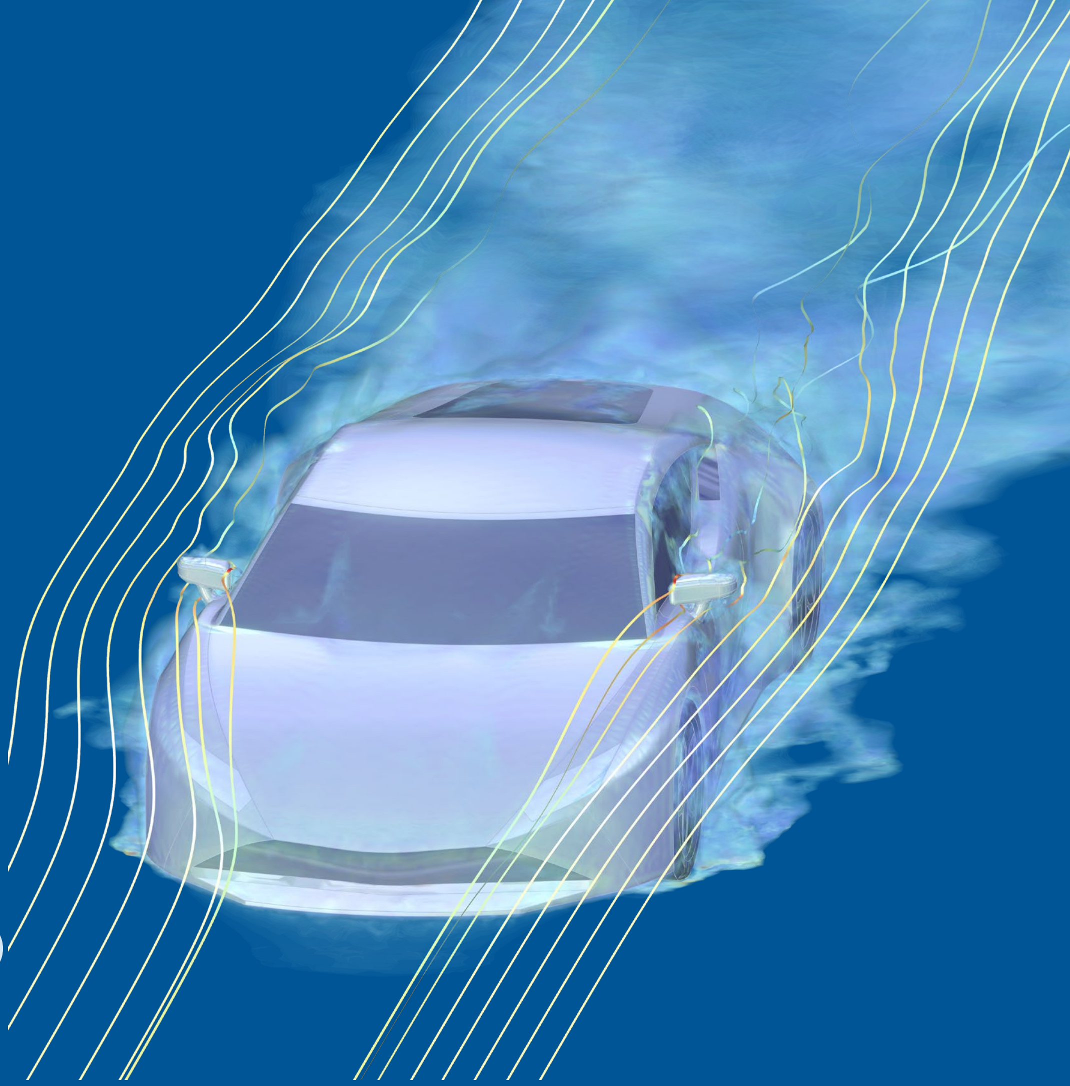


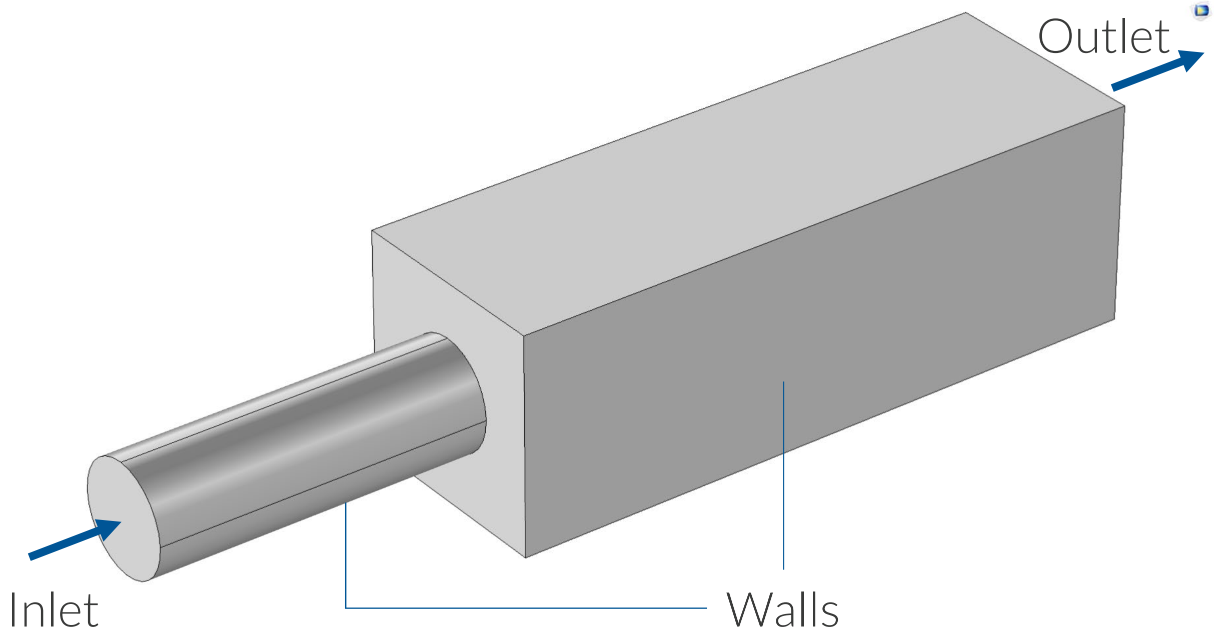
COMSOL Multiphysics®
Laminar Flow Modeling

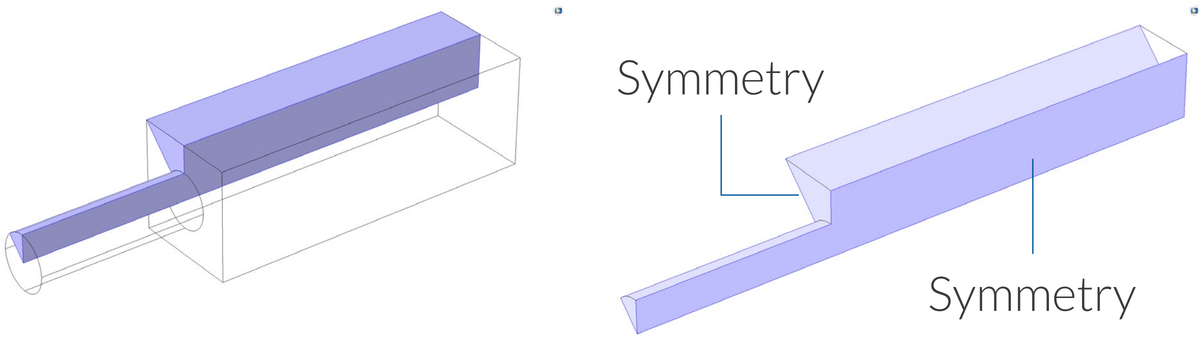
Laminar Flow

Demo

Model Definition

- Laminar flow in water
- Fully developed flow at the inlet
- Pressure condition at the outlet
- No-slip conditions at walls
- Symmetry conditions at the two lateral surfaces
- Why?
  - Typical expansion found in many systems, e.g. in medtech
  - Benchmark with flow separation





Due to symmetry, we only have to model one eight of the model domain, provided that the flow is steady and that the inlet flow is perpendicular to the inlet boundary.

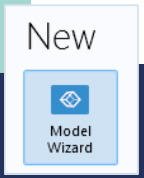
COMSOL

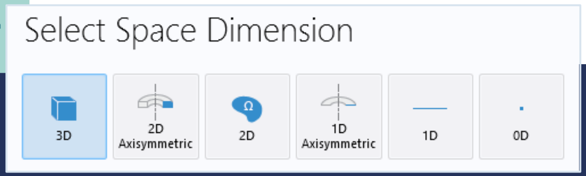
THE FIRST STEP

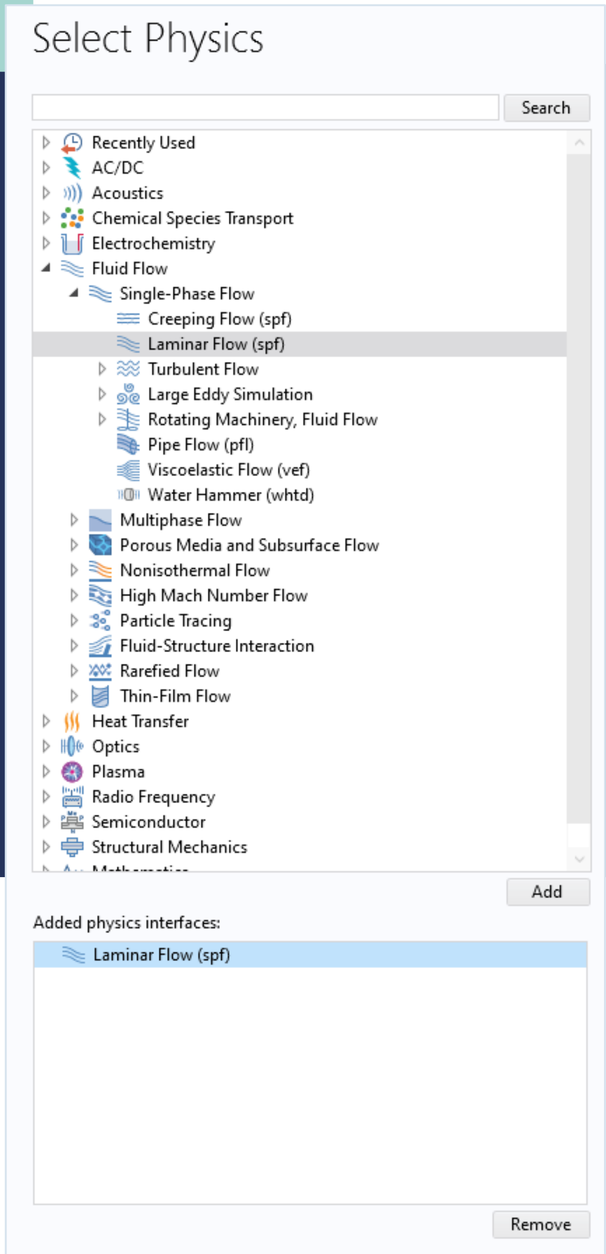
The Model Wizard

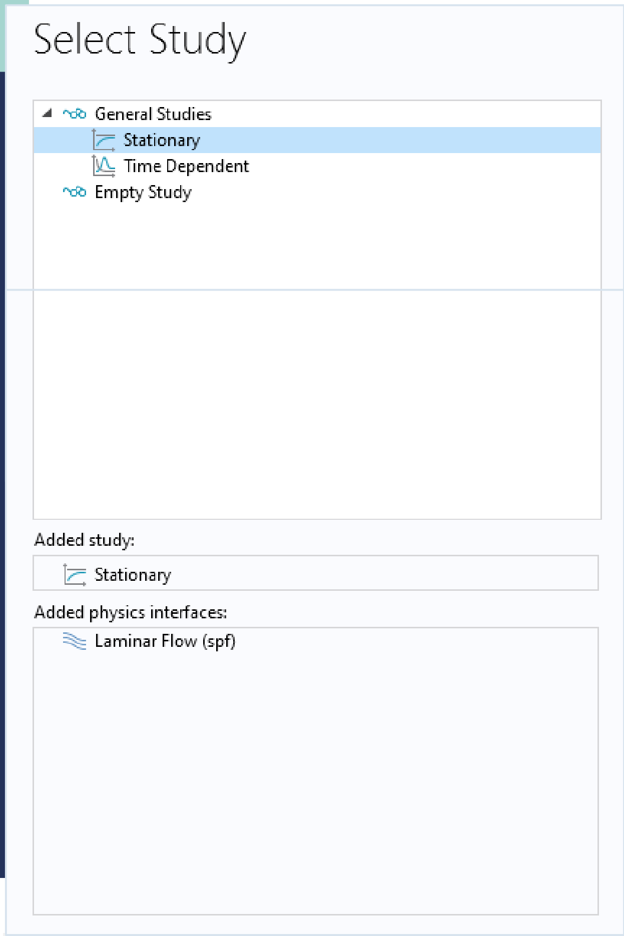
When creating a new model, the Model Wizard assists with selecting:

