

Introduction & ODE by COMSOL

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



CONTENTS

- **Introduction**
- **COMSOL desktop**
- **Mathematics module**
- **ODE examples**
- **Assignment**

- 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

www.comsol.com

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

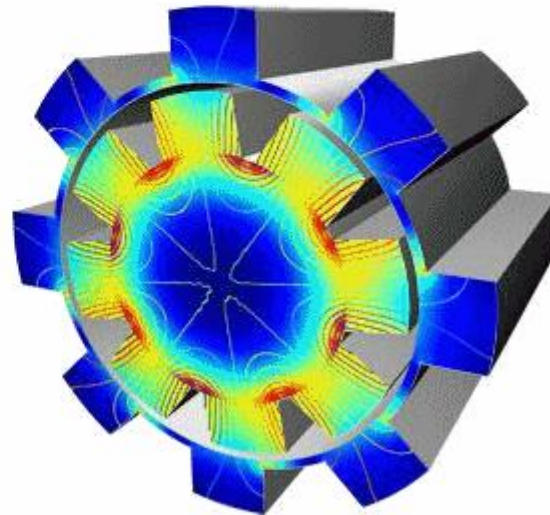
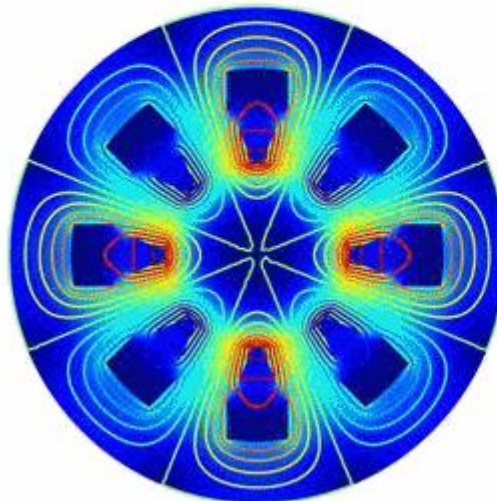
MEMS
Module

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.

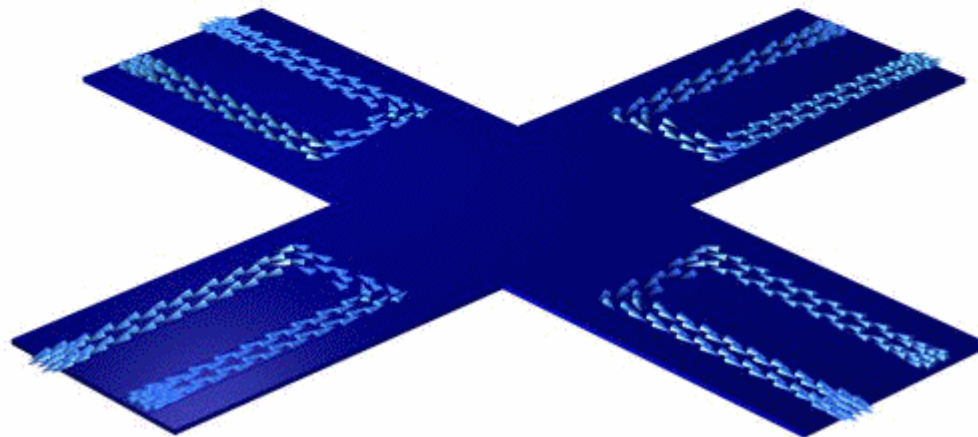


HEAT TRANSFER MODULE

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.



MECHANICAL

Heat Transfer
Module

Structural
Mechanics Module

Nonlinear Structural
Materials Module

Geomechanics
Module

Fatigue
Module

Acoustics
Module

CFD MODULE

FLUID

CFD
Module

Microfluidics
Module

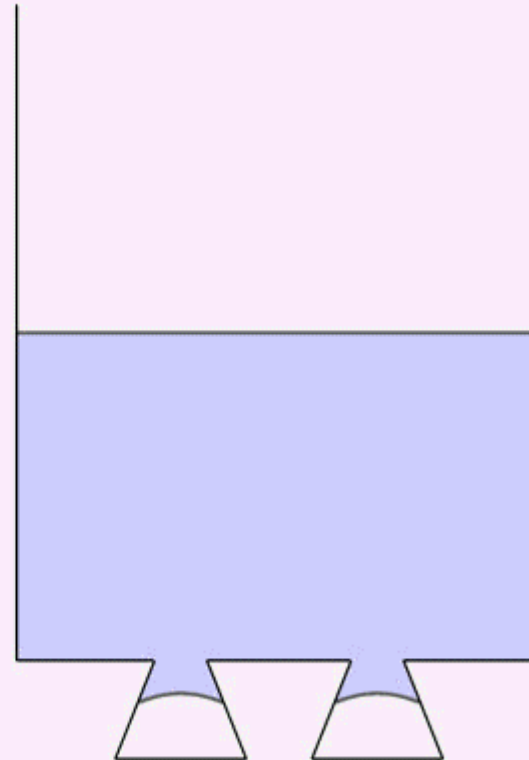
Subsurface Flow
Module

Pipe Flow
Module

Boiling Water

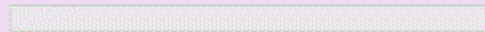
This model studies the film boiling of water. A heat flux above the Leidenfrost point is applied at the surface of two cavities. A layer of vapor is maintained at the hot surface - liquid interface where film-boiling results.

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



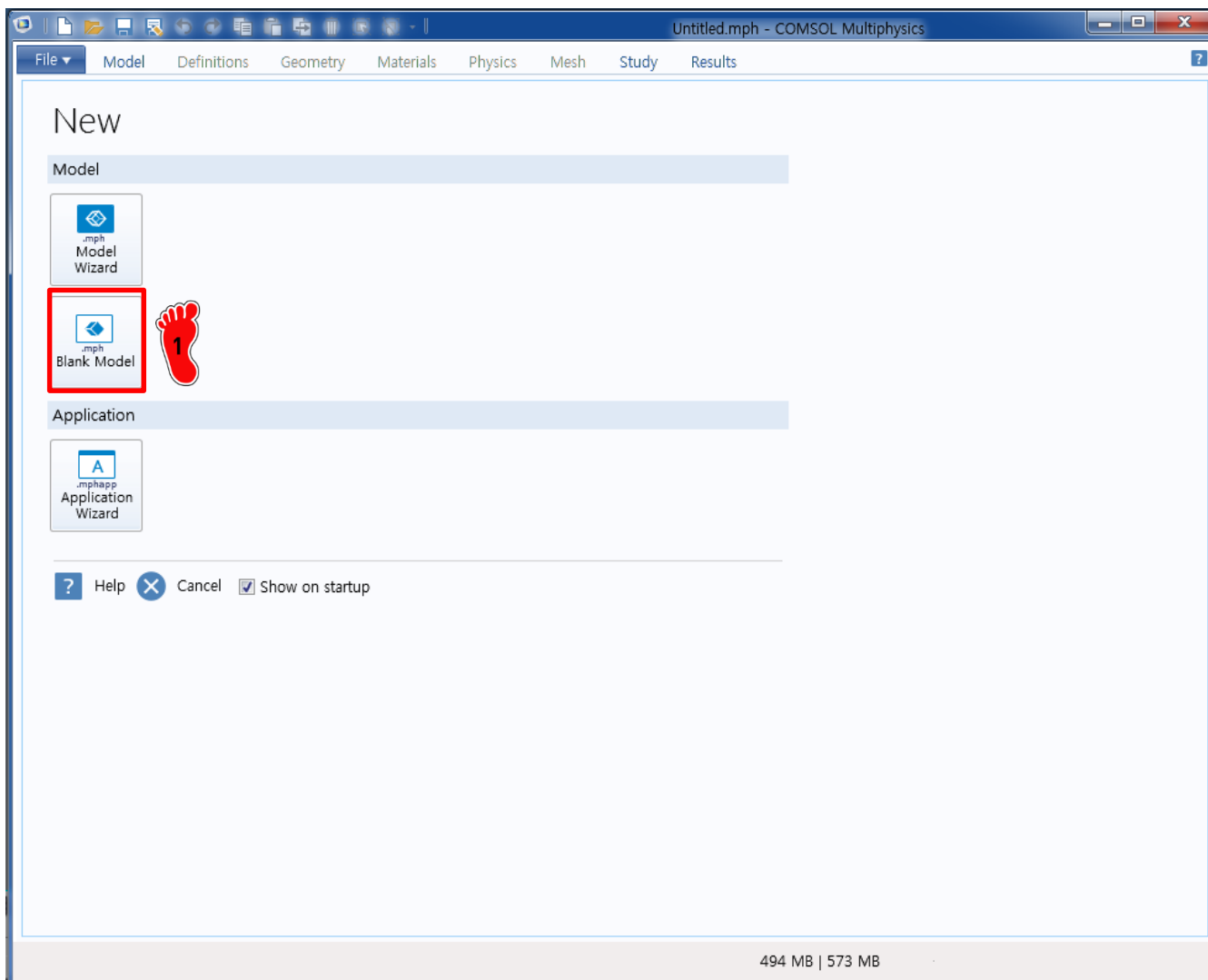
MULTIPHYSICS

COMSOL Multiphysics®



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

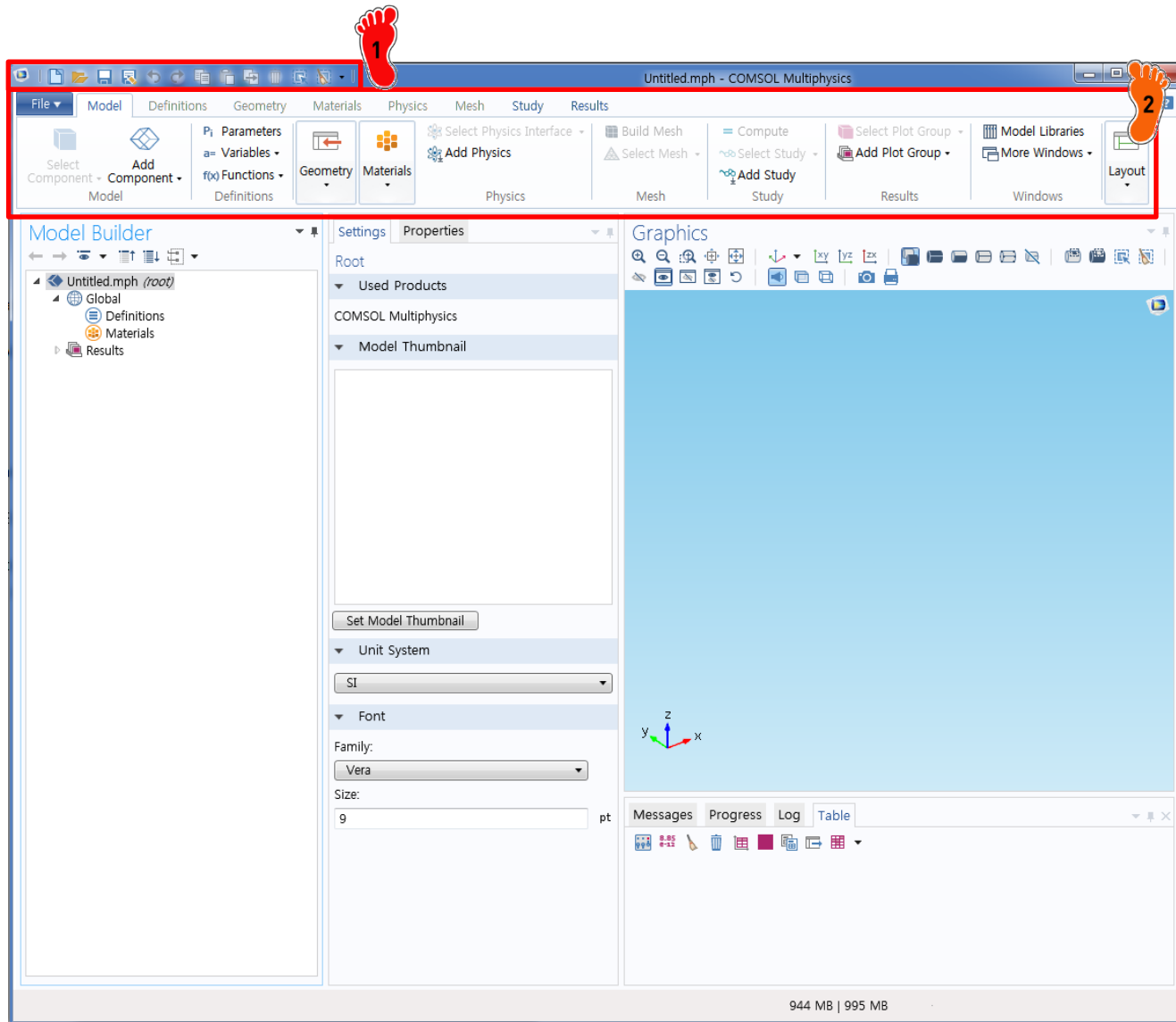
CREATING NEW MODEL



Blank Model

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

TOOLBAR & RIBBON TABS

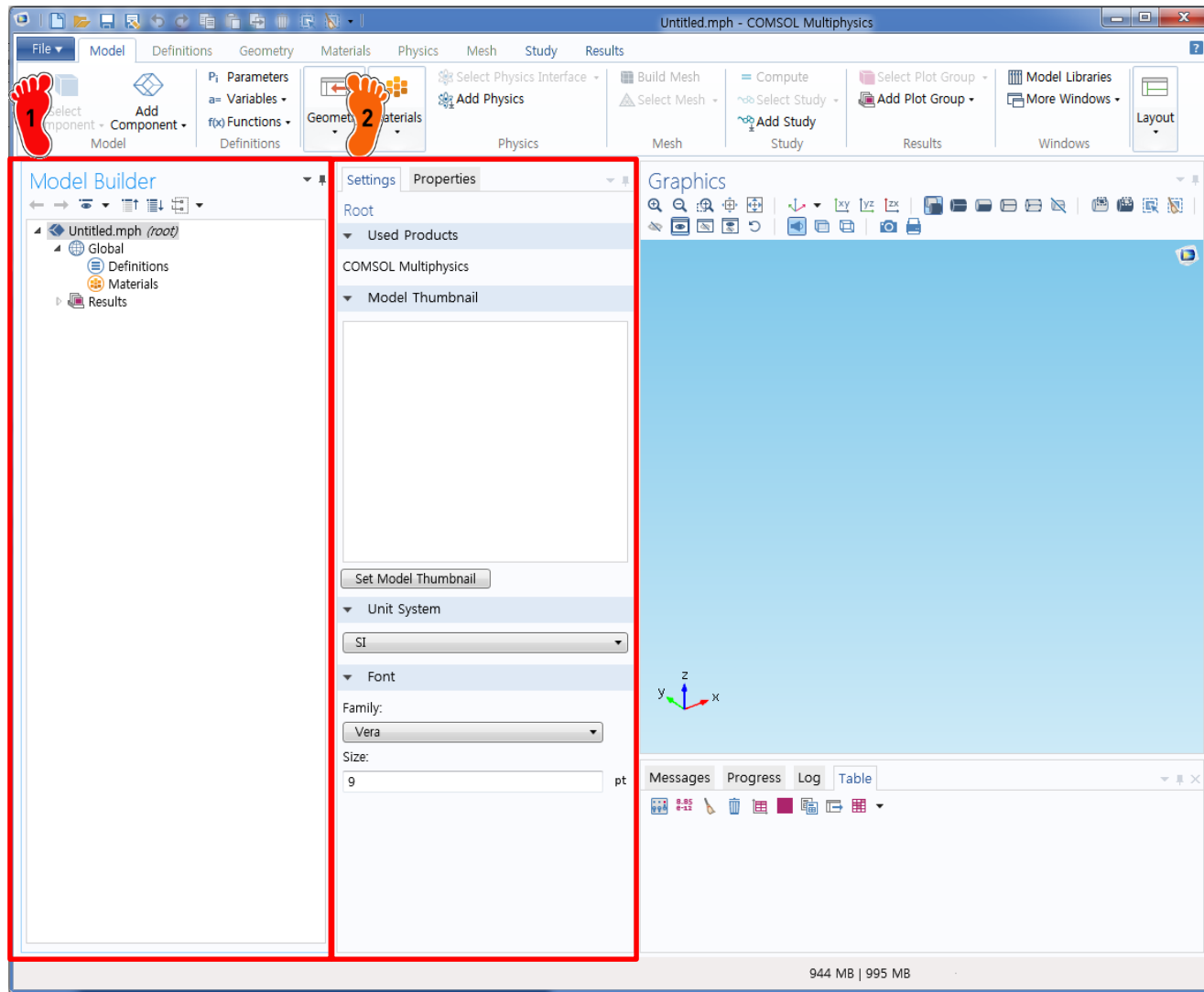


1 Quick Access Toolbar

2 Ribbon tabs

The ribbon tabs on the COMSOL Desktop environment reflect the modeling workflow and give an 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.



