



Mixer Module

User's Guide

Mixer Module User's Guide

© 1998–2018 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 5.4

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: CM023301

C o n t e n t s

Chapter 1: Introduction

About the Mixer Module	8
The Mixer Module Physics Interface Guide	11
Where Do I Access the Documentation?.	17
Overview of the User's Guide	22
Tutorial Example — Nonisothermal Mixer	23

Chapter 2: Mixer Module Part Library

Part Library Contents	40
Tank Parts	41
Impeller Parts	42
Impeller Parameters.	58

Chapter 3: Mixer Module Theory

Theory for the Free Surface Features	62
Deforming Domain	62
Free Surface Conditions	63
Contact Angle	63
Rotating Shaft Conditions	64
References for the Free Surface Features.	65

Chapter 4: Rotating Machinery, Fluid Flow

The Rotating Machinery, Fluid Flow Interfaces	69
Moving Mesh	69
The Rotating Machinery, Laminar Flow Interface	70
The Rotating Machinery, Turbulent Flow, Algebraic γ Plus Interface	72
The Rotating Machinery, Turbulent Flow, L-VEL Interface	73
The Rotating Machinery, Turbulent Flow, k - ε Interface	74
The Rotating Machinery, Turbulent Flow, Realizable k - ε Interface	75
The Rotating Machinery, Turbulent Flow, k - ω Interface	77
The Rotating Machinery, Turbulent Flow, SST Interface	78
The Rotating Machinery, Turbulent Flow, Low Re k - ε Interface	79
The Rotating Machinery, Turbulent Flow, Spalart-Allmaras Interface	80
The Rotating Machinery, Turbulent Flow, ν 2-f Interface	82
Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces	84
Stationary Free Surface	85
Free Surface.	86
Contact Angle	88
Rotating Shaft	89

Chapter 5: Rotating Machinery, Multiphase Flow Interfaces

The Rotating Machinery, Multiphase Flow, Mixture Model, Laminar Flow and Turbulent Flow Interfaces	92
The Rotating Machinery, Mixture Model, Laminar Flow Interface.	92
The Rotating Machinery, Mixture Model, Algebraic γ Plus Interface	94
The Rotating Machinery, Mixture Model, L-VEL Interface	95
The Rotating Machinery, Mixture Model, k - ε Interface.	96
The Rotating Machinery, Mixture Model, Realizable k - ε Interface.	97
The Rotating Machinery, Mixture Model, k - ω Interface	98
The Rotating Machinery, Mixture Model, SST Interface	99
The Rotating Machinery, Mixture Model, Low Re k - ε Interface	100
The Rotating Machinery, Mixture Model, Spalart-Allmaras Interface.	101

The Rotating Machinery, Mixture Model, v2-f Interface	102
Domain, Boundary, Pair, and Point Nodes for the Rotating Machinery, Mixture Model Interfaces	103
Free Surface.	104
Contact Angle	106
Rotating Shaft	106

Chapter 6: Rotating Machinery, Nonisothermal Flow Interfaces

The Rotating Machinery, Nonisothermal Flow, Laminar and Turbulent Flow Interfaces	111
Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface	111
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Algebraic yPlus Interface	112
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, L-VEL Interface.	113
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k- ϵ Interface .	115
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Realizable k- ϵ Interface	116
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k- ω Interface.	117
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, SST Interface.	118
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Low Re k- ϵ Interface.	119
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Spalart-Allmaras Interface	120
Rotating Machinery, Nonisothermal Flow, Turbulent Flow, v2-f Interface.	121
Domain, Boundary, Edge, Point, and Pair Nodes for the Rotating Machinery, Nonisothermal Flow Interfaces.	122

Chapter 7: Rotating Machinery,

Reacting Flow Interfaces

The Rotating Machinery, Reacting Flow, Laminar Flow and Turbulent Flow Interfaces	126
The Rotating Machinery, Reacting Flow, Laminar Flow Interface	126
The Rotating Machinery, Reacting Flow, Turbulent Flow, k-ε Interface. . .	127
The Rotating Machinery, Reacting Flow, Turbulent Flow, k-ω Interface . .	128
The Rotating Machinery, Reacting Flow, Turbulent Flow, SST Interface . .	130
The Rotating Machinery, Reacting Flow, Turbulent Flow, Low Re k-ε Interface.	131
Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery, Reacting Flow Interfaces	132

Introduction

This guide describes the Mixer Module, an optional add-on package for COMSOL Multiphysics® designed to assist you in setting up and solving transport problems in mixers and stirred vessels. The module is an add-on to the CFD Module and provides additional support for modeling fluid flow in rotating machinery.

This chapter introduces you to the capabilities of the module. A summary of the physics interfaces and information about where to find documentation and model examples is also included. This is followed by a brief overview with links to each chapter in the guide. The last section in this introduction contains a tutorial example from the Mixer Module application library.

This chapter contains the following sections:

- [About the Mixer Module](#)
- [Overview of the User's Guide](#)
- [Tutorial Example — Nonisothermal Mixer](#)

About the Mixer Module

The development of new products and processing lines continuously places high demands on mixer design. The Mixer Module provides engineers and scientists with the necessary simulation tools for predicting and understanding the interactions between fluid flow and chemical processes in mixers and stirred vessels. The detailed information gained from computational fluid dynamics (CFD) simulations substantially reduces the number of empirical studies needed in new product and process designs, and has eminent impact on both costs and responses to environmental considerations.

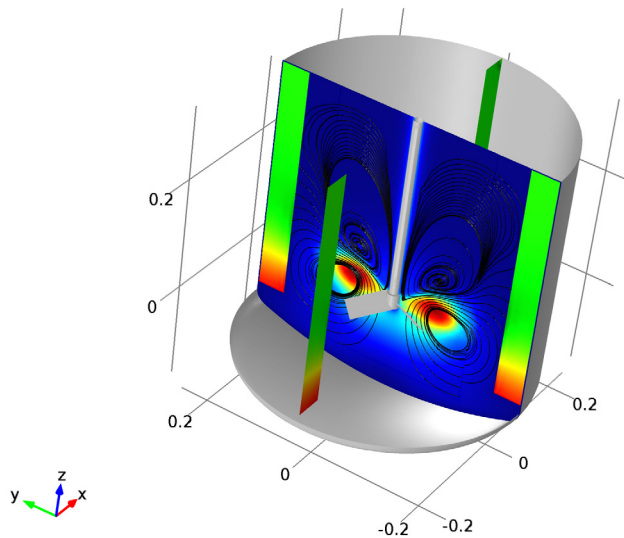


Figure 2-1: Streamlines and velocity magnitude in a cross section of a dished bottom baffled mixer with a four-bladed impeller. The color scale on the baffles represents the pressure.

The Mixer Module complements the CFD Module with additional functionality for the [Rotating Machinery, Fluid Flow](#) branch. The added functionality includes extended capability for modeling turbulence in the Rotating Machinery interfaces.

In order to facilitate fast and efficient setup of mixer geometries, the [Mixer Module Part Library](#) includes predefined geometry components typical of mixer equipment.

The part library includes impeller parts for axial impellers, radial impellers, and impellers designed for highly viscous fluids. In addition to impellers, three types of different tank geometries and a cylindrical impeller shaft geometry are available in the part library. All mixer parts are modularized through a number of input parameters corresponding to important geometrical properties of each part. These can be adjusted in order to fit the mixer system under investigation.

A high rotation rate or a strong acceleration of the rotation rate may induce a substantial deformation of the free surface in an open vessel. This topology change in turn influences the flow pattern inside the vessel. The Mixer Module includes free-surface features to capture the displacement of the liquid-air interface induced by the bulk motion in the domain, by the walls, and by the rotating shaft.

The physics interfaces define a fluid-flow problem using physical quantities such as pressure, flow rate, temperature, and species composition, as well as physical properties, such as viscosity, thermal diffusivity, and density. The different physics interfaces cover a wide range of laminar and turbulent mixer flows. The conservation laws formulated by the physics interfaces are expressed in terms of partial differential equations along with corresponding initial and boundary conditions. The equations are solved by the module using stabilized finite element formulations for fluid flow in combination with damped Newton methods and, for time-dependent problems, in combination with various time-dependent solver algorithms. The Mixer Module's general capabilities include frozen-rotor and time-dependent flows in two- and three-dimensional spaces. For a so-called frozen-rotor flow, the topology relative to the rotating reference frame is fixed ("frozen"). When the flow field is, or can be approximated to be, of this type the computational time (CPU time) can be substantially reduced using the [Frozen Rotor](#) (see the *CFD Module User's Guide*) study type.

The workflow in the Mixer Module is quite straightforward. Set up a simulation using one of the Rotating Machinery interfaces, 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, compute the solution, and visualize the results. All these steps are performed from the COMSOL Desktop. The mesh and solver steps are usually carried out automatically using default settings, which are tuned specifically for each Fluid Flow interface.

The models available in the Mixer Module application library describe the physics interfaces and their features through examples for different types of mixer flows. Here you find examples of industrial equipment and devices, tutorial models for practice,

and benchmark models for verification and validation of the Fluid Flow interfaces. Go to [The Application Libraries Window](#) to access these resources.

To help you get started, this introduction contains a list of the physics interfaces and an example, [Tutorial Example — Nonisothermal Mixer](#), to introduce you to the workflow.

ASPECTS OF MIXER SIMULATIONS

In the initial stages of development for a product, or a new process line, the focus usually lies on qualitative results such as determining whether or not the flow is well mixed, whether a heated reactor is free from hotspots, or whether the flow field contains recirculation zones, which can be inaccessible to reactants. Qualitative results such as these are usually the first step toward creating or improving a design. During the later stages of development, the focus shifts toward scale-up and optimization. For mixing of pseudoplastic slurries, for example, the yield stress has to be overcome everywhere. Bioreactors, on the other hand, should not contain regions of excessive shear. Obtaining accurate estimates for the yield stress, shear or other process parameters such as reaction rates, thermal equilibration times, torque, and energy consumption, provides developers with the edge needed to assume a competitive position within their field.

The choice of which physics to include in your simulations can be based on experimental results, experience, or a dimensional analysis. Excluding relevant physics undoubtedly leads to wrong results, while including “everything” leads to excessive computational time. The Rotating Machinery interfaces help you set up problems of varying complexity. If the mixed quantity is passive, you can use one of the Rotating Machinery, Fluid Flow interfaces to solve for the fluid flow, and then add a Transport of Diluted Species interface to determine other properties, for example, the mixing efficiency. Both reactions and thermal variations affect constitutive quantities, such as the density and viscosity of the fluid. When these effects become appreciable, you can switch to the Rotating Machinery, Reacting Flow interfaces or the Rotating Machinery, Nonisothermal Flow interfaces. When the mixed quantity consists of particles droplets or bubbles, possibly with a different density and response time compared to the surrounding fluid, you can switch to the Rotating Machinery, Multiphase Flow interfaces. Taking it one step further, COMSOL Multiphysics lets you add other physics interfaces to preexisting ones to tailor simulations to your application.

The physics interfaces in the Mixer Module are able to perform all steps in mixer analyses, from the initial idea and qualitative simulations to the final optimization of the product or process.

The Mixer Module Physics Interface Guide

The Rotating Machinery interfaces in this module are based on the laws for conservation of momentum, mass, and energy in fluids. The different physics interfaces contain different combinations and expressions of the conservation laws, which apply to the physics of the flow field being modeled. [Figure 2-2](#) shows the Rotating Machinery interfaces as they are displayed when you add a physics interface (see [Physics Interface Guide by Space Dimension and Study Type](#)). A short description of the various types of physics interfaces follows.

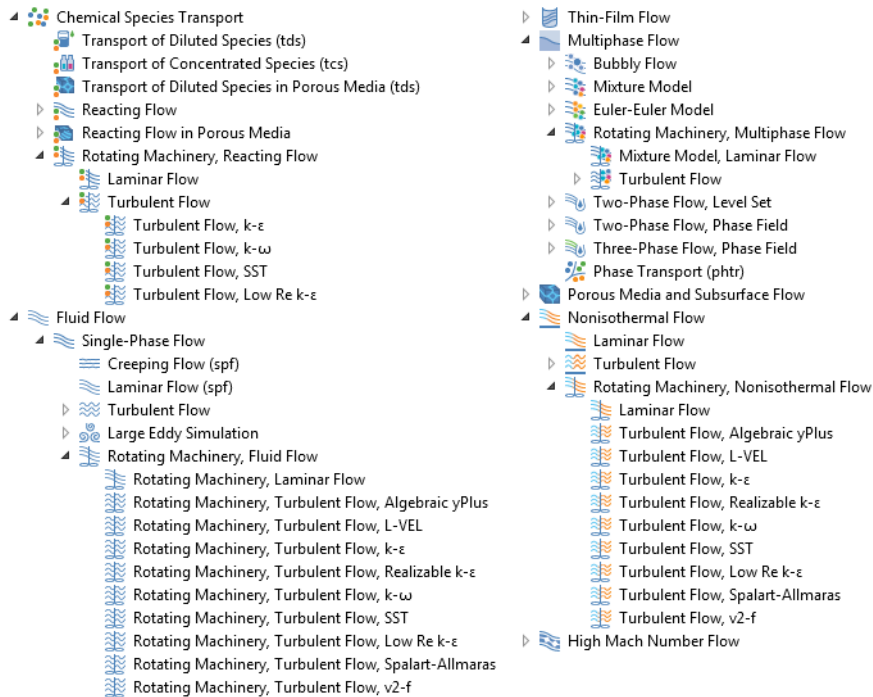




Figure 2-2: The Rotating Machinery interfaces for laminar and turbulent flow.

ROTATING MACHINERY, FLUID FLOW


The **Rotating Machinery, Laminar Flow** interface () is used primarily for modeling flows of low to intermediate Reynolds numbers. This physics interface solves the Navier-Stokes equations for incompressible (default), weakly compressible and compressible flow (up to Mach 0.3). The physics interface is also capable of simulating non-Newtonian fluid flow.

The **Rotating Machinery, Turbulent Flow** interfaces () are used to model flow at high Reynolds numbers. These physics interfaces solve the Reynolds-averaged Navier-Stokes (RANS) equations for the averaged velocity field and averaged pressure. The different physics interfaces in this branch have different models for the turbulent viscosity. There are several turbulence models available — two algebraic turbulence models, the Algebraic yPlus and L-VEL models, and seven transport-equation models, including a standard k - ϵ model, the Realizable k - ϵ model, a k - ω model, a SST model, a low Reynolds number k - ϵ model, the Spalart-Allmaras model, and the v2-f model. Similarly to the Rotating Machinery, Laminar Flow interface, compressibility is set to incompressible 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 turbulence model because it is often a good compromise between accuracy and computational cost (memory and CPU time). The Realizable k - ϵ model is similar to the standard k - ϵ model but has built-in realizability constraints, resulting in improved performance for certain flows, such as turbulent jets. 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, but requires higher resolution in the near-wall region. 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. In the v2-f model, the turbulent viscosity is based on the wall-normal velocity fluctuations, whereby wall


blockage effects and low Reynolds number effects are captured separately. The v2-f model also includes non-local effects of the fluctuating pressure on the turbulent fields.

ROTATING MACHINERY, MULTIPHASE FLOW


The **Rotating Machinery, Mixture Model** interfaces () combine the functionality of the Rotating Machinery, Fluid Flow and Mixture Model interfaces. The interfaces are designed to study the flow of a two-phase mixture, consisting of a continuous phase and a dispersed phase, in rotating and stationary domains. Both phases are assumed to be incompressible. The dispersed phase can be bubbles, liquid droplets, or solid particles, which are assumed to always travel with their terminal velocity through the continuous phase. Physics interfaces for both laminar flow and turbulent flow are available. For turbulent flow the Reynolds-averaged Navier-Stokes (RANS) version of the mixture model equations are solved, and all the turbulent models available for the Rotating Machinery, Fluid Flow interfaces are also available.

ROTATING MACHINERY, NONISOTHERMAL FLOW

The **Rotating Machinery, Nonisothermal Flow, Laminar Flow** interface () is used primarily to model laminar flow where the temperature and flow fields have to be coupled, such as in an externally heated mixer. This multiphysics interface has predefined functionality for coupling heat transfer in fluids and solids. The weakly compressible option is selected by default for the Rotating Machinery Nonisothermal flow interfaces.













The **Rotating Machinery, Nonisothermal Flow, Turbulent Flow** interfaces () solve the Reynolds-averaged Navier-Stokes (RANS) equations together with the equations for heat transfer in fluids and in solids. There is support for all the fluid-flow turbulence models - the Algebraic γ Plus model, the L-VEL model, the standard k - ϵ model, the Realizable k - ϵ model, the k - ω model, the SST model, a low Reynolds number k - ϵ model, the Spalart-Allmaras model, and the v2-f model.












ROTATING MACHINERY, REACTING FLOW













The **Rotating Machinery, Reacting Flow** interfaces () combine the functionality of the Rotating Machinery, Fluid Flow and Transport of Concentrated Species interfaces. The mass and momentum transport in a rotating reacting fluid can be modeled from a single physics interface, and the couplings between the velocity field and mixture density are set up automatically. Physics interfaces for both laminar flow and turbulent flow using the Reynolds-averaged Navier-Stokes (RANS) equations are available. There is support for the following turbulence models - the standard k - ϵ model, the k - ω model, the SST model, and the low Reynolds number k - ϵ model.










PHYSICS INTERFACE GUIDE BY SPACE DIMENSION AND STUDY TYPE

The list below shows the Rotating Machinery interfaces available with the Mixer Module. Because the Mixer Module requires a license for CFD, additional physics interfaces are listed in the *CFD Module User's Guide*.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
 Chemical Species Transport				
 Rotating Machinery, Reacting Flow				
Laminar Flow		—	3D, 2D,	frozen rotor; time dependent
 Turbulent Flow				
Turbulent Flow, k- ϵ		—	3D, 2D	frozen rotor; time dependent
Turbulent Flow, k- ω		—	3D, 2D	frozen rotor; time dependent
Turbulent Flow, SST		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
Turbulent Flow, Low Re k- ϵ		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
 Fluid Flow				
 Single-Phase Flow				
 Rotating Machinery, Fluid Flow				
Rotating Machinery, Laminar Flow		spf	3D, 2D	frozen rotor; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Rotating Machinery, Turbulent Flow, Algebraic γ Plus		spf	3D, 2D	frozen rotor with initialization; time dependent with initialization
Rotating Machinery, Turbulent Flow, L-VEL		spf	3D, 2D	frozen rotor with initialization; time dependent with initialization
Rotating Machinery, Turbulent Flow, k- ϵ		spf	3D, 2D	frozen rotor; time dependent
Rotating Machinery, Turbulent Flow, Realizable k- ϵ		spf	3D, 2D	frozen rotor; time dependent
Rotating Machinery, Turbulent Flow, k- ω		spf	3D, 2D	frozen rotor; time dependent
Rotating Machinery, Turbulent Flow, SST		spf	3D, 2D	frozen rotor with initialization; time dependent with initialization
Rotating Machinery, Turbulent Flow, Low Re k- ϵ		spf	3D, 2D	frozen rotor with initialization; time dependent with initialization
Rotating Machinery, Turbulent Flow, Spalart-Allmaras		spf	3D, 2D	frozen rotor with initialization; time dependent with initialization
Rotating Machinery, Turbulent Flow, v2-f		spf	3D, 2D	frozen rotor with initialization; time dependent with initialization
 <i>Rotating Machinery, Multiphase Flow</i>				
Mixture Model, Laminar Flow		mm	3D, 2D	frozen rotor, time dependent


PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Mixture Model, Algebraic γ Plus		mm	3D, 2D	frozen rotor with initialization; time dependent with initialization
Mixture Model, L-VEL		mm	3D, 2D	frozen rotor with initialization; time dependent with initialization
Mixture Model, k- ϵ		mm	3D, 2D	frozen rotor, time dependent
Mixture Model, Realizable k- ϵ		mm	3D, 2D	frozen rotor, time dependent
Mixture Model, k- ω		mm	3D, 2D	frozen rotor, time dependent
Mixture Model, SST		mm	3D, 2D	frozen rotor with initialization; time dependent with initialization
Mixture Model, Low Re k- ϵ		mm	3D, 2D	frozen rotor with initialization; time dependent with initialization
Mixture Model, Spalart-Allmaras		mm	3D, 2D	frozen rotor with initialization; time dependent with initialization
Mixture Model, v2-f		mm	3D, 2D	frozen rotor with initialization; time dependent with initialization
 Nonisothermal Flow				
 <i>Rotating Machinery, Nonisothermal Flow</i>				
Laminar Flow		—	3D, 2D	frozen rotor; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Turbulent Flow, Algebraic γ Plus		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
Turbulent Flow, L-VEL		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
Turbulent Flow, k- ϵ		—	3D, 2D	frozen rotor; time dependent
Turbulent Flow, Realizable k- ϵ		—	3D, 2D	frozen rotor; time dependent
Turbulent Flow, k- ω		—	3D, 2D	frozen rotor; time dependent
Turbulent Flow, SST		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
Turbulent Flow, Low Re k- ϵ		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
Turbulent Flow, Spalart-Allmaras		—	3D, 2D	frozen rotor with initialization; time dependent with initialization
Turbulent Flow, ν 2-f		—	3D, 2D	frozen rotor with initialization; time dependent with initialization

Where Do I Access the Documentation?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or


context-based) help, and the application libraries are all accessed through the COMSOL Desktop.




