PDE & Direct Stiffness Method by COMSOL

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

OUTLINE

- **Lecture Goals**
  - ✓ **COMSOL을 이용한 편미분방정식의 유한요소 해석 시뮬레이션 및 후처리 과정 전반을 실습하고 직접 강성법 예제를 통해 유한 요소 해석의 시스템 행렬과 벡터의 개념을 고찰한다.**
- **Content**
  - ✓ **PDE Examples**
  - ✓ **Direct Stiffness Method: Truss**
  - ✓ **Assignment**

- **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)

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

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

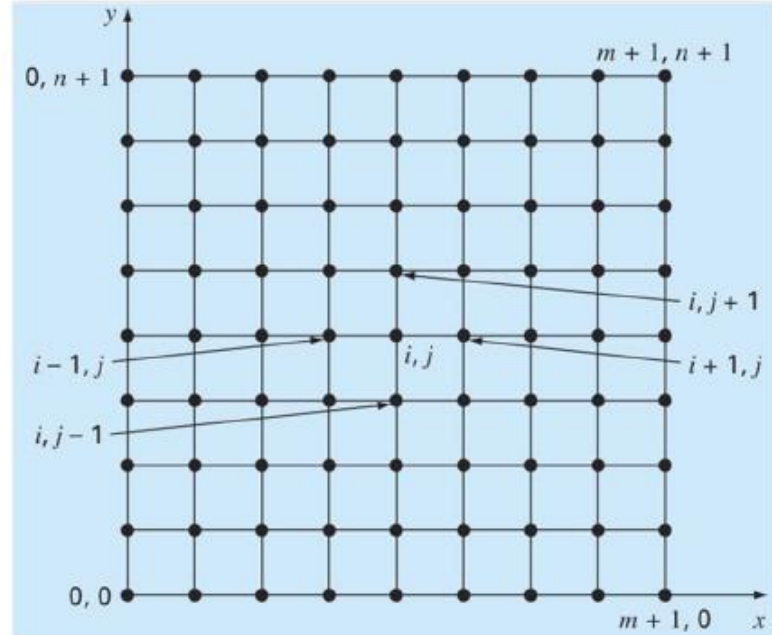
$$@(3,1): \underbrace{T_{41}}_{50°C}+T_{21}+T_{32}+\underbrace{T_{30}}_{0°C}-4T_{31}=0 \rightarrow -T_{21}+4T_{31}-T_{32}=50$$

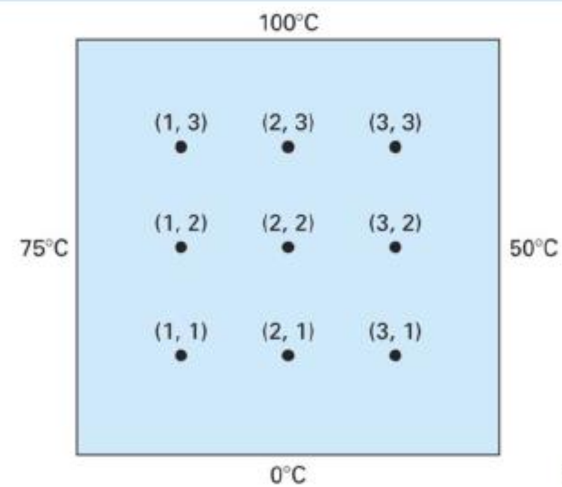
⋮

9 equations

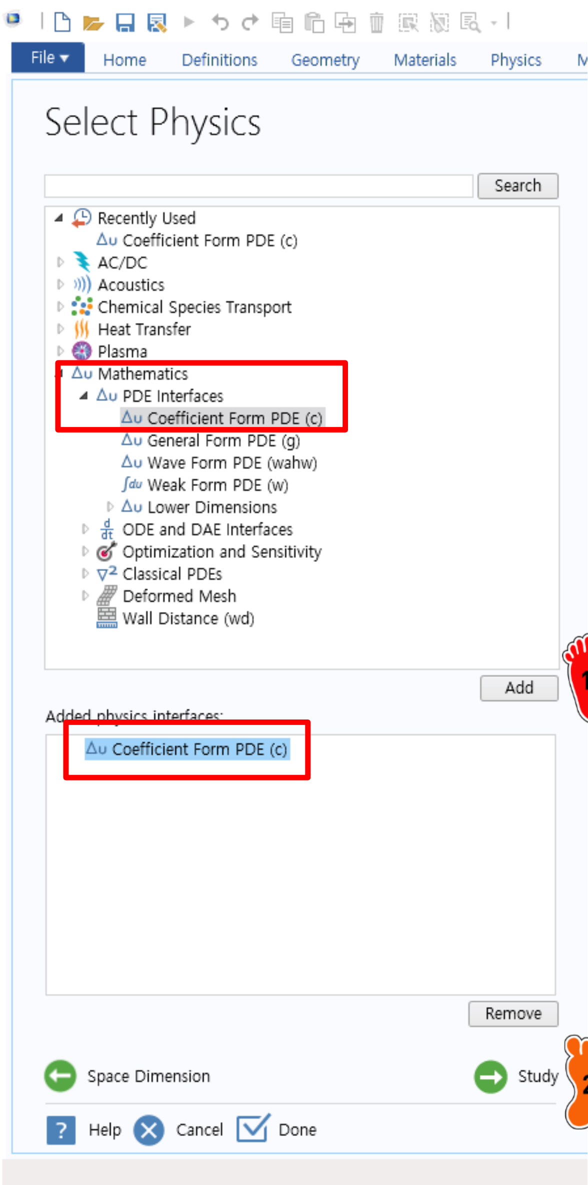
$$[K_{ij}]\{T_i\}=\{f_i\}$$

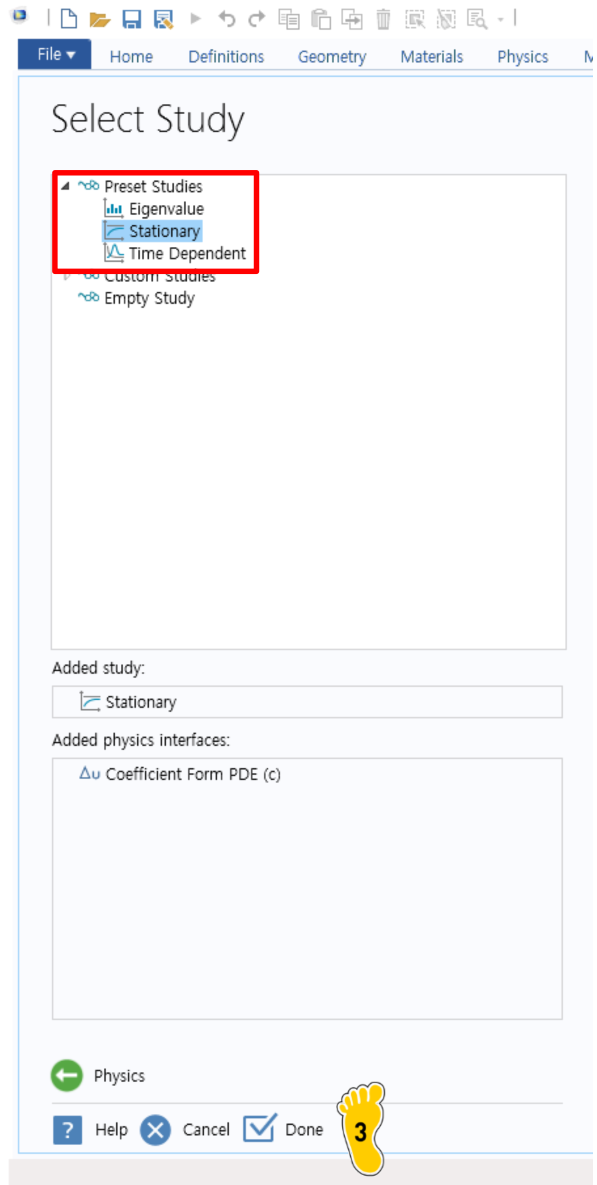
$[K_{ij}]$: coefficient matrix, $\{T_i\}$: solution vector, $\{f_i\}$: force vector





SETTING



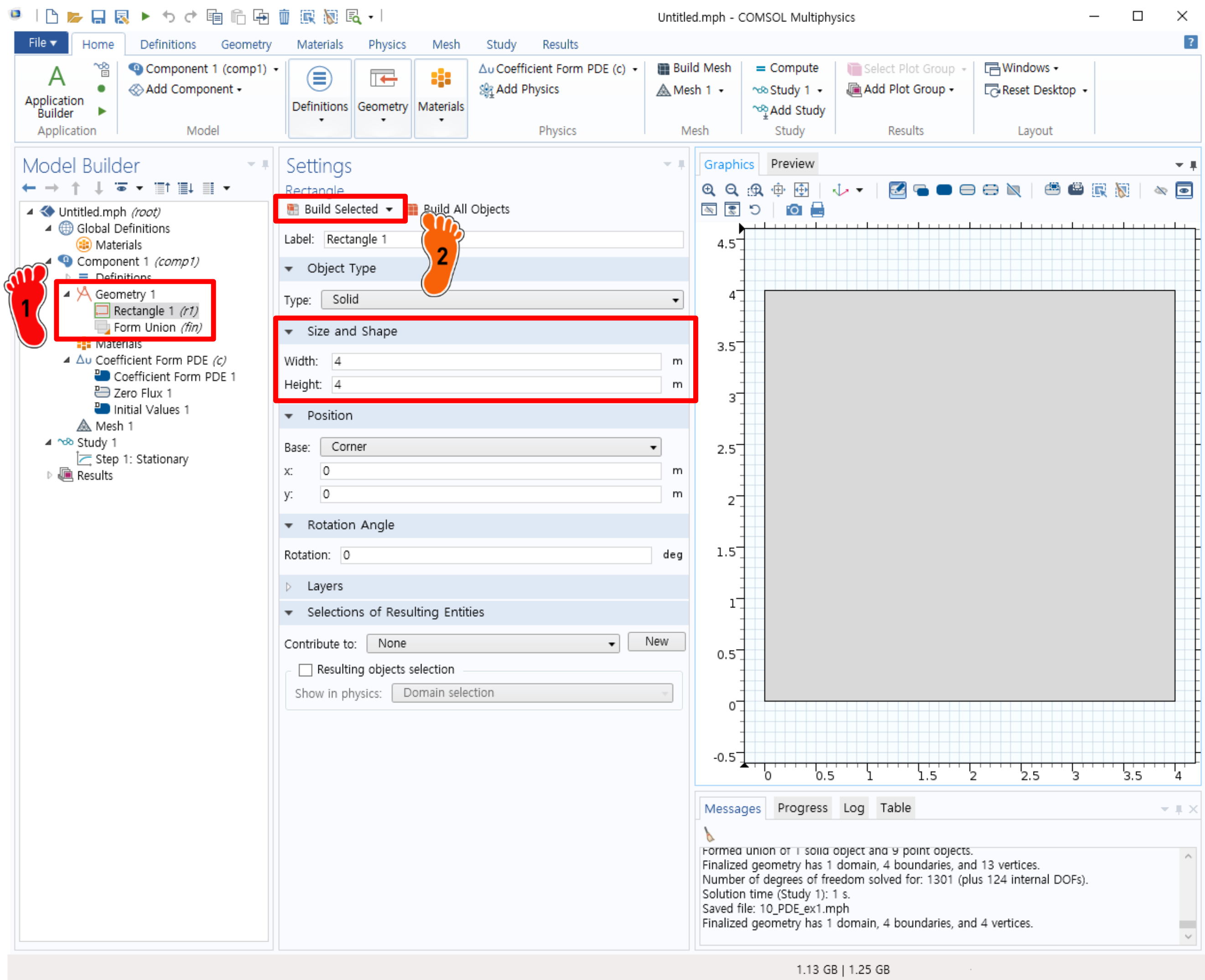


1 Model Wizard → 2D → Mathematics - PDE Interfaces – Coefficient form PDE 선택 → Add 클릭

2 Study 클릭

3 Stationary 선택 후 Done 클릭

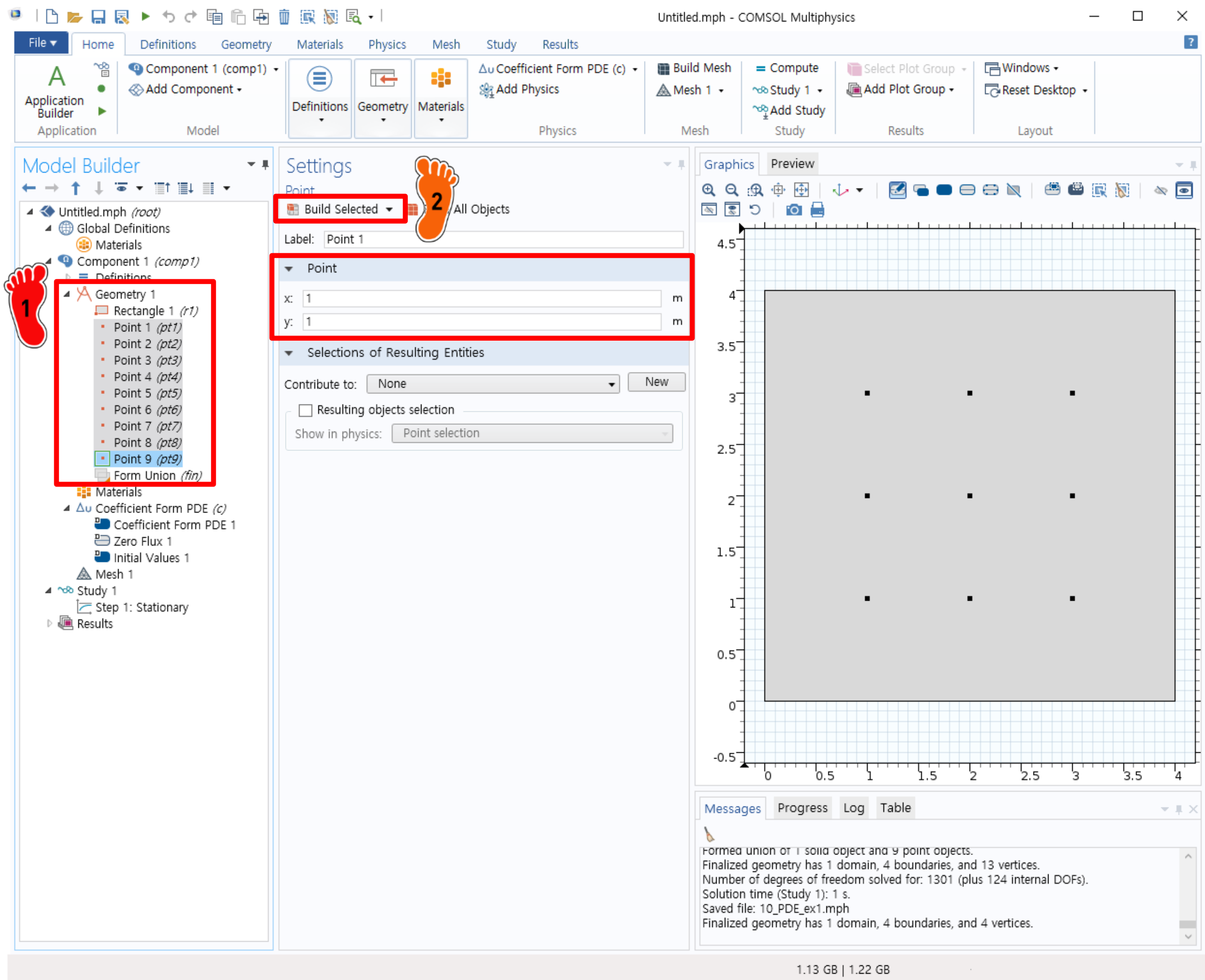
GEOMETRY CREATION



1 Geometry 1 메뉴를 마우스 우클릭 → Rectangle 선택

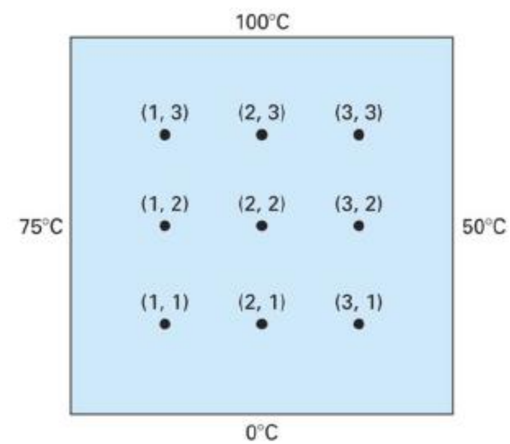
2 Width: 4, Height: 4 입력 후 Build Selected 클릭

GEOMETRY CREATION

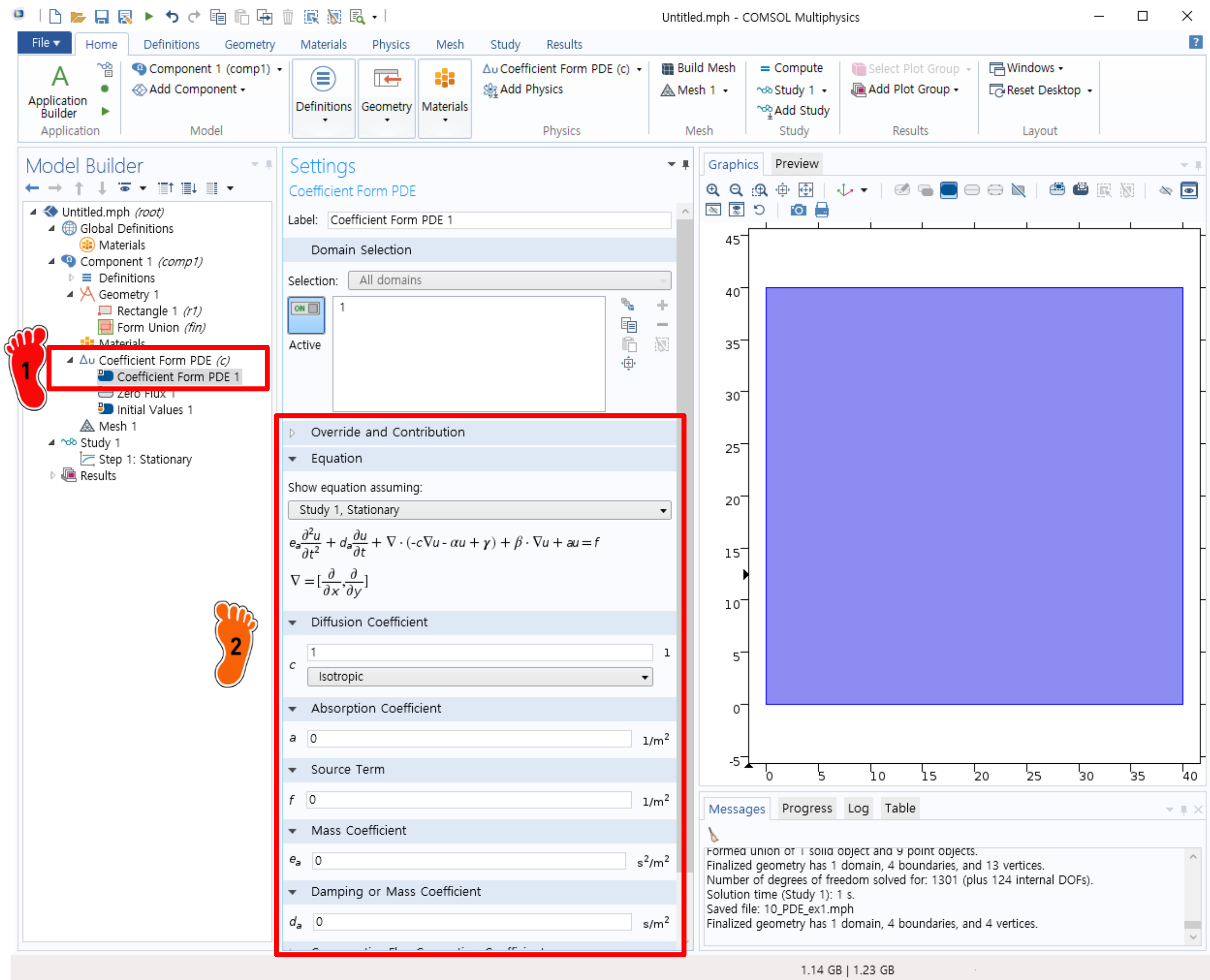


1 Geometry 1 메뉴를 마우스 우클릭 → Point 선택

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)

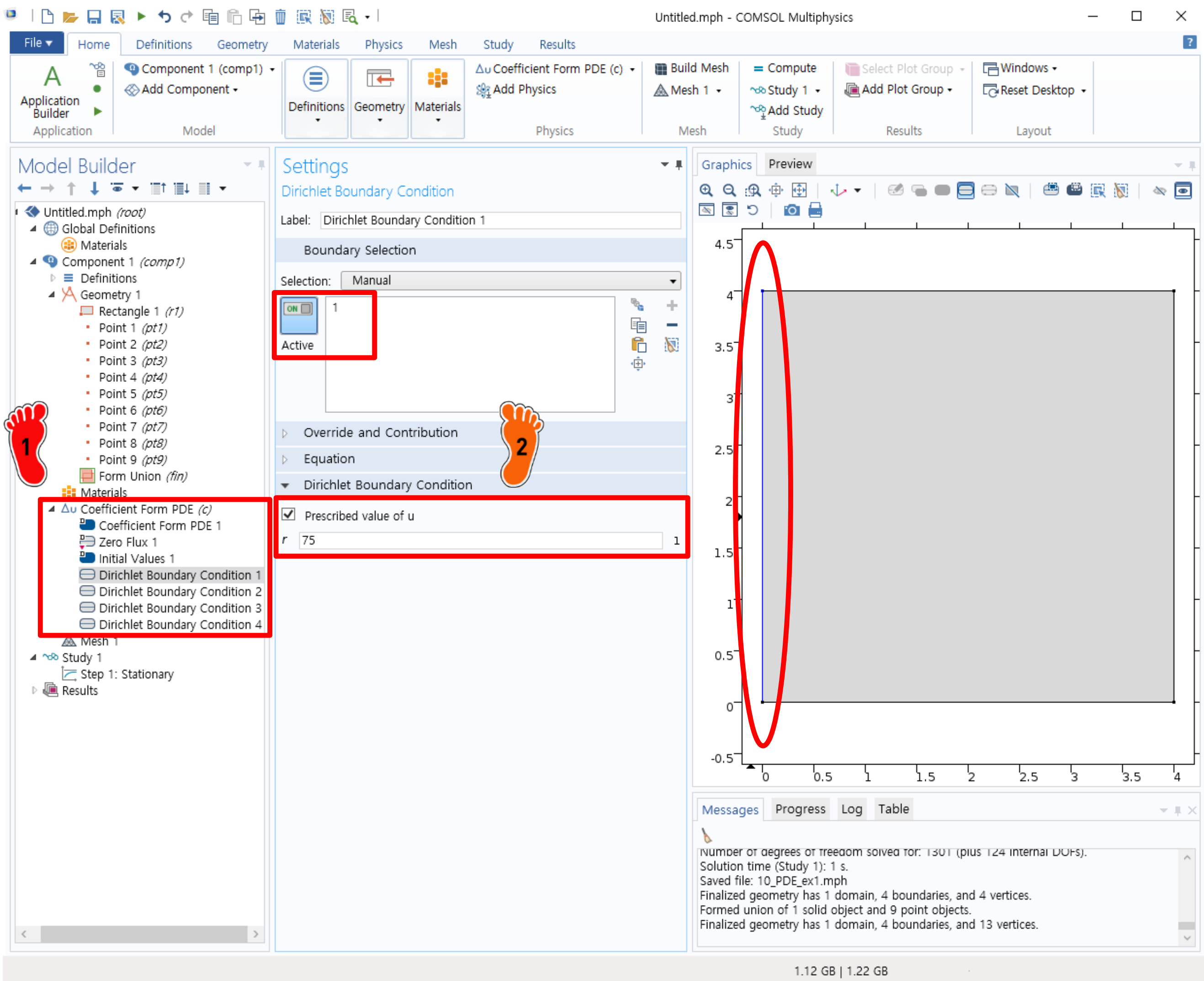
$$\nabla = \left[\frac{\partial}{\partial x}, \frac{\partial}{\partial y}\right]$$

$$\nabla^2 u = 0$$

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

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

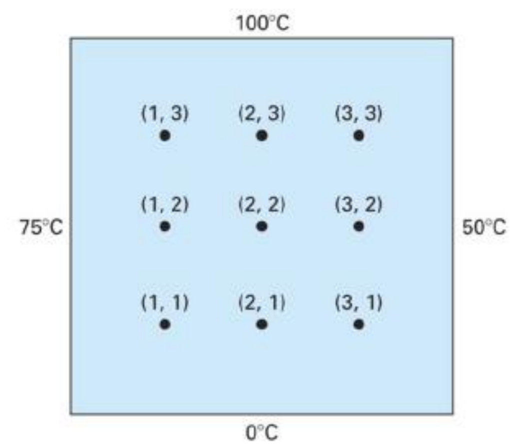
BOUNDARY CONDITION



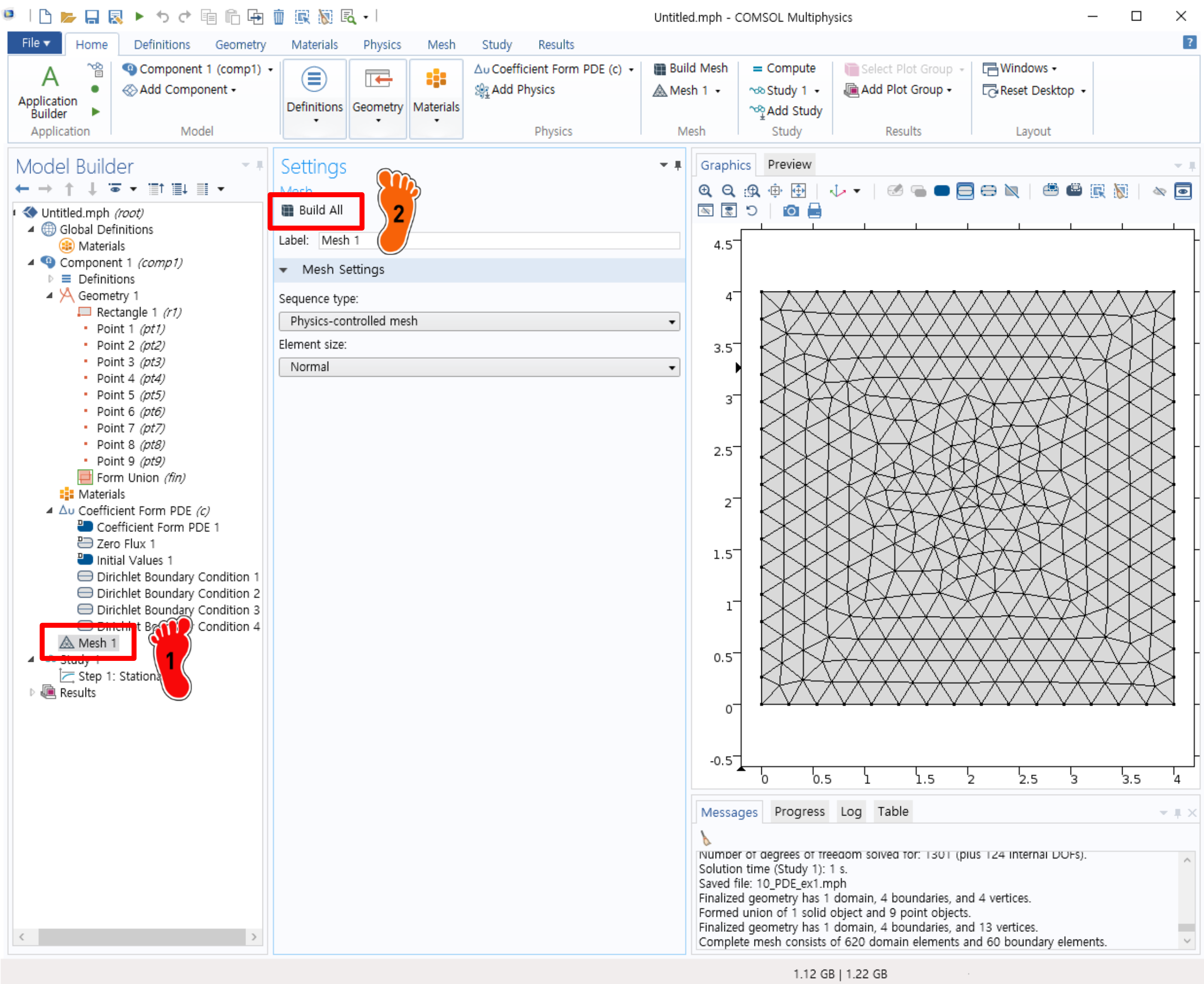
1 Coefficient Form PDE 메뉴 마우스 우클릭 → Dirichlet Boundary Condition 클릭

