\ u_{y2}^{(2)} \\ u_{x3}^{(2)} \\ u_{y3}^{(2)} \end{bmatrix}$$

$$\begin{bmatrix} f_{x1}^{(3)} \\ f_{y1}^{(3)} \\ f_{x3}^{(3)} \\ f_{y3}^{(3)} \end{bmatrix} = 20 \begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix} \begin{bmatrix} u_{x1}^{(3)} \\ u_{y1}^{(3)} \\ u_{x3}^{(3)} \\ u_{y3}^{(3)} \end{bmatrix}$$

Geometry



Stiffness matrix by COMSOL

10.0000	0.0000	-10.0000	0.0000
0.0000	0.0000	0.0000	0.0000
-10.0000	0.0000	10.0000	0.0000
0.0000	0.0000	0.0000	0.0000

0.0000	0.0000	0.0000	0.0000
0.0000	5.0000	0.0000	-5.0000
0.0000	0.0000	0.0000	0.0000
0.0000	-5.0000	0.0000	5.0000

10.0000	10.0000	-10.0000	-10.0000
10.0000	10.0000	-10.0000	-10.0000
-10.0000	-10.0000	10.0000	10.0000
-10.0000	-10.0000	10.0000	10.0000

REDUCED STIFFNESS MATRIX

$$\begin{cases} \text{Displacement BCs: } u_{x1} = u_{y1} = u_{y2} = 0 \\ \text{Force BCs: } f_{x2} = 0, f_{x3} = 2, f_{y3} = 1 \end{cases}$$

$$\rightarrow \begin{bmatrix} 20 & 10 & -10 & 0 & -10 & -10 \\ 10 & 10 & 0 & 0 & -10 & -10 \\ -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 5 & 0 & -5 \\ -10 & -10 & 0 & 0 & 10 & 10 \\ -10 & -10 & 0 & -5 & 10 & 15 \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix}$$

Strike out rows and columns pertaining to known displacements:

$$\begin{bmatrix} 10 & 0 & 0 \\ 0 & 10 & 10 \\ 0 & 10 & 15 \end{bmatrix} \begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} f_{x2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ 2 \\ 1 \end{bmatrix} \Leftrightarrow \hat{\mathbf{K}}\hat{\mathbf{u}} = \hat{\mathbf{f}}$$

Solve by Gauss elimination for unknown node displacements

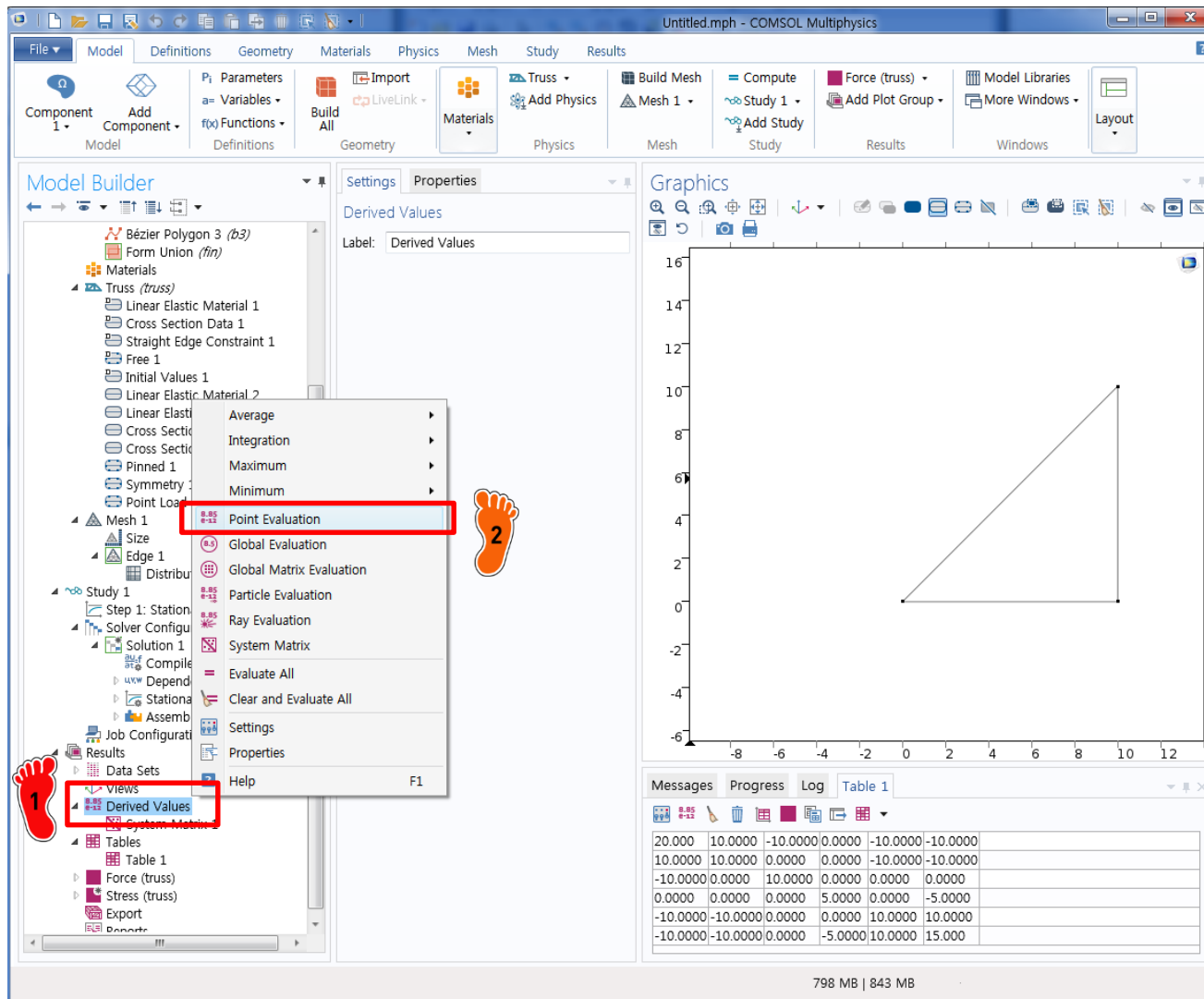
Eliminated Stiffness matrix

10.000	0.0000	0.0000
0.0000	10.000	10.000
0.0000	10.000	15.000

Eliminated Load vector

0
2
1

NODAL DISPLACEMENT



1 Derived Values 마우스 우클릭

2 Point Evaluation 클릭

NODAL DISPLACEMENT

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

- Model Builder:** Contains a hierarchy of objects including Bézier Polygon 3 (b3), Form Union (fin), Materials, Truss (truss), Mesh 1, and Study 1.
- Settings:** The 'Point Evaluation' settings are visible. The 'Data' section shows 'Study 1/Solution 1' as the data set. The 'Expression' section shows 'truss.disp' as the expression to be evaluated.
- Graphics:** A plot area showing a truss structure with a displacement field.
- Expression Selection:** A red box highlights the 'Expression' field, and a red arrow points to the 'truss.disp' expression in the 'Data' section.
- Selection Dialog:** A dialog box is open, showing a tree view of the model hierarchy. The 'Truss' object is selected, and the 'Displacement' field is highlighted. The 'truss.disp' expression is listed under 'Displacement field'.