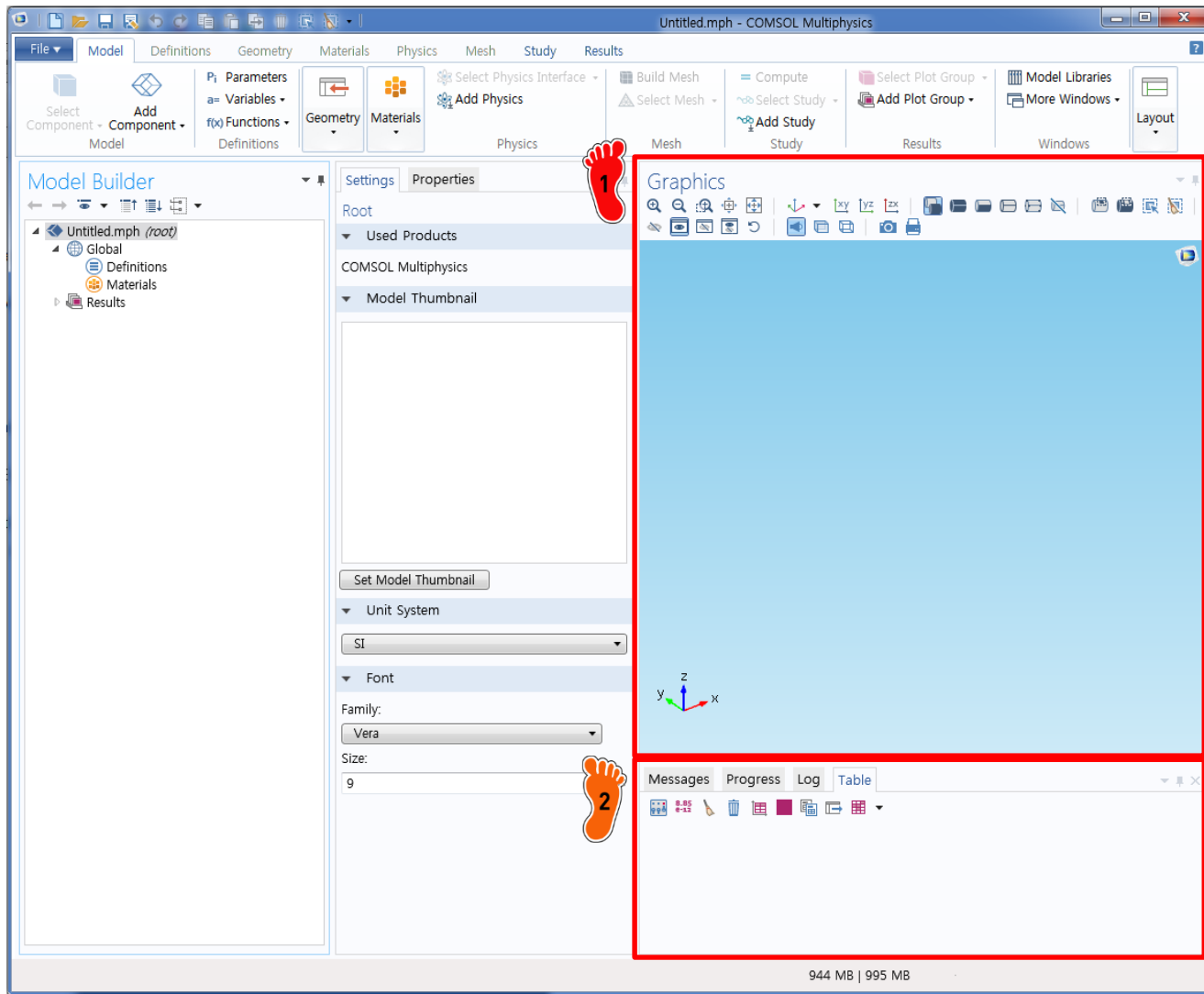
Setting window

It is here that you can edit node setting

Model library window

Easy access to COMSOL and user model libraries.

WINDOWS



Graphics window

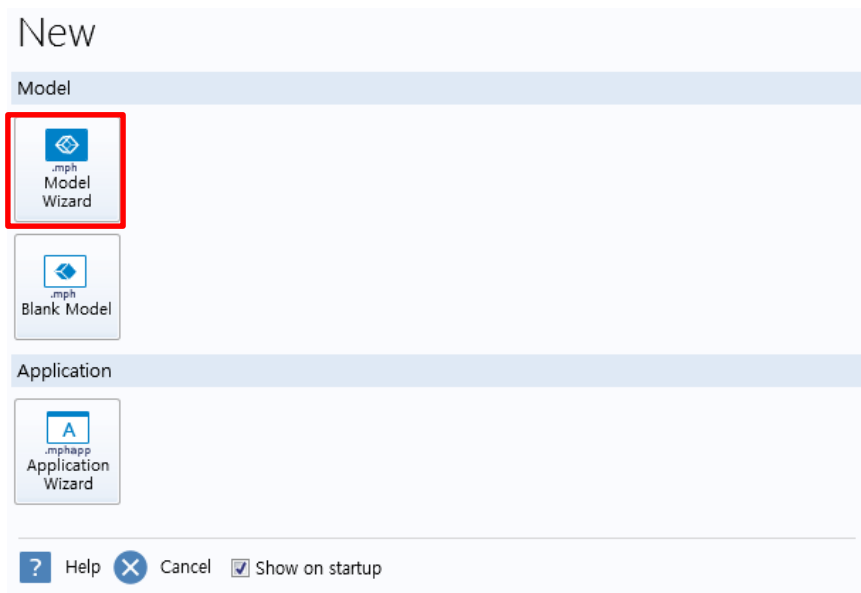
The Graphics window presents interactive graphics for Geometry, Mesh, and Results. Interactions include rotating, panning, zooming, and selecting. It is the default window for most Results and visualizations.



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



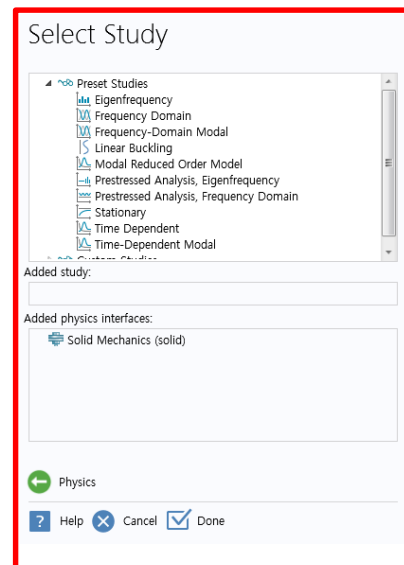
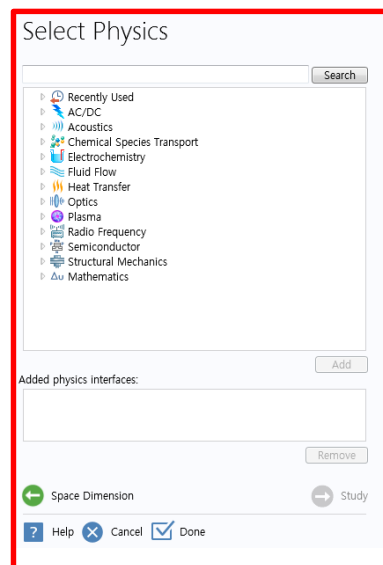
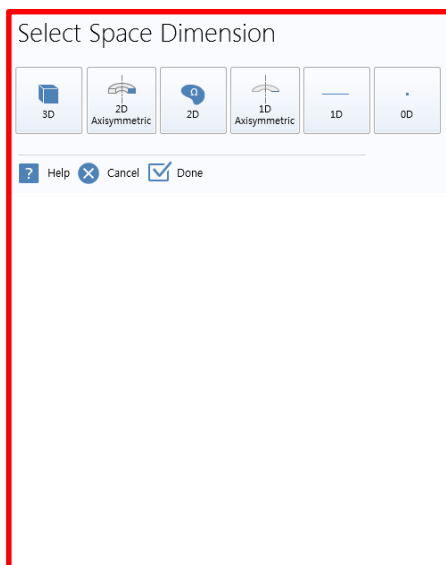
Model Wizard

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

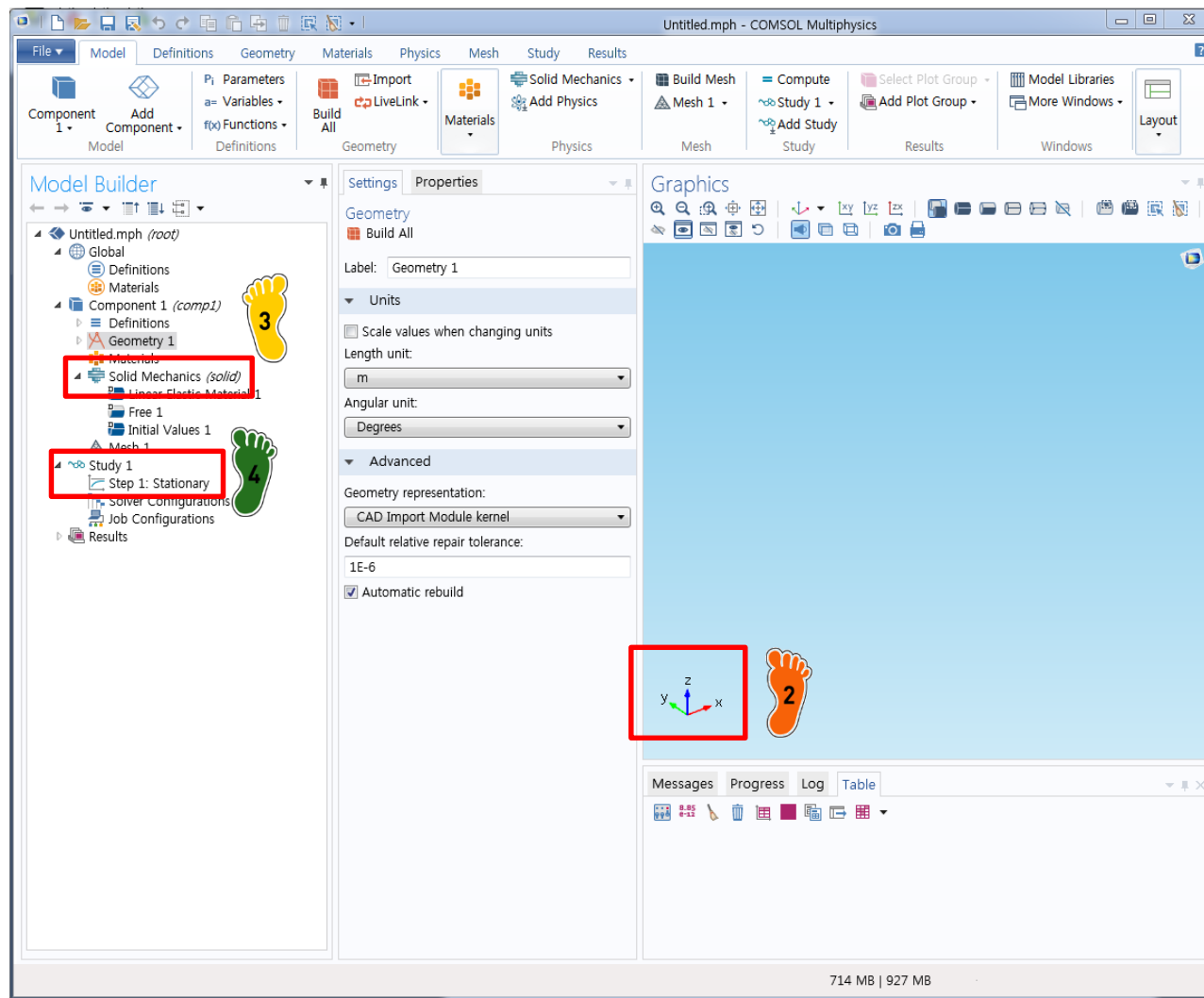
실제 모델링 전 진행 과정

1. Dimension 선택
2. 해석 물리현상(physics) 선택
3. 해석 조건(study) 선택

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



SETTING RESULT



1 전 페이지에서 선택한 setting 결과

2 3차원

3 Solid mechanics

4 Stationary

- **Mathematics module**
 - ✓ **Coefficient form PDE**
 - ✓ **PDE Interfaces**

COEFFICIENT FORM PDE

Equation of coefficient form PDE

$$\left\{ \begin{array}{ll} e_a \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot (-c \nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + a u = f & \text{in } \Omega \\ \mathbf{n} \cdot (c \nabla u + \alpha u - \gamma) + q u = g - h^T \mu & \text{on } \partial \Omega \\ 0 = R & \text{on } \partial \Omega \end{array} \right.$$

where

- Ω is the computational domain—the union of all domains
- $\partial \Omega$ is the domain boundary
- $\mathbf{n}$ is the outward unit normal vector on $\partial \Omega$

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

Diagram illustrating the physical interpretation of the coefficient form PDE terms:

- $e_a \frac{\partial^2 u}{\partial t^2}$: Mass
- $d_a \frac{\partial u}{\partial t}$: Damping/ Mass
- $\nabla \cdot (c \nabla u)$: Diffusion
- $\nabla \cdot (\alpha u)$: Convection
- $\nabla \cdot (-\gamma)$: Source
- $\beta \cdot \nabla u$: Convection
- $a u$: Source
- f : Source
- γ : Conservative Flux
- $h^T \mu$: Absorption

