Introduction to CFD Module

VERSION 4.3

COMSOL

Introduction to the CFD Module

Version: May 2012 COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM021302

Contents

Introduction

The CFD Module is used by engineers and scientists to understand, predict, and design the flow in closed and open systems. At a given cost, these types of simulations typically yield new and better products and operation of devices and processes compared to pure empirical studies involving fluid flow. As a part of an investigation, simulations give accurate estimates of flow patterns, pressure losses, forces on surfaces subjected to a flow, temperature distribution, and variations in fluid composition in a system.

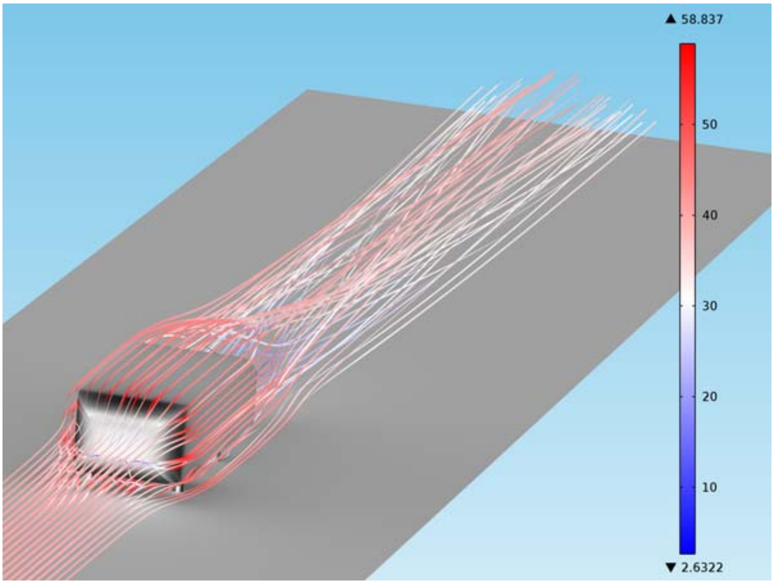


Figure 1: Flow ribbons and velocity field magnitude from the simulation of an Ahmed body. The simulation yields the flow and pressure fields and calculates the drag coefficient as a benchmark for verification and validation of turbulence models.

The CFD Module's general capabilities include stationary and time-dependent flows in two-dimensional and three-dimensional spaces. Formulations of different types of flow are predefined in a number of fluid flow interfaces, which allow you to set up and solve fluid flow problems. The fluid flow interfaces use physical quantities, such as pressure and flow rate, and physical properties, such as viscosity and density, to define a fluid flow problem. There are different fluid flow interfaces that cover a wide range of flows, for example, laminar flow, turbulent flow, single-phase flow, and multiphase flow.

The fluid flow interfaces formulate conservation laws for the momentum, mass, and energy. These laws are expressed in partial differential equations, which are solved by the module together with the corresponding initial conditions and boundary conditions. The equations are solved using stabilized finite element formulations for fluid flow, in combination with damped Newton methods and, for time-dependent problems, different time-dependent solver algorithms. The results are presented in the graphics window through predefined plots relevant for CFD, expressions of the physical quantities that you can define freely, and derived tabulated quantities (for example, average pressure at a surface and drag coefficients) obtained from a simulation.

The work flow in the CFD Module is quite straightforward and can be described by the following steps: define the geometry, select the fluid to be modeled, select the type of flow, define boundary and initial conditions, define the finite element mesh, select a solver, and visualize the results. All these steps are accessed from the COMSOL Desktop. The mesh and solver steps are usually carried out automatically using default settings, which are tuned for each specific fluid flow interface.

The CFD Module's Model Library describes the fluid flow interfaces and their different features through tutorial and benchmark examples for the different types of flows. Here you find models of industrial equipment and devices, tutorial models for education, and benchmark models for verification and validation of the fluid flow interfaces. Go to "Opening the Model Library" to access these resources.

This introduction is intended to give you an accelerated start-up in CFD model building. It contains examples of the typical use of the module, a list of all the fluid flow interfaces including a short description for each interface, and two tutorial examples that introduce the workflow. The "Tutorial Example—Backstep") solves a laminar flow problem.

Aspects of CFD Simulations

This module covers flows ranging from just above the microscale, for example, in medical applications, up to the scales of the flows in ventilation systems and buildings.

One common use of CFD simulations is to help understand the flow in a system. The flow pattern may determine the air quality in a ventilated room or it may determine the forces on a body subjected to the flow. Qualitative interpretation of the flow and pressure fields is usually the first step towards creating or improving a design.

Figure 2 shows the flow field around a solar panel. The presence of a wake in front of the panel, caused by another panel in the solar power plant, may cause lift forces that would not be present if the panel was analyzed alone in a simulation. Three-dimensional graphics such as surface plots, animations, streamlines, ribbons, arrows, and particle tracing plots are examples of the tools that you can use for qualitative studies.

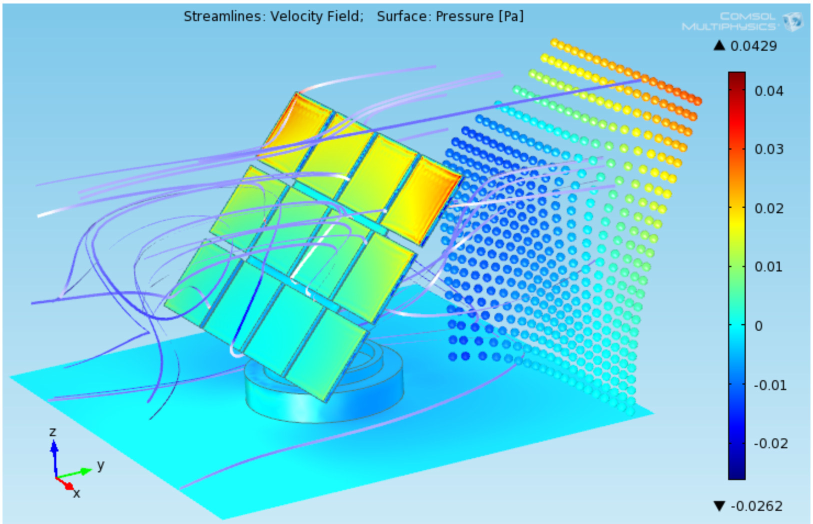


Figure 2: Turbulent fluid flow around a solar panel solved using the CFD Module.

Another common use of the CFD Module, in almost all scales and in many applications, is the prediction of quantitative estimates of properties of the system, such as the average flow at a given pressure difference and the drag and lift coefficients.

In Figure 3 and Figure 4, the pressure losses are estimated for a nozzle used in medical applications. The shear stresses and fluid forces in the nozzle system may also reveal the risks for causing blood damages in medical equipment, which have to be accounted for when controlling the flow.

