PDE & Direct Stiffness Method by COMSOL

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

CONTENTS

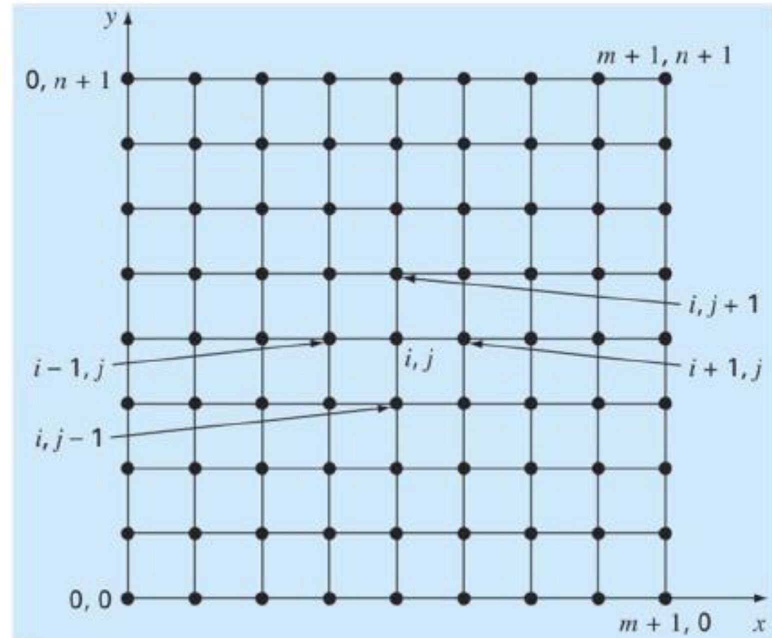
- **PDE Examples**

  - ✓ <u>**Laplace equation**</u>

  - ✓ **Heat conduction equation**

LAPLACE EQUATION

– PDE → algebraic difference equation



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

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

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

apply boundary conditions (fixed/Dirihlet)

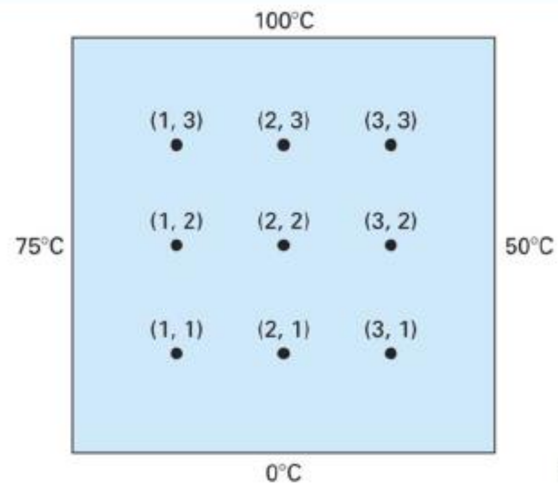
$$@(1,1): T_{21}+\underbrace{T_{01}}_{75°C}+T_{12}+\underbrace{T_{10}}_{0°C}-4T_{11}=0 \rightarrow 4T_{11}-T_{21}-T_{12}=75$$

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

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

⋮

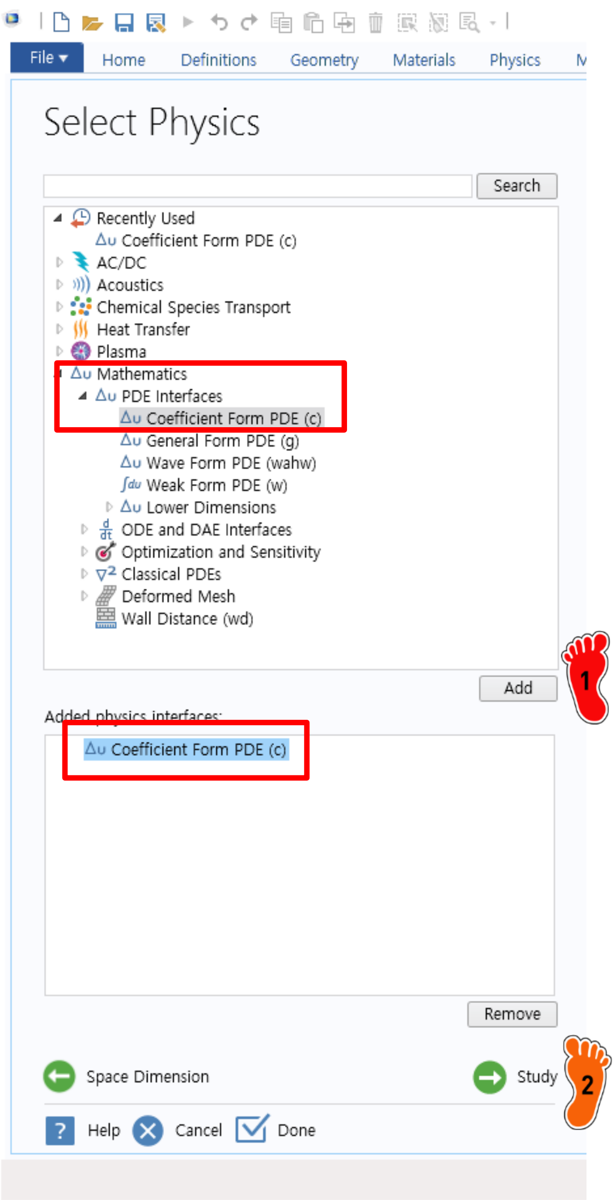
9 equations

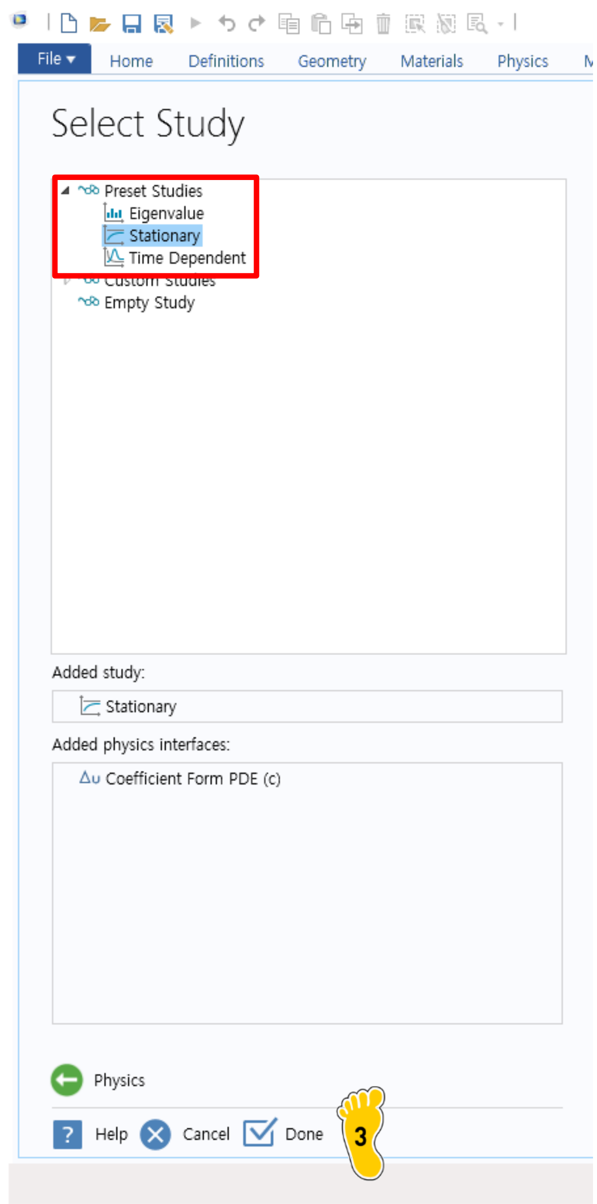


$$[K_{ij}]\{T_i\}=\{f_i\}$$

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

SETTING



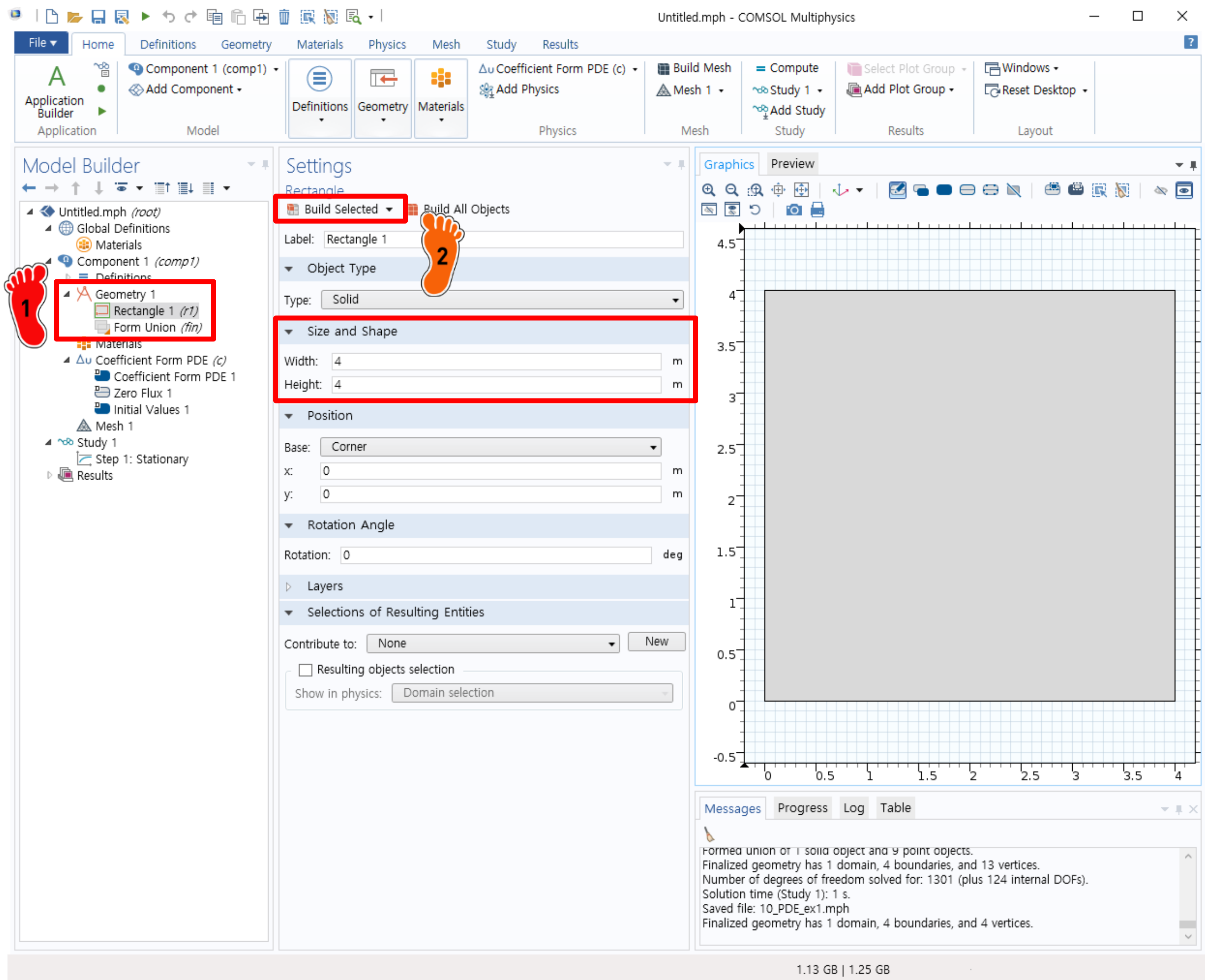


1 Model Wizard → 2D → Mathematics - PDE Interfaces – Coefficient form PDE 선택 → Add 클릭

2 Study 클릭

3 Stationary 선택 후 Done 클릭

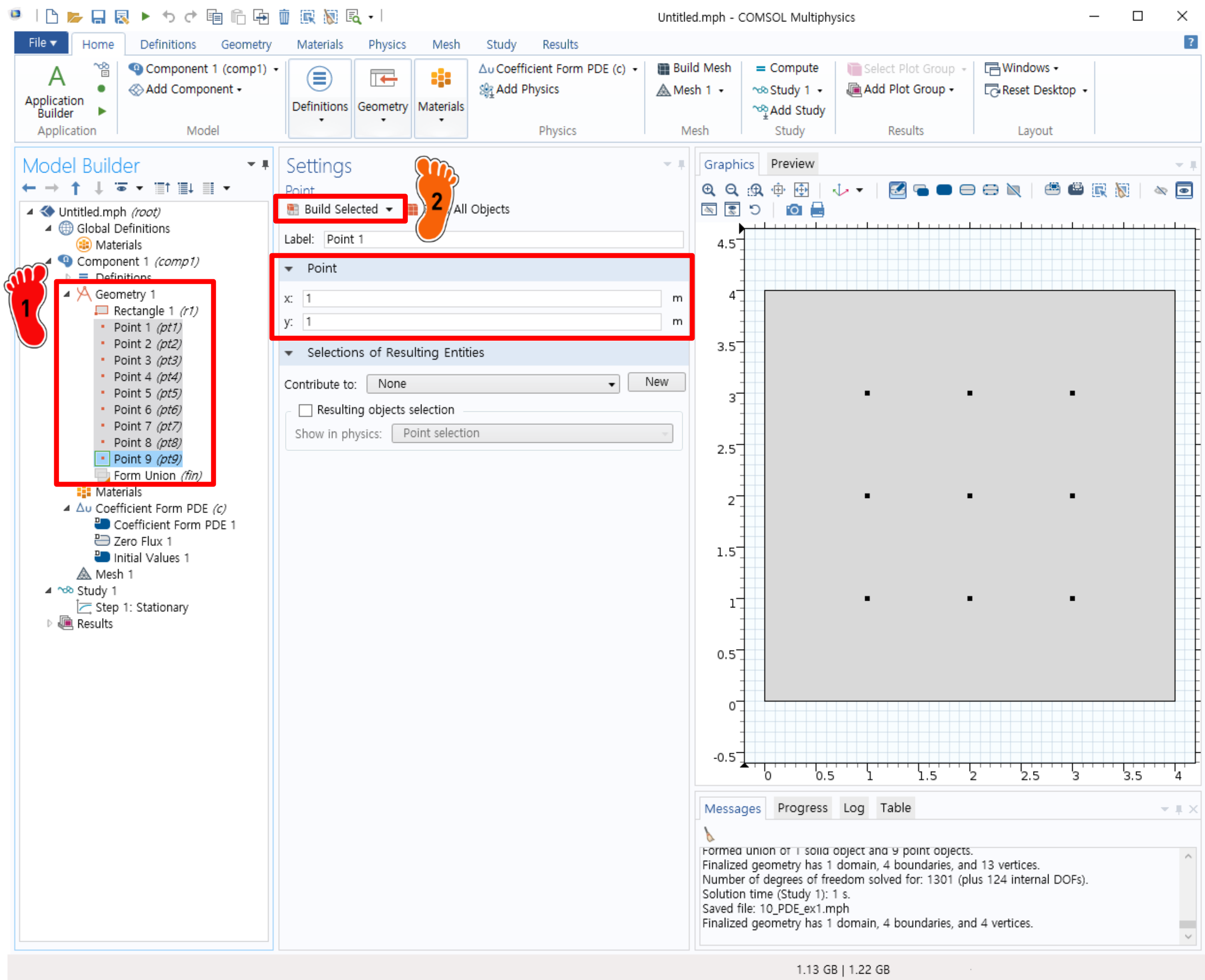
GEOMETRY CREATION



1 Geometry 1 메뉴를 마우스 우클릭 → Rectangle 선택

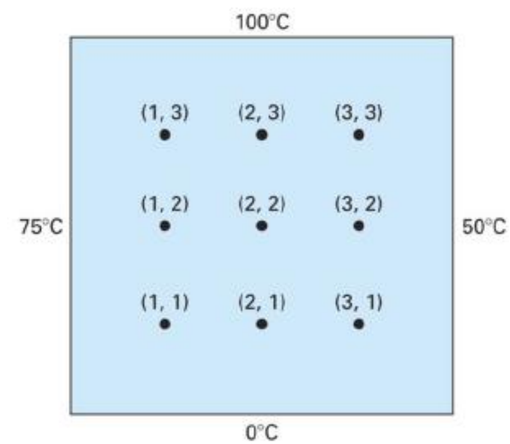
2 Width: 4, Height: 4 입력 후 Build Selected 클릭

GEOMETRY CREATION

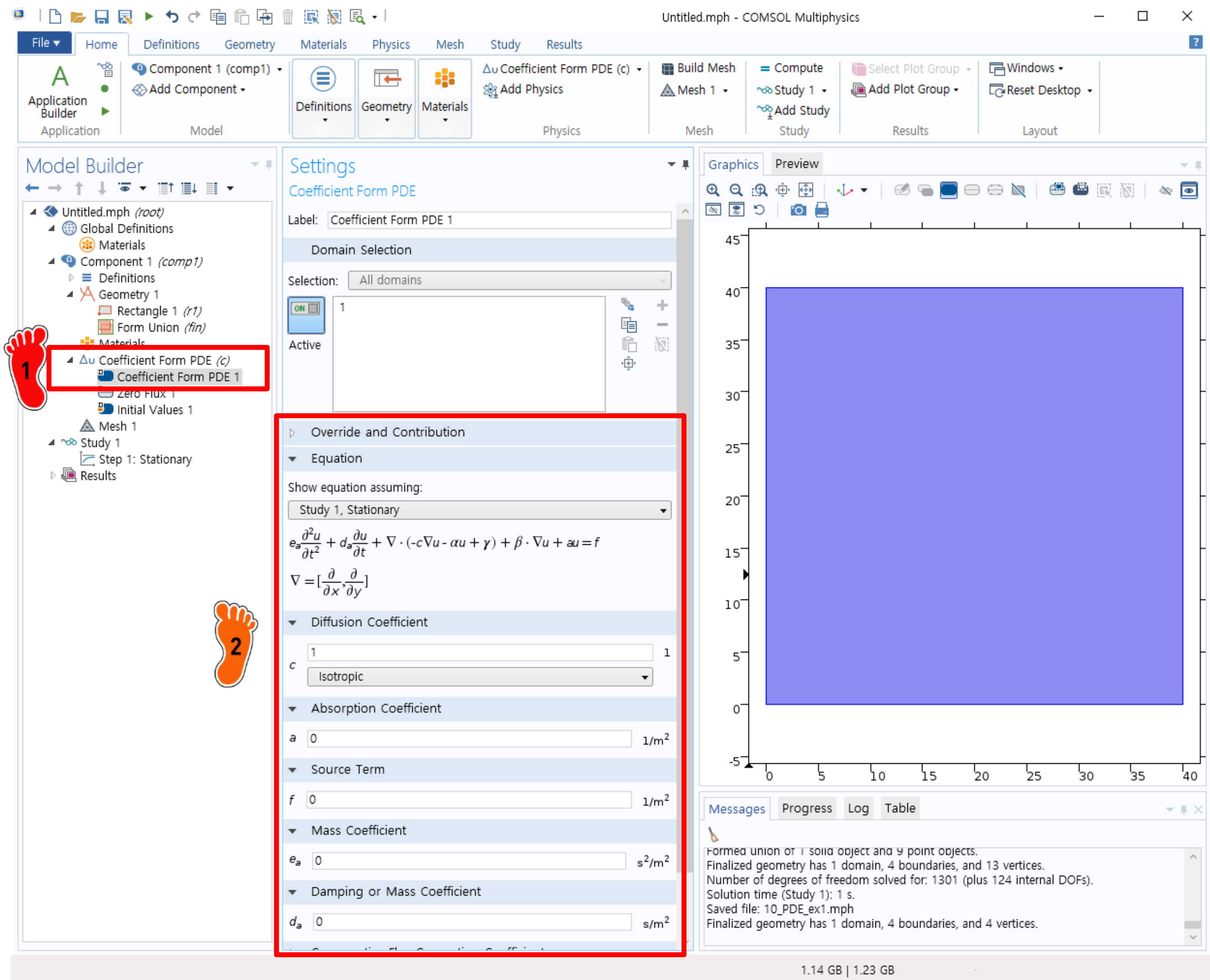


1 Geometry 1 메뉴를 마우스 우클릭 → Point 선택

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)

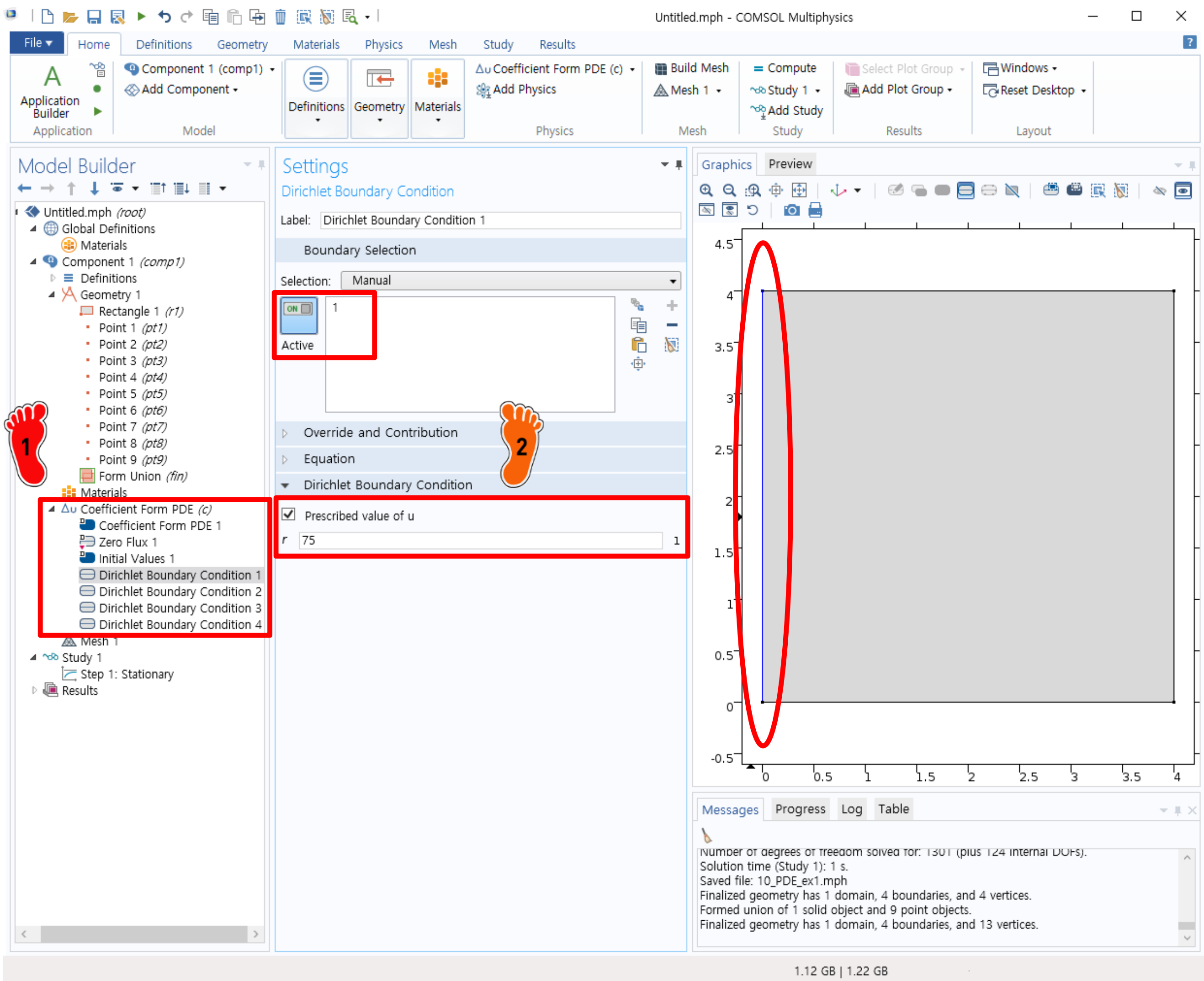
$$\nabla = \left[\frac{\partial}{\partial x}, \frac{\partial}{\partial y}\right]$$

$$\nabla^2 u = 0$$

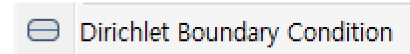
$$\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} = 0$$

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

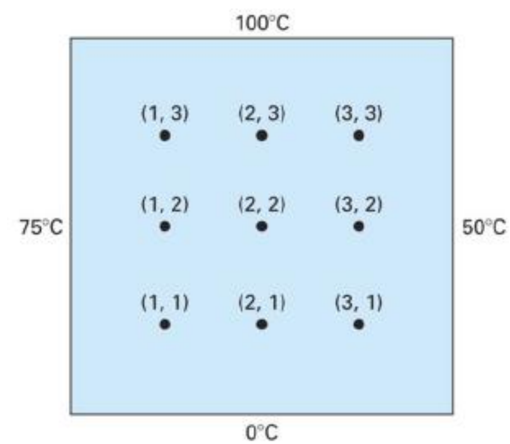
BOUNDARY CONDITION



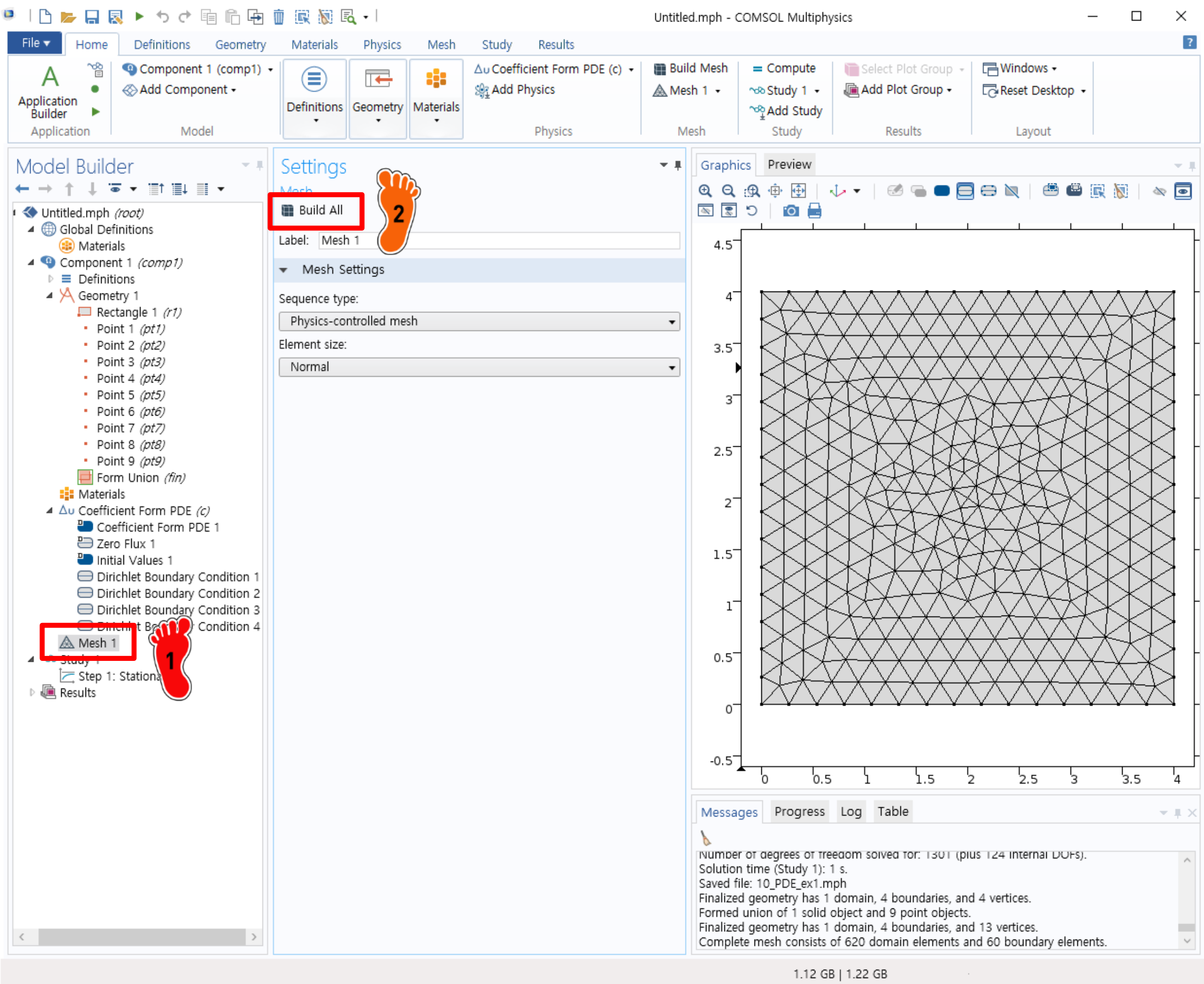
1. Coefficient Form PDE 메뉴 마우스 우클릭 → Dirichlet Boundary Condition 클릭