	If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.
---	--




THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.



Opening Topic-Based Help




The Help window is useful as it is connected to the features in the COMSOL Desktop. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (). This is called *topic-based* (or *context*) *help*.

	<p>To open the Help window:</p> <ul style="list-style-type: none">• In the Model Builder, Application Builder, or Physics Builder click a node or window and then press F1.• On any toolbar (for example, Home, Definitions, or Geometry), hover the mouse over a button (for example, Add Physics or Build All) and then press F1.• From the File menu, click Help ().• In the upper-right corner of the COMSOL Desktop, click the Help () button.
---	--

	To open the Help window:
	<ul style="list-style-type: none">• In the Model Builder or Physics Builder click a node or window and then press F1.
	<ul style="list-style-type: none">• On the main toolbar, click the Help () button.• From the main menu, select Help>Help.

Opening the Documentation Window

	To open the Documentation window:
	<ul style="list-style-type: none">• Press Ctrl+F1.• From the File menu select Help>Documentation ().

	To open the Documentation window:
	<ul style="list-style-type: none">• Press Ctrl+F1.
	<ul style="list-style-type: none">• On the main toolbar, click the Documentation () button.• From the main menu, select Help>Documentation.

THE APPLICATION LIBRARIES WINDOW







Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or app. The models and applications are available in COMSOL Multiphysics as MPH files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the model or application and its properties, including options to open it or its associated PDF document.

	The Application Libraries Window in the <i>COMSOL Multiphysics Reference Manual</i> .
---	---

Opening the Application Libraries Window

To open the **Application Libraries** window ():

	<ul style="list-style-type: none">• From the Home toolbar, Windows menu, click () Applications Libraries.• From the File menu select Application Libraries. <p>To include the latest versions of model examples, from the File>Help menu, select () Update COMSOL Application Library.</p>
	<p>Select Application Libraries from the main File> or Windows> menus.</p> <p>To include the latest versions of model examples, from the Help menu select () Update COMSOL Application Library.</p>
	

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

COMSOL ACCESS AND TECHNICAL SUPPORT

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number are sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
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

Overview of the User's Guide

The *Mixer Module User's Guide* gets you started with modeling using COMSOL Multiphysics and provides information specific to this module. Further instructions on using COMSOL in general are found in the *COMSOL Multiphysics Reference Manual*.



As detailed in the section [Where Do I Access the Documentation?](#) this information can also be found from the COMSOL Multiphysics software **Help** menu.

For general information about setting up and solving CFD applications, see the *CFD Module User's Guide*.

The last section in this chapter features a model you can access from [The Application Libraries Window](#). The [Tutorial Example — Nonisothermal Mixer](#) solves a mixer-flow problem using the Laminar Flow interface in the Rotating Machinery, Nonisothermal Flow branch.

The following chapter documents the [Mixer Module Part Library](#). It includes an overview of the all included impellers and tanks.

The third chapter introduces you to the [Theory for the Free Surface Features](#). It includes descriptions of the mesh deformation within the free surface domain and the conditions that need to be satisfied at a free surface, at a three-phase boundary, and on a rotating shaft within the free surface domain.

The remaining chapters describe the physics interfaces and features under the [Rotating Machinery, Fluid Flow](#) branch, and the additional branches exclusive to the Mixer Module: [Rotating Machinery, Multiphase Flow Interfaces](#), [Rotating Machinery, Nonisothermal Flow Interfaces](#), and [Rotating Machinery, Reacting Flow Interfaces](#).

Tutorial Example — Nonisothermal Mixer

This model demonstrates how to obtain the temperature distribution in a simplified tabletop lab mixer using the **Rotating Machinery, Nonisothermal Flow** branch in the Mixer Module. The key instructive element is a demonstration of the **Frozen Rotor** method, which substantially reduces the computational time for a mixing study.

The model geometry is shown in Figure 2-3. It represents a cross-section of a tabletop lab mixer. Small mixers are not always baffled. Instead, one or several rods are inserted through the mixer lid and function as baffles. The rods typically contain equipment to measure, for example, the temperature or pH level. In this model, the mixer is baffled by a simplified immersion heater, which has a constant surface temperature of 60°C.

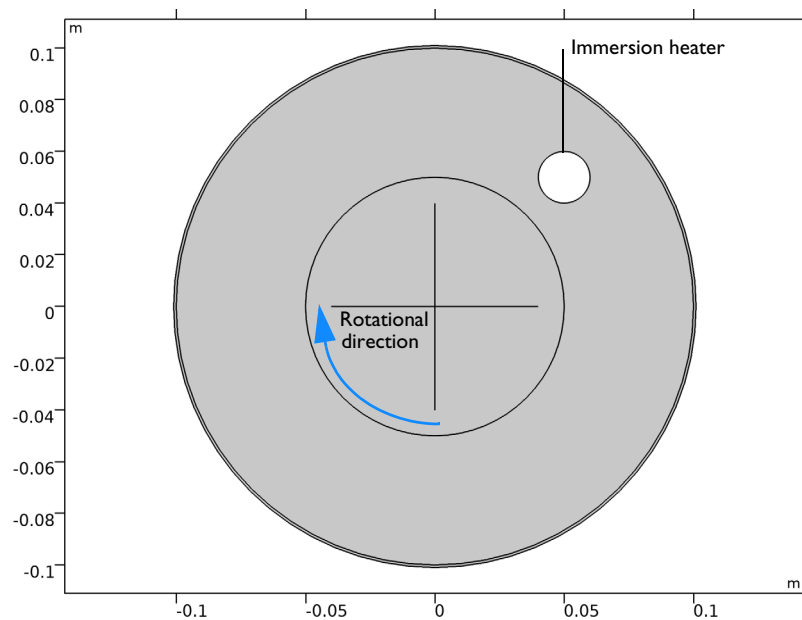


Figure 2-3: Tabletop lab mixer geometry, showing the rotational direction of the impeller and the location of the immersion heater.

The mixer tank is filled with water which is agitated by the impeller rotating clockwise (as indicated in [Figure 2-3](#)) with 20 revolutions per minute (rpm). In this model the thickness of the impeller blades is significantly smaller than the diameter of the tank, and the impeller blades are therefore modeled as infinitely thin.

The tank is made of steel and is subjected to cooling by natural convection on the outside of the mixer vessel. The surrounding conditions corresponds to a temperature of 20°C and a pressure of 1 atmosphere. The total height of the reactor is 0.2 m. These conditions are needed as input for the natural convection correlations, which are used to calculate the heat transfer coefficient from the tank wall to the surroundings.

MODEL SETUP

The Reynolds number for a mixer is commonly calculated as

$$\text{Re} = \frac{ND_a^2}{\nu} \quad (2-1)$$

where N is the number of rotations per second, D_a the impeller diameter, and ν the kinematic viscosity. A high Reynolds number means that the flow has a tendency to become turbulent. Evaluating [Equation 2-1](#) using ν at 60 °C gives $\text{Re} = 6944$. This Reynolds number indicates that the flow is at least partly turbulent. For simplicity in this instructional model, the flow is assumed to be two-dimensional and no turbulence model is used. Possible extensions of the model includes to resolve it using the full three dimensional geometry, and also to apply a turbulence model to investigate the effect of turbulent structures occurring in the flow.

The objective of this model is to obtain the temperature distribution at operating conditions. One way to get there would be to start from zero velocity and a homogeneous temperature distribution and to simulate the start-up of the mixer. This approach is simple, but requires a relatively long computation time.

A computationally more efficient method is to first simulate the flow using the frozen-rotor approach. The frozen-rotor approach is a modeling concept that treats the rotor as fixed, or frozen in space. The flow in the rotating domain is assumed to be stationary in terms of a rotating coordinate system. The effect of the rotation is then accounted for by the Coriolis and centrifugal forces. The flow in the non-rotating parts is also assumed to be stationary, but in a non-rotating coordinate system. (See [Frozen Rotor](#) in the *CFD Module User's Guide* for more information.) The result of a frozen-rotor simulation is an approximation to the flow induced by the impeller. The result depends on the angular position of the impeller and cannot represent transient effects. However, it is still a very good starting point to reach operating conditions.

Here the frozen-rotor result is used as input to a time-dependent simulation and the progress towards the operating conditions is monitored using probe plots.

RESULTS AND DISCUSSION

Figure 2-4 shows the velocity distribution obtained from the frozen-rotor simulation. As expected, the highest velocity magnitude is found at the tip of the mixer blades. Three recirculation zones can be identified: one downstream of the immersion heater, one along the top wall, and one along the bottom wall.

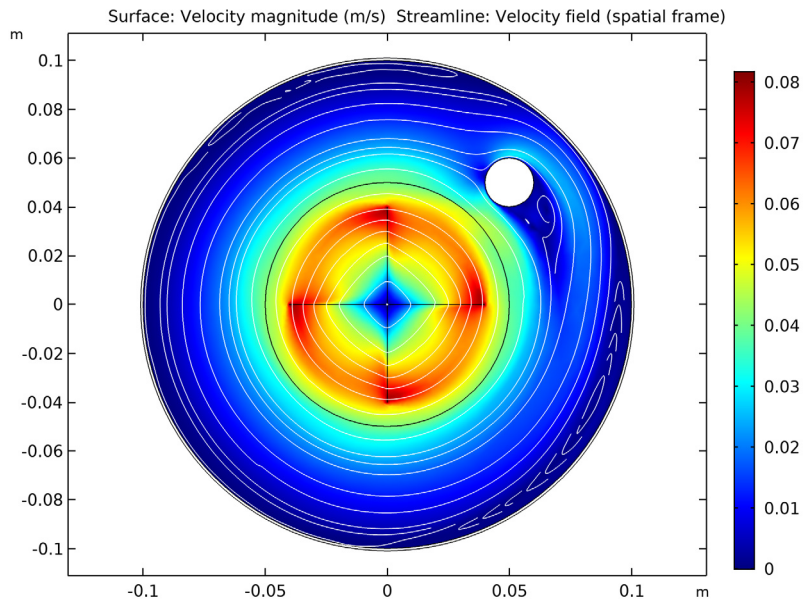


Figure 2-4: Velocity field obtained from the frozen rotor simulation.

Figure 2-5 shows the temperature distribution obtained from the frozen-rotor simulation. Streamlines are also included to visualize the flow field. The temperature is relatively homogeneous throughout the mixer. There are some cold spots in connection to the recirculation zones adjacent to the outer wall. This is expected because the fluid there has a longer residence time close to the solid wall, and therefore has less contact with the heated fluid closer to the center of the mixer.

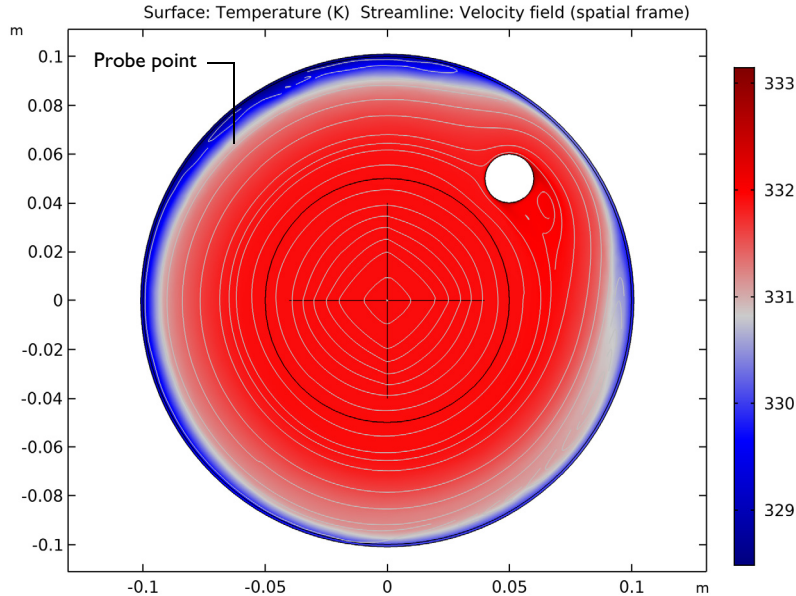


Figure 2-5: Temperature distribution obtained from the frozen-rotor simulation.

The progress of a solution can be monitored using probes (see [Probes](#) in the *COMSOL Multiphysics Reference Manual*). The velocity magnitude and temperature are probed at $(x,y) = (-0.05, 0.065)$. The location is indicated in [Figure 2-5](#), just outside the recirculation zone along the top wall.

The probe plots produced during the time-dependent simulation are shown in [Figure 2-6](#). The velocity probe plot shows that the flow pattern, after an initial transient, oscillates around the frozen-rotor result with an amplitude of about 10%. The deviations in temperature are much smaller.

The velocity probe plot clearly shows the frequency that corresponds to the passing of the blades (0.75 Hz) and an additional frequency (about 0.25 Hz) related to the intermittent boundary-layer separation. Evaluating the frequency from the Strouhal number for the immersion heater gives a value of about 0.25 Hz, suggesting that the boundary-layer separation may be induced by the Kármán vortex street behind the heater. The temperature value does not display the same frequencies and instead changes more smoothly.

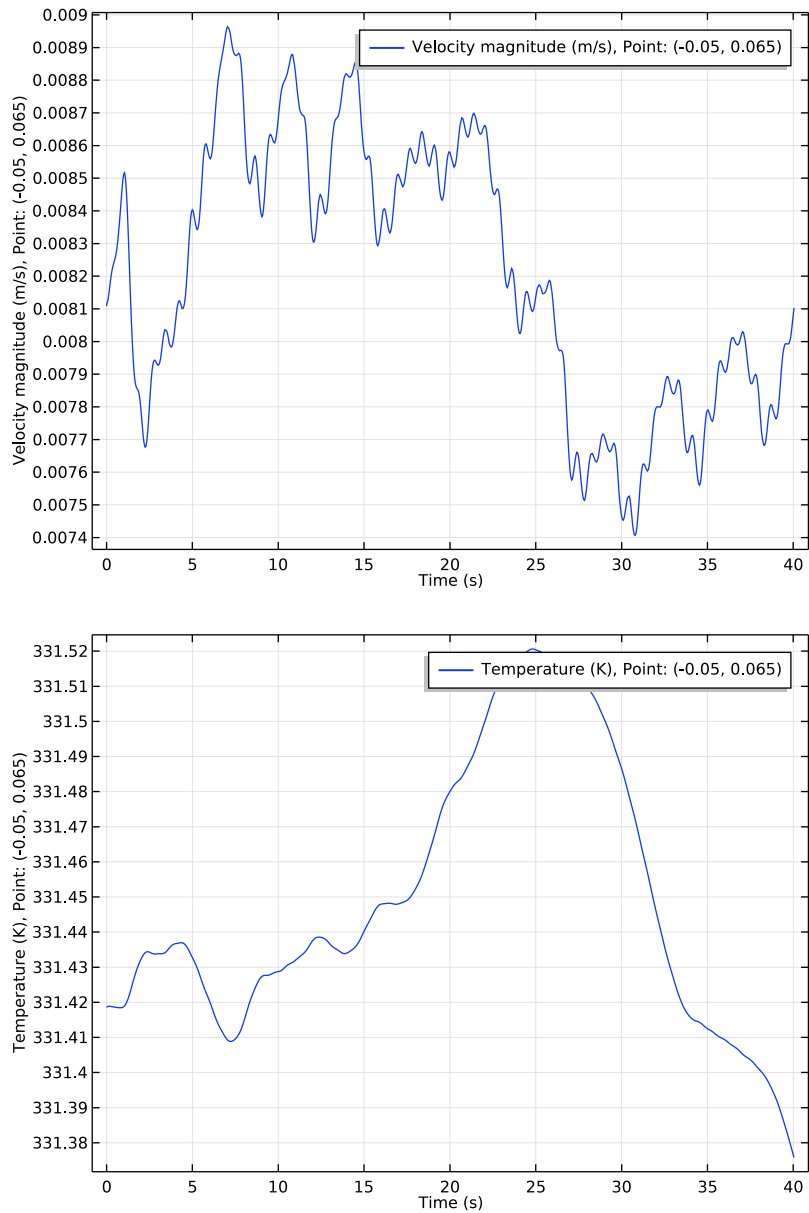


Figure 2-6: Probe plots of velocity and temperature in the mixer from the time-dependent simulation.

A more complete picture of the progress from the frozen-rotor solution toward the operating conditions can be seen through an animation. Figure 2-7 shows four snapshots from such an animation. The time runs from top left to bottom right. The most notable changes occur in the recirculation zones. The recirculation zone behind the immersion heater has two vortices which appear to be oscillating. This has no effect on the local temperature, but it significantly influences the size of the recirculation zones adjacent to the outer wall.

Looking at Figure 2-5, it can be seen that the recirculation zone along the top wall contains a single, large vortex. As the simulation progresses ($t = 20$ s to $t = 40$ s), the size and strength and position of the vortices along the top wall varies as a result of the interaction between the disturbance, produced by the immersion heater, and the outer wall.

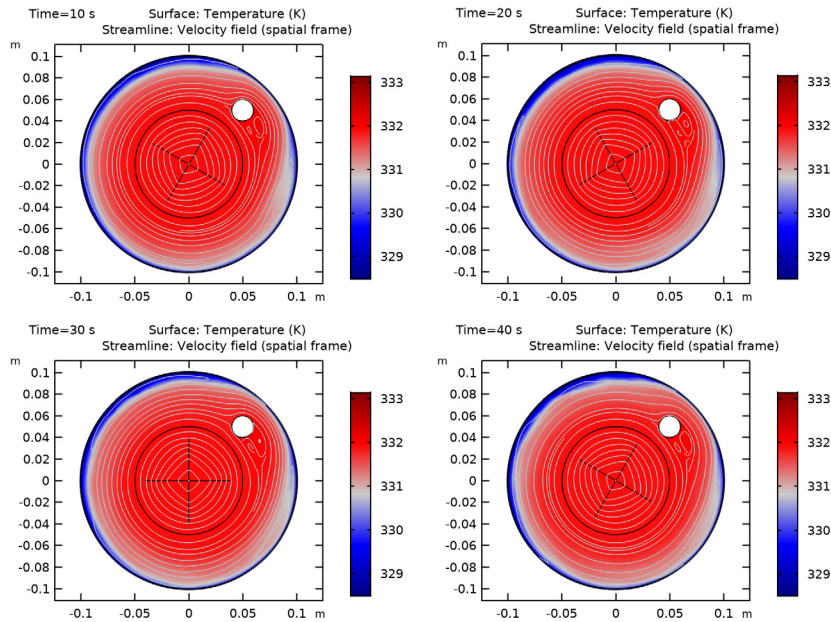


Figure 2-7: Evolution of the temperature field from frozen rotor solution toward operating conditions.







