

CFD Module

Application Library Manual





CFD Module Application Library Manual

© 1998–2016 COMSOL

Protected by U.S. 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; and 9,323,503. 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, Capture the Concept, COMSOL Desktop, LiveLink, and COMSOL Server 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 5.2a

Contact Information

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

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

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

Part number: CM021304



Airflow Over an Ahmed Body

Introduction

This example describes how to calculate the turbulent flow field around a simple car-like geometry using the CFD Module's Turbulent Flow, k- ε interface. Detailed instructions guide you through the different steps of the modeling process in COMSOL Multiphysics.

Model Definition

The Ahmed body represents a simplified, ground vehicle geometry of a bluff body type. Its shape is simple enough to allow for accurate flow simulation but retains some important practical features relevant to automobile bodies. The geometry was first defined by Ahmed, who also measured its aerodynamic properties in wind-tunnel experiments (Ref. 1). Further experiments have also been performed by Lienhart and Becker (Ref. 2). The Ahmed body has become a popular benchmark case for RANS models (Ref. 3).

GEOMETRY

The Ahmed body is presented in Figure 1. The total length (L) of the body is 1.044 m from front to end. It is 0.288 m in height and 0.389 m in width. Cylindrical legs 0.05 m in length are attached to the bottom surface. The angle of the rear slanting surface is typically varied between 0 and 40 degrees. This particular geometry has a slant angle of 25 degrees, which is the same slant angle used in Ref. 3.



Figure 1: Ahmed body with 25 degree slant of the rear face.

The body is placed in a flow domain that is 8L-by-2L-by-2L (length-by-width-by-height), with its front positioned 2L from the flow inlet face. Mirror symmetry reduces the

computational domain by half, as shown in Figure 2.



Figure 2: The size of the computational domain is reduced by mirror symmetry.

TURBULENCE MODEL

The Reynolds number based on the length of the body, L, and the inlet velocity is $2.77 \cdot 10^6$ which means that the flow is turbulent. The *k*- ε turbulence model is applied to account for the turbulence. The *k*- ε turbulence model is described in the theory section for the Turbulent Flow interfaces in the *CFD Module User's Guide*.

BOUNDARY CONDITIONS

Air enters the computational domain at a freestream velocity u_{∞} =40 m/s normal to the inlet surface. Experimental inlet conditions from Ref. 3 are used for the velocity and turbulent kinetic energy. To obtain a condition for ε , Ref. 3 suggests to set μ_T =10· μ at the inlet. At the outlet, a Pressure condition is applied.

The floor of the flow domain and surface of the Ahmed body are described by wall functions. Wall functions could also be applied to the outer wall and the ceiling of the wind tunnel. Their main effect on the flow around the body is however to keep the flow contained, therefore it suffices to model these as slip walls. The temperature is assumed to be 293 K and the reference pressure is 1 atm.

MESHING

A common mesh size in Ref. 3 is half a million cells for simulations with wall functions. However, those simulations do not include the stilts (the legs that support the body), and the computational domains are smaller. Hence, you can expect to need an even larger mesh in this simulation to resolve the flow. How large is however difficult to know in advance.

There are two important aspects of the meshing. The first is to resolve the flow in the wake. To achieve this, additional mesh control entities are introduced in the geometry. These entities are advantageous to normal geometrical entities since they are removed whence they are completely meshed. A smoothing algorithm then smooths the mesh locally in order to minimize gradients in the mesh size. Also, it is easier to introduce boundary layer mesh when the control entities are removed.

Results and Discussion

A key figure for the Ahmed body is the total drag coefficient, $C_{\rm D}$, which is defined as

$$\frac{F}{A_{\rm p}} = C_{\rm D} \frac{\rho u_{\infty}^2}{2} \tag{1}$$

where *F* is the total drag force on the body, A_p is area of the body projected on a plane perpendicular to the flow direction (that is, the *xz*-plane), ρ is the density (approximately equal to 1.2 kg/m³), and u_{∞} is the freestream velocity (equal to 40 m/s). A_p can be calculated from geometrical data and is equal to 0.115 m² including the stilts. The contributions to C_D are commonly reported as the pressure coefficients on front, slant, and base and the skin friction drag coefficient. These numbers are given in Table 1. Note that the numbers given by the postprocessing tools correspond to half the body, and hence, A_p must be replaced by $A_p/2$ when calculating the entries of Table 1.

	CP FRONT	CP SLANT	CP BASE	SKIN FRICTION	TOTAL DRAG
Measurements	0.020	0.140	0.070	0.055	0.285
k-ε	0.039	0.106	0.071	0.039	0.276

TABLE I: DF	AG CO	EFFICIENT	- 5
-------------	-------	-----------	-----

As can be seen, most contributions are in reasonable agreements with experiments. The total drag is well predicted, but the individual contributions deviate from experimental values.

The pressure coefficient on the front is too high and the skin friction too low. Ref. 4 uses two different versions of the k-e model and two different wall function formulations and

all combinations show this behavior. It can probably be attributed to the fact that wall functions are not very good at predicting the transition observed in the experiments to take place on the front and roof of the body.

The low value of the slant pressure drag coefficient can be understood by looking at Figure 3, which shows streamlines in the symmetry plane. Experimental results indicate that the flow along the slant is attached almost everywhere and that there are two small recirculation regions behind the base. The computational results capture this behavior, but the extent of the recirculation zones is somewhat overpredicted. The pressure drag coefficient, especially for the slant, is very sensitive to the exact shape and location of the recirculation regions.



Figure 3: Streamlines in the symmetry plane.

Figure 4 shows a 3D plot of the streamlines behind the Ahmed body. The thickness of the lines is given by the turbulent kinetic energy. The most notable feature of the flow field is an "empty" region behind the body. The streamlines on the edge of the region are thick but with low velocity magnitude. This region is constituted of the recirculation vortices visible in Figure 3. The region ends when vortices from the trailing edges of the body

merge into two counter rotating vortices (only one vortex is visible because the other vortex is on the other side of the symmetry plane).



Figure 4: Streamlines behind the Ahmed body. The streamlines are colored by the velocity magnitude and their thickness is proportional to the turbulent kinetic energy.

More details are visible in Figure 5 and Figure 6, which show arrow plots of the velocity in the *xz*-plane 80 mm and 200 mm downstream of the body, respectively.



Figure 5: Velocity in the xz-plane at y = L + 0.08 m.

The flow pattern 80 mm downstream of the body shows two major vortices, one emanating from the outer edge of the slant and one emanating from the interaction between the floor and the stilts. The flow is qualitatively equal to the experimental results (Ref. 2). There are however quantitative differences. The upper vortex is smaller compared

to experiments while the lower vortex is more pronounced than in the experiments.



Figure 6: Velocity in the xz-plane at y = L + 0.20 m.

The flow pattern 200 mm downstream of the body shows that one major vortex is beginning to form but remains of the separate vortices can still be detected. The formation is, however, not proceeded as far as in the experiments.

In conclusion, the major features of the flow are well-captured by the k- ε model, but there are details that deviate from experimental data. This finding is in agreement with other RANS simulations of the Ahmed body (Ref. 3).

References

1. S.R. Ahmed, G. Ramm, and G. Faltin, "Some Salient Features of the Time-Averaged Ground Vehicle Wake," *SAE Technical Paper 840300*, 1984.

2. H. Lienhart and S. Becker, "Flow and Turbulence Structure in the Wake of a Simplified Car Model," *SAE 2003 World Congress*, SAE Paper 2003-01-0656, Detroit, Michigan, 2003.

3. 9th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling, Darmstadt University of Technology, Germany, 2001.

4. T.J. Craft, S.E. Gant, H. Iacovides, B.E. Launder, and C.M.E. Robinson, "Computational Study of Flow Around the 'Ahmed' Car Body," *9th ERCOFTAC/ IAHR Workshop on Refined Turbulence Modelling*, 2001.

Application Library path: CFD_Module/Single-Phase_Benchmarks/ahmed_body

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ε (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
L	1.044[m]	I.044 m	Body length
D	0.389[m]	0.389 m	Body width
H_body	0.288[m]	0.288 m	Body height
C1	0.05[m]	0.05 m	Clearance
Sl	0.222[m]	0.222 m	Slant length

Name	Expression	Value	Description
Sb	H_body+Cl-Sl*sin(25[deg])	0.2442 m	Slant base
Rl	L-Sl*cos(25[deg])	0.8428 m	Roof length

Interpolation 1 (int1)

- I On the Home toolbar, click Functions and choose Global>Interpolation.
- 2 In the Settings window for Interpolation, locate the Definition section.
- 3 From the Data source list, choose File.
- 4 Click Browse.
- 5 Browse to the application's Application Libraries folder and double-click the file ahmed_body_kin.txt.
- 6 Click Import.
- 7 Find the Functions subsection. In the table, enter the following settings:

Function name	Position in file
kin	1

- 8 Locate the Units section. In the Arguments text field, type m.
- **9** In the **Function** text field, type m^2/s^2.

GEOMETRY I

Import I (imp1)

- I On the Home toolbar, click Import.
- 2 In the Settings window for Import, locate the Import section.
- 3 Click Browse.
- **4** Browse to the application's Application Libraries folder and double-click the file ahmed body.mphbin.
- 5 Click Import.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

Block I (blk I)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type 2*L.
- 4 In the **Depth** text field, type 8*L.

- 5 In the **Height** text field, type 2*L.
- 6 Locate the Position section. In the x text field, type -L.
- 7 In the y text field, type -2*L.
- 8 Right-click Block I (blkI) and choose Build Selected.
- 9 Click the Go to Default 3D View button on the Graphics toolbar.

Block 2 (blk2)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type L.
- 4 In the **Depth** text field, type 8*L.
- 5 In the Height text field, type 2*L.
- 6 Locate the Position section. In the x text field, type -L.
- 7 In the **y** text field, type -2*L.

Difference I (dif1)

- I Right-click Block 2 (blk2) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- **3** Select the object **blk1** only.
- 4 In the Settings window for Difference, locate the Difference section.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the objects **blk2** and **imp1** only.

Cylinder I (cyl1)

- I Right-click Difference I (difl) and choose Build Selected.
- 2 On the Geometry toolbar, click Cylinder.
- 3 In the Settings window for Cylinder, locate the Size and Shape section.
- 4 In the Radius text field, type 2.2*L.
- **5** In the **Height** text field, type L.
- 6 Locate the Position section. In the y text field, type 0.2*L.
- 7 In the z text field, type -0.1*L.
- 8 Locate the Axis section. From the Axis type list, choose x-axis.

Convert to Surface 1 (csur1)

I On the Geometry toolbar, click Conversions and choose Convert to Surface.

- 2 Select the object cyll only.
- 3 Click the Wireframe Rendering button on the Graphics toolbar.

Delete Entities I (del I)

- I In the Model Builder window, right-click Geometry I and choose Delete Entities.
- 2 On the object csurl, select Boundaries 1 and 3–6 only.

These are all surfaces of the cylinder, except the curved surface behind the body.

Union I (uni I)

- I Right-click Component I (comp1)>Geometry I>Delete Entities I (del1) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 3 Click in the Graphics window and then press Ctrl+A to select both objects.
- 4 Right-click Union I (unil) and choose Build Selected.

Delete Entities 2 (del2)

- I Right-click Geometry I and choose Delete Entities.
- 2 On the object unil, select Boundaries 10 and 16 only.

These are the boundaries that protrude above and beneath the channel.

Hexahedron I (hex1)

- I Right-click Component I (compl)>Geometry I>Delete Entities 2 (del2) and choose Build Selected.
- 2 On the Geometry toolbar, click More Primitives and choose Hexahedron.
- 3 In the Settings window for Hexahedron, locate the Vertices section.
- 4 In row I, set y to L and z to C1.
- **5** In row **2**, set **y** to **2***L and **z** to **C**1.
- 6 In row 3, set x to D/2, y to 2*L, and z to Cl.
- 7 In row 4, set x to D/2, y to L, and z to Cl.
- 8 In row 5, set y to L and z to Sb.
- 9 In row 6, set y to 2*L and z to Sb.
- **10** In row **7**, set **x** to D/2, **y** to 2*L, and **z** to Sb.
- II In row 8, set \mathbf{x} to D/2, \mathbf{y} to L, and \mathbf{z} to Sb.

Hexahedron 2 (hex2)

I On the Geometry toolbar, click More Primitives and choose Hexahedron.

- 2 In the Settings window for Hexahedron, locate the Vertices section.
- 3 In row I, set y to L and z to Sb.
- 4 In row 2, set y to 2*L and z to Sb.
- 5 In row 3, set x to D/2, y to 2*L, and z to Sb.
- 6 In row 4, set x to D/2, y to L, and z to Sb.
- 7 In row 5, set y to L and z to H_body+Cl+0.01[m].
- 8 In row 6, set y to 2*L and z to H_body+Cl+0.01[m].
- 9 In row 7, set x to D/2, y to 2*L, and z to H_body+Cl+0.01[m].
- **IO** In row **8**, set **x** to D/2, **y** to L, and **z** to H_body+Cl+0.01[m].

Hexahedron 3 (hex3)

- I On the Geometry toolbar, click More Primitives and choose Hexahedron.
- 2 In the Settings window for Hexahedron, locate the Vertices section.
- 3 In row I, set y to L and z to Sb.
- 4 In row 2, set y to L and z to H_body+Cl+0.01[m].
- **5** In row **3**, set **x** to D/2, **y** to L, and **z** to H_body+Cl+0.01[m].
- 6 In row 4, set \mathbf{x} to D/2, \mathbf{y} to L, and \mathbf{z} to Sb.
- 7 In row 5, set y to R1 and z to H_body+C1.
- 8 In row 6, set y to R1 and z to H_body+C1+0.01[m].
- 9 In row 7, set x to D/2, y to R1, and z to H_body+C1+0.01[m].
- **IO** In row **8**, set **x** to D/2, **y** to R1, and **z** to H_body+C1.

Union 2 (uni2)

- I On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 2 Select the objects hex1, hex2, and hex3 only.
- 3 In the Settings window for Union, locate the Union section.
- 4 Clear the Keep interior boundaries check box.

Ignore Edges 1 (ige1)

- I On the Geometry toolbar, click Virtual Operations and choose Ignore Edges.
- **2** On the object fin, select Edges 26, 27, 30, 34, 44, 49, 54, 55, 64, and 65 only.

Mesh Control Domains 1 (mcd1)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Domains.
- 2 On the object igel, select Domain 2 only.

Mesh Control Faces 1 (mcf1)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Faces.
- 2 On the object mcd1, select Boundary 12 only.
- 3 On the Geometry toolbar, click Build All.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 5 Click the Go to Default 3D View button on the Graphics toolbar.
- 6 In the Model Builder window, collapse the Geometry I node.
- 7 Click the **Transparency** button on the **Graphics** toolbarto return to the default state.

The model geometry is now complete.



Create an explicit selection of the boundaries of the body.

DEFINITIONS

Explicit I

- I On the Definitions toolbar, click Explicit.
- 2 In the Settings window for Explicit, locate the Input Entities section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 Click the Select Box button on the Graphics toolbar.

- **5** Select Boundaries 5–11 and 13–16 only.
- 6 Right-click Explicit I and choose Rename.
- 7 In the Rename Explicit dialog box, type Body in the New label text field.
- 8 Click OK.
- 9 Click the Transparency button on the Graphics toolbar.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

TURBULENT FLOW, $K-\epsilon$ (SPF)

Wall 2

- I On the Physics toolbar, click Boundaries and choose Wall.
- 2 In the Settings window for Wall, locate the Boundary Condition section.
- **3** From the **Boundary condition** list, choose **Slip**.
- 4 Select Boundaries 4 and 17 only.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- 2 Select Boundary 1 only.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 2 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- 4 Click the Specify turbulence variables button.
- **5** In the k_0 text field, type kin(x,z).
- **6** In the ε₀ text field, type spf.C_mu*kin(x,z)^2*spf.rho/(10*1.814e-5[Pa*s]).
- 7 Locate the Velocity section. Click the Velocity field button.

8 Specify the **u**₀ vector as

0	x
40[m/s]	у
0	z

Change to unidirectional constraints to avoid reaction forces in the pressure from the constraint for $\boldsymbol{\epsilon}$

- 9 In the Model Builder window's toolbar, click the Show button and select Advanced Physics Options in the menu.
- 10 Click to expand the Constraint settings section. Locate the Constraint Settings section. From the Apply reaction terms on list, choose Individual dependent variables.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 12 only.

MESH I

In the Model Builder window, under Component I (comp1) right-click Mesh I and choose Edit Physics-Induced Sequence.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- 4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.1.
- 5 In the Minimum element size text field, type 0.0025.
- 6 In the **Curvature factor** text field, type 0.4.
- 7 In the Resolution of narrow regions text field, type 0.5.

Size I

- I In the Model Builder window, under Component I (compl)>Mesh I click Size I.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 Click Clear Selection.
- 4 Select Boundaries 23, 25, and 27 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.

- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.05.
- 8 Click Build Selected.

Size 2

- I In the Model Builder window, right-click Mesh I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- **4** Select Boundary **3** only.
- 5 Locate the Element Size section. Click the Custom button.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.035.

Size 3

- I Right-click Mesh I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 Select Boundaries 10 and 11 only.
- 5 Locate the Element Size section. Click the Custom button.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.01.
- Size 4
- I Right-click Mesh I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 Select Boundaries 5–9, 13, and 16 only.
- 5 Locate the **Element Size** section. Click the **Custom** button.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.02.

Size 5

- I Right-click Mesh I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Edge**.

- 4 Select Edges 35 and 36 only.
- 5 Locate the Element Size section. Click the Custom button.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.01.

Corner Refinement I

In the Model Builder window, under Component I (compl)>Mesh I right-click Corner Refinement I and choose Disable.

Free Tetrahedral I

- I In the Model Builder window, under Component I (compl)>Mesh I click Free Tetrahedral I.
- 2 In the Settings window for Free Tetrahedral, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Select Domain 3 only.

Size 1

- I Right-click Component I (comp1)>Mesh 1>Free Tetrahedral I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. Select the **Maximum element growth rate** check box.
- **5** In the associated text field, type **1.03**.

Free Tetrahedral I

- I In the Model Builder window, under Component I (comp1)>Mesh I click Free Tetrahedral I.
- 2 In the Settings window for Free Tetrahedral, click Build Selected.

Free Tetrahedral 2

- I In the Model Builder window, right-click Mesh I and choose Free Tetrahedral.
- 2 In the Settings window for Free Tetrahedral, locate the Domain Selection section.
- **3** From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 1 only.
- 5 Click Build Selected.

Boundary Layers 1

I In the Settings window for Boundary Layers, locate the Domain Selection section.

2 Click Clear Selection.

3 Select Domain 1 only.

Boundary Layer Properties 1

- I In the Model Builder window, expand the Boundary Layers I node, then click Boundary Layer Properties I.
- 2 In the Settings window for Boundary Layer Properties, locate the Boundary Layer Properties section.
- 3 In the Number of boundary layers text field, type 6.
- 4 In the Thickness adjustment factor text field, type 1.5.

Boundary Layers 1

- I In the Model Builder window, under Component I (compl)>Mesh I click Boundary Layers I.
- 2 In the Settings window for Boundary Layers, click Build Selected.

Swept I

- I In the Model Builder window, right-click Mesh I and choose Swept.
- 2 In the Settings window for Swept, locate the Domain Selection section.
- 3 From the Geometric entity level list, choose Domain.
- **4** Select Domain 2 only.

Distribution I

- I Right-click Component I (compl)>Mesh I>Swept I and choose Distribution.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 28.
- 5 In the Element ratio text field, type 6.
- 6 In the Model Builder window, right-click Mesh I and choose Free Tetrahedral.
- 7 In the Settings window for Mesh, click Build All.
- 8 In the Model Builder window, collapse the Mesh I node.

COMPONENT I (COMPI)

Mesh 2 On the Mesh toolbar, click Add Mesh.

MESH 2

In the Model Builder window, under Component I (compl)>Meshes right-click Mesh 2 and choose More Operations>Reference.

Reference I

- I In the Settings window for Reference, locate the Reference section.
- 2 From the Mesh list, choose Mesh I.

Scale I

- I Right-click Component I (compl)>Meshes>Mesh 2>Reference I and choose Scale.
- 2 In the Settings window for Scale, locate the Scale section.
- 3 In the Element size scale text field, type 2.
- 4 In the Model Builder window, click Mesh 2.
- 5 In the Settings window for Mesh, click Build All.
- 6 In the Model Builder window, collapse the Mesh 2 node.

COMPONENT I (COMPI)

Mesh 3

On the Mesh toolbar, click Add Mesh.

MESH 3

In the Model Builder window, under Component I (compl)>Meshes right-click Mesh 3 and choose More Operations>Reference.

Reference I

- I In the Settings window for Reference, locate the Reference section.
- 2 From the Mesh list, choose Mesh 2.

Scale 1

- I Right-click Component I (compl)>Meshes>Mesh 3>Reference I and choose Scale.
- 2 In the Settings window for Scale, locate the Scale section.
- **3** In the **Element size scale** text field, type **2**.
- 4 In the Model Builder window, click Mesh 3.
- 5 In the Settings window for Mesh, click Build All.
- 6 In the Model Builder window, collapse the Mesh 3 node.

COMPONENT I (COMPI)

- I In the Model Builder window, collapse the Component I (compl)>Meshes node.
- 2 In the Model Builder window, collapse the Component I (comp1)>Meshes node.Show Advanced Study Options to be able to apply the manual multigrid levels.
- **3** In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Study Options** in the menu.

STUDY I

Step 1: Stationary

- I In the Model Builder window, expand the Study I node.
- 2 Right-click Step 1: Stationary and choose Multigrid Level.
- 3 In the Settings window for Multigrid Level, locate the Mesh Selection section.
- **4** In the table, enter the following settings:

Geometry	Mesh	
Geometry I	mesh2	

Solution 1 (soll)

- I In the Model Builder window, right-click Step I: Stationary and choose Multigrid Level.
- 2 On the Study toolbar, click Show Default Solver.
- 3 In the Model Builder window, expand the Solution I (soll) node.
- 4 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I>Iterative I node, then click Multigrid I.
- 5 In the Settings window for Multigrid, locate the General section.
- 6 From the Hierarchy generation method list, choose Manual.
- 7 In the Model Builder window, collapse the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I>Iterative I node.
- 8 In the Model Builder window, collapse the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I>Iterative I node.
- 9 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (solI)>Stationary Solver I>Iterative 2 node, then click Multigrid I.
- 10 In the Settings window for Multigrid, locate the General section.
- II From the Hierarchy generation method list, choose Manual.

- 12 In the Model Builder window, collapse the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I>Iterative 2 node.
- 13 In the Model Builder window, collapse the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I>Iterative 2 node.
- 14 In the Model Builder window, collapse the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I node.
- I5 In the Model Builder window, collapse the Study I>Solver Configurations>Solution I (soll)>Stationary Solver I node.
- **I6** In the **Model Builder** window, collapse the **Solution I (soll)** node.
- **I7** In the **Model Builder** window, collapse the **Solution I (soll)** node.
- **18** On the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

It is advisable to disable the Automatic update of plots when working with large 3D models.

- I In the Model Builder window, click Results.
- 2 In the Settings window for Results, locate the Result Settings section.
- **3** Clear the **Automatic update of plots** check box.

Investigate the lift-off in viscous units to verify that the Wall Resolution is sufficient.

Wall Resolution (spf)

I In the Model Builder window, under Results click Wall Resolution (spf).

2 On the Wall Resolution (spf) toolbar, click Plot.

The wall lift-off is reasonably close to the target value of 11.06 on most of the body, and can hence be considered to be acceptable.



Slice

- I In the Model Builder window, expand the Results>Velocity (spf) node, then click Slice.
- 2 In the Settings window for Slice, locate the Plane Data section.
- 3 From the Entry method list, choose Coordinates.
- 4 In the **x-coordinates** text field, type 0.15.

Velocity (spf)

- I In the Model Builder window, under Results click Velocity (spf).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 From the View list, choose View 3.

4 On the Velocity (spf) toolbar, click Plot.

The slice plot of the velocity clearly shows the recirculation zone behind the body. The result looks smooth which further supports the assumption that the resolution is acceptable.



To evaluate the input to calculate the entries in Table 1, perform the following steps:

RESULTS

Surface Integration 1

- I On the Results toolbar, click More Derived Values and choose Integration>Surface Integration.
- 2 In the Settings window for Surface Integration, locate the Expressions section.
- **3** In the table, enter the following settings:

Expression	Unit	Description
nymesh*p		

- **4** Select Boundaries 5–7 and 13 only.
- 5 Click Evaluate.
- 6 Locate the Selection section. Click Clear Selection.

- **7** Select Boundary 10 only.
- 8 Click Evaluate.
- 9 Click Clear Selection.
- **IO** Select Boundary 11 only.
- II Click Evaluate.
- 12 From the Selection list, choose Body.
- **I3** Click **Evaluate**.

Surface Integration 2

- I On the Results toolbar, click More Derived Values and choose Integration>Surface Integration.
- 2 Click the Wireframe Rendering button on the Graphics toolbar.
- 3 In the Settings window for Surface Integration, locate the Selection section.
- **4** From the **Selection** list, choose **Body**.
- 5 Locate the Expressions section. In the table, enter the following settings:

Expression	Unit	Description
<pre>spf.rho*spf.u_tau*((v-spf.nymesh*(u* spf.nxmesh+v*spf.nymesh+w*spf.nzmesh)))/</pre>	Ν	
spf.uPlus		

The wall function expression will be use directly for maximum accuracy.

6 Click Evaluate.

The following steps reproduce Figure 3:

Surface 2

- I On the Results toolbar, click More Data Sets and choose Surface.
- 2 In the Settings window for Surface, locate the Parameterization section.
- 3 From the x- and y-axes list, choose yz-plane.
- **4** Select Boundary 1 only.

2D Plot Group 4

- I On the Results toolbar, click 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Data set list, choose Surface 2.

Streamline 1

- I Right-click **2D Plot Group 4** and choose **Streamline**.
- 2 In the Settings window for Streamline, locate the Expression section.
- **3** In the **x component** text field, type **v**.
- 4 In the y component text field, type w.
- 5 Locate the Streamline Positioning section. In the Points text field, type 31.
- 6 On the 2D Plot Group 4 toolbar, click Plot.
- 7 In the Model Builder window, expand the Results>Views node.

2D Plot Group 4

- I In the Model Builder window, expand the Results>Views>View 2D 5 node, then click Results>2D Plot Group 4.
- 2 In the Settings window for 2D Plot Group, locate the Plot Settings section.
- 3 From the View list, choose View 2D 5.
- 4 On the 2D Plot Group 4 toolbar, click Plot.







On the **Results** toolbar, click **More Data Sets** and choose **Solution**.

Selection

- I On the **Results** toolbar, click **Selection**.
- 2 In the Settings window for Selection, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- **4** From the **Selection** list, choose **Body**.
- **5** Select Boundaries 3, 5–11, and 13–16 only.

3D Plot Group 5

On the **Results** toolbar, click **3D Plot Group**.

Surface 1

- I In the Model Builder window, right-click 3D Plot Group 5 and choose Surface.
- 2 In the Settings window for Surface, locate the Data section.
- 3 From the Data set list, choose Study I/Solution I (2) (soll).
- **4** Locate the **Expression** section. In the **Expression** text field, type **1**.
- 5 Locate the Coloring and Style section. Clear the Color legend check box.
- 6 From the Coloring list, choose Uniform.
- 7 From the Color list, choose Gray.

3D Plot Group 5

- I In the Model Builder window, under Results click 3D Plot Group 5.
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 Clear the Plot data set edges check box.

Streamline 1

- I Right-click Results>3D Plot Group 5 and choose Streamline.
- 2 In the Settings window for Streamline, locate the Streamline Positioning section.
- **3** From the **Positioning** list, choose **Start point controlled**.
- 4 From the Entry method list, choose Coordinates.
- 5 In the x text field, type range(0.01,0.03,0.16) range(0.01,0.03,0.16) range(0.01,0.03,0.16) range(0.01,0.03,0.16) range(0.01,0.03,0.16).
- 6 In the y text field, type -0.5*L.
- 7 In the z text field, type 0.02*1^range(1,6) 0.08*1^range(1,6) 0.14*1^range(1,
 6) 0.2*1^range(1,6) 0.26*1^range(1,6).
- 8 Locate the Coloring and Style section. From the Line type list, choose Tube.
- 9 In the Tube radius expression text field, type k*1[s^2/m].

IO Select the **Radius scale factor** check box.

II In the associated text field, type 3e-4.

Color Expression 1

- I Right-click Results>3D Plot Group 5>Streamline I and choose Color Expression.
- 2 On the 3D Plot Group 5 toolbar, click Plot.

The following steps will reproduce figures Figure 5 and Figure 6:

Cut Plane 1

- I On the **Results** toolbar, click **Cut Plane**.
- 2 In the Settings window for Cut Plane, locate the Plane Data section.
- 3 From the Plane list, choose zx-planes.
- 4 In the **y-coordinate** text field, type L+0.08.

Cut Plane 2

- I On the **Results** toolbar, click **Cut Plane**.
- 2 In the Settings window for Cut Plane, locate the Plane Data section.
- 3 From the Plane list, choose zx-planes.
- 4 In the **y-coordinate** text field, type L+0.2.

3D Plot Group 6

- I On the **Results** toolbar, click **3D Plot Group**.
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 Clear the Plot data set edges check box.

Surface 1

In the Model Builder window, under Results>3D Plot Group 5 right-click Surface I and choose Copy.

3D Plot Group 6

In the Model Builder window, under Results right-click 3D Plot Group 6 and choose Paste Surface.

Arrow Surface 1

- I Right-click **3D Plot Group 6** and choose **Arrow Surface**.
- 2 In the Settings window for Arrow Surface, locate the Data section.
- 3 From the Data set list, choose Cut Plane I.
- 4 Locate the Expression section. In the y component text field, type 0.

- 5 Locate the Coloring and Style section. From the Arrow length list, choose Logarithmic.
- 6 In the Range quotient text field, type 500.
- 7 Select the Scale factor check box.
- 8 In the associated text field, type 1.25e-3.
- 9 In the Number of arrows text field, type 2500.

IO From the **Color** list, choose **Black**.

Filter I

- I Right-click Results>3D Plot Group 6>Arrow Surface I and choose Filter.
- 2 In the Settings window for Filter, locate the Element Selection section.
- **3** In the Logical expression for inclusion text field, type (x<0.35)*(z<0.45).

3D Plot Group 6

- I In the Model Builder window, under Results click 3D Plot Group 6.
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- **3** From the **View** list, choose **View 7**.
- 4 Click to expand the Title section. From the Title type list, choose Manual.
- 5 In the Title text area, type Arrow: Velocity in xz-plane.
- 6 On the 3D Plot Group 6 toolbar, click Plot.
- 7 Right-click Results>3D Plot Group 6 and choose Duplicate.

Arrow Surface 1

- I In the Model Builder window, expand the 3D Plot Group 7 node, then click Arrow Surface
 I.
- 2 In the Settings window for Arrow Surface, locate the Data section.
- 3 From the Data set list, choose Cut Plane 2.
- 4 On the **3D Plot Group 7** toolbar, click **Plot**.

30 | AIRFLOW OVER AN AHMED BODY



Flow in an Airlift Loop Reactor

Introduction

This example illustrates multiphase flow modeling in an airlift loop reactor. Air bubbles are injected through two frits at the bottom of a water-filled reactor. Due to buoyancy, the bubbles rise, inducing a circulating motion in the liquid. There is no mass transfer between the phases.

Model Definition

The model parameters are taken from the experimental work presented in Ref. 1, and are summarized in Table 1. The liquid phase in the reactor is water while the gas phase is air.

PROPERTY	VALUE	DESCRIPTION
Η	1.75 m	Reactor height
W	0.5 m	Reactor width
T	0.08 m	Reactor thickness
d_{b}	3·10 ⁻³ m	Bubble diameter
R	0.02 m	Frit radius
L	0.16 m	Width of riser and downcomer channels
$V_{\rm in}$	0.015 m/s	Superficial velocity at inlet
$C_{ m w}$	5·10 ⁴ kg/(m ³ ·s)	Slip-velocity proportionality constant
$\rho_{g,in}$	0.9727 kg/m ³	Gas density at inlet

TABLE I: MODEL DATA.

BOUNDARY CONDITIONS

Inlet Boundary Condition

Two frits with radius 0.02 m are located at the bottom of the reactor (see Figure 1). Air bubbles with a diameter of 3 mm with a superficial speed of 0.015 m/s are injected through the frits. For the liquid, the frits are described by a wall condition.

Outlet Boundary Condition

The top of the geometry (*xz*-plane, y = 1.75 m) is a free surface. The surface motion is neglected and the surface is instead approximated by a slip condition for the liquid. The gas is free to exit the reactor through this boundary.

Symmetry Condition

Mirror symmetry is invoked in the *xy*-plane at z = 0.04 m.

Wall Condition

y zx

Other boundaries are represented by wall functions for the liquid, and with zero gas flux for the bubbles.



Figure 1: Model geometry for the airlift loop reactor.

BUBBLY FLOW INTERFACE AND TURBULENCE MODELING

The Bubbly Flow interface sets up a multiphase-flow model for gas bubbles in a liquid. The physics interface tracks the averaged gas-phase concentration rather than each bubble in detail. The physics interface solves for the liquid velocity, the pressure, and the volume fraction of the gas phase. Details of the governing equations are presented in the theory section for the Bubbly Flow interfaces in the *CFD Module User's Guide*.

For laminar flow the gas velocity \mathbf{u}_{g} is calculated from

$$\mathbf{u}_{g} = \mathbf{u}_{l} + \mathbf{u}_{slip}$$

where \mathbf{u}_{l} stands for the liquid-phase velocity, and \mathbf{u}_{slip} stands for the relative velocity between gas and liquid, the so-called slip velocity.

The slip velocity s calculated from a slip model. The Bubbly Flow interface provides several slip models. The most appropriate slip model for this reactor is a pressure-drag balance slip model with a drag coefficient tuned for large bubbles.

The experiments in Ref. 1 suggest that the Reynolds number for the fully developed flow is $2 \cdot 10^4$, and hence that the flow is turbulent. The turbulence model for bubbly flows is similar to the single-phase k- ε turbulence model (details can be found in the theory sections for the Turbulent Flow and Bubbly Flow interfaces in the *CFD Module User's Guide*). However, for the bubbly flow cases, additional source terms are added to the turbulence equations. These account for the extra production and dissipation of turbulence due to the relative motion between the gas bubbles and the liquid. The additional source term in the *k* equation, denoted S_k , accounts for the bubble-induced turbulence and is given by (see Ref. 2)

$$S_k = -C_k \phi_g \nabla p \cdot \mathbf{u}_{slip}$$

The additional source term in the ε -equation, denoted S_{ε} , accounting for the bubble-induced turbulence dissipation and is given by

$$S_{\varepsilon} = \frac{\varepsilon}{k} C_{\varepsilon} S_k$$

where C_k and C_{ε} are model constants. The values of C_k and C_{ε} are highly problem dependent but can often be tuned to obtain good agreement between experimental data and simulations (see Ref. 3). According to Ref. 2, admissible values for C_k and C_{ε} are in the ranges of [0.01, 1] and [1, 1.92], respectively. Ref. 3 does however use C_{ε} values less than 1 and they obtain good agreements between the measurements and simulations. In this example, when bubble-induced turbulence is accounted for, C_k and C_{ε} are set to 0.6 and 1.4, respectively.

To account for turbulent transport of the bubbles, a drift velocity is added for the gas-phase velocity field:

$$\mathbf{u}_{g} = \mathbf{u}_{l} + \mathbf{u}_{slip} + \mathbf{u}_{drift}$$

where

$$\mathbf{u}_{\rm drift} = -\frac{\mu_{\rm T}}{\rho_{\rm l}} \frac{\nabla \phi_{\rm g}}{\phi_{\rm g}}$$

Using the k- ε model, the turbulent dynamic viscosity is defined as

$$\mu_{\rm T} = \rho_{\rm l} C_{\mu} \frac{k^2}{\varepsilon}$$

where C_{μ} is a model constant (for details, see the theory section for the Bubbly Flow interfaces in the *CFD Module User's Guide*).
In the Bubbly Flow interface you can easily switch the k- ε turbulence model on and off. You can also control whether to include or exclude the bubble-induced turbulence term S_k by adjusting the value of C_k . A C_k equal to zero means that the bubble induced turbulence is neglected.

The Physical Model settings for the Bubbly Flow interface also provides a low gas concentration option which is active per default. This option is applicable if the gas concentration is less than 2%, in which case the transport equations can be simplified compared to cases with higher gas concentrations. Ref. 1 does not specifically report the gas concentration, but photographs of the reactor indicate that the gas concentration might be high, so the low gas concentration option is disabled in this model.

MESH GENERATION

The mesh must be very fine in the vicinity of the frits in order to resolve steep gradients in bubble concentration. The mesh also needs to be relatively fine in the interior of the reactor since the presence of bubbles creates relatively complicated flow structures.

SOLVING THE MODEL

The goal is to obtain a stationary solution, but when it comes to buoyant flows, the best way to reach it is often a time-dependent simulation. Buoyant flows can feature intricate flow structures that are in a delicate balance with each other. It can sometimes be difficult to find such flow structures with a stationary solver while a time-dependent simulation lets the structures evolve to their final state.

Results and Discussion

Figure 2 shows the gas volume fraction and velocity streamlines for the liquid in the symmetry plane at t = 30 s. The results are qualitatively in good agreement with Ref. 1 except that the experiments show a recirculation zone at the top left corner while this recirculation zone is absent in the simulation.

The maximum value of the gas concentration is about 7% close to the two frits and higher than 2% in substantial parts of the reactor. This confirms that the low-gas concentration assumption would not have been valid.



Figure 2: Results of a time-dependent simulation at t = 30 s. The surface is the gas concentration, and the streamlines are liquid velocity.

Figure 3 shows the turbulent viscosity. The effect of bubble induced turbulence can be perceived by the relatively high levels of turbulent viscosity just above the frits and also beneath the free surface. The latter can be the reason to why there is no recirculation zone by the top left corner. But it could also be that the missing recirculation zone is caused sloshing which has been neglected in this example.



Figure 3: Turbulent viscosity in the symmetry plane.