2. 4개 생성 및 경계 선택 후 경계값 입력

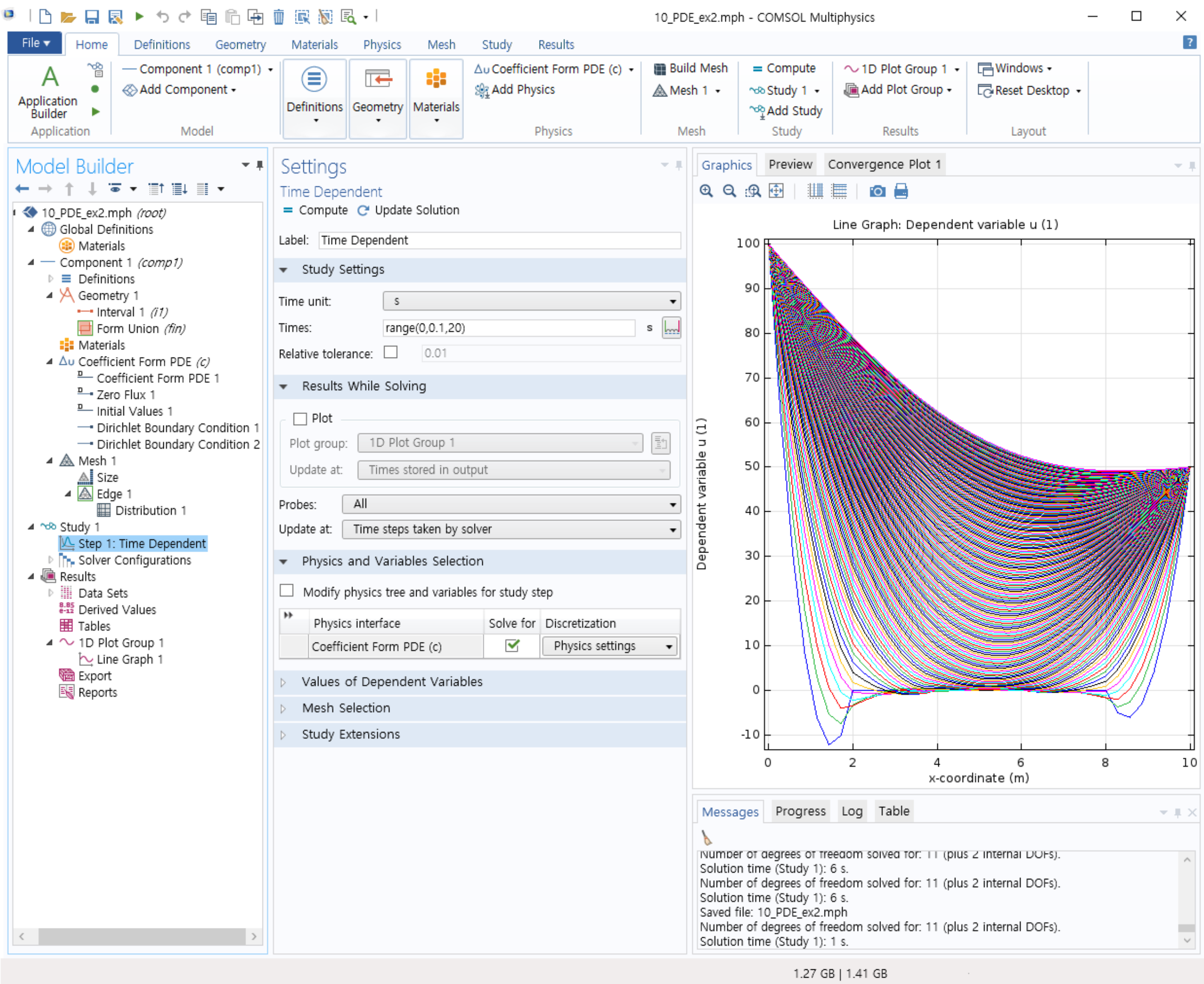


MESH CREATION



1. Mesh 1 메뉴 클릭
2. Build All 클릭

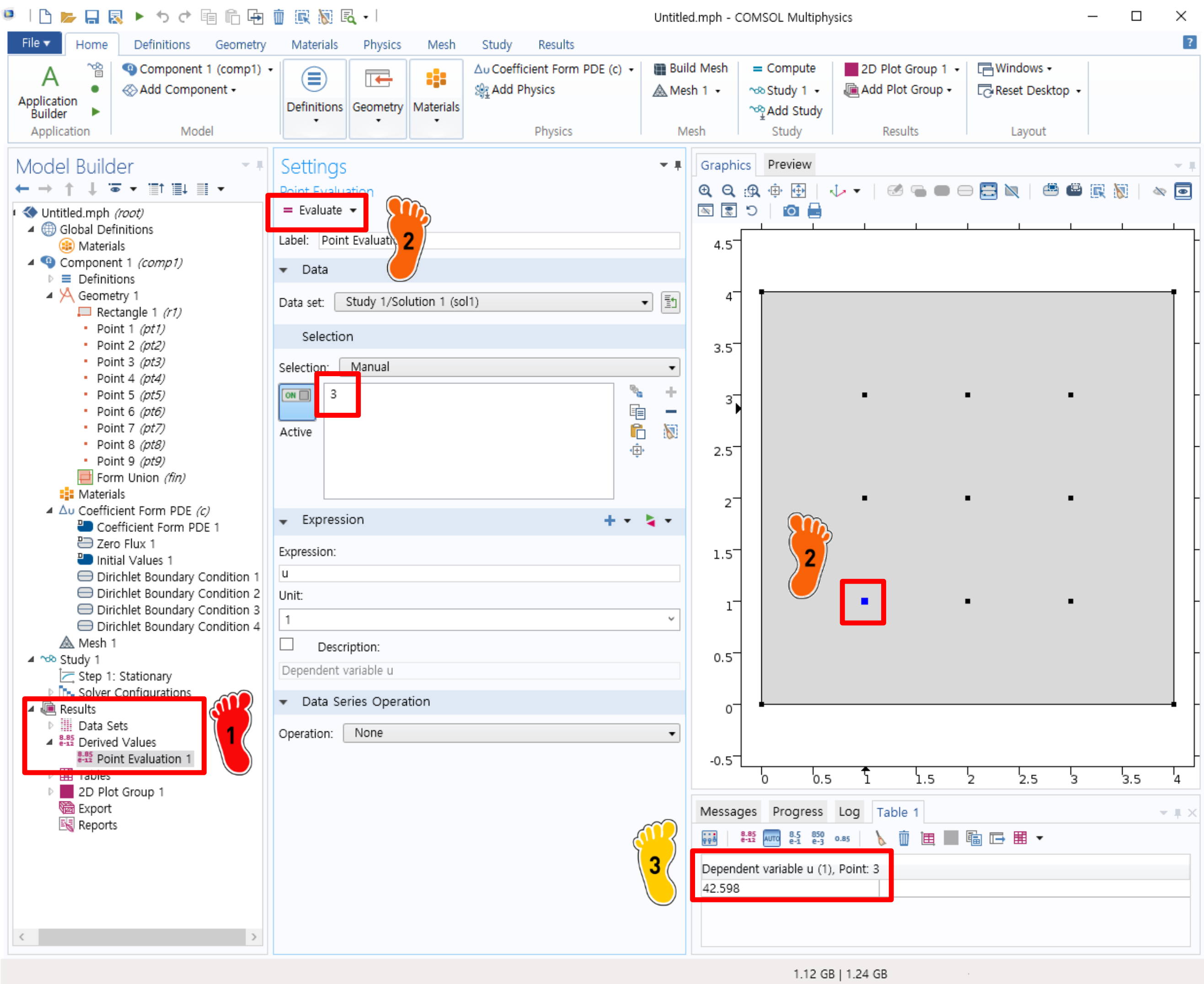
COMPUTE



1 Study 1 메뉴 선택

2 Compute 클릭

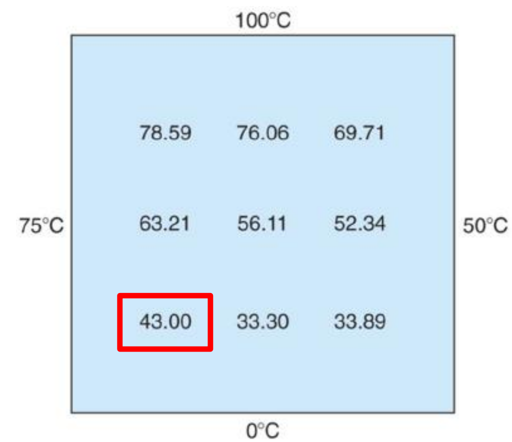
POST-PROCESSING



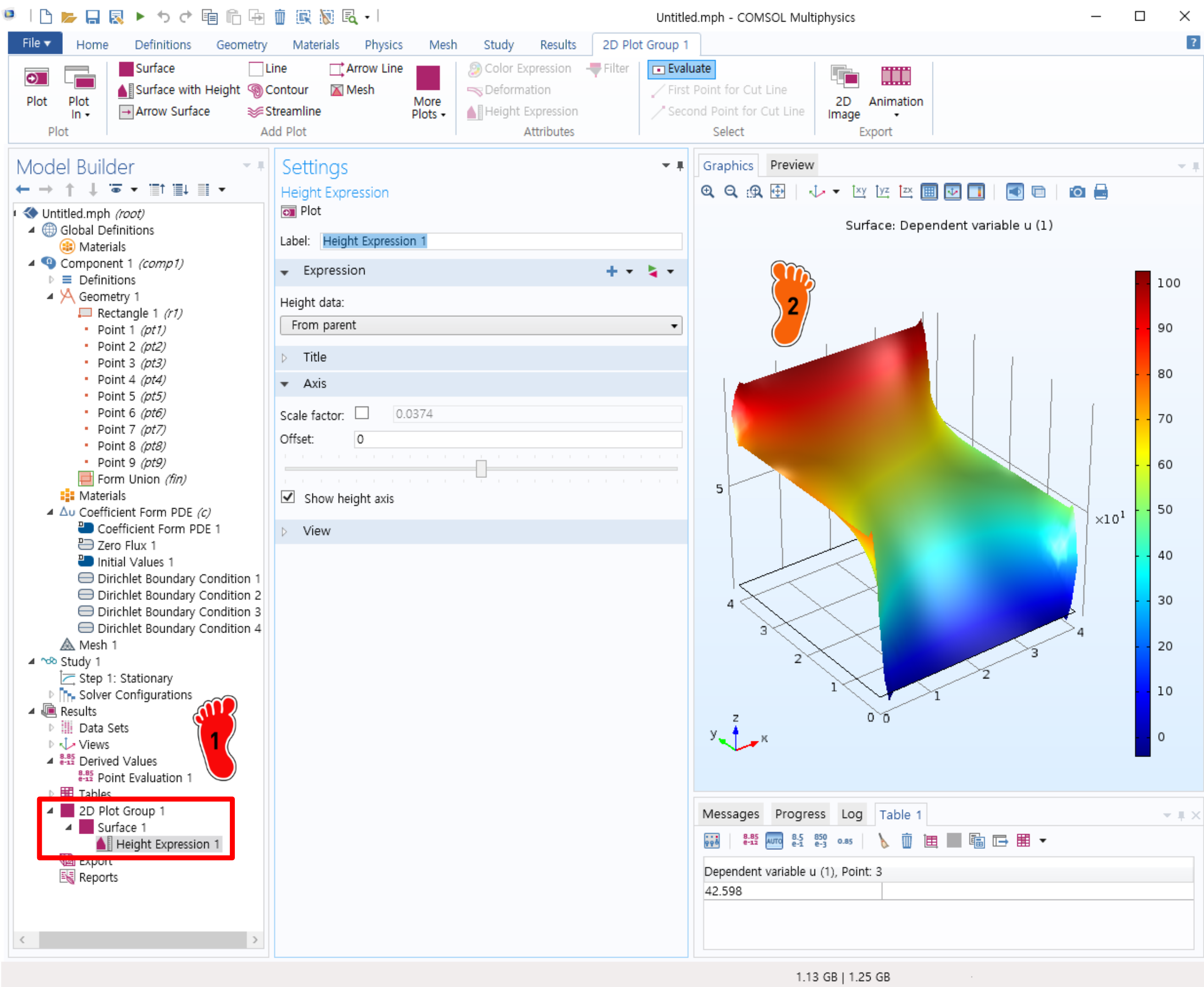
Results → Derived Values 우클릭 → Point Evaluation 선택

측정하고자 하는 Point 선택 후 Evaluate 클릭

결과값 확인

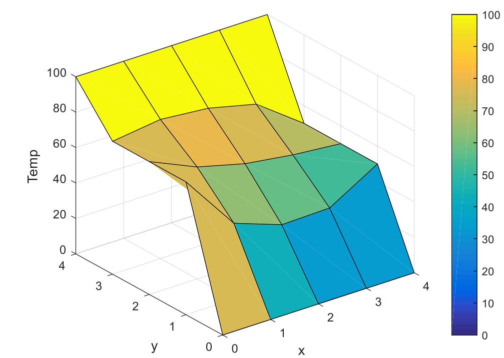


POST-PROCESSING



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

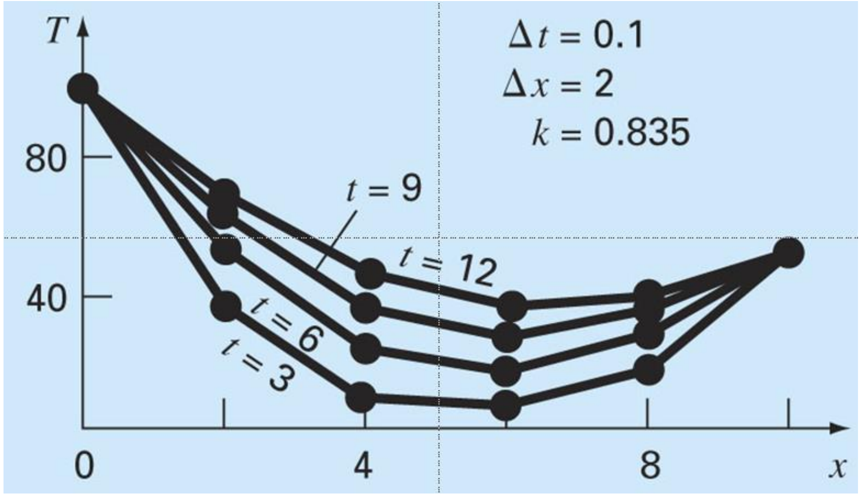
heat stored in the element over a unit time period $\Delta t = (In) - (Out)$

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

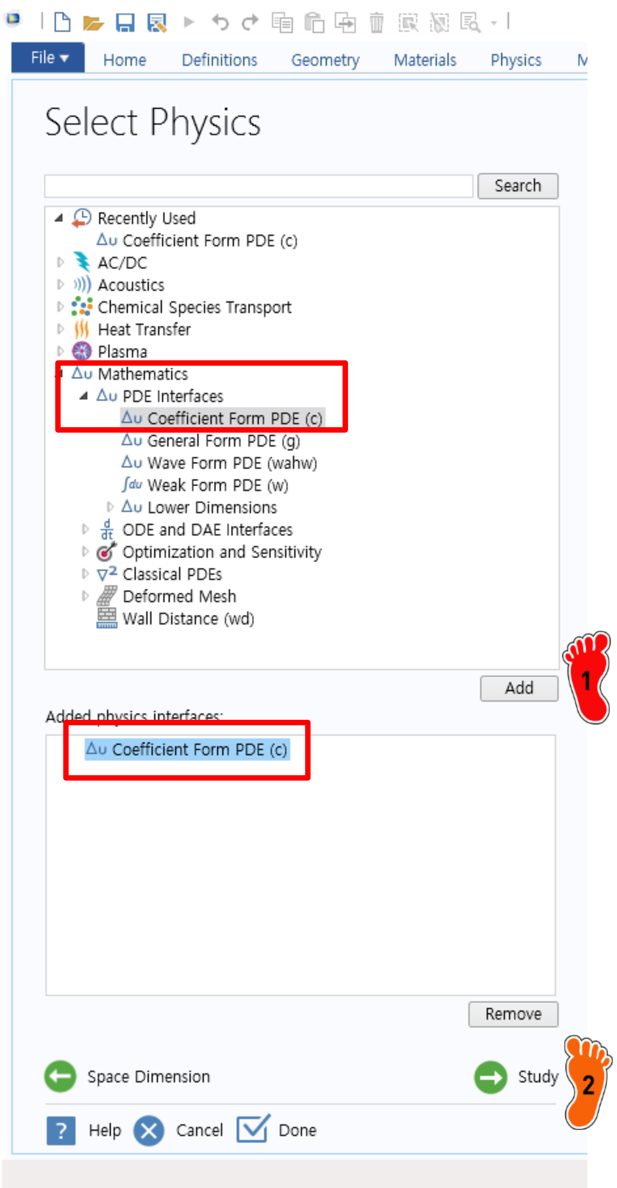
$$\frac{q(x) - q(x+\Delta x)}{\Delta x} = \rho C \frac{\Delta T}{\Delta t} \xrightarrow{\text{taking the limit}} -\frac{\partial q}{\partial x} = \rho C \frac{\partial T}{\partial t} \xrightarrow{q=-k\rho C\frac{\partial T}{\partial x}} k\frac{\partial^2 T}{\partial x^2} = \frac{\partial T}{\partial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

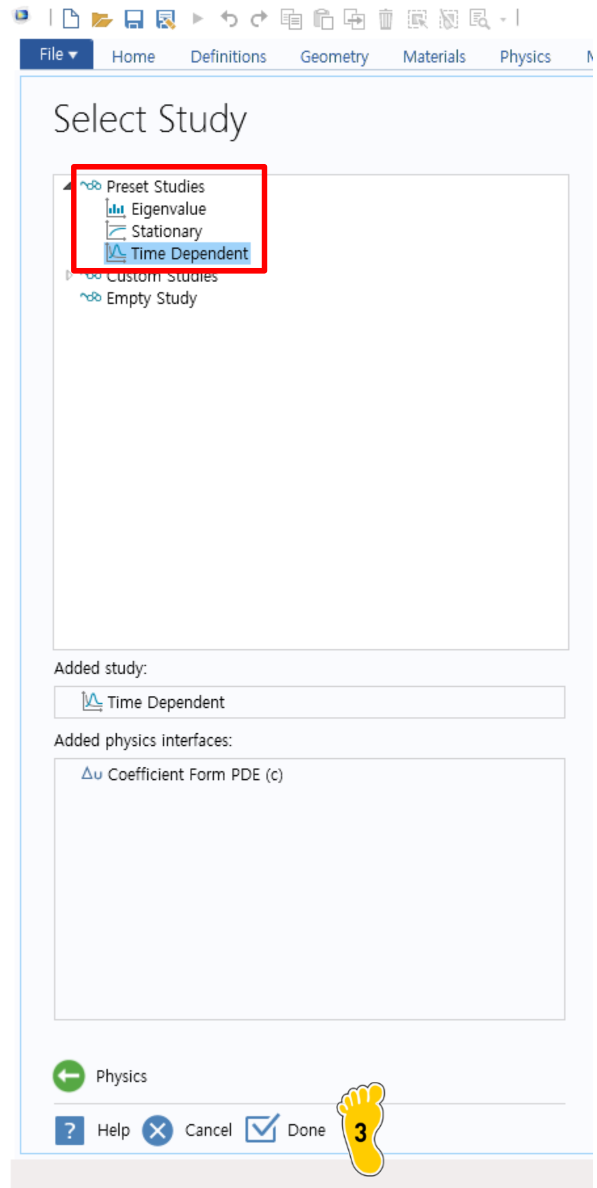
$$\begin{cases} k = 0.835 \text{ cal}/(\text{s}\cdot\text{cm}\cdot{}^\circ\text{C}) \\ \Delta x = 2 \text{ cm} \\ \Delta t = 0.1 \text{ s} \end{cases}$$

$$\begin{cases} \text{Boundary condition} \\ T(0,t) = 100^\circ\text{C} \\ T(10,t) = 50^\circ\text{C} \end{cases} \quad \begin{cases} \text{Initial condition} \\ T(i,t=0) = 0^\circ\text{C} \end{cases}$$



SETTING



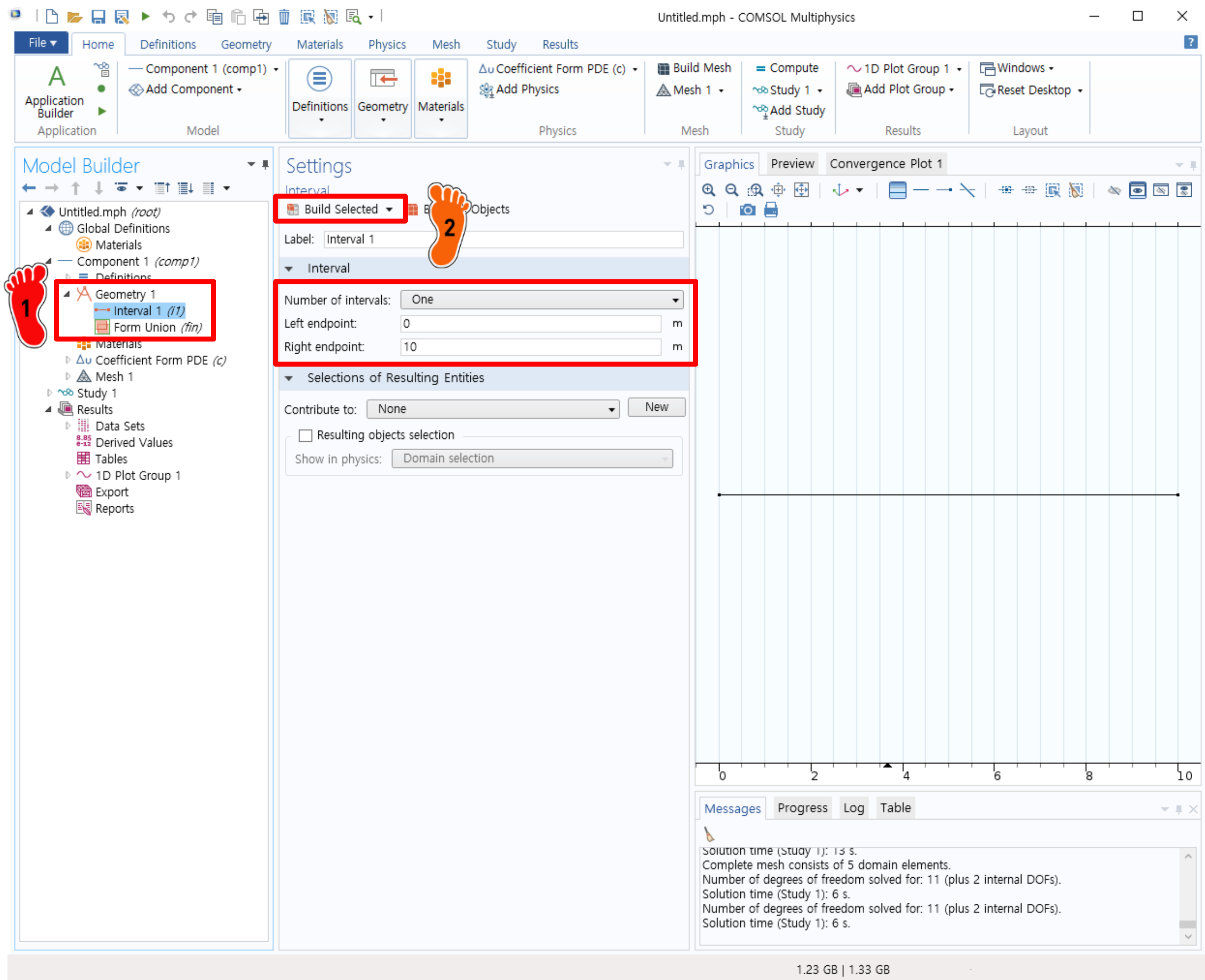


1 Model Wizard → 1D → Mathematics - PDE Interfaces – Coefficient form PDE 선택 → Add 클릭

2 Study 클릭

3 Time Dependent 선택 후 Done 클릭

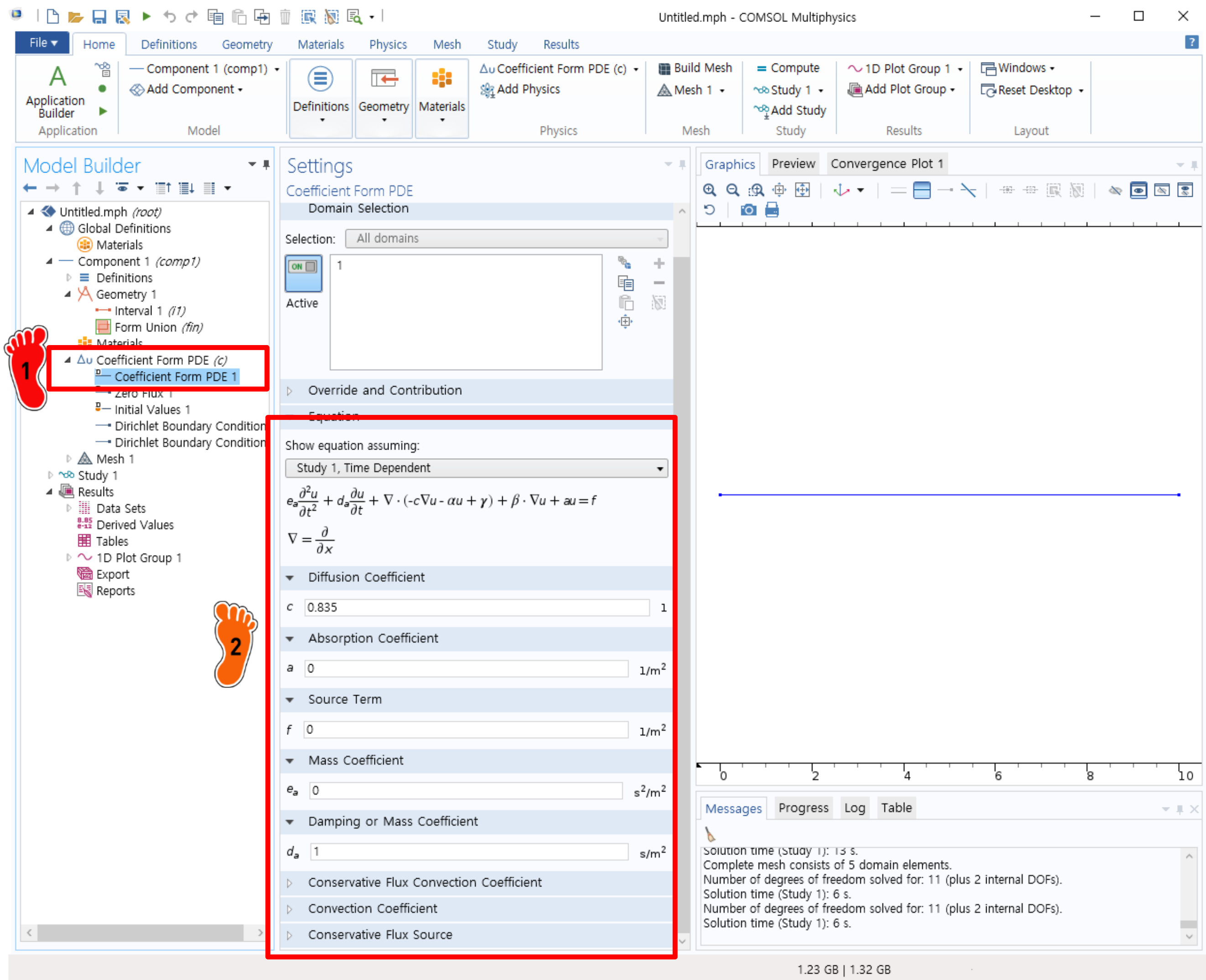
GEOMETRY CREATION



1 Geometry 1 우클릭 → Interval 선택

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

COEFFICIENT INPUT



1 Coefficient Form PDE 1 선택

2 계수 값 입력 (나머지 0)

