Direct Stiffness Method by COMSOL

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

CONTENTS

- Introduction
- COMSOL desktop
- COMSOL analysis: Heat Transfer
- COMSOL analysis: Truss structure
- Stiffness matrix confirm
- Prescribed displacement boundary condition
- Livelink with MATLAB

COMSOL MULTIPHYSICS

- Finite element analysis and simulation software package for various physics and engineering applications
- Founded by Savante Littmarck and Farhad Saeidi in 1986
- FEMLAB: Early version of COMSOL (before 2005)

Dr. h.c. Svante Littmarck
CEO of the COMSOL Group

Mr. Farhad Saeidi
President of COMSOL AB

MODULES

COMSOL Multiphysics®

ELECTRICAL	MECHANICAL	FLUID	CHEMICAL	MULTIPURPOSE	INTERFACING	
AC/DC Module	Heat Transfer Module	CFD Module	Chemical Reaction Engineering Module	Optimization Module®	LiveLink™ for MATLAB®	LiveLink™ for Excel®
RF Module	Structural Mechanics Module	Microfluidics Module	Batteries & Fuel Cells Module	Material Library	CAD Import Module	ECAD Import Module
MEMS Module	Nonlinear Structural Materials Module	Subsurface Flow Module	Electrodeposition Module	Particle Tracing Module	LiveLink™ for SolidWorks®	LiveLink™ for SpaceClaim®
Plasma Module	Geomechanics Module	Pipe Flow Module	Corrosion Module		LiveLink™ for Inventor®	LiveLink™ for AutoCAD®
	Fatigue Module				LiveLink™ for Creo™ Parametric	LiveLink™ for Pro/ENGINEER®
	Acoustics Module				LiveLink™ for Solid Edge®	File Import for CATIA®V5

AC/DC MODULE

ELECTRICAL

AC/DC Module

RF Module

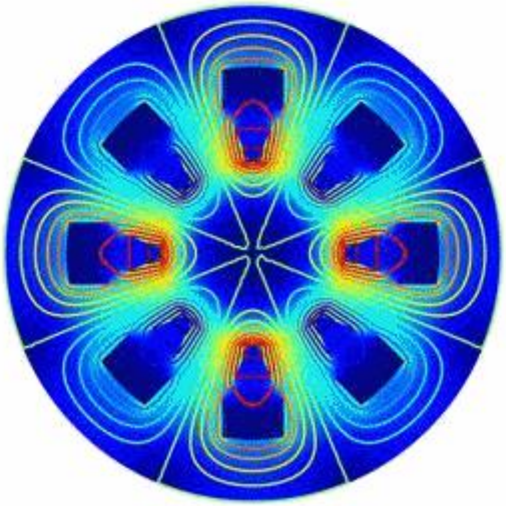
MEMS Module

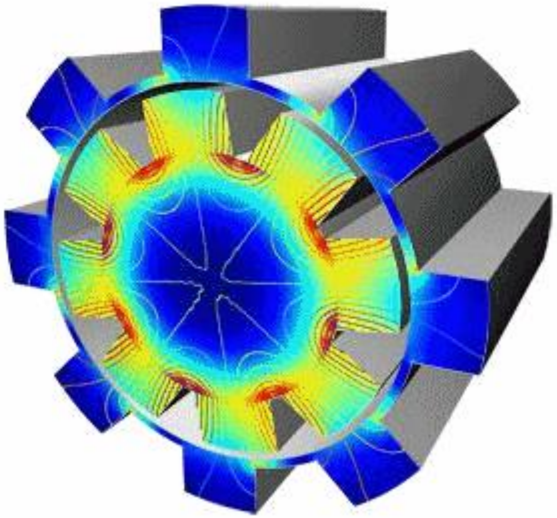
Plasma Module

Dynamics of a Generator

This example shows how the circular motion of a rotor with permanent magnets in a generator results in an induced EMF in the stator winding. The generated voltage is calculated as a function of time during the rotation.

The plot on the left shows the magnetic flux density along with a contour plot of the magnetic potential. Note the brighter regions, which indicate the position of the permanent magnets in the rotor. The figure on the right shows the geometry and a simulation of the generator in 3D.





HEAT TRANSFER MODULE

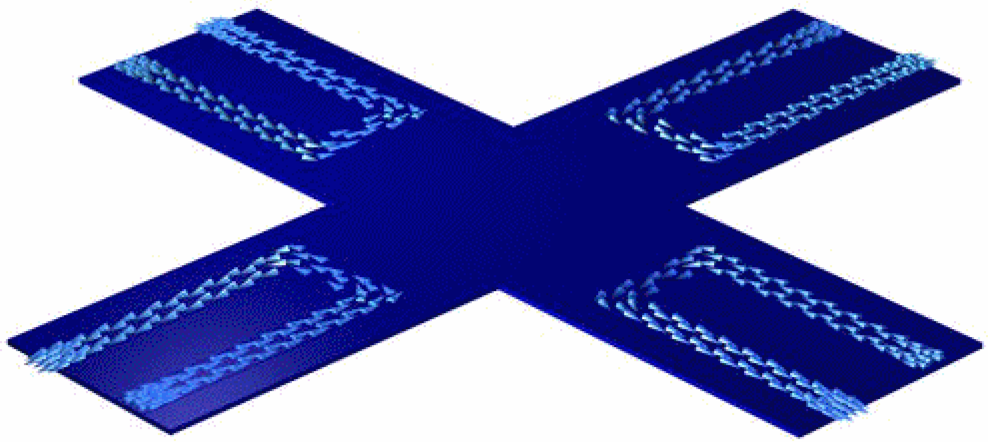
MECHANICAL

- Heat Transfer Module
- Structural Mechanics Module
- Nonlinear Structural Materials Module
- Geomechanics Module
- Fatigue Module
- Acoustics Module

Deformation of a Thermomechanical Microvalve

Thermomechanical microvalves are common flow control components in microfluidics systems. Here, an electric current generates movement by resistively heating the actuator structure, thereby causing mechanical stress and deformation.

In this example, a parametric study shows how an increasing voltage applied to each of the legs leads to temperature rise causing more and more deformation.



CFD MODULE

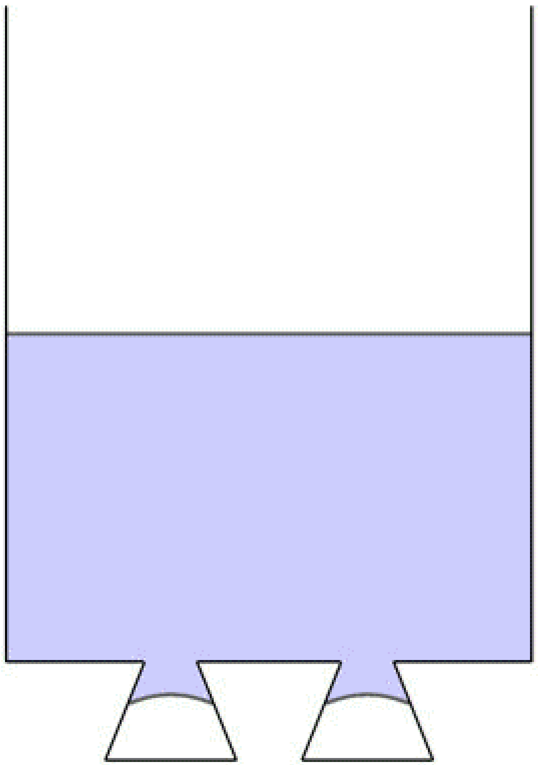
FLUID

- CFD Module
- Microfluidics Module
- Subsurface Flow Module
- Pipe Flow Module

Boiling Water

This model studies the film boiling of water. A heat flux above the Leidenfrost point is applied at the surface of two cavities. A layer of vapor is maintained at the hot surface - liquid interface where film-boiling results.

The animation shows the fluids volume fraction over time as a surface and contour plot.

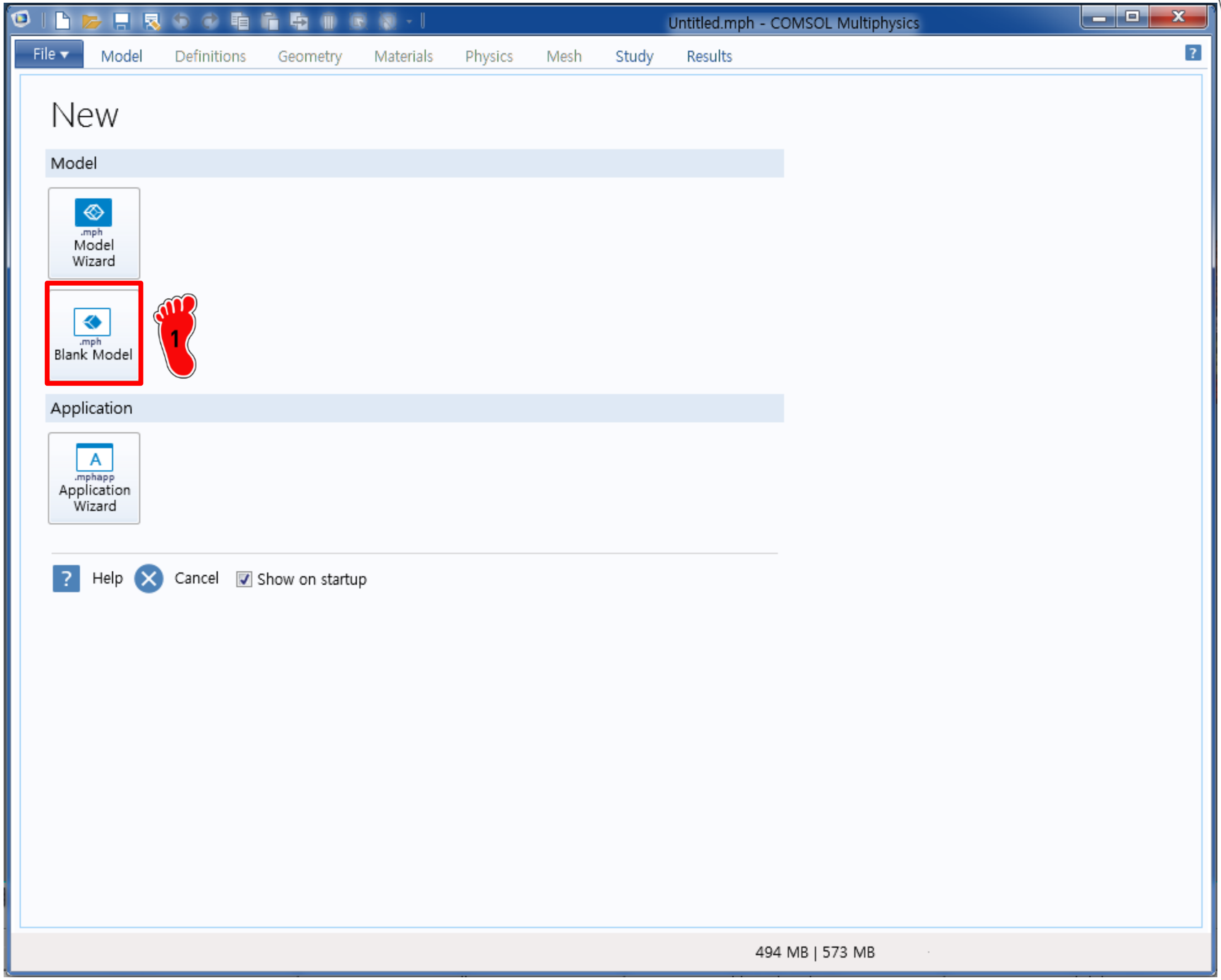


MULTIPHYSICS

COMSOL Multiphysics®

- **COMSOL desktop**
  - ✓ Toolbars & Ribbon tabs
  - ✓ Windows
  - ✓ Physics
  - ✓ Study types
  - ✓ Setting flow
  - ✓ Setting result

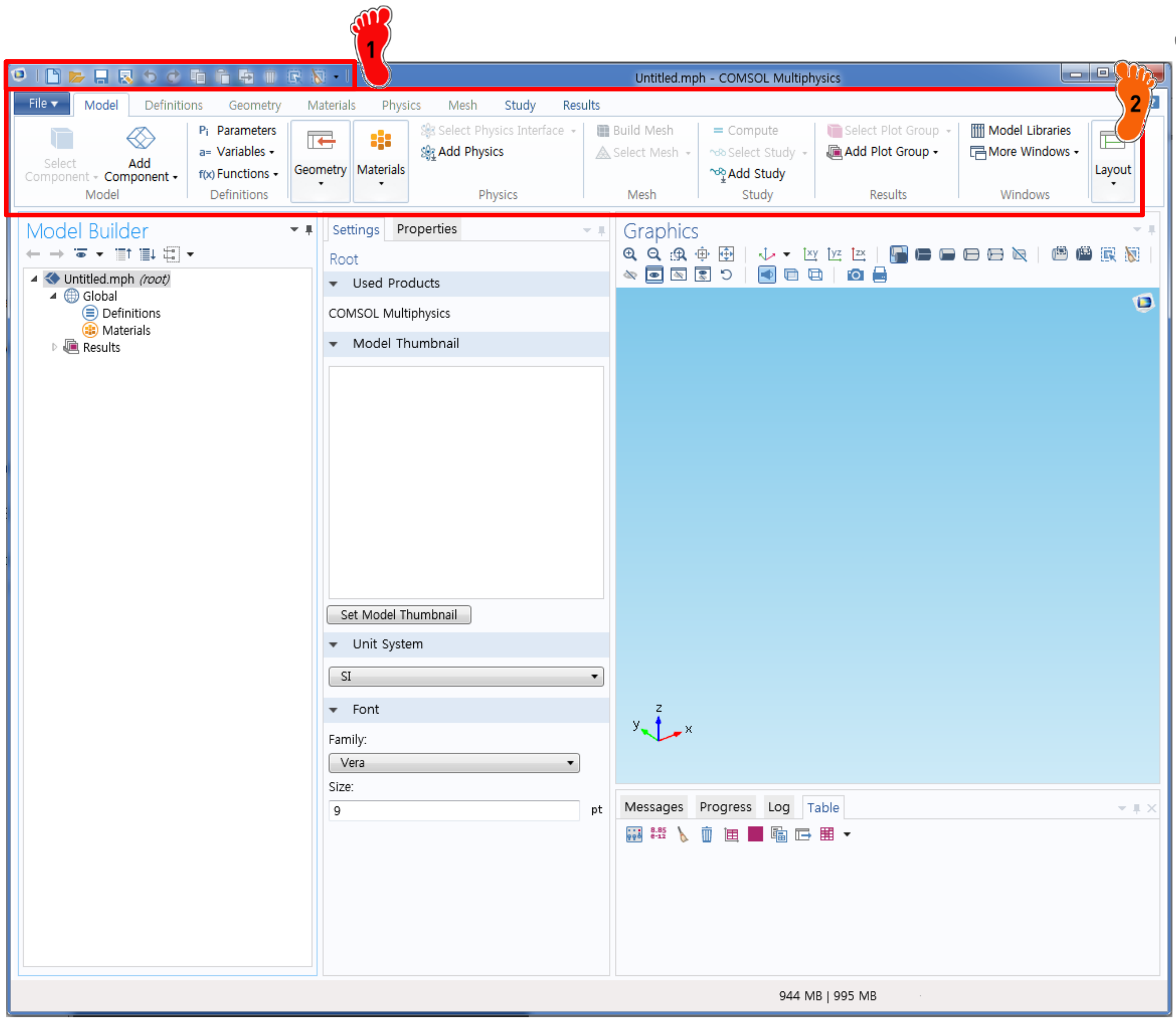
CREATING NEW MODEL



1 Blank Model

The Black Model option will open the COMSOL Desktop interface without any Component or Study.

TOOLBAR & RIBBON TABS

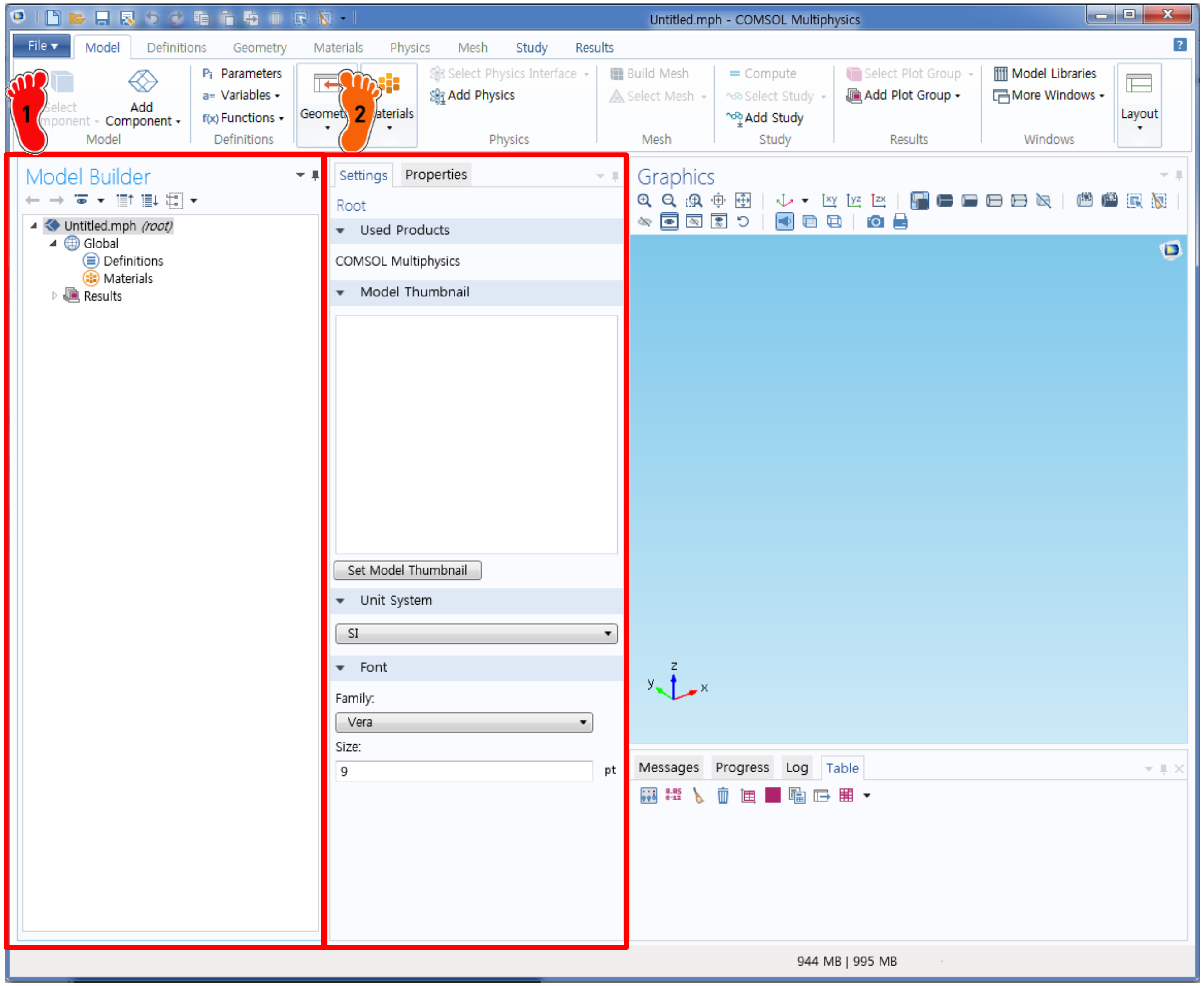


1 Quick Access Toolbar

2 Ribbon tabs

The ribbon tabs on the COMSOL Desktop environment reflect the modeling workflow and give and overview of the functionality available for each modeling

WINDOWS



Model builder window

The Model Builder is the tool where you define the model and its components.

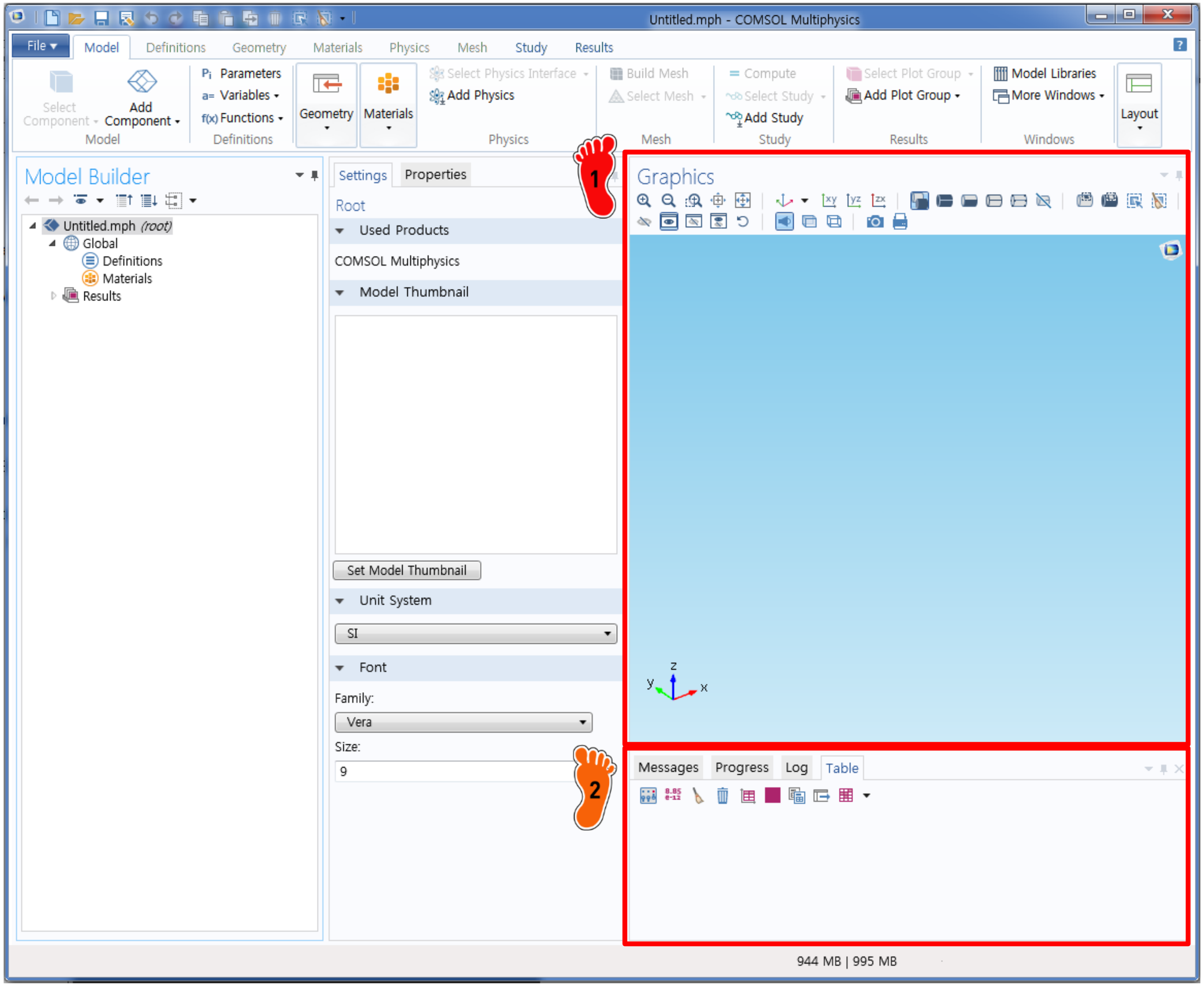
Setting window

It is here that you can edit node setting

Model library window

Easy access to COMSOL and user model libraries.

WINDOWS



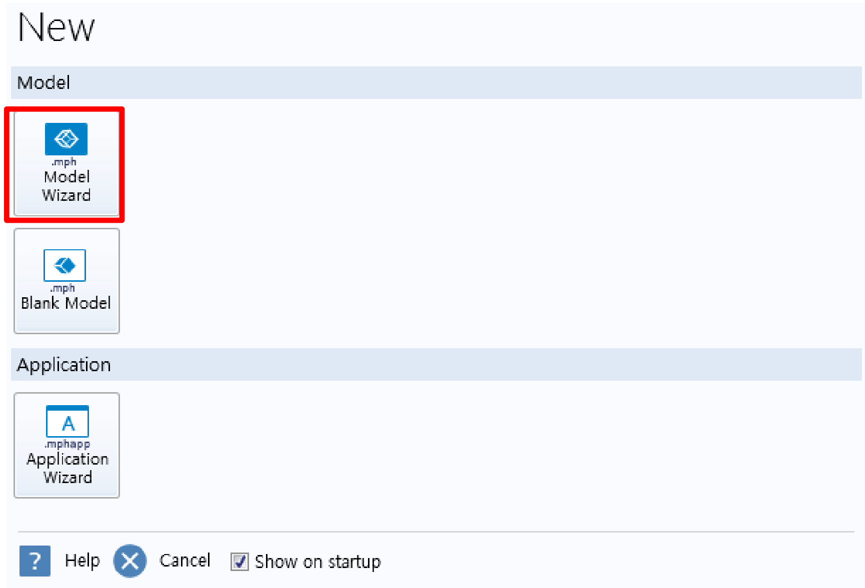
1 Graphics window

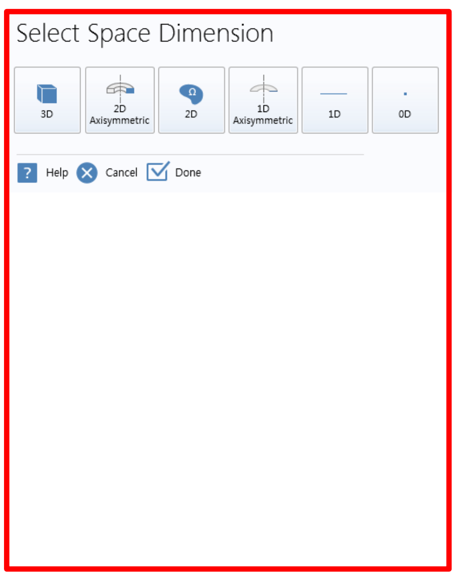
The Graphics window presents interactive graphics for Geometry, Mesh, and Results. Interactions include rotating, panning, zooming, and selecting. It is the default window for most Results and visualizations.

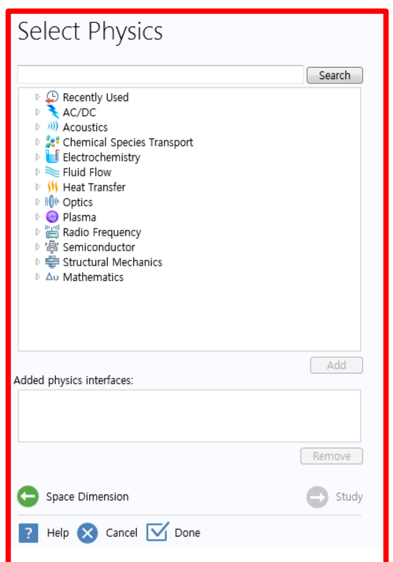
2 Information windows

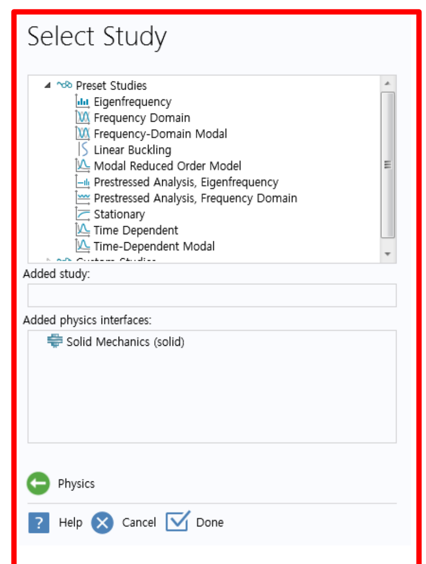
The Information windows will display vital model information during the simulation, such as the solution time, solution progress, mesh statistics and solver logs, as well as Results tables, if any.

SETTING FLOW









1 Model Wizard

The Model Wizard will guide you in setting up the space dimension, physics, and study type, in a few steps

해석을 하기 위해 준비과정은

차원 선택

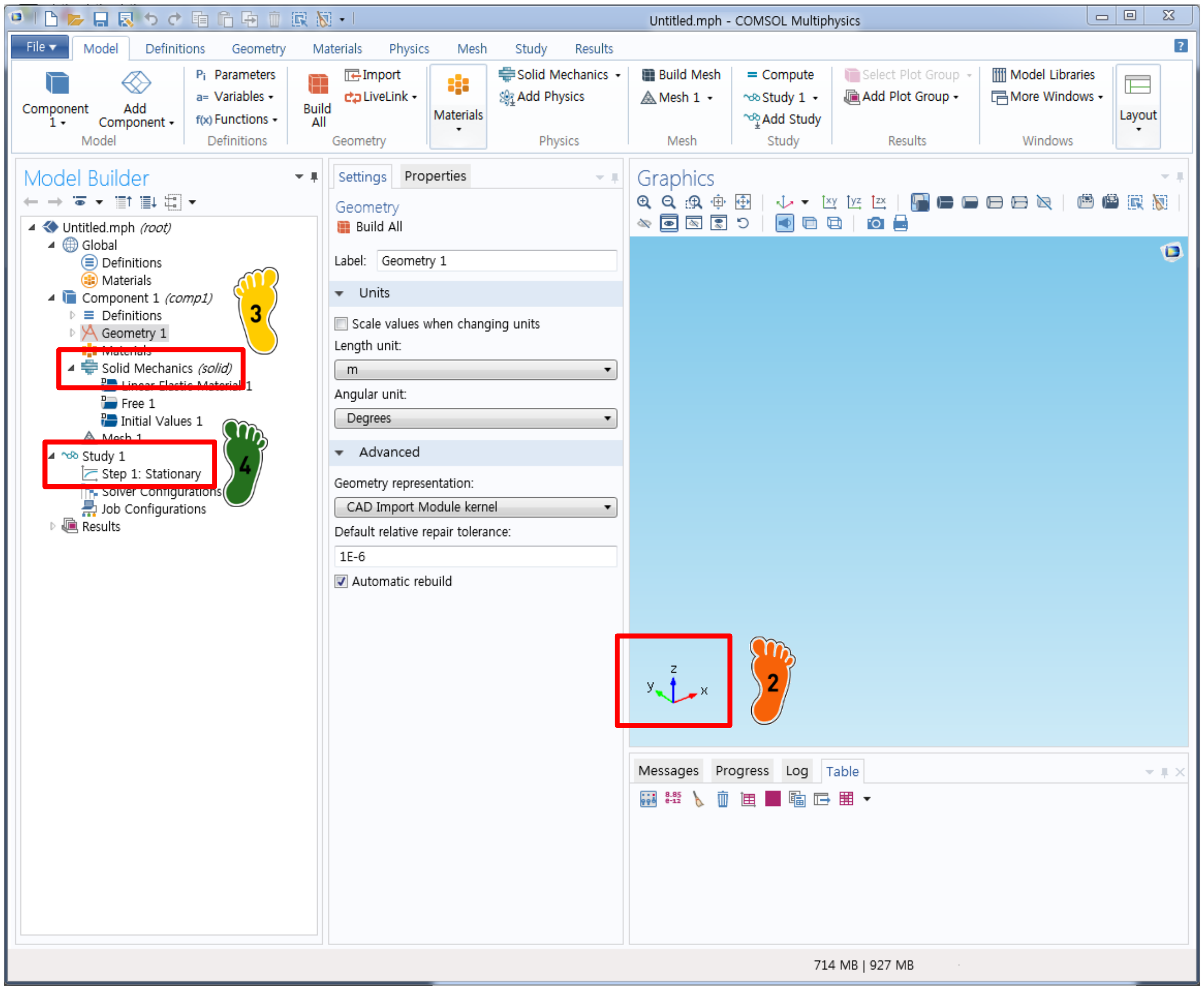
해석 하고자 하는 물리현상 선택

해석 타입 선택

으로 진행됨

ex) 3D – Solid Mechanics (solid) – Stationary

SETTING RESULT



1 전 페이지에서 선택한 setting 결과

2 3차원

3 Solid mechanics

4 Stationary

setting 결과를 확인

HEAT TRANSFER

- Heat balance for a long, thin rod
  - Not insulated along its length
  - Steady state

$$\frac{d^2T}{dx^2} + h'(T_a - T) = 0$$

$$\left.\begin{aligned} T(0) &= T_1 = 40°C \\ T(L) &= T_2 = 200°C \end{aligned}\right\} \text{boundary conditions}$$

