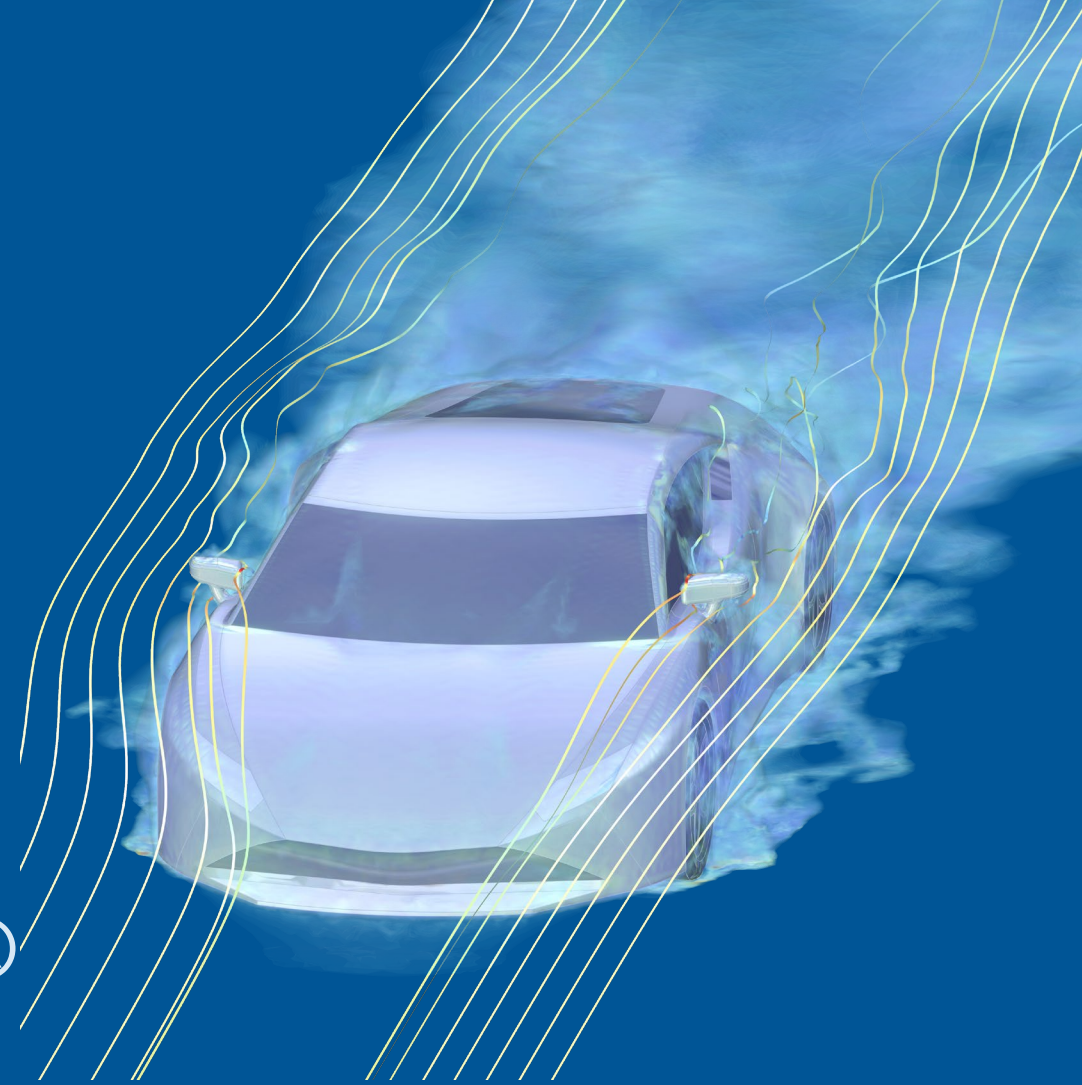


COMSOL Multiphysics® Laminar Flow Modeling

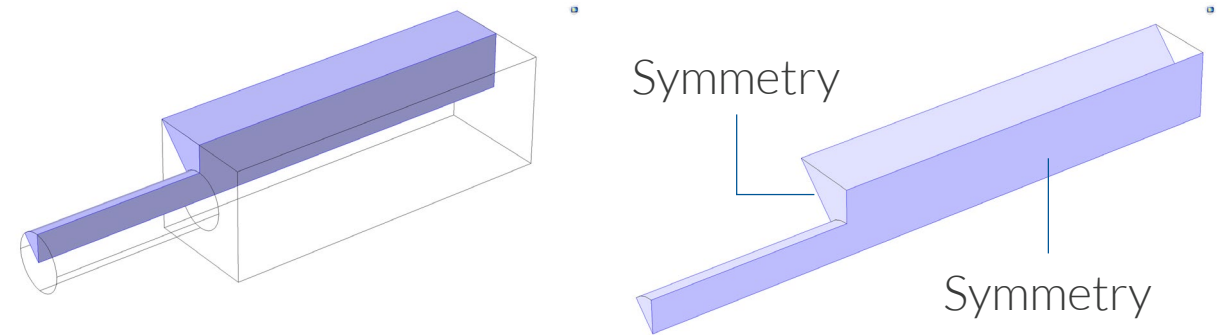
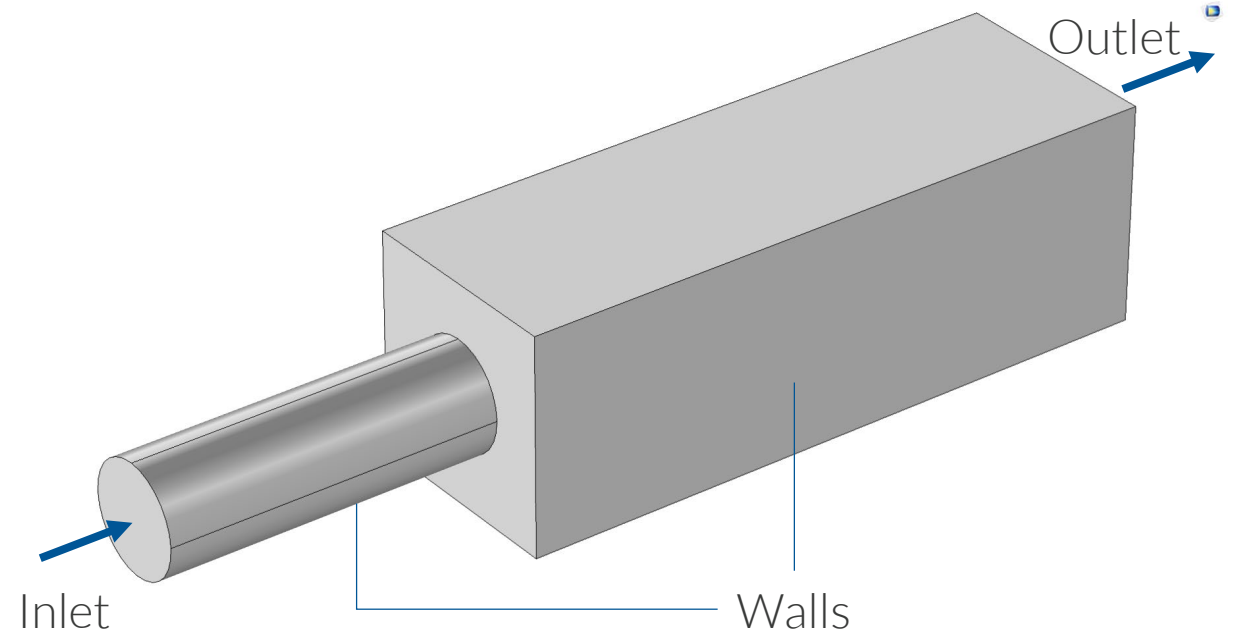


