INTRODUCTION TO

CFD Module

5.2a

COMSOL

Introduction to the CFD Module

Version: COMSOL 5.2a

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM021302

Contents

Introduction

The CFD Module is used by engineers and scientists to understand, predict, and design for fluid flow in closed and open systems. At a given cost, these types of simulations typically lead to new and better products and improved operations of devices and processes compared to purely empirical studies involving fluid flow. As a part of an investigation, simulations give accurate estimates of flow patterns, pressure losses, forces on submerged objects, temperature distributions, and variations in fluid composition within a system.

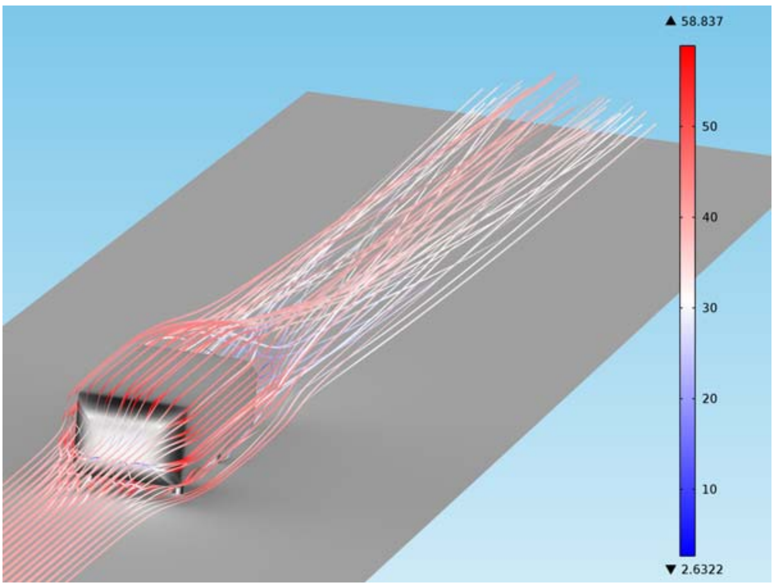


Figure 1: Flow ribbons and velocity field magnitudes from a simulation of an Ahmed body. The simulation yields the flow and pressure fields and calculates the drag coefficient as a benchmark for the verification and validation of turbulence models.

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

The physics interfaces build on conservation laws for momentum, mass, and energy. These laws are expressed in terms of partial differential equations, which are solved by the module together with the specified initial 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 physical quantities that you can freely define, and derived tabulated quantities (for example, average pressure on a surface or drag coefficients) obtained from a simulation.

The workflow in the CFD Module is quite straightforward and is 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 that are tuned for each specific Fluid Flow interface.

The CFD Module's application library describes the Fluid Flow interfaces and their different features through tutorial and benchmark examples for the different types of flow. Here you find models of industrial equipment and devices, tutorials for practice, and benchmark applications for verification and validation of the Fluid Flow interfaces. Go to Tutorial Example—Backstep for information on how to access these resources.

This introduction is intended to give you an accelerated start in CFD application building. It contains examples of the typical use of the module, a list of all the Fluid Flow interfaces including a short description of each, and two tutorial examples, Tutorial Example—Backstep and Tutorial Example—Water Purification Reactor, to introduce the workflow.

Aspects of CFD Simulations

The physical nature of a flow field may be characterized by a set of dimensionless numbers such as the Reynolds number, the Mach number, and the Grashof number. A great deal of information about the flow field can be gained by analyzing these numbers.

The Reynolds number, for example, expresses the ratio of the inertia forces to the viscous (internal friction) forces. For vanishingly small values of the Reynolds number, the inertia forces are negligible and the flow is reversible, in the sense that reversed boundary conditions lead to reversed flow. The energy dissipation is immediate. As the Reynolds number increases, viscous effects become more and more confined to boundaries, internal shear layers, and wakes. The size and other characteristics of such regions are determined by the Reynolds number. Eventually, for very large values of the Reynolds number, the flow becomes fully turbulent. In contrast to laminar flow at high Reynolds numbers, viscous dissipation is active everywhere in a turbulent flow field but acts on very small flow structures. The energy is transfered from the large-scale flow structures to the small-scale flow structures through a cascade of eddies. Due to the requirement of resolving all these flow scales, direct numerical simulation of industrially relevant turbulent flows is currently not a feasible approach. Instead, turbulence models are applied when analyzing these flows. For very small values of the Reynolds number the CFD Module offers the Creeping Flow interface; for intermediate values, the Laminar Flow interface; and for large values, the Turbulent Flow interfaces.

The Mach number expresses the ratio of the speed of the fluid to the speed of sound of the flowing medium. This dimensionless number measures the relevance of compressible effects in the flow field, predicting occurrences of shock waves and rarefaction waves. For Mach numbers greater than 0.3, the laminar and turbulent High Mach Number Flow interfaces are available.

Temperature variations caused by heat transfer, compression work, or work done by friction forces result in an inhomogeneous density field which may trigger thermal convection. The significance of thermally induced buoyancy forces in the momentum equation is characterized by the ratio of the Grashof number to the square of the Reynolds number (for large Reynolds numbers), or of the Grashof number to the Reynolds number (for small Reynolds numbers). For non-vanishing values of this ratio, the Non-Isothermal Flow interfaces are available.

You can use the Two-Phase Flow interfaces in the Multiphase Flow branch to model moving, deformable interfaces separating two different fluids. The other physics interfaces in this branch are mainly intended for modeling suspensions of many particles, droplets or bubbles. Among the latter, the Euler-Euler Model interface is able to handle high concentration levels with frequent collisions, as well as transients in the relative velocity between the phases (that is, non-vanishing ratios of the particle relaxation time to the macroscopic flow time-scale). For reacting flow and flow in porous media, the Chemical Species Transport, and Porous Media and Subsurface Flow branches are available.

Contrary to experimental analyses, which are most often performed in a laboratory where measurements are limited to a small number of points, a CFD simulation gives the "big picture" view of the flow field. A qualitative interpretation of the flow and pressure fields is usually the first step toward 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 induce lift forces that would not be present if the panel were analyzed alone.

Three-dimensional graphics such as surface, streamline, ribbon, arrow, and particle-tracing plots, as well as animations that include any combination of the aforementioned features, are examples of tools you can use for qualitative studies.

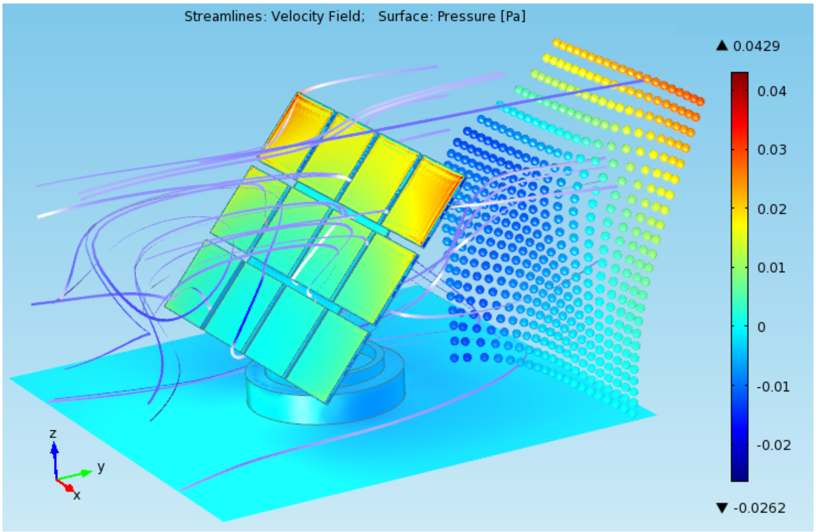


Figure 2: Turbulent fluid flow around a solar panel solved using the CFD Module.

In addition to the qualitative "big picture" view, simulations performed with the CFD Module give accurate quantitative estimates of properties of the flow field, such as the average flow at a given pressure difference, the drag and lift coefficients of bodies subjected to a flow, or the air quality in a ventilated room.