The results obtained in this model are typical for rotating-machinery models: The frozen-rotor approach can, with minimal computational effort, deliver a decent approximation of the flow and temperature fields. But transient effects can only be

captured with a time-dependent simulation, and these effects can change local temperature and velocity values significantly.

The remainder of this section consists of step-by-step instructions on setting up, solving, and analyzing the model using both analysis types.




MODEL WIZARD

The first step to build a model is to open COMSOL Multiphysics, then select the physics interface and specify the type of analysis you want to do — in this case, a frozen-rotor analysis using a Rotating Machinery, Nonisothermal Laminar Flow interface.


- 1 Open COMSOL Multiphysics. On the **New** page, click **Model Wizard** . Then click the 2D button .
- 2 In the **Select Physics** tree, under **Fluid Flow>Nonisothermal Flow>Rotating Machinery**, **Nonisothermal Flow** click **Laminar Flow** .
- 3 Click the **Add** button.
- 4 Click the **Study** button .
- 5 In the tree, select **Preset Studies for Selected Physics Interfaces>Frozen Rotor** .
- 6 Click the **Done** button .

CREATE THE GEOMETRY

To simplify this step, insert a prepared geometry sequence.

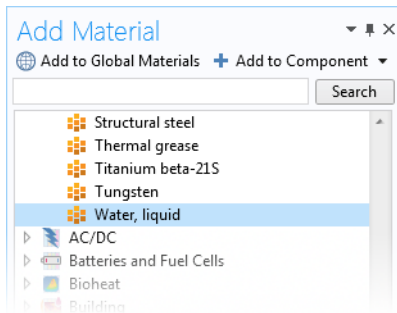
- 1 On the **Geometry** toolbar, select **Insert Sequence** .
- 2 Browse to the module's applications folder and double-click the file `nonisothermal_mixer.mph`.
- 3 Go to the **Home** toolbar and select **Build All** . The Home toolbar refers to the specific set of controls near the top of the Desktop.
- 4 Click the **Zoom Extents** button  on the **Graphics** toolbar.

MATERIALS

On the **Home** toolbar, click **Add Material** .

Water, liquid


- 1 Go to the **Add Material** window.

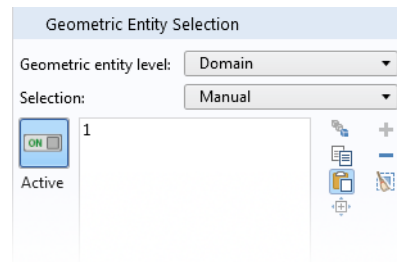
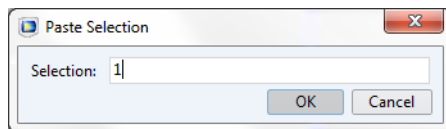


- 2 In the tree, select **Built-In>Water, liquid**.
- 3 In the **Add Material** window, click **Add to Component** + .

Steel AISI 4340


- 1 Go to the Add Material window.
- 2 In the tree, select **Built-In>Steel AISI 4340**.
- 3 In the **Add Material** window, click **Add to Component** + .
- 4 In the **Model Builder**, under **Component 1 (comp1)>Materials** click **Steel AISI 4340**.
- 5 In the **Home** toolbar, click **Add Material** again to close the **Add Material** window.
- 6 Select Domain 1, the circular outer rim of the mixer, only.

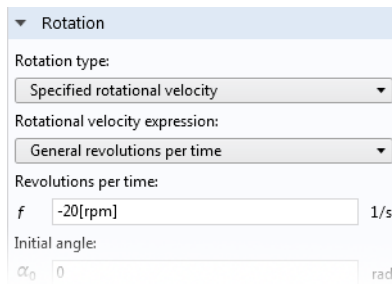
There are many ways to select geometric entities. When you know the domain to add, such as in this exercise, you can click the Paste Selection button  and enter the information in the Selection text field. In this example, enter 1 in the Paste Selection window. For more information about selecting geometric entities in the Graphics window, see the *COMSOL Multiphysics Reference Manual*.




DEFINITIONS

Rotating Domain 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Definitions**>**Moving Mesh** click **Rotating Domain 1**.
- 2 In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.
- 3 Click **Remove from Selection** . Only domain 3, the impeller domain, is selected.
- 4 In the **Settings** window for **Rotating Domain** locate the **Rotation** section.
- 5 In the **Revolutions per time** text field, type -20[rpm].



LAMINAR FLOW (SPF)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
- 2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.
- 3 In the list, select **1**.
- 4 Click **Remove from Selection** . Only domains 2 and 3, the fluid domains, are selected.

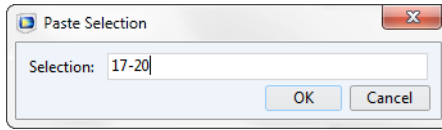
Flow Continuity 1

- 1 In the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Flow Continuity** .
- 2 In the **Settings** window for **Flow Continuity**, locate the **Pair Selection** section.
- 3 In the **Pairs** list, select **Identity Boundary Pair 1 (ap1)**.


Interior Wall 1

- 1 In the **Physics** toolbar, click **Boundaries** and choose **Interior Wall** .

- 2 Select Boundaries 17–20 only, corresponding to the impeller blades.




Pressure Point Constraint

- 1 On the **Physics** toolbar, click **Points** and select **Pressure Point Constraint** .
- 2 Select Point 5 only. This is the bottommost point in the fluid domain of the mixer.


HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.


Solid

- 1 In the **Physics** toolbar, click **Domains** and choose **Solid** .
- 2 Select Domain 1, the mixer vessel wall, only.


Continuity

- 1 In the **Physics** toolbar, in the **Boundary** section, click **Pairs** and choose **Continuity** .
- 2 In the **Settings** window for **Continuity**, locate the **Pair Selection** section.
- 3 In the **Pairs** list, select **Identity Boundary Pair 1 (ap1)**.


Temperature

- 1 In the **Physics** toolbar, click **Boundaries** and select **Temperature** .
- 2 Select Boundaries 13–16, which make up the boundary of the immersion heater.
- 3 In the **Settings** window for **Temperature**, locate the **Temperature** section.
- 4 In the T_0 text field, type 60[degC].



Heat Flux

- 1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux** .
- 2 Select Boundaries 1, 2, 7, and 12, which make up the circular outer boundary of the mixer vessel.
- 3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
- 4 Click the **Convective heat flux** button.
- 5 From the **Heat transfer coefficient** list, choose **External natural convection**.
- 6 In the L text field, type 0.2[m].

MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1** .
- 2 In the **Settings** window for **Mesh** from the **Element size** list, select **Fine**.

The default mesh generates isotropic mesh elements across the thin vessel wall. Edit the Physics-controlled mesh settings to increase the element size in the circumferential direction of the solid domain.


- 1 Go to the **Mesh** toolbar and click **Edit** .
- 2 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size** .
- 3 In the **Settings** window for **Size**, locate the **Element Size** section.
- 4 From the **Predefined** list, choose **Coarse**.

Free Triangular 1


- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1** .

Now, add a separate **Size** node with reduced resolution in narrow regions.

Size 1


- 1 Right-click **Component 1 (comp1)>Mesh 1>Free Triangular 1** and choose **Size** .
- 2 In the **Settings** window for **Size 1**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select **Domain 1**, the vessel wall, only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the **Element Size Parameters** section. Select the **Resolution of narrow regions** check box.
- 7 In the associated text field, type 0.1.


Build the **Free Triangular 1** mesh node in order to inspect the mesh at this point.

- 8 In the **Settings** window for **Size 1**, click the **Build Selected** button .

Now add the mesh boundary layers as well. Use boundary layer mesh also in the solid domain to increase the resolution there.


Boundary Layers 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Boundary Layers 1** .
- 2 In the **Settings** window for **Boundary Layers**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **All domains**.

- 4 In the **Settings** window for Boundary Layers 1, click the **Build Selected** button  .

Now that the physics is defined and the geometry is meshed, solve the model using the frozen rotor approach.



STUDY 1

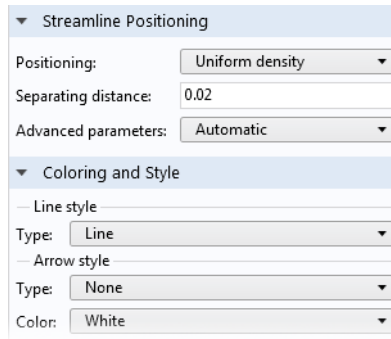
In the **Home** toolbar, click **Compute**  .



RESULTS

Re-create [Figure 2-4](#) using the following steps.

Velocity (spf)


- 1 In the **Model Builder** window, under **Results** , click the **Velocity (spf)** node.
- 1 Right-click **Velocity (spf)** and choose **Streamline** .
- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.





- 3 From the **Positioning** list, choose **Uniform density**.
- 4 In the **Separating distance** text field, type 0.02.
- 5 Locate the **Coloring and Style** section. Find the **Arrow style** subsection. From the **Color** list, choose **White**.
- 6 In the **Velocity (spf)** toolbar, click **Plot**  .
- 7 Click the **Zoom Extents** button  in the **Graphics** toolbar.

[Figure 2-5](#) can be created by the following steps.

Temperature (ht)


- 1 In the **Model Builder** window, expand the **Results>Temperature (ht)** node, then click **Surface** .
- 2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

- 3 From the **Color table** list, choose **Wave**.
- 4 Go to the **Temperature (ht)** toolbar and select **Streamline** .
- 5 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 6 From the **Positioning** list, choose **Uniform density**.
- 7 In the **Separating distance** text field, type 0.02.
- 8 Locate the **Coloring and Style** section. Find the **Arrow style** subsection. From the **Color** list, choose **Gray**.
- 9 In the **Temperature (ht)** toolbar, click **Plot** .


Add a probe to follow the development of the flow during the time-dependent simulation.

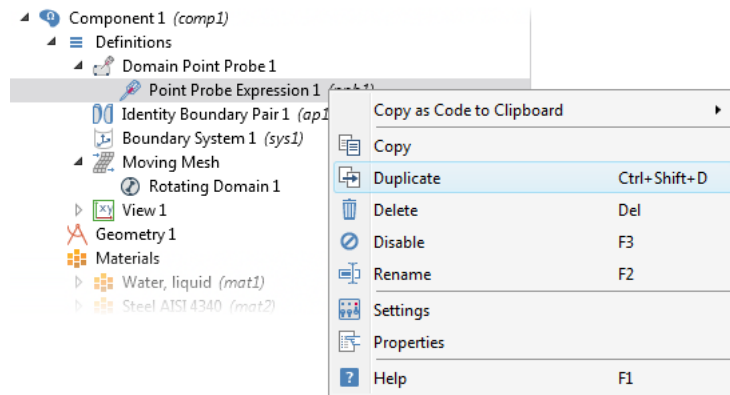
DEFINITIONS

Domain Point Probe 1

- 1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe** .
- 2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.
- 3 In the **Coordinates** text field, set **x** to -0.05[m].
- 4 In the **Coordinates** text field, set **y** to 0.065[m].

The probe is located at the outer edge of the recirculation zone that is positioned along the upper wall. By default the flow field velocity magnitude is tracked by the probe.

- 5 In the **Model Builder** window, expand the **Domain Point Probe 1** node .



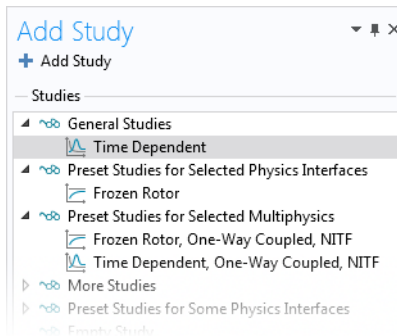
- 6 Right-click **Point Probe Expression 1 (ppb1)**  and choose **Duplicate**.




- 7 In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.
- 8 In the **Expression** text field, type T , in order to track the temperature variable.
- 9 Click to expand the **Table and Window Settings** section. From the **Plot window** list, choose **New window**.

Add a **Time Dependent** study in order to perform a transient simulation, using the previous solution as initial condition.

ADD STUDY


- 1 In the **Home** toolbar, click **Add Study**  to open the **Add Study** window.
- 2 Go to the **Add Study** window.



- 3 In the tree, select **General Studies>Time Dependent** .
- 4 In the **Add Study** window, click **Add Study** .
- 5 In the **Home** toolbar, click **Add Study**  again to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

- 1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent** .
- 2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 3 In the **Times** text field, type $\text{range}(0, 0.5, 40)$.



This means that the solution will be computed from $t = 0$ to 40 s, and the solution will be stored every 0.5 seconds.


Now, apply the solution from the previous frozen rotor analysis as initial condition for the transient simulation.

- 4 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 5 From the **Method** list, choose **Solution**.
- 6 From the **Study** list, choose **Study 1, Frozen Rotor**.

For the transient simulation, add a restriction on the time step. This will make sure that the impeller rotation in each time step is bounded, and that a high accuracy is maintained throughout the simulation. First generate the solver sequence.

Solution 2

- 1 In the **Study** toolbar, click **Show Default Solver** .
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1** .



Apply a maximum time step of 0.05 s. This is equivalent to an impeller rotation of 6°.
- 3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
- 4 From the **Maximum step constraint** list, choose **Constant**.
- 5 In the **Maximum step** text field, type 0.05.
- 6 In the **Study** toolbar, click **Compute** .

Two probe plots are automatically displayed when you start the calculation.

RESULTS



The following steps create an animation that contains the plots in [Figure 2-7](#).

Temperature (ht) 1


- 1 In the **Model Builder** window, under **Results** , click **Temperature (ht) 1** .
- 2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

Plot the data set edges in the spatial frame to make them follow the rotation.
- 3 From the **Frame** list, choose **Spatial (x, y, z)**.




Surface

- 1 In the **Model Builder** window, expand the **Temperature (ht) 1**  node, then click **Surface** .
- 2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 3 From the **Color table** list, choose **Wave**.

Streamline I

- 1 Go to the **Temperature (ht) I** toolbar and choose **Streamline** .
- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 3 From the **Positioning** list, choose **Uniform density**.
- 4 In the **Separating distance** text field, type 0.02.
- 5 Locate the **Coloring and Style** section. Find the **Arrow style** subsection. From the **Color** list, choose **Gray**.

Animation I

- 1 Go to the **Results** toolbar, click **Animation**  and choose **File** .
- 2 In the **Settings** window for **Animation**, locate the **Target** section.
- 3 From the **Target** list, choose **Player**.
- 4 Locate the **Scene** section. From the **Subject** list, choose **Temperature (ht) I**.
- 5 Locate the **Frames** section. From the **Frame selection** list, choose **All**.
- 6 Locate the **Playing** section. In the **Display each frame for** text field, type 0.25.
- 7 In the **Graphics** window, click **Play** .

Mixer Module Part Library

In this chapter:

- [Part Library Contents](#)
- [Tank Parts](#)
- [Impeller Parts](#)

Part Library Contents

The dedicated part library included in the Mixer Module contains geometrical parts typical of mixer and stirred vessel equipment. The included parts consists of impellers, tanks, and a straight cylindrical shaft. When accessing the library, the parts are sorted under the categories seen in the figure below

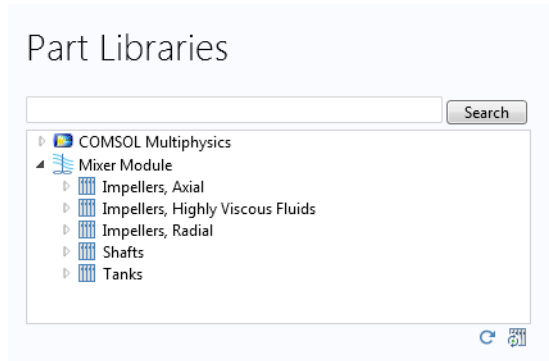


Figure 3-1: The Mixer Module Part Library

The dimensions and configuration of the different parts can be controlled to a great detail using the input parameters included in each part. For example, adding a pitched impeller part instance, you can cut different parts of the impeller, round its edges, define the pitch angle, and define asymmetrical blades, which are wider above the rotation plane than below. Also several impellers can be added, and also rotate them slightly around the rotational axis in order to evenly distribute the blades in the circumferential direction.

Tank Parts

The Mixer Module Part Library includes three different cylindrical tanks: a dished bottom tank, a flat bottom tank, and a cone bottom tank. The three tank types are shown in the figure below:

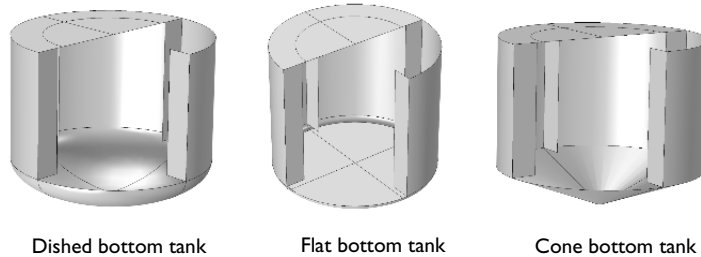


Figure 3-2: Tank parts available in the Mixer Module Part Library.

All tank parts can be defined to include an optional number of baffles in order to improve mixing. Baffles are specially required when radial impellers are used, in order to avoid a near solid body rotation of the fluid.

By default, the tank's cylindrical part is positioned with its base at $z = 0$. This implies that the dished and cone bottom sections are positioned just below the xy -plane. If you define a flat bottom tank with a fillet radius on the lower cylinder edge, then also the fillet section created is placed just below the xy -plane. If you specify a zero fillet radius for a flat bottom tank, then the base of the tank is positioned in the xy -plane, that is, at $z = 0$.

The impeller shaft's lowest part is also by default positioned at $z = 0$. You can change the position, length, and diameter of the impeller shaft. However, the impeller shaft is always placed in the middle of the tank perpendicular to the xy -plane.

Impeller Parts

As seen in [Figure 3-1](#), the impeller parts are sorted into three categories, depending on the principal direction of the flow generated by the impeller rotation (axial or radial flow), or if the impeller type is used to mix highly viscous fluids. In total, the Mixer Module Part Library includes eleven impellers: five axial impellers, four radial impellers, and two impellers for highly viscous fluids.

AXIAL AND RADIAL IMPELLERS

Axial impellers give the flow an axial component naturally achieving mixing along the z-axis of the mixer. This type of mixers is appropriate for fluids that are sensitive to high shear rates. For example, fermentation processes have living cells in the reactor solution, cells that would be killed by high shear rates. In these processes, axial impellers are often selected and baffles are usually omitted.

Radial impellers force the flow in the radial direction and only give an axial component once the flow hits the walls of the vessel. In order to achieve good mixing, these impellers rely on high shear rates and the presence of baffles, that disrupts tangential flows that would lead to poor mixing.

The figure below shows an example of an axial and a radial impeller

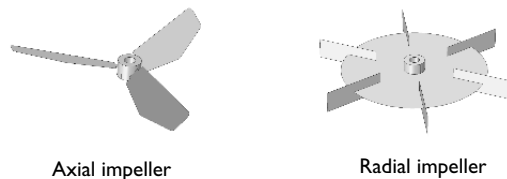


Figure 3-3: Examples of an axial impeller (pitched impeller) and a radial impeller (Rushton turbine).

PITCHED BLADE IMPELLER

The most common axial impeller is the pitched blade impeller; see [Figure 3-4](#) below. This impeller can be configured for rectangular blades or for isosceles trapezoid-shaped blades. These blade shapes can be created with wider outer or inner edges.

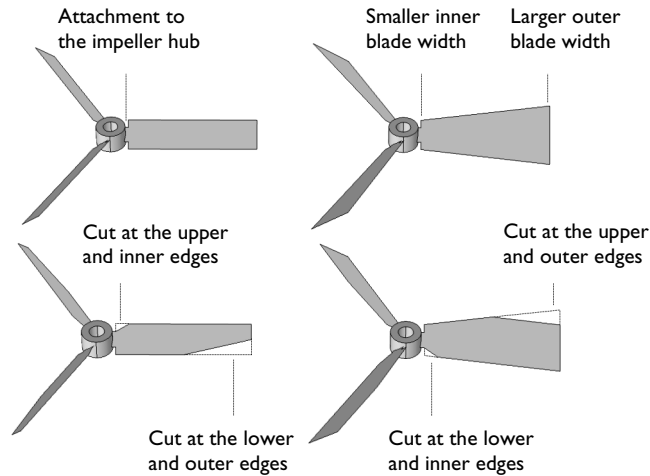


Figure 3-4: Possible configuration of impeller blades for symmetrical blades.