- Dimension (3D, 2D, 1D, or 0D)
- Physics interface(s) from the physics list
- Study for the physics interfaces

1


2


3


4


1. Select *Model Wizard*.
2. Select space dimension.
3. Select physics interfaces.
4. Select study.

COMSOL

Application Builder and Model Manager

Buttons to switch to the Application Builder or Model Manager.

Ribbon

Includes controls for all steps of the modeling process.

Graphics Window Toolbar

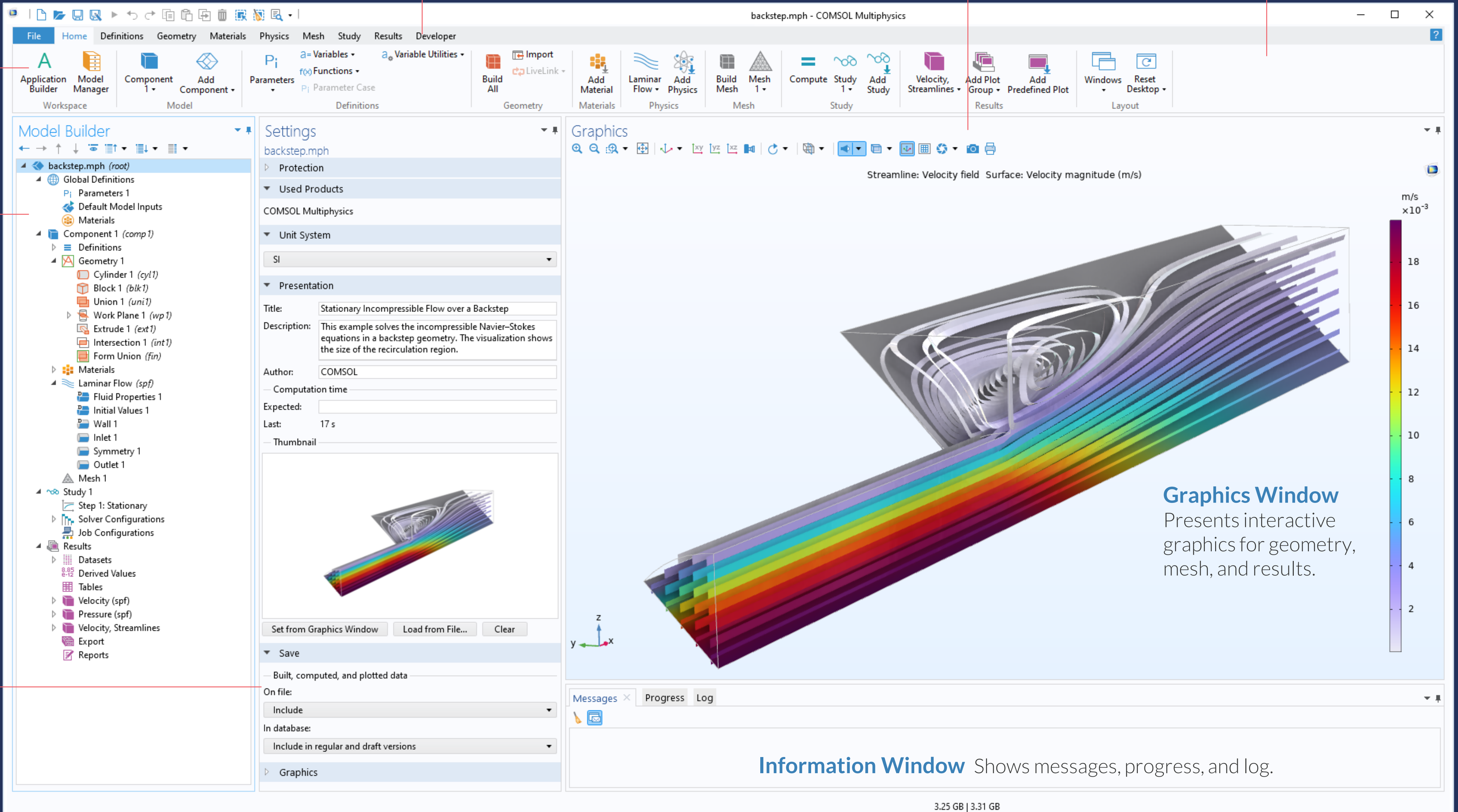
COMSOL Desktop® Model Builder

Model Builder Window

The model tree, together with the associated toolbar buttons, provides an overview of the model. The modeling process can be controlled from context-sensitive menus.

Settings Window

Shows the settings for the node that is selected in the model tree.



Graphics Window

Presents interactive graphics for geometry, mesh, and results.

Information Window Shows messages, progress, and log.