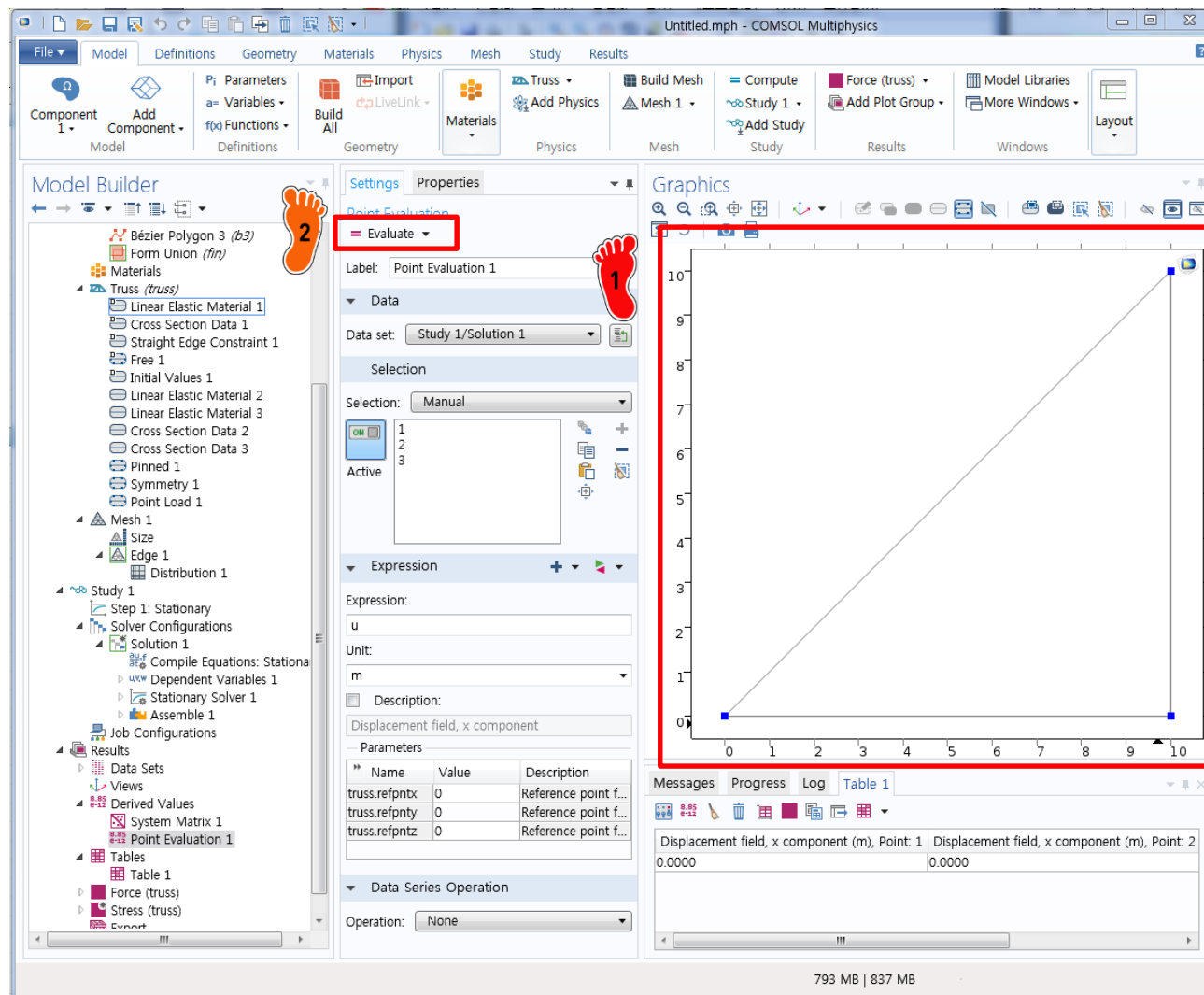
Annotations:

- 1** (Red footprint): Points to the 'Expression' field in the 'Settings' panel.
- 2** (Orange footprint): Points to the 'truss.disp' expression in the 'Data' section.

1 Replace expression

2 원하는 출력 값 선택

NODAL DISPLACEMENT



Point 선택



Evaluate 클릭

NODAL DISPLACEMENT

Strike out rows and columns pertaining to known displacements:

$$\begin{bmatrix} 10 & 0 & 0 \\ 0 & 10 & 10 \\ 0 & 10 & 15 \end{bmatrix} \begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} f_{x2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ 2 \\ 1 \end{bmatrix} \Leftrightarrow \hat{\mathbf{K}}\hat{\mathbf{u}} = \hat{\mathbf{f}}$$

Solve by Gauss elimination for unknown node displacements

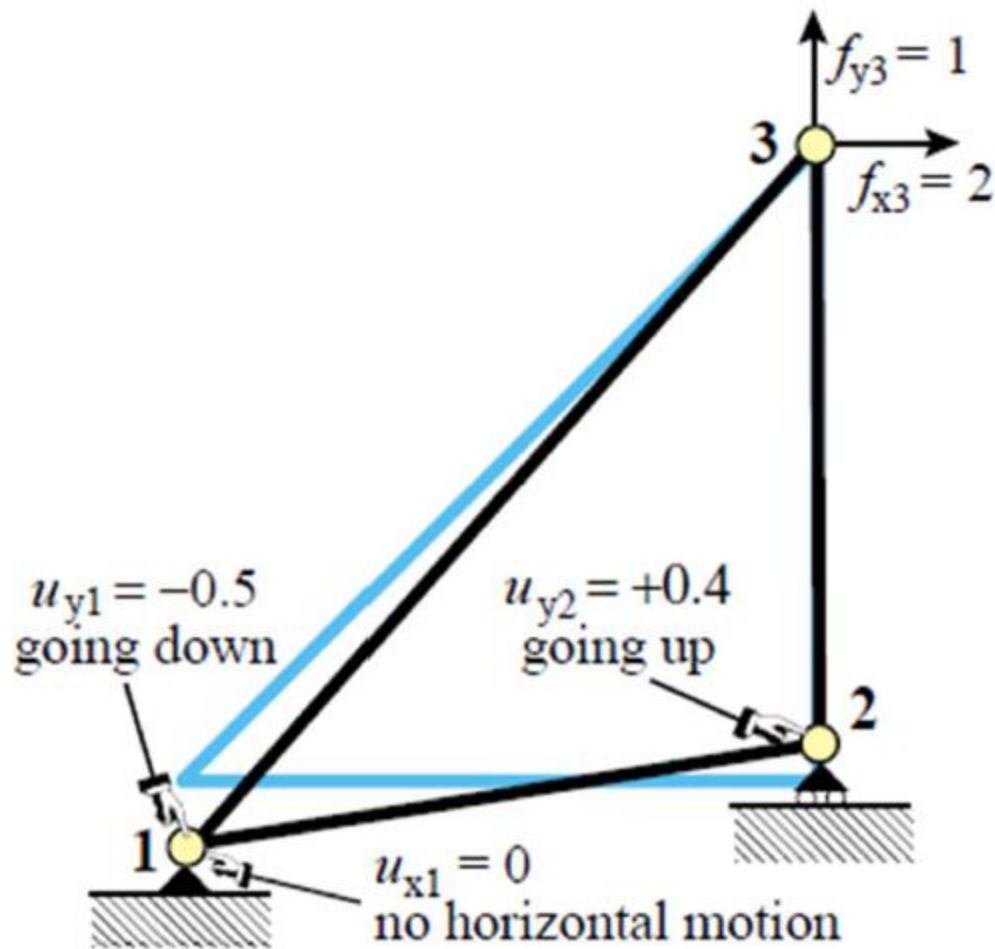
$$\begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ 0.4 \\ -0.2 \end{bmatrix} \xrightarrow{\text{expand with known displacement BCs}} \mathbf{u} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0.4 \\ -0.2 \end{bmatrix}$$