Experimental data is reported for four probe position, #3, #5, #7 and #9. They correspond to lines in the symmetry plane at different, constant heights, namely y = 300 mm, y = 650 mm, y = 1250 mm and y = 1650 mm respectively. The probes are positioned in the rising part of the reactor. Comparisons between simulation and experiments are shown in Figure 4. The agreement is good in the lower part of the reactor (positions #3 and #5) where both liquid and gas velocities from the simulation are in close quantitative agreement with their experimental counterparts. However, the agreement is less good in the upper part of the reactor due to the missing recirculation zone in the top left corner. The simulation results are still qualitatively correct at probe position #7, but the velocities are too low close to the inner wall. The lack of the recirculation zone is apparent at probe position #9 where the experiments show negative liquid velocities along the outer wall while the simulation shows positive liquid velocities.

The overall agreement must still be deemed good considering the many modeling assumptions used in this example.



Figure 4: Comparison between simulation results and experimental results for vertical velocities at four different probe positions.

References

1. S. Becker, A. Sokolichin, and G. Eigenberger, "Gas-liquid flow in bubble columns and loop reactors: Part II. Comparison of detailed experiments and flow simulations," *Chem. Eng. Sci.*, vol. 49, pp. 5747–5762, 1994.

2. D. Kuzmin and S. Turek, "Numerical simulation of turbulent bubbly flows," *Third Int.* Symposium on Two-Phase Flow Modeling and Experiment, Pisa, pp 22–24, Sept. 2004.

3. A. Sokolichin, G. Eigenberger, and A. Lapin, "Simulation of Buoyancy Driven Bubbly Flow: Established Simplification and Open Questions," *Fluid Mechanics and Transport Phenomena*, vol. 51, no. 1, pp. 24–45, 2004.

Application Library path: CFD_Module/Multiphase_Benchmarks/ airlift loop reactor

Modeling Instruction

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Bubbly Flow>Turbulent Bubbly Flow (bf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Time Dependent.
- 6 Click Done.

GLOBAL DEFINITIONS

First, define some model parameters.

Parameters

I On the Home toolbar, click Parameters.

2 In the Settings window for Parameters, locate the Parameters section.

Name	Expression	Value	Description
Н	1.75[m]	I.75 m	Reactor height
W	0.5[m]	0.5 m	Reactor width
т	0.08[m]	0.08 m	Reactor thickness
d_b	3e-3[m]	0.003 m	Bubble diameter
R	0.02[m]	0.02 m	Frit radius
L	0.16[m]	0.16 m	Width of riser and downcomer channels
V_in	0.015[m/s]	0.015 m/s	Inlet velocity
Cw	5e4[kg/ (m^3*s)]	5E4 kg/ (m³·s)	Slip-velocity proportionality constant
rhog_in	0.9727[kg/ (m^3)]	0.9727 kg/ m³	Gas density at inlet

3 In the table, enter the following settings:

GEOMETRY I

Create the geometry. To simplify this step, insert a prepared geometry sequence.

- I On the Geometry toolbar, click Insert Sequence.
- 2 Browse to the application's Application Library folder and double-click the file airlift_loop_reactor.mph.
- 3 Click Build All on the Geometry toolbar.
- **4** Hold down the left mouse button and drag in the Graphics window to rotate the geometry. Similarly, use the right mouse button to translate the geometry and the middle button to zoom.

Full geometry instructions can be found at the end of the document.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.

ADD MATERIAL

I Go to the Add Material window.

- 2 In the tree, select Built-In>Water, liquid.
- 3 Click Add to Component in the window toolbar.
- 4 On the Home toolbar, click Add Material to close the Add Material window.

TURBULENT BUBBLY FLOW (BF)

Fluid Properties 1

- I In the Model Builder window, expand the Turbulent Bubbly Flow (bf) node, then click Fluid Properties I.
- 2 In the Settings window for Fluid Properties, locate the Materials section.
- 3 From the Liquid list, choose Water, liquid (mat2).
- 4 From the Gas list, choose Air (mat1).
- 5~ Locate the Gas Properties section. From the ρ_g list, choose Calculate from ideal gas law.
- **6** In the d_b text field, type d_b.
- 7 Locate the Slip Model section. From the Slip model list, choose Pressure-drag balance.
- 8 From the Drag coefficient model list, choose Large bubbles.
- 9 In the Model Builder window, click Turbulent Bubbly Flow (bf).
- 10 In the Settings window for Turbulent Bubbly Flow, locate the Physical Model section.
- II Clear the Low gas concentration check box.
- 12 Click to expand the Turbulence model parameters section. Locate the Turbulence Model Parameters section. In the C_{ε} text field, type 1.4.
- **I3** In the C_k text field, type 0.6.

Gravity I

- I On the Physics toolbar, click Domains and choose Gravity.
- **2** Select Domain 1 only.

Wall 2

- I On the Physics toolbar, click Boundaries and choose Wall.
- 2 Click the **Zoom Extents** button on the **Graphics** toolbar.
- **3** Select Boundaries 6 and 7 only.
- 4 In the Settings window for Wall, locate the Gas Boundary Condition section.
- 5 From the Gas boundary condition list, choose Gas flux.

6 In the N_{pgdg} text field, type V_in*rhog_in*flc1hs(t[1/s]-5,5).

Here the built-in smoothed Heaviside function flc1hs is used to give a smooth initial value.

Wall 3

- I On the Physics toolbar, click Boundaries and choose Wall.
- **2** Select Boundary 5 only.
- 3 In the Settings window for Wall, locate the Liquid Boundary Condition section.
- 4 From the Liquid boundary condition list, choose Slip.
- 5 Locate the Gas Boundary Condition section. From the Gas boundary condition list, choose Gas outlet.

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Turbulent Bubbly Flow (bf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *p* text field, type g_const*bf.rhol*(1.75-y).

Pressure Point Constraint 1

- I On the Physics toolbar, click Points and choose Pressure Point Constraint.
- 2 Select Point 23 only.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- 2 Select Boundary 4 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- 3 From the Element size list, choose Coarse.

Size

- I Right-click Component I (comp1)>Mesh I and choose Edit Physics-Induced Sequence.
- 2 In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 3 In the Settings window for Size, locate the Element Size section.
- 4 From the Predefined list, choose Normal.

Boundary Layer Properties 1

- In the Model Builder window, expand the Component I (compl)>Mesh l>Boundary Layers
 I node, then click Boundary Layer Properties I.
- 2 In the Settings window for Boundary Layer Properties, locate the Boundary Layer Properties section.
- 3 In the Number of boundary layers text field, type 4.
- 4 In the Thickness adjustment factor text field, type 3.
- 5 Click Build All.
- 6 Click the Zoom Extents button on the Graphics toolbar.

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range(0,0.05,1)*30.
- 4 Select the **Relative tolerance** check box.
- **5** In the associated text field, type 0.005.

Measurement data makes it possible to estimate manual scales for velocity and pressure.

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll)>Dependent Variables I node, then click Pressure (compl.p).
- 4 In the Settings window for Field, locate the Scaling section.
- 5 From the Method list, choose Manual.
- 6 In the Scale text field, type 1.7e4.
- 7 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables I click Velocity field, liquid phase (compl.u).
- 8 In the Settings window for Field, locate the Scaling section.
- 9 From the Method list, choose Manual.
- **IO** In the **Scale** text field, type 0.5.
- II In the Model Builder window, collapse the Solution I (soll) node.

12 In the **Settings** window for Solution, click **Compute**.

The following steps reproduce Figure 2.

RESULTS

Surface 1

On the **Results** toolbar, click **More Data Sets** and choose **Surface**.

Data Sets

- I Select Boundary 4 only.
- 2 In the Settings window for Surface, locate the Parameterization section.
- 3 From the x- and y-axes list, choose xy-plane.

2D Plot Group 3

- I On the Results toolbar, click 2D Plot Group.
- 2 In the Model Builder window, right-click 2D Plot Group 3 and choose Surface.
- 3 Right-click 2D Plot Group 3 and choose Streamline.
- 4 In the Settings window for Streamline, locate the Expression section.
- **5** In the **x component** text field, type **u**.
- 6 In the y component text field, type v.
- 7 Locate the Coloring and Style section. From the Color list, choose White.
- 8 Locate the Streamline Positioning section. From the Positioning list, choose Uniform density.
- 9 In the Separating distance text field, type 0.025.
- IO On the 2D Plot Group 3 toolbar, click Plot.

The following steps reproduce Figure 3.

2D Plot Group 4

- I On the Home toolbar, click Add Plot Group and choose 2D Plot Group.
- 2 In the Model Builder window, right-click 2D Plot Group 4 and choose Surface.
- 3 In the Settings window for Surface, locate the Expression section.
- 4 In the **Expression** text field, type bf.muT.
- 5 On the 2D Plot Group 4 toolbar, click Plot.

Proceed to reproduce the 1D-plots in Figure 4. Start by importing experimental data.

Table I

I On the **Results** toolbar, click **Table**.

- 2 In the Settings window for Table, type v13 in the Label text field.
- 3 In the Settings window for Table, locate the Data section.
- 4 Click Import.
- 5 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vl_no3.txt.

Table 2

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type vg3 in the Label text field.
- 3 Locate the Data section. Click Import.
- 4 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vg_no3.txt.

Table 3

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type v15 in the Label text field.
- 3 In the Settings window for Table, locate the Data section.
- 4 Click Import.
- 5 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vl_no5.txt.

Table 4

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type vg5 in the Label text field.
- 3 In the Settings window for Table, locate the Data section.
- 4 Click Import.
- 5 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vg_no5.txt.

Table 5

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type v17 in the Label text field.
- 3 Locate the Data section. Click Import.
- **4** Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vl_no7.txt.

Table 6

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type vg7 in the Label text field.
- 3 In the Settings window for Table, locate the Data section.
- 4 Click Import.
- 5 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vg_no7.txt.

Table 7

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type v19 in the Label text field.
- 3 In the Settings window for Table, locate the Data section.
- 4 Click Import.
- 5 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vl_no9.txt.

Table 8

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, type vg9 in the Label text field.
- 3 In the Settings window for Table, locate the Data section.
- 4 Click Import.
- 5 Browse to the application's Application Libraries folder and double-click the file airlift_loop_reactor_vg_no9.txt.

Define cut lines that correspond to the experimental probe positions.

Cut Line 3D I

On the **Results** toolbar, click **Cut Line 3D**.

Data Sets

- I In the Settings window for Cut Line 3D, type No3 in the Label text field.
- 2 Locate the Line Data section. In row Point I, set y to 0.3 and z to 0.04.
- 3 In row Point 2, set x to 0.15, y to 0.3, and z to 0.04.
- 4 Right-click No3 and choose Duplicate.
- 5 In the Settings window for Cut Line 3D, type No5 in the Label text field.
- 6 Locate the Line Data section. In row Point 1, set y to 0.65.
- 7 In row **Point 2**, set **y** to 0.65.

- 8 Right-click Results>Data Sets>No5 and choose Duplicate.
- 9 In the Settings window for Cut Line 3D, type No7 in the Label text field.
- 10 Locate the Line Data section. In row Point 1, set y to 1.25.
- II In row Point 2, set y to 1.25.
- 12 Right-click Results>Data Sets>No7 and choose Duplicate.
- **I3** In the **Settings** window for Cut Line 3D, type No9 in the **Label** text field.
- 14 Locate the Line Data section. In row Point 1, set y to 1.65.
- **I5** In row **Point 2**, set **y** to **1.65**.

ID Plot Group 5

- I On the **Results** toolbar, click **ID Plot Group**.
- 2 In the Settings window for 1D Plot Group, type Probe position #3 in the Label text field.
- 3 Locate the Data section. From the Data set list, choose No3.
- 4 From the Time selection list, choose Last.
- 5 Click to expand the Title section. From the Title type list, choose Manual.
- **6** In the **Title** text area, type Vertical liquid and gas velocities at probe position #3.

Line Graph 1

On the **Probe position #3** toolbar, click **Line Graph**.

Probe position #3

- I In the Settings window for Line Graph, locate the y-Axis Data section.
- 2 In the Expression text field, type v.
- **3** Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.
- 4 Click to expand the Legends section. Select the Show legends check box.
- 5 From the Legends list, choose Manual.
- 6 In the table, enter the following settings:

Legends

Fluid velocity, simulation

7 In the Model Builder window, click Probe position #3.

Table Graph 1

On the **Probe position #3** toolbar, click **Table Graph**.

Probe position #3

- I In the Settings window for Table Graph, locate the Coloring and Style section.
- 2 Find the Line style subsection. From the Line list, choose None.
- **3** From the **Color** list, choose **Blue**.
- 4 Find the Line markers subsection. From the Marker list, choose Diamond.
- 5 From the Positioning list, choose In data points.
- 6 Click to expand the Legends section. Select the Show legends check box.
- 7 From the Legends list, choose Manual.
- 8 In the table, enter the following settings:

Legends

Fluid velocity, experiments

9 In the Model Builder window, click Probe position #3.

Line Graph 2

On the **Probe position #3** toolbar, click **Line Graph**.

Probe position #3

- I In the Settings window for Line Graph, locate the y-Axis Data section.
- 2 In the **Expression** text field, type bf.ugy.
- 3 Locate the Coloring and Style section. From the Color list, choose Red.
- 4 Locate the Legends section. Select the Show legends check box.
- 5 From the Legends list, choose Manual.
- 6 In the table, enter the following settings:

Legends

Gas velocity, simulation

7 In the Model Builder window, click Probe position #3.

Table Graph 2

On the **Probe position #3** toolbar, click **Table Graph**.

Probe position #3

I In the Settings window for Table Graph, locate the Data section.

- 2 From the Table list, choose vg3.
- **3** Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 4 From the Color list, choose Red.
- 5 Find the Line markers subsection. From the Marker list, choose Square.
- 6 From the Positioning list, choose In data points.
- 7 Locate the Legends section. Select the Show legends check box.
- 8 From the Legends list, choose Manual.
- 9 In the table, enter the following settings:

Legends

Gas velocity, experiments

- IO In the Model Builder window, click Probe position #3.
- II In the Settings window for 1D Plot Group, locate the Axis section.
- **12** Select the **Manual axis limits** check box.
- **I3** In the **x minimum** text field, type **0**.
- **I4** In the **x maximum** text field, type **0.15**.
- **I5** In the **y minimum** text field, type -0.15.
- **I6** In the **y maximum** text field, type **0.85**.
- **I7** On the **Probe position #3** toolbar, click **Plot**.
- 18 In the Model Builder window, collapse the Probe position #3 node.

Probe position #3.1

- I Right-click Probe position #3 and choose Duplicate.
- 2 In the Settings window for 1D Plot Group, type Probe position #5 in the Label text field.
- 3 Locate the Data section. From the Data set list, choose No5.
- 4 Click to expand the **Title** section. In the **Title** text area, type Vertical liquid and gas velocities at probe position #5.

Probe position #5

- I In the Model Builder window, expand the Results>Probe position #5 node, then click Table Graph I.
- 2 In the Settings window for Table Graph, locate the Data section.

- 3 From the Table list, choose vI5.
- 4 In the Model Builder window, under Results>Probe position #5 click Table Graph 2.
- 5 In the Settings window for Table Graph, locate the Data section.
- 6 From the Table list, choose vg5.
- 7 On the Probe position #5 toolbar, click Plot.
- 8 In the Model Builder window, click Probe position #5.
- 9 In the Settings window for 1D Plot Group, locate the Axis section.
- **IO** Select the **Manual axis limits** check box.
- II In the **y minimum** text field, type -0.15.
- **12** In the **y maximum** text field, type **0.85**.
- **I3** On the **Probe position #5** toolbar, click **Plot**.

14 In the Model Builder window, collapse the Probe position #5 node.

Probe position #5.1

- I Right-click Probe position #5 and choose Duplicate.
- 2 In the Settings window for 1D Plot Group, type Probe position #7 in the Label text field.
- 3 Locate the Data section. From the Data set list, choose No7.
- 4 Click to expand the **Title** section. In the **Title** text area, type Vertical liquid and gas velocities at probe position #7.

Probe position #7

- I In the Model Builder window, expand the Results>Probe position #7 node, then click Table Graph I.
- 2 In the Settings window for Table Graph, locate the Data section.
- 3 From the Table list, choose vI7.
- 4 In the Model Builder window, under Results>Probe position #7 click Table Graph 2.
- 5 In the Settings window for Table Graph, locate the Data section.
- 6 From the Table list, choose vg7.
- 7 In the Model Builder window, click Probe position #7.
- 8 In the Settings window for 1D Plot Group, locate the Axis section.
- 9 Select the Manual axis limits check box.
- **IO** In the **y minimum** text field, type -0.15.
- II In the **y maximum** text field, type 0.85.

12 On the Probe position #7 toolbar, click Plot.

I3 In the **Model Builder** window, collapse the **Probe position #7** node.

Probe position #7.1

- I Right-click Probe position #7 and choose Duplicate.
- 2 In the Settings window for 1D Plot Group, type Probe position #9 in the Label text field.
- 3 Locate the Data section. From the Data set list, choose No9.
- 4 Click to expand the **Title** section. In the **Title** text area, type Vertical liquid and gas velocities at probe position #9.

Probe position #9

- I In the Model Builder window, expand the Results>Probe position #9 node, then click Table Graph I.
- 2 In the Settings window for Table Graph, locate the Data section.
- 3 From the Table list, choose vl9.
- 4 In the Model Builder window, under Results>Probe position #9 click Table Graph 2.
- 5 In the Settings window for Table Graph, locate the Data section.
- 6 From the Table list, choose vg9.
- 7 In the Model Builder window, click Probe position #9.
- 8 In the Settings window for 1D Plot Group, locate the Axis section.
- 9 Select the Manual axis limits check box.
- **IO** In the **y minimum** text field, type -0.15.
- II In the **y maximum** text field, type **0.85**.
- 12 On the Probe position #9 toolbar, click Plot.

13 In the Model Builder window, collapse the Probe position #9 node.

Appendix. Geometry Modeling Instructions

GEOMETRY I

Work Plane I (wp1)

- I On the Geometry toolbar, click Work Plane.
- 2 In the Settings window for Work Plane, click Show Work Plane.

Polygon I (poll)

- I On the Work Plane toolbar, click Primitives and choose Polygon.
- 2 In the Settings window for Polygon, locate the Coordinates section.
- 3 In the **xw** text field, type 0 L W W 0.
- 4 In the **yw** text field, type 0 0 L H H.
- 5 On the Work Plane toolbar, click Build All.

Polygon 2 (pol2)

- I On the Work Plane toolbar, click Primitives and choose Polygon.
- 2 In the Settings window for Polygon, locate the Coordinates section.
- **3** In the **xw** text field, type L 0.34 0.34 L.
- **4** In the **yw** text field, type 0.11 0.2 1.47 1.56.
- 5 On the Work Plane toolbar, click Build All.

Difference I (dif1)

- I On the Work Plane toolbar, click Booleans and Partitions and choose Difference.
- 2 Select the object **poll** only.
- 3 In the Settings window for Difference, locate the Difference section.
- 4 Find the Objects to subtract subsection. Select the Active toggle button.
- 5 Select the object **pol2** only.
- 6 On the Work Plane toolbar, click Build All.

Work Plane I (wp1)

In the Model Builder window, under Component I (compl)>Geometry I click Work Plane I (wpl).

Extrude I (extI)

- I On the Geometry toolbar, click Extrude.
- 2 In the Settings window for Extrude, locate the Distances from Plane section.
- **3** In the table, enter the following settings:

Distances (m)

Т

4 Click Build All Objects.

Work Plane 2 (wp2)

I On the Geometry toolbar, click Work Plane.

- 2 In the Settings window for Work Plane, locate the Plane Definition section.
- 3 From the Plane list, choose zx-plane.
- 4 Click Show Work Plane.

Circle I (c1)

- I On the Work Plane toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type R.
- 4 Locate the **Position** section. In the **xw** text field, type T/2.
- **5** In the **yw** text field, type **0.11**.

Circle 2 (c2)

- I On the Work Plane toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type R.
- **4** Locate the **Position** section. In the **xw** text field, type T/2.
- **5** In the **yw** text field, type T/2.
- 6 On the Work Plane toolbar, click Build All.
- 7 In the Model Builder window, click Geometry I.

Union I (unil)

- I On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 2 Click in the Graphics window and then press Ctrl+A to select both objects.
- 3 In the Settings window for Union, click Build All Objects.

Block I (blkI)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the **Depth** text field, type 2.
- **4** Locate the **Position** section. In the **z** text field, type T/2.
- **5** Click **Build All Objects**.
- 6 Click the Zoom Extents button on the Graphics toolbar.

Difference I (dif1)

- I On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 2 Select the object unil only, to add it to the Objects to add list.

- 3 In the Settings window for Difference, locate the Difference section.
- **4** Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- **5** Select the object **blk1** only.
- 6 On the Geometry toolbar, click Build All.



Stationary Incompressible Flow over a Backstep

Introduction

This tutorial model solves the incompressible Navier-Stokes equations in a backstep geometry. A characteristic feature of fluid flow in geometries of this kind is the recirculation region that forms where the flow exits the narrow inlet region. The model clearly demonstrates the formation of such a region, which is best displayed by visualizing the flow streamlines.

Model Definition

MODEL GEOMETRY

The model consists of a pipe connected to a block-shaped tank; see Figure 1. Due to symmetry, it is sufficient to model one eighth of the full geometry. The pipe has an inlet at one end, and the tank has an outlet at the opposite end. All other boundaries are solid walls.



Figure 1: Model geometry.

DOMAIN EQUATION AND BOUNDARY CONDITIONS

The flow in the system is laminar, so the model uses the Laminar Flow interface. The inlet flow is fully developed laminar flow, described by the corresponding inlet boundary condition. The boundary condition at the outlet sets a constant relative pressure. Furthermore, the vertical and inclined boundaries along the length of the geometry are symmetry boundaries. All other boundaries are solid walls described by a no slip boundary condition.

Results

Figure 2 shows a combined surface and arrow plot of the flow velocity. This plot does not reveal the recirculation region in the tank immediately beyond the inlet pipe's end. For this purpose, a streamline plot is more useful, as demonstrated in Figure 3.



Surface: Velocity magnitude (m/s) Arrow Surface: Velocity field

Figure 2: The velocity field in the backstep geometry.



Figure 3: The recirculation region visualized using a velocity streamline plot.

Application Library path: CFD_Module/Single-Phase_Tutorials/backstep

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Laminar Flow (spf).
- 3 Click Add.
- 4 Click Study.

5 In the Select Study tree, select Preset Studies>Stationary.

6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

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

GEOMETRY I

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

- I On the Geometry toolbar, click Insert Sequence.
- 2 Browse to the application's Application Libraries folder and double-click the file backstep_geom_sequence.mph.

Cylinder I (cyl1)

- I In the Model Builder window, under Component I (comp1)>Geometry I click Cylinder I (cyl1).
- 2 In the Settings window for Cylinder, click Build All Objects.

The model geometry is now complete (Figure 1).

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Water, liquid.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

LAMINAR FLOW (SPF)

Inlet 1

I On the Physics toolbar, click Boundaries and choose Inlet.

- **2** Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Boundary Condition section.
- 4 From the list, choose Laminar inflow.
- **5** Locate the Laminar Inflow section. In the U_{av} text field, type v0.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- **2** Select Boundaries 2 and 3 only.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 7 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- 3 From the Element size list, choose Coarse.
- 4 Click Build All.

STUDY I

On the Home toolbar, click Compute.

RESULTS

Velocity (spf)

- I In the Model Builder window, expand the Velocity (spf) node.
- 2 Right-click Slice and choose Delete. Click Yes to confirm.

RESULTS

Velocity (spf)

- I In the Model Builder window, under Results right-click Velocity (spf) and choose Surface.
- 2 Right-click Velocity (spf) and choose Arrow Surface.
- 3 In the Settings window for Arrow Surface, locate the Coloring and Style section.
- 4 From the Arrow length list, choose Logarithmic.

- 5 From the Color list, choose Yellow.
- 6 Click the **Zoom Extents** button on the **Graphics** toolbar.

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

3D Plot Group 3

- I On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- 2 In the Model Builder window, right-click 3D Plot Group 3 and choose Streamline.
- **3** Select Boundary 1 only.
- 4 In the Settings window for Streamline, locate the Coloring and Style section.
- **5** From the **Line type** list, choose **Tube**.
- 6 Right-click Results>3D Plot Group 3>Streamline I and choose Color Expression.

8 | STATIONARY INCOMPRESSIBLE FLOW OVER A BACKSTEP



Laminar Flow in a Baffled Stirred Mixer

Introduction

This exercise exemplifies the use of the rotating machinery feature in the CFD Module. The Rotating Machinery physics interfaces allow you to model moving rotating parts in, for example, stirred tanks, mixers, and pumps.

The Rotating Machinery physics interfaces formulate the Navier-Stokes equations in a rotating coordinate system. Parts that are not rotated are expressed in the fixed material coordinate system. The rotating and fixed parts need to be coupled together by an identity pair, where a flux continuity boundary condition is applied.

You can use the rotating machinery predefined multiphysics coupling in cases where it is possible to divide the modeled device into rotationally invariant geometries. The desired operation can be, for example, to rotate an impeller in a baffled tank. This is exemplified in Figure 1, where the impeller rotates from position 1 to 2. The first step is to divide the geometry into two parts that are both rotationally invariant, as shown in Step 1a. The second step is to specify the parts to model using a rotating frame and the ones to model using a fixed frame (Step 1b). The predefined coupling then automatically does the coordinate transformation and the joining of the fixed and moving parts (Step 2a).



Figure 1: The modeling procedure in the Rotating Machinery physics interface in the CFD Module.

Model Definition

The model you treat in this example is that of a baffled stirred mixer. Figure 2 shows the modeled geometry. The impeller rotates counterclockwise at a speed of 10 RPM, and the fluid in the mixer is water.

The model equations are the Navier-Stokes equations formulated in a rotating frame in the inner domain and in fixed coordinates in the outer one.

At the mixer's fixed walls, no slip boundary conditions apply. The boundary condition on the rotating impeller are set to rotate with the same velocity as the no slip counterclockwise rotation conditions.

The implementation, applying the Rotating Machinery, Laminar Flow interface, is straightforward. First you draw the geometry using two separate non-overlapping domains for the fixed and rotating parts. The next step is to form an assembly and create an identity pair, which makes it possible to treat the two domains as separate parts. You then specify which part uses a rotating frame. Once you have done this, you can proceed to the usual steps of setting the fluid properties and the boundary conditions, and finally to meshing and solving the problem.



Figure 2: Geometry of the baffled stirred mixer. The inner domain is represented by a rotating (spatial) frame and the outer domain by a fixed (material) frame.

Results and Discussion

Figure 3 shows the shear rate at the last time step, when the mixer has rotated with full speed for 6 seconds. The shear rate can be important for example when the mixture consists of living cells which are sensitive to too high shear rates.



Figure 3: The shear rate after 6 seconds. The plot shows that the shear rate reaches its maximum value at the tip of the impeller blades.

In Figure 3, we can clearly see that the shear rate is highest where the impeller speed is highest, which is expected. If the shear rate is too high, consider to redesign the impeller with sharp leading and trailing edges.

Figure 4 shows the velocity field in the *yz*-plane at t = 5.5 s. It is clear that the impeller is creating a downward flow where the blades pass through the fluid.



Figure 4: Velocity field in the yz-plane at t = 5.5 s.

Use the Player button on the main toolbar to create an animation.

Application Library path: CFD_Module/Single-Phase_Tutorials/baffled_mixer

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

I In the Model Wizard window, click 3D.

- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Rotating Machinery, Fluid Flow>Rotating Machinery, Laminar Flow (rmspf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Time Dependent.
- 6 Click Done.

GEOMETRY I

Import I (imp1)

- I On the Home toolbar, click Import.
- 2 In the Settings window for Import, locate the Import section.
- 3 Click Browse.
- 4 Browse to the application's Application Libraries folder and double-click the file baffled_mixer.mphbin.
- 5 Click Import.

Use the assembly mode to create separate geometry objects. This, together with an identity pair, is needed for the sliding-mesh technique.

Form Union (fin)

- I In the Model Builder window, under Component I (comp1)>Geometry I click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, locate the Form Union/Assembly section.
- 3 From the Action list, choose Form an assembly.
- 4 Clear the **Create pairs** check box.
- 5 Right-click Component I (comp1)>Geometry 1>Form Union (fin) and choose Build Selected.

DEFINITIONS

Identity Pair I (p1)

- I On the Definitions toolbar, click Pairs and choose Identity Boundary Pair.
- 2 Click the Transparency button on the Graphics toolbar.
- 3 Click the Zoom Extents button on the Graphics toolbar.

4 Select Boundaries 8, 9, 14, and 15 only.

You can do this by first copying the text '8, 9, 14, 15' and then clicking the Paste Selection button next to the Selection box or clicking in the box and pressing Ctrl+V.

- 5 In the Settings window for Pair, locate the Destination Boundaries section.
- 6 Select the Active toggle button.
- 7 Select Boundaries 23, 24, 37, and 48 only.



Create a step function that you will use to increase the impeller rotation from zero to 10 RPM in a time of 2 seconds.

Step I (step I)

- I On the Definitions toolbar, click More Functions and choose Step.
- 2 In the Settings window for Step, locate the Parameters section.
- **3** In the **Location** text field, type **1**.
- 4 Click to expand the Smoothing section. In the Size of transition zone text field, type 2.

Variables 1

- I On the Definitions toolbar, click Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
rpm	10[1/min]*step1(t[1/s])	l/s	Revolutions per minute

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Water, liquid.
- 4 Click Add to Component in the window toolbar.

MATERIALS

Water, liquid (mat1)

On the Home toolbar, click Add Material to close the Add Material window.

ROTATING MACHINERY, LAMINAR FLOW (RMSPF)

Rotating Domain I

- I On the Physics toolbar, click Domains and choose Rotating Domain.
- 2 Select Domain 2 only.
- 3 In the Settings window for Rotating Domain, locate the Rotating Domain section.
- **4** In the *f* text field, type rpm.

Next, add the flow continuity condition on the identity pair to couple the rotating and fixed domains together.

Flow Continuity I

I On 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 Pair I (pl).

All boundaries that are adjacent to the rotating domain will by default be set to rotating boundaries. The bottom boundary should not rotate, so add an additional no-slip wall boundary condition to enforce this condition.

Wall 2

- I On the Physics toolbar, click Boundaries and choose Wall.
- **2** Select Boundary 25 only.
Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- **2** Select Boundaries 4 and 26 only.

Finally, add a pressure point constraint at one of the top boundaries.

Pressure Point Constraint I

- I On the Physics toolbar, click Points and choose Pressure Point Constraint.
- 2 Select Point 4 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- **3** From the **Element size** list, choose **Fine**.

Size

- I Right-click Component I (compl)>Mesh I and choose Edit Physics-Induced Sequence.
- 2 In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 3 In the Settings window for Size, locate the Element Size section.
- 4 Click the **Custom** button.
- **5** Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type **1.2**.
- 6 In the Minimum element size text field, type 0.002.
- 7 In the Model Builder window, click Mesh I.
- 8 In the Settings window for Mesh, click Build All.

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range(0,0.25,6).

Solution I (soll)

I On the Study toolbar, click Show Default Solver.

Before solving, display the default solver settings to be able to set a maximum time step. The time step is limited with the mesh size and rotational speed at the identity pair in mind.

- 2 In the Model Builder window, expand the Study I>Solver Configurations node.
- 3 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **4** In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
- 5 Locate the Time Stepping section. Select the Maximum step check box.
- 6 In the associated text field, type 0.1.
- 7 On the Study toolbar, click Compute.

RESULTS

Velocity (rmspf)

The default plot groups visualize the velocity field in a slice plot and the and pressure contours on the walls. Follow these steps to create a plot of the shear rate.

3D Plot Group 3

- I On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- **3** From the Frame list, choose Spatial (x, y, z).
- 4 Right-click 3D Plot Group 3 and choose Slice.
- 5 In the Settings window for Slice, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Rotating Machinery, Laminar Flow>rmspf.sr Shear rate.
- 6 Locate the Plane Data section. From the Plane list, choose xy-planes.
- 7 From the Entry method list, choose Coordinates.
- 8 On the 3D Plot Group 3 toolbar, click Plot.
- 9 Click the Transparency button on the Graphics toolbar.
- IO Click the Go to Default 3D View button on the Graphics toolbar.

II Click the Zoom Extents button on the Graphics toolbar.

Finally, create an arrow plot of the velocity field in a 2D cross section through the mixer's axis.

Cut Plane 1

On the Results toolbar, click Cut Plane.

2D Plot Group 4

- I On the Results toolbar, click 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Data set list, choose Cut Plane I.
- 4 From the Time (s) list, choose 5.5.
- 5 Locate the Plot Settings section. From the Frame list, choose Spatial (x, y, z).
- 6 Right-click 2D Plot Group 4 and choose Arrow Surface.
- 7 In the Settings window for Arrow Surface, locate the Expression section.
- 8 In the x component text field, type v.
- 9 In the y component text field, type w.
- **10** Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type **30**.
- II Find the y grid points subsection. In the Points text field, type 30.
- 12 Locate the Coloring and Style section. Select the Scale factor check box.
- **I3** In the associated text field, type 0.4.
- **I4** Click the **Zoom Extents** button on the **Graphics** toolbar.

12 | LAMINAR FLOW IN A BAFFLED STIRRED MIXER



Capillary Filling — Level Set Method

Introduction

Surface tension and wall adhesive forces are often used to transport fluid through microchannels in MEMS devices or to measure the transport and position of small amounts of fluid using micropipettes. Multiphase flow through a porous medium and droplets on solid walls are other examples where wall adhesion and surface tension strongly influence the dynamics of the flow.

This example studies a narrow vertical cylinder placed on top of a reservoir filled with water. Because of wall adhesion and surface tension at the air/water interface, water rises through the channel. The model calculates the velocity field, the pressure field, and the shape and position of the water surface.

This example demonstrates how to model the filling of a capillary channel using two multiphysics coupling features available in the CFD Module. You can use either the Two-Phase Flow, Level Set or the Two-Phase Flow, Phase Field multiphysics coupling feature. The Level Set interface uses a reinitialized level set method to represent the fluid interface between the air and the water. The Phase Field interface, on the other hand, uses a Cahn-Hilliard equation, including a chemical potential to represent a diffuse interface separating the two phases. The Navier-Stokes equations are used to describe the momentum transport and the conservation of mass.

Model Definition

The model consists of a capillary channel of radius **0.15** mm attached to a water reservoir. Water can flow freely into the reservoir. Because both the channel and the reservoir are cylindrical, you can use the axisymmetric geometry illustrated in Figure 1. Initially, the thin cylinder is filled with air. Wall adhesion causes water to creep up along the cylinder boundaries. The deformation of the water surface induces surface tension at the air/water interface, which in turn creates a pressure jump across the interface. The pressure variations cause water and air to move upward. The fluids continue to rise until the capillary forces are balanced by the gravity force building up as the water rises in the



channel. In the present example, the capillary forces dominate over gravity throughout the simulation. Consequently, the interface moves upwards during the entire simulation.

Figure 1: Axisymmetric geometry with boundary conditions.

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The Level Set interface automatically sets up the equations for the convection of the interface. The fluid interface is represented by the 0.5 contour of the level set function ϕ . In air $\phi = 0$ and in water $\phi = 1$. The level set function can thus be thought of as the volume fraction of water. The transport of the fluid interface separating the two phases is given by

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = \gamma \nabla \cdot \left(\varepsilon \nabla \phi - \phi (1 - \phi) \frac{\nabla \phi}{|\nabla \phi|} \right)$$

The ε parameter determines the thickness of the interface. When stabilization is used for the level set equation, you can typically use an interface thickness of $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The γ parameter determines the amount of reinitialization. A suitable value for γ is the maximum velocity magnitude occurring in the model. The multiphysics coupling feature defines the density and viscosity according to:

$$\rho = \rho_{air} + (\rho_{water} - \rho_{air})\phi$$
$$\mu = \mu_{air} + (\mu_{water} - \mu_{air})\phi$$

Due to these definitions, the density and viscosity vary smoothly across the fluid interface. The delta function is approximated by

$$\delta = 6|\phi(1-\phi)||\nabla\phi|$$

and the interface normal is calculated from

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|}$$

Phase Field Method

In the Phase Field interface the two-phase flow dynamics is governed by a Cahn-Hilliard equation. The equation tracks a diffuse interface separating the immiscible phases. The diffuse interface is defined as the region where the dimensionless phase field variable ϕ goes from -1 to 1. When solved in COMSOL Multiphysics, the Cahn-Hilliard equation is split up into two equations

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = \nabla \cdot \frac{\gamma \lambda}{\epsilon^2} \nabla \psi$$
$$\psi = -\nabla \cdot \epsilon^2 \nabla \phi + (\phi^2 - 1)\phi$$

where **u** is the fluid velocity (m/s), γ is the mobility (m³·s/kg), λ is the mixing energy density (N) and ϵ (m) is the interface thickness parameter. The ψ variable is referred to as the phase field help variable. The following equation relates the mixing energy density and the interface thickness to the surface tension coefficient:

$$\sigma = \frac{2\sqrt{2}\lambda}{3\epsilon}$$

You can typically set the interface thickness parameter to $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The mobility parameter γ determines the time scale of the Cahn-Hilliard diffusion and must be chosen judiciously. It must be large enough to retain a constant interfacial thickness but small enough so that the convective terms are not overly damped. The default value, $\gamma = \varepsilon^2$, is usually a good initial guess. This model uses a higher mobility to obtain the correct pressure variation over the interface.

In the Phase Field interface, the volume fractions of the individual fluids are

4 | CAPILLARY FILLING - LEVEL SET METHOD

$$V_{\rm f1} = \frac{1-\phi}{2}, \qquad V_{\rm f2} = \frac{1+\phi}{2}$$

In the present model water is defined as Fluid 1 and air as Fluid 2.

The multiphysics coupling feature defines the density (kg/m^3) and the viscosity $(Pa \cdot s)$ of the mixture to vary smoothly over the interface by letting

$$\rho = \rho_{\rm w} + (\rho_{\rm air} - \rho_{\rm w})V_{\rm f2}$$
$$\mu = \mu_{\rm w} + (\mu_{\rm air} - \mu_{\rm w})V_{\rm f2}$$

where the single phase water properties are denoted w and the air properties air.

MASS AND MOMENTUM TRANSPORT

The Navier-Stokes equations describe the transport of mass and momentum for fluids of constant density. In order to account for capillary effects, it is crucial to include surface tension in the model. The Navier-Stokes equations are then

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot [-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^{T})] + \mathbf{F}_{st} + \rho \mathbf{g}$$
$$\nabla \cdot \mathbf{u} = 0$$

Here, ρ denotes the density (kg/m³), μ equals the dynamic viscosity (Ns/m²), **u** represents the velocity (m/s), *p* denotes the pressure (Pa), and **g** is the gravity vector (m/s²). **F**_{st} is the surface tension force acting at the air/water interface.