Dirichlet Boundary Condition

2 4개 생성 및 경계 선택 후 경계값 입력



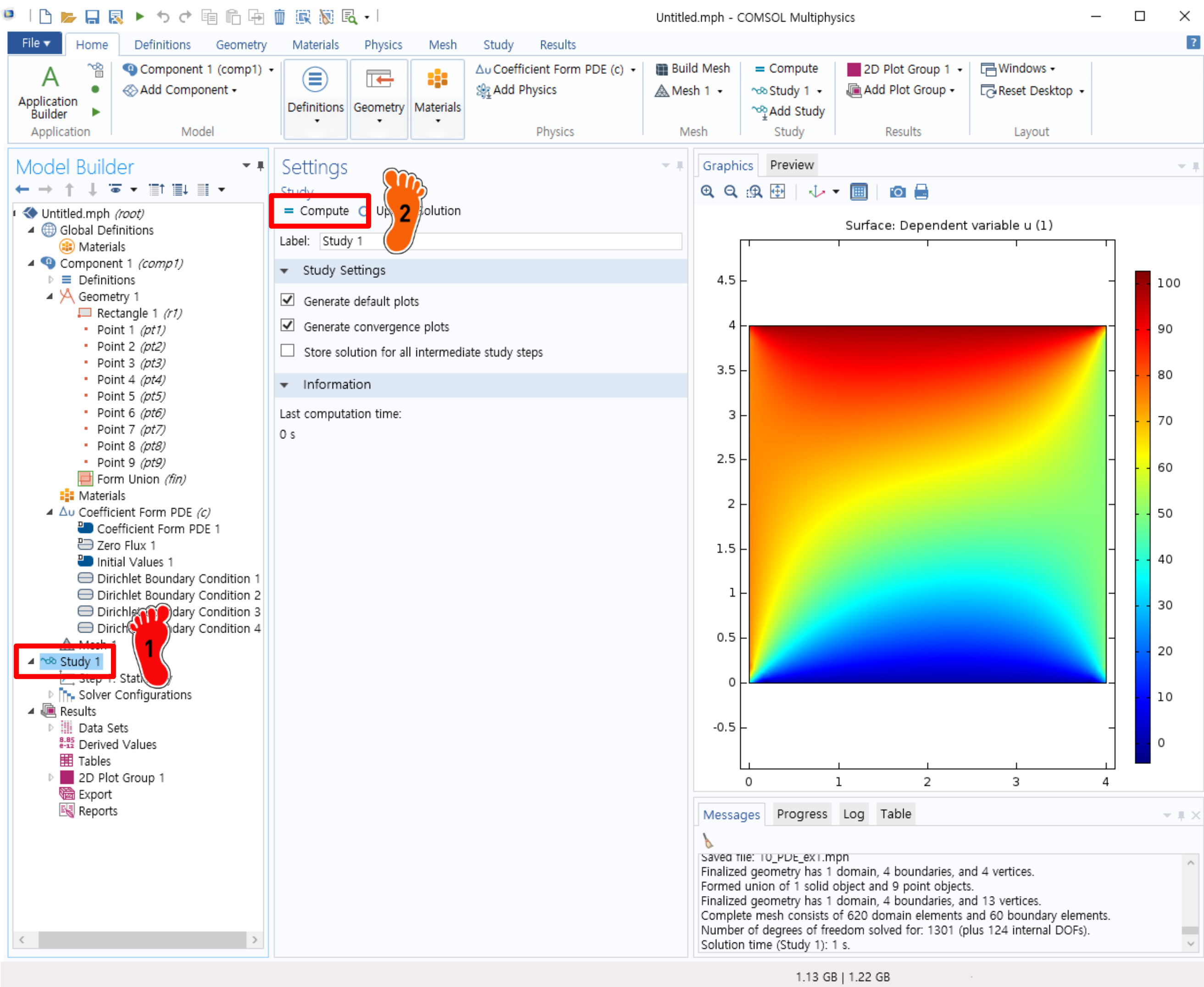
MESH CREATION



1 Mesh 1 메뉴 클릭

2 Build All 클릭

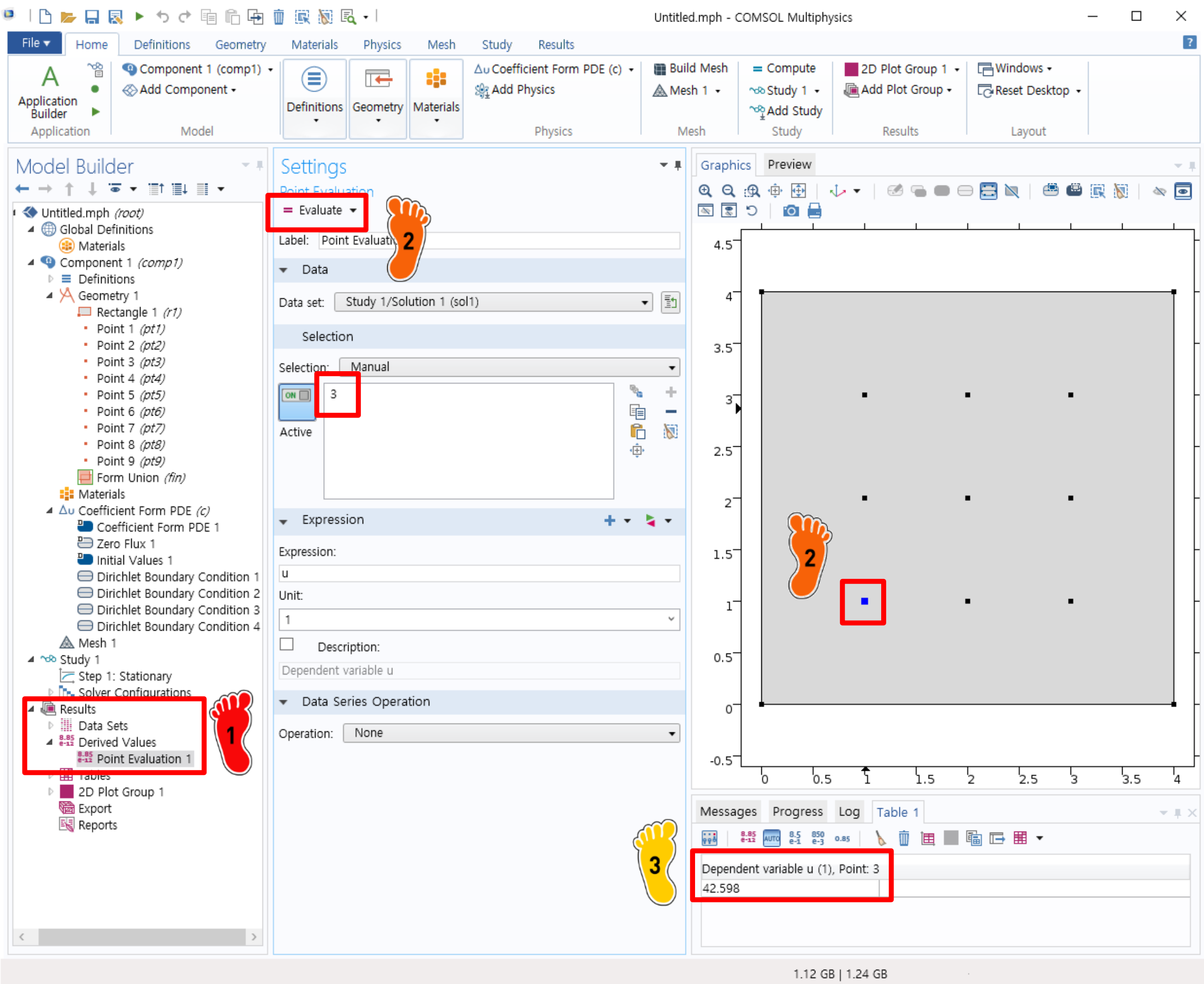
COMPUTE



1 Study 1 메뉴 선택

2 Compute 클릭

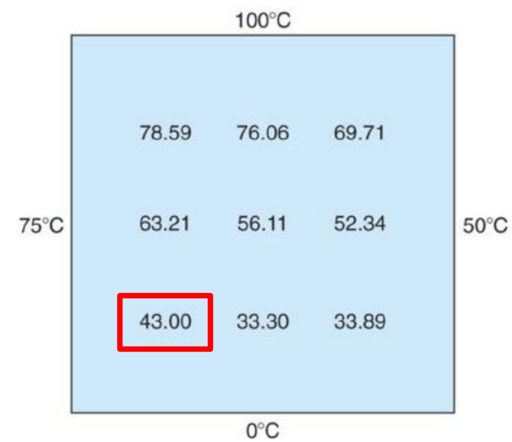
POST-PROCESSING



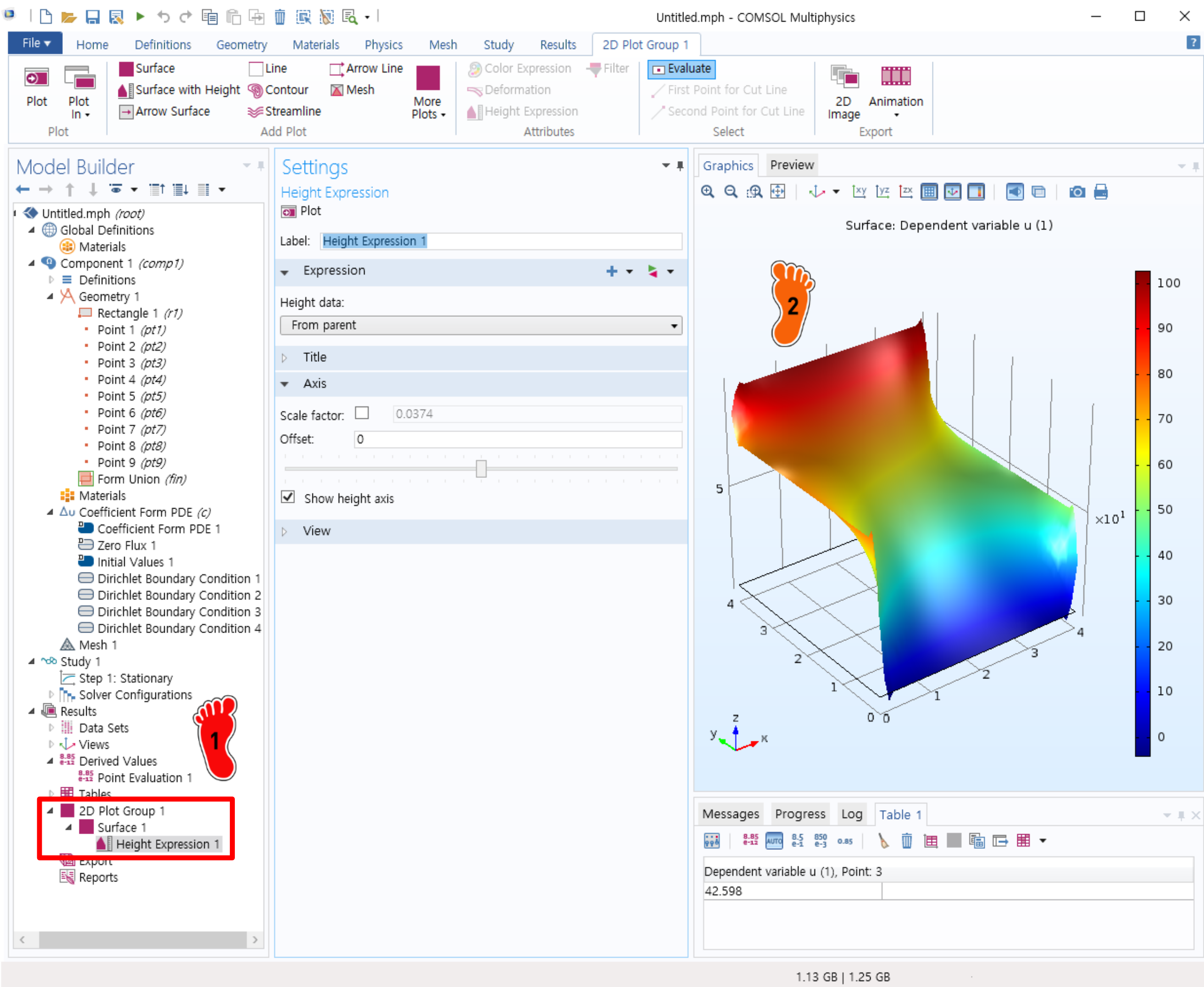
1 Results → Derived Values 우클릭 → Point Evaluation 선택

2 측정하고자 하는 Point 선택 후 Evaluate 클릭

3 결과값 확인

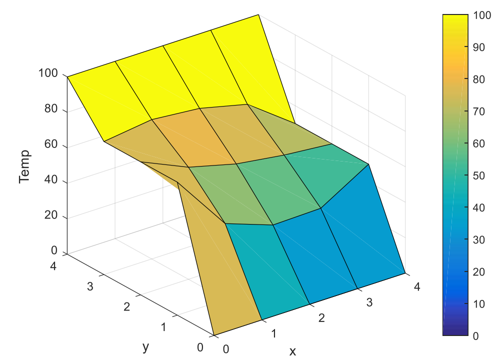


POST-PROCESSING



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

3차원 그래프 확인



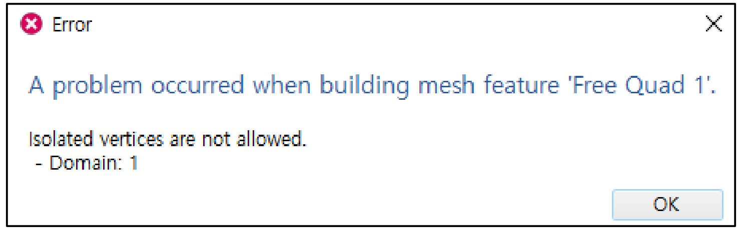
FURTHER CONSIDERATIONS

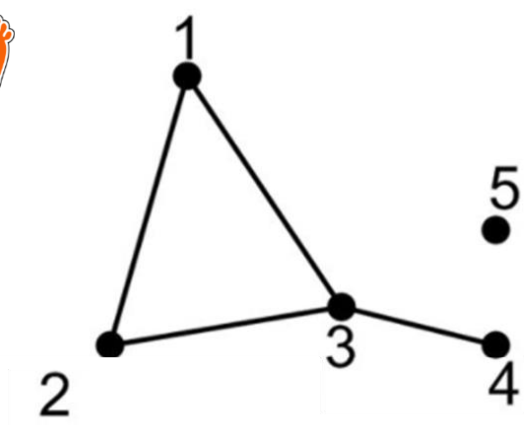
- **Mesh Type (Free Triangular → Free Quad)**

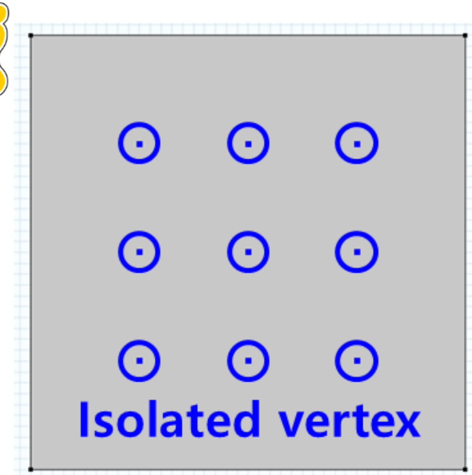
- **Solution Type**

  (Nodal Solution → Element-wise Solution)

MESH CREATION(FREE QUAD)







- A vertex is called **isolated vertex** if it does not have any edge incident to it.
- Vertex 5 is an isolated vertex.

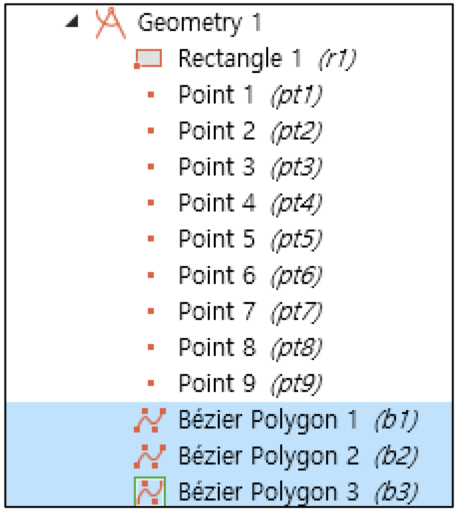
1 Mesh를 바로
Free Triangular에서
Free Quad 로 바꿔
생성을 시도하면 오류 발생

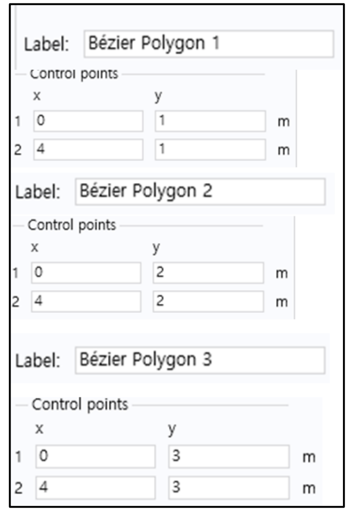
2 COMSOL은
Isolated vertices이 존재할 때
Free Quad Mesh 생성불가

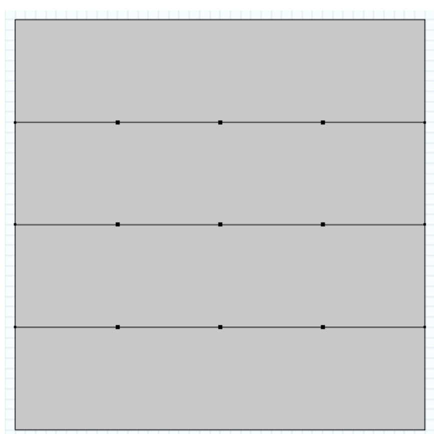
Isolated vertices의 정의는
Edge로 연결되지않은 점

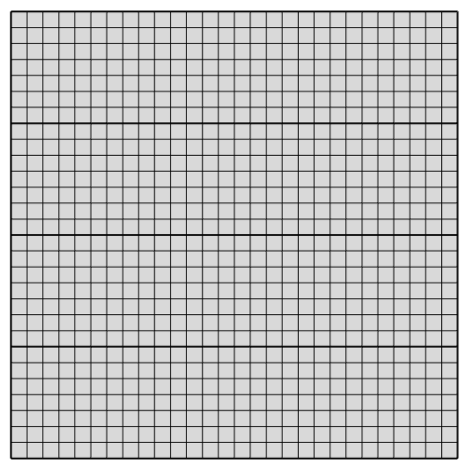
3 예제에서 오류의 원인인
Isolated vertices은
Domain내 생성된 연결되지
않은 점(9개의 점)

MESH CREATION(FREE QUAD)







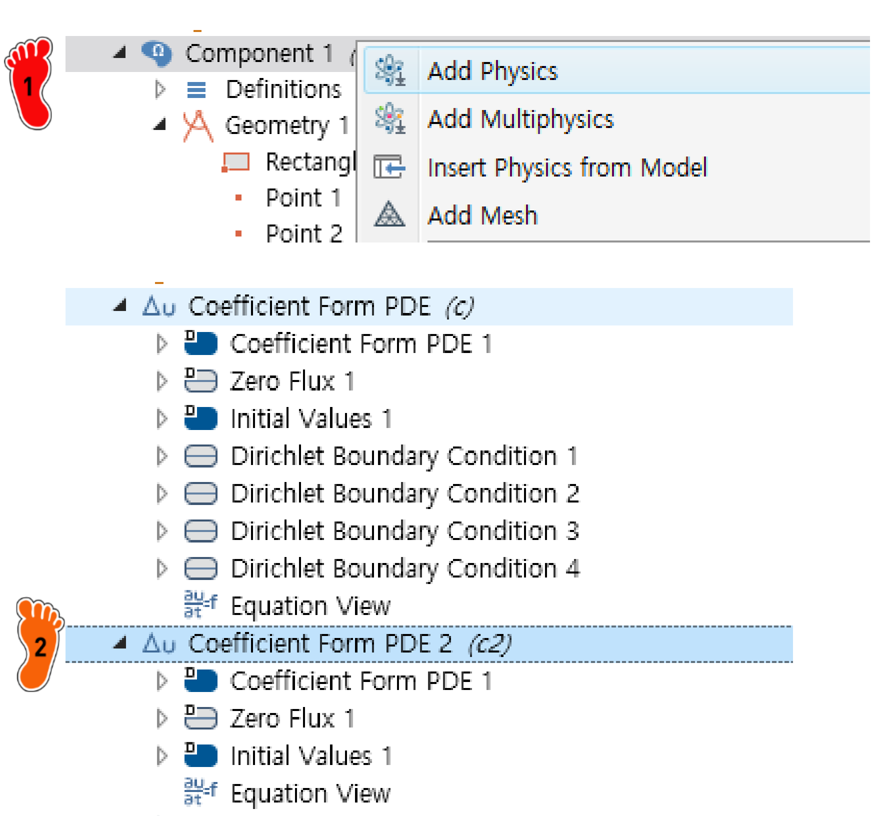


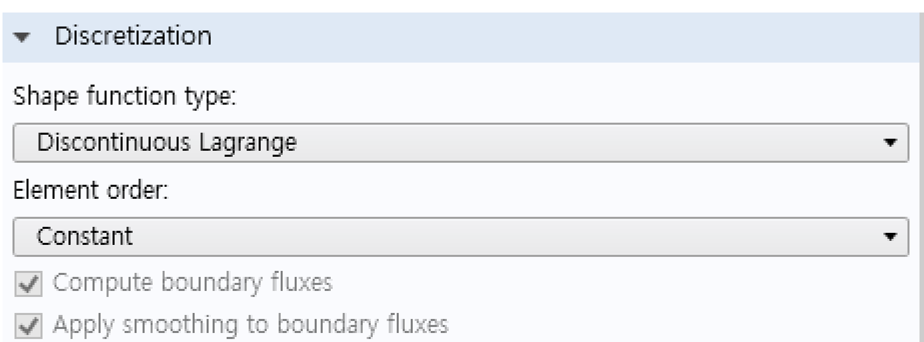
Free Quad Mesh

1. Bezier Polygon을 추가하여 설계 영역 내 Isolated vertices로 정의되는 점들을 선으로 연결

2. 선으로 연결된 점들은 더 이상 Isolated vertices로 인식되지 않아 Quad Mesh 생성가능

POST PROCESSING(ELEMENT WISE)





1 COMSOL은 기본적으로 절점 기반 솔루션 제공

요소 별 솔루션을 얻기 위해선 Coefficient Form PDE를 하나 더 추가하여 절점기반 솔루션을 후처리 작업하는 필요함

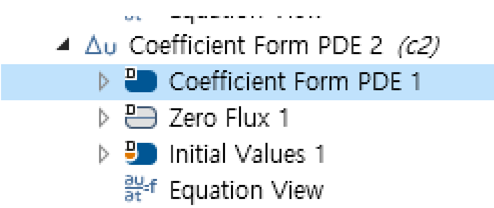
2 추가된 Coefficient Form PDE의 Discretization 설정

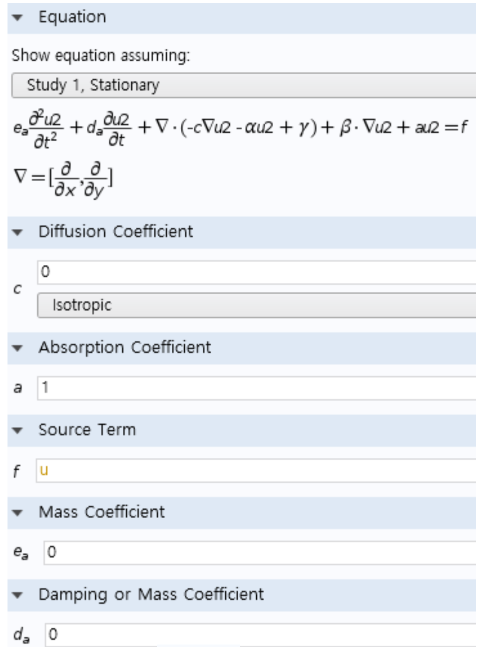
형상함수: Discontinuous Lagrange

요소차수: Constant

->요소 기반 솔루션 출력

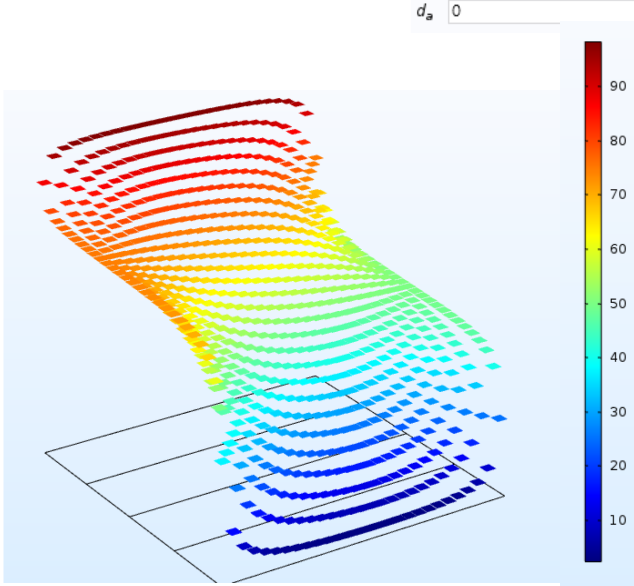
POST PROCESSING(ELEMENT WISE)





$$a = 1,\ f = u$$

$$\rightarrow u^{Node} = u_2^{Element}$$



1 후처리를 위한 지배방정식

절점 기반 솔루션 (u)
=요소 기반 솔루션 (u2)

요소 기반 솔루션의 경우

한 요소당 하나의 값을 갖고,
요소 간에 불연속한 솔루션
을 갖는다.

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

HEAT CONDUCTION EQUATION

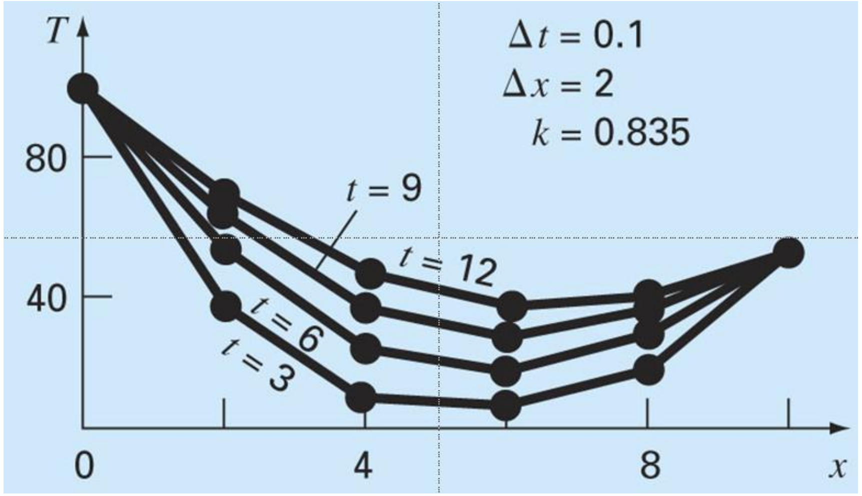
heat stored in the element over a unit time period $\Delta t = (In) - (Out)$

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

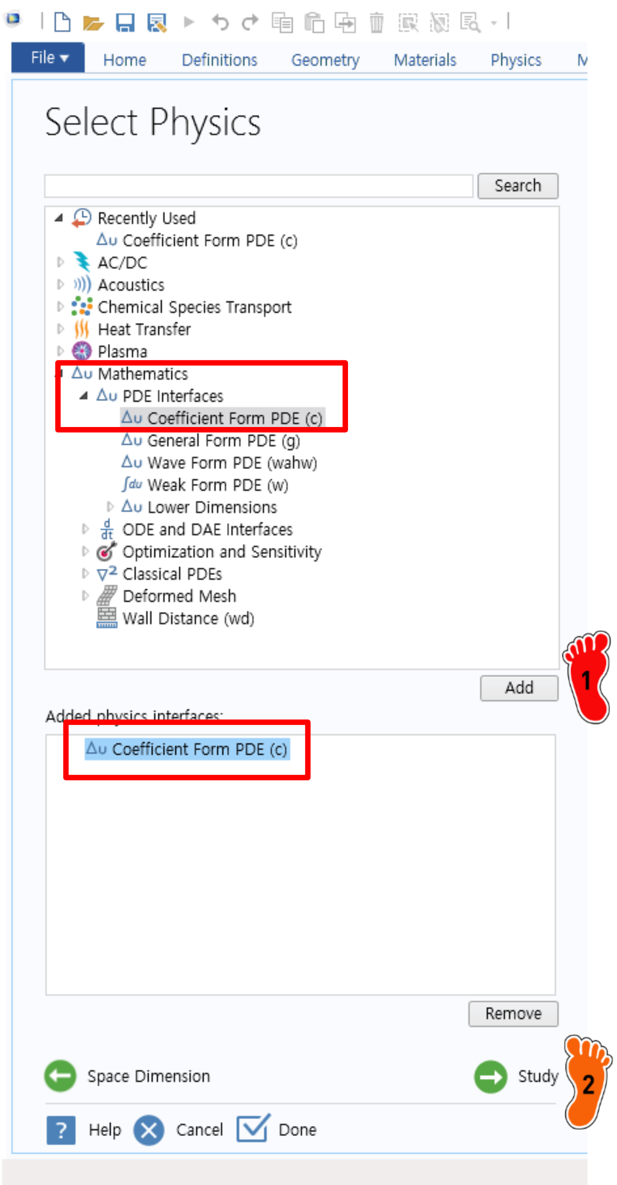
$$\frac{q(x) - q(x+\Delta x)}{\Delta x} = \rho C \frac{\Delta T}{\Delta t} \xrightarrow{\text{taking the limit}} -\frac{\partial q}{\partial x} = \rho C \frac{\partial T}{\partial t} \xrightarrow{q=-k\rho C\frac{\partial T}{\partial x}} k\frac{\partial^2 T}{\partial x^2} = \frac{\partial T}{\partial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

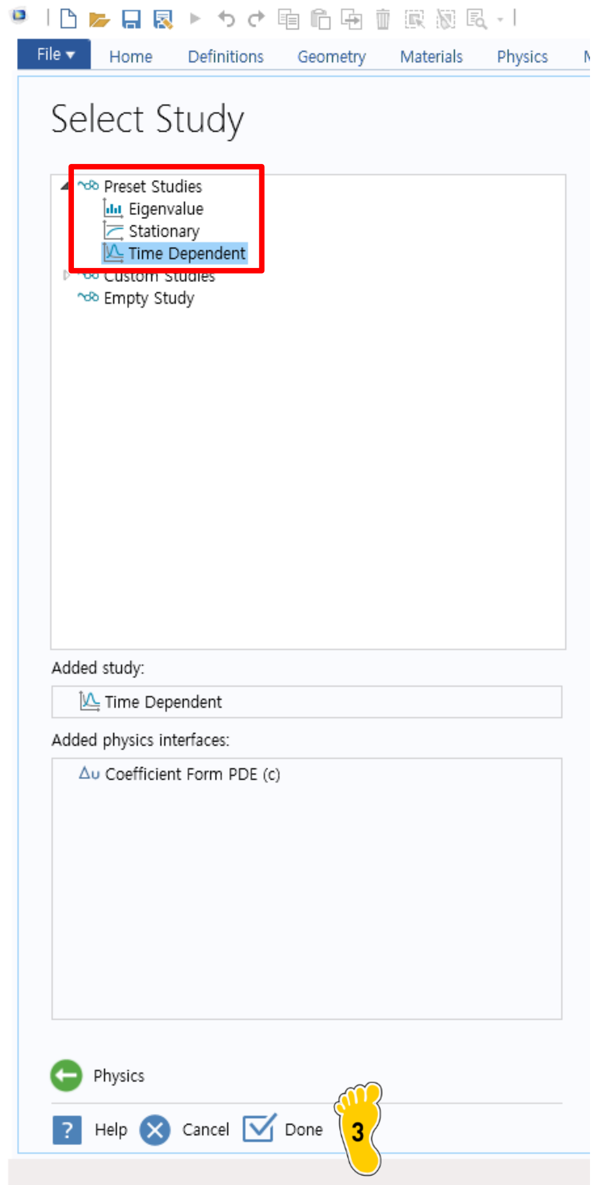
$$\begin{cases} k = 0.835 \text{ cal}/(\text{s}\cdot\text{cm}\cdot{}^\circ\text{C}) \\ \Delta x = 2 \text{ cm} \\ \Delta t = 0.1 \text{ s} \end{cases}$$

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



SETTING



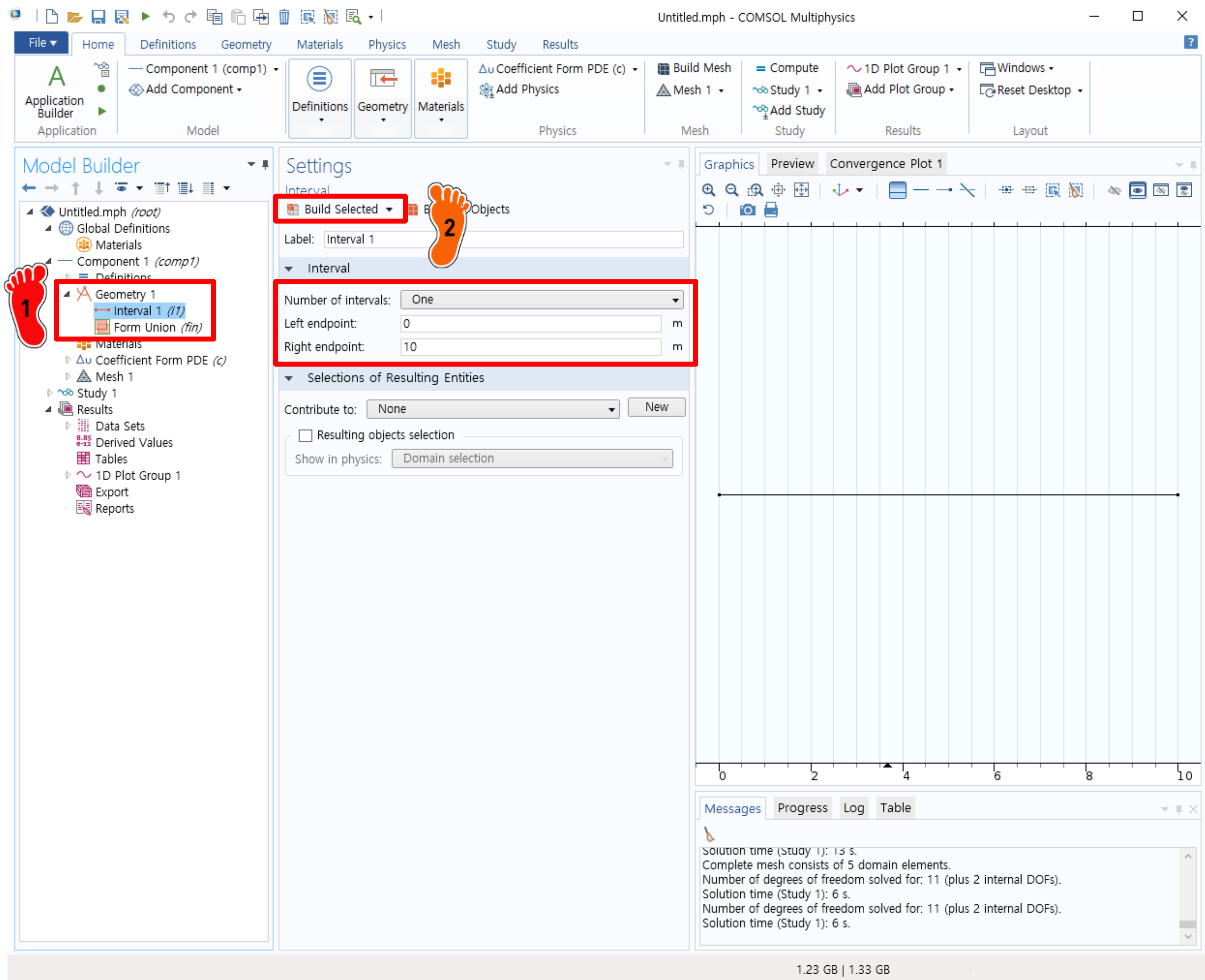


1. Model Wizard → 1D → Mathematics - PDE Interfaces – Coefficient form PDE 선택 → Add 클릭

2. Study 클릭

3. Time Dependent 선택 후 Done 클릭

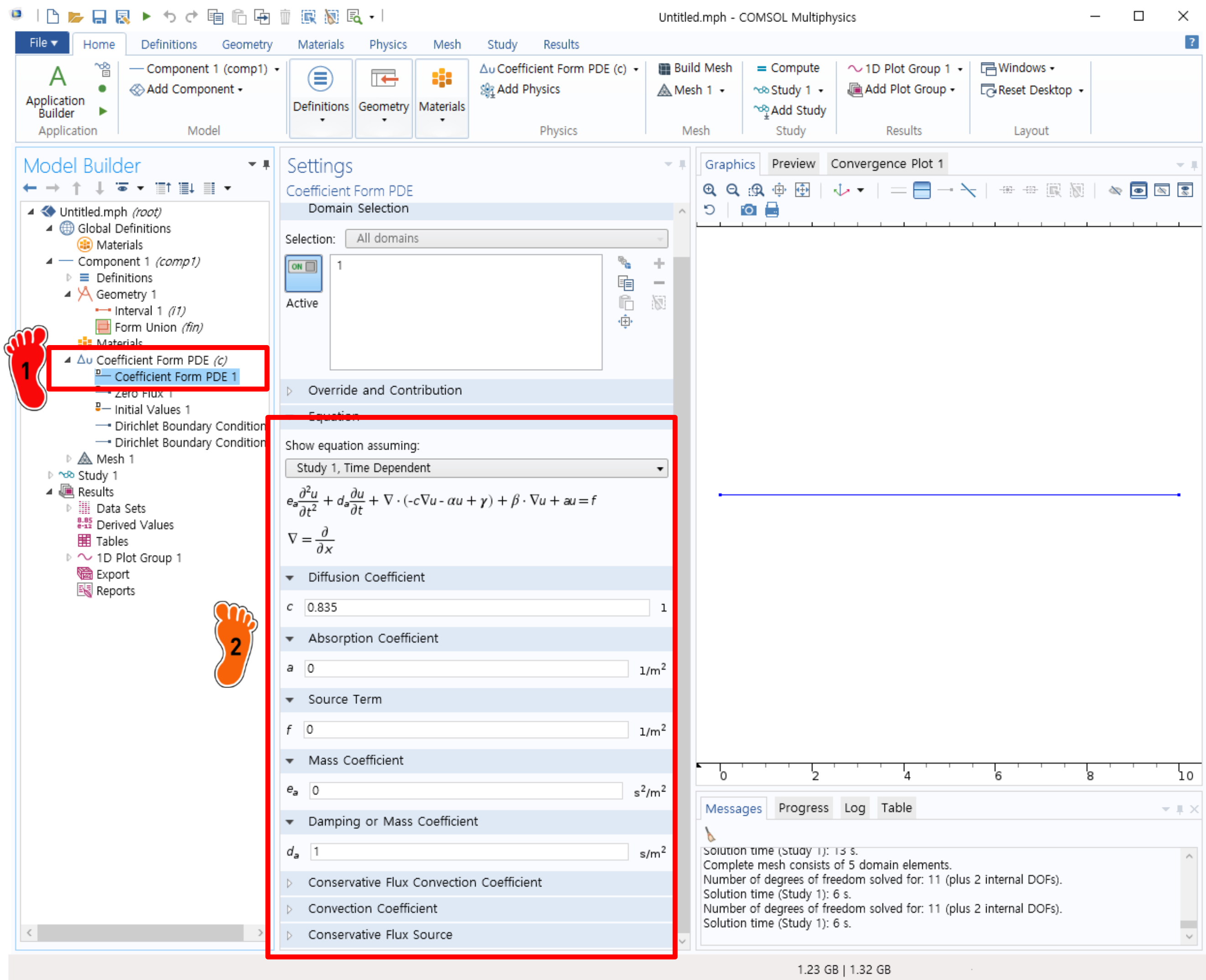
GEOMETRY CREATION



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)

$$c = 0.835$$

$$d_a = 1$$

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

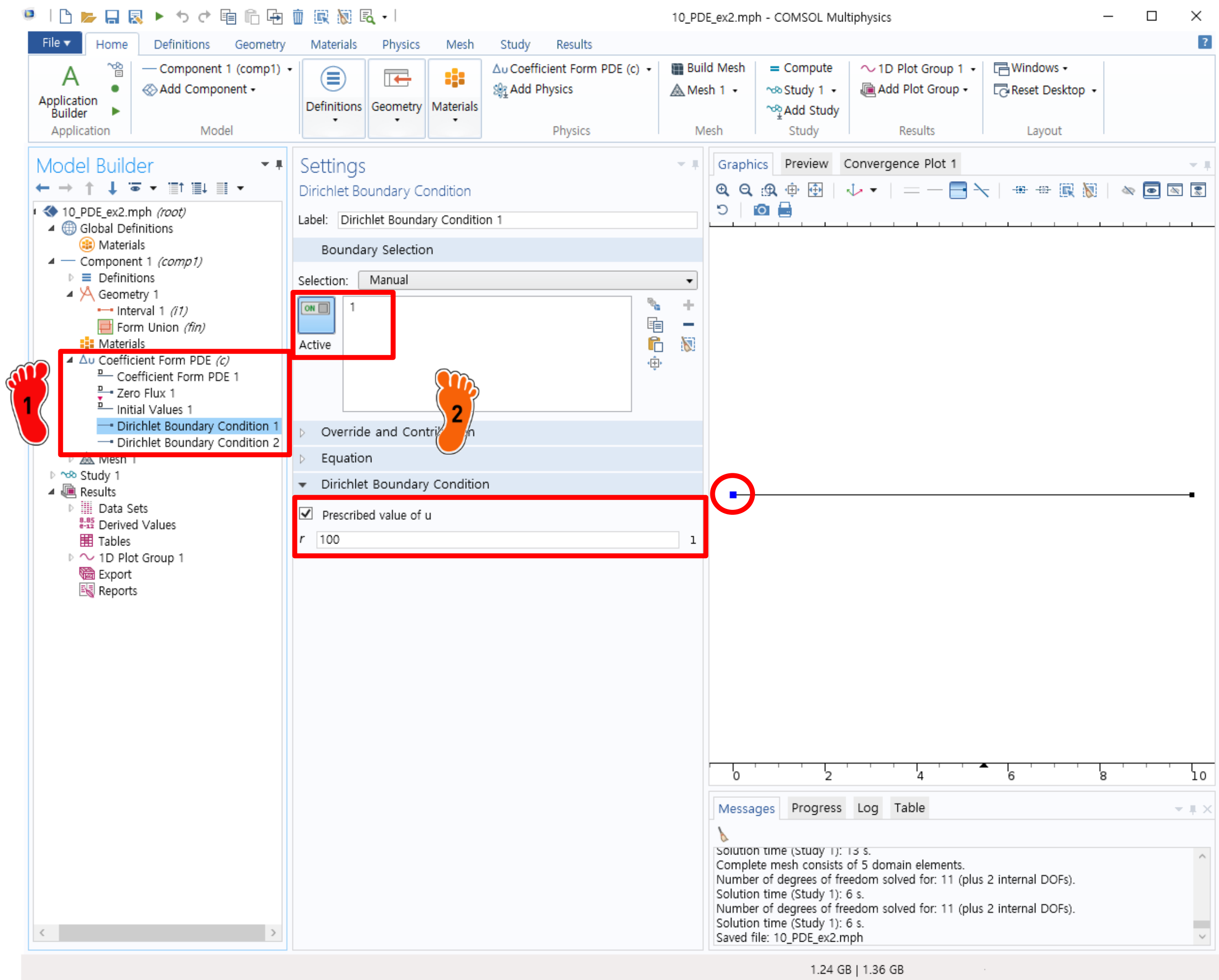
$$\updownarrow$$

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

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

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

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)