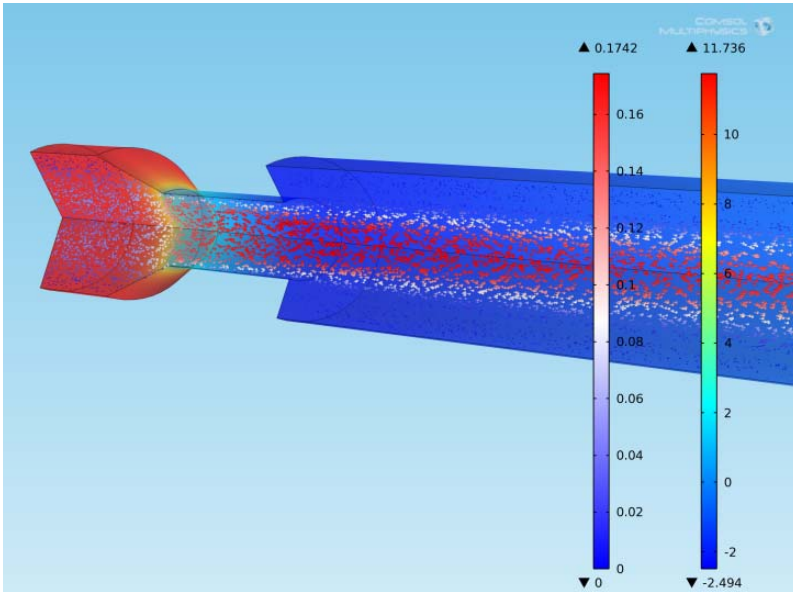


Figure 3: Pressure field and flow field in a model of a nozzle relevant for applications in medical design.

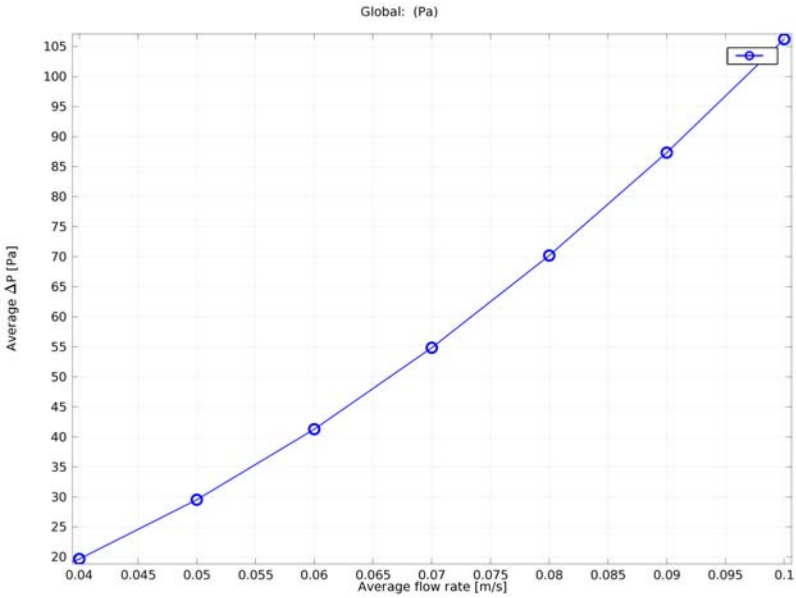


Figure 4: Pressure difference between inlet and outlet at different average flow rates through the nozzle.

For quantitative studies, the CFD Module has a vast range of tools for evaluating results. For example, to evaluate surface and volume averages, maximum and minimum values, and derived values (functions and expressions of the solution) as well as to generate tables and x-y plots. Derived values such as drag and lift coefficients and other values relevant for CFD are predefined in the module.

Typically, qualitative studies form the basis for understanding, which in turn can spark innovation. This innovation can then mean significant improvements to products and processes, often in quantum leaps. Quantitative studies, on the other hand, form the basis for optimization and control, which can also greatly improve products and processes but usually in a series of many small steps.

The CFD Module Interfaces

The fluid flow interfaces in this module are based on the laws for conservation of momentum, mass, and energy in fluids. Different combinations and expressions of the conservation laws result in the different flow models. These laws of physics are translated by the fluid flow interfaces to sets of partial differential equations with corresponding initial and boundary conditions.

A fluid flow interface defines a number of features. Each feature represents an operation that describes a term or condition in the conservation equations. Such a term or condition can be defined in a geometric entity of the model, such as a domain, boundary, edge (for 3D models), or point.

Figure 5 shows the Model Builder including a Laminar Flow interface and the settings window for the selected Fluid Properties 1 feature node. The Fluid Properties 1 node adds the terms to the model equations to a selected geometrical domain. Furthermore, the Fluid Properties 1 feature may link to the Materials feature node to obtain physical properties such as density and dynamic viscosity, in this case the fluid properties of water. The fluid properties, defined by the *Water, liquid* material, can be functions of the modeled physical quantities, such as pressure and temperature. In the same way, the Wall 1 node adds the boundary conditions to the walls of the fluid domain.

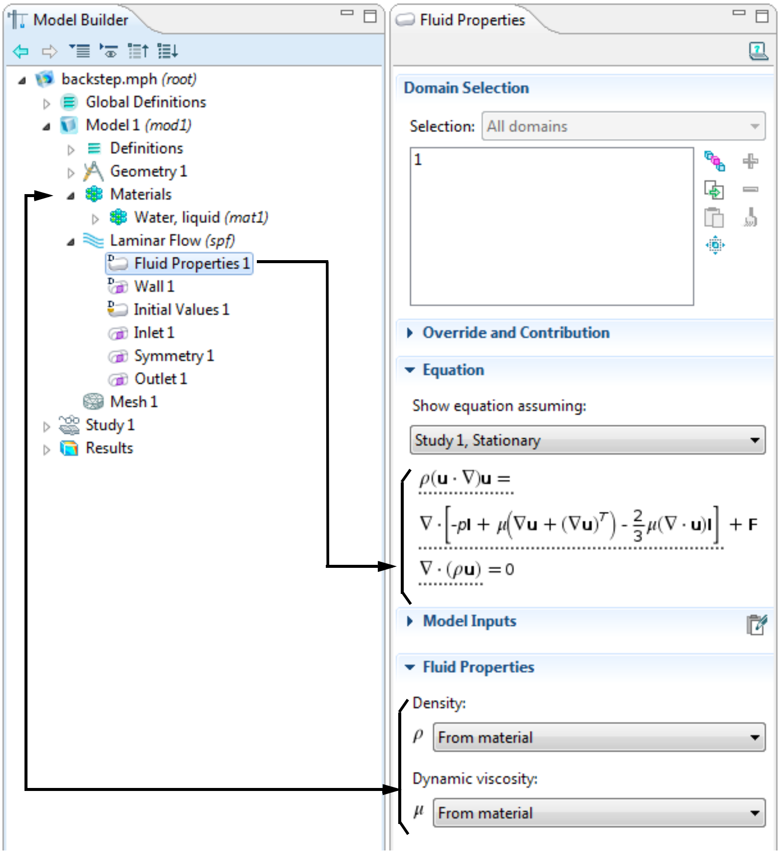


Figure 5: The Model Builder including a Laminar Flow interface (to the left), and the Fluid Properties settings window for the selected feature node (to the right).The Equation section in the settings window shows the model equations and the terms added by the Fluid Properties 1 node to the model equations. The added terms are underlined with a dotted line. The text also explains the link between the Water, liquid material node and the values for the fluid properties.