Surface Tension

In the Level Set interface the surface tension force is computed as

$$\mathbf{F}_{st} = \nabla \cdot \mathbf{T}$$
$$\mathbf{T} = \sigma(\mathbf{I} - (\mathbf{nn}^T))\delta$$

Here, **I** is the identity matrix, **n** is the interface normal, σ equals the surface tension coefficient (N/m), and δ equals a Dirac delta function that is nonzero only at the fluid interface. When you use the finite element method to solve the Navier-Stokes equations, you multiply the equations by test functions and then integrate over the computational domain. If you use integration by parts, you can move derivatives of **T** to the test functions. This i results in an integral over the computational domain plus a boundary integral of the form

$$\int_{\partial\Omega} \operatorname{test}(\mathbf{u}) \cdot [\sigma(\mathbf{n}_{\text{wall}} - (\mathbf{n}\cos\theta))\delta] dS$$
(1)

where θ is the contact angle (see Figure 2). If you apply a no slip boundary condition, the boundary term vanishes because test(**u**) = 0 on that boundary, and you cannot specify the contact angle. Instead, the interface remains fixed on the wall. However, if you allow a small amount of slip, it is possible to specify the contact angle. The Wetted Wall coupling feature adds the term given by Equation 1 and consequently makes it possible to set the contact angle.

In the Phase Field interface, the diffuse interface representation makes it possible to compute the surface tension by

$$\mathbf{F}_{st} = G\nabla\phi$$

where ϕ is the phase field parameter, and *G* is the chemical potential (J/m³)

$$G = \lambda \left[-\nabla^2 \phi + \frac{\phi(\phi^2 - 1)}{\epsilon^2} \right] = \frac{\lambda}{\epsilon^2} \psi$$

As seen above, the phase field surface tension is computed as a distributed force over the interface using only ψ and the gradient of the phase field variable. This computation avoids using the surface normal and the surface curvature, which are troublesome to represent numerically.

INITIAL CONDITIONS

Initially, the reservoir is filled with water and the capillary channel is filled with air. The initial velocity is zero.

BOUNDARY CONDITIONS

Inlet

The hydrostatic pressure, $p = \rho gz$, gives the pressure at the inflow boundary. Only water enters through the inlet, so the level set function (that is, the volume fraction of water) is 1 here.

Outlet

At the outlet, the pressure is equal to zero, that is, equal to the pressure at the top of the inflow boundary. Because it is an outflow boundary, you do not have to set any condition on the level set function.

Walls

The Wetted Wall feature is suitable for solid walls in contact with a fluid interface. It sets the velocity component normal to the wall to zero; that is,

$$\mathbf{u} \cdot \mathbf{n}_{wall} = 0$$

and adds a frictional boundary force

$$\mathbf{F}_{\mathrm{fr}} = -\frac{\mu}{\beta}\mathbf{u}$$

Here, β is the slip length. The boundary condition also allows you to specify the contact angle θ , that is, the angle between the wall and the fluid interface (see Figure 2). In this example, the contact angle is 67.5° and the slip length equals the mesh element size, *h*.



Figure 2: Definition of the contact angle θ .

Results and Discussion

The initial development of the fluid interface is shown in Figure 3. During this stage the surface changes drastically in order for it to obtain the prescribed contact angle with the wall. When this is achieved, the surface tension imposed by the surface curvature begins to pull water up through the vertical cylinder. Due to the instantaneous start, the surface oscillates slightly during the rise.



Figure 3: Snapshots of the position of the interface during the first 0.15 ms. Level Set (left) and Phase Field (right) model results.

Figure 4 shows the interface and the velocity field at three different times following the initial stage. After about 0.6 ms the shape of the water surface remains approximately constant and forms a rising concave meniscus. Comparing the velocity field in the Level Set and the Phase Field models, the Level Set results display a small velocity near the wall/

interface contact point, something that is not present in the Phase Field results. This is due to a difference in the wetted wall condition. The Level Set interface requires a wall slip length for the interface to move along the wall. As shown in Figure 4, the imposed slip velocity at the wall is small. In the Phase Field model a slip length is not necessary and the fluid velocity is truly zero on the wall.



Figure 4: Interface and velocity field at different times. Level Set (top) and Phase Field (bottom) model results.

Figure 5 shows surface plots of the pressure at t = 0.6 ms. At the fluid interface there is a pressure jump of roughly 300 Pa. The jump is caused by the surface tension, and forces the water and air to rise through the vertical cylinder.



Figure 5: Pressure at t = 0.6 ms. Level Set (top) and Phase Field (bottom) model results.

You can easily calculate the position of the interface/wall contact point by integrating the level set function along the thin cylinder wall. Figure 6 shows the position of the contact point as a function of time. The slight oscillations of the water surface noted above is seen here also in the contact point plot. The contact plots from the Level Set and Phase Field models compare very well, except for two minor points. The surface oscillation is a bit more pronounced in the Level Set model, and the surface end point is somewhat higher



up in this case. Both these differences are small and are most likely related to the different implementations of the wetted wall boundary condition.

Figure 6: Position of the interface/wall contact point as a function of time. Level Set (top) and Phase Field (bottom) model result. The velocity is approximately constant after t = 0.6 ms.

Finally, you can verify the obtained contact angle. It is defined by $\cos\theta = \mathbf{n}^T \mathbf{n}_{wall}$. In this case, the normal to the wall is $\mathbf{n}_{wall} = \mathbf{e}_r$. The contact angle is thus $\theta = \operatorname{acos} n_r$, where n_r is the radial component of the interface normal. Due to the slight oscillations of the surface, the contact angle varies during the rise. As Figure 7 shows, at t = 0.6 ms the contact angle is 67.7° for the Level Set model and 68.0° for he Phase Field models. Both



results are close to the imposed contact angle of $3\pi/8 = 1.18$ rad = 67.5° . The contact angle further approaches the imposed value if the mesh is refined.

Figure 7: Plot of $acos(n_r)$. At the wall, this gives the contact angle. In the Level Set model (top) the wall angle is 67. \mathcal{P} and in the Phase Field model (bottom) it is 68.0°.

Notes About the COMSOL Implementation

The model is straightforward to set up using either the Level Set or the Phase Field interface. At walls in contact with the fluid interface, you can use the Wetted Wall coupling feature.

The simulation procedure consists of two steps. First the phase field and level set functions are initialized, then the time-dependent calculation starts. This is automatically set up by the software. You only need to specify appropriate times for the initialization step and the time-dependent analysis.

Application Library path: CFD_Module/Multiphase_Tutorials/ capillary_filling_ls

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Two-Phase Flow, Level Set> Laminar Two-Phase Flow, Level Set.
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Transient with Phase Initialization.
- 6 Click Done.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- **3** From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.3.
- 4 In the **Height** text field, type 0.15.

5 Locate the **Position** section. In the **z** text field, type -0.15.

Rectangle 2 (r2)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Rectangle.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the Width text field, type 0.15.
- 5 In the **Height** text field, type 0.5.
- 6 Right-click Rectangle 2 (r2) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the Model Builder window, under Component I (compl)>Geometry I right-click Form Union (fin) and choose Build Selected.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.

ADD MATERIAL

- I Go to the Add Material window.
- 2 In the tree, select Built-In>Water, liquid.
- **3** Click **Add to Component** in the window toolbar.
- 4 On the Home toolbar, click Add Material to close the Add Material window.

MULTIPHYSICS

Wetted Wall I (ww1)

- I On the Physics toolbar, click Multiphysics and choose Boundary>Wetted Wall.
- 2 Select Boundaries 6 and 7 only.
- 3 In the Settings window for Wetted Wall, locate the Wetted Wall section.
- **4** In the θ_w text field, type (3*pi/8)[rad].

LEVEL SET (LS)

In the Model Builder window, under Component I (compl) click Level Set (Is).

Initial Interface 1

- I On the Physics toolbar, click Boundaries and choose Initial Interface.
- 2 Select Boundary 4 only.

MULTIPHYSICS

- I In the Model Builder window, under Component I (compl)>Multiphysics click Two-Phase Flow, Level Set I (tpfl).
- **2** In the **Settings** window for Two-Phase Flow, Level Set, locate the **Fluid I Properties** section.
- 3 From the Fluid I list, choose Air (mat1).
- 4 Locate the Fluid 2 Properties section. From the Fluid 2 list, choose Water, liquid (mat2).

LEVEL SET (LS)

Level Set Model I

- I In the Model Builder window, under Component I (compl)>Level Set (Is) click Level Set Model I.
- 2 In the Settings window for Level Set Model, locate the Level Set Model section.
- 3 In the ε_{ls} text field, type 5e-6.

Initial Values 2

- I On the Physics toolbar, click Domains and choose Initial Values.
- 2 Select Domain 1 only.
- 3 In the Settings window for Initial Values, locate the Initial Values section.
- **4** From the **Domain initially** list, choose **Fluid 2** ($\phi = I$).

Initial Values 1

- I In the Model Builder window, under Component I (comp1)>Level Set (Is) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the **Domain initially** list, choose **Fluid I** ($\phi = 0$).

LAMINAR FLOW (SPF)

On the Physics toolbar, click Level Set (Is) and choose Laminar Flow (spf).

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 In the Settings window for Inlet, locate the Boundary Condition section.
- **3** From the list, choose **Pressure**.
- 4 Select Boundary 8 only.
- **5** Locate the **Pressure Conditions** section. In the p_0 text field, type -tpf1.rho2*z* g_const.

LEVEL SET (LS)

- I In the Model Builder window, under Component I (compl) click Level Set (Is).
- 2 On the Physics toolbar, click Boundaries and choose Inlet.
- 3 Select Boundary 8 only.
- 4 In the Settings window for Inlet, locate the Inlet section.
- **5** In the ϕ text field, type 1.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 5 only.

LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 On the Physics toolbar, click Boundaries and choose Outlet.
- 3 Select Boundary 5 only.

Volume Force 1

- I On the Physics toolbar, click Domains and choose Volume Force.
- 2 In the Model Builder window, right-click Volume Force I and choose Rename.
- 3 In the Rename Volume Force dialog box, type Gravity in the New label text field.
- 4 Click OK.
- 5 In the Settings window for Volume Force, locate the Domain Selection section.
- 6 From the Selection list, choose All domains.
- 7 Locate the Volume Force section. Specify the **F** vector as

0 r -g const*spf.rho

DEFINITIONS

Next, define a variable for the contact angle using the Laminar Two-Phase Flow, Level Set interface's variable for the **r** component of the interface normal, *tpf.intnormr*.

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** In the table, enter the following settings:

Name	Expression	Unit	Description
theta	(acos(tpf1.intnormr))[1/ deg]		Contact angle expression

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Size

- I In the Settings window for Size, locate the Element Size section.
- 2 From the Predefined list, choose Extremely fine.
- 3 Click Build All.



STUDY I

Step 2: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step 2: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range(0,0.25e-4,1e-3).

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Study I>Solver Configurations node.
- **3** In the Model Builder window, expand the Solution 1 (soll) node, then click Dependent Variables **2**.
- 4 In the Settings window for Dependent Variables, locate the Scaling section.
- 5 From the Method list, choose Manual.
- 6 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (solI)>Dependent Variables 2 node, then click Pressure (compl.p).
- 7 In the Settings window for Field, locate the Scaling section.
- 8 From the Method list, choose Manual.
- 9 In the Scale text field, type 100.
- 10 In the Model Builder window, under Study 1>Solver Configurations>Solution 1 (sol1)> Dependent Variables 2 click Level set variable (comp1.phils).
- II In the Settings window for Field, locate the Scaling section.
- 12 From the Method list, choose Manual.
- **I3** On the **Study** toolbar, click **Compute**.

RESULTS

Volume Fraction of Fluid 1 (ls)

- I In the Model Builder window, expand the Volume Fraction of Fluid I (Is) node, then click Volume Fraction of Fluid I.I.
- 2 In the Settings window for Contour, locate the Coloring and Style section.
- **3** From the **Contour type** list, choose **Filled**.
- **4** From the **Coloring** list, choose **Color table**.
- **5** Select the **Color legend** check box.

- 6 From the Legend type list, choose Line.
- 7 On the Volume Fraction of Fluid I (Is) toolbar, click Plot.

Click Yes to confirm.

The fourth default plot group shows the volume fraction of air. While the position of the air/water interface appears clearly, you can obtain an even sharper interface by plotting the 0.5 level of the same quantity using a filled contour plot, as in Figure 3.

- 8 In the Model Builder window, under Results>Volume Fraction of Fluid I (Is) right-click Volume Fraction of Fluid I and choose Delete.
- 9 In the Settings window for 2D Plot Group, locate the Data section.
- **IO** From the **Time (s)** list, choose **0**.
- II On the Volume Fraction of Fluid I (Is) toolbar, click Plot.

Click the **Zoom Box** button on the Graphics toolbar, then zoom in on the lower part of the capillary. Compare the resulting plot with that in the upper-left panel of Figure 3

Reproduce the remaining plots on the left in Figure 3 by plotting the solution for the time values 5e-5, 1e-4, and 1.5e-4.

Velocity (spf)

The first default plot shows a surface plot of the velocity magnitude of the fluids combined with one contour lines to identify the air/water-interface. This plot can be changed to reproduce the combined velocity-field arrows and air/water-interface plot shown in Figure 4.

Click Yes to confirm.

- I In the Model Builder window, expand the Velocity (spf) node.
- 2 Right-click Surface and choose Delete.
- 3 Right-click Velocity (spf) and choose Arrow Surface.
- 4 In the Settings window for Arrow Surface, locate the Arrow Positioning section.
- 5 Find the z grid points subsection. In the Points text field, type 30.
- 6 Right-click Velocity (spf) and choose Contour.
- 7 In the Settings window for Contour, locate the Expression section.
- 8 In the **Expression** text field, type tpf1.Vf1.
- 9 Locate the Levels section. From the Entry method list, choose Levels.
- **IO** In the **Levels** text field, type 0.5.
- II Locate the Coloring and Style section. From the Coloring list, choose Uniform.

- 12 From the Color list, choose Gray.
- **I3** In the **Model Builder** window, click **Velocity (spf)**.
- 14 In the Settings window for 2D Plot Group, locate the Data section.
- **I5** From the **Time (s)** list, choose **2E-4**.
- **I6** On the **Velocity (spf)** toolbar, click **Plot**.

The resulting plot should closely resemble the upper-left plot in Figure 4.

Generate the remaining two plots by choosing the values 4e-4 and 6e-4 from the Time list.

Velocity (spf) 1

The third default plot group shows the air/water-interface as an isosurface plot using a revolved data set. Create an additional revolved data set to further improve the visualization of the interface.

Data Sets

- I In the Model Builder window, expand the Results>Data Sets node, then click Revolution 2D.
- 2 In the Settings window for Revolution 2D, click to expand the Revolution layers section.
- **3** Locate the **Revolution Layers** section. In the **Start angle** text field, type **0**.
- 4 In the **Revolution angle** text field, type 360.
- 5 In the Model Builder window, under Results>Data Sets click Revolution 2D 1.
- 6 In the Settings window for Revolution 2D, click to expand the Revolution layers section.
- 7 Locate the Revolution Layers section. In the Revolution angle text field, type 230.

Volume Fraction of Fluid 1 (ls) 1

- I In the Model Builder window, click Volume Fraction of Fluid I (Is) I.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Data set list, choose Revolution 2D.
- 4 From the Time (s) list, choose 3E-4.
- 5 On the Volume Fraction of Fluid I (Is) I toolbar, click Plot.
- 6 Right-click Volume Fraction of Fluid I (Is) I and choose Volume.
- 7 In the Settings window for Volume, locate the Data section.
- 8 From the Data set list, choose Revolution 2D I.
- 9 From the Time (s) list, choose 3E-4.
- **IO** Locate the **Expression** section. In the **Expression** text field, type **ls**.Vf1.
- II On the Volume Fraction of Fluid I (Is) I toolbar, click Plot.

12 Click the Zoom Extents button on the Graphics toolbar.

Next, plot the pressure at t = 0.6 ms. Compare the result with the upper plot in Figure 5.

Pressure (spf)

- I In the Model Builder window, expand the Results>Pressure (spf) node.
- 2 Right-click **Contour** and choose **Delete**.

Click **Yes** to confirm.

- 3 In the Settings window for 2D Plot Group, locate the Data section.
- 4 From the Time (s) list, choose 6E-4.
- 5 Right-click Pressure (spf) and choose Surface.
- 6 In the Settings window for Surface, locate the Expression section.
- 7 In the **Expression** text field, type p.
- 8 On the Pressure (spf) toolbar, click Plot.
- 9 Click the Zoom Extents button on the Graphics toolbar.

Derived Values

Go on to compute and plot the position of the interface/ contact point.

Line Integration 1

On the Results toolbar, click More Derived Values and choose Integration>Line Integration.

Derived Values

- I Select Boundary 6 only.
- 2 In the Settings window for Line Integration, click Replace Expression in the upper-right corner of the Expressions section. From the menu, choose Component I>Level Set>phils
 Level set variable.
- **3** Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
phils	mm	Contact point position

4 Click Evaluate.

TABLE

- I Go to the Table window.
- 2 Click **Table Graph** in the window toolbar.

RESULTS

ID Plot Group 6

Compare this graph with that in the upper panel of Figure 6.

Finally, check the value of the contact angle at t = 0.6 ms (Figure 7).

2D Plot Group 7

- I On the Results toolbar, click 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Time (s) list, choose 6E-4.
- 4 Right-click 2D Plot Group 7 and choose Contour.
- 5 In the Settings window for Contour, locate the Levels section.
- 6 From the Entry method list, choose Levels.
- 7 Locate the Expression section. In the Expression text field, type 1s.Vf1.
- 8 Locate the Levels section. In the Levels text field, type 0.5.
- 9 Right-click Results>2D Plot Group 7>Contour I and choose Color Expression.
- 10 In the Settings window for Color Expression, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Definitions> Variables>theta Contact angle expression.
- II On the 2D Plot Group 7 toolbar, click Plot.

At this instance the contact angle is 67.7 degrees which can be found by expanding the **Range** section in the **Settings Window** of the **Color Expression**.



Capillary Filling - Phase Field Method

Introduction

Surface tension and wall adhesive forces are often used to transport fluid through microchannels in MEMS devices or to measure the transport and position of small amounts of fluid using micropipettes. Multiphase flow through a porous medium and droplets on solid walls are other examples where wall adhesion and surface tension strongly influence the dynamics of the flow.

This example studies a narrow vertical cylinder placed on top of a reservoir filled with water. Because of wall adhesion and surface tension at the air/water interface, water rises through the channel. The model calculates the velocity field, the pressure field, and the shape and position of the water surface.

This example demonstrates how to model the filling of a capillary channel using two multiphysics coupling features available in the CFD Module. You can use either the Two-Phase Flow, Level Set or the Two-Phase Flow, Phase Field multiphysics coupling feature. The Level Set interface uses a reinitialized level set method to represent the fluid interface between the air and the water. The Phase Field interface, on the other hand, uses a Cahn-Hilliard equation, including a chemical potential to represent a diffuse interface separating the two phases. The Navier-Stokes equations are used to describe the momentum transport and the conservation of mass.

Model Definition

The model consists of a capillary channel of radius 0.15 mm attached to a water reservoir. Water can flow freely into the reservoir. Because both the channel and the reservoir are cylindrical, you can use the axisymmetric geometry illustrated in Figure 1. Initially, the thin cylinder is filled with air. Wall adhesion causes water to creep up along the cylinder boundaries. The deformation of the water surface induces surface tension at the air/water interface, which in turn creates a pressure jump across the interface. The pressure variations cause water and air to move upward. The fluids continue to rise until the capillary forces are balanced by the gravity force building up as the water rises in the



channel. In the present example, the capillary forces dominate over gravity throughout the simulation. Consequently, the interface moves upwards during the entire simulation.

Figure 1: Axisymmetric geometry with boundary conditions.

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The Level Set interface automatically sets up the equations for the convection of the interface. The fluid interface is represented by the 0.5 contour of the level set function ϕ . In air $\phi = 0$ and in water $\phi = 1$. The level set function can thus be thought of as the volume fraction of water. The transport of the fluid interface separating the two phases is given by

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = \gamma \nabla \cdot \left(\varepsilon \nabla \phi - \phi (1 - \phi) \frac{\nabla \phi}{|\nabla \phi|} \right)$$

The ε parameter determines the thickness of the interface. When stabilization is used for the level set equation, you can typically use an interface thickness of $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The γ parameter determines the amount of reinitialization. A suitable value for γ is the maximum velocity magnitude occurring in the model. The multiphysics coupling feature defines the density and viscosity according to:

$$\rho = \rho_{air} + (\rho_{water} - \rho_{air})\phi$$
$$\mu = \mu_{air} + (\mu_{water} - \mu_{air})\phi$$

Due to these definitions, the density and viscosity vary smoothly across the fluid interface. The delta function is approximated by

$$\delta = 6|\phi(1-\phi)||\nabla\phi|$$

and the interface normal is calculated from

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|}$$

Phase Field Method

In the Phase Field interface the two-phase flow dynamics is governed by a Cahn-Hilliard equation. The equation tracks a diffuse interface separating the immiscible phases. The diffuse interface is defined as the region where the dimensionless phase field variable ϕ goes from -1 to 1. When solved in COMSOL Multiphysics, the Cahn-Hilliard equation is split up into two equations

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = \nabla \cdot \frac{\gamma \lambda}{\epsilon^2} \nabla \psi$$
$$\psi = -\nabla \cdot \epsilon^2 \nabla \phi + (\phi^2 - 1)\phi$$

where **u** is the fluid velocity (m/s), γ is the mobility (m³·s/kg), λ is the mixing energy density (N) and ϵ (m) is the interface thickness parameter. The ψ variable is referred to as the phase field help variable. The following equation relates the mixing energy density and the interface thickness to the surface tension coefficient:

$$\sigma = \frac{2\sqrt{2}\lambda}{3\epsilon}$$

You can typically set the interface thickness parameter to $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The mobility parameter γ determines the time scale of the Cahn-Hilliard diffusion and must be chosen judiciously. It must be large enough to retain a constant interfacial thickness but small enough so that the convective terms are not overly damped. The default value, $\gamma = \varepsilon^2$, is usually a good initial guess. This model uses a higher mobility to obtain the correct pressure variation over the interface.

In the Phase Field interface, the volume fractions of the individual fluids are

$$V_{\rm f1} = \frac{1-\phi}{2}, \qquad V_{\rm f2} = \frac{1+\phi}{2}$$

In the present model water is defined as Fluid 1 and air as Fluid 2.

The multiphysics coupling feature defines the density (kg/m^3) and the viscosity $(Pa \cdot s)$ of the mixture to vary smoothly over the interface by letting

$$\rho = \rho_{\rm w} + (\rho_{\rm air} - \rho_{\rm w})V_{\rm f2}$$
$$\mu = \mu_{\rm w} + (\mu_{\rm air} - \mu_{\rm w})V_{\rm f2}$$

where the single phase water properties are denoted w and the air properties air.

MASS AND MOMENTUM TRANSPORT

The Navier-Stokes equations describe the transport of mass and momentum for fluids of constant density. In order to account for capillary effects, it is crucial to include surface tension in the model. The Navier-Stokes equations are then

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot [-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^{T})] + \mathbf{F}_{st} + \rho \mathbf{g}$$
$$\nabla \cdot \mathbf{u} = 0$$

Here, ρ denotes the density (kg/m³), μ equals the dynamic viscosity (Ns/m²), **u** represents the velocity (m/s), *p* denotes the pressure (Pa), and **g** is the gravity vector (m/s²). **F**_{st} is the surface tension force acting at the air/water interface.

Surface Tension

In the Level Set interface the surface tension force is computed as

$$\mathbf{F}_{st} = \nabla \cdot \mathbf{T}$$
$$\mathbf{T} = \sigma(\mathbf{I} - (\mathbf{nn}^T))\delta$$

Here, **I** is the identity matrix, **n** is the interface normal, σ equals the surface tension coefficient (N/m), and δ equals a Dirac delta function that is nonzero only at the fluid interface. When you use the finite element method to solve the Navier-Stokes equations, you multiply the equations by test functions and then integrate over the computational domain. If you use integration by parts, you can move derivatives of **T** to the test functions. This i results in an integral over the computational domain plus a boundary integral of the form

$$\int_{\partial\Omega} \operatorname{test}(\mathbf{u}) \cdot [\sigma(\mathbf{n}_{\text{wall}} - (\mathbf{n}\cos\theta))\delta] dS$$
(1)

where θ is the contact angle (see Figure 2). If you apply a no slip boundary condition, the boundary term vanishes because test(**u**) = 0 on that boundary, and you cannot specify the contact angle. Instead, the interface remains fixed on the wall. However, if you allow a small amount of slip, it is possible to specify the contact angle. The Wetted Wall coupling feature adds the term given by Equation 1 and consequently makes it possible to set the contact angle.

In the Phase Field interface, the diffuse interface representation makes it possible to compute the surface tension by

$$\mathbf{F}_{st} = G\nabla\phi$$

where ϕ is the phase field parameter, and *G* is the chemical potential (J/m³)

$$G = \lambda \left[-\nabla^2 \phi + \frac{\phi(\phi^2 - 1)}{\epsilon^2} \right] = \frac{\lambda}{\epsilon^2} \psi$$

As seen above, the phase field surface tension is computed as a distributed force over the interface using only ψ and the gradient of the phase field variable. This computation avoids using the surface normal and the surface curvature, which are troublesome to represent numerically.

INITIAL CONDITIONS

Initially, the reservoir is filled with water and the capillary channel is filled with air. The initial velocity is zero.

BOUNDARY CONDITIONS

Inlet

The hydrostatic pressure, $p = \rho gz$, gives the pressure at the inflow boundary. Only water enters through the inlet, so the level set function (that is, the volume fraction of water) is 1 here.

Outlet

At the outlet, the pressure is equal to zero, that is, equal to the pressure at the top of the inflow boundary. Because it is an outflow boundary, you do not have to set any condition on the level set function.

Walls

The Wetted Wall feature is suitable for solid walls in contact with a fluid interface. It sets the velocity component normal to the wall to zero; that is,

$$\mathbf{u} \cdot \mathbf{n}_{wall} = 0$$

and adds a frictional boundary force

$$\mathbf{F}_{\mathrm{fr}} = -\frac{\mu}{\beta}\mathbf{u}$$

Here, β is the slip length. The boundary condition also allows you to specify the contact angle θ , that is, the angle between the wall and the fluid interface (see Figure 2). In this example, the contact angle is 67.5° and the slip length equals the mesh element size, *h*.



Figure 2: Definition of the contact angle θ .

Results and Discussion

The initial development of the fluid interface is shown in Figure 3. During this stage the surface changes drastically in order for it to obtain the prescribed contact angle with the wall. When this is achieved, the surface tension imposed by the surface curvature begins to pull water up through the vertical cylinder. Due to the instantaneous start, the surface oscillates slightly during the rise.



Figure 3: Snapshots of the position of the interface during the first 0.15 ms. Level Set (left) and Phase Field (right) model results.

Figure 4 shows the interface and the velocity field at three different times following the initial stage. After about 0.6 ms the shape of the water surface remains approximately constant and forms a rising concave meniscus. Comparing the velocity field in the Level Set and the Phase Field models, the Level Set results display a small velocity near the wall/

interface contact point, something that is not present in the Phase Field results. This is due to a difference in the wetted wall condition. The Level Set interface requires a wall slip length for the interface to move along the wall. As shown in Figure 4, the imposed slip velocity at the wall is small. In the Phase Field model a slip length is not necessary and the fluid velocity is truly zero on the wall.



Figure 4: Interface and velocity field at different times. Level Set (top) and Phase Field (bottom) model results.

Figure 5 shows surface plots of the pressure at t = 0.6 ms. At the fluid interface there is a pressure jump of roughly 300 Pa. The jump is caused by the surface tension, and forces the water and air to rise through the vertical cylinder.



Figure 5: Pressure at t = 0.6 ms. Level Set (top) and Phase Field (bottom) model results.

You can easily calculate the position of the interface/wall contact point by integrating the level set function along the thin cylinder wall. Figure 6 shows the position of the contact point as a function of time. The slight oscillations of the water surface noted above is seen here also in the contact point plot. The contact plots from the Level Set and Phase Field models compare very well, except for two minor points. The surface oscillation is a bit more pronounced in the Level Set model, and the surface end point is somewhat higher


up in this case. Both these differences are small and are most likely related to the different implementations of the wetted wall boundary condition.

Figure 6: Position of the interface/wall contact point as a function of time. Level Set (top) and Phase Field (bottom) model result. The velocity is approximately constant after t = 0.6 ms.

Finally, you can verify the obtained contact angle. It is defined by $\cos\theta = \mathbf{n}^T \mathbf{n}_{wall}$. In this case, the normal to the wall is $\mathbf{n}_{wall} = \mathbf{e}_r$. The contact angle is thus $\theta = \operatorname{acos} n_r$, where n_r is the radial component of the interface normal. Due to the slight oscillations of the surface, the contact angle varies during the rise. As Figure 7 shows, at t = 0.6 ms the contact angle is 67.7° for the Level Set model and 68.0° for he Phase Field models. Both



results are close to the imposed contact angle of $3\pi/8 = 1.18$ rad = 67.5° . The contact angle further approaches the imposed value if the mesh is refined.

Figure 7: Plot of $acos(n_r)$. At the wall, this gives the contact angle. In the Level Set model (top) the wall angle is 67. \mathcal{P} and in the Phase Field model (bottom) it is 68.0°.

Notes About the COMSOL Implementation

The model is straightforward to set up using either the Level Set or the Phase Field interface. At walls in contact with the fluid interface, you can use the Wetted Wall coupling feature.

The simulation procedure consists of two steps. First the phase field and level set functions are initialized, then the time-dependent calculation starts. This is automatically set up by the software. You only need to specify appropriate times for the initialization step and the time-dependent analysis.

Application Library path: CFD_Module/Multiphase_Tutorials/ capillary_filling_pf

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Two-Phase Flow, Phase Field> Laminar Two-Phase Flow, Phase Field.
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Transient with Phase Initialization.
- 6 Click Done.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- **3** From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.3.
- **4** In the **Height** text field, type **0.15**.

5 Locate the Position section. In the z text field, type -0.15.

Rectangle 2 (r2)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Rectangle.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the Width text field, type 0.15.
- 5 In the **Height** text field, type 0.5.
- 6 Right-click Rectangle 2 (r2) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the Model Builder window, under Component I (compl)>Geometry I right-click Form Union (fin) and choose Build Selected.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.

ADD MATERIAL

- I Go to the Add Material window.
- 2 In the tree, select Built-In>Water, liquid.
- **3** Click **Add to Component** in the window toolbar.
- 4 On the Home toolbar, click Add Material to close the Add Material window.

PHASE FIELD (PF)

On the Physics toolbar, click Laminar Flow (spf) and choose Phase Field (pf).

In the Model Builder window, expand the Component I (compl)>Laminar Flow (spf) node, then click Component I (compl)>Phase Field (pf).

Initial Interface 1

- I On the Physics toolbar, click Boundaries and choose Initial Interface.
- **2** Select Boundary 4 only.

MULTIPHYSICS

- I In the Model Builder window, under Component I (compl)>Multiphysics click Two-Phase Flow, Phase Field I (tpfl).
- 2 In the Settings window for Two-Phase Flow, Phase Field, locate the Fluid I Properties section.
- 3 From the Fluid I list, choose Air (matl).
- 4 Locate the Fluid 2 Properties section. From the Fluid 2 list, choose Water, liquid (mat2).

PHASE FIELD (PF)

Phase Field Model 1

- I In the Model Builder window, under Component I (compl)>Phase Field (pf) click Phase Field Model I.
- 2 In the Settings window for Phase Field Model, locate the Phase Field Parameters section.
- **3** In the χ text field, type 50.
- 4 In the ε_{pf} text field, type 6.5e-6.

Initial Values 2

- I On the Physics toolbar, click Domains and choose Initial Values.
- 2 Select Domain 1 only.
- 3 In the Settings window for Initial Values, locate the Initial Values section.
- 4 From the Domain initially list, choose Fluid 2 ($\phi = 1$).

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Phase Field (pf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the **Domain initially** list, choose **Fluid I** ($\phi = -I$).

LAMINAR FLOW (SPF)

On the Physics toolbar, click Phase Field (pf) and choose Laminar Flow (spf).

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 8 only.
- 3 In the Settings window for Inlet, locate the Boundary Condition section.

- **4** From the list, choose **Pressure**.
- 5 Locate the Pressure Conditions section. In the p₀ text field, type -tpf1.rho2*z* g_const.

This is the hydrostatic pressure at the inlet.

PHASE FIELD (PF)

- I In the Model Builder window, under Component I (compl) click Phase Field (pf).
- 2 On the Physics toolbar, click Boundaries and choose Inlet.
- 3 Select Boundary 8 only.
- 4 In the Settings window for Inlet, locate the Inlet section.
- **5** In the $V_{\rm f}$ text field, type 1.

LAMINAR FLOW (SPF)

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 5 only.

PHASE FIELD (PF)

- I In the Model Builder window, under Component I (compl) click Phase Field (pf).
- 2 On the Physics toolbar, click Boundaries and choose Outlet.
- 3 Select Boundary 5 only.

Wetted Wall 2

- I On the Physics toolbar, click Boundaries and choose Wetted Wall.
- **2** Select Boundaries 6 and 7 only.
- 3 In the Settings window for Wetted Wall, locate the Wetted Wall section.
- **4** In the θ_w text field, type (3*pi/8)[rad].

LAMINAR FLOW (SPF)

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Volume Force 1

- I On the Physics toolbar, click Domains and choose Volume Force.
- 2 In the Model Builder window, right-click Volume Force I and choose Rename.
- 3 In the Rename Volume Force dialog box, type Gravity in the New label text field.

- 4 Click OK.
- 5 In the Settings window for Volume Force, locate the Domain Selection section.
- 6 From the Selection list, choose All domains.
- 7 Locate the Volume Force section. Specify the F vector as



DEFINITIONS

Next, define a variable for the contact angle.

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** In the table, enter the following settings:

Name	Expression	Unit	Description
intnormr	d(phipf,r)/sqrt(d(phipf,r)^2+ d(phipf,z)^2+eps)		Interface normal, r component
theta	(acos(intnormr))[1/deg]		Contact angle

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Size

- I In the Settings window for Size, locate the Element Size section.
- 2 From the Predefined list, choose Extremely fine.



STUDY I

Step 2: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step 2: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range(0,0.25e-4,1e-3).

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Study I>Solver Configurations node.
- **3** In the Model Builder window, expand the Solution 1 (sol1) node, then click Dependent Variables **2**.
- 4 In the Settings window for Dependent Variables, locate the Scaling section.
- 5 From the Method list, choose Manual.
- 6 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll)>Dependent Variables 2 node, then click Phase field help variable (compl.psi).
- 7 In the Settings window for Field, locate the Scaling section.
- 8 From the Method list, choose Manual.

- 9 In the Scale text field, type 0.01.
- 10 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables 2 click Pressure (compl.p).
- II In the Settings window for Field, locate the Scaling section.
- 12 From the Method list, choose Manual.
- **I3** In the **Scale** text field, type 100.
- **I4** On the **Study** toolbar, click **Compute**.

RESULTS

Volume Fraction of Fluid 1 (pf)

- I In the Model Builder window, expand the Volume Fraction of Fluid I (pf) node.
- 2 Right-click Volume Fraction of Fluid I and choose Delete.

Click **Yes** to confirm.

The third default plot group shows the volume fraction of air. While the position of the air/water interface appears clearly, you can obtain an even sharper interface by plotting the 0.5 level of the same quantity using a filled contour plot, as in Figure 3.

- 3 In the Model Builder window, under Results>Volume Fraction of Fluid I (pf) click Volume Fraction of Fluid 1.1.
- 4 In the Settings window for Contour, locate the Levels section.
- 5 From the Entry method list, choose Levels.
- 6 In the Levels text field, type 0.5.
- 7 Locate the Coloring and Style section. From the Contour type list, choose Filled.
- 8 From the Coloring list, choose Color table.
- 9 Select the Color legend check box.
- **IO** From the **Legend type** list, choose **Line**.
- II In the Model Builder window, click Volume Fraction of Fluid I (pf).
- 12 In the Settings window for 2D Plot Group, locate the Data section.
- **I3** From the **Time (s)** list, choose **0**.
- 14 On the Volume Fraction of Fluid I (pf) toolbar, click Plot.

Click the **Zoom Box** button on the Graphics toolbar, then zoom in on the lower part of the capillary. Compare the resulting plot with that in the upper-right panel of Figure 3.

Velocity (spf)

The second default plot shows a surface plot of the velocity magnitude of the fluids. This plot can be changed to reproduce the combined velocity-field arrows and air/water-interface plot shown in Figure 4.

- I In the Model Builder window, expand the Velocity (spf) node.
- 2 Right-click **Surface** and choose **Delete**.

Click **Yes** to confirm.

- 3 Right-click Velocity (spf) and choose Contour.
- 4 In the Settings window for Contour, locate the Expression section.
- **5** In the **Expression** text field, type pf.Vf1.
- 6 Locate the Levels section. From the Entry method list, choose Levels.
- 7 In the Levels text field, type 0.5.
- 8 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 9 From the Color list, choose Gray.
- **IO** On the **Velocity (spf)** toolbar, click **Plot**.
- II Click the **Zoom Extents** button on the **Graphics** toolbar.
- 12 Right-click Velocity (spf) and choose Arrow Surface.
- **I3** In the **Settings** window for Arrow Surface, locate the **Arrow Positioning** section.
- 14 Find the z grid points subsection. In the Points text field, type 30.
- **I5** In the **Model Builder** window, click **Velocity (spf)**.
- 16 In the Settings window for 2D Plot Group, locate the Data section.
- **I7** From the **Time (s)** list, choose **2E-4**.
- **18** On the **Velocity (spf)** toolbar, click **Plot**.

The resulting plot should closely resemble the upper-right plot in Figure 4.

Generate the remaining two plots by choosing the values 4e-4 and 6e-4 from the Time list.

Volume Fraction of Fluid 1 (pf) 1

The third default plot group shows the air/water-interface as an isosurface plot using a revolved data set.

I In the Model Builder window, expand the Results>Views node.

Data Sets

- I In the Model Builder window, expand the Results>Data Sets node, then click Revolution 2D.
- 2 In the Settings window for Revolution 2D, click to expand the Revolution layers section.
- 3 Locate the Revolution Layers section. In the Start angle text field, type 0.
- **4** In the **Revolution angle** text field, type **360**.
- 5 In the Model Builder window, under Results>Data Sets click Revolution 2D 1.
- 6 In the Settings window for Revolution 2D, click to expand the Revolution layers section.
- 7 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type 230.