TABLE 16-1: CLASSICAL PDES IN COMPACT AND COMPONENT NOTATION

EQUATION	COMPACT NOTATION	COMPONENT NOTATION (2D)
Laplace's equation	$-\nabla \cdot (\nabla u) = 0$	$-\frac{\partial}{\partial x} \frac{\partial u}{\partial x} - \frac{\partial}{\partial y} \frac{\partial u}{\partial y} = 0$
Poisson's equation	$-\nabla \cdot (c \nabla u) = f$	$-\frac{\partial}{\partial x} \left(c \frac{\partial u}{\partial x} \right) - \frac{\partial}{\partial y} \left(c \frac{\partial u}{\partial y} \right) = f$
Helmholtz equation	$-\nabla \cdot (c \nabla u) + a u = f$	$-\frac{\partial}{\partial x} \left(c \frac{\partial u}{\partial x} \right) - \frac{\partial}{\partial y} \left(c \frac{\partial u}{\partial y} \right) + a u = f$
Heat equation	$d_a \frac{\partial u}{\partial t} - \nabla \cdot (c \nabla u) = f$	$d_a \frac{\partial u}{\partial t} - \frac{\partial}{\partial x} \left(c \frac{\partial u}{\partial x} \right) - \frac{\partial}{\partial y} \left(c \frac{\partial u}{\partial y} \right) = f$
Wave equation	$e_a \frac{\partial^2 u}{\partial t^2} - \nabla \cdot (c \nabla u) = f$	$e_a \frac{\partial^2 u}{\partial t^2} - \frac{\partial}{\partial x} \left(c \frac{\partial u}{\partial x} \right) - \frac{\partial}{\partial y} \left(c \frac{\partial u}{\partial y} \right) = f$
Convection- diffusion equation	$d_a \frac{\partial u}{\partial t} - \nabla \cdot (c \nabla u) + \beta \cdot \nabla u = f$	$d_a \frac{\partial u}{\partial t} - \frac{\partial}{\partial x} \left(c \frac{\partial u}{\partial x} \right) - \frac{\partial}{\partial y} \left(c \frac{\partial u}{\partial y} \right) + \beta_x \frac{\partial u}{\partial x} + \beta_y \frac{\partial u}{\partial y} = f$

PDE INTERFACES

The screenshot shows the COMSOL Multiphysics interface with the following steps highlighted:

- Model Wizard**: The 'New' button is visible in the top left. The 'Select Space Dimension' dialog is open, showing the '1D' option selected with a red footprint icon.
- Module Selection**: The 'Select Physics' dialog is open. The 'Mathematics' folder is expanded, and the 'PDE Interfaces' sub-folder is highlighted with a red box and a red footprint icon. The 'Coefficient Form PDE (c)' option is selected.
- Add**: The 'Add' button is highlighted with a yellow footprint icon.
- Done**: The 'Done' button is highlighted with a green footprint icon.

The 'Review Physics Interface' panel on the right shows the 'Coefficient Form PDE (c)' settings:

- Dependent Variables**: Field name: ; Number of dependent variables: ; Dependent variables:
- Units**: Dependent variable quantity: ; Source term quantity: ; Unit:

At the bottom, the 'Space Dimension' is set to 1D, and the 'Study' button is visible.

1 Model Wizard → 1D 클릭

2 모듈 선택 메뉴에서
Mathematics - PDE
Interfaces 를 클릭

PDE Interfaces 는 PDE 를
풀기 위한 physics 모음

5가지 physics 선택 가능

Coefficient form PDE (선택)
General Form PDE
Wave Form PDE
Weak Form PDE
Lower Dimensions

3 Add 클릭

4 Done 클릭

PDE INTERFACES

Model Builder tree structure:

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Materials
 - Coefficient Form PDE 1** (highlighted with red box and red footprint icon 1)
 - Zero Flux 1
 - Initial Values 1
 - Mesh 1
 - Results

Settings window for Coefficient Form PDE:

- Label: Coefficient Form PDE 1
- Domain Selection: All domains
- Active: ON
- Override and Contribution:
 - Equation:
 - Show equation assuming:
 - $$e_s \frac{\partial^2 u}{\partial t^2} + d_s \frac{\partial u}{\partial t} + \nabla \cdot (-c \nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + au = f$$
 - $$\nabla = \frac{\partial}{\partial x}$$
 - Diffusion Coefficient:
 - c: 1
 - Absorption Coefficient:
 - a: 0 1/m²
 - Source Term:
 - f: 1 1/m²
 - Mass Coefficient:
 - e_a: 0 s²/m²

Graphics window:

- Buttons: Images, Progress, Log
- Status: COMSOL 5.2.0.166. Finalized geometry is empty. Finalized geometry is empty.

766 MB | 936 MB

1 Model builder 창 tree 에서 Coefficient form PDE 선택

2 Equation 탭 메뉴를 확장시키면 독립변수 t, x 와 종속 변수 u 에 대한 수식이 나옴

3 수식 안에 있는 계수들의 값을 입력할 수 있는 창

- **ODE examples**
 - ✓ **Example 25.5**
 - ✓ **Example 28.2: predator-prey model**
 - ✓ **Example 25.14**
 - ✓ **Example 27.10**
 - ✓ **Previous case study I**
 - ✓ **Previous case study II**

EXAMPLE 25.5



ODE 함수 예제



ordinary differential equation

$$y' = 4e^{0.8t} - 0.5y$$

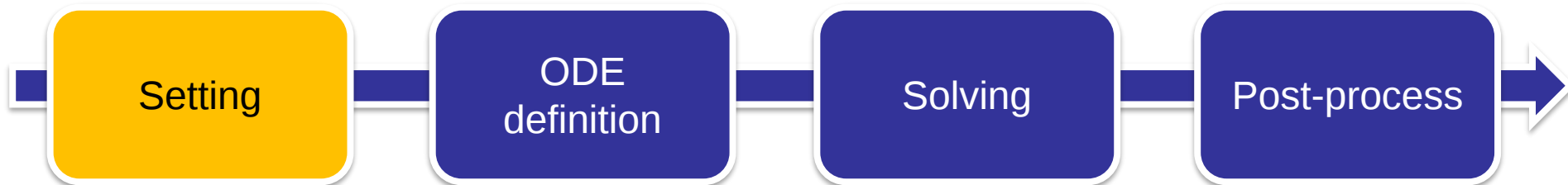
initial condition

$$t = 0, y = 2$$

analytic solution

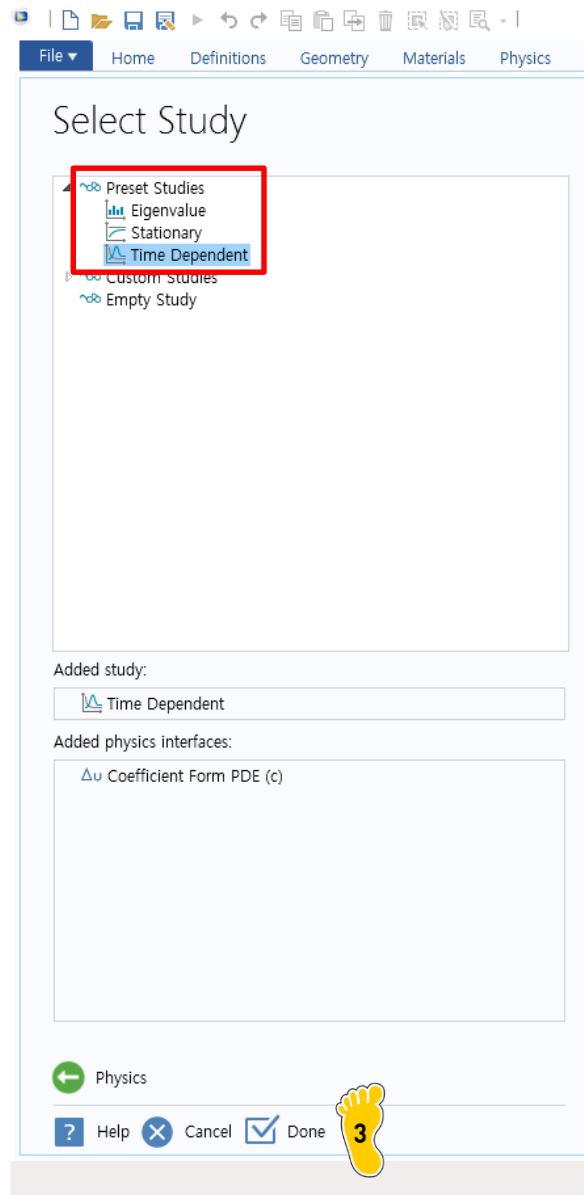
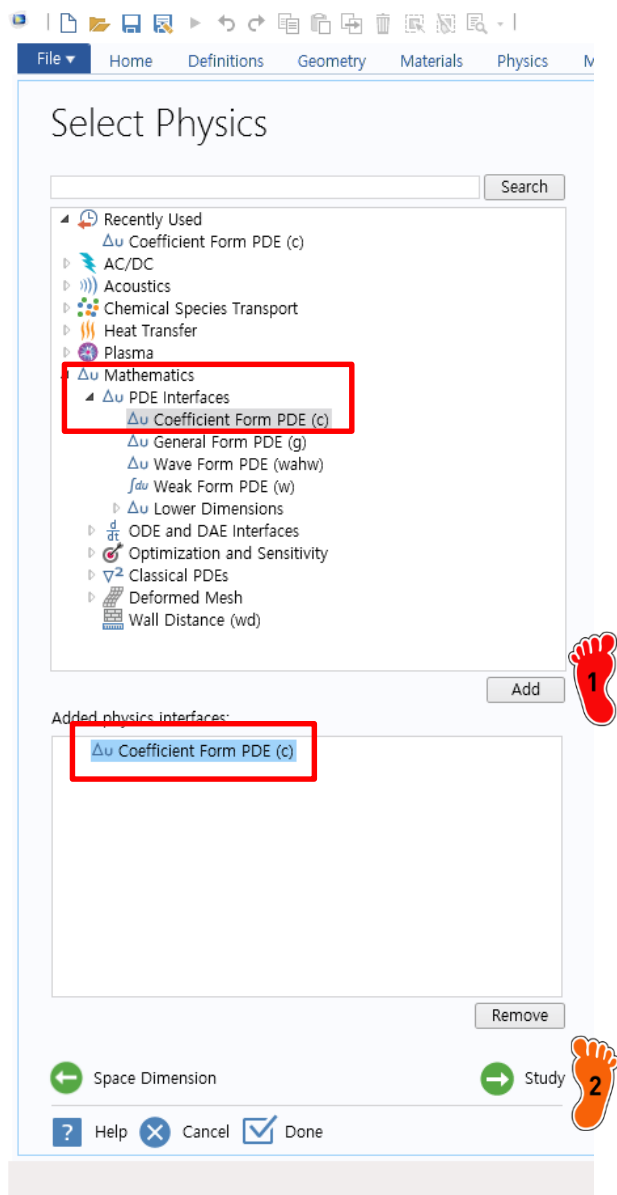
$$y = \frac{4}{1.3} (e^{0.8t} - e^{-0.5t}) + 2e^{-0.5t}$$

ANALYSIS FLOW



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

SETTING

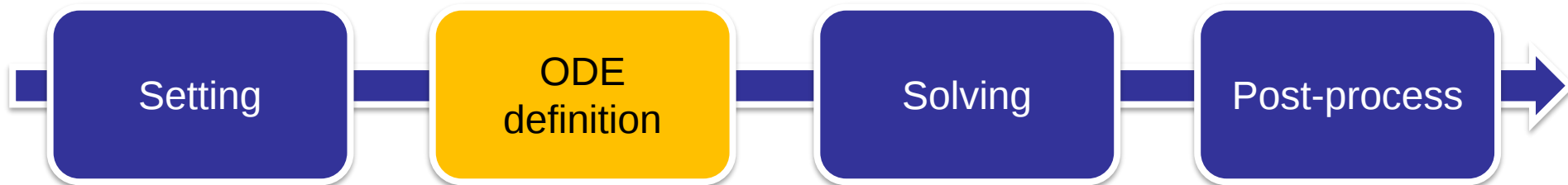


1 Model Wizard → 1D
→ Mathematics - PDE
Interfaces – Coefficient
form PDE 선택
→ Add 클릭

2 Study 클릭

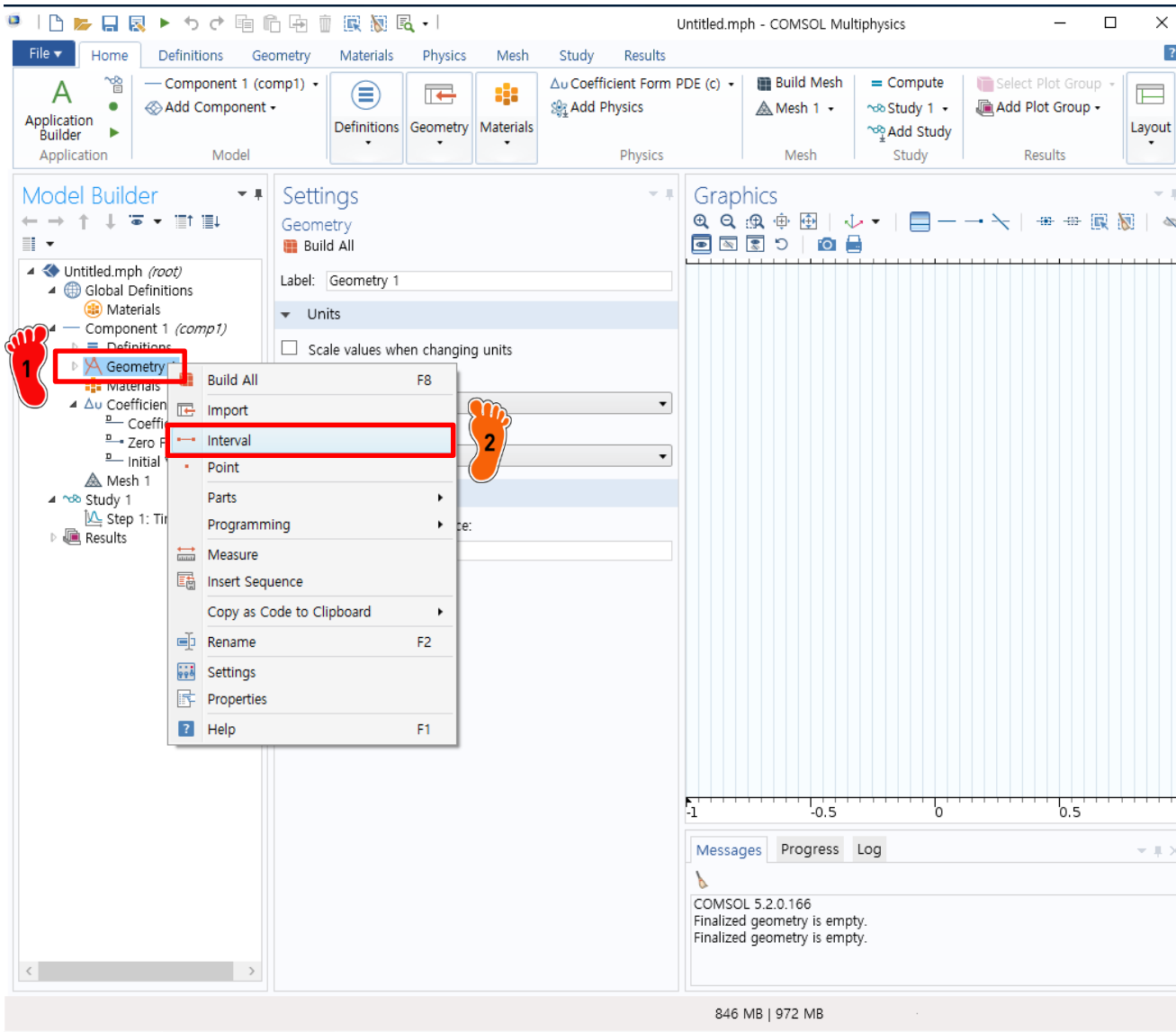
3 Time Dependent 선택 후
Done 클릭

ANALYSIS FLOW



- ✓ Geometry creation
- ✓ Coefficient input
- ✓ Initial value input

GEOMETRY CREATION



Geometry 1 메뉴를 마우스 우클릭



Interval 클릭

GEOMETRY CREATION

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Model Builder

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Interval 1 (I1)
 - Form Union (fin)
 - Materials
 - Coefficient Form PDE (c)
 - Coefficient Form PDE
 - Zero Flux 1
 - Initial Values 1
 - Mesh 1
 - Study 1
 - Step 1: Time Dependent
 - Results

Settings

Interval

Build Selected Build All Objects

Label: Interval 1

Interval

Number of intervals: One

Left endpoint: 0 m

Right endpoint: 1 m

Selections of Resulting Entities

Contribute to: None New

☐ Resulting objects selection

Show in physics: Domain selection

Graphics

0 0.2 0.4 0.6 0.8 1

Messages Progress Log

COMSOL 5.2.0.166
Finalized geometry is empty.
Finalized geometry is empty.