In addition to the symmetrical blades, asymmetrical blades can also be created. In order to create asymmetrical blades, the upper part of the blade is made wider than the lower part (see [Figure 3-5](#)). The impeller can then be flipped vertically to have the wider part of the blade facing downward. The pitch angle of the impeller blades can

be varied between 0 and 90 degrees. The outer vertices of the outer edges can also be rounded using fillets.

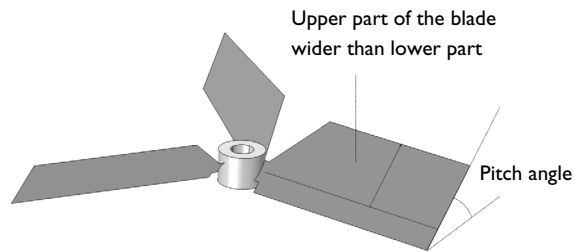


Figure 3-5: Asymmetrical blade. The upper part of the blade is wider than the lower part.

The figure below contains the notations used for the pitched impeller in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

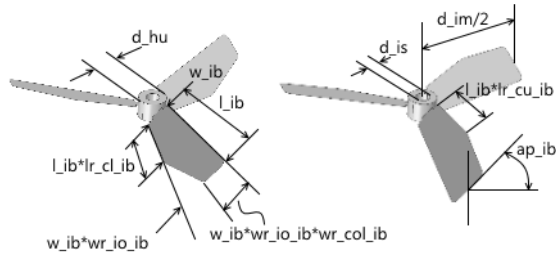


Figure 3-6: Notations used for the pitched impeller.

PITCHED IMPELLER WITH BENT BLADES

Bent blades may be used to decrease the pitch at the outer part (away from the hub) of the impeller blade. Bending these parts gives a less aggressive pitch angle at the outer edges. Using the Mixer Module, the bending is restricted to the upper and outer edges of the impeller blades. The upper part can also be made wider than the lower part to

obtain an asymmetrical blade. However, you can flip the impeller vertically to get a bend or wider part that faces downward.

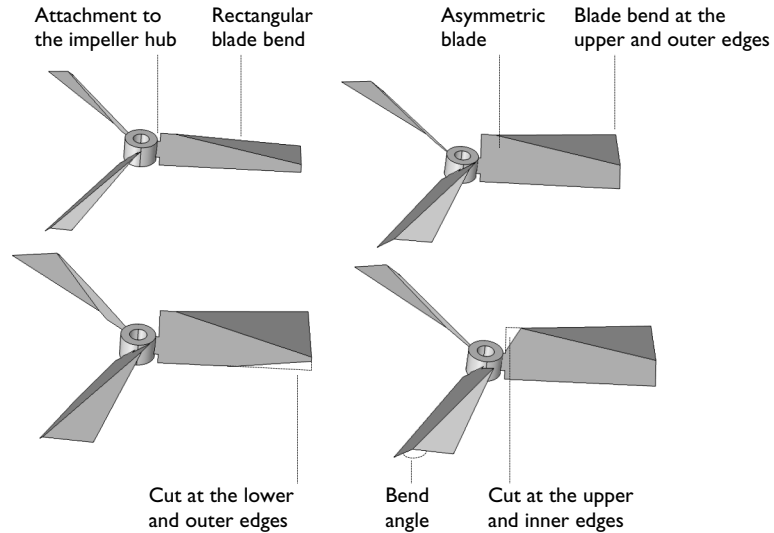


Figure 3-7: Available blade bend and cut operations.

The vertices of the outer vertical edge can be rounded using fillets. The figure below shows an impeller with asymmetrical blades and rounded vertices.

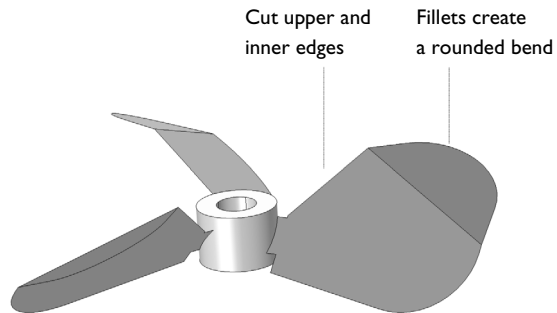


Figure 3-8: Asymmetrical impeller blades with rounded outer (away from the hub) vertices.

The figure below contains the notations used for the pitched impeller with bent blades in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

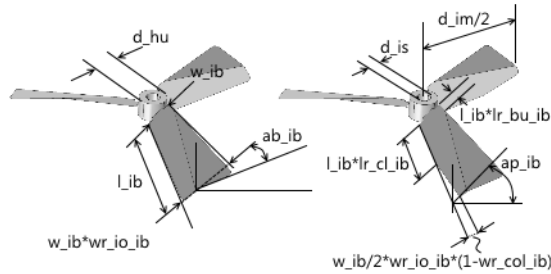


Figure 3-9: Notations used for the pitched impeller with bent blades.

PITCHED BLADE IMPELLER WITH CONSTANT PITCH

For regular pitched impellers, the pitch increases with the radius of the impeller blades, since the blade parts travel with increasing velocity with increasing radius. This means that if the impeller rotated in a fluid, but was allowed to move freely in the axial direction, the outer edge of a regular pitched impeller would want to travel at a higher axial velocity than the inner edge. This effect subjects the impeller blade to a high bending moment. The pitched impeller with constant pitch adapts the pitch angle with the increasing radius so that the outer and inner edges get the same axial velocity as the impeller rotates.

As for the regular pitched impellers, the pitched impeller with constant pitch can be designed with isosceles trapezoid-shaped blade projections (the blades are not flat).

The blades can also be cut in both upper and lower edges to achieve different impeller configurations.

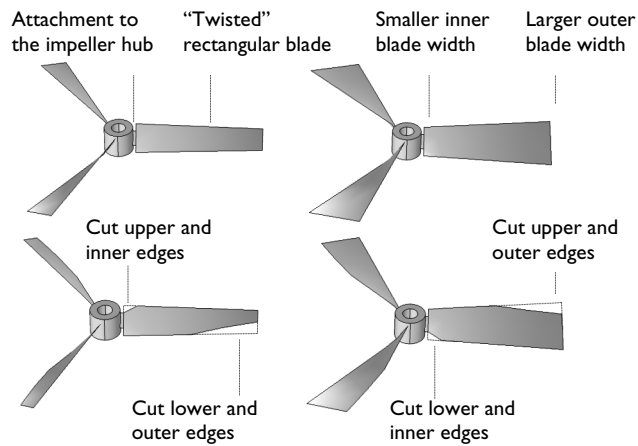


Figure 3-10: Pitched impellers with constant pitch with different blade shapes.

The inner pitch angle can be varied in order to change the axial and radial components of the flow induced by the impeller. The pitch angle is then automatically calculated to give a constant pitch.

The outer vertices of the blades can be rounded, which gives additional freedom in the design of the impeller blades. The figure below shows a propeller created by using fillets and cuts and applying those on a pitched impeller with constant pitch.

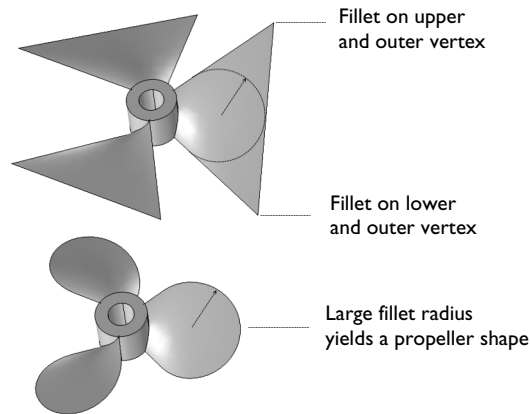


Figure 3-11: Impeller of propeller type created using isosceles trapezoid-shaped blade projections and large fillet radii for the outer vertices of the blades.

The figure below contains the notations used for the pitched impeller with constant pitch in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

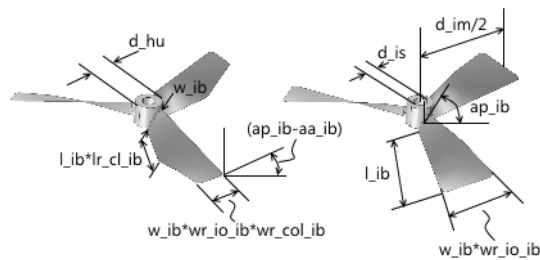


Figure 3-12: Notations used for the pitched impeller with constant pitch.

HYDROFOIL IMPELLER

Impellers with blades that are curved along the vertical edges are denoted hydrofoil impellers. These impellers are similar to the pitched blade impellers, but give an additional pressure difference across the impeller blade with an additionally lowered pressure on the convex side of the blade.

In addition to varying the curvature radius of the blades, the shape of the impeller blade projections can be varied to get the same freedom in the design of hydrofoil impellers as for the case of the pitched impeller mentioned above. This implies that isosceles trapezoid-shaped blade projections, asymmetrical blades with a wider upper part, and cut blades can be used to create the desired design. Fillets make it possible to create impellers of propeller type.

As in the case for the pitched impeller type, the pitch angle can be varied to control the relation between the radial and axial flow created as the impeller rotates.

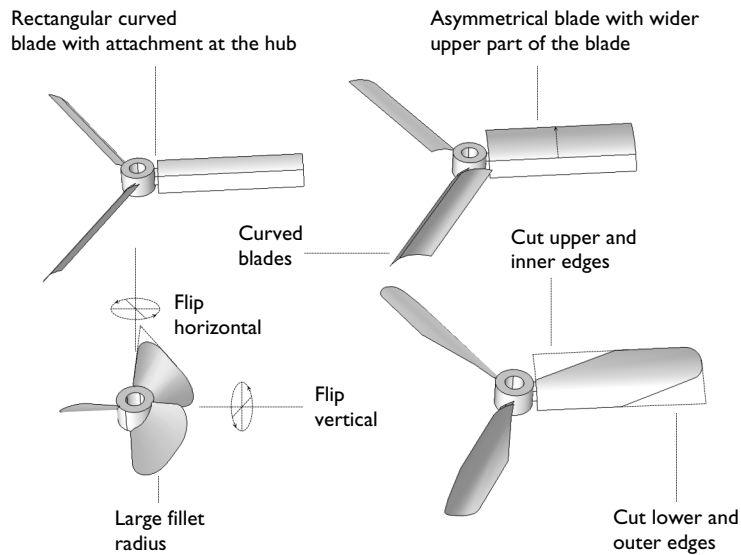


Figure 3-13: The curved blades can be shaped to create a large variety of impeller designs.

The figure below contains the notations used for the hydrofoil impeller in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

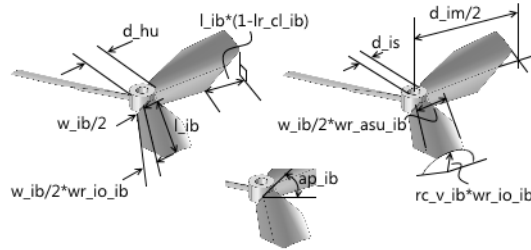


Figure 3-14: Notations used for the hydrofoil impeller.

HYDROFOIL IMPELLER WITH CONSTANT PITCH

In the same fashion as for the pitched impeller, the hydrofoil impeller creates an axial flow velocity that increases with the impeller radius as the impeller rotates. This effect subjects the impeller blade to a bending moment. A hydrofoil impeller with constant pitch adapts the pitch angle in order to obtain an almost constant pitch of the projection of the curved blade along the radius of the impeller blades. Note that it is not possible (at least not easy) to obtain an exactly constant pitch with increasing radius for the hydrofoil impeller since the blades are curved.

As in the case for the regular hydrofoil impellers, the hydrofoil impeller with constant pitch gives an additional pressure difference across the impeller blade with an additionally lowered pressure on the convex side of the blade.

The impeller blade can be designed with cut blades and asymmetrical blades. Fillets can be used to obtain impellers of propeller type.

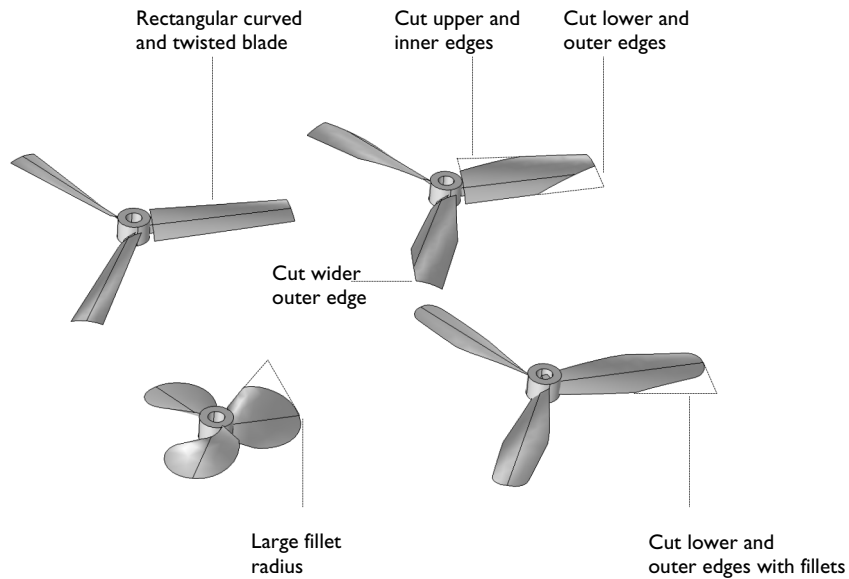


Figure 3-15: Hydrofoil impellers with constant pitch.

The figure below contains the notations used for the hydrofoil with constant pitch impeller in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

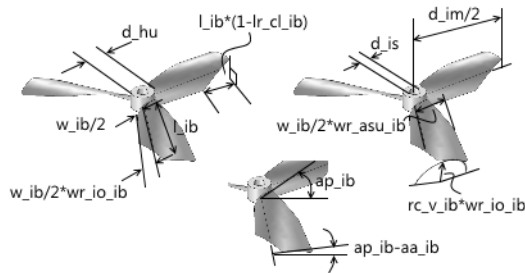


Figure 3-16: Notations used for the hydrofoil with constant pitch.

THE C-SHAPED DOUBLE BLADE IMPELLER

This is an axial impeller that is suitable for fluids with relatively high viscosity. It consists of two pitched blades equipped with a c-shaped double-blade part that adds shear at

the outer radius of the impeller. This impeller can be used to create the so-called Intermig® impeller.

The impeller can be designed with different pitch angles on the arms of the impeller. Also the angles of the c-shaped part can be varied. The profile can be changed by changing the vertical angle of the c-shaped part but keeping the upper and lower blades parallel. In addition, also the angle of the back of the “c” can be varied, keeping the lower and upper blades parallel but displacing them in the x-y plane.

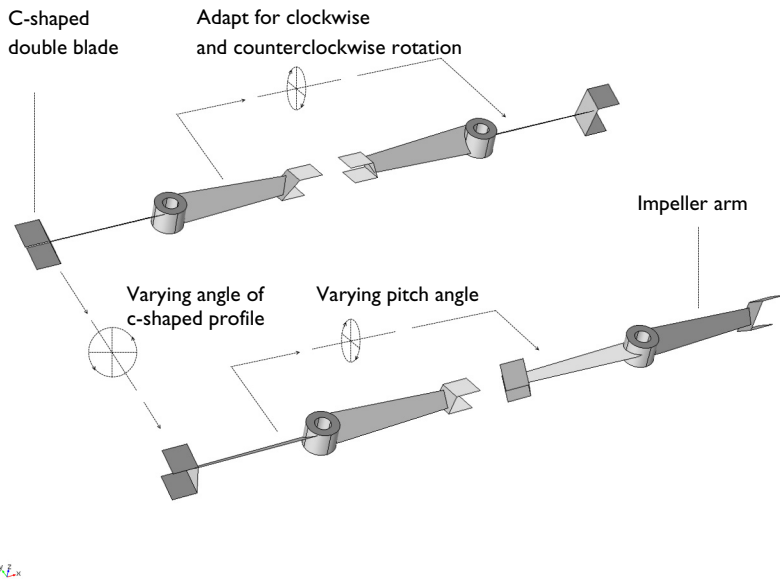


Figure 3-17: The c-shaped with double blades impeller.

The figure below contains the notations used for the c-shaped double blade impeller in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

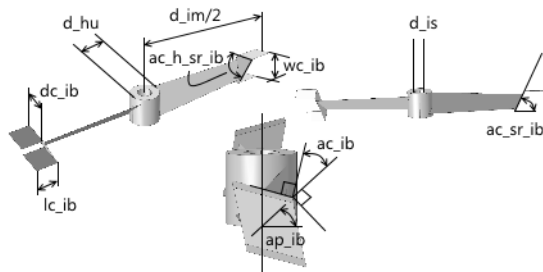


Figure 3-18: Notations used for the c-shaped double blade impeller.

RUSHTON TURBINE

Axial impellers, such as the Rushton turbine, work by pumping the fluid toward the walls of the vessel and then let the collision of the fluid with the wall supply the axial mixing and the turbulence required for mixing. In order to avoid a tangential flows that would result in a poor mixing, baffles are often used in combination with Rushton turbines. Rushton turbines induce a relatively high shear rate and therefore appropriate in processes where the fluid is not sensitive to shear rates.

The turbine can be designed with different blade lengths and also with different disk diameters.

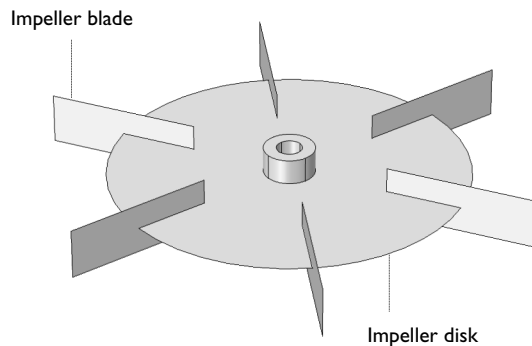


Figure 3-19: The Rushton turbine.

The figure below contains the notations used for the Rushton turbine in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

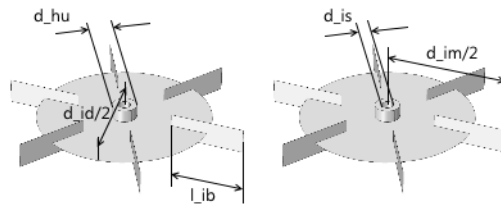


Figure 3-20: Notations used for the Rushton turbine.

THE RUSHTON TURBINE WITH BACKSWEPT BLADES

This type of impeller is similar to the Rushton turbine. The backswept blades allow for a smoother operation with lower shear rates. It is possible to vary the blade length, the disk diameter, as well as the curvature and blade angle relative to the perimeter of the disk.

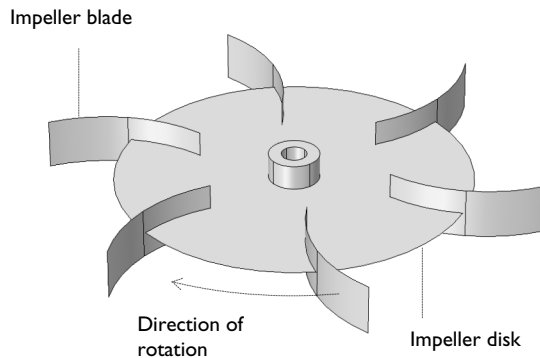


Figure 3-21: The Rushton turbine with backswept blades.