The CFD Module includes a large number of Fluid Flow interfaces for different types of flow. It also includes Chemical Species Transport interfaces for reacting flows in multicomponent solutions and interfaces for heat transfer in solids, fluids, and porous media found under the Heat Transfer branch.

Figure 6 shows the fluid flow interfaces and the other physics interfaces that are included as displayed in the Model Wizard for a 3D model. Also see "Physics List by

Space Dimension and Study Type" on page 15. A short description of the interfaces follows.

- Chemical Species Transport
  - Transport of Diluted Species (chds)
  - Transport of Concentrated Species (chcs)
  - Reacting Flow, Concentrated Species (rfcs)
  - Reacting Flow, Diluted Species (rfds)
- Fluid Flow
  - Single-Phase Flow
    - Laminar Flow (spf)
    - Turbulent Flow
    - Creeping Flow (spf)
    - Rotating Machinery, Fluid Flow
  - Thin-Film Flow
    - Lubrication Shell (tffs)
  - Multiphase Flow
    - Bubbly Flow
    - Mixture Model
    - Euler-Euler Model
    - Two-Phase Flow, Level Set
    - Two-Phase Flow, Phase Field
  - Porous Media and Subsurface Flow
    - Brinkman Equations (br)
    - Darcy's Law (dl)
    - Free and Porous Media Flow (fp)
    - Two-Phase Darcy's Law (tpdl)
  - Non-Isothermal Flow
    - Laminar Flow (nitf)
    - Turbulent Flow
  - High Mach Number Flow
    - Laminar Flow (hmnf)
    - Turbulent Flow
- Heat Transfer
  - Heat Transfer in Solids (ht)
  - Heat Transfer in Fluids (ht)
  - Heat Transfer in Porous Media (ht)
  - Conjugate Heat Transfer
  - Electromagnetic Heating

Figure 6: The 3D model physics list for the CFD Module as shown in the Model Wizard.

Single-Phase Flow

The Laminar Flow interface ( ) is used primarily to model flows of a comparably low Reynolds number. The interface solves the Navier-Stokes equations, for incompressible and weakly compressible flows (up to Mach 0.3). This fluid flow interface also allows for simulation of non-Newtonian fluid flow.

The interfaces under the Turbulent Flow branch ( ) model flow of high Reynolds numbers. The interfaces use the Reynolds-averaged Navier-Stokes (RANS) equations and solve for the averaged velocity field and averaged pressure. These fluid flow interfaces also formulate different models for the turbulent viscosity. There are several turbulence models available—a standard k-ε model, a Low Reynolds number k-ε model, a k-ω model, and the Spalart-Allmaras model. Similar to the Laminar Flow interface, compressibility (Mach < 0.3) is selected by default.

The standard k-ε model is the most widely used turbulence model since it is often a good compromise between accuracy and computational cost (memory and CPU-time). The Low Reynolds number k-ε model is more accurate than the standard model, especially close to walls, but this accuracy comes at a higher computational cost. The k-ω model is an alternative to the standard k-ε model and it can often give more accurate results. However, the k-ω model is also less robust than the standard k-ε model. The Spalart-Allmaras model is specifically designed for aerodynamic applications, such as wing profiles, but is also widely used for other applications due to its high robustness and decent accuracy.

The Creeping Flow interface ( ) approximates the Navier-Stokes equations for very low Reynolds numbers. This is often referred to as Stokes flow and is appropriate for use when viscous flow is dominant, such as in very small channels or microfluidics applications.

The Rotating Machinery interfaces ( ) are used for the modeling of flow where one or more of the boundaries rotate, for example in mixers and propellers. The physics interfaces support compressibility (Mach < 0.3), laminar non-Newtonian flow, and turbulent flow using the standard k-ε model.

Thin-Film Flow

The Lubrication Shell interface ( ) is available in 2D, 2D axisymmetric, and 3D geometries. This physics interface is a boundary interface, which means that the boundary level is the highest level; it does not have a domain level. Using boundary equations, this interface models the thickness, velocity, and pressure in narrow channels. The simulation of the flow of a lubrication oil between two rotating cylinders is an example of a possible use of this interface.

Multiphase Flow

The interfaces under the Bubbly Flow branch ( ) model two-phase flow where the fluids form a gas-liquid homogeneous mixture, and the content of the gas is less than 10%. There is support for both laminar and turbulent flows using an extended version of the k-ε turbulence model that can account for bubble induced turbulence. For laminar flows, the interface supports non-Newtonian liquids. The Bubbly Flow interfaces also allow for mass transfer between the two phases.

The interfaces under the Mixture Model branch () are similar to the Bubbly Flow interfaces but assumes that the dispersed phase consists of solid particles or liquid droplets. The continuous phase has to be a liquid. There is support for both laminar and turbulent flows using the k-ε turbulence model. For laminar flows, the interface supports non-Newtonian fluids. The Mixture Model interfaces also allow for mass transfer between the two phases.

The Euler-Euler Model interface () for two-phase flow is able to handle the same cases as the Bubbly Flow and Mixture Model interfaces but is not limited to low concentrations of the dispersed phase. In addition, the Euler-Euler Model interface can handle large differences in density between the phases, such as the case of solid particles in air. This makes the model suitable for simulations of fluidized beds.

The Two-Phase Flow, Level Set interface () and the Two-Phase Flow, Phase Field interface (), are both used primarily to model two fluids separated by a fluid interface. The moving interface is tracked in detail using the level set method and the phase field method, respectively. Similar to other fluid flow interfaces, these interfaces support both compressible (Mach < 0.3) and incompressible flow. They support laminar flow where one or both fluids can be non-Newtonian. The interfaces support turbulent flow using the standard k-ε turbulence model as well as Stokes flow.

Porous Media and Subsurface Flow

The Brinkman Equations interface () models flow through a porous medium where the influence of shear stresses are significant. The physics interface supports both the Stokes-Brinkman formulation, suitable for very low flow velocities, and Forchheimer drag, which is used to account for effects at higher velocities. The fluid can be either incompressible or compressible, given that the Mach number is less than 0.3.

The Darcy's Law interface () models relatively slow flows through porous media in the cases where the effects of shear stresses perpendicular to the flow are small.