845 MB | 973 MB

1 Left endpoint 에 "0"
Right endpoint 에 "1"
값을 입력

2 Build selected 클릭

COEFFICIENT INPUT

1

2

3

1 Coefficient Form PDE 1 선택

2 계수 값 입력 (나머지 0)

$$d_a = 1$$

$$f = 4e^{0.8t} - 0.5u$$

$$e \frac{\partial^2 u}{\partial t^2} + d_a \frac{\partial u}{\partial t} + \nabla \cdot (-c \nabla u - \alpha u + \gamma) + \beta \cdot \nabla u + a u = f \leftrightarrow \frac{\partial u}{\partial t} = 4e^{0.8t} - 0.5u$$

INITIAL VALUE INPUT

The screenshot shows the COMSOL Multiphysics interface for a new model. The **Model Builder** on the left lists the model structure: **Untitled.mph (root)** > **Global Definitions** > **Materials** > **Component 1 (comp1)** > **Definitions** > **Geometry 1** > **Interval 1 (f1)** > **Form Union (fin)** > **Materials** > **Coefficient Form PDE (c)** > **Coefficient Form PDE 1** > **Zero Flux 1** > **Initial Values 1**. A red box highlights **Initial Values 1** in the Model Builder and in the **Settings** pane. The **Settings** pane shows the **Initial Values** section with the following inputs:

- Label:** Initial Values 1
- Domain Selection:** All domains
- Active:** 1
- Initial value for u:** 2
- Initial time derivative of u:** 0

The **Graphics** pane on the right shows a plot of the initial value for u, which is a constant 2 across the domain. The **Messages** pane at the bottom shows the following text:

```
COMSOL 5.2.0.166
Finalized geometry is empty.
Finalized geometry is empty.
Finalized geometry has 1 domain and 2 boundaries.
```

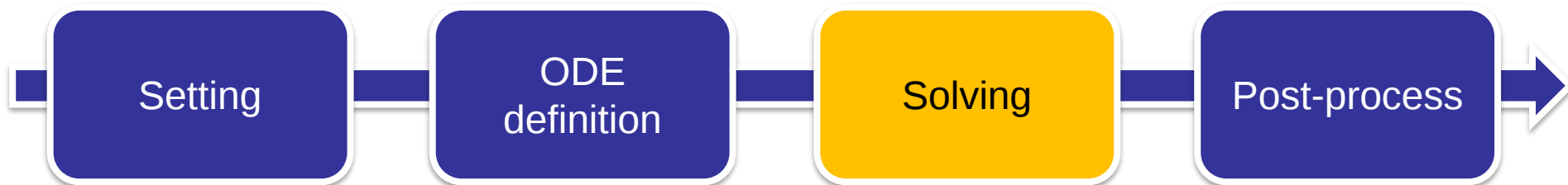


Initial Values 1 메뉴 클릭



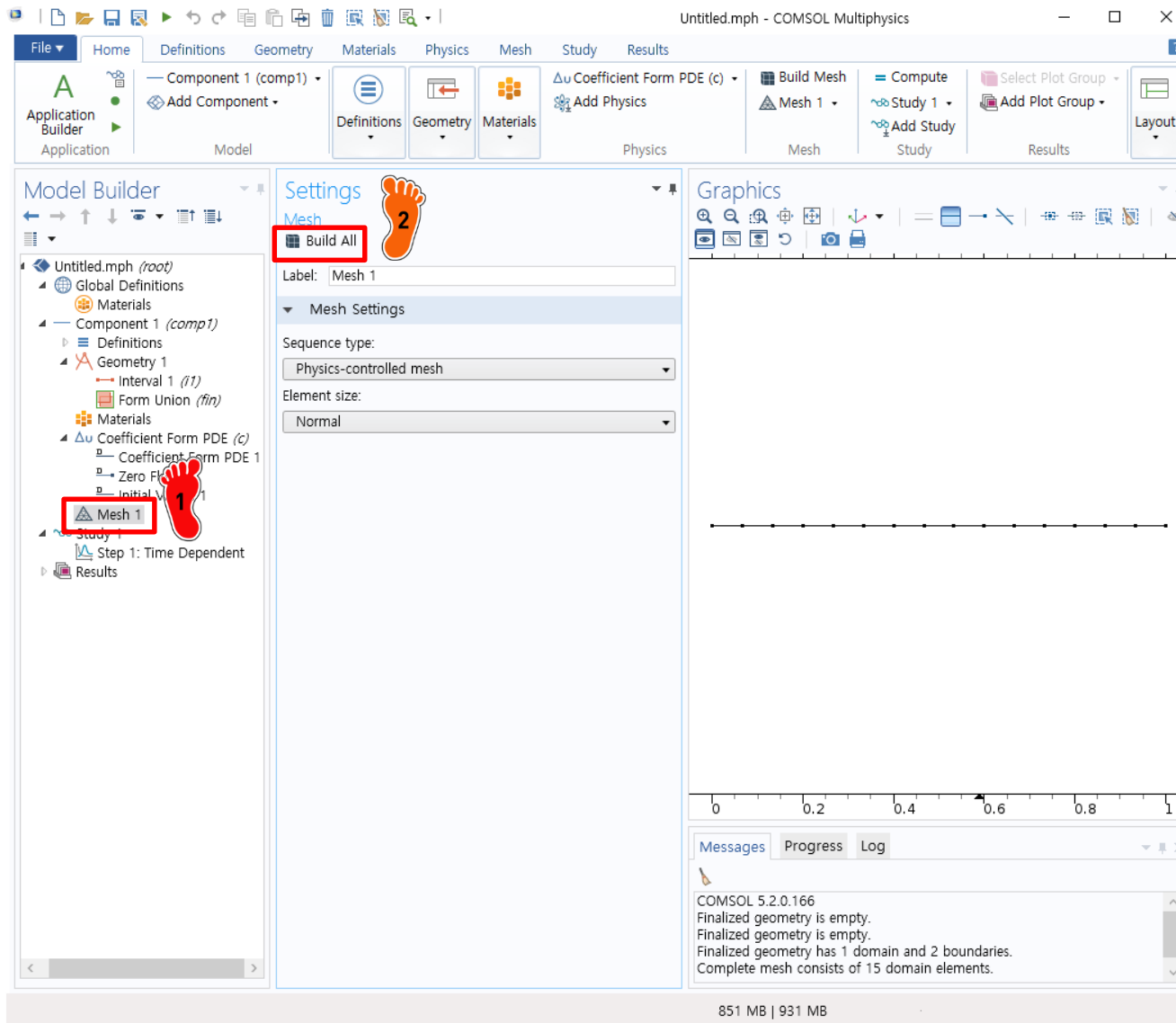
u 창에 "2" 입력

ANALYSIS FLOW



- ✓ Mesh creation
- ✓ Time range input
- ✓ Compute

MESH CREATION



1 Mesh 1 메뉴 클릭

2 Build All 클릭

TIME RANGE INPUT

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Model Builder

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Interval 1 (it)
 - Form Union (fin)
 - Materials
 - Coefficient Form PDE (c)
 - Coefficient Form PDE 1
 - Zero Flux 1
 - Initial Values 1
 - Mesh 1
 - Study 1
 - Step 1: Time Dependent
 - Results

Settings

Time Dependent

Compute

Label: Time Dependent

Study Settings

Time unit: s

Times: range(0,1,4) s

Relative tolerance: ☐ 0.01

Results While Solving

Physics and Variables Selection

Modify physics tree and variables for study step

Physics interface Solve for Discretization

Coefficient Form PDE (c) ☒ Physics settings: ▾

Values of Dependent Variables

Mesh Selection

Study Extensions

Graphics

0 0.2 0.4 0.6 0.8 1

Messages Progress Log

COMSOL 5.2.0.166
 Finalized geometry is empty.
 Finalized geometry is empty.
 Finalized geometry has 1 domain and 2 boundaries.
 Complete mesh consists of 15 domain elements.

843 MB | 936 MB

1 Study 1 → Step 1: Time Dependent 메뉴 클릭

2 Times: 창에 "range(0,1,4)" 입력

COMPUTE

The screenshot displays the COMSOL Multiphysics software interface. The top toolbar contains the 'Compute' button, which is highlighted with a red box and a red foot icon labeled '2'. The Model Builder on the left shows the 'Study 1' menu item, also highlighted with a red box and a red foot icon labeled '1'. The Settings panel in the center shows the 'Study' settings for 'Study 1', including options for 'Generate default plots' and 'Generate convergence plots'. The Graphics panel on the right shows a plot of the solution, with a red foot icon labeled '1' indicating the selection of the 'Study 1' menu item. The Messages panel at the bottom shows the status of the computation, including the version number (COMSOL 5.2.0.166) and the number of domain elements (15).



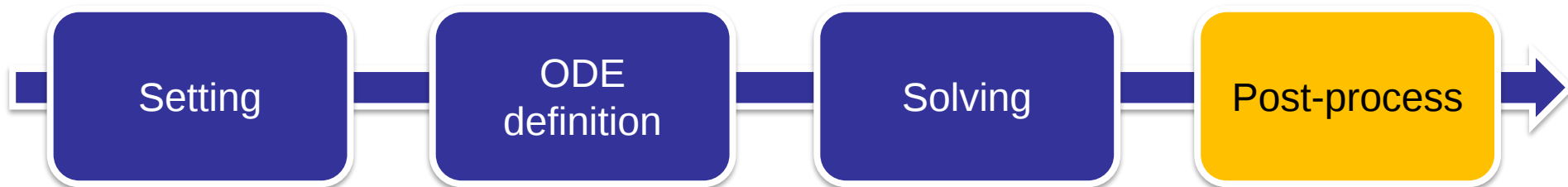
Study 1 메뉴 선택



Compute 클릭

850 MB | 944 MB

ANALYSIS FLOW



✓ Result plot

RESULT PLOT

The screenshot shows the COMSOL Multiphysics interface with the '1D Plot Group 1' context menu open. The menu options include Plot, Plot In, Line Graph, Point Graph, Global, Table Graph, Histogram, Ray, Nyquist Plot, Far Field, Octave Band Plot, Mesh, Add Image to Export, Copy as Code to Clipboard, Duplicate, Delete, Disable, Rename, Settings, and Properties. The 'Point Graph' option is highlighted with a red box and a red footprint icon labeled '2'. The '1D Plot Group 1' option in the left sidebar is also highlighted with a red box and a red footprint icon labeled '1'. The main plot area displays a line graph titled 'Line Graph: Dependent variable u (1)' with the y-axis labeled 'Dependent variable u (1)' and the x-axis labeled 'x-coordinate (m)'. The graph shows a horizontal line at y=15. The bottom status bar indicates 887 MB | 958 MB.

1D Plot Group 1 우클릭

Point Graph 클릭

RESULT PLOT

The screenshot displays the COMSOL Multiphysics interface for 'Untitled.mph - COMSOL Multiphysics'. The top toolbar includes icons for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, and Results. The '1D Plot Group 1' tab is active, showing a 'Color Expression' filter and '1D Image' and 'Animation' export options.

The 'Model Builder' on the left shows the hierarchy: Untitled.mph (root) > Global Definitions > Materials > Component 1 (comp1) > Definitions > Geometry 1 > Interval 1 (I1) > Form Union (fin) > Materials > Coefficient Form PDE (c) > Coefficient Form PDE 1 > Zero Flux 1 > Initial Values 1 > Mesh 1 > Study 1 > Step 1: Time Dependent > Solver Configurations > Results > Data Sets > Derived Values > Tables > 1D Plot Group 1 > Line Graph 1 > Point Graph 1 > Export > Reports.

The 'Settings' pane for 'Point Graph 1' is shown. The 'Label' is 'Point Graph 1'. Under 'Data', 'Data set' is 'From parent'. Under 'Selection', 'Selection' is 'Manual'. The 'Active' checkbox is checked, and a red box highlights the 'Active' checkbox and the '1' in the 'Selection' dropdown. The 'y-Axis Data' section shows 'Expression' as 'u', 'Unit' as '1', and 'Description' as 'Dependent variable u'. The 'x-Axis Data' section shows 'Parameter' as 'Time' and 'Unit' as 's'.

The 'Graphics' pane shows 'Convergence Plot 1' with a red box highlighting the plot area. The plot area shows a horizontal line at y=1, with a red box highlighting the line and a red footprint icon labeled '1'.