In Figure 3 and Figure 4, the pressure losses are estimated for a nozzle used in medical devices. The shear stresses and fluid forces in the nozzle system may damage blood cells in medical equipment, and must be accounted for when controlling the flow.

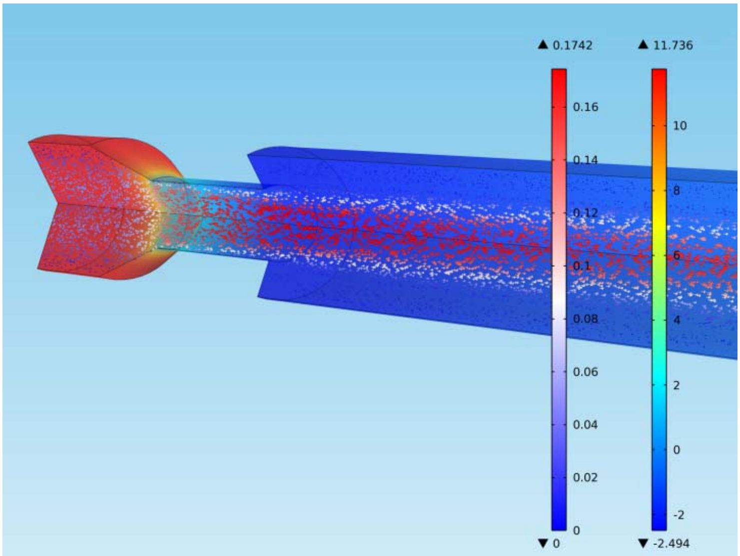


Figure 3: Pressure field and flow field in a model of a nozzle relevant for designs in medical applications.

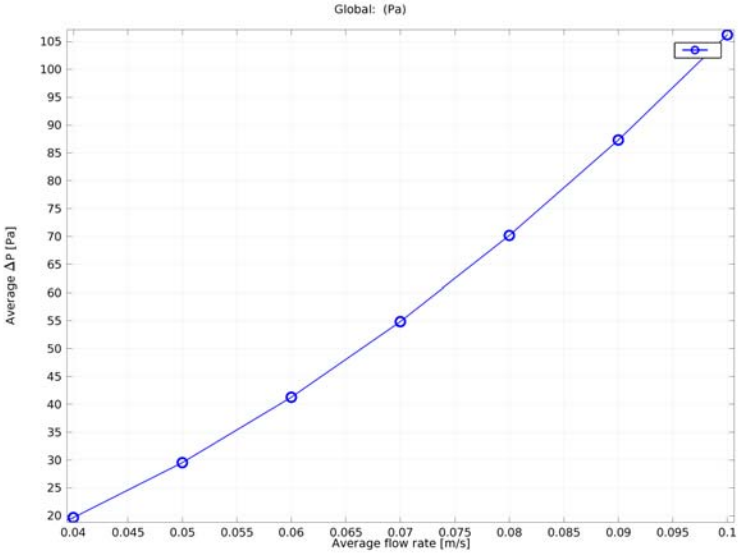


Figure 4: Pressure difference between inlet and outlet at various average flow rates through the nozzle.

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

Qualitative studies typically form the basis for understanding, which in turn can spark new ideas. These ideas can then lead to 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 do so through a series of many smaller steps.

The CFD Module Physics Interfaces

The Fluid Flow interfaces in this module are based on the laws for conservation of momentum, mass, and energy in fluids. The different flow models contain different combinations and formulations of the conservation laws that apply to the physics of the flow field. These laws of physics are translated into partial differential equations and are solved together with the specified initial and boundary conditions.

A physics interface defines a number of features. These features are used to specify the fluid properties, boundary conditions, initial conditions, and possible constraints. Each feature represents an operation describing a term or condition in the conservation equations. Such a term or condition can be defined on a geometric entity of the component, such as a domain, boundary, edge (for 3D components), 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 marked terms to the component equations in a selected geometric 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 at the walls of the fluid domain.

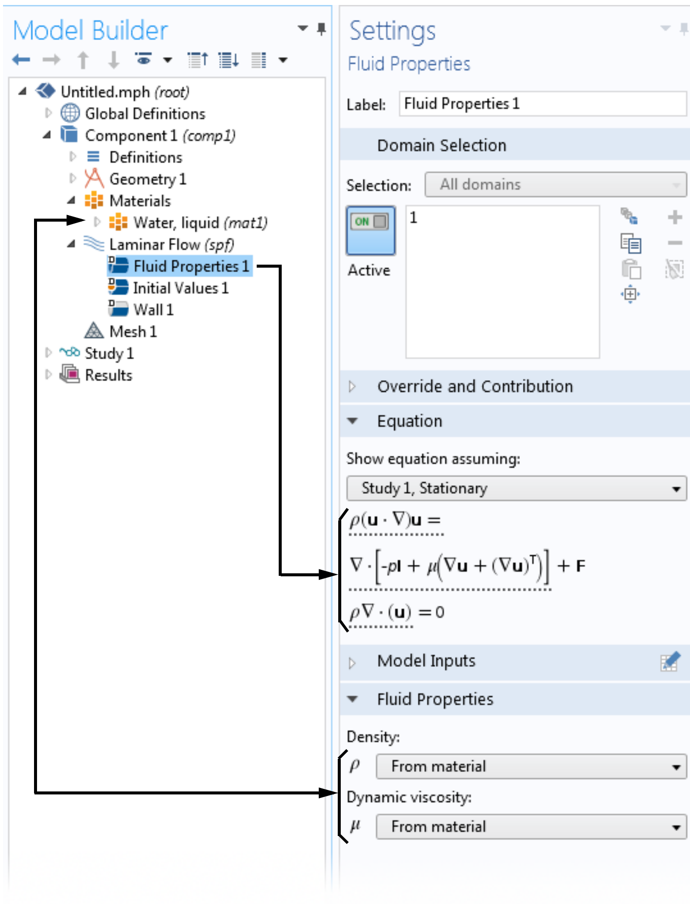


Figure 5: The Model Builder including a Laminar Flow interface (left), and the Settings window for Fluid Properties for the selected feature node (right). The Equation section in the Settings window shows the component equations and the terms added by the Fluid Properties 1. The added terms are underlined with a dotted line. The arrows also explain the link between the Materials 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 physics interfaces for heat transfer in solids, fluids, and porous media found under the Heat Transfer branch.

[Figure 6](#) shows the Fluid Flow interfaces as they are displayed when you add a physics interface (see also Physics Interface Guide by Space Dimension and Study Type for further information). A short description of the physics interfaces follows.

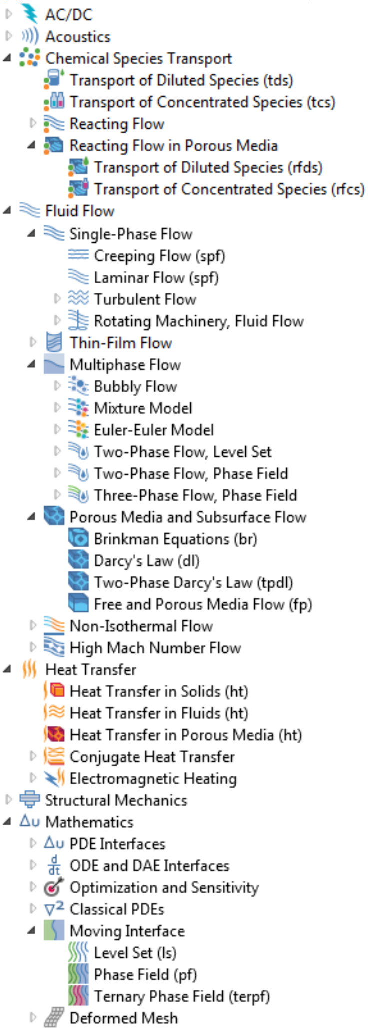


Figure 6: The physics interfaces for the CFD Module as shown in the Model Wizard.