The Free and Porous Media Flow interface () models porous media that contain open channels connected to the porous media, such as in fixed-bed reactors and catalytic converters.

The Two-Phase Darcy's Law interface () sets up two Darcy's law, one for each fluid phase in porous media, and couples the two, for example using capillary expressions. It is tailored to model effects such as moisture transport in porous media.

Nonisothermal Flow

The Non-Isothermal Flow, Laminar Flow interface ( ) is used primarily to model slow-moving flow in environments where the temperature and flow fields have to be coupled. A typical example is natural convection. The physics interface has predefined functionality to couple heat transfer in fluids and solids.

The Non-Isothermal Flow, Turbulent Flow interfaces ( ) use the Reynolds-Averaged Navier-Stokes (RANS) equations coupled to heat transfer in fluids and in solids. There is support for additional RANS turbulence models – the standard k-ε model, a Low Reynolds number k-ε model, a k-ω model, and the Spalart-Allmaras model.

The Conjugate Heat Transfer interface ( ) is also included in the CFD Module and is almost identical to the Non-Isothermal Flow interface, since they only differ in the default settings.

High Mach Number Flow

The High Mach Number Flow, Laminar Flow interface ( ) solves momentum and energy equations for fully compressible laminar flow. The physics interface is typically used to model low-pressure systems, where the flow velocity can be very large but where the flow stays laminar.

The High Mach Number Flow, Turbulent Flow interfaces ( ) solve momentum and energy equations for fully compressible turbulent flow coupled to a RANS turbulence model. There are two versions: one that couples to the k-ε turbulence model and one that couples to the Spalart-Allmaras turbulence model.

Physics List by Space Dimension and Study Type

The table lists the physics interfaces available with this module in addition to those included with the COMSOL basic license.

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Chemical Species Transport				
Transport of Concentrated Species		chcs	all dimensions	stationary; time dependent
Reacting Flow, Concentrated Species		rfcs	3D, 2D, 2D axisymmetric	stationary; time dependent
Reacting Flow, Diluted Species		rfds	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Fluid Flow				
Single-Phase Flow				
Single-Phase Flow, Laminar Flow*		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ε		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ω		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Spalart-Allmaras		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Creeping Flow		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Rotating Machinery, Laminar Flow		rmspf	3D, 2D	time dependent
Rotating Machinery, Turbulent Flow, k-ε		rmspf	3D, 2D	time dependent
Thin-Film Flow				
Lubrication Shell		tffs	3D, 2D, 2D axisymmetric	stationary; eigenfrequency; frequency domain; frequency domain modal; time dependent; time dependent modal; frequency-domain, perturbation
Thin-Film Flow		tff	2D	stationary; eigenfrequency; frequency domain; frequency domain modal; time dependent; time dependent modal; frequency-domain, perturbation
Multiphase Flow				
Bubbly Flow				
Laminar Bubbly Flow		bf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Bubbly Flow		bf	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Mixture Model				
Mixture Model, Laminar Flow		mm	3D, 2D, 2D axisymmetric	stationary; time dependent
Mixture Model, Turbulent Flow		mm	3D, 2D, 2D axisymmetric	stationary; time dependent
Euler-Euler Model				
Euler-Euler Model, Laminar Flow		ee	3D, 2D, 2D axisymmetric	stationary; time dependent
Two-Phase Flow, Level Set				
Laminar Two-Phase Flow, Level Set		tpf	3D, 2D, 2D axisymmetric	transient with initialization
Turbulent Two-Phase Flow, Level Set		tpf	3D, 2D, 2D axisymmetric	transient with initialization
Two-Phase Flow, Phase Field				
Laminar Two-Phase Flow, Phase Field		tpf	3D, 2D, 2D axisymmetric	transient with initialization
Turbulent Two-Phase Flow, Phase Field		tpf	3D, 2D, 2D axisymmetric	transient with initialization
Porous Media and Subsurface Flow				
Brinkman Equations		br	3D, 2D, 2D axisymmetric	stationary; time dependent
Darcy's Law		dl	all dimensions	stationary; time dependent
Free and Porous Media Flow		fp	3D, 2D, 2D axisymmetric	stationary; time dependent
Two-Phase Darcy's Law		tpdl	3D, 2D, 2D axisymmetric	stationary; time dependent
Non-Isothermal Flow				
Laminar Flow		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, k-ε		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ω		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Turbulent Flow, Low Re k-ε		nitf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Spalart-Allmaras		nitf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Rotating Machinery, Non-Isothermal Flow				
Laminar Flow		rmnitf	3D, 2D	time dependent
Turbulent Flow, k-ε		rmnitf	3D, 2D	time dependent
High Mach Number Flow				
Laminar Flow		hmnf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ε		hmnf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Spalart-Allmaras		hmnf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Heat Transfer				
Heat Transfer in Fluids*		ht	all dimensions	stationary; time dependent
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent
Conjugate Heat Transfer				
Laminar Flow		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, k-ε		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ω		nitf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε		nitf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Spalart-Allmaras		nitf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Mathematics				
Moving Interface				
Level Set		ls	all dimensions	transient with initialization
Phase Field		pf	all dimensions	time dependent

* This is an enhanced interface, which is included with the base COMSOL package but has added functionality for this module.

Opening the Model Library

To open a CFD Module Model Library model, select **View > Model Library** from the main menu in COMSOL Multiphysics. In the Model Library window that opens, expand the CFD Module folder and browse or search the contents. Click **Open Model and PDF** to open the model in COMSOL Multiphysics and a PDF to read background theory about the model including the step-by-step instructions to build it.

The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the icon. If the MPH-file's size exceeds 25MB, a tip with the text "Large file" and the file size appears when you position the cursor at the model's node in the Model Library tree.
- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the icon. If you position the cursor at a compact model in the Model Library window, a No solutions stored message appears. If a full MPH-file is available for download, the corresponding node's context menu includes a Model Library Update item.

The rest of this guide uses a model from the Model Library. The "Tutorial Example—Backstep", which starts on the next page, solves a laminar flow problem.

Tutorial Example—Backstep

This tutorial model solves the incompressible Navier-Stokes equations in a backstep geometry. A characteristic feature of fluid flow in geometries of this kind is the recirculation region that forms where the flow exits the narrow inlet region. The model demonstrates the modeling procedure for laminar flows in the CFD Module.

Model Geometry

The model consists of a pipe connected to a block-shaped duct (see Figure 7). Due to symmetry, it is sufficient to model one eighth of the full geometry.