$$c = 0.835$$

$$d_a = 1$$

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

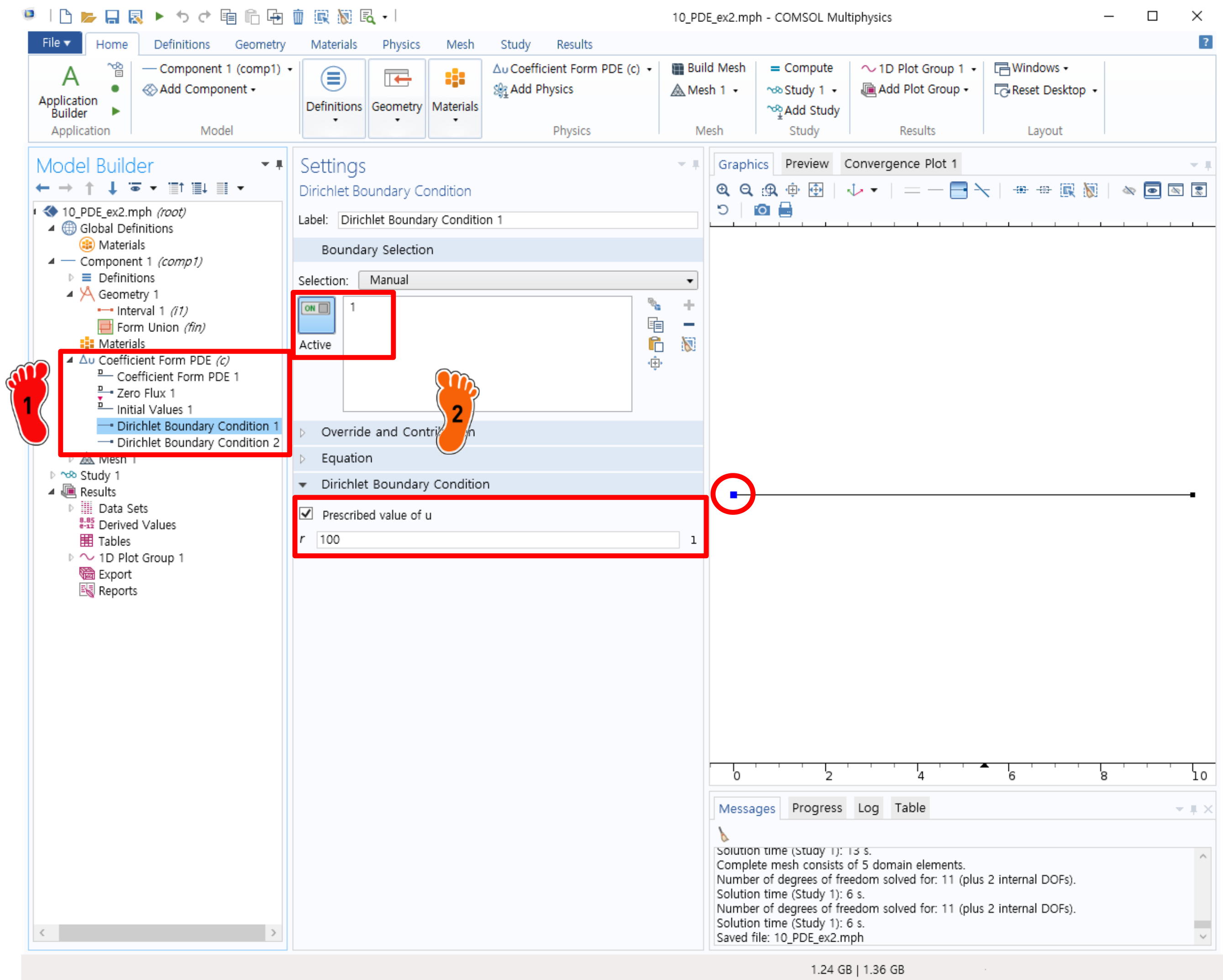
$$\updownarrow$$

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

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

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

BOUNDARY CONDITION



1 Coefficient Form PDE 우클릭 → Dirichlet Boundary Condition 클릭

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

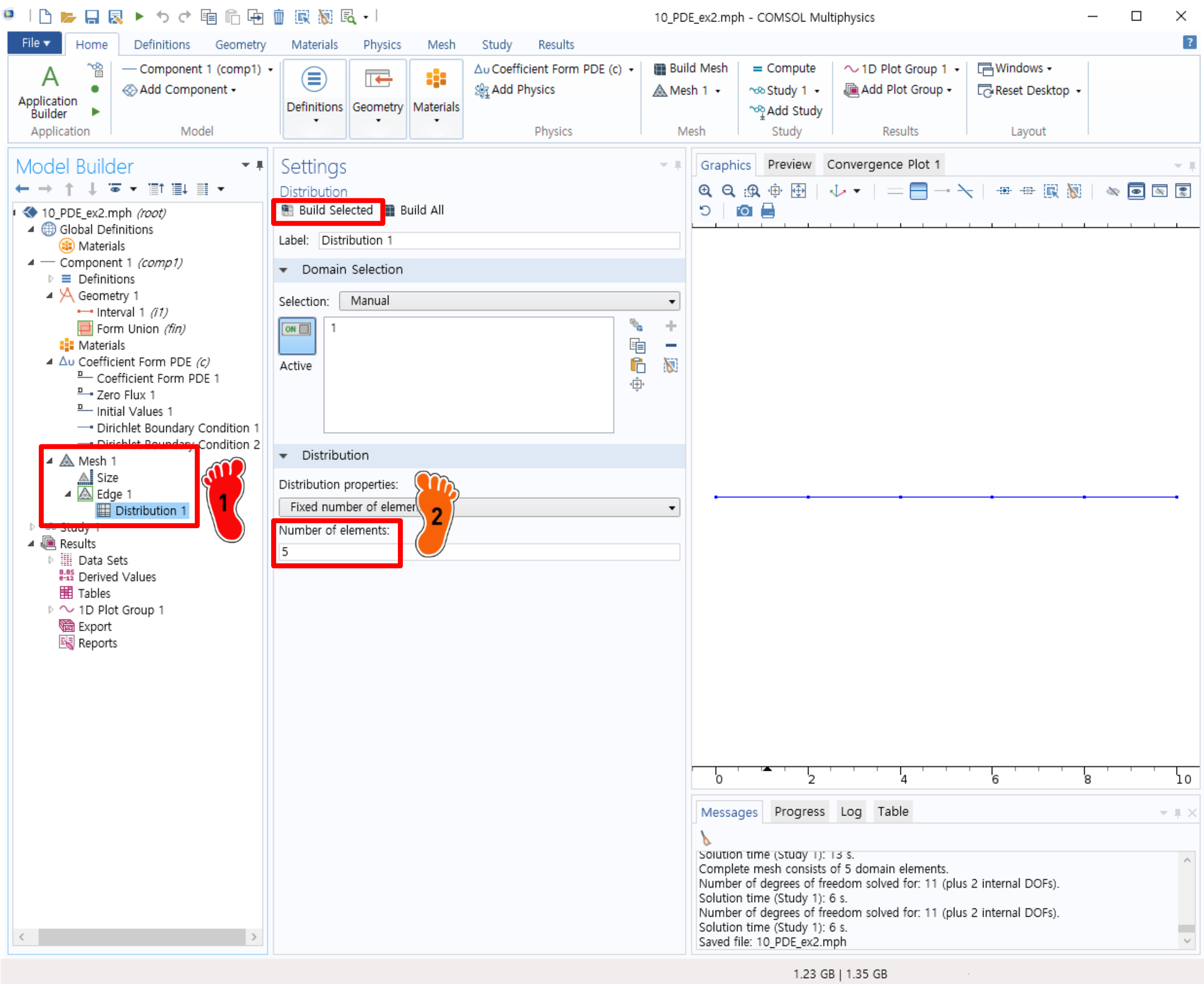
Boundary condition

$$\boxed{\begin{aligned} T(0,t) &= 100°C \\ T(10,t) &= 50°C \end{aligned}}$$

Initial condition

$$T(i, t=0) = 0°C$$

MESH CREATION

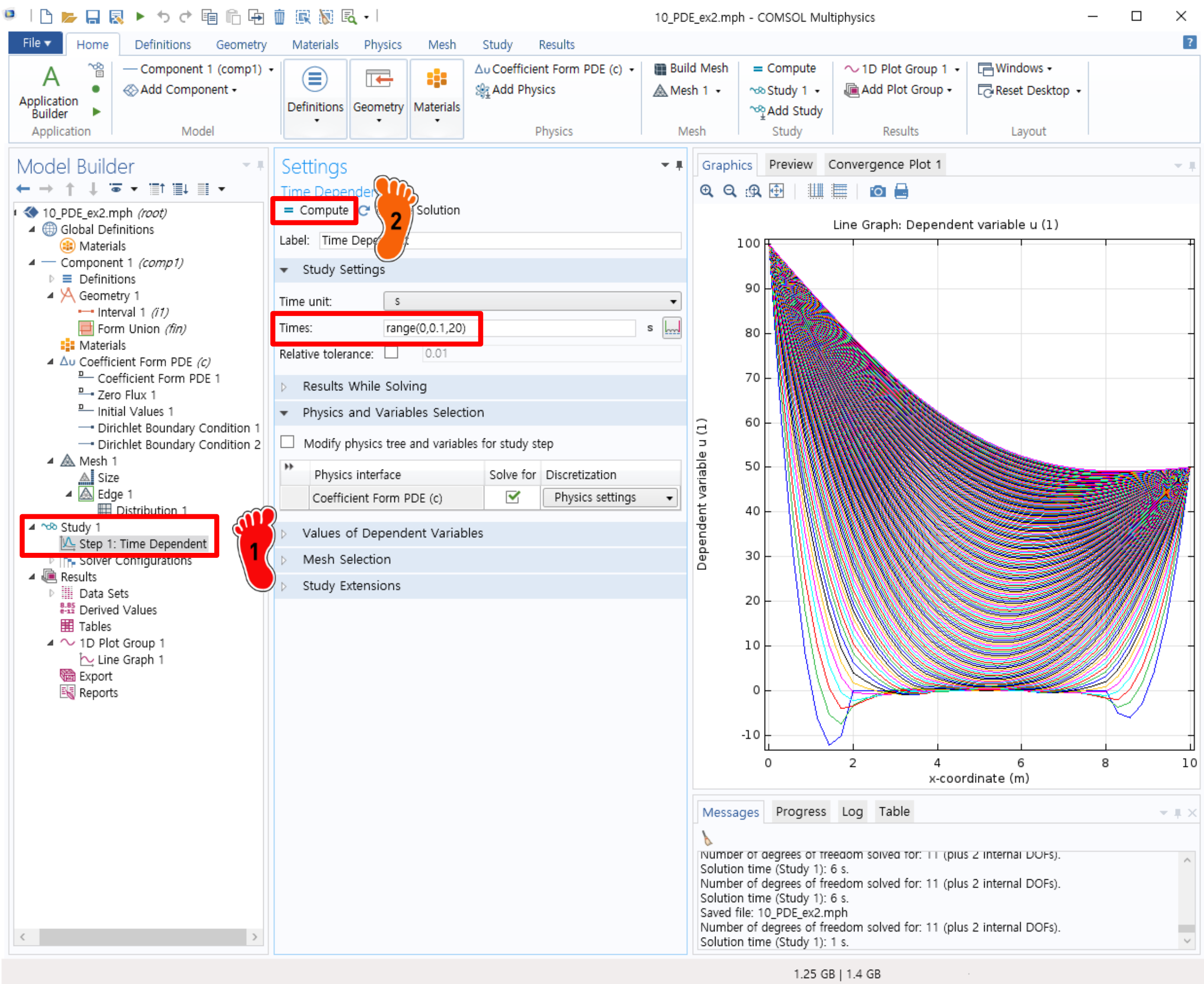


1 Mesh 1 메뉴 우클릭
→ Edge 선택
→ Edge 메뉴 우클릭
→ Distribution 선택

2 Number of elements에 5
입력 → Build Seleted 클릭

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

COMPUTE

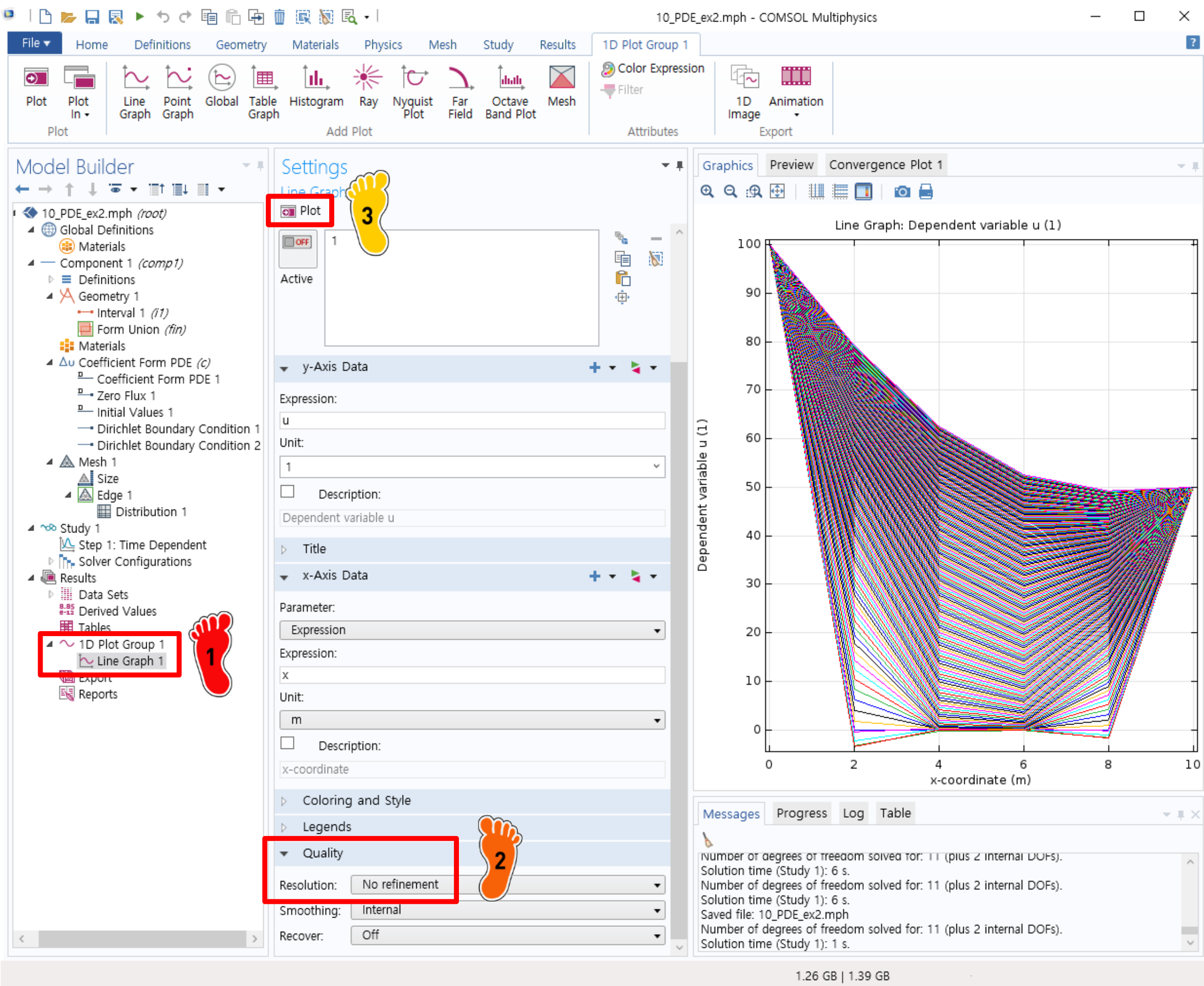


1. Study 1 → Step 1: Time Dependent 클릭

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

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

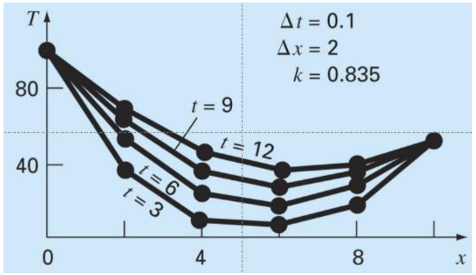
POST-PROCESSING



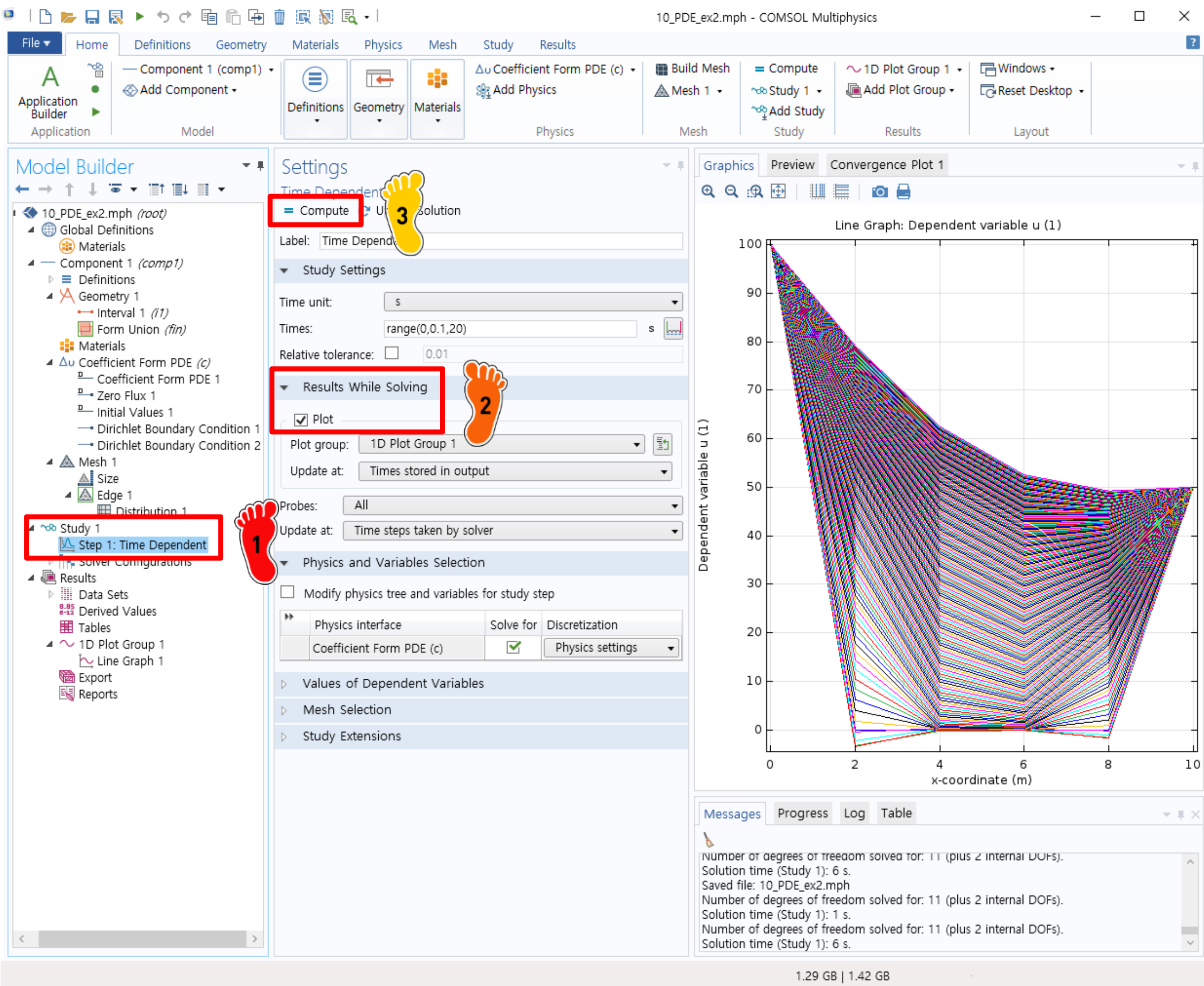
1 Line Graph 1 클릭

2 Quality → Resolution 에서 No refinement로 변경

3 Plot 클릭



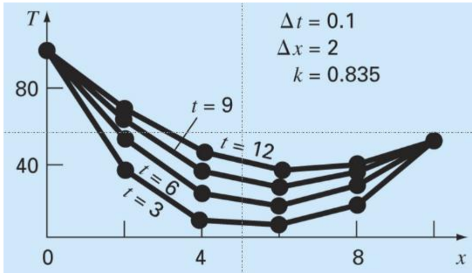
POST-PROCESSING



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**

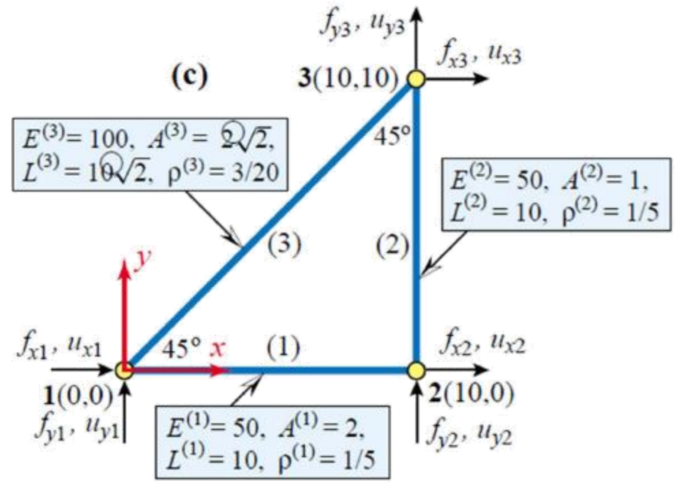
  ✓ **Cross section property**

  ✓ **Boundary condition**

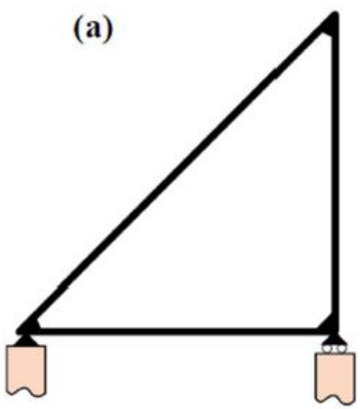
  ✓ **Nodal force**

  ✓ **Analysis**

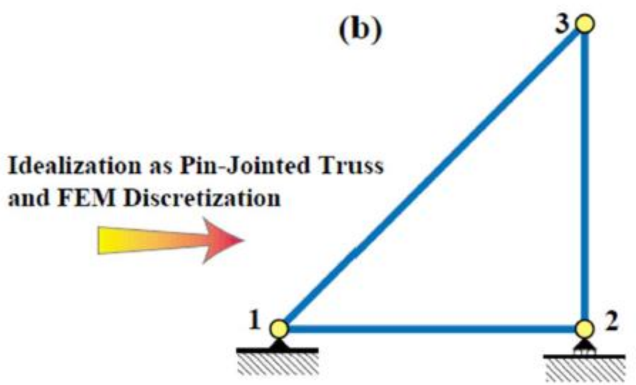
EXAMPLE TRUSS STRUCTURE



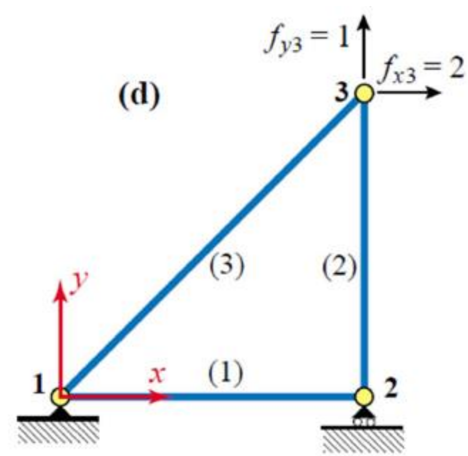
Geometric, material and fabrication properties



Physical structure



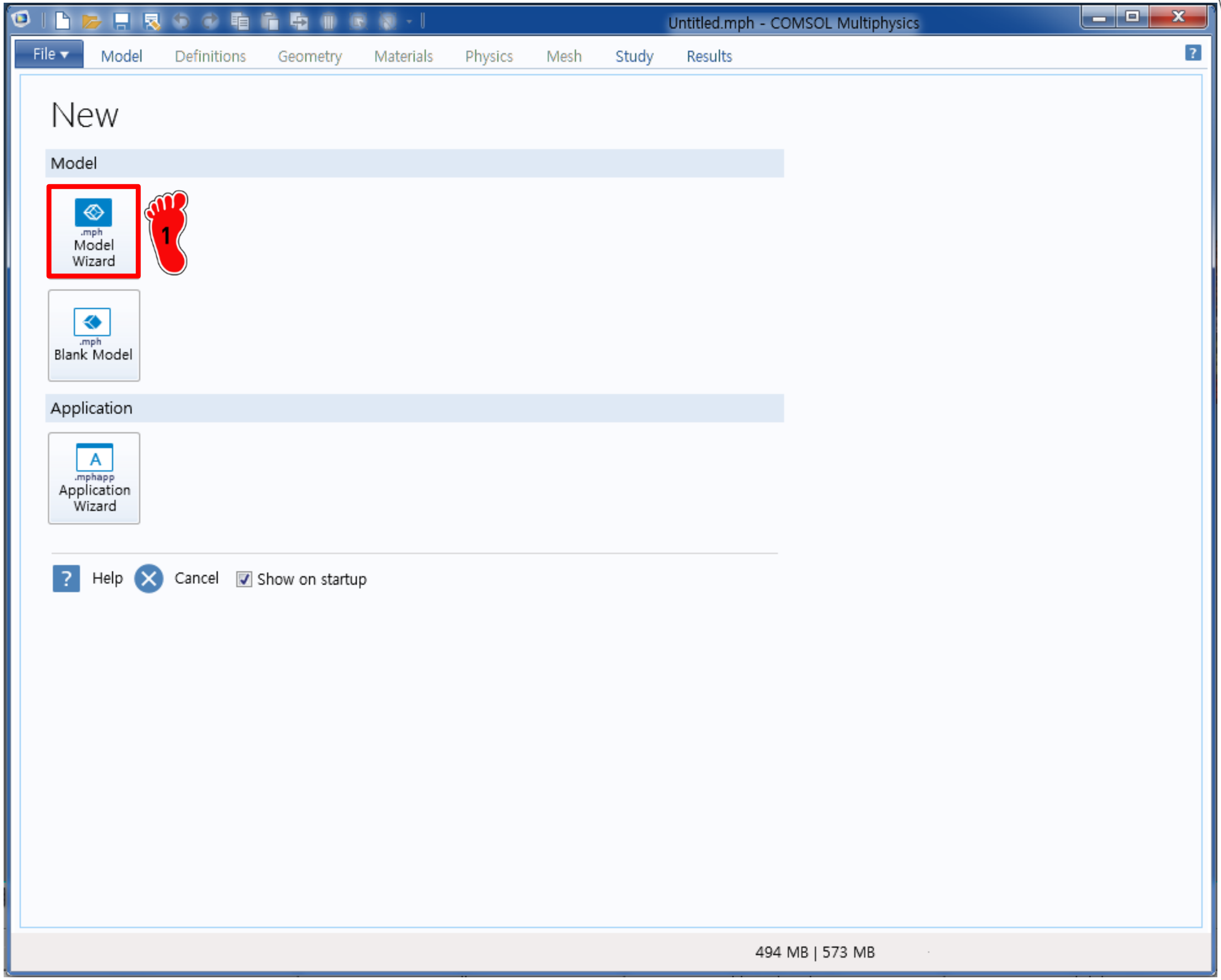
Idealization as a pin-jointed bar assemblage