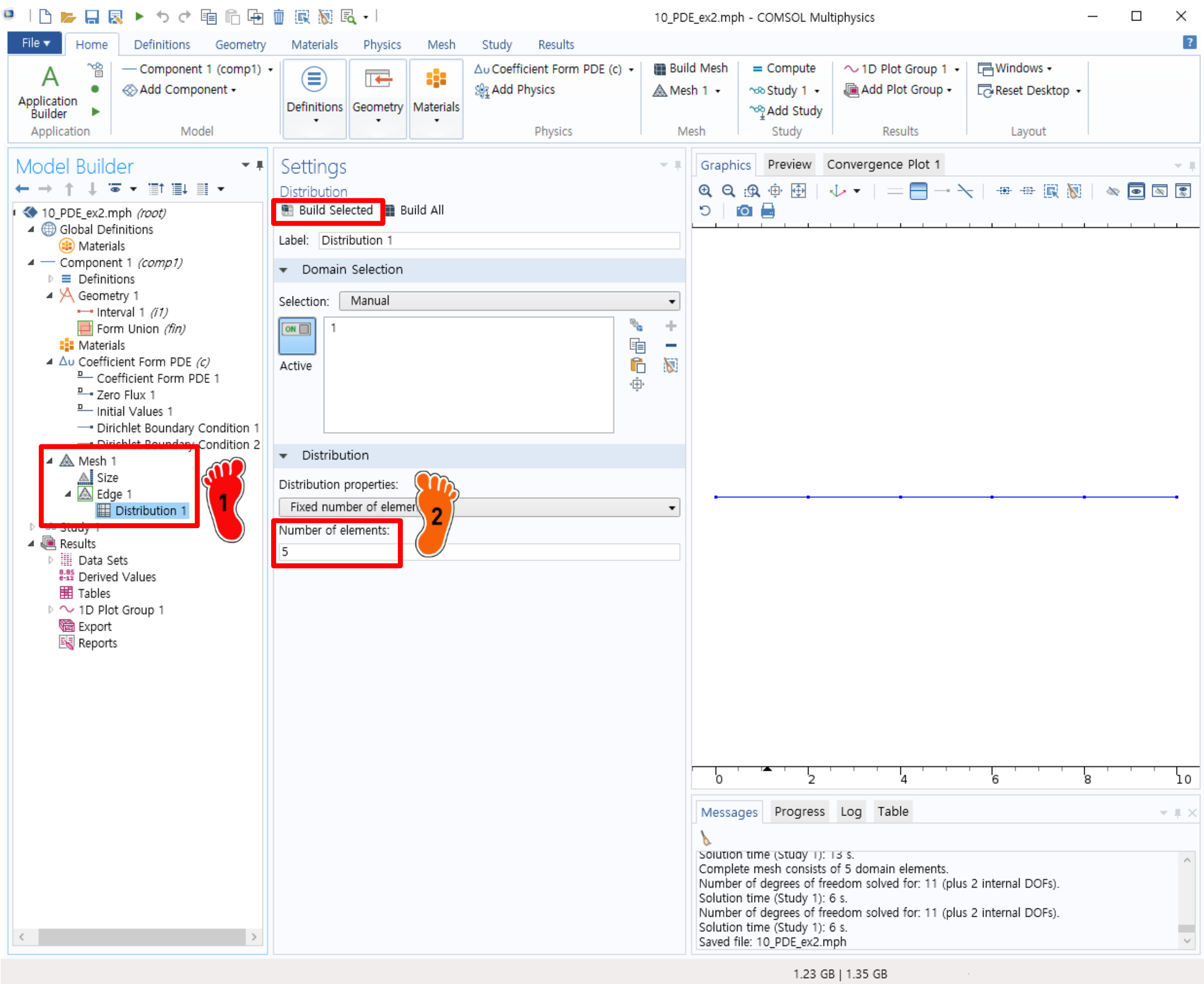
Boundary condition

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

Initial condition

$$\begin{cases} T(i,t=0) = 0°C \end{cases}$$

MESH CREATION

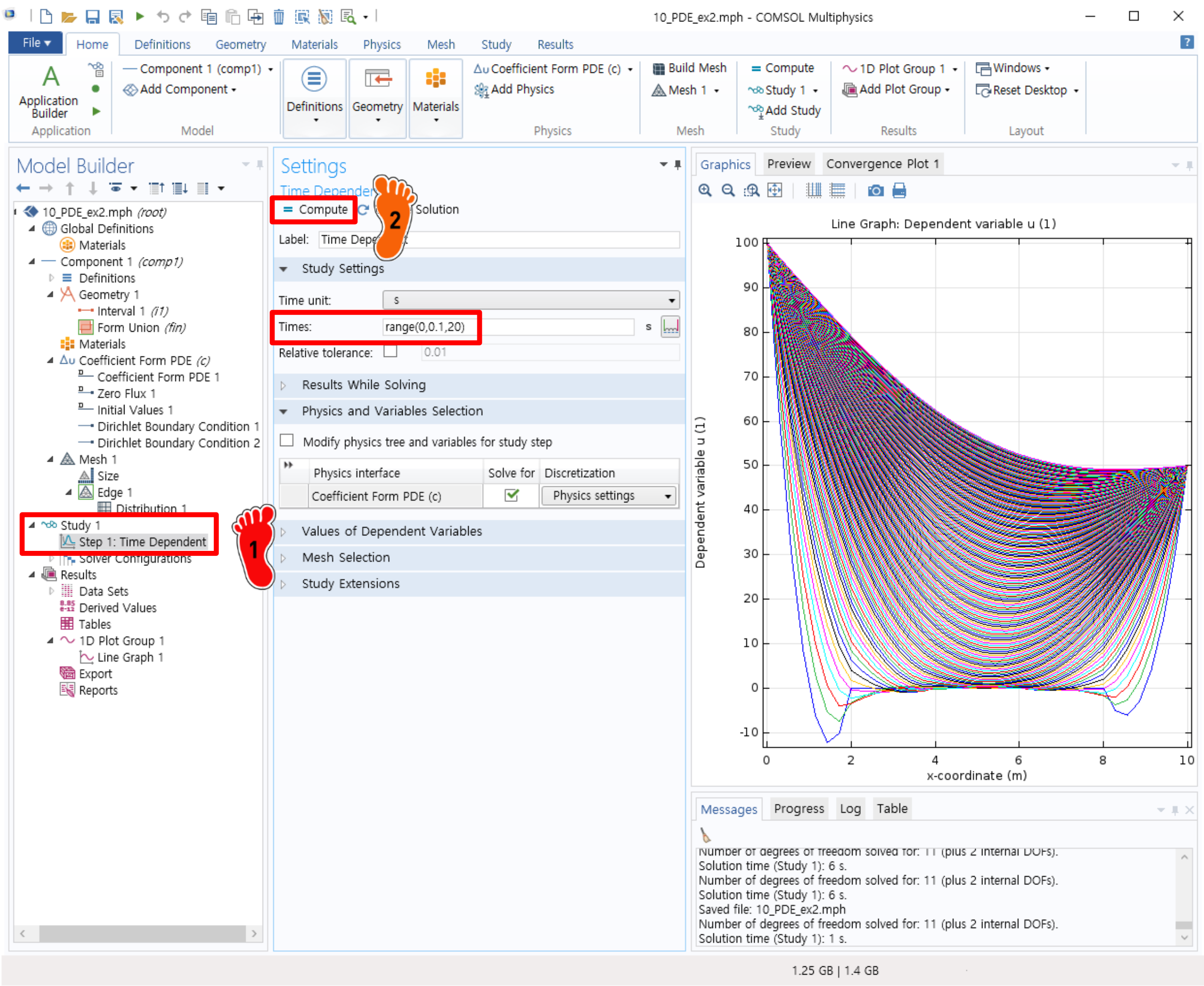


1 Mesh 1 메뉴 우클릭
→ Edge 선택
→ Edge 메뉴 우클릭
→ Distribution 선택

2 Number of elements에 5 입력 → Build Seleted 클릭

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

COMPUTE

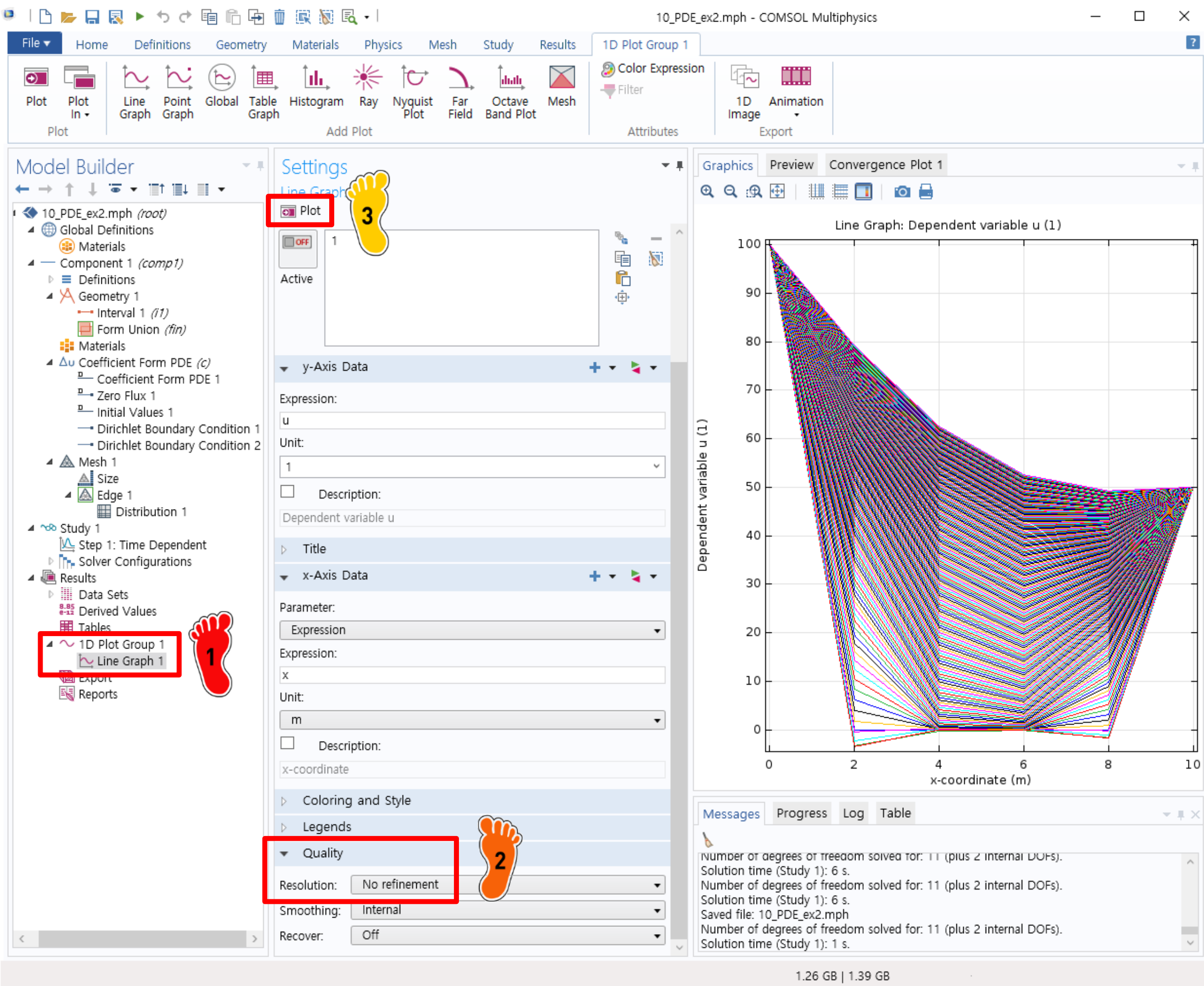


1 Study 1 → Step 1: Time Dependent 클릭

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

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

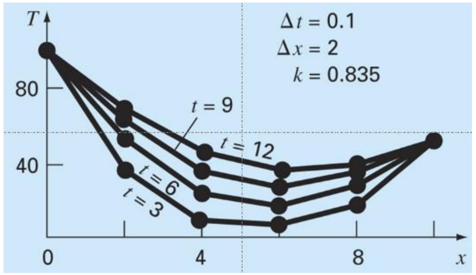
POST-PROCESSING



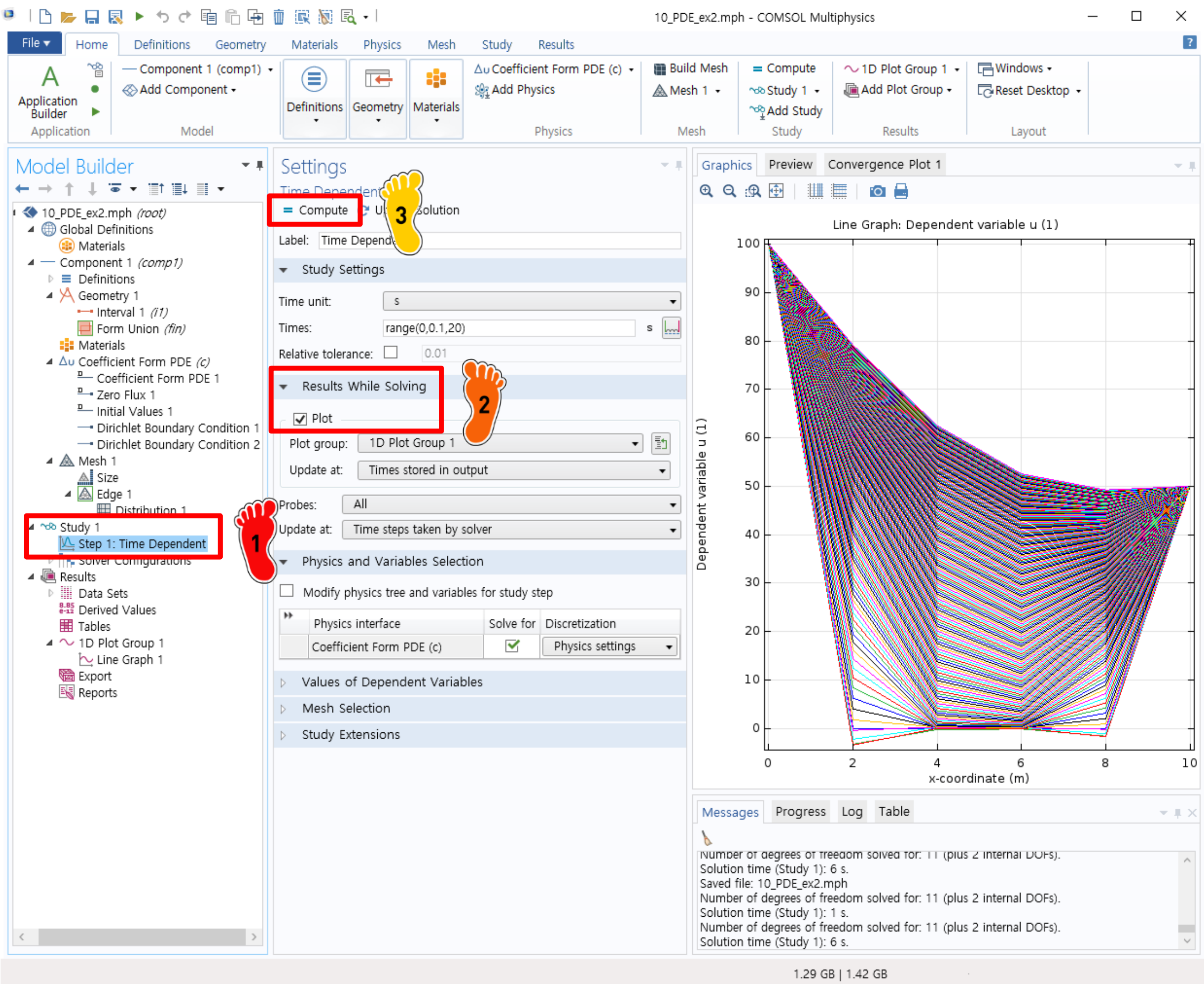
1 Line Graph 1 클릭

2 Quality → Resolution 에서 No refinement로 변경

3 Plot 클릭



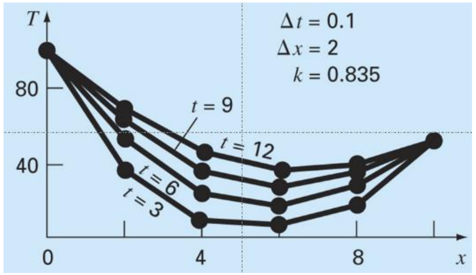
POST-PROCESSING



1. Study 1 → Step 1: Time Dependent 클릭

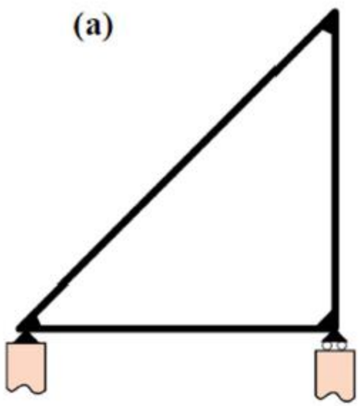
2. Results While Solving에서 Plot 선택

3. Compute 클릭 (시간에 따른 변화 확인)

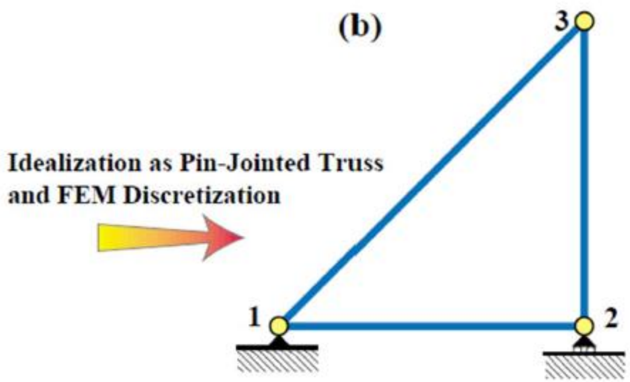


- **Direct Stiffness Method: Truss**
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ **Analysis**