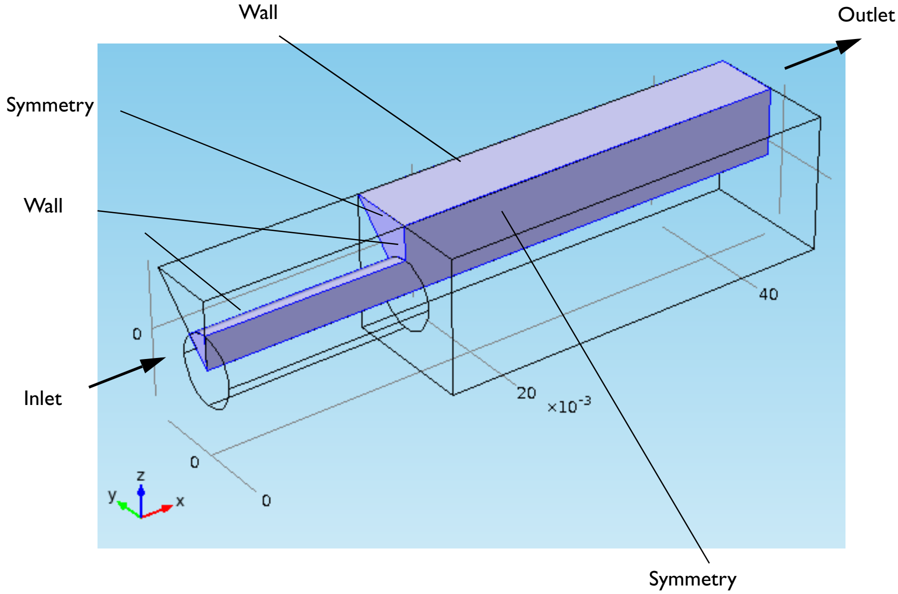


Figure 7: The model geometry showing the symmetry.

Domain Equation and Boundary Conditions

The flow in the system is laminar and therefore the Laminar Flow interface is used.

The inlet flow is fully developed laminar flow, described by the corresponding inlet boundary condition. This boundary condition computes the flow profile for fully developed laminar flow in channels of an arbitrary cross section. The boundary condition at the outlet sets a constant relative pressure. Furthermore, the vertical and inclined boundaries along the length of the geometry are symmetry boundaries. All other boundaries are solid walls described by a non-slip boundary condition.

Results

Figure 8 shows a combined surface and arrow plot of the flow velocity. This plot does not reveal the recirculation region in the duct immediately beyond the inlet pipe's end. For this purpose, a streamline plot is more useful, as shown in Figure 9.

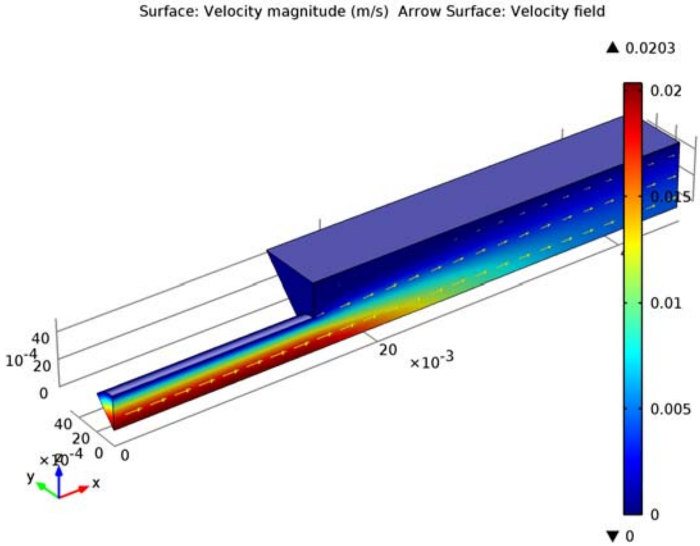


Figure 8: The velocity field in the backstep geometry.

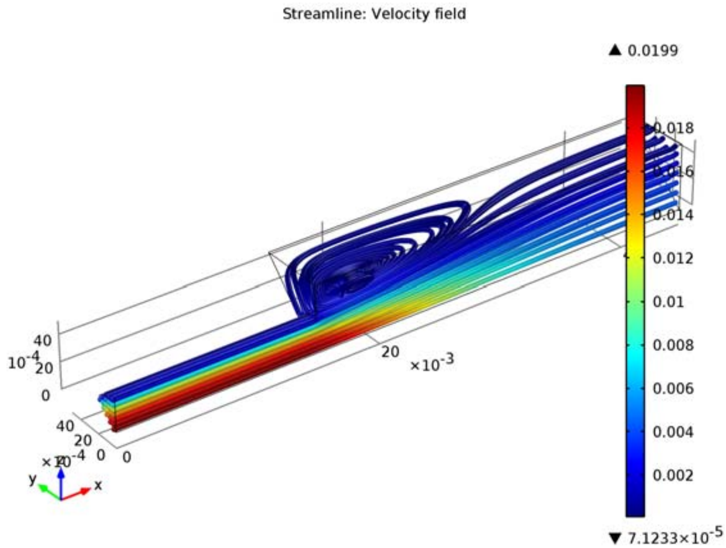


Figure 9: The recirculation region visualized using a velocity streamline plot.

The following instructions show how to formulate, solve, and reproduce these plots.

MODEL WIZARD

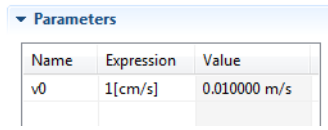
The first step is to select the space dimension and the Laminar Flow interface for stationary studies.

1. Open COMSOL Multiphysics. In the **Model Wizard** the **Space Dimension** defaults to **3D**. Click **Next**.
2. In the **Add physics** tree under **Fluid Flow>Single-Phase Flow**, double-click **Laminar Flow (spf)** to add it to the **Selected physics** list. Click **Next**.
3. Under **Studies>Preset Studies** select **Stationary**.
4. Click **Finish**.

GLOBAL DEFINITIONS - PARAMETERS

The first task is to define a parameter for the inlet velocity then use this parameter to run parametric studies.

1 In the **Model Builder**, right-click **Global Definitions** and choose **Parameters** .
2 Go to the **Parameters** settings window. In the table, enter the following settings:
   - In the **Name** field, enter `v0`
   - In the **Expression** field, enter `1[cm/s]`
   - In the **Description** field, enter `Inlet velocity`

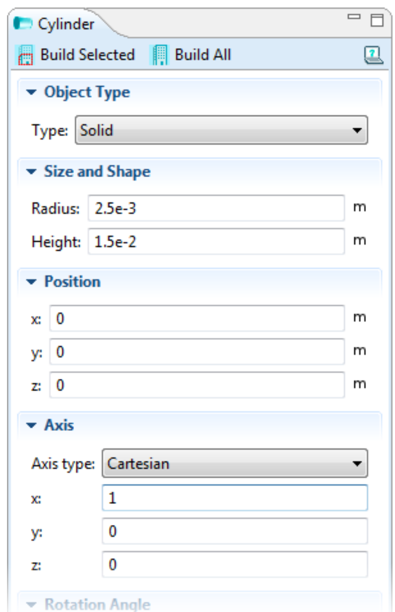


GEOMETRY 1

This section defines the geometry for the model.