Nodal displacement by COMSOL

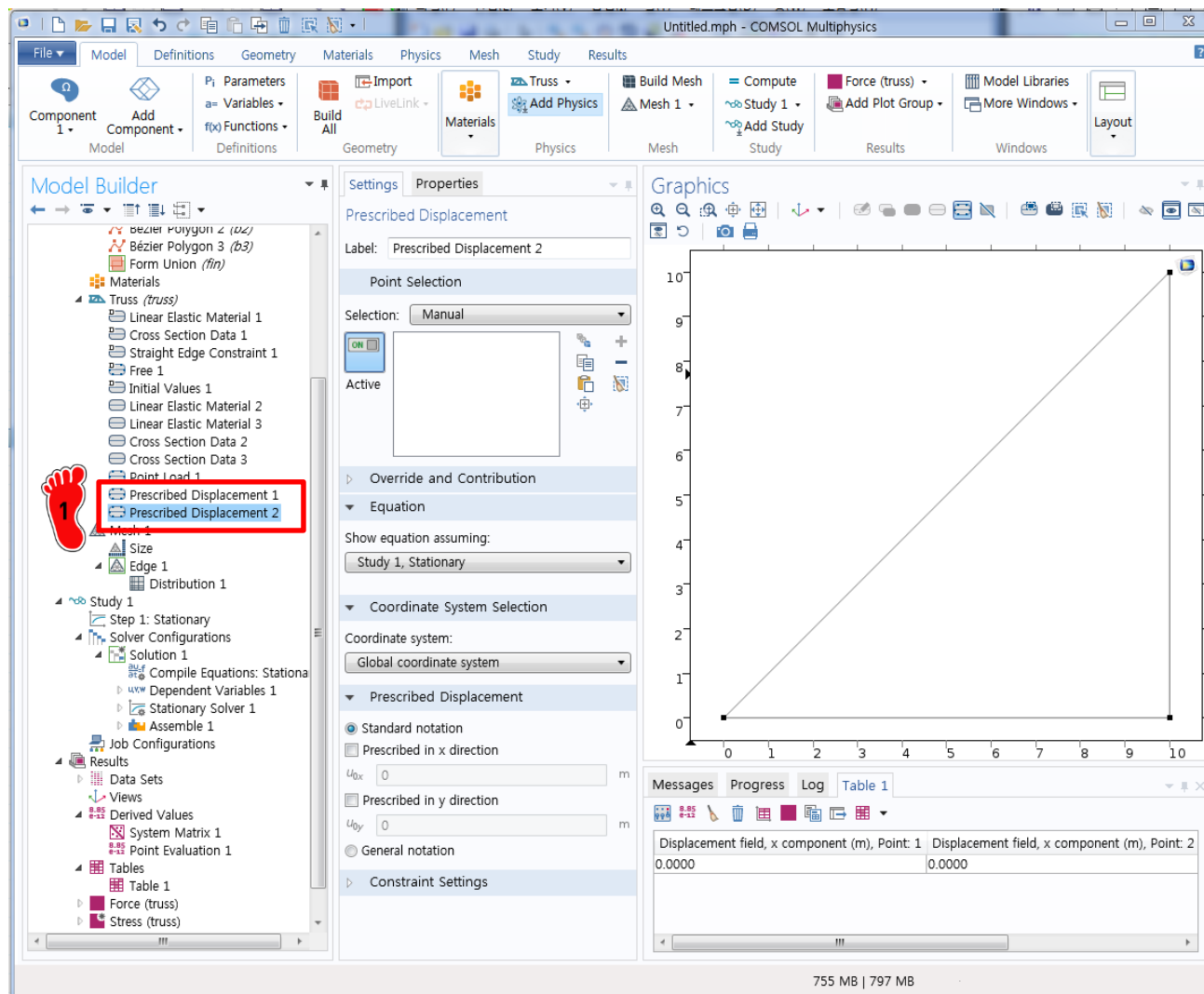
Displacement field, x component (m), Point: 1	Displacement field, x component (m), Point: 2	Displacement field, x component (m), Point: 3
0.0000	0.0000	0.40000

Displacement field, y component (m), Point: 1	Displacement field, y component (m), Point: 2	Displacement field, y component (m), Point: 3
0.0000	0.0000	-0.20000