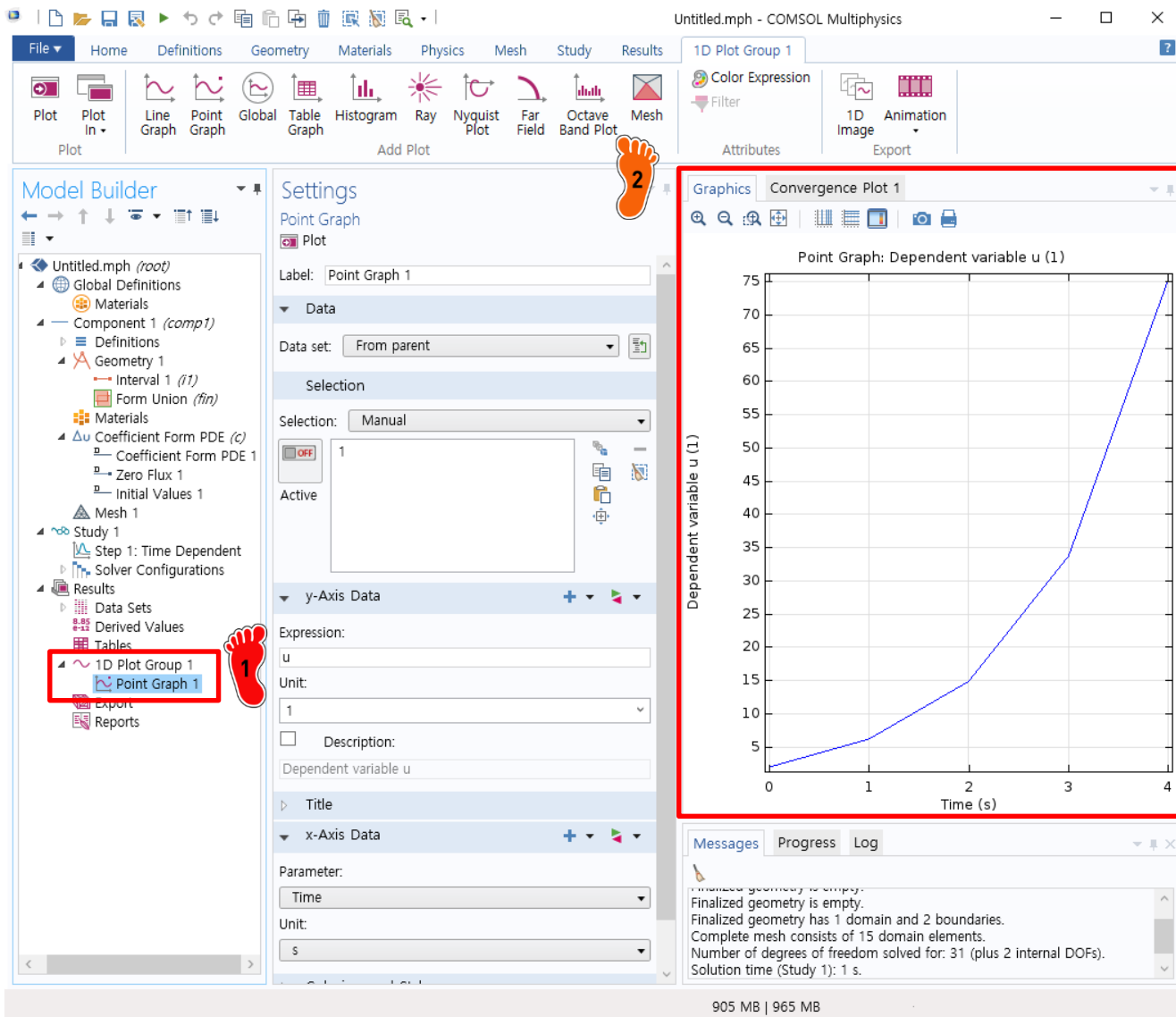
The 'Messages' pane at the bottom shows the following text: 'Finalized geometry is empty. Finalized geometry is empty. Finalized geometry has 1 domain and 2 boundaries. Complete mesh consists of 15 domain elements. Number of degrees of freedom solved for: 31 (plus 2 internal DOFs). Solution time (Study 1): 1 s.'

1 왼쪽 point 클릭

2 Selection에 "1" 확인

3 Plot 클릭

RESULT PLOT



1 기존 Line Graph 1 삭제
(우클릭 → Delete 또는
선택 후 "Del" 키)

2 결과 그래프 확인

- **ODE examples**
 - ✓ **Example 25.5**
 - ✓ **Example 28.2: predator-prey model**
 - ✓ **Example 25.14**
 - ✓ **Example 27.10**
 - ✓ **Previous case study I**
 - ✓ **Previous case study II**

EXAMPLE 28.2

[predator – prey model]

nonlinear ordinary differential equations

$$\begin{cases} \frac{dy_1}{dt} = ay_1 - by_1y_2 \\ \frac{dy_2}{dt} = -cy_2 + dy_1y_2 \end{cases}$$

$$a = 1.2, b = 0.6, c = 0.8, d = 0.3$$

initial condition

$$t = 0, y_1 = 2, y_2 = 1$$



Predator-prey model developed by the Italian mathematician Vito Volterra and the American biologist Alfred J. Lotka.

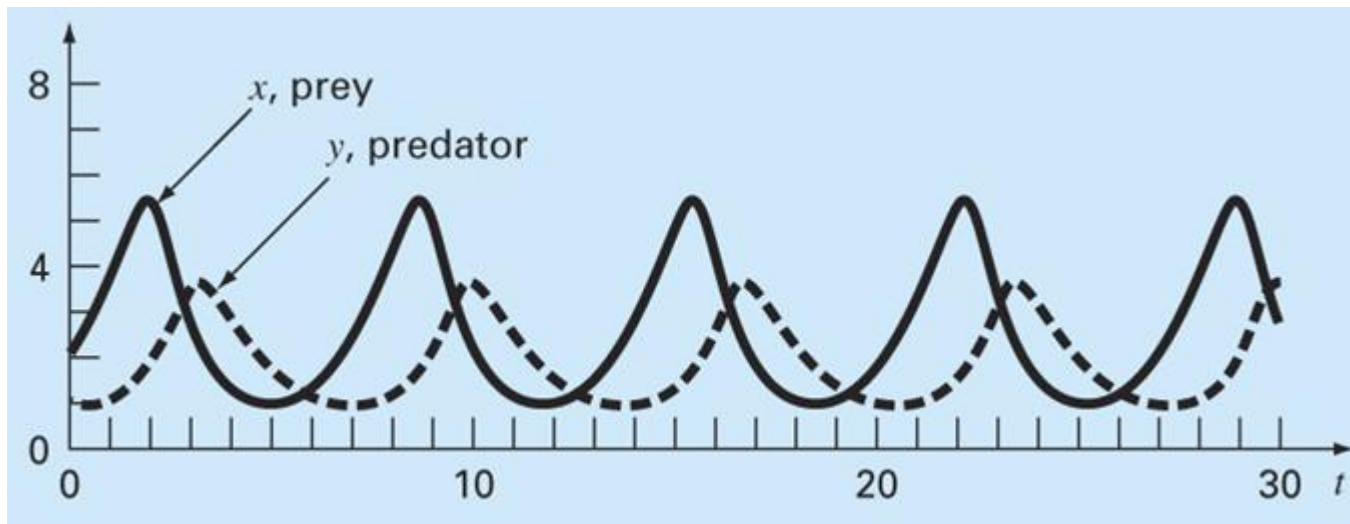
먹이사슬에 관한 미분방정식

a = the prey growth rate

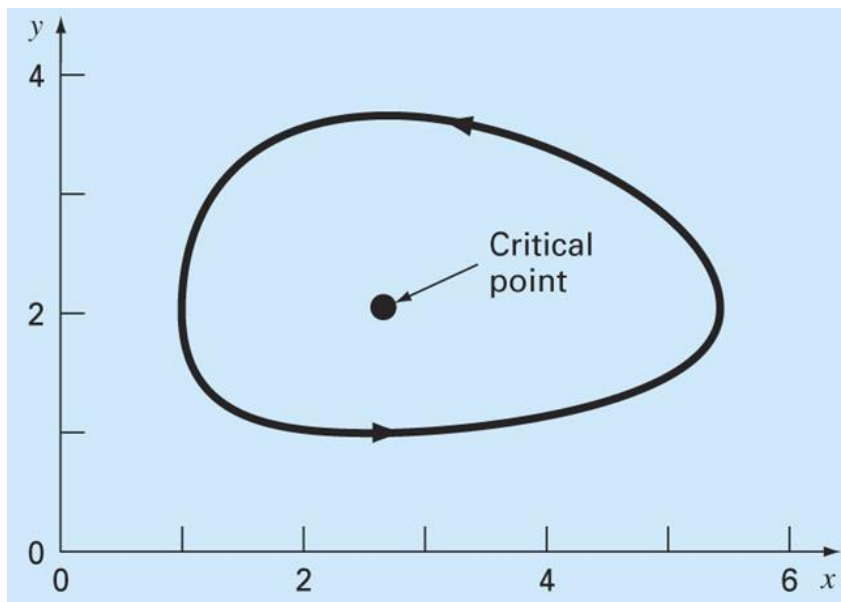
c = the predator death rate

$b=d$ = the rate characterizing the effect of the predator-prey interaction on prey death and predator growth

EXAMPLE 28.2



Parameters: change of magnitudes of peaks, lags, period



[state-space representation]

$$\frac{dx}{dt} = \frac{dy}{dt} = 0 \rightarrow (x, y) = (0, 0) \text{ and } \left(\frac{c}{d}, \frac{a}{b}\right)$$

$\left(\frac{0.8}{0.3}, \frac{1.2}{0.6}\right)$ populations will remain constant

SETTING

09_25p5.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results 1D Plot Group 1

Select Physics

Search

- Recently Used
- AC/DC
- Acoustics
- Chemical Species Transport
- Heat Transfer
- Plasma
- Mathematics
 - Δu PDE Interfaces**
 - Δu Coefficient Form PDE (c)**
 - Δu General Form PDE (g)
 - Δu Wave Form PDE (wahw)
 - ∫du Weak Form PDE (w)
 - Δu Lower Dimensions
 - ODE and DAE Interfaces
 - Optimization and Sensitivity
 - Classical PDEs
 - Deformed Mesh
 - Wall Distance (wd)

Add

Added physics interfaces:

- Δu Coefficient Form PDE (c)

Remove

Space Dimension → Study

Help Cancel Done

Review Physics Interface

Coefficient Form PDE (c)

Dependent Variables

Field name: u

Number of dependent variables: 2

Dependent variables: u1, u2

Units

Dependent variable quantity: Dimensionless (1)

Source term quantity: None

Unit: m⁻²

1D → Coefficient form PDE
선택 → Add 클릭

Number of dependent
variables: "2" 입력

Study 클릭
→ Time Dependent 선택
→ Done 클릭

899 MB | 929 MB

COEFFICIENT INPUT

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Component 1 (comp1) Add Component

Definitions Geometry Materials

Physics

Build Mesh Mesh

Compute Study 1 Add Study

Select Plot Group Add Plot Group Results

Layout

Model Builder

Untitled.mph (root)

Global Definitions

Materials

Component 1 (comp1)

Definitions

Geometry 1

Interval 1 (f1)

Form Union (fin)

Materials

Δ Coefficient Form PDE (c)

Coefficient Form PDE 1

Zero Flux 1

Initial Values 1

Mesh 1

Study 1

Step 1: Time Dependent

Results

Settings

Coefficient Form PDE

c

0 1 0 1

0 1 0 1

Absorption Coefficient

a

0 1/m² 0 1/m²

0 1/m² 0 1/m²

Source Term

f

1.2*u1-0.6*u1*u2 1/m²

-0.8*u2+0.3*u1*u2 1/m²

Mass Coefficient

e_a

0 s²/m² 0 s²/m²

0 s²/m² 0 s²/m²

Damping or Mass Coefficient

d_a

1 s/m² 0 s/m²

0 s/m² 1 s/m²

Conservative Flux Convection Coefficient

α

0 1/m 0 1/m

0 1/m 0 1/m

Convection Coefficient

β

0 1/m 0 1/m

0 1/m 0 1/m

Conservative Flux Source

γ

0 1/m

0 1/m

Graphics

Convergence Plot 1

Messages

Progress

Log

Number of degrees of freedom: 10000 (approx. 2.5 million elements)

Solution time (Study 1): 1 s.

Saved file: 09_25p2.mph

Saved file: 09_25p5.mph

Finalized geometry has 1 domain and 2 boundaries.

892 MB | 929 MB

1 전 예제와 동일한 방식으로 geometry 생성

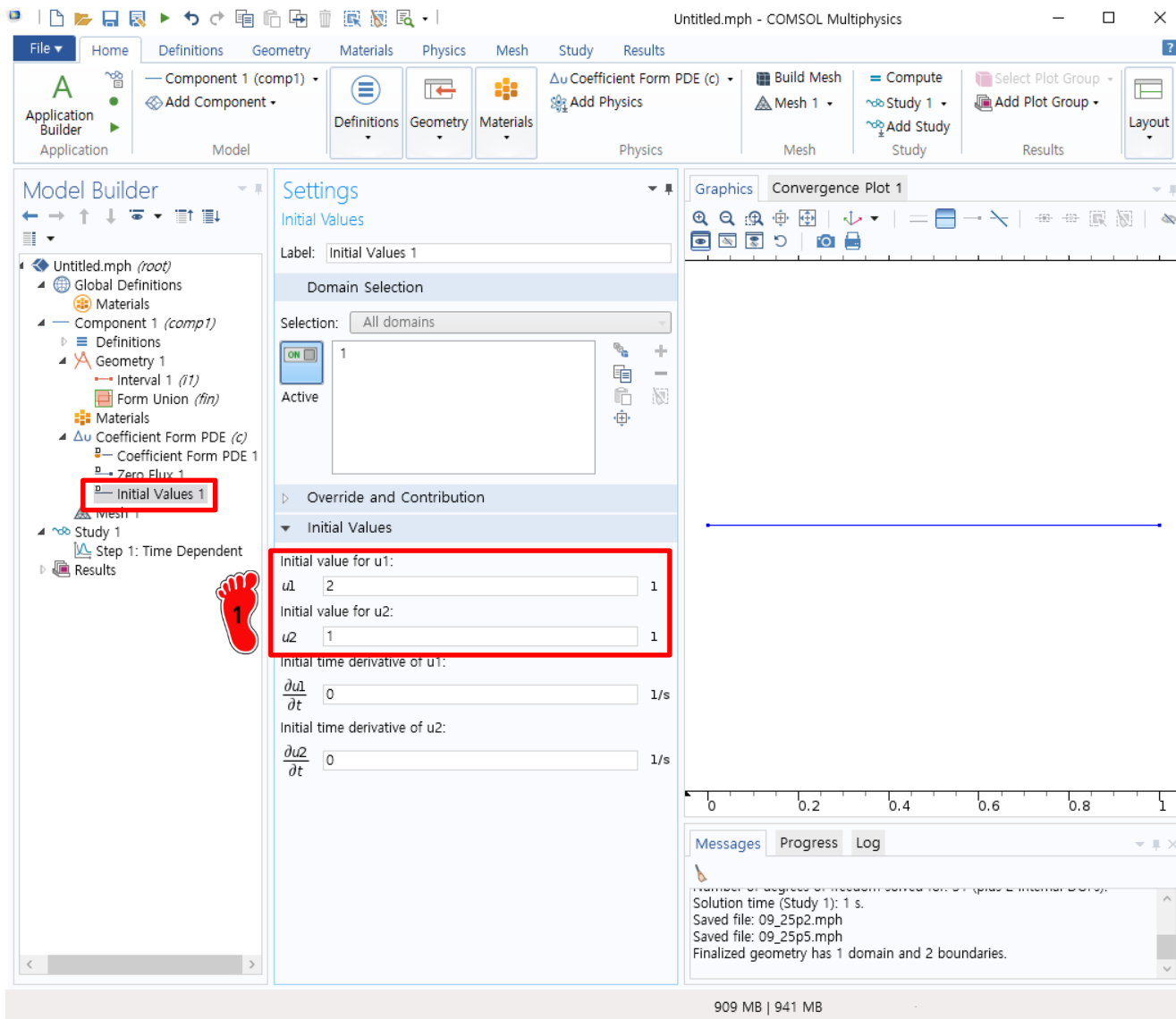
2 Coefficient Form PDE 1에 계수 입력

종속변수가 2개 이므로 벡터 또는 행렬로 입력 됨 (나머지 0)

$$d_a = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

$$f = \begin{bmatrix} 1.2u_1 - 0.6u_1u_2 \\ -0.8u_2 + 0.3u_1u_2 \end{bmatrix}$$

INITIAL VALUE INPUT



Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Component 1 (comp1) Add Component

Definitions Geometry Materials

Physics

Build Mesh Mesh 1 Compute Study 1 Add Study

Select Plot Group Add Plot Group Layout

Model Builder

Untitled.mph (root)

- Global Definitions
- Materials
- Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Interval 1 (i1)
 - Form Union (fin)
 - Materials
 - Δu Coefficient Form PDE (c)
 - Coefficient Form PDE 1
 - Zero Flux 1
 - Initial Values 1**
 - Mesh 1
 - Study 1
 - Step 1: Time Dependent
 - Results

Settings

Initial Values

Label: Initial Values 1

Domain Selection

Selection: All domains

1 Active

Override and Contribution

Initial Values

Initial value for u1:

u1 2 1

Initial value for u2:

u2 1 1

Initial time derivative of u1:

$\frac{\partial u_1}{\partial t}$ 0 1/s

Initial time derivative of u2:

$\frac{\partial u_2}{\partial t}$ 0 1/s

Graphics Convergence Plot 1

0 0.2 0.4 0.6 0.8 1

Messages Progress Log

Number of degrees of freedom solved for: 1 (plus 2 internal DOFs).