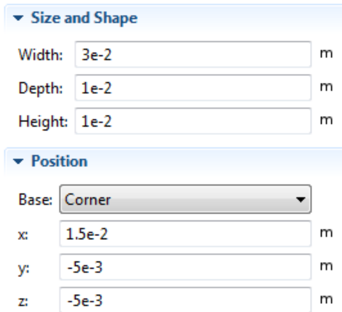
Cylinder 1

1 In the **Model Builder**, right-click **Geometry 1** and choose **Cylinder** .
2 Go to the **Cylinder** settings window. Under **Size and Shape** in the:
   - **Radius** field, enter `2.5e-3`
   - **Height** field, enter `1.5e-2`
3 Under **Axis** in the:
   - **x** field, enter `1` (replace the default)
   - **z** field, enter `0` (replace the default)
4 Click the **Build Selected** button on the **Cylinder** window toolbar.



Block 1

1. In the **Model Builder**, right-click **Geometry 1** and choose **Block**.
2. Go to the **Block** settings window. Under **Size and Shape** in the:
   - **Width** field, enter `3e-2`
   - **Depth** field, enter `1e-2`
   - **Height** field, enter `1e-2`
3. Under **Position** in the:
   - **x** field, enter `1.5e-2`
   - **y** field, enter `-5e-3`
   - **z** field, enter `-5e-3`
4. Click the **Build Selected** button on the **Block** window toolbar.
5. Click the **Zoom Extents** button on the **Graphics** toolbar.

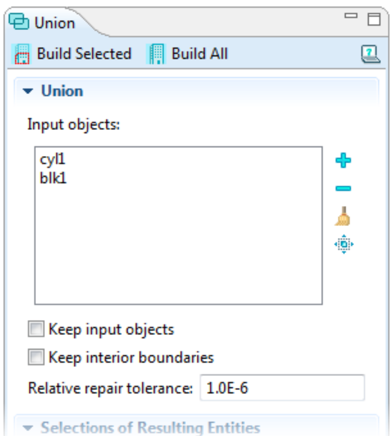


The next step is to form the composite geometry of the inlet and outlet channel.

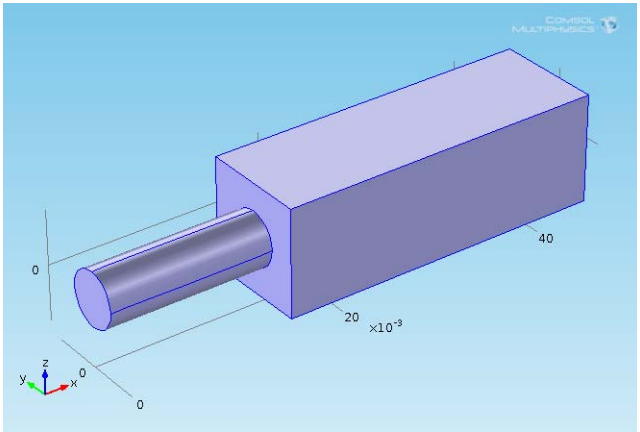
Union 1

1. In the **Model Builder**, right-click **Geometry 1** and choose **Boolean Operations>Union**.
2. Select the objects **cyl1** and **blk1** only.

   There are different ways to select objects. For example, in the **Graphics** window, click **cyl1** to highlight it in red and then right-click to change the color to blue. This adds it to the selection list. Or click the object and then click the **Add to Selection** button to do the same thing.
3. Go to the **Union** settings window. Under **Union** click to clear the **Keep interior boundaries** check box.
4. Click the **Build Selected** button.



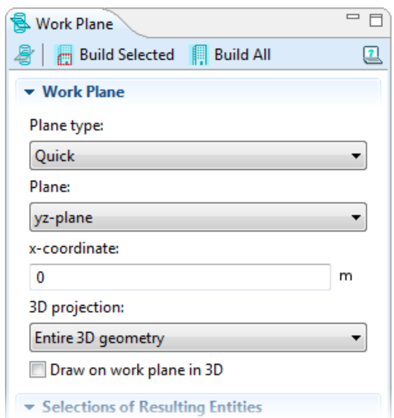
The model geometry should match this figure:



Due to symmetry (see Figure 7 on page 20), one eighth of the geometry is cut out to represent the full solution. This is done by first creating a prism of one eighth of a box around the geometry and then by using the intersection Boolean operation.

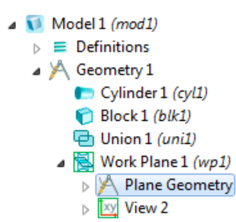
Work Plane 1

1 In the **Model Builder**, right-click **Geometry 1** and choose **Work Plane**.
2 Go to the **Work Plane** settings window. Under **Work Plane** from the **Plane** list, select **yz-plane**.
3 From the **3D projection** list, select **Entire 3D geometry**.
4 Click the **Build Selected** button.



Bézier Polygon 1

1 In the **Model Builder**, under **Work Plane 1**, click the **Plane Geometry** node and then the **Zoom Extents** button on the **Graphics** toolbar.
2 Right-click **Plane Geometry** and choose **Bézier Polygon**.



3 Go to the **Bézier Polygon** settings window. Under **Polygon Segments** click **Add Linear**.

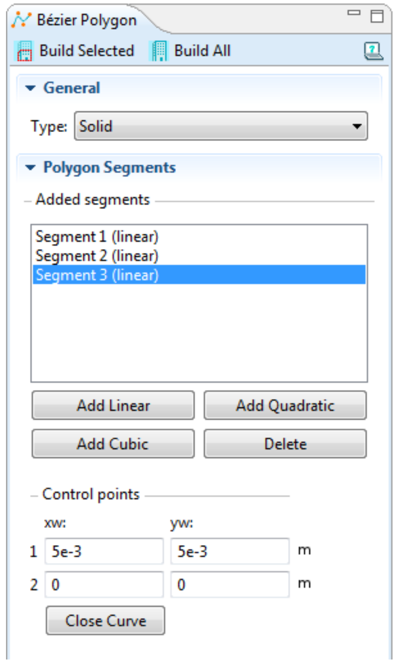
4 Find the **Control points** subsection. In row **2**, set **yw** to `5e-3`.