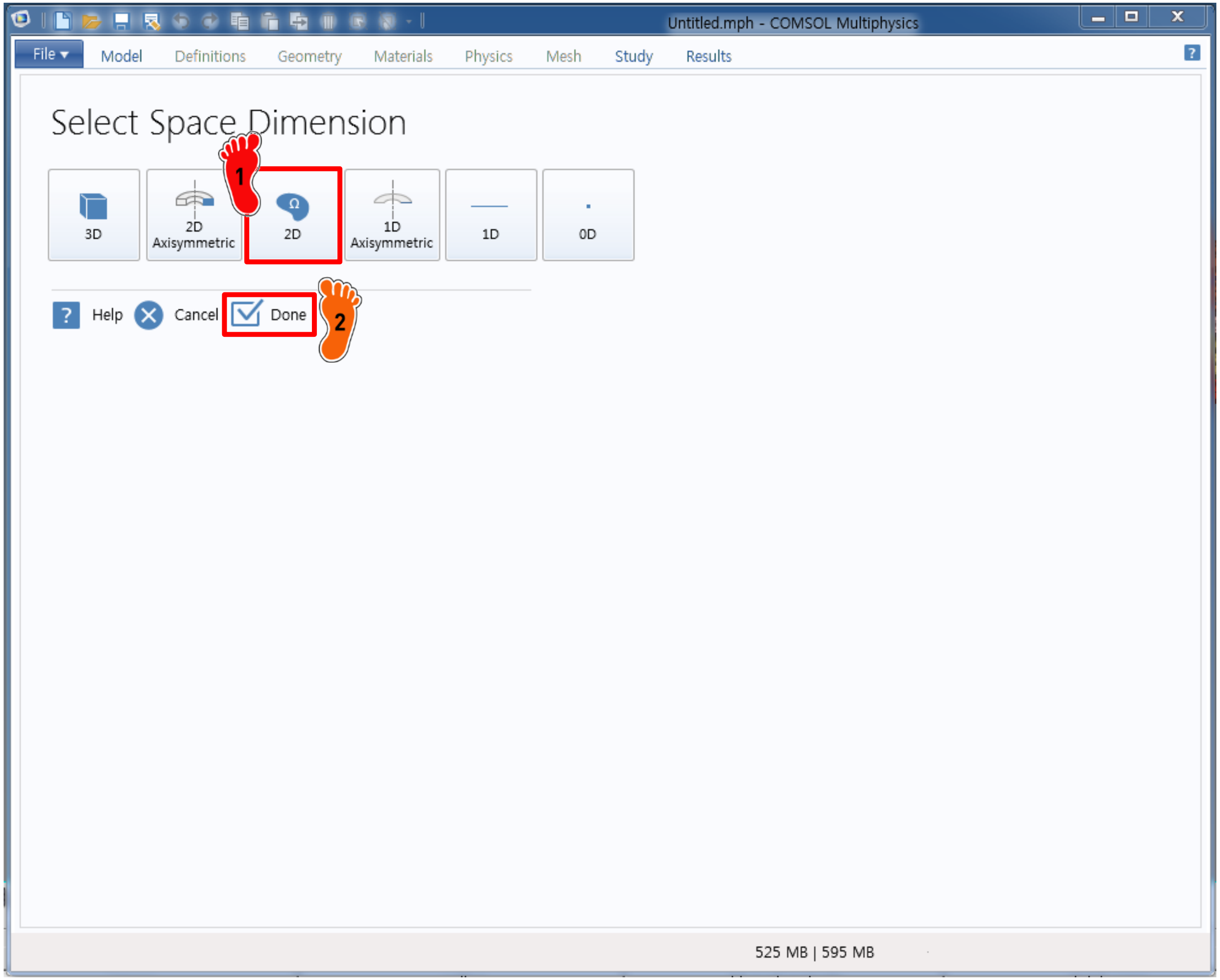
Support conditions and applied loads

MODEL WIZARD

1 Model Wizard 선택



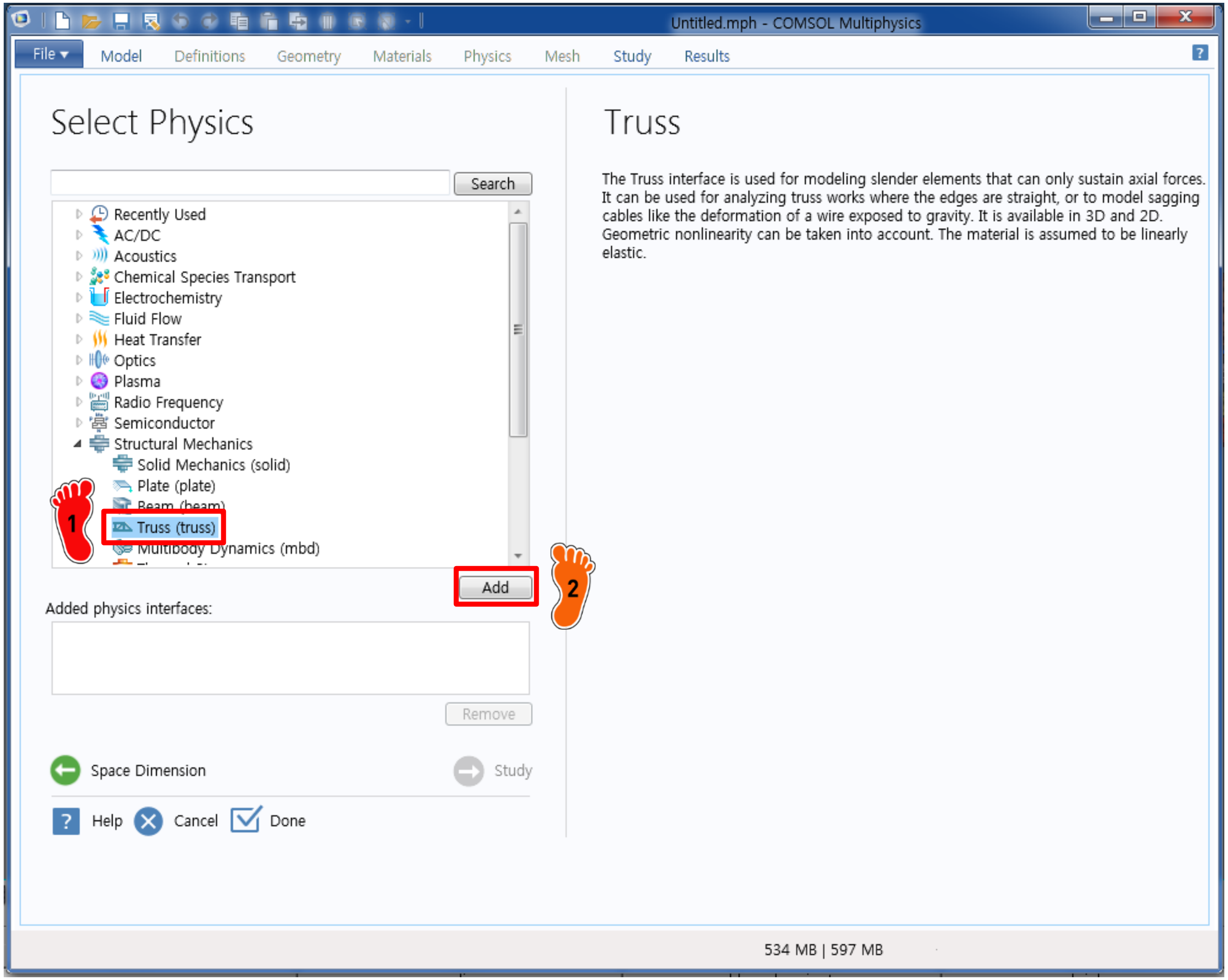
DIMENSION SELECTION



1 2D 선택

2 Done 클릭

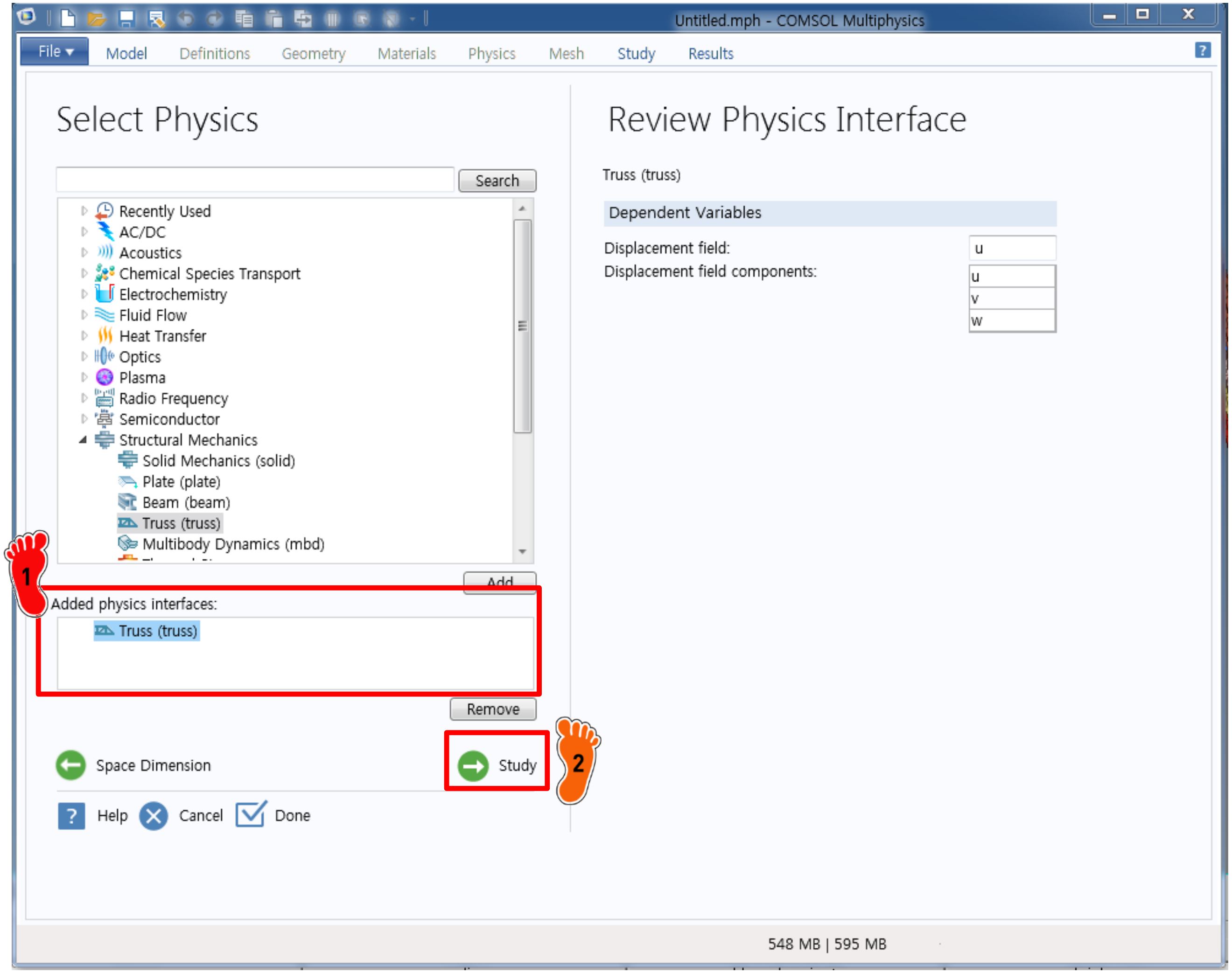
PHYSICS SELECTION



1 Structural Mechanics 모듈의 Truss physics 선택

2 Add 클릭

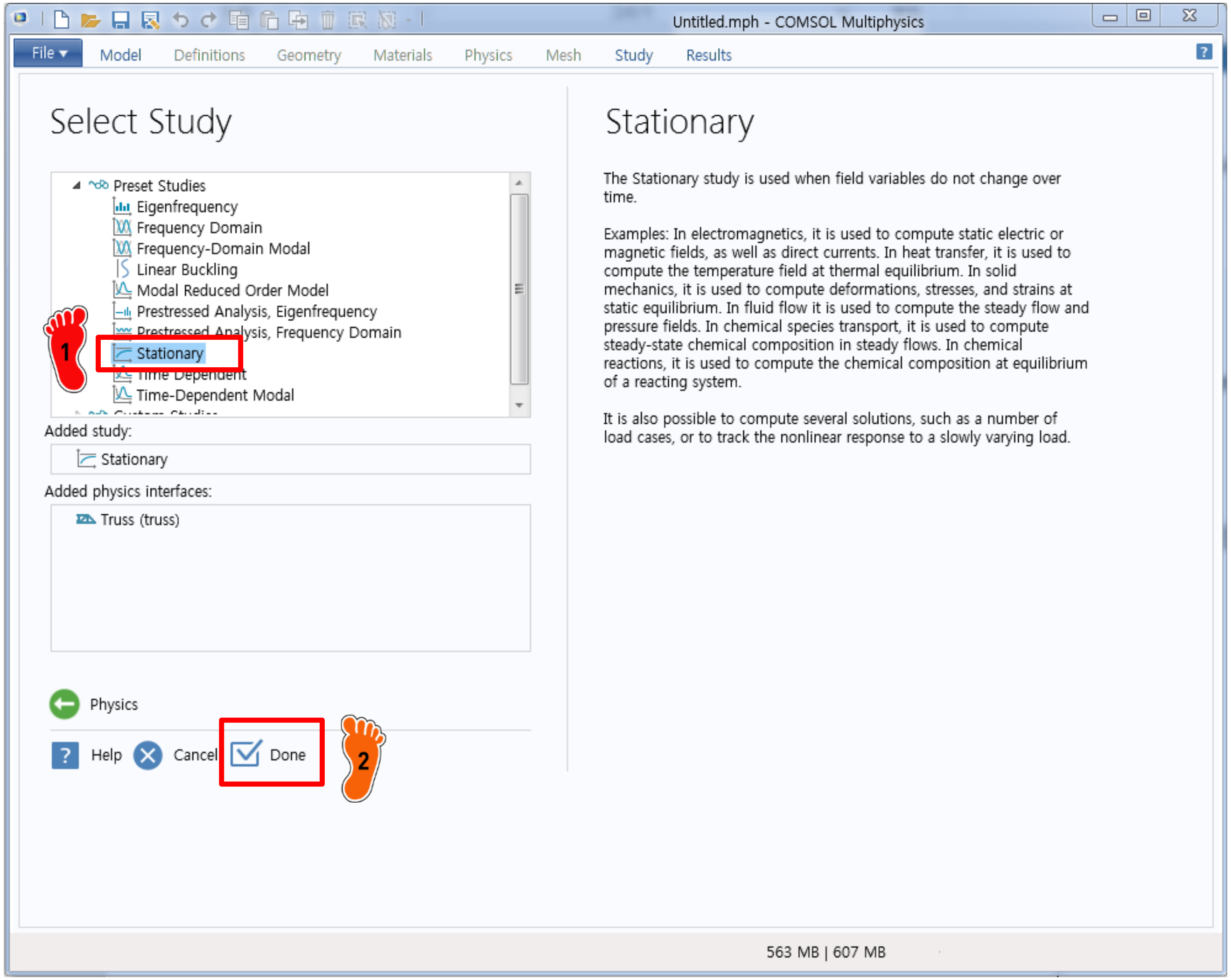
PHYSICS SELECTION



1 Added physics interfaces에서 추가된 physics 확인

2 Study 클릭

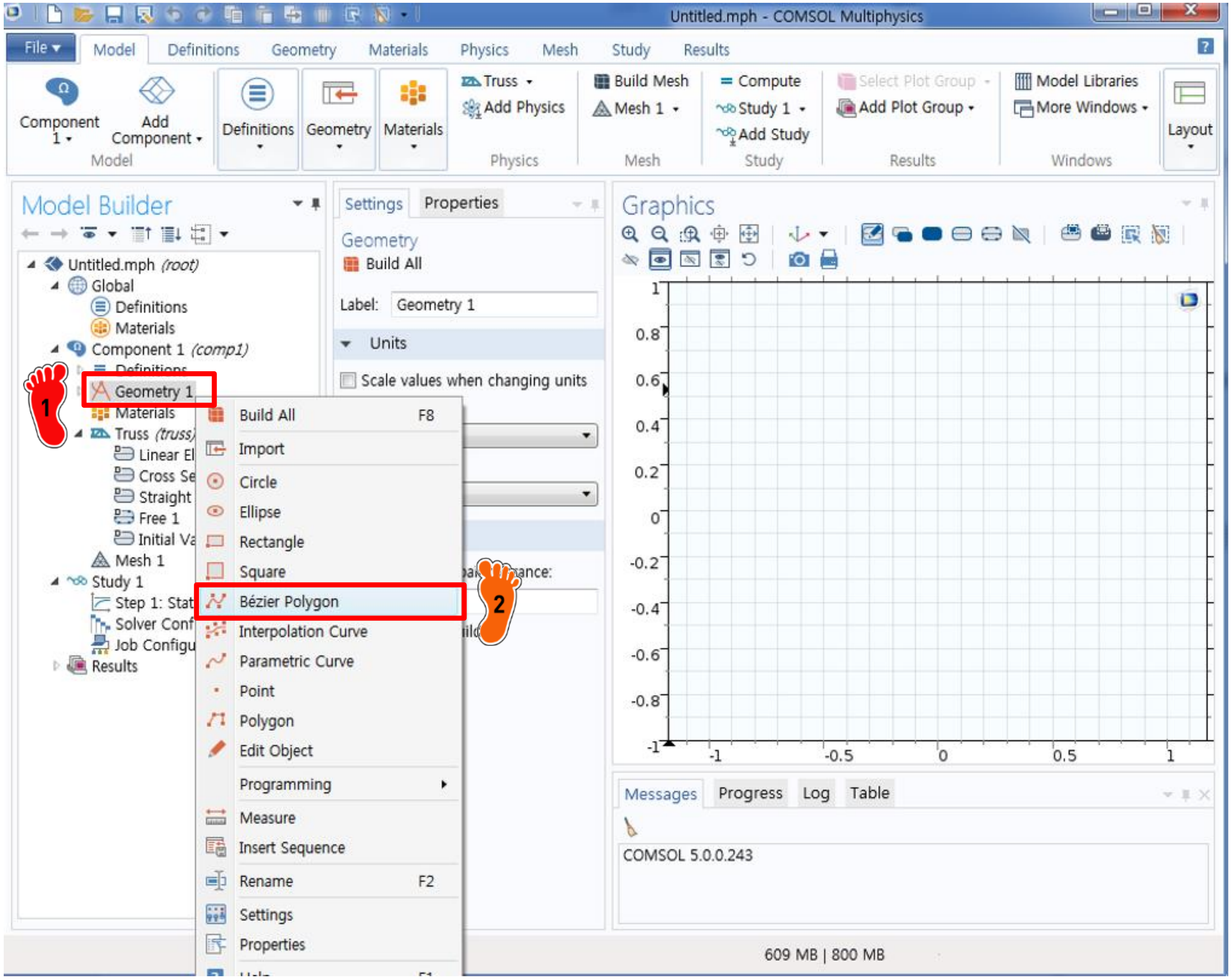
STUDY TYPE SELECTION



1 Preset Studies에서 Stationary 선택

2 Done 클릭

GEOMETRY CREATION

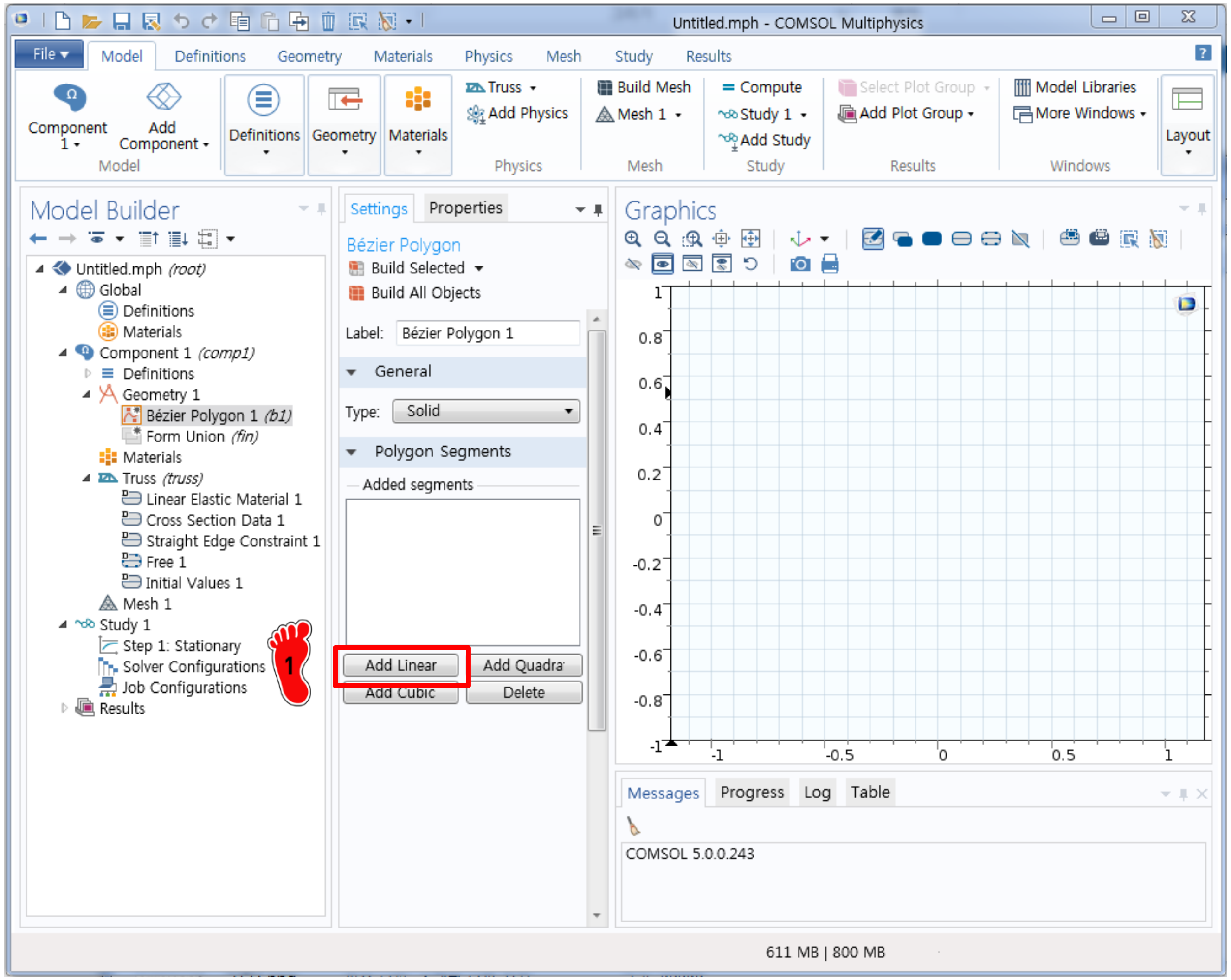


1 Geometry 1 마우스 우클릭

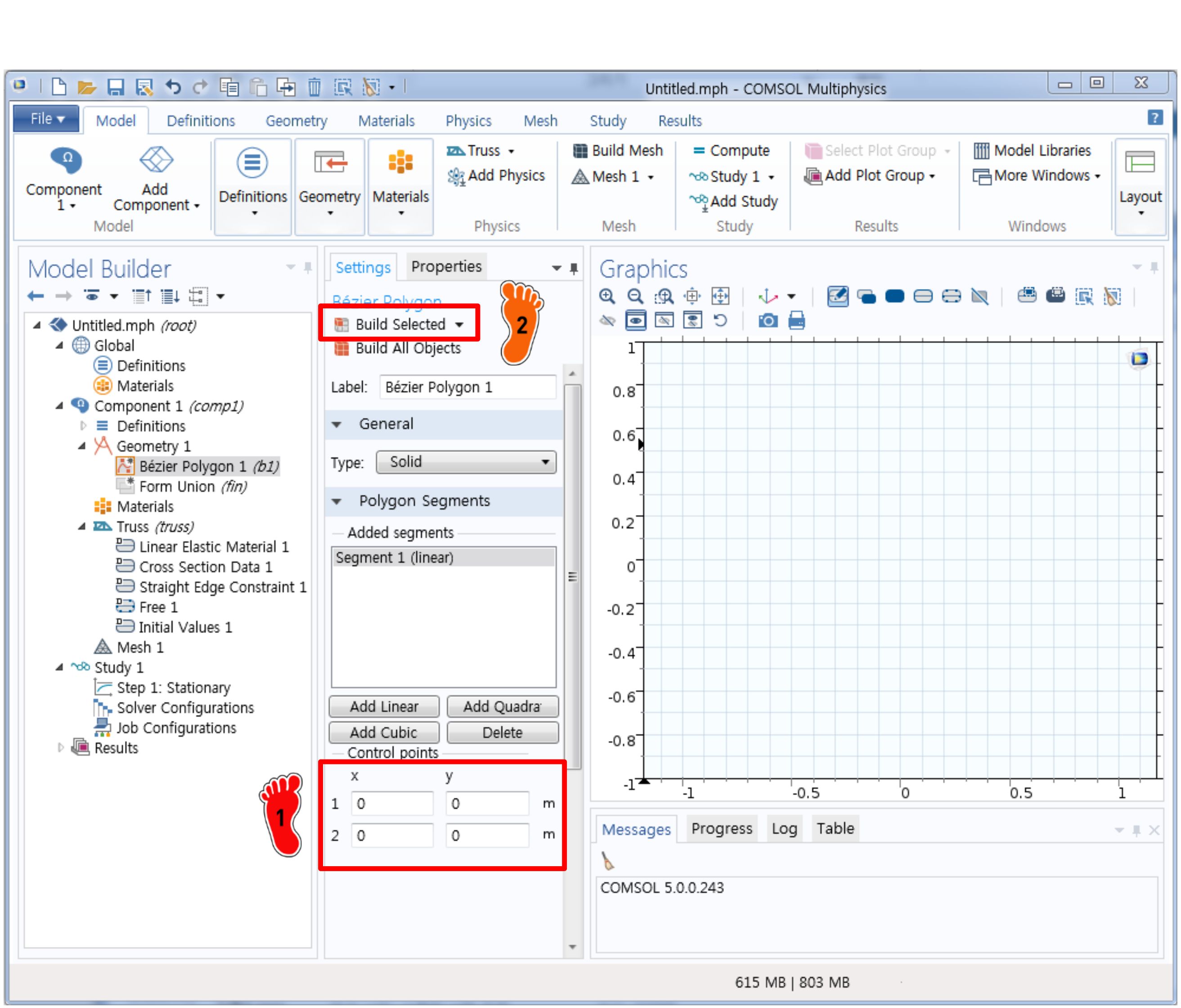
2 Bezier Polygon 클릭

GEOMETRY CREATION

1 Add Linear 클릭



GEOMETRY CREATION



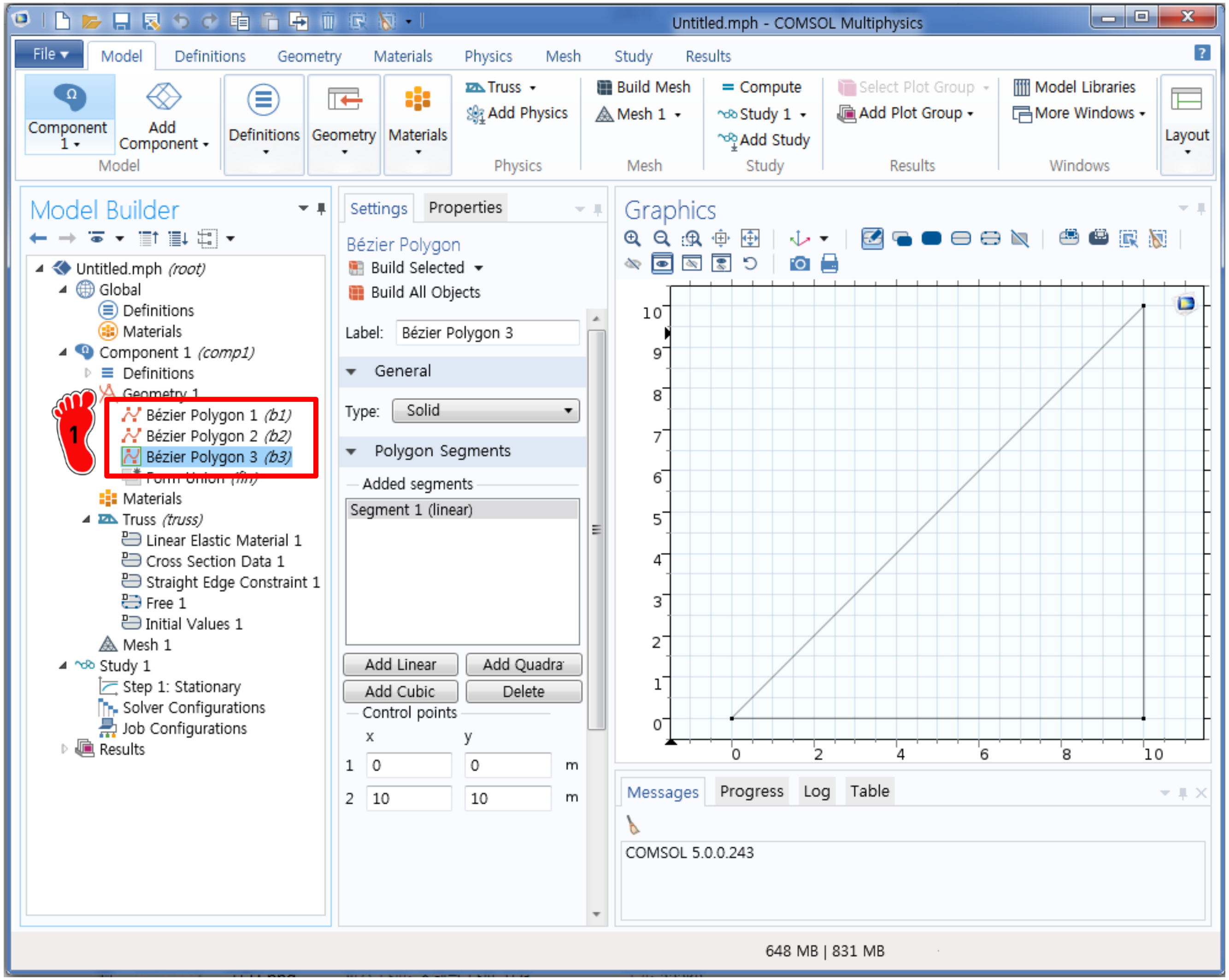
1 Control points 시작점 (0,0) 끝나는 점 (10,0) 을 입력

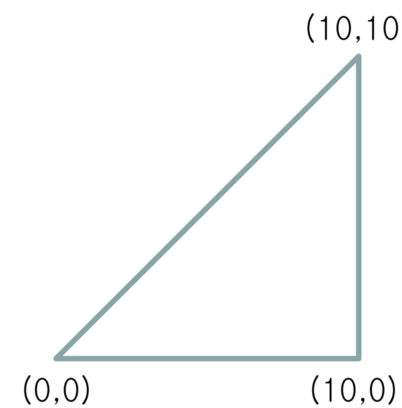
옆의 숫자 1 은 시작점 좌표, 2는 끝나는 점을 의미

2 Build Selected 클릭

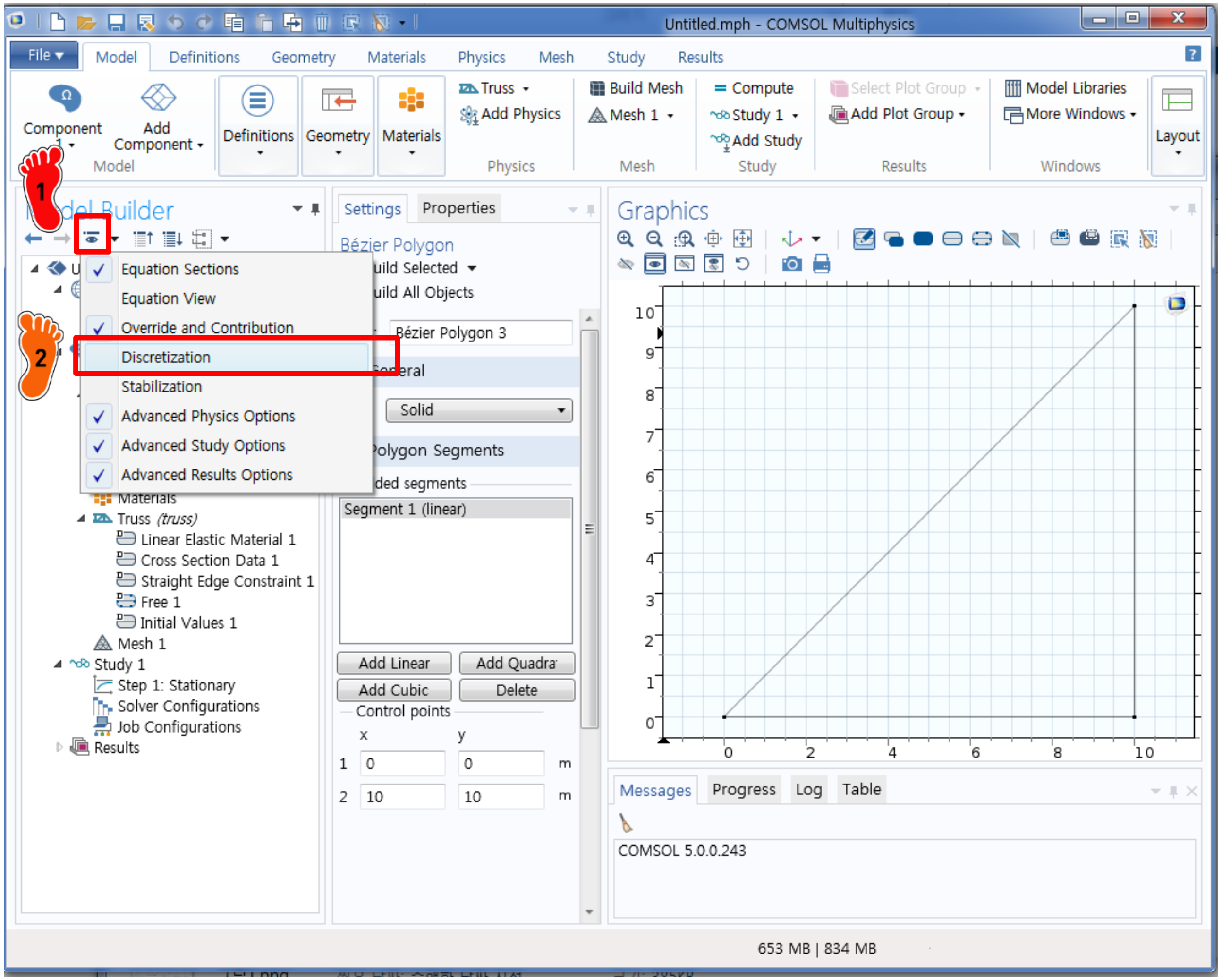
GEOMETRY CREATION

1 같은 방식으로 총 3개의 직선을 생성





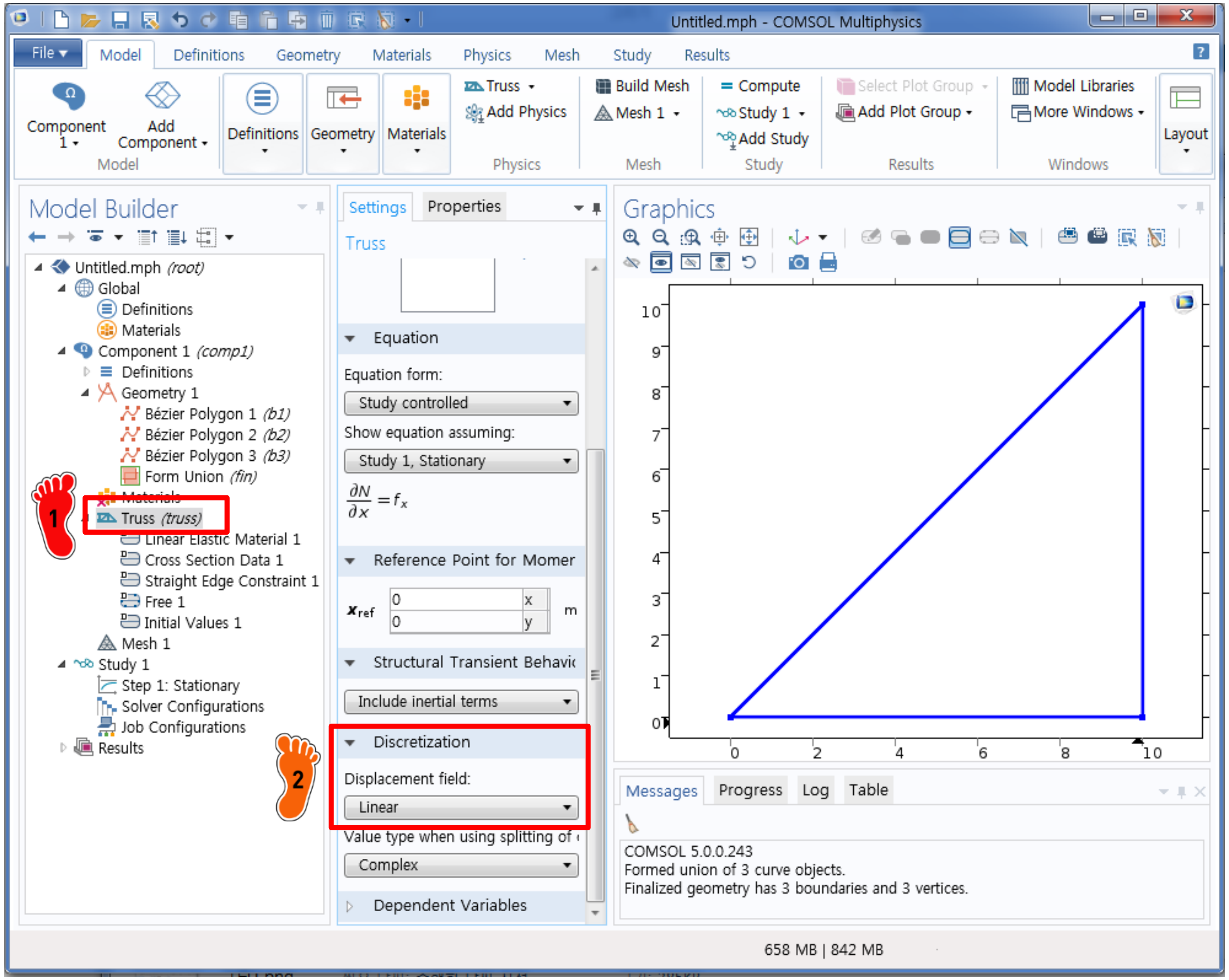
ELEMENT ORDER CHANGING



1 Model Builder 밑에 있는 눈 모양 메뉴 클릭

2 Discretization 클릭

ELEMENT ORDER CHANGING



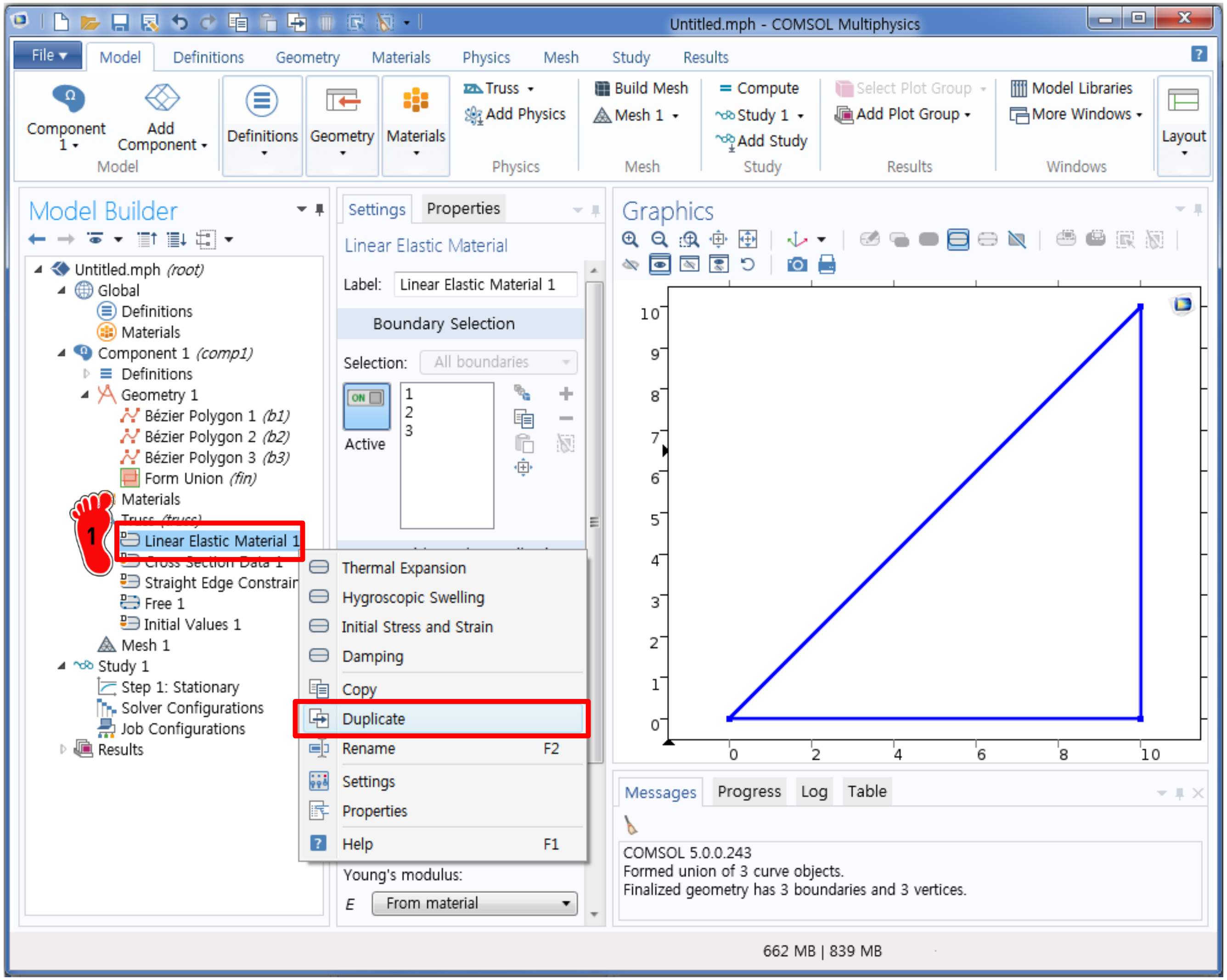
1 Truss physics 클릭

2 Displacement field 를 Linear 로 변경

• Direct Stiffness Method: Truss

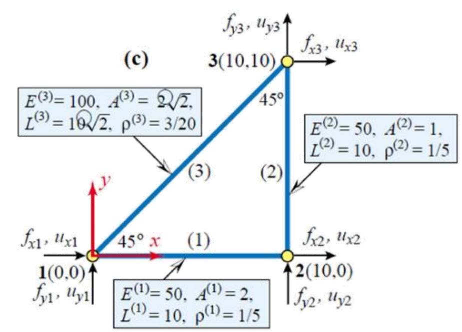
- ✓ Geometry creation
- ✓ Material property
- ✓ Cross section property
- ✓ Boundary condition
- ✓ Nodal force
- ✓ Analysis