$$T_a = 20°C\left(\text{temperature of the surrounding air}\right)$$

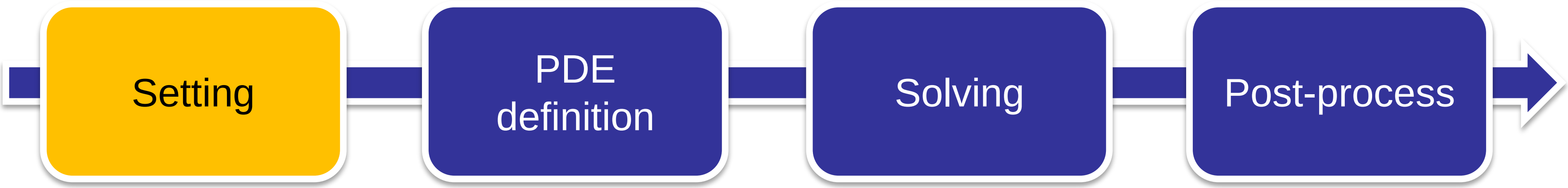
$$L = 10\,m$$

$$h' = 0.01\,m^{-2}\left(\text{heat transfer coefficient}\right)$$

rate of heat dissipation to the surrounding air

analytic solution: $T = 73.4523e^{0.1x} - 53.4523e^{-0.1x} + 20$

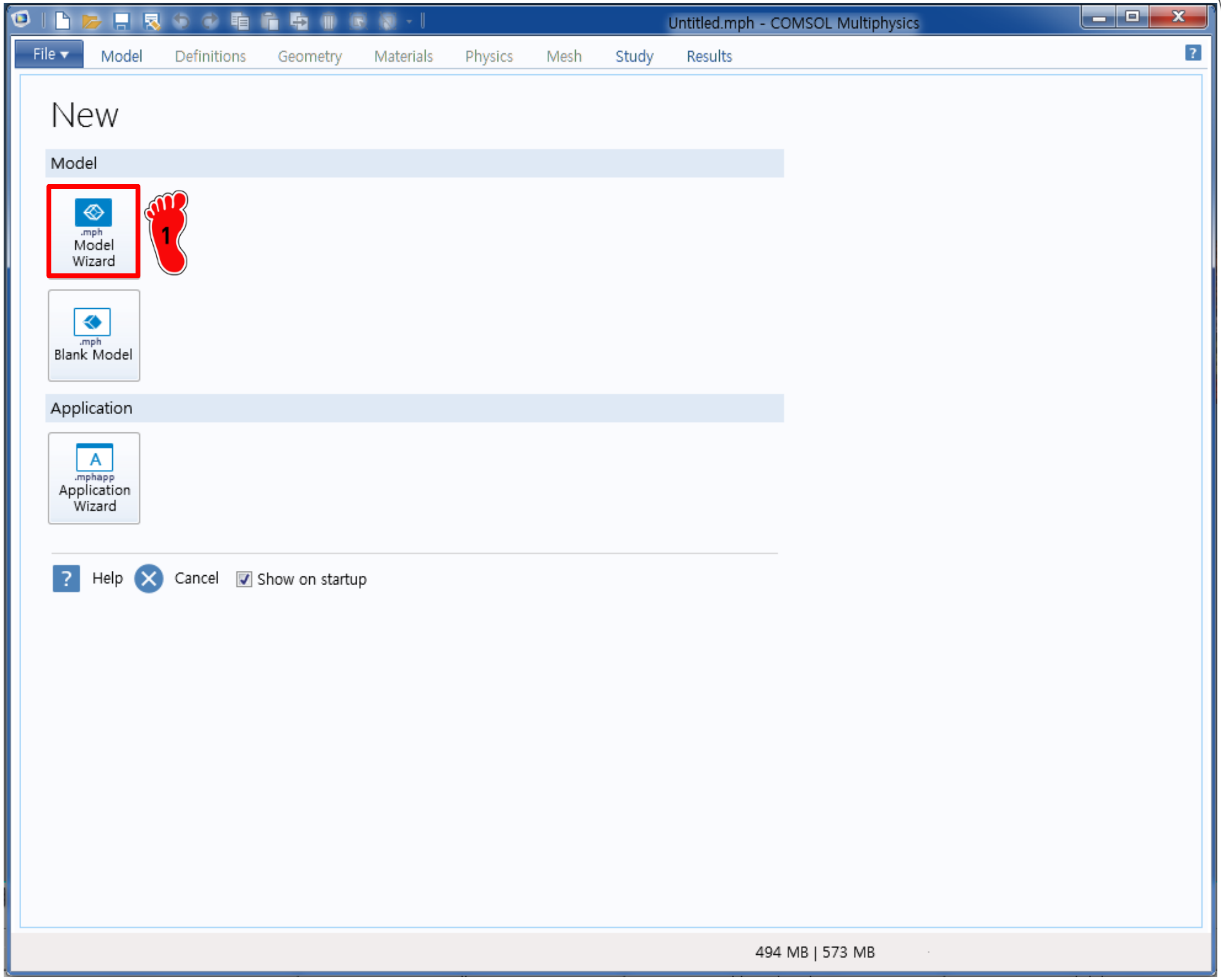
ANALYSIS FLOW



- ✓ Dimension selection
- ✓ Physics selection
- ✓ Study type selection

MODEL WIZARD

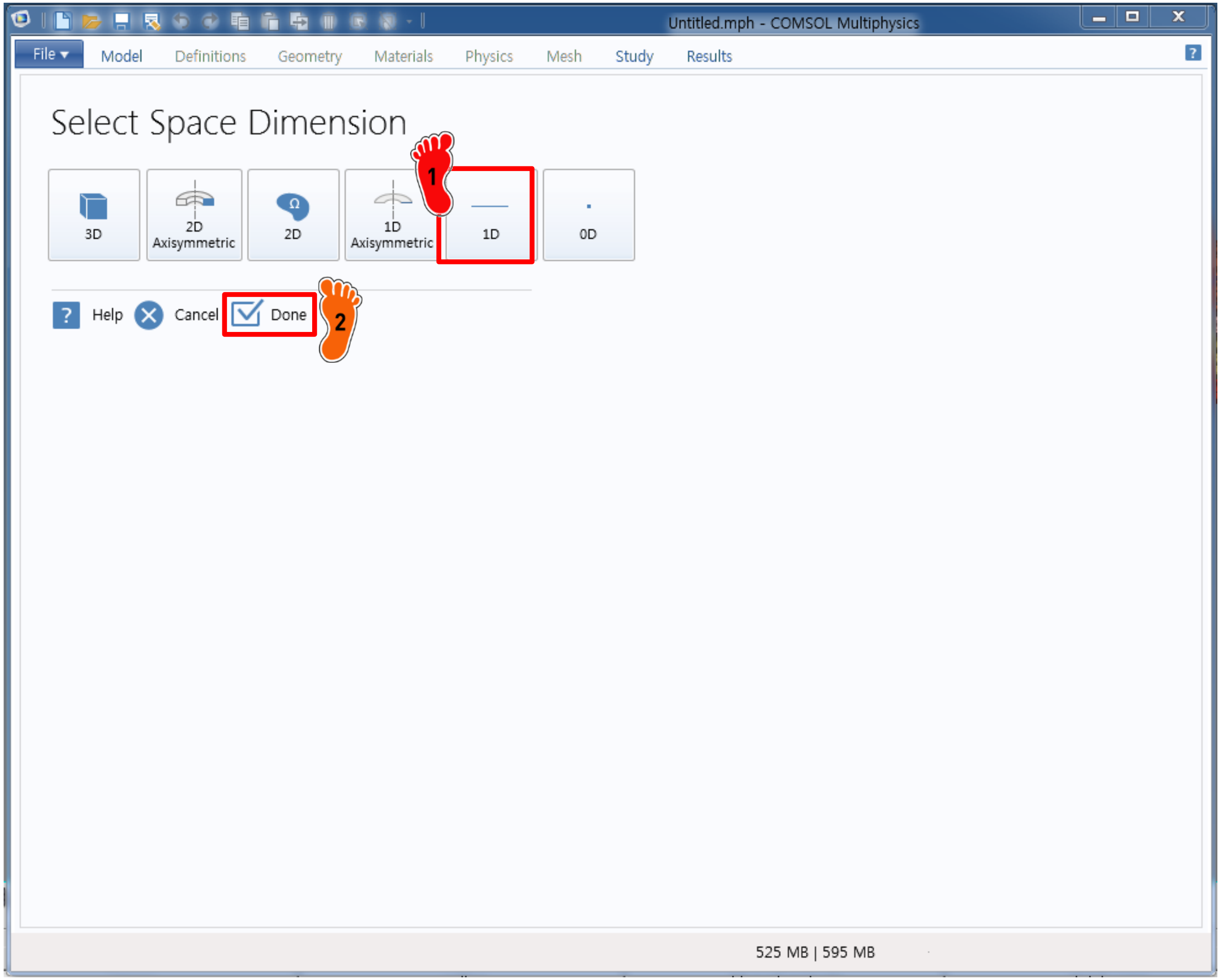
1 Model Wizard 선택



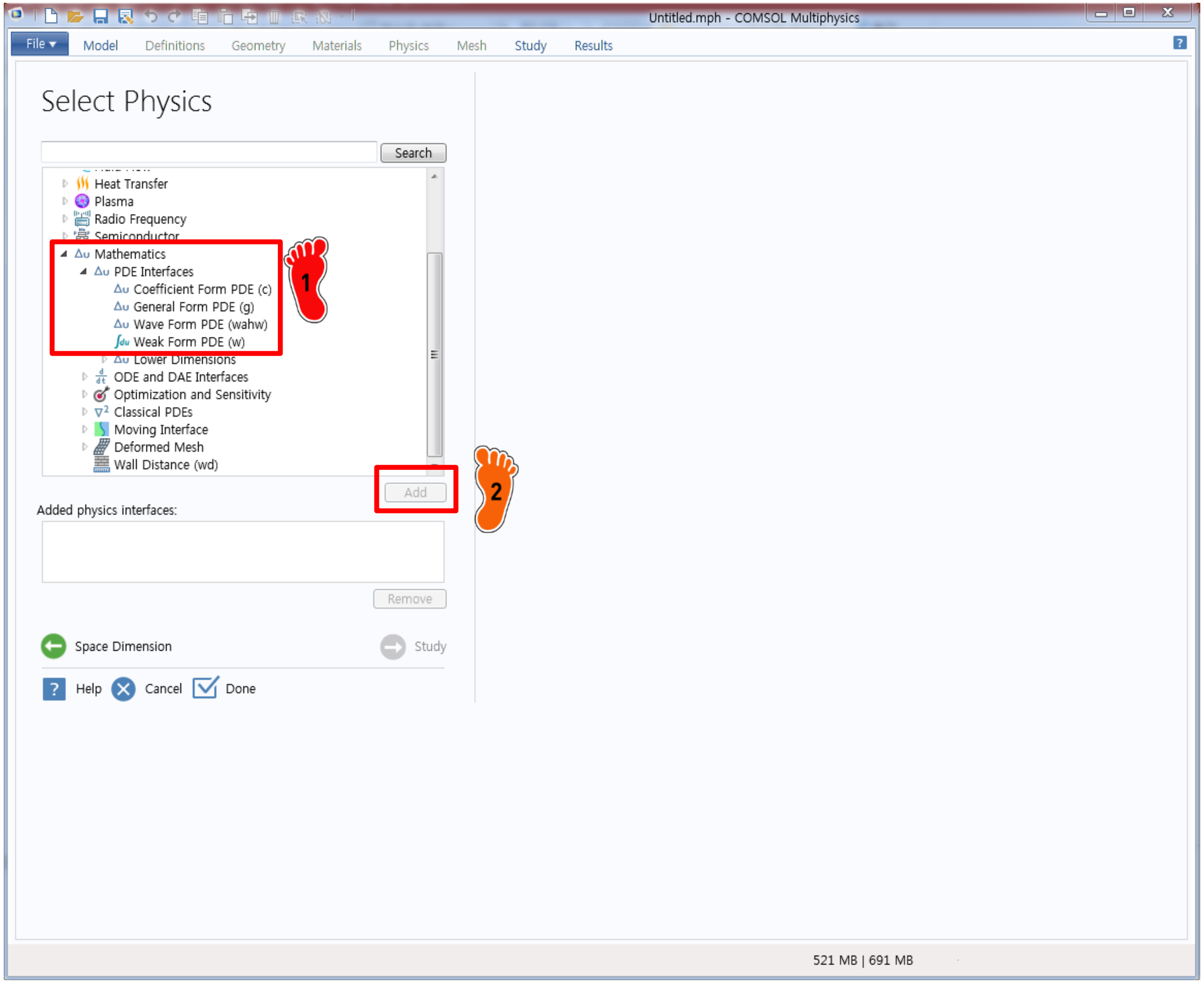
DIMENSION SELECTION

1 1D 선택

2 Done 클릭

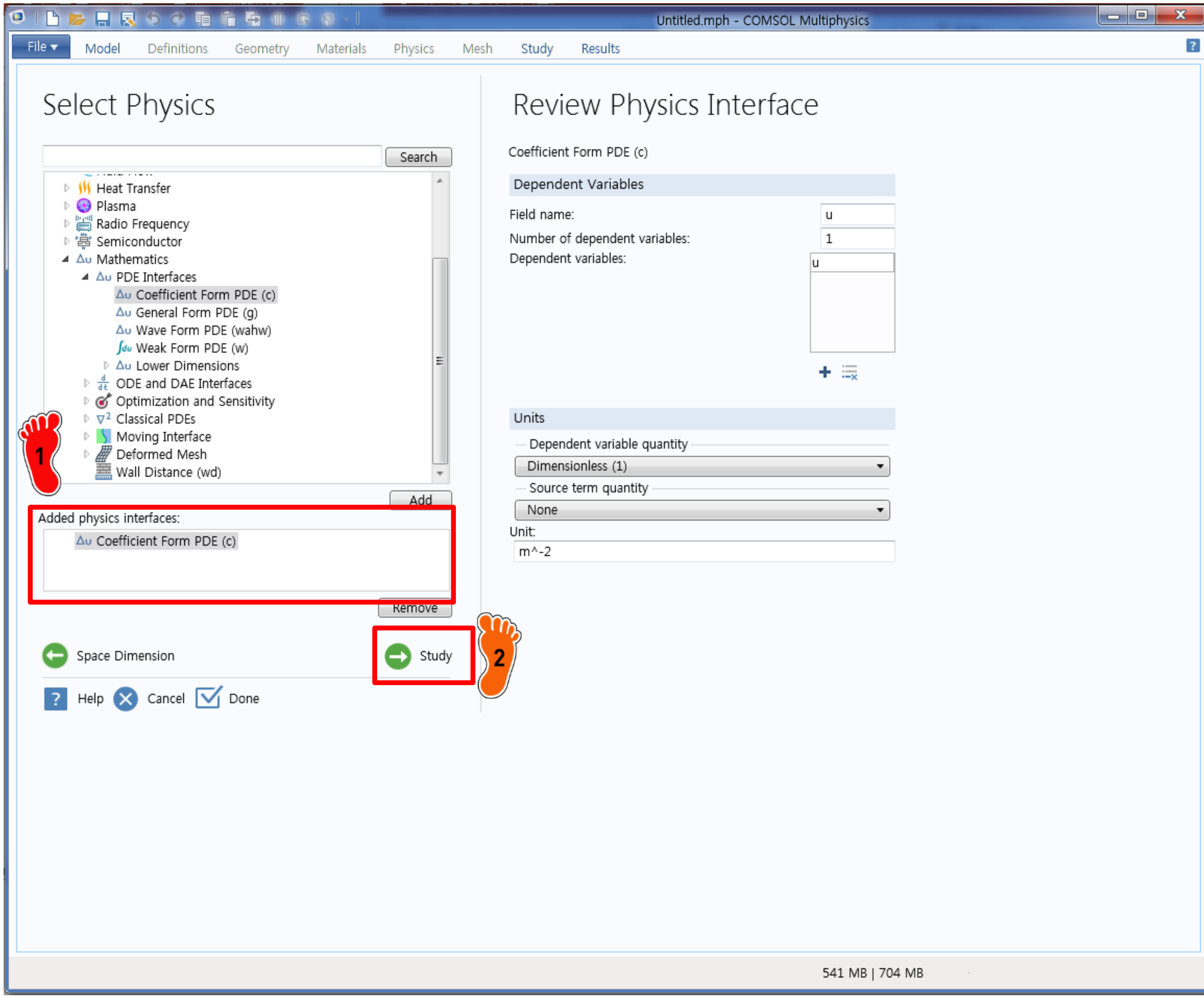


PHYSICS SELECTION



1. Mathematics에서 PDE Interface -> coefficient Form PDE (c) 클릭
2. Add 클릭

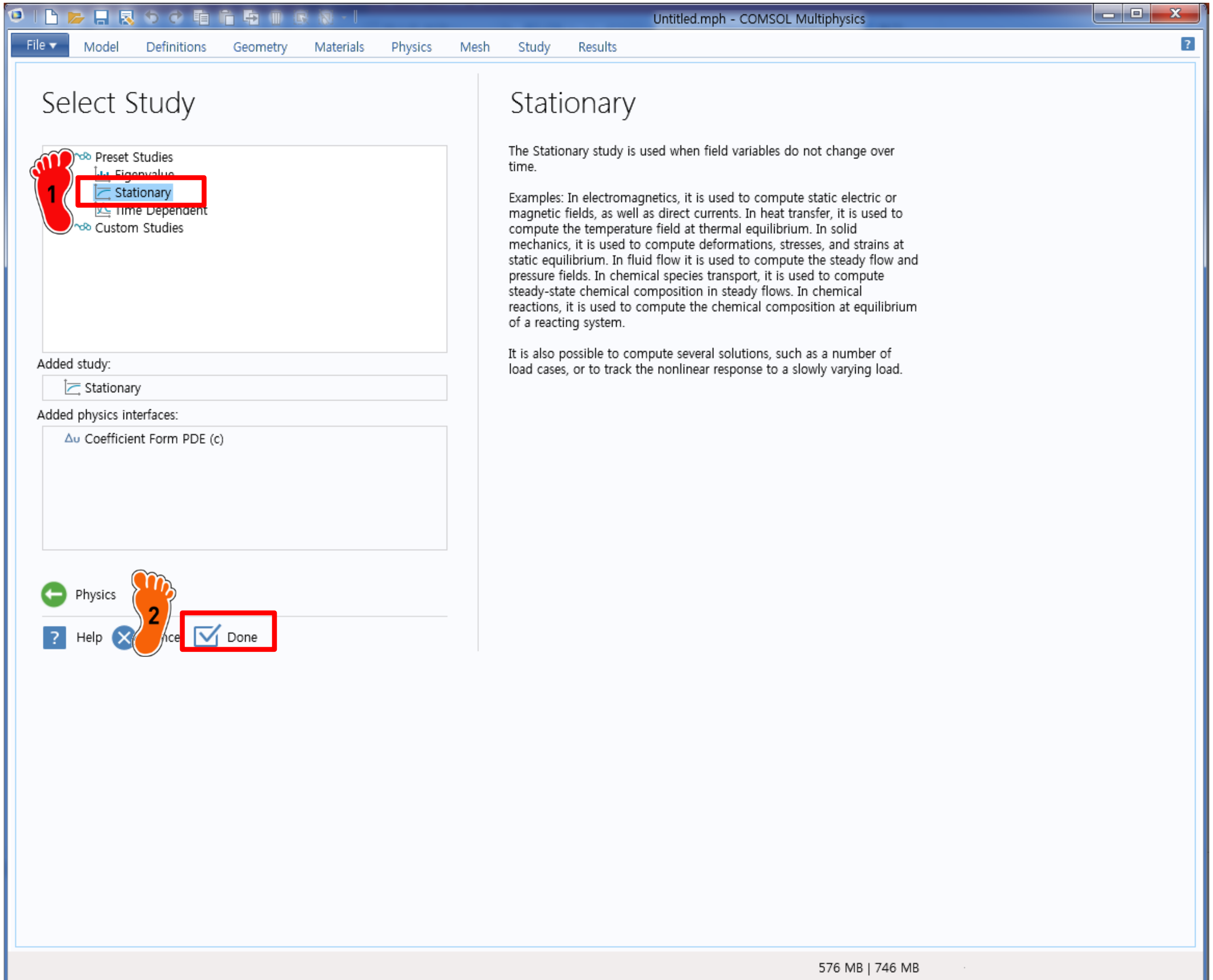
PHYSICS SELECTION



1 Added physics interfaces에서 추가된 physics 확인

2 Study 클릭

STUDY TYPE SELECTION



1 Staionary클릭

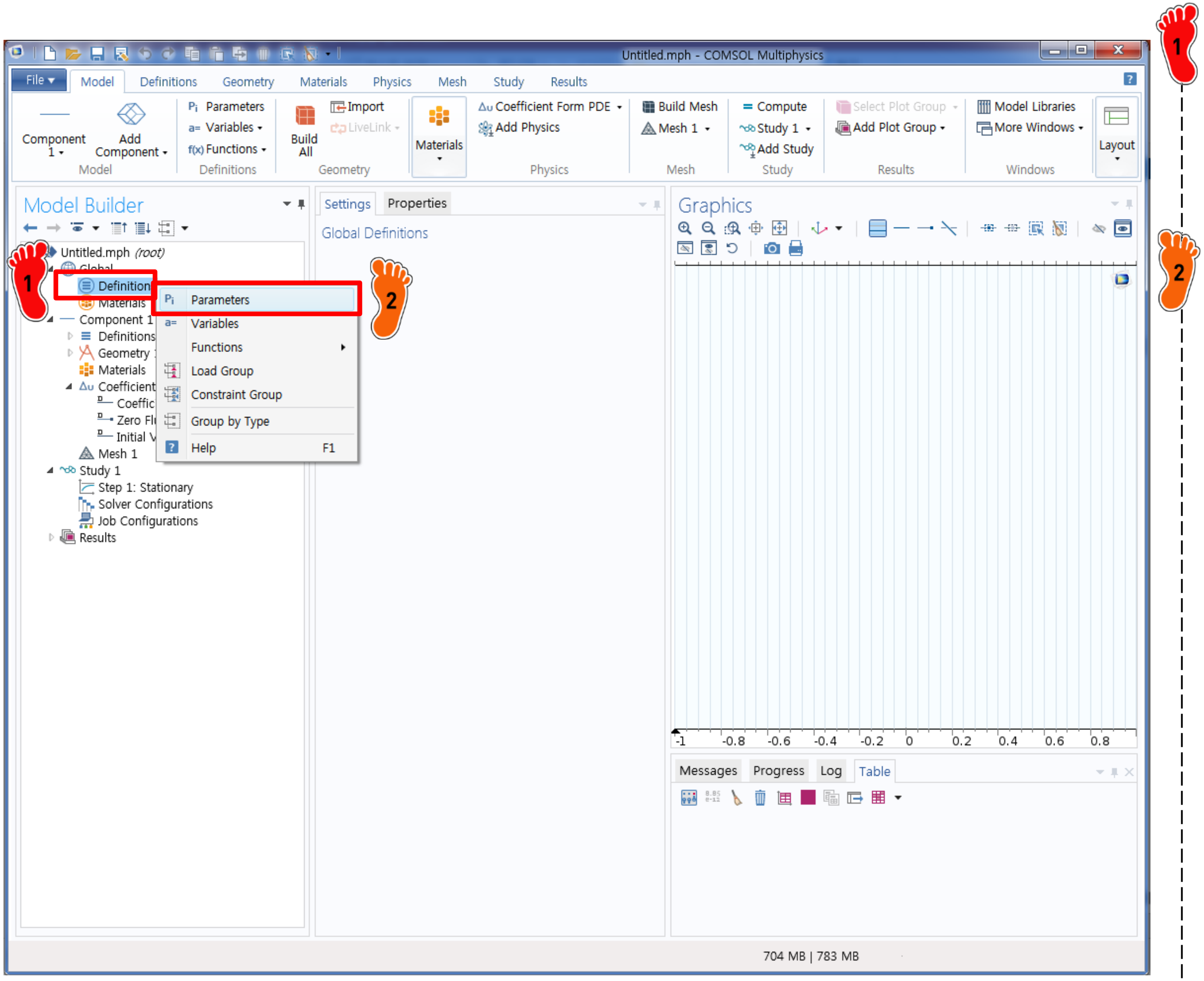
2 Done 클릭

ANALYSIS FLOW

Setting → PDE definition → Solving → Post-process

- ✓ Geometry creation
- ✓ Coefficient input
- ✓ Boundary condition input

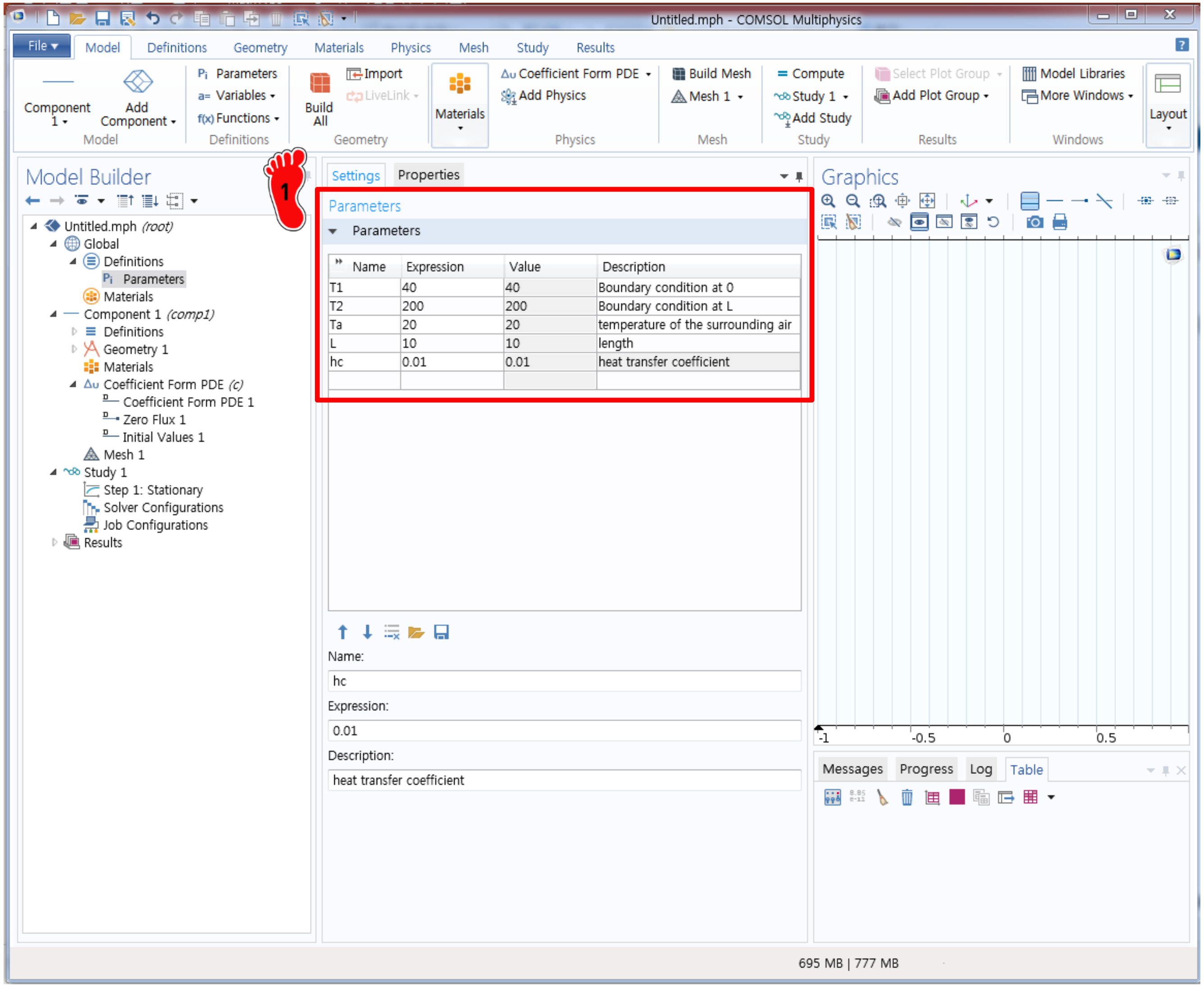
PARAMETER INPUT



1 Definitions 마우스 우클릭

2 Parameters 클릭

PARAMETER INPUT



1 Parameter 값 입력

T1 = 40

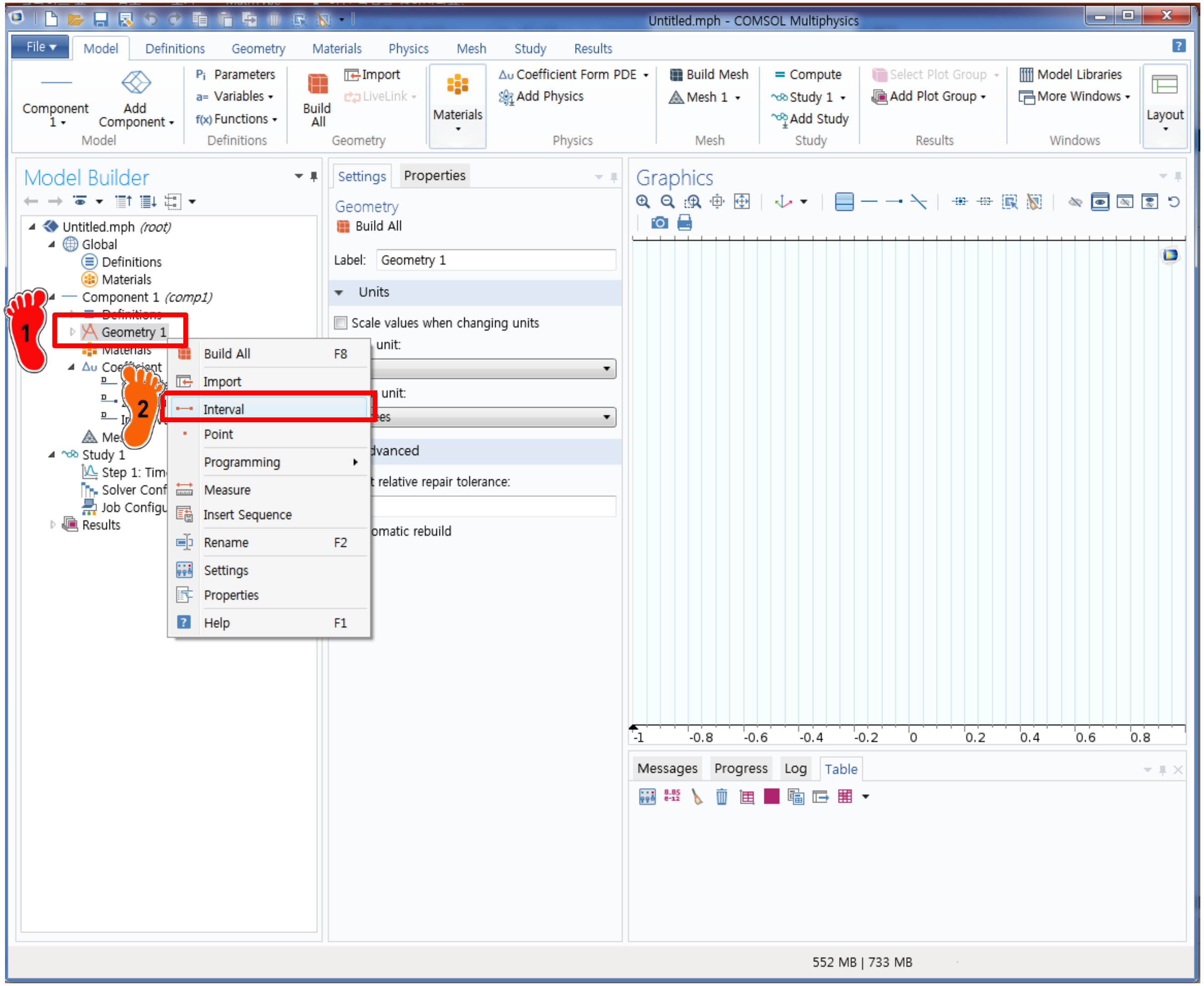
T2 = 200

Ta = 20

L = 10

hc = 0.01

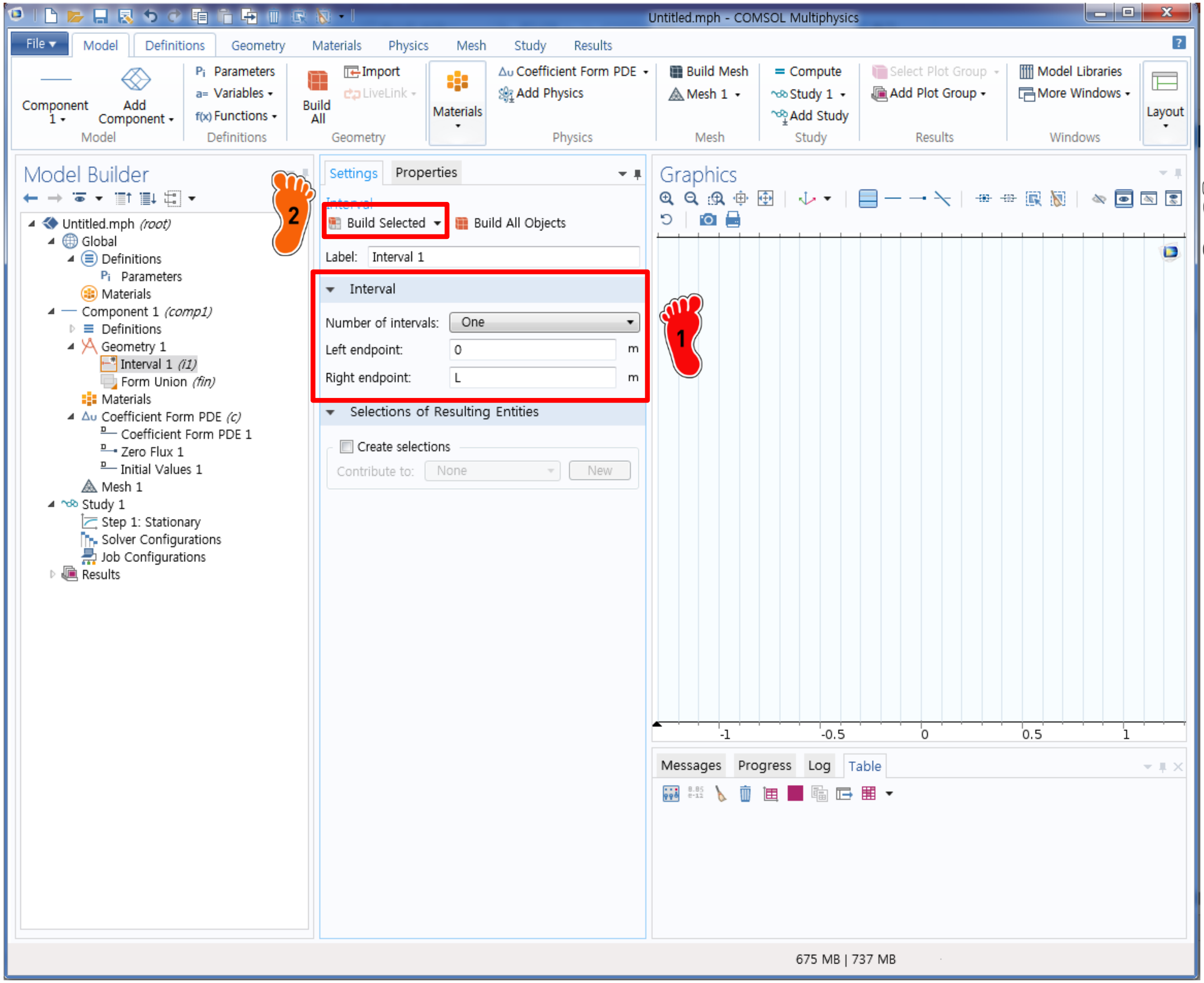
GEOMETRY CREATION



1 Geometry 1 마우스 우클릭

2 Interval 클릭

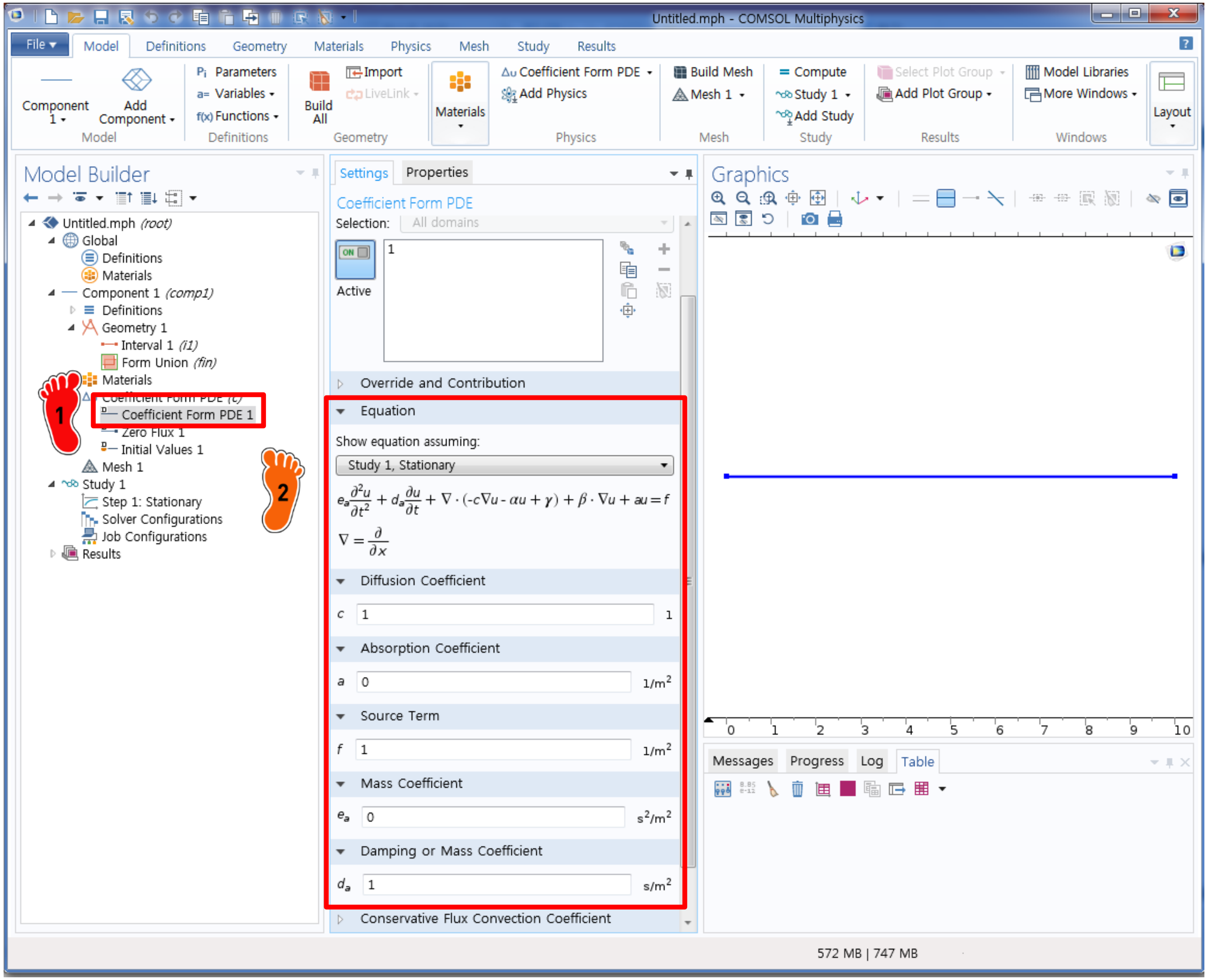
GEOMETRY CREATION



1 Left endpoint에 0, Right endpoint에 L 값을 입력

2 Build selected 클릭

COEFFICIENT INPUT



1 Coefficient Form PDE 1 클릭

2 계수 값 입력

c = -1

a = -hc

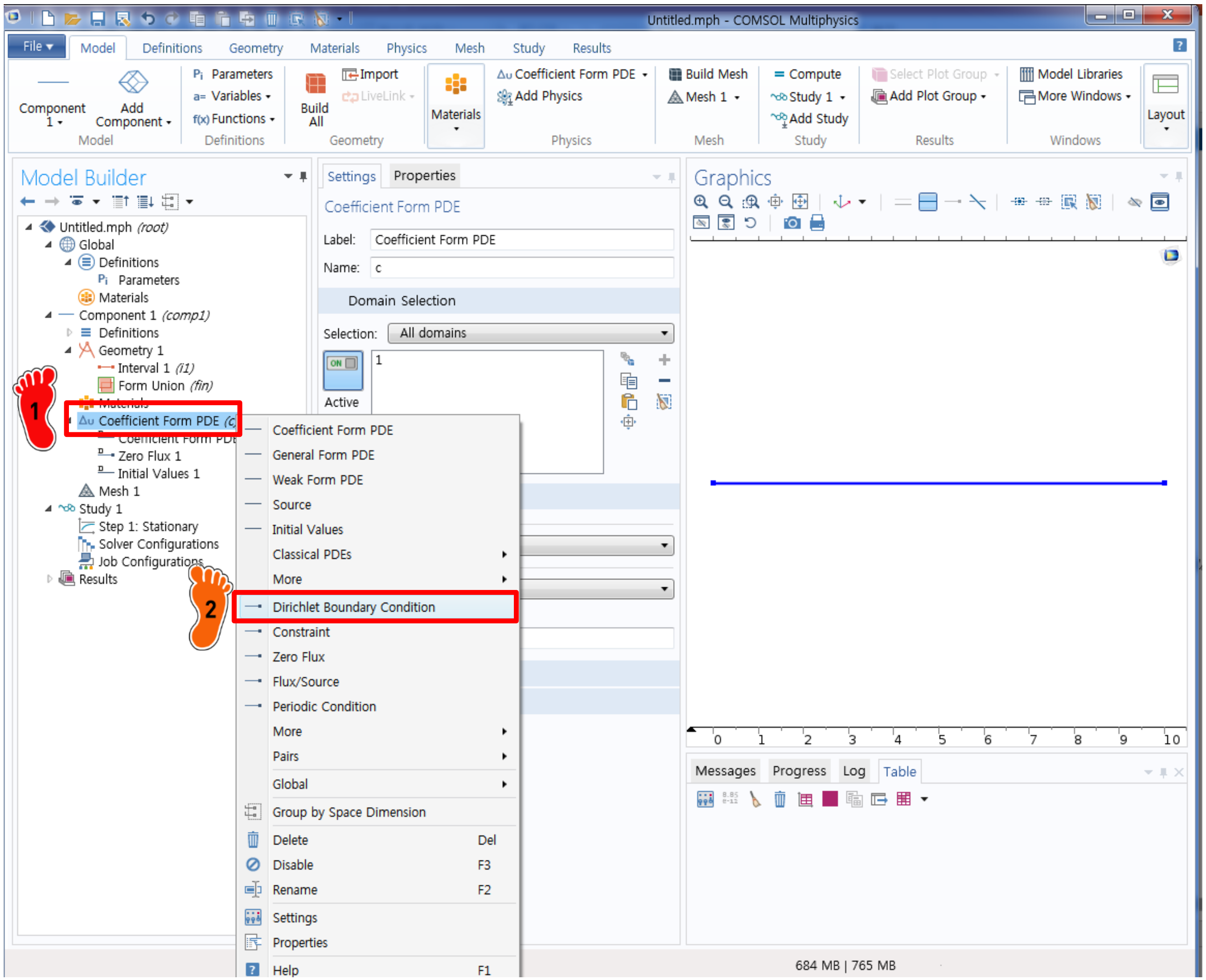
f = -hc*Ta

$$\frac{d^2T}{dx^2} + h'(T_a - T) = 0$$

$$\frac{d^2u}{dx^2} + h'(T_a - u) = 0 \quad (T \to u)$$

$$e\frac{\partial^2 u}{\partial t^2} + d_a\frac{\partial u}{\partial t} + \nabla\cdot\left(-c\nabla u - \alpha u + \gamma\right) + \beta\cdot\nabla u + au = f \leftrightarrow \frac{d^2u}{dx^2} - h'u = -h'T_a$$