5 Click **Add Linear**.

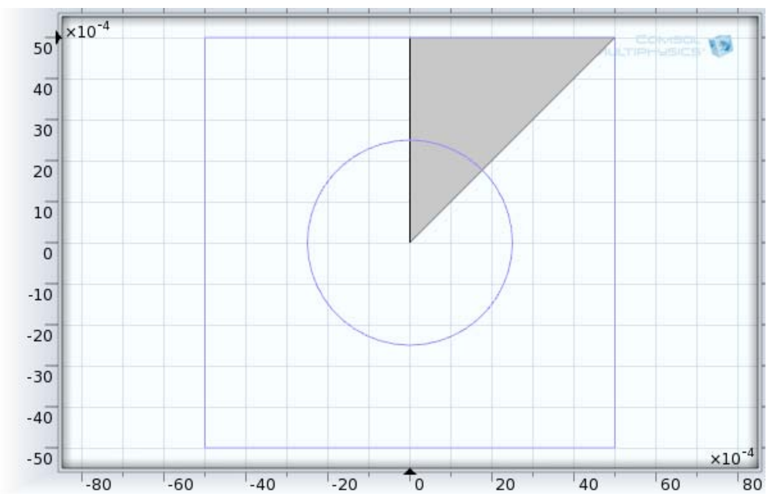
6 In row **2**, set **xw** to `5e-3`.

7 Click **Add Linear** and **Close Curve**.

8 Click the **Build Selected** button on the settings window, then the **Zoom Extents** button on the **Graphics** tool bar.

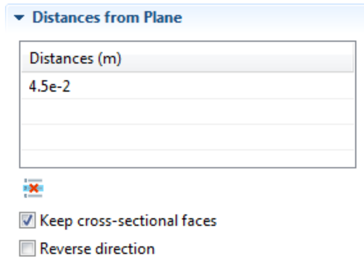


The triangle created, using the polygon tool, overlaps with one eighth of a fictive box around the geometry. This is used to create the prism for the intersection Boolean operation.

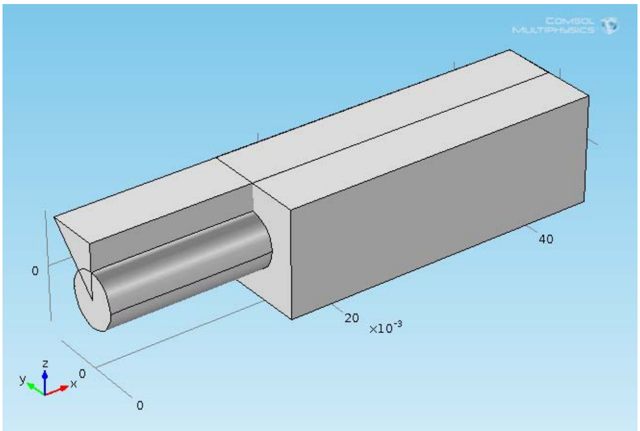


Extrude 1

1. In the **Model Builder**, right-click **Work Plane 1** and choose **Extrude**.
2. Go to the **Extrude** settings window. Under **Distances from Plane** enter `4.5e-2` in the table.
3. Click the **Build Selected** button and then the **Zoom Extents** button on the **Graphics** window toolbar.

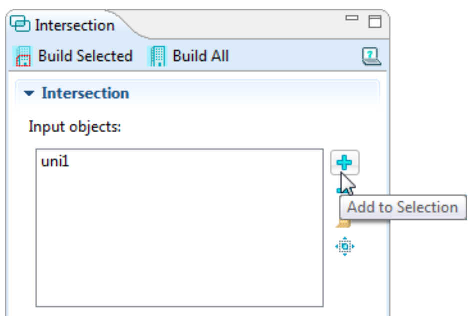


The geometry should match this figure.



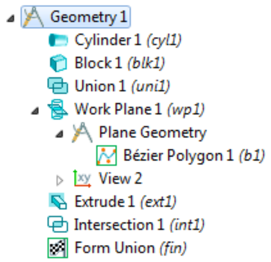
Intersection 1

1. In the **Model Builder**, right-click **Geometry 1** and choose **Boolean Operations>Intersection**.
2. In the **Intersection** settings window, add the objects **uni1** and **ext1** to the **Input objects** list.
3. Click the **Build Selected** button and then the **Zoom Extents** button on the **Graphics** window toolbar.



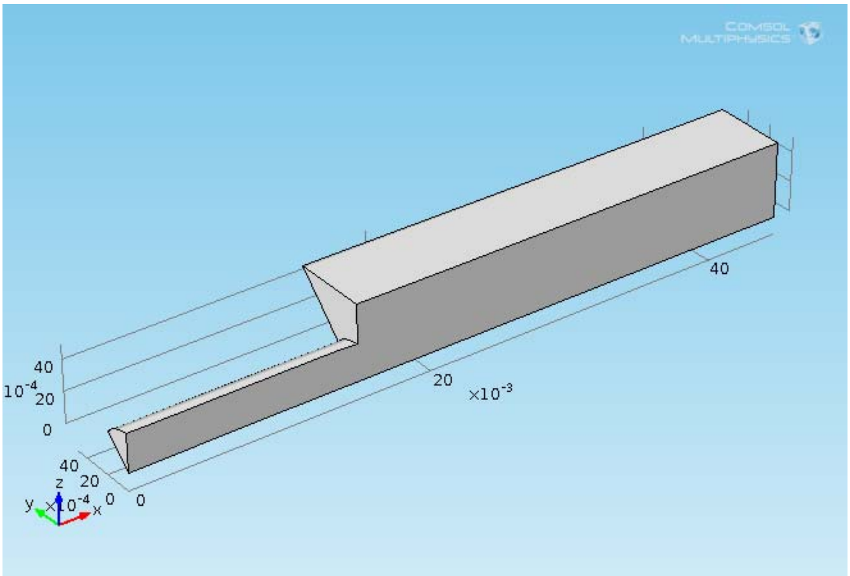
Form Union

1. In the **Model Builder**, right-click **Form Union** and choose **Build Selected**.



The final **Geometry** node sequence in the **Model Builder** should match the figure.

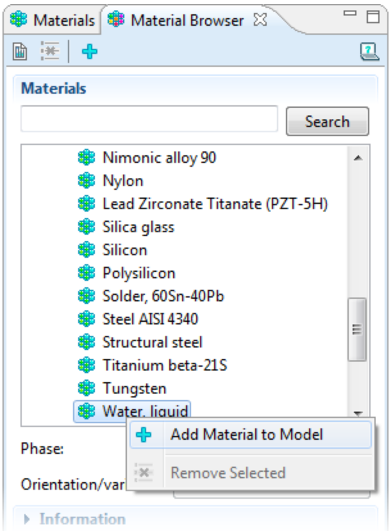
The model geometry is now complete.



MATERIALS

- From the main menu, select **View>Material Browser**.

2 In the **Material Browser** window, under **Built-In** right-click **Water, liquid** and choose **Add Material to Model** from the menu.



The physical properties are now available for the CFD simulation. This also defines the domain settings. The next step is to specify the boundary conditions.