Solution time (Study 1): 1 s.

Saved file: 09_25p2.mph

Saved file: 09_25p5.mph

Finalized geometry has 1 domain and 2 boundaries.

909 MB | 941 MB



Initial Values 1 클릭 후
u1: 2, u2: 1 입력

TIME RANGE INPUT

The screenshot shows the COMSOL Multiphysics interface for a new model named 'Untitled.mph'. The 'Model Builder' on the left shows the hierarchy: Component 1 (comp1) > Geometry 1 > Interval 1 (t1) > Form Union (fin) > Materials > Coefficient Form PDE (c) > Coefficient Form PDE 1 > Zero Flux 1 > Initial Values 1 > Mesh 1 > Study 1. The 'Study 1' node is highlighted with a red box and a red footprint icon with the number 1. The 'Settings' pane for 'Study 1' is open, showing the 'Time Dependent' study type. The 'Times' field is highlighted with a red box and a red footprint icon with the number 1, showing the input 'range(0,0.1,20)'. The 'Physics and Variables Selection' table shows 'Coefficient Form PDE (c)' selected. The 'Graphics' pane on the right shows a 'Convergence Plot 1' with a horizontal axis from 0 to 1. The 'Messages' pane at the bottom shows the solution time and saved files.

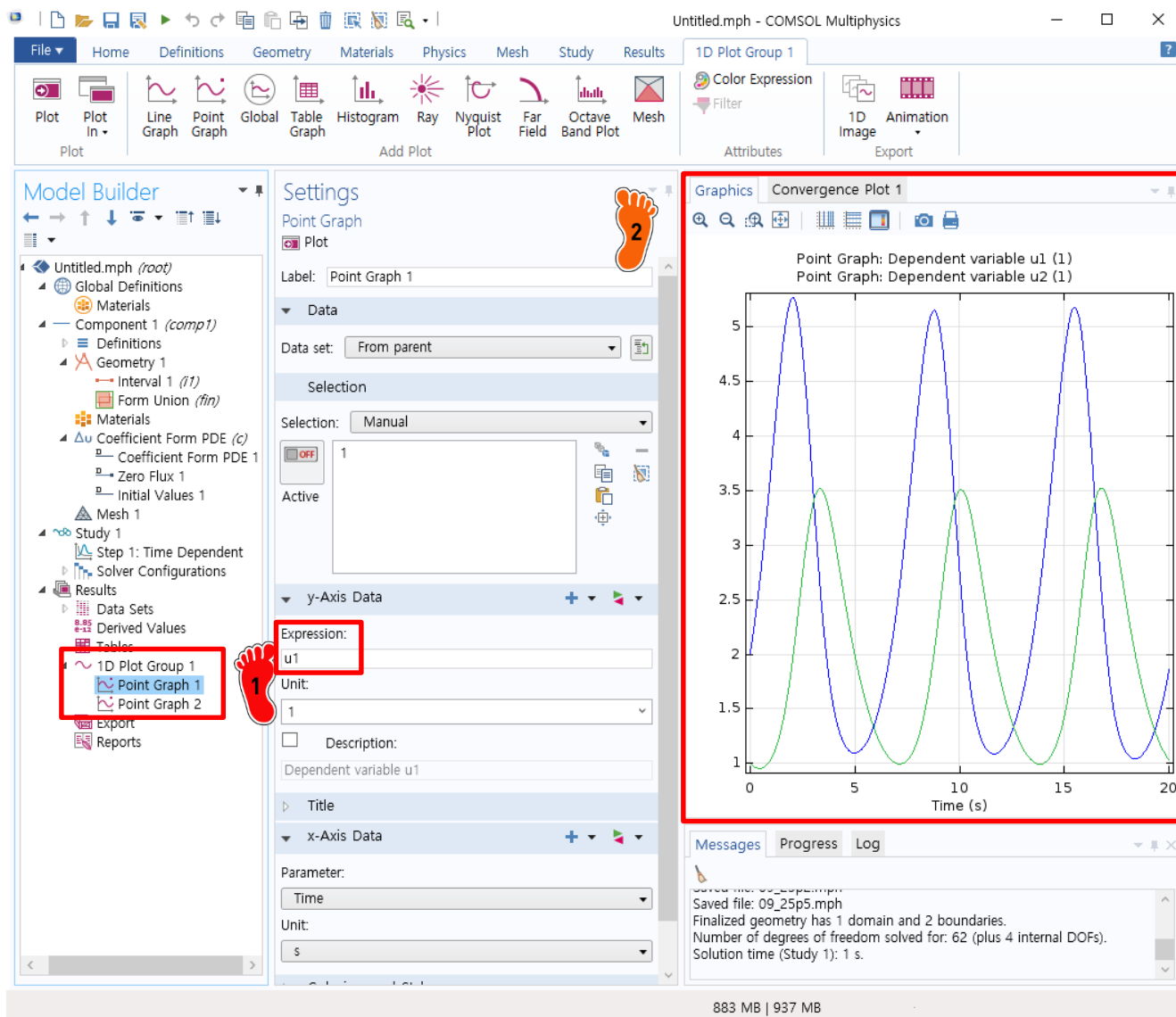


Step 1: Time Dependent 메뉴 선택 후

Times: range(0,0.1,20) 입력

이 후 Study 선택 후
Compute 클릭

RESULT PLOT



1 두 개의 Point Graph 생성
(Expression에 u1, u2 입력)

2 결과 그래프 확인

- **ODE examples**
 - ✓ **Example 25.5**
 - ✓ **Example 28.2: predator-prey model**
 - ✓ **Example 25.14**
 - ✓ **Example 27.10**
 - ✓ **Previous case study I**
 - ✓ **Previous case study II**

EXAMPLE 25.14

ordinary differential equation

$$\frac{dy}{dt} = 10e^{-(t-2)^2 / [2(0.075)^2]} - 0.6y$$

initial condition

$$t = 0, y = 0.5$$



Adaptive ODE solver 를 보여주기 위한 미분방정식



모델링 입력 항목



Coefficient Input

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

Absorption Coefficient

a 0.6 1/m²

Source Term

f $10 \cdot \exp(-(t-2)^2 / (2 \cdot (0.075)^2))$ 1/m²

Damping or Mass Coefficient

d_a 1 s/m²

Initial Value Input

Initial Values

Initial value for u:

u 0.5

Time Range Input

Study Settings

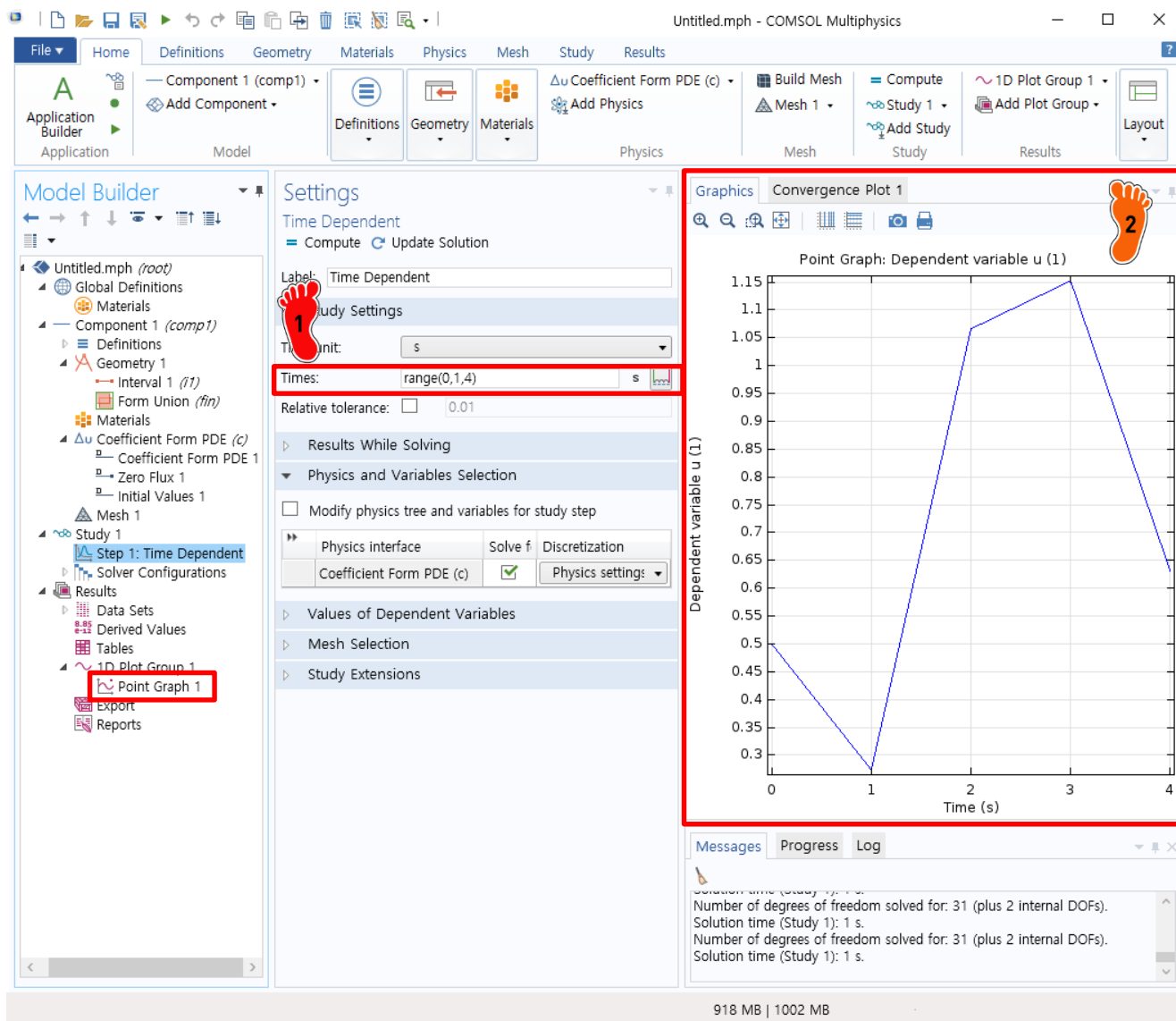
Time unit:

s

Times:

range(0,1,4)

RANGE INPUT



1 range(0,1,4) 로 입력할 경우 step size 가 상대적으로 큼

2 결과 그래프에서 데이터는 step size 1 로 결정이 됨

ADAPTIVE OPTION SELECTION

The screenshot shows the COMSOL Multiphysics interface. The top toolbar includes icons for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, and Results. The left sidebar shows the Model Builder tree with the following structure:

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Interval 1 (t1)
 - Form Union (fin)
 - Materials
 - Coefficient Form PDE (c)
 - Coefficient Form PDE 1
 - Zero Flux 1
 - Initial Values 1
 - Mesh 1
 - Study 1
 - Step 1: Time Dependent
 - Solver Configurations
 - Solution 1 (sol1)
 - Compile Equations: Time 1
 - Dependent Variables 1
 - Time-Dependent Solver 1

The right sidebar shows the Settings tab for the Time-Dependent Solver. The 'General' section is expanded, showing the following settings:

- Label: Time-Dependent Solver 1
- Defined by study step: Step 1: Time Dependent
- Time unit: s
- Time: range(0,1,4)
- Relative tolerance: 0.01
- Absolute Tolerance
- Time Stepping
- Results While Solving
- Output
 - Times to store: Steps taken by solver
 - ☒ Store reaction forces
 - ☐ Use lumping when computing fluxes
 - ☒ Store time derivatives
 - ☒ Store solution on disk
 - ☐ Store solution before and after events
- Advanced
- Log
 - ☐ Keep warnings in stored log

The bottom of the Settings tab shows the Log section with the following text:

```
Time-Dependent Solver 1 in Study 1/Solution 1 (
Time-dependent solver (BDF)
Number of degrees of freedom solved for: 31 (plus 2
Symmetric matrices found.
Scales for dependent variables:
Dependent variable u (comp1.u): 0.5
Step      Time      Stepsize  Res  Jac  Sol
```

The right sidebar also shows the Graphics tab with a Point Graph titled 'Point Graph: Dependent variable u (1)'. The graph shows a blue line with circular markers representing the dependent variable u (1) over time (s). The x-axis ranges from 0 to 2, and the y-axis ranges from 0.2 to 1.8. The graph shows a sharp peak around t=1.5.

1 Time-Dependent Solver 1 메뉴 선택

2 Times to store: Steps taken by solver 변경

3 step size 가 adaptive 하게 변경이 됨

Point Graph 1

The screenshot shows the 'Coloring and Style' dialog box for the Point Graph. The 'Line style' section is expanded, showing the following settings:

- Line: Solid
- Color: Cycle
- Width: 1

The 'Line markers' section is also expanded, showing the following settings:

- Marker: Circle
- Positioning: In data points

- **ODE examples**
 - ✓ **Example 25.5**
 - ✓ **Example 28.2: predator-prey model**
 - ✓ **Example 25.14**
 - ✓ **Example 27.10**
 - ✓ **Previous case study I**
 - ✓ **Previous case study II**

EXAMPLE 27.10



[Van der Pol equation]

$$\frac{d^2 y_1}{dt^2} - \mu(1 - y_1^2) \frac{dy_1}{dt} + y_1 = 0$$

initial condition

$$t = 0, y_1 = 1, \frac{dy_1}{dt} = 1$$

convert process →

$$\begin{cases} \frac{dy_1}{dt} = y_2 \\ \frac{dy_2}{dt} = \mu(1 - y_1^2)y_2 - y_1 \end{cases}$$



Dependent Variables

Dependent Variables	
Field name:	u
Number of dependent variables:	2
Dependent variables:	u1 u2

Coefficient Input

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

Source Term	
f	1/(1-u1^2)*u2-u1
	1/m^2
	1/m^2

Damping or Mass Coefficient	
d _a	1 s/m ²
	0 s/m ²
	1 s/m ²

Initial Value Input

Initial Values	
Initial value for u1:	1
Initial value for u2:	1

Time Range Input

Study Settings	
Time unit:	s
Times:	range(0,0.1,20)



stiff 한 정도가 mu 값에 따라서 변하는 van der Pol equation.