Volume Fraction of Fluid I (pf) I

- I In the Model Builder window, expand the Volume Fraction of Fluid I (pf) I node, then click Isosurface I.
- 2 In the Settings window for Isosurface, locate the Data section.
- 3 From the Data set list, choose Revolution 2D.
- 4 From the Time (s) list, choose 3E-4.
- **5** In the Model Builder window, right-click Volume Fraction of Fluid I (pf) I and choose Volume.
- 6 In the Settings window for Volume, locate the Data section.
- 7 From the Data set list, choose Revolution 2D I.
- 8 From the Time (s) list, choose 3E-4.
- 9 Locate the Expression section. In the Expression text field, type pf.Vf1.

IO On the Volume Fraction of Fluid I (pf) I toolbar, click Plot.

Next, plot the pressure at t = 0.6 ms. Compare the result with the upper plot in Figure 5.

2D Plot Group 6

- I On the Home toolbar, click Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Time (s) list, choose 6E-4.
- 4 Right-click 2D Plot Group 6 and choose Surface.
- 5 In the Settings window for Surface, locate the Expression section.
- 6 In the **Expression** text field, type p.
- 7 On the 2D Plot Group 6 toolbar, click Plot.
- 8 Click the Zoom Extents button on the Graphics toolbar.

Derived Values

Go on to compute and plot the position of the interface/wall contact point.

Line Integration 1

On the Results toolbar, click More Derived Values and choose Integration>Line Integration.

Derived Values

- I Select Boundary 6 only.
- 2 In the Settings window for Line Integration, locate the Expressions section.
- **3** In the table, enter the following settings:

Expression	Unit	Description
pf.Vf2	mm	Volume fraction of fluid 2

4 Click Evaluate.

TABLE

- I Go to the Table window.
- 2 Click Table Graph in the window toolbar.

RESULTS

ID Plot Group 7

Compare this graph with that in the lower panel of Figure 6.

Finally, check the value of the contact angle at t = 0.6 ms (Figure 7).

2D Plot Group 8

- I On the **Results** toolbar, click **2D Plot Group**.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Time (s) list, choose 6E-4.
- 4 Right-click 2D Plot Group 8 and choose Contour.
- **5** In the **Settings** window for Contour, locate the **Expression** section.
- 6 In the **Expression** text field, type pf.Vf1.
- 7 Locate the Levels section. From the Entry method list, choose Levels.
- 8 In the Levels text field, type 0.5.
- 9 Right-click Results>2D Plot Group 8>Contour I and choose Color Expression.

- 10 In the Settings window for Color Expression, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Definitions> Variables>theta Contact angle.
- II On the 2D Plot Group 8 toolbar, click Plot.

At this instance the contact angle is 68.0 degrees which can be found by expanding the **Range** section in the **Settings Window** of the **Color Expression** node created.



Two-Phase Flow Modeling of a Dense Suspension

Introduction

Liquid-solid mixtures (suspensions) are important in a variety of industrial fields, such as oil and gas refinement, paper manufacturing, food processing, slurry transport, and wastewater treatment. Several different modeling approaches have been developed, ranging from discrete, particle-based methods to macroscopic, semi-empirical two-phase descriptions. Particle-based methods are suitable when there is a limited number of solid particles. When, on the other hand, there are many particles, it is better to use a macroscopic, or averaged, model that tracks the volume fractions of the phases.

The following example illustrates how you can set up a macroscopic two-phase flow model in COMSOL Multiphysics using the Mixture Model, Laminar Flow interface. The model is based on the "diffusive flux" model described in Ref. 1, Ref. 2, and Ref. 3, suitable for liquid-solid mixtures with high concentrations of solid particles. It accounts for not only buoyancy effects but also shear-induced migration; that is, the tendency of particles to migrate toward regions of lower shear rates.

The model simulates the flow of a dense suspension consisting of light, solid particles in a liquid placed between two concentric cylinders. The inner cylinder rotates while the outer one is fixed.

Model Definition

A suspension is a mixture of solid particles and a liquid. The dynamics of a suspension can be modeled by a momentum transport equation for the mixture, a continuity equation, and a transport equation for the solid phase volume fraction. The Mixture Model, Laminar Flow interface automatically sets up these equations. It uses the following equation to model the momentum transport:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p - \nabla \cdot (\rho c_s(1 - c_s)\mathbf{u}_{\text{slip}}\mathbf{u}_{\text{slip}}) + \nabla \cdot [\eta(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \rho \mathbf{g}$$

where **u** is the mass averaged mixture velocity (m/s), *p* denotes the pressure (Pa), **g** refers to the acceleration of gravity (m/s²), *c_s* is the dimensionless particle mass fraction, and **u**_{slip} gives the relative velocity between the solid and the liquid phases (m/s). Further, $\rho = (1 - \phi_s)\rho_f + \phi_s\rho_s$ is the mixture density, where ρ_f and ρ_s are the pure-phase densities (kg/m³) of liquid and solids, respectively, and ϕ_s is the solid-phase volume fraction (m³/m³). Finally, η represents the mixture viscosity (Ns/m²) according to the Krieger-type expression

$$\eta = \eta_f \left(1 - \frac{\phi_s}{\phi_{max}} \right)^{-2.5\phi_{max}}$$
(1)

where η_f is the dynamic viscosity of the pure fluid and ϕ_{max} is the maximum packing concentration.

The mixture model uses the following form of the continuity equation

$$(\rho_{\rm f} - \rho_{\rm s}) [\nabla \cdot (\phi_{\rm s} (1 - c_{\rm s}) \mathbf{u}_{\rm slip})] + \rho_{\rm f} (\nabla \cdot \mathbf{u}) = 0$$
⁽²⁾

The transport equation for the solid-phase volume fraction is

$$\frac{\partial \phi_{\rm s}}{\partial t} + \nabla \cdot (\phi_{\rm s} \mathbf{u}_{\rm s}) = 0 \tag{3}$$

The solid-phase velocity, \mathbf{u}_s , is given by $\mathbf{u}_s = \mathbf{u} + (1 - c_s) \mathbf{u}_{slip}$. Consequently, Equation 3 is equivalent to

$$\frac{\partial \phi_{\rm s}}{\partial t} + \nabla \cdot (\phi_{\rm s} \mathbf{u} + \phi_{\rm s} (1 - c_s) \mathbf{u}_{\rm slip}) = 0$$
(4)

Rao and others (Ref. 2) formulate the continuity equation and the particle transport in a slightly different way. Instead of the slip velocity, \mathbf{u}_{slip} , they define a particle flux, \mathbf{J}_s (kg $/(\text{m}^2 \cdot \text{s})$), and write the continuity equation as

$$\nabla \cdot \mathbf{u} = \frac{\rho_{\rm s} - \rho_{\rm f}}{\rho_{\rm s} \rho_{\rm f}} (\nabla \cdot \mathbf{J}_{\rm s})$$
(5)

and the solid phase transport according to

$$\frac{\partial \phi_{\rm s}}{\partial t} + \nabla \cdot (\phi_{\rm s} \mathbf{u}) = -\frac{\nabla \cdot \mathbf{J}_{\rm s}}{\rho_{\rm s}}$$
(6)

By comparing Equation 5 and Equation 6 with Equation 2 and Equation 4, it is clear that they are equivalent if

$$\mathbf{u}_{\rm slip} = \frac{\mathbf{J}_{\rm s}}{\phi_{\rm s}\rho_{\rm s}(1-c_{\rm s})}$$

In this example you use the model for the particle flux, J_s , as suggested by Subia and others (Ref. 3) and Rao and others (Ref. 2), but the open and editable format of COMSOL Multiphysics makes it possible to specify the expression arbitrarily.

Following Rao and others, the particle flux is

$$\frac{\mathbf{J}_{\rm s}}{\rho_{\rm s}} = - [\phi \mathbf{D}_{\phi} \nabla(\dot{\gamma} \phi) + \phi^2 \dot{\gamma} \mathbf{D}_{\eta} \nabla(\ln \eta)] + f_{\rm h} \mathbf{u}_{\rm st} \phi$$

Here, \bm{u}_{st} is the settling velocity (m/s) of a single particle surrounded by fluid and D_{ϕ} and D_{η} are empirically fitted parameters (m^2) given by

$$D_{\phi} = 0.41a^2$$
$$D_{\eta} = 0.62a^2$$

where a is the particle radius (m).

The shear rate tensor, $\dot{\gamma}$ (1/s), is given by

$$\dot{\boldsymbol{\gamma}} = \nabla \mathbf{u} + (\nabla \mathbf{u})^T$$

and its magnitude by

$$\dot{\gamma} = \sqrt{\frac{1}{2}}(\dot{\dot{\gamma}}:\dot{\check{\gamma}})$$

which for a 2-dimensional problem is

$$\dot{\gamma} = \sqrt{\frac{1}{2}(4u_x^2 + 2(u_y + v_x)^2 + 4v_y^2)}$$

The settling velocity, \mathbf{u}_{st} , for a single spherical particle surrounded by pure fluid is given by

$$\mathbf{u}_{\rm st} = \frac{2}{9} \frac{a^2(\rho_{\rm s} - \rho_{\rm f})}{\eta_0} \mathbf{g}$$

For several particles in a fluid, the settling velocity is lower. To account for the surrounding particles, the settling velocity for a single particle is multiplied by the hindering function, $f_{\rm h}$, defined as

$$f_{\rm h} = \frac{\eta_{\rm f}(1-\phi_{\rm av})}{\eta}$$

where ϕ_{av} is the average solid phase volume fraction in the suspension, η_f is the dynamic viscosity of the pure fluid (Ns/m²), and η is the mixture viscosity (Equation 1).

NAME	VALUE	DESCRIPTION
$\rho_{\rm s}$	1180 kg/m ³	Density of particles
ρ_{f}	1250 kg/m ³	Density of pure fluid
a	678 μm	Particle radius
$\eta_{\rm f}$	0.589 Pa·s	Viscosity of pure fluid

The following table gives the physical properties of the solid and the liquid phases:

BOUNDARY CONDITIONS

The suspension is placed in a Couette device, that is, between two concentric cylinders. The inner cylinder rotates while the outer one is fixed. The radii of the two cylinders are 0.64 cm and 2.54 cm, respectively. The inner cylinder rotates at a steady rate of 55 rpm. With the cylinder centered at (0,0), this corresponds to a velocity of

$$(u, v) = \frac{110\pi}{60}(y, -x)$$

The fluid and particle motion is small along the direction of the cylinder axis. You can therefore use a 2-dimensional model. Figure 1 shows the corresponding geometry.



Figure 1: Geometry of the Couette device. The inner cylinder rotates, the outer one is fixed.

There is no particle flux through the boundaries, and the suspension velocity satisfies no slip conditions at all walls.

INITIAL CONDITIONS

There are two different initial particle distributions. In the first example, the particles are evenly distributed within the device. In the second example, the particles are initially gathered at the top of the device.

CASE I—INITIALLY EVENLY DISTRIBUTED PARTICLES

A suspension with particles lighter than the fluid is placed in a concentric Couette device. Initially, the particles are evenly distributed with a constant volume fraction of 0.35. The shear rate in the device varies radially across the gap and thus it is expected that particles migrate (shear-induced migration) from regions of high shear to regions of low shear (toward the outer wall). Because the particles are lighter than the fluid, they also rise.



Figure 2: The particle concentration ϕ_s at different times. The particles move to regions with lower shear rate and rise because of buoyancy.

Figure 2 shows the particle concentration ϕ_s in the device at t = 0 s, t = 30 s, t = 100 s and t = 1000 s. The migration of the particles toward the outer wall is apparent. As a result of the shear induced migration and gravity, the solid phase volume fraction approaches the value for maximum packing close to the upper right outer wall. The suspension viscosity thus becomes high in this region. The results compare well with those presented in Ref. 2.

CASE 2-PARTICLES INITIALLY GATHERED AT THE TOP OF THE DEVICE

In this case the particles are initially gathered at the top of the device. The particle volume fraction is initially zero in the lower part, while it is 0.59 at the top.



Figure 3: Particle concentrations for t = 0 s, 10 s, 20 s, and 100 s with particles initially at the top. Note that the same color range values are used in each of the plots.

Figure 3 shows the numerically predicted particle concentration at times 0 s, 10 s, 20 s, and 100 s. Initially, the particle motion is dominated by inertia and the effect of the shear-induced migration is not visible. At later times, shear-induced migration causes the particles to move toward the outer boundary. In this case also, the results agree well with the results in Ref. 2.

References

1. R.J. Phillips, R.C. Armstrong, R.A. Brown, A.L. Graham, and J.R. Abbot, "A Constitutive Equation for Concentrated Suspensions that Accounts for Shear-induced Particle Migration," *Phys. Fluids A*, vol. 4, pp. 30–40, 1992.

2. R. Rao, L. Mondy, A. Sun, and S. Altobelli, "A Numerical and Experimental Study of Batch Sedimentation and Viscous Resuspension," *Int. J. Num. Methods in Fluids*, vol. 39, pp. 465–483, 2002.

3. S.R. Subia, M.S. Ingber, L.A. Mondy, S.A. Altobelli, and A.L. Graham, "Modelling of Concentrated Suspensions Using a Continuum Constitutive Equation," *J. Fluid Mech.*, vol. 373, pp. 193–219, 1998.

Notes About the COMSOL Implementation

To set up the model with COMSOL Multiphysics, open the Mixture Model, Laminar Flow interface. The shear rate is discretized as an additional equation to improve accuracy because the particle flux contains derivatives of this quantity, which in turn depend on the derivatives of the velocity.

Application Library path: CFD_Module/Multiphase_Benchmarks/

dense_suspension

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Mixture Model>Mixture Model, Laminar Flow (mm).
- 3 Click Add.
- 4 In the Select Physics tree, select Mathematics>PDE Interfaces>General Form PDE (g).
- 5 Click Add.
- 6 In the Dependent variables table, enter the following settings:

gamma

7 Click Study.

- 8 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Time Dependent.
- 9 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Click Load from File.
- **4** Browse to the application's Application Library folder and double-click the file dense_suspension_parameters.txt.

GEOMETRY I

Circle 1 (c1)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.0064.

Circle 2 (c2)

- I Right-click Circle I (cl) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Circle.
- 3 In the Settings window for Circle, locate the Size and Shape section.
- 4 In the **Radius** text field, type 0.0254.

Compose I (col)

- I Right-click Circle 2 (c2) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Compose.
- 3 Click in the Graphics window and then press Ctrl+A to select both objects.
- 4 In the Settings window for Compose, locate the Compose section.
- 5 In the Set formula text field, type c2-c1.
- 6 Right-click Compose I (col) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Define an integration coupling operator that you will use to specify the point constraint.

Integration 1 (intop1)

- I On the Definitions toolbar, click Component Couplings and choose Integration.
- 2 In the Settings window for Integration, locate the Source Selection section.
- 3 From the Selection list, choose All domains.

Variables I

- I On the **Definitions** toolbar, click **Local Variables**.
- 2 In the Settings window for Variables, locate the Variables section.
- 3 Click Load from File.
- 4 Browse to the application's Application Library folder and double-click the file dense_suspension_variables.txt.

MIXTURE MODEL, LAMINAR FLOW (MM)

Now, define the Slip model, the Mixture Properties, and the Initial Values.

- I In the Model Builder window, under Component I (compl) click Mixture Model, Laminar Flow (mm).
- **2** In the **Settings** window for Mixture Model, Laminar Flow, locate the **Physical Model** section.
- 3 From the Slip model list, choose User defined.

Mixture Properties 1

- In the Model Builder window, under Component I (compl)>Mixture Model, Laminar Flow (mm) click Mixture Properties I.
- **2** In the **Settings** window for Mixture Properties, locate the **Continuous Phase Properties** section.
- 3 From the ρ_c list, choose User defined. In the associated text field, type rho_f.
- 4 From the μ_c list, choose User defined. In the associated text field, type eta_f.
- 5 Locate the Dispersed Phase Properties section. From the ρ_d list, choose User defined. In the associated text field, type rho_s.
- **6** Locate the **Mixture Model** section. Specify the \mathbf{u}_{slip} vector as

J1/(phid*rho_s*(1-mm.cd)) x J2/(phid*rho_s*(1-mm.cd)) y

7 In the ϕ_{max} text field, type phi_max.

Initial Values 1

- In the Model Builder window, under Component I (compl)>Mixture Model, Laminar Flow (mm) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the ϕ_d text field, type phi0.

Gravity I

- I On the Physics toolbar, click Domains and choose Gravity.
- 2 In the Settings window for Gravity, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundaries 3, 4, 6, and 7 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- 4 Click the Velocity field button.
- **5** Specify the **u**₀ vector as

c vel*y x

-c_vel*x y

6 Locate the Dispersed Phase Boundary Condition section. From the Dispersed phase boundary condition list, choose No dispersed phase flux.

For the continuous phase, the default boundary condition, No slip, is correct for the remaining boundaries. For the dispersed phase, the default condition, **No dispersed phase flux**, is correct for all boundaries.

7 In the Model Builder window's toolbar, click the Show button and select Advanced Physics Options in the menu.

Weak Constraint I

- I On the Physics toolbar, click Points and choose Weak Constraint.
- 2 Select Point 1 only.
- 3 In the Settings window for Weak Constraint, locate the Weak Constraint section.
- **4** From the **Apply reaction terms on** list, choose **User defined**.
- **5** In the **Constraint expression** text field, type **Fp_tot**.

6 In the **Constraint force expression** text field, type test(Fp_tot).

The constraint ensures that the integral of the pressure is equal to zero.

GENERAL FORM PDE (G)

Specify an appropriate unit for shear rate variable gamma.

- I In the Model Builder window, expand the Component I (compl) node, then click General Form PDE (g).
- 2 In the Settings window for General Form PDE, locate the Units section.
- 3 Find the Dependent variable quantity subsection. From the list, choose None.
- 4 In the Unit text field, type 1/s.
- 5 Find the Source term quantity subsection. In the Unit text field, type 1/s.

General Form PDE 1

- I In the Model Builder window, under Component I (compl)>General Form PDE (g) click General Form PDE I.
- 2 In the Settings window for General Form PDE, locate the Conservative Flux section.
- **3** Specify the Γ vector as



- 4 Locate the Source Term section. In the f text field, type gamma-sqrt(0.5*(4*ux^2+2* (uy+vx)^2+4*vy^2)+eps).
- **5** Locate the **Damping or Mass Coefficient** section. In the d_a text field, type **0**.
- 6 In the Model Builder window, click General Form PDE (g).

Flux/Source 1

- I On the Physics toolbar, click Boundaries and choose Flux/Source.
- 2 In the Settings window for Flux/Source, locate the Boundary Selection section.
- **3** From the Selection list, choose All boundaries.

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Free Triangular.

Size

I In the Settings window for Size, locate the Element Size section.

- 2 From the Calibrate for list, choose Fluid dynamics.
- 3 From the Predefined list, choose Finer.
- 4 Click the **Custom** button.
- **5** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.0012.
- 6 Click Build All.

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- **3** In the **Times** text field, type 0 30 100 1000.

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Study I>Solver Configurations node.
- 3 In the Model Builder window, expand the Solution I (soll) node.
- 4 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll) click Time-Dependent Solver I.
- **5** In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
- **6** Locate the **Time Stepping** section. Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.
- 7 On the Study toolbar, click Compute.

RESULTS

Mixture (mm)

To visualize the volume fraction of the **Dispersed Phase** as a surface plot along with the mixture velocity field as an arrow surface plot, follow the steps given below.

Dispersed Phase Volume Fraction

- I In the Model Builder window, expand the Dispersed Phase (mm) node, then click Dispersed Phase Volume Fraction.
- 2 In the Settings window for Surface, click to expand the Range section.

3 Select the Manual color range check box.

Fix the color range so that the solutions for different time values use the same color legend. The following values cover both cases.

- **4** In the **Minimum** text field, type 0.
- 5 In the Maximum text field, type 0.45.

Dispersed Phase (mm)

In the Model Builder window, under Results right-click Dispersed Phase (mm) and choose Arrow Surface.

Arrow Surface 1

On the Dispersed Phase (mm) toolbar, click Plot.

Dispersed Phase (mm)

- I In the Model Builder window, under Results click Dispersed Phase (mm).
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- **3** From the **Time (s)** list, choose **0**.
- 4 On the Dispersed Phase (mm) toolbar, click Plot.
- 5 From the Time (s) list, choose 30.
- 6 On the Dispersed Phase (mm) toolbar, click Plot.
- 7 From the Time (s) list, choose 100.
- 8 On the Dispersed Phase (mm) toolbar, click Plot.
- 9 From the Time (s) list, choose 1000.
- 10 On the Dispersed Phase (mm) toolbar, click Plot.

This completes Case 1. Now model the case with the particles initially gathered at the top.

DEFINITIONS

Define the particle concentration distribution using the **Step** function.

Step I (step I)

- I On the Home toolbar, click Functions and choose Local>Step.
- 2 In the Settings window for Step, click to expand the Smoothing section.
- **3** In the Size of transition zone text field, type 2*2.

Variables I

I In the Model Builder window, under Component I (compl)>Definitions click Variables I.

2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
phi0_2	step1(y[1/mm]-8)*0.59		Initial concentration

MIXTURE MODEL, LAMINAR FLOW (MM)

On the Physics toolbar, click General Form PDE (g) and choose Mixture Model, Laminar Flow (mm).

In the Model Builder window, under Component I (compl) click Mixture Model, Laminar Flow (mm).

Initial Values 2

- I On the Physics toolbar, click Domains and choose Initial Values.
- 2 In the Settings window for Initial Values, locate the Domain Selection section.
- 3 From the Selection list, choose All domains.
- **4** Locate the **Initial Values** section. In the ϕ_d text field, type phi0_2.

In order to avoid dividing by zero in the region where the particle volume fraction is initially zero, modify the expressions for the slip velocity.

Mixture Properties 1

- In the Model Builder window, under Component I (compl)>Mixture Model, Laminar Flow (mm) click Mixture Properties I.
- 2 In the Settings window for Mixture Properties, locate the Mixture Model section.
- **3** Specify the \mathbf{u}_{slip} vector as

J1/(phid*rho_s*(1-mm.cd)+eps) x J2/(phid*rho_s*(1-mm.cd)+eps) y

COMPONENT I (COMPI)

Create a finer mesh compared to the one used in the previous case.

Mesh 2

On the Mesh toolbar, click Add Mesh.

MESH 2

In the Model Builder window, under Component I (compl)>Meshes right-click Mesh 2 and choose Free Triangular.

Size

- I In the Settings window for Size, locate the Element Size section.
- 2 From the Predefined list, choose Extra fine.
- 3 Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.0006.
- 5 Click Build All.

Next, add a Time Dependent study to compute the particle distribution.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select Preset Studies>Time Dependent.
- 4 Click Add Study in the window toolbar.
- 5 On the Home toolbar, click Add Study to close the Add Study window.

STUDY 2

Step 1: Time Dependent

- I In the Model Builder window, under Study 2 click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the **Times** text field, type range(0,10,100).
- **4** Click to expand the **Mesh selection** section. Locate the **Mesh Selection** section. In the table, enter the following settings:

Geometry	Mesh	
Geometry I	mesh2	

Use manual **Scaling** of the variables to improve the convergence.

Solution 2 (sol2)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Study 2>Solver Configurations node.
- 3 In the Model Builder window, expand the Solution 2 (sol2) node.
- 4 In the Model Builder window, expand the Study 2>Solver Configurations>Solution 2 (sol2)>Dependent Variables I node, then click Velocity field, mixture (compl.u).

- 5 In the Settings window for Field, locate the Scaling section.
- 6 From the Method list, choose Manual.
- 7 In the Scale text field, type 0.1.
- 8 In the Model Builder window, under Study 2>Solver Configurations>Solution 2 (sol2)> Dependent Variables I click Pressure (compl.p).
- 9 In the Settings window for Field, locate the Scaling section.
- **IO** From the **Method** list, choose **Manual**.
- II In the Scale text field, type 100.

The default absolute tolerance for phid is set assuming that phid is nowhere near the packing limit and hence, the tolerance becomes too stringent in this case.

- 12 In the Model Builder window, under Study 2>Solver Configurations>Solution 2 (sol2) click Time-Dependent Solver 1.
- **13** In the **Settings** window for Time-Dependent Solver, locate the **Absolute Tolerance** section.
- 14 In the Variables list, select Volume fraction, dispersed phase (compl.phid).
- **I5** In the **Tolerance** text field, type 1e-4.
- **I6** Locate the **Time Stepping** section. Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

To ensure that the volume fraction of particles has a positive value, do the following:

- 17 In the Model Builder window, expand the Study 2>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver I>Segregated I node.
- 18 Right-click Study 2>Solver Configurations>Solution 2 (sol2)>Time-Dependent Solver I> Segregated I and choose Lower Limit.
- 19 In the Settings window for Lower Limit, locate the Lower Limit section.
- **20** In the Lower limits (field variables) text field, type comp1.phid 0.
- **2** On the **Study** toolbar, click **Compute**.

RESULTS

Mixture (mm) 1

To visualize the volume fraction of the dispersed phase as a surface plot along with the arrow plot of the mixture velocity, follow the steps given below.

Dispersed Phase (mm) I

I In the Model Builder window, under Results click Dispersed Phase (mm) I.

- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Time (s) list, choose 0.

Dispersed Phase Volume Fraction

- I In the Model Builder window, expand the Dispersed Phase (mm) I node, then click Dispersed Phase Volume Fraction.
- 2 In the Settings window for Surface, locate the Range section.
- 3 Select the Manual color range check box.
- **4** In the **Minimum** text field, type 0.
- 5 In the Maximum text field, type 0.67.

Dispersed Phase (mm) I

In the Model Builder window, under Results right-click Dispersed Phase (mm) I and choose Arrow Surface.

Arrow Surface 1

On the Dispersed Phase (mm) I toolbar, click Plot.

Dispersed Phase (mm) 1

- I In the Model Builder window, under Results click Dispersed Phase (mm) I.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- **3** From the **Time (s)** list, choose **10**.
- 4 On the Dispersed Phase (mm) I toolbar, click Plot.
- 5 From the Time (s) list, choose 20.
- 6 On the Dispersed Phase (mm) I toolbar, click Plot.
- 7 From the Time (s) list, choose 100.
- 8 On the Dispersed Phase (mm) I toolbar, click Plot.



Displacement Ventilation

Introduction

The present example investigates the performance of a displacement ventilation system. Given measured values for inlet velocity, inlet temperature, and heat flux, this simulation yields field configurations of air temperature and velocity that are consistent with experimental measurements and analytic global models (Ref. 1).

Model Definition

In general, there are two classes of ventilation: mixing ventilation and displacement ventilation. In displacement ventilation, air enters a room at the floor level and displaces warmer air to achieve the desired temperature. Heating sources in the room can include running electronic devices, or inlet jets of warm air. A potential issue with the displacement ventilation approach is that significant temperature variation and strong stratification may arise.

The model geometry consists of a test chamber with the dimensions 2.5 m by 3.65 m by 3 m. A warm jet enters the chamber from an inlet located at the floor center. Fresh air, at constant temperature and relatively low velocity, is supplied through a wall inlet. Heat exits the chamber through an exhaust located in the center of the ceiling. The walls of the chambers are almost perfectly insulated.

Symmetry reduces the modeling domain to half of the chamber, as shown in Figure 1. The warm jet feeds 0.028 m³/s of air at a temperature of 45 °C into the room. The temperature of the fresh air is 21 °C and has a flow rate of 0.05 m³/s.



Figure 1: The modeling domain is reduced to half the chamber size due to symmetry.

Convection of heat can be either forced or free. Forced convection occurs if

$$\frac{g\alpha\Delta T}{U^2/L} \ll 1 \tag{1}$$

where g is the gravity (m/s^2) , $\alpha(1/K)$ is the coefficient of thermal expansion, T (K) the temperature, U (m/s) the velocity, and L (m) refers to the characteristic length. Equation 1 states that the buoyancy force is small compared to the inertial force. In such a situation, the character of the flow field is described by the Reynolds number, Re = UL/v where v (m²/s) is the kinematic viscosity. Natural convection occurs if Equation 1 is not fulfilled, in which case the flow field character is described by the Grashof number,

$$Gr = \frac{g\alpha\Delta TL^3}{v^2}$$

If the convective forces and buoyant forces are of the same order of magnitude, then $Gr^{1/2}$ can be interpreted as the ratio between the inertial and viscous forces. That is, when the Grashof number is large, the flow becomes turbulent.

To investigate if Equation 1 holds, the air can approximated as an ideal gas in which case $\alpha = 1/T$. Furthermore, $\Delta T \approx 20$ K, $U \approx 1$ m/s, and $L \approx 2$ m. This gives

$$\frac{g\alpha\Delta T}{U^2/L} \approx \frac{9.8 \cdot 20 \cdot 2}{1^2 \cdot 300} = 1.3$$

Hence, it is the Grashof number that determines whether the flow is turbulent or laminar. Using the same approximations as above:

$$Gr \approx \frac{9.8 \cdot 20 \cdot 2^3}{(1.6 \cdot 10^{-5})^2 \cdot 300} = 2 \cdot 10^{10}$$
(2)

Equation 2 clearly indicates that the flow is turbulent.

Modeling Considerations

You model the flow using the $k \cdot \omega$ model. The main reason for using the $k \cdot \omega$ model over the $k \cdot \varepsilon$ model is that former is in general more reliable when it comes to predicting the spreading rate of jets (Ref. 2). The $k \cdot \omega$ model uses wall functions which is quite all right in this case since all walls are almost insulated and there would not be much benefit from using the more expensive low-Re $k \cdot \varepsilon$ model.

As can be seen in Figure 1, the inlets and the outlet have been extended with small domains. This is to avoid having velocity conditions perpendicular to the no-penetration conditions of the walls, which often turns out to be numerically unstable.

The model is solved in two steps. In the first step, the viscosity is set to approximately 25 times higher than the physical viscosity. This factor is lowered to the physical viscosity in a second computational step which uses the result from the first step as initial guess. This step-wise procedure has the benefit that it is often easier to solve a model with lower Reynolds number. The resulting intermediate solutions can also be used to investigate the possible need for mesh refinement before moving on towards the correct Reynolds number.
Figure 2 shows a streamline plot colored by the temperature. As expected, there is a stratification at $z \approx 1$ m with a complicated recirculation pattern above.



Figure 2: Streamlines colored by the temperature illustrating the velocity field.

A more quantitative picture is given in Figure 3 which shows isosurfaces of the temperature field. The stratification is even more clearly visible here. The result compares well with the experimental results in Ref. 1, although the stratification is located a few

decimeter too low.



Figure 3: Isosurfaces of the temperature.

Figure 4 shows a comparison of the computed and measured temperature along the line $(1.25,0,0) \rightarrow (1.25,0,3)$, that is through the center of the jet. While the computational result captures the main trend with decreasing temperature with height, it still over predicts the experimental result with 2 °C at *z*=2.6 m. There are two possible reasons for this. The first is, as mentioned in Ref. 1, that the test chamber is not as well insulated as intended. The other possible explanation is that the buoyancy induced production in the *k* and ω equations (see for example Ref. 3) must be included in order to reproduce the

experimental results more accurately.



Figure 4: Plume temperature.

References

1. D. Mazoni and P. Guitton, "Validation of Displacement Ventilation Simplified Models, "*Proceedings of 'Building Simulation '97'*, the Fifth International IBSPA Conference, vol. I, pp. 233–239, International Building Performance Simulation Association (IBSPA), 1997.

2. D.C. Wilcox, *Turbulence Model for CFD*, 2nd ed., DCW Industries, La Canada, CA, 1998.

3. S. Tieszen, A. Ooi, P. Durbin, and M. Behnia, "Modeling of Natural Convection Heat Transfer," *Proceedings of the Summer Program 1998*, Stanford: Center for Turbulence Research, 1998.

Application Library path: CFD_Module/Non-Isothermal_Flow/ displacement_ventilation From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Non-Isothermal Flow>Turbulent Flow> Turbulent Flow, k-ω.
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
Н	3[m]	3 m	Room height
D	2.5[m]	2.5 m	Room depth
W	3.65[m]	3.65 m	Room width
Hd	0.5[m]	0.5 m	Diffuser inlet height
Ad	1.7[m^2]	1.7 m ²	Diffuser inlet area
As	0.0324[m^2]	0.0324 m ²	Source inlet area
Ao	0.04[m^2]	0.04 m ²	Outlet area

DEFINITIONS

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

Name	Expression	Unit	Description
Ms	0.028[m^3/s]	m³/s	Volume flow rate at source
Md	0.051[m^3/s]	m³/s	Volume flow rate at diffuser
Us	Ms/As	m/s	Source inlet velocity
Ud	Md/Ad	m/s	Diffuser inlet velocity
Tdiff	21[degC]	К	Diffuser air temperature
Tsource	45[degC]	К	Source air temperature
Tout	17[degC]	К	Outside temperature

3 In the table, enter the following settings:

GEOMETRY I

Block I (blkI)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type D.
- 4 In the Depth text field, type W.
- 5 In the **Height** text field, type 2*H.
- 6 Locate the Position section. In the y text field, type -W/2.
- 7 In the z text field, type -H.

Block 2 (blk2)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type 0.05[m].
- 4 In the **Height** text field, type 2*Hd.
- 5 In the **Depth** text field, type Ad/Hd.
- 6 Locate the **Position** section. In the **x** text field, type -0.05[m].
- 7 In the y text field, type -Ad/Hd/2.
- 8 In the z text field, type -Hd.

Union I (uni I)

- I On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 2 In the Settings window for Union, locate the Union section.

- **3** Clear the Keep interior boundaries check box.
- 4 Click in the Graphics window and then press Ctrl+A to select both objects.

Block 3 (blk3)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- **3** In the **Width** text field, type 4[m].
- 4 In the **Depth** text field, type 4[m].
- 5 In the **Height** text field, type 4[m].
- 6 Locate the Position section. In the x text field, type -1[m].
- 7 In the y text field, type -2[m].
- 8 In the z text field, type -4[m].

Difference I (dif I)

- I On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 2 In the Settings window for Difference, locate the Difference section.
- **3** Clear the **Keep interior boundaries** check box.
- 4 Select the object unil only.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the object **blk3** only.

Block 4 (blk4)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- **3** In the **Width** text field, type **3**[m].
- 4 In the **Depth** text field, type 2[m].
- **5** In the **Height** text field, type 5[m].
- 6 Locate the **Position** section. In the **x** text field, type -0.2[m].
- 7 In the y text field, type -2[m].
- 8 In the z text field, type -1[m].

Difference 2 (dif2)

- I On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 2 In the Settings window for Difference, locate the Difference section.
- **3** Clear the Keep interior boundaries check box.

- 4 Select the object difl only.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the object **blk4** only.

Block 5 (blk5)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- **3** In the **Width** text field, type sqrt(As).
- 4 In the **Depth** text field, type sqrt(As)/2.
- 5 In the Height text field, type H/2.
- 6 Locate the **Position** section. In the **x** text field, type D/2-sqrt(As)/2.
- 7 In the z text field, type -0.05[m].

Block 6 (blk6)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- **3** In the **Width** text field, type sqrt(Ao).
- 4 In the **Depth** text field, type sqrt(Ao)/2.
- 5 In the **Height** text field, type 0.45[m].
- 6 Locate the Position section. In the x text field, type D/2-sqrt(Ao)/2.
- **7** In the **y** text field, type 0.
- 8 In the z text field, type H-0.3[m].

Mesh Control Domains 1 (mcd1)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Domains.
- 2 On the object fin, select Domains 2 and 5 only.
- 3 On the Geometry toolbar, click Build All.
- 4 In the Model Builder window, collapse the Geometry I node.
- 5 In the Model Builder window, collapse the Geometry I node.
- 6 Click the Zoom Extents button on the Graphics toolbar.

The model geometry is now complete and should look like Figure 1.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.

- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

TURBULENT FLOW, $K-\omega$ (SPF)

Fluid Properties 1

- I In the Model Builder window, expand the Component I (comp1)>Turbulent Flow, k-ω (spf) node, then click Fluid Properties 1.
- 2 In the Settings window for Fluid Properties, locate the Fluid Properties section.
- 3 From the μ list, choose User defined. In the associated text field, type 5e-4[Pa*s].
- **4** In the Model Builder window, click Turbulent Flow, k-ω (spf).
- 5 In the Settings window for Turbulent Flow, $k-\omega$, locate the Physical Model section.
- 6 Select the Include gravity check box.
- 7 Specify the \mathbf{r}_{ref} vector as

0	x
0	у
H+0.15[m]	z

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 13 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the U_0 text field, type Us.
- **5** Locate the **Turbulence Conditions** section. In the $I_{\rm T}$ text field, type 0.13.
- 6 In the $L_{\rm T}$ text field, type 0.01[m].

The air probably enters through a grid. It is therefore appropriate to set a high inlet intensity and short length scale.

Inlet 2

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- **4** In the $I_{\rm T}$ text field, type 0.05.

- **5** In the $L_{\rm T}$ text field, type 0.01[m].
- 6 Locate the Velocity section. In the U_0 text field, type Ud.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 10 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- 2 Select Boundary 2 only.

HEAT TRANSFER IN FLUIDS (HT)

- I In the Model Builder window, under Component I (compl) click Heat Transfer in Fluids (ht).
- 2 On the Physics toolbar, click Boundaries and choose Symmetry.
- **3** Select Boundary 2 only.

Temperature 1

- I On the Physics toolbar, click Boundaries and choose Temperature.
- 2 Select Boundary 1 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the T_0 text field, type Tdiff.

Temperature 2

- I On the Physics toolbar, click Boundaries and choose Temperature.
- 2 Select Boundary 13 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the T_0 text field, type Tsource.

Outflow I

- I On the Physics toolbar, click Boundaries and choose Outflow.
- 2 Select Boundary 10 only.

Heat Flux 1

- I On the Physics toolbar, click Boundaries and choose Heat Flux.
- 2 In the Settings window for Heat Flux, locate the Heat Flux section.

- 3 Click the Convective heat flux button.
- **4** In the *h* text field, type $0.4[W/(m^2*K)]$.
- **5** In the T_{ext} text field, type Tout.
- 6 Select Boundaries 6–8 and 17 only.

Add an initial pressure profile that is consistent both with the outlet pressure condition and the gravity force.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- 3 From the Element size list, choose Fine.