MATERIAL PROPERTY



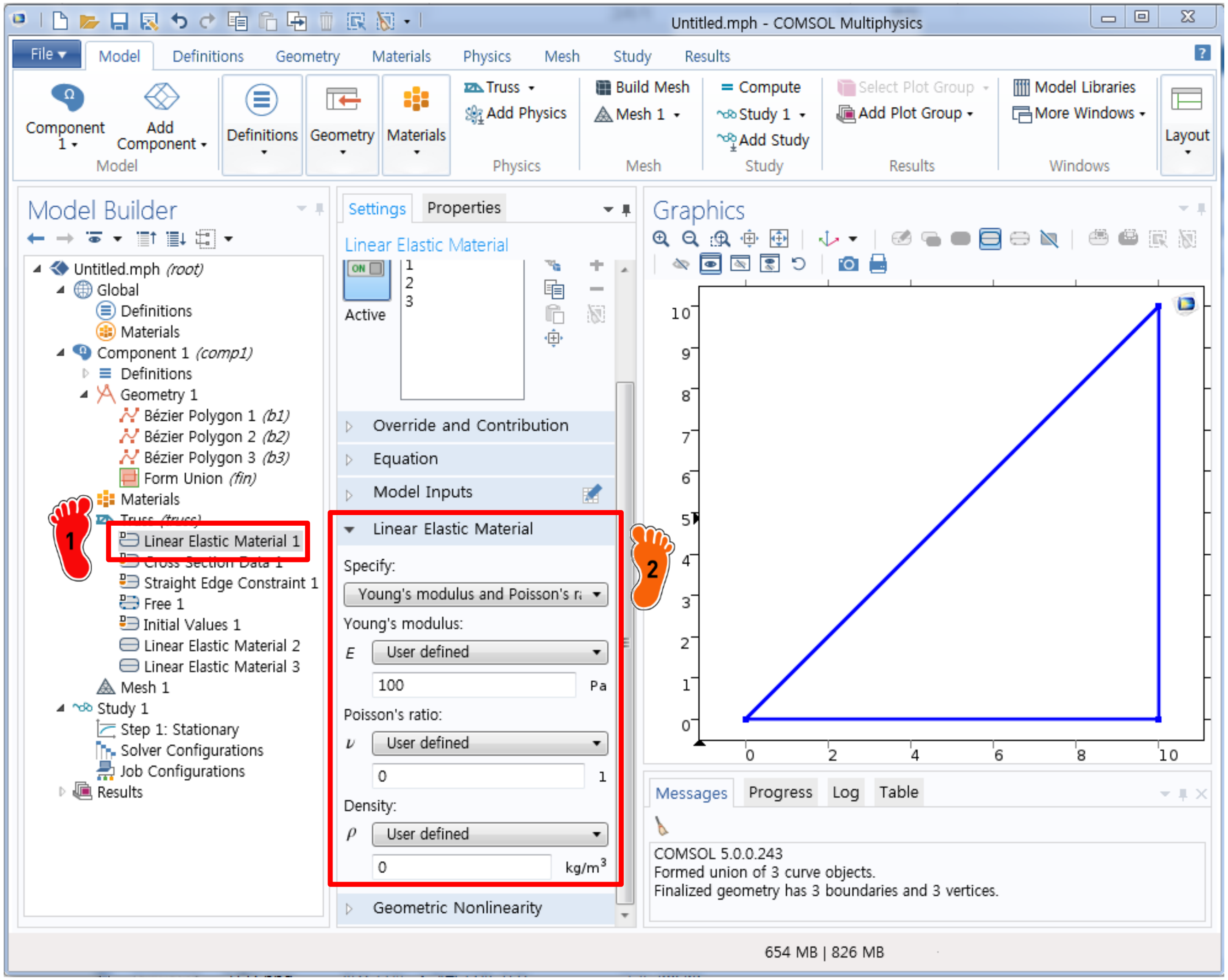
1 Linear Elastic Material 마우스 우클릭

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



Geometric, material and fabrication properties

MATERIAL PROPERTY

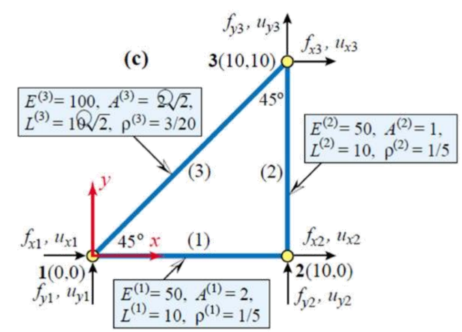


1. 첫 번째 Linear Elastic Material 클릭

2. From material을 User defined로 변경 후

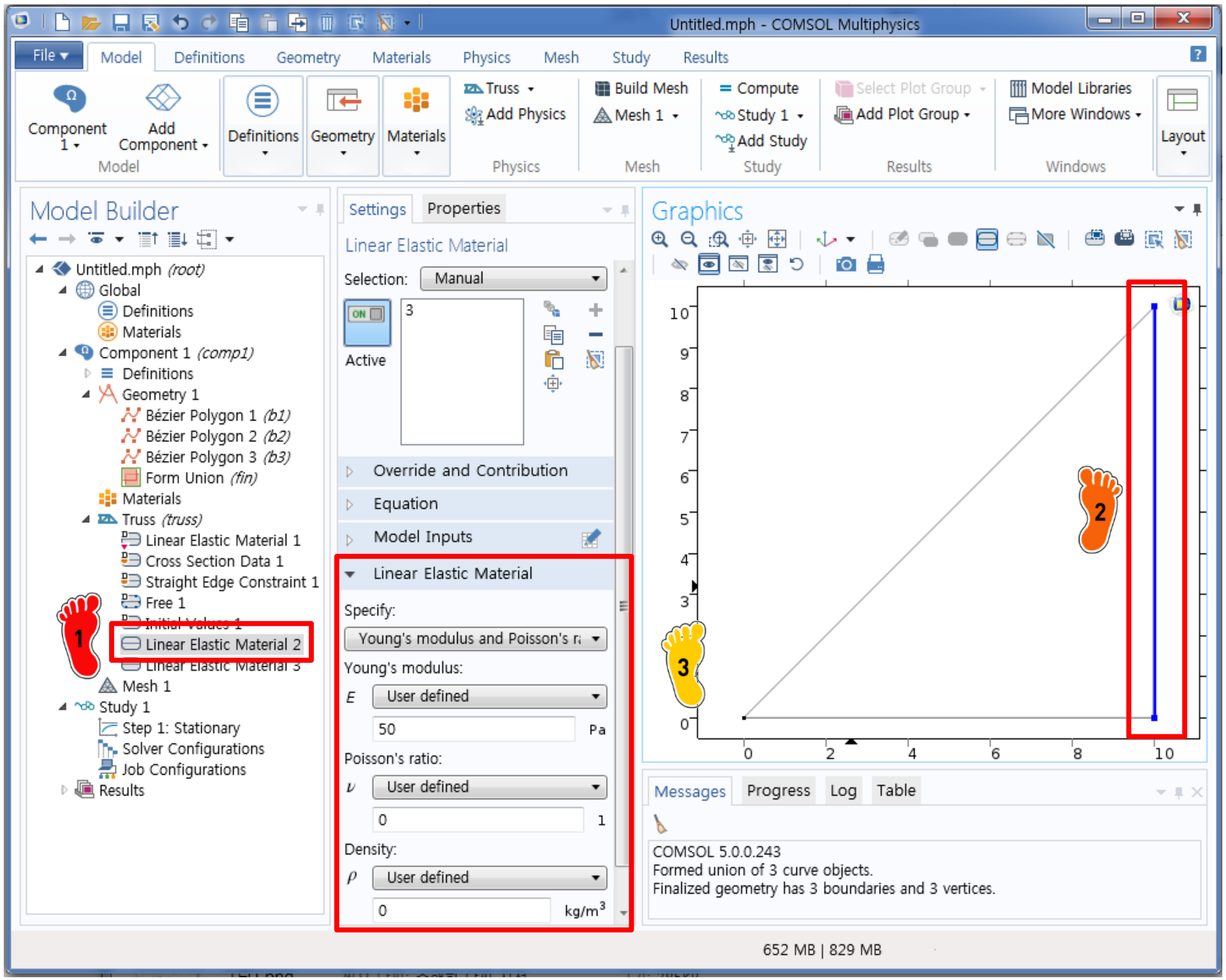
Young`s modulus 100 입력

푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

MATERIAL PROPERTY

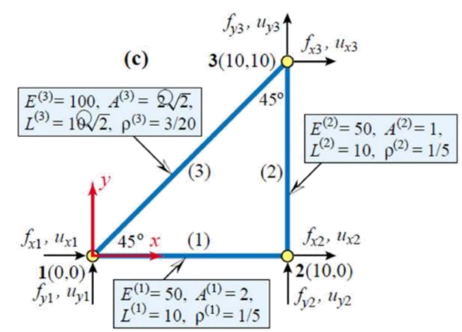


1 두 번째 Linear Elastic Material 클릭

2 제일 오른쪽 부재 선택

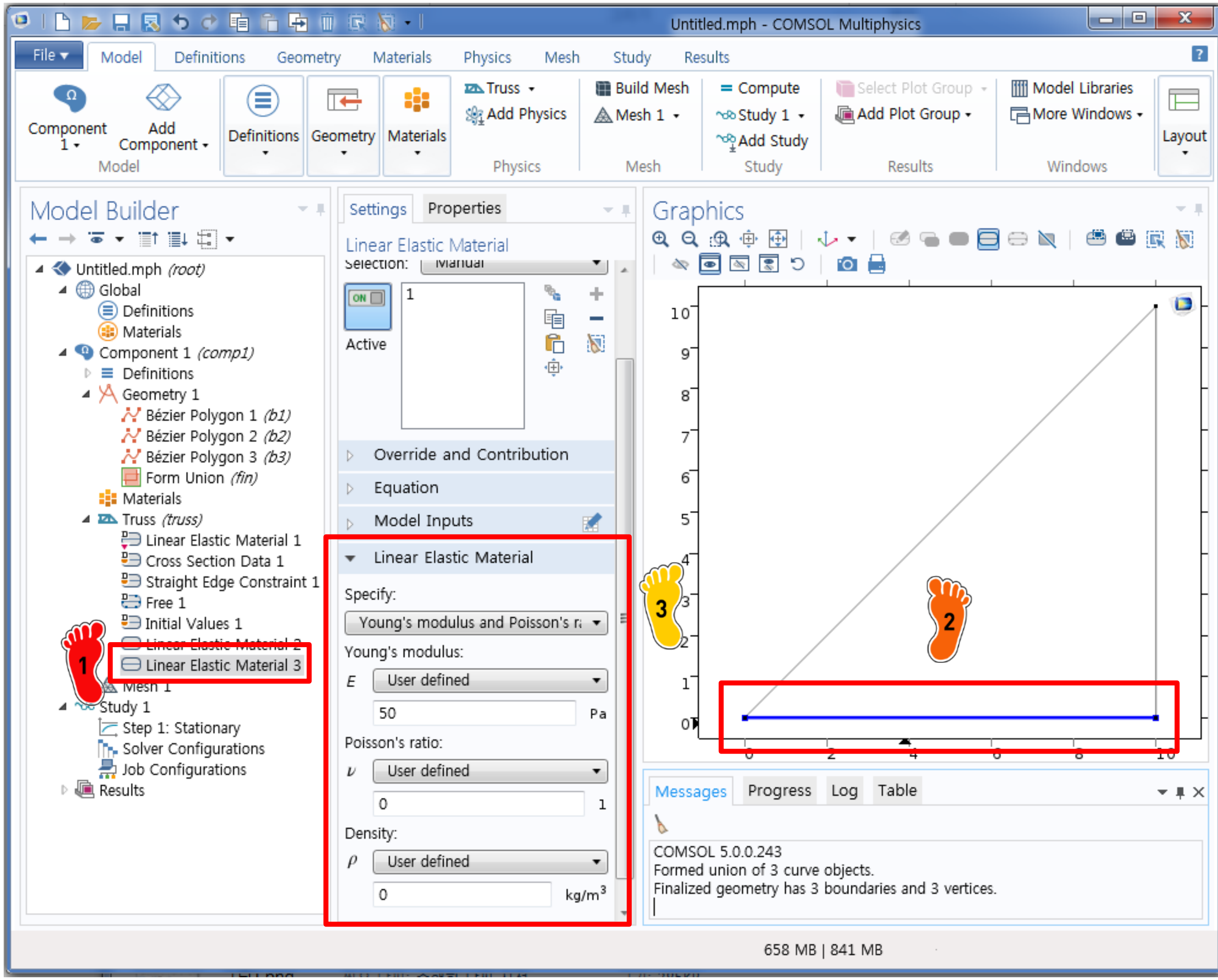
3 Young`s modulus 50 입력

푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

MATERIAL PROPERTY

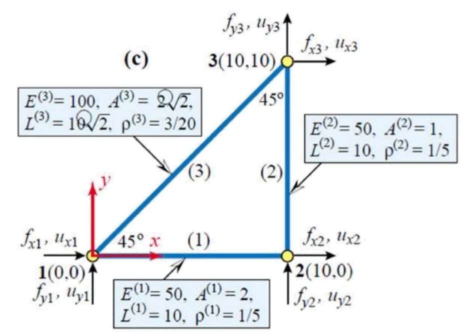


1 마지막 Linear Elastic Material 클릭

2 제일 아래 부재 선택

3 Young`s modulus 50 입력

푸아송비와 밀도는 0 입력

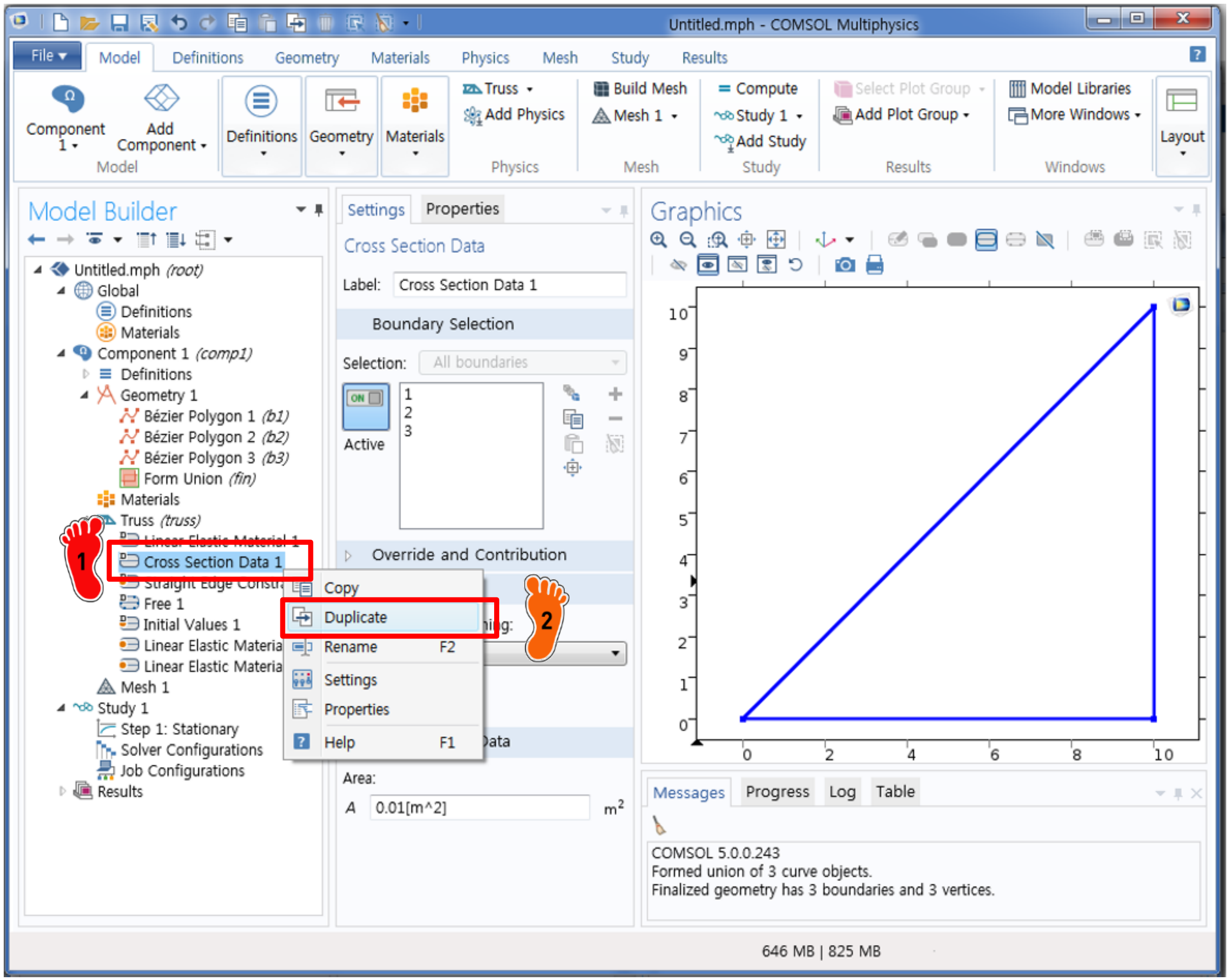


Geometric, material and fabrication properties

- **Direct Stiffness Method: Truss**

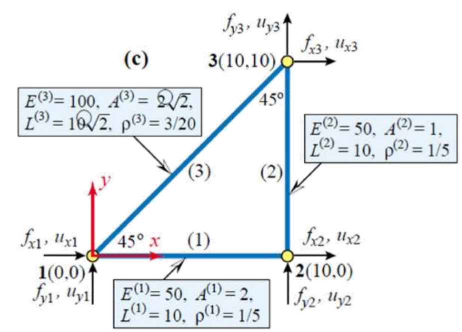
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ **Analysis**

SECTION PROPERTY



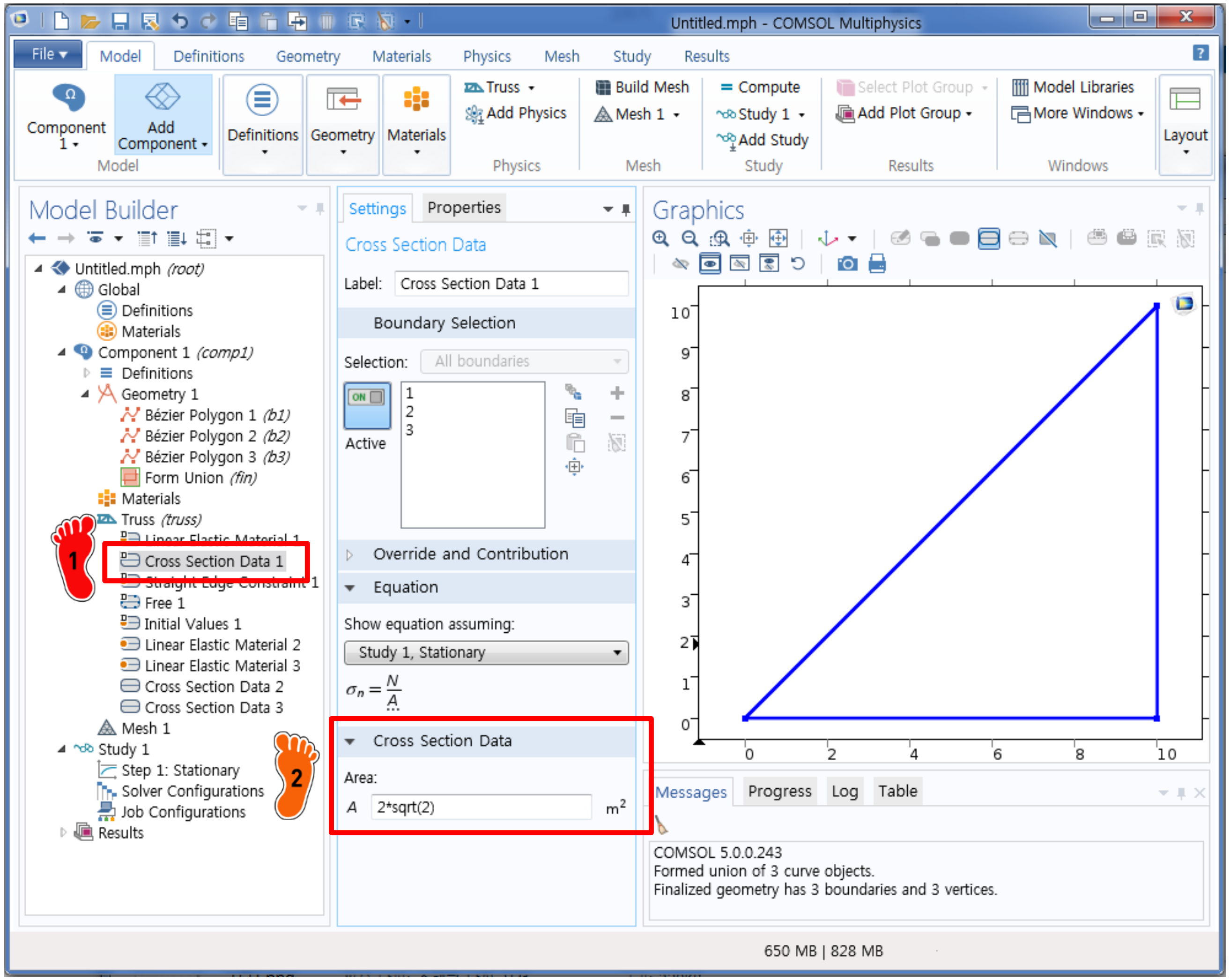
1. Cross Section Data 마우스 우클릭

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



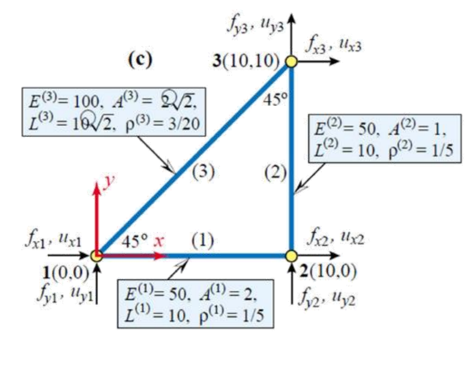
Geometric, material and fabrication properties

SECTION PROPERTY



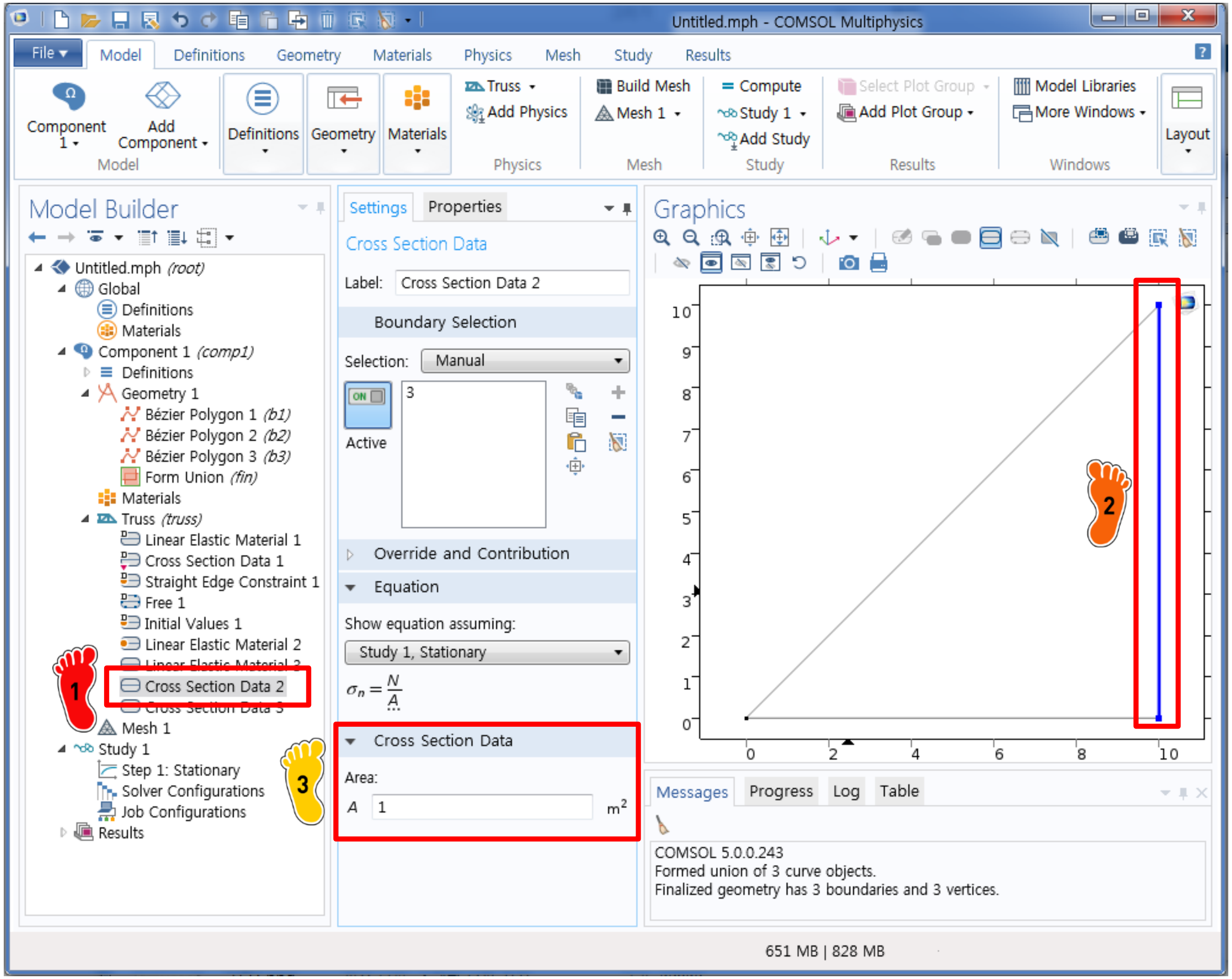
1. 첫 번째 Cross Section Data 클릭

2. Area 에 2*sqrt(2) 입력

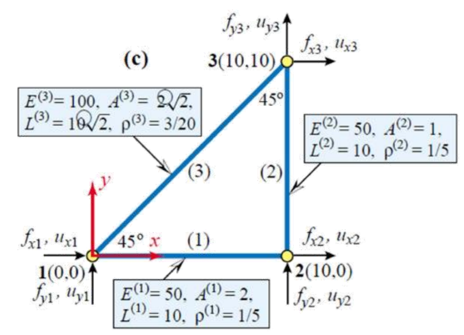


Geometric, material and fabrication properties

SECTION PROPERTY

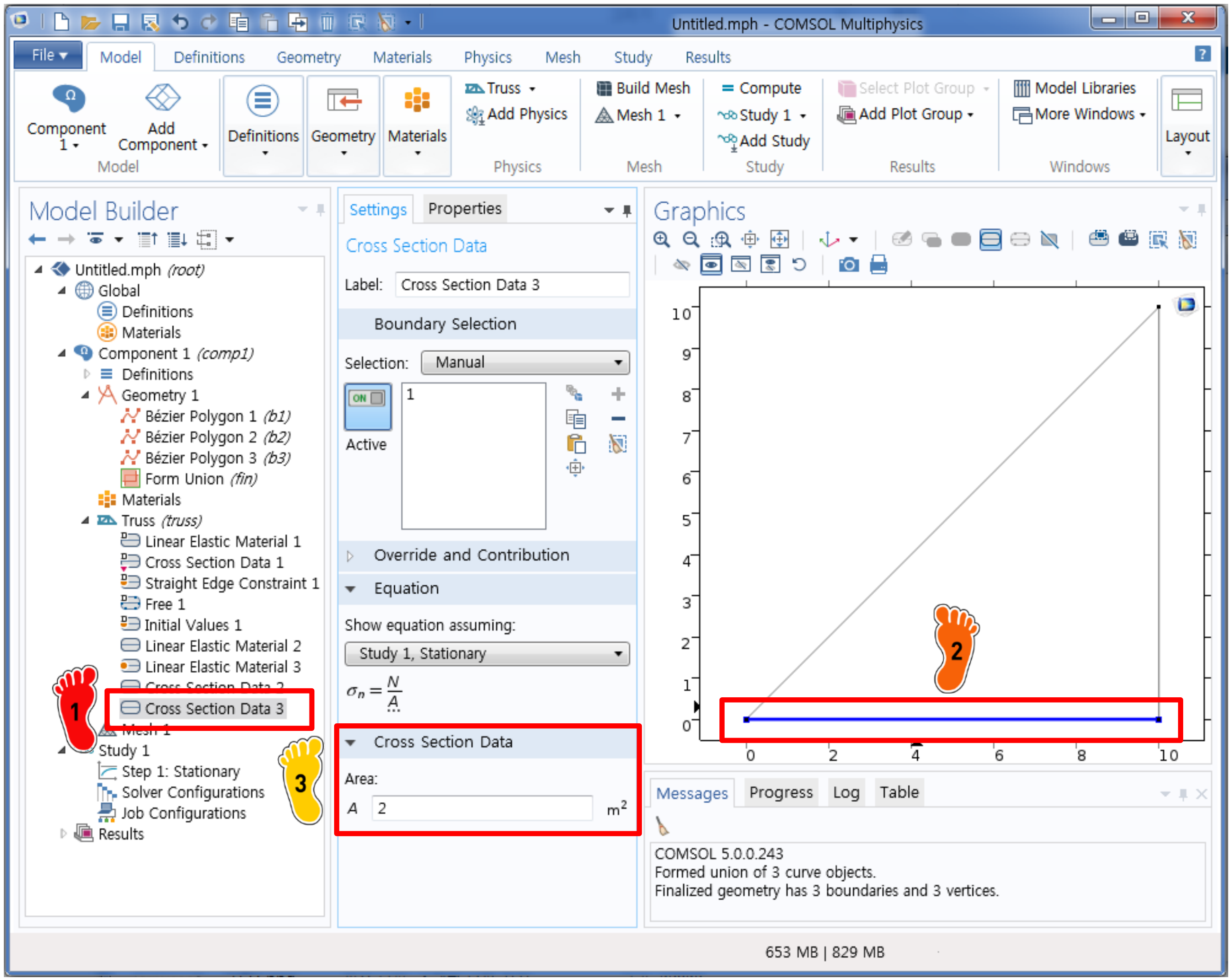


1. 두 번째 Cross Section Data 클릭
2. 제일 오른쪽 부재 선택
3. Area 에 1 입력



Geometric, material and fabrication properties

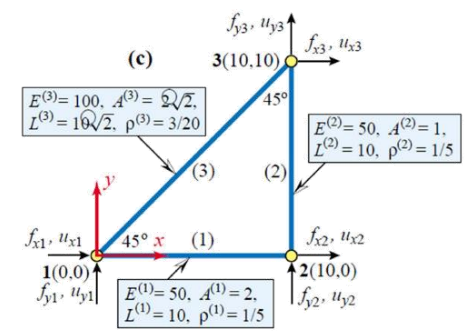
SECTION PROPERTY



1 마지막 Cross Section Data 클릭

2 제일 아래 부재 선택

3 Area 에 2 입력



Geometric, material and fabrication properties

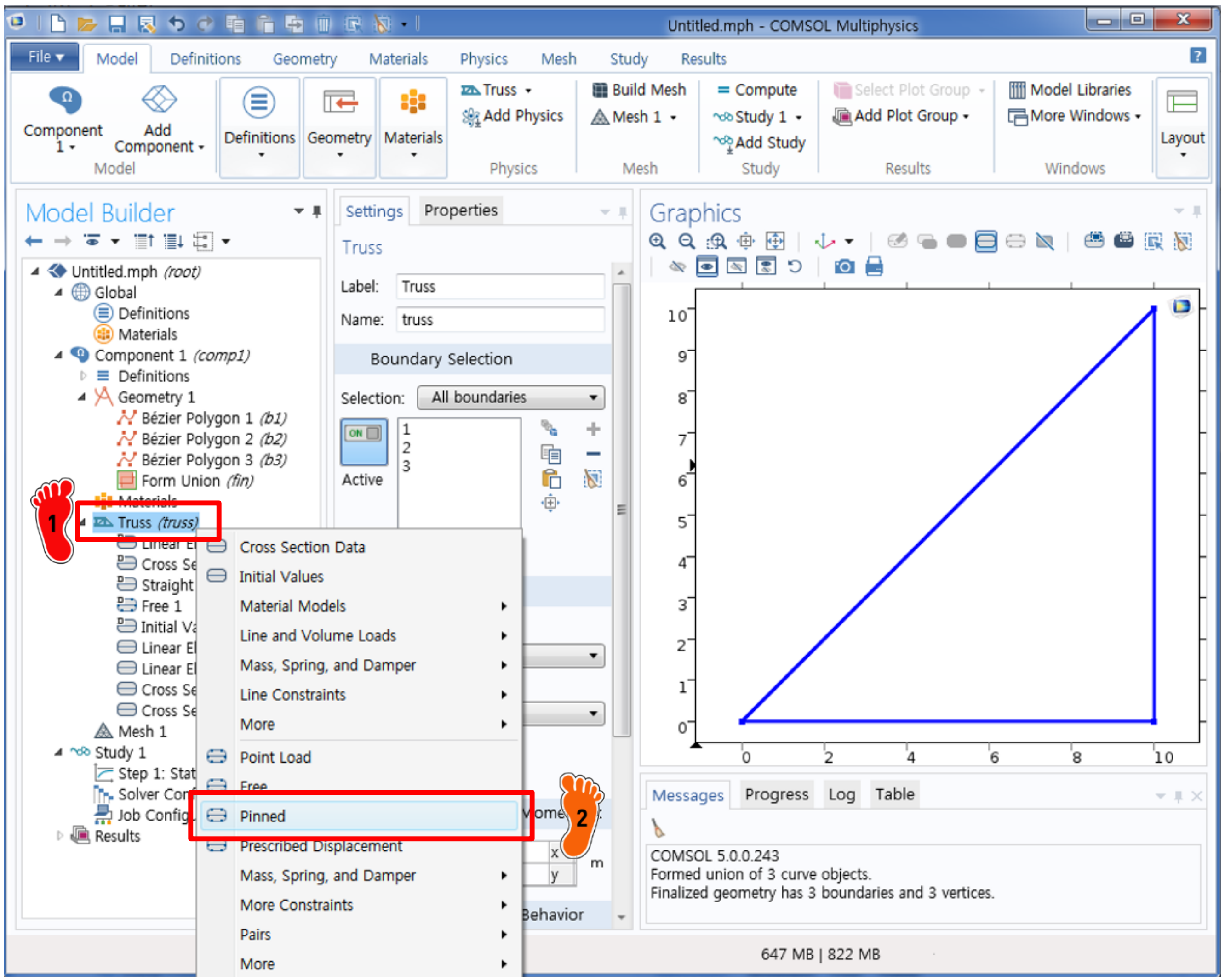
- **Direct Stiffness Method: Truss**

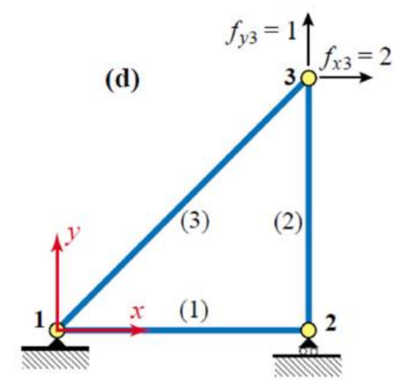
  - ✓ Geometry creation
  - ✓ Material property
  - ✓ Cross section property
  - ✓ <u>Boundary condition</u>
  - ✓ Nodal force
  - ✓ Analysis