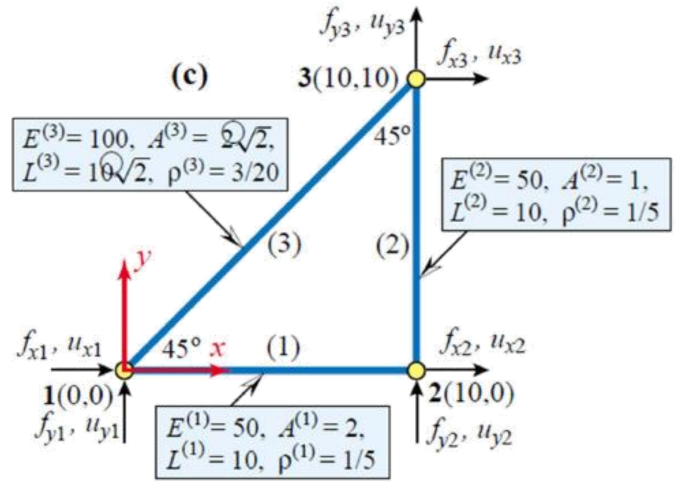
EXAMPLE TRUSS STRUCTURE



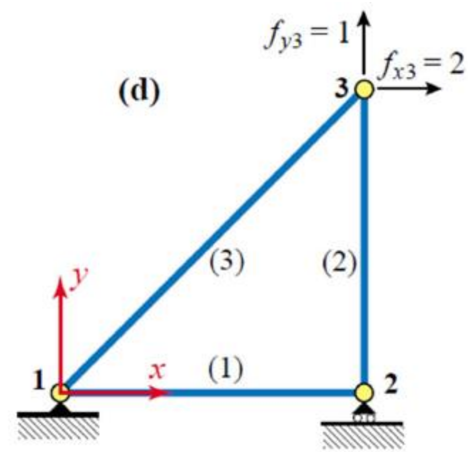
Physical structure



Idealization as a pin-jointed bar assemblage

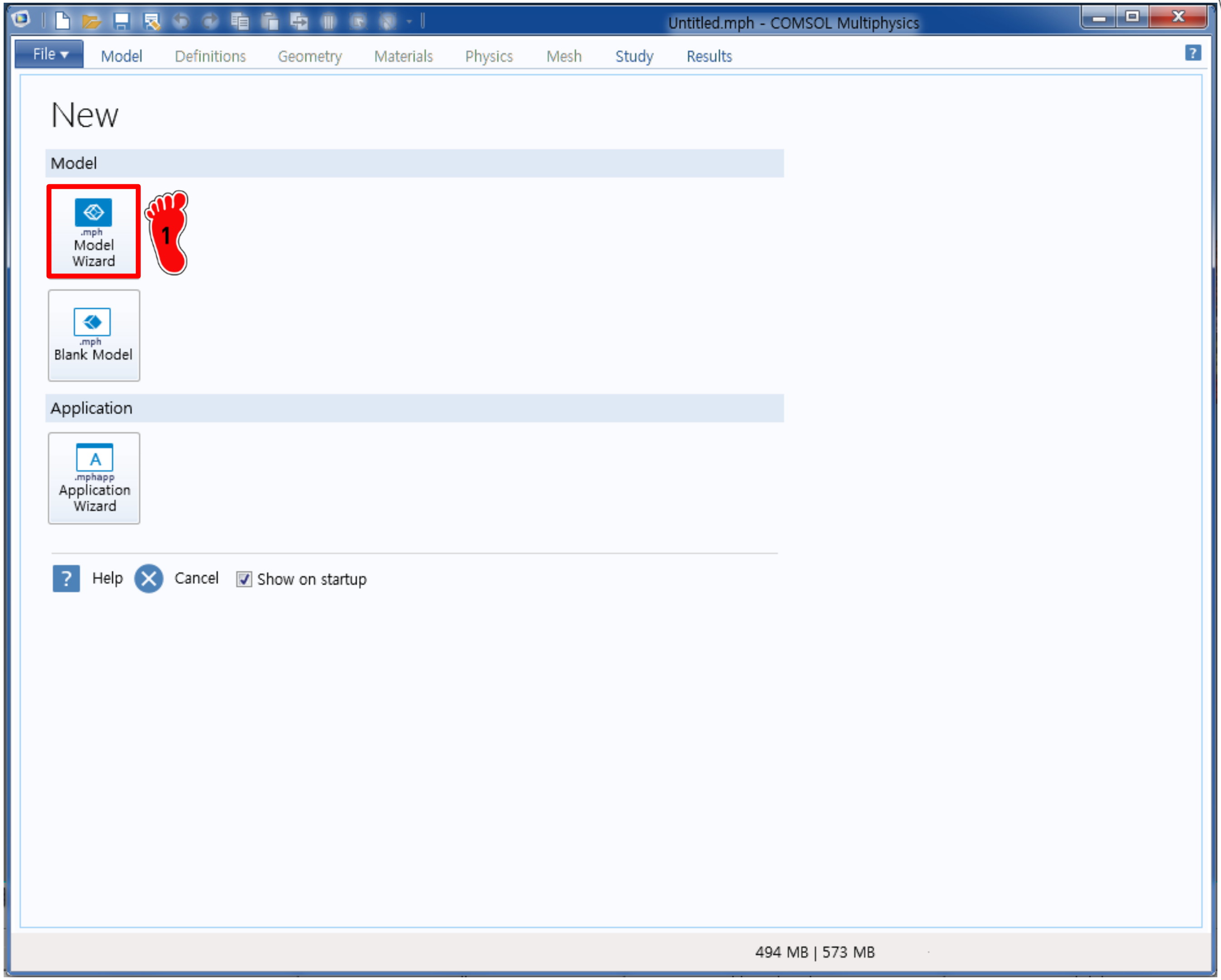


Geometric, material and fabrication properties



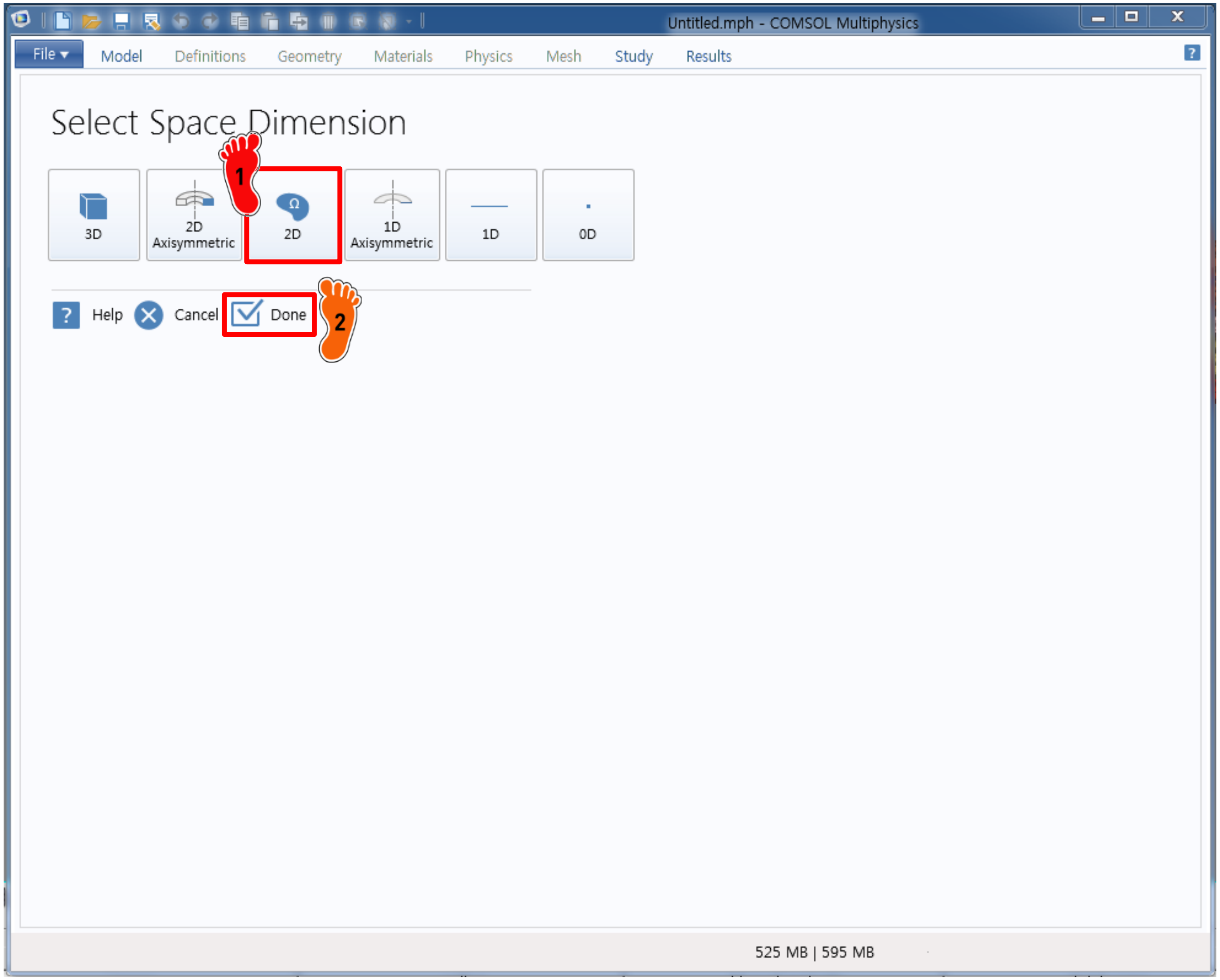
Support conditions and applied loads

MODEL WIZARD



1 Model Wizard 선택

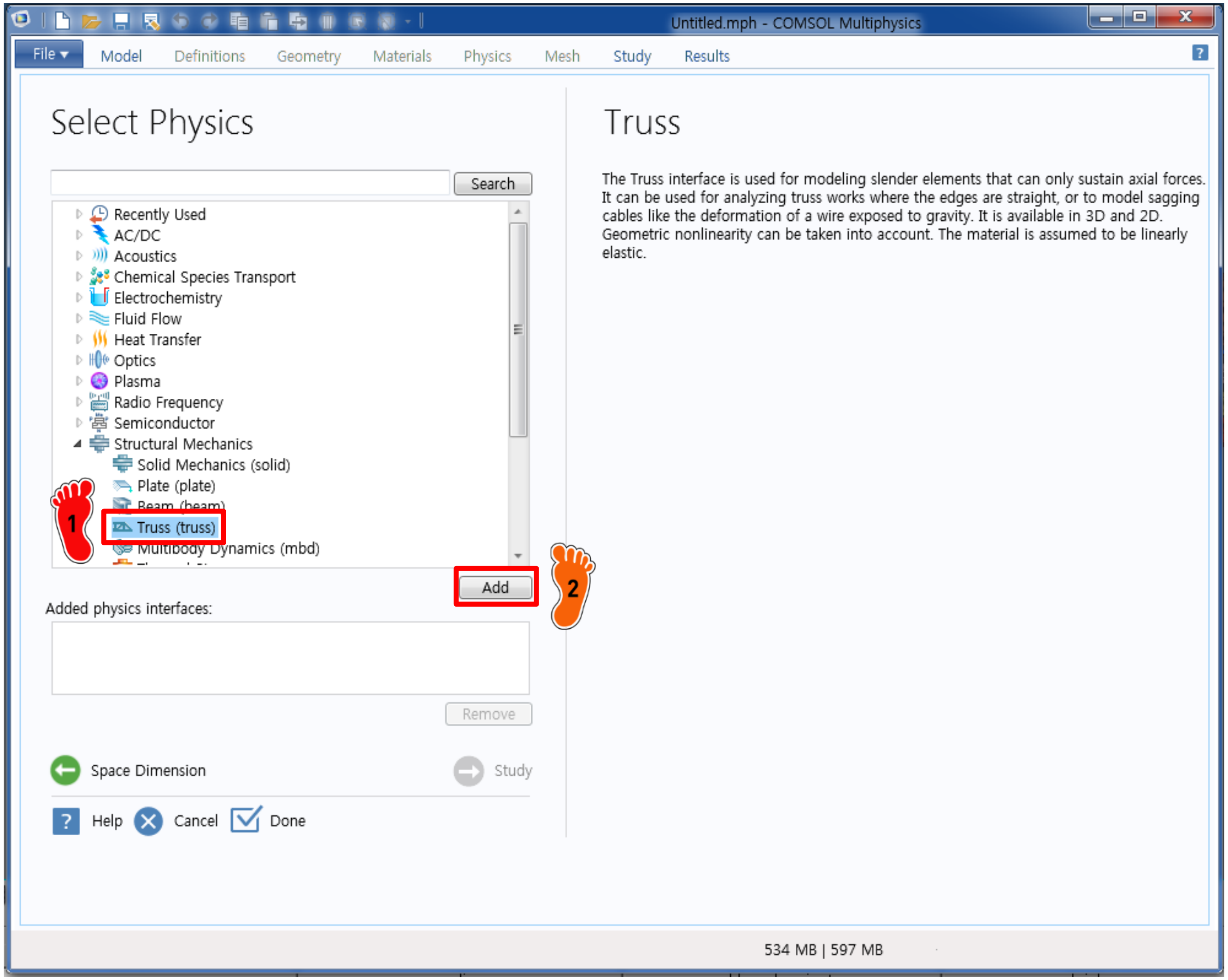
DIMENSION SELECTION



1 2D 선택

2 Done 클릭

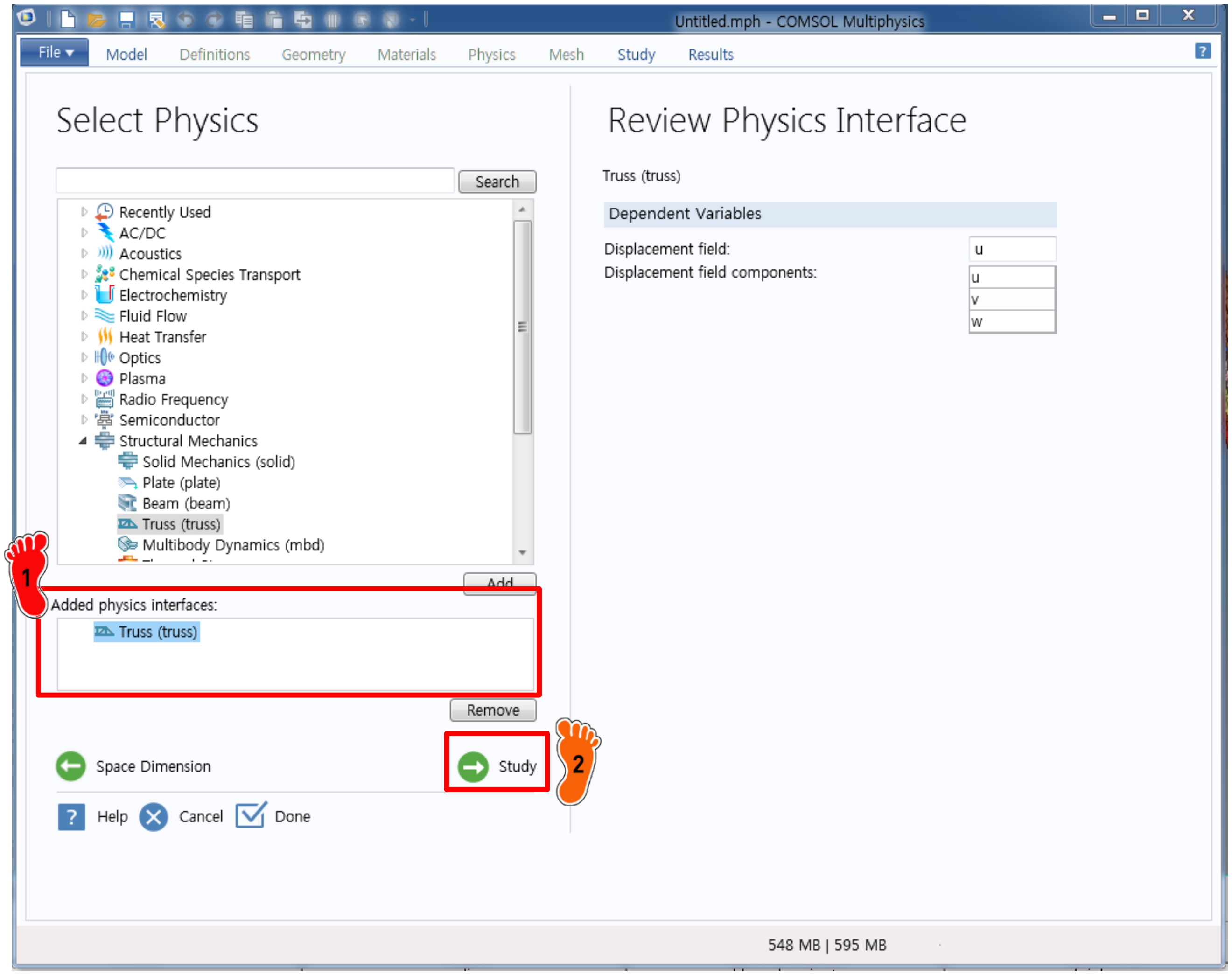
PHYSICS SELECTION



1 Structural Mechanics 모듈의 Truss physics 선택

2 Add 클릭

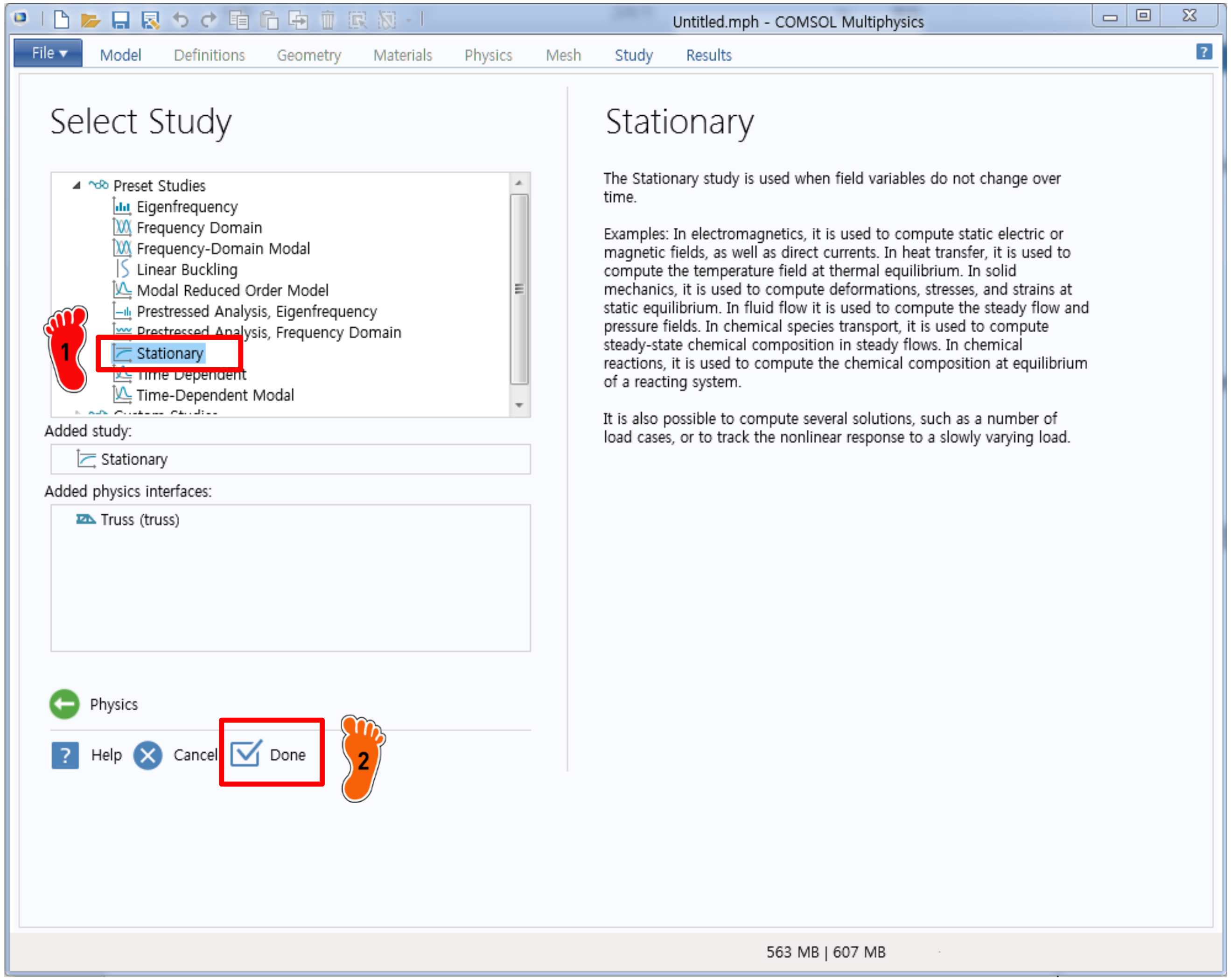
PHYSICS SELECTION



1 Added physics interfaces에서 추가된 physics 확인

2 Study 클릭

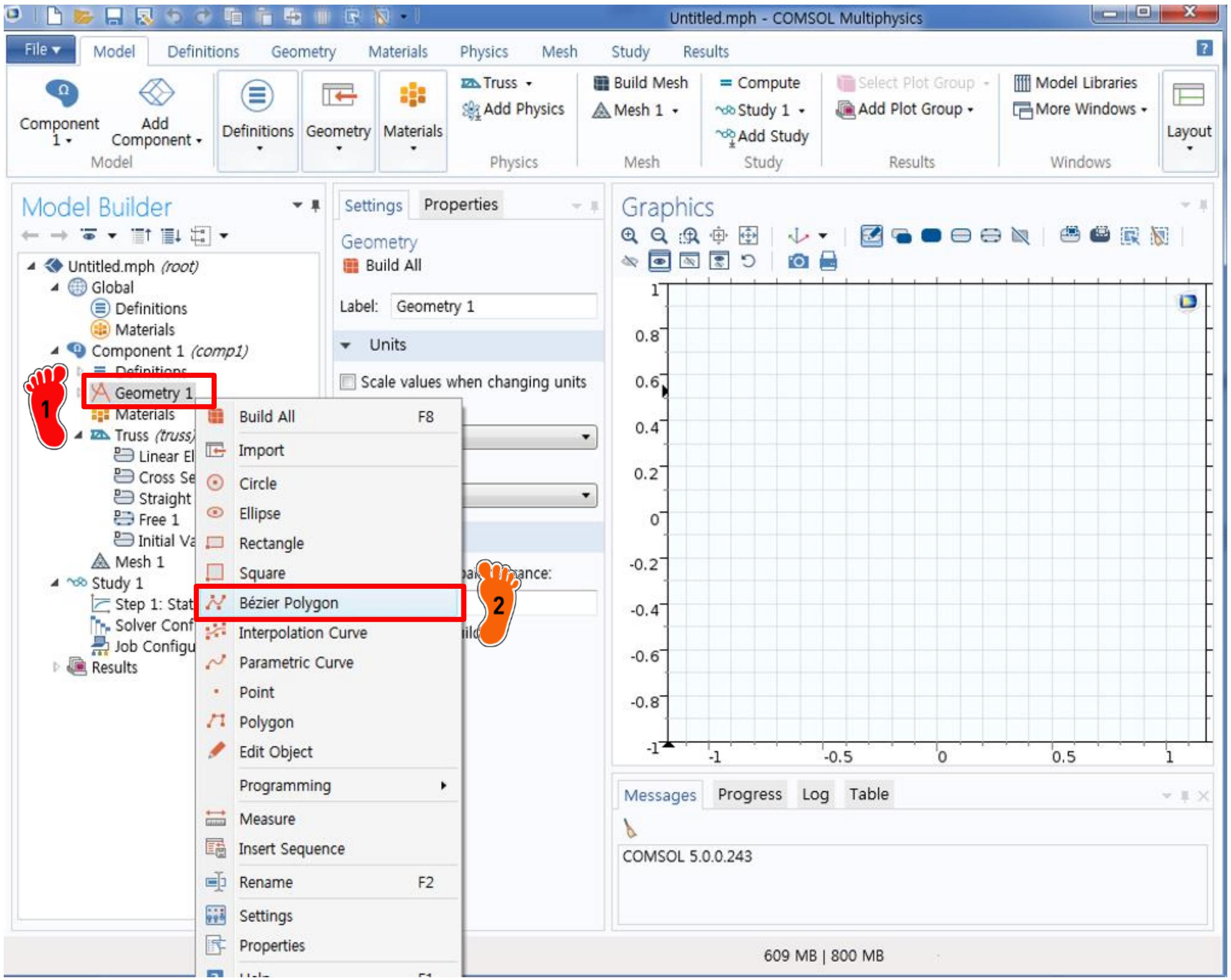
STUDY TYPE SELECTION



1 Preset Studies에서 Stationary 선택

2 Done 클릭

GEOMETRY CREATION

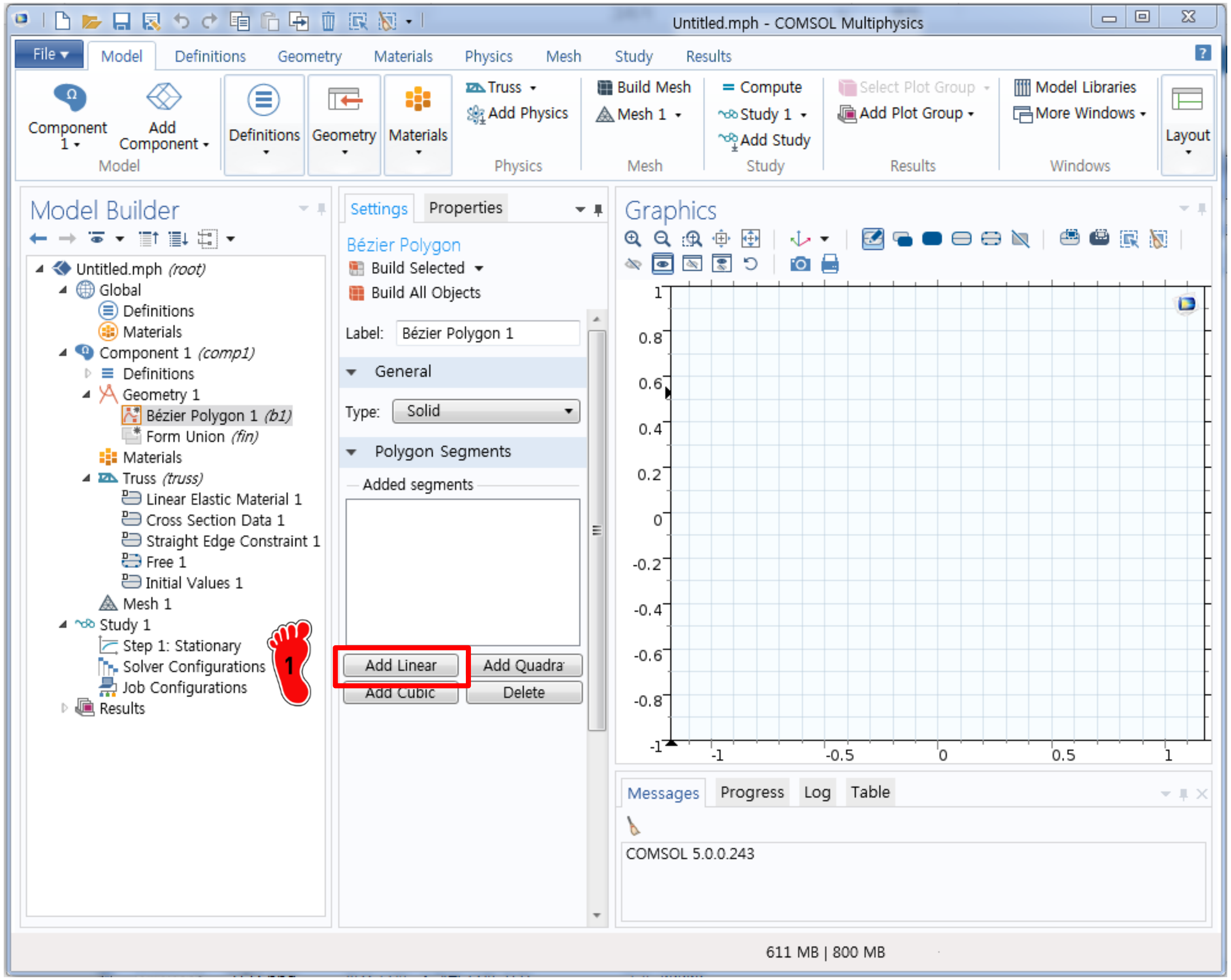


1 Geometry 1 마우스 우클릭

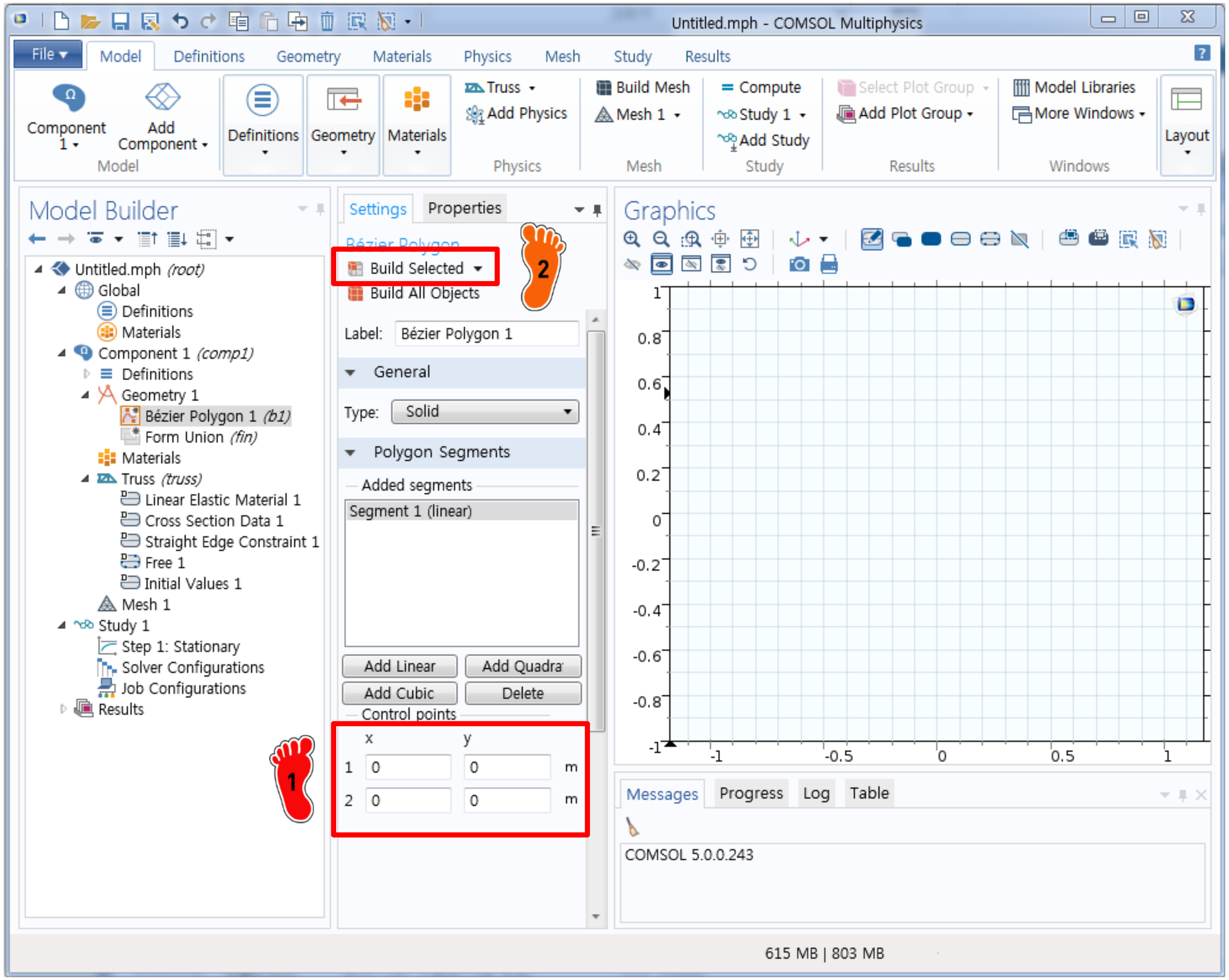
2 Bezier Polygon 클릭

GEOMETRY CREATION

1 Add Linear 클릭



GEOMETRY CREATION



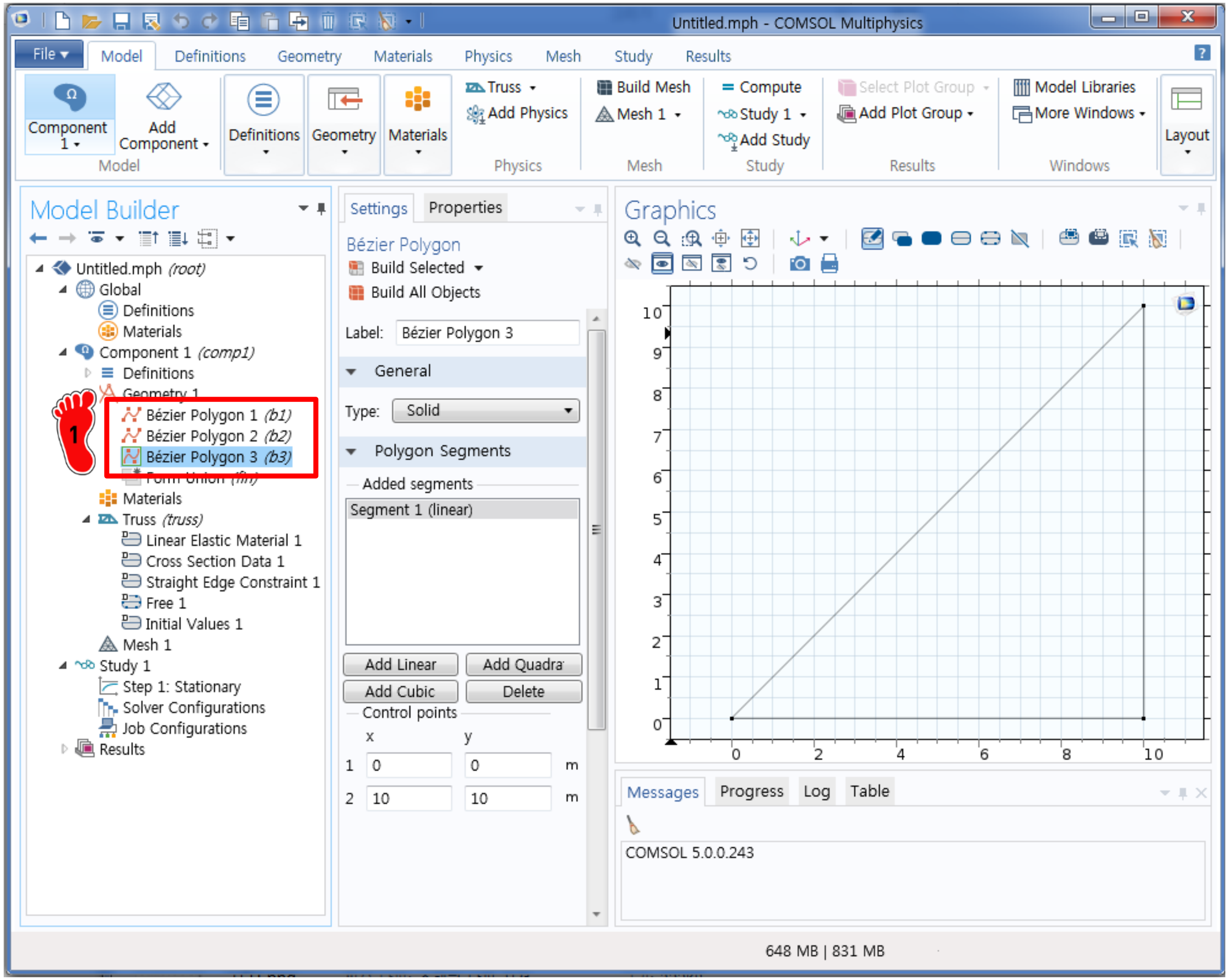
1 Control points 시작점 (0,0) 끝나는 점 (10,0) 을 입력

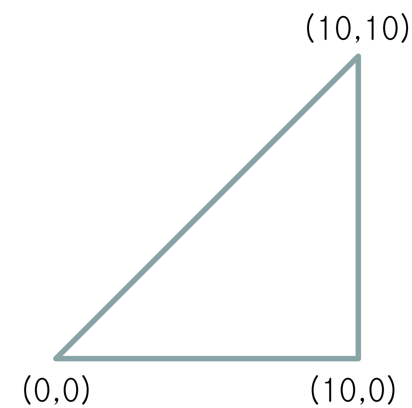
옆의 숫자 1 은 시작점 좌표, 2는 끝나는 점을 의미

2 Build Selected 클릭

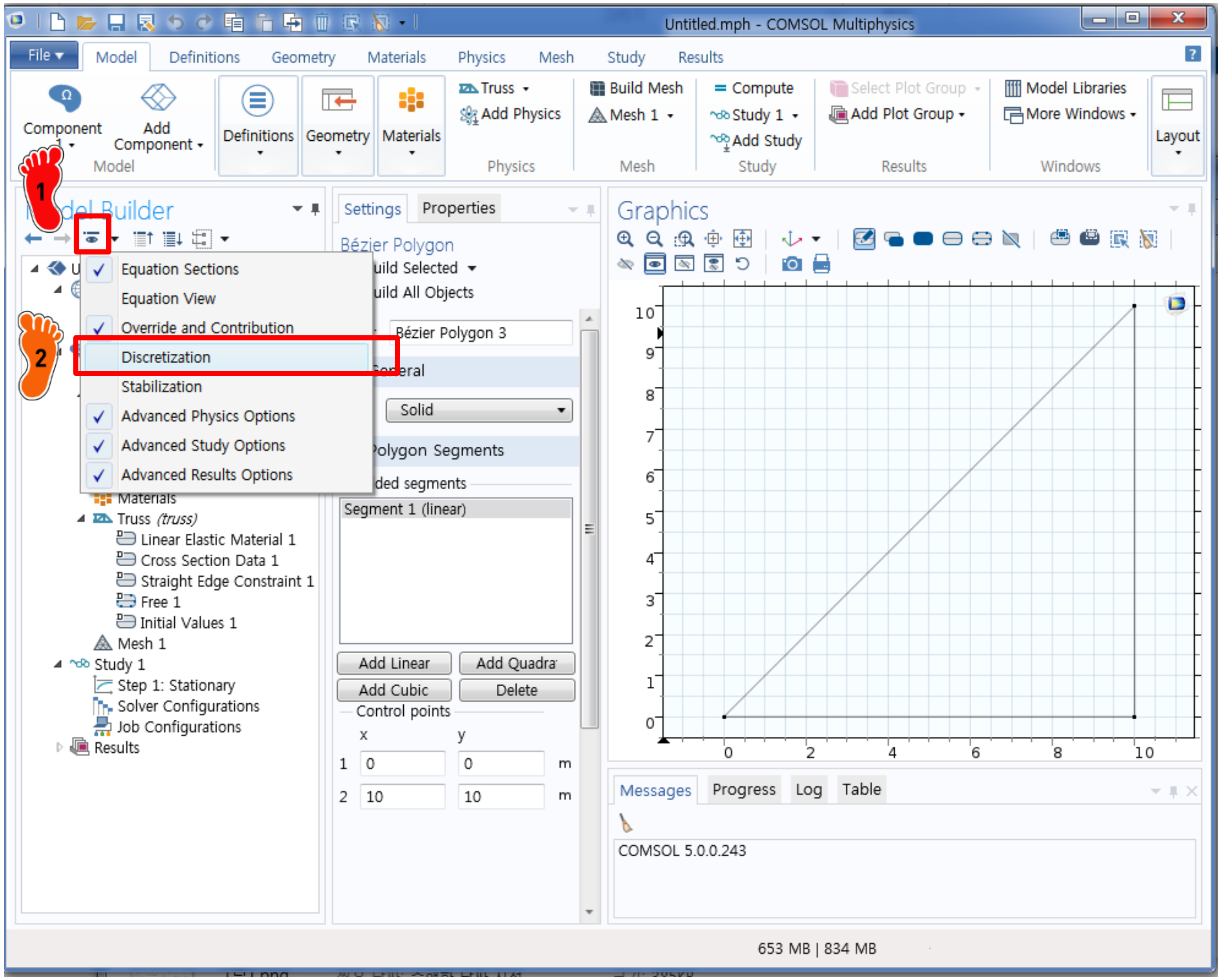
GEOMETRY CREATION

1 같은 방식으로 총 3개의 직선을 생성





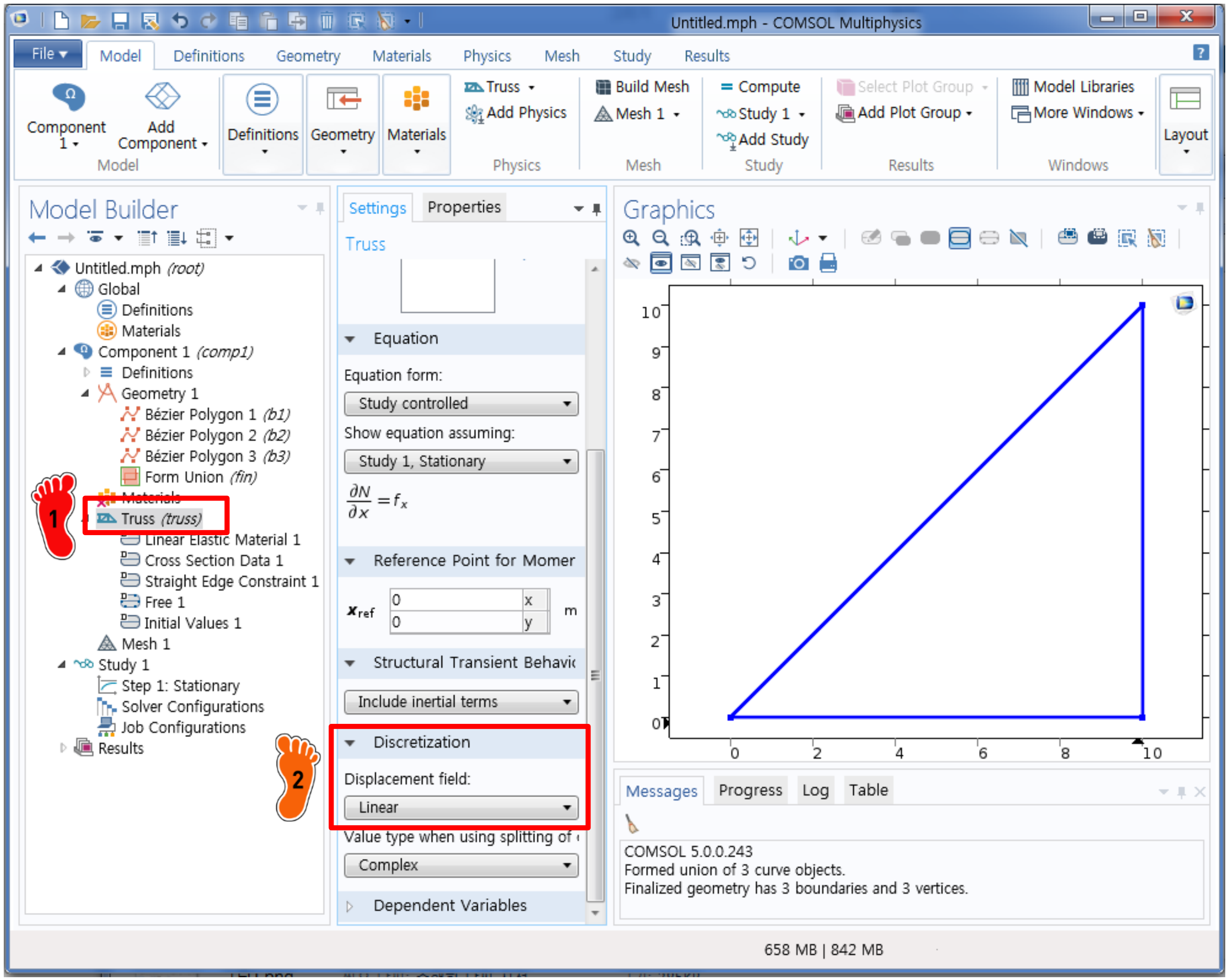
ELEMENT ORDER CHANGING



1 Model Builder 밑에 있는 눈 모양 메뉴 클릭

2 Discretization 클릭

ELEMENT ORDER CHANGING



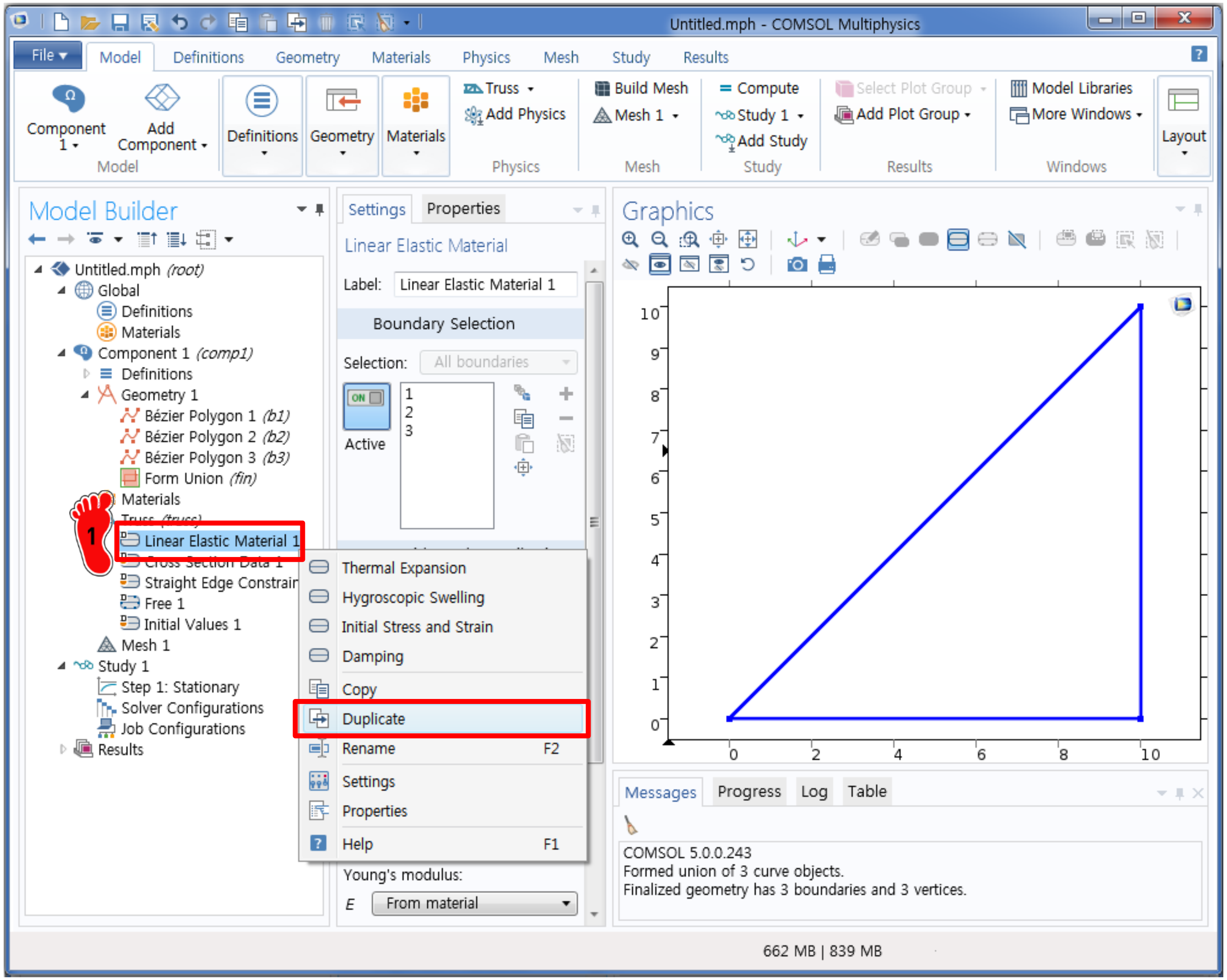
1 Truss physics 클릭

2 Displacement field 를 Linear 로 변경

- **Direct Stiffness Method: Truss**

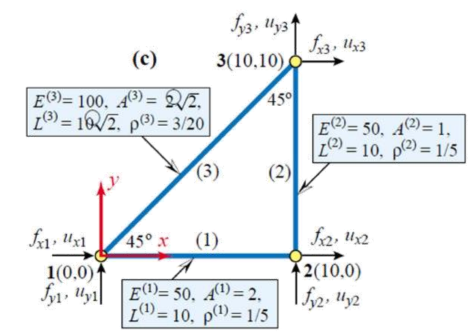
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ **Analysis**