The figure below contains the notations used for the Rushton turbine with backswept blades in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

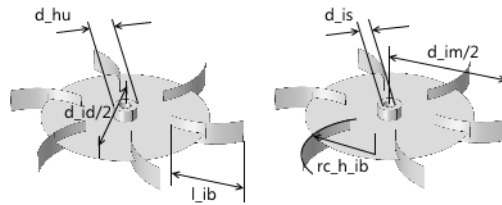


Figure 3-22: Notations used for the Rushton turbine with backswept blades.

THE SMITH TURBINE

This is also an impeller that resembles the Rushton turbine. It has the same properties as the Rushton turbine but with a somewhat smoother operation with lower shear rates. The length of the blades, the diameter of the disk, as well as the radius of the curvature of the blades can be varied when creating the impeller geometry.

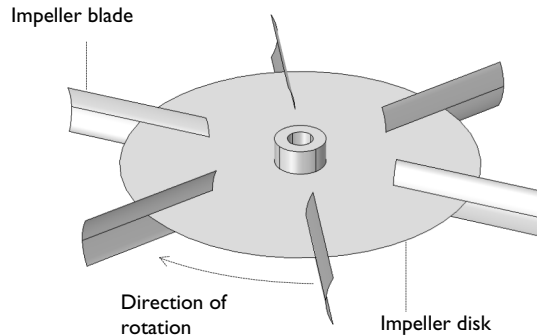


Figure 3-23: The Smith turbine.

The figure below contains the notations used for the Smith turbine in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

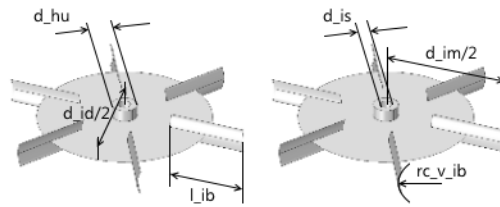


Figure 3-24: Notations used for the Smith turbine.

THE BACKSWEEPED BLADE IMPELLER

This impeller type is suitable for fluids with low viscosity and density (for example, gases). The curvature of the blades create smooth operating conditions with relatively small shear rates compared to straight blades. The impeller can be designed with different blade lengths and curvatures of the blades.

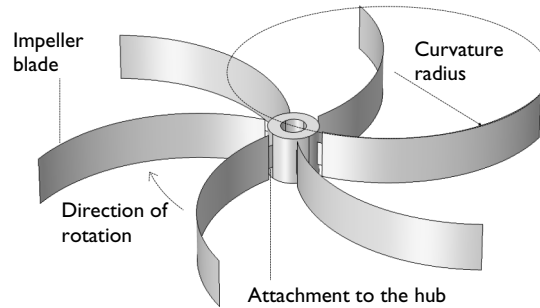


Figure 3-25: The backswept impeller.

The figure below contains the notations used for the backswept blade impeller in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

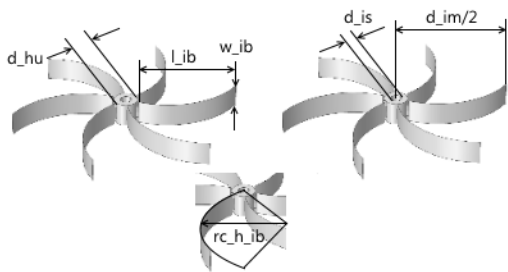


Figure 3-26: Notations used for the backswept blade impeller.

ANCHOR IMPELLER

This impeller type is suitable for mixing fluids of very high viscosity. For example, this impeller is common when mixing Portland concrete and paints in small scales. The lower part of the impeller is shaped as an ellipse that is cut in the middle. The major axis of the ellipse equals the impeller diameter while the minor axis equals to the dish tank where the impeller is placed minus the clearance between the impeller and tank wall.

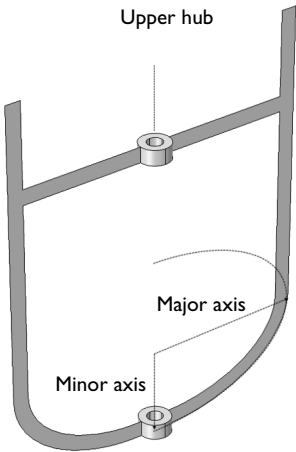


Figure 3-27: Anchor impeller.

The figure below contains the notations used for the anchor impeller in the part library. The full list of parameters together with their description is found in [Table 3-1](#)

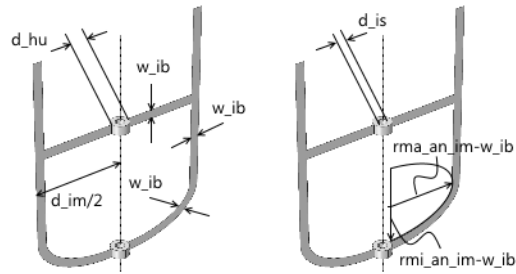


Figure 3-28: Notations used for the anchor impeller.

Impeller Parameters

The table below contains the names and descriptions of the input parameter used for the impellers in the Mixer Module Part Library.

TABLE 3-1: MIXER MODULE PART LIBRARY PARAMETERS

ab_ib	Bending angle of folded section
ac_h_s_ib	Shear angle, back of c-shape blades
ac_ib	Vertical angle between arm and c-shaped blade
ac_s_ib	Internal angle, c-shaped profile
ap_ib	Pitch angle of impeller blade
ar_ib	Rake angle of impeller
as_ib	Relative rotation angle around shaft
d_a_ib	Diameter of attachment section
d_hu	Impeller hub diameter
d_id	Diameter of impeller dish
d_im	Impeller diameter
d_is	Impeller shaft diameter
d_ta	Tank diameter
Da d_im	Input diameter for turbulence model
db_im	Distance of first impeller hub from bottom
dc_ib	Blade width, c-shaped blades
h_an_im	Height of anchor impeller
hp_im	Impeller position

TABLE 3-1: MIXER MODULE PART LIBRARY PARAMETERS

hp_is	Position of the lowest part of the impeller shaft
l_ib	Length of the impeller blades, Rushton, Smith
lc_ib	Blade length, c-shaped blades
lr_bu_ib	Relative position of the bending point along the length of the upper edge of the impeller blade
lr_cl_ib	Relative position of cutting point along the length of the lower edge
lr_cu_ib	Relative position of cutting point along the length of the upper edge
n_ib	Number of blades
pa_rd_im	Parameter that controls the impeller blade direction: 1 = clockwise and -1 = counterclockwise
pa_ud_im	Flip impeller vertically
rc_h_ib	Curvature of backswept Rushton, Smith, and Backswept impeller
rc_v_ib	Hydrofoil curvature radius closest to the hub
rf_ib	Fillet radius of outer vertices
w_a_ib	Blade attachment width
w_bou_ib	Width of the folded section at the outer edge
w_cil_ib	Width of inner and lower cut
w_ciu_ib	Width of inner and upper cut
w_col_ib	Width of outer and lower cut
w_cou_ib	Width of outer and upper cut
w_ib	Impeller blade width
w_o_ib	Outer impeller width
wc_ib	Blade distance, c-shaped blades
wc_o_ib	Outer arm width for c-shaped impeller
wr_ai_ib	Ratio between blade width and blade attachment
wr_asu_ib	Asymmetry factor for upper part of the width of the impeller blade: 1 = symmetry, >1 = wider upper part

Mixer Module Theory

In this chapter:

- [Theory for the Free Surface Features](#)

Theory for the Free Surface Features

In this section:

- [Deforming Domain](#)
- [Free Surface Conditions](#)
- [Contact Angle](#)
- [Rotating Shaft Conditions](#)
- [References for the Free Surface Features](#)



[Theory for the Rotating Machinery Interfaces](#) and [Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces](#) in the *CFD Module User's Guide*.



The links to the physics features described in other guides do not work in the PDF, only from the online help in COMSOL Multiphysics.

Deforming Domain

The mesh within the free surface domain is deformed to account for the movement of the free surface. This mesh movement is accomplished using a moving mesh approach. The software perturbs the mesh nodes so that they conform with the free surface and with other moving or stationary boundaries in the model. The boundary displacement is propagated throughout the domain to obtain a smooth mesh deformation everywhere. This is done by solving PDEs for the mesh displacements, using the movement of the free surface as boundary condition. Taking two dimensions as an example, a location in the deformed mesh with coordinates (x, y) can be related to its coordinates in the original undeformed mesh (X, Y) by a function on the form:

$$x = x(X, Y, t), \quad y = y(X, Y, t)$$

The original, undeformed, mesh is referred to as the *material frame* (or reference frame), while the deformed mesh is called the *spatial frame*. COMSOL Multiphysics also defines *geometry* and *mesh frames*, which are coincident with the material frame for this physics interface.

For the Rotating Machinery interfaces the fluid-flow equations (along with other coupled equations such as heat or chemical species transport) are solved in the spatial frame in which the mesh is perturbed.

A number of mesh smoothing methods are available: Yeoh (the default), Laplace, Winslow, and Hyperelastic. Note that the equations used for each smoothing type have different properties.



Deformed Geometry and Moving Mesh in the *COMSOL Multiphysics Reference Manual*

Free Surface Conditions

The Free Surface feature assumes that the density and viscosity of the fluid outside the domain are negligible compared to the corresponding values inside the domain. On the free surface the following condition for the stress, τ , along the surface is applied:

$$\mathbf{n}_i \cdot \tau = -p_{\text{ext}} \mathbf{n}_i + \mathbf{f}_{\text{st}} = -p_{\text{ext}} \mathbf{n}_i + \sigma (\nabla_s \cdot \mathbf{n}_i) \mathbf{n}_i - \nabla_s \sigma \quad (4-1)$$

Here p_{ext} is the pressure outside the free surface domain (SI unit: Pa) and \mathbf{f}_{st} denotes the surface tension forces (SI unit: N/m²). In the surface tension terms, ∇_s is the surface gradient operator ($\nabla_s = (\mathbf{I} - \mathbf{n}_i \mathbf{n}_i^T) \nabla$ where \mathbf{I} is the identity matrix) and σ is the surface tension coefficient (SI unit: N/m).

The mesh velocity at the free surface is defined as the fluid velocity in the direction normal to the surface:

$$(\mathbf{u}_{\text{mesh}} \cdot \mathbf{n}) = (\mathbf{u} \cdot \mathbf{n}) \quad (4-2)$$

Contact Angle

At a three-phase boundary, it is necessary to add force terms to ensure that the fluid maintains a consistent contact angle. The forces acting at the contact point are applied to the fluid by the [Contact Angle](#) node (added per default under a [Free Surface](#) node). In equilibrium, the surface tension forces and the normal restoring force from the surface are in balance at a contact angle (θ_c), as shown in [Figure 4-1](#). This equilibrium is expressed by Young's equation, which considers the components of the forces in the plane of the surface:

$$\sigma \cos(\theta_c) + \gamma_{s1} = \gamma_{s2} \quad (4-3)$$

where σ is the surface tension between the two fluids, γ_{s1} is the surface energy density on the wetted side, and γ_{s2} the surface energy density on the other side of the two-phase interface.

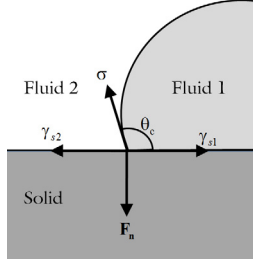


Figure 4-1: The forces acting at a contact point. In equilibrium, the surface tension forces and the normal restoring force from the surface are in balance at a contact angle θ_c .

There is still debate in the literature as to precisely what occurs in non-equilibrium situations (for example, drop impact) when the physical contact angle deviates from the contact angle specified by Young's equation. A simple approach, is to assume that the unbalanced part of the in-plane Young force acts on the fluid to move the contact angle toward its equilibrium value (Ref. 1). COMSOL Multiphysics employs this approach as it is physically motivated and is consistent with the thermodynamics allowed form of the boundary condition (Ref. 2, Ref. 3).

The normal force balance at the solid surface is handled by the wall boundary condition, which automatically prevents fluid flow across the solid boundary through a no-penetration condition. The wall fluid interface feature applies a force, \mathbf{f}_{wf} , on the fluid at the interface:

$$\mathbf{f}_{wf} = \sigma(\cos(\theta_c) - \cos(\theta))\mathbf{m}_s$$

where θ is the actual contact angle and \mathbf{m}_s is the binormal to the solid surface.

Rotating Shaft Conditions

Using a Rotating Shaft feature, the velocity of the shaft boundary is defined by assuming solid body rotation of the shaft. The wall velocity is hence a function of the angular velocity w (SI unit: rad/s) of the shaft, and the (spatial frame) position \mathbf{x} :


$$\mathbf{u}_w = w \frac{\mathbf{r}_{ax}}{|\mathbf{r}_{ax}|} \times (\mathbf{x} - \mathbf{r}_{bp}) \quad (4-4)$$

where \mathbf{r}_{ax} and \mathbf{r}_{bp} is the rotation axis direction and rotation base point, respectively.

References for the Free Surface Features

1. J. Gerbeau and T. Lelievre, “Generalized Navier Boundary Condition and Geometric Conservation Law for Surface Tension,” *Computer Methods In Applied Mechanics and Engineering*, vol. 198, pp. 644–656, 2009.
2. W. Ren and E. Weinan, “Boundary Conditions for the Moving Contact Line Problem,” *Physics of Fluids*, vol. 19, p. 022101, 2007.
3. W. Ren and D. Hu, “Continuum Models for the Contact Line Problem,” *Physics of Fluids*, vol. 22, p. 102103, 2010.

Rotating Machinery, Fluid Flow

The **Rotating Machinery, Laminar Flow (spf)** and **Rotating Machinery, Turbulent Flow (spf)** interfaces, found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, are used for modeling flow where one or more of the boundaries rotate in a periodic fashion. This is used for mixers and propellers.

The physics interfaces support compressible, weakly compressible and incompressible flow, the flow of non-Newtonian fluids described by the Power Law, Carreau Bingham-Papanastasiou, Herschel-Bukley-Papanastasiou, and Casson-Papanastasiou models, as well as turbulent flow. In addition, the physics interfaces support creeping flow, although the shallow channel approximation is redundant.

In this chapter:

- [Moving Mesh](#)
- [The Rotating Machinery, Laminar Flow Interface](#)
- [The Rotating Machinery, Turbulent Flow, Algebraic yPlus Interface](#)
- [The Rotating Machinery, Turbulent Flow, L-VEL Interface](#)
- [The Rotating Machinery, Turbulent Flow, k-ε Interface](#)
- [The Rotating Machinery, Turbulent Flow, Realizable k-ε Interface](#)

- The Rotating Machinery, Turbulent Flow, $k-\omega$ Interface
- The Rotating Machinery, Turbulent Flow, SST Interface
- The Rotating Machinery, Turbulent Flow, Low Re $k-\epsilon$ Interface
- The Rotating Machinery, Turbulent Flow, Spalart-Allmaras Interface
- The Rotating Machinery, Turbulent Flow, $v2-f$ Interface
- Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces

The Rotating Machinery, Fluid Flow Interfaces

Moving Mesh

The Rotating Machinery, Fluid Flow interfaces rely on ALE (Arbitrary Lagrangian Eulerian) moving mesh functionality for their rotating domain and free surface implementation.

ROTATING DOMAINS

When a **Rotating Machinery, Fluid Flow** interface is added using the **Model Wizard**, a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions>Moving Mesh** node. It contains settings for specifying the rotation of one or several domains. By default, **All domains** are selected. Under the **Rotating Domain**, you can exclude any non-rotating fluid domains. For both 3D and 2D components, the **Rotation type** should be set to **Specified rotational velocity**.

Select a **Rotational velocity expression** — **General revolutions per time** (the default), **Constant revolutions per time**, **General angular velocity**, or **Constant angular velocity**.

- For **Constant revolutions per time**, or **General revolutions per time**, enter a value or expression respectively in the **Revolutions per time** input field and optionally an **Initial angle**. The angular velocity in this case is defined as the input multiplied by $2\cdot\pi$.
- For **Constant angular velocity**, or **General angular velocity** enter a value or an expression respectively in the **Angular velocity** input field and optionally an **Initial angle**.

3D Components

For 3D components, select the **Rotation axis base point**, \mathbf{r}_{ax} . The default is the origin. Select the **Rotation axis**, \mathbf{u}_{rot} , the z -axis is the default. If the x -axis is selected, it corresponds to a rotational axis (1, 0, 0) with the origin as the base point, and correspondingly for the y -axis and z -axis.

2D Components

For 2D components, enter coordinates for the **Rotation axis base point**, \mathbf{r}_{ax} . The default is the origin (0, 0).

DEFORMING DOMAIN

When using the **Free Surface** feature, a **Deforming Domain** must first be added in the **Model Builder** under the **Definitions>Moving Mesh** node. Within the **Deforming Domain**, the mesh position is solved for to match the current position of the free surface. On external boundaries not defined as a free surface, the mesh is constrained to move only in the tangential direction. On boundaries which are internal with respect to the fluid (separating deforming domains from ordinary fluid domains), the mesh is constrained to its initial position.

If there is more than one deforming domain, these must not intersect. Also a deforming domain cannot intersect with a rotating domain.



Mesh Smoothing Type

This setting controls the method used to solve for the mesh displacement within deforming domains. The **Mesh smoothing type** can be chosen to be **Laplace**, **Winslow**, **Hyperelastic**, or **Yeoh** (the default) smoothing. Note that the equations used for each smoothing type have different properties. For the Yeoh mesh smoothing type, also specify a **Stiffening factor** (default: 10). See [Smoothing Methods](#) in the *COMSOL Multiphysics Reference Manual* for more information.





Free Surface

The Rotating Machinery, Laminar Flow Interface

The **Rotating Machinery, Laminar Flow (spf)** interface () is found under the **Single-Phase Flow>Rotating Machinery** branch () and is used to simulate flow at low to moderate Reynolds numbers in geometries with one or more rotating parts. The physics interface supports incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3). It also supports modeling of non-Newtonian fluids. The physics interface is available for 3D and 2D components.

There are two study types available for this physics interface. Using the **Time Dependent** study type, rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study type, the rotating parts are kept frozen in position, and rotation is accounted for by the inclusion of centrifugal and Coriolis forces. In both types, the momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default physics nodes are also added in the **Model Builder** under **Laminar Flow** — **Fluid Properties**, **Wall** and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Laminar Flow** to select physics features from the context menu. See [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

	Pseudo Time Stepping for Laminar Flow Models in the <i>CFD Module User's Guide</i>
	<i>Laminar Flow in a Baffled Stirred Mixer</i> : Application Library path <code>CFD_Module/Single-Phase_Tutorials/baffled_mixer</code>

SETTINGS


The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `spf`.

In addition to the settings described below, see [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide* for all the other settings available. See [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#) for links to all the physics nodes.

ADVANCED SETTINGS



To display this section, click the **Show** button () and select **Advanced Physics Options**. Normally these settings do not need to be changed.

Pseudo time stepping

Select the **Use pseudo time stepping for stationary equation form** check box to add pseudo time derivatives to the equation when the **Frozen Rotor** equation form is used. (Frozen rotor is a pseudo stationary formulation.) When selected, also choose a **CFL**

number expression — **Automatic** (the default) or **Manual**. **Automatic** sets the local CFL number (from the Courant–Friedrichs–Lewy condition) to the built-in variable `CFLCMP` which in turn triggers a PID regulator for the CFL number. For **Manual** enter a **Local CFL number** CFL_{loc} (dimensionless).

The Rotating Machinery, Turbulent Flow, Algebraic yPlus Interface

The **Rotating Machinery, Turbulent Flow, Algebraic yPlus (spf)** interface () , found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface is suitable for incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3).

The equations solved by the Turbulent Flow, Algebraic yPlus interface are the Reynolds-averaged Navier-Stokes (RANS) equations for conservation of momentum, the continuity equation for conservation of mass, and an algebraic equation for the scaled wall distance. Turbulence effects are included using an enhanced viscosity model based on the scaled wall distance, and the physics interface therefore includes a wall distance equation.

Two study types are available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. Both study types includes an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, Algebraic yPlus — Fluid Properties, Wall and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, Algebraic yPlus** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the settings below, see the advanced settings for [The Rotating Machinery, Laminar Flow Interface](#), and [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS



The turbulence model parameters for the Algebraic yPlus model, κ_a and B_a , have been optimized for turbulent pipe flow. For certain cases, better performance can be obtained by tuning the model parameters.