BOUNDARY CONDITION

1 Truss 마우스 우클릭

2 Pinned 클릭



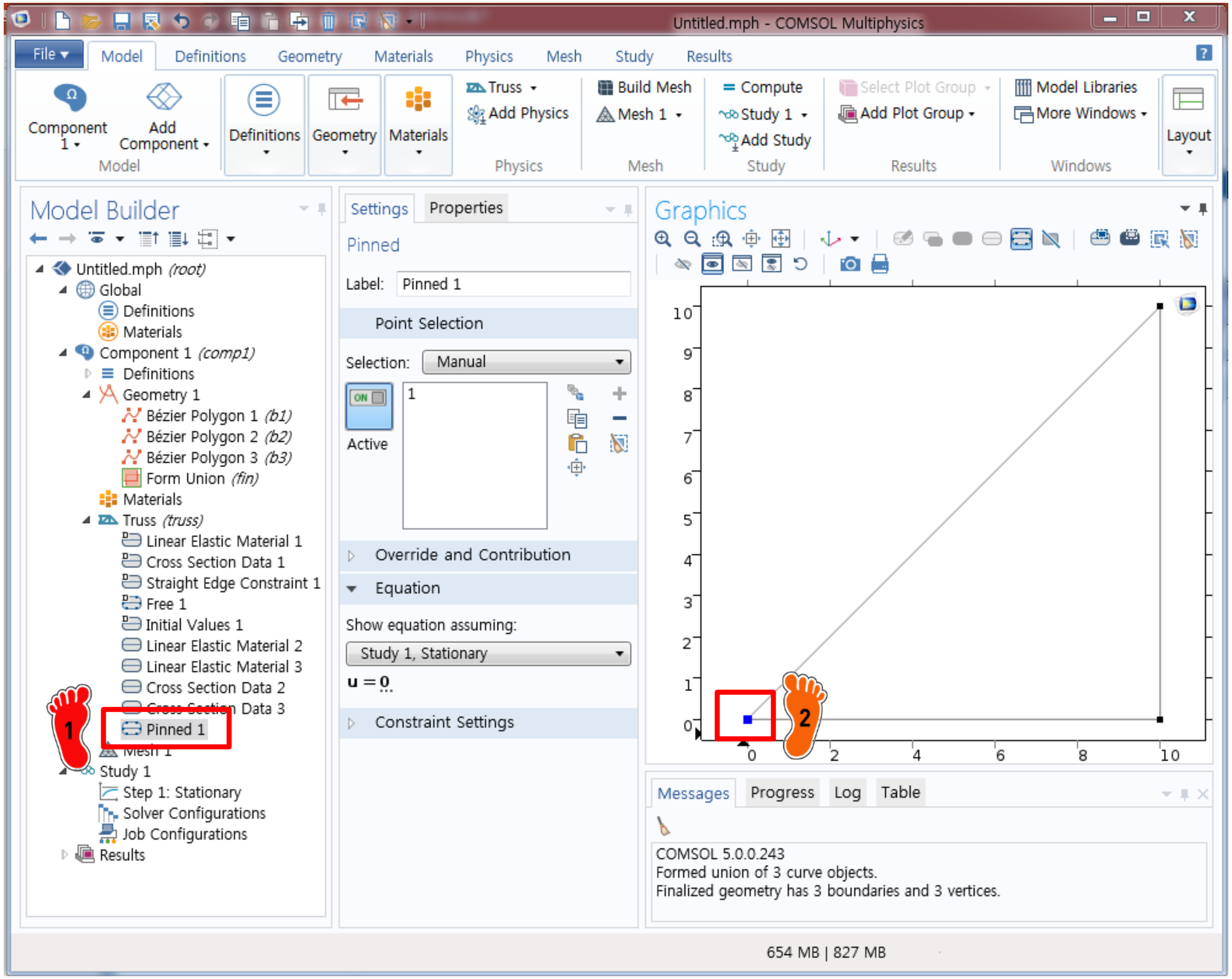


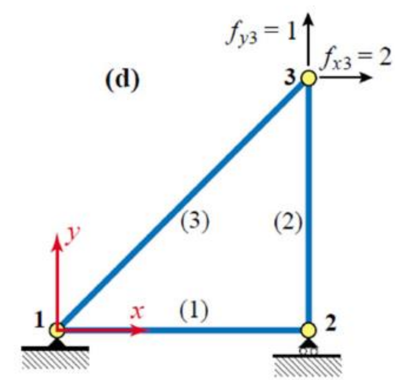
Support conditions and applied loads

BOUNDARY CONDITION

1 Pinned 클릭

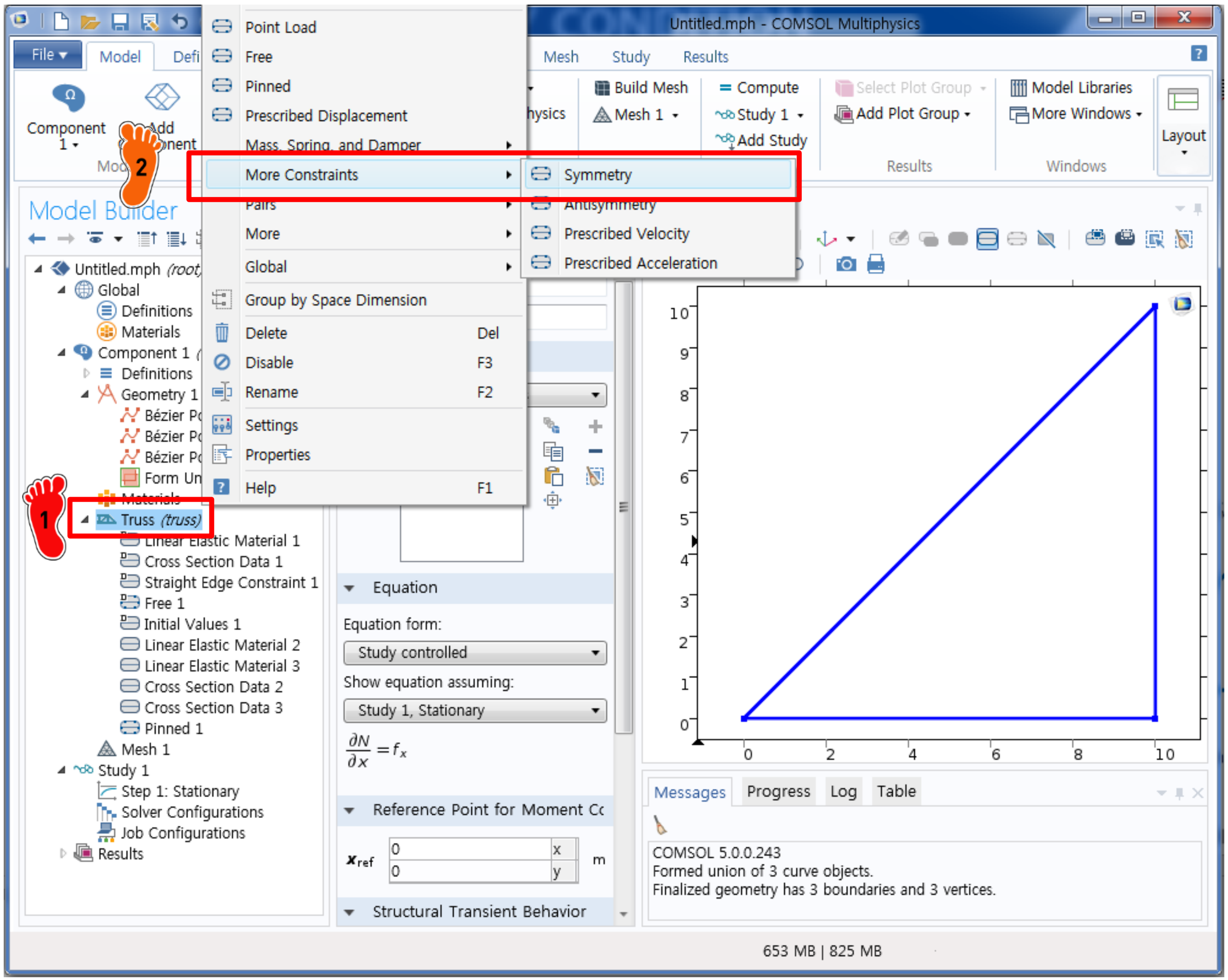
2 왼쪽 아래 절점 선택





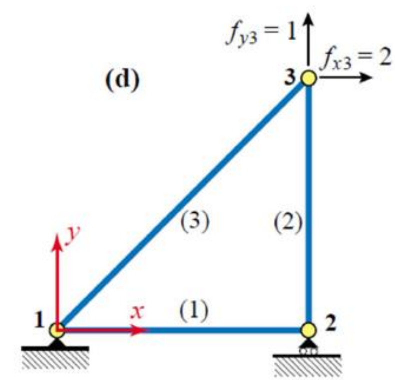
Support conditions and applied loads

BOUNDARY CONDITION



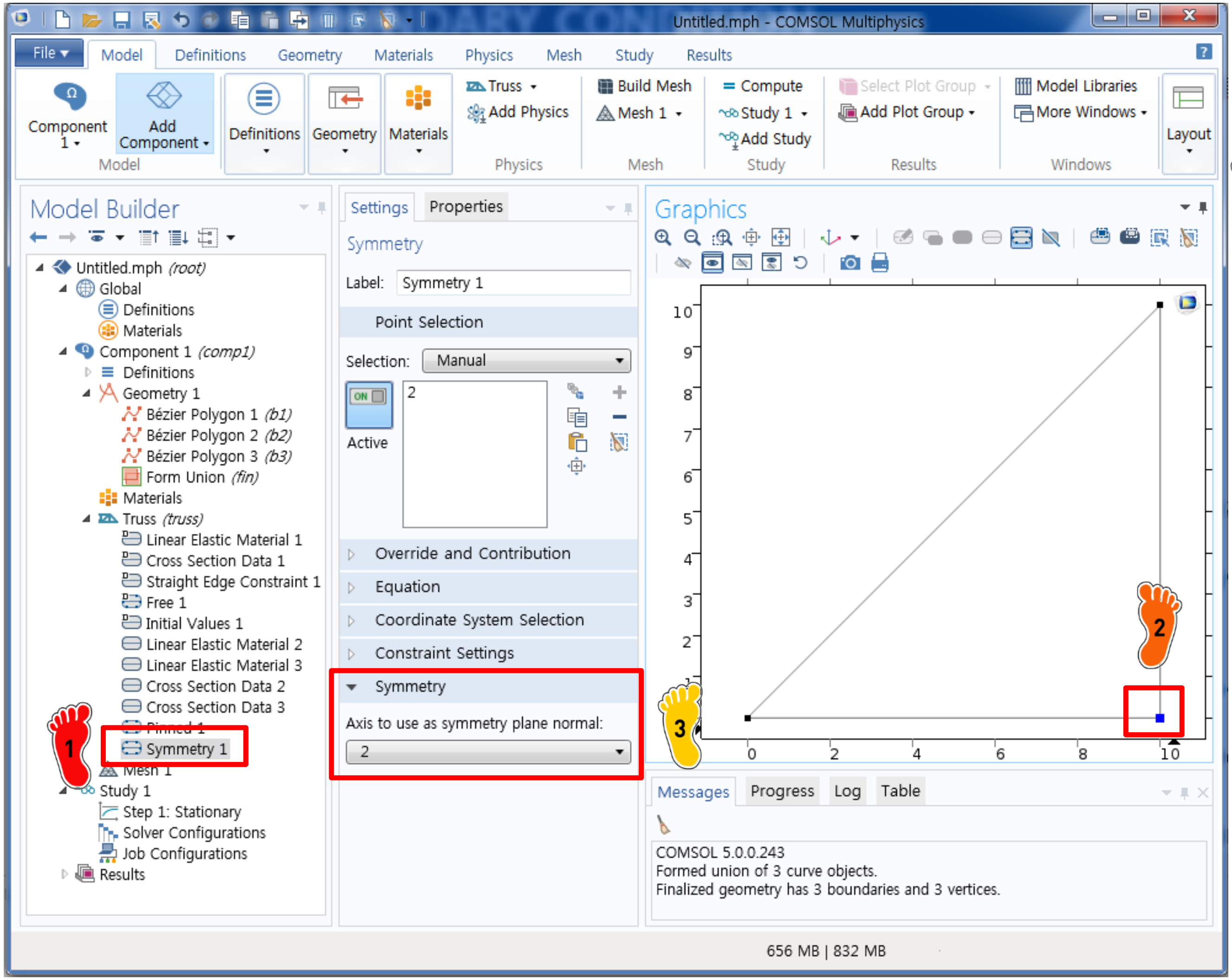
1 Truss 마우스 우클릭

2 More Constraints에서 Symmetry 클릭



Support conditions and applied loads

BOUNDARY CONDITION

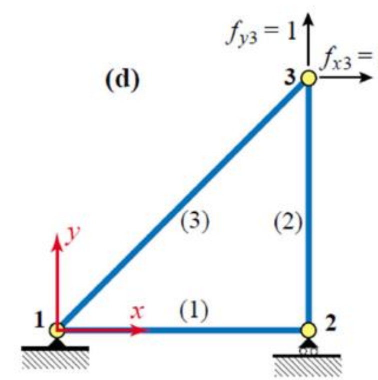


1 Symmetry 1 클릭

2 오른쪽 아래 절점 선택

3 2 로 변경

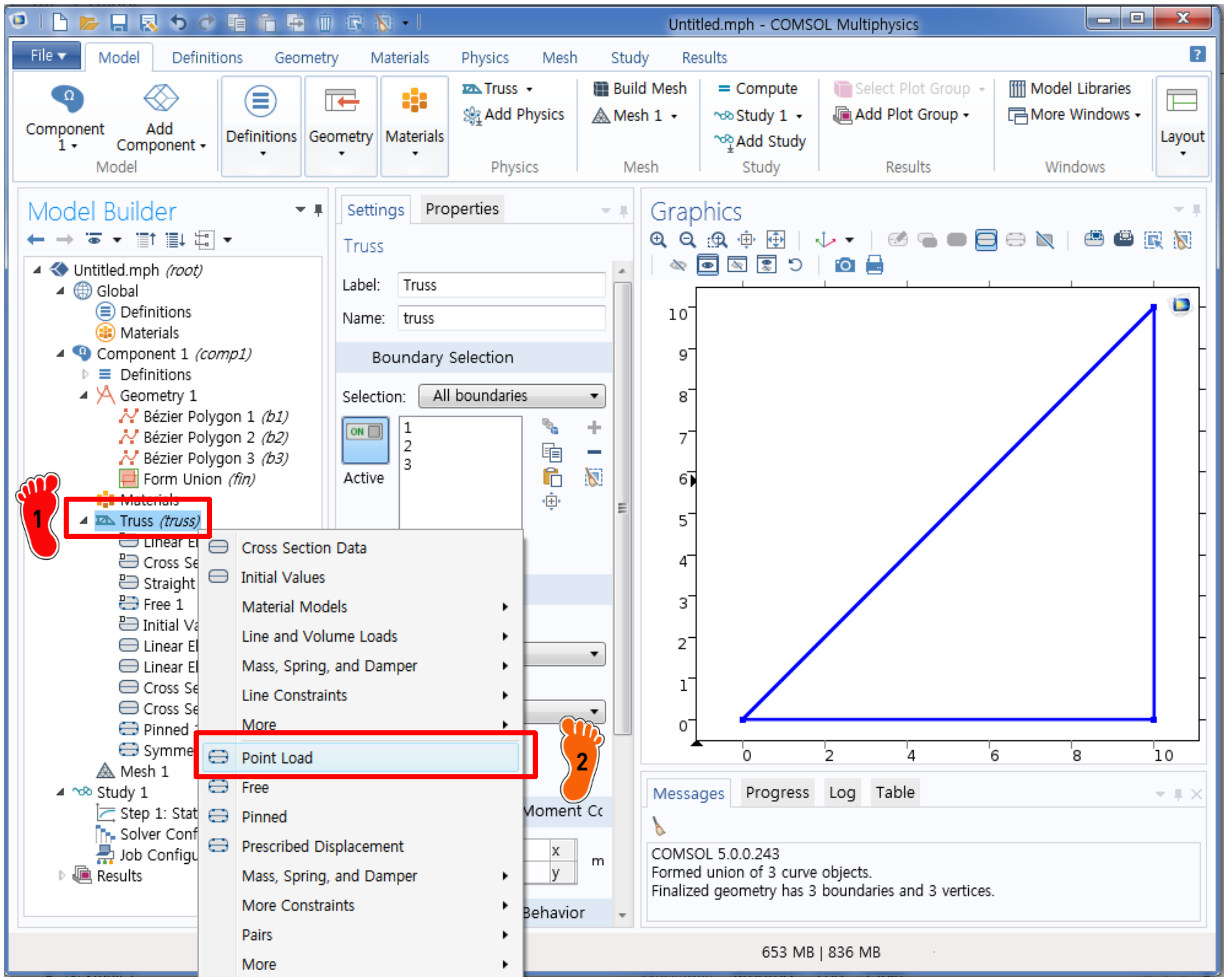
1의 의미는 x 방향에 수직한 평면, 2는 y 방향에 수직한 평면



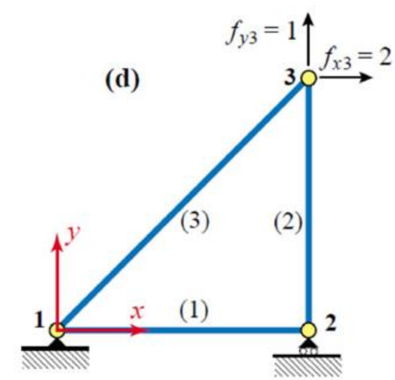
Support conditions and applied loads

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

POINT LOAD

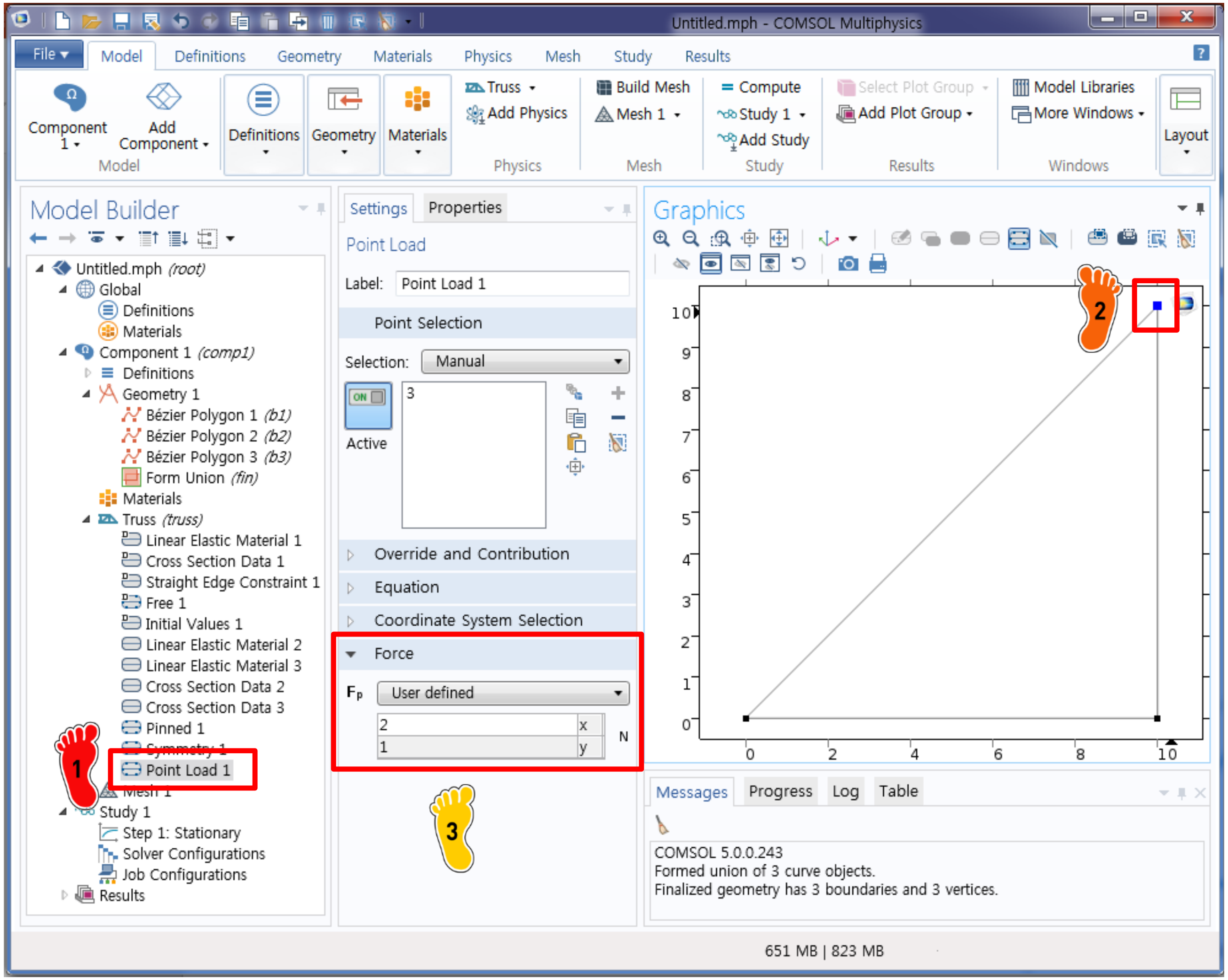


1. Truss 마우스 우클릭
2. Point Load 클릭



Support conditions and applied loads

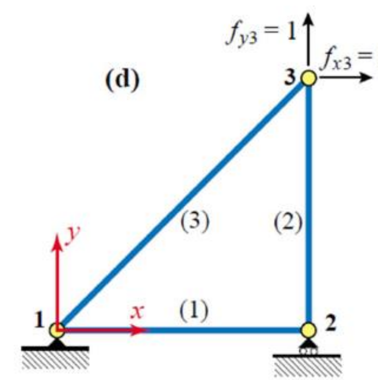
POINT LOAD



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**

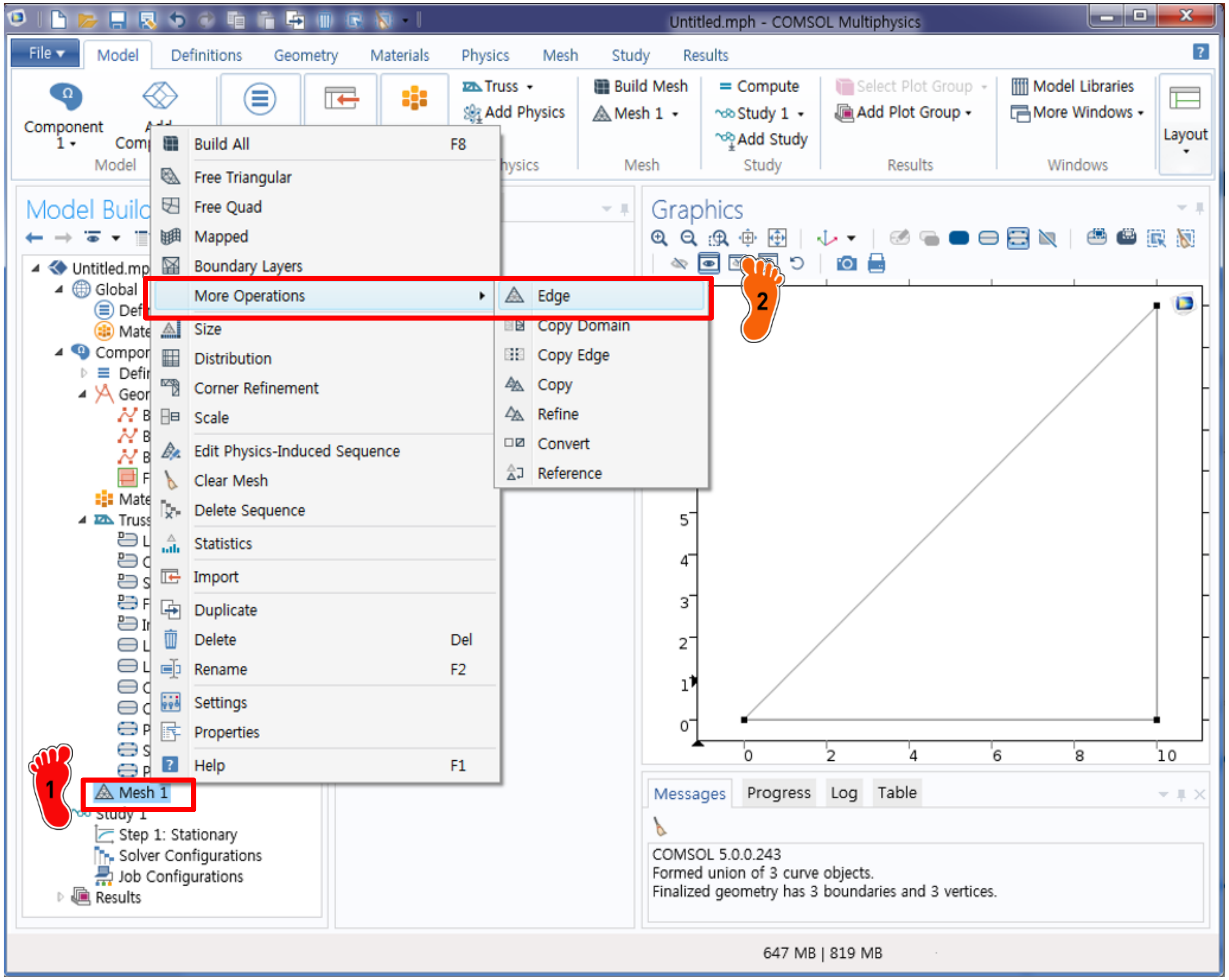
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ <u>**Analysis**</u>

MESH CREATION

Mesh 1 마우스 우클릭

More Operations에서

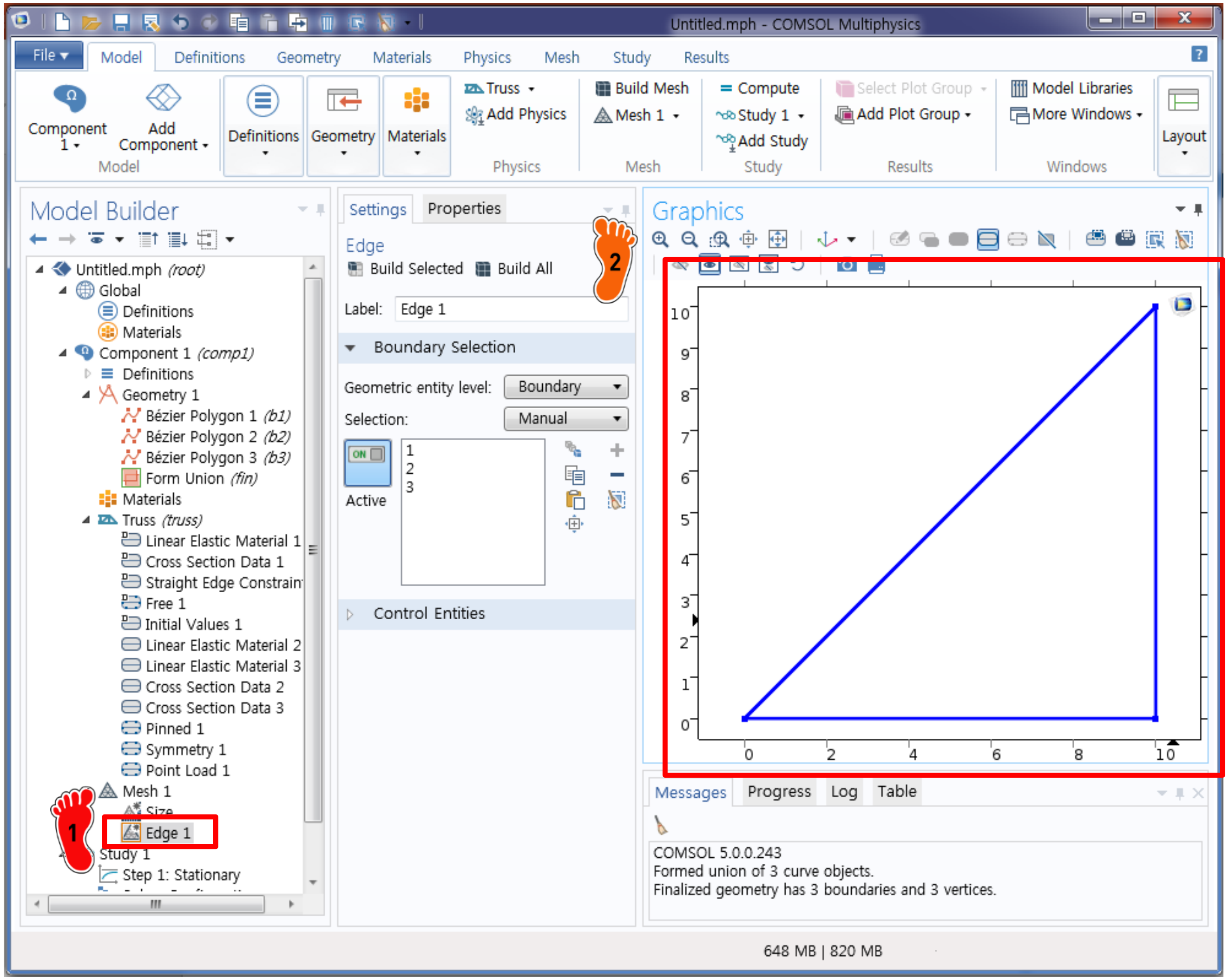
Edge 클릭



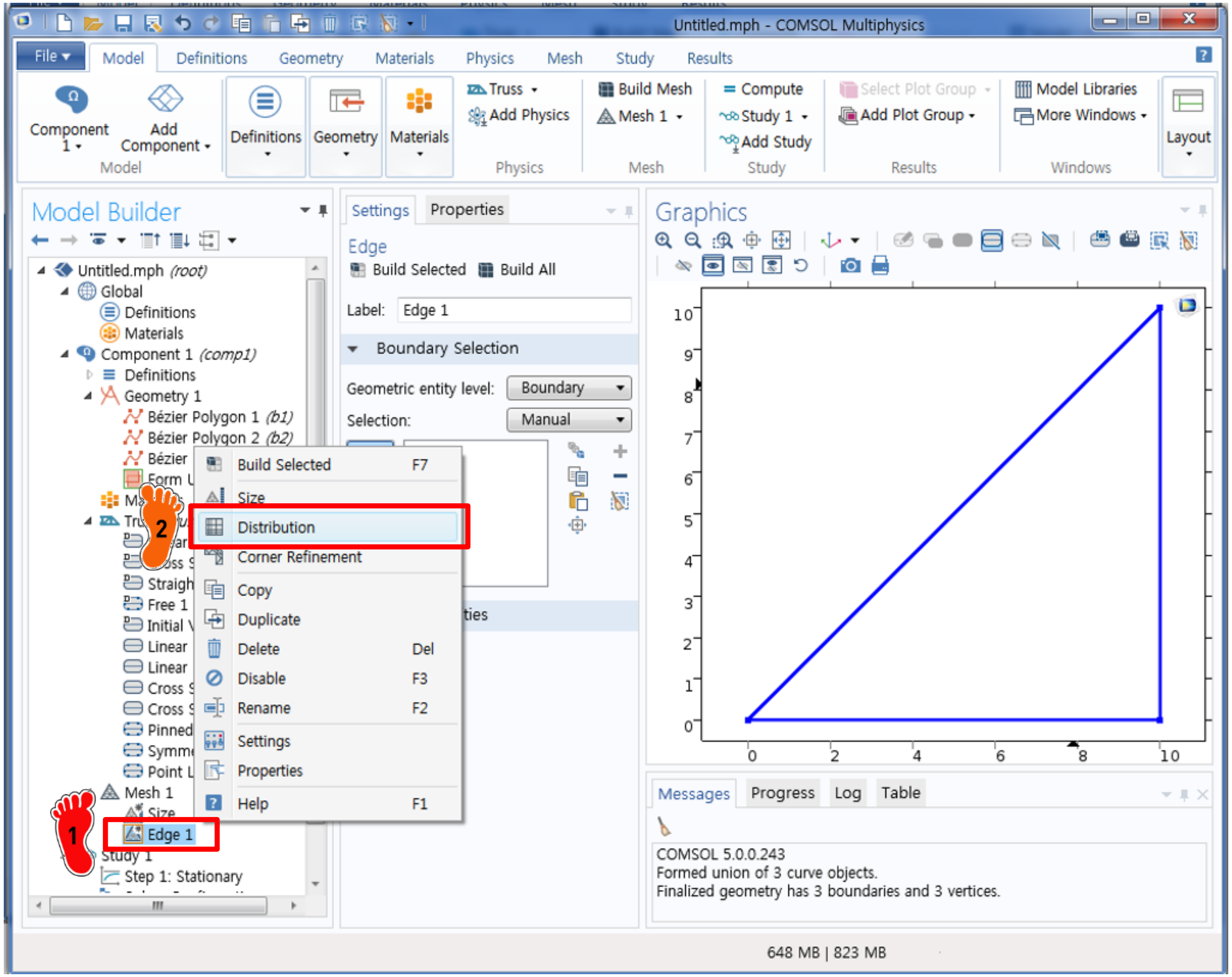
MESH CREATION

1 Edge 1 클릭

2 부재 세 개 선택



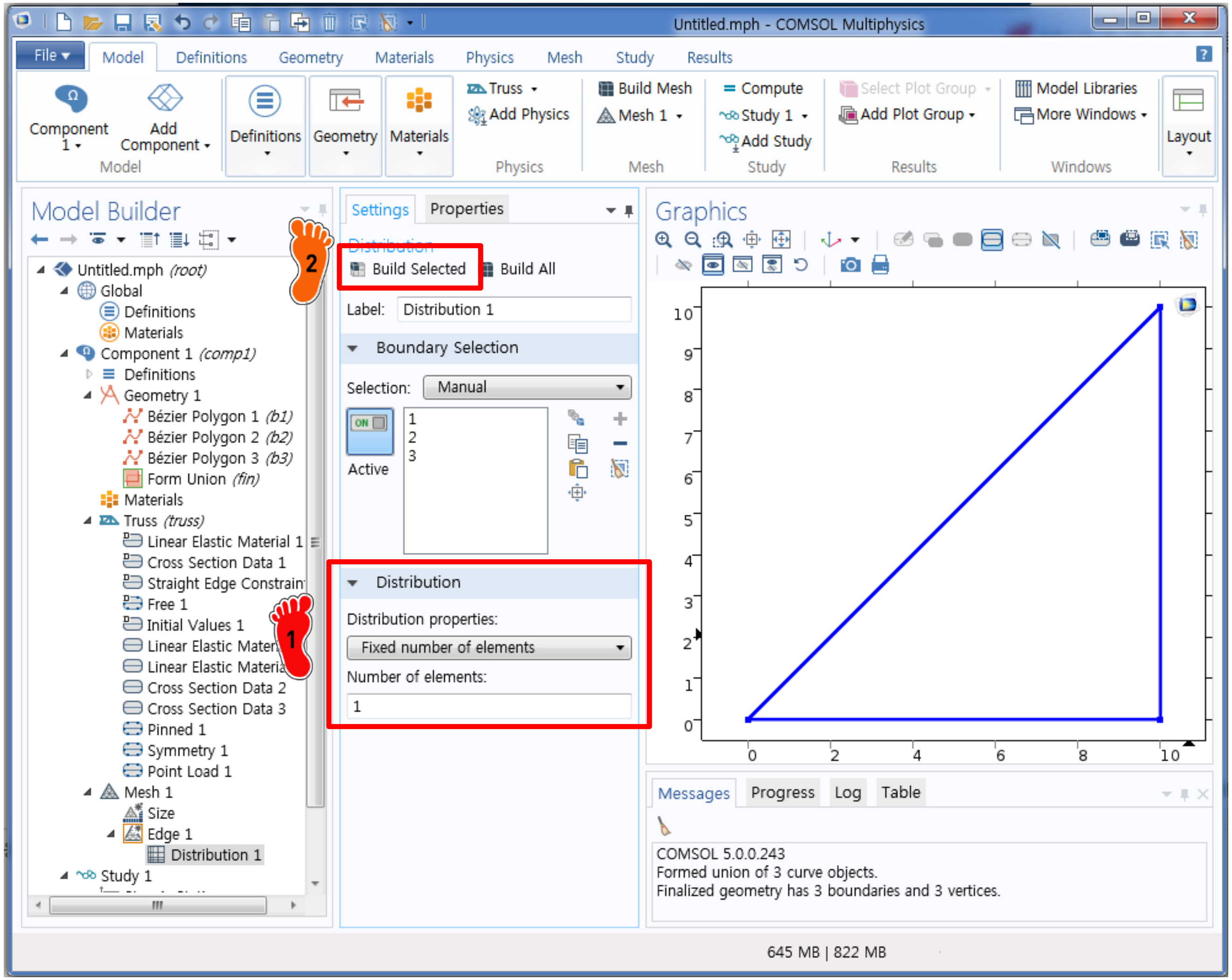
MESH CREATION



1 Edge 1 우클릭

2 Distribution 클릭

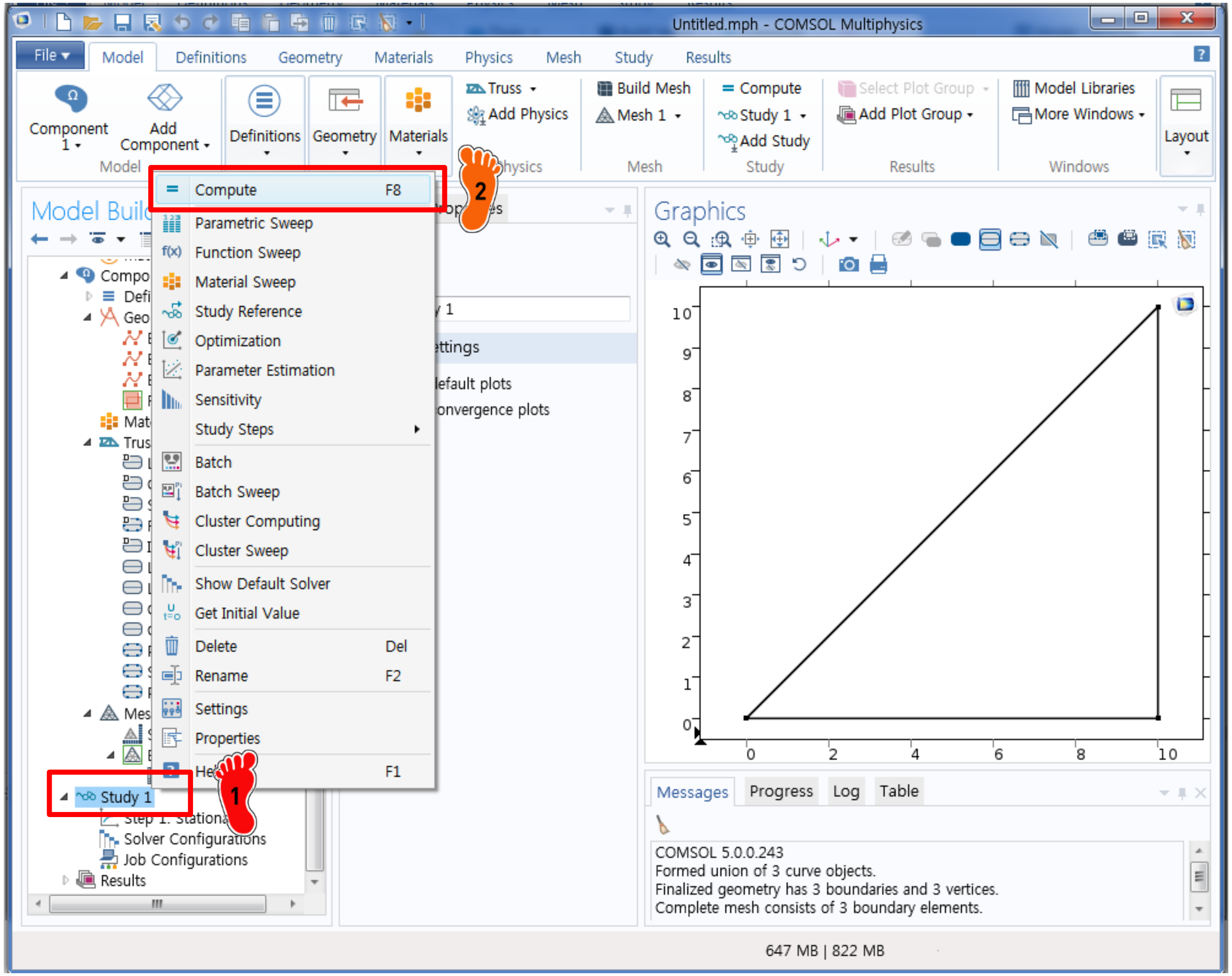
MESH CREATION



1 Number of elements 1 입력

2 Build Selected 클릭

COMPUTE



1 Study 1 마우스 우클릭

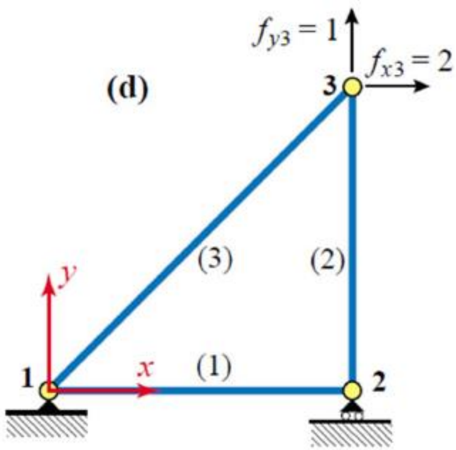
2 Compute 클릭

- **Stiffness matrix**

  - ✓ **<u>Global stiffness matrix by hand</u>**

  - ✓ **Global stiffness matrix by COMSOL**

ASSEMBLE OPTION



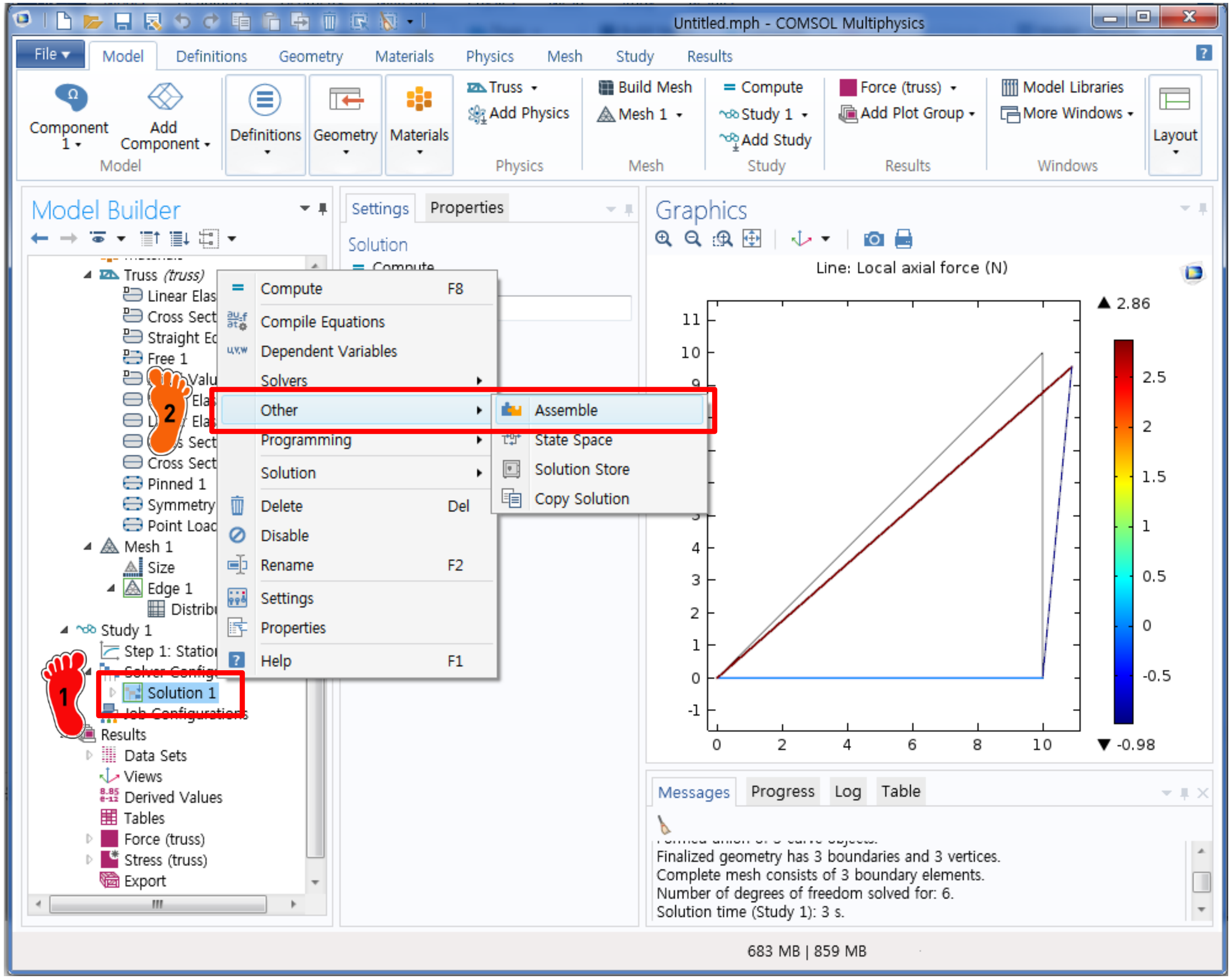
Support conditions and applied loads

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

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

- **Stiffness matrix confirm**
  - ✓ **Global stiffness matrix by hand**
  - ✓ **<u>Global stiffness matrix by COMSOL</u>**