EXAMPLE TRUSS STRUCTURE

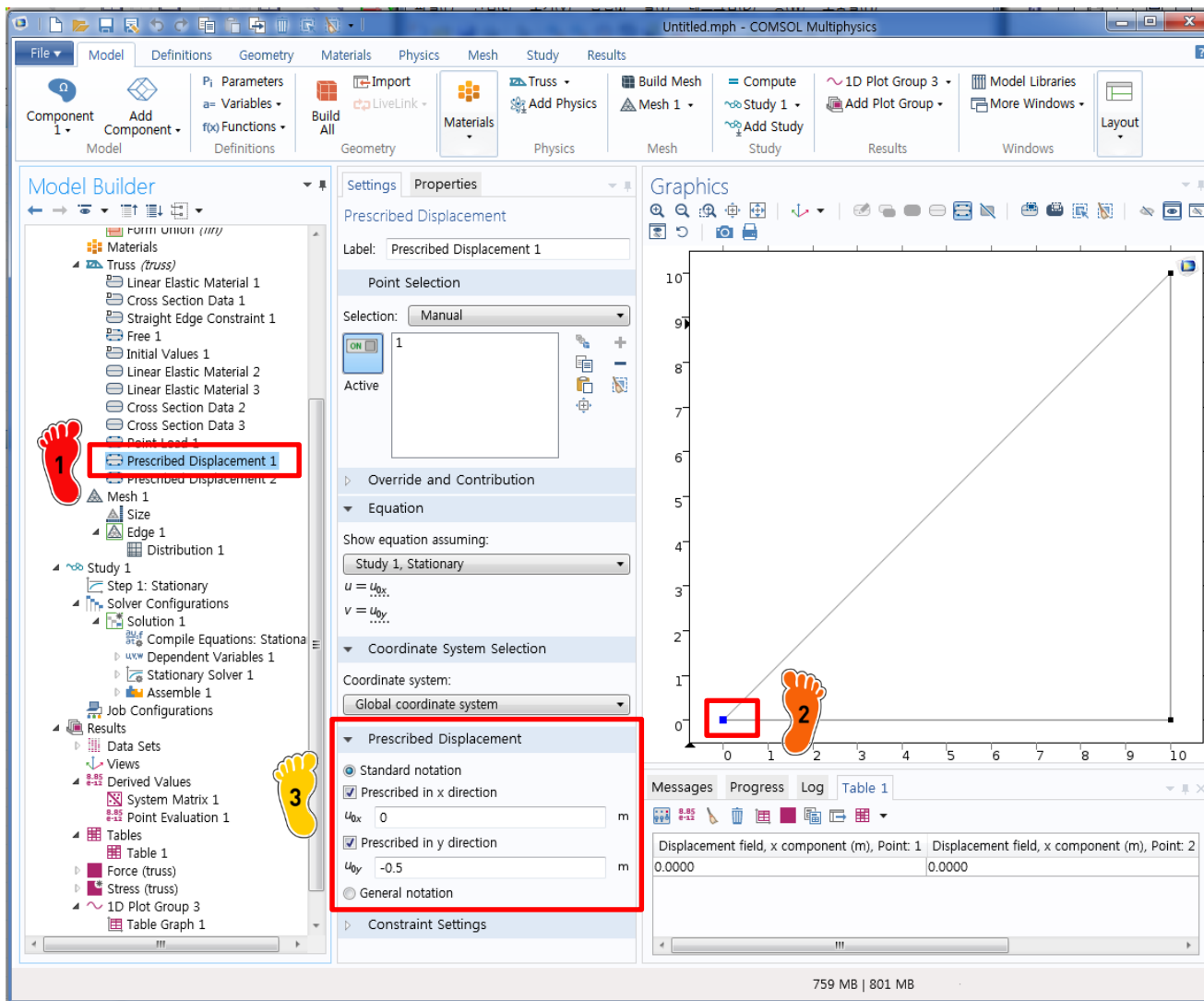


BOUNDARY CONDITION



1 기존 경계조건인 pinned 와 symmetry 조건 삭제 후 Prescribed Displacement 경계조건 2 개 생성

BOUNDARY CONDITION



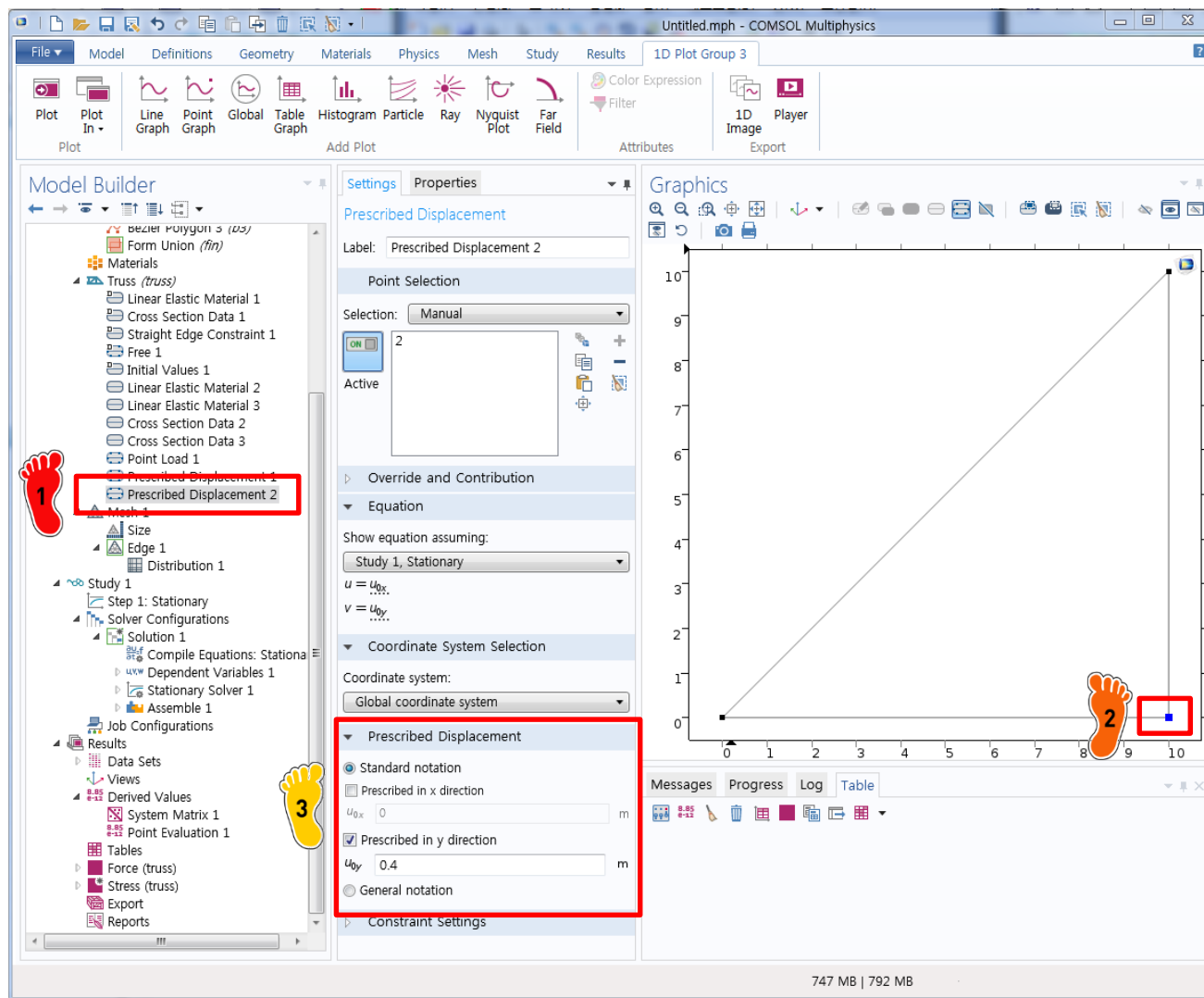
1 첫 번째 Prescribed Displacement 클릭

2 왼쪽 아래 절점 선택

3 $u_{0x} : 0$
 $u_{0y} : -0.5$

입력

BOUNDARY CONDITION



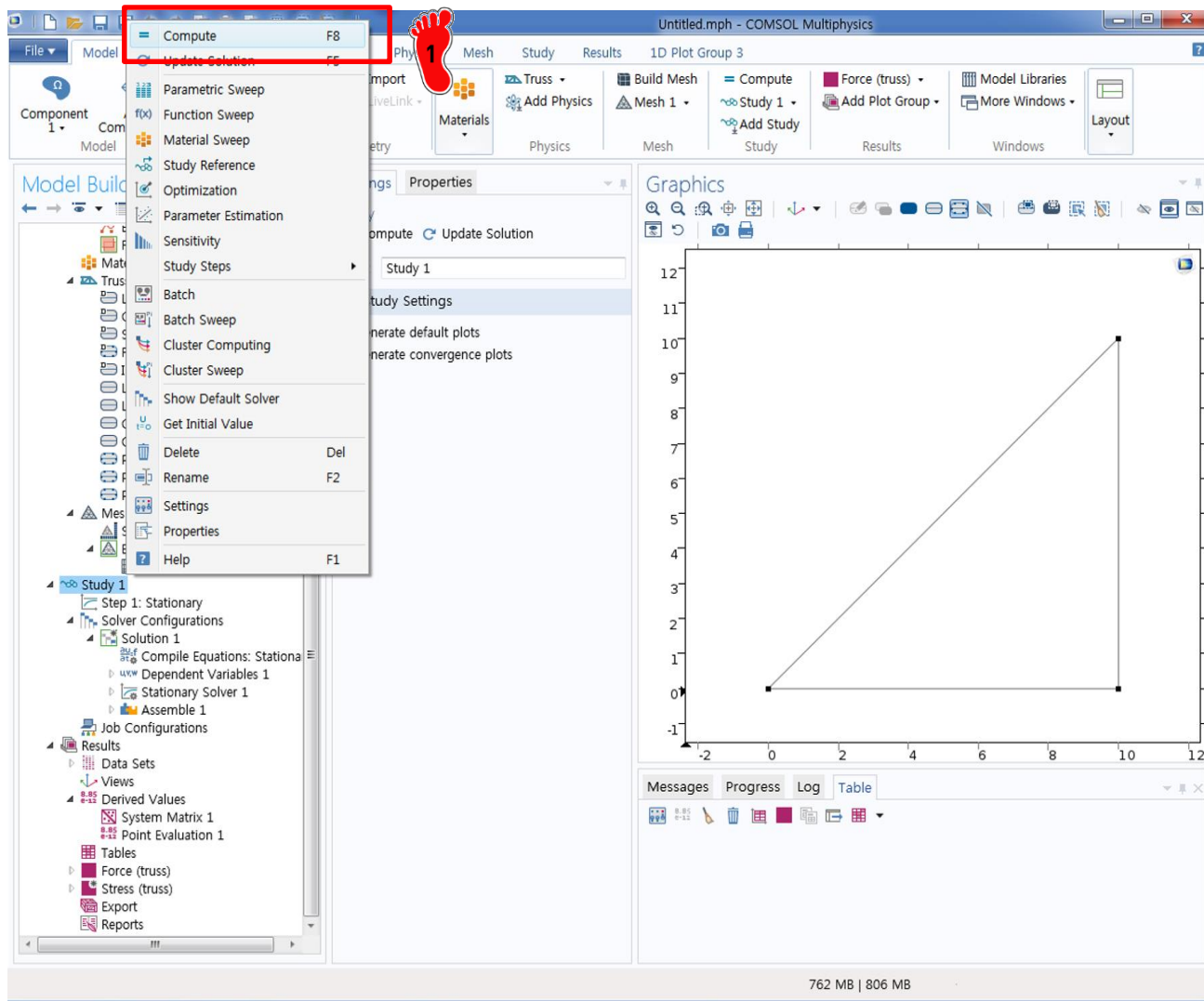
1 두 번째 Prescribed Displacement 클릭

2 오른쪽 아래 절점 선택

3 u_{0x} : none
 u_{0y} : 0.4

입력

COMPUTE



1 Compute 클릭

RESULT

Transfer effect of known displacements to RHS, and delete columns:

$$\begin{bmatrix} 10 & 0 & 0 \\ 0 & 10 & 10 \\ 0 & 10 & 15 \end{bmatrix} \begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ 2 \\ 1 \end{bmatrix} - \begin{bmatrix} (-10) \times 0 + 0 \times (-0.5) + 0 \times 0.4 \\ (-10) \times 0 + (-10) \times (-0.5) + 0 \times 0.4 \\ (-10) \times 0 + (-10) \times (-0.5) + (-5) \times 0.4 \end{bmatrix} = \begin{bmatrix} 0 \\ -3 \\ -2 \end{bmatrix}$$