Definitions Geometry Materials Physics Mesh Study Results

Ribbon tabs for all steps in the modeling process

Model tree shows sequences of operations

Definitions

Geometry

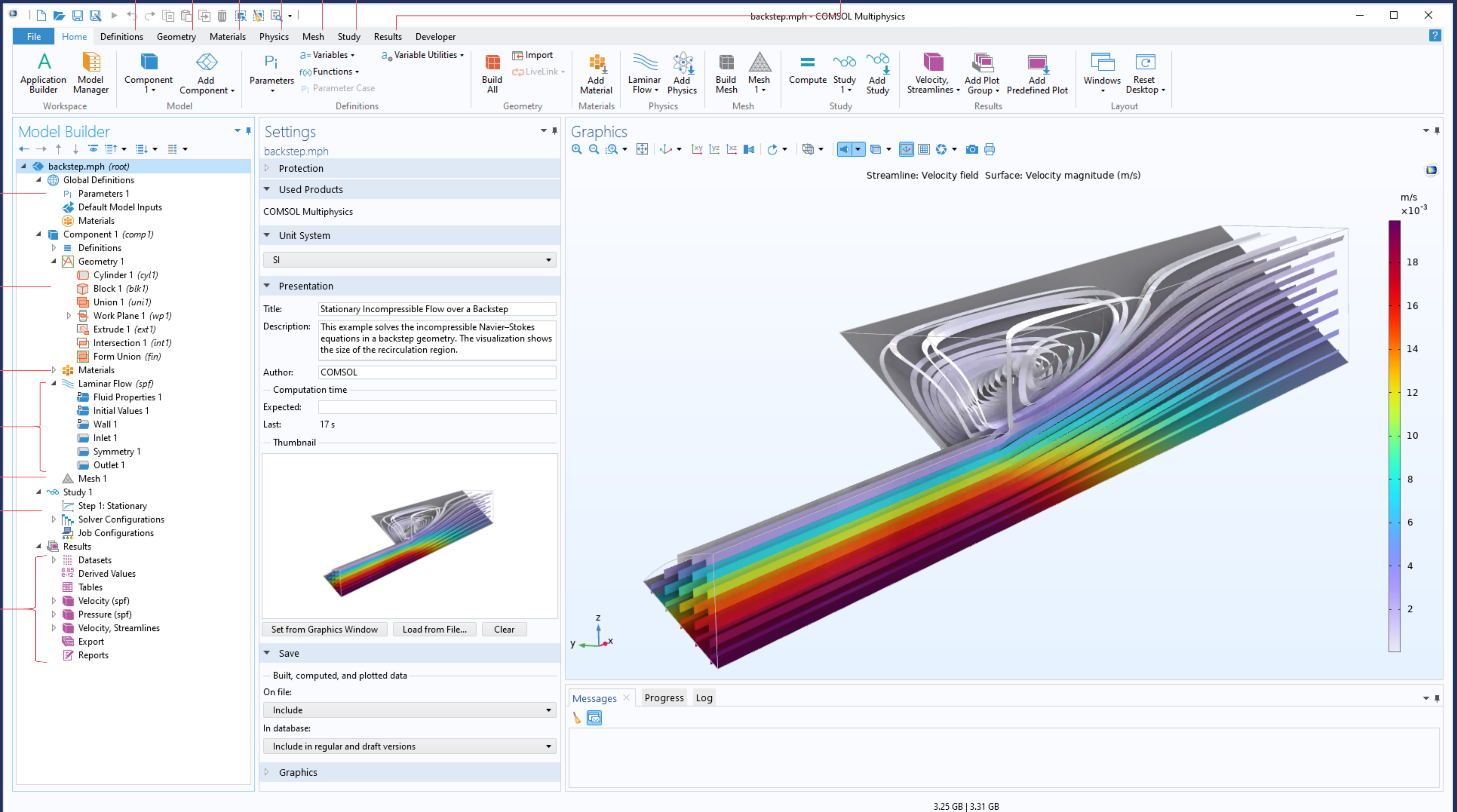
Materials

Physics

Mesh

Study

Results



Definitions Geometry Materials Physics Mesh Study Results

Home tab with the most common commands in the modeling process

Model tree shows sequences of operations

Definitions

Geometry

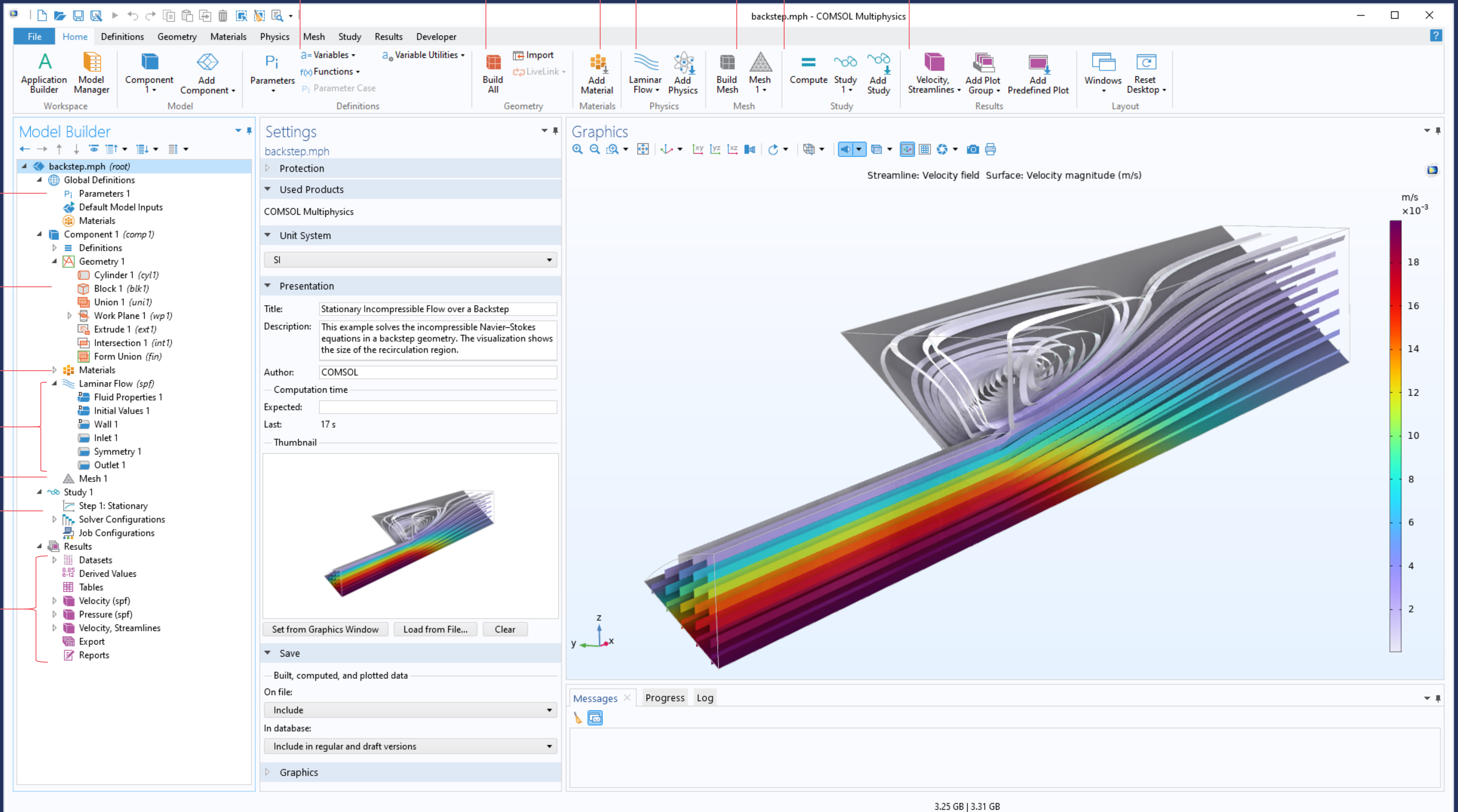
Materials

Physics

Mesh

Study

Results



Model Implementation

First model example:

- Define the model and solve the problem in a minimum of steps
- Once solved, we will go back and briefly review the procedure

Definitions:
- Load parameters from file

Geometry:
- Import sequence

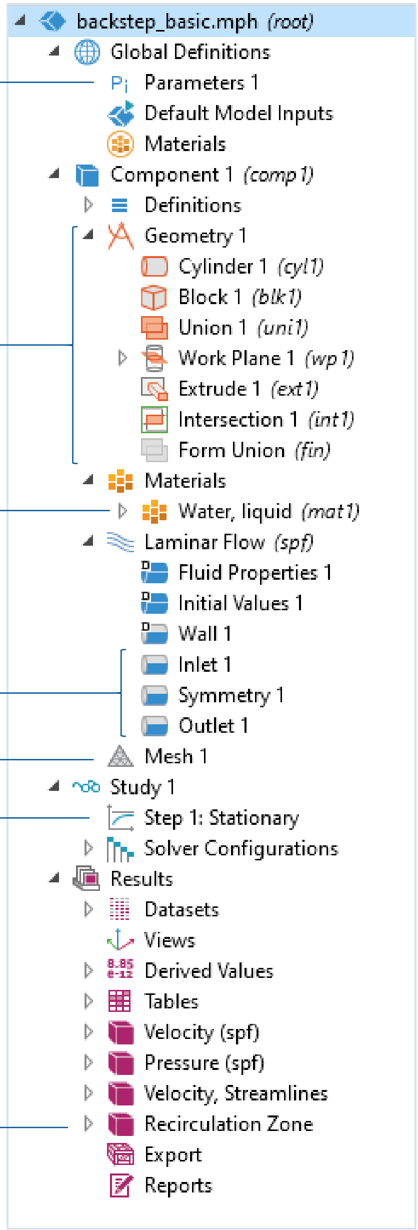
Materials:
- Load from Material Library

Physics:
- Define inlet, symmetry, and outlet

Mesh:
- Select mesh

Study:
- Select compute

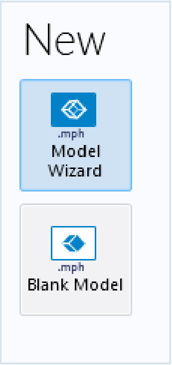
Results:
- Plot streamlines



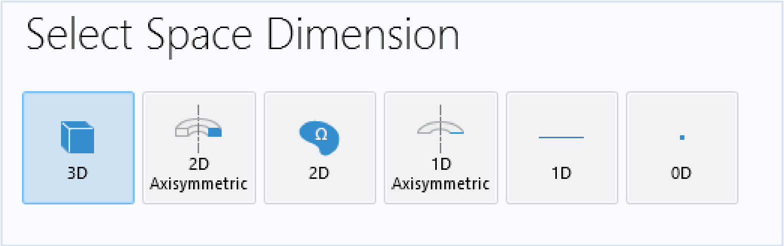
Overview of procedure for first model example