Size 1

- I Right-click Component I (compl)>Mesh I and choose Edit Physics-Induced Sequence.
- 2 In the Model Builder window, under Component I (compl)>Mesh I click Size I.
- 3 In the Settings window for Size, locate the Geometric Entity Selection section.
- 4 Click Clear Selection.
- **5** Select Boundaries 4, 5, 9, 11, 12, and 14–16 only.
- 6 Locate the **Element Size** section. Click the **Custom** button.
- 7 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 8 In the associated text field, type 0.015.

Free Tetrahedral I

In the Model Builder window, under Component I (compl)>Mesh I right-click Free Tetrahedral I and choose Size.

Size 1

- I In the Settings window for Size, locate the Geometric Entity Selection section.
- 2 From the Geometric entity level list, choose Domain.
- 3 Select Domains 2–5 only.
- 4 Locate the **Element Size** section. Click the **Custom** button.
- **5** Locate the **Element Size Parameters** section. Select the **Maximum element growth rate** check box.
- 6 In the associated text field, type 1.05.

14 | DISPLACEMENT VENTILATION

Boundary Layer Properties 1

- I In the Model Builder window, expand the Component I (compl)>Mesh l>Boundary LayersI node, then click Boundary Layer Properties I.
- 2 In the Settings window for Boundary Layer Properties, locate the Boundary Layer Properties section.
- 3 In the Thickness adjustment factor text field, type 3.
- 4 In the Number of boundary layers text field, type 4.
- 5 In the Model Builder window, click Mesh I.
- 6 In the Settings window for Mesh, click Build All.

RESULTS

Large models may be easier to work with if the plots are only updated on request.

- I In the Settings window for Results, locate the Result Settings section.
- 2 Clear the Automatic update of plots check box.

STUDY I

- I In the Model Builder window, right-click Study I and choose Rename.
- 2 In the Rename Study dialog box, type mu=5e-4[Pa*s] in the New label text field.
- 3 Click OK.
- 4 On the Home toolbar, click Compute.

RESULTS

Velocity (spf)

The Cell Reynolds number can be of help to determine the need for mesh refinement before you lower the viscosity again.

Slice

- I In the Model Builder window, expand the Velocity (spf) node, then click Slice.
- 2 In the Settings window for Slice, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Turbulent Flow, k-ω>Auxiliary variables>spf.cellRe Cell Reynolds number.

3 On the Velocity (spf) toolbar, click Plot.

The Cell Reynolds number is at most approximately 20, which can be considered well resolved. The mesh will probably suffice for the final Reynolds number as well.

```
16
                                            14
      1.5
             0
                                            12
              3
              2
                                            10
              1
                                            8
                                            6
         2
    1
0
                                            4
                                            2
```

Slice: Cell Reynolds number (1)

Cell Reynolds number for $\mu = 5e^{-4}$ [Pa.s].

TURBULENT FLOW, $K - \omega$ (SPF)

On the Physics toolbar, click Heat Transfer in Fluids (ht) and choose Turbulent Flow, $k-\omega$ (spf).

Fluid Properties 1

y _ _ x

- I In the Model Builder window, under Component I (compl)>Turbulent Flow, k-ω (spf) click Fluid Properties I.
- 2 In the Settings window for Fluid Properties, locate the Fluid Properties section.
- **3** From the μ list, choose **From material**.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 In the Select Study tree, select Preset Studies>Stationary.
- 4 Click Add Study in the window toolbar.

5 On the Home toolbar, click Add Study to close the Add Study window.

STUDY 2

Step 1: Stationary

- I In the Model Builder window, under Study 2 click Step I: Stationary.
- **2** In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.
- **3** Locate the Values of Dependent Variables section. From the Settings list, choose User controlled.
- 4 From the Method list, choose Solution.
- 5 From the Study list, choose mu=5e-4[Pa*s], Stationary.
- 6 In the Model Builder window, right-click Study 2 and choose Rename.
- 7 In the Rename Study dialog box, type mu from material in the New label text field.
- 8 Click OK.
- **9** On the **Home** toolbar, click **Compute**.

RESULTS

Temperature (ht) 1 Proceed to reproduce Figure 2.

Surface

- I In the Model Builder window, expand the Temperature (ht) I node, then click Surface.
- 2 In the Settings window for Surface, locate the Data section.
- 3 From the Data set list, choose Exterior Walls I.
- 4 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 5 From the Color list, choose Gray.

Temperature (ht) 1

In the Model Builder window, under Results right-click Temperature (ht) I and choose Streamline.

Streamline I

- I In the **Settings** window for Streamline, locate the **Data** section.
- 2 From the Data set list, choose mu from material/Solution 2 (sol2).
- **3** Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

- 4 In the **Separating distance** text field, type 0.07.
- 5 Locate the Coloring and Style section. From the Line type list, choose Ribbon.

Color Expression 1

- I Right-click Results>Temperature (ht) I>Streamline I and choose Color Expression.
- 2 In the Settings window for Color Expression, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Heat Transfer in Fluids>Temperature>T Temperature.
- 3 Locate the Coloring and Style section. From the Color table list, choose Thermal.
- 4 Click to expand the Range section. Select the Manual color range check box.
- 5 In the Minimum text field, type 293.
- 6 In the Maximum text field, type 300.

Temperature (ht) 1

Execute the following steps to reproduce Figure 3.

3D Plot Group 11

- I On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Data set list, choose mu from material/Solution 2 (sol2).

Isosurface 1

- I Right-click **3D Plot Group II** and choose Isosurface.
- 2 In the Settings window for Isosurface, locate the Expression section.
- **3** In the **Expression** text field, type T.
- 4 From the Unit list, choose degC.
- 5 Locate the Levels section. From the Entry method list, choose Levels.
- 6 In the Levels text field, type 22 23 24 25 26.
- 7 Locate the Coloring and Style section. From the Color table list, choose ThermalLight.
- 8 On the 3D Plot Group 11 toolbar, click Plot.

The following steps will reproduceFigure 4.

Table I

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.

4 Browse to the application's Application Library folder and double-click the file displacement_ventilation_exp.txt.

TABLE

- I Go to the Table window.
- 2 Click the right end of the Display Table I split button in the window toolbar.
- **3** From the menu, choose **Table Graph**.

RESULTS

Table Graph 1

- I In the Model Builder window, under Results>ID Plot Group 12 click Table Graph 1.
- 2 In the Settings window for Table Graph, locate the Data section.
- 3 From the x-axis data list, choose Column 2.
- 4 Locate the Coloring and Style section. From the Line list, choose None.
- 5 From the Marker list, choose Circle.
- 6 From the Positioning list, choose In data points.

Cut Line 3D I

- I On the **Results** toolbar, click **Cut Line 3D**.
- 2 In the Settings window for Cut Line 3D, locate the Data section.
- 3 From the Data set list, choose mu from material/Solution 2 (sol2).
- 4 Locate the Line Data section. In row Point I, set x to D/2.
- 5 In row Point 2, set x to D/2 and z to H.

ID Plot Group 12

- I In the Model Builder window, under Results click ID Plot Group 12.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Cut Line 3D I.

Line Graph I

- I On the ID Plot Group 12 toolbar, click Line Graph.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type z.
- 4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- **5** In the **Expression** text field, type T.

- 6 From the **Unit** list, choose **degC**.
- 7 Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. In the **Width** text field, type **3**.

ID Plot Group 12

- I In the Model Builder window, under Results click ID Plot Group 12.
- 2 In the Settings window for 1D Plot Group, click to expand the Title section.
- **3** From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
- **5** In the associated text field, type T [degC].
- 6 Select the y-axis label check box.
- 7 In the associated text field, type z [m].
- 8 Locate the Axis section. Select the Manual axis limits check box.
- **9** In the **x minimum** text field, type **0**.
- **IO** In the **x maximum** text field, type **46**.
- **II** In the **y minimum** text field, type **0**.
- **12** In the **y maximum** text field, type **3**.
- **I3** On the **ID Plot Group 12** toolbar, click **Plot**.



Transient Elastohydrodynamic Squeeze-Film Interaction

This model is licensed under the COMSOL Software License Agreement 5.2a. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This benchmark model computes the transient pressure distribution and film height in a squeeze-film bearing for lubrication in a nonconformal conjunction of a solid sphere and an elastic wall separated by a lubricant film.

Lubrication between mechanical parts prevents wear and tear due to friction.

Elastohydrodynamic contact between mechanical parts refers to the interaction between a lubricant and elastic bodies. The pressure developed in the lubricant and the mechanical stresses near the contact center are important concerns in elastohydrodynamic interaction. Solving such problems numerically involves modeling the elastohydrodynamic interaction by solving Reynold's equation and solid mechanics as a coupled problem.



Figure 1: This example considers the case of an equivalent rigid sphere and an elastic wall. The equivalent model does not require modeling the rigid sphere. Because the model is symmetric, it is sufficient to model one quarter of the above geometry.

This example solves the benchmark case of hydrodynamic interaction between a solid sphere and a wall separated by a lubricant film, and extends the benchmark case to include elastic deformation and stresses on the contacting wall. The model setup involves a solid sphere being pushed by an external force towards a solid plane wall. The lubricant layer gets squeezed by the approaching ball, which leads to a rise in the pressure in the lubricant. The calculated maximum lubricant pressure and the change in film height with time are compared with analytical solutions.

Figure 1 shows the scenario of an elastic sphere pushed by an external force towards an elastic wall covered by a thin lubricant layer. This model computes the time-dependent pressure developed in the lubricant and the position of the sphere relative to the elastic wall. The scenario in Figure 1 is reduced to an equivalent model with a rigid sphere and an elastic wall with an equivalent Young's modulus given by (Ref. 1)

$$E = \frac{2}{\frac{1 - v_1^2}{E_1} + \frac{1 - v_2^2}{E_2}}$$

where E_1 and E_2 are the Young's moduli and v1 and v2 are the Poisson's ratios of the two elastic bodies. Because of symmetry, the model uses only one quarter of the geometry shown in Figure 1.

For non-slip boundary conditions at the wall and the base, Reynolds equation takes the form

$$\begin{split} \frac{\partial}{\partial t}(\rho h) + \nabla_t \cdot (h\rho \mathbf{v}_{av}) - \rho(\mathbf{v}_w \cdot \nabla_t h_w + \mathbf{v}_b \cdot \nabla_t h_b) &= 0\\ \mathbf{v}_{av} &= \frac{1}{2} (\mathbf{I} - \mathbf{n}_r \mathbf{n}_r^T) (\mathbf{v}_w + \mathbf{v}_b) - \frac{h^2}{12\mu} \nabla_t p_f \end{split}$$

where *h* is the film thickness, ρ is the fluid density, μ is the viscosity, and *p_f*—the dependent variable in the Thin-Film Flow, Shell user interface—is the pressure developed as a result of the flow. For further details, see the theory section for the Thin-Film Flow interfaces in the *CFD Module User's Guide*.



Figure 2: 2D representation of the distance of the sphere from the solid wall.

Figure 2 shows a 2D representation of the sphere at some distance from the solid wall with an exaggerated view of the film height between the sphere and the solid wall. The sphere has a radius a and the center of the sphere is initially located at a distance a + b from the surface of the solid. For a thin-film approximation, the film thickness h is given by the expression

$$h = b(t) + \frac{r^2}{2a}$$

where $r = \sqrt{x^2 + y^2}$ is the horizontal radial distance measured from the center of the sphere.

The external force, F, is counterbalanced by the pressure in the lubricant. This is imposed as a constraint:

$$\int_{\partial\Omega} p_f dS - F = 0 \tag{1}$$

The hydrodynamic pressure exerted by the lubricant causes elastic deformation of the two surfaces containing the lubricant. In this model, the surface of interest is the elastic wall. The hydrodynamic pressure is therefore used as a mechanical load on the elastic wall to calculate its deformation using a Solid Mechanics interface. However, the elastic deformation in this example is negligibly small in comparison with the change in film height due to the squeezing motion of the sphere against the elastic wall. Therefore, the results for pressure developed in the lubricant and the change in film height can be compared with the solution to the benchmark hydrodynamic problem of a solid sphere

pushed against a wall with a lubricant layer between the sphere and the wall. Because the problem is axisymmetric, the Reynolds equation can be greatly simplified and written as

$$\frac{1}{r\frac{\partial}{\partial r}}\left(rh^{3}\frac{\partial p_{f}}{\partial r}\right) = 12\mu\frac{\partial h}{\partial t}$$
(2)

Restricting the lubrication calculations to the range 0 < r < a, Equation 2 is solved with boundary conditions $\partial p_f / \partial r = 0$ at r = 0 and $p_f = 0$ at r = a to give the pressure developed in the film as (see Ref. 2)

$$p_{f}(r) = -6\mu \frac{\partial b}{\partial t} \left(\frac{2a^{3}}{(2ab+r^{2})^{2}} - \frac{2a^{3}}{(2ab+a^{2})^{2}} \right)$$
(3)

Given this expression for the pressure distribution, the hydrodynamic force can be calculated using Equation 1 and is given by the following expression (see Ref. 2)

$$F = \frac{6\pi\mu \dot{b}a^2}{b} \left(1 - \frac{2ab}{(a^2 + 2ab)} - \frac{2a^3b}{(a^2 + 2ab)^2} \right)$$
(4)

Equation 4 is an ordinary differential equation that can be solved for the analytical change in the film height, b. The values of b thus obtained can then be substituted in Equation 3 to solve for the analytical pressure developed in the lubricant film.

Results and Discussion

Figure 3 shows that the pressure distribution in the lubricant is concentrated near the center of the wall with the maximum pressure due to the squeezing action developing at the center. Figure 4 shows the von Mises stress distribution on the elastic wall resulting from the fluid load due to increased pressure in the lubricant.

Figure 5 and Figure 6 show the results for the maximum film pressure and the change in film height with time, respectively, together with the corresponding analytical solutions obtained by solving Equation 3 and Equation 4. As expected, as the gap between the sphere and the wall decreases, the film pressure increases. The figures also show a very good match between the numerical and analytical solutions.



Figure 3: Pressure distribution in the lubricant.



Figure 4: von Mises stress plot on the boundaries of the elastic solid.



Figure 5: Comparison between calculated and analytical values of maximum pressure.



Figure 6: Comparison between the calculated and analytical values of change in film height.

Notes About the COMSOL Implementation

To resolve the high pressure gradients at the center of the wall, the mesh is customized to be fine in this region. This is important for getting results with higher accuracy. As the wall deforms, the film height changes by an additional amount equal to the wall displacement along the surface normal. This is accounted for in the settings of the Thin-Film Flow, Shell user interface by choosing the displacement field as an additional wall displacement. However, in this example this change in film height is negligibly small in comparison to the change in film height due to the external force.

References

1. A.Z. Szeri, *Fluid Film Lubrication: Theory and Design*, Cambridge University Press, 1998.

2. L.G. Leal, Advanced Transport Phenomena: Fluid Mechanics and Convective Transport Processes, Cambridge University Press, 2007.

Application Library path: CFD_Module/Thin-Film_Flow/ elastohydrodynamic_interaction

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Thin-Film Flow>Thin-Film Flow, Shell (tffs).
- 3 Click Add.
- 4 In the Select Physics tree, select Structural Mechanics>Solid Mechanics (solid).
- 5 Click Add.
- 6 Click Study.

- 7 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Time Dependent.
- 8 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
a	0.02[m]	0.02 m	Sphere radius
extent	а	0.02 m	Extent of lubricated area
Force	1.5[N]	1.5 N	Applied force
b0	a/10	0.002 m	Initial film height
visc_mat2	0.8[Pa*s]	0.8 Pa·s	Lubricant viscosity
density_mat2	860[kg/m^3]	860 kg/m³	Lubricant density
timescale	6*pi*visc_mat2* a^2/Force	0.004021 s	Timescale
nu_steel	0.28	0.28	Poisson's ratio
E_steel	205e9[Pa]	2.05E11 Pa	Young's modulus
dens_steel	7850[kg/m^3]	7850 kg/m³	Density
E_eqv	E_steel/ (1-nu_steel^2)	2.224E11 Pa	Equivalent Young's modulus

GEOMETRY I

Work Plane I (wp1)

- I On the Geometry toolbar, click Work Plane.
- 2 In the Model Builder window, right-click Work Plane I (wpl) and choose Show Work Plane.

Bézier Polygon I (b1)

- I On the Work Plane toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.

- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the Control points subsection. In row 2, set xw to extent.
- 5 Right-click Bézier Polygon I (b1) and choose Build Selected.

Work Plane I (wp1)

In the Model Builder window, under Component I (compl)>Geometry I click Work Plane I (wpl).

Revolve I (rev1)

- I On the Geometry toolbar, click Revolve.
- 2 In the Settings window for Revolve, locate the Revolution Axis section.
- **3** From the **Axis type** list, choose **3D**.
- **4** Find the **Direction of revolution axis** subsection. In the **y** text field, type **0**.
- **5** In the **z** text field, type **1**.
- 6 Locate the **Revolution Angles** section. Click the **Angles** button.
- 7 In the End angle text field, type 90.
- 8 Right-click Revolve I (revI) and choose Build Selected.
- 9 Click the Zoom Extents button on the Graphics toolbar.

Block I (blkI)

- I On the Geometry toolbar, click Block.
- 2 In the Settings window for Block, locate the Size and Shape section.
- 3 In the Width text field, type 3*a.
- 4 In the **Depth** text field, type 3*a.
- 5 In the Height text field, type 6*a.
- 6 Locate the **Position** section. In the **z** text field, type -6*a.
- 7 Click Build All Objects.

DEFINITIONS

View I

Modify the view settings.

Use the mouse to rotate the image so that you can see the lubricant boundary.

I In the Model Builder window, expand the Component I (compl)>Definitions node, then click View I.

- 2 In the Settings window for View, locate the View section.
- **3** Select the **Lock camera** check box.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Explicit I

- I On the Definitions toolbar, click Explicit.
- 2 In the Settings window for Explicit, locate the Input Entities section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 Select Boundary 4 only.
- 5 Right-click Explicit I and choose Rename.
- 6 In the Rename Explicit dialog box, type Lubricant in the New label text field.
- 7 Click OK.

Integration 1 (intop1)

- I On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the Settings window for Integration, locate the Source Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 From the Selection list, choose Lubricant.

Variables 1

- I On the Definitions toolbar, click Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** In the table, enter the following settings:

Name	Expression	Unit	Description
forcetot	4*intop1(pfilm)	N	Net force in lubricant
r	sqrt(x^2+y^2)	m	Radial distance

MATERIALS

In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.

Material I (mat1)

I In the Settings window for Material, locate the Material Contents section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Young's modulus	E	E_eqv	Pa	Basic
Poisson's ratio	nu	nu_steel	1	Basic
Density	rho	dens_steel	kg/m³	Basic

Material 2 (mat2)

- I Right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 From the Selection list, choose Lubricant.
- 5 Locate the Material Contents section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Dynamic viscosity	mu	visc_mat2	Pa·s	Basic
Density	rho	density_mat2	kg/m³	Basic

THIN-FILM FLOW, SHELL (TFFS)

- I In the Model Builder window, under Component I (compl) click Thin-Film Flow, Shell (tffs).
- 2 In the Settings window for Thin-Film Flow, Shell, locate the Boundary Selection section.
- 3 From the Selection list, choose Lubricant.

Fluid-Film Properties 1

- I In the Model Builder window, under Component I (compl)>Thin-Film Flow, Shell (tffs) click Fluid-Film Properties I.
- 2 In the Settings window for Fluid-Film Properties, locate the Wall Properties section.
- **3** In the h_{w1} text field, type b+r^2/(2*a).
- **4** From the \mathbf{u}_w list, choose **Displacement field (solid)**.

Symmetry I

- I On the Physics toolbar, click Edges and choose Symmetry.
- 2 Select Edges 4 and 5 only.
- **3** In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

I On the Physics toolbar, click Global and choose Global Equations.

2 In the Settings window for Global Equations, locate the Global Equations section.

Name	f(u,ut,utt,t) (l)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
b	Force-for cetot	b0	b0/ timescale	Change in film height

3 In the table, enter the following settings:

4 Locate the Units section. Find the Dependent variable quantity subsection. From the list, choose Length (m).

5 Find the Source term quantity subsection. From the list, choose Force load (N).

Global Equations 2

I On the Physics toolbar, click Global and choose Global Equations.

2 In the Settings window for Global Equations, locate the Global Equations section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (I)	Initial value (u_t0) (1/s)	Description
k	<pre>timescale* kt+k/(1-2* a*k/ (extent^2+ 2*a* k)-extent^ 2*(2*a*k)/ (extent^2+ 2*a*k)/2)</pre>	b0	b0/ timescale	Analytical change in film height

- 4 Locate the Units section. Find the Dependent variable quantity subsection. From the list, choose Length (m).
- 5 Find the Source term quantity subsection. From the list, choose Length (m).

DEFINITIONS

Variables I

- I In the Model Builder window, under Component I (compl)>Definitions click Variables I.
- 2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
analytical_p	-6*visc_mat2*kt*(2* a^3/(2*a*k+r^2)^2-2* a^3/(2*a*k+ extent^2)^2)	Pa	Analytical pressure

SOLID MECHANICS (SOLID)

On the Physics toolbar, click Thin-Film Flow, Shell (tffs) and choose Solid Mechanics (solid).

In the Model Builder window, under Component I (compl) click Solid Mechanics (solid).

Boundary Load I

- I On the Physics toolbar, click Boundaries and choose Boundary Load.
- 2 In the Settings window for Boundary Load, locate the Boundary Selection section.
- 3 From the Selection list, choose Lubricant.
- 4 Locate the Force section. From the F_A list, choose Fluid load on wall (tffs/ffpl).

Fixed Constraint I

- I On the Physics toolbar, click Boundaries and choose Fixed Constraint.
- 2 Select Boundary 3 only.

Symmetry I

The symmetry boundary condition applied in the next step requires either the Structural Mechanics module or the MEMS module. An alternative is to use prescribed displacement boundary conditions to constrain displacements normal to the symmetry boundaries.

I On the Physics toolbar, click Boundaries and choose Symmetry.

2 Select Boundaries 1 and 2 only.

MESH I

In the Model Builder window, under Component I (comp1) right-click Mesh I and choose More Operations>Free Triangular.

Free Triangular 1

I In the Settings window for Free Triangular, locate the Boundary Selection section.

2 From the Selection list, choose Lubricant.

Size 1

I Right-click Component I (comp1)>Mesh I>Free Triangular I and choose Size.

- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Point**.
- **4** Select Point 2 only.
- 5 Locate the Element Size section. From the Predefined list, choose Extremely fine.
- 6 Click the **Custom** button.
- 7 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 8 In the associated text field, type 1.92e-4.
- 9 Select the Maximum element growth rate check box.
- **IO** In the associated text field, type 1.15.

II In the Model Builder window, right-click Mesh I and choose Free Tetrahedral.

Free Tetrahedral I Click Build All.

STUDY I

Step 1: Time Dependent

- I In the Settings window for Time Dependent, locate the Study Settings section.
- 2 Click Range.
- 3 In the Range dialog box, type 2e-4 in the Step text field.
- 4 In the Stop text field, type 6e-3.
- 5 Click Replace.
- 6 In the Settings window for Time Dependent, locate the Study Settings section.
- 7 Select the **Relative tolerance** check box.
- 8 In the associated text field, type 0.0001.

Solution I (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **3** In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.
- 4 Locate the Absolute Tolerance section. In the Tolerance text field, type 0.00001.

The fully coupled solver performs better for this model and is therefore enabled in the next step.

- 5 Right-click Study I>Solver Configurations>Solution I (sol1)>Time-Dependent Solver I and choose Fully Coupled.
- 6 On the Study toolbar, click Compute.

RESULTS

Fluid Pressure (tffs)

The first default plot group shows a surface plot of the fluid pressure for the final time step (figure pressure).

Stress (solid)

The second default plot group shows a surface plot of the von Mises stress and a deformation plot (exaggerated) of the elastic wall displacement. To reproduce Figure 4 do as follows.

- I In the Model Builder window, expand the Stress (solid) node.
- 2 In the Model Builder window, expand the Results>Stress (solid)>Surface I node.
- 3 Right-click **Deformation** and choose **Disable**.

To reproduce Figure 5 and Figure 6 do as follows.

ID Plot Group 3

On the Home toolbar, click Add Plot Group and choose ID Plot Group.

Point Graph 1

On the ID Plot Group 3 toolbar, click Point Graph.

ID Plot Group 3

- I Select Point 2 only.
- 2 In the Settings window for Point Graph, click to expand the Legends section.
- 3 Select the Show legends check box.
- 4 From the Legends list, choose Manual.
- **5** In the table, enter the following settings:

Legends

Calculated maximum pressure

- 6 On the ID Plot Group 3 toolbar, click Plot.
- 7 Right-click Point Graph I and choose Rename.
- 8 In the Rename Point Graph dialog box, type Calculated in the New label text field.

9 Click OK.

Point Graph 2

On the ID Plot Group 3 toolbar, click Point Graph.

ID Plot Group 3

- I Select Point 2 only.
- 2 In the Settings window for Point Graph, locate the y-Axis Data section.
- 3 In the **Expression** text field, type analytical_p.
- 4 Locate the Legends section. Select the Show legends check box.
- 5 From the Legends list, choose Manual.
- 6 In the table, enter the following settings:

Legends

Analytical maximum pressure

- 7 Click to expand the Coloring and style section. Locate the Coloring and Style section. Find the Line style subsection. From the Line list, choose None.
- 8 Find the Line markers subsection. From the Marker list, choose Asterisk.
- **9** In the **Number** text field, type **20**.
- IO On the ID Plot Group 3 toolbar, click Plot.
- II From the Color list, choose Black.
- 12 Right-click Point Graph 2 and choose Rename.
- **I3** In the **Rename Point Graph** dialog box, type Analytical in the **New label** text field.

I4 Click OK.

- IS In the Settings window for 1D Plot Group, click to expand the Title section.
- **I6** From the **Title type** list, choose **Manual**.
- 17 In the Title text area, type Maximum pressure (Pa).
- **18** Click to expand the **Legend** section. From the **Position** list, choose **Upper left**.
- 19 Right-click 1D Plot Group 3 and choose Rename.
- **20** In the **Rename ID Plot Group** dialog box, type Maximum pressure in the **New label** text field.
- 2I Click OK.

ID Plot Group 4

On the Home toolbar, click Add Plot Group and choose ID Plot Group.

Global I

On the ID Plot Group 4 toolbar, click Global.

ID Plot Group 4

- I In the Settings window for Global, type Calculated in the Label text field.
- 2 Click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component I>Thin-Film Flow, Shell>b Change in film height.
- 3 On the ID Plot Group 4 toolbar, click Plot.
- 4 Right-click Calculated and choose Duplicate.
- 5 In the Settings window for Global, type Analytical in the Label text field.
- 6 Click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component I>Thin-Film Flow, Shell>k Analytical change in film height.
- 7 Click to expand the Coloring and style section. Locate the Coloring and Style section. Find the Line style subsection. From the Line list, choose None.
- 8 Find the Line markers subsection. From the Marker list, choose Asterisk.
- 9 From the Color list, choose Black.
- **IO** In the **Number** text field, type **20**.
- II On the ID Plot Group 4 toolbar, click Plot.
- 12 In the Model Builder window, click ID Plot Group 4.
- **I3** In the **Settings** window for 1D Plot Group, locate the **Title** section.
- **I4** From the **Title type** list, choose **Manual**.
- IS In the Title text area, type Change in film height (m).
- 16 Right-click 1D Plot Group 4 and choose Rename.
- 17 In the Rename ID Plot Group dialog box, type Change in film height in the New label text field.
- I8 Click OK.



Fluid Damper

Introduction

Fluid dampers are used in military devices for shock isolation and in civil structures for suppressing earthquake-induced shaking and wind-induced vibrations, among many other applications. Fluid dampers work by dissipating the mechanical energy into heat (Ref. 1). This example shows the phenomenon of viscous heating and consequent temperature increase in a fluid damper. Viscous heating is also important in microflow devices, where a small cross-sectional area and large length of the device can generate significant heating and affect the fluid flow consequently (Ref. 2).

Model Definition

The structural elements of a fluid damper are relatively few. Figure 1 depicts a schematic of the fluid damper modeled herein with its main components: damper cylinder housing, piston rod, piston head, and viscous fluid in the chamber. There is a small annular space between the piston head and the inside wall of the cylinder housing. This acts as an effective channel for the fluid. As the piston head moves back and forth inside the damper cylinder, fluid is forced to pass through the annular channel with large shear rate, which leads to significant heat generation. The heat is transferred in both the axial and radial directions. In the radial direction, the heat is conducted through the cylinder house wall and convected to the air outside the damper, which is modeled using the Newton's convective cooling law.



Figure 1: A sketch of a typical fluid damper with its major components

You make use of the axially symmetric nature of the fluid damper and model it in a 2D-axisymmetric geometry as shown in Figure 2. The geometric dimensions and other parameters of the damper are taken according to Ref. 1 to represent the smaller, 15 kip damper experimentally studied therein. Thus, the piston head has a diameter of 8.37 cm, the piston rod diameter is 2.83 cm, and the gap thickness is about 1/100 of the piston
head diameter. The damper has the maximum stroke U_0 of 0.1524 m. The damper solid parts are made of steel, and the damping fluid is silicone oil.



Figure 2: Geometry and mesh. The domains (from left to right) represent: piston rod, piston head and damping fluid space, the damper outer wall.

FLUID FLOW

The fluid flow in the fluid damper is described by the weakly compressible Navier-Stokes equations, solving for the velocity field $\mathbf{u} = (u, w)$ and the pressure p:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nabla \cdot \left(\mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right)$$
$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

The density is assumed independent of the temperature, while the temperature dependence of the fluid viscosity is taken into account as:

$$\mu = \mu_0 - \alpha (T - T_0) \tag{1}$$

The reference material properties of silicone oil are used.

No Slip wall boundary conditions are applied for both ends of the damper cylinder and on the inner wall of the damper cylinder house. Moving/sliding wall with the given velocity is applied on the boundaries of the piston head and on the piston rod.

CONJUGATE HEAT TRANSFER

The conjugate heat transfer is solved both in the fluid domain and the damper cylinder house wall: heat transfer by convection and conduction in the fluid domain, heat transfer by conduction only in the solid domain, and the temperature field is continuous between the fluid and solid domains. In the fluid domain, the viscous heating is activated and pressure work can be included when the slight compressibility of the damper fluid needs to be considered:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = -\alpha_p T \frac{\partial p}{\partial t} + \mu \left[\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}} - \frac{2}{3} (\nabla \cdot \mathbf{u}) \mathbf{I} \right] : \nabla \mathbf{u} + Q$$

where the first term and second terms on the right-hand side represent the heat source from pressure work and viscous dissipation, respectively. Hence, the problem is a fully coupled fluid-thermal interaction problem.

In the solid domain of the cylinder house wall, this equation reduces to conductive heat transfer equation without any heating source.

The heat flux boundary condition based on the Newton's cooling law is applied on the outside boundaries of the cylinder house wall. The temperature field is continuous between the fluid and solid domains. The ends of the damper connected to the structures outside are kept at constant temperature.

The piston head movement is provided as harmonic oscillations with given amplitude and frequency, $z = a_0 \sin(2\pi f)$. The motion is modeled using the arbitrary Lagrangian-Eulerian (ALE) deformed mesh. The ALE method handles the dynamics of the deforming geometry and the moving boundaries with a moving grid. The Navier-Stokes equations for fluid flow and heat equations for temperature variation are formulated in these moving coordinates.

Results and Discussion

The modeled loading has the amplitude of 0.127 m, and the excitation frequency is 0.4 Hz. This represents the long-stroke loading experiment performed in Ref. 1. The loading time period is 40 s.

Note that the simulation results for the temperature are presented in degrees Fahrenheit for the sake of easier comparison with the experimental measurements.

Figure 3 gives the temperature field in the damper at the end the loading. It also shows a typical streamline configuration for the flow induced in the damping fluid.



Figure 3: Temperature field in the damper at the end of simulation.

Figure 4 shows the temperature of the inner wall of the damper at the end-of-stroke position $z = U_0$. This corresponds to the internal probe position under experiments performed in Ref. 1. The simulation results show very good agreement with the experimental measurements (see Fig. 9 in Ref. 1).



Figure 4: Temperature at the probe position.

Figure 5 shows the temperature variation along the inner wall of the damper after 10 s and 40 s of loading. It clearly shows that the temperature at the probe position does not represent the maximum temperature within the damper. This supports the conclusion drawn in Ref. 1, where the choice of the probe positioning was limited by the construction of the outer shell of the damper. Figure 5 also shows that the temperature near the center of the damper increases by more than 100 degrees already after few loading cycles.



Figure 5: Temperature of the damper inner wall. The probe position corresponds to $z/U_0 = 1$.

Notes About the COMSOL Implementation

You decompose the computational domain into several parts and mesh the domains with mapped meshes to resolve the very thin annular space. For the moving mesh you prescribe the displacement of the mesh in each domain so that their alignment remains unchanged with a zero displacement at the top and the bottom of the damper cylinder housing connecting to the high-performance seal, and the displacement equal to that of the piston head is used for the domain lined up with the piston head. This is achieved by specifying the mesh displacement field as a linear function of the deformed mesh frame coordinate and the reference (material) frame coordinate.

The steel material needed for the damper solid parts is available in the built-in material library. You create a user-defined material for the silicone oil. Such damping fluids are typically characterized by the density, kinematic viscosity at the temperature 25° C, and so-called *viscosity temperature coefficient*, VTC = 1–(viscosity at 98.9° C)/(viscosity at 37.8° C). Using this parameters, you create the linear correlation for the dynamic viscosity given by Equation 1.

References

1. C.J. Black and N. Makris, "Viscous Heating of Fluid Dampers Under Small and Large Amplitude Motions: Experimental Studies and Parametric Modeling," *J. Eng. Mech.*, vol. 133, pp. 566–577, 2007.

2. G.L. Morini, "Viscous Heating in Liquid Flows in Micro-Channels," *Int. J. Heat Mass Transfer*, vol. 48, pp. 3637–3647, 2005.

Application Library path: CFD_Module/Non-Isothermal_Flow/fluid_damper

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Heat Transfer>Conjugate Heat Transfer>Laminar Flow.
- 3 Click Add.
- 4 In the Select Physics tree, select Mathematics>Deformed Mesh>Moving Mesh (ale).
- 5 Click Add.
- 6 Click Study.
- 7 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Time Dependent.
- 8 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- 3 Click Load from File.

4 Browse to the application's Application Libraries folder and double-click the file fluid_damper_parameters.txt.

DEFINITIONS

Variables 1

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.
- 3 Click Load from File.
- **4** Browse to the application's Application Libraries folder and double-click the file fluid_damper_variables.txt.

GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type Dr/2.
- 4 In the **Height** text field, type 2*Ld.
- 5 Locate the Position section. In the z text field, type -Ld.

Rectangle 2 (r2)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type Dp/2.
- 4 In the **Height** text field, type 2*Ld.
- **5** Locate the **Position** section. In the **z** text field, type -Ld.

Rectangle 3 (r3)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type Dd/2-Hw.
- 4 In the **Height** text field, type 2*Ld.
- **5** Locate the **Position** section. In the **z** text field, type -Ld.

Rectangle 4 (r4)

I On the Geometry toolbar, click Primitives and choose Rectangle.

- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type Dd/2.
- 4 In the **Height** text field, type 2*Ld.
- 5 Locate the **Position** section. In the **z** text field, type -Ld.

Rectangle 5 (r5)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type Dd/2.
- 4 In the **Height** text field, type 2*Lp.
- **5** Locate the **Position** section. In the **z** text field, type -Lp.
- 6 On the Geometry toolbar, click Build All.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

The model geometry is now complete.

LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 Select Domains 4 and 6–9 only.

HEAT TRANSFER (HT)

On the Physics toolbar, click Laminar Flow (spf) and choose Heat Transfer (ht).

Fluid I

- I In the Model Builder window, under Component I (compl)>Heat Transfer (ht) click Fluid I.
- 2 Select Domains 4 and 6–9 only.
- 3 In the Settings window for Fluid, locate the Thermodynamics, Fluid section.
- **4** From the γ list, choose **User defined**.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Steel AISI 4340.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

MATERIALS

In the following steps, you create a new material for the damping fluid, Silicone Oil.

Material 2 (mat2)

- I In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.
- 2 In the Settings window for Material, type Silicon Oil in the Label text field.
- **3** Select Domains 4 and 6–9 only.

Silicon Oil (mat2)

- I In the Model Builder window, expand the Component I (compl)>Materials>Silicon Oil (mat2) node, then click Basic (def).
- **2** In the **Settings** window for Property Group, locate the **Output Properties and Model Inputs** section.
- 3 Find the Quantities subsection. In the tree, select Model Inputs>Temperature.
- 4 Click Add.
- **5** Locate the **Local Properties** section. In the **Local properties** table, enter the following settings:

Property	Expression	Unit
nu_25C	0.0125[m^2/s]	m²/s
VTC	0.6[1]	

6 In the Model Builder window, click Silicon Oil (mat2).

7 In the Settings window for Material, locate the Material Contents section.

8 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	22.5	W/(m·K)	Basic
Density	rho	950	kg/m³	Basic

Property	Name	Value	Unit	Property group
Heat capacity at constant pressure	Ср	2e3	J/(kg·K)	Basic
Dynamic viscosity	mu	nu_25C* rho* (1-VTC* (T-311[K])/ (61[K]))/ (1+VTC* 0.2107)	Pa [.] s	Basic

MOVING MESH (ALE)

On the Physics toolbar, click Heat Transfer (ht) and choose Moving Mesh (ale).

- I In the Model Builder window, under Component I (compl) click Moving Mesh (ale).
- 2 In the Settings window for Moving Mesh, locate the Frame Settings section.
- 3 From the Geometry shape order list, choose I.

Prescribed Deformation I

- I On the Physics toolbar, click Domains and choose Prescribed Deformation.
- **2** Select Domains 2, 5, 8, and 11 only.
- **3** In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.
- **4** In the d_z text-field array, type zp on the 2nd row.

Prescribed Deformation 2

- I On the Physics toolbar, click Domains and choose Prescribed Deformation.
- 2 Select Domains 1, 4, 7, and 10 only.
- **3** In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.
- **4** In the d_z text-field array, type zlin1 on the 2nd row.

Prescribed Deformation 3

- I On the Physics toolbar, click Domains and choose Prescribed Deformation.
- 2 Select Domains 3, 6, 9, and 12 only.
- **3** In the **Settings** window for Prescribed Deformation, locate the **Prescribed Mesh Displacement** section.