BOUNDARY CONDITION

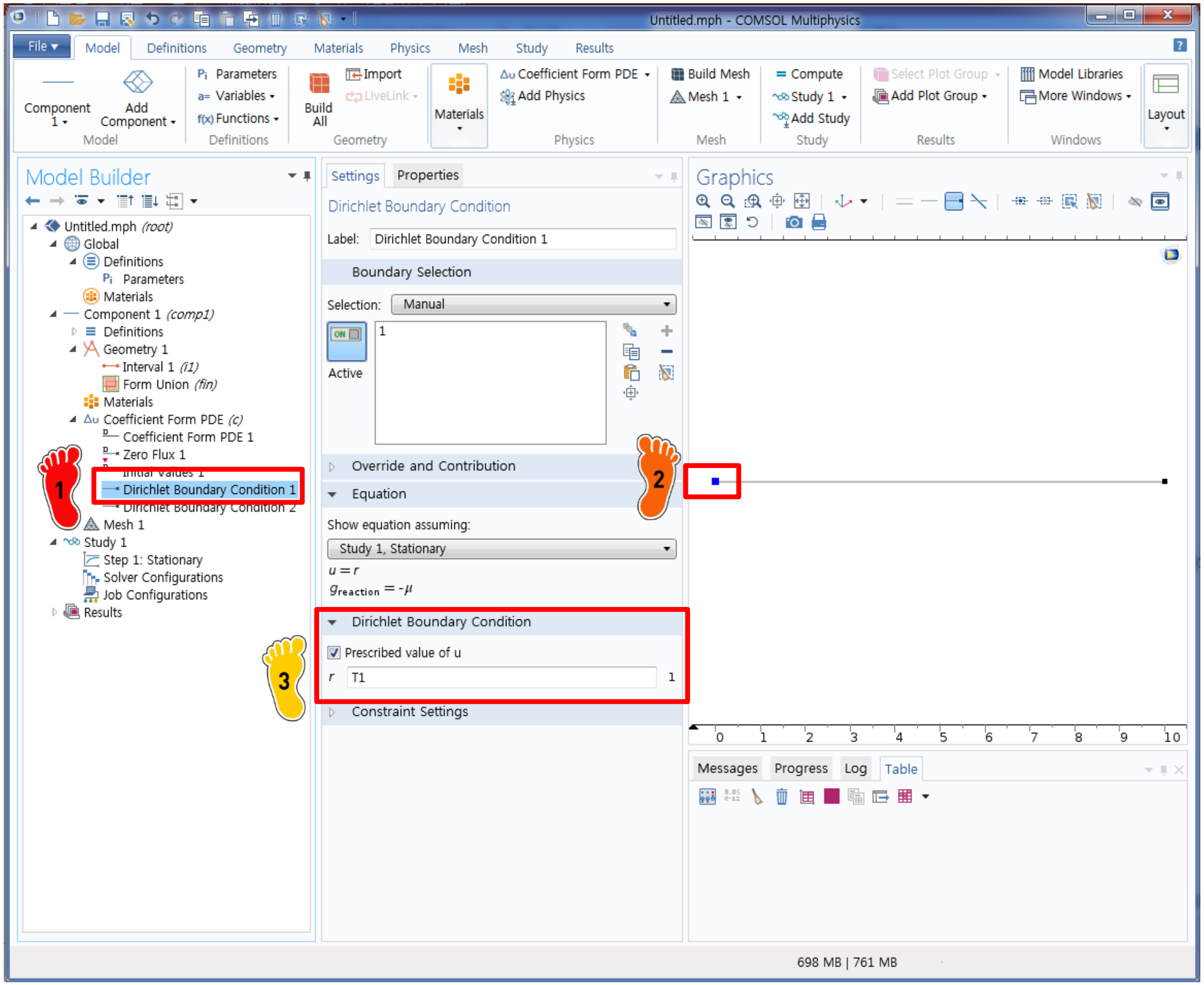


1 Coefficient Form PDE 마우스 우클릭

2 Dirichlet Boudary Condition 선택

2개 생성

BOUNDARY CONDITION



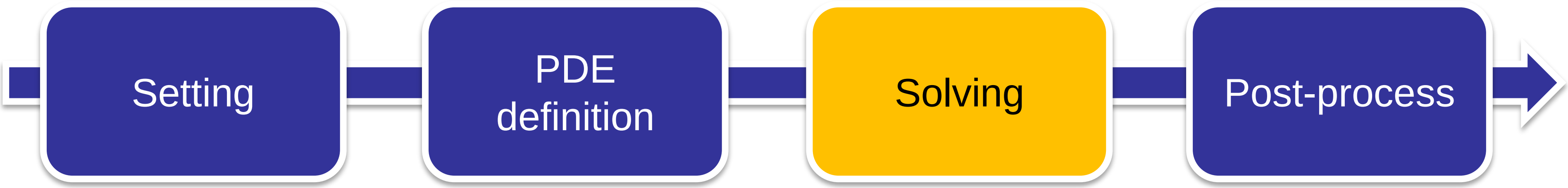
1 Dirichlet Boudary Condition 1 선택

2 x=0인 점 선택

3 경계조건 T(0)값 T1을 입력

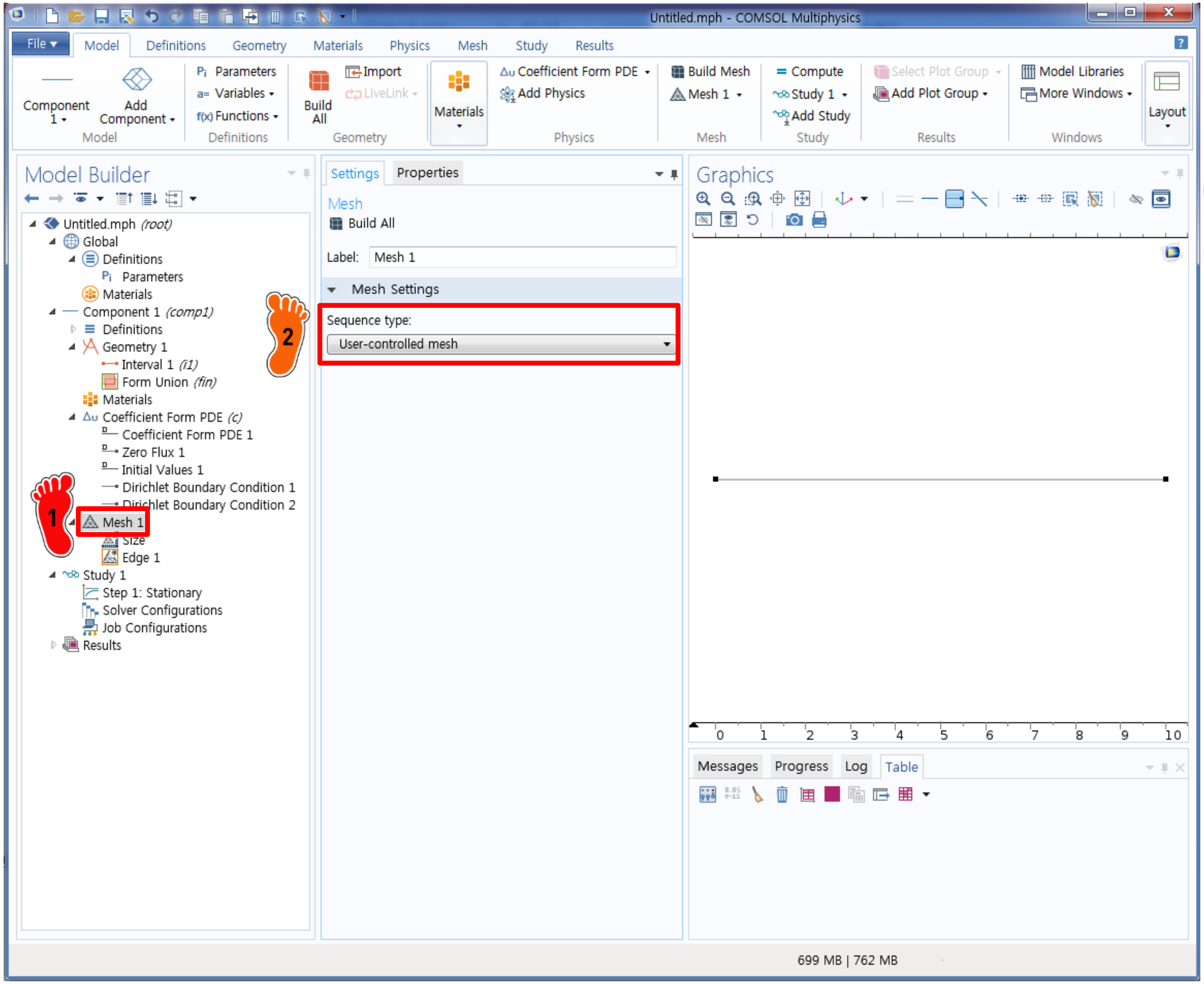
Dirichlet Boudary Condition 2를 이용하여 x=10 인 점의 경계 조건 입력

ANALYSIS FLOW



- ✓ Mesh creation
- ✓ Compute

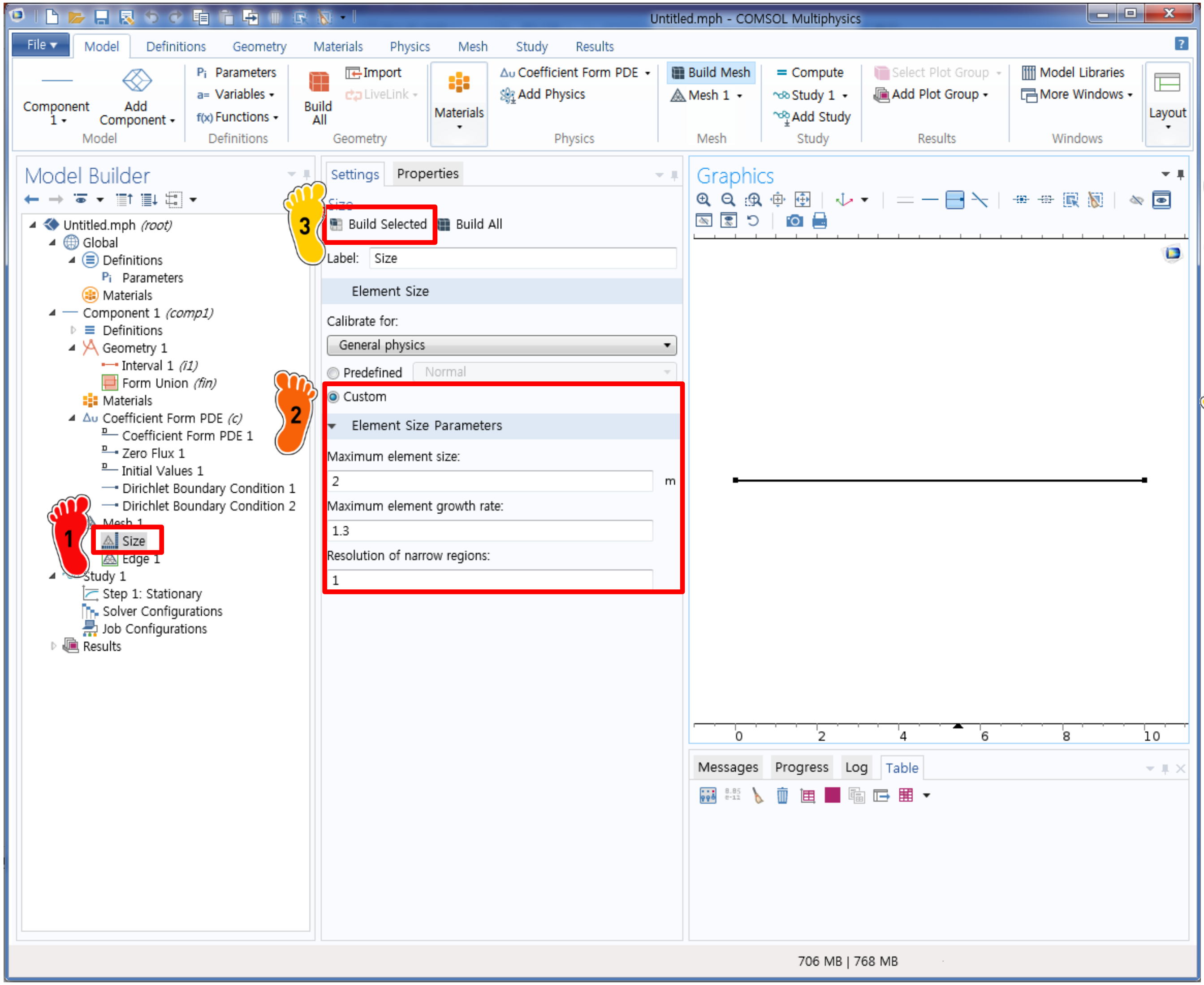
MESH CREATION



1. mesh 1 선택

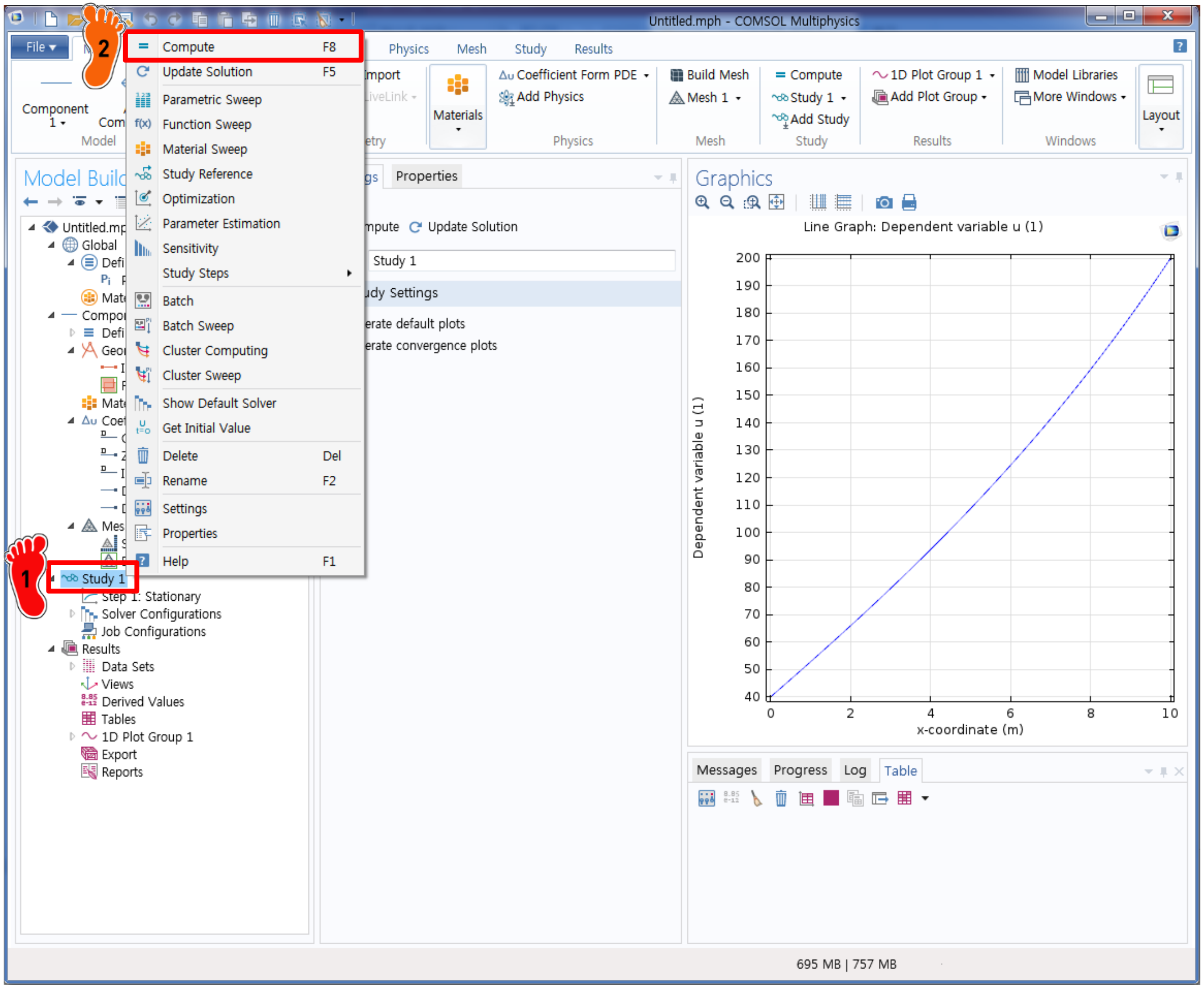
2. Mesh Setting에서 User-controlled mesh 선택

MESH CREATION



1. Size 선택
2. Custom 선택 후 Maximum element size에 2 입력
3. Build Selected 클릭

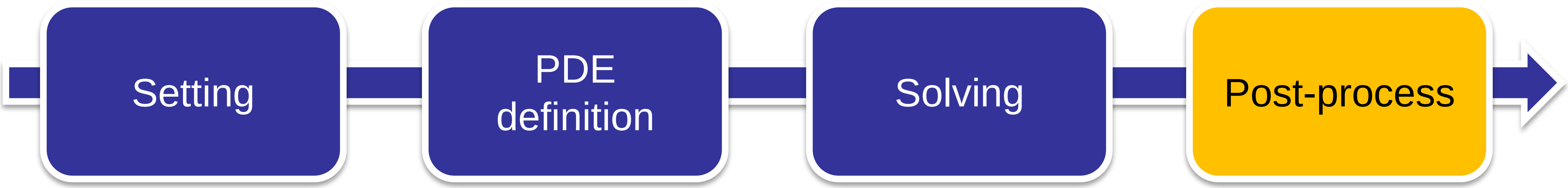
COMPUTE



1 Study 마우스 우클릭

2 Compute를 클릭

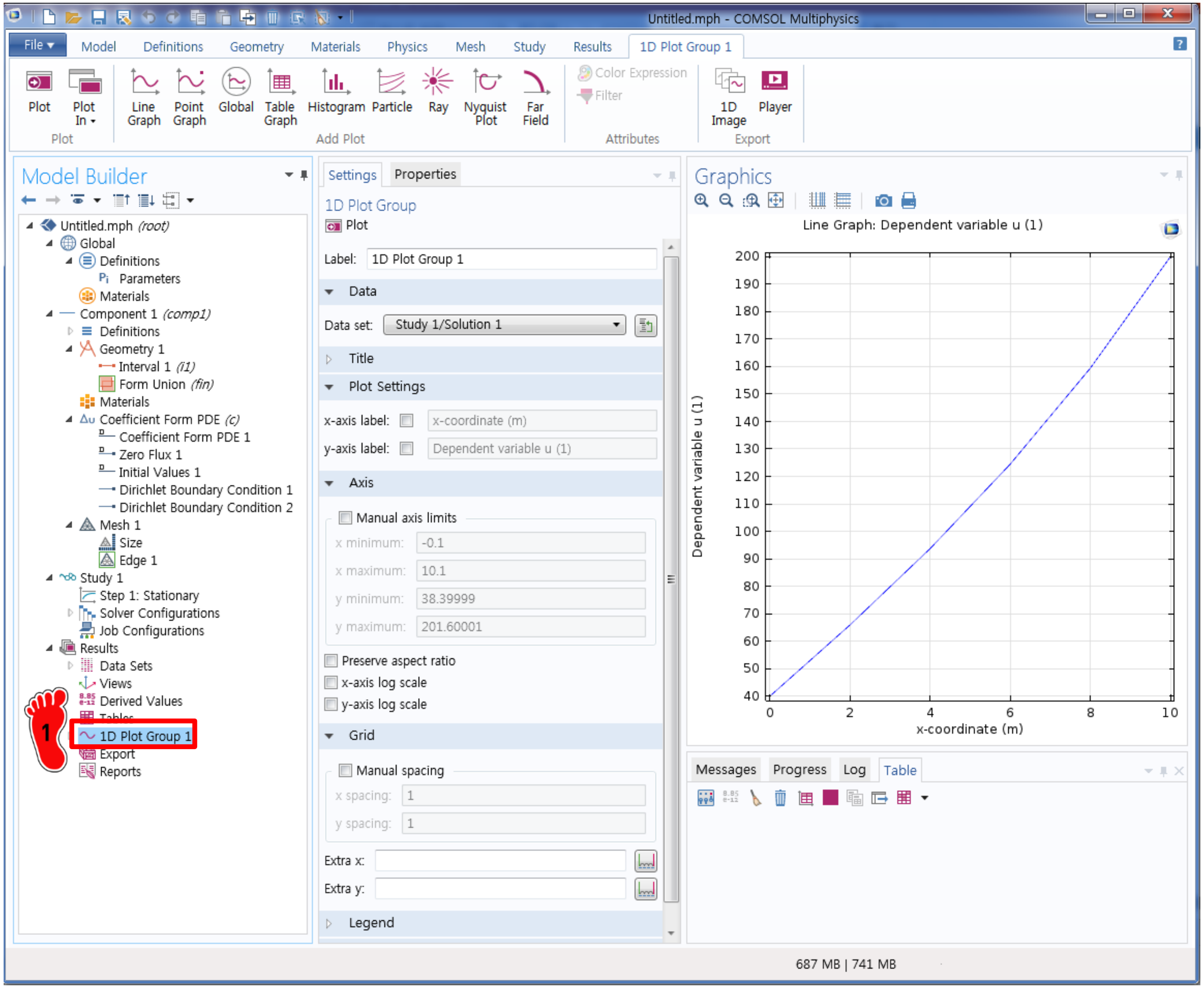
ANALYSIS FLOW

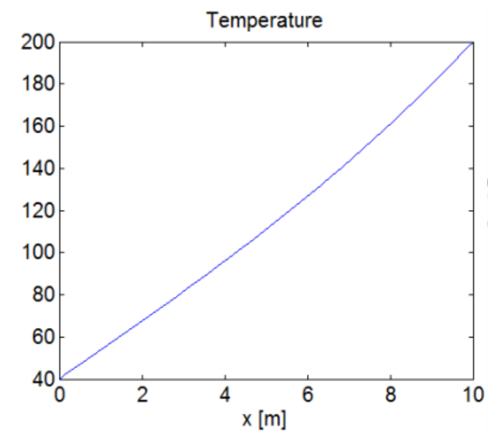


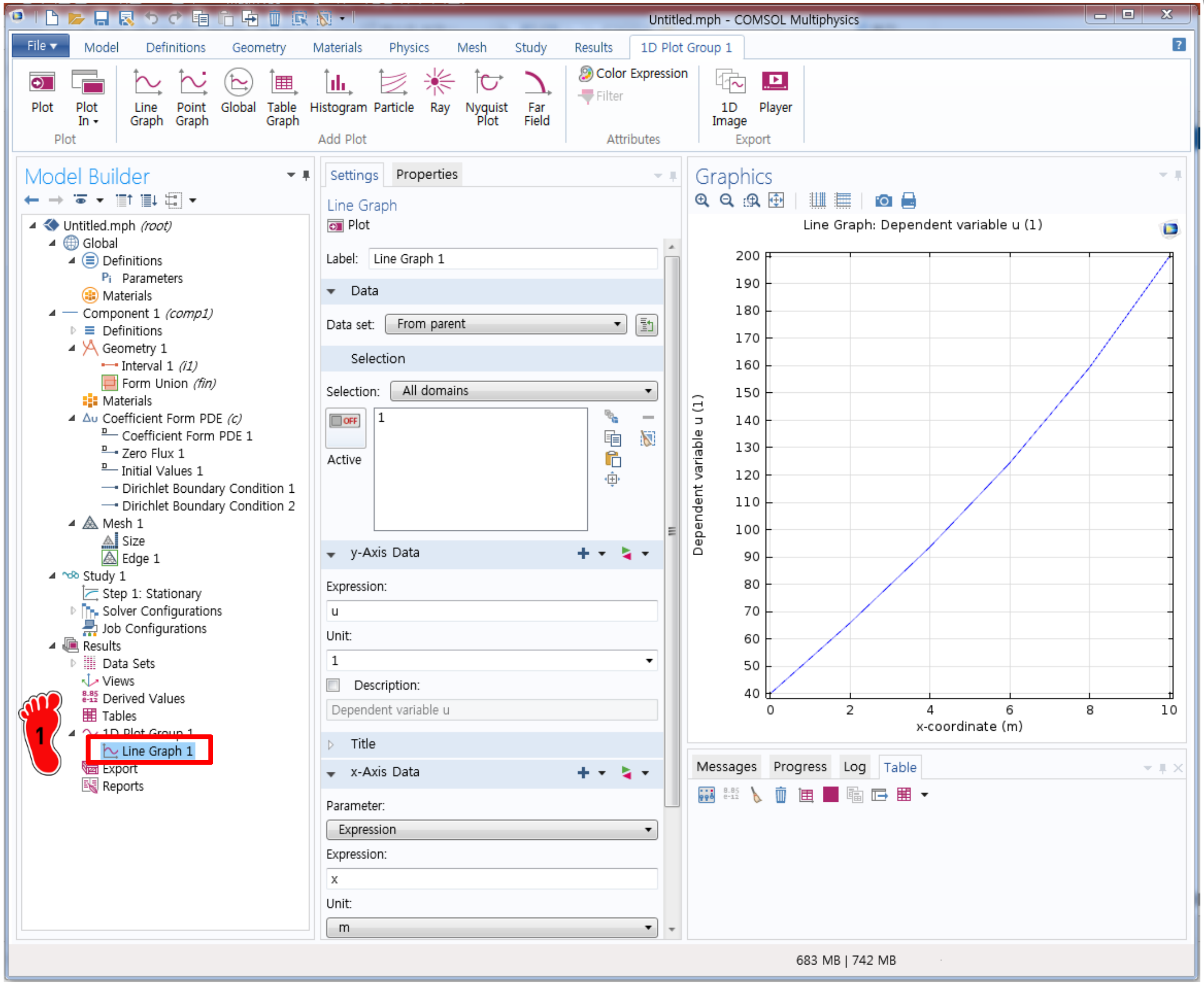
✓ Result plot

RESULT PLOT

1 Result 아래에 생성된 1D Plot Group 1을 클릭하여 결과 확인





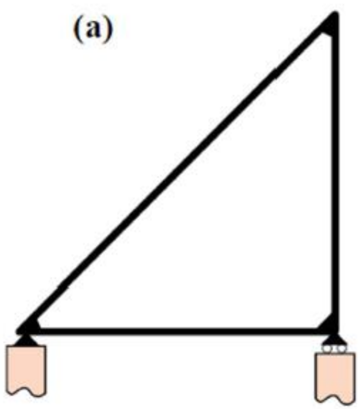


1 Line Graph1 클릭

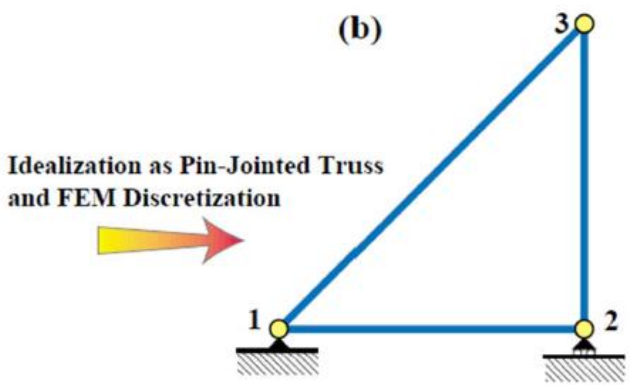
Setting 창에서 설정을 다르게 하여 그래프를 변경 할 수 있다.

- **COMSOL analysis: Truss structure**
  - ✓ **<u>Geometry creation</u>**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **Nodal force**
  - ✓ **Analysis**