Laminar Flow

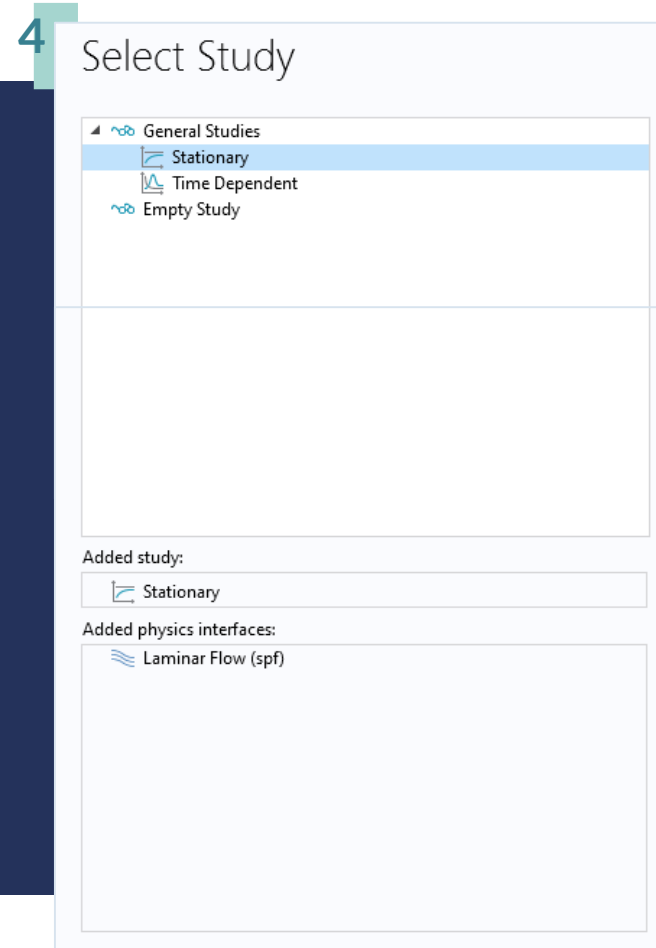
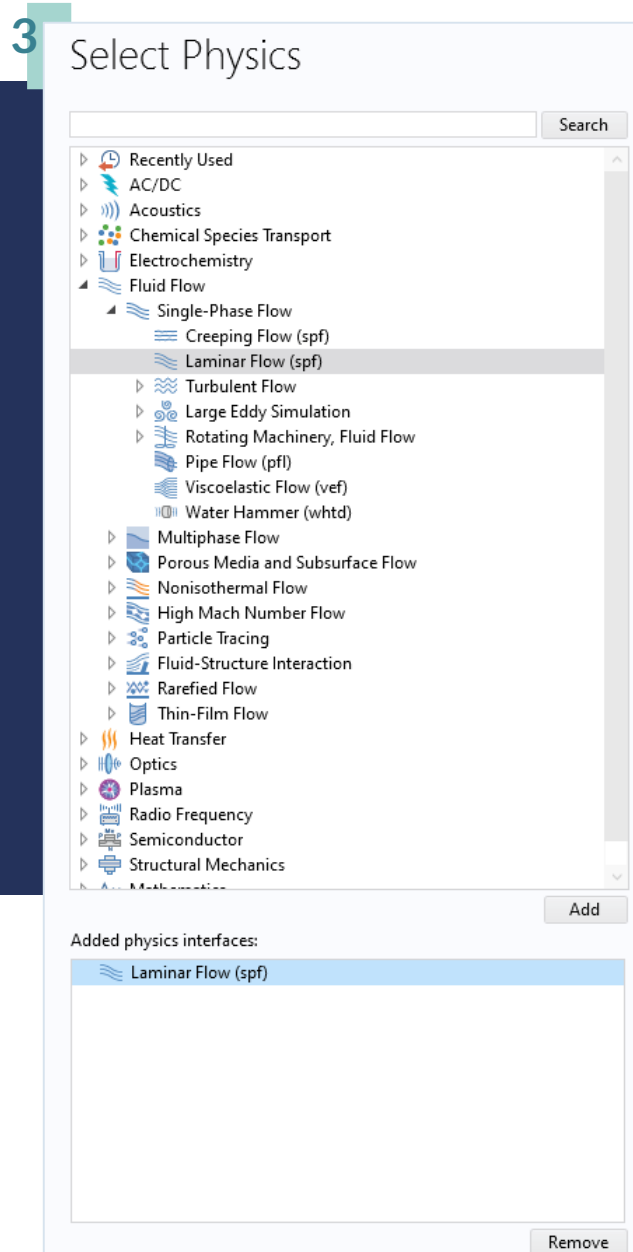
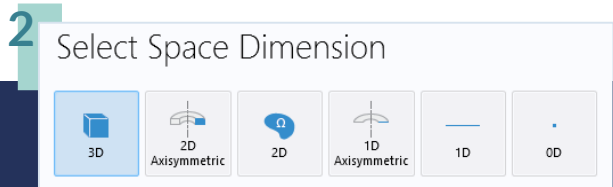
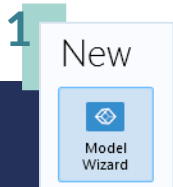
Demo

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 eighth of the model domain, provided that the flow is steady and that the inlet flow is perpendicular to the inlet boundary.



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

Application Builder and Model Manager

Buttons to switch to the Application Builder or Model Manager.

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

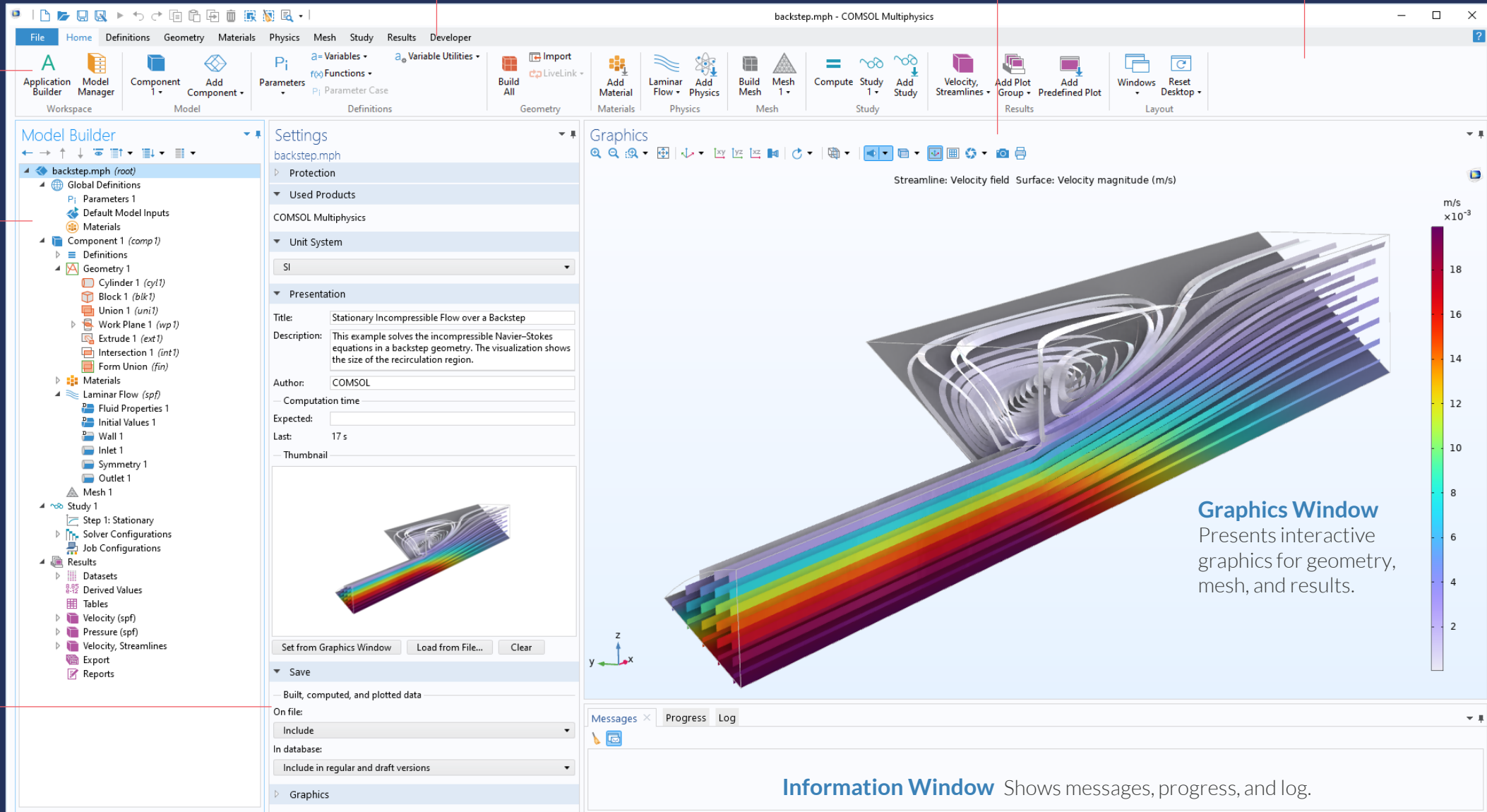
Shows the settings for the node that is selected in the model tree.

Ribbon

Includes controls for all steps of the modeling process.

Graphics Window Toolbar

COMSOL Desktop® Model Builder



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

The screenshot displays the COMSOL Multiphysics software interface. At the top, a ribbon contains tabs for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer. Below the ribbon is a toolbar with icons for various modeling operations. On the left, the Model Builder tree shows a hierarchical structure of the model, including Global Definitions, Component 1 (with sub-entities for Geometry, Materials, and Physics), Mesh 1, and Study 1. The central Settings window is open to the 'backstep.mph' model, showing details for Protection, Used Products, Unit System (SI), and Presentation. The right side of the interface features a Graphics window displaying a 3D visualization of the velocity field in a backstep geometry. The visualization uses a color scale from blue (low velocity) to red (high velocity), with a legend on the right indicating values in m/s $\times 10^{-3}$ ranging from 2 to 18. A smaller thumbnail of the same visualization is shown in the Settings window. At the bottom, a Messages window is visible with tabs for Messages, Progress, and Log.