MATERIAL PROPERTY



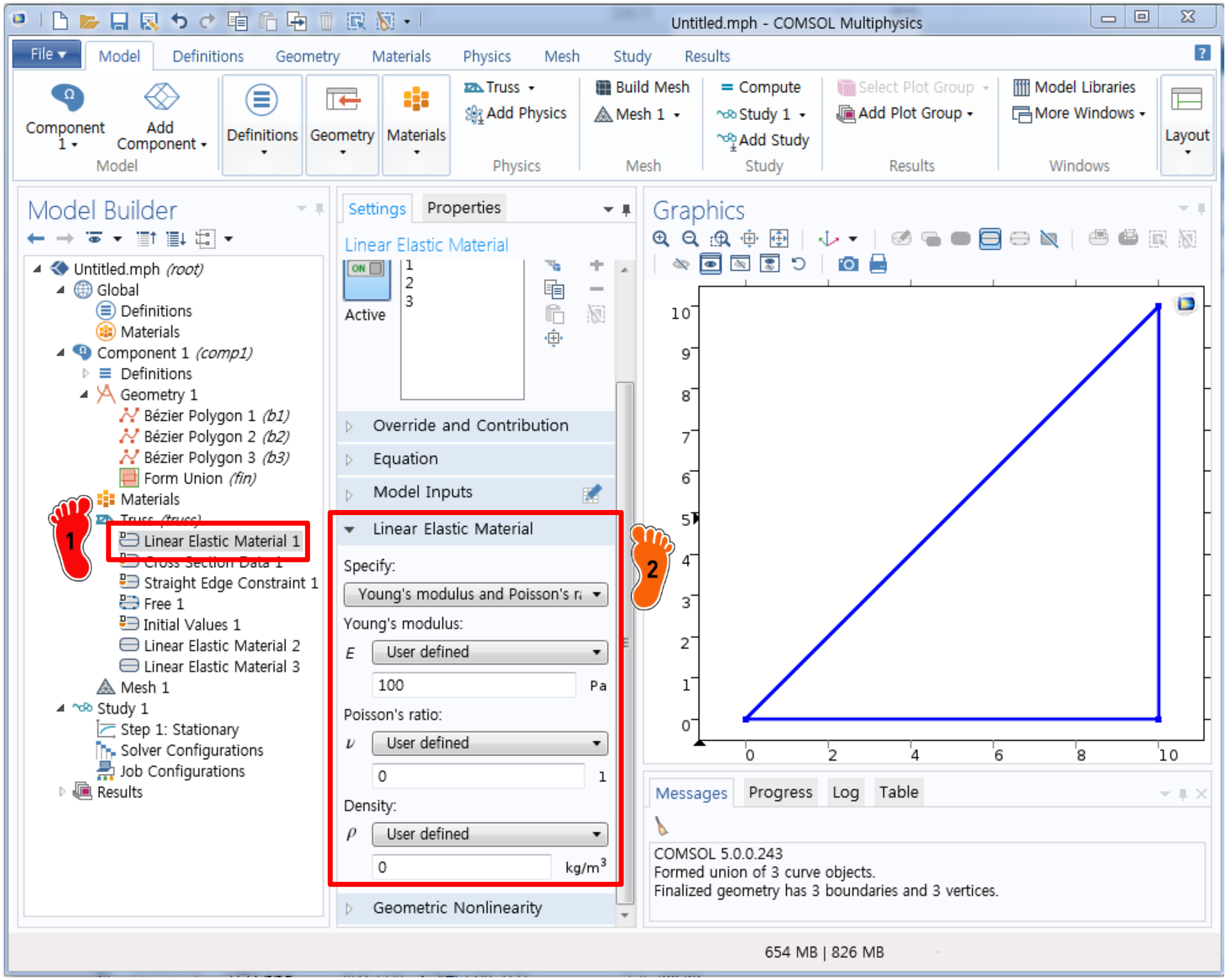
1 Linear Elastic Material 마우스 우클릭

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



Geometric, material and fabrication properties

MATERIAL PROPERTY

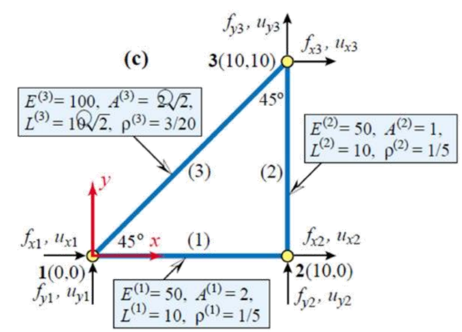


1 첫 번째 Linear Elastic Material 클릭

2 From material을 User defined로 변경 후

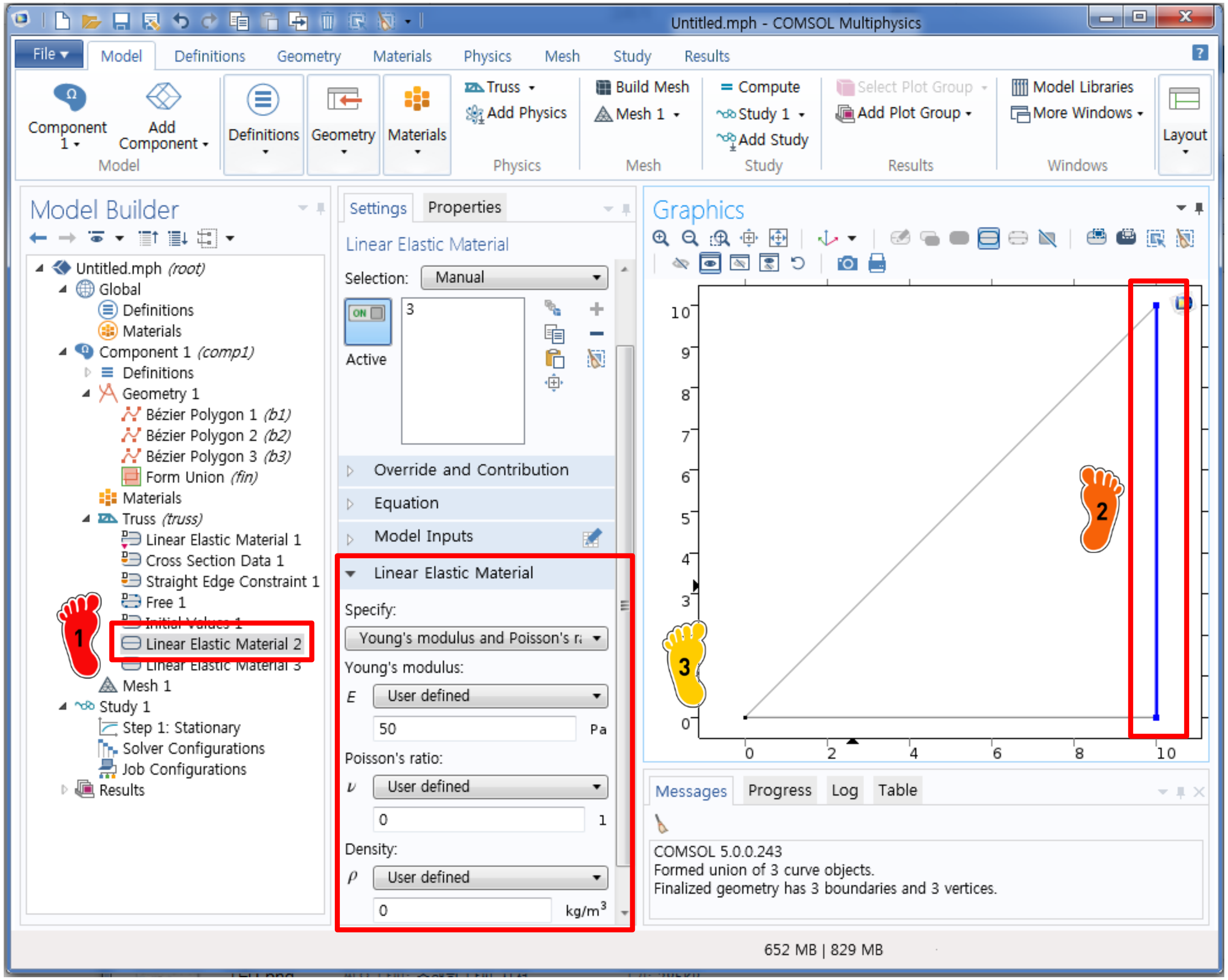
Young`s modulus 100 입력

푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

MATERIAL PROPERTY

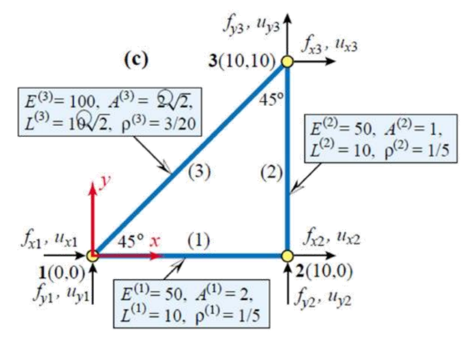


1 두 번째 Linear Elastic Material 클릭

2 제일 오른쪽 부재 선택

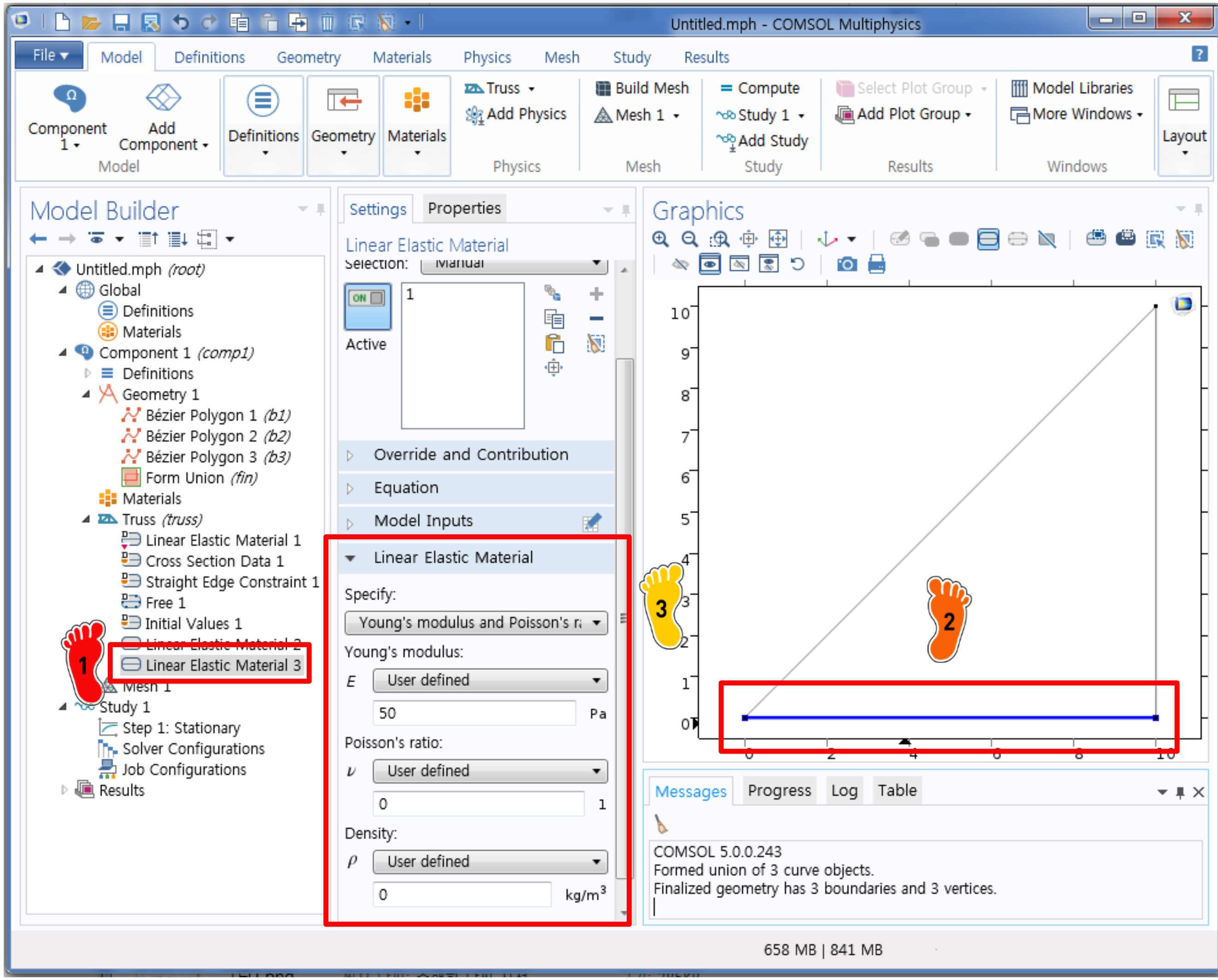
3 Young`s modulus 50 입력

푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

MATERIAL PROPERTY

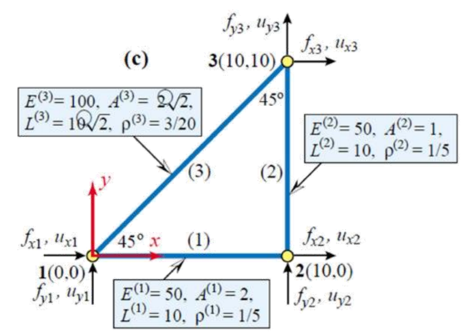


1. 마지막 Linear Elastic Material 클릭

2. 제일 아래 부재 선택

3. Young`s modulus 50 입력

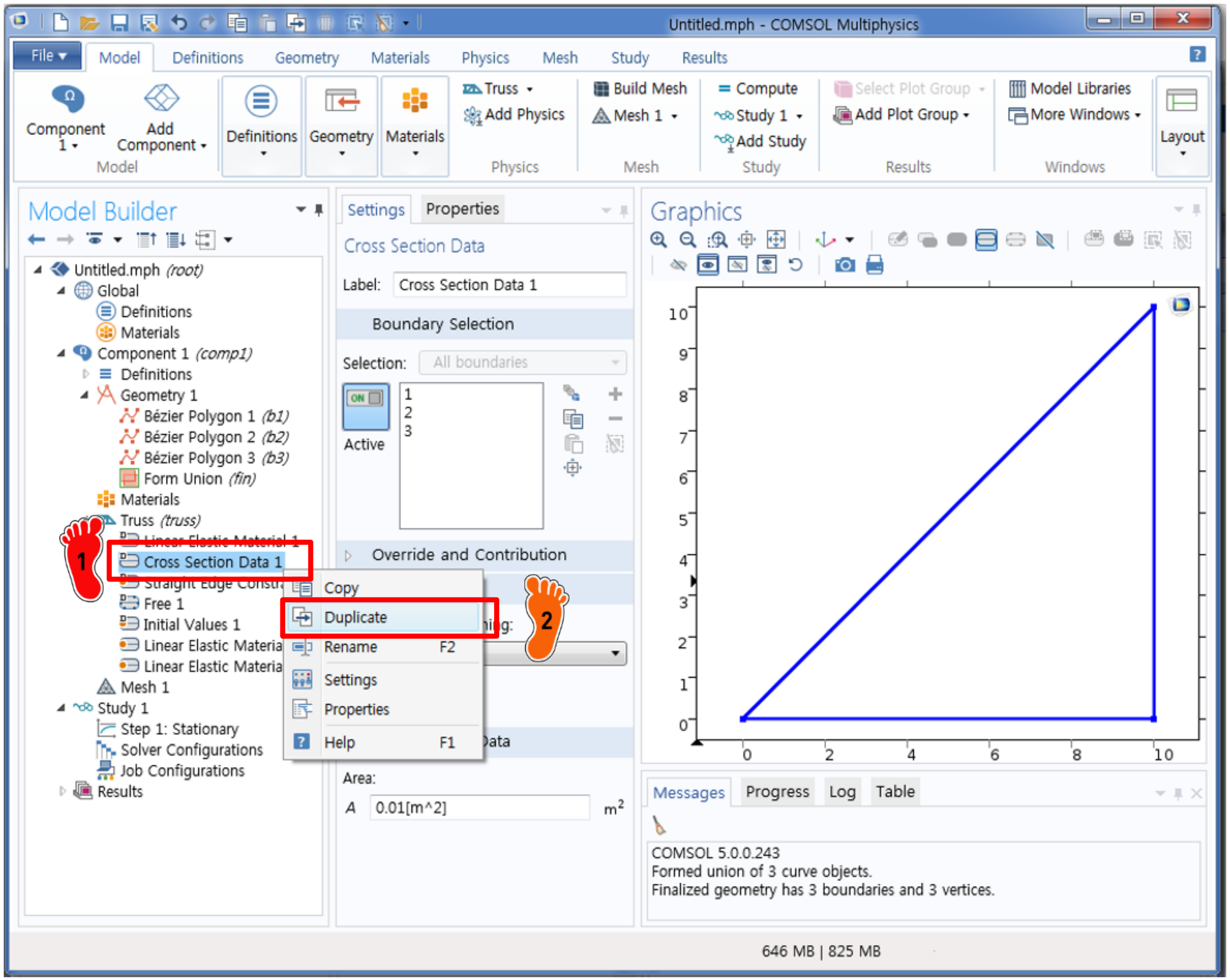
푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

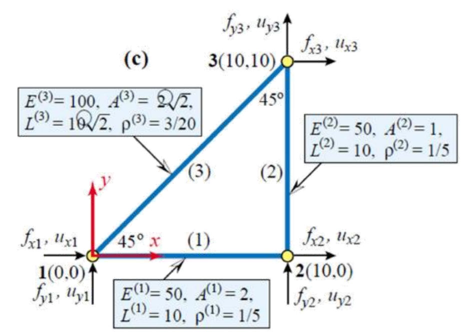
- **Direct Stiffness Method: Truss**
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **<u>Cross section property</u>**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ **Analysis**

SECTION PROPERTY



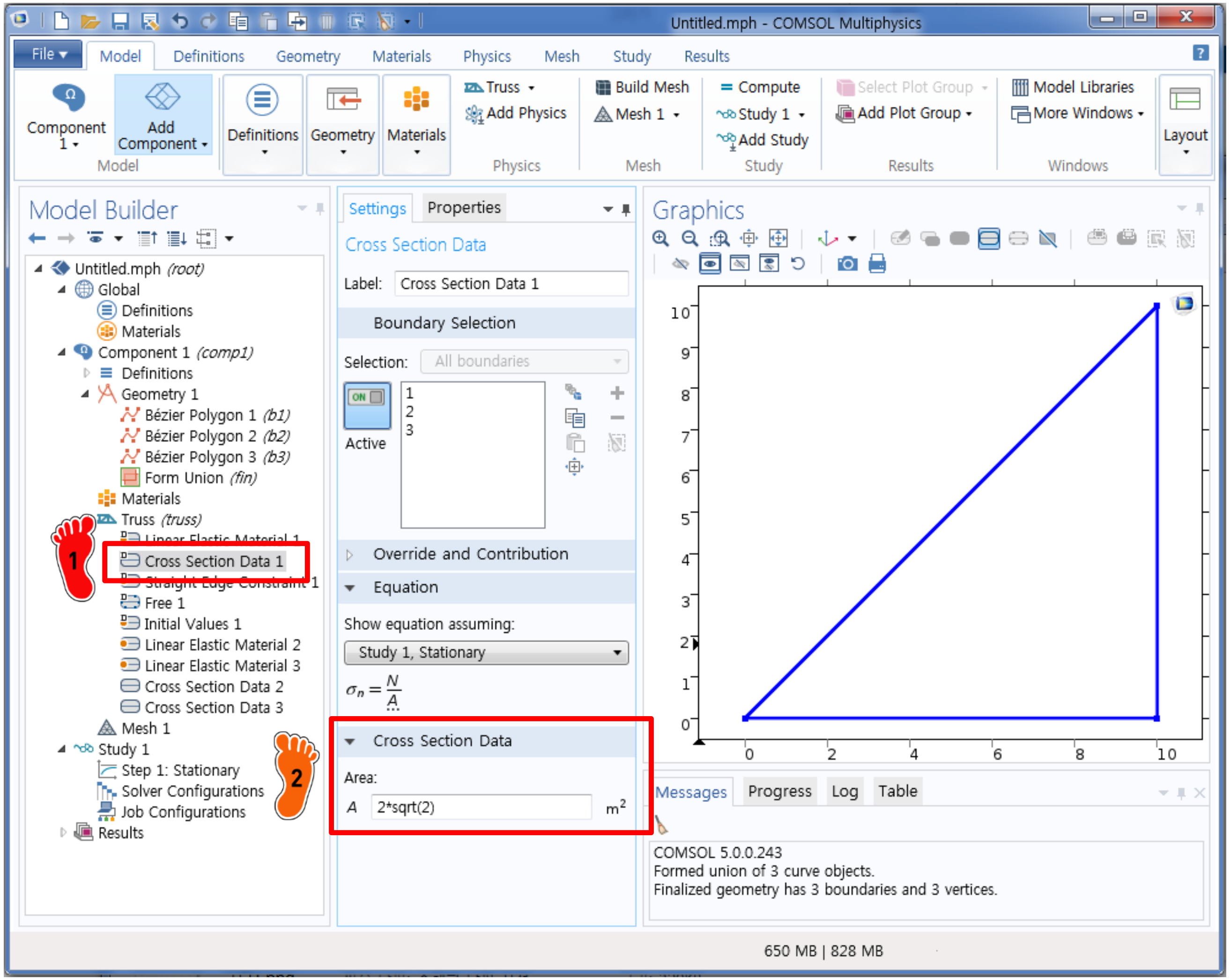
1 Cross Section Data 마우스 우클릭

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



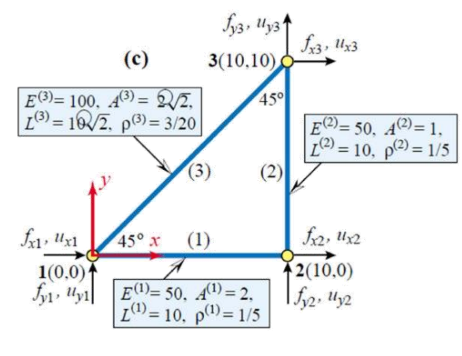
Geometric, material and fabrication properties

SECTION PROPERTY



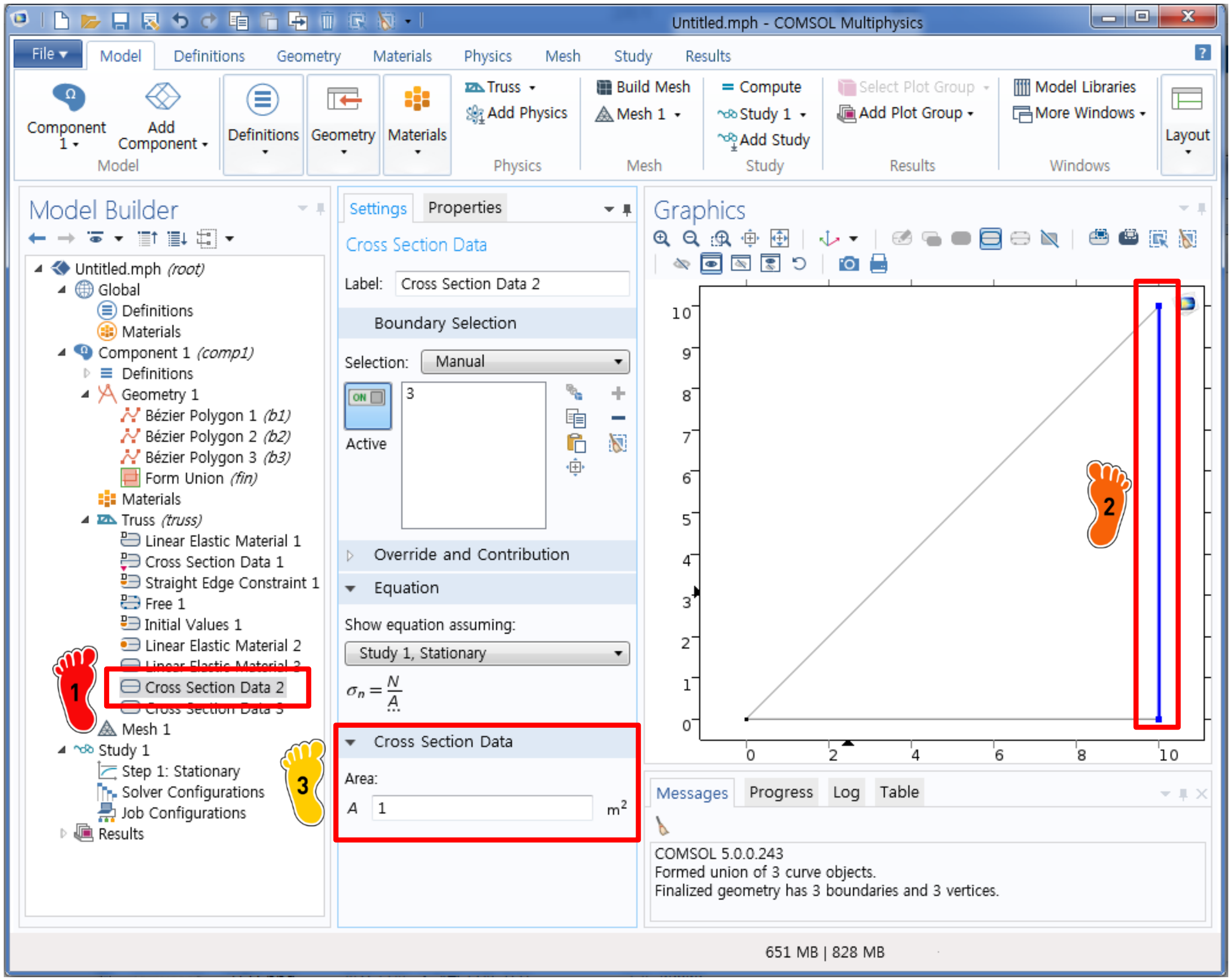
1 첫 번째 Cross Section Data 클릭

2 Area 에 2*sqrt(2) 입력



Geometric, material and fabrication properties

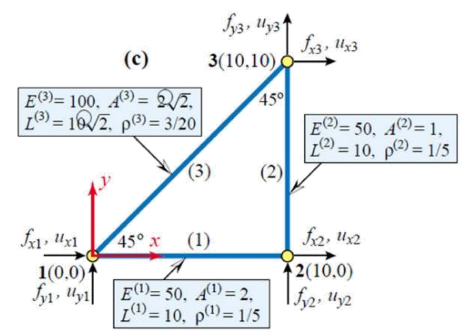
SECTION PROPERTY



1 두 번째 Cross Section Data 클릭

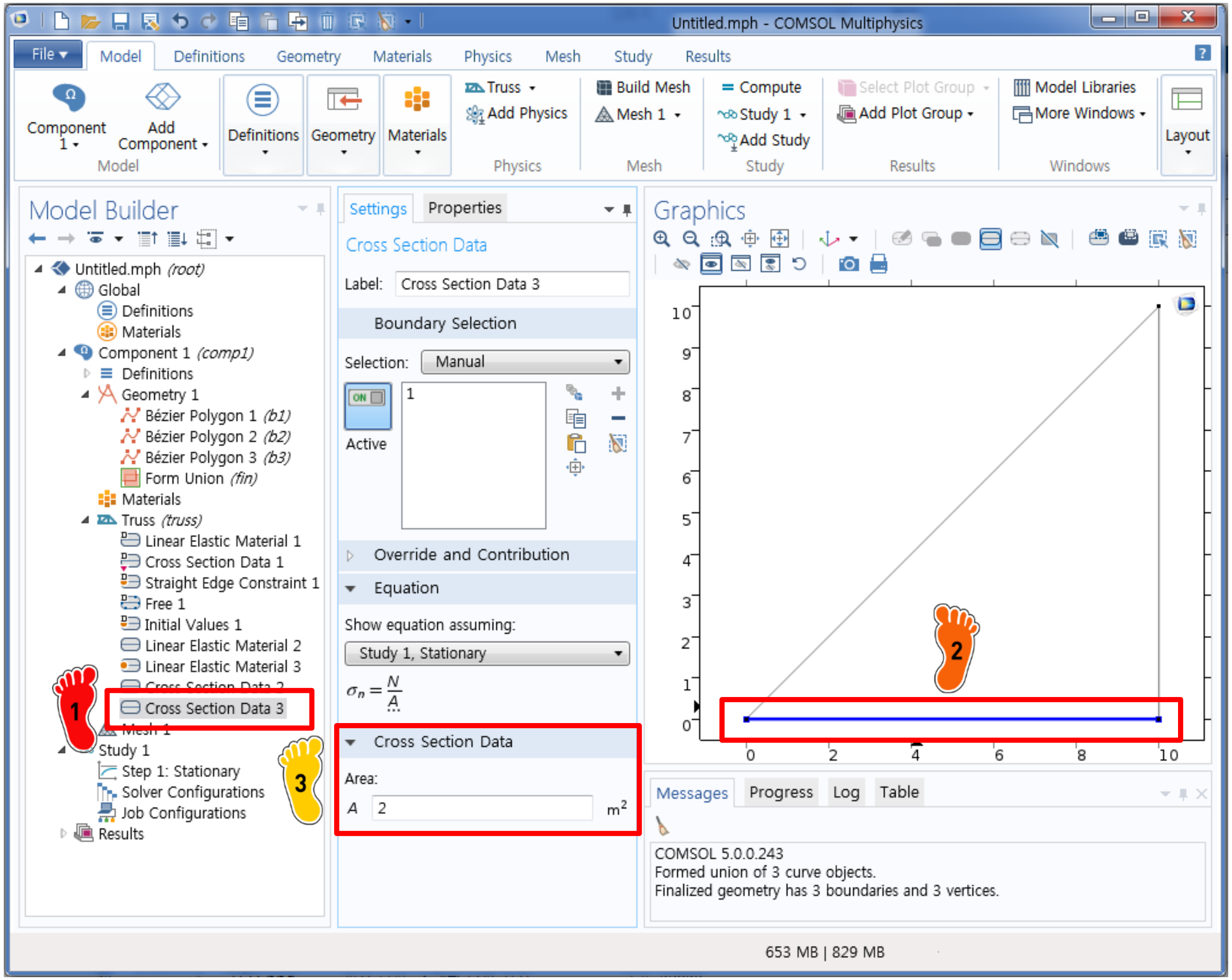
2 제일 오른쪽 부재 선택

3 Area 에 1 입력



Geometric, material and fabrication properties

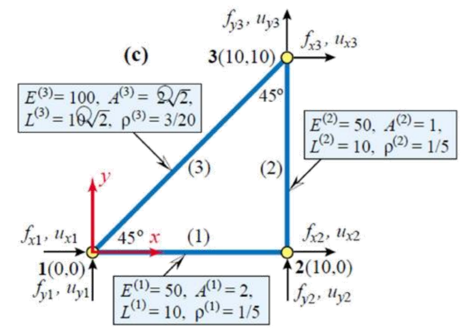
SECTION PROPERTY



1 마지막 Cross Section Data 클릭

2 제일 아래 부재 선택

3 Area 에 2 입력



Geometric, material and fabrication properties

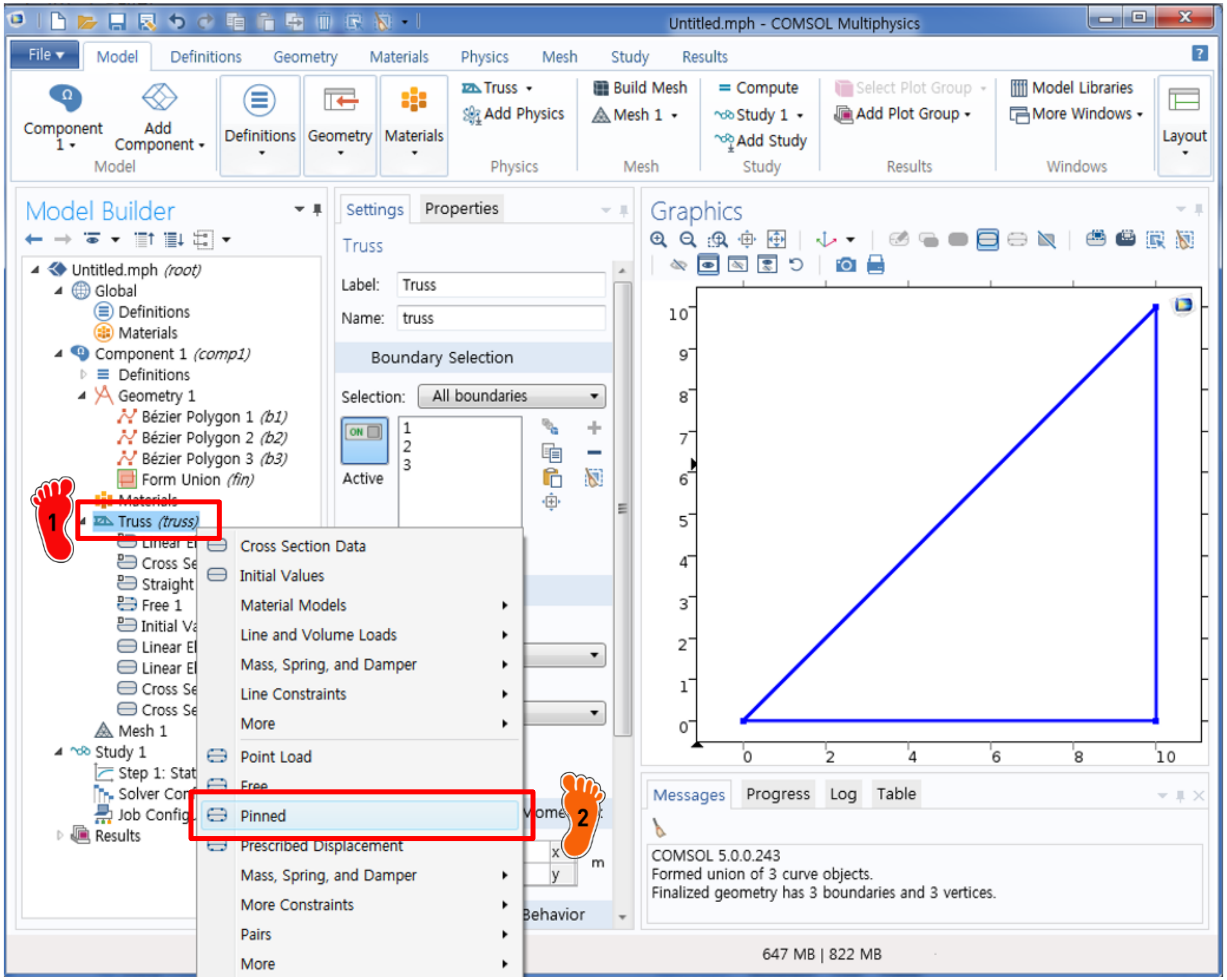
- **Direct Stiffness Method: Truss**

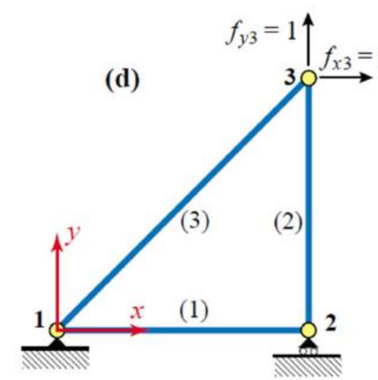
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ **Analysis**