SINGLE-PHASE FLOW

The Laminar Flow interface ( ) is primarily applied to flows at low to intermediate Reynolds numbers. This physics interface solves the Navier-Stokes equations, for incompressible, weakly compressible and compressible flows (up to

Mach 0.3). The Laminar Flow interface also allows for simulation of non-Newtonian flow.

The physics interfaces under the Turbulent Flow branch ( ) model flows at high Reynolds numbers. These physics interfaces solve the Reynolds-averaged Navier-Stokes (RANS) equations for the averaged velocity and pressure fields. The turbulent flow interfaces provide different options for modeling the turbulent viscosity. There are several turbulence models available—two algebraic turbulence models, the Algebraic yPlus and L-VEL models, and five transport-equation models, including a standard k-ε model, a k-ω model, an SST (Shear Stress Transport) model, a low Reynolds number k-ε model and the Spalart-Allmaras model. Similarly to the Laminar Flow interface, incompressible flow is selected by default.

The Algebraic yPlus and L-VEL turbulence models are so-called enhanced viscosity models. A turbulent viscosity is computed from the local distance to the nearest wall. For this reason, the algebraic turbulence models are best suited for internal flows, such as in electronic cooling applications. Algebraic turbulence models are computationally economical, and more robust but, in general, less accurate than transport-equation models. Among the transport-equation turbulence models, the standard k-ε model is the most widely used since it often is a good compromise between accuracy and computational cost (memory and CPU time). The k-ω model is an alternative to the standard k-ε model and often gives more accurate results, especially in recirculation regions and close to solid walls. However, the k-ω model is also less robust than the standard k-ε model. The SST model combines the robustness of the k-ε model with the accuracy of the k-ω model, making it applicable to a wide variety of turbulent flows. The Low Reynolds number k-ε model is more accurate than the standard k-ε model, especially close to walls. The Spalart-Allmaras model is specifically designed for aerodynamic applications, such as flow around wing profiles, but is also widely used in other applications due to its high robustness and decent accuracy. Higher resolution is needed in the near-wall region for the SST model, the Low Reynolds number k-ε model, and the Spalart-Allmaras model. Thus, the better accuracy provided by these models comes at a higher computational cost.

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

The Rotating Machinery interfaces ( ) are applicable to fluid-flow problems where one or more of the boundaries rotate, for example in mixers and around propellers. The physics interfaces support incompressible, weakly compressible and compressible (Mach < 0.3) flows, laminar Newtonian and non-Newtonian flows, and turbulent flow using the standard k-ε model or either of the two algebraic turbulence models (Algebraic yPlus or L-VEL).

Multiphase Flow

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

The physics interfaces under the Mixture Model branch ( ) are similar to the Bubbly Flow interfaces but assume 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 flow and turbulent flow using the k-ε turbulence model. 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, for example, fluidized beds. There is support for both laminar flow and turbulent flow using either a mixture or phase-specific k-ε turbulence model.

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

The Laminar Three-Phase Flow, Phase Field interface ( ) models laminar flow of three incompressible phases which may be either Newtonian or non-Newtonian. The moving fluid-fluid interfaces between the three phases are tracked in detail using the phase-field method.

Nonisothermal Flow

The Non-Isothermal Flow, Laminar Flow interface ( ) is primarily applied to model flow at low to intermediate Reynolds numbers in situations where the temperature and flow fields have to be coupled. A typical example is natural convection, where thermal buoyancy forces drive the flow. This is a multiphysics interface for which the component couplings between fluid flow and heat transfer are set up automatically.

The Non-Isothermal Flow, Turbulent Flow interfaces ( ) solve the Reynolds-Averaged Navier-Stokes (RANS) equations coupled to heat transfer in fluids and in solids. There is support for all the fluid-flow turbulence models – the Algebraic yPlus model, the L-VEL model, the standard k-ε model, a k-ω model, an SST model, a low Reynolds number k-ε model, and the Spalart-Allmaras model.

The Conjugate Heat Transfer interfaces ( ) are also included with the CFD Module and are almost identical to the Non-Isothermal Flow interfaces. They only differ in the default domain feature selected -Heat transfer in Solids instead of Fluid.

High Mach Number Flow

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

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

Porous Media and Subsurface Flow

The Brinkman Equations interface ( ) models flow through porous media where the influence of shear stresses is significant. This physics interface supports the Stokes-Brinkman formulation, suitable for very low flow velocities, as well as the full Brinkman equations including convective terms and Forchheimer drag, which is used to account for effects at high interstitial velocities. The flow can be either incompressible or compressible, provided the Mach number is less than 0.3.

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

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

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

Reacting Flow

The Laminar Flow interface ( ) under the Reacting Flow branch combines the functionality of the Single-Phase Flow and Transport of Concentrated Species interfaces. The physics interface is primarily applied to model flow at low to intermediate Reynolds numbers in situations where the mass transport and flow fields have to be coupled. is primarily applied to model flow at low to intermediate Reynolds numbers in situations where the temperature and flow fields have to be coupled

The Turbulent Flow interfaces ( ) under the Reacting Flow branch apply the Reynolds-Averaged Navier-Stokes (RANS) equations together with the functionality in the Transport of Concentrated Species interface. They model mass and momentum transport in turbulent reacting fluid flow. The supported RANS models comprise of the standard k-ε model, a k-ω model, and a low Reynolds number k-ε model.

Reacting Flow in Porous Media

The Reacting Flow in Porous Media, Transport of Diluted Species interface ( ) and the Reacting Flow in Porous Media, Transport of Concentrated Species interface ( ) model diluted respectively concentrated reacting mixtures transported by free and/or porous media flow. Effective diffusion coefficients in a porous matrix can be calculated from the porosity.

Thin-Film Flow

The Thin Film Flow interfaces ( ) model the flow of liquids or gases confined in a thin layer on a surface. Applying equations defined on a surface, these physics interfaces compute the average velocity and pressure across the layer in narrow planar structures. The physics interfaces are thus boundary physics interfaces, which means that the boundary level is the highest level; they do not have a domain level. The simulation of the flow of a lubrication oil between two rotating cylinders is an example of a possible application for this physics interface.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Chemical Species Transport				
Transport of Diluted Species$^{(1)}$		tds	all dimensions	stationary; time dependent
Transport of Concentrated Species		tcs	all dimensions	stationary; time dependent
Reacting Flow				
Laminar Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, k-ε		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ω		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, SST		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Low Re k-ε		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Reacting Flow in Porous Media				
Transport of Diluted Species		rfds	3D, 2D, 2D axisymmetric	stationary; time dependent
Transport of Concentrated Species		rfcs	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Fluid Flow				
Single-Phase Flow				
Creeping Flow		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Laminar Flow[(1)]		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, Algebraic yPlus		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
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, SST		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
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
Rotating Machinery, Fluid Flow				
Rotating Machinery, Laminar Flow		rmspf	3D, 2D	frozen rotor; time dependent
Rotating Machinery, Turbulent Flow, Algebraic yPlus		rmspf	3D, 2D	frozen rotor with initialization; transient with initialization