Home tab with the most common commands in the modeling process

Model tree shows sequences of operations

Definitions

Geometry

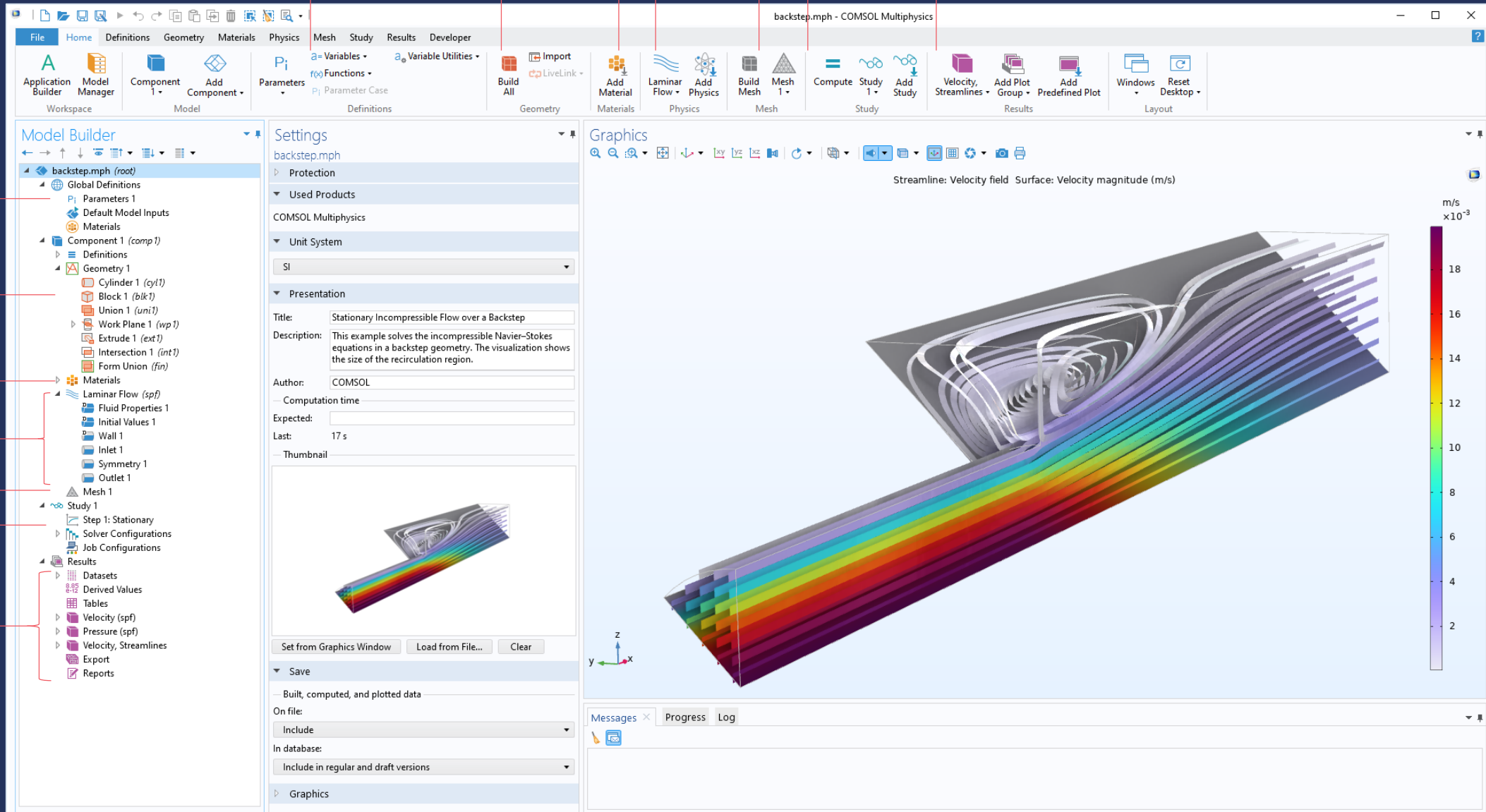
Materials

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:

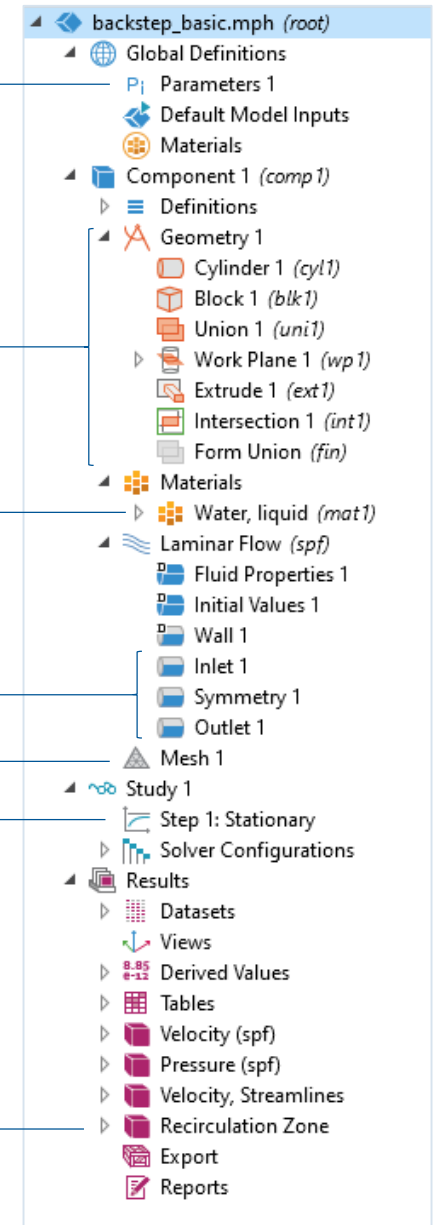
- Select mesh

Study:

- Select compute

Results:

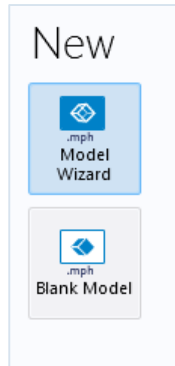
- Plot streamlines



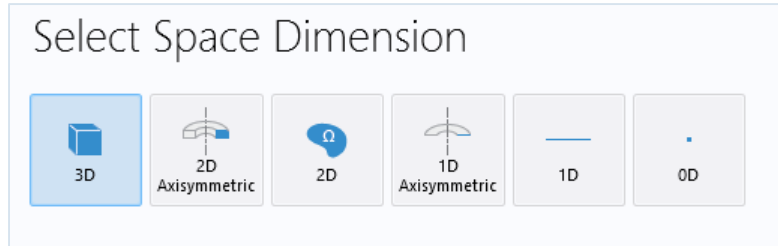
Overview of procedure for first model example