BOUNDARY CONDITION

1 Truss 마우스 우클릭

2 Pinned 클릭



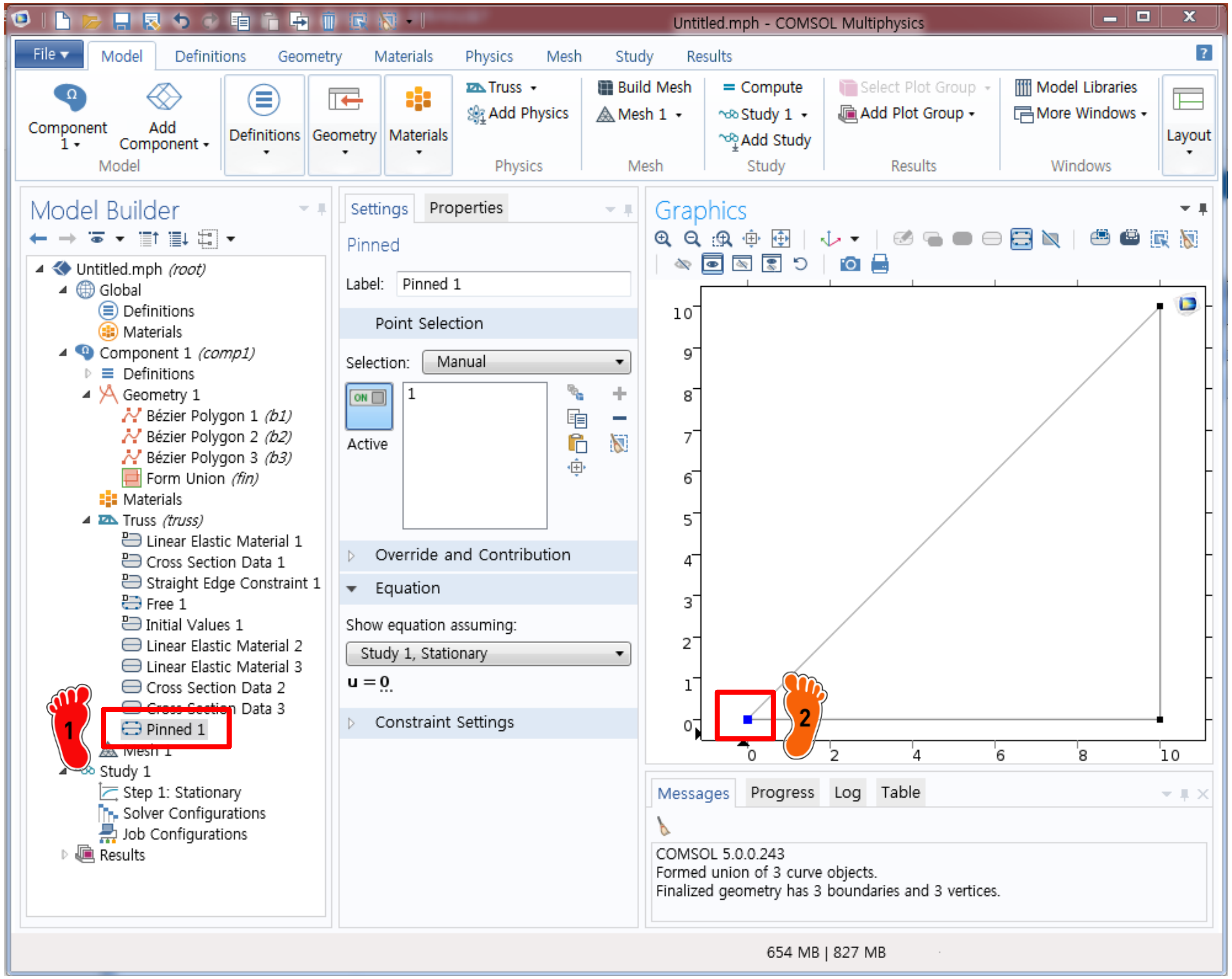


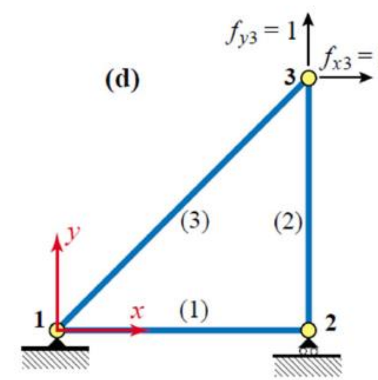
Support conditions and applied loads

BOUNDARY CONDITION

1 Pinned 클릭

2 왼쪽 아래 절점 선택



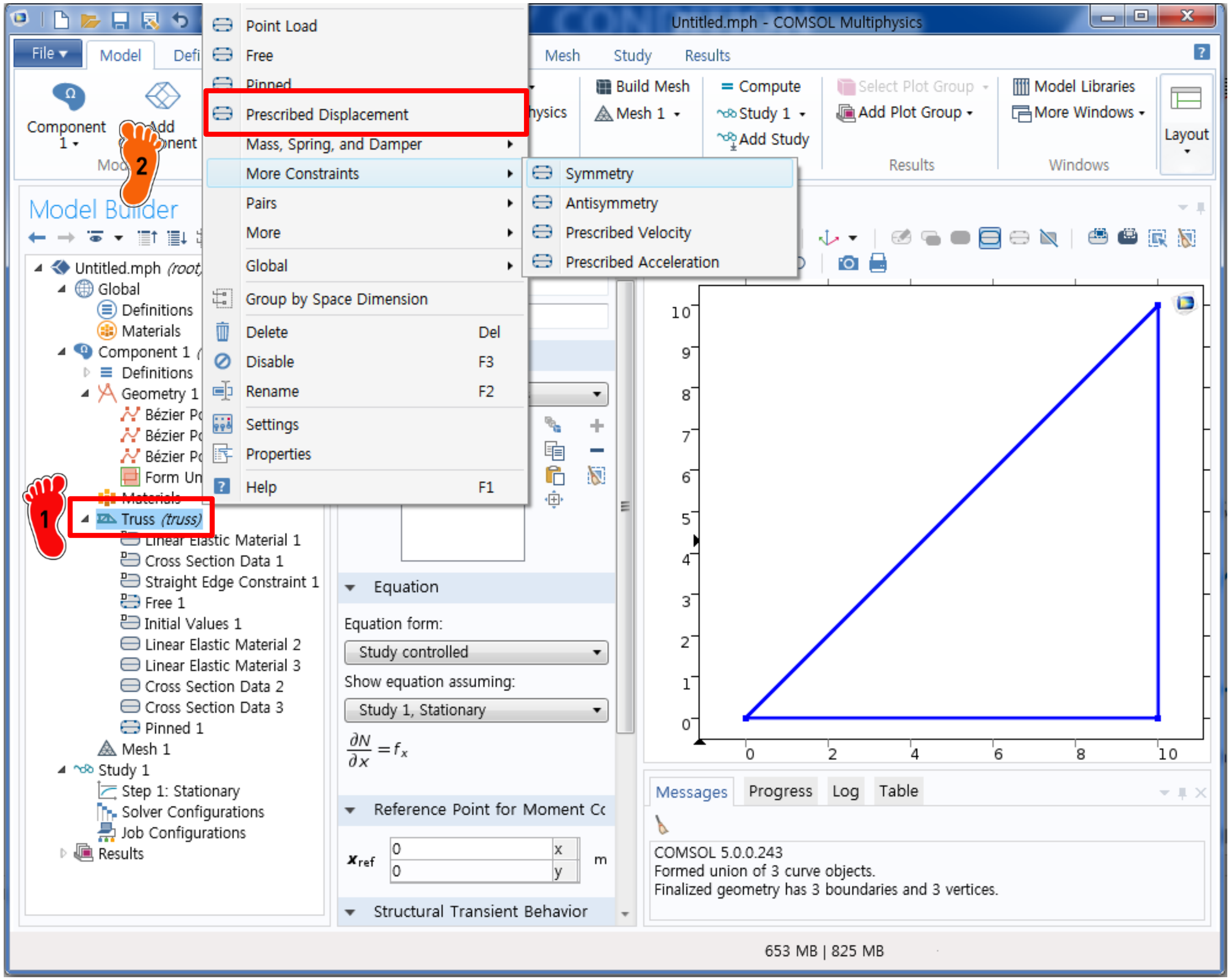


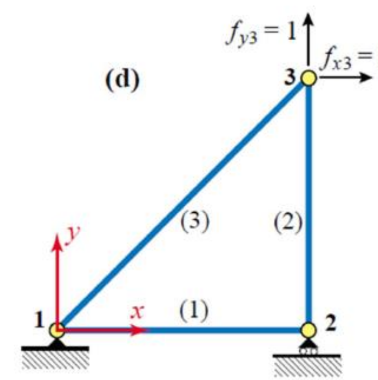
Support conditions and applied loads

BOUNDARY CONDITION

1 Truss 마우스 우클릭

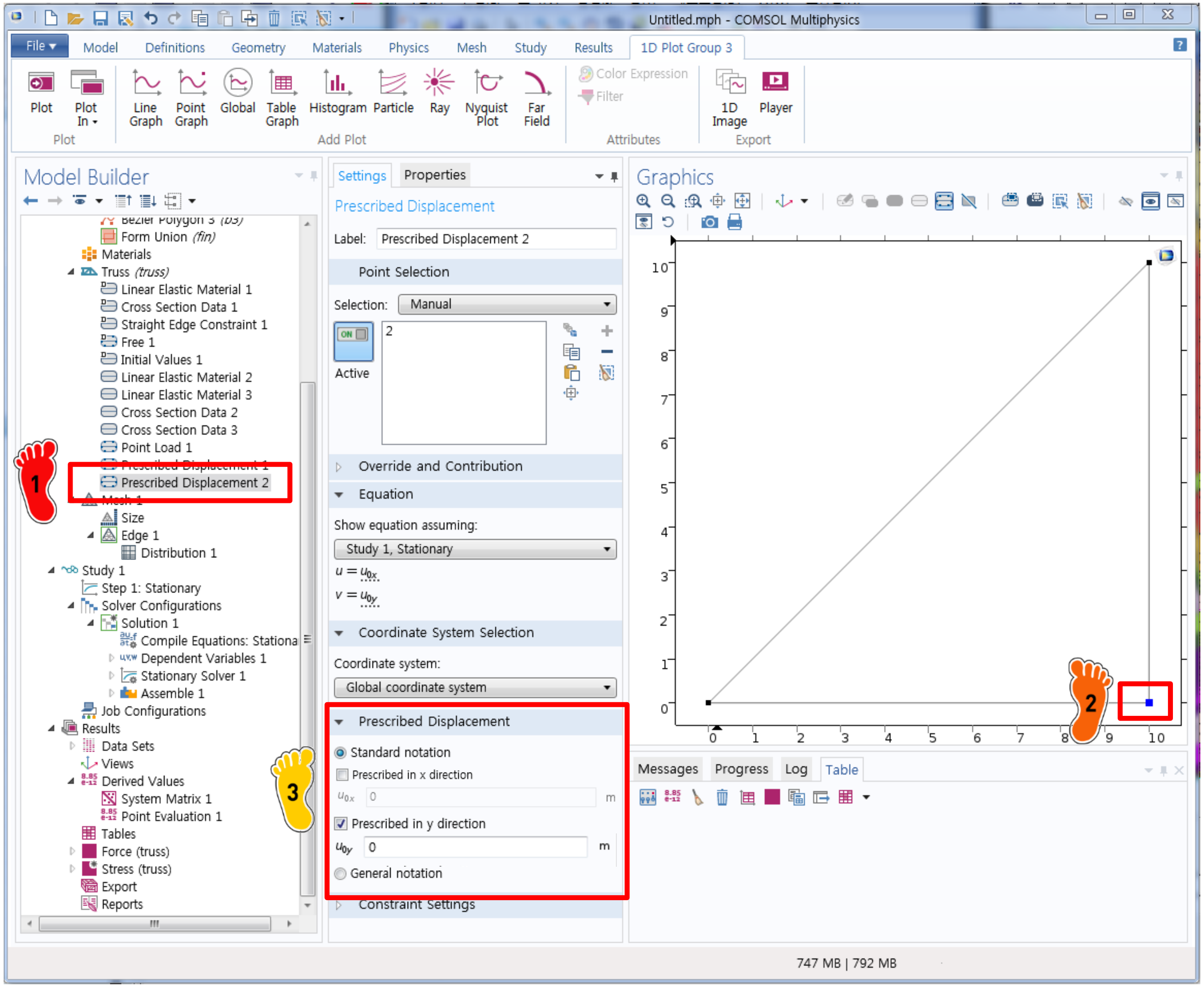
2 Prescribed Displacement





Support conditions and applied loads

BOUNDARY CONDITION



1 Prescribed Displacement 클릭

2 오른쪽 아래 절점 선택

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

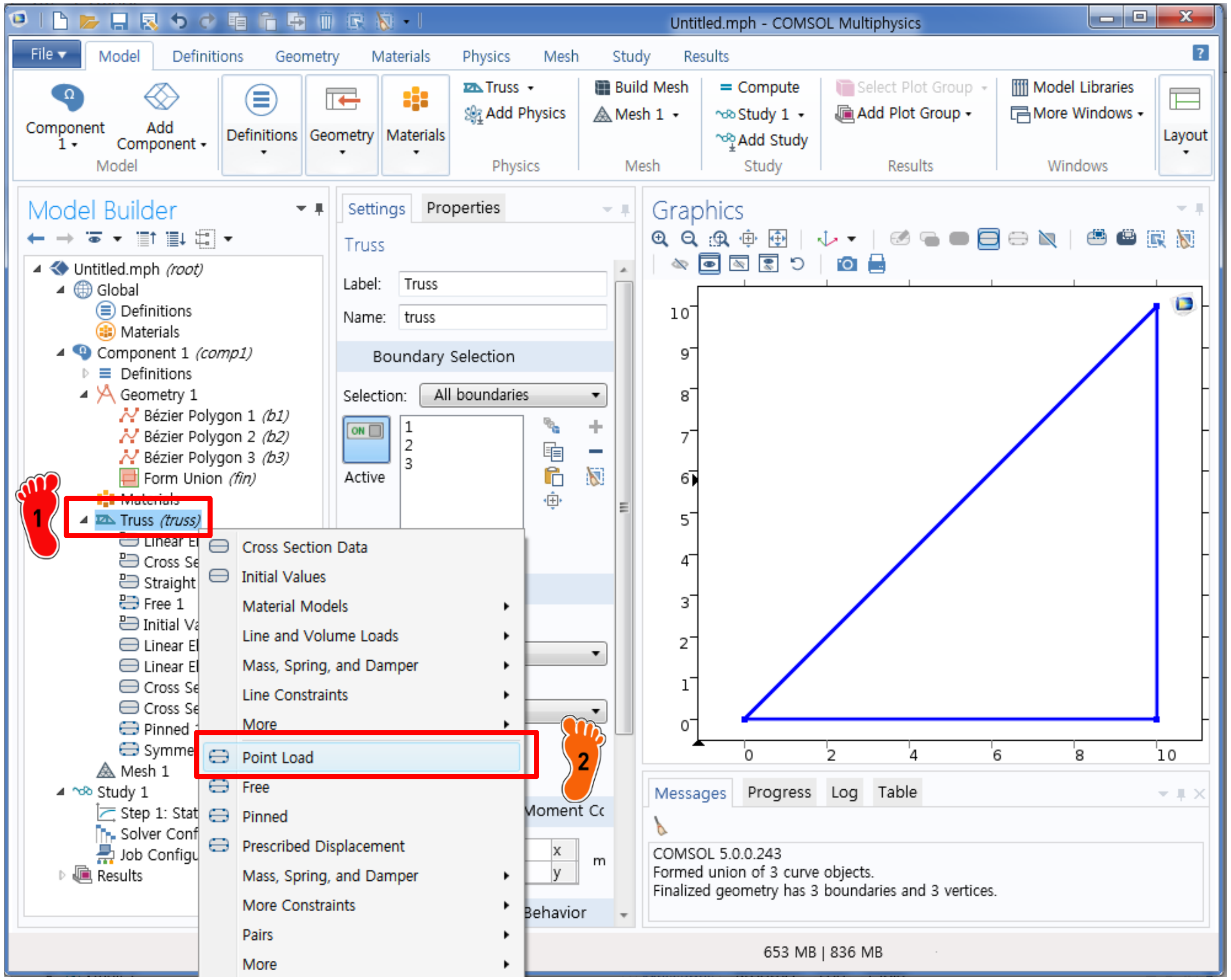
입력

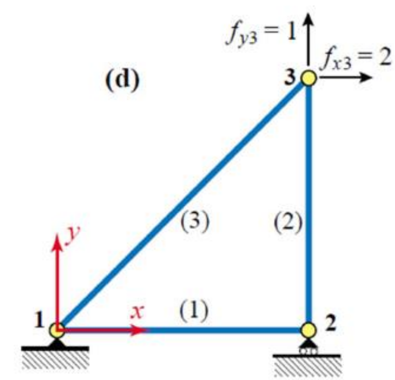
- **Direct Stiffness Method: Truss**
  - ✓ Geometry creation
  - ✓ Material property
  - ✓ Cross section property
  - ✓ Boundary condition
  - ✓ Nodal force
  - ✓ Analysis