Demo: Model Wizard

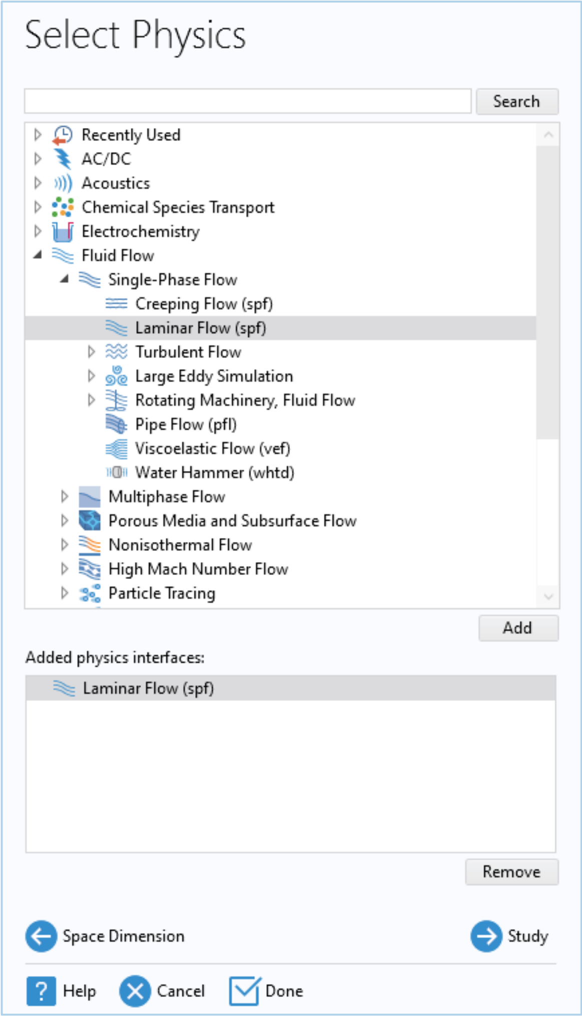
1. Select Model Wizard



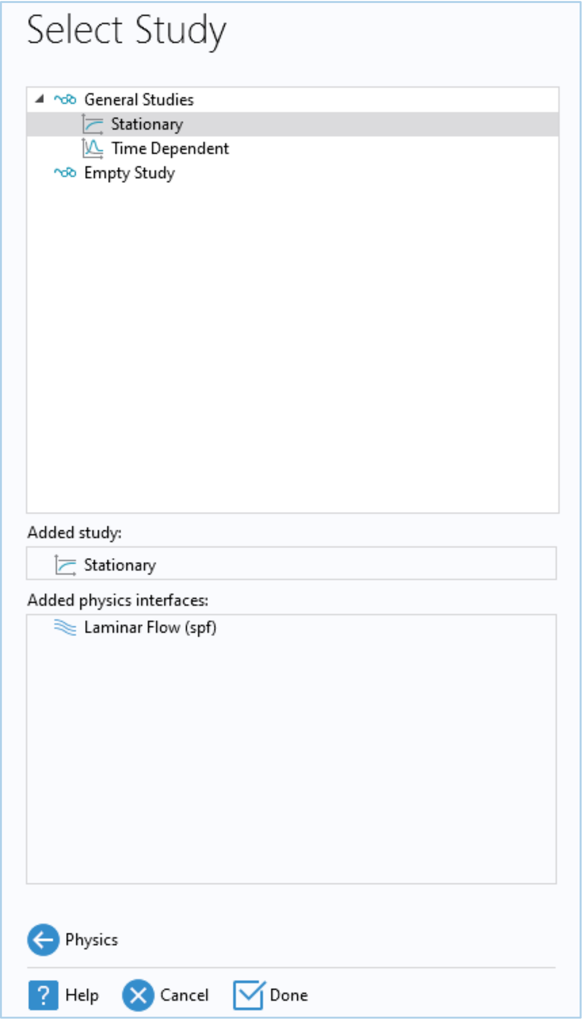
2. Select space dimension



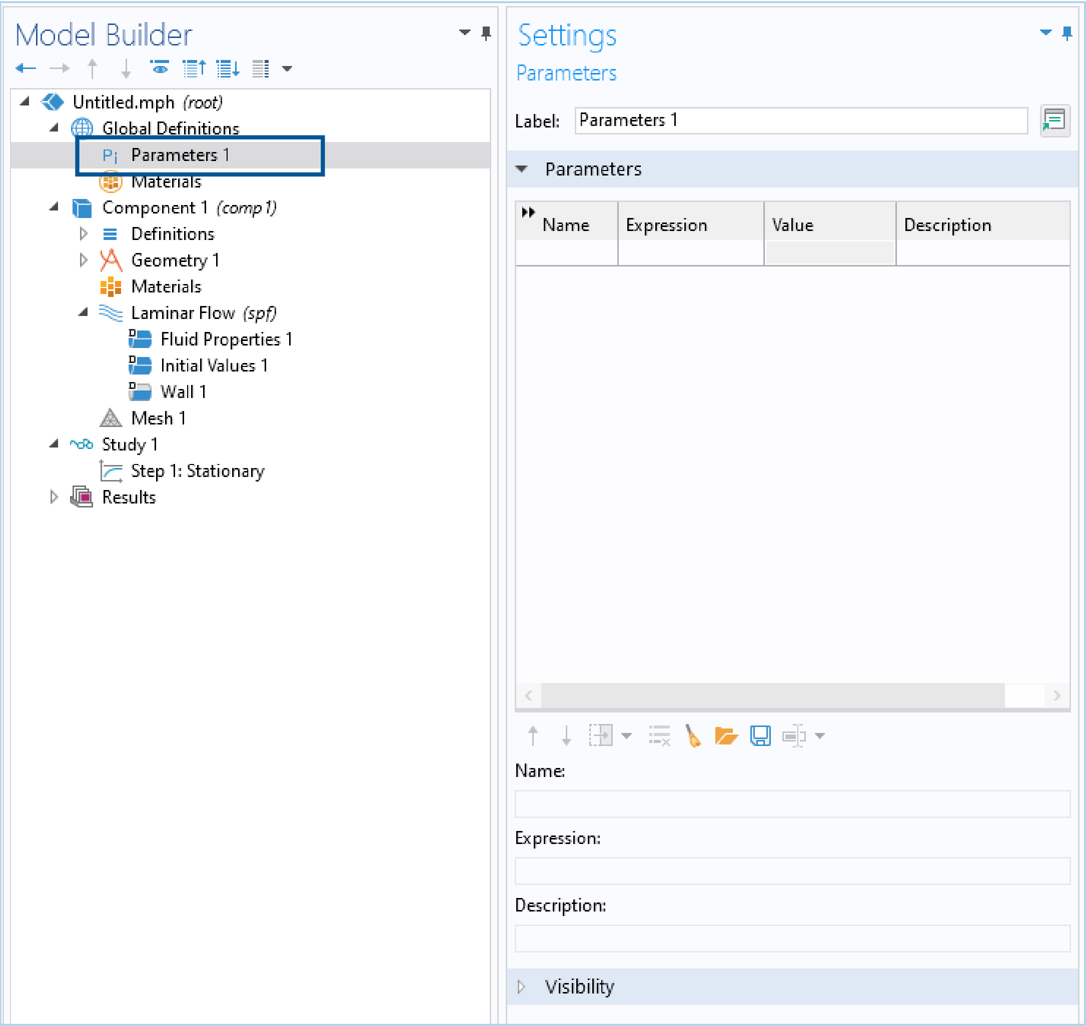
3. Select physics interfaces



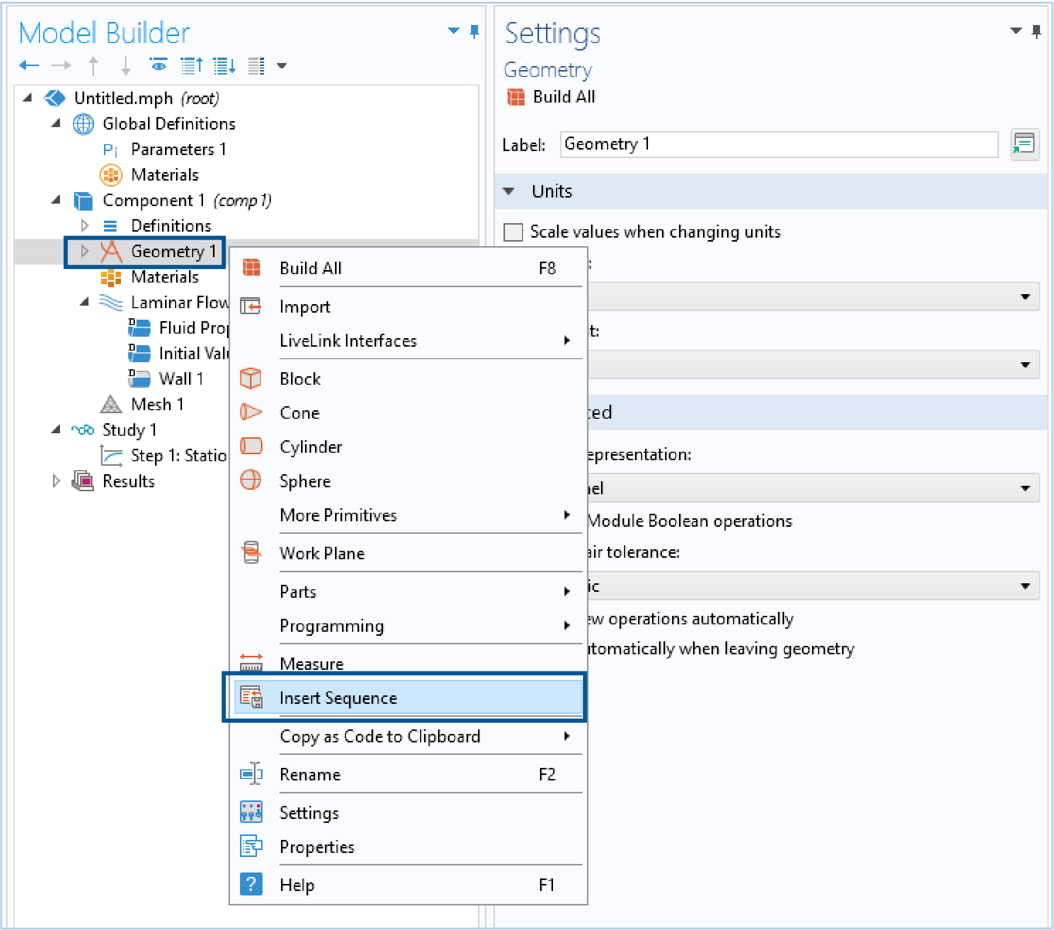
4. Select study



Demo: Parameters & Geometry

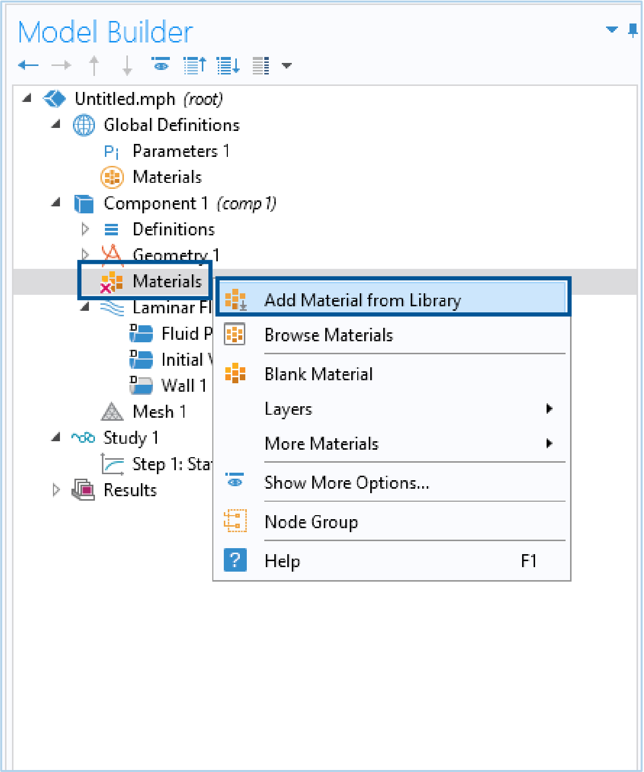


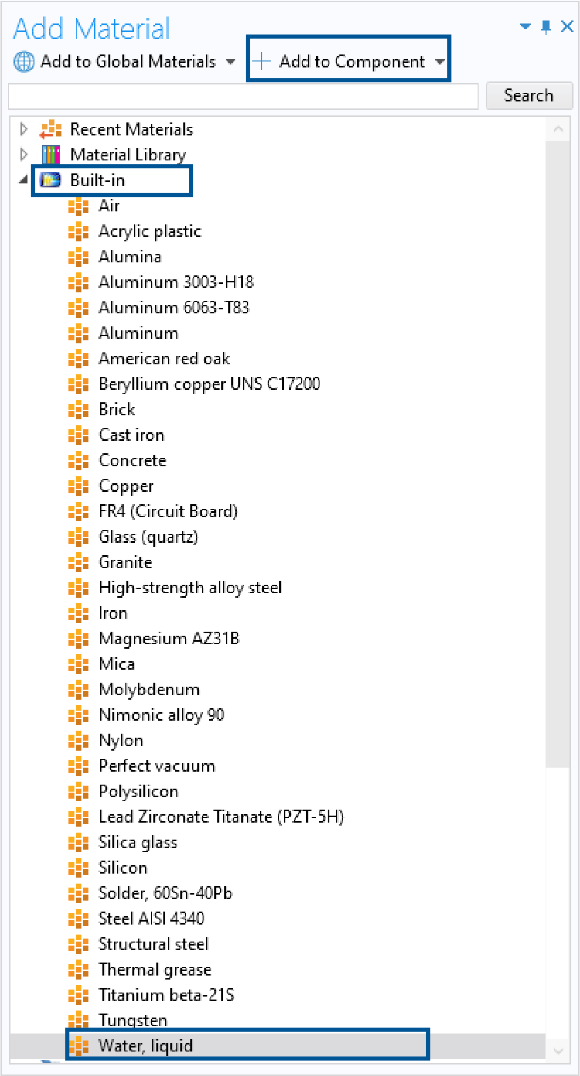
Navigate the course material and choose "backstep_parameters.txt"



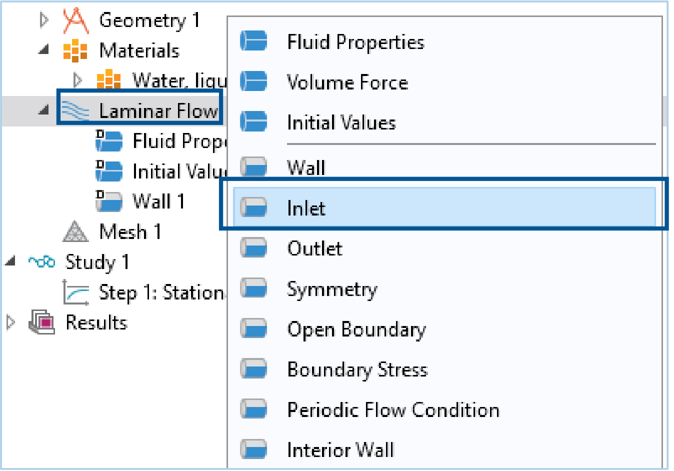
Navigate the course material and choose "backstep_geom_sequence.mph"