Solving gives

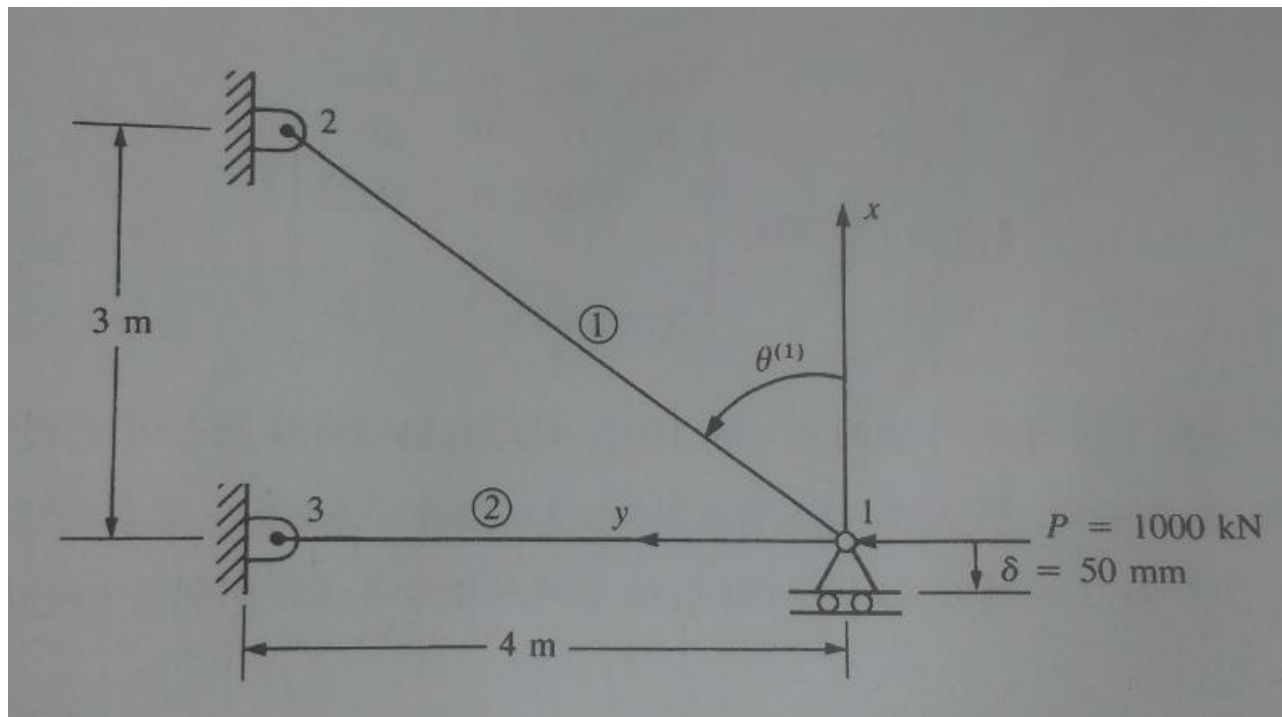
$$\begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ -0.5 \\ 0.2 \end{bmatrix} \xrightarrow{\text{Complete the displacement vector with known values}} \mathbf{u} = \begin{bmatrix} 0 \\ -0.5 \\ 0 \\ 0.4 \\ -0.5 \\ 0.2 \end{bmatrix}$$

In summary, the only changes to the SDM is in the application of displacement boundary conditions before solve

Nodal displacement by COMSOL

Displacement field, x component (m), Point: 1	Displacement field, x component (m), Point: 2	Displacement field, x component (m), Point: 3
0.0000	0.0000	-0.50000
Displacement field, y component (m), Point: 1	Displacement field, y component (m), Point: 2	Displacement field, y component (m), Point: 3
-0.50000	0.40000	0.20000

ASSIGNMENT



$$E = 210 \text{ GPa}$$

$$A = 6 \times 10^{-4} \text{ m}^2$$

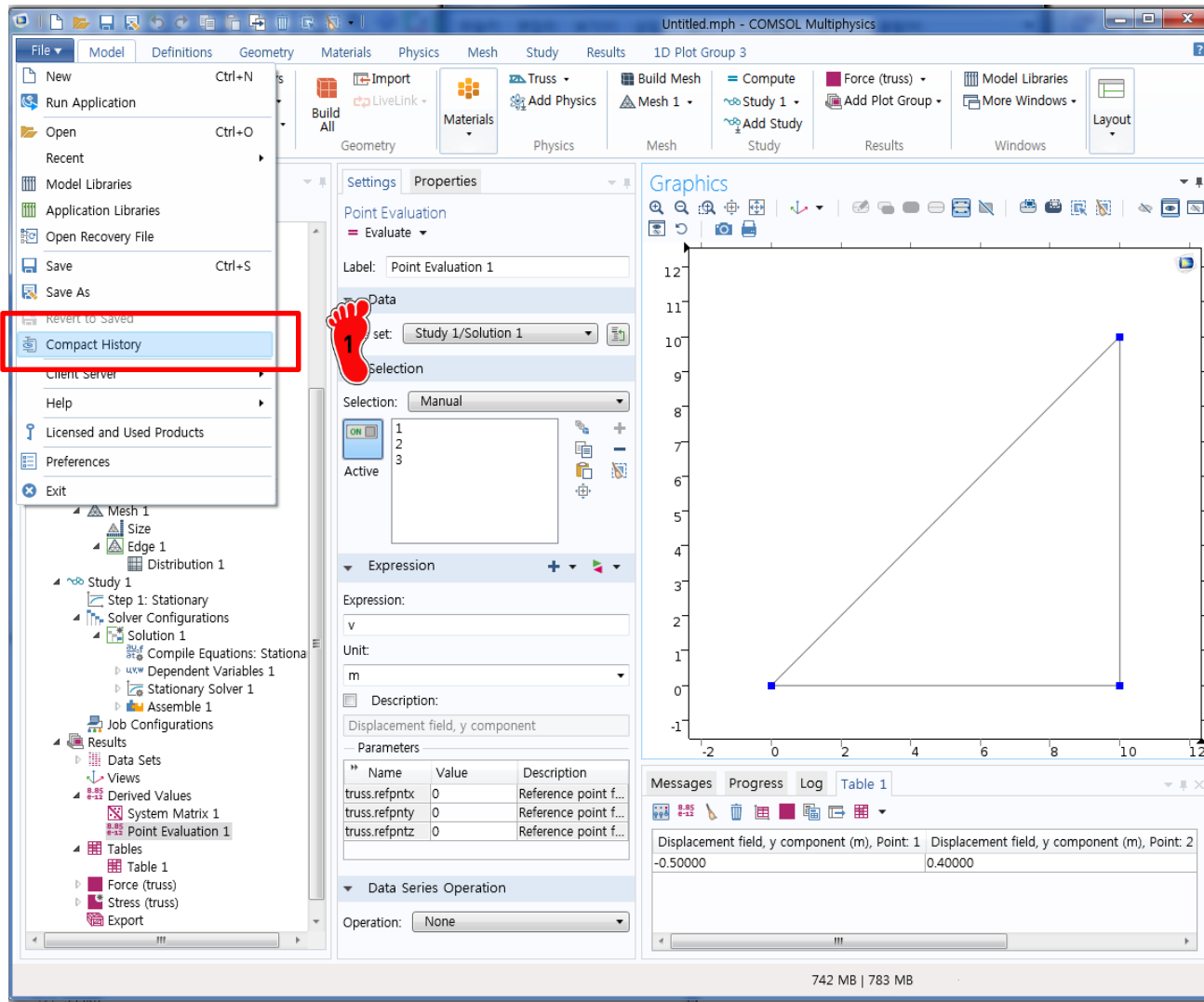
$$\therefore d_{1y} = 0.0337 \text{ m}$$