Demo: Model Wizard

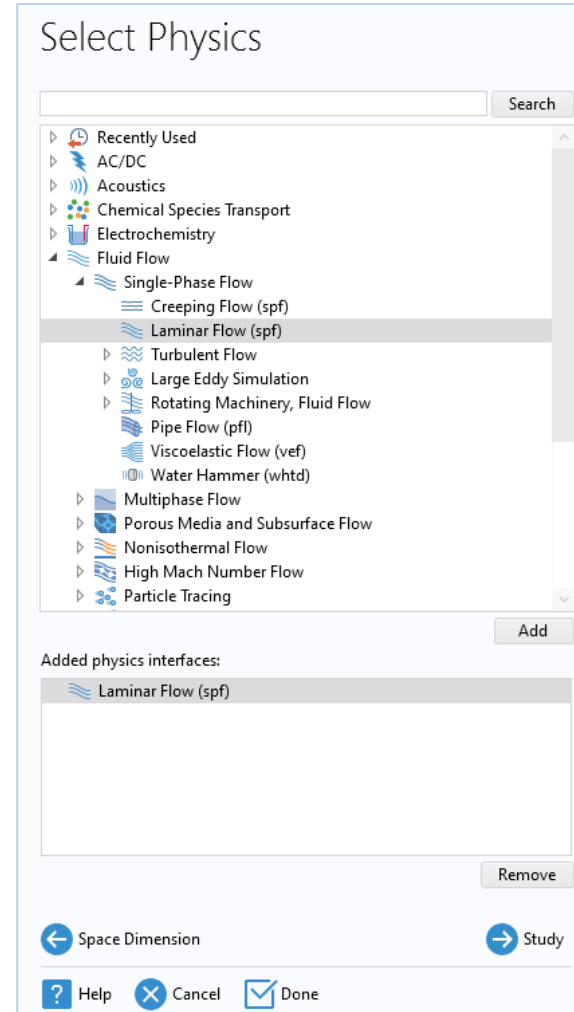
1. Select Model Wizard



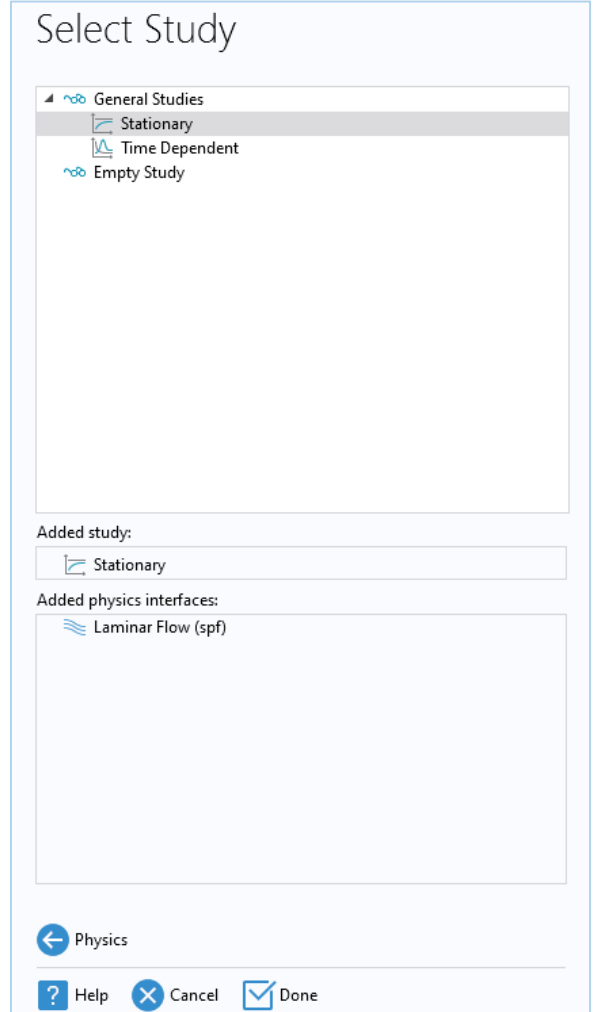
2. Select space dimension

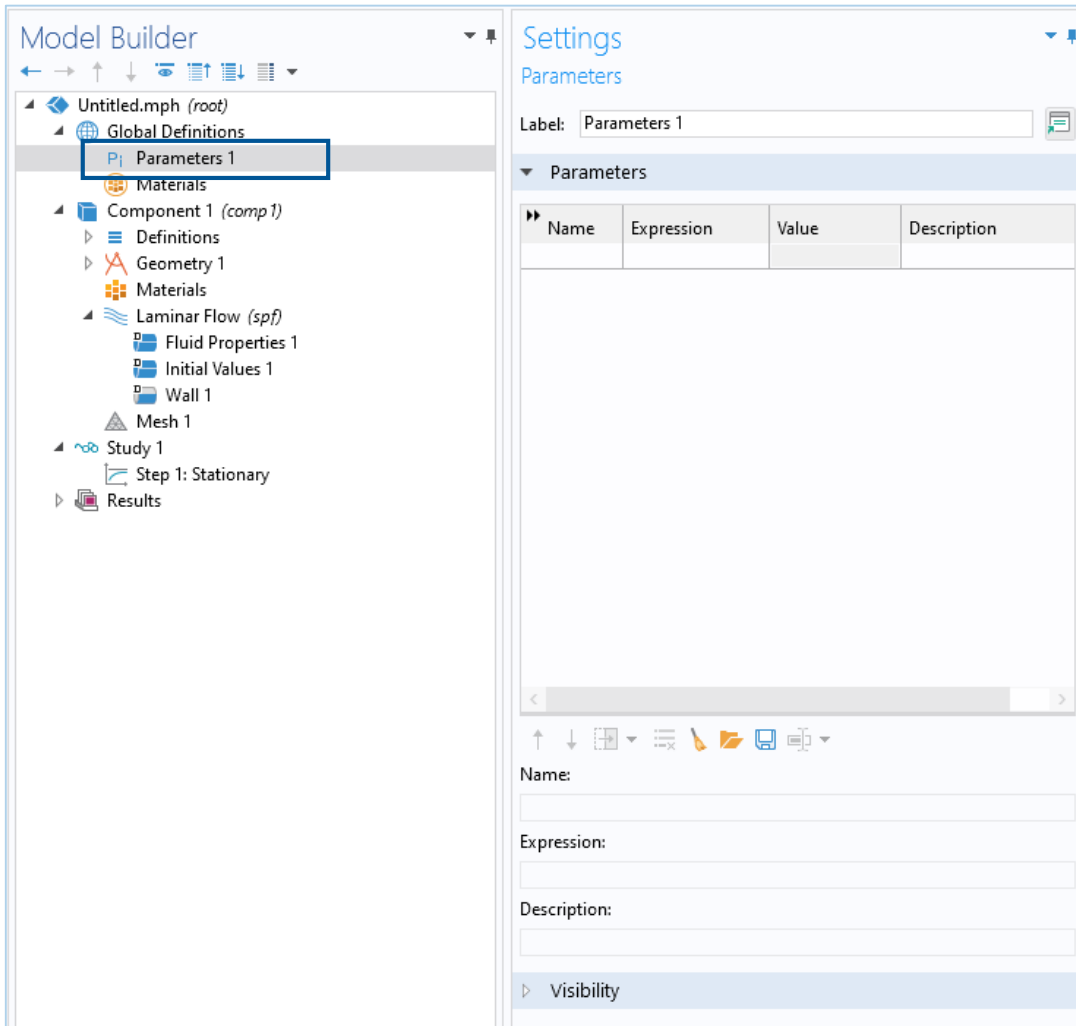


3. Select physics interfaces

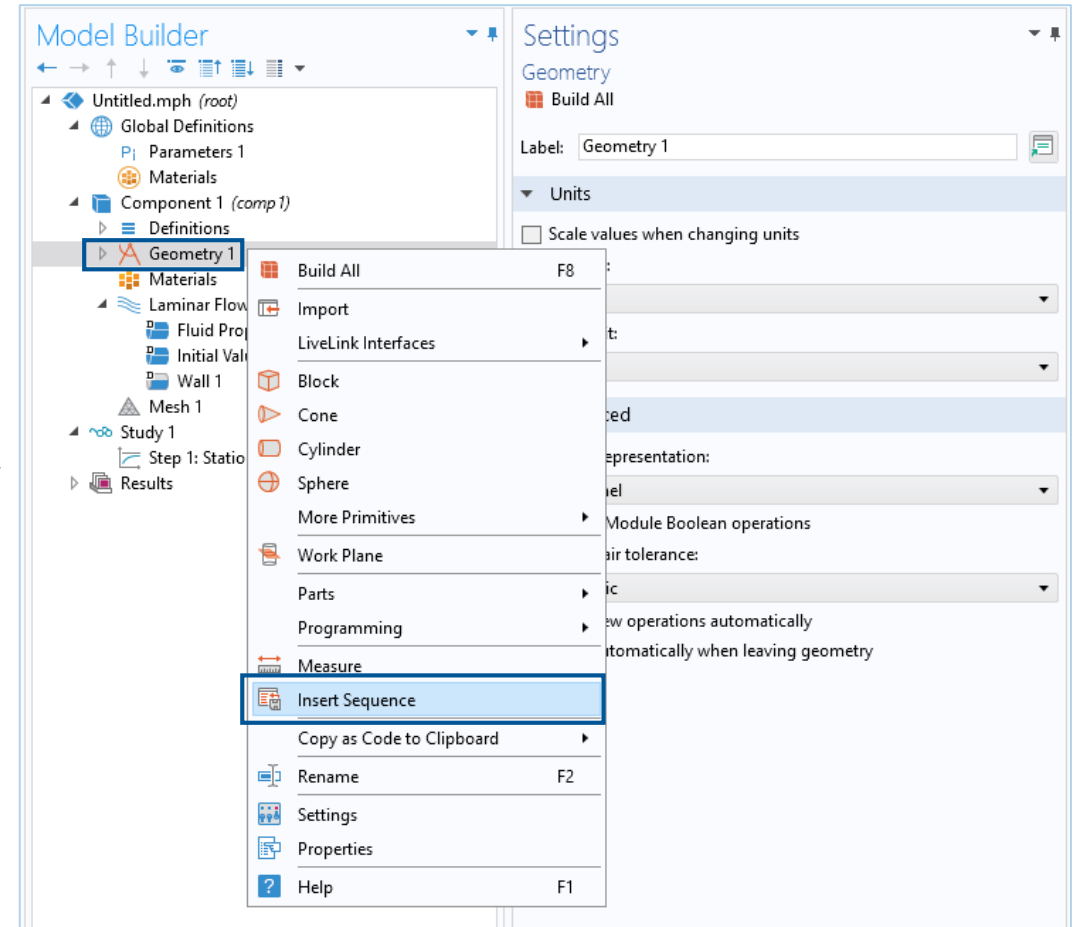


4. Select study

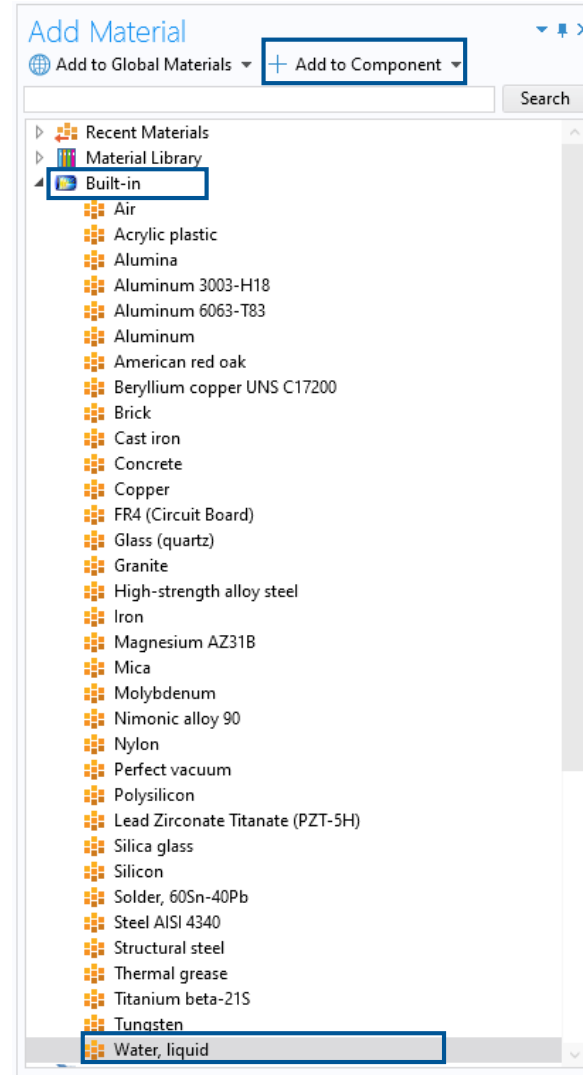
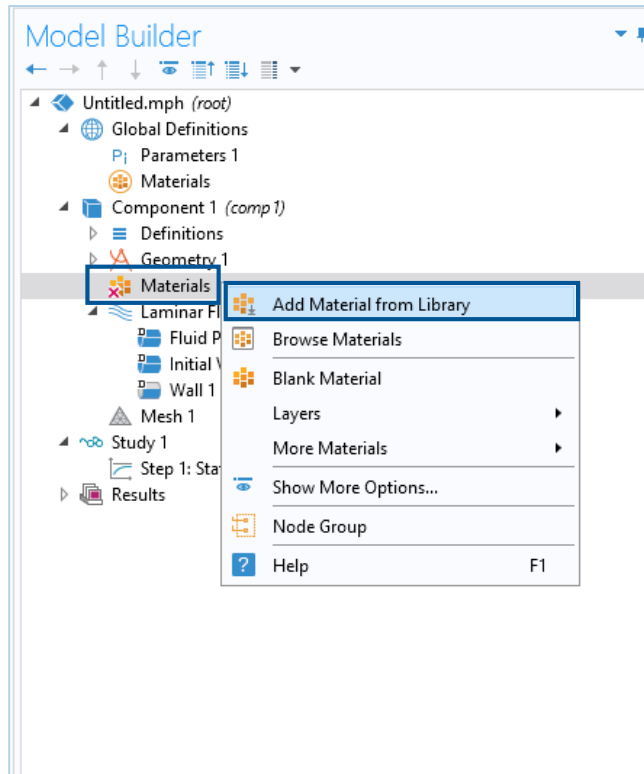




Navigate the course material and choose
“backstep_parameters.txt”



Navigate the course material and choose
“backstep_geom_sequence.mph”



The software tree on the left shows the following structure:

- Geometry 1
 - Materials
 - Water, liquid
 - Laminar Flow** (highlighted)
 - Fluid Properties
 - Initial Values
 - Wall
 - Inlet** (highlighted)
 - Outlet
 - Symmetry
 - Open Boundary
 - Boundary Stress
 - Periodic Flow Condition
 - Interior Wall
 - Mesh 1
- Study 1
 - Step 1: Stationary
 - Results

The Settings panel for the Inlet boundary condition is shown. The label is "Inlet 1". The boundary selection is "Manual" with a selection of "1". The boundary condition is set to "Fully developed flow". The option "Apply condition on each disjoint selection separately" is checked. Under "Fully Developed Flow", "Average velocity" is selected. The average velocity is set to $U_{av} = u_0$ m/s.