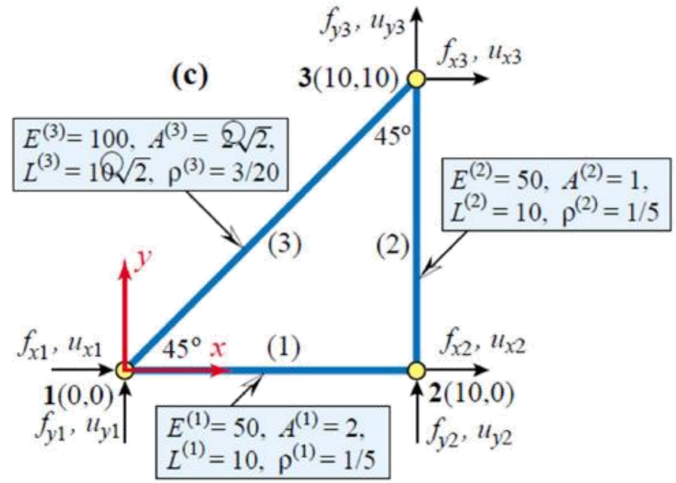
EXAMPLE TRUSS STRUCTURE



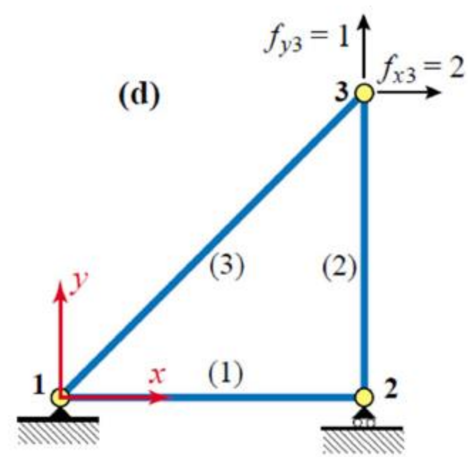
Physical structure



Idealization as a pin-jointed bar assemblage

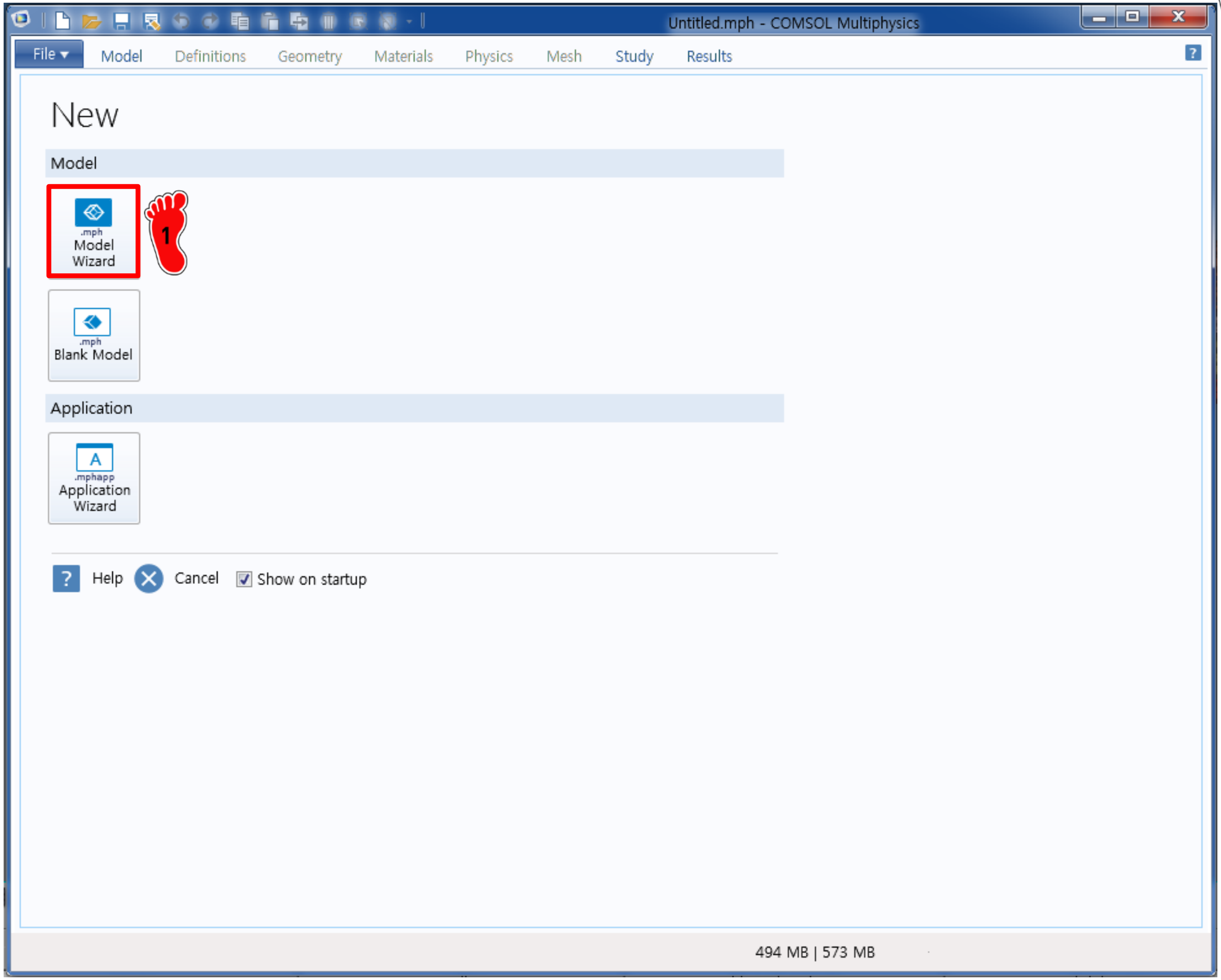


Geometric, material and fabrication properties



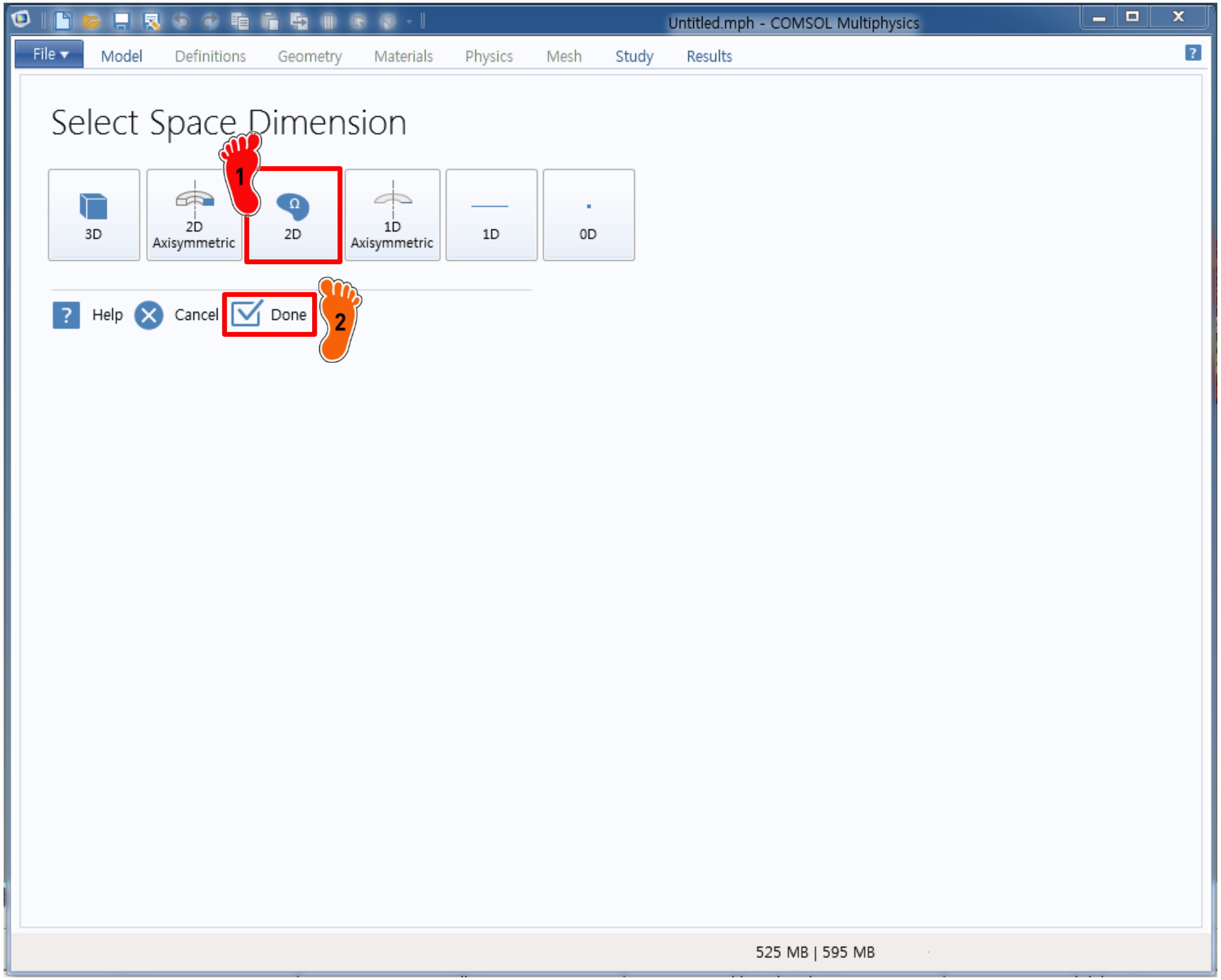
Support conditions and applied loads

MODEL WIZARD



1 Model Wizard 선택

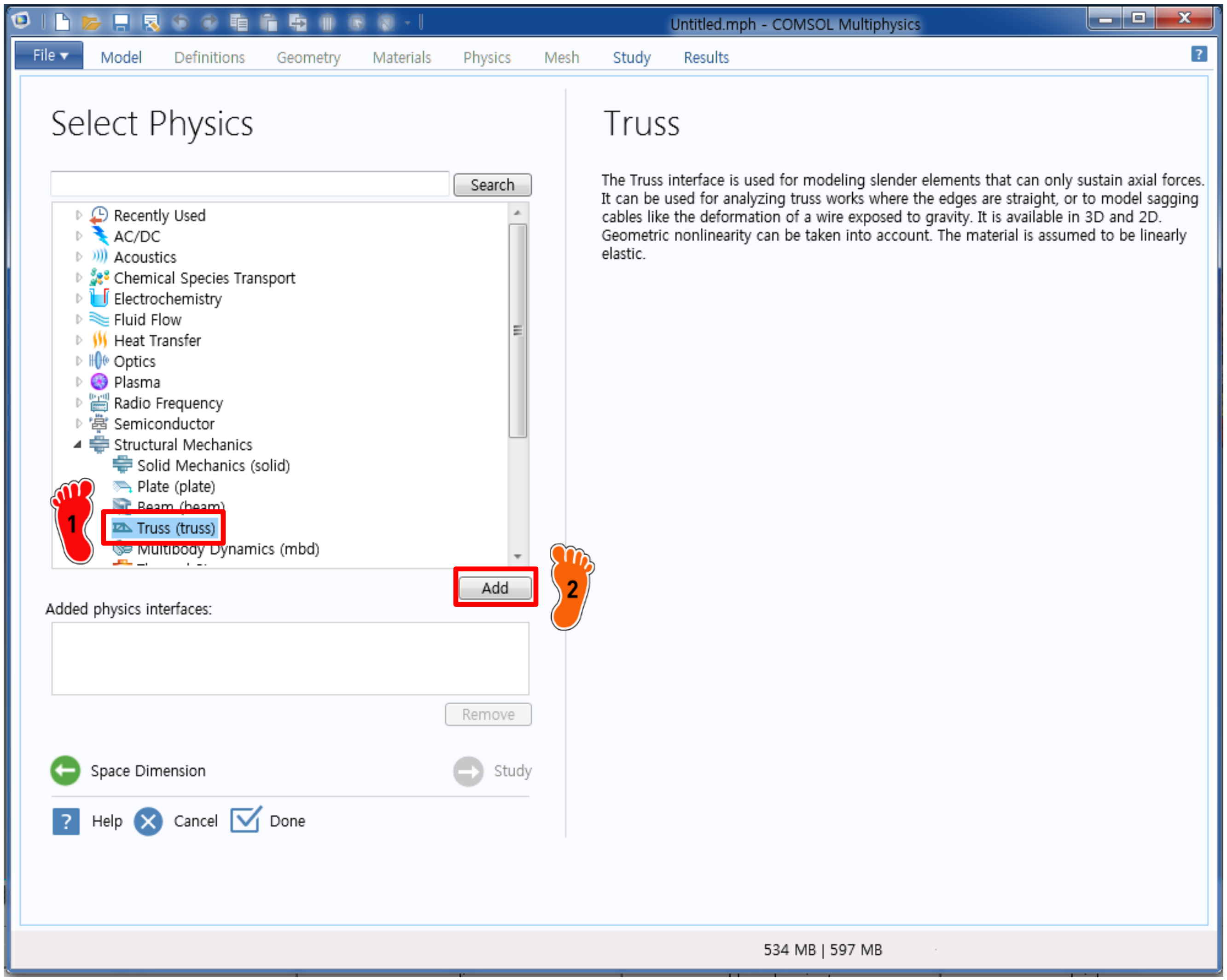
DIMENSION SELECTION



1 2D 선택

2 Done 클릭

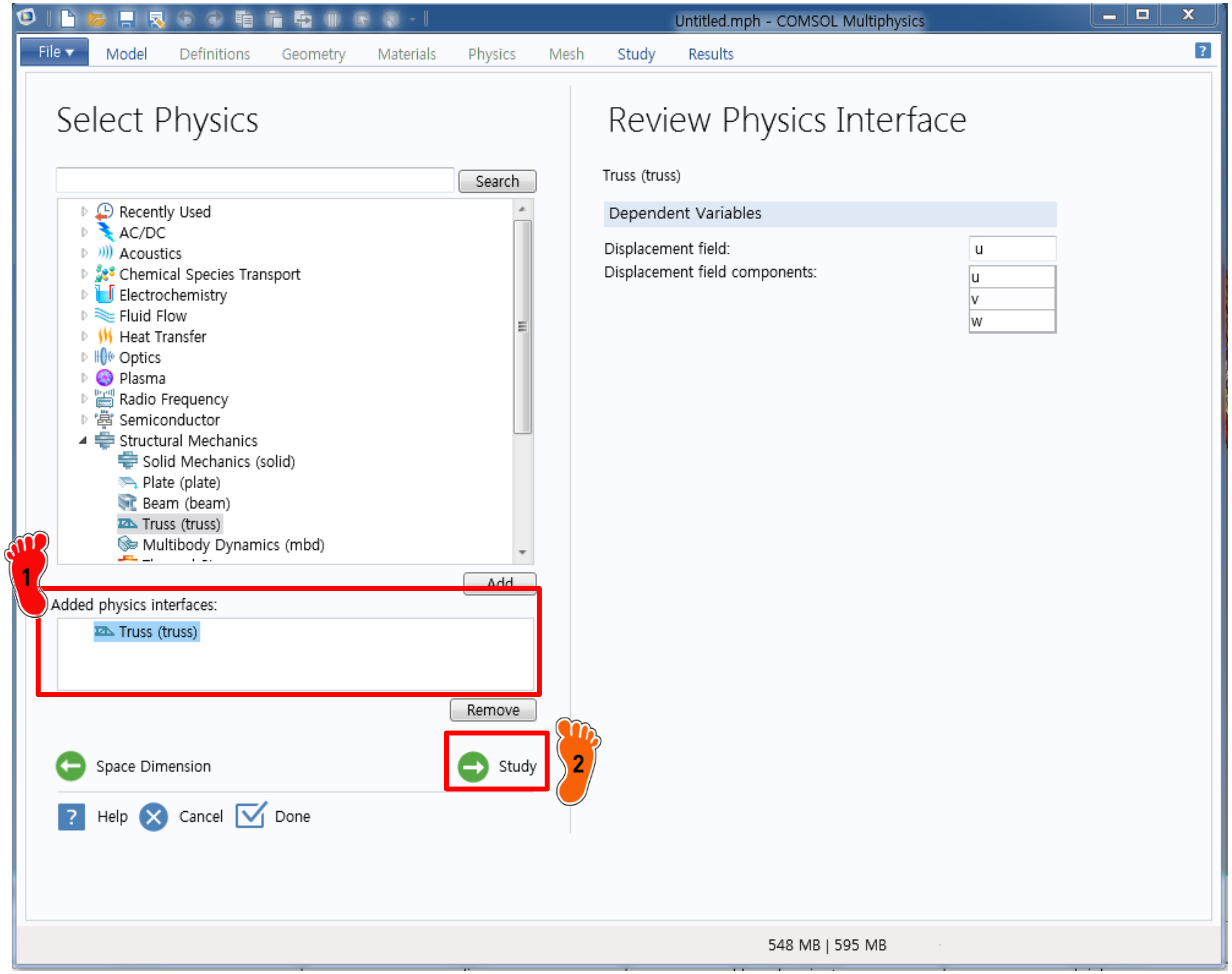
PHYSICS SELECTION



1 Structural Mechanics 모듈의 Truss physics 선택

2 Add 클릭

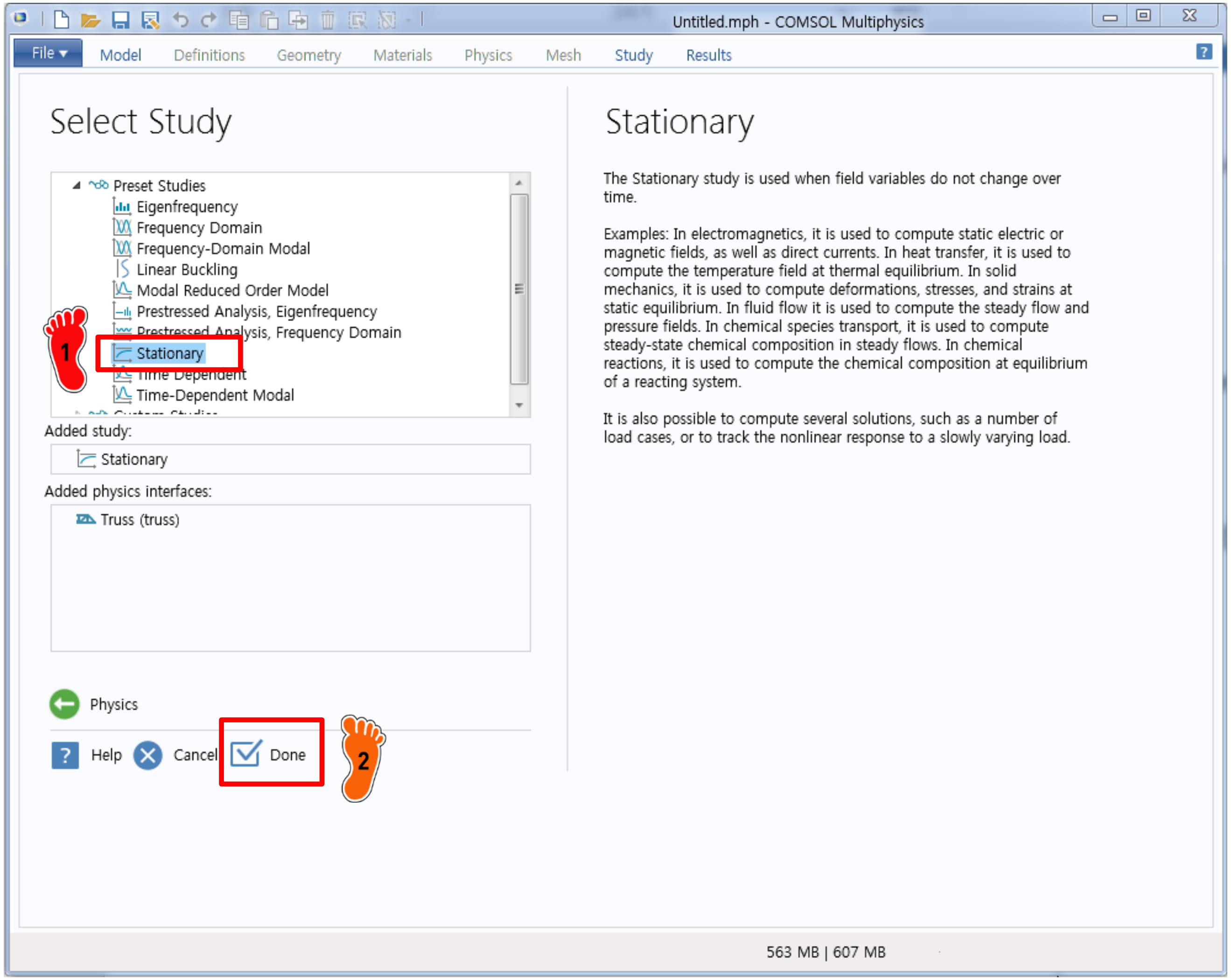
PHYSICS SELECTION



1 Added physics interfaces에서 추가된 physics 확인

2 Study 클릭

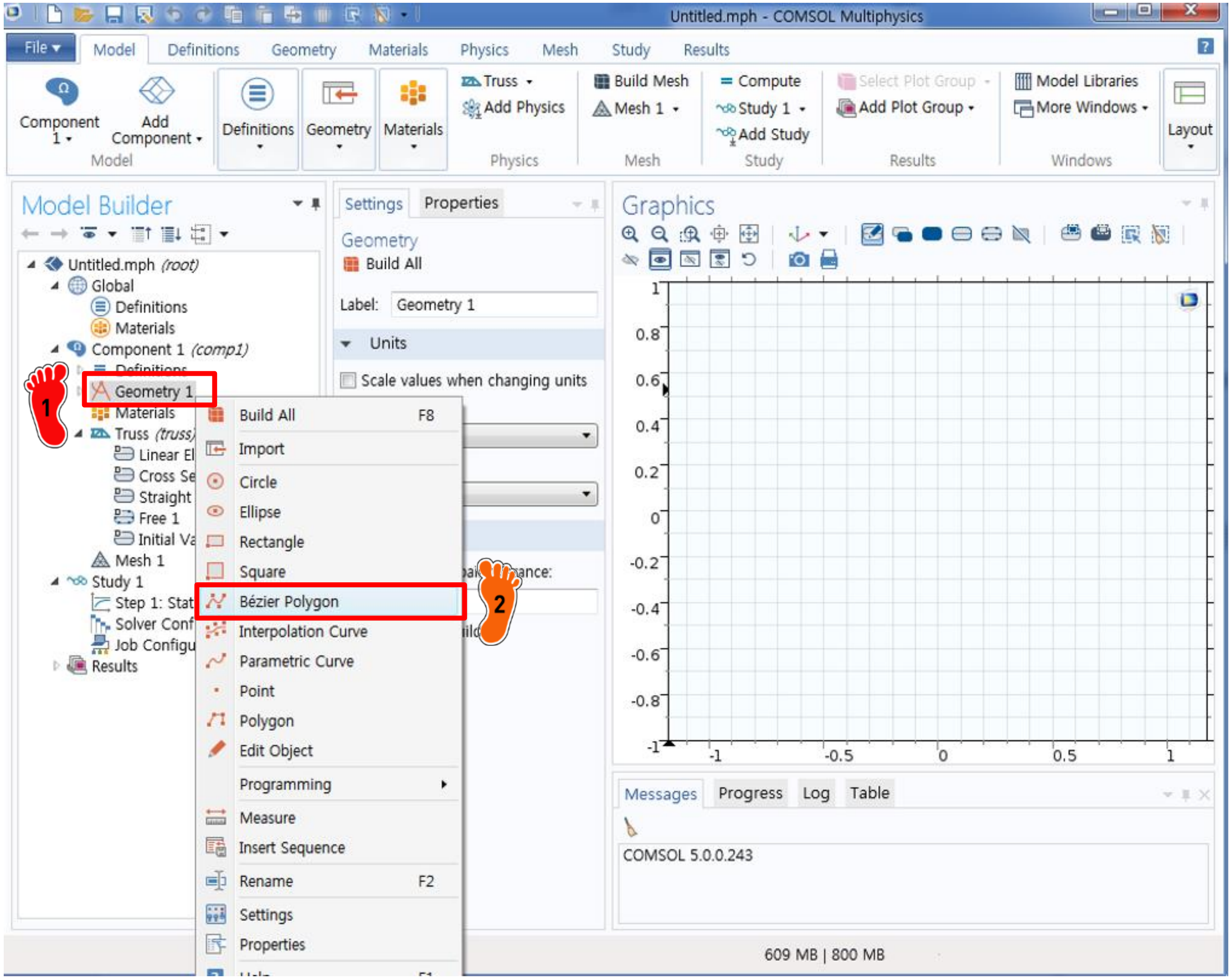
STUDY TYPE SELECTION



1 Preset Studies에서 Stationary 선택

2 Done 클릭

GEOMETRY CREATION

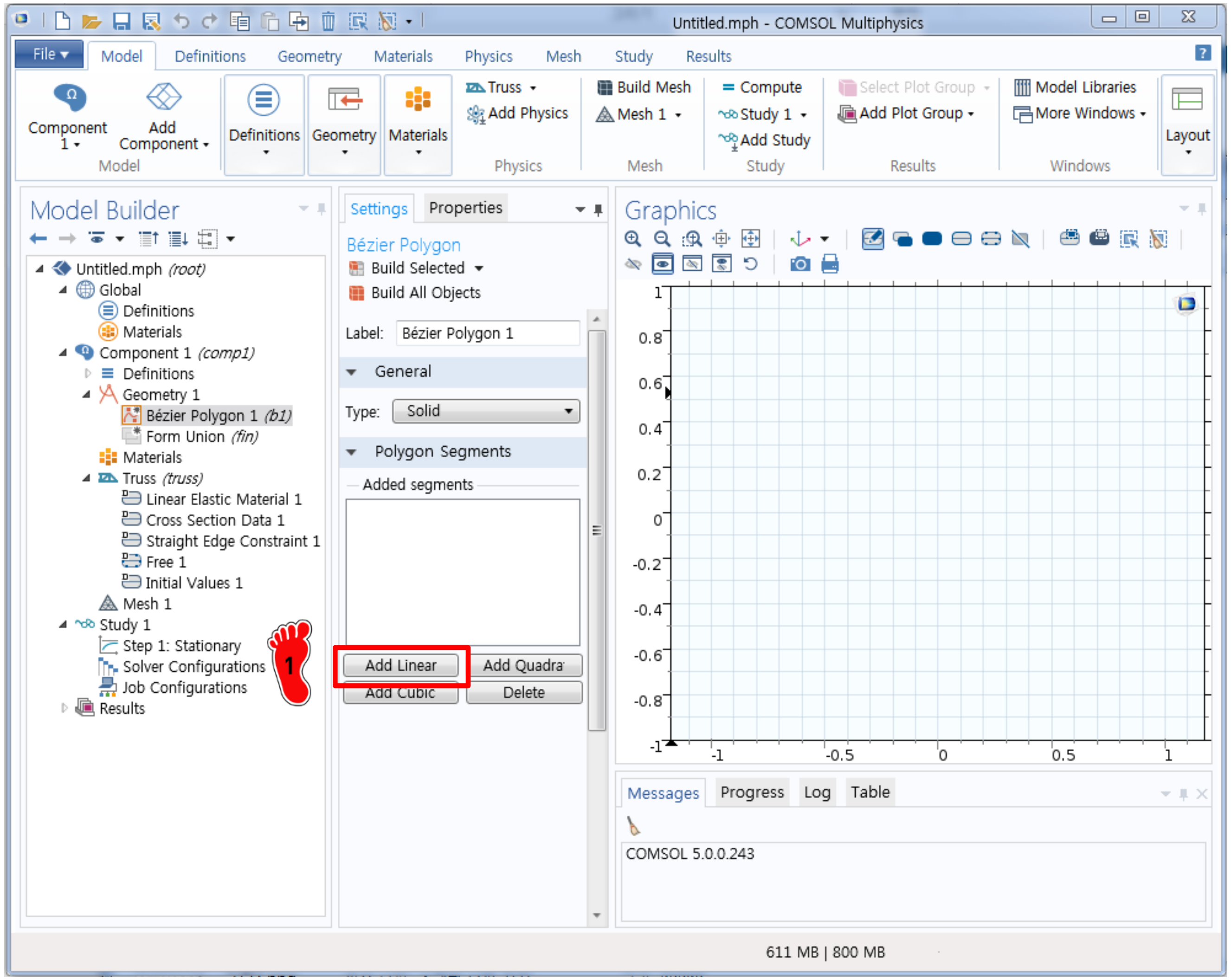


1 Geometry 1 마우스 우클릭

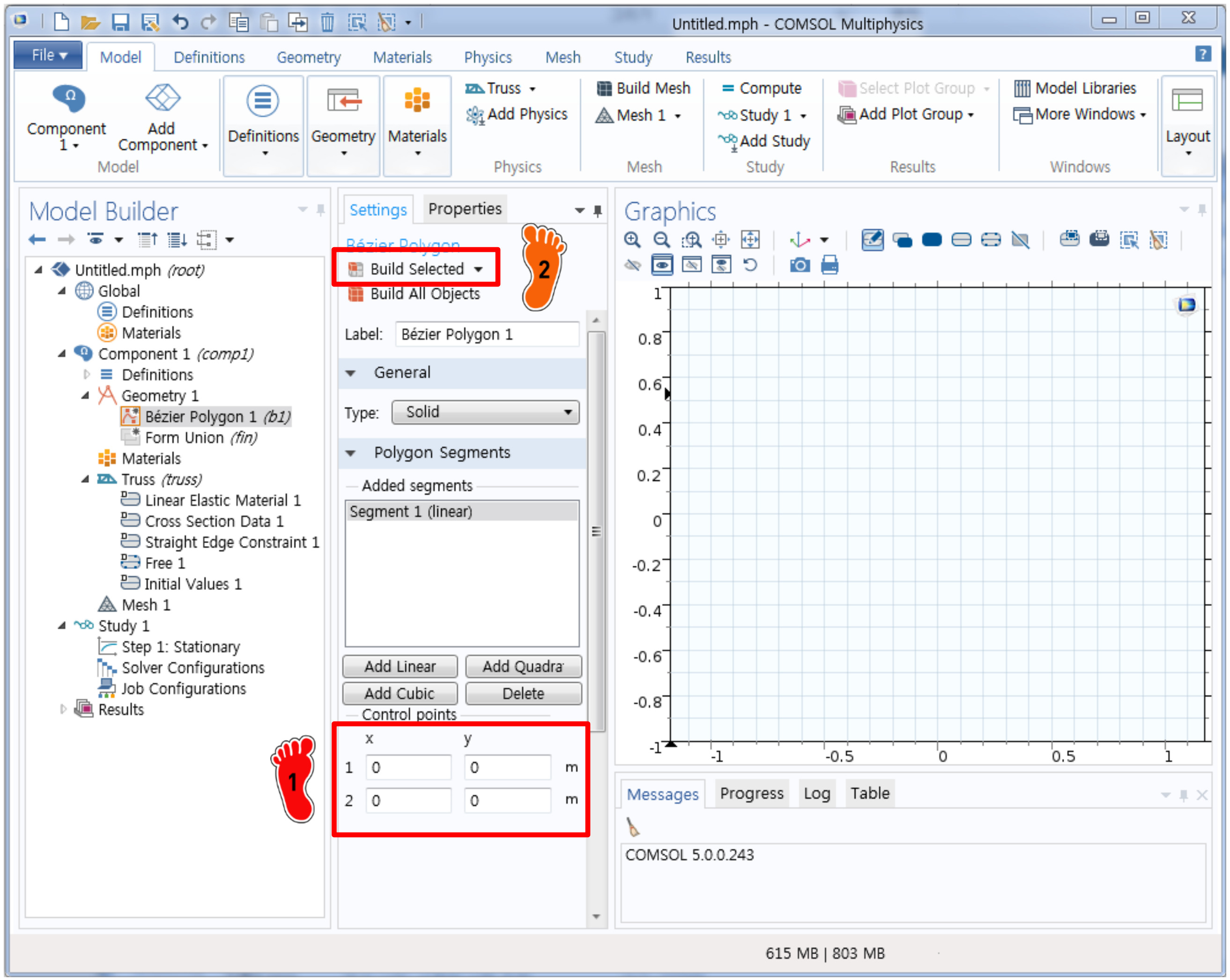
2 Bezier Polygon 클릭

GEOMETRY CREATION

1 Add Linear 클릭



GEOMETRY CREATION

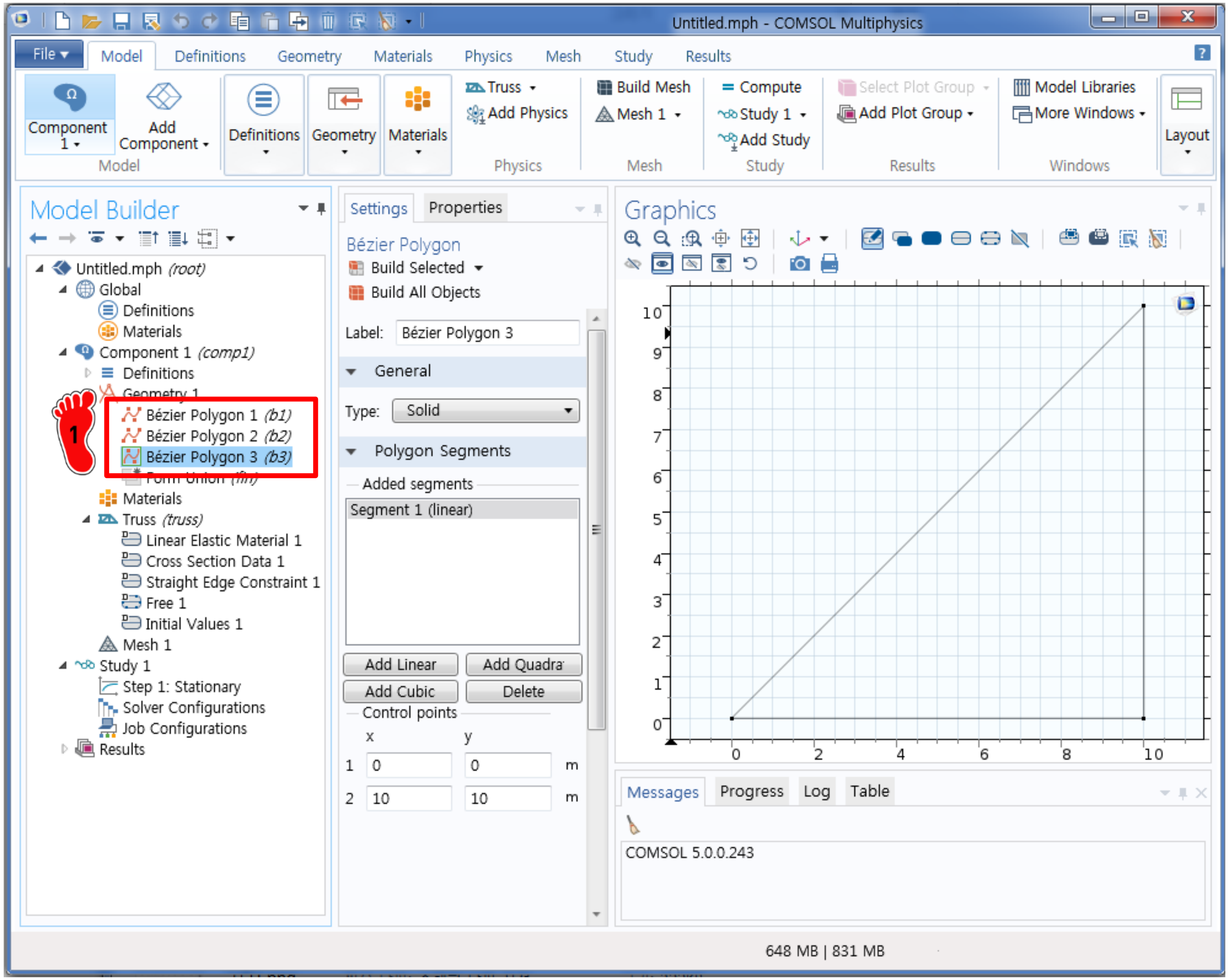


Control points 시작점 (0,0) 끝나는 점 (10,0) 을 입력

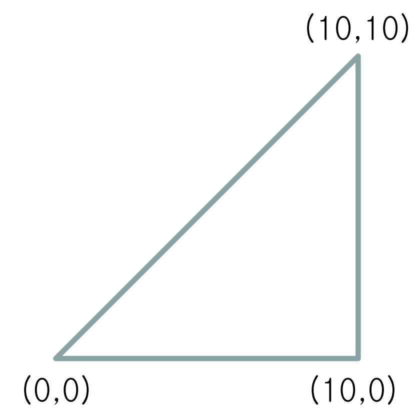
옆의 숫자 1 은 시작점 좌표, 2는 끝나는 점을 의미

Build Selected 클릭

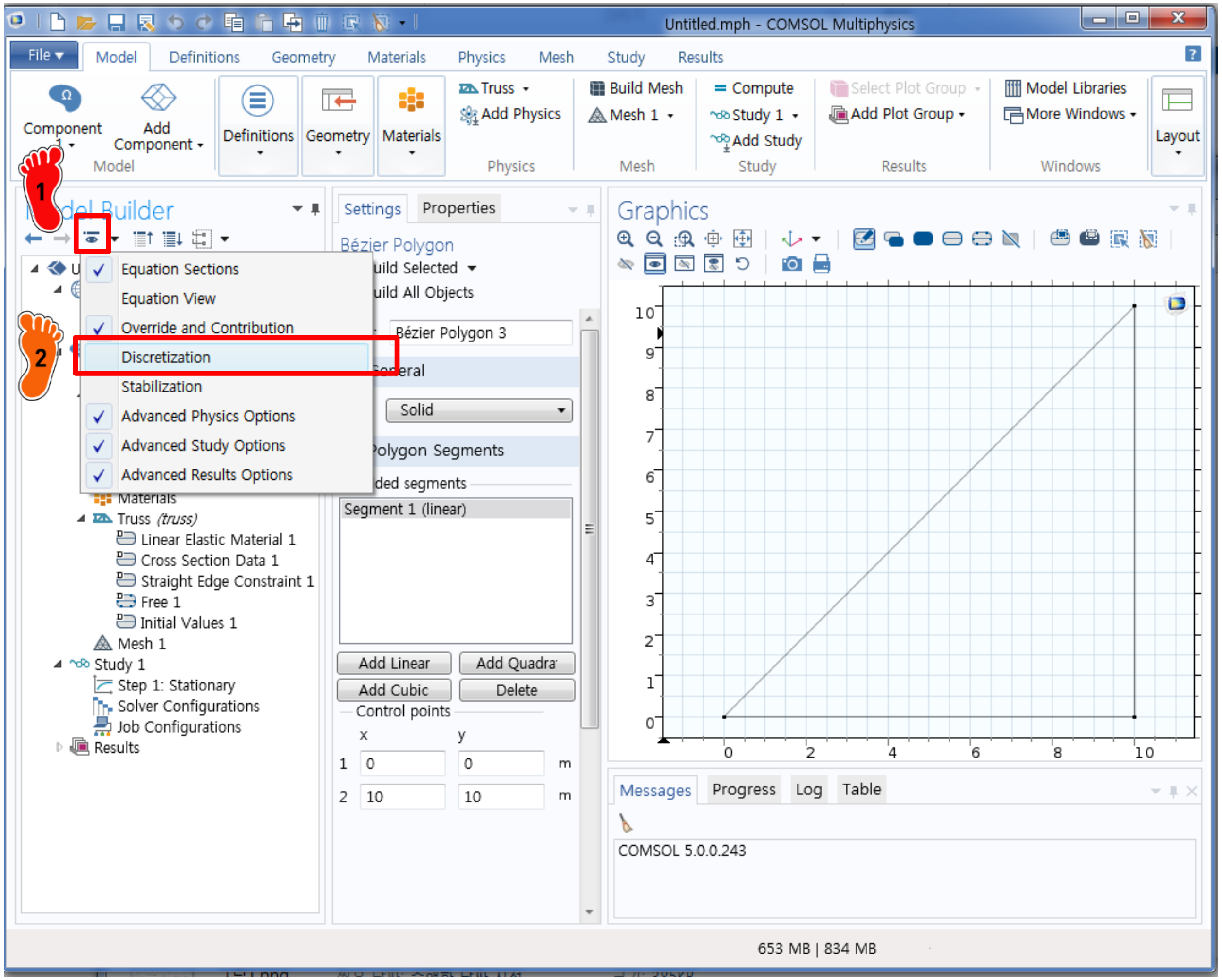
GEOMETRY CREATION



1 같은 방식으로 총 3개의 직선을 생성



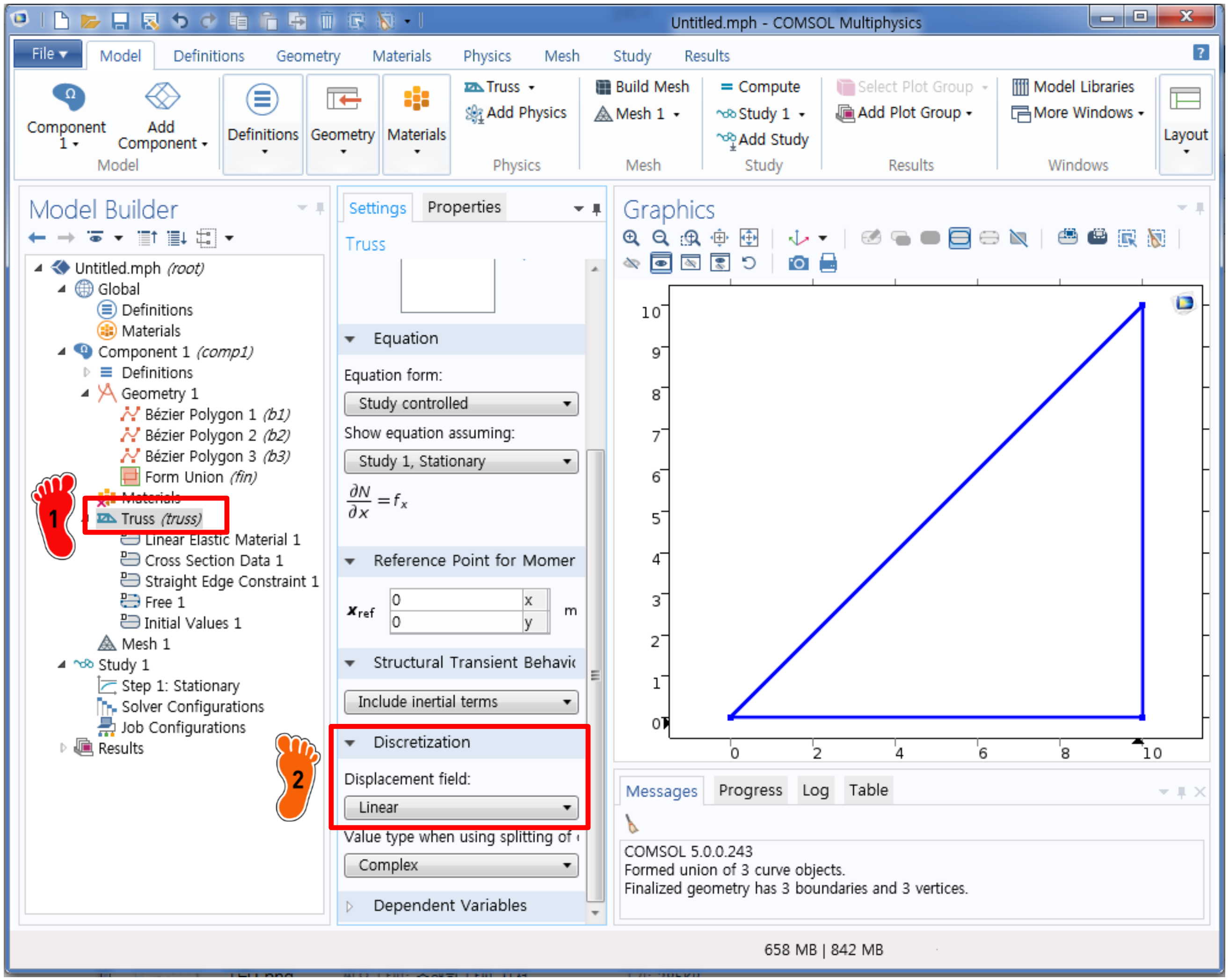
ELEMENT ORDER CHANGING



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

- **COMSOL analysis: Truss structure**

  ✓ **Geometry creation**

  ✓ **Material property**

  ✓ **Cross section property**

  ✓ **Boundary condition**

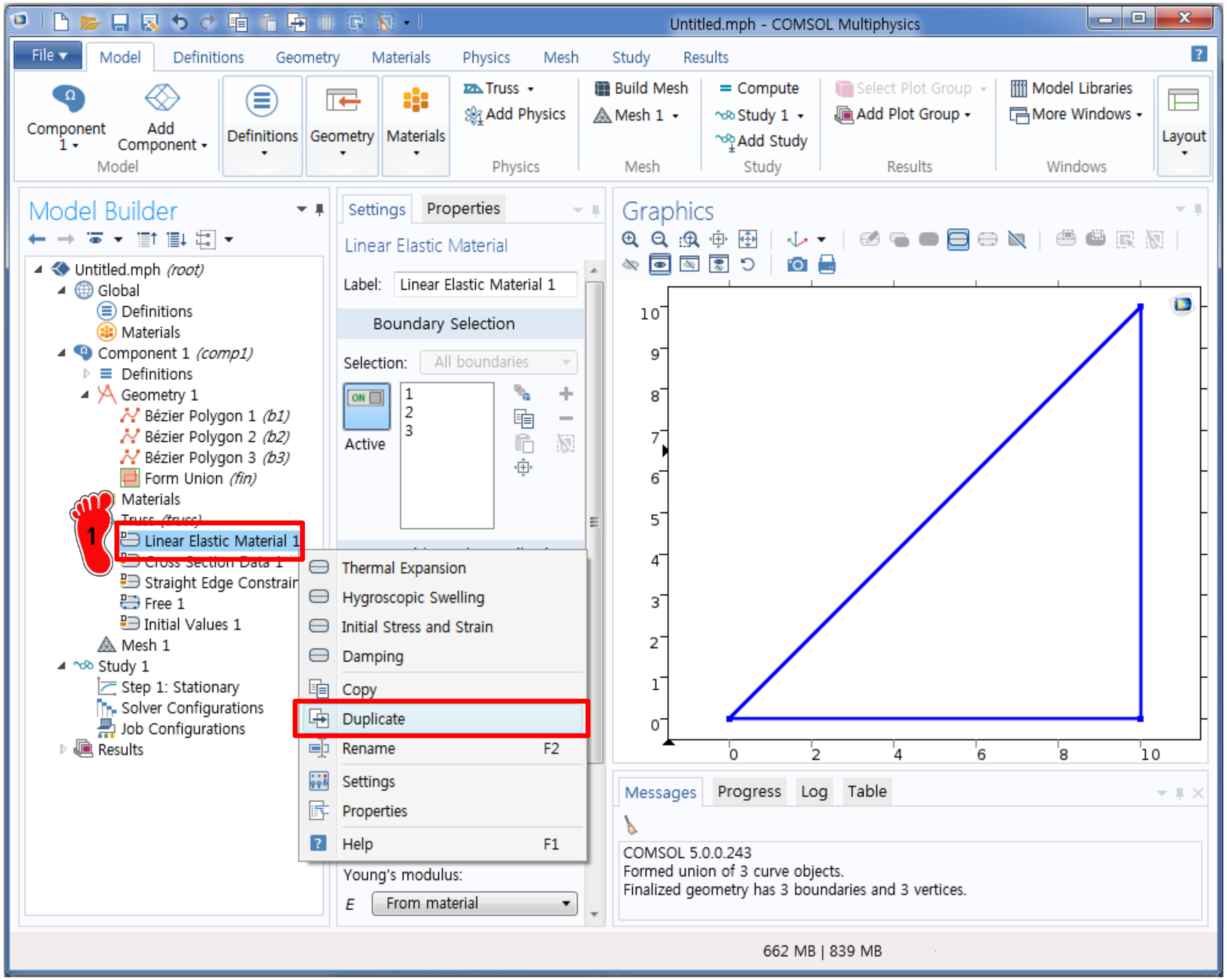
  ✓ **Nodal force**

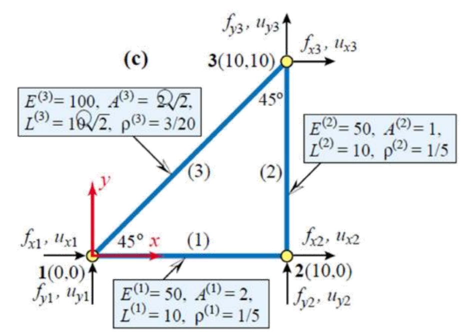
  ✓ **Analysis**

MATERIAL PROPERTY

1 Linear Elastic Material 마우스 우클릭

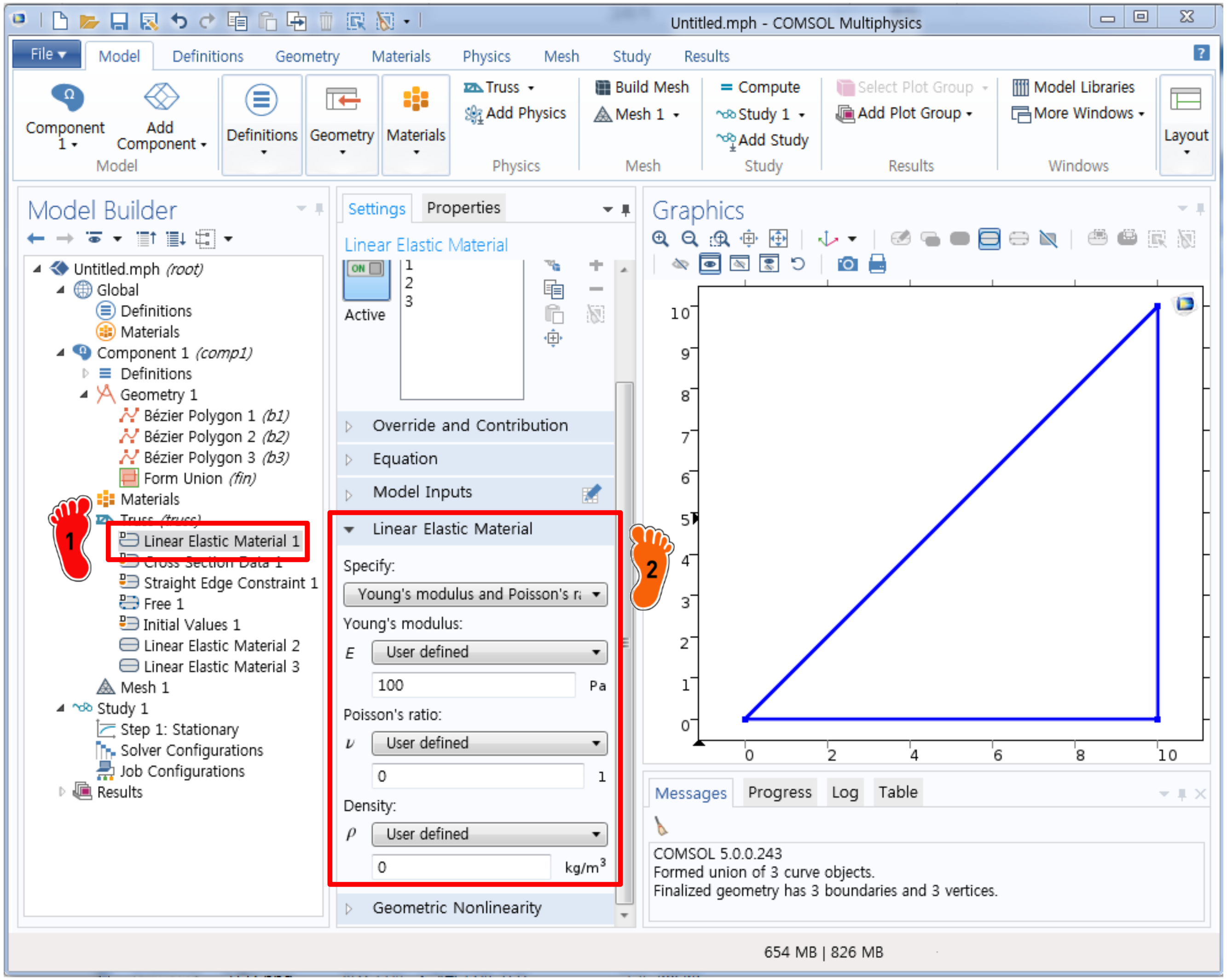
2 Duplicate 를 이용하여 Linear Elastic Material 메뉴 2 개 더 생성