POINT LOAD

1 Truss 마우스 우클릭

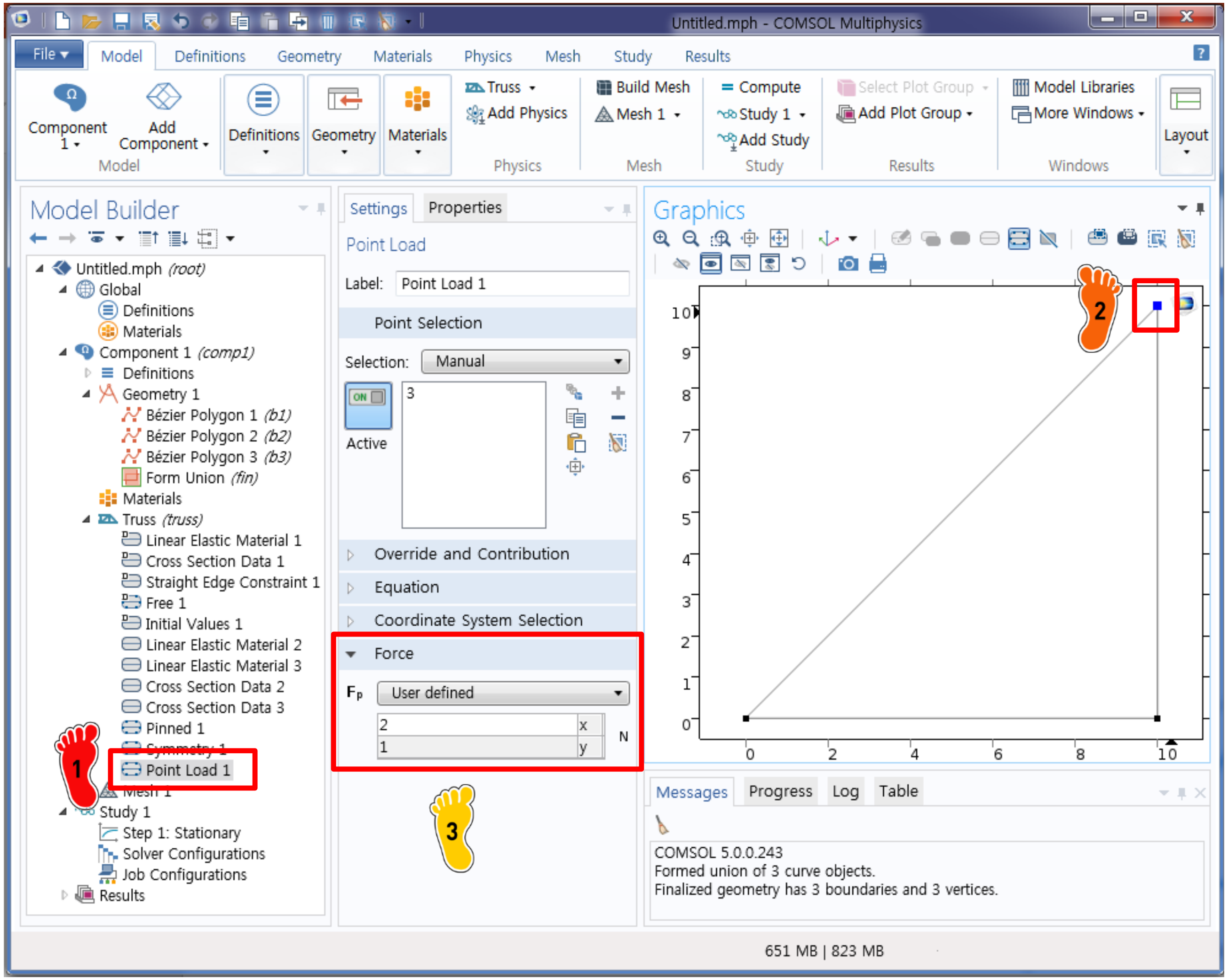
2 Point Load 클릭





Support conditions and applied loads

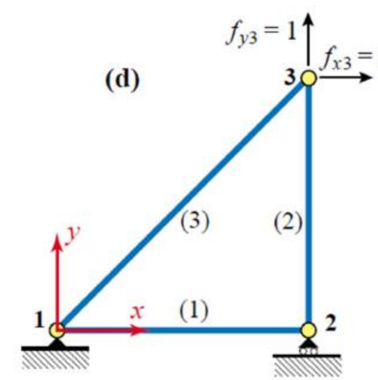
POINT LOAD



1 Point Load 1 클릭

2 오른쪽 위 절점 선택

3 x 방향 2, y 방향 1 입력

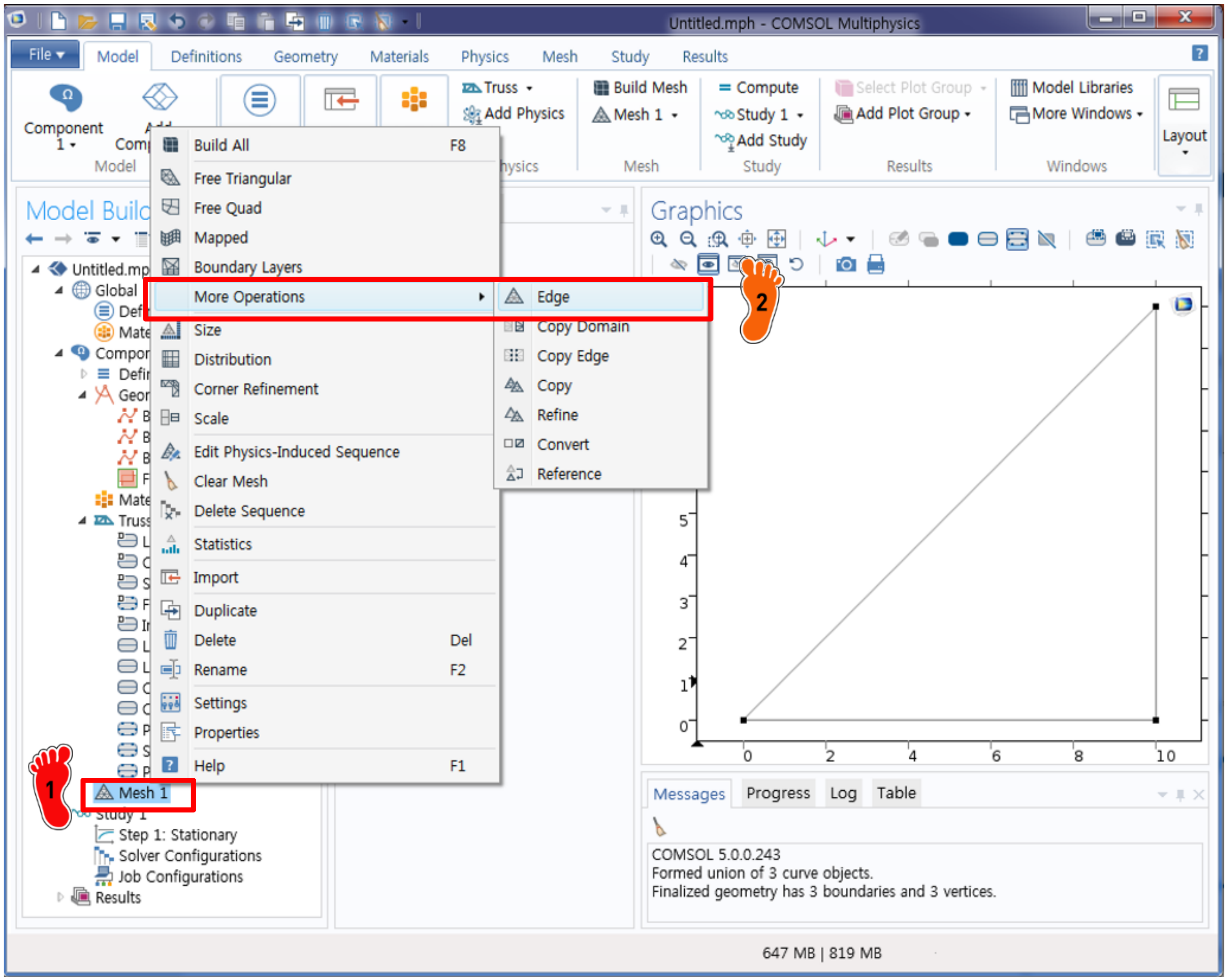


Support conditions and applied loads

- # Direct Stiffness Method: Truss

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

MESH CREATION

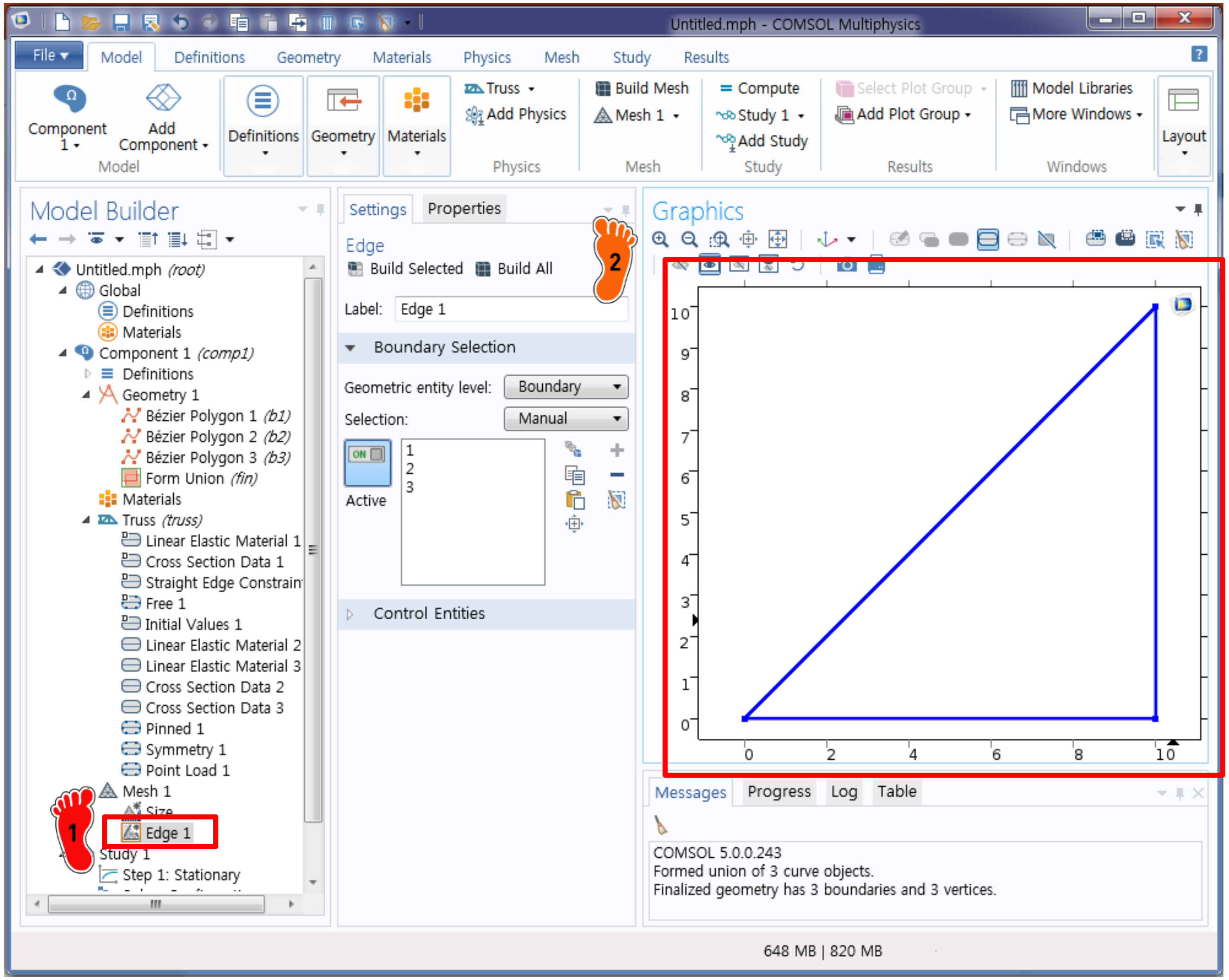


1 Mesh 1 마우스 우클릭

2 More Operations에서

Edge 클릭

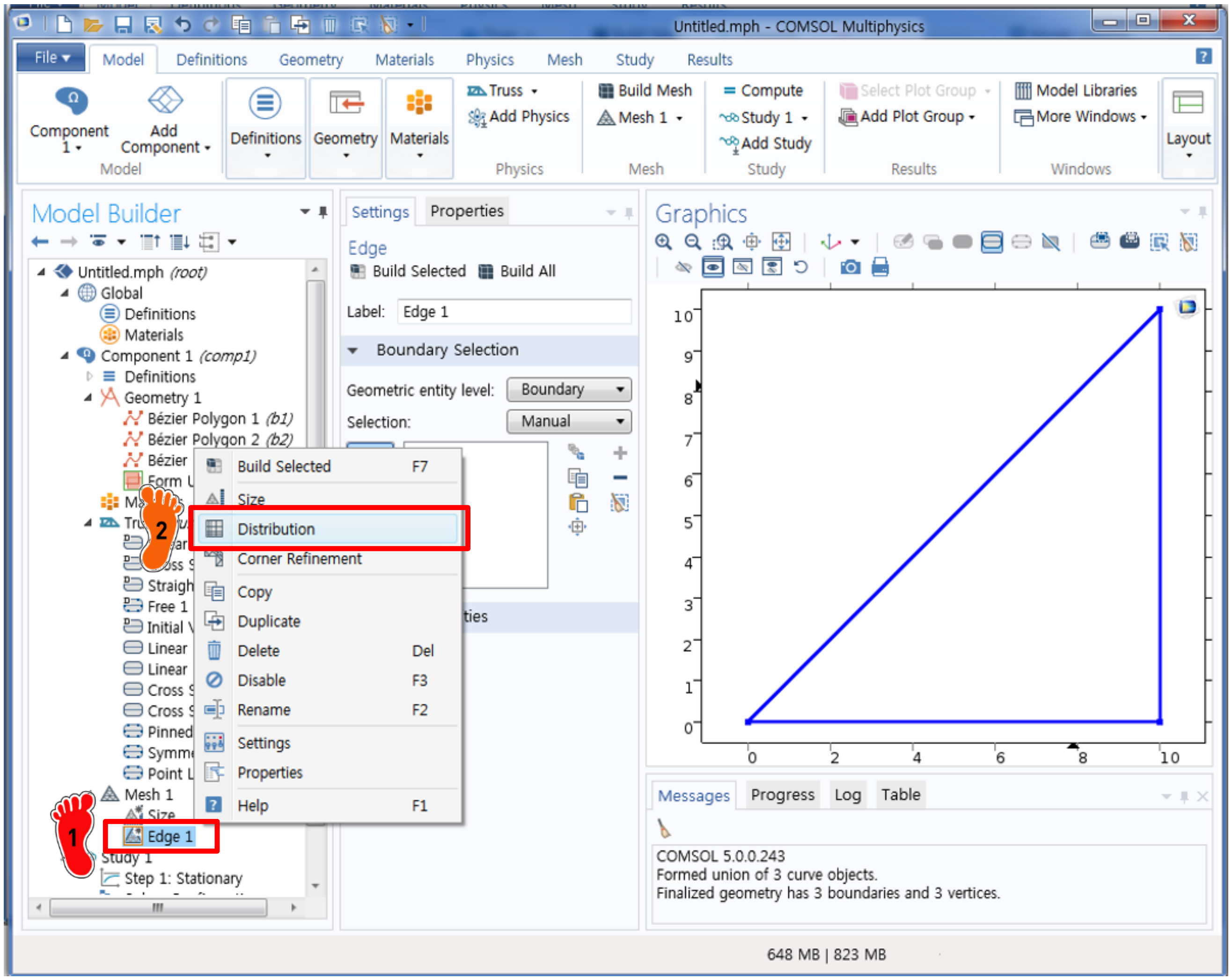
MESH CREATION



1 Edge 1 클릭

2 부재 세 개 선택

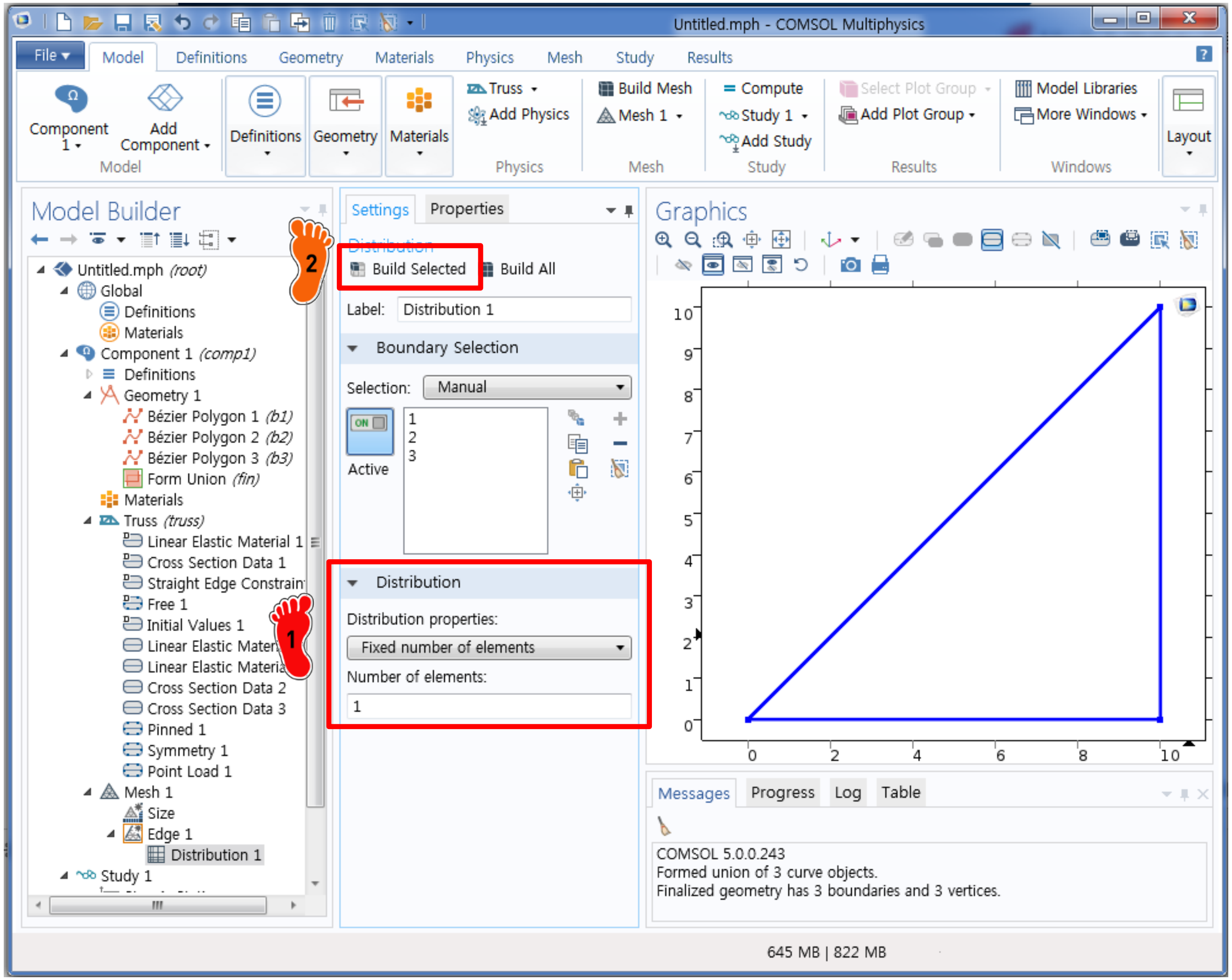
MESH CREATION



1 Edge 1 우클릭

2 Distribution 클릭

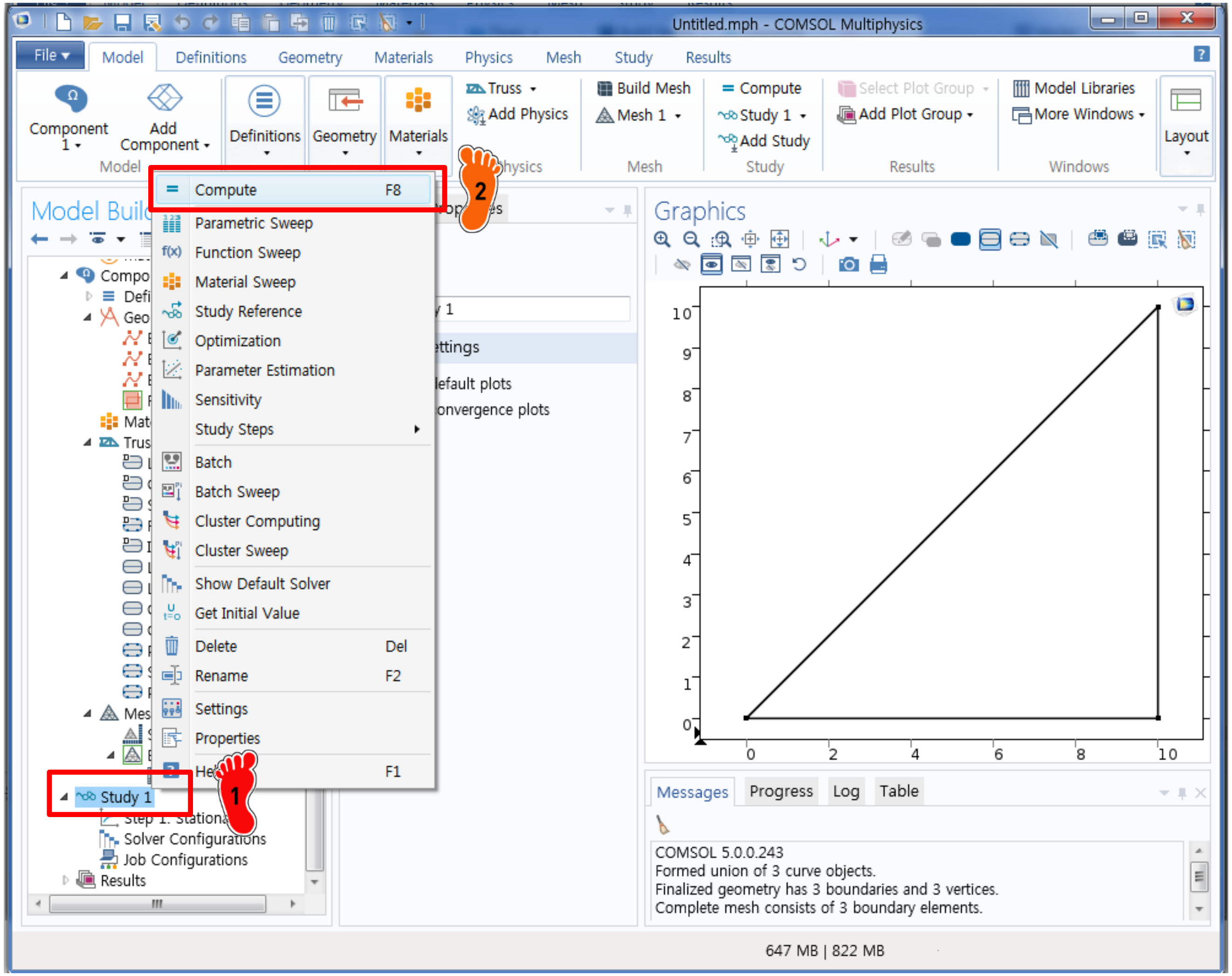
MESH CREATION



1 Number of elements 1 입력

2 Build Selected 클릭

COMPUTE

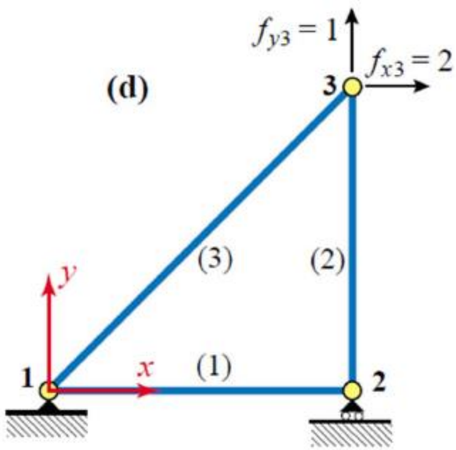


1 Study 1 마우스 우클릭

2 Compute 클릭

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

ASSEMBLE OPTION



Support conditions and applied loads

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

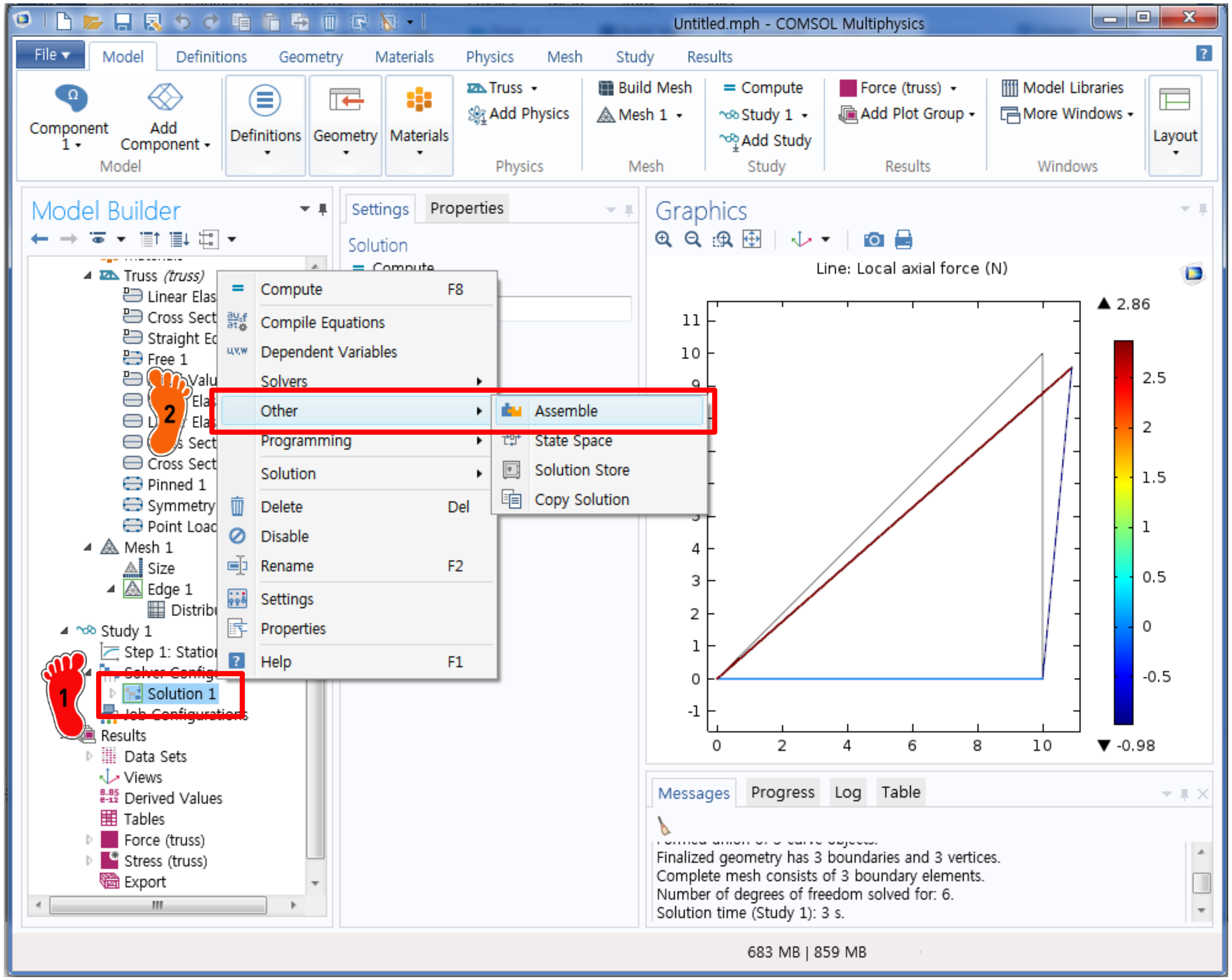
$$\Downarrow$$

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

- **Stiffness matrix confirm**

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

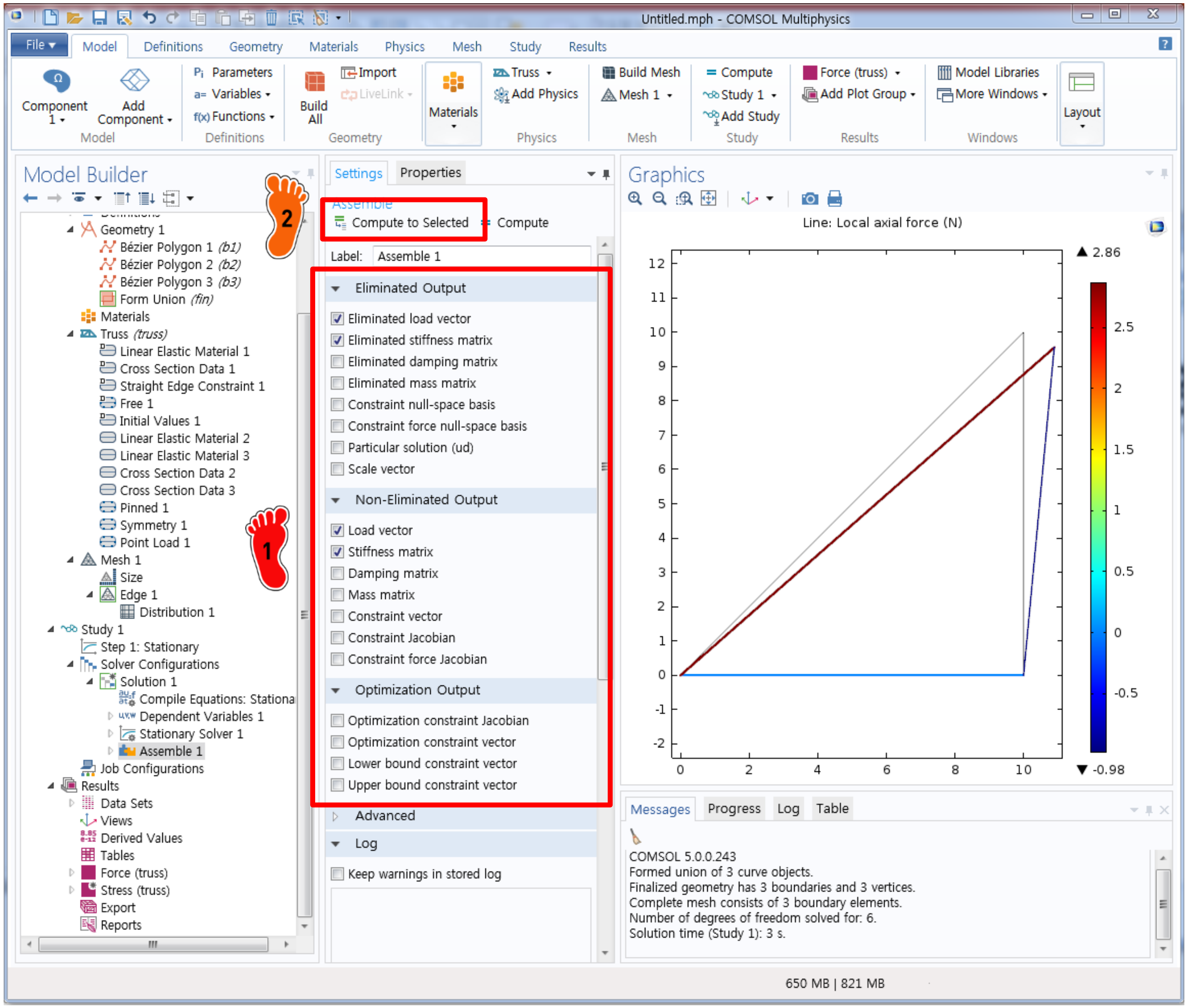
ASSEMBLE OPTION



1 Solver 1 마우스 우클릭

2 Other에서 Assemble 클릭

ASSEMBLE OPTION



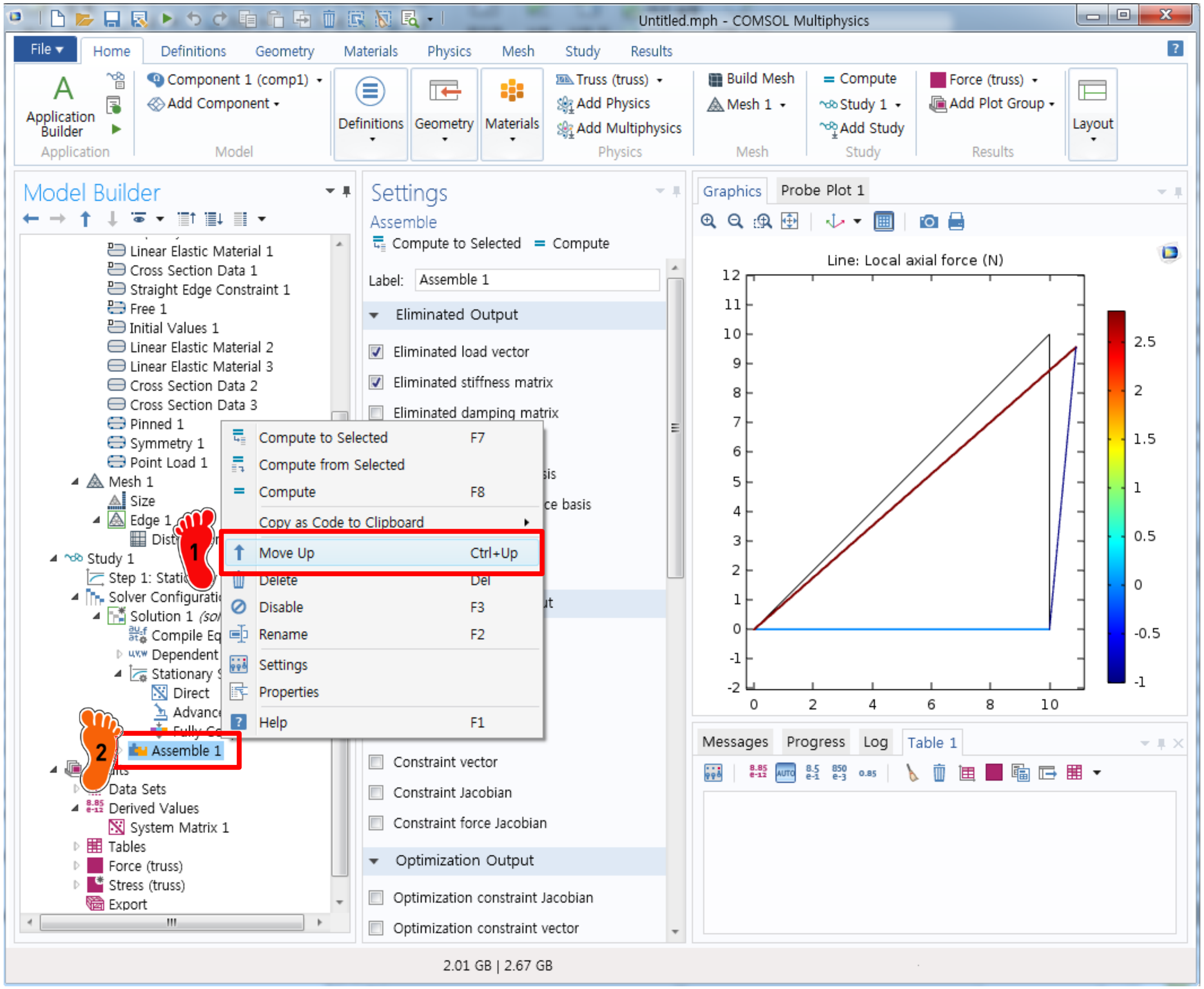
1 Eliminated load vector
Eliminated stiffness matrix

Load vector
Stiffness matrix

체크박스에 체크

2 Compute to Selected 클릭

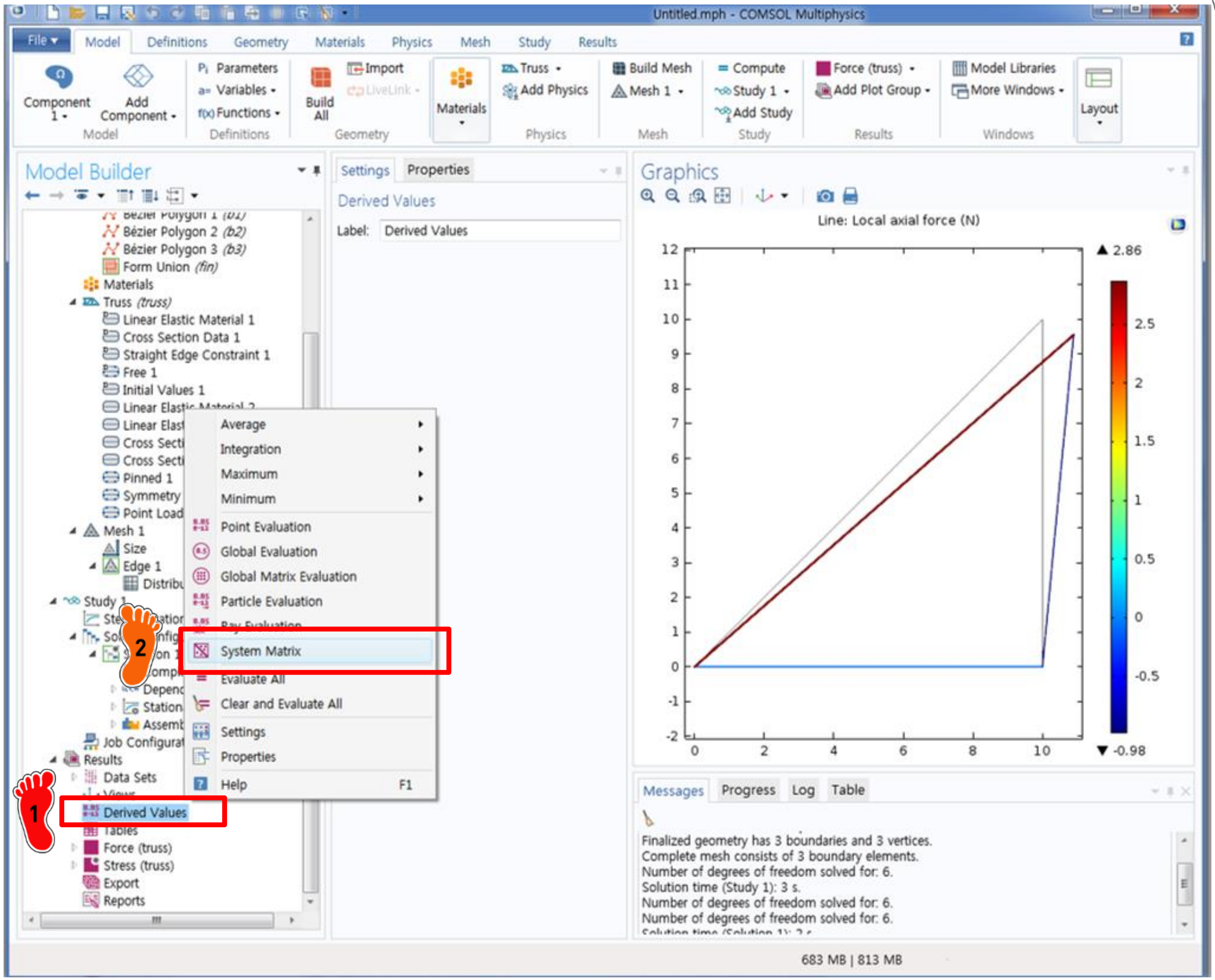
ASSEMBLE OPTION



1 Assemble 1 마우스 우클릭

2 Move up을 클릭하여 Assemble 노드가 Solver 노드 위에 위치하도록 이동

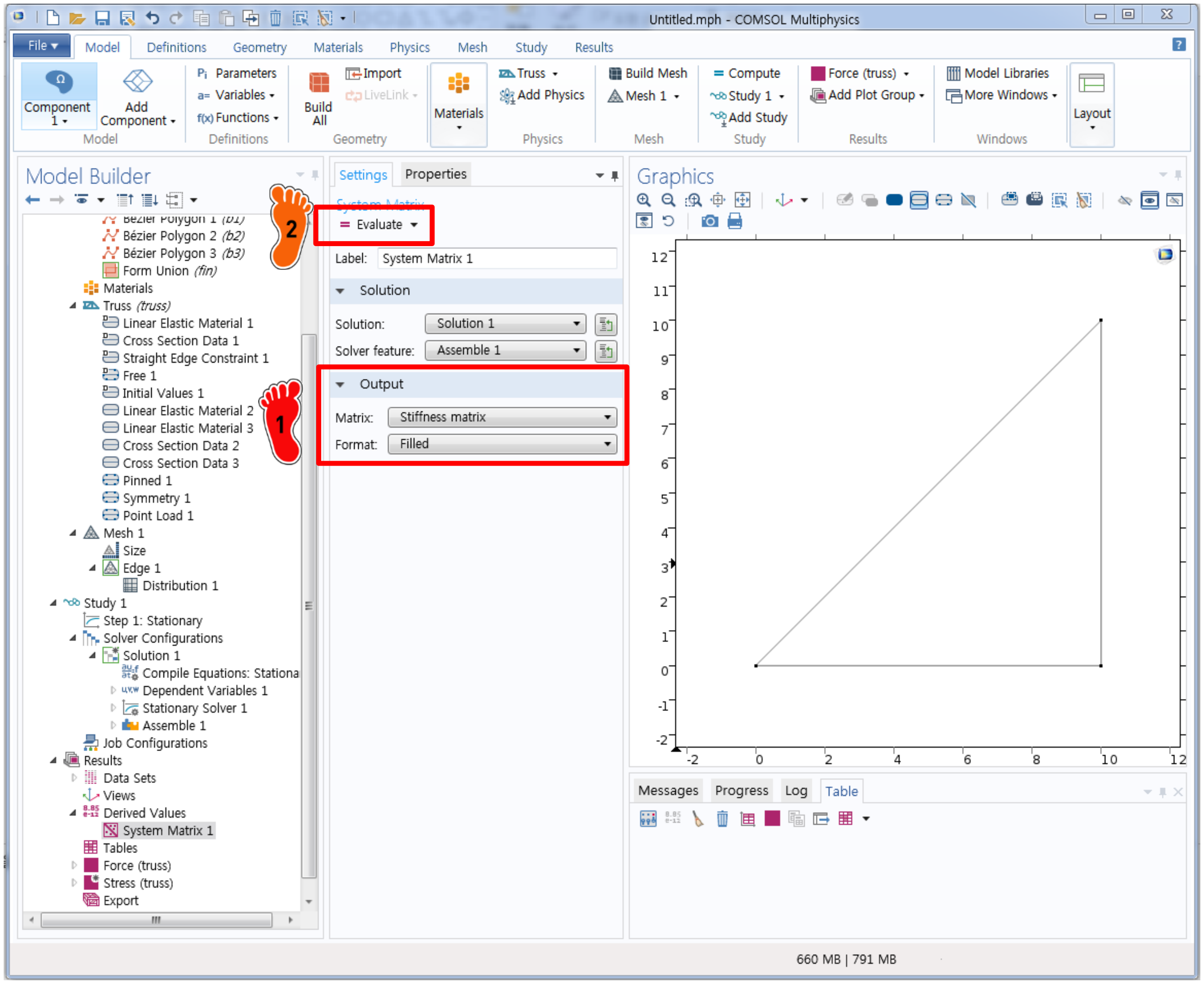
SYSTEM MATRIX



1 Derived Values 마우스 우클릭

2 System Matrix 클릭

SYSTEM MATRIX

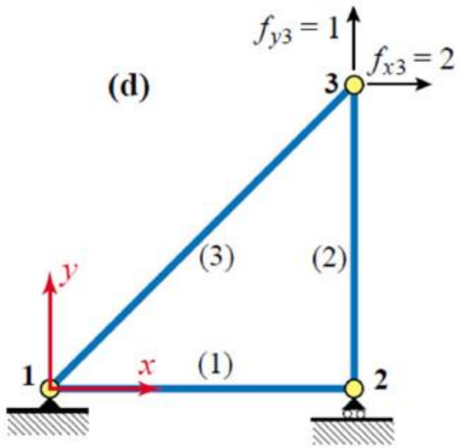


1 Matrix : Stiffness matrix
Format : Filled
선택

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

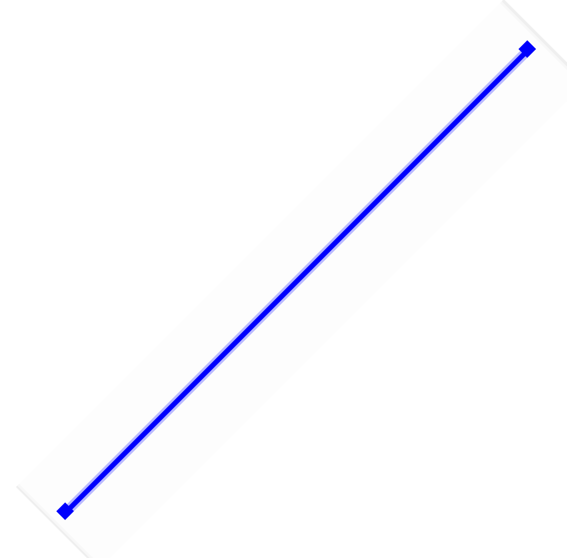
20.000	10.0000	-10.0000	0.0000	-10.0000	-10.0000
10.0000	10.0000	0.0000	0.0000	-10.0000	-10.0000
-10.0000	0.0000	10.0000	0.0000	0.0000	0.0000
0.0000	0.0000	0.0000	5.0000	0.0000	-5.0000
-10.0000	-10.0000	0.0000	0.0000	10.0000	10.0000
-10.0000	-10.0000	0.0000	-5.0000	10.0000	15.000

LOCAL STIFFNESS MATRIX

Stiffness matrix by Hand

$$\begin{bmatrix} f_{x1}^{(1)} \\ f_{y1}^{(1)} \\ f_{x2}^{(1)} \\ f_{y2}^{(1)} \end{bmatrix} = 10 \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_{x1}^{(1)} \\ u_{y1}^{(1)} \\ u_{x2}^{(1)} \\ u_{y2}^{(1)} \end{bmatrix} \quad \begin{bmatrix} f_{x2}^{(2)} \\ f_{y2}^{(2)} \\ f_{x3}^{(2)} \\ f_{y3}^{(2)} \end{bmatrix} = 5 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_{x2}^{(2)} \\ u_{y2}^{(2)} \\ u_{x3}^{(3)} \\ u_{y3}^{(3)} \end{bmatrix} \quad \begin{bmatrix} f_{x1}^{(3)} \\ f_{y1}^{(3)} \\ f_{x3}^{(3)} \\ f_{y3}^{(3)} \end{bmatrix} = 20 \begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix} \begin{bmatrix} u_{x1}^{(3)} \\ u_{y1}^{(3)} \\ u_{x3}^{(3)} \\ u_{y3}^{(3)} \end{bmatrix}$$

Geometry



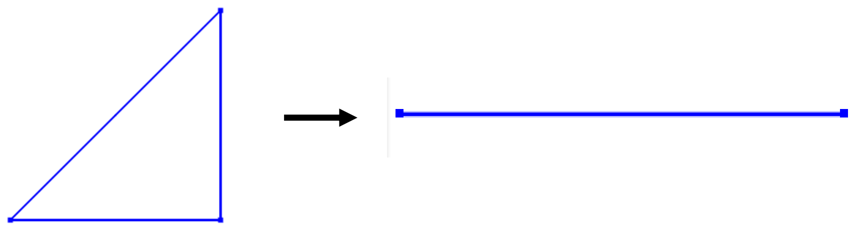
Stiffness matrix by COMSOL

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

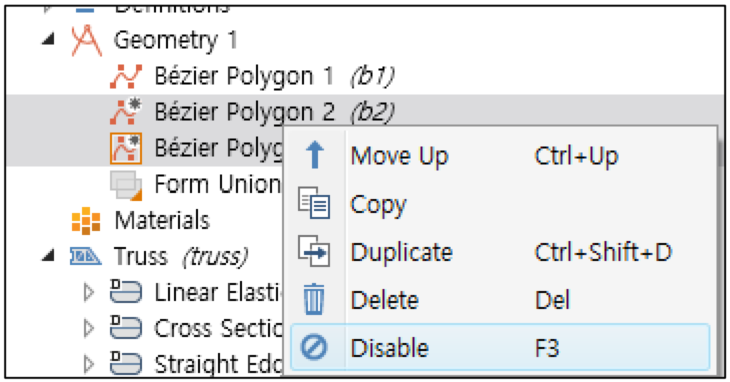
0.0000	0.0000	0.0000	0.0000
0.0000	5.0000	0.0000	-5.0000
0.0000	0.0000	0.0000	0.0000
0.0000	-5.0000	0.0000	5.0000

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

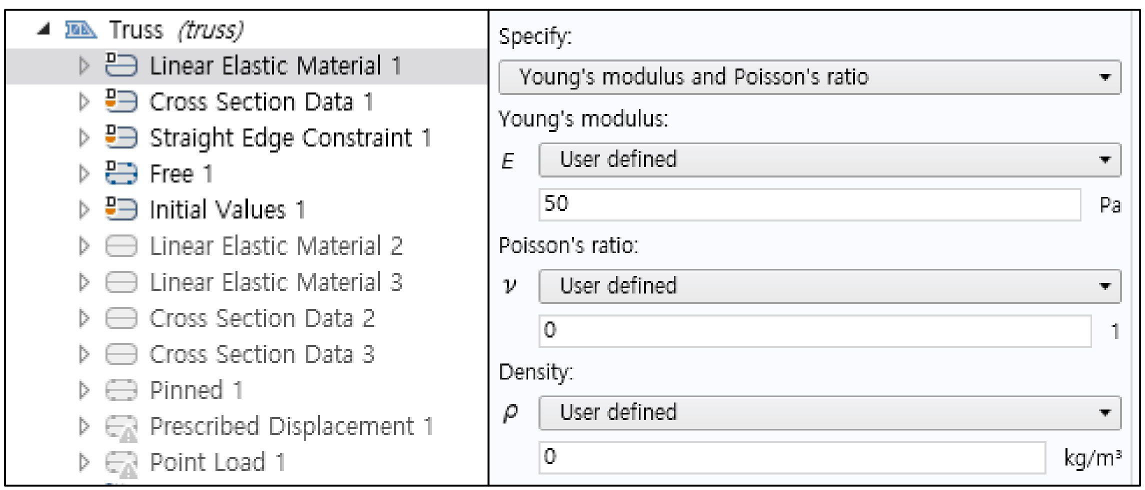
LOCAL STIFFNESS MATRIX



1 각 Truss의
Local Stiffness Matrix를
얻기 위해선 하나의
Truss만 남기고
Geometry에서 비활성화



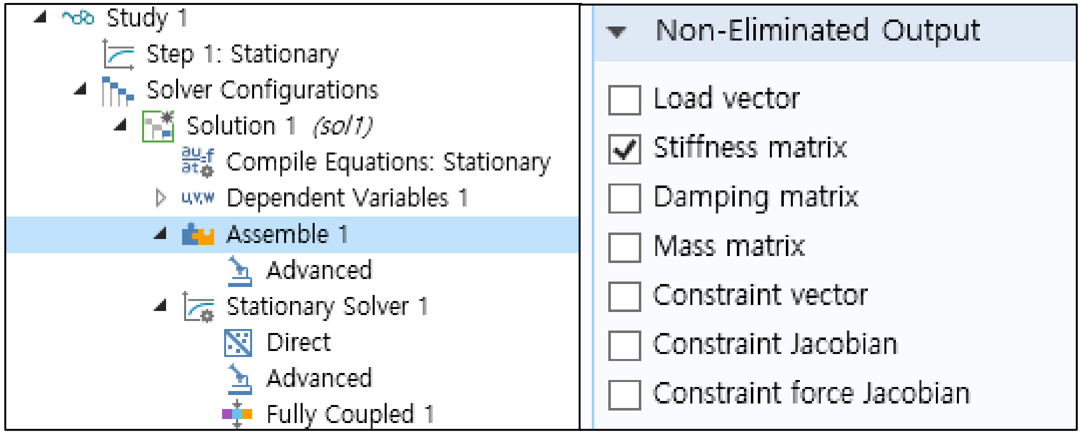
2 Truss Physics에서
다른 Truss들의 재료 물성
및 경계조건 설정 비활성화



Local Stiffness Matrix를
계산을 위한 Truss에 대한
재료물성과 단면정보 입력

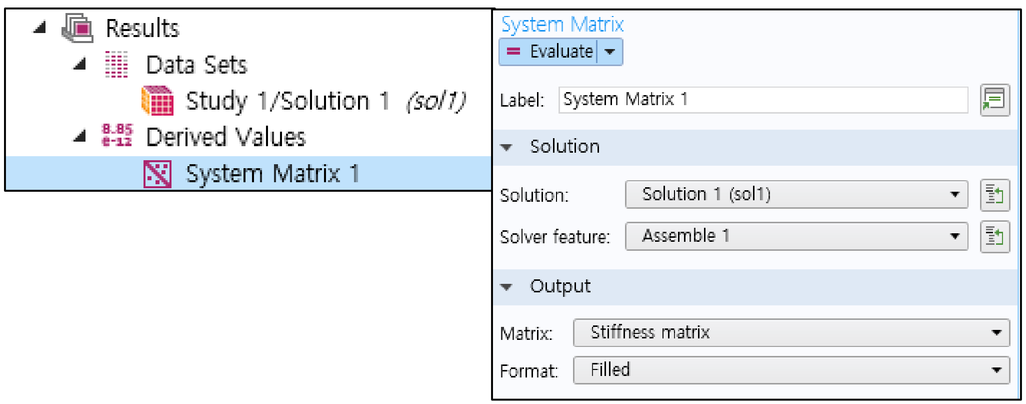
LOCAL STIFFNESS MATRIX

1



1 Study
하부 항목 중
Assemble ->
Stiffness matrix 체크

2



2 Results->Derived values
-> System matrix에서
Evaluate 실행

Table에서
Local Stiffness Matrix 확인

다른 Truss에 대해서도
같은 방법으로 수행

10.000	10.000	-10.000	-10.000
10.000	10.000	-10.000	-10.000
-10.000	-10.000	10.000	10.000
-10.000	-10.000	10.000	10.000