The Graphics window displays a 3D model of a channel with a mesh. The channel has a length of 0.04 m, a width of 0.02 m, and a height of 0.004 m. The mesh is refined near the inlet. The coordinate system (x, y, z) is shown at the bottom left.

▸ Geometry 1
 ▸ Materials
 ▸ Water, liqu
 ▸ Laminar Flow
 ▸ Fluid Prop
 ▸ Initial Valu
 ▸ Wall 1
 ▸ Inlet 1
 ▸ Outlet
 ▸ Mesh 1
 ▸ Study 1
 ▸ Step 1: Station
 ▸ Results

- ▢ Fluid Properties
- ▢ Volume Force
- ▢ Initial Values
- ▢ Wall
- ▢ Inlet
- ▢ Outlet
- ▢ Symmetry
- ▢ Open Boundary
- ▢ Boundary Stress
- ▢ Periodic Flow Condition
- ▢ Interior Wall

Settings

Outlet

Label: Outlet 1

Boundary Selection

Selection: Manual

7

▸ Override and Contribution
 ▸ Equation
 ▾ Boundary Condition
 Pressure

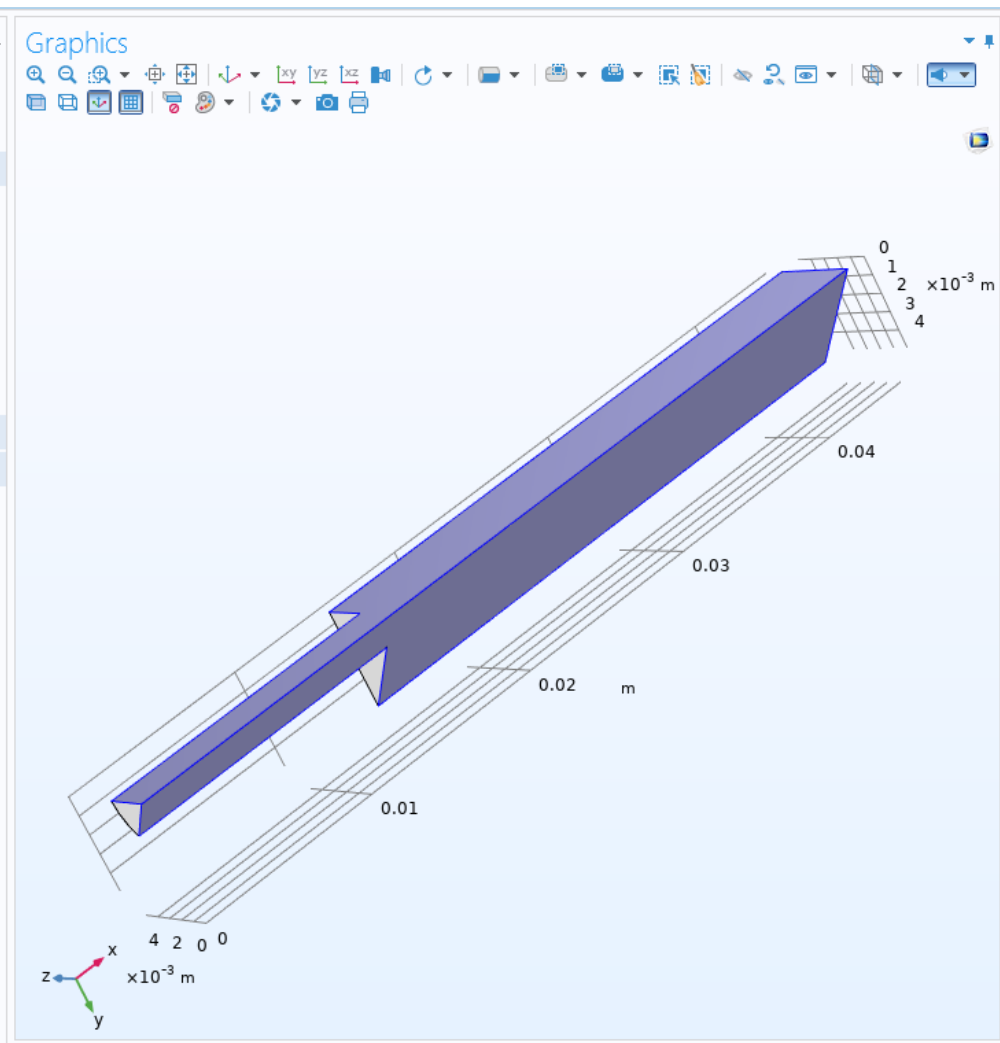
▾ Pressure Conditions
 Pressure:
 Static
 p_0 0 Pa
 Normal flow
 Suppress backflow

Graphics

0.01 0.02 0.03 0.04 m
 4×10^{-3} m
 2 1 0
 0 2 4
 0 1
 x y z

Geometry 1
 Materials
 Water, liquid
 Laminar Flow
 Fluid Properties
 Initial Values
 Wall
 Inlet
 Outlet
Symmetry
 Open Boundary
 Boundary Stress
 Periodic Flow Condition
 Interior Wall
 Mesh 1
 Study 1
 Step 1: Stationary
 Results

Settings
 Symmetry
 Label: Symmetry 1
 Boundary Selection
 Selection: Manual
 2
 3
 Override and Contribution
 Equation



The image shows two side-by-side screenshots of the COMSOL Model Builder interface, illustrating the workflow from meshing to solving a study. An arrow points from the left screenshot to the right one.