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field u**
- **Pressure p**
- **Reciprocal wall distance G**
- **Wall distance in viscous units yPlus** (dimensionless)

The Rotating Machinery, Turbulent Flow, L-VEL Interface

The **Rotating Machinery, Turbulent Flow, L-VEL (spf)** interface () found under the **Single-Phase Flow > Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface is suitable for incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3).

The equations solved by the Turbulent Flow, L-VEL interface are the Reynolds-averaged Navier-Stokes (RANS) equations for conservation of momentum, the continuity equation for conservation of mass, and an algebraic equation for the scaled fluid flow velocity tangential to the nearest wall. Turbulence effects are included using an enhanced viscosity model based on the scaled wall distance, and the physics interface therefore includes a wall distance equation.

Two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. Both study types includes an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, L-VEL — Fluid Properties, Wall, and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes

that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, L-VEL** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the interface settings defined below, see the advanced settings for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS



The turbulence model parameters for the L-VEL model, κ_l and E_l , have been optimized for internal flow. For certain cases, better performance can be obtained by tuning the model parameters.

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field u**
- **Pressure p**
- **Reciprocal wall distance G**
- **Tangential velocity in viscous units $uPlus$ (dimensionless)**

The Rotating Machinery, Turbulent Flow, k - ϵ Interface

The **Rotating Machinery, Turbulent Flow, k - ϵ (spf)** interface () found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface is suitable for incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3).

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using the standard two-equation k - ϵ model with realizability constraints. Flow close to walls is modeled using wall functions.

There are two study types available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis

forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, k- ϵ — Fluid Properties, Wall** and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, k- ϵ** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the defined settings below, see the advanced settings for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS

Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters.





For this physics interface the parameters are C_{e1} , C_{e2} , C_μ , σ_k , σ_ϵ , κ_v , and B .

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Turbulent kinetic energy k**
- **Turbulent dissipation rate ϵ_p**

The Rotating Machinery, Turbulent Flow, Realizable k- ϵ Interface

The **Rotating Machinery, Turbulent Flow, Realizable k- ϵ (spf)** interface () () () found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface is suitable for incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3).

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using the Realizable two-equation k - ϵ model. Flow close to walls is modeled using wall functions.

There are two study types available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, Realizable k- ϵ — Fluid Properties, Wall and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, Realizable k- ϵ** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the defined settings below, see the advanced settings for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS

Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters.



For this physics interface the parameters are C_2 , A_0 , σ_k , σ_ϵ , κ_v , and B .

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Turbulent kinetic energy k**
- **Turbulent dissipation rate ϵ_p**

The Rotating Machinery, Turbulent Flow, k - ω Interface

The **Rotating Machinery, Turbulent Flow, k - ω (spf)** interface () found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface supports incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3). The physics interface is available for 3D and 2D components.

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using Wilcox revised two-equation k - ω model with realizability constraints. Flow close to walls is modeled using wall functions.

There are two study types available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, k - ω — Fluid Properties, Wall and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, k - ω** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the settings defined below, see the advanced settings for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS

Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters.



For this physics interface the parameters are α , σ_k^* , σ_w , β_0 , β_0^* , κ_v , and B .

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Turbulent kinetic energy k**
- **Specific dissipation rate ω**

The Rotating Machinery, Turbulent Flow, SST Interface

The **Rotating Machinery, Turbulent Flow, SST (spf)** interface () is found under the **Single-Phase Flow > Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface supports incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3). The physics interface is available for 3D and 2D components.

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using the SST two-equation model with realizability constraints. The SST model is a so-called low-Reynolds number model, which means that it resolves the flow all the way down to the wall. The SST model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*. In both study types initialization is performed by first solving for the distance to the closest wall.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, SST — Fluid Properties, Wall** and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, SST** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the interface settings defined below, see the advanced settings for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS






Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters. For this physics interface the parameters are a_1 , β_0^* , β_1 , β_2 , γ_1 , γ_2 , σ_{k1} , σ_{k2} , σ_{w1} , and σ_{w2} .

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Turbulent kinetic energy k**
- **Specific dissipation rate ω**
- **Reciprocal wall distance G**

The Rotating Machinery, Turbulent Flow, Low Re k - ϵ Interface

The **Rotating Machinery, Turbulent Flow, Low Re k - ϵ (spf)** interface () () () () is found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface supports incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3). The physics interface is available for 3D and 2D components.

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using the AKN two-equation k - ϵ model with realizability constraints. The AKN model is a so-called low-Reynolds number model, which means that it resolves the flow all the way down to the wall. The AKN model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization**

study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*. In both study types initialization is performed by first solving for the distance to the closest wall.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, Low Re k-ε**— **Fluid Properties**, **Wall** and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, Low Re k-ε** to select physics features from the context menu. For information about the available physics features, see [Domain](#), [Boundary](#), [Point](#), and [Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the interface settings defined below, see the advanced settings defined for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS

Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters.



For this physics interface the parameters are C_{e1} , C_{e2} , C_μ , σ_k , σ_ϵ , and κ_v .

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Turbulent kinetic energy k**
- **Turbulent dissipation rate ϵ_p**
- **Reciprocal wall distance G**

The Rotating Machinery, Turbulent Flow, Spalart-Allmaras Interface

The **Rotating Machinery, Turbulent Flow, Spalart-Allmaras (spf)** interface () is found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics

interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface supports incompressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3). The physics interface is available for 3D and 2D components.

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using the Spalart-Allmaras one-equation model. The Spalart-Allmaras model is a so-called low-Reynolds number model, which means that it resolves the flow all the way down to the wall. The Spalart-Allmaras model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*. In both study types initialization is performed by first solving for the distance to the closest wall.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, Spalart-Allmaras — Fluid Properties, Wall, and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, Spalart-Allmaras** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).


A part for the interface settings defined below, see the advanced settings defined for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS

Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters.

For this physics interface the parameters are C_{b1} , C_{b2} , C_{v1} , σ_v , C_{w2} , C_{w3} , κ_v , and C_{rot} .

ADVANCED SETTINGS



To display this section, click the **Show** button () and select **Advanced Physics Options**. Under the **Turbulence variables scale parameters** subsection, the default ν_{scale} is of the same order of magnitude as the kinematic viscosity of air at room temperature.

DEPENDENT VARIABLES

The following physics dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Reciprocal wall distance G**
- **Undamped turbulent kinematic viscosity nutilde**

The Rotating Machinery, Turbulent Flow, v2-f Interface

The **Rotating Machinery, Turbulent Flow, v2-f (spf)** interface (), found under the **Single-Phase Flow>Rotating Machinery** branch () when adding a physics interface, is used to simulate flow at high Reynolds numbers in geometries with one or more rotating parts. The physics interface supports incompressible, weakly compressible, weakly compressible and compressible flows at low Mach numbers (typically less than 0.3). The physics interface is available for 3D and 2D components.

The momentum balance is governed by the Navier-Stokes equations, and the mass conservation is governed by the continuity equation. Turbulence effects are modeled using the v2-f transport-equation model with realizability constraints. The v2-f model is a so-called low-Reynolds number model, which means that it can resolve the flow all the way down to the wall. The physics interface includes a wall distance equation.

There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*. In both study types initialization is performed by first solving for the distance to the closest wall.

When this physics interface is added, the following physics nodes are also added in the **Model Builder** under **Turbulent Flow, v2-f — Fluid Properties, Wall** and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes

that implement, for example, boundary conditions and volume forces. You can also right-click **Turbulent Flow, v2-f** to select physics features from the context menu. For information about the available physics features, see [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

A part for the interface settings defined below, see the advanced settings defined for [The Rotating Machinery, Laminar Flow Interface](#), and the settings in [The Creeping Flow, Laminar Flow, Turbulent Flow, and Large Eddy Simulation Interfaces](#) in the *CFD Module User's Guide*.

TURBULENCE MODEL PARAMETERS

Turbulence model parameters are optimized to fit as many flow types as possible, but for some special cases, better performance can be obtained by tuning the model parameters.

For this physics interface the parameters are $C_{\mu v2f}$, C_τ , C_η , C_L , C_1 , C_2 , $C_{\epsilon1v2f}$, $C_{\epsilon2v2f}$, A_1 , σ_{kv2f} , $\sigma_{\epsilon v2f}$, $\sigma_{\zeta v2f}$, κ_v , and B .

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface:

- **Velocity field \mathbf{u}** and its components
- **Pressure p**
- **Turbulent kinetic energy k**
- **Turbulent dissipation rate ϵ**
- **Turbulent relative fluctuations ζ**
- **Elliptic blending function α**
- **Reciprocal wall distance G**

Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces

All versions of the Rotating Machinery, Fluid Flow interfaces use the following domain, boundary, point, and pair physics nodes.

- [Contact Angle](#)
- [Free Surface](#)
- [Rotating Shaft](#)
- [Stationary Free Surface](#)

The nodes are available under the option **Rotating Machinery** from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

The following nodes, available in the Fluid Flow interfaces, are described for the Laminar Flow interface in the *CFD Module User's Guide*:

- | | |
|-------------------------------------|---|
| • Initial Values | • Open Boundary |
| • Wall | • Outlet |
| • Interior Wall | • Periodic Flow Condition |
| • No Viscous Stress | • Point Mass Source |
| • Flow Continuity | • Pressure Point Constraint |
| • Fluid Properties | • Screen |
| • Inlet | • Symmetry |
| • Line Mass Source | • Volume Force |



[References for the Single-Phase Flow, Turbulent Flow Interfaces](#)

Stationary Free Surface

This postprocessing feature is used to estimate the deformation of a free surface from the pressure distribution on the boundary when using the **Frozen Rotor** or **Frozen Rotor with Initialization** study types. When computing the flow field, a slip condition together with a constant pressure level p_{ext} is applied on the boundary. The free surface deformation η_{FS} is evaluated from the linearized free surface condition in a separate study step, [Stationary Free Surface](#), using the pressure from the Frozen Rotor study step.

$$p(\mathbf{x}_0) - p_{\text{ext}} + \hat{\mathbf{n}} \cdot \nabla p \Big|_{\mathbf{x} = \mathbf{x}_0} \eta_{\text{FS}} = -\sigma \nabla_S^2 \eta_{\text{FS}} \quad (5-1)$$

σ is the surface tension coefficient, $\mathbf{x} = \mathbf{x}_0$ represents the position of the undisturbed surface - the feature selection, and $\hat{\mathbf{n}}$ is its unit normal. The correct study sequence can be triggered by toggling the study window once a selection for the Stationary Free Surface feature has been made.

The Stationary Free Surface feature is supported by laminar and turbulent flow, and is applicable for small surface deformations (η_{FS} and its gradient should be small compared to the dimensions of the computational domain). If the surface deformation is large, a time-dependent study with a **Deforming Domain** and a [Free Surface](#) feature should be used instead.

Iterative solvers are necessary to reduce the cost of models with large number of degrees of freedom. The fluid flow interfaces use a **Smoothed aggregation AMG** solver per default. It is recommendable to switch to a **Geometric multigrid** solver when working with the Stationary Free Surface feature to ensure robustness of the model.



[The Choice of Solver and Solver Settings](#) in the *CFD Module User's Guide* and [Studies and Solvers](#) in the *COMSOL Multiphysics Reference Manual*

STATIONARY FREE SURFACE

Enter an **External pressure** p_{ext} corresponding to the average pressure level on the free surface.

SURFACE TENSION

The **Surface tension coefficient** σ (SI unit: N/m) can be specified from predefined libraries, by selecting **Library coefficient, liquid/gas interface** (the default) or **Library coefficient, liquid/liquid interface**, or be set to **User defined**.

- For **Library coefficient, liquid/gas interface** select an option from the list — **Water/Air** (the default), **Acetone/Air**, **Acetic acid/Air**, **Ethanol/Air**, **Ethylene glycol/Ethylene glycol vapor**, **Diethyl ether/Air**, **Glycerol/Air**, **Heptane/Nitrogen**, **Mercury/Mercury vapor**, or **Toluene/Air**.
- For **Library coefficient, liquid/liquid interface** select an option from the list — **Benzene/Water, 20°C**, **Corn oil/Water, 20°C**, **Ether/Water, 20°C**, **Hexane/Water, 20°C**, **Mercury/Water, 20°C**, or **Olive oil/Water, 20°C**.

Only one Stationary Free Surface feature can be applied in a Rotating Machinery, Fluid Flow interface, and it requires the study to be stationary (Frozen Rotor).

LOCATION IN USER INTERFACE

Right click the physics interface and select **Rotating Machinery>Stationary Free Surface**

Ribbon

Physics tab with the physics interface selected:

Boundaries>Rotating Machinery>Stationary Free Surface

Free Surface

This feature is used to define the two-phase interface separating the modeled fluid in a free surface domain from the outer fluid. By applying this boundary condition the motion of the two-phase interface can be tracked. The free surface follows the fluid motion in the normal direction. In the tangential direction the free surface motion is governed by the fluid motion in combination with the surface tension of the fluid-fluid interface.

The Free Surface node can only be applied to outer boundaries of a **Deforming Domain** that do not have any boundary condition assigned under **Moving Mesh**.

To control the attachment angle between the free surface and adjacent walls, a [Contact Angle](#) node is by default added under the **Free Surface** node.

The Free Surface feature is supported by laminar flow and by turbulence models using wall functions or automatic wall treatment but not low-Reynolds number wall

treatment. It also requires the study to be time dependent. The node is inactive for Frozen Rotor studies.

FREE SURFACE

Enter an **External pressure** p_{ext} (SI unit: Pa) corresponding to the pressure level in the external fluid phase.

SURFACE TENSION

The **Surface tension coefficient** σ (SI unit: N/m) can be specified from predefined libraries, by selecting **Library coefficient, liquid/gas interface** (the default) or **Library coefficient, liquid/liquid interface**, or set to **User defined**.

- For **Library coefficient, liquid/gas interface** select an option from the list — **Water/Air** (the default), **Acetone/Air**, **Acetic acid/Air**, **Ethanol/Air**, **Ethylene glycol/Ethylene glycol vapor**, **Diethyl ether/Air**, **Glycerol/Air**, **Heptane/Nitrogen**, **Mercury/Mercury vapor**, or **Toluene/Air**.
- For **Library coefficient, liquid/liquid interface** select an option from the list — **Benzene/Water, 20°C**, **Corn oil/Water, 20°C**, **Ether/Water, 20°C**, **Hexane/Water, 20°C**, **Mercury/Water, 20°C**, or **Olive oil/Water, 20°C**.

LOCATION IN USER INTERFACE

Context Menus

Right click the physics interface and select **Rotating Machinery>Free Surface**

Ribbon

Physics tab with the physics interface selected:

Boundaries>Rotating Machinery>Free Surface



The **Free Surface** feature imposes a mesh deformation in the normal direction. On adjacent boundaries, the mesh displacement parallel to those boundaries cannot be fixed if the contact point is moving. In this case a [Mesh Slip](#), [Symmetry/Roller](#), [Prescribed Normal Mesh Displacement](#), or [Prescribed Normal Mesh Velocity](#) boundary condition under **Moving Mesh** should be selected. A [Mesh Slip](#) condition is automatically used on boundaries that are adjacent to the Free Surface and do not have a prescribed Moving Mesh boundary condition.



If a **Free Surface** feature is in contact with a solid boundary, the movement of the contact point is not compatible with the no slip condition. For laminar flow the **Wall** feature with the **Navier Slip** option should be selected. For turbulence models with wall functions or automatic wall treatment the **Wall** feature with the **No Slip** option can be selected.

Contact Angle

This feature specifies the contact angle between the free surface and a solid wall. The Contact Angle feature is a subfeature to, and added under a [Stationary Free Surface](#) or a Free Surface feature.

CONTACT ANGLE

Select an option from the **Specify contact angle list** — **Directly** (the default) or **Through Young's equation**.

- For **Directly** enter a **Contact angle** θ_w (SI unit: rad). The default is $\pi/2$.
- For **Through Young's equation** enter values or expressions for **Phase 1-Solid surface energy density** γ_{s1} (SI unit: J/m²) and **Phase 2-Solid surface energy density** γ_{s2} (SI unit: J/m²).

The remainder of this section applies when the Contact Angle feature is a subfeature to the [Free Surface](#) feature.

The condition applies forces on the wall that balance the surface tension for the prescribed contact angle. For laminar flow, the Contact Angle node should be used on edges or points where a Wall feature with the Navier Slip option is prescribed adjacent to the free surface. In this case, the Navier Slip option is required for the contact line to move along the wall.

The representation of the contact angle is dependent on the mesh resolution in the region where the fluid-fluid interface attaches to the wall. If, during the simulation, the contact angle is found to fluctuate around a given value, this indicates that the resolution needs to be improved. This can for example be achieved by adding more mesh boundary layers on the wall.



The moving mesh boundary condition must be compatible with the motion of the wall for the problem to have a valid solution.

Rotating Shaft

This boundary feature specifies flow conditions on boundaries where the fluid is subjected to a solid shaft rotation. This feature can be applied to rotating shaft boundaries in a free-surface domain modeled by a **Deforming Domain** that connects to a rotating shaft in a **Rotating Domain**.

The flow condition prescribed by the feature corresponds to a rotational velocity defined from an existing **Rotating Domain**. In order for the contact line, defined by intersection of the rotating shaft and the free surface, to be able to move along the rotating shaft the velocity condition is applied using a [Wall](#) boundary condition with the Navier Slip option for laminar flow. For turbulent flow, wall functions or automatic wall treatment must be applied.

ROTATING SHAFT

Axis of Rotation

Select the **Axis of rotation**. If *x*-axis is selected, this corresponds to a rotational axis (1, 0, 0) with the origin as the base point. Correspondingly, if *y*-axis or *z*-axis is selected, the rotational axis is defined by a unit vector in the selected direction together with a base point at the origin. For **User defined** enter values into these fields: **Rotation axis base point** and **Rotation axis direction**. The *z*-axis is the default choice and the default values in the fields for **User defined** correspond to those for the *z*-axis.

Angular Velocity

Select the rotating shaft **Rotational frequency** from the list:

- Select the **Revolution per time** option to manually define the rotational velocity (SI unit: rad/s) and select the **Rotational direction** as **Positive angular velocity** or **Negative angular velocity**. This is the default setting.
- Select the **Angular velocity** option to manually define the angular velocity (SI unit: rad/s).

LOCATION IN USER INTERFACE

Context Menus


Right click the physics interface and select **Rotating Machinery>Rotating Shaft**

Ribbon

Physics tab with the physics interface selected:

Boundaries>Rotating Machinery>Rotating Shaft


Rotating Machinery, Multiphase Flow Interfaces

This chapter describes the physics interfaces found under the **Fluid Flow>Multiphase Flow>Rotating Machinery, Multiphase Flow** branch (). In this chapter:

- The Rotating Machinery, Mixture Model, Laminar Flow Interface
- The Rotating Machinery, Mixture Model, Algebraic yPlus Interface
- The Rotating Machinery, Mixture Model, L-VEL Interface
- The Rotating Machinery, Mixture Model, k- ϵ Interface
- The Rotating Machinery, Mixture Model, Realizable k- ϵ Interface
- The Rotating Machinery, Mixture Model, k- ω Interface
- The Rotating Machinery, Mixture Model, SST Interface
- The Rotating Machinery, Mixture Model, Low Re k- ϵ Interface
- The Rotating Machinery, Mixture Model, Spalart-Allmaras Interface
- The Rotating Machinery, Mixture Model, v2-f Interface
- Domain, Boundary, Pair, and Point Nodes for the Rotating Machinery, Mixture Model Interfaces

The Rotating Machinery, Multiphase Flow, Mixture Model, Laminar Flow and Turbulent Flow Interfaces

The Rotating Machinery, Mixture Model, Laminar Flow Interface

The **Laminar Flow** version of the **Rotating Machinery, Mixture Model (mm)** interface () is used to model low to moderate Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.

There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.