4 In the d_z text-field array, type zlin2 on the 2nd row.

LAMINAR FLOW (SPF)

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Wall 2

- I On the Physics toolbar, click Boundaries and choose Wall.
- 2 Select Boundaries 11 and 13 only.
- 3 In the Settings window for Wall, locate the Boundary Condition section.
- 4 From the Boundary condition list, choose Moving wall.
- **5** Specify the $\mathbf{u}_{\mathbf{w}}$ vector as

0	r
d(zp,t)	z

Wall 3

- I On the Physics toolbar, click Boundaries and choose Wall.
- 2 Select Boundaries 8, 12, and 17 only.
- 3 In the Settings window for Wall, locate the Boundary Condition section.
- 4 From the Boundary condition list, choose Sliding wall.
- **5** In the $U_{\rm w}$ text field, type d(zp,t).

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Laminar Flow (spf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *p* text field, type p0.

HEAT TRANSFER (HT)

On the Physics toolbar, click Laminar Flow (spf) and choose Heat Transfer (ht).

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Heat Transfer (ht) click Initial Values I.
- 2 In the Settings window for Initial Values, type T0 in the T text field.
- 3 In the Model Builder window, click Heat Transfer (ht).

Temperature 1

- I On the Physics toolbar, click Boundaries and choose Temperature.
- 2 Select Boundaries 2, 7, 9, 14, 16, 21, 23, and 28 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the T_0 text field, type T0.

Heat Flux 1

- I On the Physics toolbar, click Boundaries and choose Heat Flux.
- 2 Select Boundaries 29–31 only.
- 3 In the Settings window for Heat Flux, locate the Heat Flux section.
- 4 Click the **Convective heat flux** button.
- **5** In the T_{ext} text field, type T0.
- **6** In the *h* text field, type hwall.

LAMINAR FLOW (SPF)

Because the damper is a closed container, you need to pin-point the pressure level within. To achieve that, use the point constraint as follows.

I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Pressure Point Constraint I

- I On the Physics toolbar, click Points and choose Pressure Point Constraint.
- 2 Select Point 12 only.
- **3** In the **Settings** window for Pressure Point Constraint, locate the **Pressure Constraint** section.
- **4** In the p_0 text field, type p0.

MULTIPHYSICS

- I In the Model Builder window, under Component I (compl)>Multiphysics click Non-Isothermal Flow I (nitfl).
- 2 In the Settings window for Non-Isothermal Flow, locate the Flow Heating section.
- 3 Select the Include work done by pressure changes check box.
- 4 Select the Include viscous dissipation check box.

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Mapped I

In the Model Builder window, under Component I (compl)>Mesh I right-click Mapped I and choose Distribution.

Distribution I

- I Select Boundaries 23, 25, 27, and 28 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 4.
- 5 In the Element ratio text field, type 4.
- 6 From the Distribution method list, choose Geometric sequence.
- 7 Select the **Reverse direction** check box.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 2

- I Select Boundaries 1, 5, 8, 12, 15, 19, 22, 26, 29, and 31 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 32.
- 5 In the Element ratio text field, type 8.
- **6** From the **Distribution method** list, choose **Geometric sequence**.
- 7 Select the Symmetric distribution check box.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 3

- I Select Boundaries 9, 11, 13, and 14 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 30.
- 5 In the Element ratio text field, type 10.
- 6 From the Distribution method list, choose Geometric sequence.
- 7 Select the Symmetric distribution check box.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 4

- I Select Boundaries 16, 18, 20, and 21 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** In the Number of elements text field, type 8.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 5

- I Select Boundaries 3, 10, 17, 24, and 30 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- 3 In the Number of elements text field, type 32.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 6

- I Select Boundaries 2, 4, 6, and 7 only.
- 2 In the Model Builder window, click Mesh I.
- 3 In the Settings window for Mesh, click Build All.

The mesh is now complete. It should look similar to that shown in Figure 2.

STUDY I

Step 1: Time Dependent

Start the simulation when the piston is in the lowest position consistent with the steady flow initial conditions.

This will not give constant temperature at t = 0, but the overall effect is small.

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the **Times** text field, type -0.25/f range(0,tstep,tmax).
- 4 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Study I>Solver Configurations node.
- 3 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **4** In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.

To control the time-step manually, use the Generalized alpha time stepping method.

- 5 Locate the Time Stepping section. From the Method list, choose Generalized alpha.
- 6 From the Steps taken by solver list, choose Manual.
- 7 In the **Time step** text field, type tstep/20.
- 8 In the Amplification for high frequency text field, type 0.5.

Before computing the solution, set up some plots, including the one to display in the **Graphics** window while solving.

RESULTS

Cut Point 2D 1 On the **Results** toolbar, click **Cut Point 2D**.

Data Sets

- I In the Settings window for Cut Point 2D, locate the Point Data section.
- 2 In the r text field, type Dd/2-Hw.
- **3** In the **z** text field, type U0.

2D Plot Group 1

- I On the **Results** toolbar, click **2D** Plot Group.
- 2 In the Settings window for 2D Plot Group, type Temperature Surface and Velocity Streamlines, 2D in the Label text field.

Temperature Surface and Velocity Streamlines, 2D

- I Right-click Temperature Surface and Velocity Streamlines, 2D and choose Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degF.
- **4** In the **Expression** text field, type T.
- 5 Right-click Temperature Surface and Velocity Streamlines, 2D and choose Streamline.

- 6 In the Settings window for Streamline, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>Heat Transfer> ht.ur,ht.uz Velocity field (Spatial).
- 7 Locate the Streamline Positioning section. From the Positioning list, choose Start point controlled.
- ID Plot Group 2
- I On the **Results** toolbar, click **ID Plot Group**.
- 2 In the Settings window for 1D Plot Group, type Temperature along Inner Wall in the Label text field.

Line Graph 1

On the Temperature along Inner Wall toolbar, click Line Graph.

Temperature along Inner Wall

- I In the Settings window for Line Graph, locate the y-Axis Data section.
- **2** In the **Expression** text field, type T.
- **3** Select Boundaries 22, 24, and 26 only.
- 4 From the Unit list, choose degF.
- 5 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- **6** In the **Expression** text field, type z/U0.
- 7 In the Model Builder window, click Temperature along Inner Wall.
- 8 In the Settings window for 1D Plot Group, click to expand the Title section.
- 9 From the Title type list, choose Manual.
- **IO** In the **Title** text area, type Temperature along inner wall.
- II Locate the Plot Settings section. Select the x-axis label check box.
- **12** In the associated text field, type z/U0.
- **I3** Select the **y-axis label** check box.
- **I4** In the associated text field, type T (degF).

ID Plot Group 3

- I On the Home toolbar, click Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for 1D Plot Group, type Inner Wall Temperature at End-of-Stroke Position in the Label text field.

Point Graph 1

On the Inner Wall Temperature at End-of-Stroke Position toolbar, click Point Graph.

Inner Wall Temperature at End-of-Stroke Position

- I In the Settings window for Point Graph, locate the y-Axis Data section.
- **2** In the **Expression** text field, type T.
- 3 Locate the Data section. From the Data set list, choose Cut Point 2D I.
- 4 Locate the y-Axis Data section. From the Unit list, choose degF.
- 5 In the Model Builder window, click Inner Wall Temperature at End-of-Stroke Position.
- 6 In the Settings window for 1D Plot Group, click to expand the Axis section.
- 7 Locate the Title section. From the Title type list, choose Manual.
- 8 In the Title text area, type Temperature of inner wall at end-of-stroke position.
- 9 Locate the Plot Settings section. Select the x-axis label check box.
- **IO** In the associated text field, type time (s).
- II Select the y-axis label check box.
- **12** In the associated text field, type T (degF).

STUDY I

Now return to the **Study I** branch to compute the solution.

Step 1: Time Dependent

- I In the Model Builder window, under Study I click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Results While Solving section.
- 3 From the Plot group list, choose Inner Wall Temperature at End-of-Stroke Position.
- 4 On the Home toolbar, click Compute.

RESULTS

Temperature Surface and Velocity Streamlines, 2D

During the solution time, a plot of the temperature at the probe position will be displayed and updated following the solver time steps.

When the solution is finished, click the **Zoom Extents** button on the **Graphics** toolbar. This will produce a plot of the temperature field and the flow streamlines within the damper, which should appear similar to that shown in Figure 3.

Temperature along Inner Wall

- I In the Model Builder window, under Results click Temperature along Inner Wall.
- 2 In the Settings window for 1D Plot Group, locate the Data section.

- **3** From the **Time selection** list, choose **From list**.
- 4 In the Times (s) list, choose 10 and 40.
- 5 On the Temperature along Inner Wall toolbar, click Plot.

This will show the temperature distribution along the damper inner wall at times 10 s and 40 s, it should look similar to that shown in Figure 5.

Inner Wall Temperature at End-of-Stroke Position

- I In the Model Builder window, under Results click Inner Wall Temperature at End-of-Stroke Position.
- 2 On the Inner Wall Temperature at End-of-Stroke Position toolbar, click Plot.

This will show the temperature variation at the probe position aver the complete loading time period, it should look similar to that shown in Figure 4.



Flow Through a Uniform Inclined Screen

Introduction

This example simulates the flow through a uniform inclined screen using the Screen feature in Single-Phase Flow physics and compares the results with an analytic solution due to Elder (Ref. 1). The Screen feature is a tool for modeling wire gauzes, perforated plates etc without resolving their geometric complexity (see the *CFD Module User's Guide* for further details).

Model Definition

The model geometry is shown in Figure 1.



Figure 1: Model geometry showing flow direction and screen inclination.

Air at a temperature of T = 20 °C enters the channel on the left with a uniform inlet velocity of $u_{in} = 1$ m/s and exits on the right at uniform pressure, $p_0 = 0$ Pa. The flow through the channel is obstructed by a screen inclined at an angle θ . The combined effect of resistance and refraction (suppression of the tangential velocity component) creates a non-uniform velocity profile on the downstream side of the screen. An asymptotic solution valid for small inclinations is (Ref. 1),

$$\frac{(u/u_{\rm in}-1)(1+\eta+K\cos^2\theta)}{(1-\eta)\tan\theta\cdot K\cos^2\theta} = \frac{2}{\pi}\log\left(\cot\left(\frac{\pi y}{2}\right)\right)$$

where K and η are the screen resistance and refraction coefficients. To facilitate comparison with the asymptotic solution, assume that the flow is incompressible and apply free-slip boundary conditions on the channel walls. Choose the user-defined option for both the screen type and refraction in order to set the resistance coefficient K to 2.2 and the refraction coefficient η to 0.78.

Results and Discussion

The study performs a Parametric Sweep with the angle θ taking the values,

$$\theta = \frac{\pi}{18}, \frac{\pi}{9}, \frac{\pi}{6}, \frac{2\pi}{9}, \frac{\pi}{4} \qquad (\theta = 10^{\circ}, 20^{\circ}, 30^{\circ}, 40^{\circ}, 45^{\circ})$$

Figure 2 shows the outlet velocity scaled according to the left-hand side of Equation 1 together with the asymptotic solution on the right-hand side.



Figure 2: Comparison between the asymptotic solution (blue) and the simulations (red).

The agreement between the asymptotic solution and the simulations is good, surprisingly so even for $\theta = \pi/4$ (45°). Figure 3 shows a surface plot of the pressure field together with velocity vectors on the upstream and downstream side of the screen. The velocity vectors are displaced from the screen for clarity. You can easily distinguish the induced pressure jump, the flow distribution and deflection. See Ref. 1 for asymptotic solutions to other

related screen-flow problems if you are looking to extend the analysis to screens of varying shape and/or resistance.



Figure 3: Pressure drop, flow distribution and deflection for a screen inclined at an angle of 45° to the incoming flow.

Notes About the COMSOL Implementation

The model uses the Screen feature together with a Parametric Sweep to vary the inclination angle of the screen.

Reference

1. J.W. Elder, "Steady Flow Through Non-Uniform Gauzes of Arbitrary Shape," J. Fluid Mech., vol 5, pp 355–363, 1959.

Application Library path: CFD_Module/Single-Phase_Benchmarks/ inclined_screen From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Laminar Flow (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
theta	pi/18	0.1745	Angle of inclination
u_in	1[m/s]	l m/s	Inlet velocity

GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 4.
- 4 Locate the Position section. In the x text field, type -2.

Bézier Polygon I (b1)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.

- 4 Find the **Control points** subsection. In row I, set x to -0.5*tan(theta).
- 5 In row 2, set x to 0.5*tan(theta) and y to 1.
- 6 On the Geometry toolbar, click Build All.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

LAMINAR FLOW (SPF)

Wall I

- I In the Model Builder window, under Component I (compl)>Laminar Flow (spf) click Wall I.
- 2 In the Settings window for Wall, locate the Boundary Condition section.
- **3** From the **Boundary condition** list, choose **Slip**.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the U_0 text field, type u_in.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- **2** Select Boundary 7 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

Screen 1

- I On the Physics toolbar, click Boundaries and choose Screen.
- **2** Select Boundary 4 only.
- 3 In the Settings window for Screen, locate the Screen Type section.

- 4 From the Screen type list, choose User defined. Locate the Parameters section. In the *K* text field, type 2.2.
- **5** From the **Refraction** list, choose **User defined**. In the η text field, type **0.78**.

STUDY I

Parametric Sweep

- I On the Study toolbar, click Parametric Sweep.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click Add.
- **4** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
theta	pi/18, pi/9, pi/6, 2*pi/9, pi/4	

5 On the Study toolbar, click Compute.

RESULTS

Velocity (spf)

Create a new plot group to reproduce Figure 2.

ID Plot Group 3

On the Home toolbar, click Add Plot Group and choose ID Plot Group.

Line Graph I

On the ID Plot Group 3 toolbar, click Line Graph.

ID Plot Group 3

I Select Boundary 7 only.

Type in the analytic solution.

- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- 3 In the Expression text field, type 2/pi*log(cot(pi*y/2)).
- 4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- **5** In the **Expression** text field, type y.
- 6 On the ID Plot Group 3 toolbar, click Plot.
- 7 In the Model Builder window, click ID Plot Group 3.
- 8 In the Settings window for 1D Plot Group, click to expand the Axis section.

- 9 Select the Manual axis limits check box.
- **IO** In the **x minimum** text field, type **0**.
- II In the **x maximum** text field, type 1.
- **12** In the **y minimum** text field, type -3.
- **I3** In the **y maximum** text field, type **3**.
- 14 In the Model Builder window, under Results>1D Plot Group 3 right-click Line Graph 1 and choose Duplicate.
- 15 In the Settings window for Line Graph, locate the Data section.
- 16 From the Data set list, choose Study 1/Parametric Solutions 1 (sol2).
- 17 From the Parameter selection (theta) list, choose From list.
- 18 In the Parameter values (theta) list, select 0.1745.

Scale the solutions for comparison with the analytic solution.

- I9 Locate the y-Axis Data section. In the Expression text field, type (u/u_in-1)/ (1-0.78)/2.2/cos(theta)^2*(1+0.78+2.2*cos(theta)^2)/tan(theta).
- **20** Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 2I From the Color list, choose Red.
- 22 Find the Line markers subsection. From the Marker list, choose Square.
- 23 Click to expand the Legends section. Select the Show legends check box.
- 24 From the Legends list, choose Manual.
- **25** In the table, enter the following settings:

Legends

pi/18

- 26 Right-click Results>ID Plot Group 3>Line Graph 2 and choose Duplicate.
- **27** In the **Settings** window for Line Graph, locate the **Data** section.
- 28 In the Parameter values (theta) list, select 0.3491.
- **29** Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.
- **30** Locate the Legends section. In the table, enter the following settings:

Legends
pi/9

- 3I Right-click Results>ID Plot Group 3>Line Graph 3 and choose Duplicate.
- 32 In the Settings window for Line Graph, locate the Data section.
- **33** In the **Parameter values (theta)** list, select **0.5236**.
- **34** Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.
- **35** Locate the **Legends** section. In the table, enter the following settings:

Legends

pi/6

- **36** Right-click **Results>ID Plot Group 3>Line Graph 4** and choose **Duplicate**.
- 37 In the Settings window for Line Graph, locate the Data section.
- 38 In the Parameter values (theta) list, select 0.6981.
- **39** Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.
- **40** Locate the **Legends** section. In the table, enter the following settings:

Legends

2*pi/9

- 4 Right-click Results>ID Plot Group 3>Line Graph 5 and choose Duplicate.
- **42** In the **Settings** window for Line Graph, locate the **Data** section.
- 43 In the Parameter values (theta) list, select 0.7854.
- **44** Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
- **45** Locate the **Legends** section. In the table, enter the following settings:

Legends

pi/4

- **46** In the **Model Builder** window, click **ID Plot Group 3**.
- **47** In the **Settings** window for 1D Plot Group, click to expand the **Title** section.
- **48** From the **Title type** list, choose **Manual**.
- 49 On the ID Plot Group 3 toolbar, click Plot.
- **50** In the **Title** text area, type Normalized streamwise velocity component downstream.

Data Sets

To generate Figure 3, continue with the steps below.

Study I/Solution I (3) (soll)

On the Results toolbar, click More Data Sets and choose Solution.

Data Sets

- I In the Settings window for Solution, locate the Solution section.
- 2 From the Solution list, choose Parametric Solutions I (sol2).

Selection

- I On the **Results** toolbar, click **Selection**.
- 2 In the Settings window for Selection, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- **4** Select Boundary 4 only.

Velocity (spf)

- I In the Model Builder window, expand the Results>Velocity (spf) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type p.
- 4 In the Model Builder window, right-click Velocity (spf) and choose Arrow Line.
- 5 In the Settings window for Arrow Line, locate the Data section.
- 6 From the Data set list, choose Study I/Parametric Solutions I (3) (sol2).
- 7 Locate the **Expression** section. In the **x component** text field, type up(u).
- 8 In the y component text field, type up(v).
- 9 Locate the Coloring and Style section. From the Arrow base list, choose Head.
- **IO** Select the **Scale factor** check box.
- II In the associated text field, type 0.25.
- **12** In the **Number of arrows** text field, type **30**.
- **I3** From the **Color** list, choose **Black**.
- **I4** Right-click **Results>Velocity (spf)>Arrow Line I** and choose **Deformation**.
- **I5** In the **Settings** window for Deformation, locate the **Expression** section.
- **I6** In the **x component** text field, type -0.05.
- **I7** In the **y component** text field, type **0**.
- **I8** Locate the **Scale** section. Select the **Scale factor** check box.

- **19** In the associated text field, type 1.
- 20 Right-click Arrow Line I and choose Duplicate.
- 21 In the Settings window for Arrow Line, locate the Expression section.
- **2** In the **x** component text field, type down(u).
- **23** In the **y component** text field, type down(v).
- 24 Locate the Coloring and Style section. From the Arrow base list, choose Tail.
- **25** In the Model Builder window, expand the Results>Velocity (spf)>Arrow Line 2 node, then click Deformation 1.
- **26** In the **Settings** window for Deformation, locate the **Expression** section.
- **27** In the **x component** text field, type 0.05.
- **28** In the **Model Builder** window, click **Velocity (spf)**.
- **29** In the **Settings** window for 2D Plot Group, click to expand the **Title** section.
- **30** From the **Title type** list, choose **Manual**.
- **3I** In the **Title** text area, type Pressure drop (Pa) and up/downstream velocity vectors on screen.
- 32 On the Velocity (spf) toolbar, click Plot.



Inkjet Nozzle — Level Set Method

Introduction

Inkjet printers are attractive tools for printing text and images because they combine low cost and high resolution with acceptable speed. The working principle behind inkjet technology is to eject small droplets of liquid from a nozzle onto a sheet of paper. Important properties of a printer are its speed and the resolution of the final images. Designers can vary several parameters to modify a printer's performance. For instance, they can vary the inkjet geometry and the type of ink to create droplets of different sizes. The size and speed of the ejected droplets are also strongly dependent on the speed at which ink is injected into the nozzle. Simulations can be useful to improve the understanding of the fluid flow and to predict the optimal design of an inkjet for a specific application.

Although initially invented to produce images on paper, the inkjet technique has since been adopted for other application areas. Instruments for the precise deposition of microdroplets often employ inkjets. These instruments are used within the life sciences for diagnosis, analysis, and drug discovery. Inkjets have also been used as 3D printers to synthesize tissue from cells and to manufacture microelectronics. For all of these applications it is important to be able to accurately control the inkjet performance.

This example demonstrates how to model the fluid flow within an inkjet using the Laminar Two-Phase Flow, Level Set interface.

Model Definition

Figure 1 shows the geometry of the inkjet studied in this example. Because of its symmetry you can use an axisymmetric 2D model. Initially, the space between the inlet and the nozzle is filled with ink. Additional ink is injected through the inlet during a period of 10 μ s, and it consequently forces ink to flow out of the nozzle. When the injection stops, a droplet of ink snaps off and continues to travel until it hits the target.





REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

Level Set Method

The Laminar Two-Phase Flow, Level Set interface uses a reinitialized, conservative level set method to describe and convect the fluid interface. The 0.5 contour of the level set function ϕ defines the interface, where ϕ equals 0 in air and 1 in ink. In a transition layer close to the interface, ϕ goes smoothly from 0 to 1. The interface moves with the fluid velocity, **u**, at the interface. The following equation describes the convection of the reinitialized level set function:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \mathbf{u}) + \gamma \left[\left(\nabla \cdot \left(\phi (1 - \phi) \frac{\nabla \phi}{|\nabla \phi|} \right) \right) - \varepsilon \nabla \cdot \nabla \phi \right] = 0$$

The thickness of the transition layer is proportional to ε . For this model you can use $\varepsilon = h_c/2$, where h_c is the typical mesh size in the region passed by the droplet.

The parameter γ determines the amount of reinitialization. A suitable value for γ is the maximum magnitude occurring in the velocity field.

Beside defining the fluid interface, the level set function is used to smooth the density and viscosity jumps across the interface through the definitions

$$\rho = \rho_{air} + (\rho_{ink} - \rho_{air})\phi$$
$$\mu = \mu_{air} + (\mu_{ink} - \mu_{air})\phi$$

TRANSPORT OF MASS AND MOMENTUM

The incompressible Navier-Stokes equations, including surface tension, describe the transport of mass and momentum. Both ink and air can be considered incompressible as long as the fluid velocity is small compared to the speed of sound. The Navier-Stokes equations are

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) - \nabla \cdot \left(\mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \right) + \nabla p = \mathbf{F}_{st}$$
$$(\nabla \cdot \mathbf{u}) = 0$$

Here, ρ denotes density (kg/m³), μ equals the dynamic viscosity (N·s/m²), **u** represents the velocity (m/s), *p* denotes pressure (Pa), and **F**_{st} is the surface tension force.

The surface tension force is computed as

$$\mathbf{F}_{st} = \nabla \cdot \mathbf{T}$$
$$\mathbf{T} = \sigma(\mathbf{I} - (\mathbf{nn}^T))\delta$$

where **I** is the identity matrix, **n** is the interface normal, σ is the surface tension coefficient (N/m), and δ equals a Dirac delta function that is nonzero only at the fluid interface. The normal to the interface is

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|}$$

while the delta function is approximated by

$$\delta = 6 |\phi(1 - \phi)| |\nabla \phi|$$

The following table gives the physical parameters of ink and air used in the model:

MEDIUM	DENSITY	DYNAMIC VISCOSITY	SURFACE TENSION
ink	10 ³ kg/m ³	0.01 N·s/m ²	0.07 N/m
air	1.225 kg/m ³	1.789·10 ⁻⁵ N·s/m ²	

INITIAL CONDITIONS

Figure 2 shows the initial distribution (t = 0) of ink and air. The velocity is initially 0.



Figure 2: Initial distribution of ink. Black corresponds to ink and white corresponds to air.

BOUNDARY CONDITIONS

Inlet

The inlet velocity in the z direction increases from 0 to the parabolic profile

$$v(r) = 4.5 \left(\frac{r+0.1 \text{ mm}}{0.2 \text{ mm}}\right) \left(1 - \frac{r+0.1 \text{ mm}}{0.2 \text{ mm}}\right) \text{ m/s}$$

during the first 2 μ s. The velocity is then v(r) for 10 μ s and finally decreases to 0 for another 2 μ s. The time-dependent velocity profile in the *z* direction can then be defined as

$$v(r,t) = (\operatorname{step}(t - 1 \cdot 10^{-6}) - \operatorname{step}(t - 13 \cdot 10^{-6})) \cdot v(r)$$

where t is given in seconds and step(t) is a smooth step function (see Figure 3).



Figure 3: Smooth step function.

Use $\phi = 1$ as the inlet boundary condition for the level set variable.

Outlet

Set a constant pressure at the outlet. The value of the pressure given here is not important because the velocity depends only on the pressure gradient. You thus obtain the same velocity field regardless of whether the pressure is set to 1 atm or to 0.

Walls

On all other boundaries except the target, set No slip conditions. Use the Wetted wall condition on the target, with a contact angle of $\pi/2$ and a slip length of 10 μ m.

Results and Discussion

Figure 4 shows the ink surface and the velocity field at different times.


Figure 4: Position of air/ink interface and velocity field at various times.

Figure 5 illustrates the mass of ink that is further than 0.7 mm from the inlet. The figure shows that the mass of the ejected droplet is approximately $1.9 \cdot 10^{-10}$ kg.



Figure 5: Amount of ink from just above the nozzle.

This example studies only one inkjet model, but it is easy to modify the model in several ways. You can, for example, change properties such as the geometry or the inlet velocity and study the influence on the size and the speed of the ejected droplets. You can also investigate how the inkjet would perform if the ink were replaced by a different fluid. It is also easy to add forces such as gravity to the model.

Notes About the COMSOL Implementation

You can readily set up the model using the Laminar Two-Phase Flow, Level Set interface. This interface adds the equations automatically, and you need only specify physical parameters of the fluids and the initial and boundary conditions.

In order to accurately resolve the interface between the air and ink, use the adaptive meshing. This means that as the interface moves during the simulation, the mesh is updated in order to keep the mesh refined in the interface region.

The simulation procedure involves two consecutive computations. First you calculate a smooth initial solution for the level set variable. Using this initial solution, you then start the time-dependent simulation of the fluid motion.

To calculate the droplet's mass, use an integration coupling operator. To visualize the droplet in 3D, revolve the 2D axially symmetric solution to a 3D geometry.

References

1. J.-T. Yeh, "A VOF-FEM Coupled Inkjet Simulation," *Proc. ASME FEDSM'01*, New Orleans, Louisiana, 2001.

2. E. Olsson and G. Kreiss, "A Conservative Level Set Method for Two Phase Flow," J. Comput. Phys., vol. 210, pp. 225–246, 2005.

3. P. Yue, J. Feng, C. Liu, and J. Shen, "A Diffuse-Interface Method for Simulating Two-Phase Flows of Complex Fluids," *J. Fluid Mech.*, vol. 515, pp. 293–317, 2004.

Application Library path: CFD_Module/Multiphase_Tutorials/inkjet_nozzle_ls

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Two-Phase Flow, Level Set> Laminar Two-Phase Flow, Level Set.
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Transient with Phase Initialization.
- 6 Click Done.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.

3 From the Length unit list, choose mm.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.1.
- 4 In the **Height** text field, type 0.2.

Bézier Polygon I (b1)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 4 Find the Added segments subsection. Click Add Linear.
- 5 Find the **Control points** subsection. In row I, set r to 0.1 and z to 0.2.
- 6 In row 2, set z to 0.2.
- 7 Find the Added segments subsection. Click Add Linear.
- 8 Find the Control points subsection. In row 2, set z to 0.575.
- 9 Find the Added segments subsection. Click Add Linear.
- **IO** Find the **Control points** subsection. In row **2**, set **r** to **0**.025.
- II Find the Added segments subsection. Click Add Linear.
- 12 Find the Control points subsection. Click Close Curve.
- **I3** Right-click **Bézier Polygon I (bI)** and choose **Build Selected**.
- I4 Click the Zoom Extents button on the Graphics toolbar.

Rectangle 2 (r2)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.025.
- 4 In the **Height** text field, type 1.025.
- 5 Locate the Position section. In the z text field, type 0.575.
- 6 Right-click Rectangle 2 (r2) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Rectangle 3 (r3)

I On the Geometry toolbar, click Primitives and choose Rectangle.

- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.1**.
- 4 In the **Height** text field, type 1.
- **5** Locate the **Position** section. In the **r** text field, type **0**.
- 6 In the z text field, type 0.6.

Rectangle 4 (r4)

- I Right-click Rectangle 3 (r3) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Rectangle.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the **Width** text field, type 0.2.
- 5 In the **Height** text field, type 0.1.
- 6 Locate the **Position** section. In the **z** text field, type 1.5.

Form Union (fin)

- I Right-click Rectangle 4 (r4) and choose Build Selected.
- 2 In the Model Builder window, under Component I (compl)>Geometry I right-click Form Union (fin) and choose Build Selected.

This completes the geometry modeling stage.



MATERIALS

In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.

Material I (mat1)

I In the Settings window for Material, locate the Material Contents section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1e3[kg/m^3]	kg/m³	Basic
Dynamic viscosity	mu	1e-2[Pa*s]	Pa·s	Basic

3 Right-click Component I (compl)>Materials>Material I (matl) and choose Rename.

4 In the Rename Material dialog box, type Ink in the New label text field.

5 Click OK.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.

DEFINITIONS

Now, define a step function to use when defining the time dependence of the inlet velocity.

Step I (step I)

- I On the Home toolbar, click Functions and choose Local>Step.
- 2 In the Settings window for Step, click to expand the Smoothing section.
- 3 In the Size of transition zone text field, type 2*1e-6.
- 4 Click Plot.

Next, define an integration operator that you will use when defining a variable for the droplet mass.

Integration 1 (intop1)

- I On the **Definitions** toolbar, click **Component Couplings** and choose **Integration**.
- 2 In the Settings window for Integration, locate the Source Selection section.
- **3** From the **Selection** list, choose **All domains**.

After these preliminaries, you can define variables for the inlet velocity and the droplet mass.

Variables I

- I On the **Definitions** toolbar, click **Local Variables**.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** In the table, enter the following settings:

Name	Expression	Unit	Description
v_inr	4.5[m/s]*((r+0.1[mm])/ 0.2[mm])*(1-((r+ 0.1[mm])/0.2[mm]))	m/s	r-dependent inlet velocity factor
inlett	step1(t[1/ s]-1e-6)-step1(t[1/ s]-13e-6)		t-dependent inlet velocity factor
v_in	v_inr*inlett	m/s	Inlet velocity
m_d	intop1(1e3[kg/m^3]* phils*(z>0.7[mm])*2*pi* r)	kg	Droplet mass

4 In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

LEVEL SET (LS)

- I In the Model Builder window's toolbar, click the Show button and select Advanced Physics Options in the menu.
- 2 In the Model Builder window, under Component I (compl) click Level Set (Is).
- 3 In the Settings window for Level Set, click to expand the Advanced settings section.
- 4 Locate the Advanced Settings section. From the Convective term list, choose Conservative form.
- 5 In the Model Builder window, click Level Set (Is).

Initial Interface 1

- I On the Physics toolbar, click Boundaries and choose Initial Interface.
- **2** Select Boundary 8 only.
- 3 In the Model Builder window, click Level Set (Is).
- 4 In the Settings window for Level Set, click to expand the Discretization section.
- 5 In the Settings window for Level Set, locate the Discretization section.

6 From the Level set variable list, choose Linear.

LAMINAR FLOW (SPF)

On the Physics toolbar, click Level Set (Is) and choose Laminar Flow (spf).

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 In the Settings window for Laminar Flow, click to expand the Discretization section.
- 3 From the Discretization of fluids list, choose PI+PI.

The model utilizes adaptive meshing and linear elements will then suffice.

MULTIPHYSICS

- I In the Model Builder window, under Component I (compl)>Multiphysics click Two-Phase Flow, Level Set I (tpfl).
- **2** In the **Settings** window for Two-Phase Flow, Level Set, locate the **Fluid I Properties** section.
- 3 From the Fluid I list, choose Air (mat2).
- 4 Locate the Fluid 2 Properties section. From the Fluid 2 list, choose lnk (matl).
- 5 Locate the Surface Tension section. From the Surface tension coefficient list, choose User defined. In the σ text field, type 0.07.

LEVEL SET (LS)

On the Physics toolbar, click Laminar Flow (spf) and choose Level Set (ls).

Level Set Model 1

- I In the Model Builder window, under Component I (compl)>Level Set (Is) click Level Set Model I.
- 2 In the Settings window for Level Set Model, locate the Level Set Model section.
- 3 In the ε_{ls} text field, type 2.5e-6.
- **4** In the γ text field, type 10.

Initial Values 2

- I On the Physics toolbar, click Domains and choose Initial Values.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the **Domain initially** list, choose **Fluid 2** ($\phi = I$).
- **4** Select Domains 1–3 only.

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Level Set (Is) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the **Domain initially** list, choose **Fluid I** ($\phi = 0$).

LAMINAR FLOW (SPF)

On the Physics toolbar, click Level Set (Is) and choose Laminar Flow (spf).

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundary 2 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the U_0 text field, type v_in.

LEVEL SET (LS)

- I In the Model Builder window, under Component I (compl) click Level Set (Is).
- 2 On the Physics toolbar, click Boundaries and choose Inlet.
- 3 Select Boundary 2 only.
- 4 In the Settings window for Inlet, locate the Inlet section.
- **5** In the ϕ text field, type 1.

LAMINAR FLOW (SPF)

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 24 only.

LEVEL SET (LS)

- I In the Model Builder window, under Component I (compl) click Level Set (Is).
- 2 On the Physics toolbar, click Boundaries and choose Outlet.
- **3** Select Boundary 24 only.

MULTIPHYSICS

Wetted Wall I (ww1)

- I On the Physics toolbar, click Multiphysics and choose Boundary>Wetted Wall.
- 2 Select Boundaries 11, 18, and 23 only.
- 3 In the Settings window for Wetted Wall, locate the Wetted Wall section.
- **4** In the β text field, type 10[um].

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, click Build All.

STUDY I

Step 2: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step 2: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range(0,10e-6,200e-6).
- 4 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.

This choice means that the **Graphics** window will show a contour line of the volume function of Fluid 1 and velocity field while solving, and this plot will be updated at each output time step.

- 5 From the Plot group list, choose Default.
- 6 Click to expand the **Study extensions** section. Locate the **Study Extensions** section. Select the **Adaptive mesh refinement** check box.

By adjusting the scaling of the fields manually, you can reduce the computation time

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll)>Dependent Variables 2 node, then click Velocity field (compl.u).
- 4 In the Settings window for Field, locate the Scaling section.
- 5 From the Method list, choose Manual.

- 6 In the Scale text field, type 10.
- 7 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables 2 click Pressure (compl.p).
- 8 In the Settings window for Field, locate the Scaling section.
- 9 From the Method list, choose Manual.
- **IO** In the **Scale** text field, type 1e4.
- II In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables 2 click Level set variable (compl.phils).
- 12 In the Settings window for Field, locate the Scaling section.
- **I3** From the **Method** list, choose **Manual**.
- **I4** On the **Study** toolbar, click **Compute**.

RESULTS

Volume Fraction of Fluid 1 (Is) 1

- I In the Model Builder window, expand the Volume Fraction of Fluid I (Is) I node.
- 2 Right-click Results>Volume Fraction of Fluid I (Is) I and choose Slice.
- 3 In the Settings window for Slice, locate the Plane Data section.
- 4 From the Plane list, choose zx-planes.
- 5 In the Planes text field, type 1.
- 6 In the Model Builder window, click Volume Fraction of Fluid I (Is) I.
- 7 In the Settings window for 3D Plot Group, locate the Data section.
- 8 From the Time (s) list, choose 4E-5.
- 9 On the Volume Fraction of Fluid I (Is) I toolbar, click Plot.
- **IO** Click the **Zoom Extents** button on the **Graphics** toolbar.
- II In the Model Builder window, click Volume Fraction of Fluid I (Is) I.
- 12 In the Settings window for 3D Plot Group, locate the Data section.
- **I3** From the **Time (s)** list, choose **0**.
- 14 On the Volume Fraction of Fluid I (Is) I toolbar, click Plot.

Compare the resulting plot with that in the upper panel of Figure 4.To create the remaining plots, plot the solution for the time values 2e-5, 4e-5, 8e-5, 1.2e-4, 1.6e-4, and 2e-4.

ID Plot Group 6

- I On the Home toolbar, click Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Study I/Refined Mesh Solution I (sol3).

Global I

On the ID Plot Group 6 toolbar, click Global.

ID Plot Group 6

- In the Settings window for Global, click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component I>Definitions>Variables>m_d
 Droplet mass.
- 2 On the ID Plot Group 6 toolbar, click Plot.



Journal Bearing

This model is licensed under the COMSOL Software License Agreement 5.2a. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Journal bearings are used to carry radial loads, for example, to support a rotating shaft.

A simple journal bearing consists of two rigid cylinders. The outer cylinder (bearing) wraps the inner rotating journal (shaft). Normally, the position of the journal center is eccentric with the bearing center. A lubricant fills the small annular gap or clearance between the journal and the bearing. The amount of eccentricity of the journal is related to the pressure that is generated in the bearing to balance the radial load. The lubricant is supplied through a hole or a groove and may or may not extend all around the journal.

Under normal operating conditions, the gases dissolved in the lubricant cause cavitation in the diverging clearance between the journal and the bearing. This happens because the pressure in the lubricant drops below the saturation pressure for the release of dissolved gases. The saturation pressure is normally similar to the ambient pressure. The following model does not account for cavitation and therefore predicts sub-ambient pressures. Such sub-ambient pressures are the result of the so-called Sommerfeld boundary condition. For practical purposes, these sub-ambient pressures should be neglected.

Model Definition

The pressure in the lubricant (SAE 10 at 70° C) is governed by the Reynolds equation. For an incompressible fluid with no slip condition, the stationary Reynolds equation in the continuum range is given by

$$\nabla_{\mathrm{T}} \cdot \left(\frac{-\rho h^3}{12\mu} \nabla_{\mathrm{T}} p + \frac{\rho h}{2} (v_{\mathrm{a}} + v_{\mathrm{b}}) \right) - \rho((\nabla_{\mathrm{T}} b \cdot v_{\mathrm{b}}) - (\nabla_{\mathrm{T}} a \cdot v_{\mathrm{a}})) = 0$$

In this equation, ρ is the density (SI unit: kg/m³), *h* is the lubricant thickness (SI unit: m), μ is the viscosity (SI unit: Pa·s), *p* is the pressure (SI unit: Pa), *a* is the location (m) of the channel base, v_a is the tangential velocity (SI unit: m/s) of the channel base, *b* is the location (SI unit: m) of the solid wall, and v_b is the tangential velocity (SI unit: m/s) of the solid wall.