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Rotating Machinery, Turbulent Flow, L-VEL		rmspf	3D, 2D	frozen rotor with initialization; transient with initialization
Rotating Machinery, Turbulent Flow, k-ε		rmspf	3D, 2D	frozen rotor; time dependent
Thin-Film Flow				
Thin-Film Flow, Shell		tffs	3D	stationary; time dependent; frequency domain; eigenfrequency
Thin-Film Flow, Domain		tff	2D	stationary; time dependent; frequency domain; eigenfrequency
Thin-Film Flow, Edge		tffs	2D and 2D axisymmetric	stationary; time dependent; frequency domain; eigenfrequency
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
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
Euler-Euler Model, Turbulent Flow		ee	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Two-Phase Flow, Level Set				
Laminar Two-Phase Flow, Level Set		—	3D, 2D, 2D axisymmetric	transient with phase initialization
Turbulent Two-Phase Flow, Level Set		—	3D, 2D, 2D axisymmetric	transient with phase initialization
Two-Phase Flow, Phase Field				
Laminar Two-Phase Flow, Phase Field		—	3D, 2D, 2D axisymmetric	transient with phase initialization
Turbulent Two-Phase Flow, Phase Field		—	3D, 2D, 2D axisymmetric	transient with phase initialization
Three-Phase Flow, Phase Field				
Laminar, Three-Phase Flow, Phase Field		—	3D, 2D	time dependent
Porous Media and Subsurface Flow				
Brinkman Equations		br	3D, 2D, 2D axisymmetric	stationary; time dependent
Darcy's Law		dl	all dimensions	stationary; time dependent
Two-Phase Darcy's Law		tpdl	3D, 2D, 2D axisymmetric	stationary; time dependent
Free and Porous Media Flow		fp	3D, 2D, 2D axisymmetric	stationary; time dependent
Non-Isothermal Flow				
Laminar Flow(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Turbulent Flow				
Turbulent Flow, Algebraic yPlus[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ω[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, SST[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Low Re k-ε[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Spalart-Allmaras[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
High Mach Number Flow				
High Mach Number 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[1]		ht	all dimensions	stationary; time dependent
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Conjugate Heat Transfer				
Laminar Flow(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, Algebraic yPlus		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, k-ω(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, SST(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Spalart-Allmaras(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Mathematics				
Moving Interface				
Level Set		ls	all dimensions	transient with phase initialization
Phase Field		pf	all dimensions	time dependent; transient with phase initialization

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Ternary Phase Field		terpf	3D, 2D, 2D axisymmetric	time dependent

(1) This physics interface is included with the core COMSOL package but has added functionality for this module.

(2) This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.

Tutorial Example—Backstep

This tutorial 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 application 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.

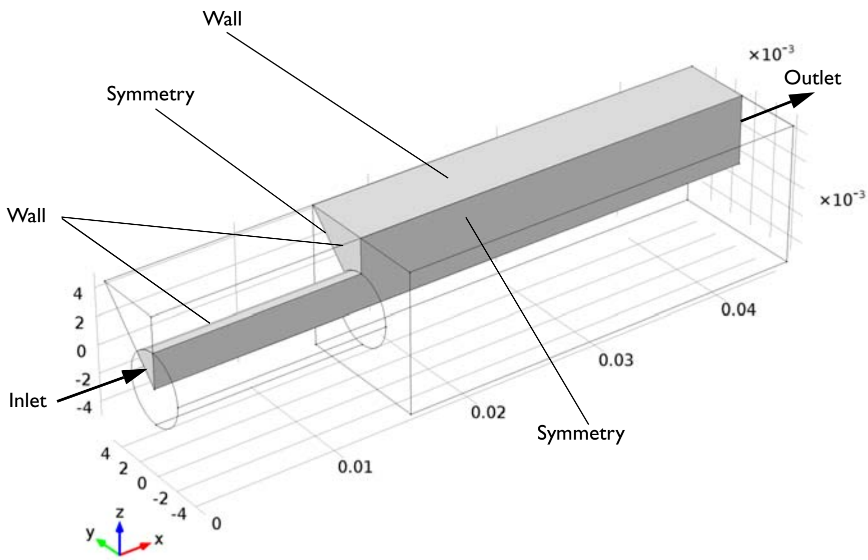


Figure 7: The model geometry showing the symmetry.

Domain Equations and Boundary Conditions

The flow in this system is laminar and you should therefore use the Laminar Flow interface.

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 a channel of 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 no-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.

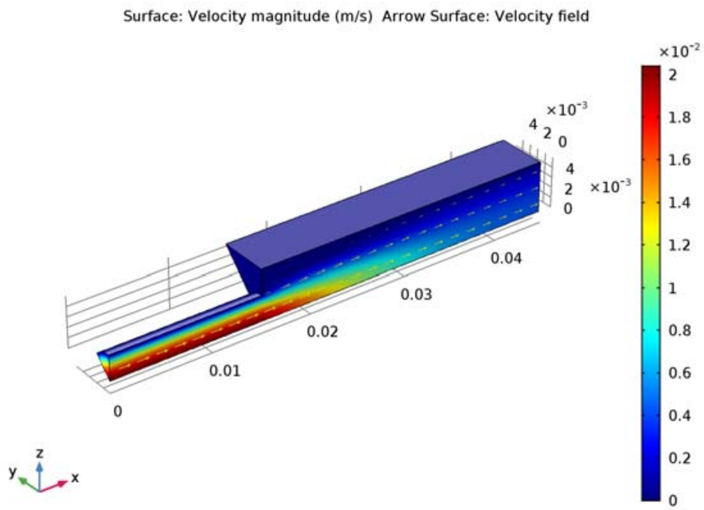


Figure 8: The velocity field in the backstep geometry.

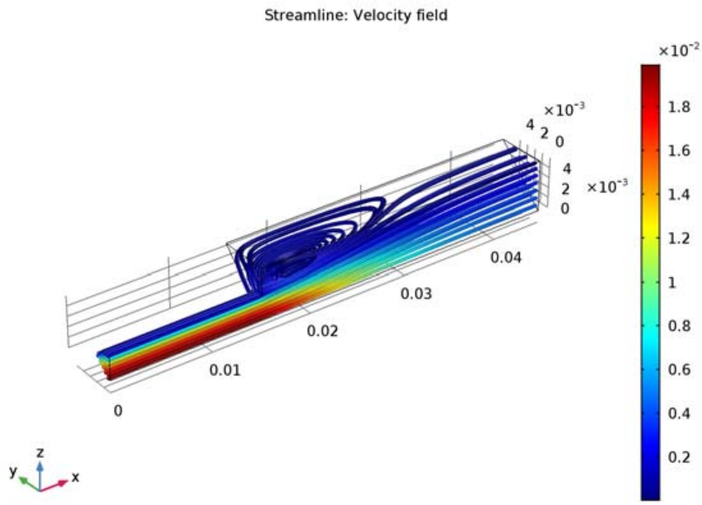


Figure 9: The recirculation region visualized using a velocity streamline plot.

The following instructions show how to set up the model, solve it, and reproduce these plots.

Model Wizard

The first step to build an application is to open COMSOL, then select the physics interface and specify the type of analysis you want to do—in this case, a stationary, Laminar Flow analysis.

Note: These instructions are for the user interface on Windows but also apply, with minor differences, to Linux and Mac.

1. To open the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the Model Wizard to create a new COMSOL application or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

   The Model Wizard guides you through the first steps of setting up an application. If COMSOL is already open, you can start the Model Wizard by selecting New from the File menu and then clicking Model Wizard.

   The next window lets you select the dimension of the modeling space.

2 In the Select Space Dimension window click the 3D button .

3 In the Select Physics tree under Fluid Flow>Single-Phase Flow click Laminar Flow (spf) .

4 Click Add and then click the Study button .

5 In the tree under Preset Studies, click Stationary .

6 Click the Done button .

Global Definitions - Parameters

The first task is to define a parameter for the inlet velocity. Then you will use this parameter to run a parametric study.

1 On the Home toolbar click Parameters .

Note: On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

2 In the Settings window for Parameters locate the Parameters section. In the table enter the following settings:

- In the Name text field, enter `v0`
- In the Expression text field, enter `1[cm/s]`
- In the Description text field, enter `Inlet velocity`

Parameters

Name	Expression	Value	Description
v0	1[cm/s]	0.01 m/s	Inlet velocity

Geometry 1

You can build the backstep geometry from geometric primitives. Here, instead use a file containing the sequence of geometry features that has been provided for convenience.

Note: The location of the file used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path might be similar to
`C:\Program Files\COMSOL\COMSOL52a\Multiphysics\applications\`.