- ① derive K matrix
- ② calculate y-displacement at node 1
- ③ compare with COMSOL result

CONTENTS

- **Livelihood with MATLAB**

RESET HISTORY

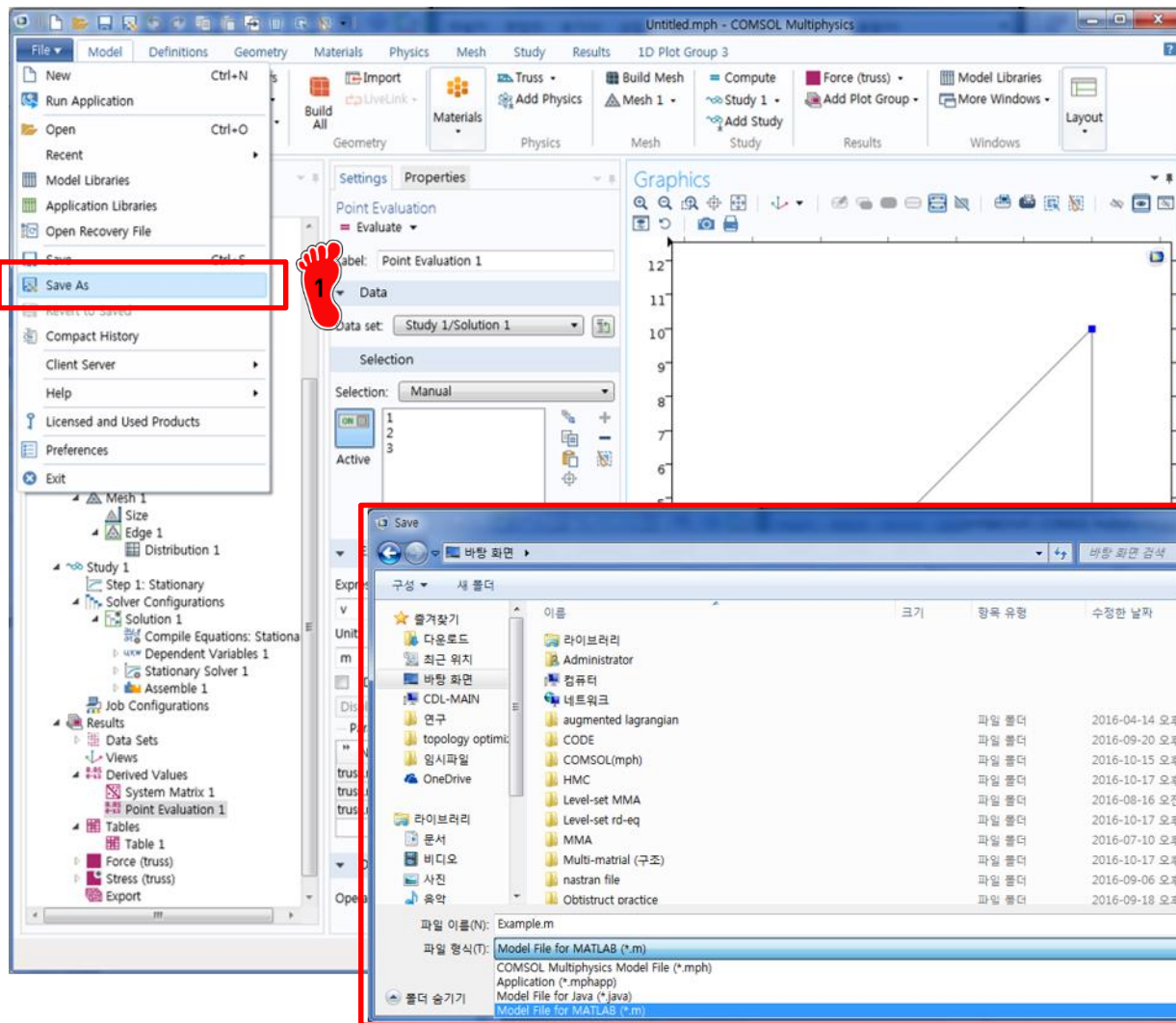


Reset History 클릭

COMSOL에서 m-file 로 저장할 때 작업 history 모두 저장됨

따라서 불필요한 history 를 삭제하기 위하여 Compact history 기능을 이용

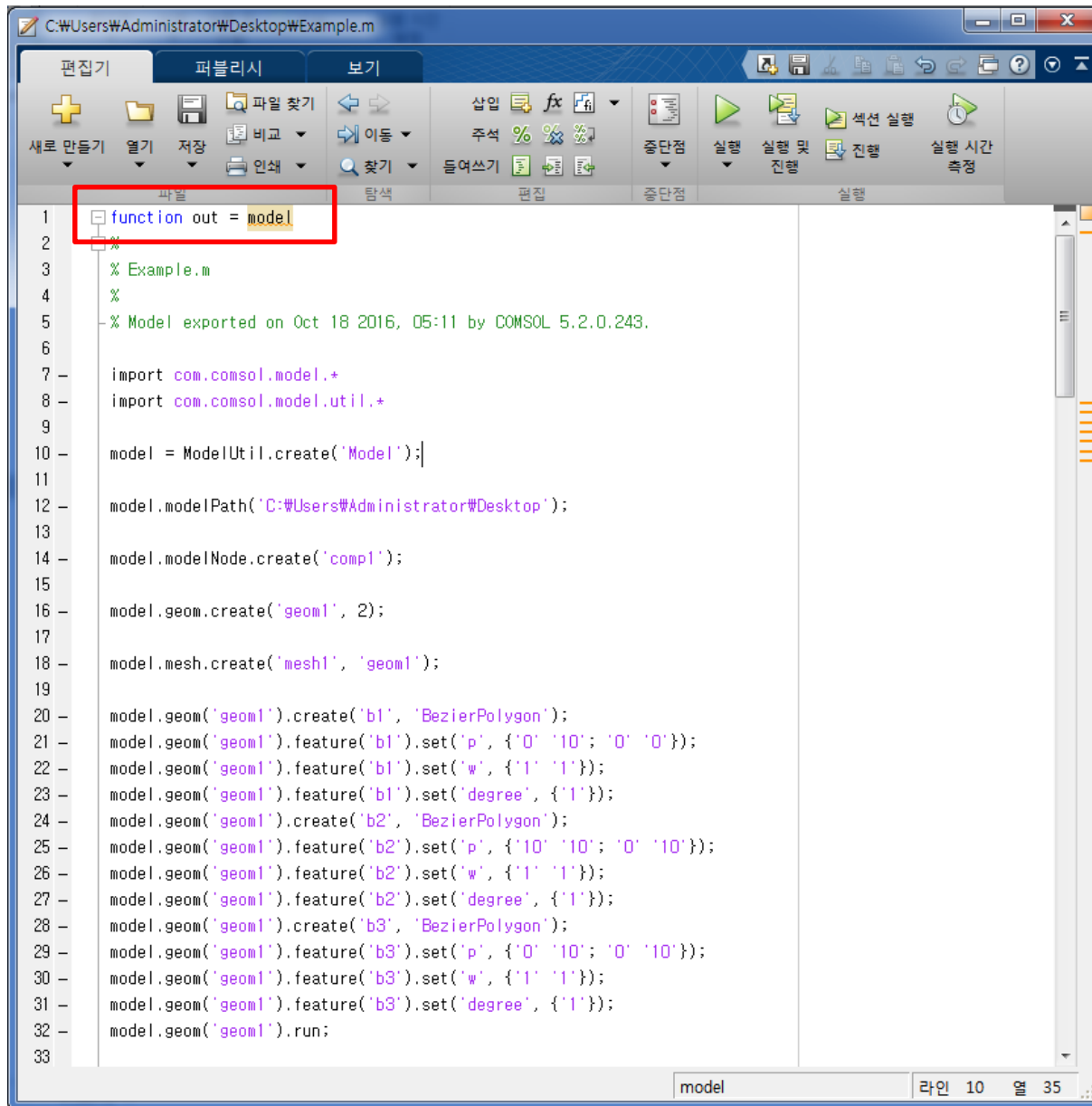
SAVE AS M-FILE



1 Save As 클릭

2 m-file 로 저장

COMSOL M-FILE STRUCTURE



```

1 function out = model
2 %
3 % Example.m
4 %
5 % Model exported on Oct 18 2016, 05:11 by COMSOL 5.2.0.243.
6
7 import com.comsol.model.*
8 import com.comsol.model.util.*
9
10 model = ModelUtil.create('Model');
11
12 model.modelPath('C:\Users\Administrator\Desktop');
13
14 model.modelNode.create('comp1');
15
16 model.geom.create('geom1', 2);
17
18 model.mesh.create('mesh1', 'geom1');
19
20 model.geom('geom1').create('b1', 'BezierPolygon');
21 model.geom('geom1').feature('b1').set('p', {'0' '10'; '0' '0'});
22 model.geom('geom1').feature('b1').set('w', {'1' '1'});
23 model.geom('geom1').feature('b1').set('degree', {'1'});
24 model.geom('geom1').create('b2', 'BezierPolygon');
25 model.geom('geom1').feature('b2').set('p', {'10' '10'; '0' '10'});
26 model.geom('geom1').feature('b2').set('w', {'1' '1'});
27 model.geom('geom1').feature('b2').set('degree', {'1'});
28 model.geom('geom1').create('b3', 'BezierPolygon');
29 model.geom('geom1').feature('b3').set('p', {'0' '10'; '0' '10'});
30 model.geom('geom1').feature('b3').set('w', {'1' '1'});
31 model.geom('geom1').feature('b3').set('degree', {'1'});
32 model.geom('geom1').run;
33
  