LAMINAR FLOW

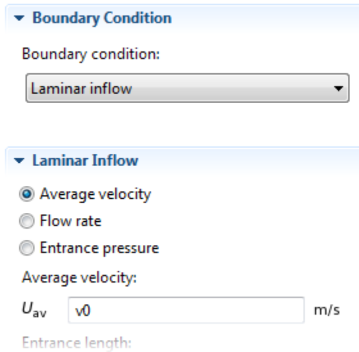
Inlet 1

1 In the **Model Builder**, right-click **Laminar Flow** and choose **Inlet** .

2 Go to the **Inlet** settings window. Select Boundary **1**, which represents the inlet.

3 Under **Boundary Condition** from the **Boundary condition** list, select **Laminar inflow**.

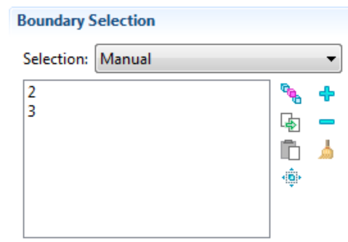
4 Under **Laminar Inflow** in the U_{av} field, enter v0 (defined as a Global Parameter).



Symmetry 1

1 In the **Model Builder**, right-click **Laminar Flow** and choose **Symmetry**.

2 Go to the **Symmetry** settings window. Select Boundaries 2 and 3 only.

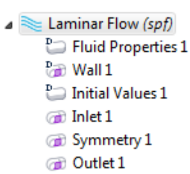


Outlet 1

1 In the **Model Builder**, right-click **Laminar Flow** and choose **Outlet**.

The default outlet condition specifies a zero relative pressure.

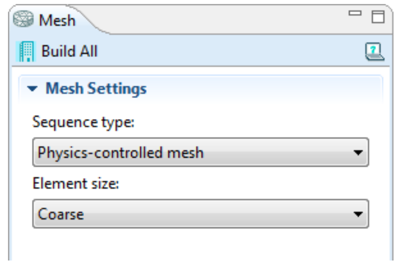
2 Go to the **Outlet** settings window. Select Boundary 7 only.



The sequence of nodes in the **Model Builder** under **Laminar Flow** should match the figure. The 'D' in the upper left corner of a node means it is a default node.

All other boundaries have now the default wall condition.

MESH 1

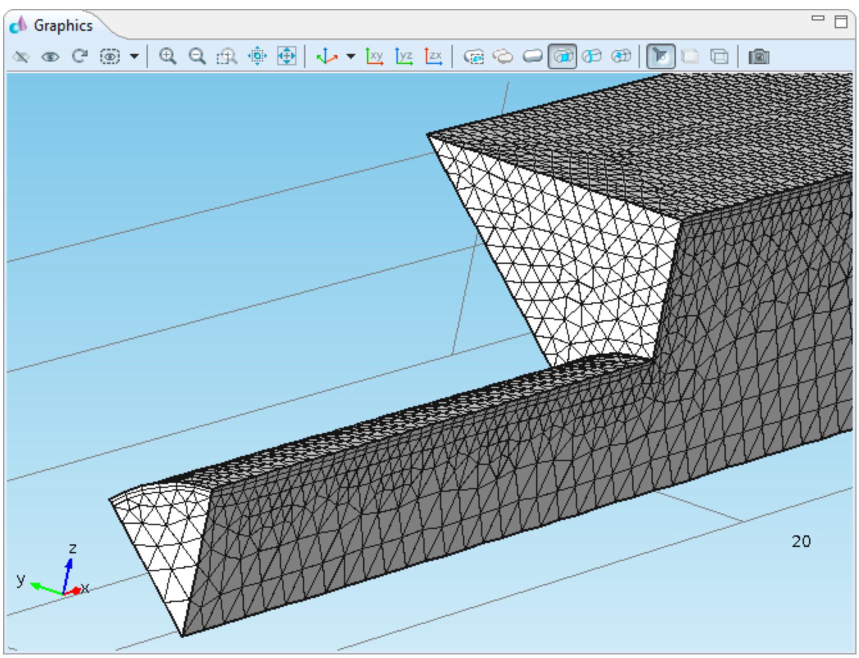


1. In the **Model Builder** under **Model 1**, click **Mesh 1** .
2. Go to the **Mesh** settings window. Under **Mesh Settings** from the **Element size** list, select **Coarse**.

The physics induced mesh automatically introduces a mesh that is a bit finer on the walls compared to the free stream mesh. The following figure shows the boundary layer mesh at the walls.

3. Click the **Build All** button on the settings window.

Zoom into the mesh in the **Graphics** window to see it matches the figure.



STUDY 1

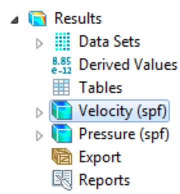
1 In the **Model Builder**, right-click **Study 1** and choose **Compute** .

When **Compute** is selected, COMSOL automatically uses a suitable solver for the problem.

RESULTS

Two plots are automatically created, one slice plot for the velocity and one pressure contour plot on the wall.

Velocity (spf)



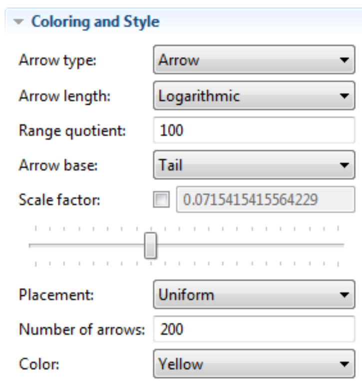
1 In the **Model Builder** under **Results**, expand the **Velocity (spf)** node.

2 Right-click **Slice 1** and choose **Delete**. Click **Yes** to confirm.

3 In the **Model Builder**, right-click **Velocity (spf)** and choose **Surface** .

4 Right-click **Velocity (spf)** and choose **Arrow Surface** .

5 Go to the **Arrow Surface** settings window. Under **Coloring and Style** from the **Arrow length** list, select **Logarithmic**.



6 From the **Color** list, select **Yellow**.

7 Click the **Zoom Extents** button on the **Graphics** window toolbar.

The plot in Figure 8 on page 21 displays in the **Graphics** window. To see the recirculation effects, create a streamline plot of the velocity field.

3D Plot Group 3

1 In the **Model Builder**, right-click **Results** and choose **3D Plot Group** .

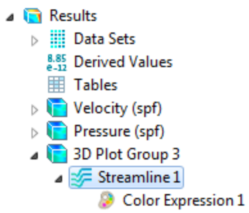
2 Right-click **3D Plot Group 3** and choose **Streamline** .

3 In the **Graphics** window, select Boundary 1 only and right-click it to add this boundary to the selection list in the **Streamline** settings window. The streamlines now start at this boundary.

4 Go to the **Streamline** settings window. Under **Coloring and Style** from the **Line type** list, select **Tube**.

5 Right-click **Streamline 1** and choose **Color Expression** .

6 Click the **Plot** button .



The plot in Figure 9 on page 22 displays in the **Graphics** window.