모델링 입력 항목

TIME DEPENDENT OPTION

The screenshot shows the COMSOL Multiphysics interface for a time-dependent study. The 'Study' tab is active, and the 'Time Dependent' option is selected in the 'Study Settings' section. A red box highlights the 'Results While Solving' checkbox, which is checked. The 'Plot' checkbox is also checked. The 'Time unit' is set to 's' (seconds), and the 'Times' range is 'range(0,0.1,20)'. The 'Relative tolerance' is set to '0.01'. The 'Physics and Variables Selection' section shows the 'Coefficient Form PDE (c)' interface selected. The 'Values of Dependent Variables' section is expanded, showing 'u1' and 'u2'. The 'Mesh Selection' and 'Study Extensions' sections are also visible. The 'Graphics' window shows a 'Convergence Plot 1' with two data series: 'Point Graph: Dependent variable u1 (1)' (blue circles) and 'Point Graph: Dependent variable u2 (1)' (green circles). The plot shows oscillatory behavior over time (0 to 20 seconds). The 'Messages' window at the bottom displays the following information:

```

Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).
Solution time (Solution 1 (sol1)): 6 s.
Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).
Solution time (Study 1): 6 s.
  
```

1.11 GB | 1.17 GB



Step 1: Time Dependent
 → Result While Solving
 → Plot 체크 후 Compute

ADAPTIVE OPTION SELECTION

The screenshot displays the COMSOL Multiphysics interface for a time-dependent study. The left sidebar shows the Model Builder tree with the following structure:

- Untitled.mph (root)
 - Global Definitions
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Interval 1 (t1)
 - Form Union (fin)
 - Materials
 - Coefficient Form PDE (c)
 - Coefficient Form PDE 1
 - Zero Flux 1
 - Initial Values 1
 - Mesh 1
 - Study 1
 - Step 1: Time Dependent
 - Solver Configurations
 - Solution 1 (sol1)
 - Compile Equations: 1
 - Dependent Variables: 1
 - Time-Dependent Solver 1** (highlighted with a red box and a red footprint icon labeled '1')

The right sidebar shows the Settings window for the Time-Dependent Solver 1. The 'General' tab is active, showing the following settings:

- Label: Time-Dependent Solver 1
- Defined by study step: Step 1: Time Dependent
- Time unit: s
- Time: range(0,0.1,20)
- Relative tolerance: 0.01
- Absolute Tolerance
- Time Stepping
- Results While Solving
- Output
 - Times to store: Steps taken by solver (highlighted with a red box)
 - ☒ Store reaction forces
 - ☐ Use lumping when computing fluxes
 - ☒ Store time derivatives
 - ☒ Store solution on disk
 - ☐ Store solution before and after events
- Advanced
- Log
 - ☐ Keep warnings in stored log

The bottom of the Settings window shows the following information:

- Time-Dependent Solver 1 in Study 1/Solution 1
- Time-dependent solver (BDF)
- Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).
- Nonsymmetric matrix found.
- Scales for dependent variables:
- Dependent variable u1 (comp1.u1): 1
- Dependent variable u2 (comp1.u2): 1

The right sidebar also shows the Graphics window with a Convergence Plot 1. The plot shows the dependent variables u1 (blue line with circles) and u2 (green line with circles) over time (s). The x-axis ranges from 0 to 20, and the y-axis ranges from -2.5 to 2.5. The plot shows oscillatory behavior for both variables.

The bottom status bar shows the memory usage: 1.11 GB | 1.17 GB.

1 Time-Dependent Solver 1
메뉴 선택 → Steps taken
by solver 선택

2 Print step size 비교

STIFF COEFFICIENT INPUT



mu 값을 1000으로 변경

Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results

Application Builder Model

Component 1 (comp1) Add Component

Definitions Geometry Materials

Physics

Build Mesh Mesh 1

Compute Study 1 Add Study

1D Plot Group 1 Add Plot Group

Layout

Model Builder

Untitled.mph (root)

Global Definitions

Materials

Component 1 (comp1)

Definitions

Geometry 1

Interval 1 (f1)

Form Union (fin)

Materials

Coefficient Form PDE (c)

Coefficient Form PDE 1

Zero Flux 1

Initial Values 1

Mesh 1

Study 1

Step 1: Time Dependent

Solver Configurations

Solution 1 (sol1)

Compile Equations: Time

Dependent Variables 1

Time-Dependent Solution 1

Results

Data Sets

Derived Values

Tables

1D Plot Group 1

Point Graph 1

Point Graph 2

Export

Reports

Settings

Coefficient Form PDE

Equation

Show equation assuming:

Study 1, Time Dependent

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

$$\mathbf{u} = [u_1, u_2]^T$$

$$\nabla = \frac{\partial}{\partial x}$$

Diffusion Coefficient

c	0	1	0	1
	0	1	0	1

Absorption Coefficient

a	0	1/m ²	0	1/m ²
	0	1/m ²	0	1/m ²

Source Term

f	u2	1/m ²
	1000*(1-u1^2)*u2-u1	1/m ²

Mass Coefficient

e_a	0	s ² /m ²	0	s ² /m ²
	0	s ² /m ²	0	s ² /m ²

Damping or Mass Coefficient

d_a	1	s/m ²	0	s/m ²
	0	s/m ²	1	s/m ²

Conservative Flux Convection Coefficient

Graphics

Convergence Plot 1

Messages Progress Log

Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).
Solution time (Study 1): 6 s.
Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).
Solution time (Solution 1 (sol1)): 5 s.

1.08 GB | 1.16 GB

TIME RANGE INPUT

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

- Model Builder:** The left sidebar shows the model tree. The 'Study 1' node is expanded, and 'Step 1: Time Dependent' is highlighted with a red box and a red footprint icon labeled '1'.
- Settings:** The 'Time Dependent' settings are shown. The 'Times' field is highlighted with a red box and contains the input `range(0,0.1,2000)`. The 'Time unit' is set to 's'.
- Graphics:** The 'Convergence Plot 1' is displayed, showing two oscillating curves (blue and green) over time (0 to 20 seconds). The y-axis ranges from -2.5 to 2.5.
- Messages:** The bottom panel shows the progress log, indicating the solution time and degrees of freedom for both Solution 1 and Study 1.

1.16 GB | 1.21 GB



range(0,0.1,2000)으로 변경
→ Compute

PLOT GROUP DISABLE

Untitled.mph - COMSOL Multiphysics

1D Plot Group 1

Color Expression

Filter

Attributes

1D Image

Animation

Export

Model Builder

Settings

Point Graph

Plot

Label: Point Graph 2

Data

Data set: From parent

Plot

Plot In

Color Expression

Add Plot Data to Export

Copy as Code to Clipboard

Move Up

Ctrl+Up

Copy

Duplicate

Delete

Del

Disable

F3

Rename

F2

Settings

Properties

Help

F1

Point Graph 2

Convergence Plot 1

Point Graph: Dependent variable u1 (1)

Point Graph: Dependent variable u2 (1)

Time (s)

Messages

Progress

Log

Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).

Solution time (Study 1): 14 s.

Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).

Solution time (Solution 1 (sol1)): 1 s.

Number of degrees of freedom solved for: 62 (plus 4 internal DOFs).

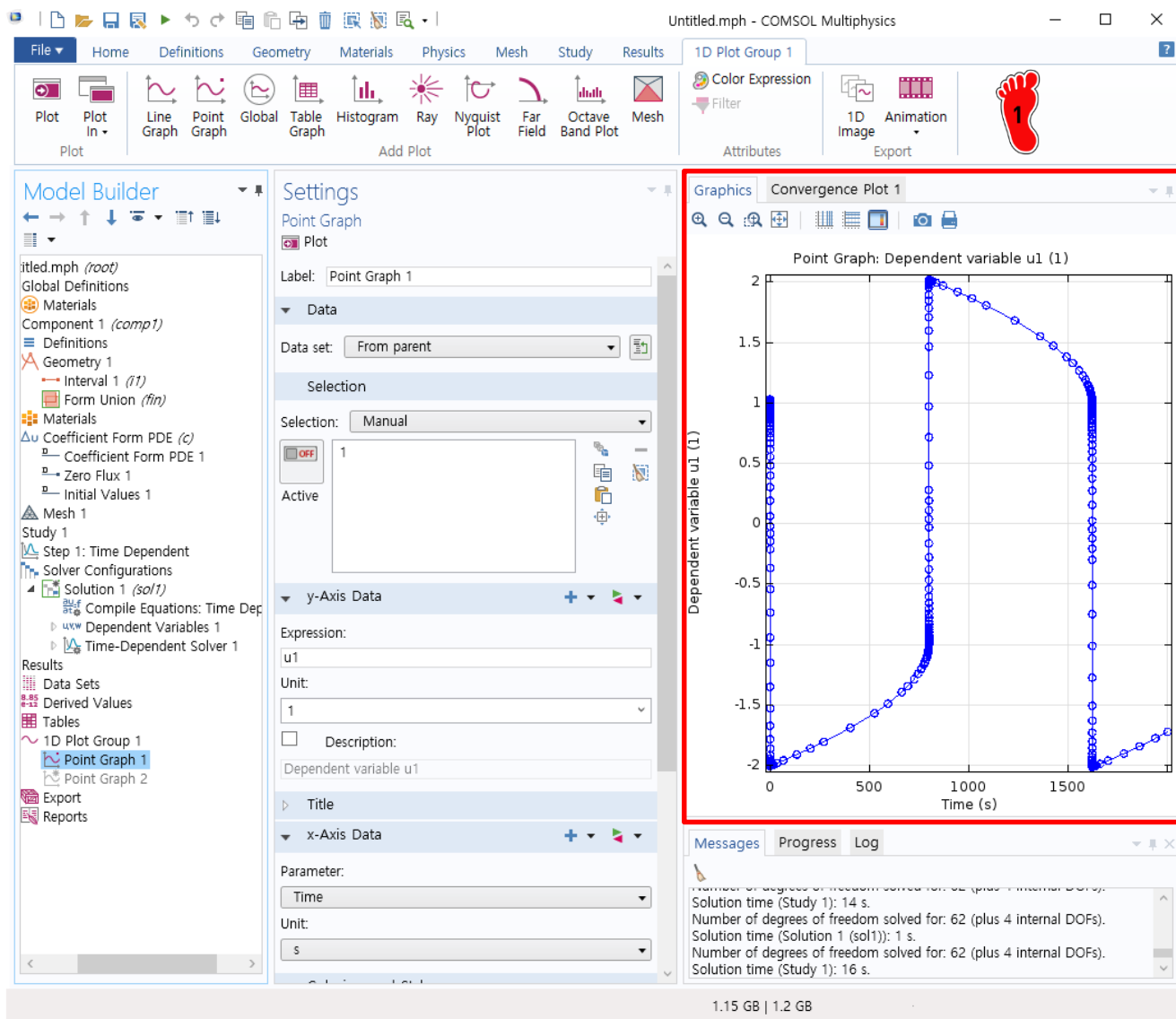
Solution time (Study 1): 16 s.

1.15 GB | 1.2 GB

1 Point Group 2 우클릭

2 Disable 클릭
(u2의 경우 stiff한 구간에서 값이 급격하게 증가하기 때문에(결과 그래프 참조) u1 결과를 동시에 확인 어려움)

RESULT PLOT



결과 확인

stiff 한 시스템인 경우 시간이 소요되는 부분은 기울기가 급격히 변화하는 부분임을 확인

Times to store: Steps taken by solver
☒ Store reaction Specified values
☐ Use lumping when computing fluxes Steps taken by solver

옵션에 따른 결과 확인

- **ODE examples**
 - ✓ **Example 25.5**
 - ✓ **Example 28.2: predator-prey model**
 - ✓ **Example 25.14**
 - ✓ **Example 27.10**
 - ✓ **Previous case study I**
 - ✓ **Previous case study II**

CASE STUDY I

Background. Electric circuits where the current is time-variable rather than constant are common. A transient current is established in the right-hand loop of the circuit shown in Fig. 28.11 when the switch is suddenly closed.

Equations that describe the transient behavior of the circuit in Fig. 28.11 are based on Kirchhoff's law, which states that the algebraic sum of the voltage drops around a closed loop is zero (recall Sec. 8.3). Thus,

$$L \frac{di}{dt} + Ri + \frac{q}{C} - E(t) = 0 \quad (28.9)$$

where $L(di/dt)$ = voltage drop across the inductor, L = inductance (H), R = resistance (Ω), q = charge on the capacitor (C), C = capacitance (F), $E(t)$ = time-variable voltage source (V), and

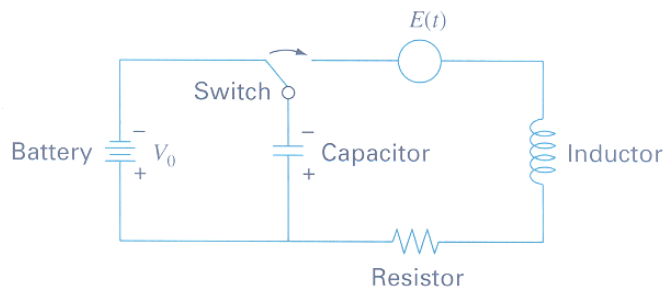
$$i = \frac{dq}{dt} \quad (28.10)$$

Equations (28.9) and (28.10) are a pair of first-order linear differential equations that can be solved analytically. For example, if $E(t) = E_0 \sin \omega t$ and $R = 0$,

$$q(t) = \frac{-E_0}{L(p^2 - \omega^2)} \frac{\omega}{p} \sin pt + \frac{E_0}{L(p^2 - \omega^2)} \sin \omega t \quad (28.11)$$

FIGURE 28.11

An electric circuit where the current varies with time.