Left Screenshot (Meshing):

- Model Builder:** The tree view shows the hierarchy: *Untitled.mph (root)* > *Global Definitions* > *Parameters 1*, *Default Model Inputs*, *Materials*; *Component 1 (comp 1)* > *Definitions*, *Geometry 1*, *Materials* > *Water, liquid (mat 1)*; *Laminar Flow (spf)* > *Fluid Properties 1*, *Initial Values 1*, *Wall 1*, *Inlet 1*, *Outlet 1*, *Symmetry 1*. The **Mesh 1** node is selected and highlighted.
- Settings (Mesh):** The **Build All** button is highlighted. The **Label** is "Mesh 1". Under **Mesh Settings**, the **Sequence type** is "Physics-controlled mesh". Under **Physics-Controlled Mesh**, the **Element size** is set to "Coarser". The **Contributor** is "Laminar Flow (spf)" with a "Use" checkbox checked.

Right Screenshot (Solving):

- Model Builder:** The tree view is identical to the left screenshot, but the **Study 1** node is now selected and highlighted.
- Settings (Study):** The **Compute** button is highlighted. The **Label** is "Study 1". Under **Study Settings**, the checkboxes for "Generate default plots" and "Generate convergence plots" are checked, while "Store solution for all intermediate study steps" and "Plot the location of undefined values" are unchecked. Under **Information**, the **Last computation time** is displayed as "5 s".

Study 1
 Step 1: Stationary
 Solver Configurations
 Results
 3D Plot Group
 2D Plot Group
 1D Plot Group
 Polar Plot Group
 Smith Plot Group
 Evaluation Group
 Parameters
 Plot All
 Close All Plot Windows
 Show More Options...
 Node Group
 Copy as Code to Clipboard
 Help F1



Study 1
 Step 1: Stationary
 Solver Configurations
 Results
 Datasets
 Views
 Derived Values
 Tables
 Velocity (spf)
 Pressure (spf)
 3D Plot Group 3
 Export
 Reports
 Plot F8
 Plot In
 Volume
 Arrow Volume
 Surface
 Slice
 Isosurface
 Arrow Surface
 Image
 Line
 Contour
 Streamline
 Arrow Line
 Particle Trajectories
 Mesh
 Annotation



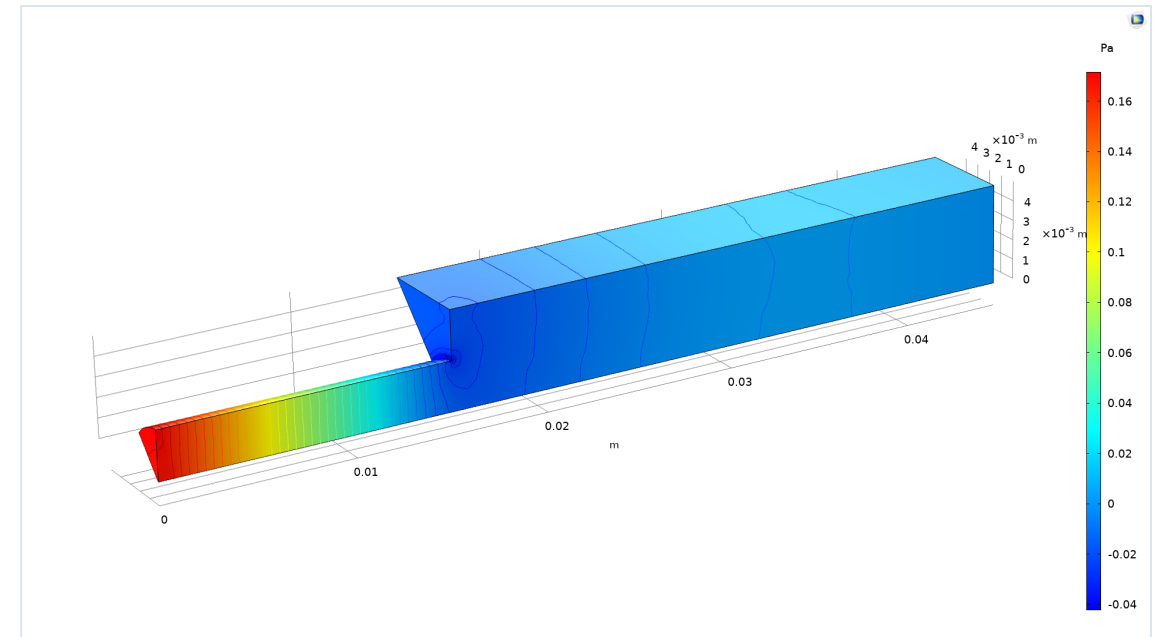
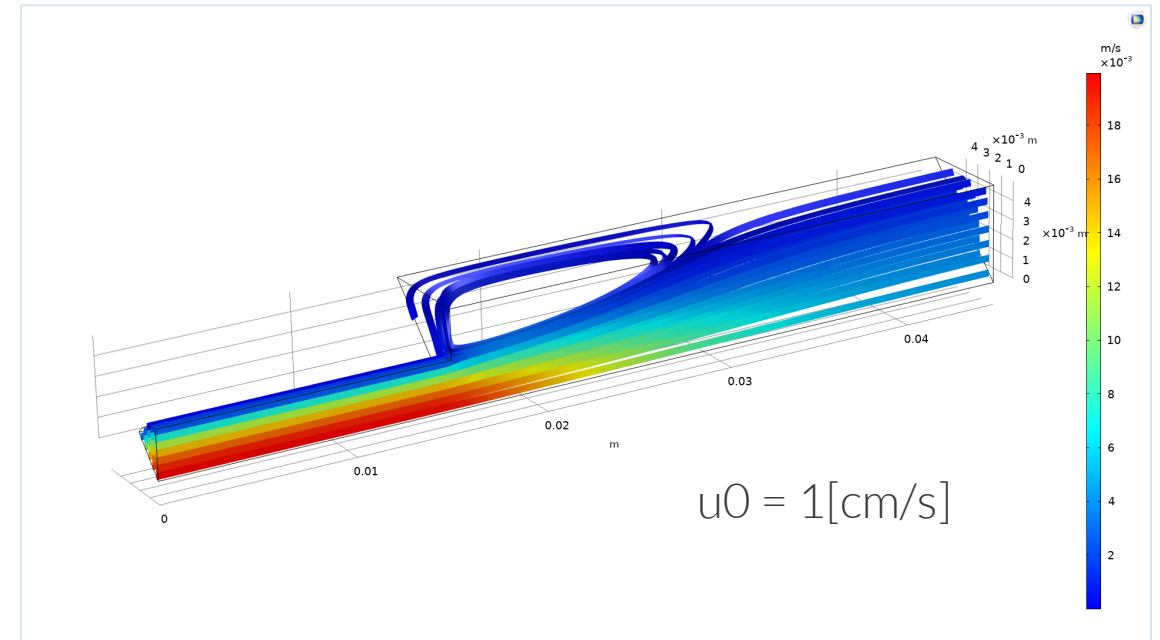
Settings
 Streamline
 Plot
 Label: Streamline 1
 Data
 Dataset: From parent
 Expression
 x component: u m/s
 y component: v m/s
 z component: w m/s
 Description: Velocity field
 Title
 Streamline Positioning
 Positioning: Magnitude controlled
 Min distance: 0.04
 Max distance: 0.08
 Advanced parameters: Automatic
 Coloring and Style
 Line style
 Type: Ribbon
 Width expression: 1 m
 Width scale factor: 1.8E-4
 Point style
 Type: Arrow
 Arrow distribution: Equal arc length
 Number of arrows: 130
 Arrow type: Arrowhead
 Arrow length: Normalized
 Scale factor: 0.0785



Velocity, Streamlines
 Streamline 1
 Recirculation Zone
 Export
 Reports
 Plot
 Plot In
 Color Expression
 Deformation
 Export Expressions
 Filter