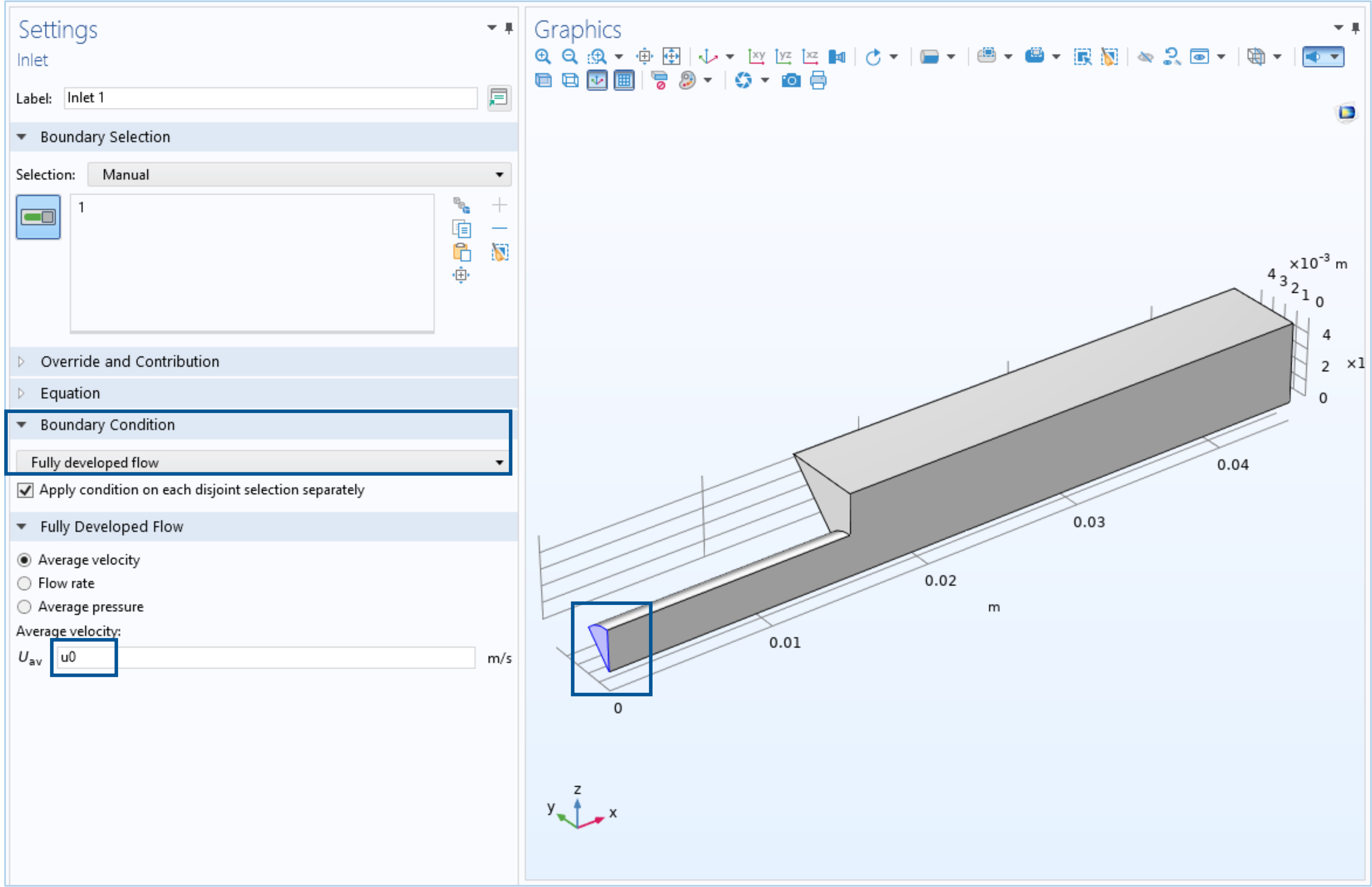
Demo: Materials



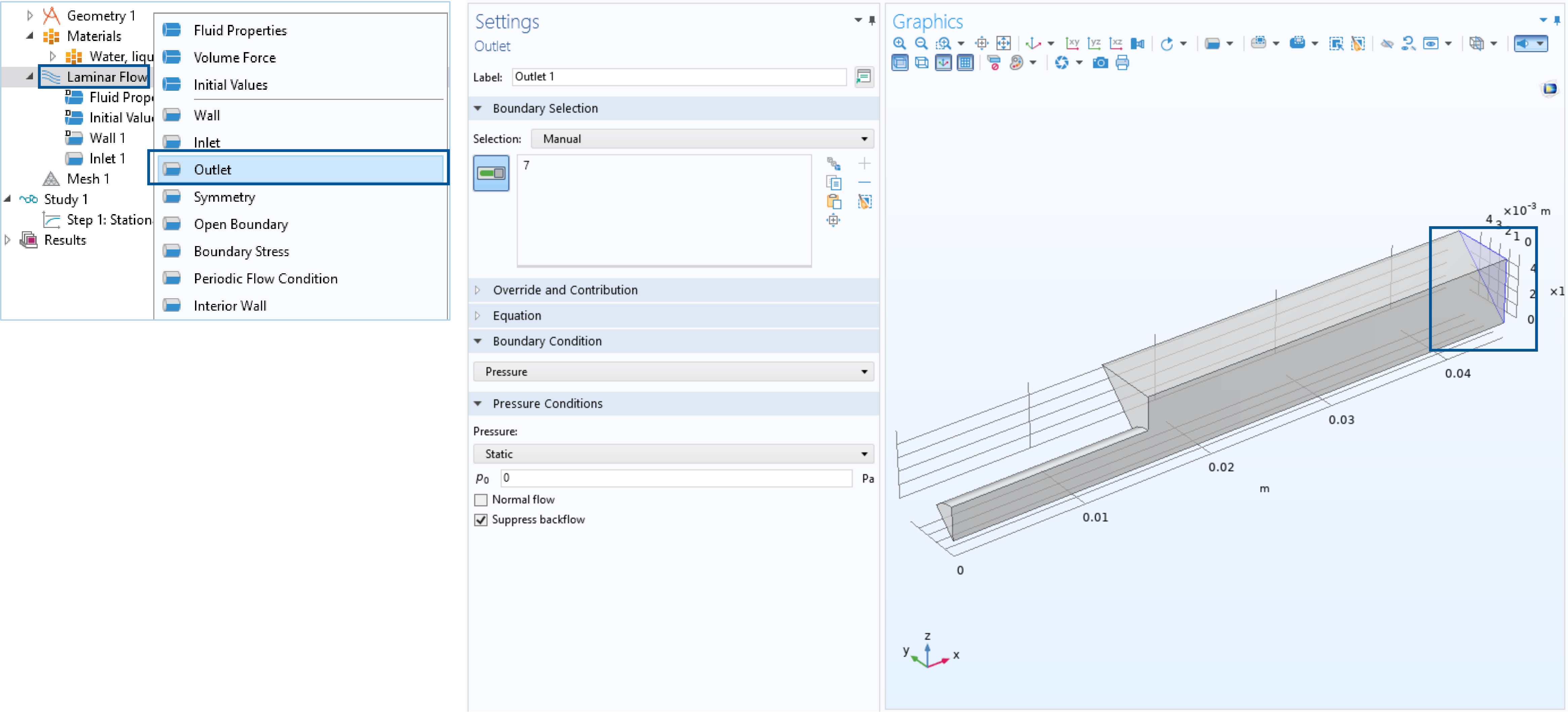


Demo: Laminar Flow - Inlet

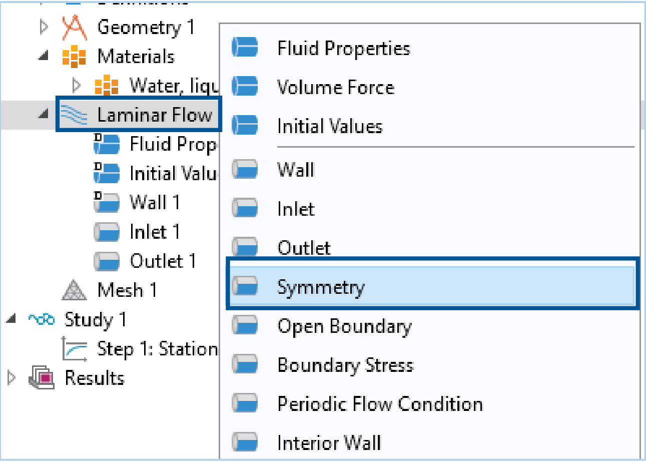


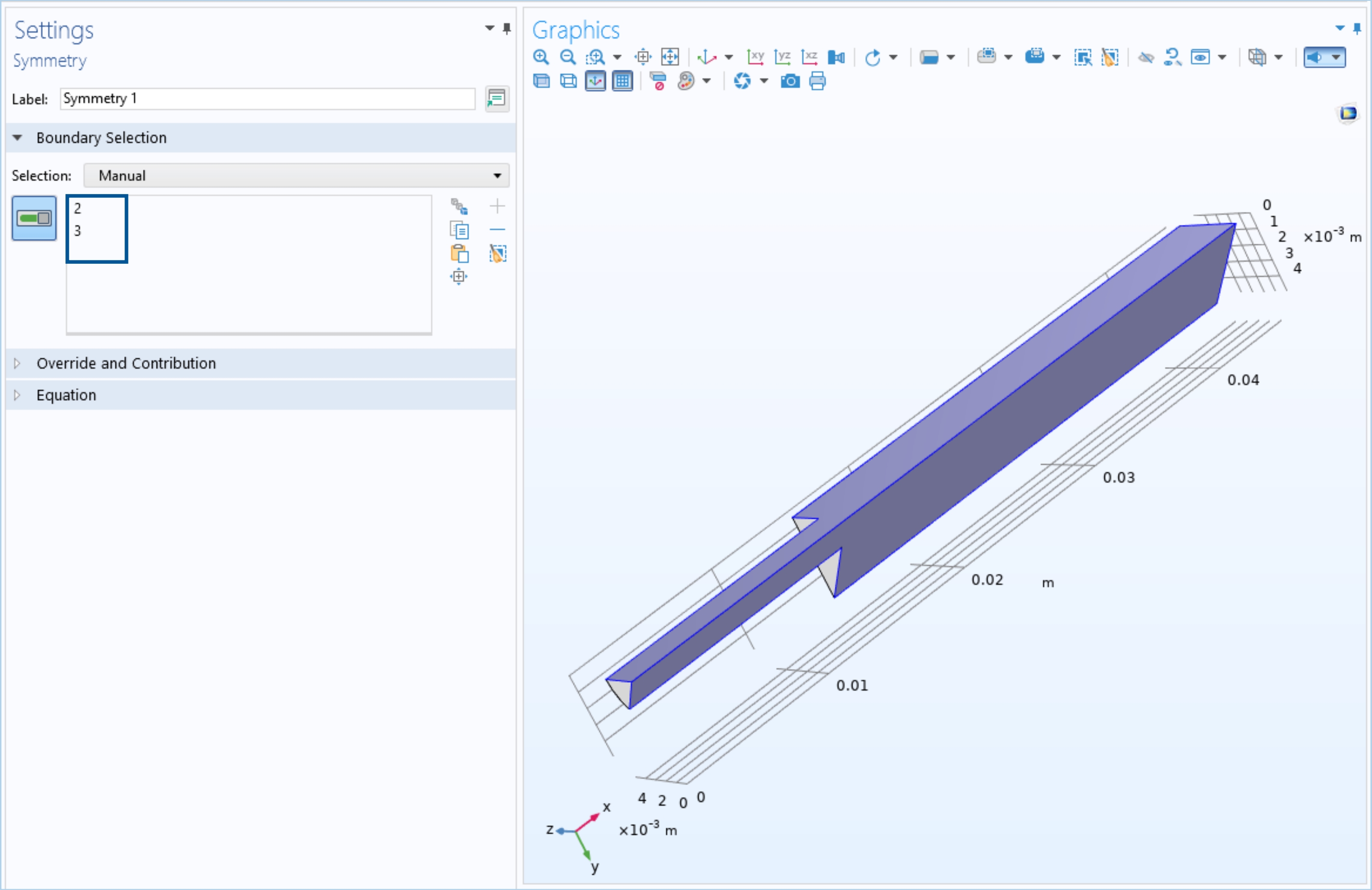


Demo: Laminar Flow - Outlet

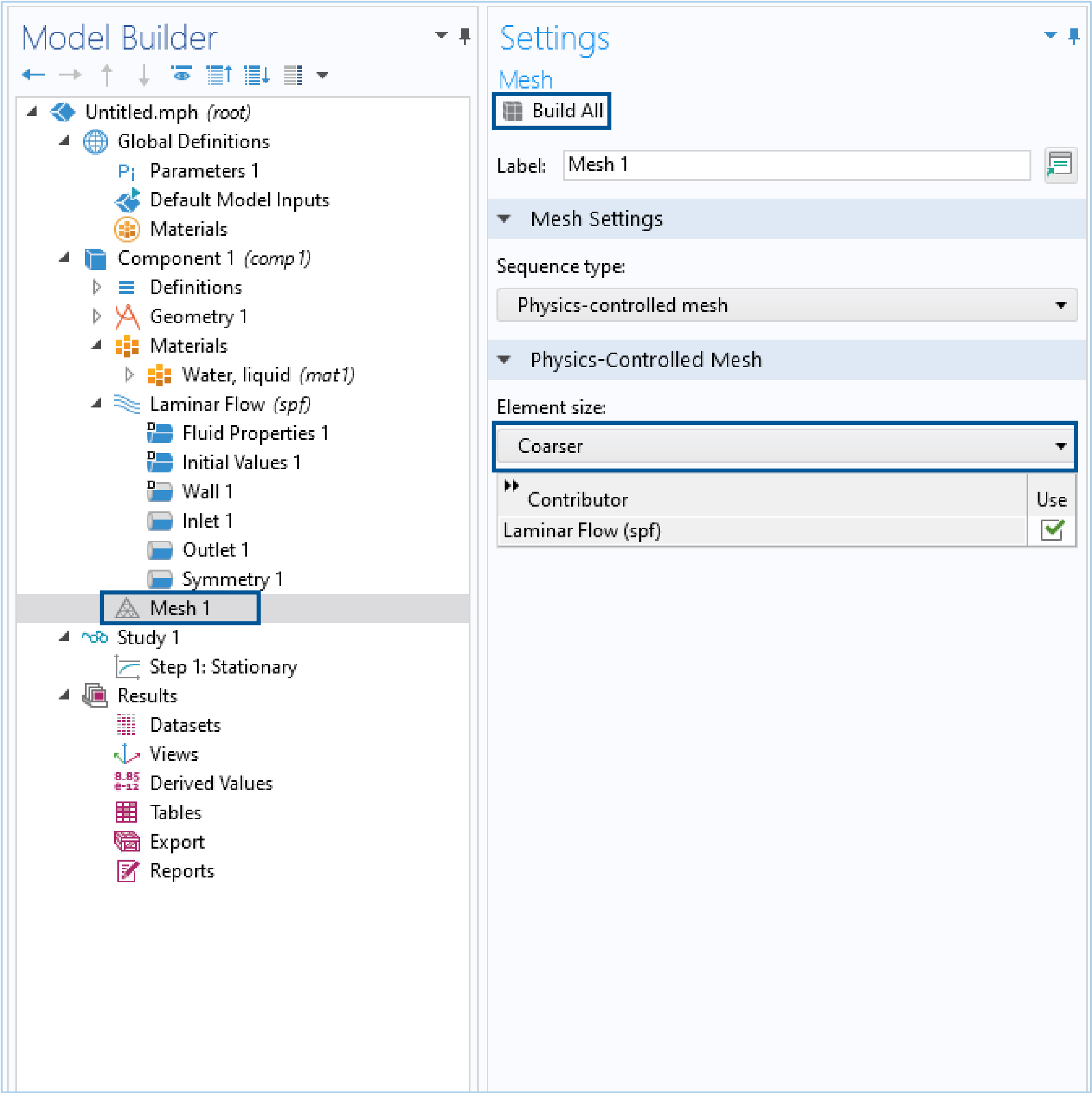


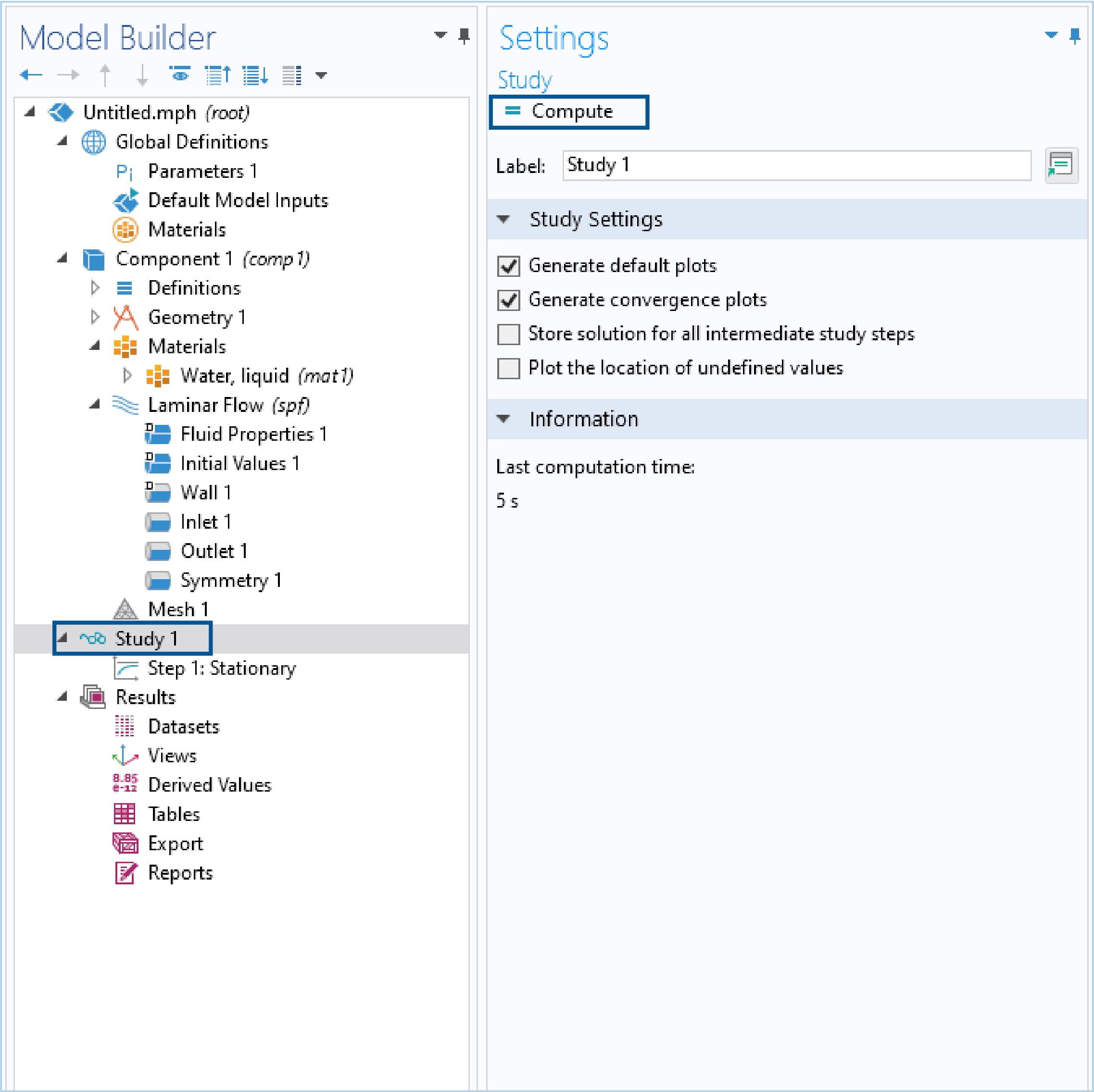
Demo: Laminar Flow - Symmetry



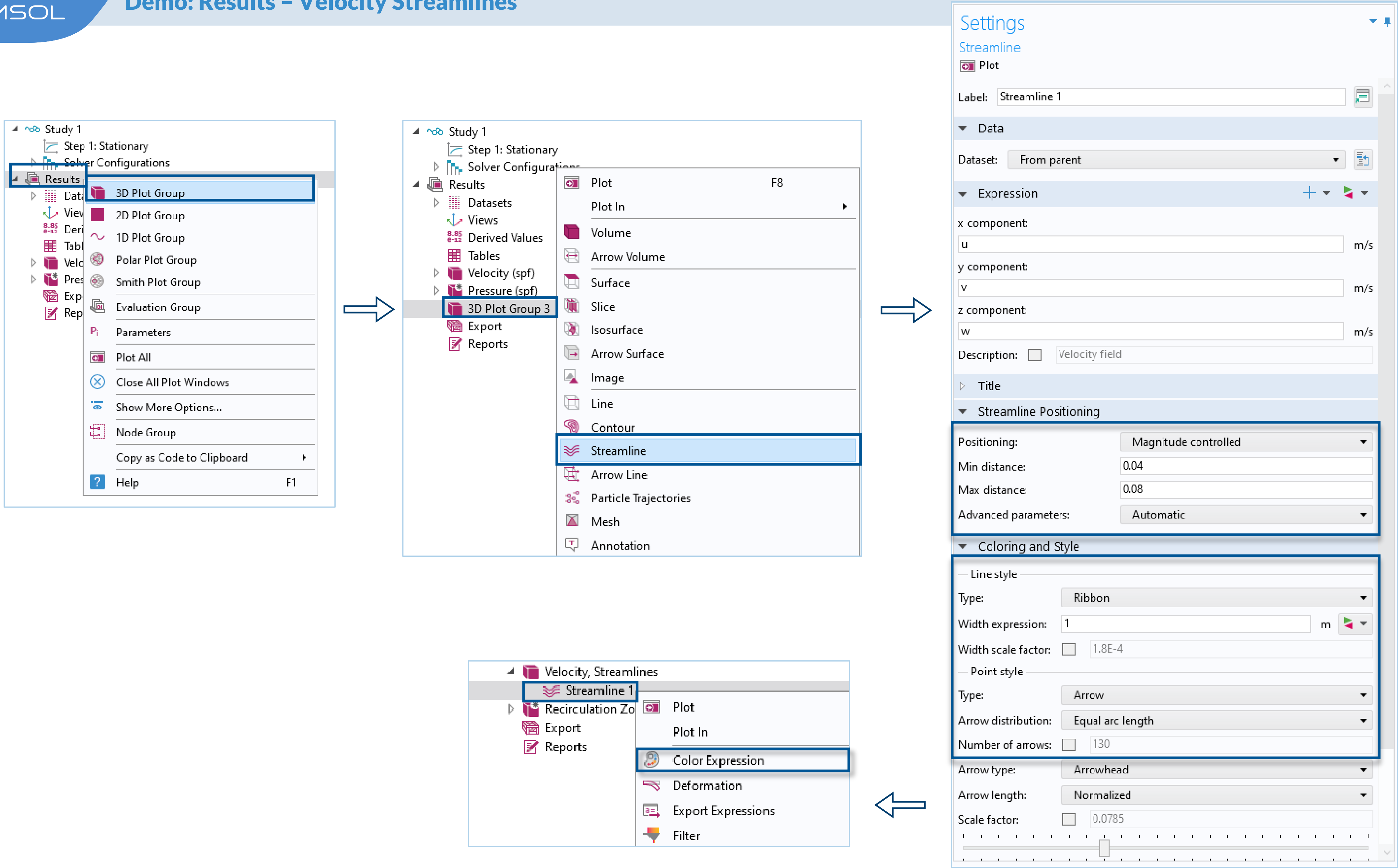


Demo: Mesh & Study





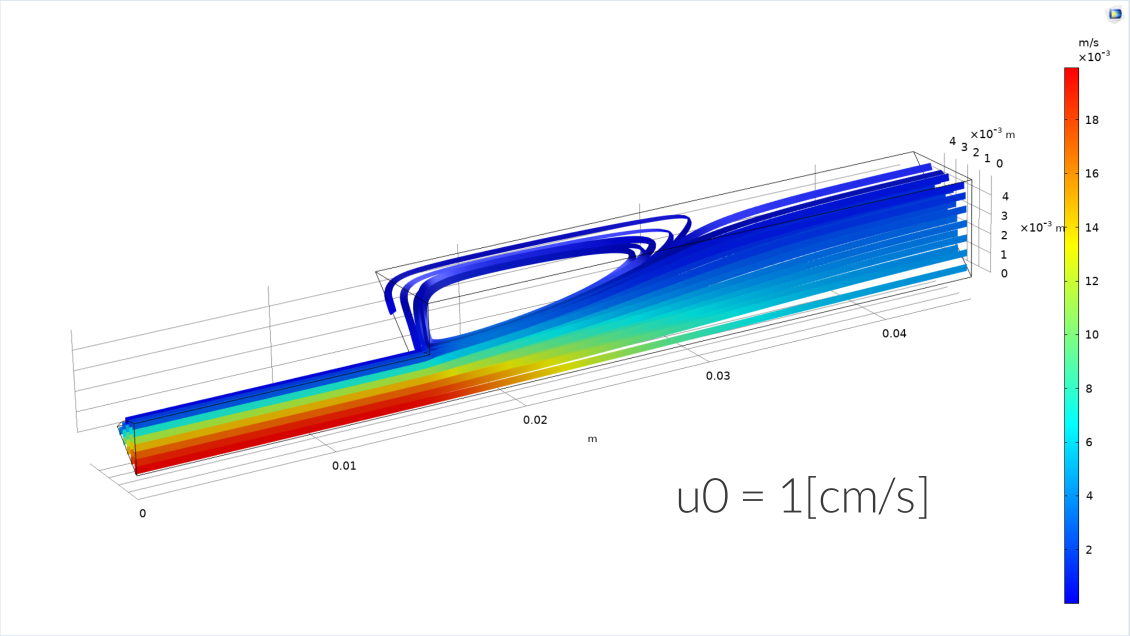
Demo: Results – Velocity Streamlines