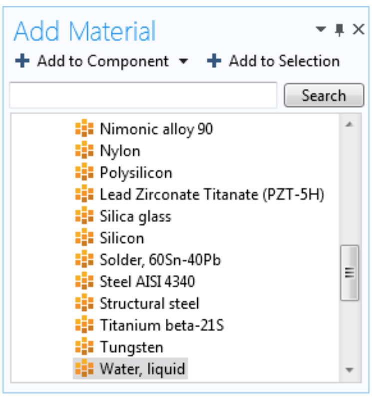
1 On the Geometry toolbar choose Insert Sequence .

2 Browse to the applications library folder and double-click the file `\CFD_Module\Single-Phase_Tutorials\backstep_geom_sequence.mph`.
3 Go to the Home toolbar and click Build All.

The geometry sequence is now inserted into your component and should look like the geometry in Figure 7.

Materials

1 On the Home toolbar click Add Material.
2 Go to the Add Material window. In the tree under Built-In click Water, liquid.



3 In the Add Material window, click Add to Component.
4 On the Home toolbar click Add Material again to close the window.

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.

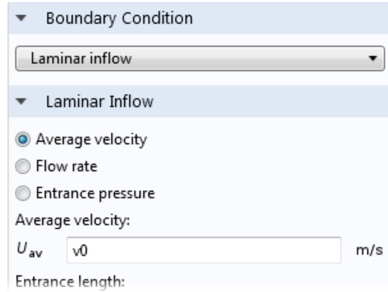
Laminar Flow

Inlet 1

1 On the Physics toolbar click Boundaries and choose Inlet.
2 Select Boundary 1, which represents the inlet.

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

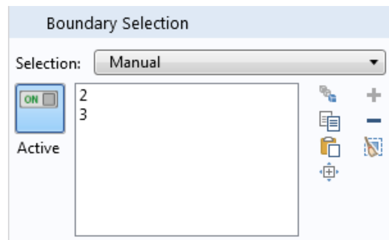
4 Under Laminar Inflow in the U_{av} text field, type `v0` (which you defined as a Global Parameter).



Symmetry 1

1 On the Physics toolbar click Boundaries and choose Symmetry.

2 Select Boundaries 2 and 3 only.



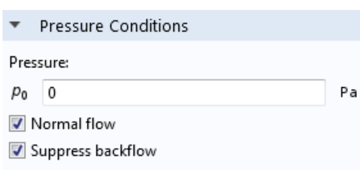
Outlet 1

1 On the Physics toolbar click Boundaries and choose Outlet.

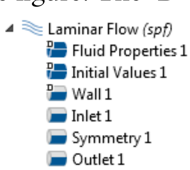
The default outlet condition specifies a zero relative pressure.

2 Go to the Settings window for Outlet. Select Boundary 7 only.

3 On the Settings window for Outlet, locate the Pressure Conditions section. Select the Normal flow check box.



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.

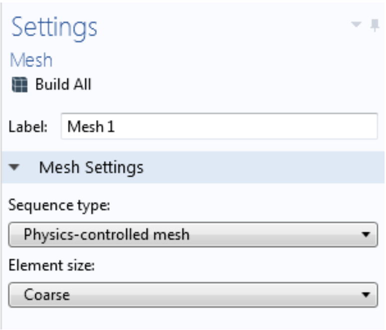


All other boundaries now have the default wall condition.

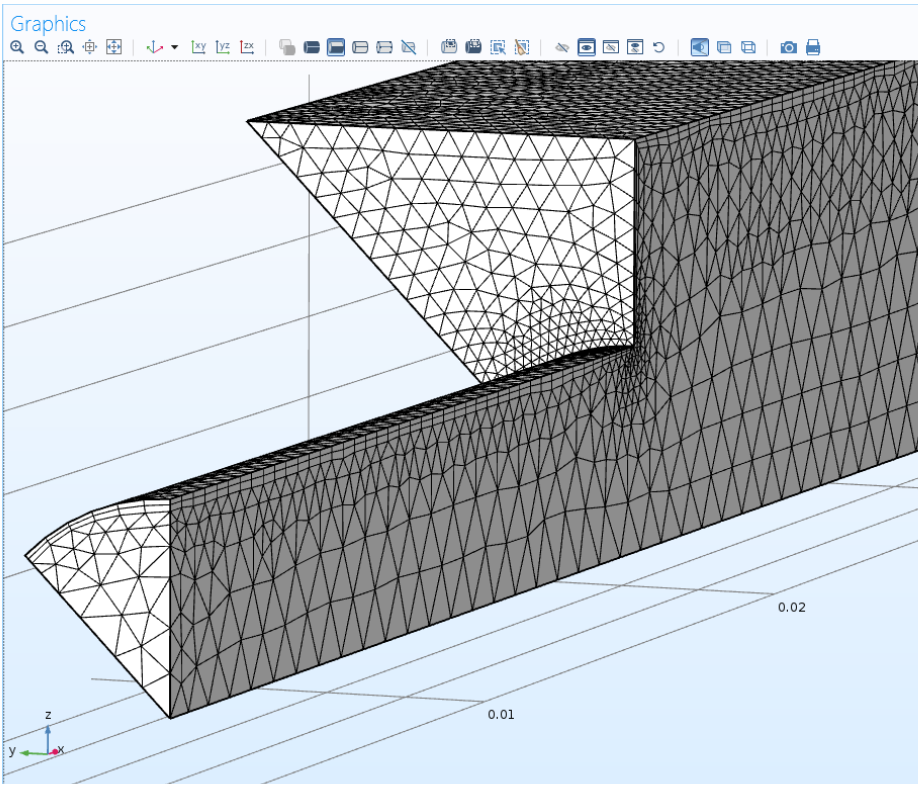
Mesh 1

1. In the Model Builder click Mesh 1.
2. In the Settings window for Mesh locate the Mesh Settings section. From the Element size list, choose Coarse.

   The physics-induced mesh automatically introduces a mesh that is a bit finer on the walls than the free stream mesh.
3. Click the Build All button.



The figure below shows the boundary layer mesh at the walls. Zoom in to the mesh using the zoom function on the Graphics toolbar to confirm that it matches the figure.



Study 1

1 On the Home toolbar click Compute.

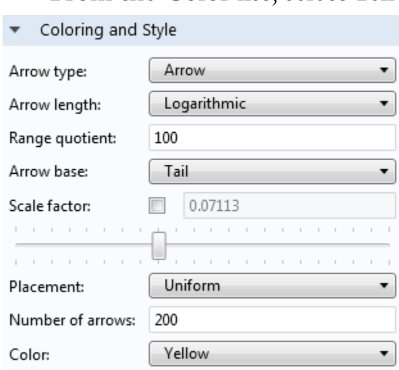
When Compute is selected, COMSOL Multiphysics automatically chooses a suitable solver for the problem.

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.
3 Go to the Velocity (spf) toolbar and click Surface.
4 On the Velocity (spf) toolbar, click Arrow Surface.
5 Go to the Settings window for Arrow Surface.
   - Under Coloring and Style from the Arrow length list, select Logarithmic.
   - From the Color list, select Yellow.



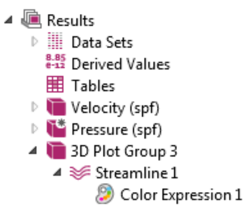
6 Click the Plot button followed by the Zoom Extents button on the Graphics window toolbar.

The plot in Figure 8 displays in the Graphics window.

To see the recirculation effects, create a streamline plot of the velocity field.

3D Plot Group 3

1. On the Home toolbar click Add Plot Group and choose 3D Plot Group .
2. Go to the 3D Plot Group 3 toolbar and click Streamline .
3. In the 3D Plot Group Settings window, scroll to the Selection section, and click the Active button next to the Selection list.
4. Select Boundary 1 (the inflow boundary) only.
5. In the Settings window for Streamline locate the Coloring and Style section. From the Line type list, choose Tube.
6. Right-click Streamline 1 and choose Color Expression .



The plot in Figure 9 displays in the Graphics window.