The rotating journal is considered to be the solid wall. Figure 1 shows the rotating journal wall on which you solve the Reynolds equation. Because the pressure is constant through the lubricant film thickness, COMSOL uses the tangential projection of the gradient operator, $\nabla_{\rm T}$, to calculate the pressure distribution on the lubricant surface. Note that in this case the term $\rho((\nabla_{\rm T} b \cdot v_{\rm b}) - (\nabla_{\rm T} a \cdot v_{\rm a}))$ equates to 0, so the governing equation simplifies to

$$\nabla_{\mathrm{T}} \cdot \left(\frac{-\rho h^{3}}{12\mu} \nabla_{\mathrm{T}} p + \frac{\rho h}{2} (v_{\mathrm{a}} + v_{\mathrm{b}})\right) = 0$$

The lubricant thickness, h, is defined as

$$h = c(1 + \varepsilon \cos \theta)$$

where $c \equiv R_{\rm B} - R_{\rm J}$ is the difference between the bearing radius and the journal radius, ε is the eccentricity, and θ is the polar angular coordinate of a point on the lubricant. Figure 2 shows the converging and diverging lubricant thickness around the journal.



Figure 1: Geometry (cylindrical journal) showing the base velocity direction with red arrows.



Figure 2: The lubricant thickness around the rotating journal.

BORDER CONDITIONS

The pressure at the ends of the cylindrical journal is assumed to be similar to the ambient pressure. Therefore, the border conditions are

$$p = 0$$
 at $z = 0, L$

where L is the length of the cylindrical journal.

Results and Discussion

Figure 3 shows the calculated pressure distribution and pressure contours. As expected, the maximum pressure is reached in a region closer to the minimum lubricant thickness. Sub-ambient or negative pressure also results due to approximate boundary conditions. For a more accurate modeling of pressure distribution, gaseous cavitation has to be taken into account.



Figure 3: Pressure distribution and pressure contours on the journal.

Application Library path: CFD_Module/Thin-Film_Flow/journal_bearing

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Thin-Film Flow>Thin-Film Flow, Shell (tffs).
- 3 Click Add.
- 4 Click Study.

5 In the Select Study tree, select Preset Studies>Stationary.

6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
R	0.03[m]	0.03 m	Journal radius
Н	0.05[m]	0.05 m	Journal height
С	0.03[mm]	3E-5 m	Clearance between the bearing and the journal
omega	1500/60*2*pi[rad/s]	157.1 rad/s	Journal angular velocity

GEOMETRY I

Cylinder I (cyl1)

- I On the Geometry toolbar, click Cylinder.
- 2 In the Settings window for Cylinder, locate the Object Type section.
- 3 From the Type list, choose Surface.
- 4 Locate the Size and Shape section. In the Radius text field, type R.
- **5** In the **Height** text field, type H.
- 6 Click Build All Objects.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Variables 1

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
angle	atan2(y,x)[rad]	rad	Angle along circumference
h	c*(1+0.6*cos(angle))	m	Lubricant film thickness
u	-omega*R*sin(angle)	m/s	x-component of journal velocity
v	omega*R*cos(angle)	m/s	y-component of journal velocity

THIN-FILM FLOW, SHELL (TFFS)

Fluid-Film Properties 1

- I In the Model Builder window, expand the Component I (compl)>Thin-Film Flow, Shell (tffs) node, then click Fluid-Film Properties I.
- 2 In the Settings window for Fluid-Film Properties, locate the Fluid Properties section.
- **3** From the ρ list, choose **User defined**. In the associated text field, type 860[kg/m^3].
- **4** From the μ list, choose **User defined**. In the associated text field, type **0.01**[Pa*s].
- **5** Locate the **Wall Properties** section. In the h_{w1} text field, type h.
- 6 Click to expand the Base properties section. Locate the Base Properties section. From the v_b list, choose User defined. Specify the vector as
- u x
- V y
- 0 z

Border I

As you can see in the **Border Settings** section, the default condition that applies at the cylinder ends is **Zero pressure**.

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Build All.

STUDY I

On the Home toolbar, click Compute.

RESULTS

Fluid Pressure (tffs)

The default plot group shows the pressure field as a surface plot and the displacement of the solid wall as a deformation plot. Add a contour plot of the same quantity to reproduce the plot in Figure 3.

- I In the Model Builder window, expand the Fluid Pressure (tffs) node, then click Surface 1.1.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose MPa.
- 4 In the Model Builder window, right-click Fluid Pressure (tffs) and choose Contour.
- 5 In the Settings window for Contour, locate the Expression section.
- 6 From the Unit list, choose MPa.
- 7 Locate the Coloring and Style section. From the Color table list, choose GrayScale.
- 8 Clear the **Color legend** check box.
- 9 In the Model Builder window, click Fluid Pressure (tffs).
- 10 In the Settings window for 3D Plot Group, click to expand the Title section.
- II From the Title type list, choose Manual.
- 12 In the Title text area, type Pressure (MPa).
- I3 On the Fluid Pressure (tffs) toolbar, click Plot.
- **I4** Click the **Zoom Extents** button on the **Graphics** toolbar.

To see the bearing from different angles just click and drag in the Graphics window.

The following steps will reproduce Figure 1.

3D Plot Group 2

- I On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- 2 In the Model Builder window, right-click 3D Plot Group 2 and choose Surface.
- 3 In the Settings window for Surface, locate the Coloring and Style section.
- 4 From the Coloring list, choose Uniform.
- 5 From the Color list, choose Gray.
- 6 Right-click 3D Plot Group 2 and choose Arrow Surface.
- 7 In the Settings window for Arrow Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Thin-Film Flow, Shell>Wall and base properties>tffs.vbx,...,tffs.vbz Velocity of base.
- 8 In the Model Builder window, click 3D Plot Group 2.

- 9 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- **IO** Clear the **Plot data set edges** check box.
- II Click to expand the Title section. From the Title type list, choose None.

3D Plot Group 3

Reproduce Figure 2 by the following steps.

- I On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- 2 In the Model Builder window, right-click 3D Plot Group 3 and choose Surface.
- 3 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Thin-Film Flow, Shell>Wall and base properties>tffs.h - Total gap height.
- 4 Locate the Expression section. From the Unit list, choose µm.
- 5 On the 3D Plot Group 3 toolbar, click Plot.
- 6 In the Model Builder window, click 3D Plot Group 3.
- 7 In the Settings window for 3D Plot Group, click to expand the Color legend section.
- 8 Locate the Color Legend section. Select the Show maximum and minimum values check box.

10 | JOURNAL BEARING



Journal Bearing with Cavitation

Introduction

Journal bearings are used to carry radial loads, for example, to support a rotating shaft.

A simple journal bearing consists of two rigid cylinders. The outer cylinder (bearing) wraps the inner rotating journal (shaft). Normally, the position of the journal center is eccentric with the bearing center. A lubricant fills the small annular gap or clearance between the journal and the bearing. The amount of eccentricity of the journal is related to the pressure that is generated in the bearing to balance the radial load. The lubricant is supplied through a hole or a groove and may or may not extend all around the journal.

If the bearing is not designed correctly, the gases dissolved in the lubricant can cause cavitation in the diverging clearance between the journal and the bearing. This happens because the pressure in the lubricant drops below the saturation pressure for the release of dissolved gases. The saturation pressure is normally similar to the ambient pressure. Cavitation can cause damage to the bearing components leading to premature failure.

The following model predicts the onset and extent of cavitation in the lubrication layer. The onset and extent of gaseous cavitation in a journal bearing determine the load that can be applied to the bearing.

This example is based on the Journal Bearing model, that does not include cavitation effects; review that model before beginning this one.

Model Definition

The governing equation, geometry and boundary conditions are discussed for the Journal Bearing model.

With the cavitation feature enabled, the flow in the journal bearing is divided in two regions:

- A full film region where the pressure varies but is limited from below by the cavitation pressure.
- A cavitation region where only part of the volume is occupied by the fluid. Because of the presence of the gas in the void fraction, the pressure in this region is assumed to be constant and equal to the cavitation pressure.

Elrod and Adams derived a general form of the Reynolds equation by introducing a switch function, *g*, equal to 1 in the full film region ($\theta \ge 1$) and 0 in the cavitation region ($\theta < 1$). This switch function allows for solving a single equation for both the full film and the

cavitation region and leads to a modified version of the average velocity used in the Reynold's equation:

$$\mathbf{v}_{av} = \mathbf{v}_{av,c} - g v_{av,p} \nabla_t p_f$$

where the first and second terms on the right-hand side correspond to the average Couette and average Poiseuille velocities, respectively. This switch function sets the average Poiseuille velocity is to zero in the cavitation region.

Because the average Poiseuille velocity is set to zero in the cavitation region, the density needs to be a function of the pressure variable and could be defined as

$$\rho = \rho_c e^{\frac{p-p_c}{\beta}}$$

A density that is not pressure dependent would lead to empty equations in the cavitation region since the pressure variable p would no longer be present in the governing equations.

Results and Discussion

While the pressure is constant and equal to the cavitation pressure in the cavitation region, the computed pressure, pfilm, is negative in this region. The value of this negative pressure can be used to derive the volume fraction of fluid in the cavitation region. The actual or physical pressure, available in the post processing section as tffs.p, is equal to the computed pressure in the full film region and equal to the cavitation pressure in the cavitation region. Figure 1 shows this physical pressure, tffs.p. The maximum pressure



is reached in a region closer to the minimum lubricant thickness.

Figure 1: Pressure distribution and pressure contours on the journal.

Figure 2 shows the fluid mass fraction. The mass fraction is equal to 1 in the full film region and less than 1 in the cavitation region (where only part of the volume is occupied by the fluid). It is computed as the minimum value between 1 and the ratio ρ/ρ_{cav} , where ρ and ρ_{cav} represent the fluid density and the density at the cavitation pressure, respectively.



Figure 2: Fluid mass fraction.

Application Library path: CFD_Module/Thin-Film_Flow/ journal_bearing_cavitation

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

I In the Model Wizard window, click 3D.

2 In the Select Physics tree, select Fluid Flow>Thin-Film Flow>Thin-Film Flow, Shell (tffs).

- 3 Click Add.
- 4 Click Study.

5 In the Select Study tree, select Preset Studies>Stationary.

6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
R	0.03[m]	0.03 m	Journal radius
Н	0.05[m]	0.05 m	Journal height
С	0.03[mm]	3E-5 m	Clearance between the bearing and the journal
omega	1500/60*2*pi[rad/s]	157.1 rad/s	Journal angular velocity

GEOMETRY I

Cylinder I (cyl1)

- I On the Geometry toolbar, click Cylinder.
- 2 In the Settings window for Cylinder, locate the Object Type section.
- 3 From the Type list, choose Surface.
- 4 Locate the Size and Shape section. In the Radius text field, type R.
- 5 In the Height text field, type H.
- 6 Click Build All Objects.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
angle	atan2(y,x)[rad]	rad	Angle along circumference
h	c*(1+0.6*cos(angle))	m	Lubricant film thickness
u	-omega*R*sin(angle)	m/s	x-component of journal velocity
v	omega*R*cos(angle)	m/s	y-component of journal velocity

THIN-FILM FLOW, SHELL (TFFS)

- I In the Model Builder window's toolbar, click the Show button and select Advanced Physics Options in the menu.
- 2 In the Model Builder window, under Component I (compl) click Thin-Film Flow, Shell (tffs).
- 3 In the Settings window for Thin-Film Flow, Shell, click to expand the Cavitation section.
- 4 Select the Cavitation check box.

Fluid-Film Properties 1

- I In the Model Builder window, under Component I (compl)>Thin-Film Flow, Shell (tffs) click Fluid-Film Properties I.
- 2 In the Settings window for Fluid-Film Properties, locate the Fluid Properties section.
- **3** From the μ list, choose **User defined**. In the associated text field, type 0.01[Pa*s].
- **4** Locate the **Wall Properties** section. In the h_{w1} text field, type h.
- 5 Click to expand the Base properties section. Locate the Base Properties section. From the v_b list, choose User defined. Specify the vector as

u	x
v	у
0	z

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Build All.

STUDY I

On the Home toolbar, click Compute.

RESULTS

Fluid Pressure (tffs)

The default plot group shows the pressure field as a surface plot. Add a contour plot of the same quantity to reproduce the plot in Figure 1.

- I In the Model Builder window, expand the Fluid Pressure (tffs) node, then click Surface 1.1.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type tffs.p.
- 4 From the Unit list, choose MPa.
- 5 On the Fluid Pressure (tffs) toolbar, click Plot.
- 6 In the Model Builder window, right-click Fluid Pressure (tffs) and choose Contour.
- 7 In the Settings window for Contour, locate the Expression section.
- 8 In the **Expression** text field, type tffs.p.
- 9 From the Unit list, choose MPa.
- 10 Locate the Coloring and Style section. From the Color table list, choose GrayScale.
- II Clear the **Color legend** check box.
- 12 In the Model Builder window, click Fluid Pressure (tffs).
- 13 In the Settings window for 3D Plot Group, click to expand the Title section.
- **I4** From the **Title type** list, choose **Manual**.
- **I5** In the **Title** text area, type **Pressure** (MPa).
- 16 On the Fluid Pressure (tffs) toolbar, click Plot.
- **17** Click the **Zoom Extents** button on the **Graphics** toolbar.

To see the bearing from different angles just click and drag in the Graphics window.

3D Plot Group 2

Reproduce Figure 2 by the following these steps.

- I On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- 2 In the Model Builder window, right-click 3D Plot Group 2 and choose Rename.
- 3 In the Rename 3D Plot Group dialog box, type Mass Fraction in the New label text field.
- 4 Click OK.
- 5 On the Mass Fraction toolbar, click Surface.

Mass Fraction

I In the Model Builder window, under Results>Mass Fraction click Surface I.

- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Thin-Film Flow, Shell> Cavitation (Cavitation)>tffs.theta Mass fraction.
- **3** On the Mass Fraction toolbar, click Plot.

10 | JOURNAL BEARING WITH CAVITATION



Particle Tracing in a Micromixer

Introduction

Micromixers can either be static or dynamic depending on the required mixing time and length scale. For static mixers, the Reynolds number has to be suitably high to induce turbulence-enhanced mixing. Often micromixers operate in the laminar flow regime due to their small characteristic size. The diffusivity of a solute in the flowing fluid may also be extremely small, on the order of 10^{-10} m²/s. This results in mixing length scales on the order of meters—clearly unacceptable for a microscale device. One way to alleviate this problem is to add mixing elements to induce vorticity into the flow. A dynamic mixer uses rotating blades to enhance the mixing process, allowing for smaller-scale devices. The one big disadvantage of a dynamic mixer is that moving parts are required.

Note: This application requires the Particle Tracing Module.

Model Definition

This example examines how mixing between microscopic particles occurs in a micromixer. Particles enter the mixer through 3 **Inlet** features and exit through the **Outlet** feature. The particles enter the modeling domain through the inlets in a continuous stream. A new set of particles is released every 50 milliseconds for a total duration of one second. After this, no more particles are released but the model is solved for an additional second. For each release inlet and each release time, 50 particles are released with an initial velocity equal to the fluid velocity, so a total of $3 \times 21 \times 50 = 3150$ particles are released.

The geometry is an assembly containing stationary and rotating domains. The particles are free to cross the pair boundary between the stationary and moving domains as it if were invisible, provided that the **Pair Continuity** feature is used in the Particle Tracing for Fluid Flow interface.

The blades are rotating at a constant angular velocity of 1 revolution per second in the anti-clockwise direction.



Figure 1: Plot of the model geometry. The geometry length unit is millimeters.

The particles obey Newton's second law:

$$\frac{d}{dt}(m_p \mathbf{v}) = m_p F_D(\mathbf{u} - \mathbf{v})$$

where **u** is the fluid velocity (SI unit: m/s), m_p is the particle mass (SI unit: kg), **v** is the particle velocity (SI unit: m/s) and F_D is the drag force per unit mass (SI unit: 1/s). When the relative Reynolds number between the particles and fluid is small, as is the case here, the drag force per unit mass can be written as:

$$F_D = \frac{18\mu}{\rho_p d_p^2}$$

where μ is the fluid viscosity (SI unit Pa s), ρ_p is the particle density (SI unit: kg/m³) and d_p is the particle diameter (SI unit: m). To compute the trajectory length of each of the particles the following ordinary differential equation is solved for each particle:

$$\frac{d}{ds}(tl) = 1$$

where tl is the trajectory length and *s* is the tangential direction of particle motion at any given instant.

The mesh needs to be quite fine on the stationary/sliding interface so that the fluid motion remains continuous. The mesh used in this model is plotted in Figure 2.



Figure 2: The mesh is quite fine on the pair boundary to accurately resolve the flow field.

The flow field is computed using the Rotating Machinery, Laminar Flow interface. The force exerted on the fluid from the particles is neglected in this model. So, it is possible to solve for the flow field only in one study, then use a separate study to compute the particle trajectories based on that flow field. This is usually the recommended approach, if the field is computed from a stationary study. In this case, there are very strong transients in the model, meaning that a huge number of timesteps have to be stored if the model is to be solved sequentially. It is more attractive to solve for the particle trajectories and flow field in a single study step.

Results and Discussion

The location of the particles at different snapshots in time is plotted in Figure 3. The particle color is different for each release feature, which conveniently allows the effect of the mixing to be visualized. The particles make their way normally inwards from the inlets and, like the fluid velocity, begin to assume a parabolic velocity flow profile. The particles
entering from the left (the blue particles) are then swept downwards due to the presence of the rotating blades. The particles entering from the right (the red particles) are swept upwards, but the momentum they acquire from the blades means that very few reach the outlet. Between 0.6 and one second, the particles begin to reach the outlet. Mixing of the three particle streams continues until the particle stream is shut off. By two seconds the particles mainly accumulate in the top left hand corner of the geometry, where they flow towards the outlet in the opposite direction to that of the blades. This is because liquid continues to flow in from all of the inlets after the particle stream is terminated.



Figure 3: Plot of the particle coordinates at different stages of the mixing process. The color is different for each of the inlet features.

This example uses the assembly option when finalizing the geometry. The assembly option automatically creates pair features which allows the mesh in a rotating domain to slide with respect to the stationary domain. For the fluid flow, the Flow Continuity feature must be added on the pair boundaries. For the particle tracing, the Particle Continuity feature must be used on pairs.

Reference

1. G. Karniadakis, A. Beskok, and N. Aluru, Microflows and Nanoflows, Springer, 2005.

Application Library path: CFD_Module/Particle_Tracing/ micromixer particle tracing

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Rotating Machinery, Fluid Flow>Rotating Machinery, Laminar Flow (rmspf).
- 3 Click Add.
- 4 In the Select Physics tree, select Fluid Flow>Particle Tracing>Particle Tracing for Fluid Flow (fpt).
- 5 Click Add.
- 6 Click Study.
- 7 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Time Dependent.
- 8 Click Done.

GEOMETRY I

The micromixer is only a few millimeters in size, so change the geometry length unit to millimeters:

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Circle I (c1)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 3.
- 4 Click Build All Objects.

Circle 2 (c2)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 2.75.
- 4 Click Build All Objects.

Difference I (dif1)

- I On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 2 Select the object **cl** only.
- 3 In the Settings window for Difference, locate the Difference section.
- 4 Find the **Objects to subtract** subsection. Select the **Active** toggle button.
- 5 Select the object c2 only.
- 6 Click Build All Objects.

Circle 3 (c3)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 2.75.
- 4 Click Build All Objects.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type **0.2**.
- 4 In the **Height** text field, type 5.25.
- 5 Locate the Position section. From the Base list, choose Center.

6 Click Build All Objects.

Rectangle 2 (r2)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 5.25.
- 4 In the **Height** text field, type 0.2.
- 5 Locate the Position section. From the Base list, choose Center.
- 6 Click Build All Objects.

Rectangle 3 (r3)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the **Height** text field, type 0.5.
- 4 Locate the **Position** section. In the **x** text field, type -3.4.
- 5 From the Base list, choose Center.
- 6 Click Build All Objects.

Rotate | (rot |)

- I On the Geometry toolbar, click Transforms and choose Rotate.
- 2 Select the object **r3** only.
- 3 In the Settings window for Rotate, locate the Rotation Angle section.
- 4 In the Rotation text field, type 90 180 270.
- 5 Locate the Input section. Select the Keep input objects check box.
- 6 Click Build All Objects.

Union I (uni I)

- I On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 2 Select the objects rot1(2), r3, rot1(1), rot1(3), and dif1 only.
- 3 In the Settings window for Union, locate the Union section.
- 4 Clear the Keep interior boundaries check box.
- **5** Click **Build All Objects**.

Difference 2 (dif2)

- I On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 2 Select the object c3 only.

- 3 In the Settings window for Difference, locate the Difference section.
- 4 Find the Objects to subtract subsection. Select the Active toggle button.
- 5 Select the objects rl and r2 only.
- 6 Click Build All Objects.

Form Union (fin)

The **Rotating Machinery, Laminar Flow** interface requires that a pair is present between the stationary and rotating domains. In order to do this, use the **Assembly** option. This will automatically create **Pair** boundaries between the stationary and rotating domains.

- I In the Model Builder window, under Component I (compl)>Geometry I click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, locate the Form Union/Assembly section.
- 3 From the Action list, choose Form an assembly.
- 4 Right-click Component I (compl)>Geometry I>Form Union (fin) and choose Build Selected.
- 5 Click the Zoom Extents button on the Graphics toolbar.

DEFINITIONS

It is usually convenient to define an explicit selection for the pair boundaries.

Explicit I

- I On the Definitions toolbar, click Explicit.
- 2 In the Settings window for Explicit, locate the Input Entities section.
- 3 From the Geometric entity level list, choose Boundary.

4 Select Boundaries 15–18 and 33–36 only.

The easiest way to select these boundaries is to copy the text '15-18, 33-36', click in the selection box, and then press Ctrl+V. Alternatively, click the **Paste Selection** button and type or paste the boundary numbers in the dialog box that appears.



- 5 Right-click Explicit I and choose Rename.
- 6 In the Rename Explicit dialog box, type Pair boundaries in the New label text field.
- 7 Click OK.

Now define a **Ramp** function for the inlet velocity. The boundary condition for the inlet velocity must be consistent with the initial condition for the velocity. The initial velocity in this model will be zero so the inlet velocity must be ramped up from zero to its maximum value over a certain period of time. In this case the ramp time is 0.01 seconds. To achieve this, the **ramp** function is used with a **slope** of 100, meaning that the ramp function reaches its maximum value after 0.01 seconds.

Ramp I (rm I)

- I On the Definitions toolbar, click More Functions and choose Ramp.
- 2 In the Settings window for Ramp, locate the Parameters section.
- **3** In the **Slope** text field, type 100.
- 4 Select the **Cutoff** check box.
- 5 Click to expand the Smoothing section. Select the Smooth at cutoff check box.
- 6 In the Size of transition zone text field, type 0.001.

Now that the ramp function is defined, create an expression for the inlet velocity which will ramp up over 0.01 seconds.

Variables 1

- I On the **Definitions** toolbar, click **Local Variables**.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** In the table, enter the following settings:

Name	Expression	Unit	Description
uin	0.02[m/s]*rm1(t[1/s])	m/s	Inlet velocity

MATERIALS

In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.

Material I (mat1)

I In the Settings window for Material, locate the Material Contents section.

2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1E3	kg/m³	Basic
Dynamic viscosity	mu	1E-3	Pa∙s	Basic

ROTATING MACHINERY, LAMINAR FLOW (RMSPF)

Add a feature which designates the rotating domain. The speed of revolution is also specified, in this case one revolution per unit time. This means the blade system will undergo one complete revolution (360 degrees) per second.

Rotating Domain 1

- I On the Physics toolbar, click Domains and choose Rotating Domain.
- 2 Select Domain 2 only.
- 3 In the Settings window for Rotating Domain, locate the Rotating Domain section.
- **4** In the *f* text field, type 1.
- 5 From the Rotational direction list, choose Counterclockwise.

Inlet I

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundaries 1, 5, and 12 only.

- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the U_0 text field, type uin.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 7 only.

The flow continuity boundary condition is necessary on **Pairs** so that the velocity field in the rotating domain can be matched to the velocity field in the stationary domain.

Flow Continuity I

- I On 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 Pair I (apl).

PARTICLE TRACING FOR FLUID FLOW (FPT)

Wall I

- I In the Model Builder window, expand the Component I (comp1)>Particle Tracing for Fluid Flow (fpt) node, then click Wall I.
- 2 In the Settings window for Wall, locate the Wall Condition section.
- **3** From the **Wall condition** list, choose **Bounce**.

Start by adding the drag force on the particles. This requires input of the fluid velocity and viscosity.

4 In the Model Builder window, click Particle Tracing for Fluid Flow (fpt).

Drag Force 1

- I On the Physics toolbar, click Domains and choose Drag Force.
- 2 In the Settings window for Drag Force, locate the Domain Selection section.
- **3** From the Selection list, choose All domains.
- 4 Locate the Drag Force section. From the u list, choose Velocity field (rmspf/fpl).
- **5** From the μ list, choose **Dynamic viscosity (rmspf/fp1)**.

Much like the flow continuity boundary condition which was added earlier, add a boundary condition for the particles on the pairs which ensures that the particles pass through as if the boundary was invisible.

Particle Continuity I

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

Now define a stream of particles over the first second for each inlet, with 50 particles per inlet and a new release every 50 milliseconds. Defining 3 separate inlet features will allow for improved visualization during results processing.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Initial Position section.
- 4 From the Initial position list, choose Uniform distribution.
- **5** In the N text field, type 50.
- 6 Locate the Initial Velocity section. From the u list, choose Velocity field (rmspf/fpl).
- 7 Locate the Release Times section. Click Range.
- 8 In the Range dialog box, type 0 in the Start text field.
- 9 In the **Stop** text field, type 1.
- **IO** In the **Step** text field, type 0.05.

II Click Replace.

- Inlet 2
- I Right-click Inlet I and choose Duplicate.
- **2** Select Boundary 5 only.
- 3 Right-click Component I (comp1)>Particle Tracing for Fluid Flow (fpt)>Inlet 2 and choose Duplicate.

Inlet 3 Select Boundary 12 only.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 7 only.

Particle Properties 1

- I In the Model Builder window, under Component I (comp1)>Particle Tracing for Fluid Flow (fpt) click Particle Properties 1.
- 2 In the Settings window for Particle Properties, locate the Particle Properties section.
- 3 From the Particle property specification list, choose Specify particle mass and density.

MESH I

A reasonably fine mesh is needed on the interface between the stationary and rotating domains.

Edge 1

- I In the Model Builder window, under Component I (compl) right-click Mesh I and choose More Operations>Edge.
- 2 In the Settings window for Edge, locate the Boundary Selection section.
- 3 From the Selection list, choose All boundaries.

Size I

- I Right-click Component I (compl)>Mesh I>Edge I and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 From the **Predefined** list, choose **Extra fine**.
- 4 In the Model Builder window, right-click Mesh I and choose Free Triangular.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 From the **Predefined** list, choose **Finer**.
- 4 Click Build All.
- 5 Click the **Zoom Extents** button on the **Graphics** toolbar.

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the **Times** text field, type range(0,0.02,2).
- 4 Select the **Relative tolerance** check box.

In order to make the model solve efficiently some of the default solver settings need to be modified. This is because the optimum solver settings for a fluid flow problem are very different to the optimum settings for a particle tracing model. The default absolute tolerances for a particle tracing model are very strict and need to be relaxed; otherwise the fluid flow part of the problem will not solve. For both particles and fluids, updating the Jacobian once per time step ensures that the velocity continuity and the particle continuity across the rotating/non-rotating interface are accurate. Also solve for best performance, solve for the particle tracing variables in the last group.

Solution 1 (sol1)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **3** In the **Settings** window for Time-Dependent Solver, click to expand the **Absolute tolerance** section.
- 4 Locate the Absolute Tolerance section. In the Tolerance text field, type 1E-3.
- 5 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (solI)>Time-Dependent Solver I>Segregated I node, then click Segregated Step I.
- **6** In the **Settings** window for Segregated Step, click to expand the **Method and termination** section.
- 7 Locate the Method and Termination section. From the Jacobian update list, choose Once per time step.
- 8 Right-click Study I>Solver Configurations>Solution I (solI)>Time-Dependent Solver I> Segregated I>Segregated Step I and choose Move Down.
- 9 Right-click Study I>Solver Configurations>Solution I (solI)>Time-Dependent Solver I> Segregated I>Segregated Step I and choose Move Down.
- **IO** On the **Study** toolbar, click **Compute**.

RESULTS

Particle Trajectories (fpt)

The predefined variable fpt.prf can be used to place colors on a particle based on the inlet where it appeared. This allows you to visualize the effect of the mixing between the three inlets.

I Click the Zoom Extents button on the Graphics toolbar.

Particle Trajectories 1

In the Model Builder window, expand the Particle Trajectories (fpt) node.

Color Expression 1

- I In the Model Builder window, expand the Particle Trajectories I node, then click Color Expression I.
- 2 In the Settings window for Color Expression, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Model>Component I>Particle Tracing for Fluid Flow>Particle statistics>fpt.prf Particle release feature.
- 3 On the Particle Trajectories (fpt) toolbar, click Plot.

Hide the pair boundary using the **Hide Geometric Entities** option in the **View** node.

DEFINITIONS

In the Model Builder window, expand the Component I (compl)>Definitions node.

View I

In the Model Builder window, expand the Component I (compl)>Definitions>View I node, then click View I.

Hide for Physics 1

- I On the View I toolbar, click Hide for Physics.
- **2** In the **Settings** window for Hide for Physics, locate the **Geometric Entity Selection** section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 From the Selection list, choose Pair boundaries.

RESULTS

Particle Trajectories (fpt)

You can reproduce the results in Figure 3 by selecting different values for Time. A better way of visualizing the results is to click the **Player** button, in which case the following instructions can be skipped.

- I In the Model Builder window, under Results click Particle Trajectories (fpt).
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Time (s) list, choose 0.2.
- 4 On the Particle Trajectories (fpt) toolbar, click Plot.

Repeat last two steps for the time values 0.4, 0.6, 0.8, 1, and 2 s.



Flow Around an Inclined NACA 0012 Airfoil

Introduction

This example simulates the flow around an inclined NACA 0012 airfoil using the SST turbulence model and compares the results with the experimental lift data of Ladson (Ref. 1) and pressure data of Gregory and O'Reilly (Ref. 2). The SST model combines the near-wall capabilities of the k- ω model with the superior free-stream behavior of the k- ε model to enable accurate simulations of a wide variety of internal and external flow problems. See the theory for the SST turbulence model in the *CFD Module User's Guide* for further information.

Model Definition

Consider the flow relative to a reference frame fixed on a NACA 0012 airfoil with chord-length c=1.8 m. The temperature of the ambient air is 20 °C and the relative free-stream velocity is $U_{\infty}=50$ m/s resulting in a Mach number of 0.15. The Reynolds number based on the chord length is roughly 6×10^6 , so you can assume that the boundary layers are turbulent over practically the entire airfoil. The airfoil is inclined at an angle α to the oncoming stream,

$$(u_{\infty}, v_{\infty}) = U_{\infty}(\cos\alpha, \sin\alpha)$$

To obtain a sharp trailing edge, the airfoil is slightly altered from its original shape (Ref. 3),

$$y = \pm c \cdot 0.594689181 \cdot \left(0.298222773 \cdot \sqrt{\frac{x}{c}} - 0.127125232 \cdot \frac{x}{c} - 0.357907906 \cdot \left(\frac{x}{c}\right)^2 + 0.291984971 \cdot \left(\frac{x}{c}\right)^3 - 0.105174696 \cdot \left(\frac{x}{c}\right)^4 \right)$$

The upstream, top and bottom edges of the computational domain are located 100 chord-lengths away from the trailing edge of the airfoil and the downstream edge is located 200 chord-lengths away. This is to minimize the effect of the applied boundary conditions.



Figure 1 shows the flow domain and the applied far-field boundary conditions,

Figure 1: Flow domain and far-field boundary conditions.

Ref. 4 provides far-field values for the turbulence variables,

$$\omega_{\infty} = (1 \rightarrow 10) \frac{U_{\infty}}{L}, \qquad \frac{v_{T_{\infty}}}{v_{\infty}} = 10^{-(2 \rightarrow 5)}$$

where the free-stream value of the turbulence kinetic energy is given by,

$$k_{\infty} = v_{T_{\infty}}\omega_{\infty}$$

and L is the appropriate length of the computational domain. The current model applies the upper limit of the provided free-stream turbulence values,

$$\omega_{\infty} = 10 \frac{U_{\infty}}{L}, \qquad \qquad k_{\infty} = 0.1 \frac{v_{\infty} U_{\infty}}{L}$$



Figure 2 shows a close-up of the airfoil section. A no slip condition is applied on the surface of the airfoil.

Figure 2: Close-up of the airfoil section.

The computations employ a structured mesh with a high size-ratio between the outermost and wall-adjacent elements.

Results and Discussion

The study performs a Parametric Sweep with the angle of attack α taking the values,

 $\alpha \, = \, 0^{\circ}, 2^{\circ}, 4^{\circ}, 6^{\circ}, 8^{\circ}, 10^{\circ}, 12^{\circ}, 14^{\circ}$

Figure 3 shows the velocity magnitude and the streamlines for the steady flow around the NACA 0012 profile at α =14 °.



Figure 3: Velocity magnitude and streamlines for the flow around a NACA 0012 airfoil.

A small separation bubble appears at the trailing edge for higher values of α and the flow is unlikely to remain steady and two-dimensional hereon. Ref. 1 provides experimental data for the lift coefficient versus the angle of attack,

$$C_L(\alpha) = \oint_c (c_p(s)/c)(n_y(s)\cos(\alpha) - n_x(s)\sin(\alpha)) \, ds$$

where the pressure coefficient is defined as,

$$c_p(s) = \frac{p(s) - p_{\infty}}{\frac{1}{2}\rho_{\infty}U_{\infty}^2}$$

and c is the chord length. Note that the normal is directed outwards from the flow domain (into the airfoil). Figure 4 shows computational and experimental results for the lift coefficient versus angle of attack.



Figure 4: Computational (solid) and experimental (dots) results for the lift coefficient vs. angle of attack.

No discernible discrepancy between the computational and experimental results occurs within the range of α values used in the computations. The experimental results continue through the parameter regime where the airfoil stalls. Figure 5 shows a comparison between the computed pressure coefficient at α =10 ° and the experimental results in Ref.



Figure 5: Computational (solid) and experimental (dots) results for the pressure coefficient along the airfoil.

Experimental data is only available on the low-pressure side of the airfoil. The agreement between the computational and experimental results is very good.

Notes About the COMSOL Implementation

The model uses the SST turbulence model together with a Parametric Sweep for the angle of attack to compute the different flows on a Mapped Mesh.

References

1. C.L. Ladson, "Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section," *NASA TM* 4074, 1988.

2. N. Gregory and C. L. O'Reilly, "Low-Speed Aerodynamic Characteristics of NACA 0012 Aerofoil Section, including the Effects of Upper-Surface Roughness Simulating Hoar Frost," *A.R.C.*, R. & M. no. 3726, 1970.

3. NASA Langley Research Centre, Turbulence Modeling Resource, "2D NACA 0012 Airfoil Validation Case," http://turbmodels.larc.nasa.gov/naca0012_val.html

4. F.R. Menter, "Two-Equation Eddy-Viscosity Models for Engineering Applications," *AIAA Journal*, vol. 32, no. 8, pp. 1598–1605, 1994.

Application Library path: CFD_Module/Single-Phase_Benchmarks/ naca0012_airfoil

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, SST (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary with Initialization.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
U_inf	50[m*s^-1]	50 m/s	Free-stream velocity
rho_inf	1.2043[kg*m^-3]	1.204 kg/m³	Free-stream density

Name	Expression	Value	Description
mu_inf	1.81397e-5[kg* m^-1*s^-1]	1.814E-5 kg/(m·s)	Free-stream dynamic viscosity
L	180[m]	180 m	Domain reference length
С	1.8[m]	I.8 m	Chord length
k_inf	0.1*mu_inf*U_inf/ (rho_inf*L)	4.184E-7 m ² /s ²	Free-stream turbulent kinetic energy
om_inf	10*U_inf/L	2.778 I/s	Free-stream specific dissipation rate
alpha	0	0	Angle of attack

GEOMETRY I

Circle I (cl)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type L.
- 4 In the Sector angle text field, type 90.
- 5 Locate the Rotation Angle section. In the Rotation text field, type 90.

Parametric Curve 1 (pc1)

- I On the Geometry toolbar, click Primitives and choose Parametric Curve.
- 2 In the Settings window for Parametric Curve, locate the Expressions section.
- **3** In the **x** text field, type c*s.
- 4 In the y text field, type c*0.594689181*(0.298222773*sqrt(s)-0.127125232* s-0.357907906*s^2+0.291984971*s^3-0.105174696*s^4).
- 5 Locate the **Position** section. In the **x** text field, type -c.

Bézier Polygon I (b1)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the Control points subsection. In row I, set x to -c.

Union I (uni I)

- I Right-click Bézier Polygon I (b1) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Union.
- **3** Select the objects **b1** and **pc1** only.

Convert to Solid I (csoll)

- I Right-click Union I (unil) and choose Build Selected.
- 2 On the Geometry toolbar, click Conversions and choose Convert to Solid.
- 3 Select the object unil only.

Difference I (dif I)

- I Right-click Convert to Solid I (csoll) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- **3** Select the object **cl** only.
- 4 In the Settings window for Difference, locate the Difference section.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the object csoll only.

Rectangle 1 (r1)

- I Right-click Difference I (difl) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Rectangle.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- **4** In the **Width** text field, type 2*L.
- **5** In the **Height** text field, type L.
- 6 Right-click Rectangle I (rI) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Mirror I (mir I)

- I On the Geometry toolbar, click Transforms and choose Mirror.
- 2 In the Settings window for Mirror, locate the Input section.
- **3** Select the **Keep input objects** check box.
- 4 Click in the Graphics window and then press Ctrl+A to select both objects.
- 5 Locate the Normal Vector to Line of Reflection section. In the x text field, type 0.
- 6 In the y text field, type 1.
- 7 Right-click Mirror I (mirl) and choose Build Selected.

8 Click the Zoom Extents button on the Graphics toolbar.

Mesh Control Edges 1 (mcel)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Edges.
- 2 On the object fin, select Boundaries 1, 2, 4, and 5 only.
- 3 On the Geometry toolbar, click Build All.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Liquids and Gases>Gases>Air.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

TURBULENT FLOW, SST (SPF)

Since the density variation is not small, the flow can not be regarded as incompressible. Therefore set the flow to be compressible.