Geometric, material and fabrication properties

MATERIAL PROPERTY

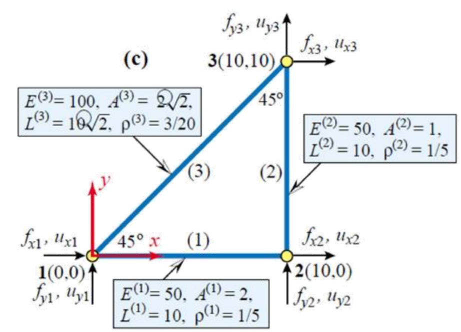


1 첫 번째 Linear Elastic Material 클릭

2 From material을 User defined로 변경 후

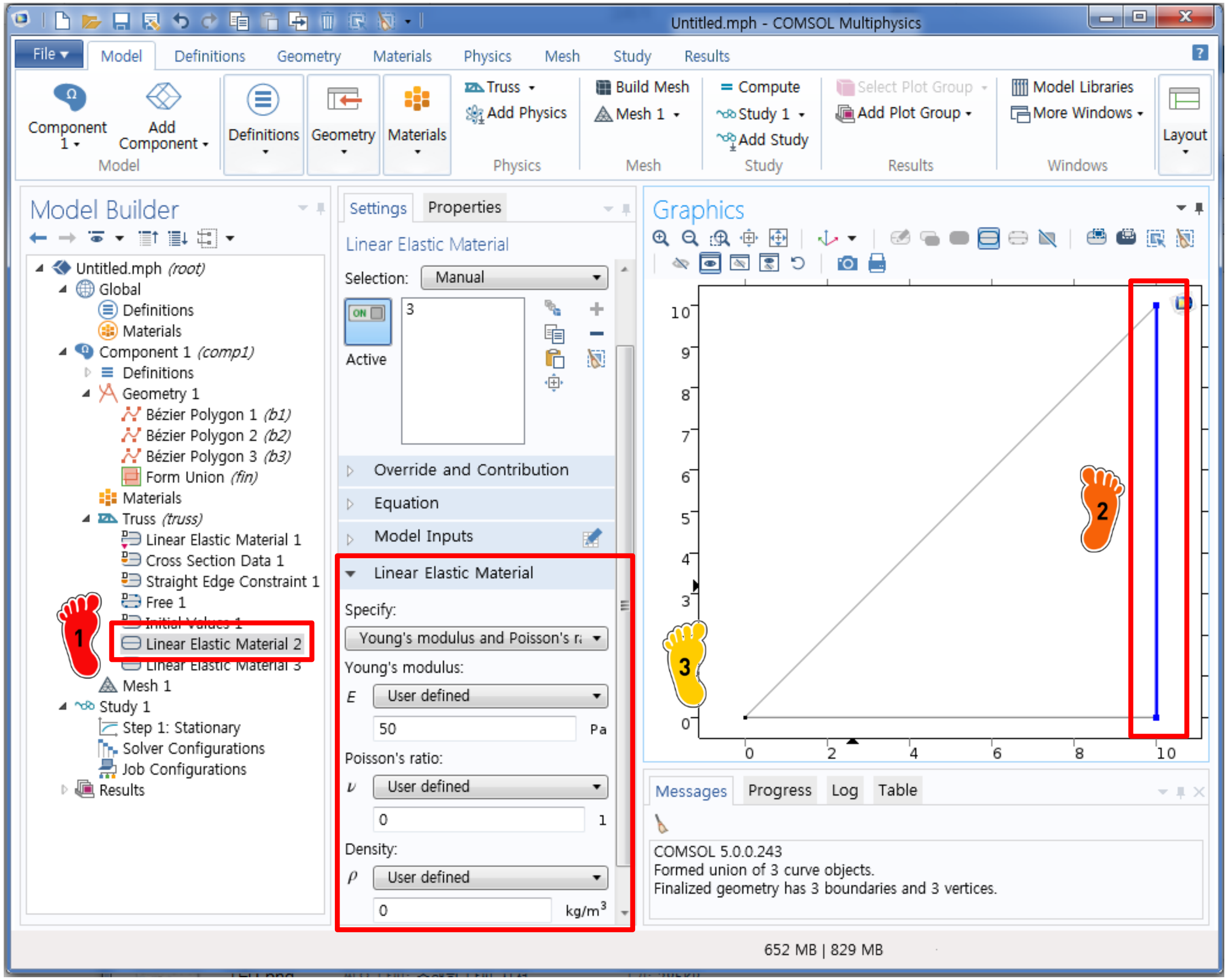
Young`s modulus 100 입력

푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

MATERIAL PROPERTY

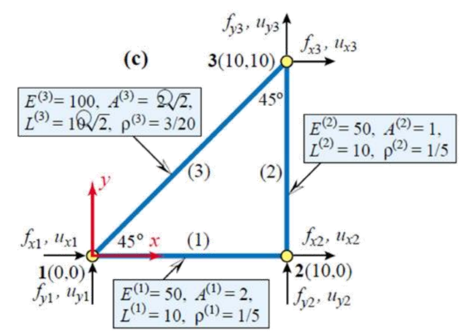


1 두 번째 Linear Elastic Material 클릭

2 제일 오른쪽 부재 선택

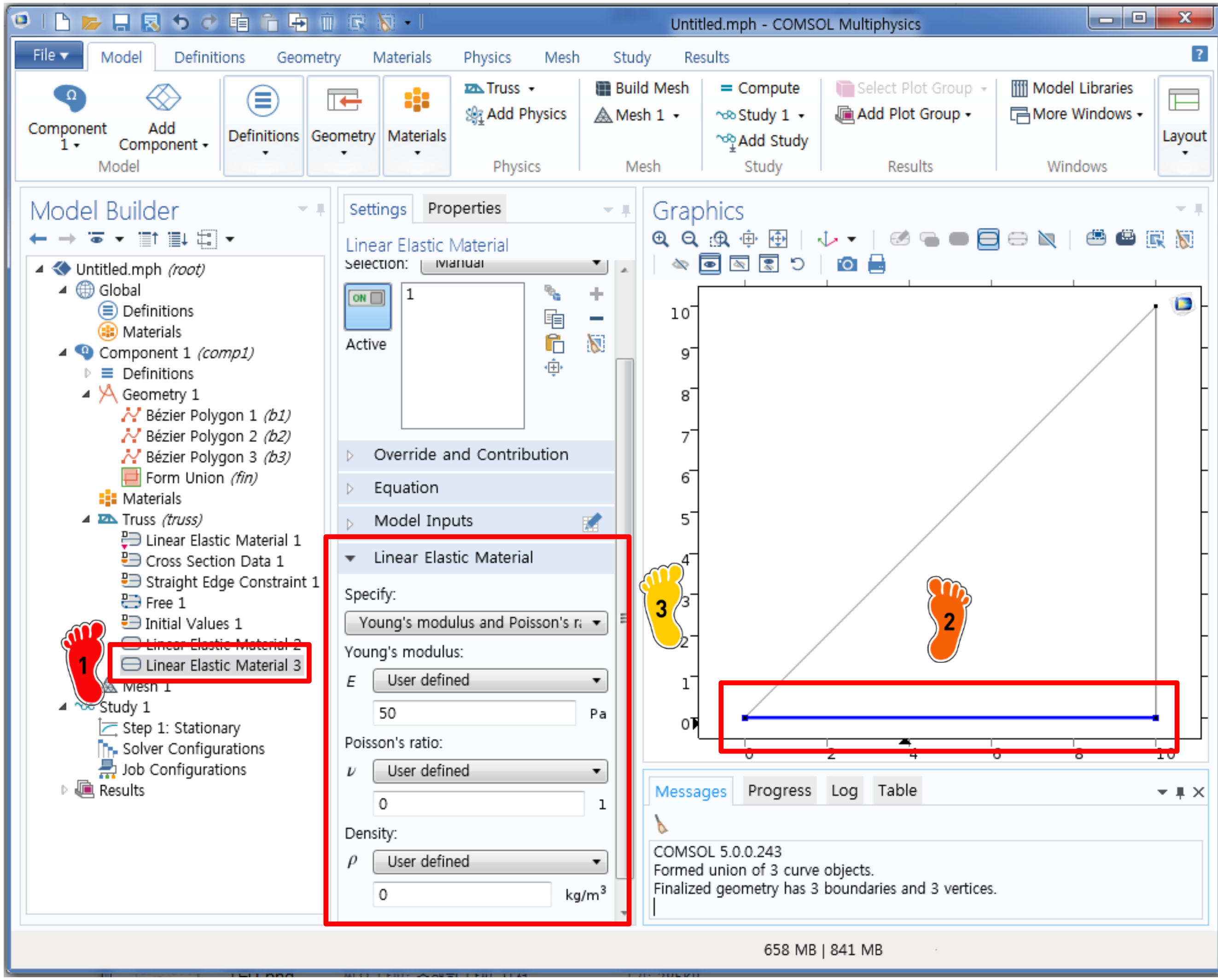
3 Young`s modulus 50 입력

푸아송비와 밀도는 0 입력



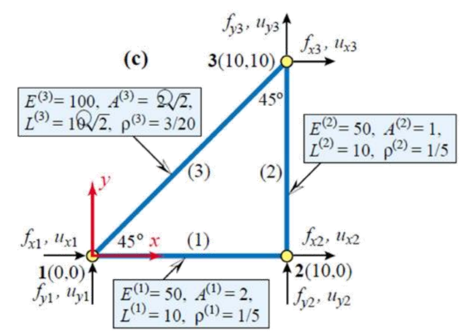
Geometric, material and fabrication properties

MATERIAL PROPERTY



1. 마지막 Linear Elastic Material 클릭
2. 제일 아래 부재 선택
3. Young`s modulus 50 입력

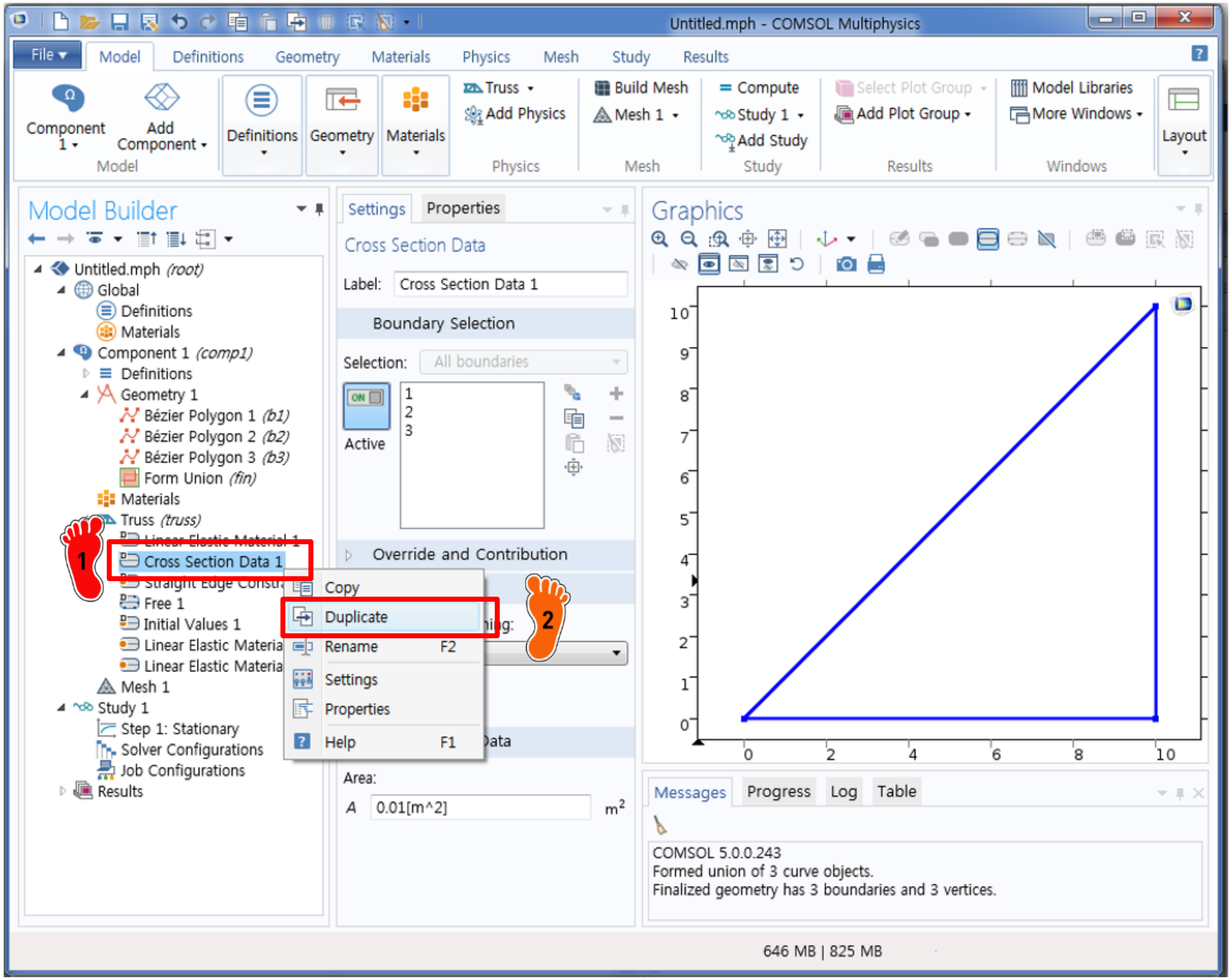
푸아송비와 밀도는 0 입력



Geometric, material and fabrication properties

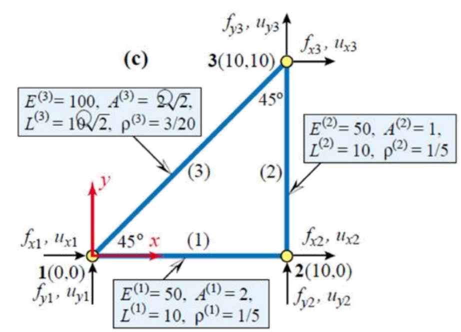
- **COMSOL analysis: Truss structure**
  - ✓ Geometry creation
  - ✓ Material property
  - ✓ Cross section property
  - ✓ Boundary condition
  - ✓ Nodal force
  - ✓ Analysis

SECTION PROPERTY



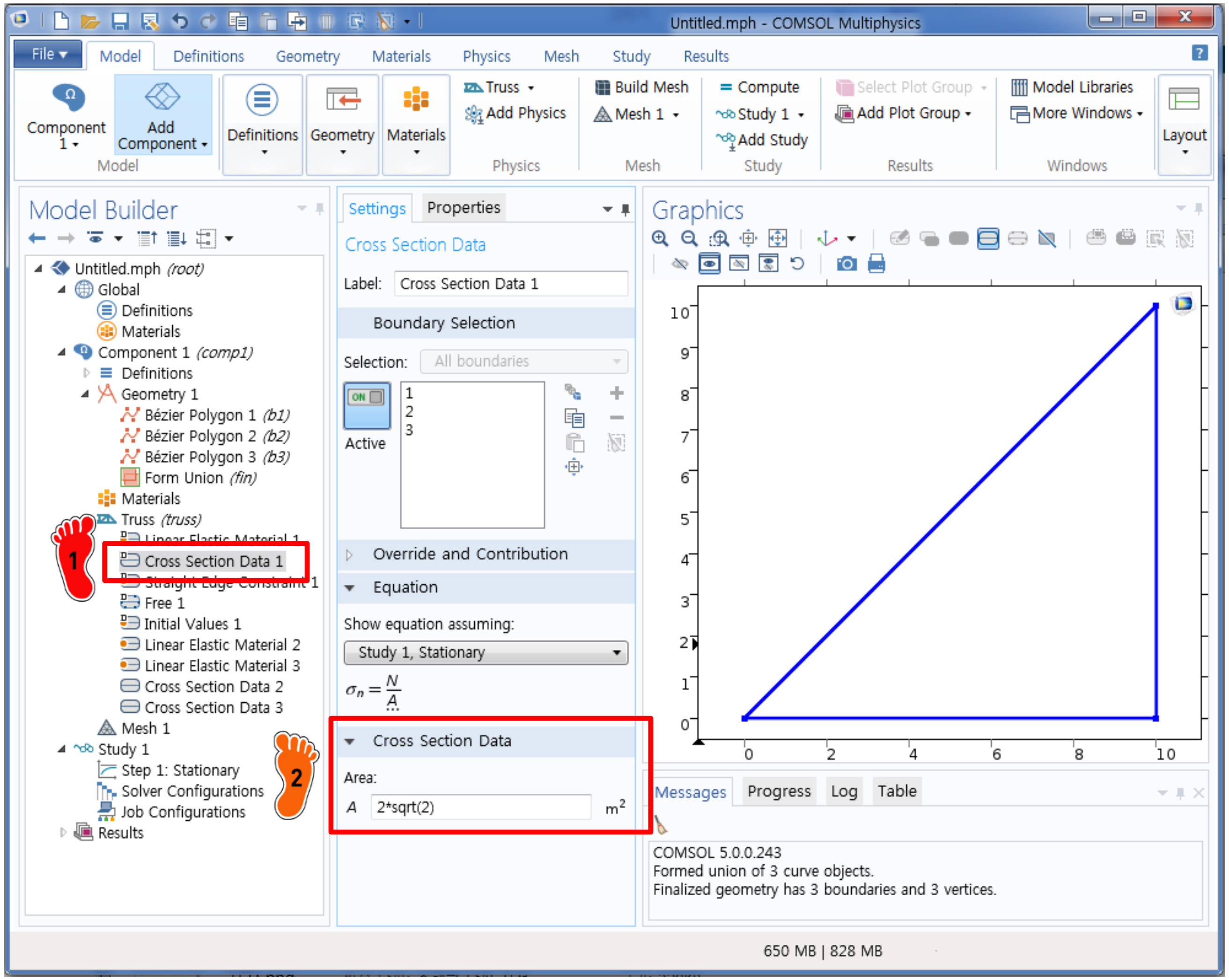
1 Cross Section Data 마우스 우클릭

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



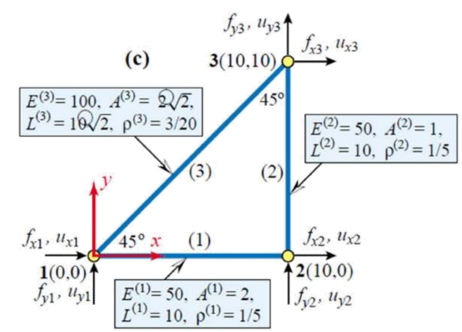
Geometric, material and fabrication properties

SECTION PROPERTY



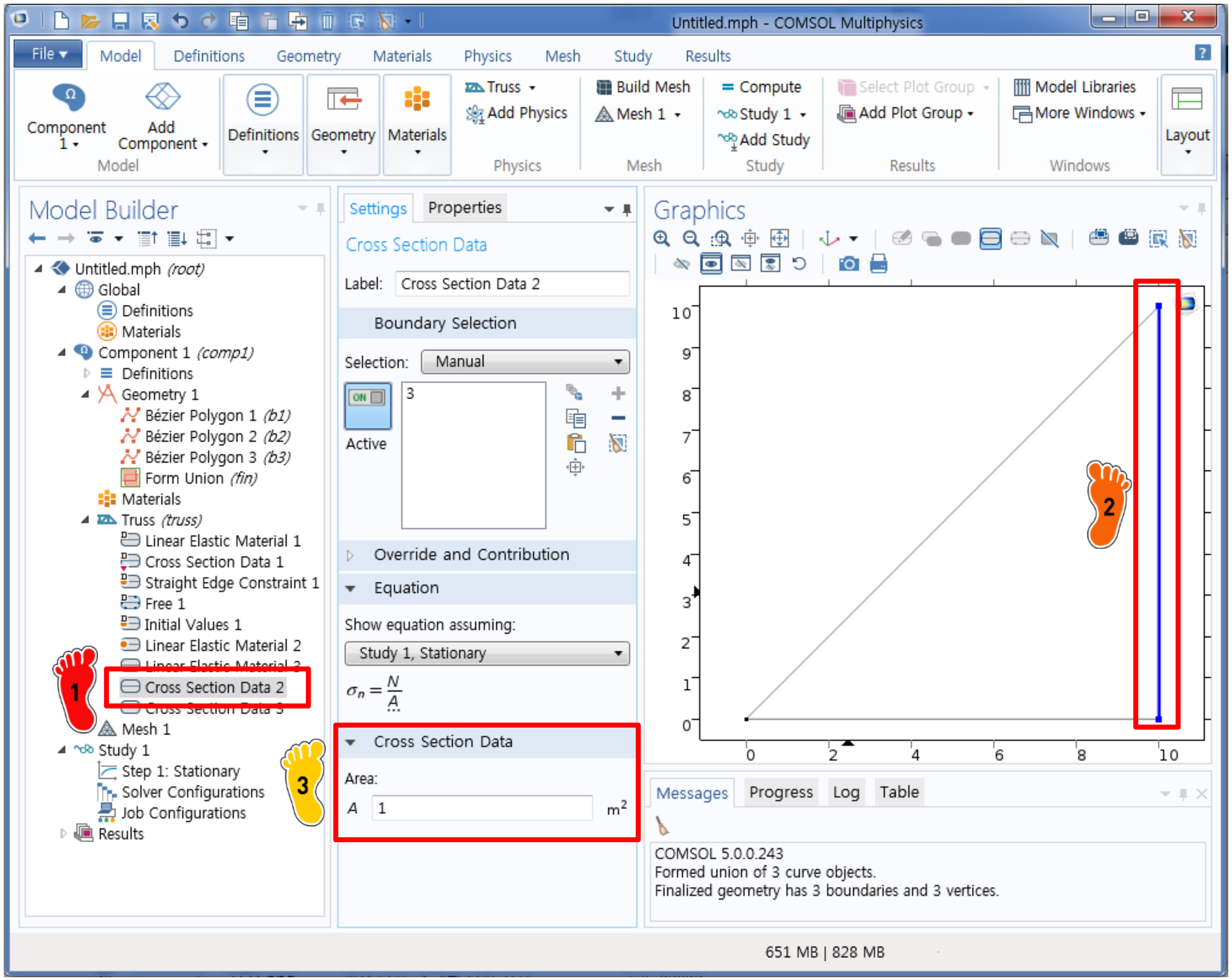
첫 번째 Cross Section Data 클릭

Area 에 2*sqrt(2) 입력



Geometric, material and fabrication properties

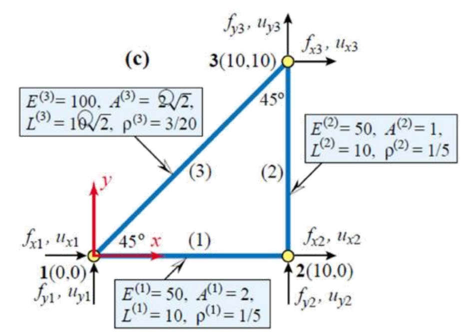
SECTION PROPERTY



1 두 번째 Cross Section Data 클릭

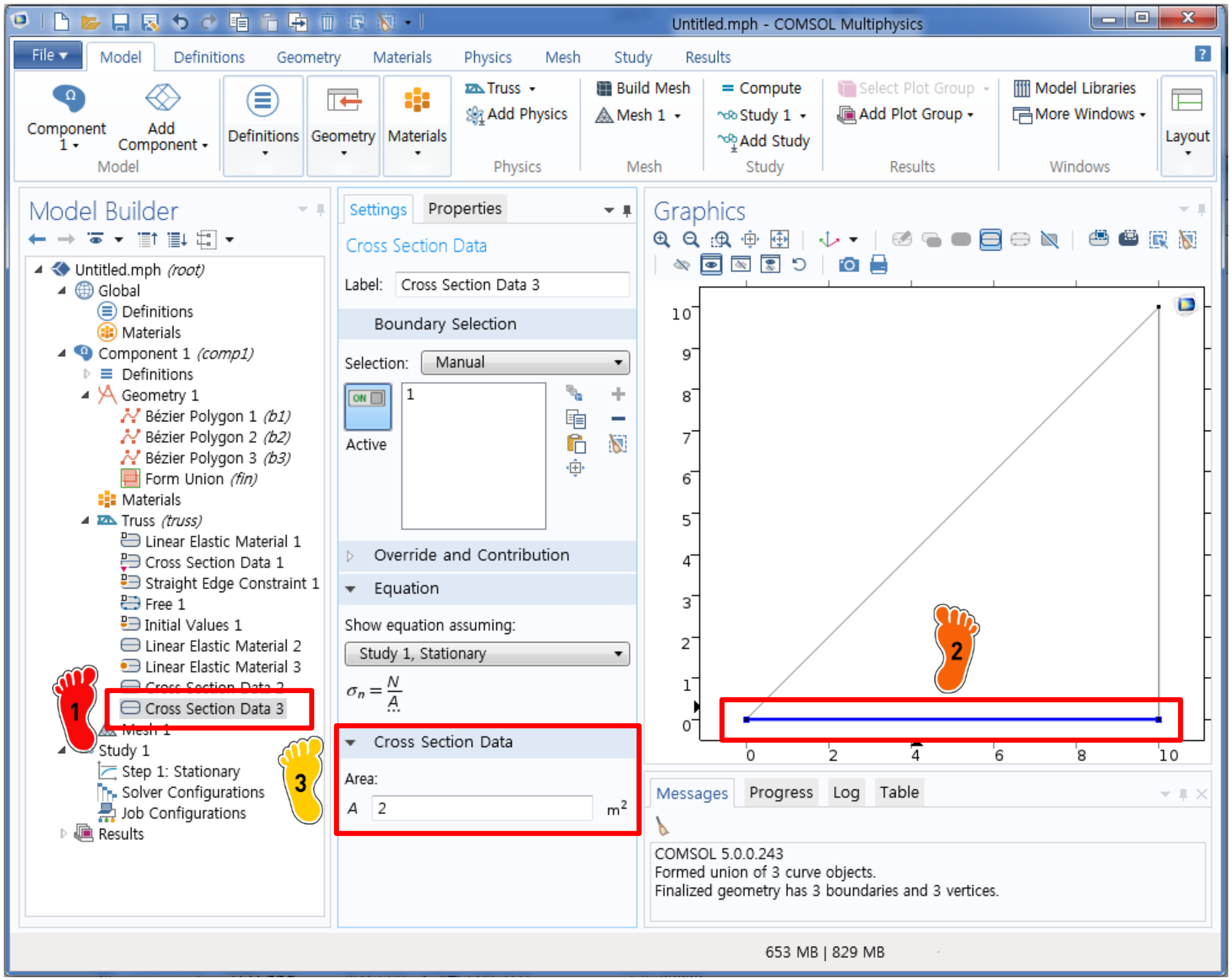
2 제일 오른쪽 부재 선택

3 Area 에 1 입력



Geometric, material and fabrication properties

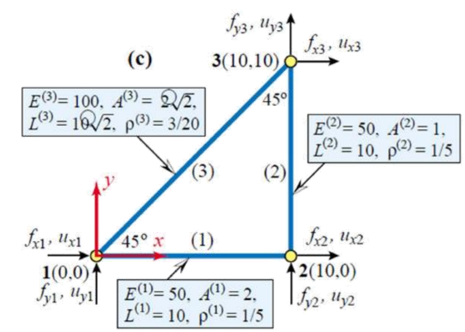
SECTION PROPERTY



1 마지막 Cross Section Data 클릭

2 제일 아래 부재 선택

3 Area 에 2 입력



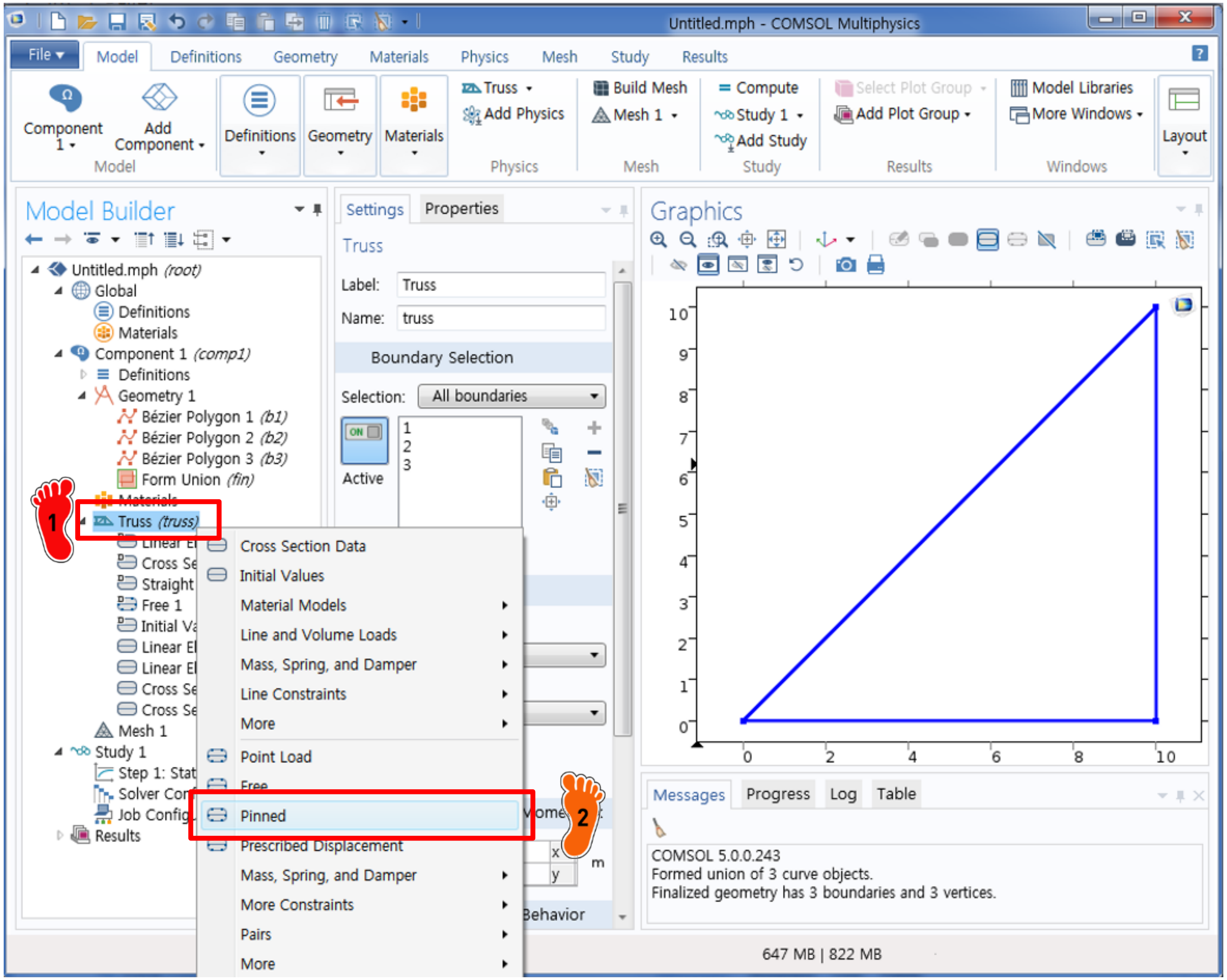
Geometric, material and fabrication properties

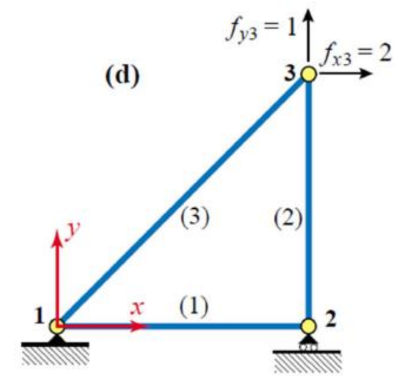
- **COMSOL analysis: Truss structure**
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **<u>Boundary condition</u>**
  - ✓ **Nodal force**
  - ✓ **Analysis**