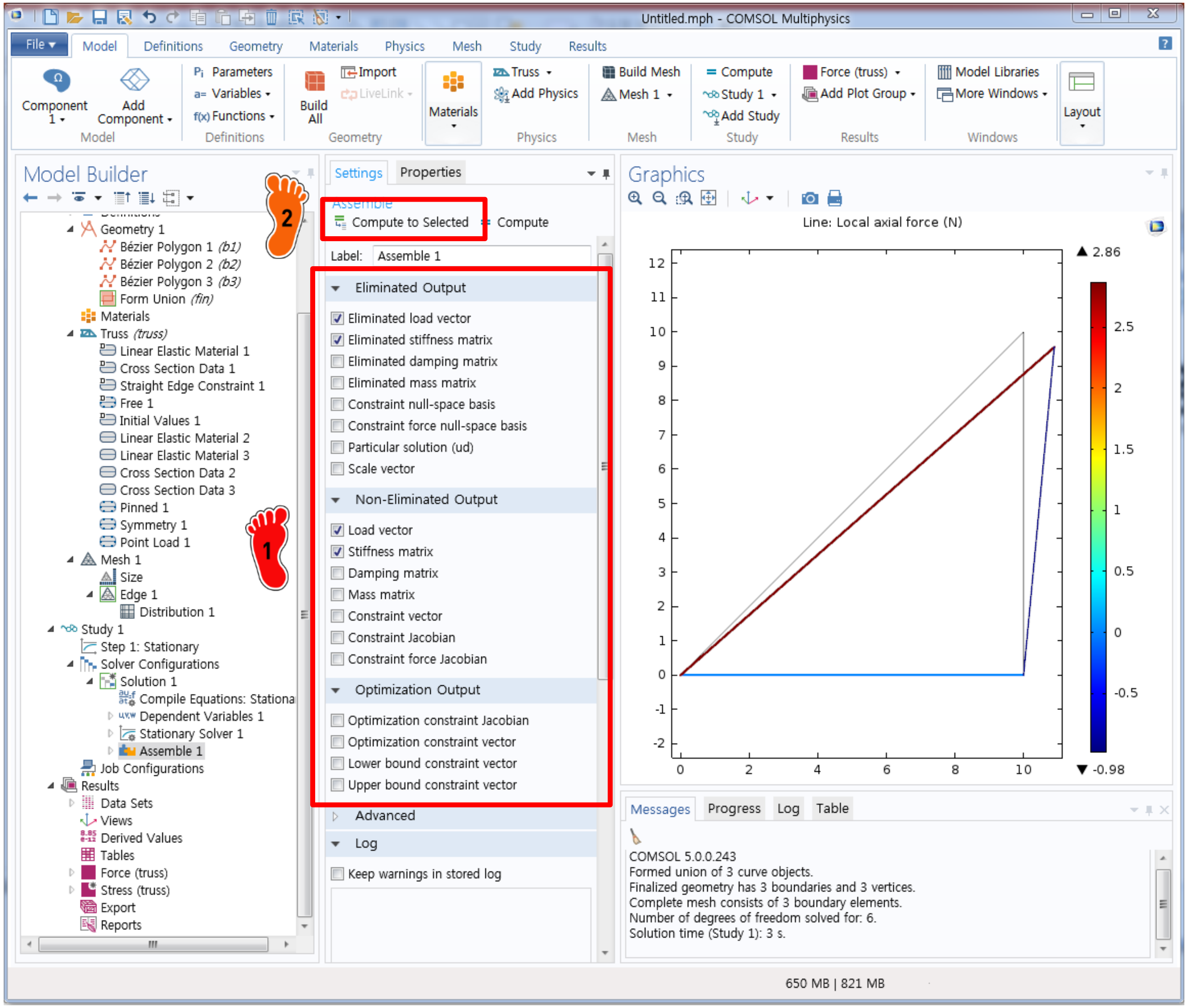
ASSEMBLE OPTION



1 Solver 1 마우스 우클릭

2 Other에서 Assemble 클릭

ASSEMBLE OPTION



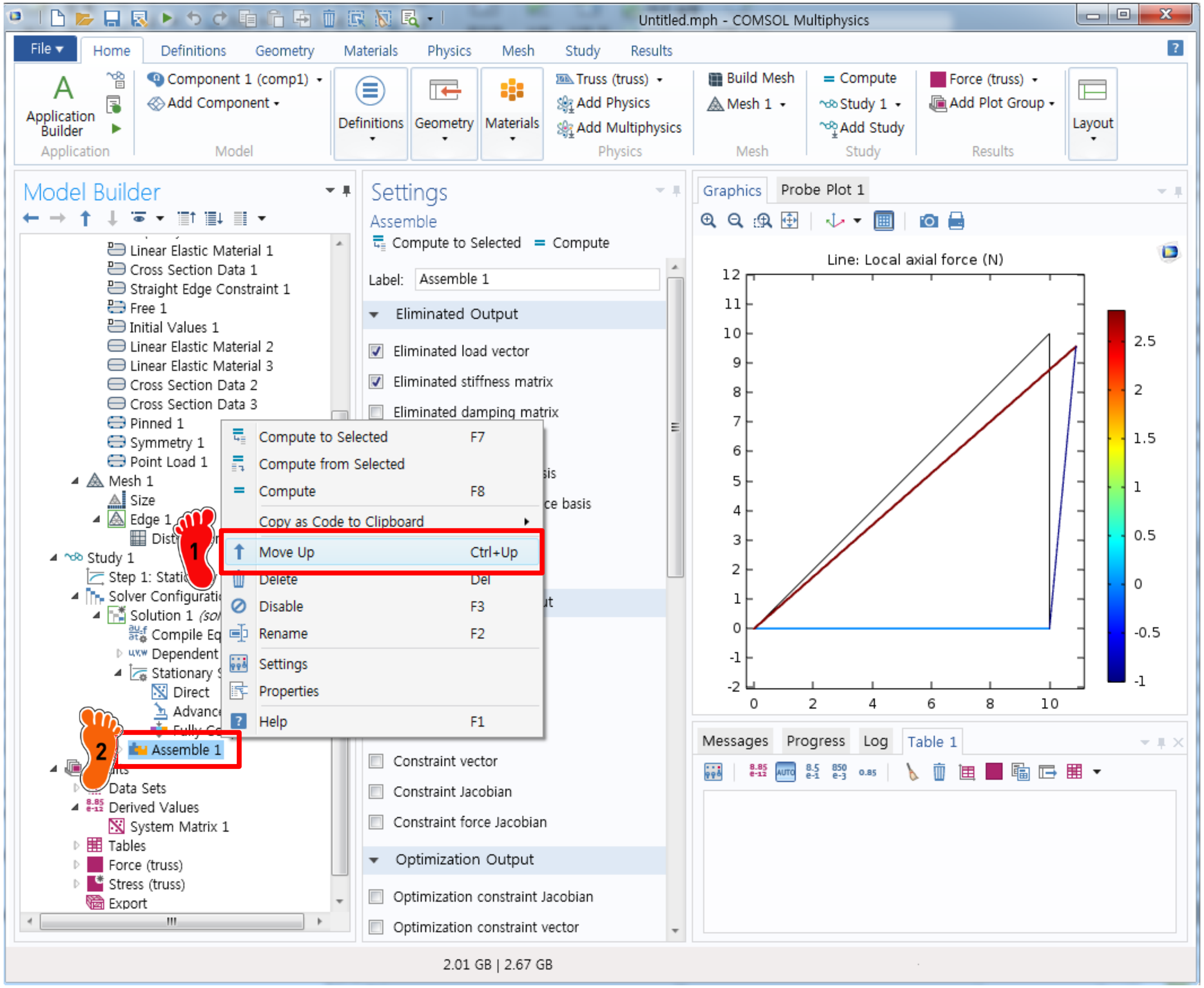
1 Eliminated load vector
Eliminated stiffness matrix

Load vector
Stiffness matrix

체크박스에 체크

2 Compute to Selected 클릭

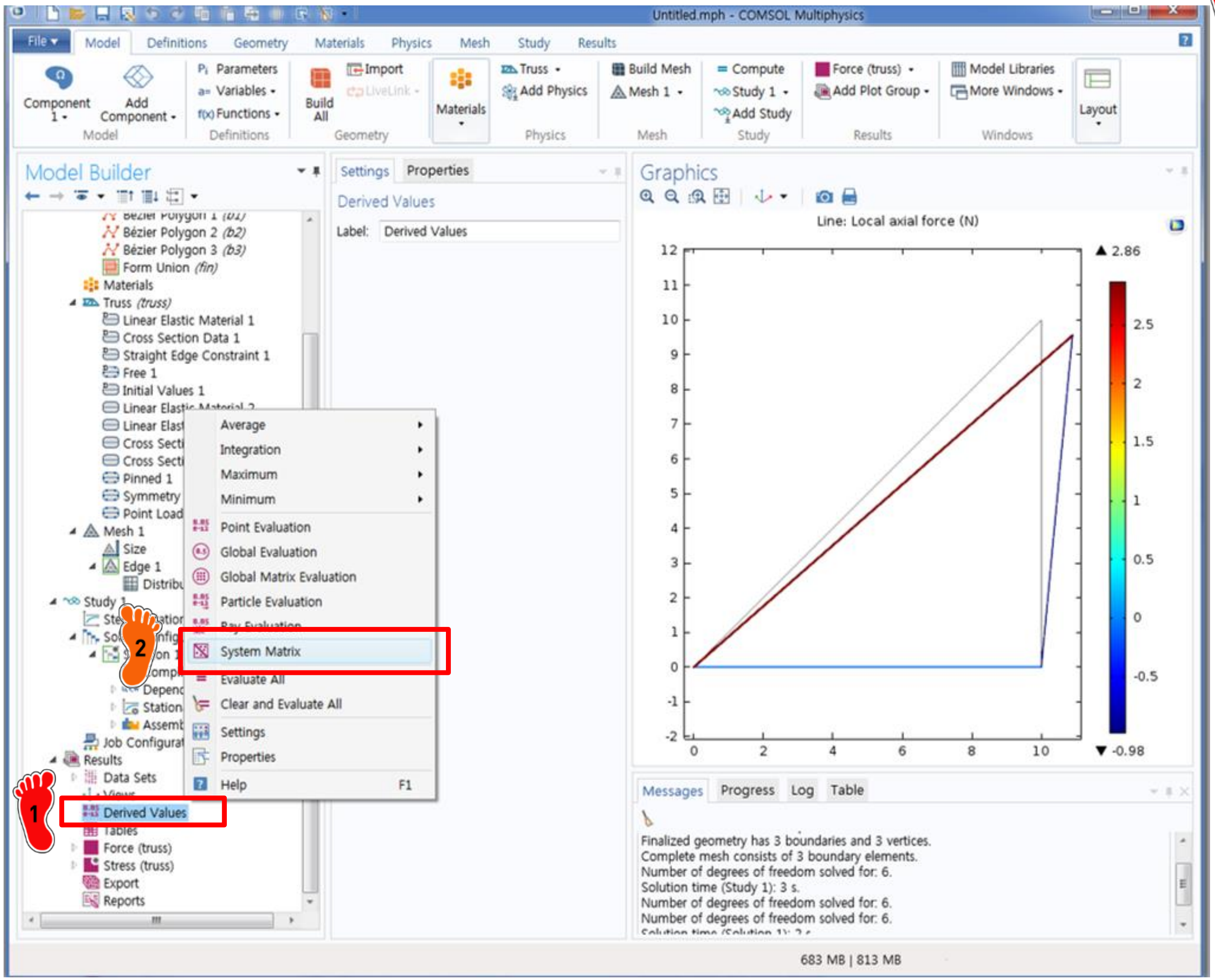
ASSEMBLE OPTION



1 Assemble 1 마우스 우클릭

2 Move up을 클릭하여 Assemble 노드가 Solver 노드 위에 위치하도록 이동

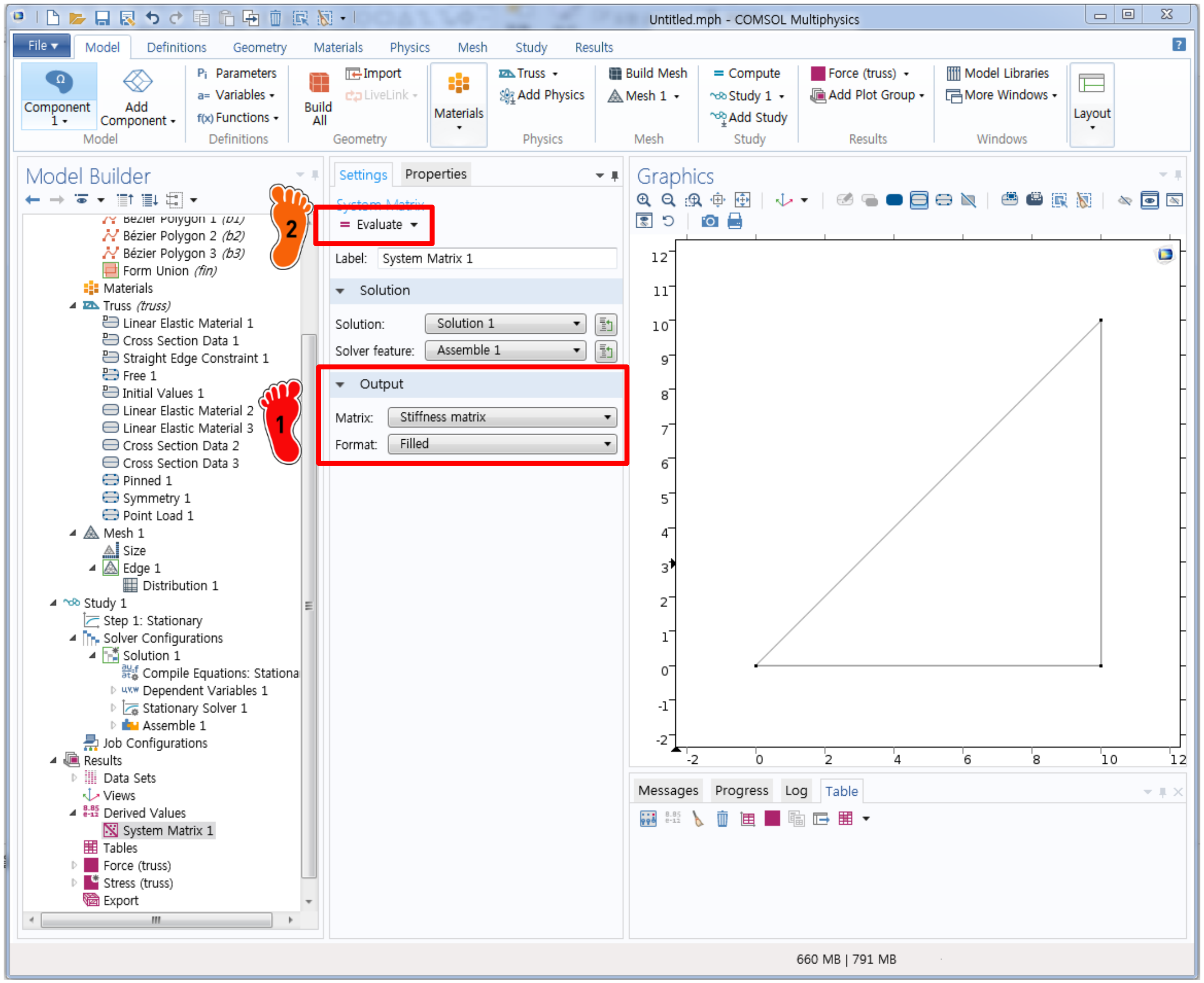
SYSTEM MATRIX



1 Derived Values 마우스 우클릭

2 System Matrix 클릭

SYSTEM MATRIX

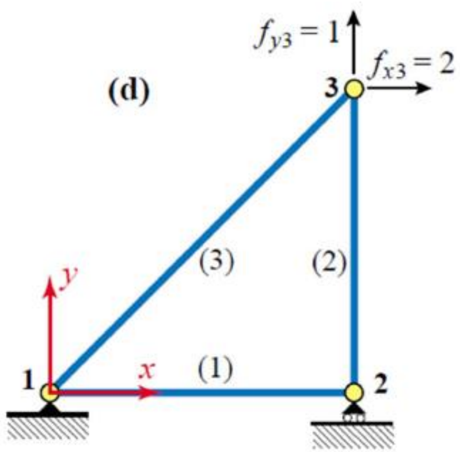


1 Matrix : Stiffness matrix
Format : Filled
선택

2 Evaluate 클릭

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

COMPARISON



Support conditions and applied loads

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

$$\Downarrow$$

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

COMSOL Result

Load Vector

0
0
0
0
2
1

Stiffness matrix

20.000	10.0000	-10.0000	0.0000	-10.0000	-10.0000
10.0000	10.0000	0.0000	0.0000	-10.0000	-10.0000
-10.0000	0.0000	10.0000	0.0000	0.0000	0.0000
0.0000	0.0000	0.0000	5.0000	0.0000	-5.0000
-10.0000	-10.0000	0.0000	0.0000	10.0000	10.0000
-10.0000	-10.0000	0.0000	-5.0000	10.0000	15.000

LOCAL STIFFNESS MATRIX

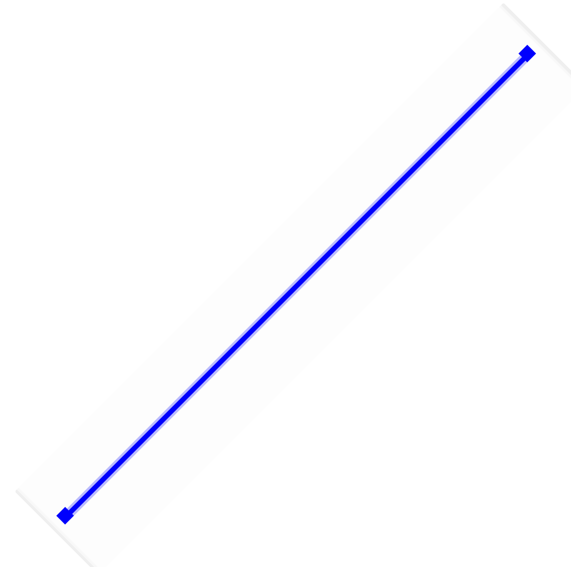
Stiffness matrix by Hand

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

$$\begin{bmatrix} f_{x2}^{(2)} \\ f_{y2}^{(2)} \\ f_{x3}^{(2)} \\ f_{y3}^{(2)} \end{bmatrix} = 5 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_{x2}^{(2)} \\ u_{y2}^{(2)} \\ u_{x3}^{(3)} \\ u_{y3}^{(3)} \end{bmatrix}$$

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

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

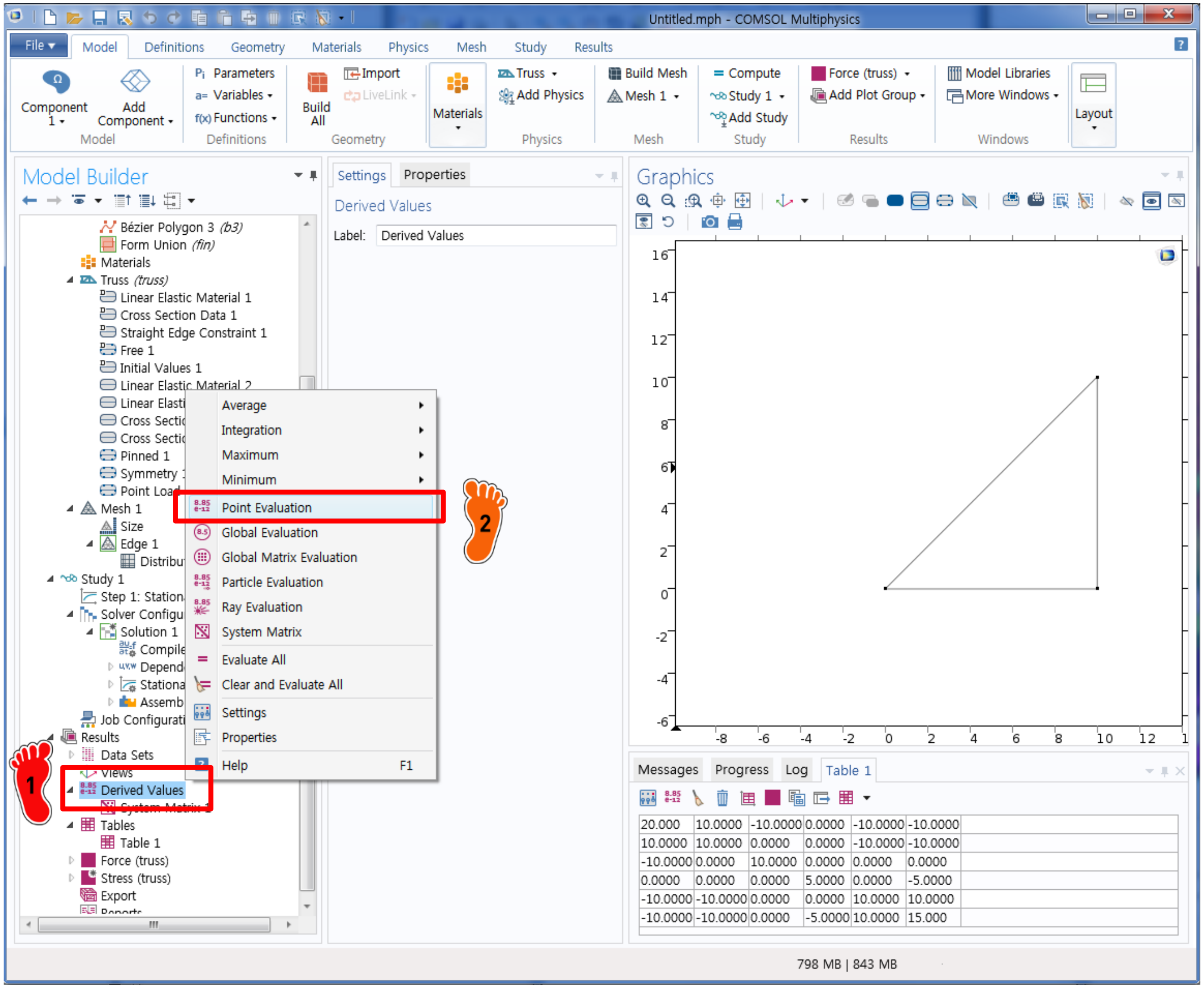
Eliminated Stiffness matrix

10.000	0.0000	0.0000
0.0000	10.000	10.000
0.0000	10.000	15.000

Eliminated Load vector

0
2
1

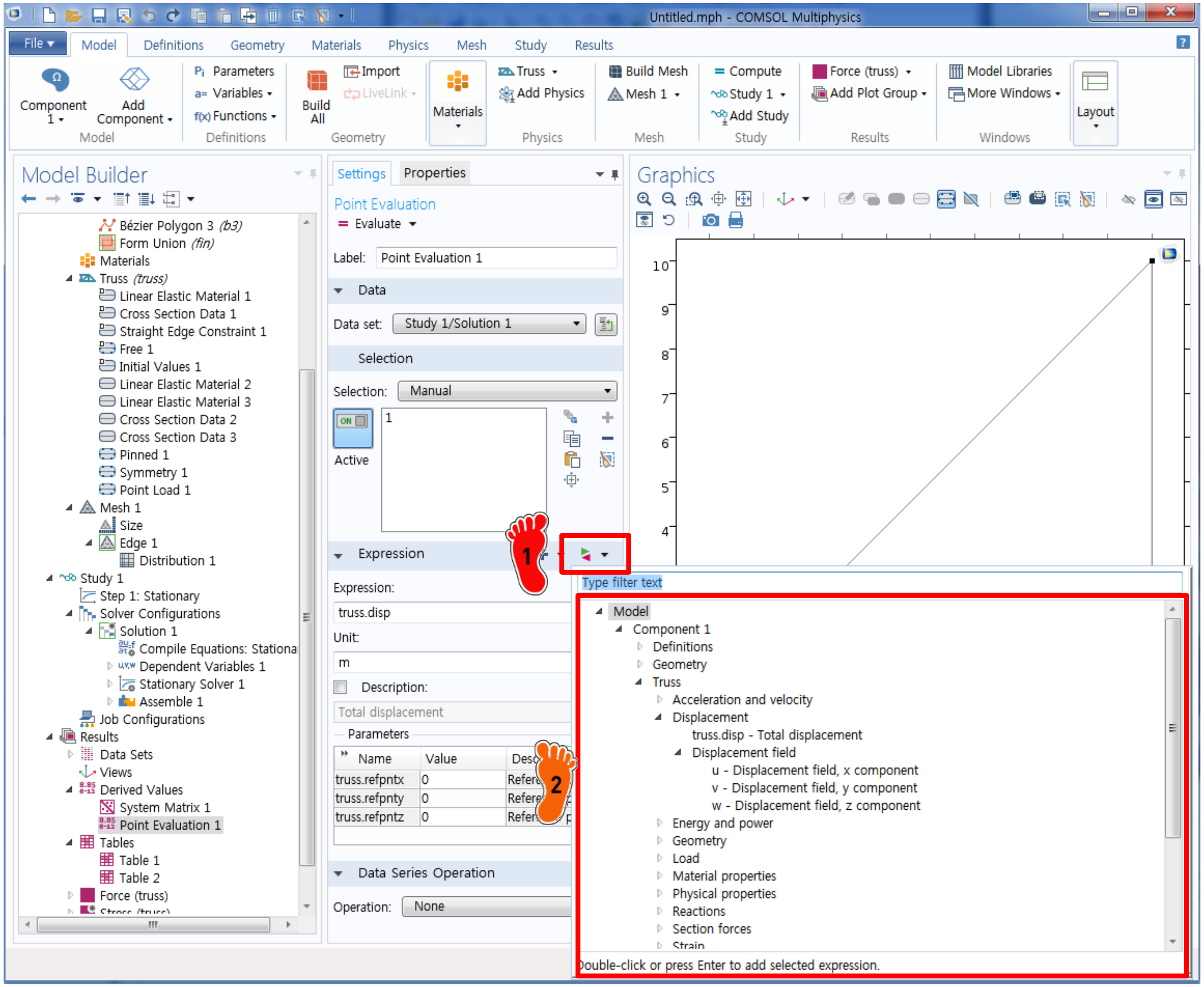
NODAL DISPLACEMENT



1 Derived Values 마우스 우클릭

2 Point Evaluation 클릭

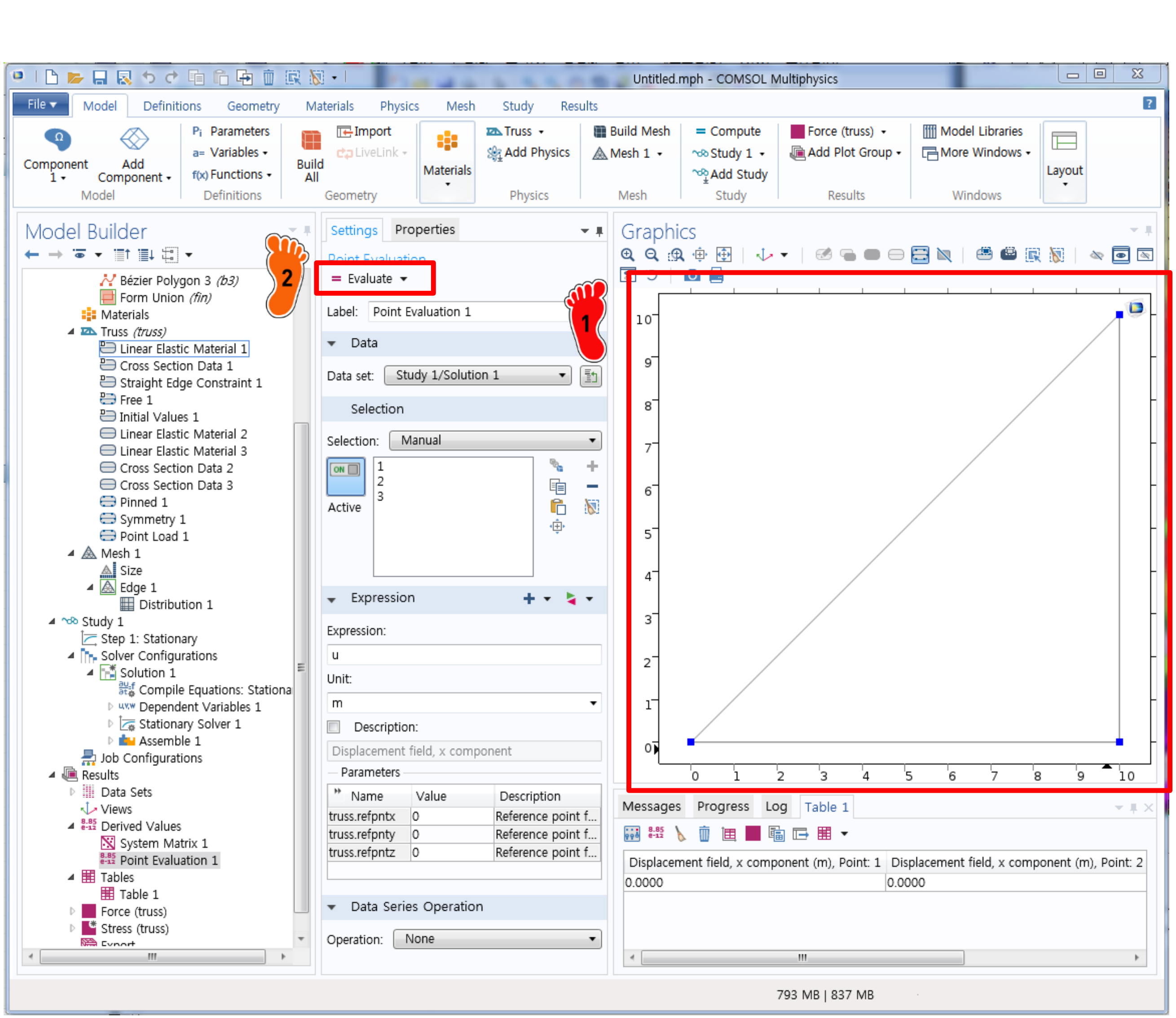
NODAL DISPLACEMENT



1 Replace expression

2 원하는 출력 값 선택

NODAL DISPLACEMENT



1 Point 선택

2 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

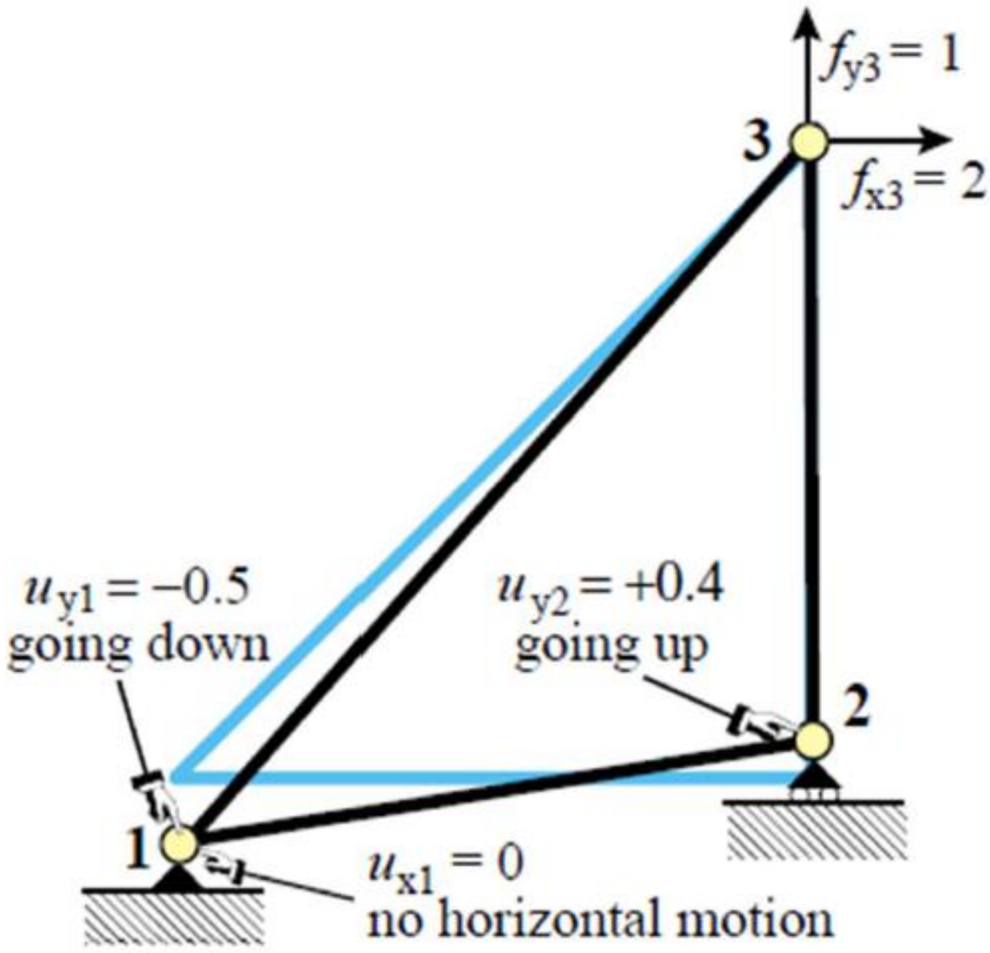
$$\begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ 0.4 \\ -0.2 \end{bmatrix} \xrightarrow{\text{expand with known displacement BCs}} \mathbf{u} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0.4 \\ -0.2 \end{bmatrix}$$

Nodal displacement by COMSOL

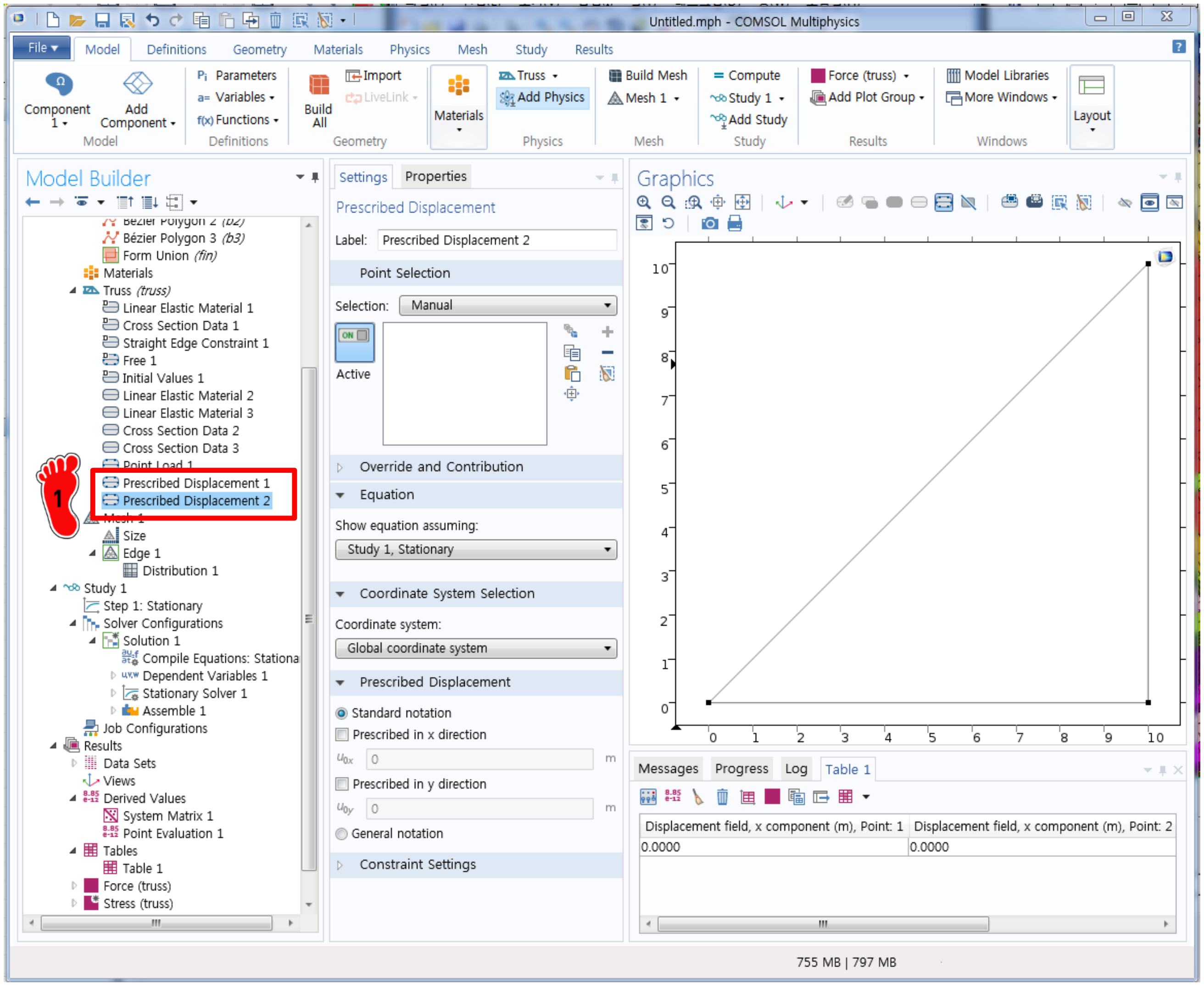
Displacement field, x component (m), Point: 1	Displacement field, x component (m), Point: 2	Displacement field, x component (m), Point: 3
0.0000	0.0000	0.40000