$$E = E_0 \sin(\omega t)$$

$$L = 1 \text{ H}$$

$$E_0 = 1 \text{ V}$$

$$C = 0.25 \text{ C}$$

$$\omega^2 = 3.5 \text{ rad/s}^2$$

$$R = 0$$

$$q = y_1$$

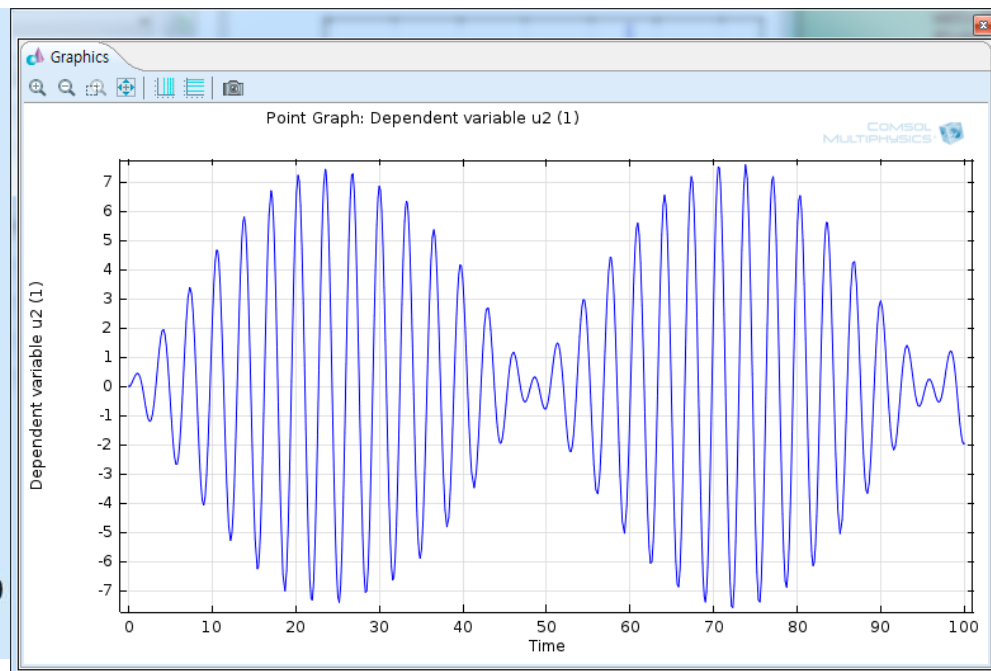
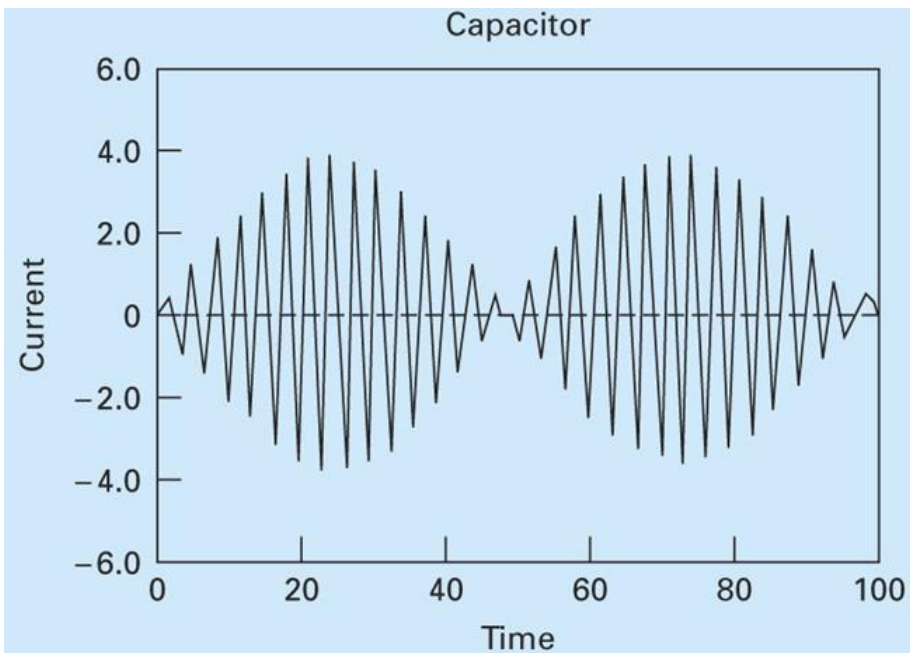
$$i = \frac{dq}{dt} = \frac{dy_1}{dt} = y_2$$

$$L \frac{di}{dt} + Ri + \frac{q}{C} - E(t) = 0$$

$$\rightarrow \frac{dy_2}{dt} = \frac{1}{L} \left(E(t) - Ri - \frac{y_1}{C} \right)$$

$$\begin{cases} \frac{dy_1}{dt} = y_2 \\ \frac{dy_2}{dt} = \frac{1}{L} \left(E(t) - Ri - \frac{y_1}{C} \right) \end{cases}$$

CASE STUDY I: RESULT



- **ODE examples**
 - ✓ **Example 25.5**
 - ✓ **Example 28.2: predator-prey model**
 - ✓ **Example 25.14**
 - ✓ **Example 27.10**
 - ✓ **Previous case study I**
 - ✓ **Previous case study II**

CASE STUDY II

Background. Mechanical engineers (as well as all other engineers) are frequently faced with problems concerning the periodic motion of free bodies. The engineering approach to such problems ultimately requires that the position and velocity of the body be known as a function of time. These functions of time invariably are the solution of ordinary differential equations. The differential equations are usually based on Newton's laws of motion.

As an example, consider the simple pendulum shown previously in Fig. PT7.1. The particle of weight W is suspended on a weightless rod of length l . The only forces acting on the particle are its weight and the tension R in the rod. The position of the particle at any time is completely specified in terms of the angle θ and l .

The free-body diagram in Fig. 28.16 shows the forces on the particle and the acceleration. It is convenient to apply Newton's laws of motion in the x direction tangent to the path of the particle:

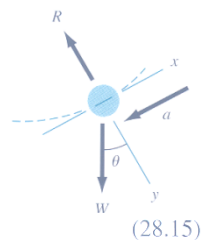
$$\Sigma F = -W \sin \theta = \frac{W}{g} a$$

where g = the gravitational constant (32.2 ft/s²) and a = the acceleration in the x direction. The angular acceleration of the particle (α) becomes

$$\alpha = \frac{a}{l}$$

FIGURE 28.16

A free-body diagram of the swinging pendulum showing the forces on the particle and the acceleration.



Therefore, in polar coordinates ($\alpha = d^2\theta/dt^2$),

$$-W \sin \theta = \frac{Wl}{g} \alpha = \frac{Wl}{g} \frac{d^2\theta}{dt^2}$$

or

$$\frac{d^2\theta}{dt^2} + \frac{g}{l} \sin \theta = 0$$

This apparently simple equation is a second-order nonlinear differential equation. In general, such equations are difficult or impossible to solve analytically. You have two choices regarding further progress. First, the differential equation might be reduced to a form that can be solved analytically (recall Sec. PT7.1.1), or second, a numerical approximation technique can be used to solve the differential equation directly. We will examine both of these alternatives in this example.

$$\sin \theta \approx \theta$$

$$g = 9.81 \text{ m/s}^2$$

$$\theta_0 = \pi / 4$$

$$l = 0.6096 \text{ m}$$

$$\theta = y_1$$

$$\frac{d\theta}{dt} = \frac{dy_1}{dt} = y_2$$

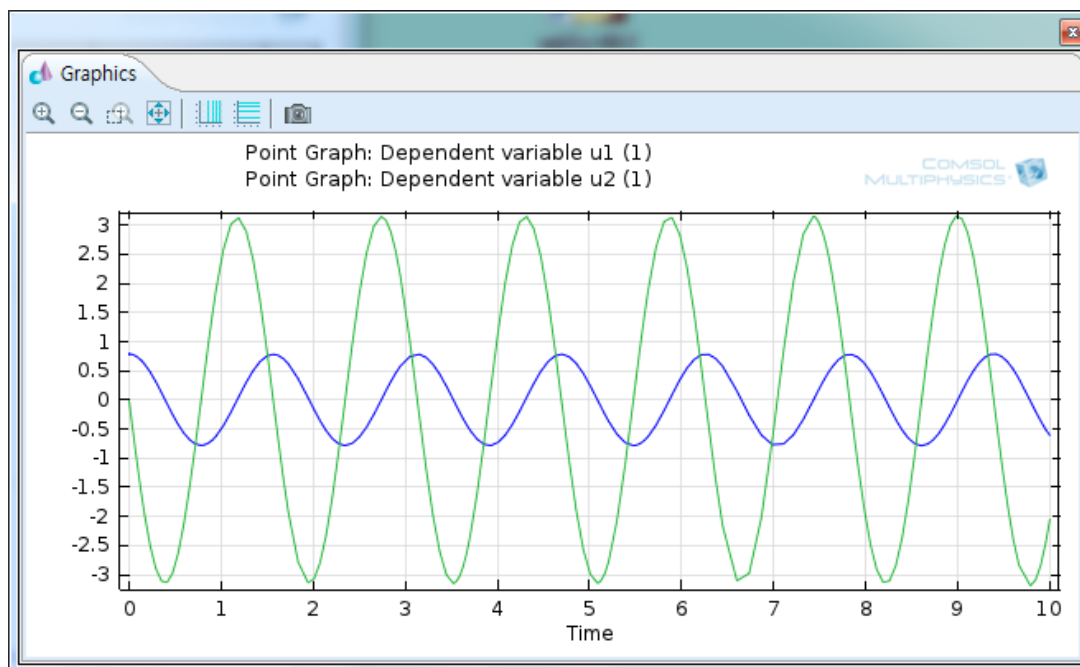
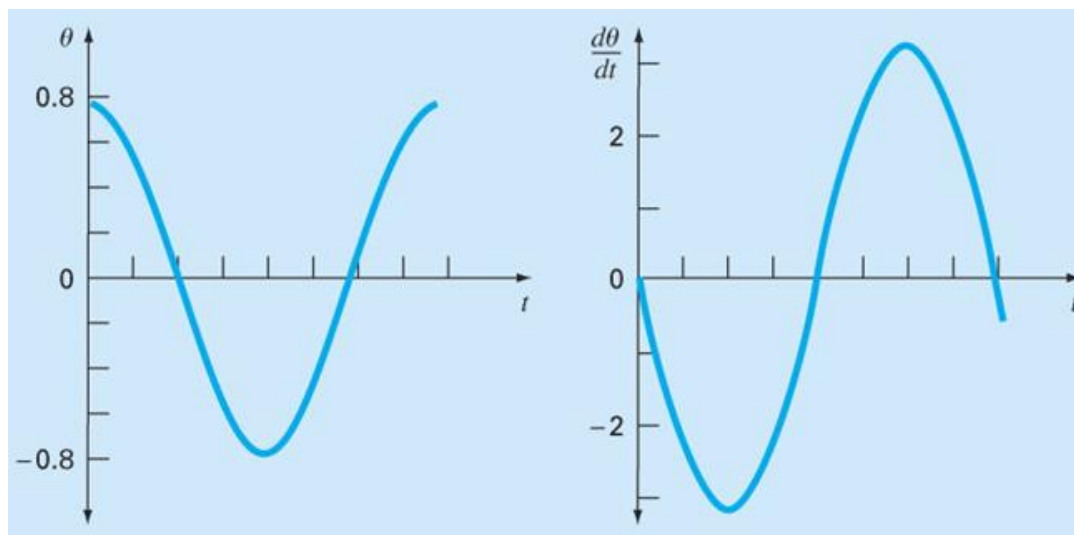
$$\frac{d^2\theta}{dt^2} + \frac{g}{l} \sin \theta = 0$$

$$\rightarrow \frac{dy_2}{dt} = -\frac{g}{l} \sin y_1$$

$$\rightarrow \frac{dy_2}{dt} = -\frac{g}{l} y_1$$

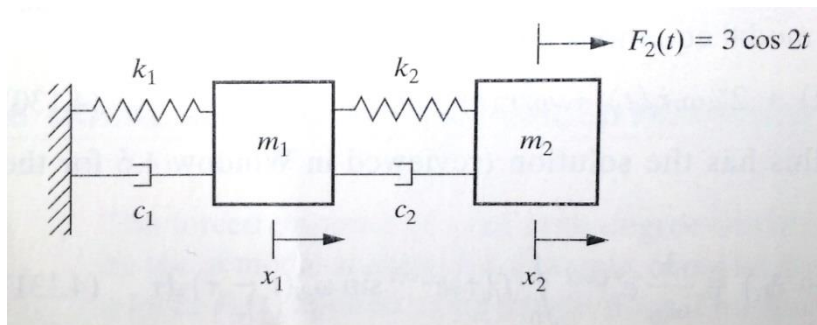
$$\begin{cases} \frac{dy_1}{dt} = y_2 \\ \frac{dy_2}{dt} = -\frac{g}{l} y_1 \end{cases}$$

CASE STUDY II: RESULT



ASSIGNMENT(3RD WEEK: ODE)

2-DOF system



$$m_1 = 9 \text{ kg}, m_2 = 1 \text{ kg}$$

$$k_1 = 24 \text{ N/m}, k_2 = 3 \text{ N/m}$$

$$c_1 = 2.4 \text{ Ns/m}, c_2 = 0.3 \text{ Ns/m}$$

$$F_2(t) = 3 \cos 2t$$

※ Ref. : Daniel J. Inman, "Engineering Vibration",
Prentice Hall International, Inc., pp 296-298, 2001

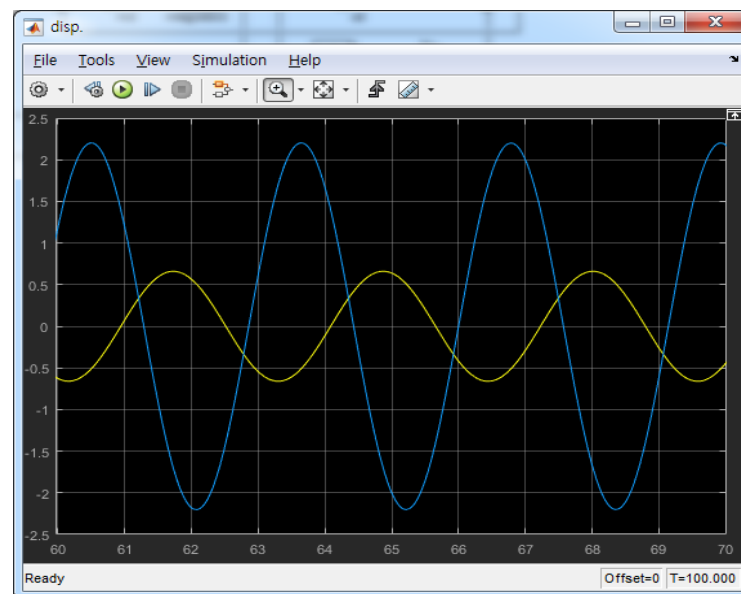
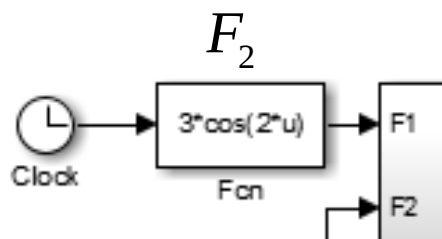
$$m_1 \ddot{x}_1 = -k_2(x_1 - x_2) - c_2(\dot{x}_1 - \dot{x}_2) - k_1 x_1 - c_1 \dot{x}_1$$

$$m_2 \ddot{x}_2 = k_2(x_1 - x_2) + c_2(\dot{x}_1 - \dot{x}_2) + F(t)$$

※ Analytic Solution (Steady State)

$$x_1(t) = 0.2451 \cos(2t - 0.1974) - 0.6249 \sin 2t$$

$$x_2(t) = 0.7354 \cos(2t - 0.1974) + 1.8749 \sin 2t$$



ASSIGNMENT(3RD WEEK: ODE)