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

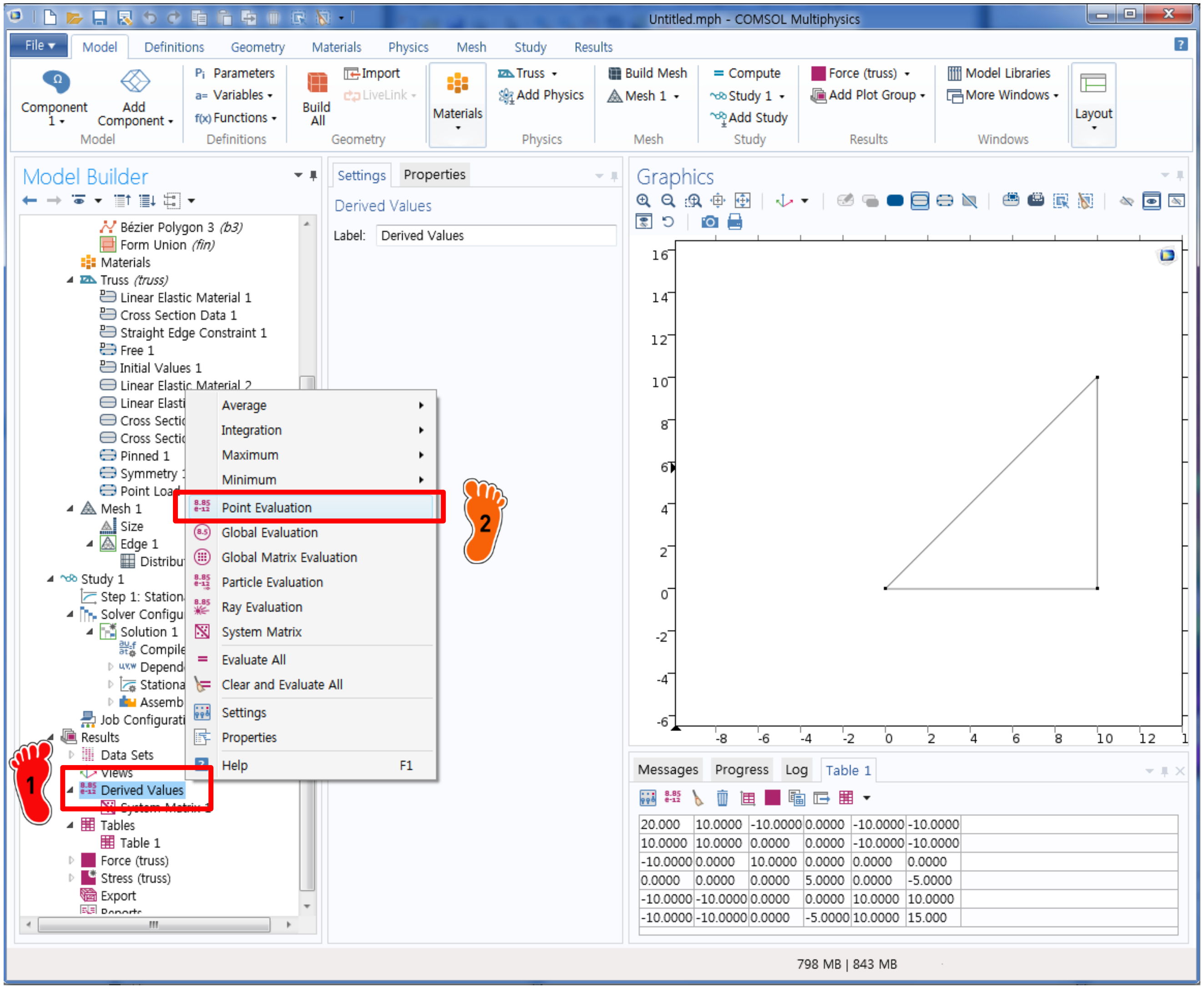
Eliminated Stiffness matrix

10.000	0.0000	0.0000
0.0000	10.000	10.000
0.0000	10.000	15.000

Eliminated Load vector

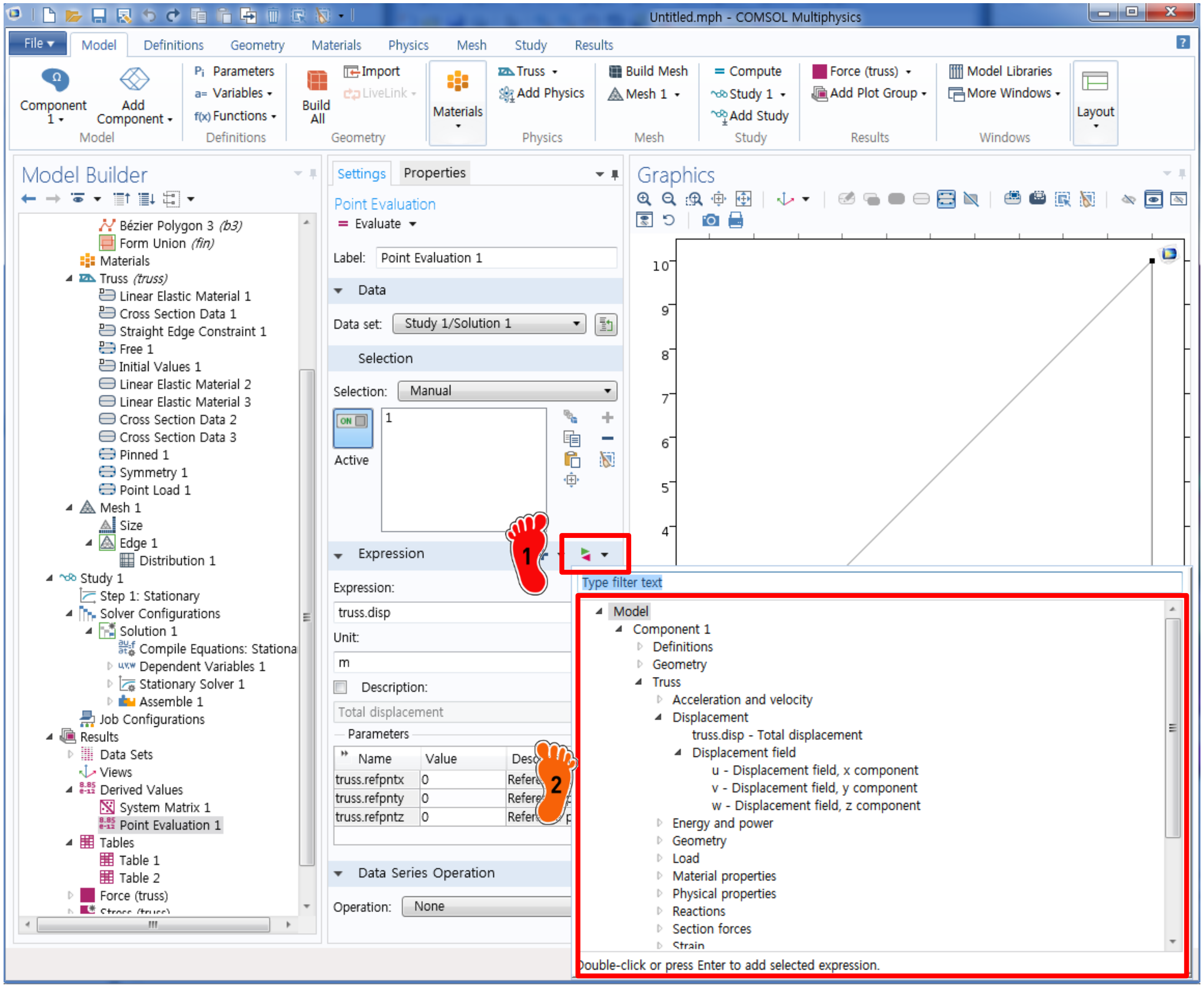
0
2
1

NODAL DISPLACEMENT



1. Derived Values 마우스 우클릭
2. Point Evaluation 클릭

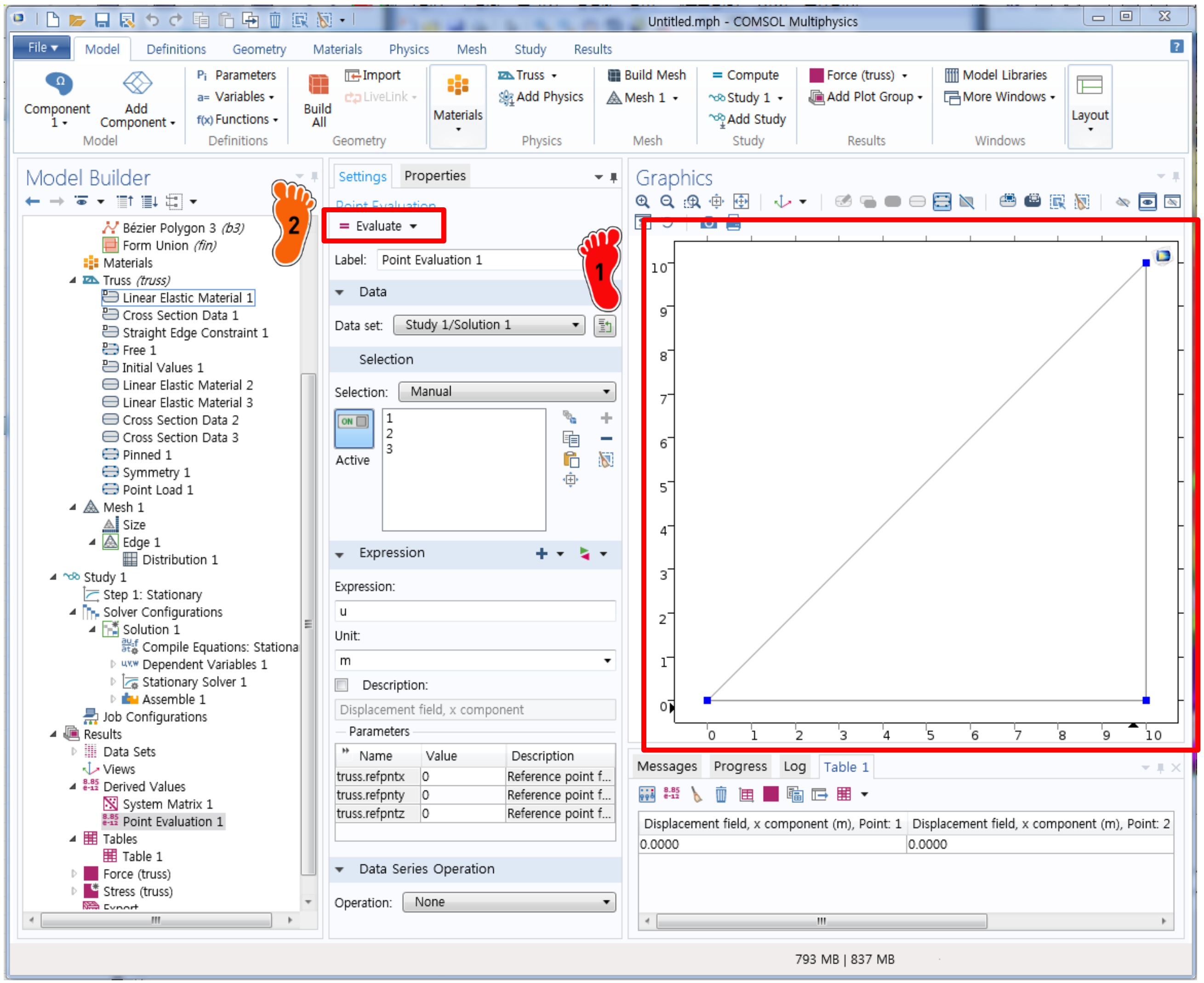
NODAL DISPLACEMENT



1 Replace expression

2 원하는 출력 값 선택

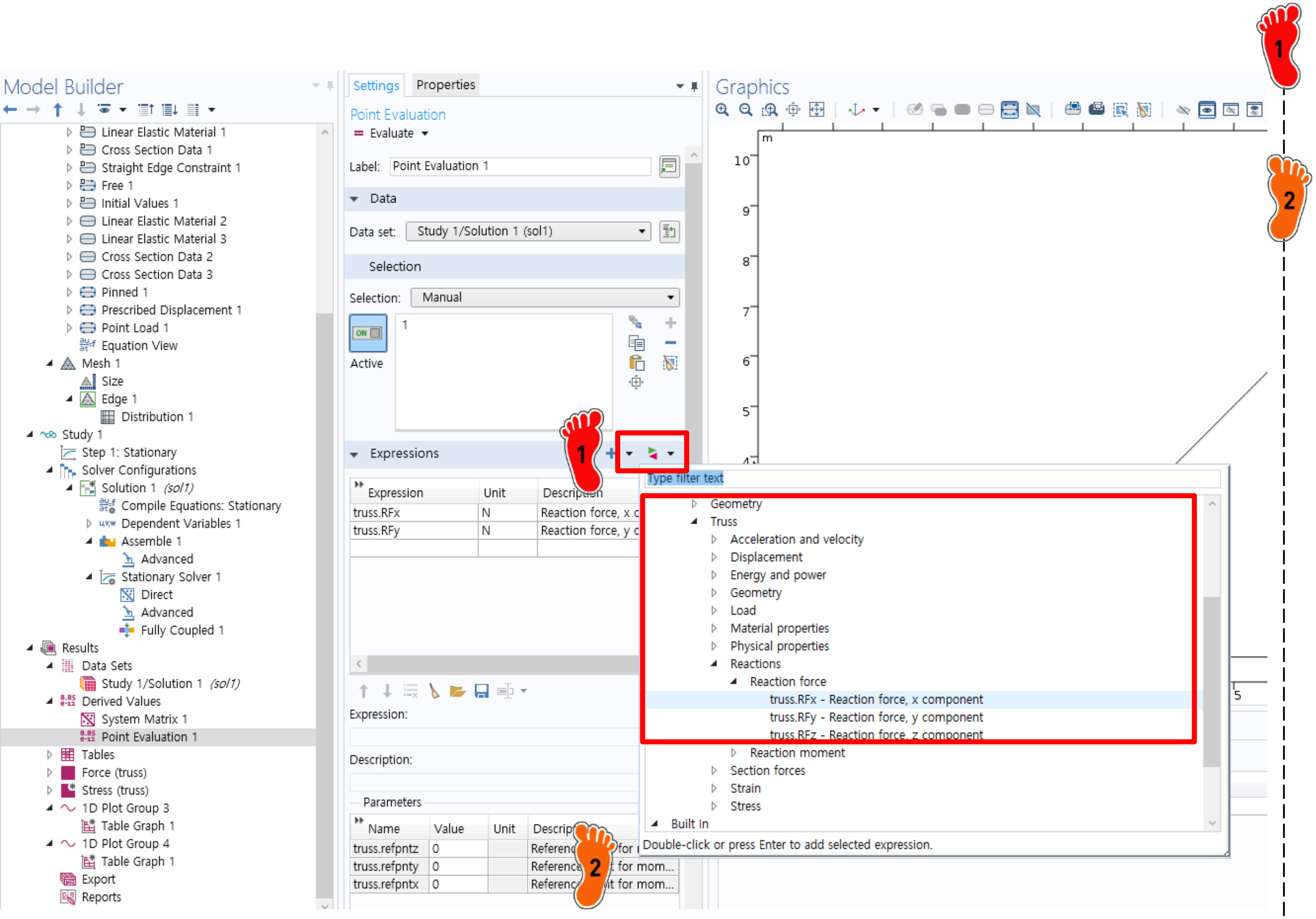
NODAL DISPLACEMENT



1 Point 선택

2 Evaluate 클릭

NODAL REACTION FORCE



1 Replace expression

2 Truss>Reaction
>Reaction force

Truss.RF(x,y,z)
Reaction force 계산

NODAL REACTION FORCE

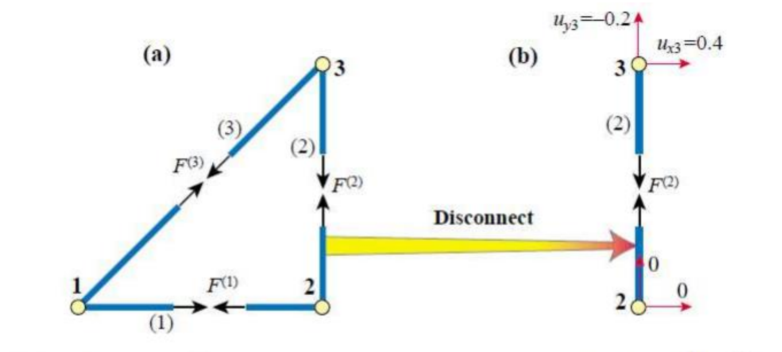


FIGURE 3.3. Internal force recovery for example truss: (a) member axial forces $F^{(1)}$, $F^{(2)}$ and $F^{(3)}$, with arrow directions pertaining to tension; (b) details of computation for member (2).

$$\mathbf{f} = \mathbf{K}\mathbf{u} = \begin{bmatrix} 20 & 10 & -10 & 0 & -10 & -10 \\ 10 & 10 & 0 & 0 & -10 & -10 \\ -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 5 & 0 & -5 \\ -10 & -10 & 0 & 0 & 10 & 10 \\ -10 & -10 & 0 & -5 & 10 & 15 \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0.4 \\ -0.2 \end{bmatrix} = \underbrace{\begin{bmatrix} -2 \\ -2 \\ 0 \\ 1 \\ 2 \\ 1 \end{bmatrix}}_{\text{Reaction Forces}}$$

COMSOL의 Load벡터는 Reaction force와 B.C인 Point load를 구분해서 출력

Reaction force, x component (N), Point: 1	Reaction force, y component (N), Point: 1
-2.0000	-2.0000

Reaction force, x component (N), Point: 2	Reaction force, y component (N), Point: 2
0.0000	1.0000

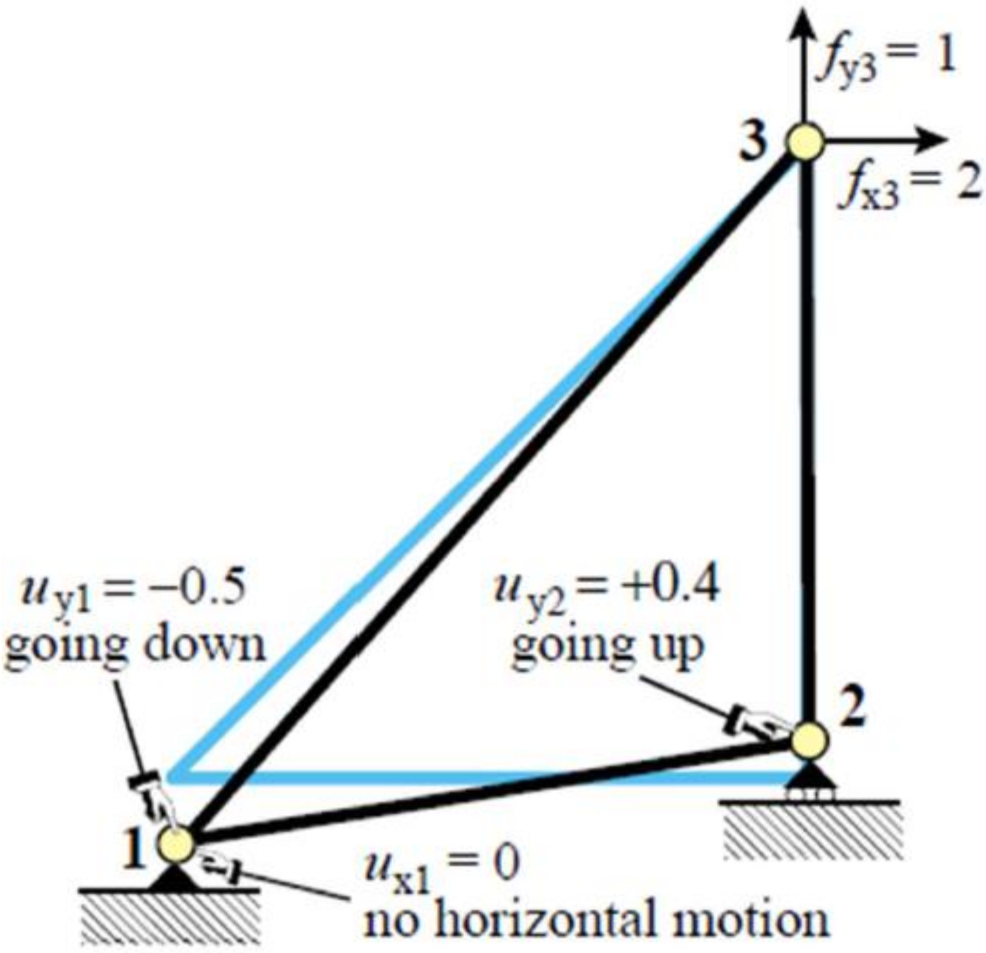
Reaction force, x component (N), Point: 3	Reaction force, y component (N), Point: 3
0.0000	0.0000

Type filter text

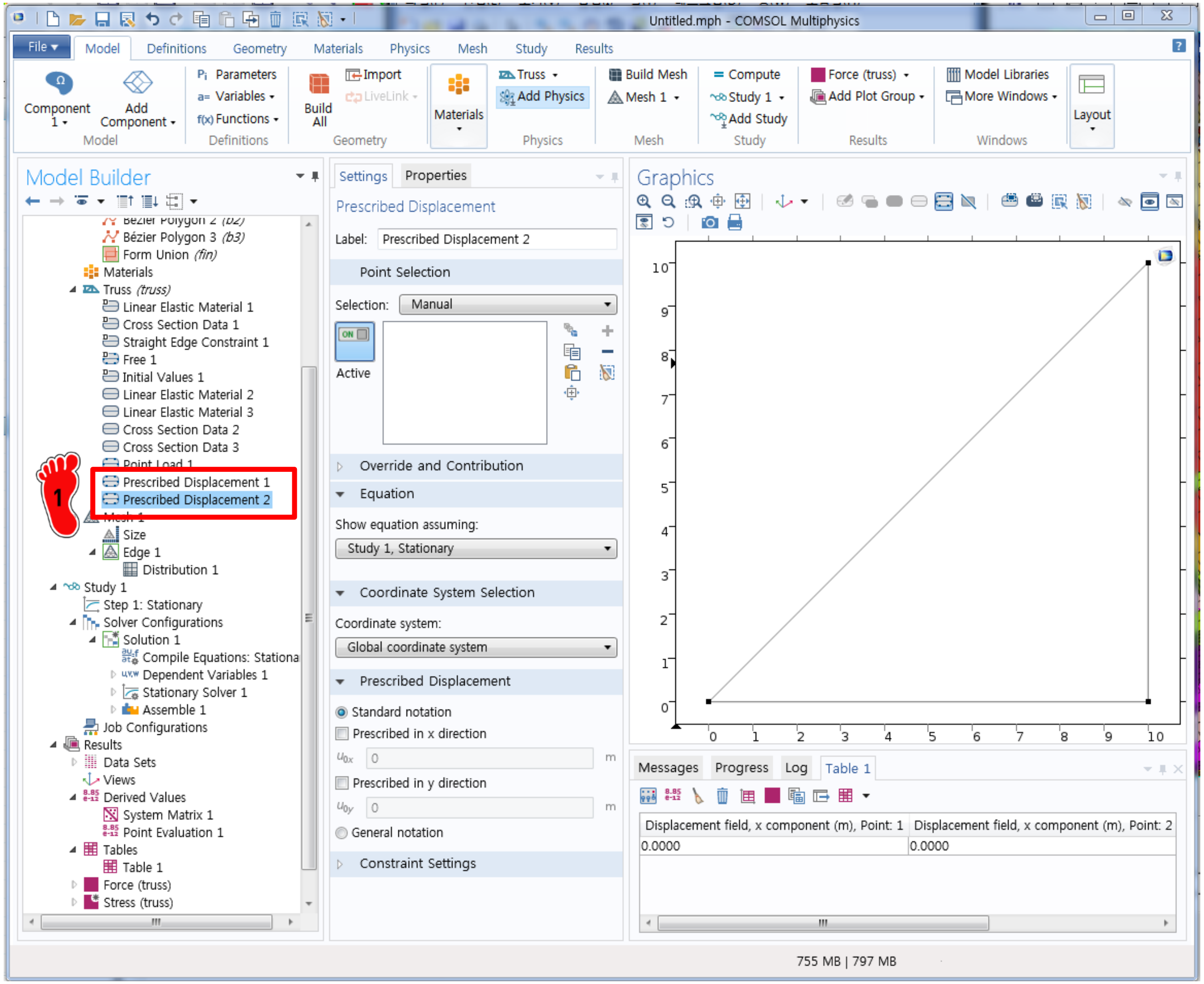
- Component 1
  - Definitions
  - Geometry
  - Truss
    - Acceleration and velocity
    - Displacement
    - Energy and power
    - Geometry
    - Load
      - truss.FpMag - Point load magnitude
      - Point load
        - truss.Fpx - Point load, x component
        - truss.Fpy - Point load, y component

Point load, x component (N), Point: 3	Point load, y component (N), Point: 3
2.0000	1.0000

EXAMPLE TRUSS STRUCTURE

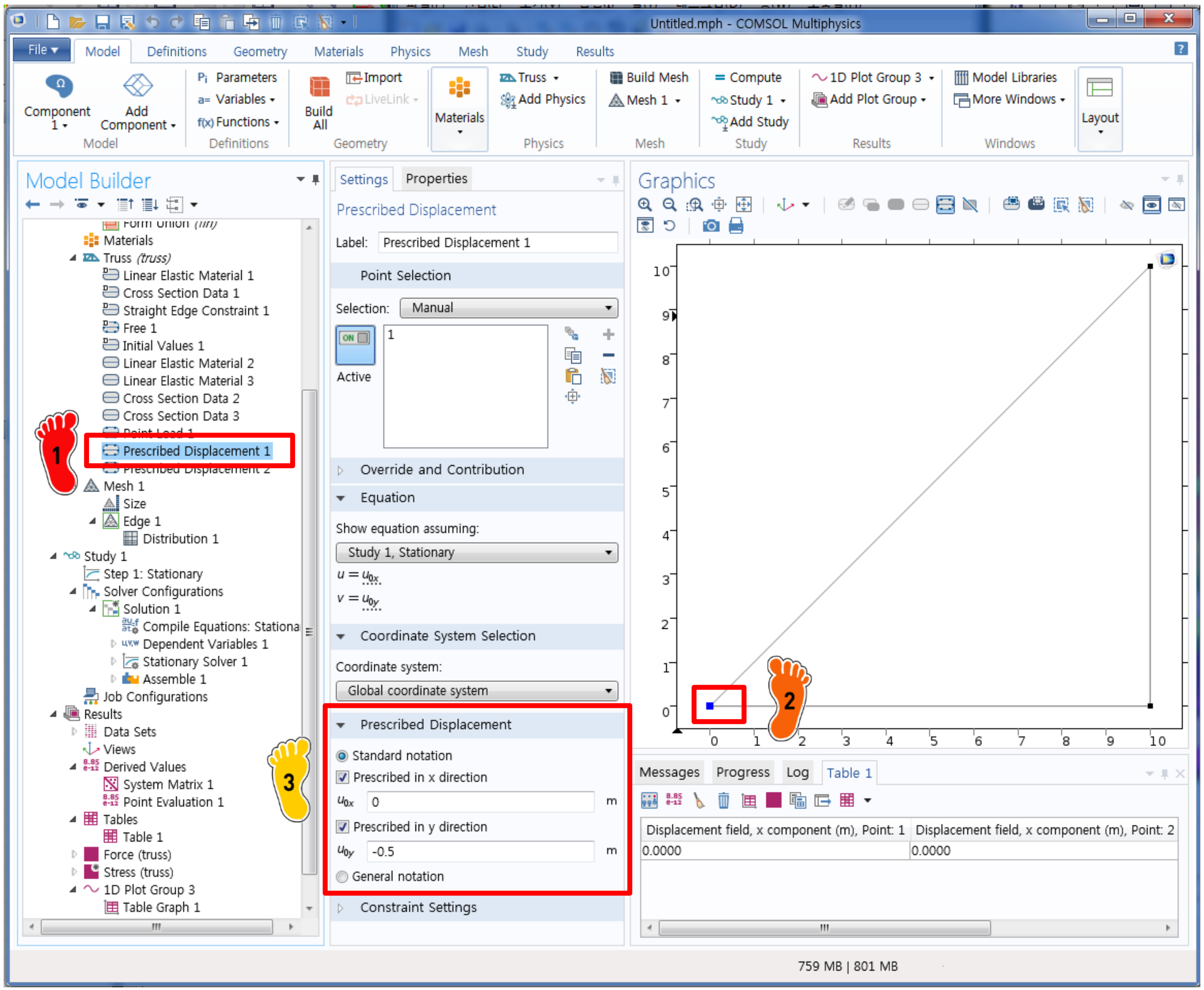


BOUNDARY CONDITION



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

BOUNDARY CONDITION



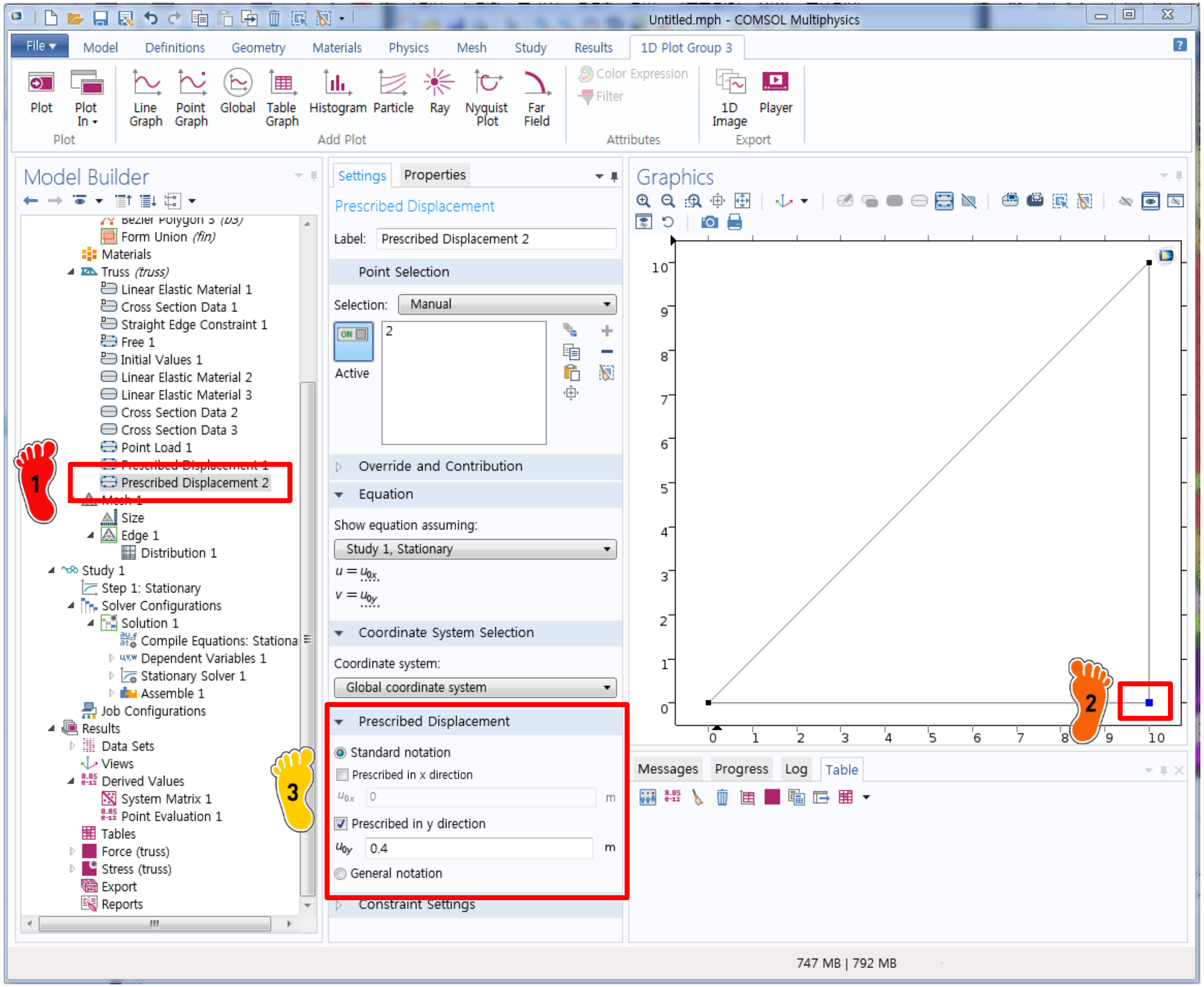
첫 번째 Prescribed Displacement 클릭

왼쪽 아래 절점 선택

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

입력

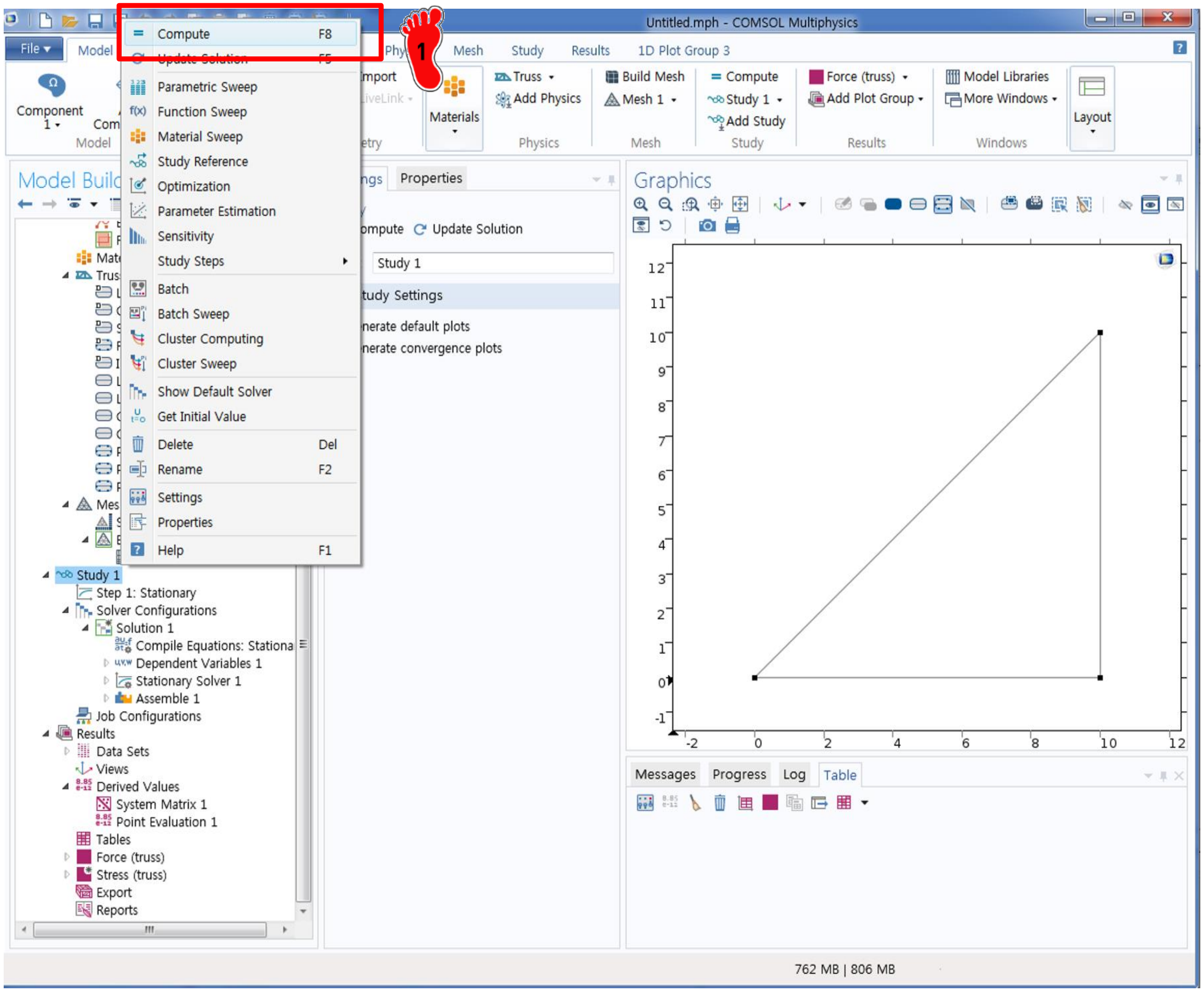
BOUNDARY CONDITION



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

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

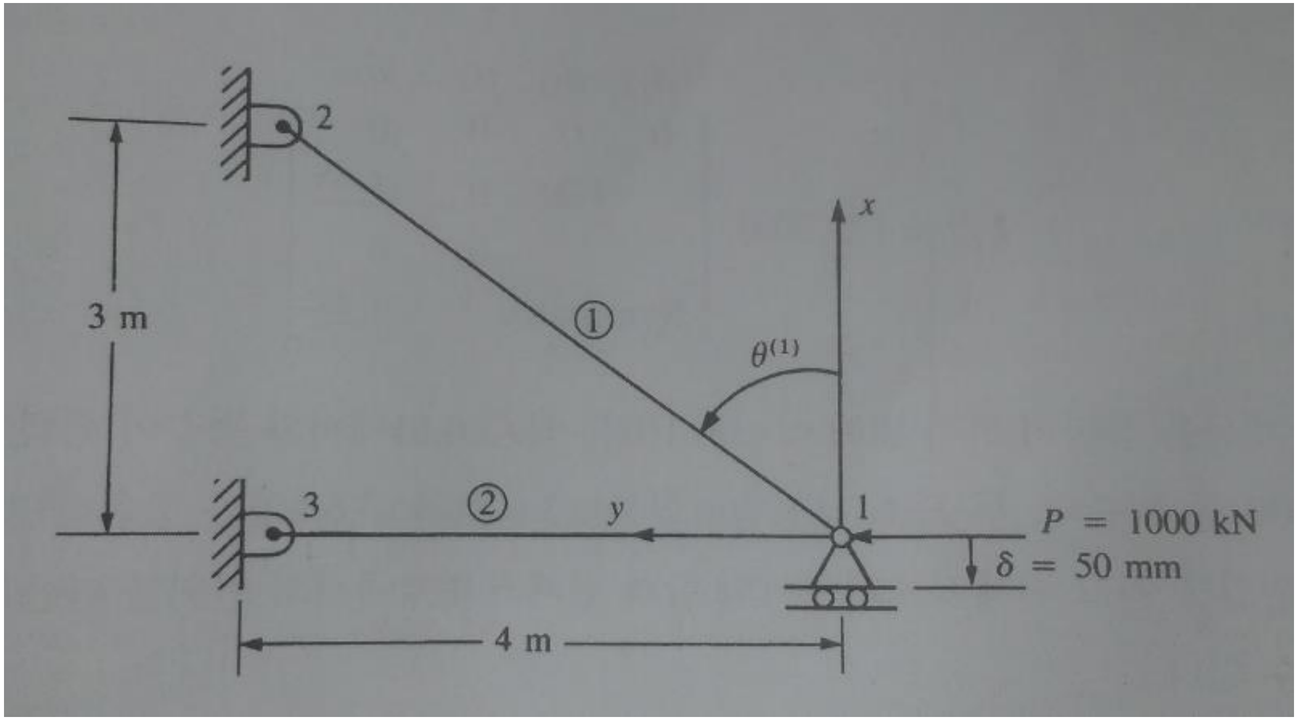
In summary, the only changes to the SDM is in the application of displacement boundary conditions before solve

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