When this physics interface is added, the following default nodes are also added in the **Model Builder** under **Mixture Model, Laminar Flow** — **Mixture Properties**, **Wall**, and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other

nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model**, **Laminar Flow** to select physics features from the context menu.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Mixture Model (rmmf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.

SETTINGS


The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `mm`.

Most of the physics interface settings are the same as described for [The Mixture Model, Laminar Flow Interface](#) in the *CFD Module User's Guide*, and for [Moving Mesh](#). Only unique settings are discussed below.

ADVANCED SETTINGS

To display this section, click the **Show** button () and select **Advanced Physics Options**. Normally these settings do not need to be changed.

Pseudo Time Stepping

Select the **Use pseudo time stepping for stationary equation form** check box to add pseudo time derivatives to the equation when the **Frozen Rotor** equation form is used. (Frozen rotor is a pseudo stationary formulation.) When selected, also choose a **CFL number expression**—**Automatic** (the default) or **Manual**. **Automatic** sets the local CFL number (from the Courant–Friedrichs–Lewy condition) to the built-in variable

CFLCMP which in turn triggers a PID regulator for the CFL number. For **Manual** enter a **Local CFL number** CFL_{loc} (dimensionless).




DEPENDENT VARIABLES

Enter values for the dependent variables (field variables):

- **Velocity field, mixture u** (SI unit: m/s)
- **Pressure p** (SI unit: Pa)
- **Volume fraction, dispersed phase ϕ_{id}** (dimensionless)
- **Squared slip velocity $slip_{vel}$**
- **Number density, dispersed phase nd** (SI unit: $1/m^3$)

The names can be changed, but the names of fields and dependent variables must be unique within a component.

The Rotating Machinery, Mixture Model, Algebraic yPlus Interface

The **Rotating Machinery, Mixture Model, Algebraic yPlus (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are included using an enhanced viscosity model based on the scaled wall distance, and the physics interface therefore includes a wall distance equation.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.


There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh

functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, Algebraic yPlus — Mixture Properties, Wall, and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, Algebraic yPlus** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, Algebraic yPlus Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, L-VEL Interface

The **Rotating Machinery, Mixture Model, L-VEL (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are included using an enhanced viscosity model based on the local wall distance, and the physics interface therefore includes a wall distance equation.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.


There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization**

study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, L-VEL — Mixture Properties, Wall, and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, L-VEL** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, L-VEL Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, k - ϵ Interface

The **Rotating Machinery, Mixture Model, k - ϵ (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the standard two-equation k - ϵ model with realizability constraints. Flow close to walls is modeled using wall functions.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.

There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, k - ϵ — Mixture Properties, Wall, and Initial Values**.


A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, k- ϵ** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, k- \$\epsilon\$ Interface](#) in the *CFD Module User's Guide*.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Mixture Model (rmmf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.

The Rotating Machinery, Mixture Model, Realizable k- ϵ Interface

The **Rotating Machinery, Mixture Model, Realizable k- ϵ (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the realizable two-equation k - ϵ model. Flow close to walls is modeled using wall functions.


The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.

There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, Realizable $k-\epsilon$ — Mixture Properties, Wall**, and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, Realizable $k-\epsilon$** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, Realizable \$k-\epsilon\$ Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, $k-\omega$ Interface

The **Rotating Machinery, Mixture Model, $k-\omega$ (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the Wilcox revised two-equation $k-\omega$ model with realizability constraints. The $k-\omega$ model is a so-called low-Reynolds-number model, which means that it can resolve the flow all the way down to the wall.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.


There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, $k-\omega$ — Mixture Properties, Wall**, and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other

nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, k- ω** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, k- \$\omega\$ Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, SST Interface

The **Rotating Machinery, Mixture Model, SST (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the Menter shear-stress transport (SST) two-equation model from 2003 with realizability constraints. The SST model is a so-called low-Reynolds-number model, which means that it can resolve the flow all the way down to the wall. The SST model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.


There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, SST — Mixture Properties, Wall, and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other

nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, SST** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, SST Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, Low Re k - ϵ Interface

The **Rotating Machinery, Mixture Model, Low Re k - ϵ (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the AKN two-equation k - ϵ model with realizability constraints. The AKN model is a so-called low-Reynolds-number model, which means that it can resolve the flow all the way down to the wall. The AKN model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.


There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, Low Re k - ϵ — Mixture Properties, Wall, and Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other

nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, Low Re k-ε** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, Low Re k-ε Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, Spalart-Allmaras Interface

The **Rotating Machinery, Mixture Model, Spalart-Allmaras (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the Spalart-Allmaras one-equation model. The Spalart-Allmaras model is a so-called low-Reynolds-number model, which means that it can resolve the flow all the way down to the wall. The Spalart-Allmaras model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.


There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, Spalart-Allmaras — Mixture Properties, Wall**, and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added

automatically in the **Model Builder** under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model**, **Spalart-Allmaras** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, Spalart-Allmaras Interface](#) in the *CFD Module User's Guide*.

The Rotating Machinery, Mixture Model, v2-f Interface

The **Rotating Machinery, Mixture Model, v2-f (mm)** interface () is used to model high Reynolds number flow of a multiphase mixture in equipment containing one or more rotating parts. The mixture consists of a liquid with dispersed particles, bubbles, liquid droplets, or solid particles, which are assumed to always travel with terminal velocity.

The interface solves one set of Navier-Stokes equations for the momentum of the mixture. The pressure distribution is calculated from a mixture-averaged continuity equation and the velocity of the dispersed phase is described by a slip model. The dispersed particles are tracked by solving a transport equation for the volume fraction of the dispersed phase. Turbulence effects are modeled using the v2-f transport-equation model with realizability constraints. The v2-f model is a so-called low-Reynolds-number model, which means that it can resolve the flow all the way down to the wall. The v2-f model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

The physics interface can also model the distribution of the number density, which in turn can be used to calculate the interfacial area, useful when simulating chemical reactions in the mixture.

There are two study types available for this physics interface. Using the **Time Dependent with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When this physics interface is added, the following default nodes are also added in the **Model Builder**, under **Mixture Model, v2-f — Mixture Properties, Wall**, and **Initial Values**. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the

Model Builder under the **Definitions** node. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Mixture Model, v2-f** to select physics features from the context menu.

The physics interface settings are the same as described for [The Rotating Machinery, Mixture Model, Laminar Flow Interface](#), for [Moving Mesh](#), and for the [The Mixture Model, v2-f Interface](#) in the *CFD Module User's Guide*.

Domain, Boundary, Pair, and Point Nodes for the Rotating Machinery, Mixture Model Interfaces

All versions of the Rotating Machinery, Mixture Model interfaces include the following domain, boundary, pair, and point nodes:

- [Contact Angle](#)
- [Free Surface](#)
- [Rotating Shaft](#)

The nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

The following nodes (listed in alphabetical order) are described for the Mixture Model interface in the *CFD Module User's Guide*:

- | | |
|--------------------------------------|--|
| • Gravity | • Outlet |
| • Initial Values | • Periodic Flow Condition |
| • Inlet | • Pressure Point Constraint ¹ |
| • Interior Wall | • Symmetry |
| • Mass Transfer | • Volume Force ¹ |
| • Mixture Properties | • Wall |
| • Mixture Continuity | |

¹ Described for the Laminar Flow interface in the *CFD Module User's Guide*.

Free Surface

The **Free Surface** node is used to define the free surface separating an interior modeled fluid from the outer fluid. The free surface follows the fluid motion, in the normal direction, and is governed by the fluid motion in the free surface domain and the surface tension of the fluid-fluid interface. The Free Surface node can only be applied to outer boundaries of a **Deforming Domain** that do not have any boundary condition assigned under **Moving Mesh**.

To control the attachment angle between the free surface and adjacent walls, a **Contact Angle** node is by default added under the **Free Surface** node.

The Free Surface feature is supported by laminar flow and by turbulence models using wall functions or automatic wall treatment, but not low-Reynolds number wall treatment. It also requires the study to be time dependent. The node is inactive for Frozen Rotor studies.

FREE SURFACE

Enter an **External pressure** p_{ext} (SI unit: Pa) corresponding to the pressure level in the external fluid phase.

SURFACE TENSION

The **Surface tension coefficient** σ (SI unit: N/m) can be specified from predefined libraries, by selecting **Library coefficient, liquid/gas interface** (the default) or **Library coefficient, liquid/liquid interface**, or set to **User defined**.

- For **Library coefficient, liquid/gas interface** select an option from the list — **Water/Air** (the default), **Acetone/Air**, **Acetic acid/Air**, **Ethanol/Air**, **Ethylene glycol/Ethylene glycol vapor**, **Diethyl ether/Air**, **Glycerol/Air**, **Heptane/Nitrogen**, **Mercury/Mercury vapor**, or **Toluene/Air**.
- For **Library coefficient, liquid/liquid interface** select an option from the list — **Benzene/Water, 20°C**, **Corn oil/Water, 20°C**, **Ether/Water, 20°C**, **Hexane/Water, 20°C**, **Mercury/Water, 20°C**, or **Olive oil/Water, 20°C**.

DISPERSED PHASE BOUNDARY CONDITION

Select an option from the **Dispersed phase boundary condition** list—**No dispersed phase flux** (the default), **Dispersed phase concentration**, **Dispersed phase outlet**, **Dispersed phase flux**, or **Symmetry**.

- For **Dispersed phase concentration** enter a **Dispersed phase volume fraction** ϕ_{d0} (dimensionless). The default is 0. When **Solve for interfacial area** is selected under

the **Physical Model** section, select either the **Dispersed phase number density** n_0 (SI unit: $1/\text{m}^3$) or **Diameter of particles/droplets** d_d (SI unit: m) button and enter a value or expression for n_0 (the default is $5 \cdot 10^6 \text{ } 1/\text{m}^3$) or d_d (the default is 1 mm).

- For **Dispersed phase flux** enter values or expression for the **Dispersed phase flux** $N_{\phi d}$ (SI unit: m/s) and, if **Solve for interfacial area** is selected under the **Physical Model** section, the **Number density flux** N_n (SI unit: $1/(\text{m}^2 \cdot \text{s})$). The defaults are 0 m/s and 0 $1/(\text{m}^2 \cdot \text{s})$, respectively.



[Dispersed Phase Boundary Conditions Equations](#) in the *CFD Module User's Guide*

LOCATION IN USER INTERFACE

Context Menus

Right click the physics interface and select **Rotating Machinery>Free Surface**

Ribbon

Physics tab with the physics interface selected:

Boundaries>Rotating Machinery>Free Surface



The **Free Surface** feature imposes a mesh deformation in the normal direction. On adjacent boundaries, the mesh displacement parallel to those boundaries cannot be fixed if the contact point is moving. In this case a [Mesh Slip](#), [Symmetry/Roller](#), [Prescribed Normal Mesh Displacement](#), or [Prescribed Normal Mesh Velocity](#) boundary condition under **Moving Mesh** should be selected. A [Mesh Slip](#) condition is automatically used on the boundaries that are adjacent to the Free Surface and do not have a prescribed Moving Mesh boundary condition.



If a **Free Surface** feature is in contact with a solid boundary, the movement of the contact point is not compatible with the no slip condition. For laminar flow the **Wall** feature with the **Navier Slip** option should be selected. For turbulence models with wall functions or automatic wall treatment the **Wall** feature with the **No Slip** option can be selected.

Contact Angle

This feature specifies the contact angle between the free surface and a solid wall. The Contact Angle feature is a subfeature to, and added under a [Free Surface](#) feature.

CONTACT ANGLE

Select an option from the **Specify contact angle list**—**Directly** (the default) or **Through Young's equation**.

- For **Directly** enter a **Contact angle** θ_w (SI unit: rad). The default is $\pi/2$.
- For **Through Young's equation** enter values or expressions for **Phase 1-Solid surface energy density** γ_{s1} (SI unit: J/m²) and **Phase 2-Solid surface energy density** γ_{s2} (SI unit: J/m²).

This condition applies forces on the wall that balance the surface tension for the prescribed contact angle.

For laminar flow, the Contact Angle node should be used on edges or points where a Navier Slip wall boundary condition is applied adjacent to the free surface. In this case, the Navier Slip condition is required for the contact line to move along the wall.

The representation of the contact angle is dependent on the mesh resolution in the region where the fluid-fluid interface attaches to the wall. If, during the simulation, the contact angle is found to fluctuate around a given value, this indicates that the resolution needs to be improved. This can for example be achieved by adding more mesh boundary layers on the wall.



The moving mesh boundary condition must be compatible with the motion of the wall for the problem to have a valid solution.

Rotating Shaft

Use the **Rotating Shaft** node to define boundaries where the fluid is subjected to a solid shaft rotation. This feature should be applied to boundaries outside of, but connecting to, a **Rotating Domain**. This feature can be applied to rotating shaft boundaries in a free-surface domain modeled by a **Deforming Domain**.

ROTATING SHAFT

Axis of Rotation

Select the **Axis of rotation**. If x -axis is selected, this corresponds to a rotational axis $(1, 0, 0)$ with the origin as the base point. Correspondingly, if y -axis or z -axis is selected, the rotational axis is defined by a unit vector in the selected direction together with a base point at the origin. For **User defined** enter values into these fields: **Rotation axis base point** and **Rotation axis direction**. The z -axis is the default choice and the default values in the fields for **User defined** correspond to those for the z -axis.

Angular Velocity

Select the rotating shaft **Rotational frequency** from the list:

- Select the **Revolution per time** option to manually define the rotational velocity (SI unit: rad/s) and select the **Rotational direction** as **Positive angular velocity** or **Negative angular velocity**. This is the default setting.
- Select the **Angular velocity** option to manually define the angular velocity (SI unit: rad/s).

DISPERSED PHASE BOUNDARY CONDITION

Select an option from the **Dispersed phase boundary condition** list—**No dispersed phase flux** (the default), **Dispersed phase concentration**, **Dispersed phase outlet**, **Dispersed phase flux**, or **Symmetry**.

- For **Dispersed phase concentration** enter a **Dispersed phase volume fraction** ϕ_{d0} (dimensionless). The default is 0. When **Solve for interfacial area** is selected under the **Physical Model** section, select either the **Dispersed phase number density** n_0 (SI unit: $1/\text{m}^3$) or **Diameter of particles/droplets** d_d (SI unit: m) button and enter a value or expression for n_0 (the default is $5 \cdot 10^6 \text{ 1/m}^3$) or d_d (the default is 1 mm).
- For **Dispersed phase flux** enter values or expression for the **Dispersed phase flux** $N_{\phi d}$ (SI unit: m/s) and, if **Solve for interfacial area** is selected under the **Physical Model** section, the **Number density flux** N_n (SI unit: $1/(\text{m}^2 \cdot \text{s})$). The defaults are 0 m/s and 0 $1/(\text{m}^2 \cdot \text{s})$, respectively.



Dispersed Phase Boundary Conditions Equations in the *CFD Module User's Guide*

LOCATION IN USER INTERFACE

Context Menus


Right click the physics interface and select **Rotating Machinery>Rotating Shaft**

Ribbon

Physics tab with the physics interface selected:

Boundaries>Rotating Machinery>Rotating Shaft

Rotating Machinery, Nonisothermal Flow Interfaces



This chapter describes the multiphysics interfaces found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch (). In this chapter:

- Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Algebraic γ Plus Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, L-VEL Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k - ϵ Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Realizable k - ϵ Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k - ω Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, SST Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Low Re k - ϵ Interface
- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k - ω Interface

- Rotating Machinery, Nonisothermal Flow, Turbulent Flow, v2-f Interface
- Domain, Boundary, Edge, Point, and Pair Nodes for the Rotating Machinery, Nonisothermal Flow Interfaces

The Rotating Machinery, Nonisothermal Flow, Laminar and Turbulent Flow Interfaces

Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface



The **Laminar Flow** version of the **Rotating Machinery, Nonisothermal Flow** multiphysics interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate laminar flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Laminar Flow interface and the Heat Transfer in Fluids interface, and can be used to simulate fluid flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



Four study types are available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow.

When adding the multiphysics interface, the **Laminar Flow** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the

Model Builder under **Laminar Flow** — **Fluid Properties**, **Wall**, and **Initial Values**; and under **Heat Transfer in Fluids** — **Fluid**, **Thermal Insulation**, and **Initial Values**.

	<p>In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called Rotating Machinery, Nonisothermal Flow (rmnltf) was added to the Model Builder. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see The Multiphysics Branch and Multiphysics Modeling Approaches in the <i>COMSOL Multiphysics Reference Manual</i>.</p>
	<ul style="list-style-type: none">• Domain, Boundary, Edge, Point, and Pair Nodes for the Rotating Machinery, Nonisothermal Flow Interfaces• Moving Mesh• Theory for the Free Surface Features <p>In the <i>CFD Module User's Guide</i>:</p> <ul style="list-style-type: none">• Modeling Heat Transfer in the CFD Module• The Nonisothermal Flow and Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces• Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Algebraic yPlus Interface

The **Turbulent Flow, Algebraic yPlus** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, Algebraic yPlus interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in

participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



The physics interface solves for conservation of energy, mass, and momentum in fluids, and for conservation of energy in solids. The interface also solves an algebraic equation for the scaled wall distance. Turbulence effects are included using an enhanced viscosity model based on the scaled wall distance, and the physics interface therefore includes a wall distance equation.

Four study types available for this physics interface. Using the **Transient with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When adding the multiphysics interface, the **Turbulent Flow, Algebraic yPlus** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, Algebraic yPlus — Fluid Properties, Wall, and Initial Values**; and under **Heat Transfer in Fluids — Fluid, Thermal Insulation, and Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, L-VEL Interface

The **Turbulent Flow, L-VEL** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, L-VEL interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



The physics interface solves for conservation of energy, mass, and momentum in fluids, and for conservation of energy in solids. The interface also solves an algebraic equation for the scaled fluid flow velocity tangential to the nearest wall. Turbulence effects are included using an enhanced viscosity model based on the scaled wall distance, and the physics interface therefore includes a wall distance equation.

Four study types available for this physics interface. Using the **Transient with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When adding the multiphysics interface, the **Turbulent Flow, L-VEL** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, L-VEL — Fluid Properties, Wall**, and **Initial Values**; and under **Heat Transfer in Fluids— Fluid, Thermal Insulation**, and **Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k - ϵ Interface

The **Turbulent Flow, k - ϵ** version of the **Rotating Machinery, Nonisothermal Flow (rmnntf)** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, k - ϵ interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number fluid flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



The physics interface solves for conservation of energy, mass, and momentum in fluids, and for conservation of energy in solids. Turbulence effects are modeled using the standard two-equation k - ϵ model with realizability constraints. Flow and heat transfer close to walls are modeled using wall functions.

Four study types are available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow.

When adding the multiphysics interface, the **Turbulent Flow, k - ϵ** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, k - ϵ — Fluid Properties, Wall, and Initial Values**; and under **Heat Transfer in Fluids — Fluid, Thermal Insulation, and Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Realizable k - ϵ Interface

The **Turbulent Flow, Realizable k - ϵ** version of the **Rotating Machinery, Nonisothermal Flow (rmnitf)** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, Realizable k - ϵ interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number fluid flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



The physics interface solves for conservation of energy, mass, and momentum in fluids, and for conservation of energy in solids. Turbulence effects are modeled using the Realizable two-equation k - ϵ model. Flow and heat transfer close to walls are modeled using wall functions.

Four study types are available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow.

When adding the multiphysics interface, the **Turbulent Flow, Realizable k - ϵ** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, Realizable k - ϵ** — **Fluid Properties**, **Wall**, and **Initial Values**; and under **Heat Transfer in Fluids** — **Fluid**, **Thermal Insulation**, and **Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, k - ω Interface

The **Turbulent Flow, k - ω** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, k - ω interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number fluid flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



The physics interface solves for conservation of energy, mass, and momentum in fluids and for conservation of energy in solids. Turbulence effects are modeled using the Wilcox revised two-equation k - ω model with realizability constraints. The k - ω model is a so-called low-Reynolds number model, which means that it can resolve the flow and heat transfer all the way down to the wall.

Four study types are available for this physics interface. Using the **Time Dependent** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor** study, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow.