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:

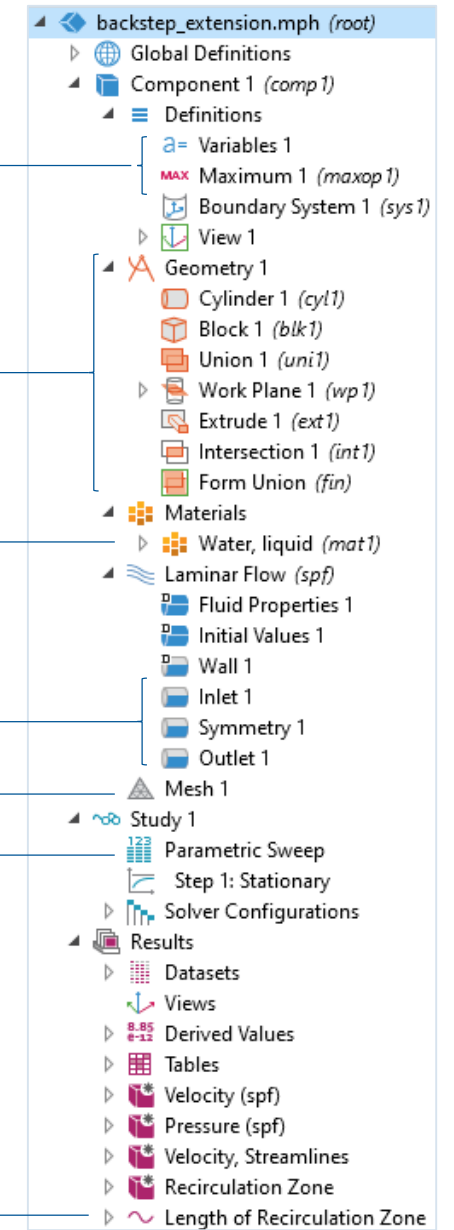
- Unchanged

Study:

- Add a parametric sweep

Results:

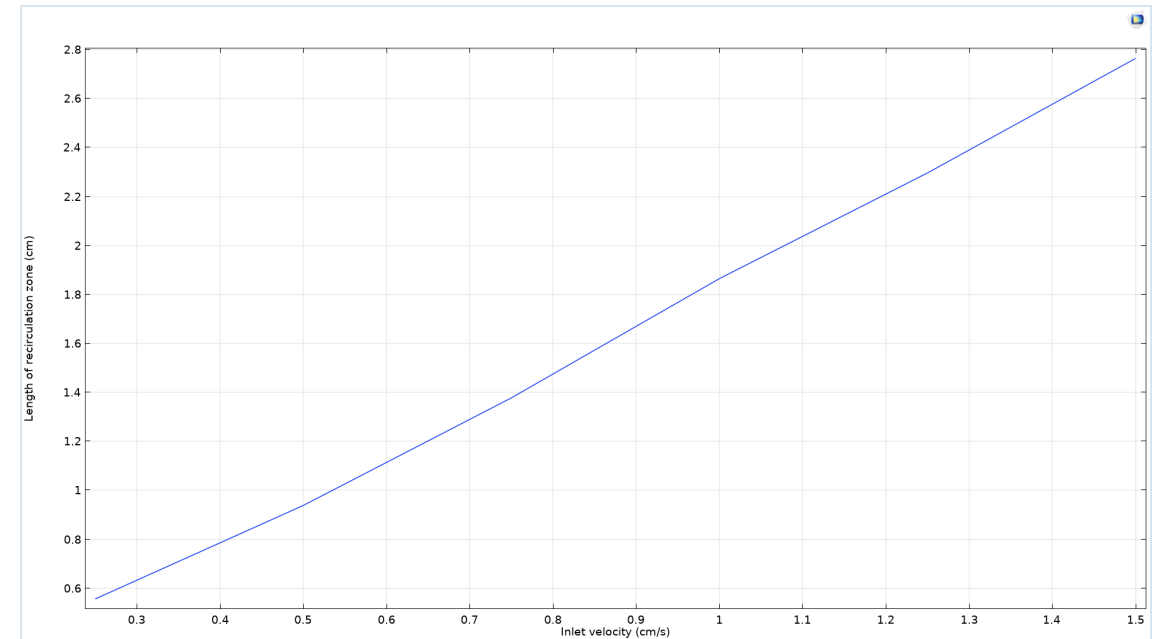
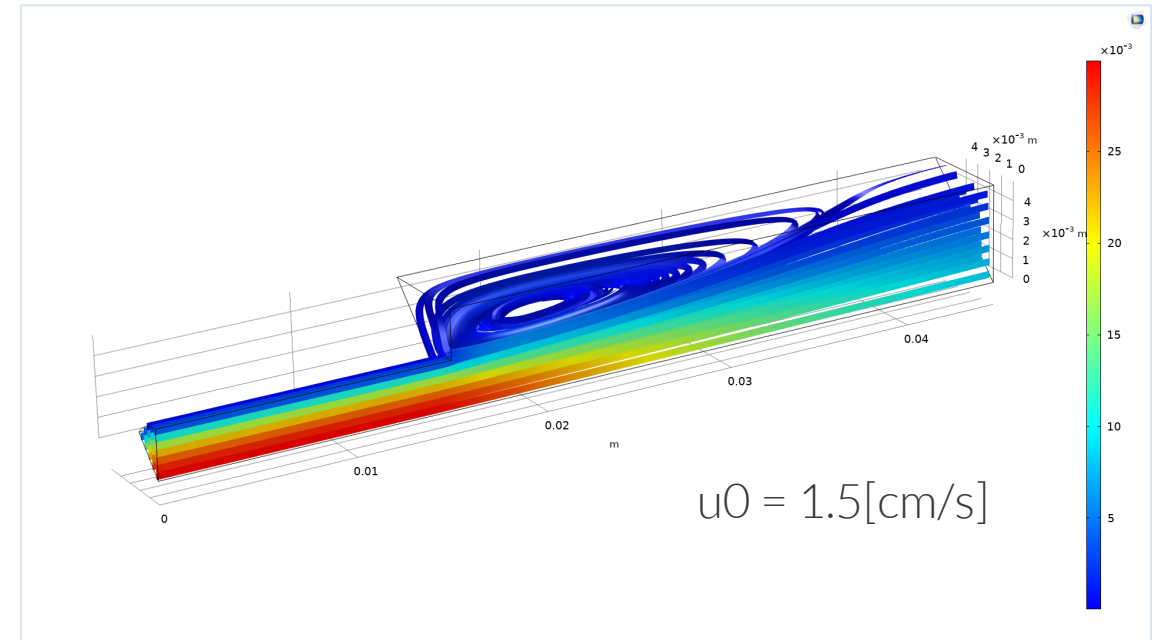
- Plot length of recirculation zone