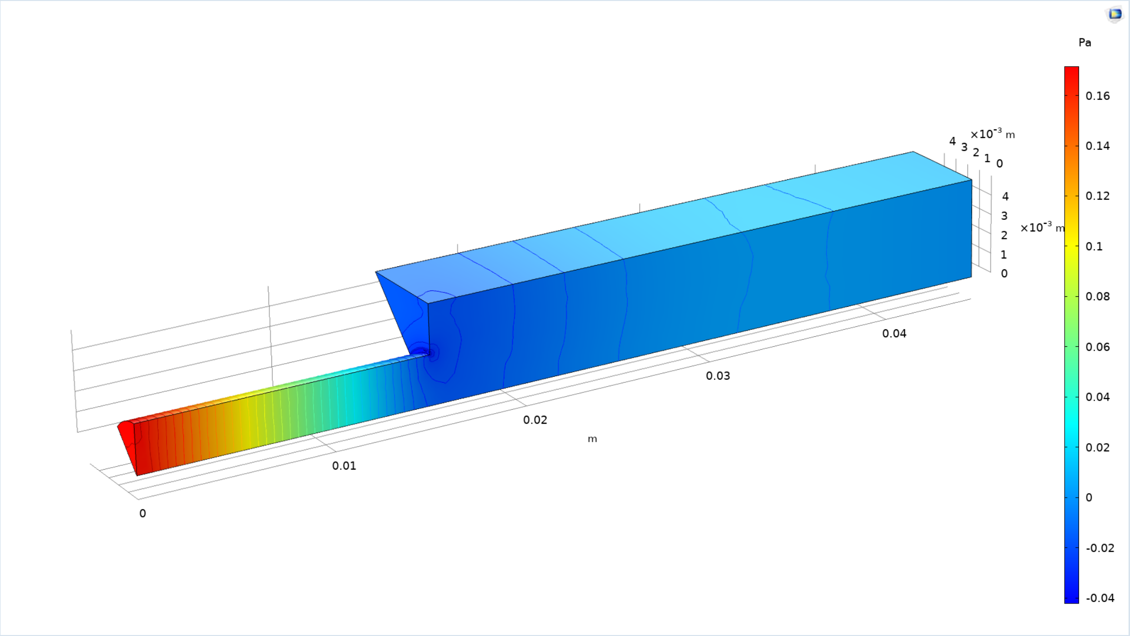


COMSOL

Results

- Flow and pressure fields
- Length of recirculation zone
- Total pressure loss
- Extending the model:
  - How is the recirculation zone affected by the inlet velocity?
  - When do we need to elongate the outlet section?
  - When do we need to use a turbulence model?
  - How can we create an app?





Model Implementation

- Model extension:
  - Variables for computing recirculation zone
  - Parametric sweep of inlet velocity
  - Mesh convergence study
  - Create an app