Tutorial Example—Water Purification Reactor

Water purification is a multiple step process for turning natural water into drinking water. At least one step must be a disinfectant step. One way to achieve efficient disinfection in an environmentally friendly way is to use ozone. A typical ozone purification reactor is about 40 m long and resembles a mace with partial walls or baffles that divide the space into room-sized compartments (Ref. 1). When water flows through the reactor, turbulent flow is created along its winding path around the baffles towards the exit pipe. The turbulence mixes the water with ozone gas, which enters through diffusers just long enough to inactivate micro-pollutants. When the water leaves the reactor, the remaining purification steps filter off or otherwise remove the reacted pollutants.

In analyzing an ozone purification reactor, the first step is to get an overview of the turbulent flow field. The results from the turbulent-flow simulation can then be used for further analyses of residence time and chemical species transport and reactions. This requires adding more physics features to the model. The current application solves for turbulent flow in a water treatment reactor using the Turbulent Flow, k-ε interface.

Model Geometry

The model geometry along with some boundary conditions is shown in Figure 10. The full reactor has a symmetry plane, which is utilized to reduce the size of the component.

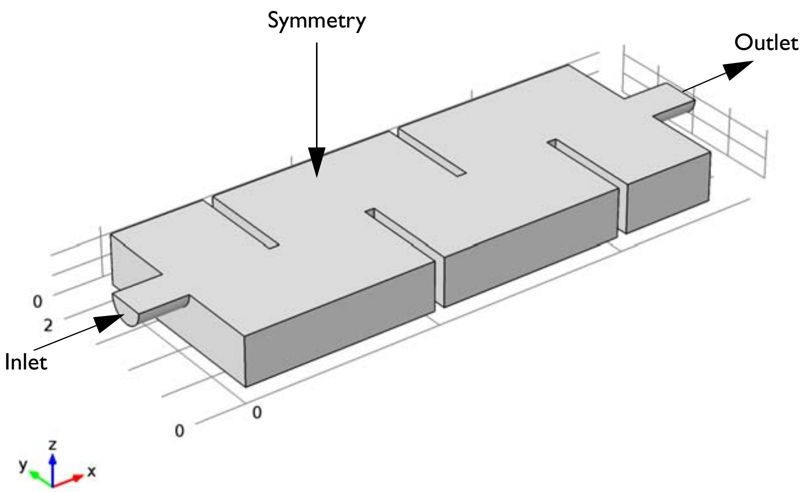


Figure 10: Model geometry. All boundaries except the inlet, outlet and symmetry plane are walls.

Domain Equations and Boundary Conditions

Based on the inflow velocity, which is 0.1 m/s, and a length scale L equal to the diameter of the inlet, the Reynolds number is

$$\mathrm{Re} = \frac{U \cdot L}{\nu} = \frac{0.1 \cdot 0.4}{1 \cdot 10^{-6}} = 4 \cdot 10^5$$

Here ν is the kinematic viscosity. The high Reynolds number clearly indicates that the flow is turbulent and a turbulence model must be applied. In this case, you will use the k-ε model. It is commonly used in industrial applications, because it is both relatively robust and computationally inexpensive compared to more advanced turbulence models. One major reason the k-ε model is inexpensive is that it employs wall functions to describe the flow close to walls instead of resolving the very steep gradients there. All boundaries are walls in Figure 10 except the inlet, the outlet, and the symmetry plane.

The inlet velocity is prescribed as a plug flow profile. The turbulent intensity is set to 5 % and the turbulent length scale is specified according to Table 3-5 in Theory for the Turbulent Flow Interfaces in the *CFD Module User's Guide*. A constant pressure is prescribed on the outlet.

Notes About the Implementation

A three-dimensional turbulent flow can take a rather long time to solve, even using a turbulence model with wall functions. To make this tutorial feasible, the mesh is deliberately selected to be relatively coarse and the results are hence not mesh-independent. In any component, the effect of refining the mesh should be investigated in order to ensure that the model is well-resolved.

Results

The velocity field in the symmetry plane is shown in Figure 11. The jet from the inlet hits the top of the first baffle, which splits the jet. One half creates a strong recirculation zone in the first "chamber". The other half continues downstream into the reactor and gradually spreads out. The velocity magnitude decreases as more fluid is entrained into the jet.

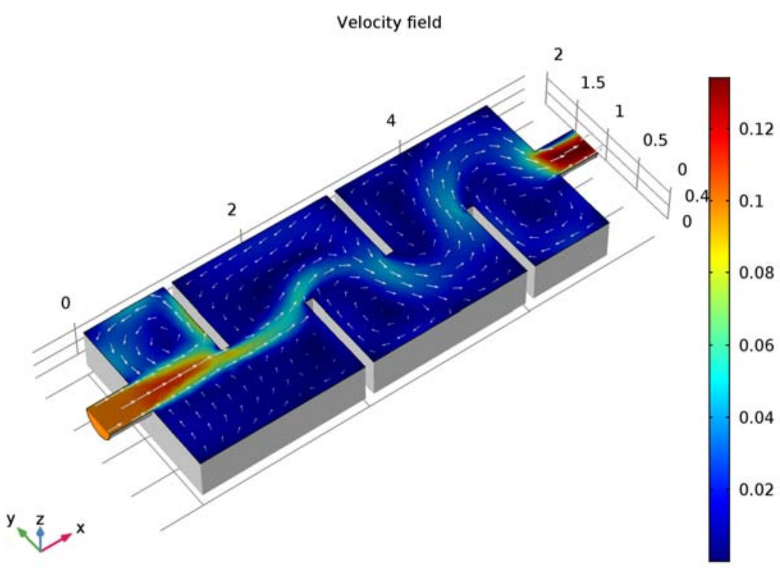


Figure 11: Velocity field in the symmetry plane.

Figure 12 gives a more complete picture of the mixing process in the reactor. The streamlines are colored by the velocity magnitude, and their widths are proportional to the turbulent viscosity. Wide lines hence indicate a high degree of mixing. The turbulence in this model is mainly produced in the shear layers between the central jet and the recirculation zones. The mixing can be seen to be relatively weak near the entrance to the reactor and to increase further downstream.

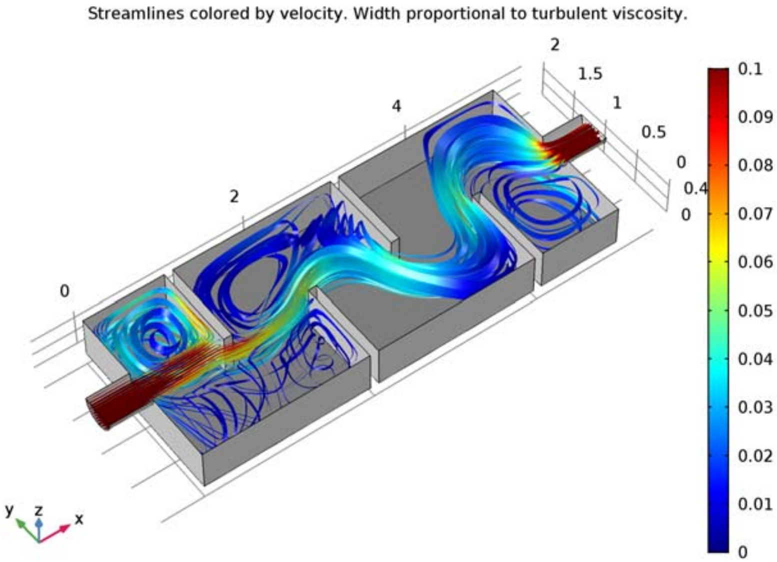


Figure 12: Streamlines colored by velocity. The width of the streamlines is proportional to the turbulent viscosity.

Reference

1. http://www.comsol.com/stories/hofman_water_purification/full/

Model Wizard

The first step to build an application is to open COMSOL, then select the physics interface and specify the type of analysis you want to do—in this case, a stationary, Turbulent Flow, k-ε analysis.