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

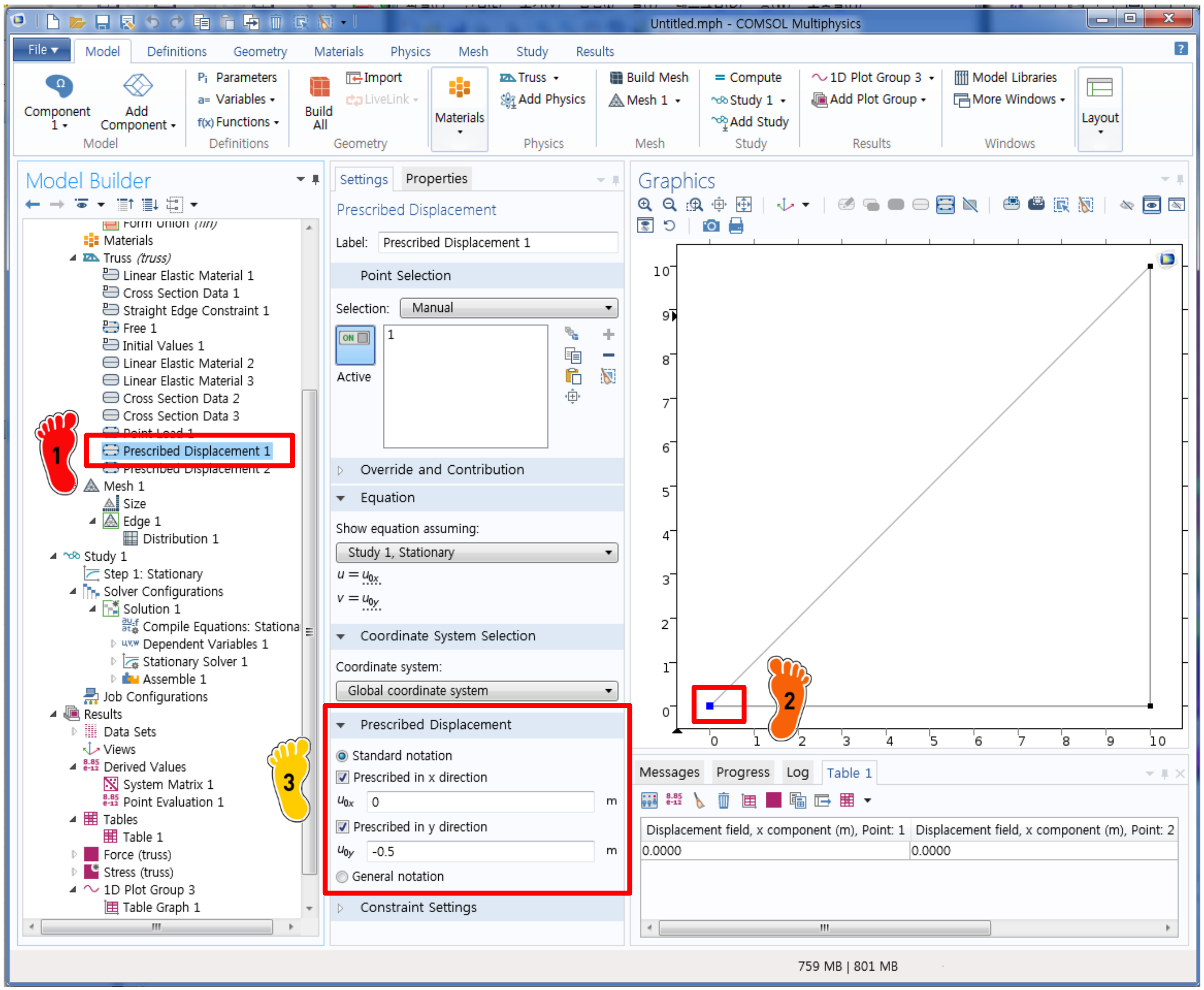


BOUNDARY CONDITION



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

BOUNDARY CONDITION



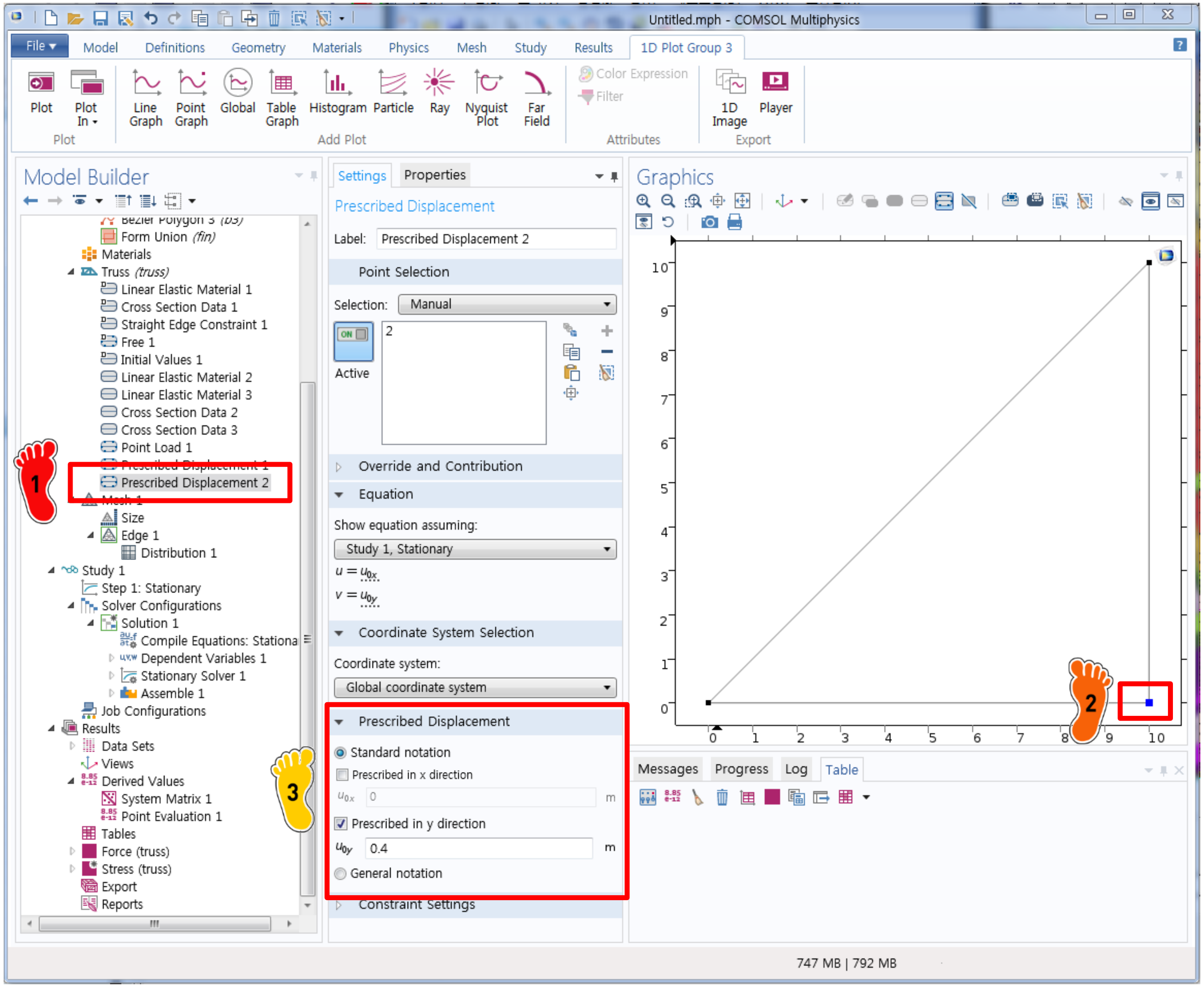
1 첫 번째 Prescribed Displacement 클릭

2 왼쪽 아래 절점 선택

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

입력

BOUNDARY CONDITION



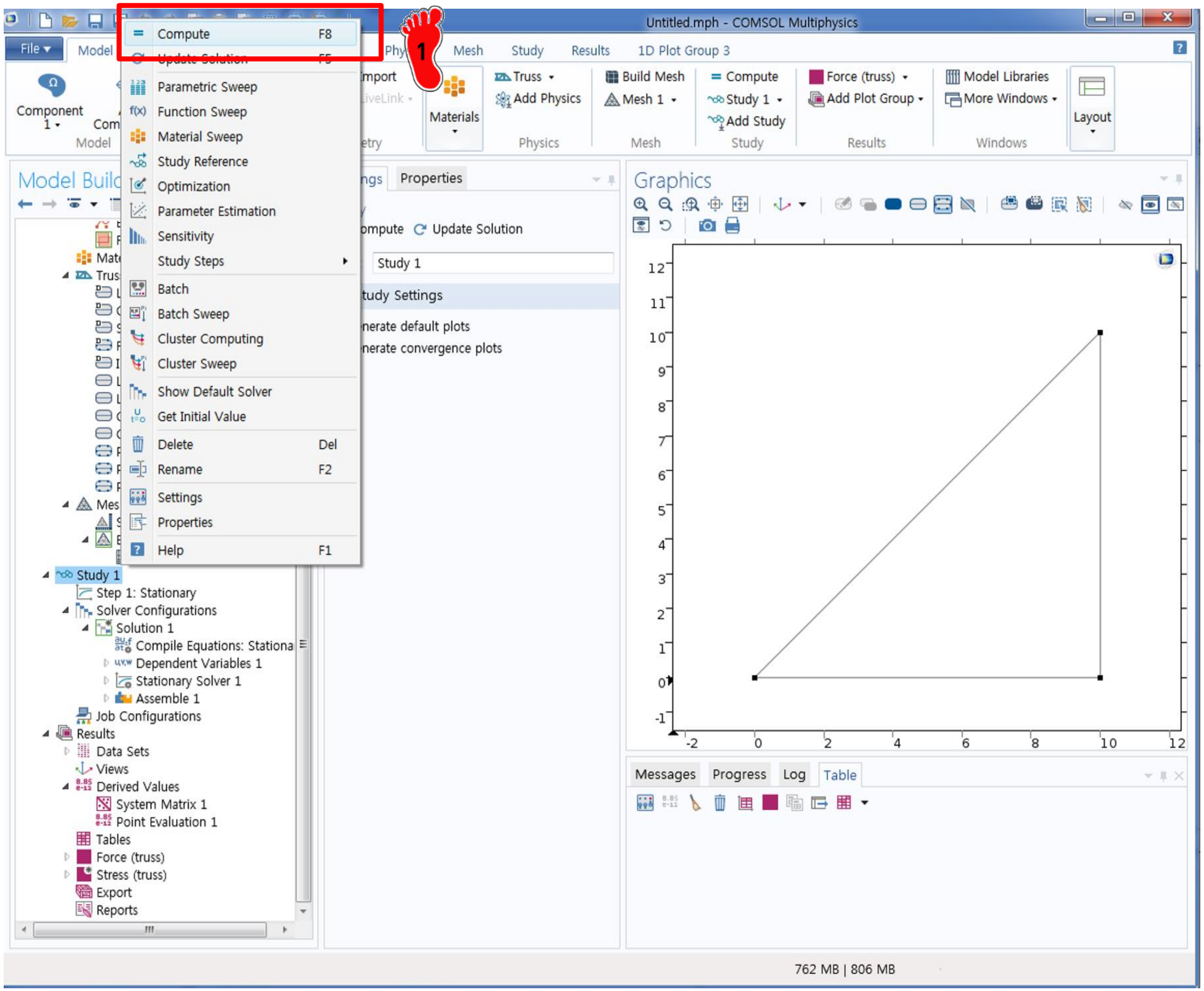
1 두 번째 Prescribed Displacement 클릭

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

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

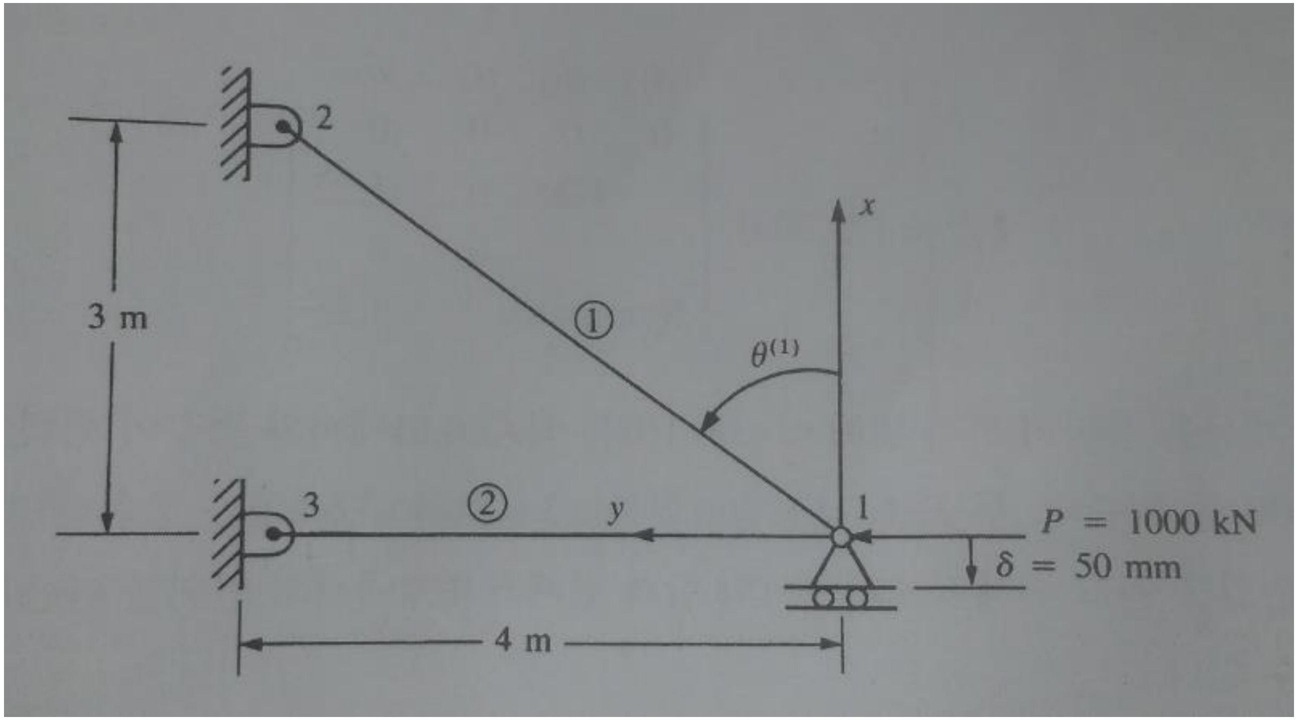
In summary, the only changes to the SDM is in the application of displacement boundary conditions before solve

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



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

$$A = 6\times10^{-4} \text{ m}^2$$

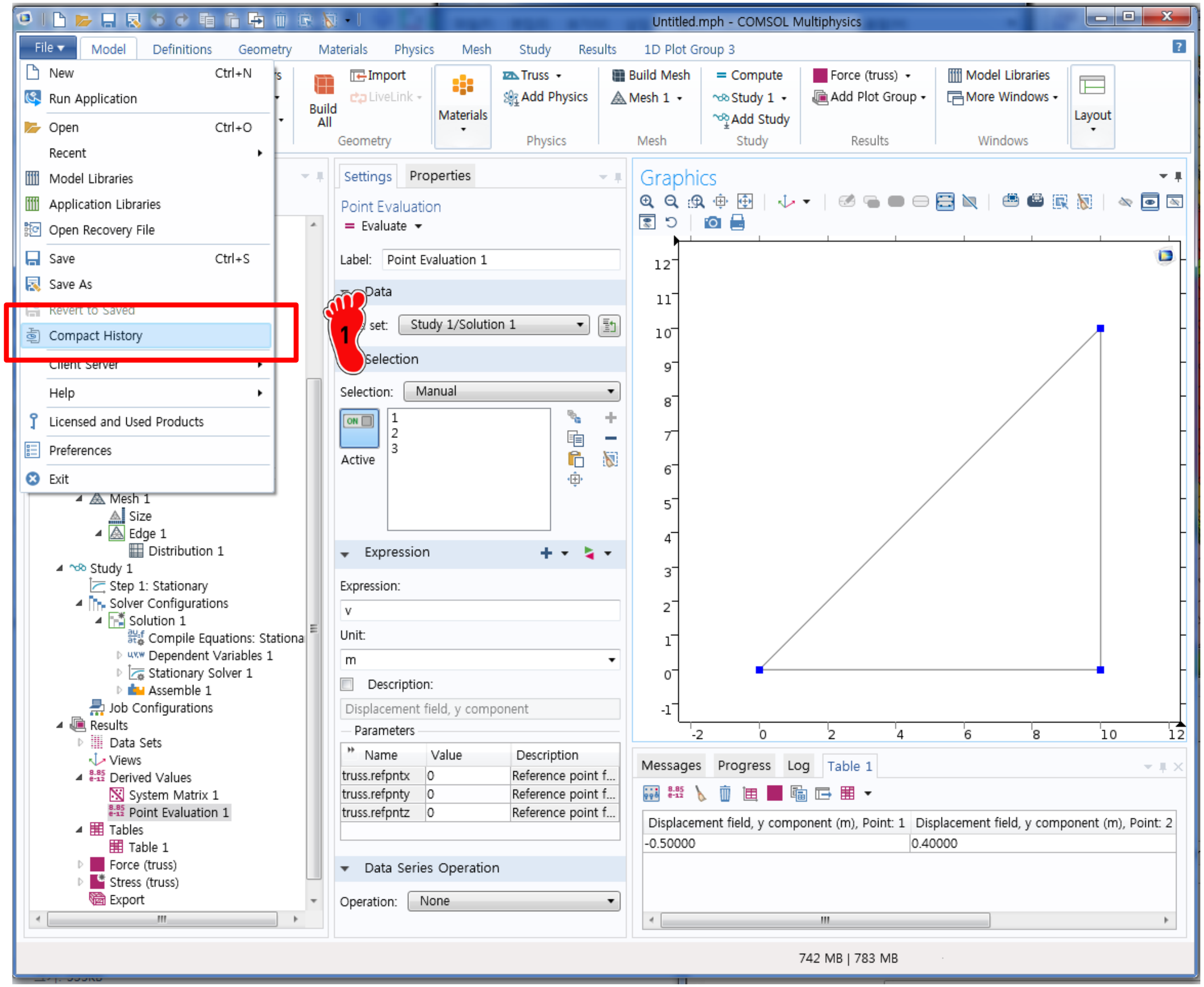
$$\therefore\ d_{1y} = 0.0337 \text{ m}$$

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

CONTENTS

- **Livelink with MATLAB**

RESET HISTORY



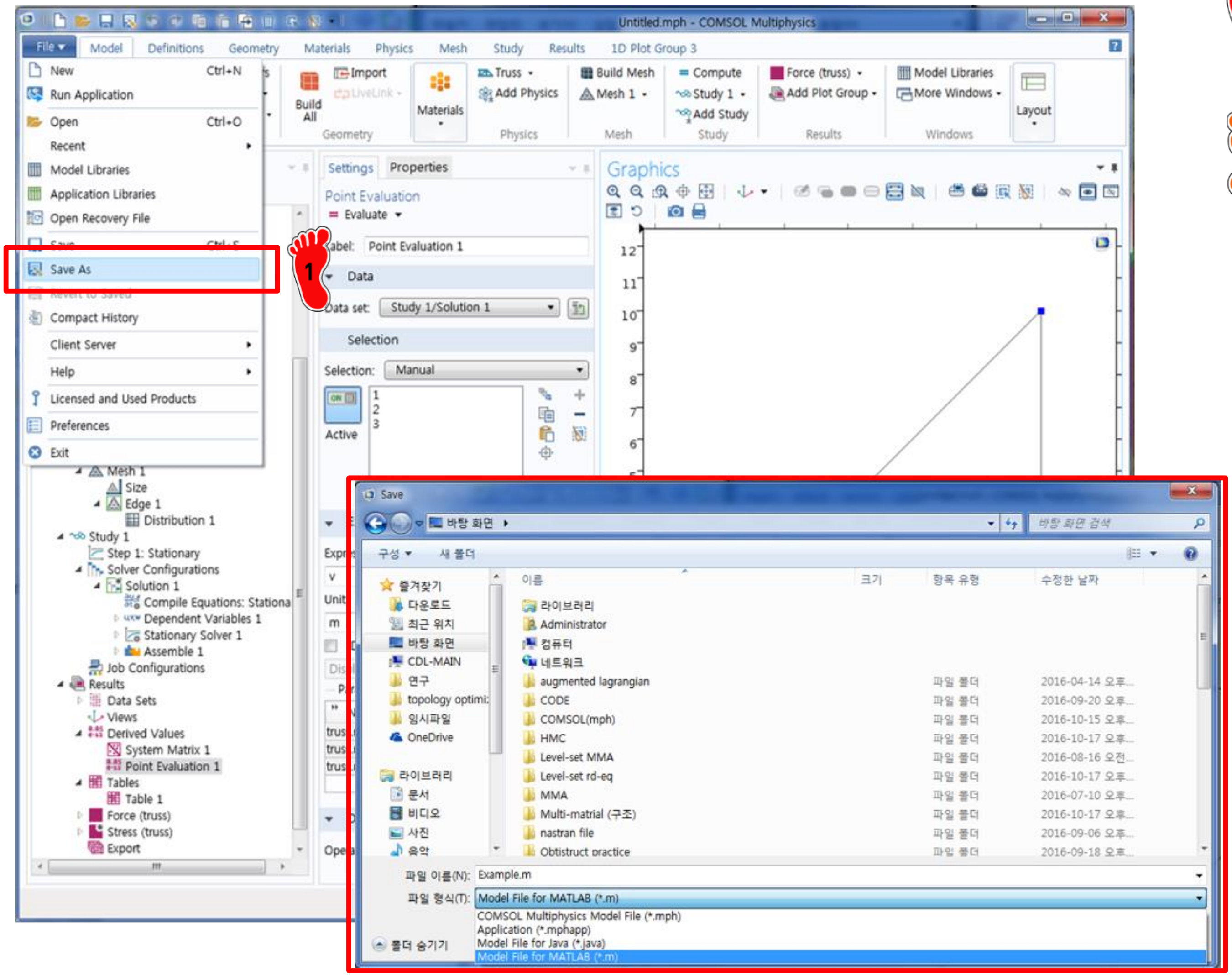
1

Reset History 클릭

COMSOL에서 m-file 로 저장할 때 작업 history 모두 저장됨

따라서 불필요한 history 를 삭제하기 위하여 Compact history 기능을 이용

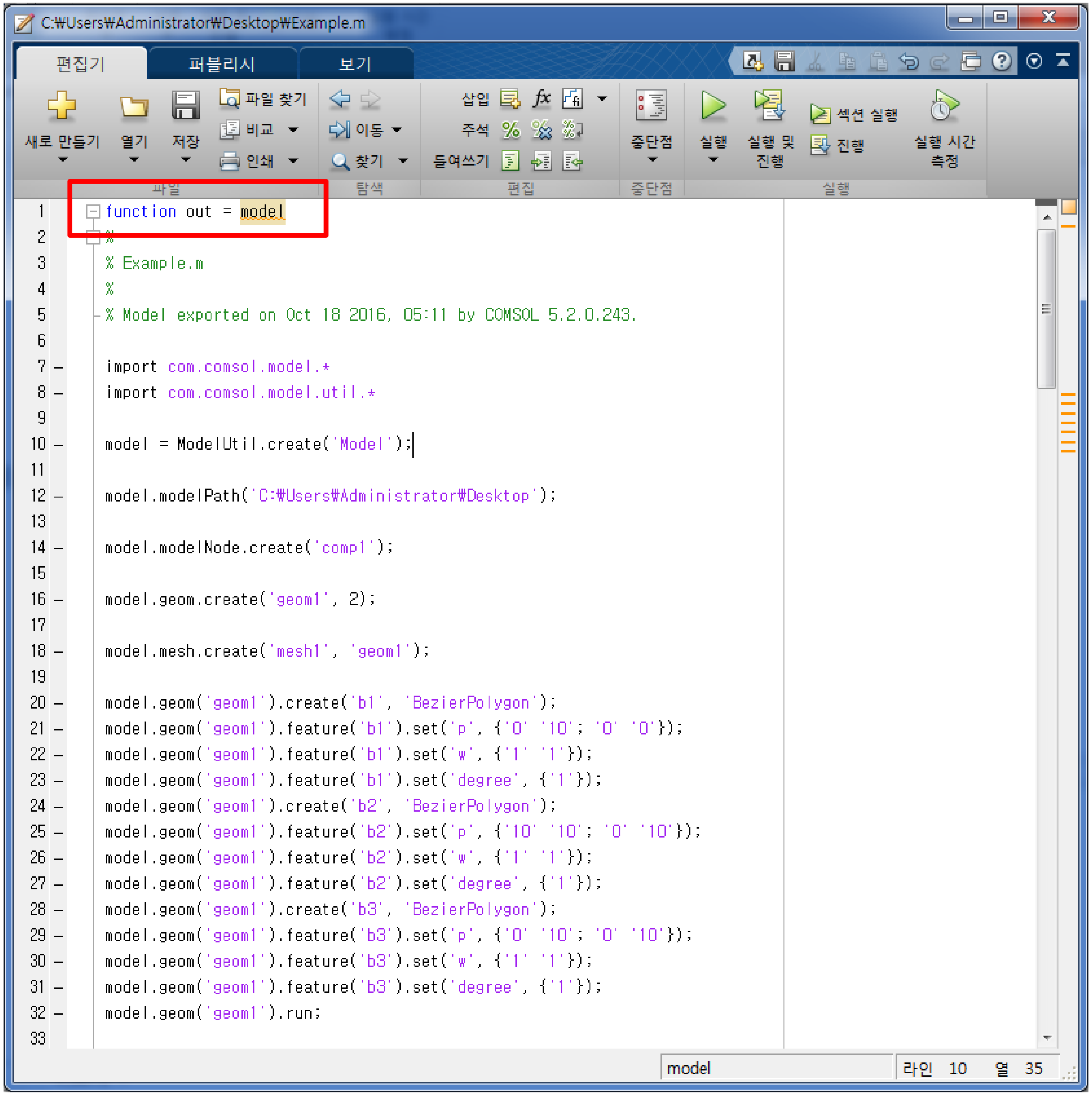
SAVE AS M-FILE



1 Save As 클릭

2 m-file 로 저장

COMSOL M-FILE STRUCTURE

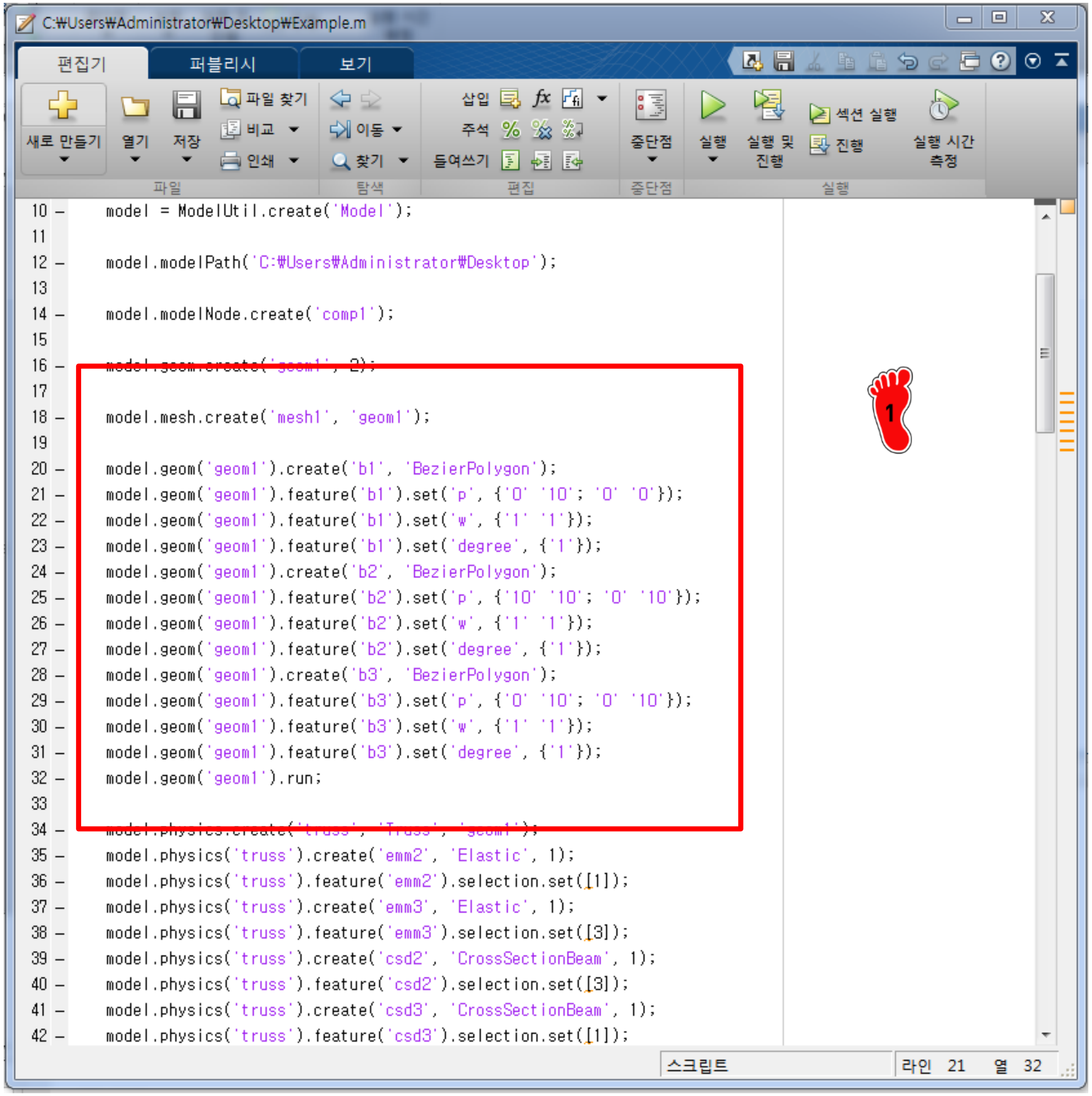


1

function 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



```
model = ModelUtil.create('Model');

model.modelPath('C:\Users\Administrator\Desktop');

model.modelNode.create('comp1');

model.geom.create('geom1', 2);

model.mesh.create('mesh1', 'geom1');

model.geom('geom1').create('b1', 'BezierPolygon');
model.geom('geom1').feature('b1').set('p', {'0' '10'; '0' '0'});
model.geom('geom1').feature('b1').set('w', {'1' '1'});
model.geom('geom1').feature('b1').set('degree', {'1'});
model.geom('geom1').create('b2', 'BezierPolygon');
model.geom('geom1').feature('b2').set('p', {'10' '10'; '0' '10'});
model.geom('geom1').feature('b2').set('w', {'1' '1'});
model.geom('geom1').feature('b2').set('degree', {'1'});
model.geom('geom1').create('b3', 'BezierPolygon');
model.geom('geom1').feature('b3').set('p', {'0' '10'; '0' '10'});
model.geom('geom1').feature('b3').set('w', {'1' '1'});
model.geom('geom1').feature('b3').set('degree', {'1'});
model.geom('geom1').run;

model.physics.create('truss', 'Truss', 'geom1');
model.physics('truss').create('emm2', 'Elastic', 1);
model.physics('truss').feature('emm2').selection.set([1]);
model.physics('truss').create('emm3', 'Elastic', 1);
model.physics('truss').feature('emm3').selection.set([3]);
model.physics('truss').create('csd2', 'CrossSectionBeam', 1);
model.physics('truss').feature('csd2').selection.set([3]);
model.physics('truss').create('csd3', 'CrossSectionBeam', 1);
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

C:₩Users₩Administrator₩Desktop₩Example.m

```
model.sol('sol1').runAll;

model.result.numerical('sys1').set('format', 'filled');
model.result.numerical('sys1').set('matrixassem', 'elimstiffness');
model.result.numerical('sys1').set('table', 'tbl1');
model.result.numerical('pev1').set('expr', 'v');
model.result.numerical('pev1').set('descr', 'Displacement field, y component');
model.result.numerical('pev1').set('table', 'tbl2');
model.result.numerical('sys1').setResult;
model.result.numerical('pev1').setResult;
model.result('pg1').label('Force (truss)');
model.result('pg1').feature('line1').set('tuberadiusscale', '0.042156294610216236');
model.result('pg1').feature('line1').set('expr', 'truss.Nxl');
model.result('pg1').feature('line1').set('resolution', 'extrafine');
model.result('pg1').feature('line1').set('linetype', 'tube');
model.result('pg1').feature('line1').set('radiusexpr', 'sqrt(truss.area/pi)');
model.result('pg1').feature('line1').set('unit', 'N');
model.result('pg1').feature('line1').set('descr', 'Local axial force');
model.result('pg1').feature('line1').set('tuberadiusscaleactive', false);
model.result('pg1').feature('line1').feature('def1').set('scale', '1.856953381770518');
model.result('pg1').feature('line1').feature('def1').set('scaleactive', false);
model.result('pg2').label('Stress (truss)');
model.result('pg2').feature('line1').set('expr', 'truss.sn');
model.result('pg2').feature('line1').set('resolution', 'extrafine');
model.result('pg2').feature('line1').set('linetype', 'tube');
model.result('pg2').feature('line1').set('radiusexpr', 'sqrt(truss.area/pi)');
model.result('pg2').feature('line1').set('unit', 'N/m^2');
model.result('pg2').feature('line1').set('descr', 'Axial stress at center line');

% out = model;

stiffness_matrix = model.result.table('tbl1').getReal;

```

스크립트 라인 151 열 52

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

2 model.result.table('tbl1').getReal 명령어로 table 1 의 데이터를 저장