Definitions:
- Variable for evaluation

Geometry:
- Unchanged

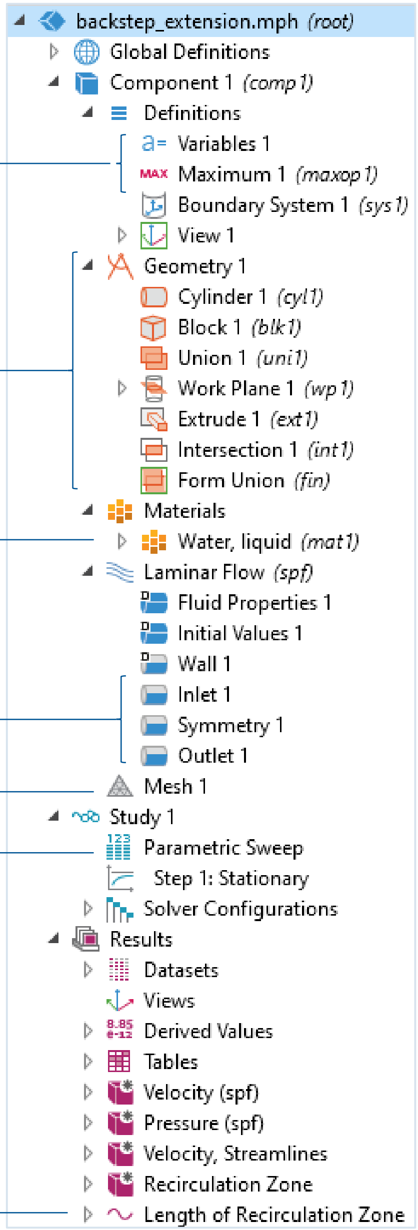
Materials:
- Unchanged

Physics:
- Unchanged

Mesh:
- Unchanged

Study:
- Add a parametric sweep

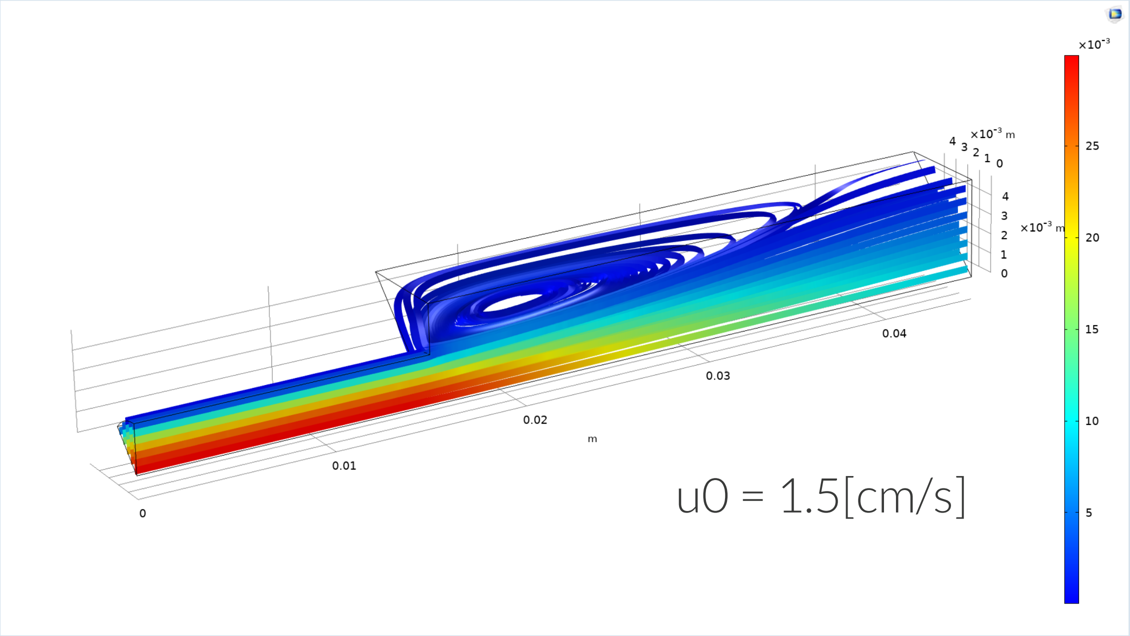
Results:
- Plot length of recirculation zone

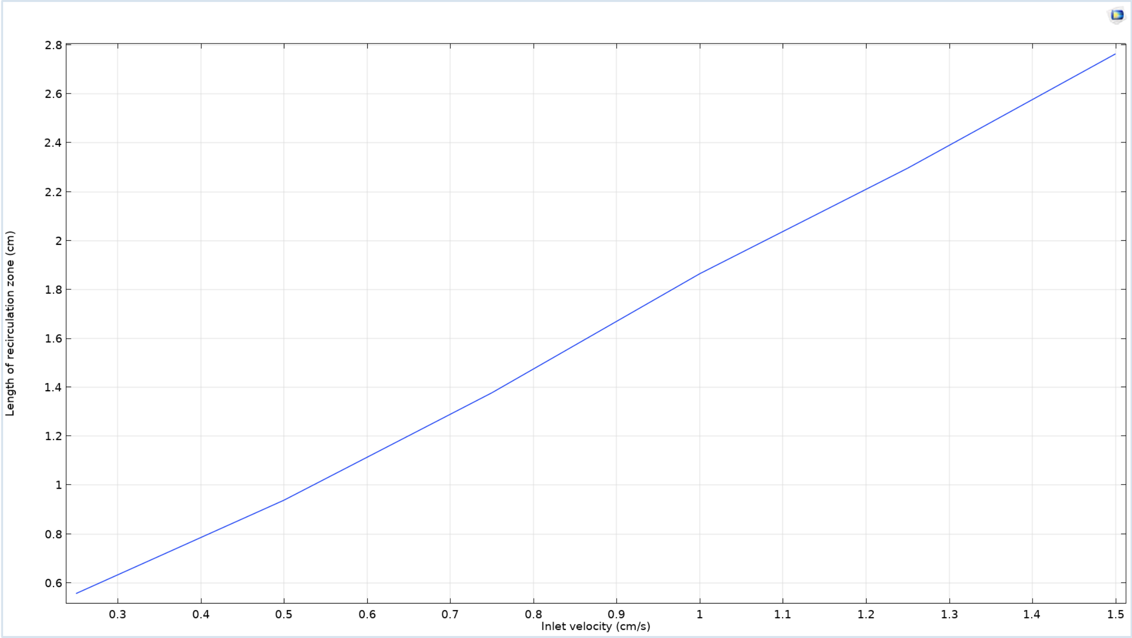


Overview of procedure for extended model example

Results

- Flow and pressure fields
- Length of recirculation zone
- Note:
  - Recirculation reaches the outlet
    -> elongate the outlet section