1 Open COMSOL Multiphysics. On the New page click Model Wizard. Then click the 3D button.
2 In the Select Physics tree under Fluid Flow>Single-Phase Flow>Turbulent Flow, click Turbulent Flow, k-ε (spf).
3 Click Add and then click the Study button
4 In the tree under Preset Studies, click Stationary.
5 Click the Done button.

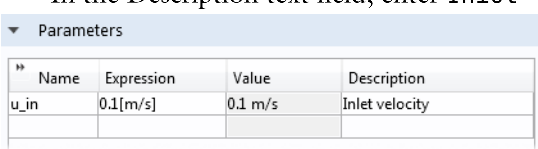
Global Definitions - Parameters

The first task is to define a parameter for the inlet velocity. Parameters can be used to run parametric studies.

1 On the Home toolbar click Parameters P_i.

   The Home toolbar refers to the specific set of controls near the top of the Desktop.

2 Go to the Settings window for Parameters. In the table, enter the following settings:
   - In the Name text field, enter `u_in`
   - In the Expression text field, enter `0.1[m/s]`
   - In the Description text field, enter `Inlet velocity`



Name	Expression	Value	Description
u_in	0.1[m/s]	0.1 m/s	Inlet velocity

Geometry 1

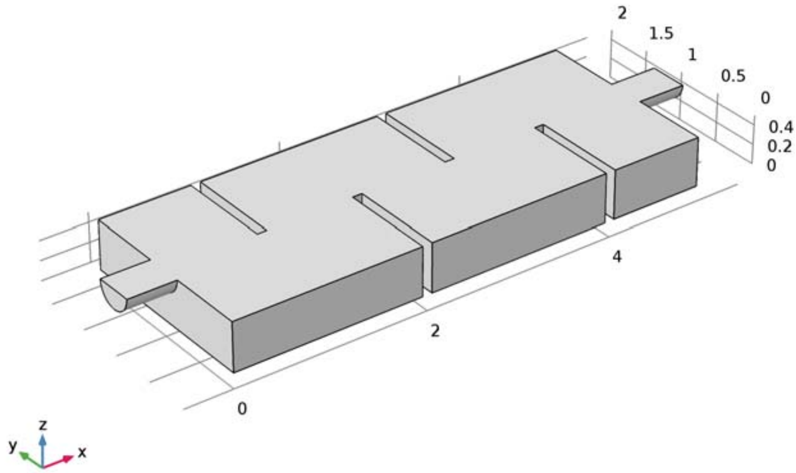
You can build the reactor geometry from geometric primitives. Here, instead use a file containing the sequence of geometry features that has been provided for convenience.

Note: The location of the file used in this exercise varies based on your installation. For example, if the installation is on your hard drive, the file path might be similar to
`C:\Program Files\COMSOL\COMSOL52a\Multiphysics\applications\`.

1 On the Geometry toolbar click Insert Sequence.

2 Browse to the applications library folder and double-click the file `\CFD_Module\Single-Phase_Tutorials\water_purification_reactor_geom_sequence.mph`.
3 On the Home toolbar click Build All .

The geometry sequence is now inserted into your component and should look like the figure below.



Materials

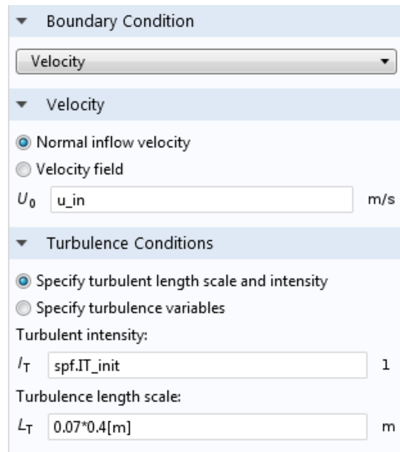
1 On the Home toolbar click Add Material .
2 Go to the Add Material window. In the tree under Built-In click Water, liquid .
3 In the Add Material window, click Add to Component.
4 On the Home toolbar, click Add Material again to close the window.

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.

Turbulent Flow, k-ε

Inlet 1

1. On the Physics toolbar click Boundaries and choose Inlet.
2. Select Boundary 1, which represents the inlet.
3. In the Settings window for Inlet locate the Velocity section. In the U_0 text field, type `u_in`.
4. Locate the Turbulence Conditions section. In the L_T text field, type `0.07*0.4[m]` where `0.4[m]` is the diameter of the inlet (see Inlet Values for the Turbulence Length Scale and Intensity).



Symmetry 1

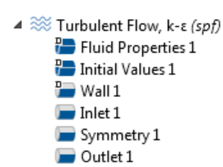
1. On the Physics toolbar click Boundaries and choose Symmetry.
2. Select Boundary 3 only.

Outlet 1

1. On the Physics toolbar click Boundaries and choose Outlet.
2. Select Boundary 28 only.

The sequence of nodes in the Model Builder under Turbulent Flow, k-ε should match the figure. The 'D' in the upper left corner of a node means it is a default node.

All boundaries not selected in Inlet 1, Symmetry 1, or Outlet 1 now have the default wall condition.



Mesh 1

The physics-induced mesh automatically introduces a mesh that is a bit finer on the walls than the free stream mesh. It also refines the mesh at sharp corners and adds a boundary layer mesh. The finer mesh on the walls is not critical in this model since most of the turbulence is produced in the shear layers between the jet and the recirculation zones. The boundary layer mesh can also be coarsened in order to save computational time.

1 In the Model Builder under Component 1 click Mesh 1.
2 In the Settings window for Mesh, from the Element size list, choose Coarser.

Size 1

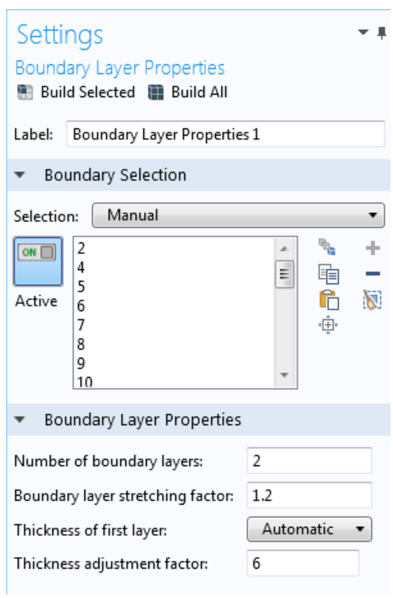
1 Go to the Mesh toolbar and click Edit.

A mesh sequence as shown below appears. It contains suggestions made by the physics interface. The asterisk on each of the mesh features indicates that the features are not yet built.

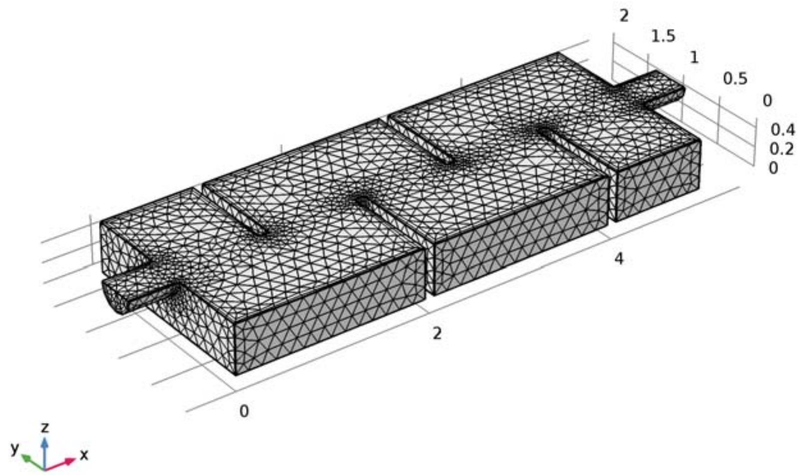
2 Right-click Size 1 and choose Disable.