BOUNDARY CONDITION

1 Truss 마우스 우클릭

2 Pinned 클릭



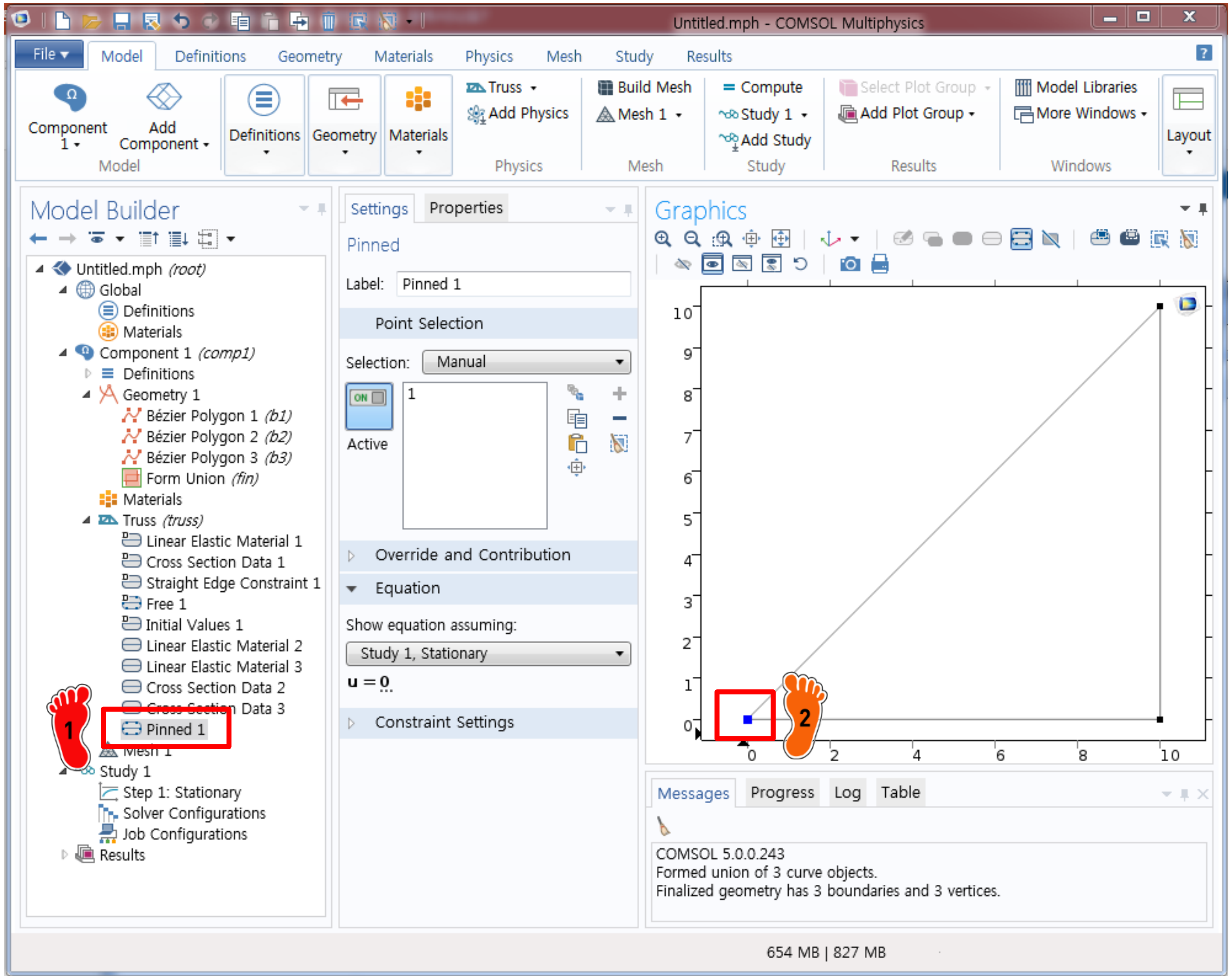


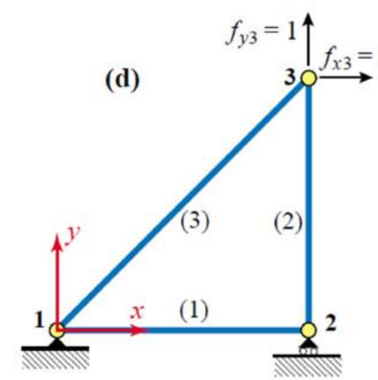
Support conditions and applied loads

BOUNDARY CONDITION

1 Pinned 클릭

2 왼쪽 아래 절점 선택



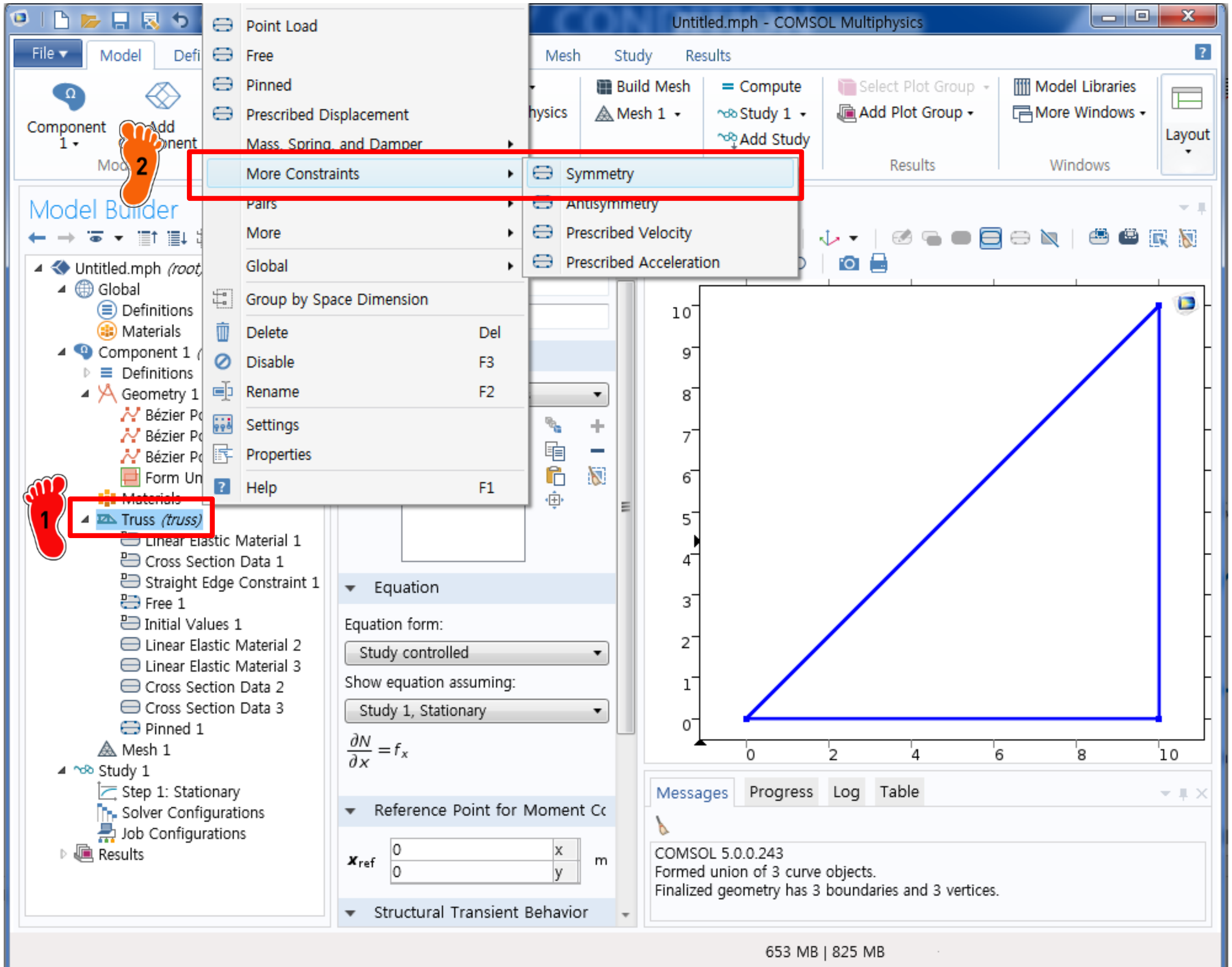


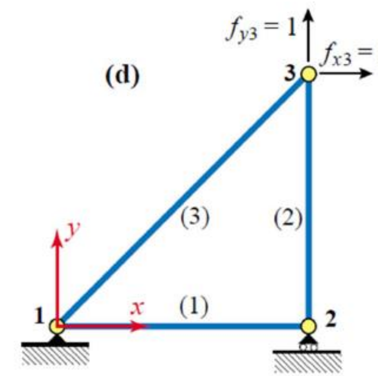
Support conditions and applied loads

BOUNDARY CONDITION

1 Truss 마우스 우클릭

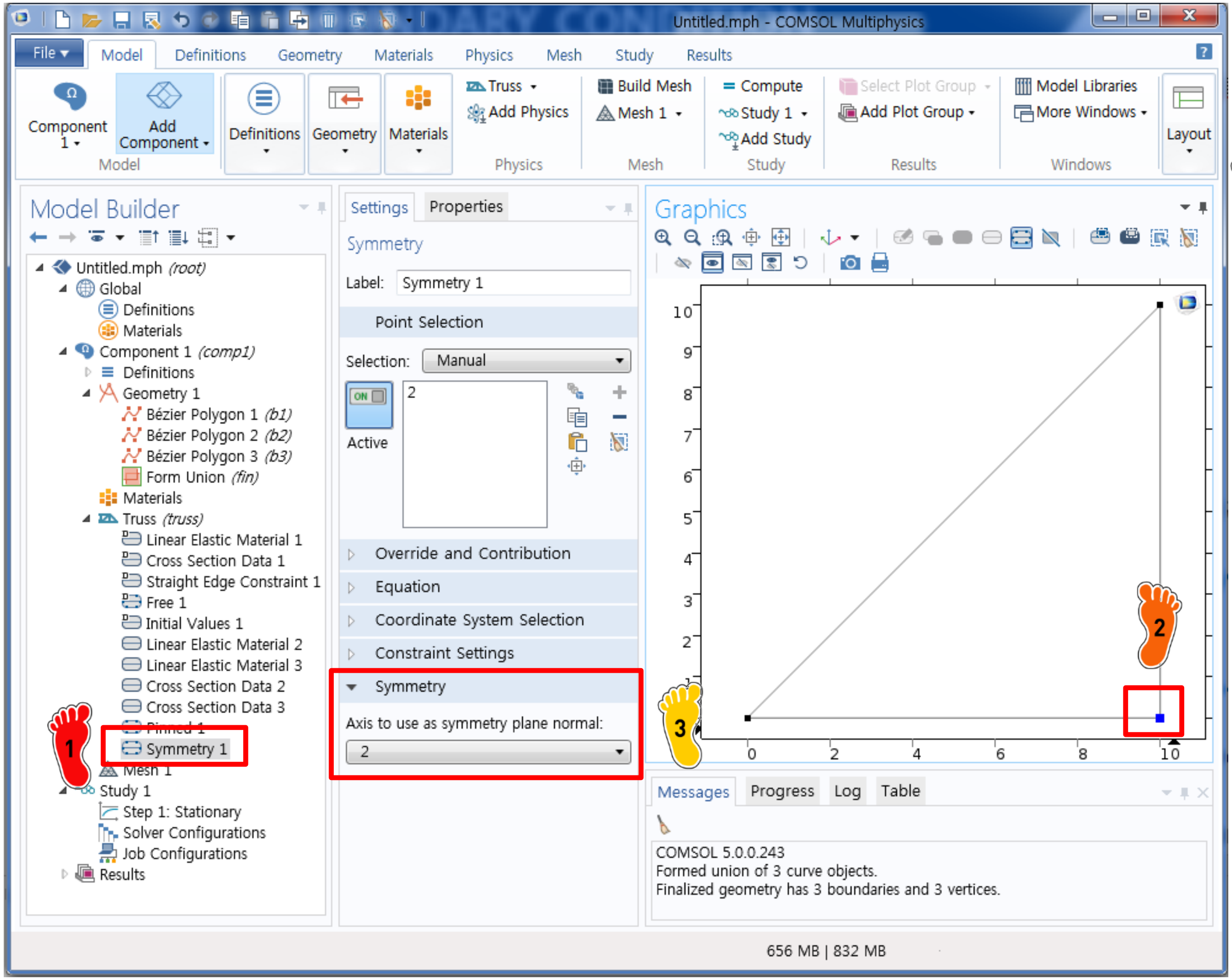
2 More Constraints에서 Symmetry 클릭





Support conditions and applied loads

BOUNDARY CONDITION

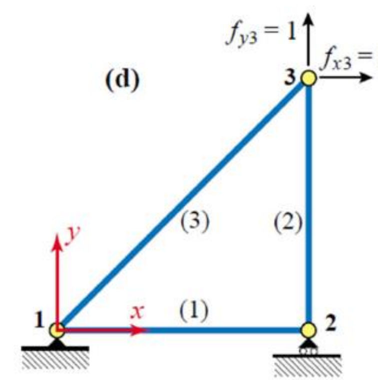


1 Symmetry 1 클릭

2 오른쪽 아래 절점 선택

3 2 로 변경

1의 의미는 x 방향에 수직한 평면, 2는 y 방향에 수직한 평면



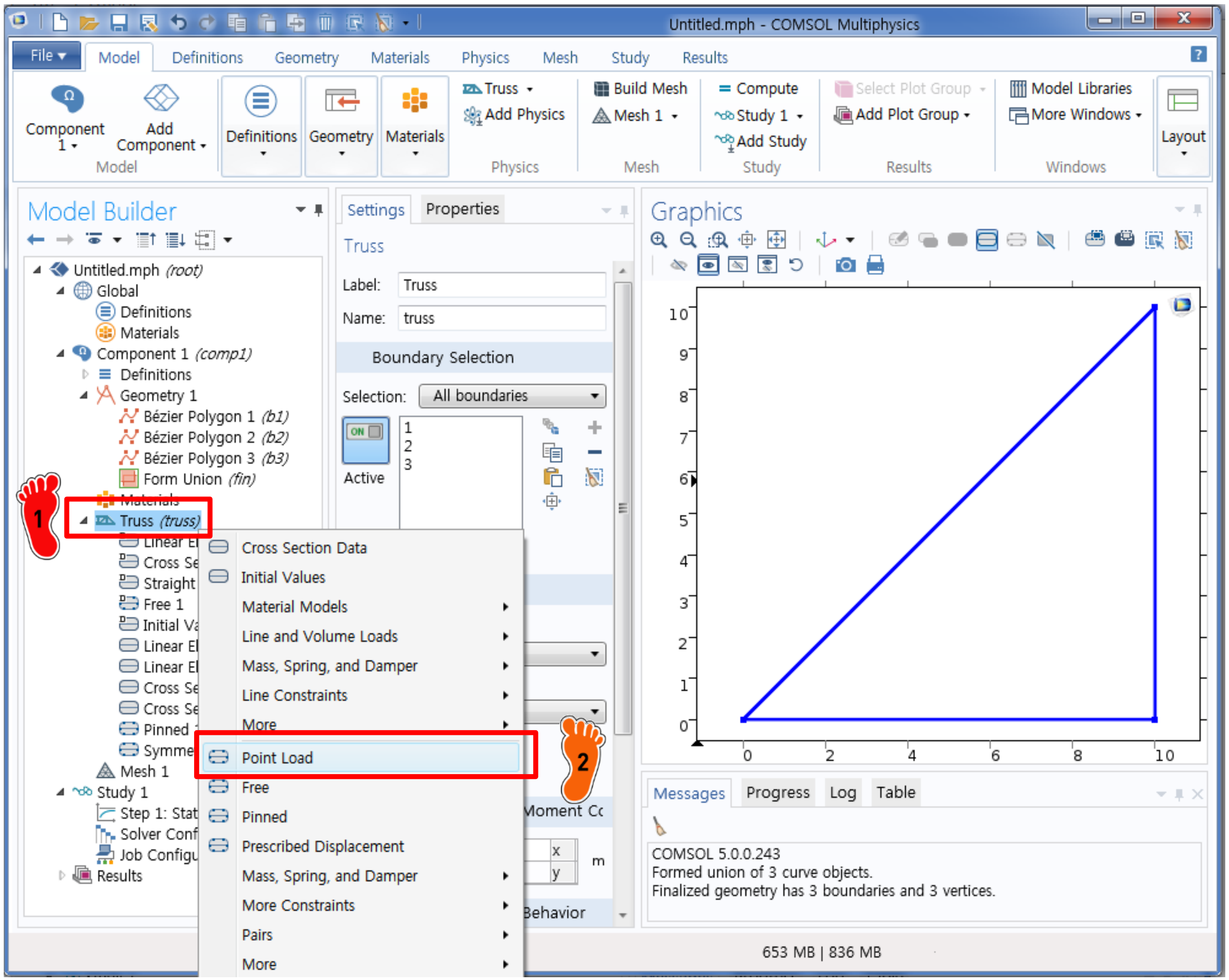
Support conditions and applied loads

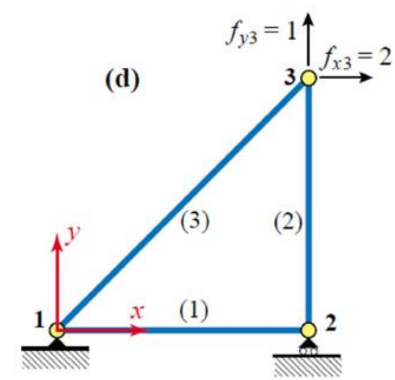
- **COMSOL analysis: Truss structure**
  - ✓ **Geometry creation**
  - ✓ **Material property**
  - ✓ **Cross section property**
  - ✓ **Boundary condition**
  - ✓ **<u>Nodal force</u>**
  - ✓ **Analysis**