```

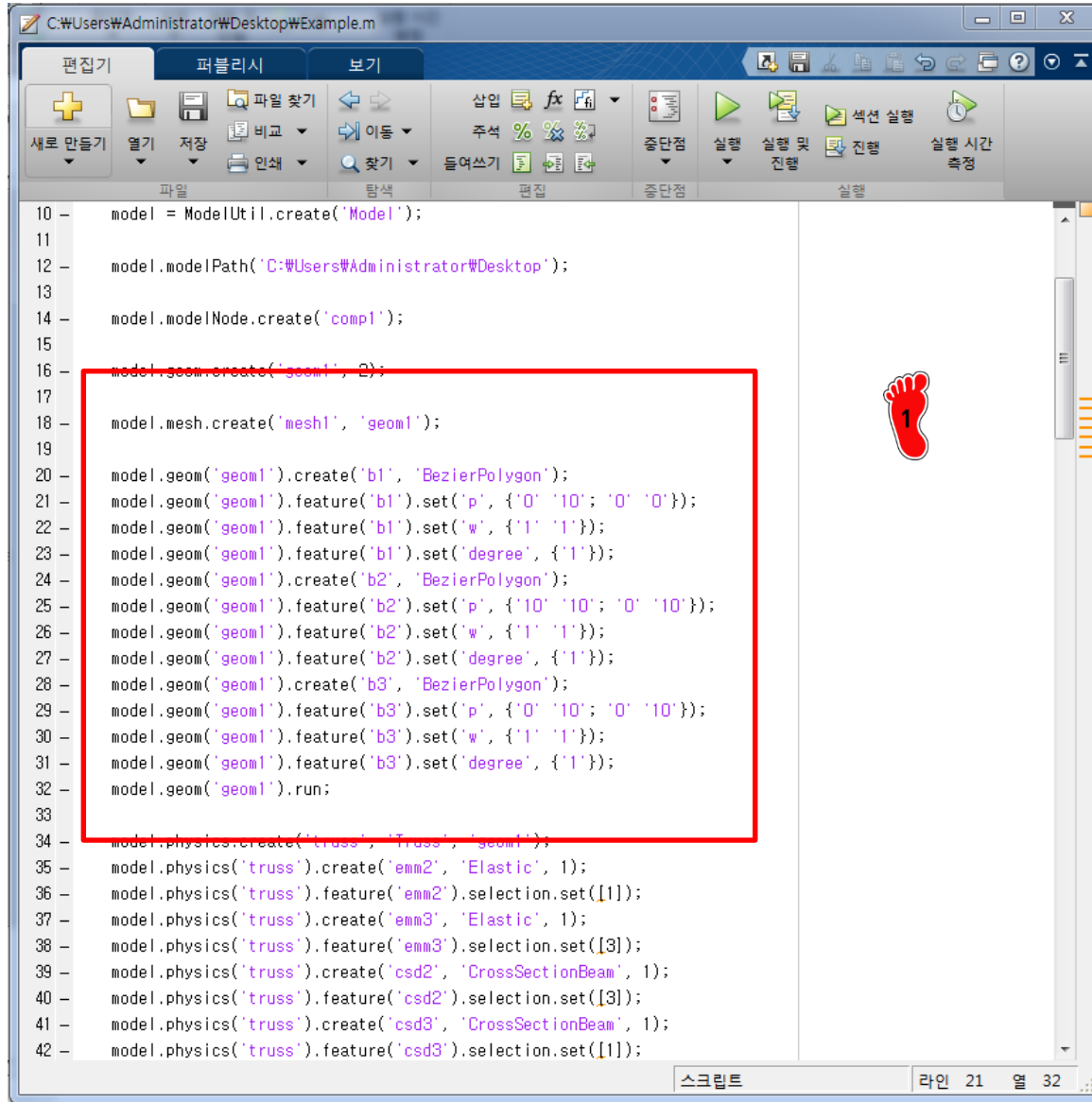


function 부분을 삭제 후 m-file 을 실행하면 COMSOL desktop 결과와 동일한 결과를 출력

단, COMSOL 5.2a with MATLAB 아이콘을 실행시켜 나온 MATLAB 창을 이용해야함



COMSOL M-FILE STRUCTURE



```

10 - model = ModelUtil.create('Model');
11
12 - model.modelPath('C:\Users\Administrator\Desktop\Example.m');
13
14 - model.modelNode.create('comp1');
15
16 - model.geom.create('geom1', 2);
17
18 - model.mesh.create('mesh1', 'geom1');
19
20 - model.geom('geom1').create('b1', 'BezierPolygon');
21 - model.geom('geom1').feature('b1').set('p', {'0' '10'; '0' '0'});
22 - model.geom('geom1').feature('b1').set('w', {'1' '1'});
23 - model.geom('geom1').feature('b1').set('degree', {'1'});
24 - model.geom('geom1').create('b2', 'BezierPolygon');
25 - model.geom('geom1').feature('b2').set('p', {'10' '10'; '0' '10'});
26 - model.geom('geom1').feature('b2').set('w', {'1' '1'});
27 - model.geom('geom1').feature('b2').set('degree', {'1'});
28 - model.geom('geom1').create('b3', 'BezierPolygon');
29 - model.geom('geom1').feature('b3').set('p', {'0' '10'; '0' '10'});
30 - model.geom('geom1').feature('b3').set('w', {'1' '1'});
31 - model.geom('geom1').feature('b3').set('degree', {'1'});
32 - model.geom('geom1').run;
33
34 - model.physics.create('truss', 'Truss', 'geom1');
35 - model.physics('truss').create('emm2', 'Elastic', 1);
36 - model.physics('truss').feature('emm2').selection.set([1]);
37 - model.physics('truss').create('emm3', 'Elastic', 1);
38 - model.physics('truss').feature('emm3').selection.set([3]);
39 - model.physics('truss').create('csd2', 'CrossSectionBeam', 1);
40 - model.physics('truss').feature('csd2').selection.set([3]);
41 - model.physics('truss').create('csd3', 'CrossSectionBeam', 1);
42 - model.physics('truss').feature('csd3').selection.set([1]);
  
```



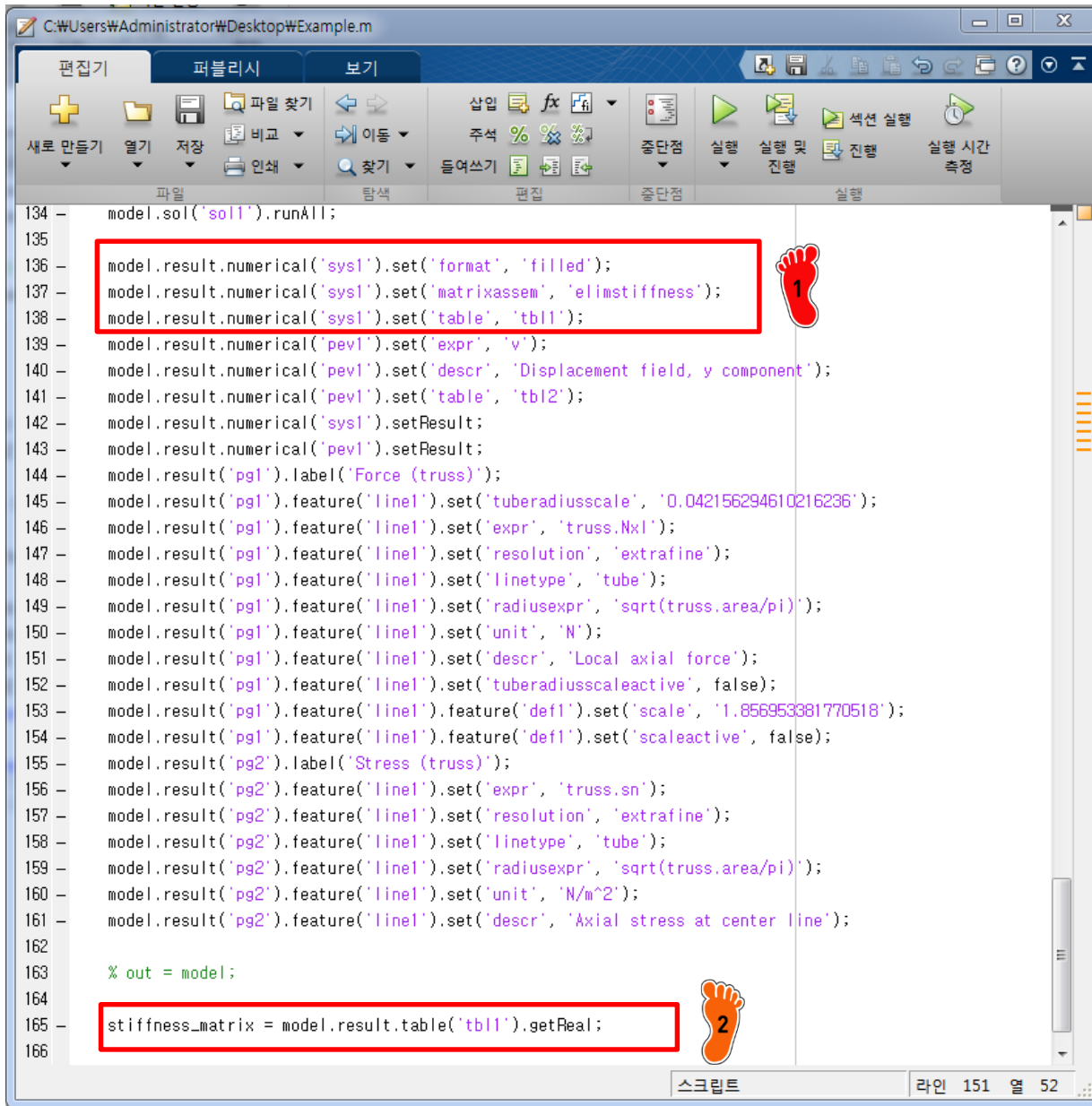
model.geom 부분

COMSOL 작업창에서 생성한 geometry 정보

그 외
model.physics
model.mesh
model.study
model.sol
model.result

등 model 뒤에 붙어있는 구조로 어떤 정보를 저장하고 있는지 파악 가능

COMSOL M-FILE STRUCTURE



```

134 - model.sol('sol1').runAll();
135
136 - model.result.numerical('sys1').set('format', 'filled');
137 - model.result.numerical('sys1').set('matrixassem', 'elimstiffness');
138 - model.result.numerical('sys1').set('table', 'tbl1');
139 - model.result.numerical('pev1').set('expr', 'v');
140 - model.result.numerical('pev1').set('descr', 'Displacement field, y component');
141 - model.result.numerical('pev1').set('table', 'tbl2');
142 - model.result.numerical('sys1').setResult;
143 - model.result.numerical('pev1').setResult;
144 - model.result('pg1').label('Force (truss)');
145 - model.result('pg1').feature('line1').set('tuberadiusscale', '0.042156294610216236');
146 - model.result('pg1').feature('line1').set('expr', 'truss.Nx1');
147 - model.result('pg1').feature('line1').set('resolution', 'extrafine');
148 - model.result('pg1').feature('line1').set('linetype', 'tube');
149 - model.result('pg1').feature('line1').set('radiusexpr', 'sqrt(truss.area/pi)');
150 - model.result('pg1').feature('line1').set('unit', 'N');
151 - model.result('pg1').feature('line1').set('descr', 'Local axial force');
152 - model.result('pg1').feature('line1').set('tuberadiusscaleactive', false);
153 - model.result('pg1').feature('line1').feature('def1').set('scale', '1.856953381770518');
154 - model.result('pg1').feature('line1').feature('def1').set('scaleactive', false);
155 - model.result('pg2').label('Stress (truss)');
156 - model.result('pg2').feature('line1').set('expr', 'truss.sn');
157 - model.result('pg2').feature('line1').set('resolution', 'extrafine');
158 - model.result('pg2').feature('line1').set('linetype', 'tube');
159 - model.result('pg2').feature('line1').set('radiusexpr', 'sqrt(truss.area/pi)');
160 - model.result('pg2').feature('line1').set('unit', 'N/m^2');
161 - model.result('pg2').feature('line1').set('descr', 'Axial stress at center line');
162
163 - % out = model;
164
165 - stiffness_matrix = model.result.table('tbl1').getReal;
166
  
```



model.result 부분에
elimstiffness 가 table 1 에
입력된 것을 확인

model.result.table('tbl1').ge
tReal 명령어로 table 1 의
데이터를 저장