Boundary Layer Properties 1

1 In the Model Builder, expand the Component 1>Mesh 1> Boundary Layers 1 node, then click Boundary Layer Properties 1.
2 In the Settings window for Boundary Layer Properties locate the Boundary Layer Properties section.
   - In the Number of boundary layers text field, type 2.
   - In the Thickness adjustment factor text field, type 6.
3 Click the Build All button.
4 In the Model Builder, collapse the Mesh 1 node.



The mesh is now complete and should match the figure below. The mesh can differ slightly depending on which computer architecture you use. The mesh in the figure is built on a Windows computer, and will look similar, but not identical, if built on, for example, a Mac or Linux computer.



Study 1

Next, solve for the flow field. This takes approximately 15 minutes on a quad-core desktop computer.

1 On the Home toolbar click Compute.

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

Results

Three plots are automatically created, one slice plot for the velocity, one pressure contour plot on the walls and one boundary plot of the wall lift-off in viscous units for the wall functions. The last one is important since it gives an indication of how well resolved the flow is at the walls. See Theory for the Turbulent Flow Interfaces in the *CFD Module User's Guide* for further details on wall functions.

The following steps reproduce Figure 11.

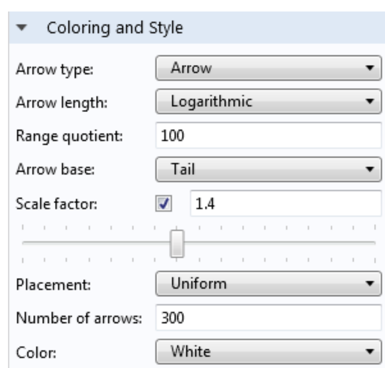
First, create a data set that corresponds to the non-wall boundaries.

Data Sets

1 On the Results toolbar click More Data Sets and choose Surface.
2 Select Boundaries `1`, `3`, and `28` only which correspond to the non-wall boundaries.

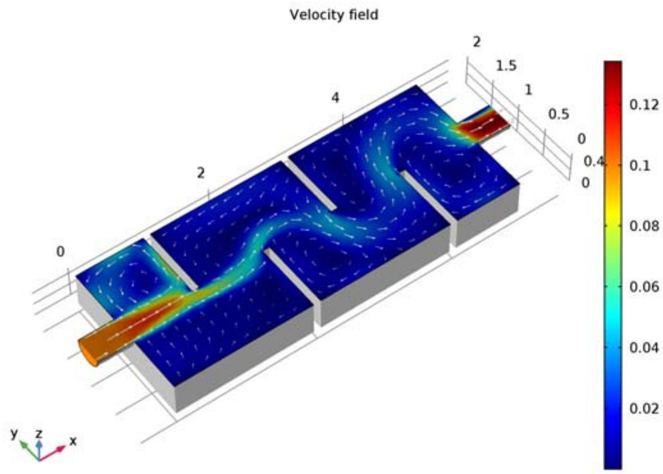
Velocity (spf)

1 In the Model Builder expand the Results>Velocity (spf) node.
2 Right-click Slice 1 and choose Disable.
3 In the Model Builder click Velocity (spf).
4 In the Settings window for 3D Plot Group locate the Data section. From the Data set list, choose Surface 2.
5 On the Velocity (spf) toolbar click Surface.
6 In the Settings window for Surface locate the Data section. From the Data set list, choose Exterior Walls.
7 Locate the Coloring and Style section.
- From the Coloring list, choose Uniform.
- From the Color list, choose Gray.
8 On the Velocity (spf) toolbar click Surface to generate a surface plot of the velocity magnitude.
9 On the Velocity (spf) toolbar click Arrow Surface.
10 In the Settings window for Arrow Surface locate the Coloring and Style section.
- From the Arrow length list, choose Logarithmic.
- Select the Scale factor check box. In the associated text field, type `1.4`.
- In the Number of arrows text field, type `300`.
- From the Color list, choose White.



11 In the Model Builder click Velocity (spf).
12 In the Settings window for 3D Plot Group click to expand the Title section.
- From the Title type list, choose Manual.
- In the Title text area, type `Velocity field`.

13 Click the Plot button.



Also reproduce Figure 12 as follows.

1 In the Model Builder under Velocity (spf), right-click Surface 1 and choose Copy.

3D Plot Group 4

1 On the Home toolbar click Add Plot Group and choose 3D Plot Group.

2 In the Model Builder right-click 3D Plot Group 4 and choose Paste Surface.

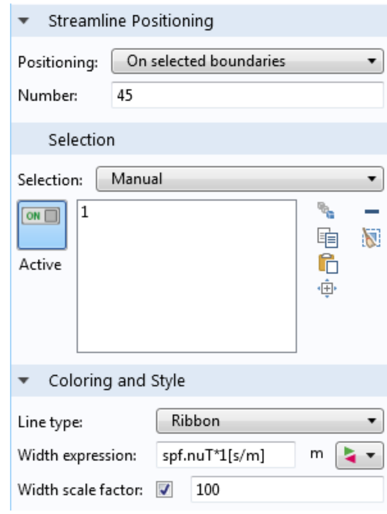
3 On the 3D Plot Group 4 toolbar click Streamline.

4 In the Settings window for Streamline go to the Selection section and click the Active button next to the Selection list.

5 Select Boundary 1 which is the inlet. The streamlines now start at this boundary.

6 In the Settings window for Streamline:

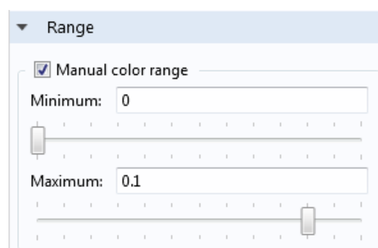
- Locate the Streamline Positioning section. In the Number text field, type `45`.
- Locate the Coloring and Style section. From the Line type list, choose Ribbon.
- In the Width expression text field, type `spf.nuT*1[s/m]`. The width of the streamlines is set to the local value of the turbulent viscosity and the factor `1[s/m]` is used to get the right dimension.
- Select the Width scale factor check box. In the associated text field, type `100`.



7 Right-click Results>3D Plot Group 4> Streamline 1 and choose Color Expression.

8 In the Settings window for Color Expression click to expand the Range section.

- Select the Manual color range check box.
- In the Minimum text field, type `0`.
- In the Maximum text field, type `0.1`.

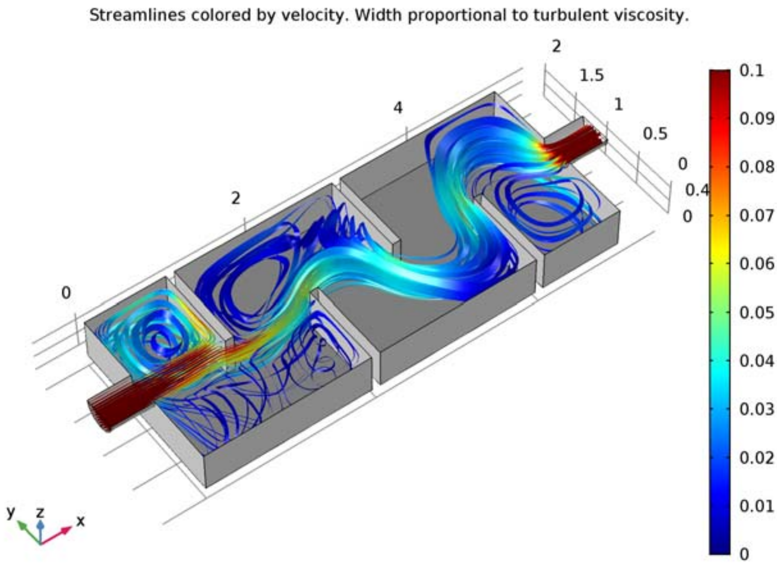


9 In the Model Builder click 3D Plot Group 4.

10 In the Settings window for 3D Plot Group click to expand the Title section.

- From the Title type list, choose Manual.
- In the Title text area, type `Streamlines colored by velocity. Width proportional to turbulent viscosity.`

On the 3D Plot Group 4 toolbar click Plot .



This concludes the introduction to the COMSOL CFD Module.