POINT LOAD

1 Truss 마우스 우클릭

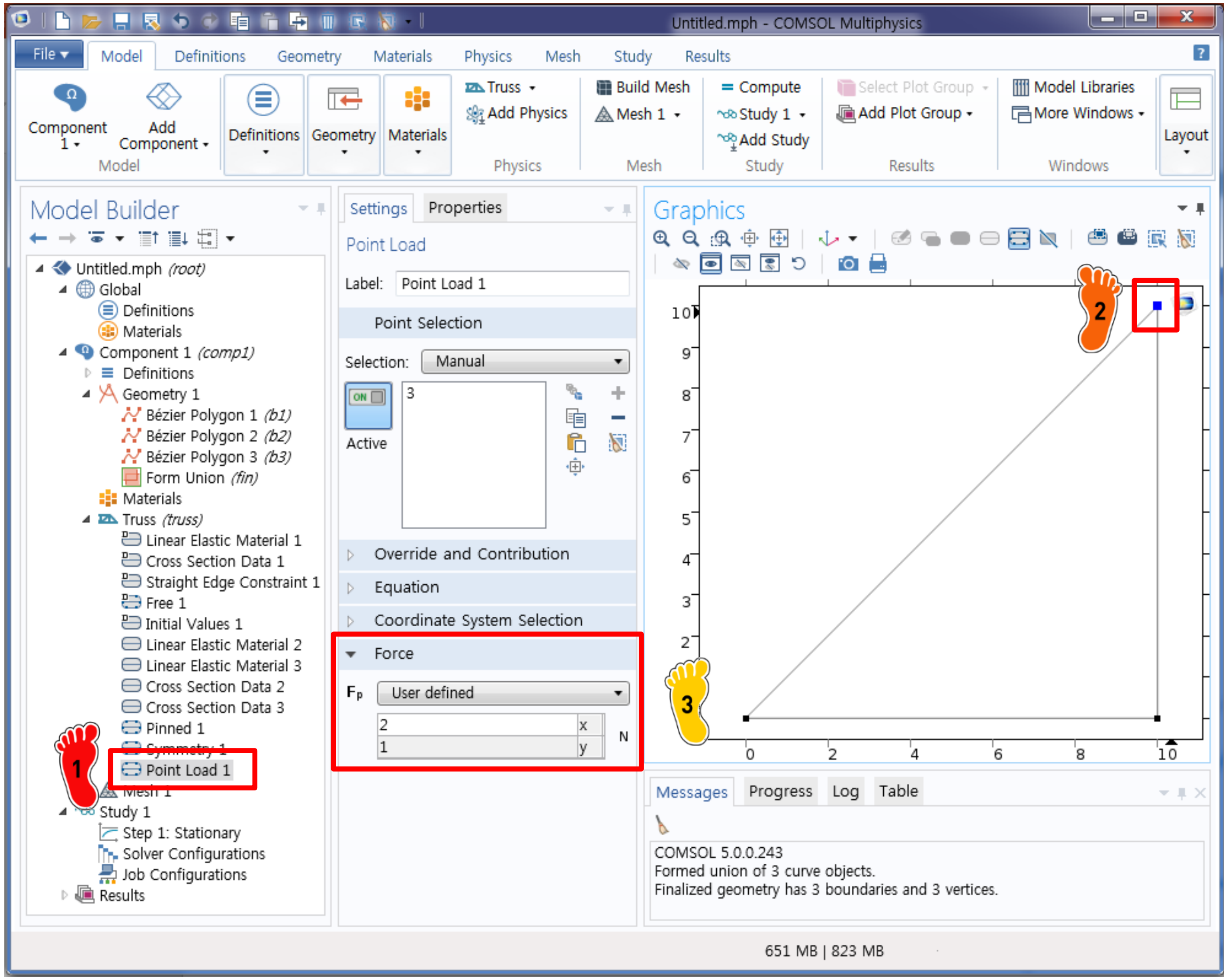
2 Point Load 클릭



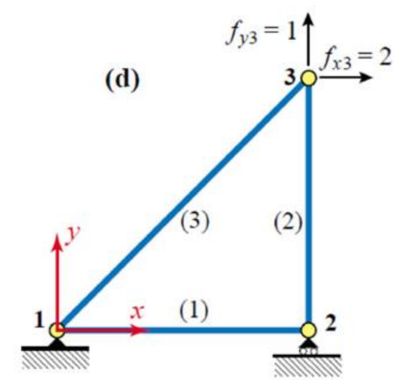


Support conditions and applied loads

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

- **COMSOL analysis: Truss structure**

  ✓ Geometry creation

  ✓ Material property

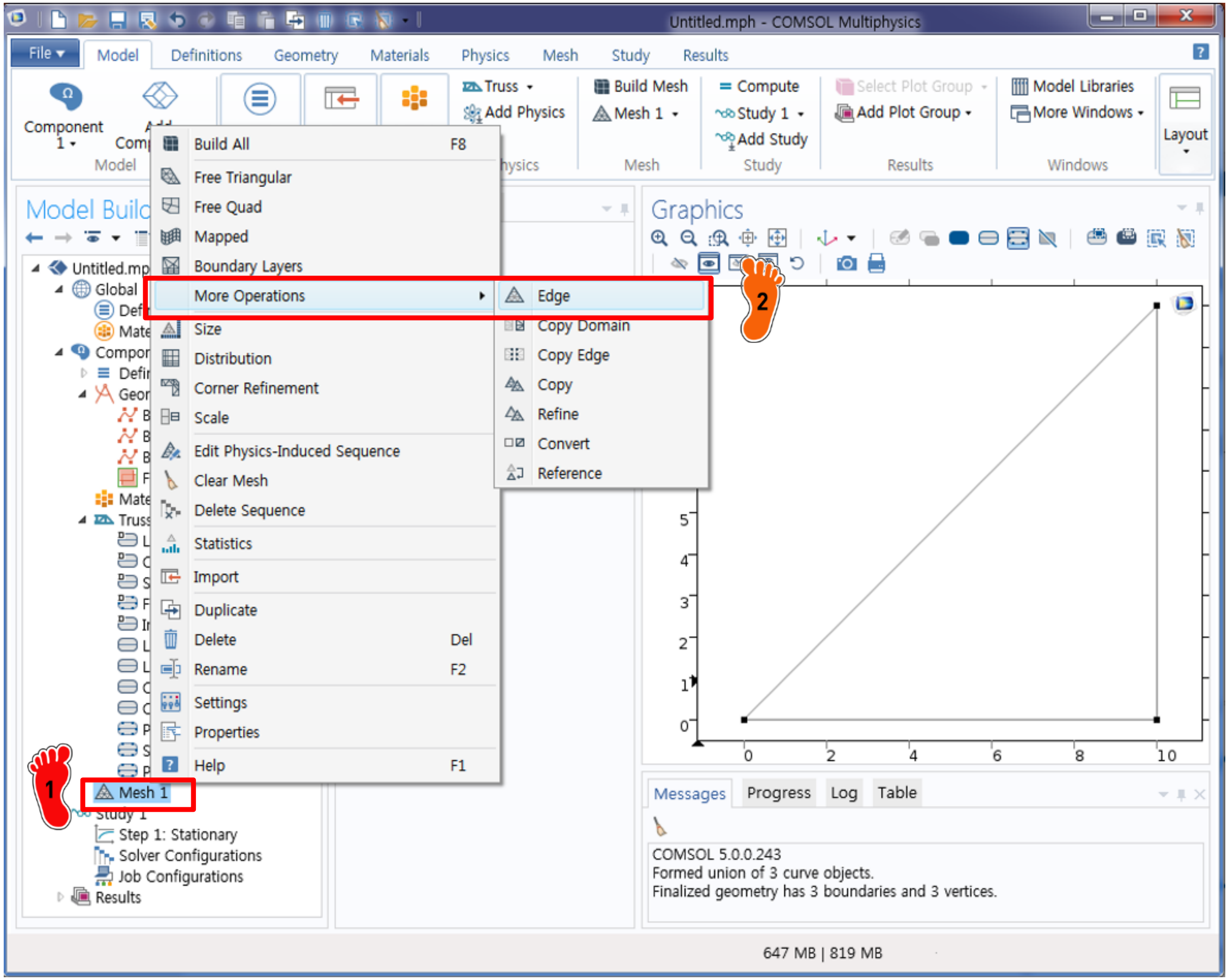
  ✓ Cross section property

  ✓ Boundary condition

  ✓ Nodal force

  ✓ <u>Analysis</u>

MESH CREATION

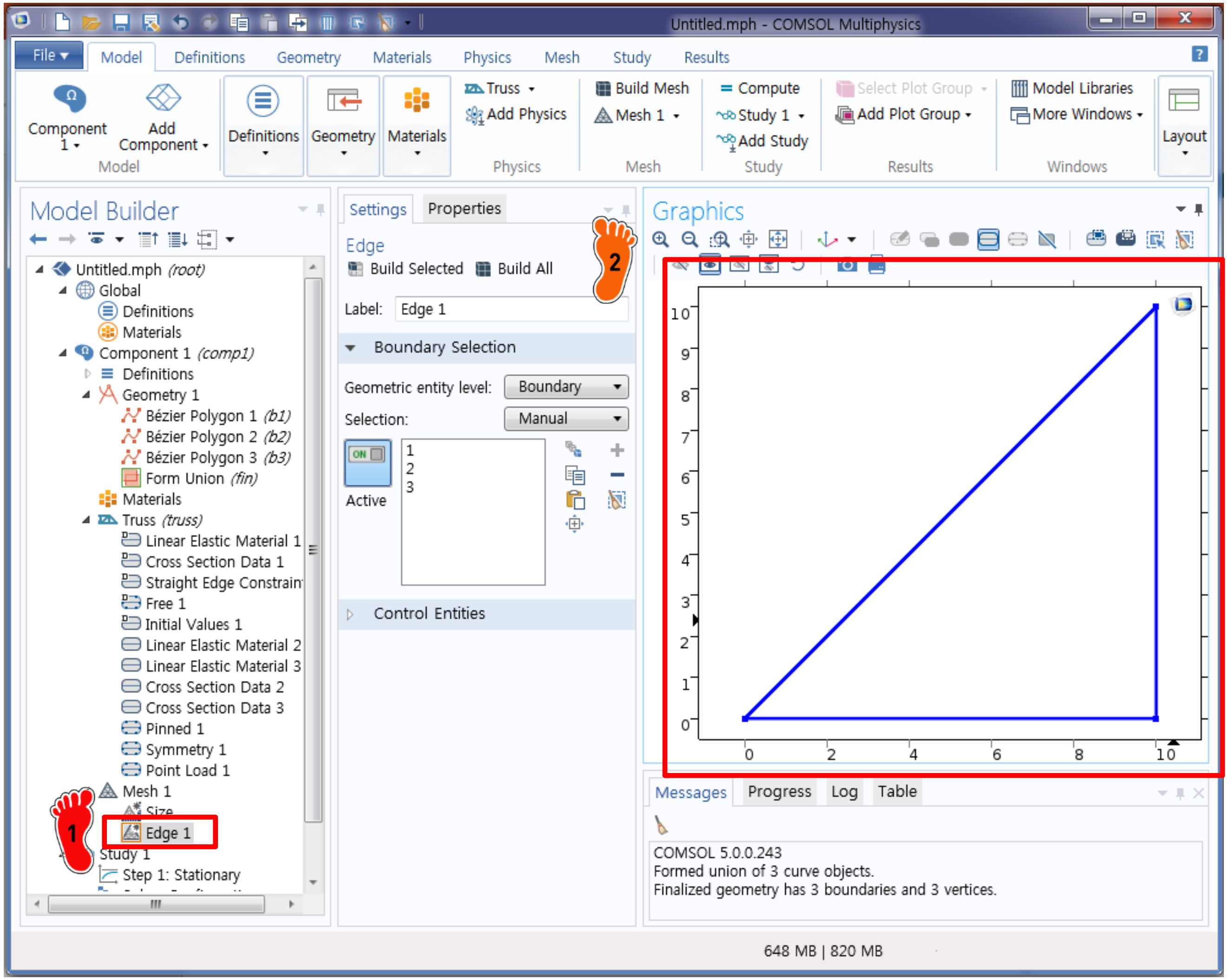


1 Mesh 1 마우스 우클릭

2 More Operations에서

Edge 클릭

MESH CREATION



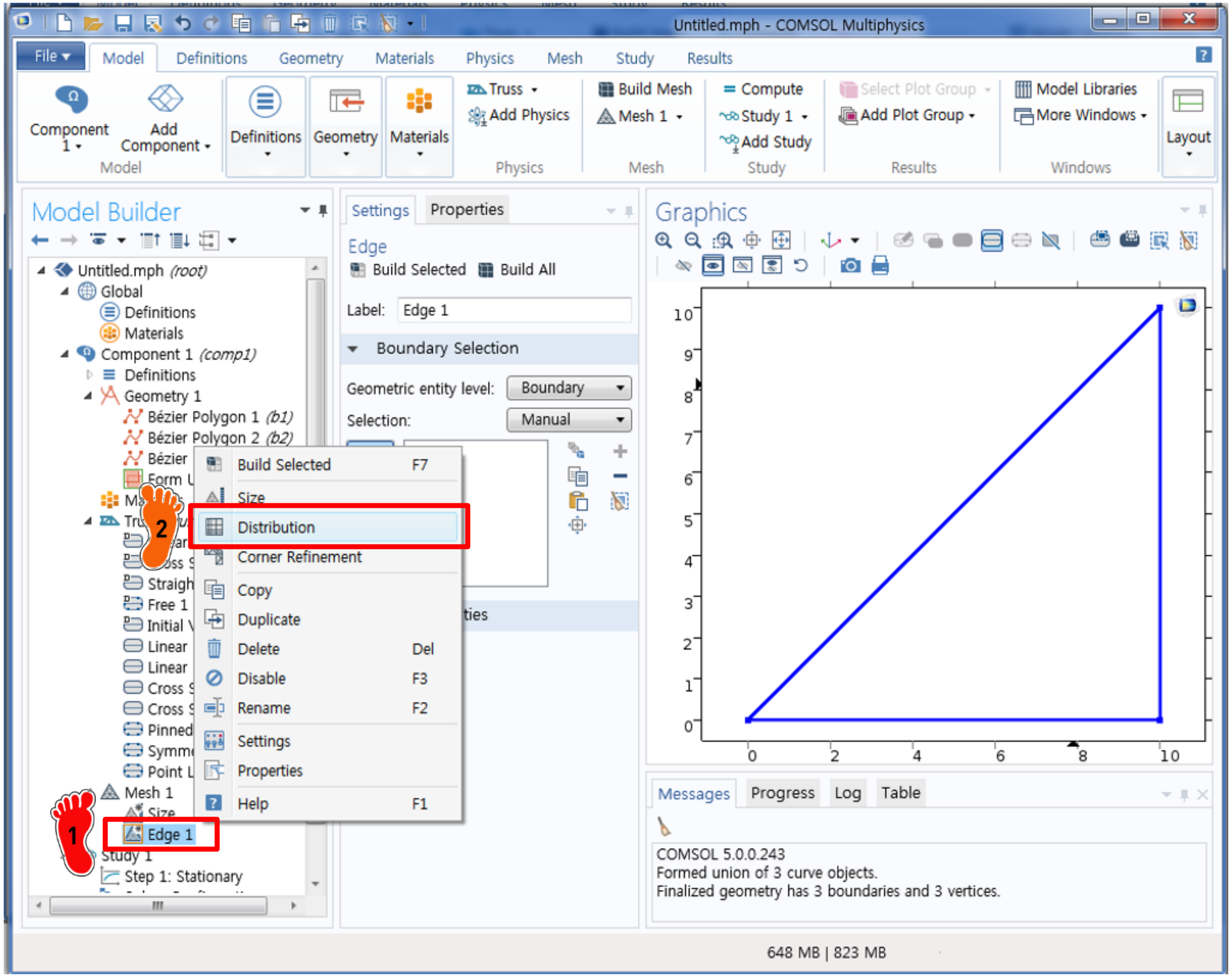
1 Edge 1 클릭

2 부재 세 개 선택

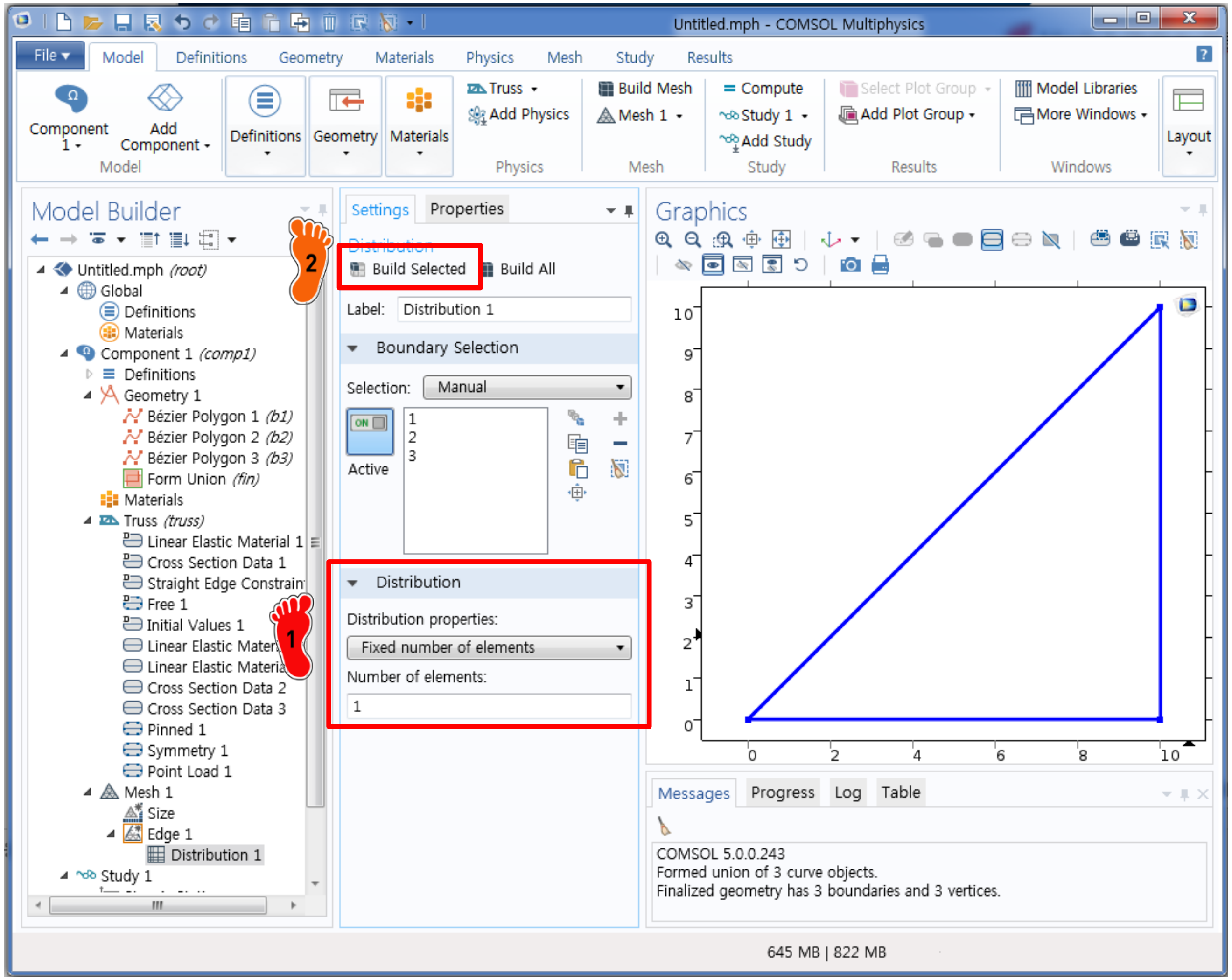
MESH CREATION

1 Edge 1 우클릭

2 Distribution 클릭

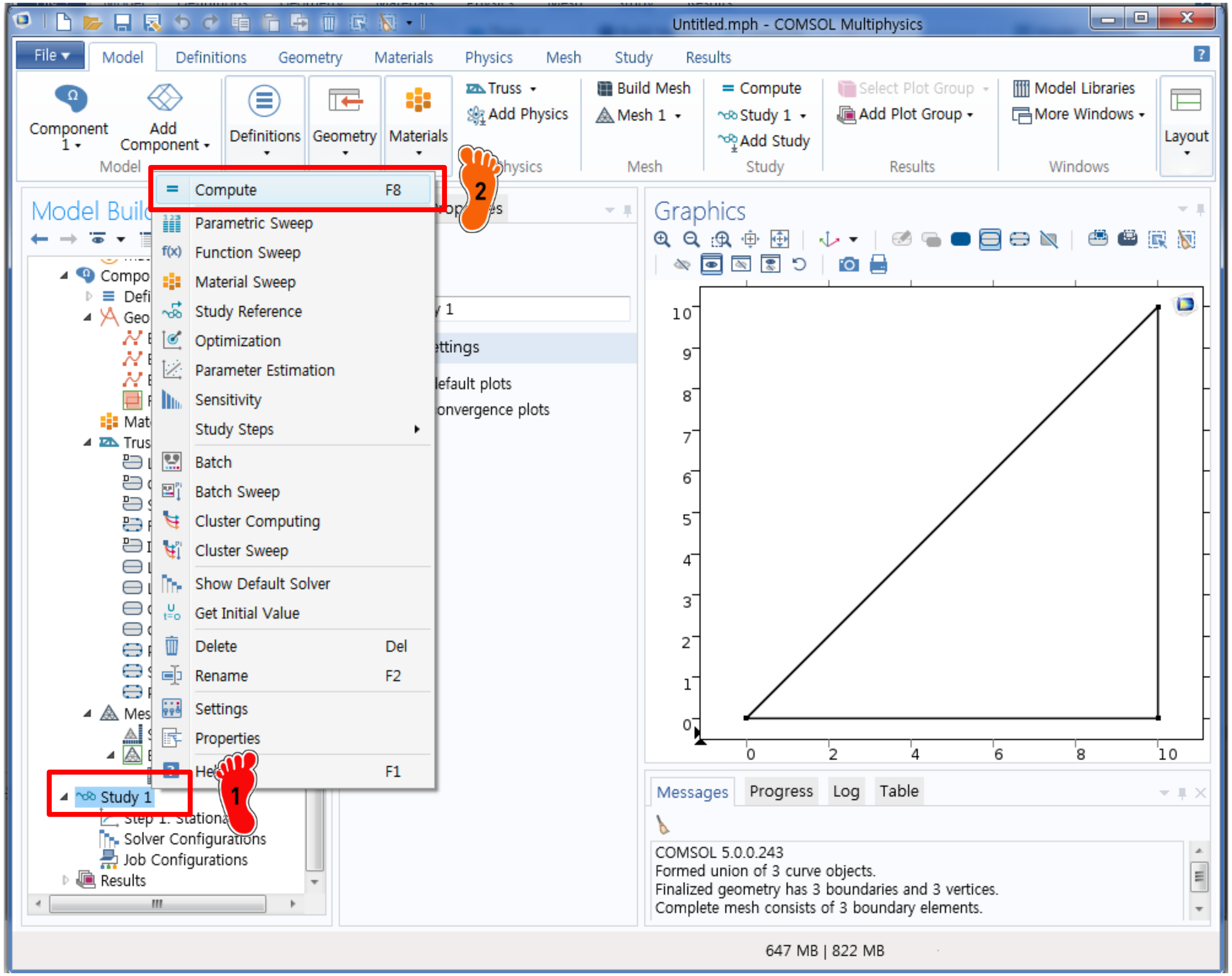


MESH CREATION



1. Number of elements 1 입력
2. Build Selected 클릭

COMPUTE

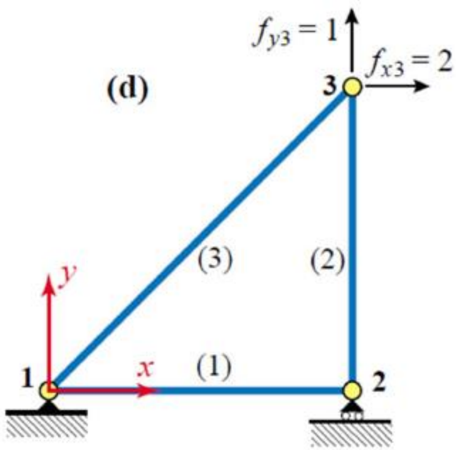


1 Study 1 마우스 우클릭

2 Compute 클릭

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

ASSEMBLE OPTION



Support conditions and applied loads

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

$$\Downarrow$$

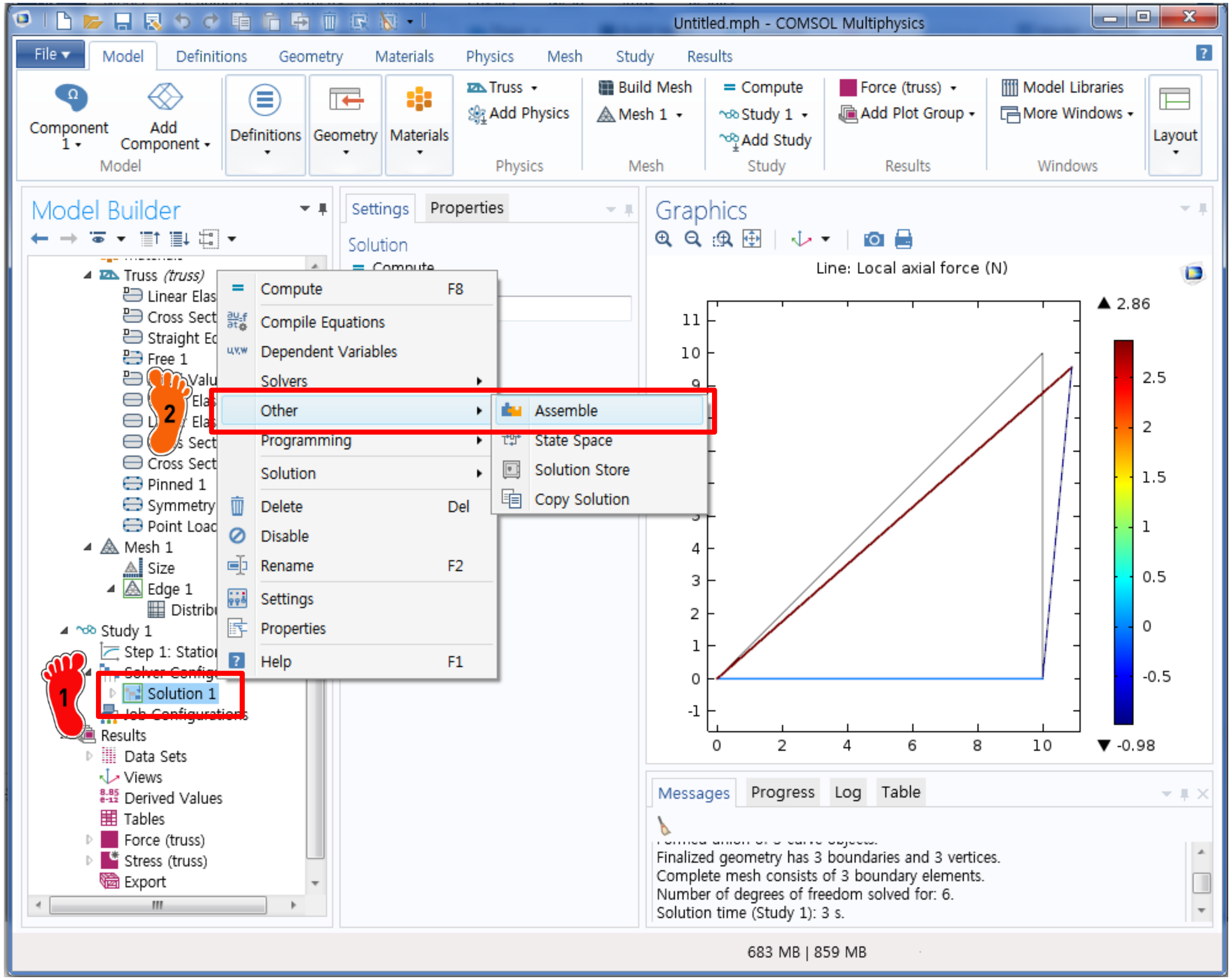
$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \boxed{\begin{bmatrix} 20 & 10 & -10 & 0 & -10 & -10 \\ 10 & 10 & 0 & 0 & -10 & -10 \\ -10 & 0 & 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 5 & 0 & -5 \\ -10 & -10 & 0 & 0 & 10 & 10 \\ -10 & -10 & 0 & -5 & 10 & 15 \end{bmatrix}}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

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

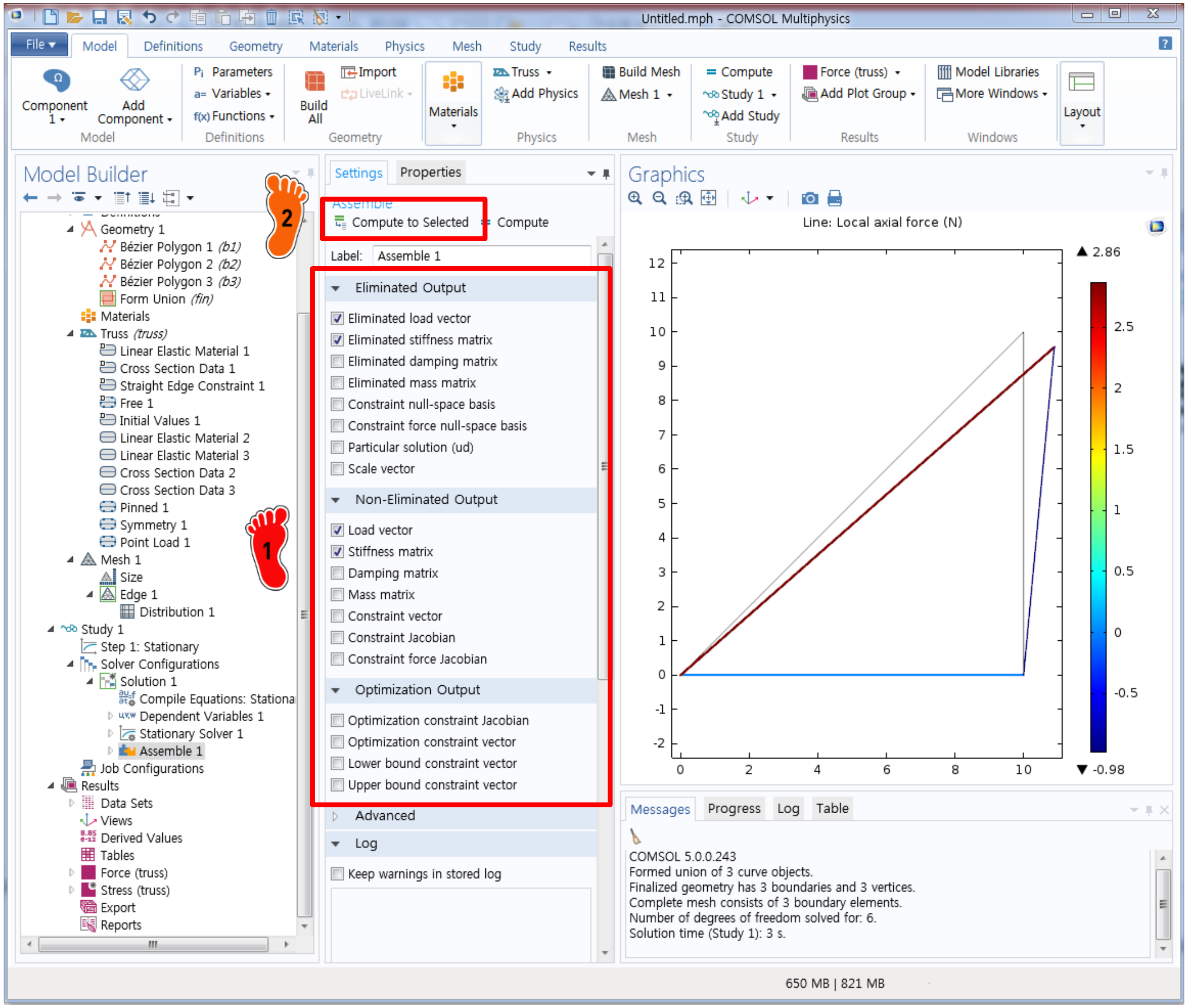
ASSEMBLE OPTION

Solver 1 마우스 우클릭

Other에서 Assemble 클릭



ASSEMBLE OPTION



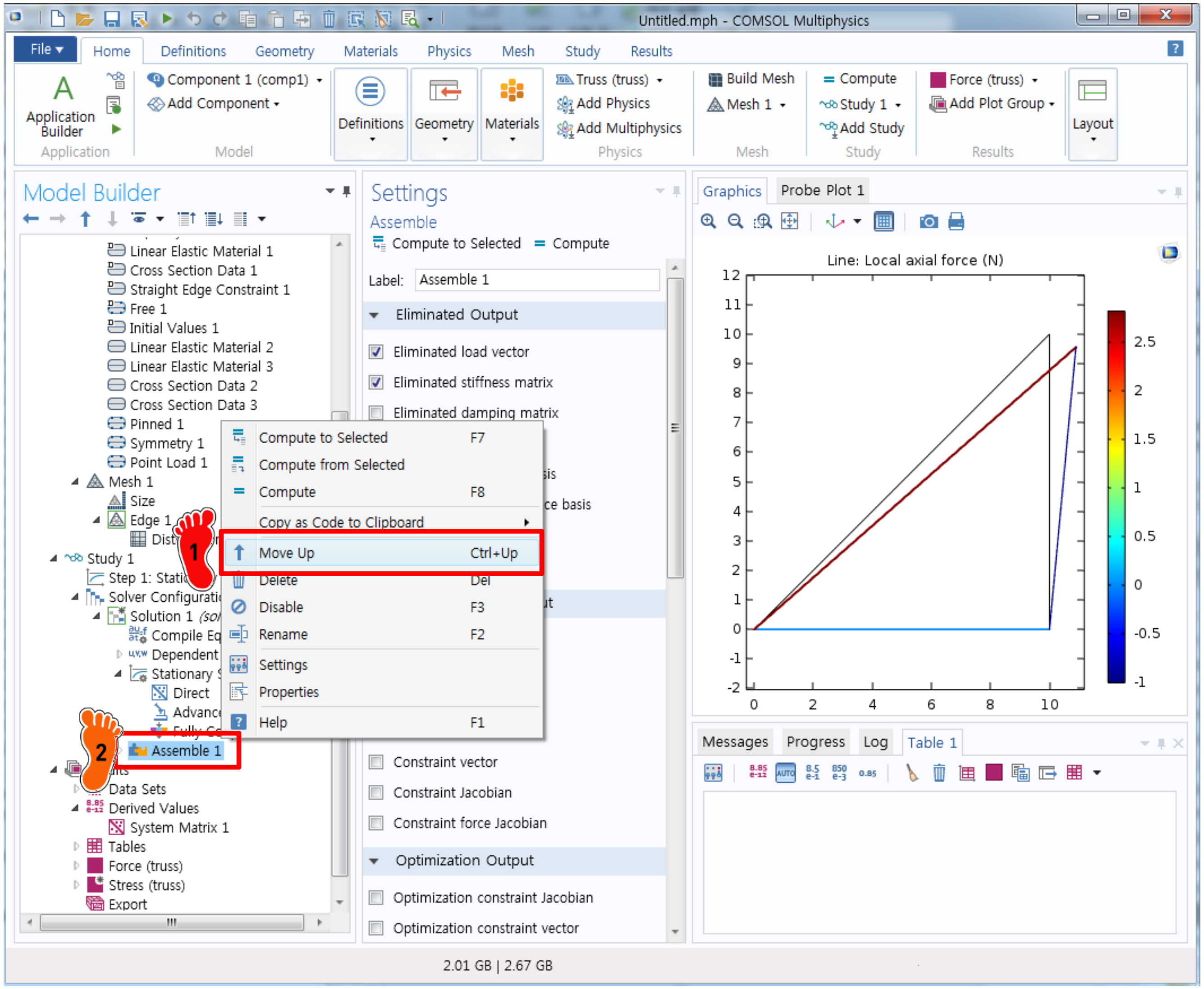
1 Eliminated load vector
Eliminated stiffness matrix

Load vector
Stiffness matrix

체크박스에 체크

2 Compute to Selected 클릭

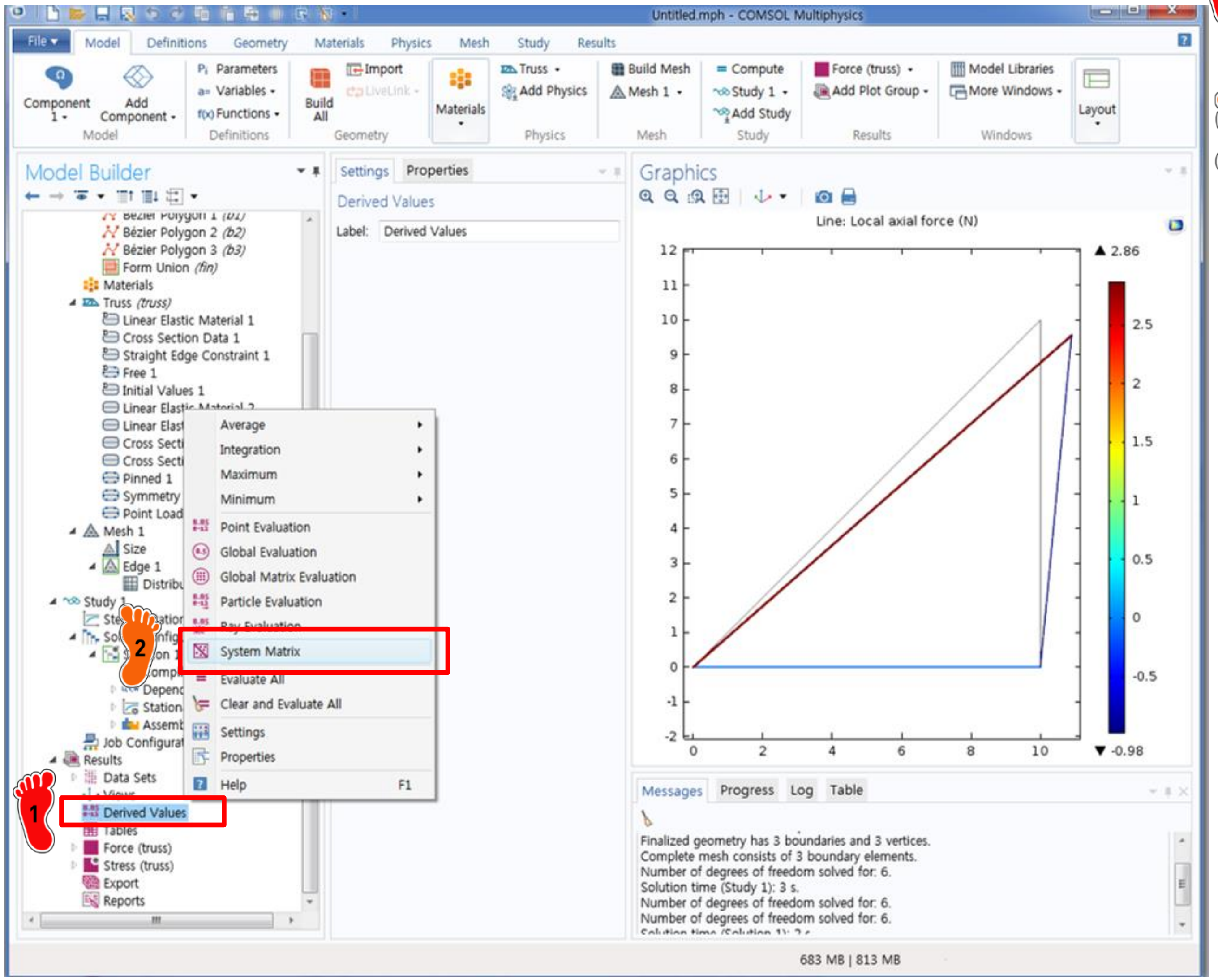
ASSEMBLE OPTION



1 Assemble 1 마우스 우클릭

2 Move up을 클릭하여
Assemble 노드가
Solver 노드 위에 위치하도
록 이동

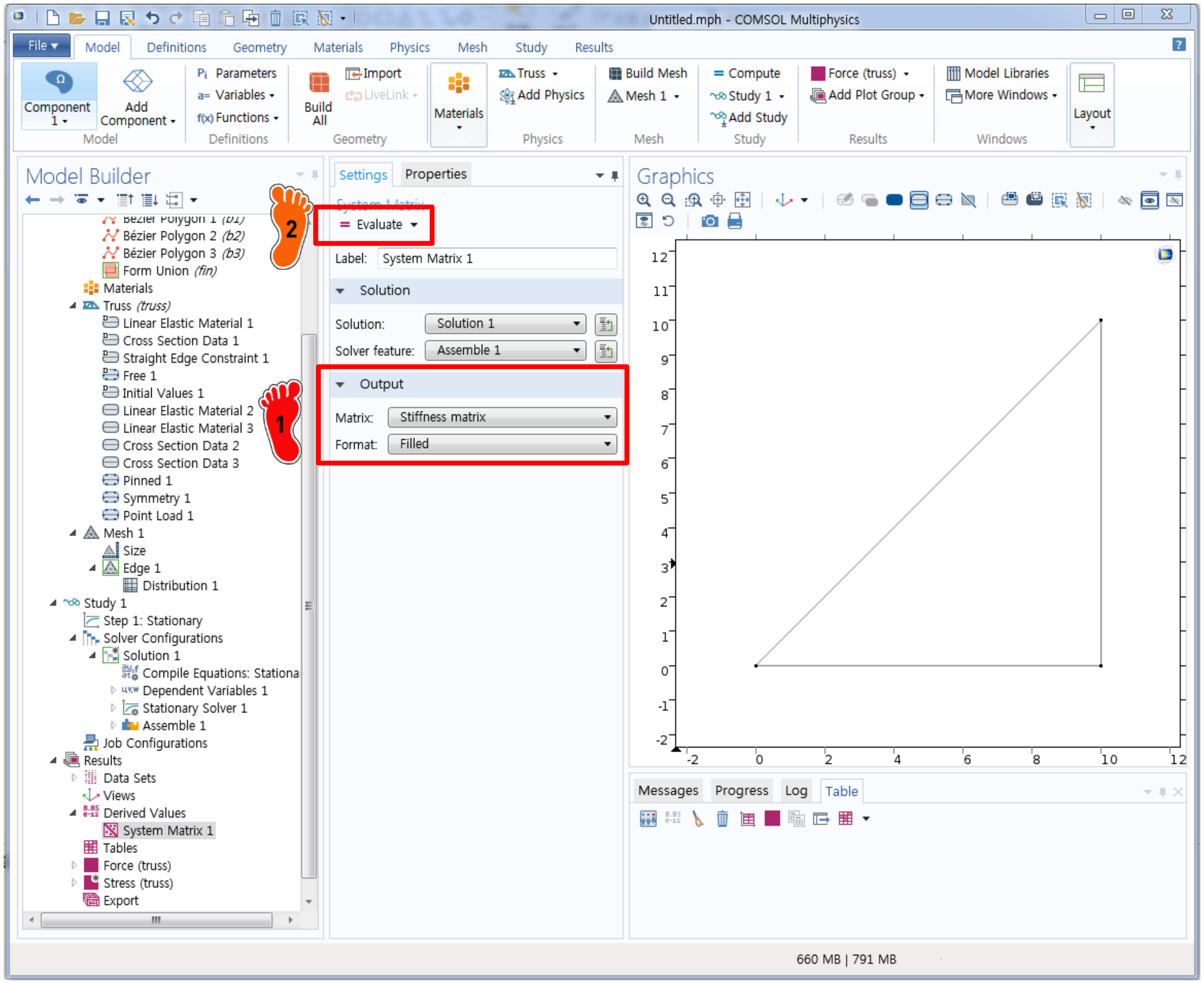
SYSTEM MATRIX



1 Derived Values 마우스 우클릭

2 System Matrix 클릭

SYSTEM MATRIX

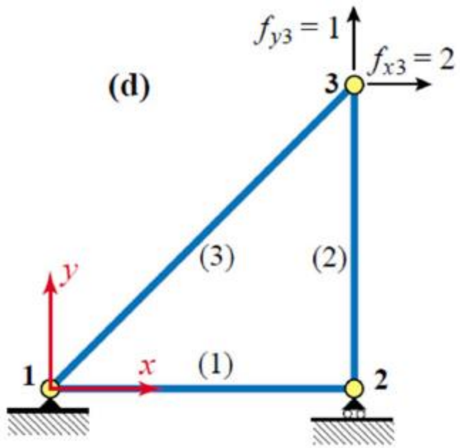


1 Stiffness matrix
Filled
선택

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

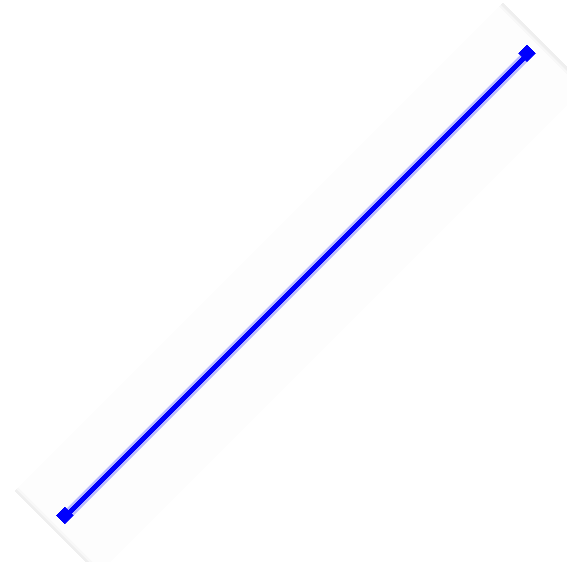
Stiffness matrix by Hand

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

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

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

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

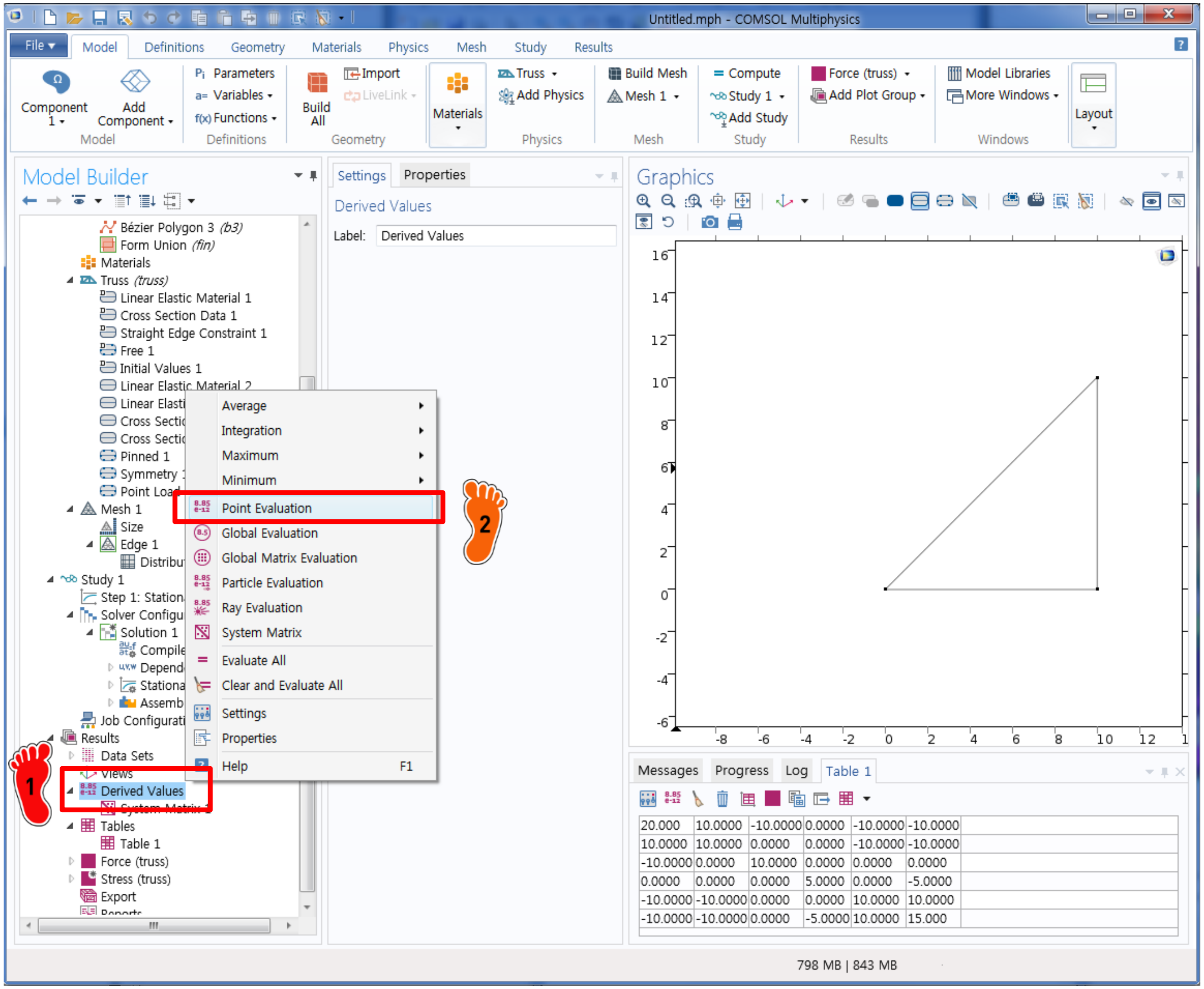
Eliminated Stiffness matrix

10.000	0.0000	0.0000
0.0000	10.000	10.000
0.0000	10.000	15.000

Eliminated Load vector

0
2
1

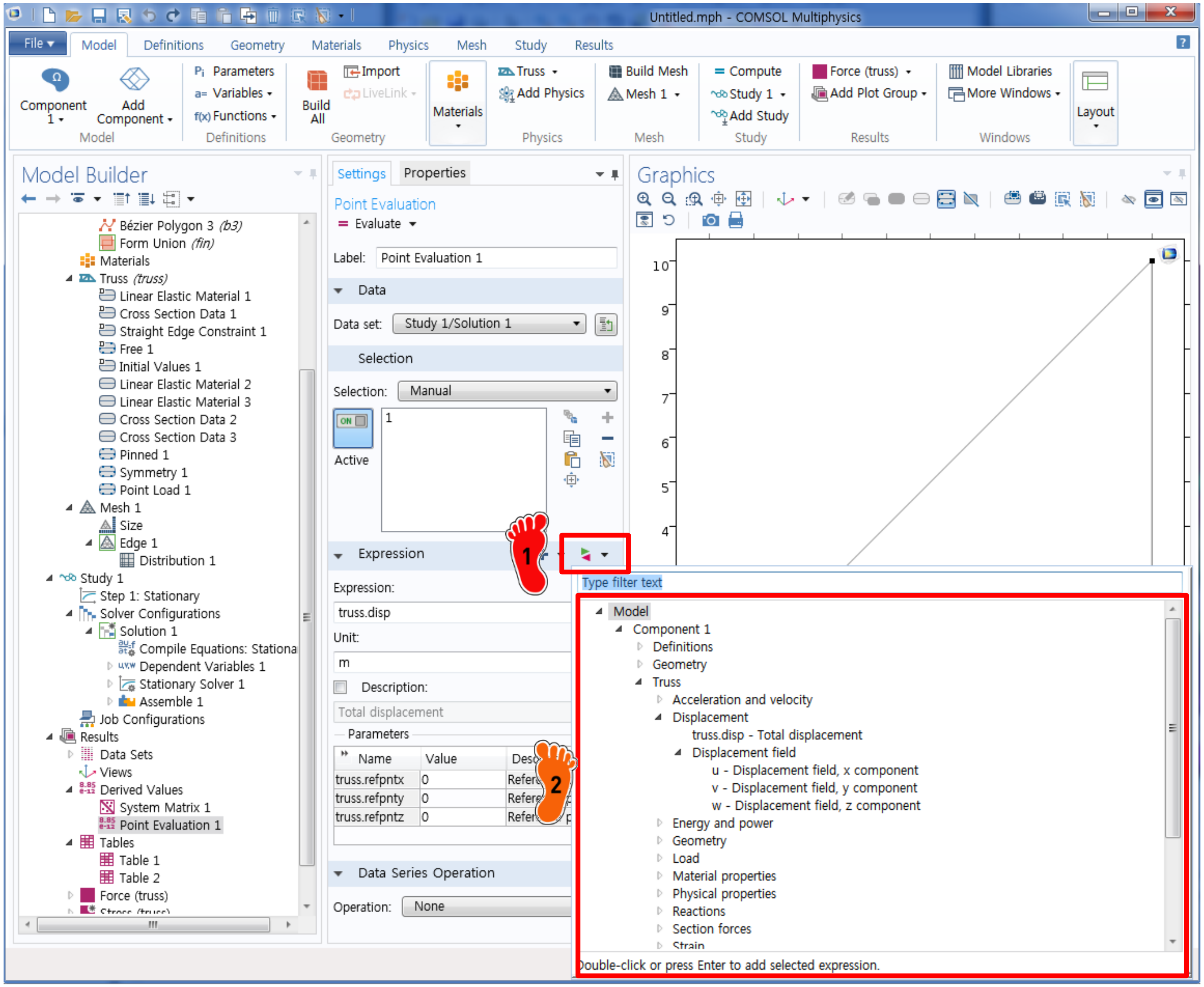
NODAL DISPLACEMENT



1 Derived Values 마우스 우클릭

2 Point Evaluation 클릭

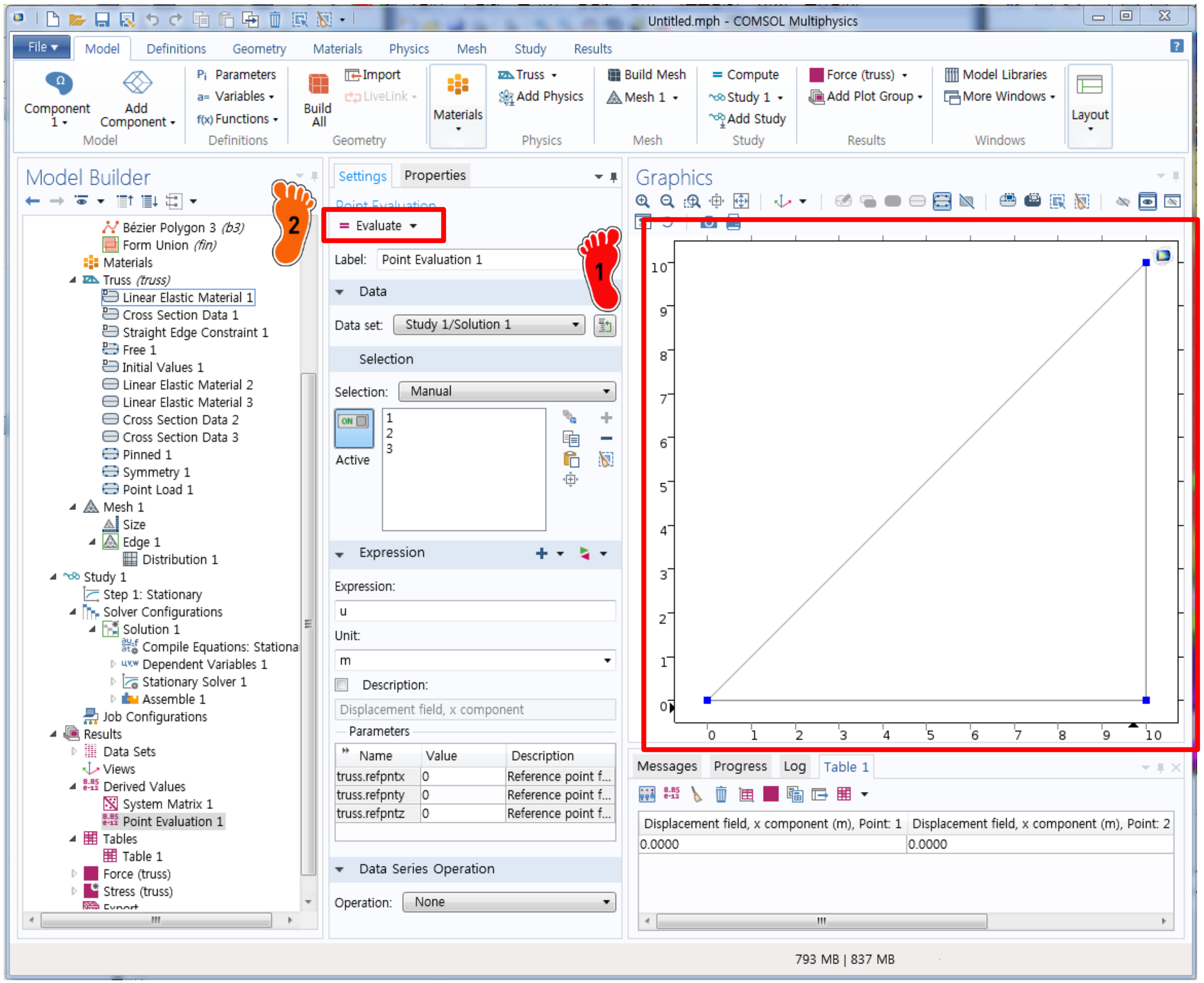
NODAL DISPLACEMENT



1 Replace expression

2 보고 싶은 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

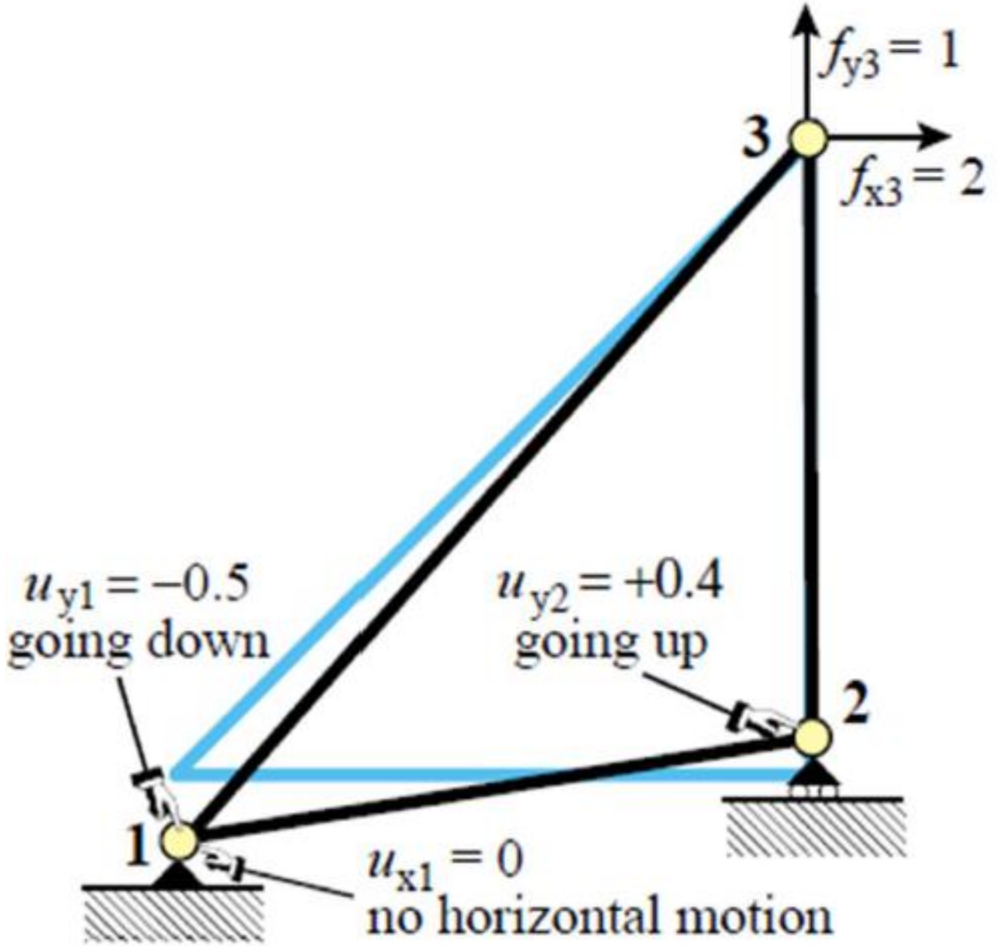
$$\begin{bmatrix} u_{x2} \\ u_{x3} \\ u_{y3} \end{bmatrix} = \begin{bmatrix} 0 \\ 0.4 \\ -0.2 \end{bmatrix} \xrightarrow{\text{expand with known displacement BCs}} \mathbf{u} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0.4 \\ -0.2 \end{bmatrix}$$