When adding the multiphysics interface, the **Turbulent Flow, k - ω** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, k - ω** — **Fluid Properties**, **Wall**, and **Initial Values**; and under **Heat Transfer in Fluids** — **Fluid**, **Thermal Insulation**, and **Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, SST Interface

The **Turbulent Flow, SST** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, SST interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number fluid flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.

The physics interface solves for conservation of energy, mass, and momentum in fluids and for conservation of energy in solids. Turbulence effects are modeled using the SST two-equation model with realizability constraints. The SST model is a so-called low-Reynolds number model, which means that it can resolve the flow and heat transfer all the way down to the wall. The SST model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.



Four study types available for this physics interface. Using the **Transient with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When adding the multiphysics interface, the **Turbulent Flow, SST** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also

added in the **Model Builder** under **Turbulent Flow, SST — Fluid Properties, Wall, and Initial Values**; and under **Heat Transfer in Fluids— Fluid, Thermal Insulation, and Initial Values**.

See [Rotating Machinery](#), [Nonisothermal Flow](#), [Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Low Re k - ϵ Interface

The **Turbulent Flow, Low Re k - ϵ** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, Low Re k - ϵ interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number fluid flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.



The physics interface solves for conservation of energy, mass, and momentum in fluids and for conservation of energy in solids. Turbulence effects are modeled using the AKN two-equation k - ϵ model with realizability constraints. The AKN model is a so-called low-Reynolds number model, which means that it can resolve the flow and heat transfer all the way down to the wall. The AKN model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

Four study types available for this physics interface. Using the **Transient with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When adding the multiphysics interface, the **Turbulent Flow, low Re k-ε** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, low Re k-ε — Fluid Properties**, **Wall**, and **Initial Values**; and under **Heat Transfer in Fluids — Fluid, Thermal Insulation**, and **Initial Values**.

See [Rotating Machinery](#), [Nonisothermal Flow](#), [Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, Spalart-Allmaras Interface

The **Turbulent Flow, Spalart-Allmaras** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, Spalart-Allmaras interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.

The physics interface solves for conservation of energy, mass, and momentum in fluids and for conservation of energy in solids. Turbulence effects are modeled using the Spalart-Allmaras one-equation model. The Spalart-Allmaras model is a so-called low-Reynolds number model, which means that it can resolve the flow and heat transfer all the way down to the wall. The Spalart-Allmaras model depends on the distance to the closest wall, and the physics interface therefore includes a wall distance equation.



Four study types available for this physics interface. Using the **Transient with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the

inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When adding the multiphysics interface, the **Turbulent Flow, Spalart-Allmaras** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, Spalart-Allmaras** — **Fluid Properties**, **Wall**, and **Initial Values**; and under **Heat Transfer in Fluids** — **Fluid**, **Thermal Insulation**, and **Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Rotating Machinery, Nonisothermal Flow, Turbulent Flow, v2-f Interface

The **Turbulent Flow, v2-f** version of the **Rotating Machinery, Nonisothermal Flow** interface () , found under the **Fluid Flow>Nonisothermal Flow>Rotating Machinery, Nonisothermal Flow** branch () when adding a physics interface, is used to simulate turbulent flow and heat transfer in equipment containing one or more rotating parts.

This physics interface combines the capabilities of the Rotating Machinery, Turbulent Flow, v2-f interface and the Heat Transfer in Fluids interface, and can be used to simulate high Reynolds number flows where the fluid properties depend on the temperature. Furthermore it is also possible to include heat transfer in solids, stationary and rotating, as well as surface-to-surface radiation and radiation in participating media. The physics interface is available in 2D and 3D and supports low Mach number (typically less than 0.3) flows, as well as non-Newtonian fluids.

The physics interface solves for conservation of energy, mass, and momentum in fluids and for conservation of energy in solids. Turbulence effects are modeled using the v2-f transport-equation model with realizability constraints. The v2-f model is a so-called low-Reynolds number model, which means that it can resolve the flow and heat

transfer all the way down to the wall. The physics interface includes a wall distance equation.

Four study types available for this physics interface. Using the **Transient with Initialization** study type, the rotation is achieved through moving mesh functionality, also known as sliding mesh. Using the **Frozen Rotor with Initialization** study type, the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. The **Time Dependent, One-Way Coupled, NITF** and **Frozen Rotor, One-Way Coupled, NITF** study are used for frozen rotor and time dependent nonisothermal flow computations where the model is one-way coupled in the sense that the temperature variations are so small that they do not affect the flow. All study types include an initial Wall Distance Initialization step in order to compute the wall distance. See [Theory for the Rotating Machinery Interfaces](#) in the *CFD Module User's Guide*.

When adding the multiphysics interface, the **Turbulent Flow, v2-f** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Nonisothermal Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, v2-f — Fluid Properties, Wall, and Initial Values**; and under **Heat Transfer in Fluids— Fluid, Thermal Insulation, and Initial Values**.

See [Rotating Machinery, Nonisothermal Flow, Laminar Flow Interface](#) for details.

Domain, Boundary, Edge, Point, and Pair Nodes for the Rotating Machinery, Nonisothermal Flow Interfaces

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using.

HEAT TRANSFER IN FLUIDS (NONISOTHERMAL FLOW)

The available physics features for [The Heat Transfer in Fluids Interface](#) are listed in the section [Feature Nodes for the Heat Transfer in Fluids Interface](#). Also see [Fluid](#) for details about that feature.

LAMINAR FLOW

The available physics features for [The Rotating Machinery, Laminar Flow Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, ALGEBRAIC YPLUS

The available physics features for [The Rotating Machinery, Turbulent Flow, Algebraic yPlus Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, L-VEL

The available physics features for [The Rotating Machinery, Turbulent Flow, L-VEL Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, k-ε

The available physics features for [The Rotating Machinery, Turbulent Flow, k-ε Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, REALIZABLE k-ε

The available physics features for [The Rotating Machinery, Turbulent Flow, Realizable k-ε Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, k-ω

The available physics features for [The Rotating Machinery, Turbulent Flow, k-ω Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, SST

The available physics features for [The Rotating Machinery, Turbulent Flow, SST Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, LOW RE k - ϵ

The available physics features for [The Rotating Machinery, Turbulent Flow, Low Re \$k\$ - \$\epsilon\$ Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).


TURBULENT FLOW, SPALART-ALLMARAS

The available physics features for [The Rotating Machinery, Turbulent Flow, Spalart-Allmaras Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, v_2 -f

The available physics features for [The Rotating Machinery, Turbulent Flow, \$v_2\$ -f Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

Rotating Machinery, Reacting Flow Interfaces


This chapter describes the physics interfaces found under the **Chemical Species Transport>Rotating Machinery, Reacting Flow** branch ()

In this chapter:

- The Rotating Machinery, Reacting Flow, Laminar Flow Interface
- The Rotating Machinery, Reacting Flow, Turbulent Flow, k- ϵ Interface
- The Rotating Machinery, Reacting Flow, Turbulent Flow, k- ω Interface
- The Rotating Machinery, Reacting Flow, Turbulent Flow, SST Interface
- The Rotating Machinery, Reacting Flow, Turbulent Flow, Low Re k- ϵ Interface
- Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery, Reacting Flow Interfaces

The Rotating Machinery, Reacting Flow, Laminar Flow and Turbulent Flow Interfaces

The Rotating Machinery, Reacting Flow, Laminar Flow Interface

The **Laminar Flow** version of the **Rotating Machinery, Reacting Flow** interface () is used to simulate the laminar flow and transport of chemical species in equipment containing one or more rotating parts. Reactions between the participating species can also be studied.

The physics interface combines the functionality of the **Rotating Machinery, Laminar Flow** and the **Transport of Concentrated Species** interfaces. The **Reacting Flow** multiphysics coupling, which is added automatically, couples fluid flow and mass transport. The instantaneous velocity and pressure fields are solved together with an arbitrary number of mass fractions. The momentum balance is governed by the Navier-Stokes equations, and the total mass conservation is governed by the continuity equation. For the chemical species, the governing transport equations include convection, diffusion and, optionally, migration in an electric field.

There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type (see [Frozen Rotor](#) in the *CFD Module User's Guide*), the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.

When adding the multiphysics interface, the **Laminar Flow** and **Transport of Concentrated Species** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Reacting Flow**. The following default nodes are also added in the **Model Builder** under **Laminar Flow** — **Fluid Properties**, **Wall**, and

Initial Values; and under **Transport of Concentrated Species — Transport Properties, No Flux**, and **Initial Values**.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Reacting Flow (rmrf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.





- [The Rotating Machinery, Fluid Flow Interfaces](#)
- [Moving Mesh](#)
- [Theory for the Free Surface Features](#)

In the *CFD Module User's Guide*:

- [The Reacting Flow Multiphysics Interfaces](#)
- [The Transport of Concentrated Species Interface](#)
- [Theory for the Rotating Machinery Interfaces](#)
- [Theory for the Reacting Flow Interfaces](#)
- [Theory for the Transport of Concentrated Species Interface](#)

The Rotating Machinery, Reacting Flow, Turbulent Flow, k-ε Interface

The **Turbulent Flow, k-ε** version of the **Rotating Machinery, Reacting Flow** interface () found under the **Rotating Machinery, Reacting Flow>Turbulent Flow** branch () when adding a physics interface, is used to simulate the turbulent flow and mixing of chemical species in equipment containing one or more rotating parts. Reactions between the participating species can also be studied.

This physics interface combines the functionality of the **Rotating Machinery, Turbulent Flow, k-ε** and the **Transport of Concentrated Species** interfaces. The **Reacting Flow** multiphysics coupling, which is added automatically, couples fluid flow and mass transport. The averaged velocity and pressure fields are solved together with an arbitrary number of averaged mass fractions. The momentum balance is governed by

the averaged Navier-Stokes equations, and the total mass conservation is governed by the continuity equation. The fluid-flow turbulence is modeled using the standard two-equation k - ε model with realizability constraints. For the chemical species, the governing transport equations include convection, diffusion and, optionally, migration in an electric field. Flow close to solid walls is modeled using wall functions for the flow and species transport. Turbulent reactions are modeled using the eddy dissipation model.

There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.



When adding the multiphysics interface, the **Turbulent Flow, k - ε** and **Transport of Concentrated Species** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Reacting Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, k - ε** — **Fluid Properties**, **Wall**, and **Initial Values**; and under **Transport of Concentrated Species** — **Transport Properties**, **No Flux**, and **Initial Values**.

See [The Rotating Machinery, Reacting Flow, Laminar Flow Interface](#) for details.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Reacting Flow (rmrf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.

The Rotating Machinery, Reacting Flow, Turbulent Flow, k - ω Interface

The **Turbulent Flow k - ω** version of the **Rotating Machinery, Reacting Flow** interface () , found under the **Rotating Machinery, Reacting Flow>Turbulent Flow** branch () when adding a physics interface, is used to simulate the turbulent flow and mixing of

chemical species in equipment containing one or more rotating parts. Reactions between the participating species can also be studied.

The physics interface combines the functionality of the **Rotating Machinery, Turbulent Flow k- ω** and the **Transport of Concentrated Species** interfaces. The **Reacting Flow** multiphysics coupling, which is added automatically, couples fluid flow and mass transport. The averaged velocity and pressure fields are solved together with an arbitrary number of averaged mass fractions. The momentum balance is governed by the averaged Navier-Stokes equations, and the total mass conservation is governed by the continuity equation. The fluid-flow turbulence is modeled using the Wilcox revised k - ω model with realizability constraints. The k - ω model is a so-called low-Reynolds number model, which means that it can resolve the velocity, pressure and mass fractions all the way down to the wall. For the chemical species, the governing transport equations include convection, diffusion and, optionally, migration in an electric field. Turbulent reactions are modeled using the eddy dissipation model.

There are two study types available for this physics interface. For the **Time Dependent** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces.



When adding the multiphysics interface, the **Turbulent Flow, k- ω** and **Transport of Concentrated Species** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Reacting Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, k- ω — Fluid Properties, Wall, and Initial Values**; and under **Transport of Concentrated Species — Transport Properties, No Flux, and Initial Values**.

See [The Rotating Machinery, Reacting Flow, Laminar Flow Interface](#) for details.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Reacting Flow (rmrf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.

The Rotating Machinery, Reacting Flow, Turbulent Flow, SST Interface

The **Turbulent Flow, SST** version of the **Rotating Machinery, Reacting Flow** interface () , found under the **Rotating Machinery, Reacting Flow>Turbulent Flow** branch () when adding a physics interface, is used to simulate the turbulent flow and mixing of chemical species in equipment containing one or more rotating parts. Reactions between the participating species can also be studied.

The physics interface combines the functionality of the **Rotating Machinery, Turbulent Flow, SST** and the **Transport of Concentrated Species** interface. The **Reacting Flow** multiphysics coupling, which is added automatically, couples fluid flow and mass transport. The averaged velocity and pressure fields are solved together with an arbitrary number of averaged mass fractions. The momentum balance is governed by the averaged Navier-Stokes equations, and the total mass conservation is governed by the continuity equation. The fluid-flow turbulence is modeled using the SST two-equation model with realizability constraints. The SST model is a so-called low-Reynolds number model, which means that it can resolve the velocity, pressure and mass fractions all the way down to the wall. For this reason the physics interface can be used to study mass transfer at high Schmidt numbers. The SST model depends on the distance to the closest wall, and the physics interface therefore includes a wall distance equation. For the chemical species, the governing transport equations include convection, diffusion and, optionally, migration in an electric field. Turbulent reactions are modeled using the eddy dissipation model.

There are two study types available for this physics interface. For the **Time Dependent with Initialization** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor with Initialization** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. In both study types initialization is performed by first solving for the distance to the closest wall.



When adding the multiphysics interface, the **Turbulent Flow, SST** and **Transport of Concentrated Species** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Reacting Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, SST — Fluid Properties, Wall, and Initial Values**; and under **Transport of Concentrated Species — Transport Properties, No Flux, and Initial Values**.

See [The Rotating Machinery, Reacting Flow, Laminar Flow Interface](#) for details.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Reacting Flow (rmrf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.

The Rotating Machinery, Reacting Flow, Turbulent Flow, Low Re k - ϵ Interface

The **Turbulent Flow, Low Re k - ϵ** version of the **Rotating Machinery, Reacting Flow** interface () , found under the **Rotating Machinery, Reacting Flow>Turbulent Flow** branch () when adding a physics interface, is used to simulate the turbulent flow and mixing of chemical species in equipment containing one or more rotating parts. Reactions between the participating species can also be studied.

The physics interface combines the functionality of the **Rotating Machinery, Turbulent Flow, Low Re k - ϵ** and the **Transport of Concentrated Species** interfaces. It solves for the averaged velocity and pressure fields, together with an arbitrary number of averaged mass fractions. The momentum balance is governed by the averaged Navier-Stokes equations, and the total mass conservation is governed by the continuity equation. The fluid-flow turbulence is modeled using the AKN low-Reynolds number k - ϵ model. The low-Reynolds number model can resolve the velocity, pressure and mass fractions all the way down to the wall. For this reason the physics interface is suited for studying mass transfer at high Schmidt numbers. The AKN model depends on the distance to the closest wall, and the physics interface therefore includes a wall distance equation. For the chemical species, the governing transport equations include convection, diffusion and, optionally, migration in an electric field. Turbulent reactions are modeled using the eddy dissipation model.

There are two study types available for this physics interface. For the **Time Dependent with Initialization** study type the rotation is achieved by moving mesh functionality, also known as sliding mesh. For the **Frozen Rotor with Initialization** study type the rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. In both study types initialization is performed by first solving for the distance to the closest wall.

When adding the multiphysics interface, the **Turbulent Flow, Low Re k - ϵ** and **Transport of Concentrated Species** interfaces are added to the Model Builder. A **Moving Mesh** interface with a **Rotating Domain** node is added automatically in the **Model Builder** under the **Definitions** node. In addition, the **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Reacting Flow**. The following default nodes are also added in the **Model Builder** under **Turbulent Flow, Low Re k - ϵ — Fluid Properties, Wall, and Initial Values**; and under **Transport of Concentrated Species — Transport Properties, No Flux, and Initial Values**.

See [The Rotating Machinery, Reacting Flow, Laminar Flow Interface](#) for details.



In previous versions of COMSOL Multiphysics (prior to version 5.4), a specific physics interface called **Rotating Machinery, Reacting Flow (rmrf)** was added to the **Model Builder**. Now, a predefined multiphysics coupling approach is used, improving the flexibility and design options for your modeling. For specific details, see [The Multiphysics Branch](#) and [Multiphysics Modeling Approaches](#) in the *COMSOL Multiphysics Reference Manual*.

Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery, Reacting Flow Interfaces

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using.

LAMINAR FLOW

The available physics features for [The Rotating Machinery, Laminar Flow Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, k - ϵ

The available physics features for [The Rotating Machinery, Turbulent Flow, \$k\$ - \$\epsilon\$ Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, k - ω

The available physics features for [The Rotating Machinery, Turbulent Flow, \$k\$ - \$\omega\$ Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, SST

The available physics features for [The Rotating Machinery, Turbulent Flow, SST Interface](#) interface are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TURBULENT FLOW, LOW RE k - ϵ

The available physics features for [The Rotating Machinery, Turbulent Flow, Low Re \$k\$ - \$\epsilon\$ Interface](#) are listed in the section [Domain, Boundary, Point, and Pair Nodes for the Rotating Machinery Interfaces](#).

TRANSPORT OF CONCENTRATED SPECIES

The available physics features for [The Transport of Concentrated Species Interface](#) are listed in the section [Domain, Boundary, and Pair Nodes for the Transport of Concentrated Species Interface](#) in the *CFD Module User's Guide*.

I n d e x

- A** Application Libraries window 19
 - application library examples
 - rmspf interfaces 71
- B** boundary nodes
 - rmmf interfaces 103
 - rmspf interfaces 84
- C** CFL number
 - pseudo time stepping, and 72, 93
 - contact angle (node) 88, 106
- D** documentation 18
 - domain nodes
 - rmmf interfaces 103
 - rmspf interfaces 84
- E** emailing COMSOL 20
- F** free surface (node) 86, 104
 - free surface domain 62
- G** geometry frames 62
- I** internet resources 17
- K** knowledge base, COMSOL 21
- L** laminar flow (rmnitf) interface 111
 - laminar flow (rmrf) interface 126
 - local CFL number 72, 93
- M** material frames 62
 - mesh frames 62
 - moving mesh 62
 - MPH-files 19
- P** pair nodes
 - rmmf interfaces 103
 - rmspf interfaces 84
 point nodes
 - rmmf interfaces 103
 - rmspf interfaces 84
- R** rotating machinery, laminar flow (rmspf)
 - interface 70
 rotating machinery, mixture model, algebraic yplus interface 94
 - rotating machinery, mixture model, k-eps interface 96
 - rotating machinery, mixture model, k-omega interface 98
 - rotating machinery, mixture model, laminar flow interface 92
 - rotating machinery, mixture model, Low Re k-eps interface 100
 - rotating machinery, mixture model, l-vel interface 95
 - rotating machinery, mixture model, realizable k-eps interface 97
 - rotating machinery, mixture model, Spalart-Allmaras interface 101
 - rotating machinery, mixture model, SST interface 99
 - rotating machinery, mixture model, turbulent flow interface 96
 - rotating machinery, mixture model, v2-f interface 102
 - rotating machinery, turbulent flow (rmspf) interface 72–75, 77–80, 82
 - rotating shaft (node) 89, 106
- S** spatial frames 62
- T** technical support, COMSOL 20
 - turbulent flow k-omega (rmrf) interface 128
 - turbulent flow low re k-epsilon (rmnitf) interface 119–121
 - turbulent flow low re k-epsilon (rmrf) interface 130–131

- turbulent flow sst interface 118
- turbulent flow, algebraic yplus interface 112
- turbulent flow, k-epsilon (rmnitf) interface 115
- turbulent flow, k-epsilon (rmrf) interface 127
- turbulent flow, k-omega (rmnitf) interface 117
- turbulent flow, l-vel interface 113
- turbulent flow, realizable k-epsilon interface 116

W websites, COMSOL 20