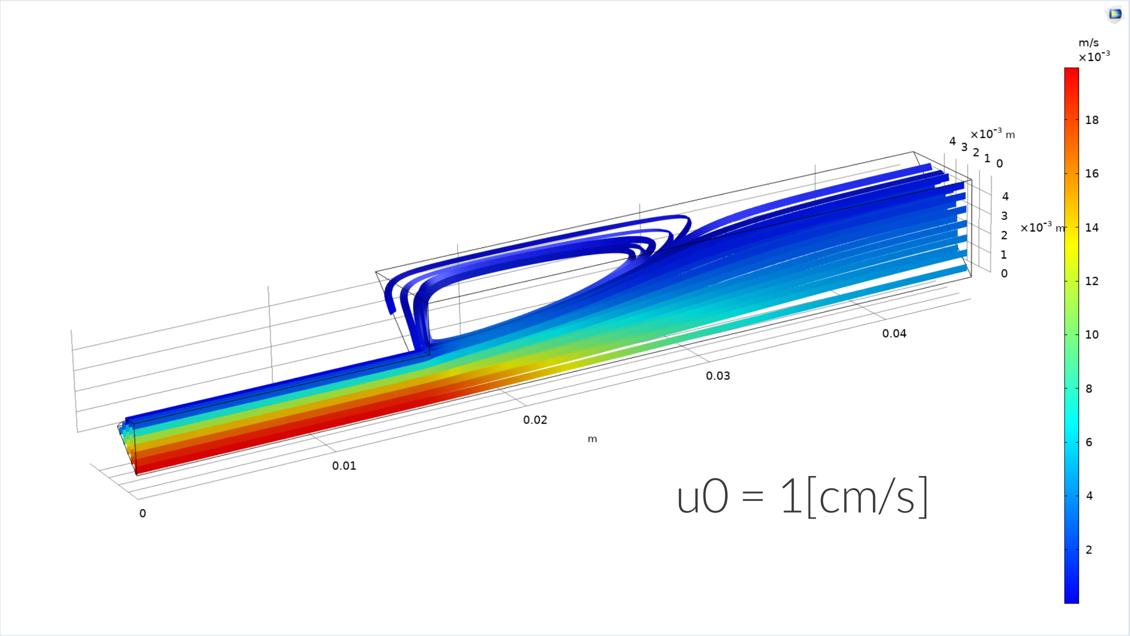


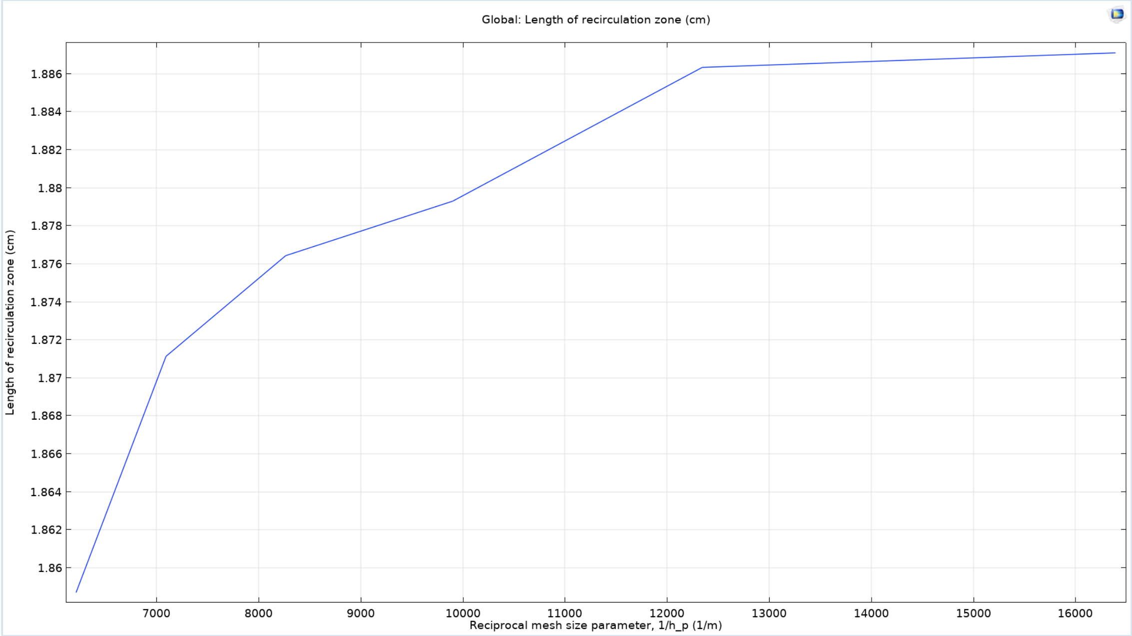


COMSOL

Results

- Flow and pressure fields
- Length of recirculation zone
- Next possible step:
  - Mesh convergence analysis, how does the length of the zone change with mesh size?
  - Seems to converge around a value of 1.89 cm



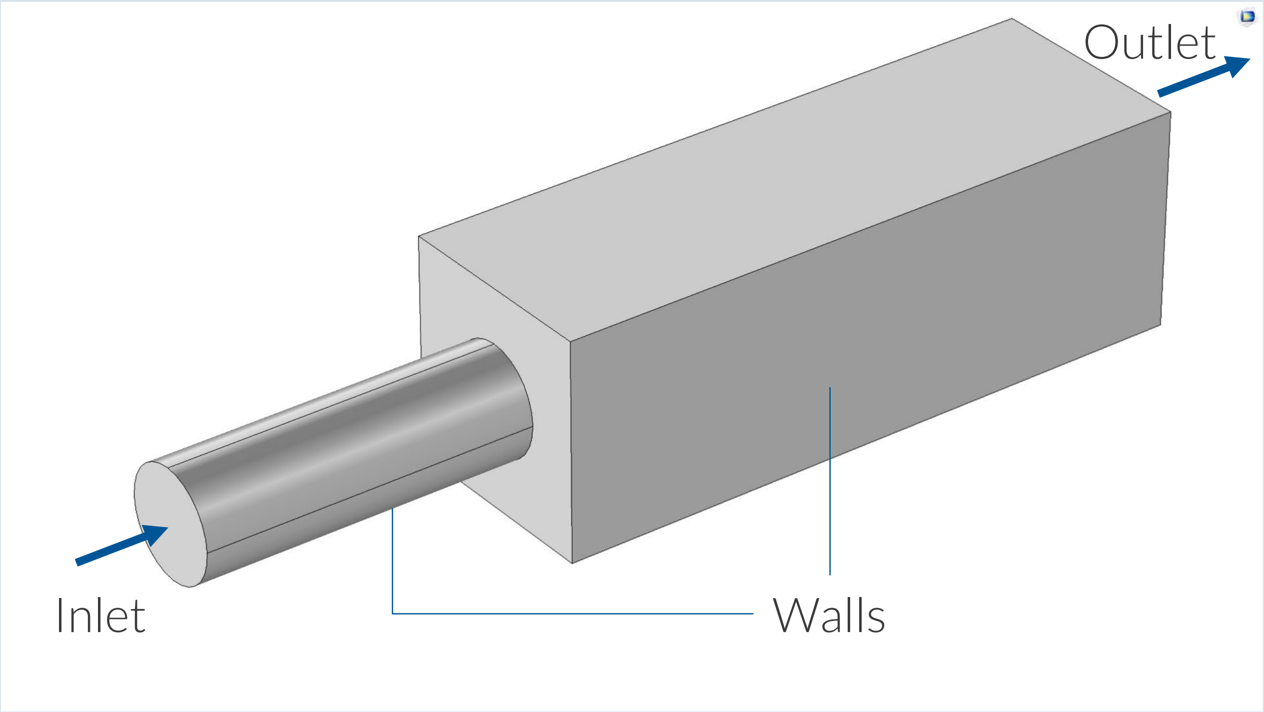


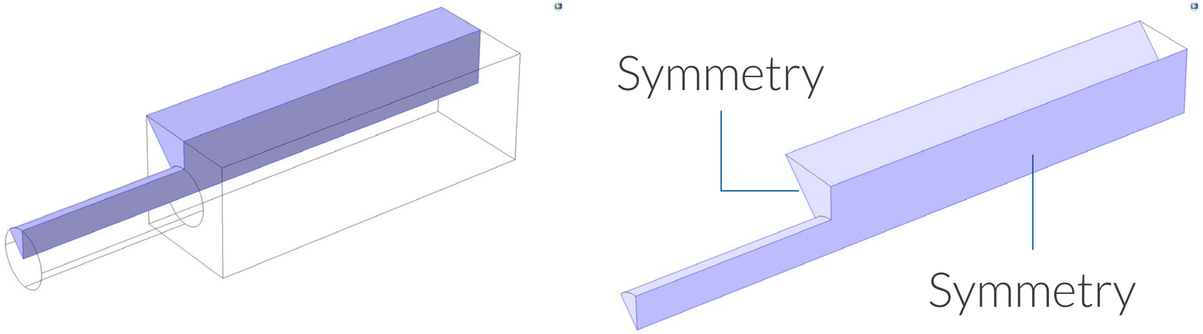
Exercise

Reproduce backstep model with varying inlet velocity

Model Definition

- Laminar flow in water
- Fully developed flow at the inlet
- Pressure condition at the outlet
- No-slip conditions at walls
- Symmetry conditions at the two lateral surfaces
- Why?
  - Typical expansion found in many systems, e.g. in medtech
  - Benchmark with flow separation





Due to symmetry, we only have to model one eight of the model domain, provided that the flow is steady and that the inlet flow is perpendicular to the inlet boundary.

Model Implementation

- First step:
  - Define the model and solve the problem for one parameter value
  - Variable for computing the length of the recirculation zone
- Second step:
  - Extend the model with a parameter sweep
  - Plot recirculation zone as a function of inlet velocity

Definitions:
- Load parameters from file

Geometry:
- Import sequence

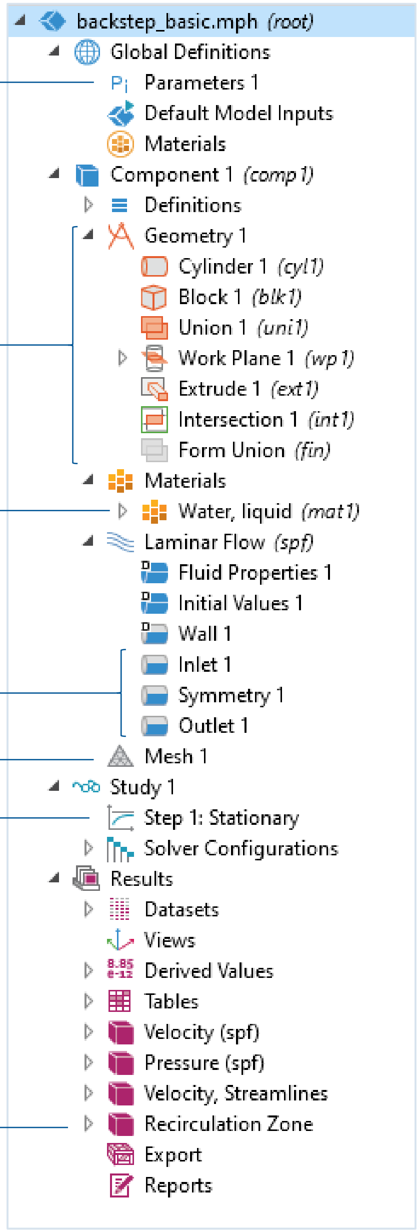
Materials:
- Load from Material Library

Physics:
- Define inlet, symmetry, and outlet

Mesh:
- Select mesh

Study:
- Select compute

Results:
- Plot streamlines



Overview of procedure for first model example