Nodal displacement by COMSOL

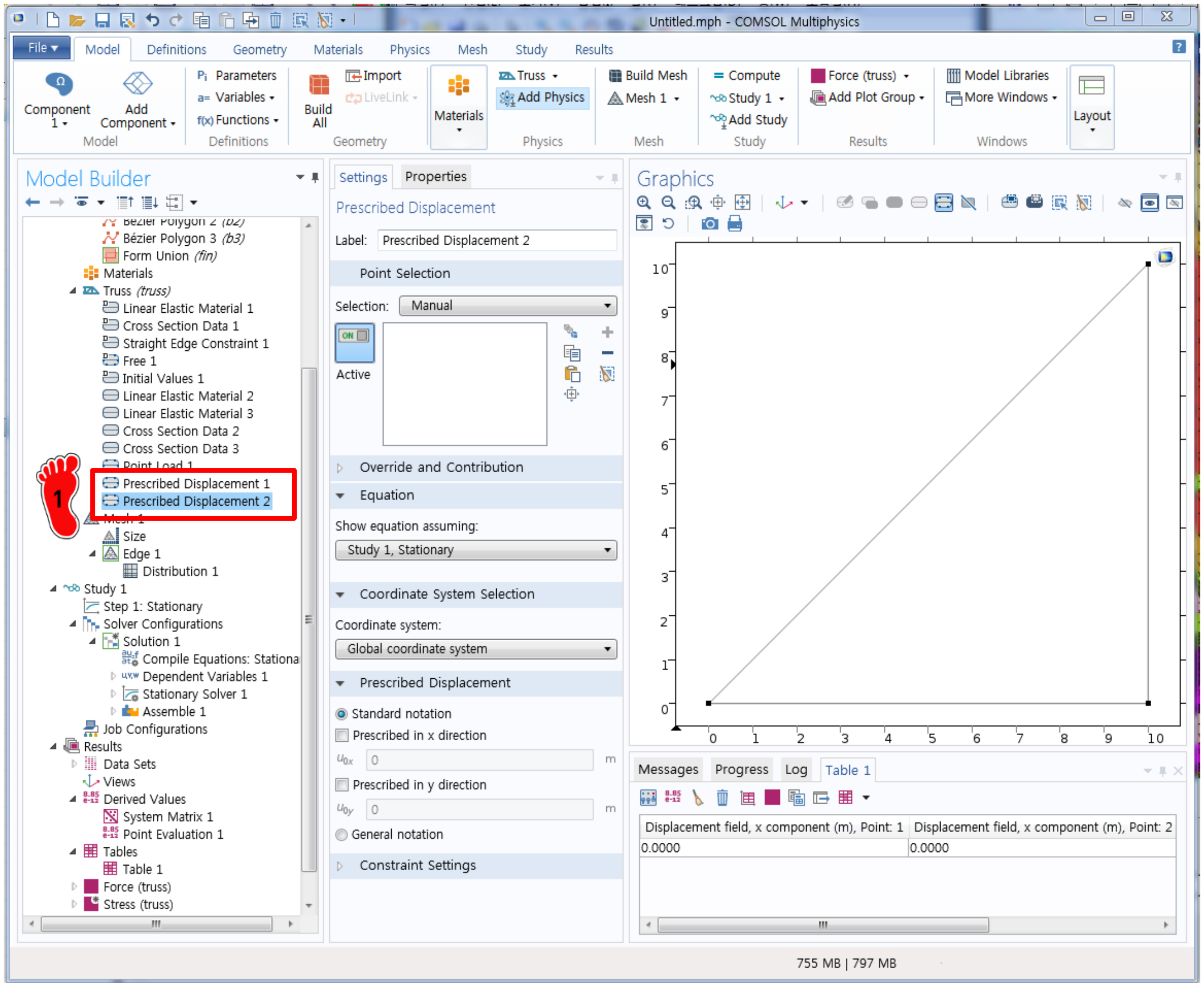
Displacement field, x component (m), Point: 1	Displacement field, x component (m), Point: 2	Displacement field, x component (m), Point: 3
0.0000	0.0000	0.40000

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

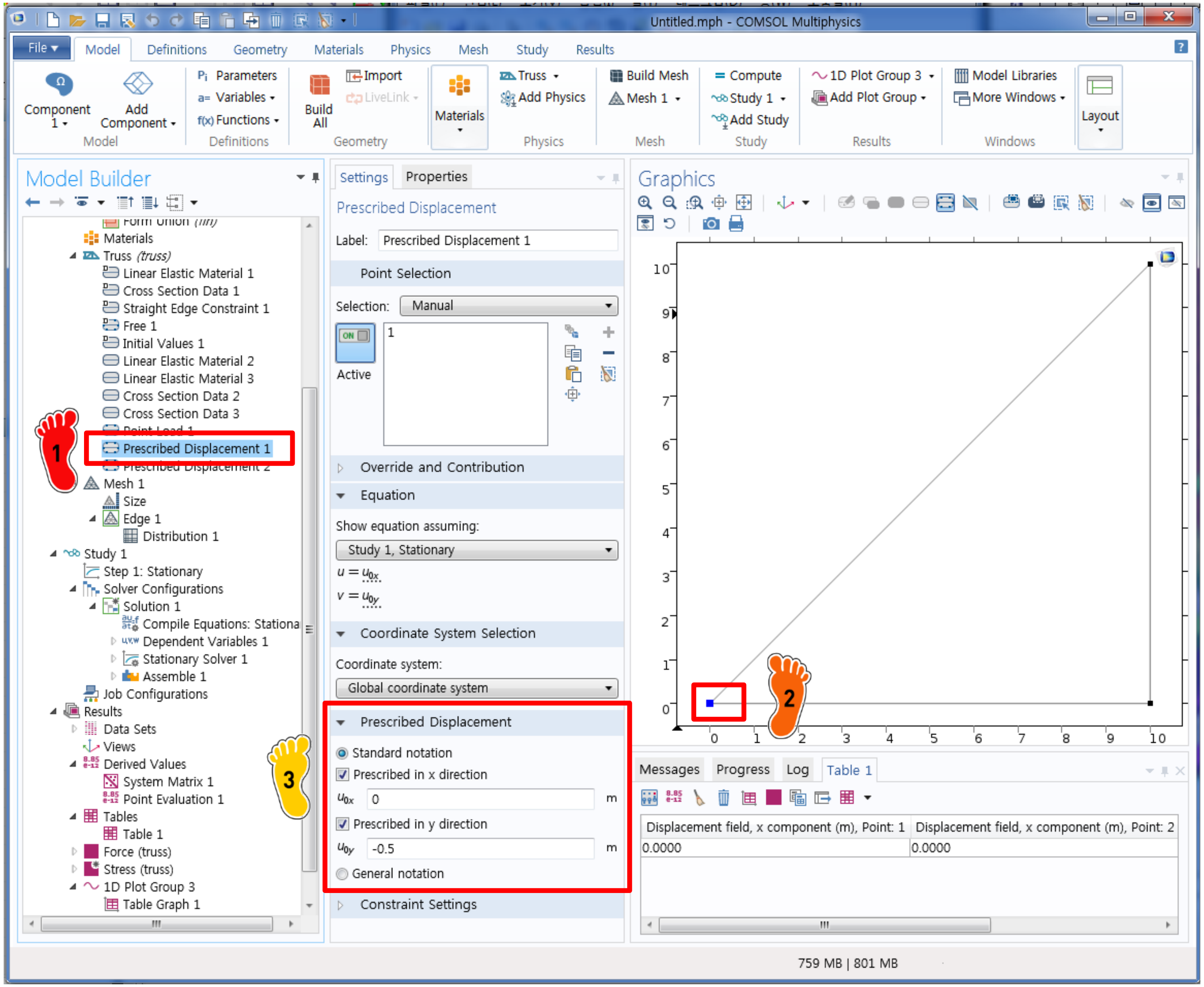


BOUNDARY CONDITION



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

BOUNDARY CONDITION



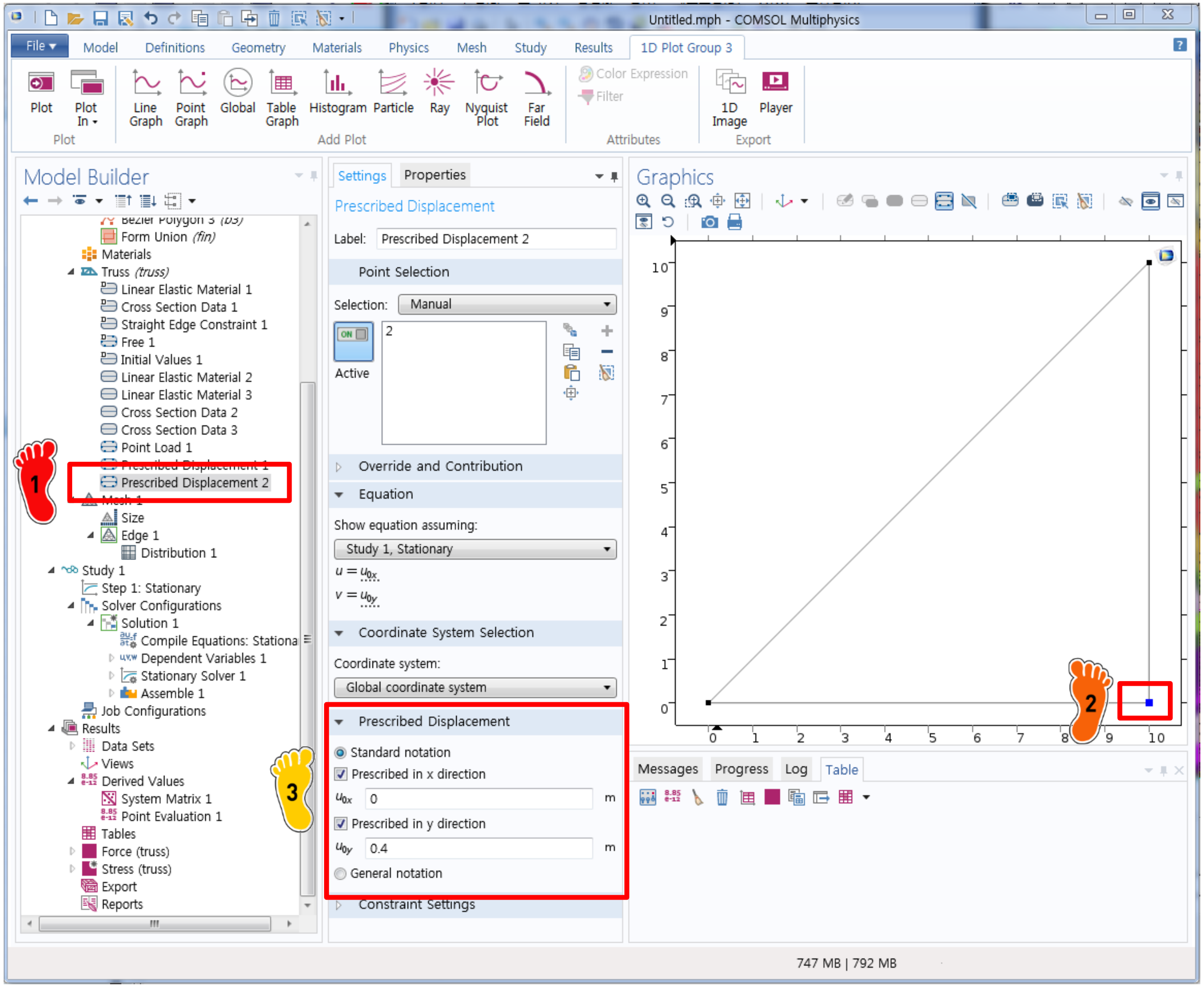
1 첫 번째 Prescribed Displacement 클릭

2 왼쪽 아래 절점 선택

3 u_0 : 0
v_0 : -0.5

입력

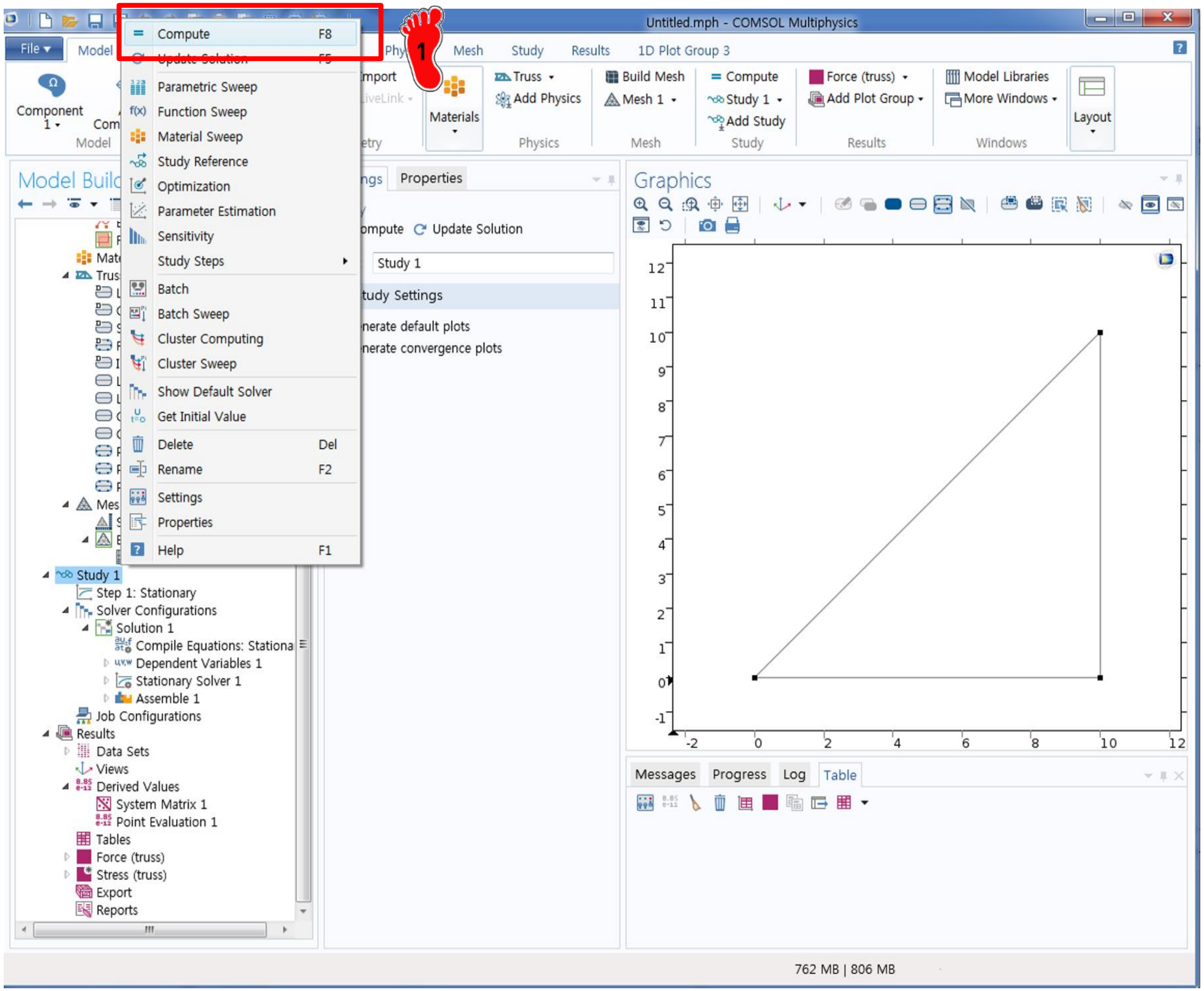
BOUNDARY CONDITION



1. 두 번째 Prescribed Displacement 클릭
2. 오른쪽 아래 절점 선택
3. u_0 : none
   v_0 : 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

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

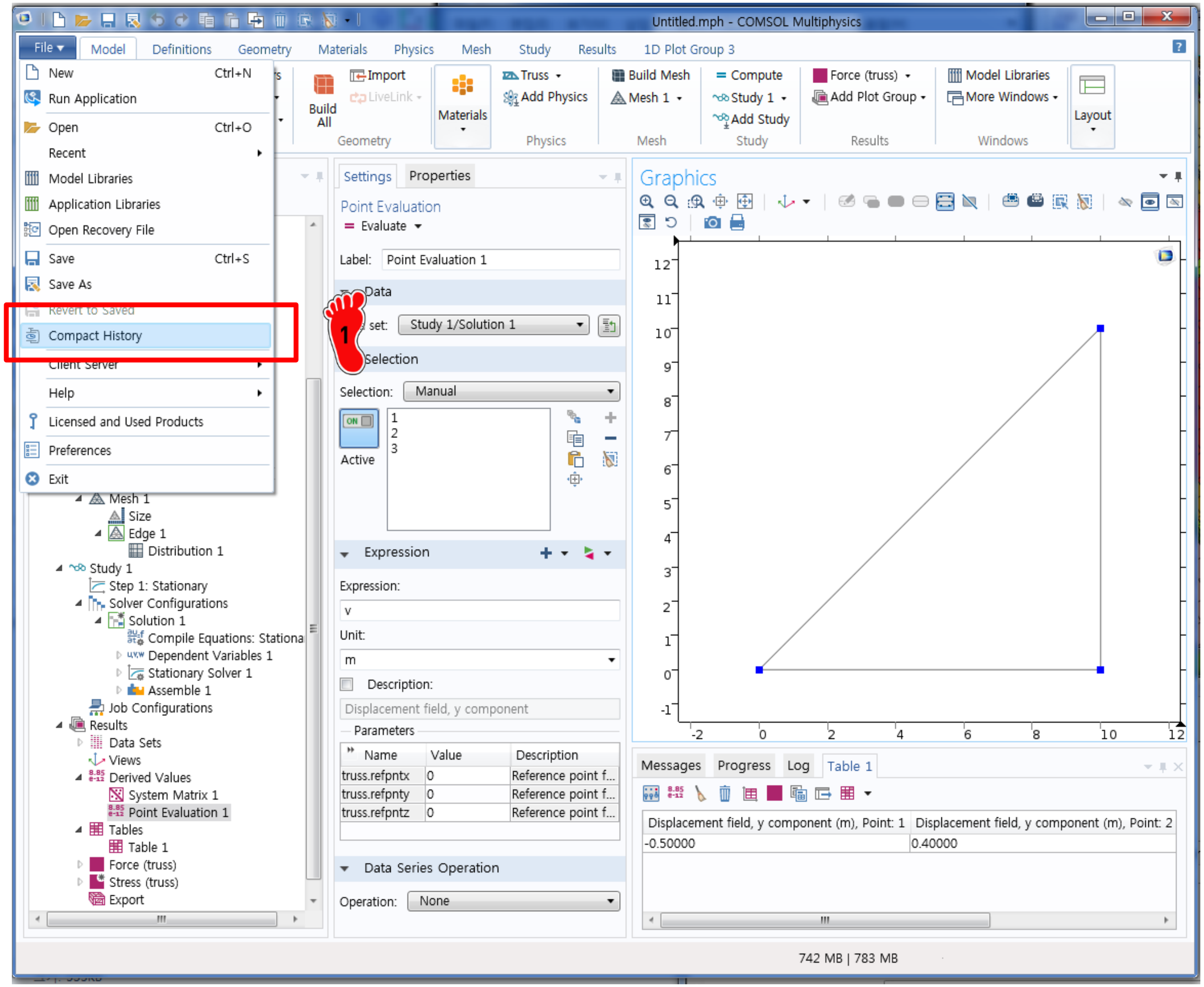
In summary, the only changes to the SDM is in the application of displacement boundary conditions before solve

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

RESET HISTORY

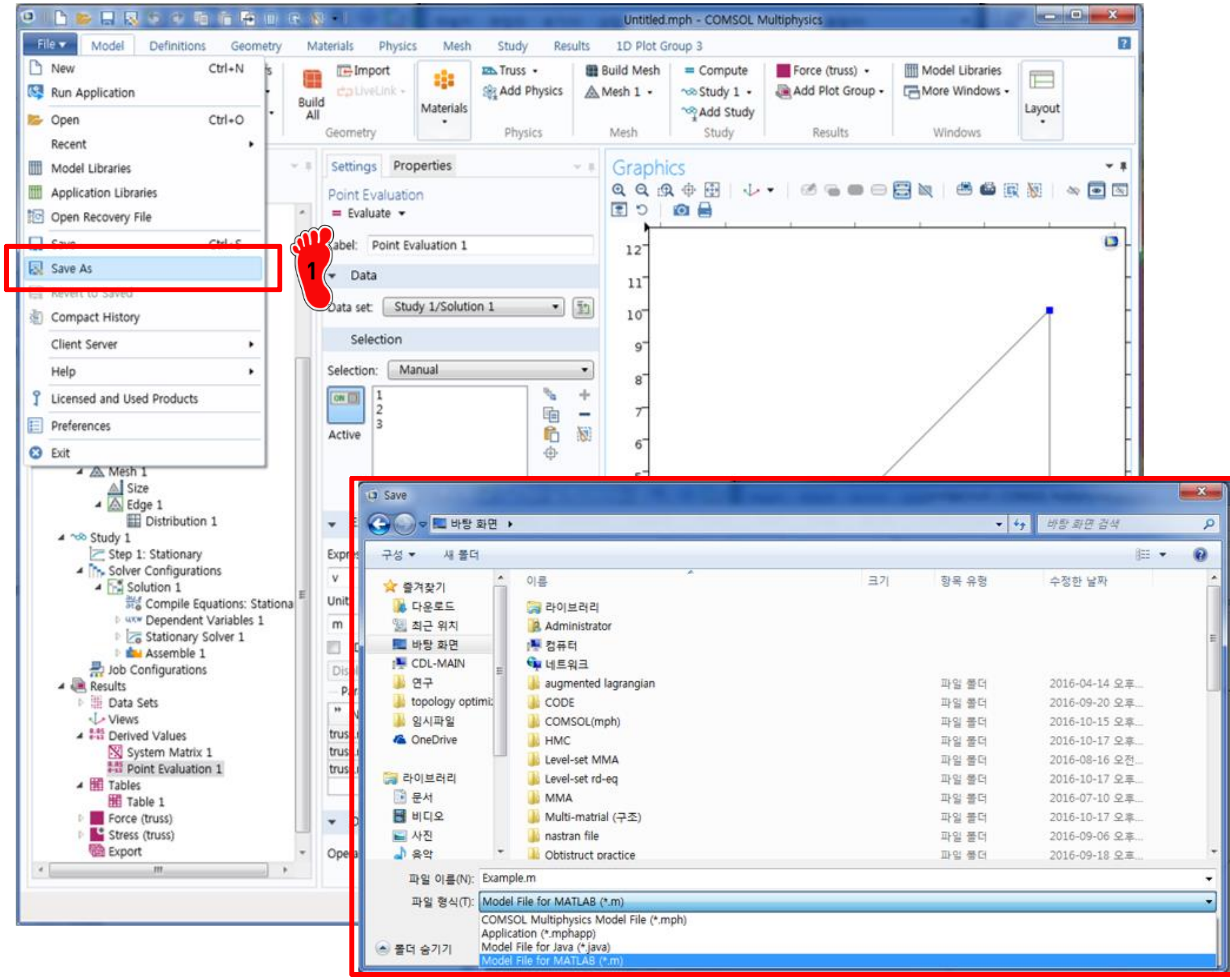


1 Reset History 클릭

COMSOL에서 m-file 로 저장할 때 작업 history 모두 저장됨

따라서 불필요한 history 를 삭제하기 위하여 Compact history 기능을 이용

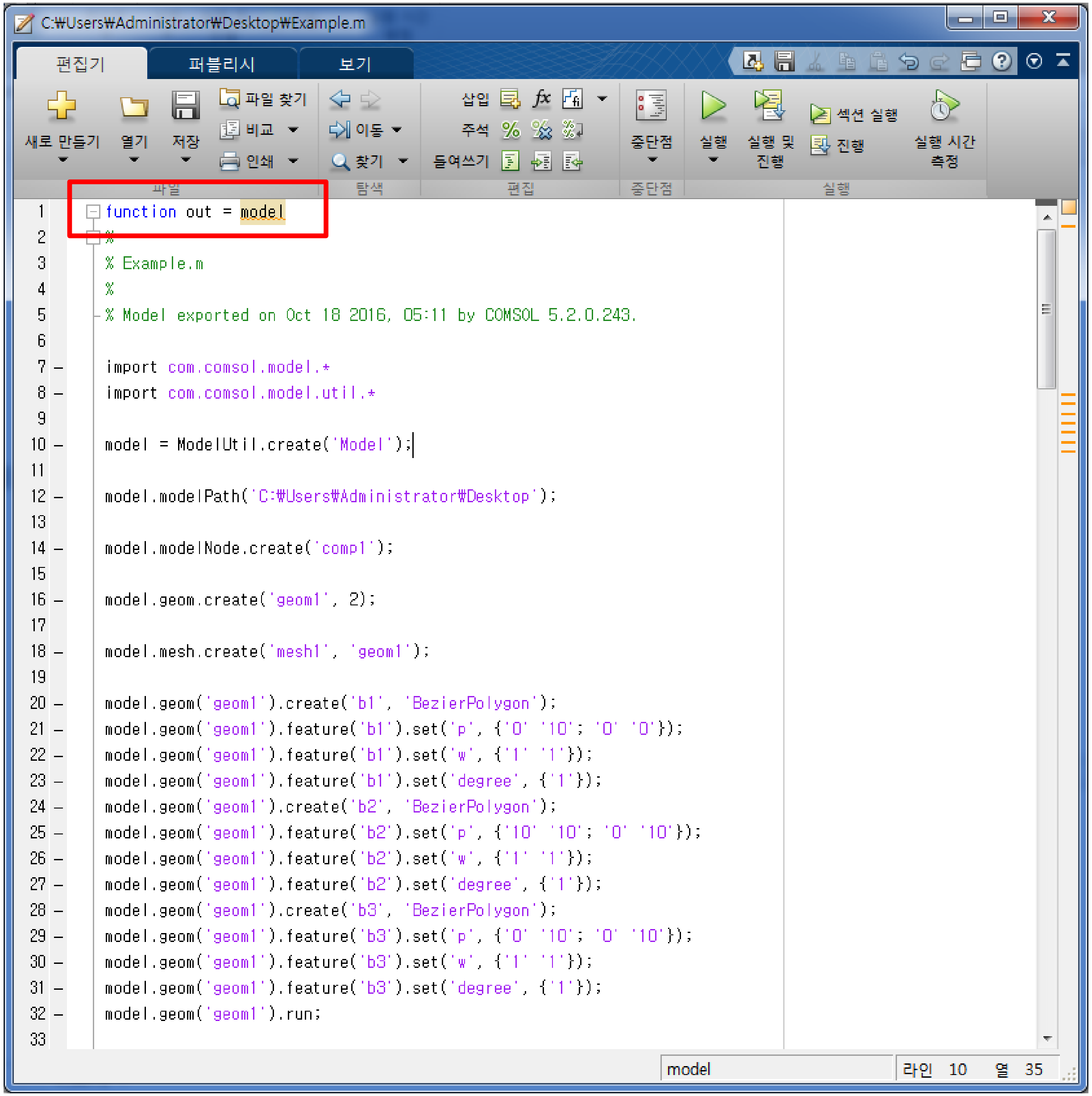
SAVE AS M-FILE



1 Save As 클릭

2 m-file 로 저장

COMSOL M-FILE STRUCTURE

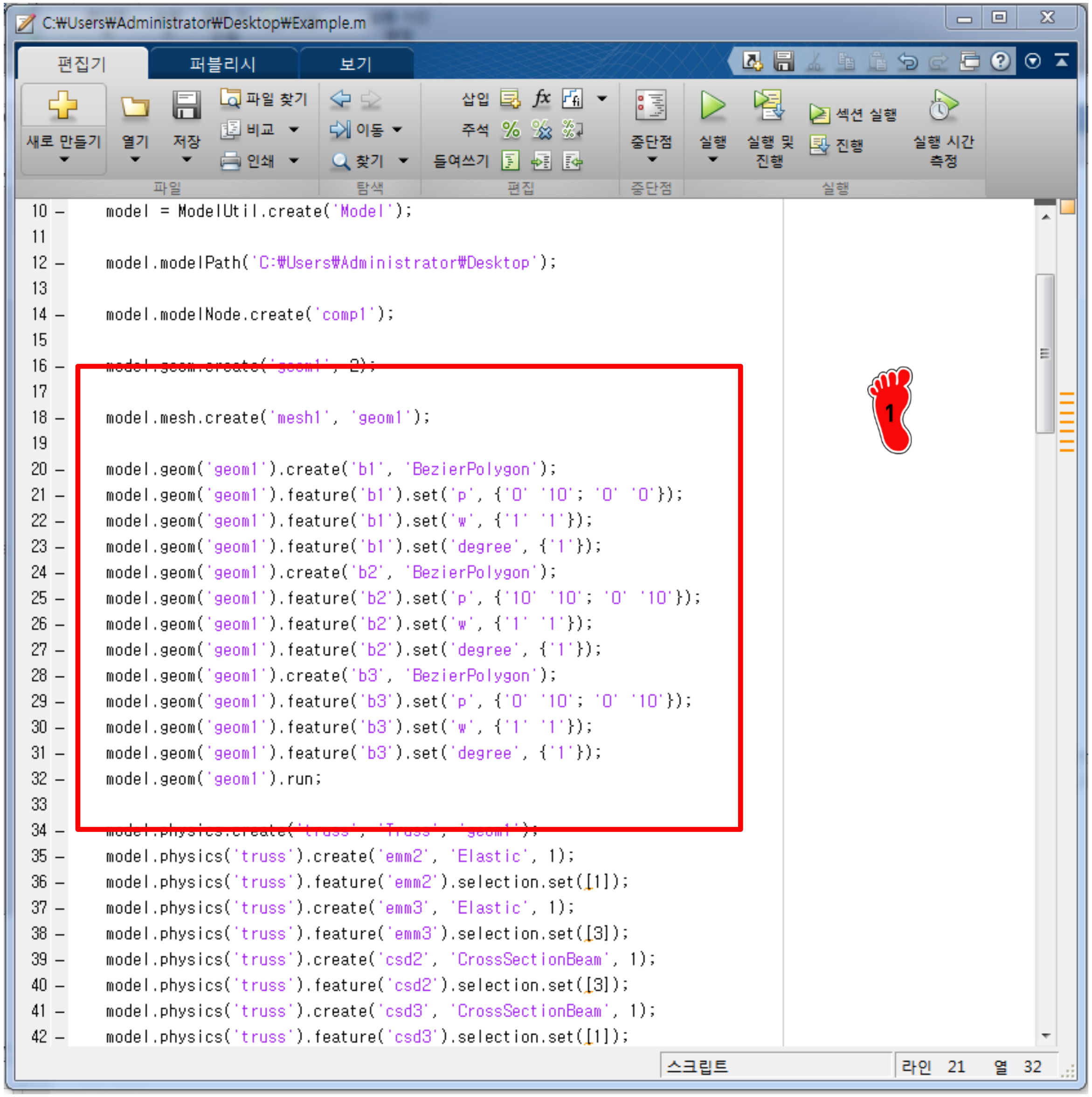


1

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



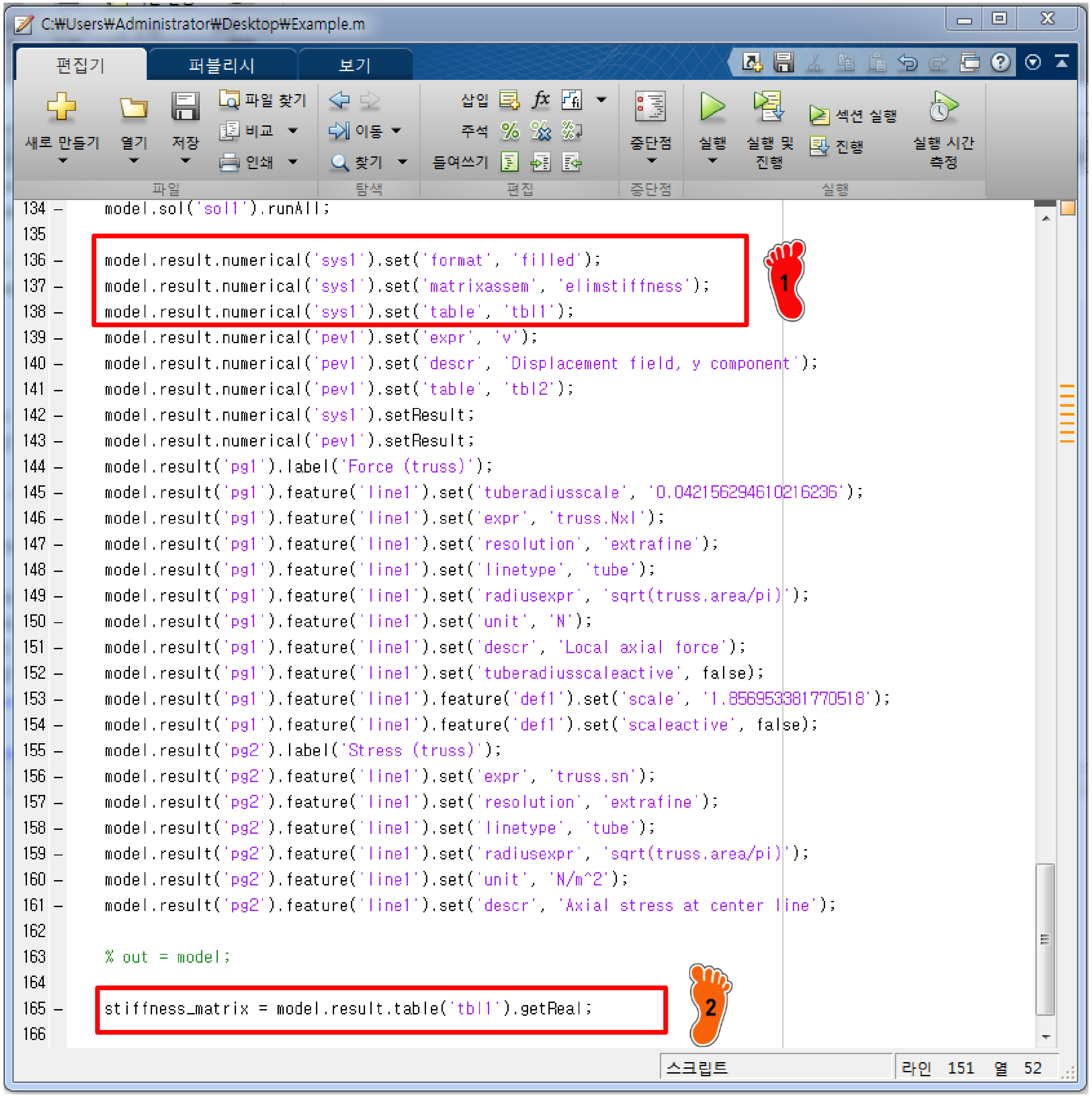
1 model.geom 부분

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

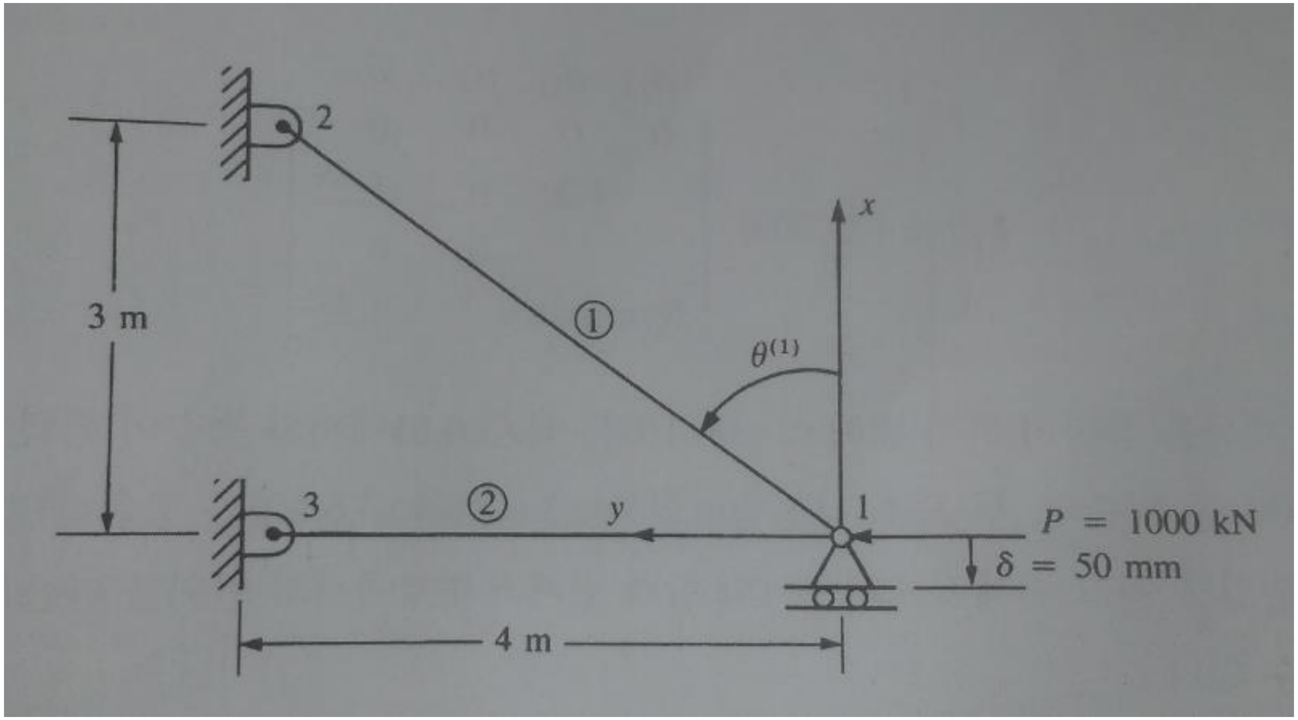


1

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

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

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