Overview of procedure for extended model example

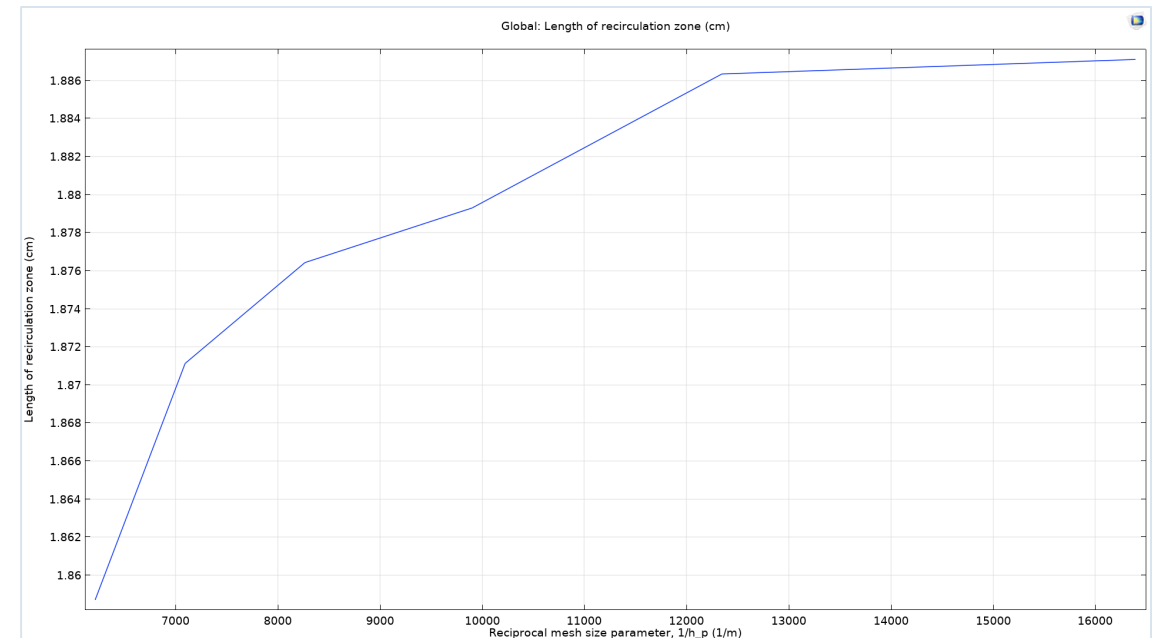
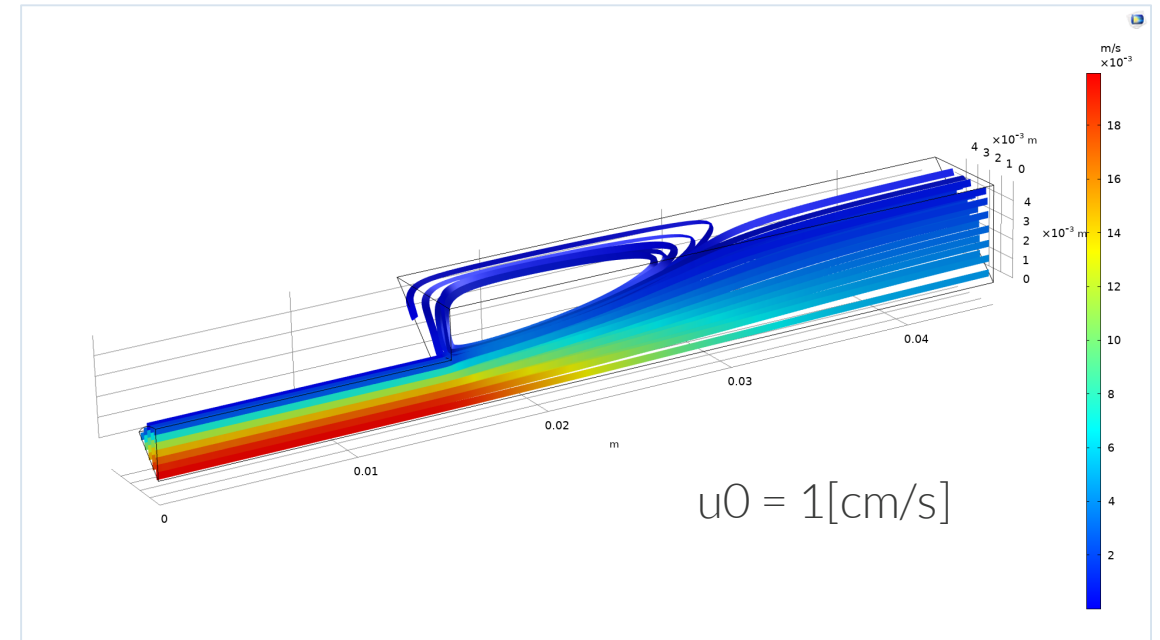
Results

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



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

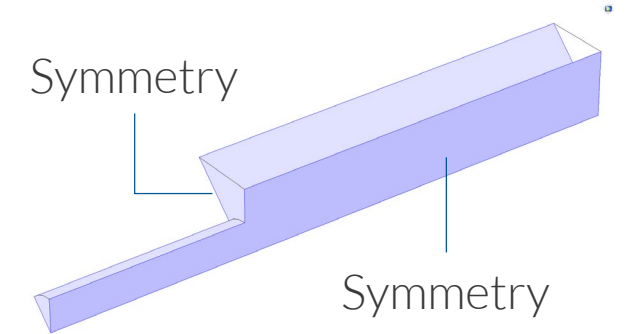
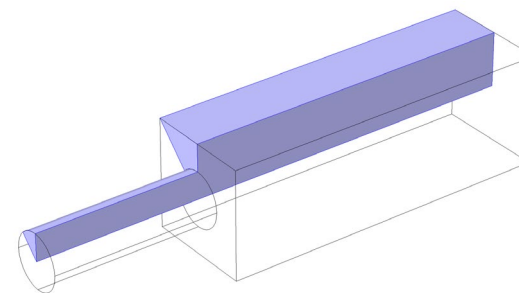
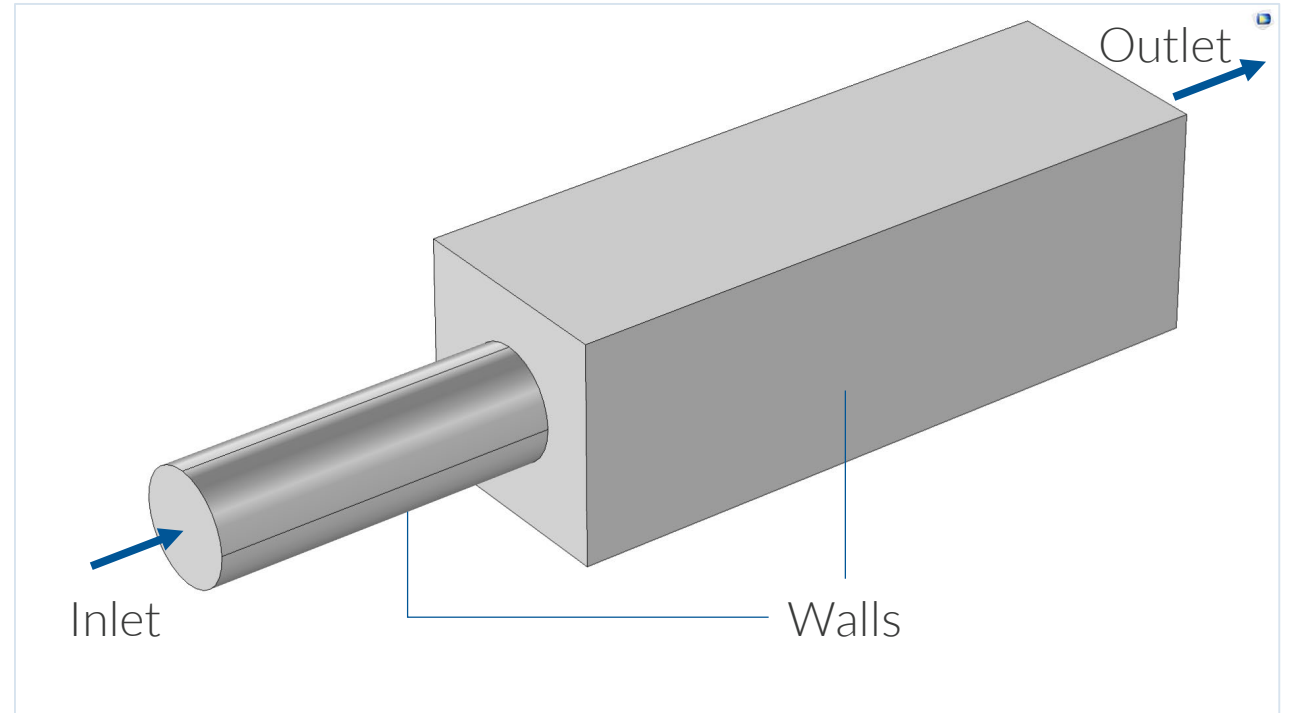


Exercise

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 eighth 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:

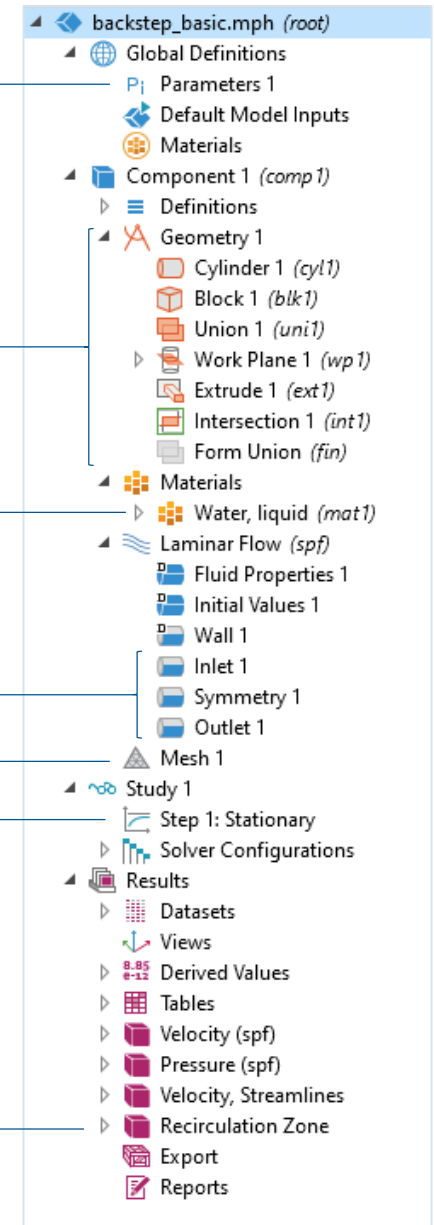
- Select mesh

Study:

- Select compute

Results:

- Plot streamlines



Overview of procedure for first model example