- I In the Model Builder window, expand the Component I (compl)>Turbulent Flow, SST (spf) node, then click Turbulent Flow, SST (spf).
- 2 In the Settings window for Turbulent Flow, SST, locate the Physical Model section.
- **3** From the **Compressibility** list, choose **Compressible flow (Ma<0.3)**.

Fluid Properties 1

- I In the Model Builder window, under Component I (compl)>Turbulent Flow, SST (spf) click Fluid Properties I.
- 2 In the Settings window for Fluid Properties, locate the Distance Equation section.
- **3** From the l_{ref} list, choose Manual.
- 4 In the associated text field, type 0.2.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- 4 Click the Specify turbulence variables button.
- **5** In the k_0 text field, type k_inf.
- **6** In the ω_0 text field, type om_inf.

- 7 Locate the Velocity section. Click the Velocity field button.
- **8** Specify the \mathbf{u}_0 vector as

U_inf*cos(alpha*pi/180)	x
U_inf*sin(alpha*pi/180)	у

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Turbulent Flow, SST (spf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** Specify the **u** vector as

U_inf*cos(alpha*pi/180)	x
U_inf*sin(alpha*pi/180)	у

Open Boundary I

- I On the Physics toolbar, click Boundaries and choose Open Boundary.
- **2** Select Boundary 2 only.

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Mapped I

- I In the Settings window for Mapped, locate the Domain Selection section.
- 2 From the Geometric entity level list, choose Domain.
- **3** Select Domain 3 only.
- **4** Click to expand the **Control entities** section. Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution I

- I Right-click Component I (compl)>Mesh I>Mapped I and choose Distribution.
- 2 Select Boundaries 2 and 11 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- 4 From the Distribution properties list, choose Predefined distribution type.
- 5 In the Number of elements text field, type 100.
- 6 In the Element ratio text field, type 15000000.

- 7 From the Distribution method list, choose Geometric sequence.
- 8 Select the Reverse direction check box.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 2

- I Select Boundary 7 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 50.
- 5 In the Element ratio text field, type 25.
- **6** From the **Distribution method** list, choose **Geometric sequence**.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 3

- I Select Boundary 12 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 50.
- 5 In the **Element ratio** text field, type 480000.
- 6 From the Distribution method list, choose Geometric sequence.
- 7 Select the **Reverse direction** check box.

Mapped 2

- I In the Model Builder window, right-click Mesh I and choose Mapped.
- 2 In the Settings window for Mapped, locate the Domain Selection section.
- **3** From the Geometric entity level list, choose Domain.
- **4** Select Domains 1 and 4 only.
- **5** Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution I

- I Right-click Component I (compl)>Mesh I>Mapped 2 and choose Distribution.
- **2** Select Boundaries 9–11 only.

- 3 In the Settings window for Distribution, locate the Distribution section.
- **4** From the **Distribution properties** list, choose **Predefined distribution type**.
- **5** In the **Number of elements** text field, type 100.
- 6 In the **Element ratio** text field, type 15000000.
- 7 From the Distribution method list, choose Geometric sequence.

Mapped 2

Right-click Mapped 2 and choose Distribution.

Distribution 2

- I Select Boundaries 3 and 4 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 256.
- 5 In the **Element ratio** text field, type 256.
- 6 From the Distribution method list, choose Geometric sequence.
- 7 Select the Symmetric distribution check box.

Mapped 3

- I In the Model Builder window, right-click Mesh I and choose Mapped.
- 2 In the Settings window for Mapped, locate the Domain Selection section.
- **3** From the Geometric entity level list, choose Domain.
- **4** Select Domain 2 only.
- **5** Locate the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

Distribution I

- I Right-click Component I (comp1)>Mesh 1>Mapped 3 and choose Distribution.
- **2** Select Boundaries 8 and 10 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- **4** From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the Number of elements text field, type 100.
- 6 In the **Element ratio** text field, type 15000000.
- 7 From the Distribution method list, choose Geometric sequence.

Mapped 3

Right-click Mapped 3 and choose Distribution.

Distribution 2

- I Select Boundary 1 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 50.
- 5 In the Element ratio text field, type 25.
- **6** From the **Distribution method** list, choose **Geometric sequence**.
- 7 Select the **Reverse direction** check box.
- 8 In the Model Builder window, click Mesh I.
- 9 In the Settings window for Mesh, click Build All.

STUDY I

Step 2: Stationary

- I In the Model Builder window, expand the Study I node, then click Step 2: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study extensions section.
- 3 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 4 Click Add.
- **5** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alpha	0,2,4,6,8,10,12,14	

Velocity (spf)

On the Study toolbar, click Get Initial Value.

RESULTS

Velocity (spf)

Set up a velocity-magnitude and streamline plot for the region close to the airfoil and display it during the computations

- I In the Model Builder window, under Results 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 **Start point controlled**.
- 4 From the Entry method list, choose Coordinates.
- **5** In the **x** text field, type **0**.
- **6** In the **y** text field, type range(-2,0.025,2).
- 7 In the Model Builder window, click Velocity (spf).
- 8 In the Settings window for 2D Plot Group, locate the Plot Settings section.
- 9 From the View list, choose View I.

DEFINITIONS

In the Model Builder window, expand the Component I (compl)>Definitions node.

Axis

- I In the Model Builder window, expand the Component I (comp1)>Definitions>View I node, then click Axis.
- 2 In the Settings window for Axis, locate the Axis section.
- **3** In the **x minimum** text field, type -2.2.
- 4 In the **x maximum** text field, type 0.6.
- **5** In the **y minimum** text field, type -1.
- 6 In the **y maximum** text field, type 1.
- 7 Click Update.

View 1

- I In the Model Builder window, under Component I (compl)>Definitions click View I.
- 2 In the Settings window for View, locate the View section.
- **3** Select the **Lock axis** check box.

STUDY I

In the Model Builder window, expand the Study I>Solver Configurations node.

Solution I (soll)

- I In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll)>Stationary Solver 2 node, then click Segregated I.
- 2 In the Settings window for Segregated, click to expand the Results while solving section.
- 3 Locate the Results While Solving section. Select the Plot check box.
- 4 On the Study toolbar, click Compute.

RESULTS

Line Integration 1

On the Results toolbar, click More Derived Values and choose Integration>Line Integration.

Derived Values

- I Select Boundaries 3 and 4 only.
- 2 In the Settings window for Line Integration, locate the Expressions section.
- **3** In the table, enter the following settings:

Expression	Unit	Description
<pre>p/(1/2*rho_inf*U_inf^2)/c*(spf.nymesh* cos(alpha*pi/180)-spf.nxmesh*sin(alpha*pi/ 180))</pre>	1	

4 Click Evaluate.

TABLE

- I Go to the Table window.
- 2 Click Table Graph in the window toolbar.

RESULTS

Table 2

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- **4** Browse to the application's Application Library folder and double-click the file naca0012_airfoil_Ladson_CL.dat.

ID Plot Group 4

In the Model Builder window, under Results click ID Plot Group 4.

Table Graph 2

On the ID Plot Group 4 toolbar, click Table Graph.

ID Plot Group 4

- I In the Settings window for Table Graph, locate the Data section.
- 2 From the Table list, choose Table 2.
- **3** Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

- **4** From the **Color** list, choose **Blue**.
- 5 Find the Line markers subsection. From the Marker list, choose Point.
- 6 From the Positioning list, choose In data points.
- 7 In the Model Builder window, click ID Plot Group 4.
- 8 In the Settings window for 1D Plot Group, click to expand the Title section.
- 9 From the Title type list, choose Manual.
- **IO** In the **Title** text area, type Lift vs. angle of attack.
- II Locate the Plot Settings section. Select the x-axis label check box.
- **12** In the associated text field, type \alpha.
- **I3** Select the **y-axis label** check box.
- 14 In the associated text field, type CL.
- 15 On the 1D Plot Group 4 toolbar, click Plot.

Table 3

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- **4** Browse to the application's Application Library folder and double-click the file naca0012_airfoil_Gregory_OReilly_Cp.dat.

TABLE

- I Go to the Table window.
- 2 Click Table Graph in the window toolbar.

RESULTS

ID Plot Group 5

- I In the Model Builder window, under Results>ID Plot Group 5 click Table Graph I.
- 2 In the Settings window for Table Graph, locate the Coloring and Style section.
- **3** Find the Line style subsection. From the Line list, choose None.
- 4 From the **Color** list, choose **Blue**.
- 5 Find the Line markers subsection. From the Marker list, choose Point.
- 6 From the Positioning list, choose In data points.
- 7 In the Model Builder window, click ID Plot Group 5.

Line Graph I

On the ID Plot Group 5 toolbar, click Line Graph.

ID Plot Group 5

- I In the Settings window for Line Graph, locate the Data section.
- 2 From the Data set list, choose Study I/Solution I (soll).
- **3** From the **Parameter selection (alpha)** list, choose **Manual**.
- 4 In the Parameter values (alpha) (1-8) text field, type 6.
- 5 Select Boundaries 3 and 4 only.
- 6 Locate the y-Axis Data section. In the Expression text field, type -p/(1/2*rho_inf* U_inf^2).
- 7 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 8 In the **Expression** text field, type (x+c)/c.
- 9 Click to expand the Coloring and style section. Locate the Coloring and Style section. From the Color list, choose Blue.
- IO In the Model Builder window, click ID Plot Group 5.
- II In the Settings window for 1D Plot Group, locate the Title section.
- **12** From the **Title type** list, choose **None**.
- **I3** Locate the **Plot Settings** section. Select the **x-axis label** check box.
- **I4** In the associated text field, type (x-xLE)/c.
- **I5** Select the **y-axis label** check box.
- **I6** In the associated text field, type cp.
- 17 On the 1D Plot Group 5 toolbar, click Plot.



Non-Newtonian Flow

Introduction

This example shows the influence of shear rate dependent viscosity on the flow of a linear polystyrene solution. For this type of flow, you can use the Carreau viscosity model. Due to rotational symmetry, it is possible to reduce the model dimensions from 3D to axisymmetric 2D (see Figure 1).

Model Definition

For non-Newtonian flow, μ denotes the viscosity (SI unit: kg/(m·s)), **u** the velocity (SI unit: m/s), ρ the density of the fluid (SI unit: kg/m³) and *p* the pressure (SI unit: Pa). The equations to solve are the momentum and continuity equations.

$$\rho \frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = 0$$
$$\nabla \cdot \mathbf{u} = 0$$

In the Carreau model, the viscosity depends on the shear rate, $\dot{\gamma}$, which for an axisymmetric model in cylindrical coordinates is defined according to Equation 1:

$$\dot{\gamma} = \sqrt{\frac{1}{2} \left(\left(2u_r \right)^2 + 2\left(u_z + v_r \right)^2 + \left(2v_z \right)^2 + 4\left(\frac{u}{r}\right)^2 \right)}$$
(1)

The viscosity is given by

$$\mu \,=\, \mu_\infty + (\mu_0 - \mu_\infty) [1 + (\lambda \dot{\gamma})^2]^{\frac{(n-1)}{2}}$$

where μ_{∞} is the infinite shear rate viscosity, μ_0 is the zero shear rate viscosity, λ is a parameter with units of time, and *n* is a dimensionless parameter. A solution of linear polystyrene in 1-chloronaphthalene has the properties listed in Table 1 (Ref. 1).

TABLE I: PROPERTIES OF A SOLUTION OF LINEAR POLYSTYRENE IN I-CHLORONAPHTALENE.

PARAMETER	VALUE
μ_{∞}	0
μ ₀	166 Pa·s
λ	1.73·10 ⁻² s
n	0.538
ρ	450 kg/m ³
The model domain is shown in Figure 1.



Figure 1: Model domain. The geometry can be simplified assuming axisymmetry.

The boundary conditions at the inlet and the outlet are set to fixed pressures and vanishing viscous stresses:

$$p = p_{in}$$

 $p = 0$

and

$$\mathbf{n} \cdot [\mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] = 0$$

To study the effect on viscosity at different inlet pressures, the model makes use of the parametric solver to vary p_{in} from 10 kPa to 210 kPa. The axis of rotation requires the axial symmetry condition:

 $\mathbf{u} \cdot \mathbf{n} = 0$

while all other boundaries impose the no slip condition:

 $\mathbf{u} = \mathbf{0}$

Results and Discussion

Figure 2 shows that the velocity distribution is more pronounced at the inlet compared to the outlet. This is because the cross section is greater at the outlet. The figure also shows

that the region with greatest velocity gradient is in the contraction, which means that the shear rate is largest there.



Figure 2: Velocity field in the modeling domain.

Because the fluid is shear thinning, the viscosity depends on the shear rate and is shown in Figure 3. It reaches its lowest value close to the wall in the contraction between the piston and the wall.



Figure 3: Viscosity distribution in the domain. The lowest viscosity occurs at the wall in the contraction region.

Showing the result of a parametric study of the inlet pressure, Figure 4 contains a cross-sectional plot of the viscosity across the contraction. Sweeping through a range of inlet pressures imposes greater velocities on the non-Newtonian fluid. As the velocity increases, the shear rate also increases and the viscosity decreases. An optimal condition is to have as flat a viscosity profile as possible. This is hindered by also wanting to put through as high a flow rate as possible.



Figure 4: Parametric study of the process, sweeping the inlet pressure from 10 kPa to 210 kPa, while investigating a cross-sectional viscosity plot. A greater inlet pressure (and pressure differential) results in lower values for the viscosity and greater variations in its distribution through the cross section.

Reference

1. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, John Wiley & Sons, 1960.

Application Library path: CFD_Module/Single-Phase_Tutorials/ non_newtonian_flow

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Laminar Flow (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary.
- 6 Click Done.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 9.
- 4 In the **Height** text field, type 21.

Rectangle 2 (r2)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Rectangle.
- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the Width text field, type 12.
- 5 In the Height text field, type 27.
- 6 Locate the **Position** section. In the **r** text field, type 6.
- 7 In the z text field, type -21.

Bézier Polygon I (b1)

- I Right-click Rectangle 2 (r2) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.

- 4 Find the Added segments subsection. Click Add Quadratic.
- 5 Find the Control points subsection. In row 1, set z to 3.
- 6 In row 2, set r to 12 and z to 3.
- 7 In row 3, set r to 12 and z to -9.
- 8 Find the Added segments subsection. Click Add Quadratic.
- 9 Find the Control points subsection. In row 2, set r to 6.
- **10** In row **3**, set **r** to **6** and **z** to **-15**.
- II Find the Added segments subsection. Click Add Linear.
- 12 Find the Control points subsection. In row 2, set r to 0.
- **I3** Find the **Added segments** subsection. Click **Add Linear**.
- 14 Find the Control points subsection. Click Close Curve.
- **I5** Right-click **Bézier Polygon I (bI)** and choose **Build Selected**.
- **I6** Click the **Zoom Extents** button on the **Graphics** toolbar.

Bézier Polygon 2 (b2)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Quadratic.
- 4 Find the **Control points** subsection. In row 1, set r to 9 and z to 6.
- **5** In row **2**, set **r** to **9** and **z** to **3**.
- 6 In row 3, set r to 12 and z to 3.
- 7 Find the Added segments subsection. Click Add Quadratic.
- 8 Find the Control points subsection. In row 2, set r to 18.
- **9** In row **3**, set **r** to **18** and **z** to **-3**.
- **IO** Find the **Added segments** subsection. Click **Add Linear**.
- II Find the **Control points** subsection. In row **2**, set **z** to **6**.
- 12 Find the Added segments subsection. Click Add Linear.
- **I3** Find the **Control points** subsection. Click **Close Curve**.
- 14 Right-click Bézier Polygon 2 (b2) and choose Build Selected.
- **I5** Click the **Zoom Extents** button on the **Graphics** toolbar.

Compose I (col)

I On the Geometry toolbar, click Booleans and Partitions and choose Compose.

- 2 Click in the Graphics window and then press Ctrl+A to select all objects.
- 3 In the Settings window for Compose, locate the Compose section.
- 4 In the Set formula text field, type r1+r2-b1-b2.
- **5** Clear the **Keep interior boundaries** check box.
- 6 Right-click Compose I (col) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union (fin)

In the Model Builder window, under Component I (compl)>Geometry I right-click Form Union (fin) and choose Build Selected.

GLOBAL DEFINITIONS

Add a global parameter for the inlet pressure. You will use this as the parameter in a parametric sweep.

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
p_in	10[kPa]	10000 Pa	Inlet pressure

LAMINAR FLOW (SPF)

Fluid Properties 1

- I In the Model Builder window, expand the Component I (comp1)>Laminar Flow (spf) node, then click Fluid Properties 1.
- 2 In the Settings window for Fluid Properties, locate the Fluid Properties section.
- **3** From the ρ list, choose **User defined**. In the associated text field, type **450**.
- 4 From the μ list, choose Non-Newtonian Carreau model.
- **5** In the μ_0 text field, type 166.
- 6 In the λ text field, type 1.73e-2.
- **7** In the *n* text field, type **0.538**.

Inlet 1

I On the Physics toolbar, click Boundaries and choose Inlet.

- **2** Select Boundary 2 only.
- 3 In the Settings window for Inlet, locate the Boundary Condition section.
- **4** From the list, choose **Pressure**.
- **5** Locate the **Pressure Conditions** section. In the *p*₀ text field, type **p_in**.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 4 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- **3** From the **Element size** list, choose **Extra fine**.
- 4 Click Build All.

STUDY I

Parametric Sweep

- I On the Study toolbar, click Parametric Sweep.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click Add.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_in	range(10000,40000,210000)	

5 On the Study toolbar, click Compute.

RESULTS

Velocity (spf)

The default plot groups shows the velocity magnitude in 2D (compare with Figure 2) and 3D as well as the pressure field in 2D.

Visualize the viscosity distribution in a separate plot group with the following steps:

2D Plot Group 4

- I On the Home toolbar, click Add Plot Group and choose 2D Plot Group.
- 2 In the Model Builder window, right-click 2D Plot Group 4 and choose Surface.

- 3 On the 2D Plot Group 4 toolbar, click Plot.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 5 In the Model Builder window, under Results>2D Plot Group 4 click Surface 1.
- 6 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Laminar Flow>Material properties>spf.mu - Dynamic viscosity.
- 7 On the 2D Plot Group 4 toolbar, click Plot.

Compare the resulting plot with that in Figure 3.

8 In the Model Builder window, right-click 2D Plot Group 4 and choose Rename.

9 In the Rename 2D Plot Group dialog box, type Viscosity in the New label text field.

IO Click OK.

Finally, plot the viscosity across the contraction as in Figure 4.

Cut Line 2D I

On the Results toolbar, click Cut Line 2D.

Data Sets

- I In the Settings window for Cut Line 2D, locate the Line Data section.
- 2 In row **Point I**, set **r** to 7.55 and **z** to 0.32.
- **3** In row **Point 2**, set **r** to **9.97** and **z** to **3.79**.
- 4 Click Plot.



ID Plot Group 5

On the **Results** toolbar, click **ID Plot Group**.

Line Graph 1

On the **ID Plot Group 5** toolbar, click **Line Graph**.

ID Plot Group 5

- I In the Settings window for Line Graph, locate the Data section.
- 2 From the Data set list, choose Cut Line 2D I.
- 3 Click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component I>Laminar Flow>Material properties>spf.mu - Dynamic viscosity.
- 4 Click to expand the Quality section. From the Recover list, choose Within domains.
- 5 On the ID Plot Group 5 toolbar, click Plot.



Rising Bubble

Introduction

This example shows how to model two immiscible fluids, tracking the fluid-fluid interface. An oil bubble rises through water and merges with oil already residing at the top of the container. Initially three different regions exist: the initially still oil bubble, the oil at the top of the container, and the water surrounding the bubble (see Figure 1). The container is cylindrical with a diameter of $1 \cdot 10^{-2}$ m and a height of 1.5×10^{-2} m. The oil phase has a viscosity of 0.0208 Pa·s and a density of 879 kg/m³. For water the viscosity is 1.01×10^{-3} Pa·s and the density is 1000 kg/m^3 . Buoyancy effects cause the oil bubble to rise through the water phase. As the bubble reaches the liquid-liquid interface, it merges with the oil phase



Figure 1: Initial bubble position. The geometry is axisymmetric.

As outlined above, the topology of the fluid interface changes with time. You start with three separate fluid regions and end up with two. The level set method as well as the phase field method are both well suited for modeling moving boundaries where topology changes occur. Both methods are available in the CFD Module as predefined physics interfaces. This example shows you how to use the Laminar Two-Phase Flow, Level Set interface.

Model Definition

REPRESENTATION AND CONVECTION OF THE FLUID INTERFACE

The Level Set interface finds the fluid interface by tracing the isolines of the level set function, ϕ . The level set or isocontour $\phi = 0.5$ determines the position of the interface. The equation governing the transport and reinitialization of ϕ is

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = \gamma \nabla \cdot \left(\epsilon \nabla \phi - \phi (1 - \phi) \frac{\nabla \phi}{|\nabla \phi|} \right)$$

where **u** (SI unit: m/s) is the fluid velocity, and γ (SI unit: m/s) and ε (SI unit: m) are reinitialization parameters. The ε parameter determines the thickness of the layer around the interface where ϕ goes from zero to one. When stabilization is used for the level set equation, you can typically use an interface thickness of $\varepsilon = h_c/2$, where h_c is the characteristic mesh size in the region passed by the interface. The γ parameter determines the amount of reinitialization. A suitable value for γ is the maximum velocity magnitude occurring in the model.

Because the level set function is a smooth step function, it is also used to determine the density and dynamic viscosity globally by

$$\rho = \rho_{w} + (\rho_{o} - \rho_{w})\phi$$

and

$$\mu = \mu_{w} + (\mu_{o} - \mu_{w})\phi,$$

Here ρ_w , μ_w , ρ_o , and μ_o denote the constant density and viscosity of water and oil, respectively.

MASS AND MOMENTUM TRANSPORT

In the Laminar Two-Phase Flow, Level Set interface, the transport of mass and momentum is governed by the incompressible Navier-Stokes equations, including surface tension:

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot \mu (\nabla \mathbf{u} + \nabla \mathbf{u}^{T}) + \rho \mathbf{g} + \mathbf{F}_{st}$$
$$\nabla \cdot \mathbf{u} = 0$$

In the above equations, ρ (SI unit: kg/m³) denotes the density, **u** is the velocity (SI unit: m/s), *t* equals time (SI unit: s), *p* is the pressure (SI unit: Pa), and μ denotes the viscosity (SI unit: Pa·s). The momentum equations contain gravity, ρ **g**, and surface tension force components, denoted by **F**_{st}.

Surface Tension

The surface tension force is defined by

$$\mathbf{F}_{st} = \nabla \cdot \mathbf{T} = \nabla \cdot [\sigma \{\mathbf{I} + (-\mathbf{nn}^T)\} \delta]$$

where σ is the surface tension coefficient, **I** is the identity matrix, **n** is the interface unit normal, and δ is a Dirac delta function, nonzero only at the fluid interface. The interface normal is calculated from

$$\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|}$$

The level set parameter ϕ is also used to approximate the delta function by a smooth function defined by

$$\delta = 6|\phi(1-\phi)||\nabla\phi|$$

INITIAL CONDITION

At t = 0, the velocity is zero. Figure 2 shows the initial level set function. This is automatically computed using a Phase Initialization study step by solving for the geometrical distance to the initial interface, D_{wi} . The initialized level set function is then defined from the analytical steady state solution for a straight fluid-fluid interface:

$$\phi_{1,0} = \frac{1}{1 + e^{D_{wi}/\varepsilon}}, \quad \phi_{2,0} = \frac{1}{1 + e^{-D_{wi}/\varepsilon}},$$

in the domains initially filled with Fluid 1 and Fluid 2 respectively,



Figure 2: A surface and contour plot of the initialized level set function.

BOUNDARY CONDITIONS

Use no slip conditions, $\mathbf{u} = 0$ at the top and bottom and a wetted wall condition on the right boundary. The left boundary corresponds to the symmetry axis.

Results and Discussion

Figure 3 and Figure 4 contain snapshots of the fluid interface. The snapshots show how the bubble travels up through the water and merges with the oil above. As the bubble rises, its shape remains spherical due to the surface tension and the high viscosity of the oil. As the droplet hits the water surface, it merges with the oil above and creates waves on the surface.



Figure 3: Snapshots showing the interface prior to and just after the bubble hits the surface.



Figure 4: Snapshots showing the interface after the bubble has merged with the oil above.

One way to investigate the quality of the numerical results is to check the conservation of mass. Because there are no reactions and no flow through the boundaries, the total mass of each fluid should be constant in time. Figure 5 shows the total mass of oil as a function



of time. The mass loss during simulation is less than 0.2%, showing that the model conserve mass well.

Figure 5: Total mass of oil as a function of time. The total mass loss during the simulation is very small, less than 0.2%.

Notes About the COMSOL Implementation

The model is straightforward to set up and solve using either the Laminar Two-Phase Flow, Level Set interface. Automatically, two study steps are created. The first one initializes the level set function, and the second one calculates the dynamic two phase flow problem.

Application Library path: CFD_Module/Multiphase_Tutorials/ rising_bubble_2daxi

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Two-Phase Flow, Level Set> Laminar Two-Phase Flow, Level Set.
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies for Selected Physics Interfaces>Transient with Phase Initialization.
- 6 Click Done.

GEOMETRY I

- I In the Model Builder window, under Component I (compl) click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose mm.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 5.
- 4 In the **Height** text field, type 15.

Polygon I (poll)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Polygon.
- 3 In the Settings window for Polygon, locate the Coordinates section.
- 4 In the r text field, type 0 5.
- **5** In the **z** text field, type **10**.

Circle I (c1)

- I Right-click Polygon I (poll) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Circle.
- 3 In the Settings window for Circle, locate the Size and Shape section.
- 4 In the Radius text field, type 2.

- 5 In the Sector angle text field, type 180.
- 6 Locate the **Position** section. In the **z** text field, type 4.
- 7 Locate the Rotation Angle section. In the Rotation text field, type -90.
- 8 Right-click Circle I (cl) and choose Build Selected.

Form Union (fin)

In the Model Builder window, under Component I (compl)>Geometry I right-click Form Union (fin) and choose Build Selected.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Liquids and Gases>Liquids>Transformer oil.
- 4 Click Add to Component in the window toolbar.

ADD MATERIAL

- I Go to the Add Material window.
- 2 In the tree, select Liquids and Gases>Liquids>Water.
- 3 Click Add to Component in the window toolbar.
- 4 On the Home toolbar, click Add Material to close the Add Material window.

You can leave the **Geometric Entity Selection** empty at this stage; it will be defined when you use this material in the Fluid Properties feature.

LEVEL SET (LS)

In the Model Builder window, under Component I (compl) click Level Set (Is).

Initial Interface 1

- I On the Physics toolbar, click Boundaries and choose Initial Interface.
- 2 Select Boundaries 7, 11, and 12 only.

MULTIPHYSICS

- I In the Model Builder window, under Component I (compl)>Multiphysics click Two-Phase Flow, Level Set I (tpfl).
- **2** In the **Settings** window for Two-Phase Flow, Level Set, locate the **Fluid I Properties** section.
- 3 From the Fluid I list, choose Transformer oil (mat I).

- 4 Locate the Fluid 2 Properties section. From the Fluid 2 list, choose Water (mat2).
- 5 Locate the Surface Tension section. From the Surface tension coefficient list, choose Library coefficient, liquid/liquid interface.
- 6 Select Olive oil/Water, 20 C^o from the list.

LEVEL SET (LS)

Level Set Model I

- I In the Model Builder window, under Component I (compl)>Level Set (Is) click Level Set Model I.
- 2 In the Settings window for Level Set Model, locate the Level Set Model section.
- **3** In the γ text field, type 0.1.

Initial Values 2

- I On the Physics toolbar, click Domains and choose Initial Values.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the **Domain initially** list, choose **Fluid 2** ($\phi = I$).
- 4 Select Domain 1 only.

Initial Values 1

- I In the Model Builder window, under Component I (comp1)>Level Set (Is) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** From the **Domain initially** list, choose **Fluid I** ($\phi = 0$).

LAMINAR FLOW (SPF)

On the Physics toolbar, click Level Set (Is) and choose Laminar Flow (spf).

In the Model Builder window, under Component I (compl) click Laminar Flow (spf).

Volume Force 1

- I On the Physics toolbar, click Domains and choose Volume Force.
- 2 In the Model Builder window, right-click Volume Force I and choose Rename.
- 3 In the Rename Volume Force dialog box, type Gravity in the New label text field.
- 4 Click OK.
- 5 In the Settings window for Volume Force, locate the Domain Selection section.
- 6 From the Selection list, choose All domains.

7 Locate the Volume Force section. Specify the F vector as

0 r -g_const*spf.rho z

Pressure Point Constraint I

- I On the Physics toolbar, click Points and choose Pressure Point Constraint.
- 2 Select Point 7 only.

MULTIPHYSICS

Wetted Wall I (ww1)

- I On the Physics toolbar, click Multiphysics and choose Boundary>Wetted Wall.
- 2 Select Boundaries 9 and 10 only.

DEFINITIONS

Before creating the mesh, add a variable for computing the mass of oil in the model domain. You will use this variable later to test mass conservation.

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
rho_oil	tpf1.rho1*tpf1.Vf1	kg/m³	Oil mass per unit volume

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Free Triangular.

Size

- I In the Settings window for Size, locate the Element Size section.
- 2 Click the **Custom** button.
- **3** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type 0.2.
- 4 Click Build All.

STUDY I

Step 2: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step 2: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range (0, 0.5/50, 0.5).

Solution I (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll)>Dependent Variables 2 node, then click Velocity field (compl.u).
- 4 In the Settings window for Field, locate the Scaling section.
- 5 From the Method list, choose Manual.
- 6 In the Scale text field, type 0.01.
- 7 In the Model Builder window, under Study I>Solver Configurations>Solution I (soll)> Dependent Variables 2 click Pressure (compl.p).
- 8 In the Settings window for Field, locate the Scaling section.
- 9 From the Method list, choose Manual.
- **IO** In the **Scale** text field, type 100.
- II In the Model Builder window, click Solution I (soll).
- **12** In the **Settings** window for Solution, click **Compute**.

RESULTS

Next, test to what degree the total mass of oil is conserved.

Surface Integration 1

On the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

Derived Values

- I In the Settings window for Surface Integration, locate the Selection section.
- 2 From the Selection list, choose All domains.
- 3 Click Replace Expression in the upper-right corner of the Expressions section. From the menu, choose Component I>Definitions>Variables>rho_oil Oil mass per unit volume.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
rho_oil	g	Oil mass per unit volume

5 Locate the Integration Settings section. Select the Compute volume integral check box.

6 Click Evaluate.

TABLE

- I Go to the Table window.
- 2 Click **Table Graph** in the window toolbar.

RESULTS

ID Plot Group 6

Compare the result to that in Figure 5. As the plot shows, mass is conserved to within 0.1% accuracy.

Volume Fraction of Fluid 1 (ls)

- I In the Model Builder window, under Results click Volume Fraction of Fluid I (Is).
- 2 In the Settings window for 2D Plot Group, locate the Plot Settings section.
- 3 Clear the Plot data set edges check box.
- 4 In the Model Builder window, expand the Volume Fraction of Fluid I (Is) node, then click Volume Fraction of Fluid I.
- 5 In the Settings window for Surface, locate the Coloring and Style section.
- 6 From the Color table list, choose WaveLight.

To reproduce the plots of reproduced in Figure 2 and Figure 3, plot the solution for the time values 0 0.08 0.12, 0.16, 0.20, 0.26, and 0.32.

- 7 In the Model Builder window, click Volume Fraction of Fluid I (Is).
- 8 In the Settings window for 2D Plot Group, locate the Data section.
- 9 From the Time (s) list, choose 0.
- 10 On the Volume Fraction of Fluid 1 (Is) toolbar, click Plot.

Repeat last two steps for the time values 0.08 0.12, 0.16, 0.20, 0.26, and 0.32 s.

Volume Fraction of Fluid 1 (Is) 1

Add a slice plot of the velocity magnitude to the axisymmetric model revolved into 3D

I In the Model Builder window, expand the Results>Velocity (spf) I node.

- 2 Right-click Results>Volume Fraction of Fluid I (Is) I and choose Slice.
- 3 In the Settings window for Slice, locate the Plane Data section.
- 4 From the Plane list, choose zx-planes.
- **5** In the **Planes** text field, type **1**.
- 6 In the Model Builder window, click Volume Fraction of Fluid I (Is) I.
- 7 In the Settings window for 3D Plot Group, locate the Data section.
- 8 From the Time (s) list, choose 0.11.
- 9 On the Volume Fraction of Fluid I (Is) I toolbar, click Plot.

Volume Fraction of Fluid I (Is) I

- I In the Model Builder window, under Results click Volume Fraction of Fluid I (Is) I.
- 2 On the Volume Fraction of Fluid I (Is) I toolbar, click Plot.

Finally, create a movie using the current plot group.

3 Click Animation and choose Player.

Export

In the Model Builder window, under Results>Export right-click Animation I and choose Play.



Swirl Flow Around a Rotating Disk

Introduction

This example models a rotating disk in a tank. The model geometry is shown in Figure 1. Because the geometry is rotationally symmetric, it is possible to model it as a 2D cross section. However, the velocities in the angular direction differ from zero, so the model must include all three velocity components, even though the geometry is in 2D.



Figure 1: The original 3D geometry can be reduced to 2D because the geometry is rotationally symmetric.

Model Definition

DOMAIN EQUATIONS

The flow is described by the Navier-Stokes equations:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot [-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \mathbf{F}$$
$$\nabla \cdot \mathbf{u} = 0$$

In these equations, **u** denotes the velocity (SI unit: m/s), ρ the density (SI unit: kg/m³), μ the dynamic viscosity (SI unit: Pa·s), and *p* the pressure (SI unit: Pa). For a stationary, axisymmetric flow the equations reduce to (Ref. 1):

$$\begin{split} \rho \Big(u \frac{\partial u}{\partial r} - \frac{v^2}{r} + w \frac{\partial u}{\partial z} \Big) + \frac{\partial p}{\partial r} &= \mu \bigg[\frac{1}{r} \frac{\partial}{\partial r} \Big(r \frac{\partial u}{\partial r} \Big) - \frac{u}{r^2} + \frac{\partial^2 u}{\partial z^2} \bigg] + F_r \\ \rho \Big(u \frac{\partial v}{\partial r} + \frac{uv}{r} + w \frac{\partial v}{\partial z} \Big) &= \mu \bigg[\frac{1}{r} \frac{\partial}{\partial r} \Big(r \frac{\partial v}{\partial r} \Big) - \frac{v}{r^2} + \frac{\partial^2 v}{\partial z^2} \bigg] + F_{\varphi} \\ \rho \Big(u \frac{\partial w}{\partial r} + w \frac{\partial w}{\partial z} \Big) + \frac{\partial p}{\partial z} &= \mu \bigg[\frac{1}{r} \frac{\partial}{\partial r} \Big(r \frac{\partial w}{\partial r} \Big) + \frac{\partial^2 w}{\partial z^2} \bigg] + F_z \end{split}$$

Here *u* is the radial velocity, *v* the rotational velocity, and *w* the axial velocity (SI unit: m/s). In the model you set the volumetric force components F_r , F_{ϕ} , and F_z to zero. The swirling flow is 2D even though the model includes all three velocity components.

BOUNDARY CONDITIONS

Figure 2 below shows the boundary conditions.



Figure 2: Boundary conditions.

On the stirrer, use the sliding wall boundary condition to specify the velocities. The velocity components in the plane are zero, and that in the angular direction is equal to the angular velocity, ω , times the radius, *r*:

$$w_{\rm w} = r\omega$$

At the boundaries representing the cylinder surface a no slip condition applies, stating that all velocity components equal zero:

$$\mathbf{u} = (0, 0, 0)$$

At the boundary corresponding to the rotation axis, use the axial symmetry boundary condition allowing flow in the tangential direction of the boundary but not in the normal direction. This is obtained by setting the radial velocity to zero:

u = 0

On the top boundary, which is a free surface, use the Symmetry condition to allow for flow in the axial and rotational directions only. The boundary condition is mathematically similar to the axial symmetry condition.

POINT SETTINGS

In this model you need to lock the pressure to a reference value in a point. The reason for this is that the model does not contain any boundary condition where the pressure is specified (this is often done at outlets). Also the fluid density is constant, which means that the pressure level is not coupled to the density. In this model, set the pressure to zero in the top right corner.

Results

The parametric solver provides the solution for four different angular velocities. Figure 3 shows the results for the smallest angular velocity, $\omega = 0.25\pi$ rad/s.



Figure 3: Results for angular velocity $\omega = 0.25\pi$ rad/s. The surface plot shows the magnitude of the velocity field and the white lines are streamlines of the velocity field.

The shape of the two recirculation zones, which are visualized with streamlines, changes as the angular velocity increases. Figure 4 shows the streamlines of the velocity field for higher angular velocities.



Figure 4: Results for angular velocities $\omega = 0.5\pi$, 2π , and 4π rad/s. The surface plot shows the magnitude of the velocity and the white lines are streamlines of the velocity field.

Figure 5 and Figure 6 show isocontours of the rotational velocity component together with surface plots of the velocity magnitude for different angular velocities.



Figure 5: Isocontours for the azimuthal velocity component for angular velocity $\omega = 0.25\pi$ rad/s. The surface plot shows the magnitude of the velocity.



Figure 6: Magnitude of the velocity field (surface) and isocontours for the azimuthal velocity component for angular velocities (left to right) $\omega = 0.5\pi$, 2π , and 4π rad/s.

Figure 7 shows the turbulent viscosity and flow fields for the angular velocity to $\omega = 500\pi$ rad/s and turbulent flow in the mixer volume.



Figure 7: Results for angular velocity $\omega = 500\pi$ rad/s. The surface plot shows the turbulent viscosity and the white lines are streamlines of the velocity field.

Reference

1. P.M. Gresho and R.L. Sani, *Incompressible Flow and the Finite Element Method*, vol. 2, p. 469, John Wiley & Sons, 1998.

Application Library path: CFD_Module/Single-Phase_Tutorials/rotating_disk

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Laminar Flow (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
omega	0.25*pi[rad/s]	0.7854 rad/s	Angular velocity

GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.02.
- **4** In the **Height** text field, type 0.04.
- **5** Click **Build All Objects**.
- 6 Click the Zoom Extents button on the Graphics toolbar.

Rectangle 2 (r2)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type 0.008.
- 4 In the **Height** text field, type 0.003.
- **5** Locate the **Position** section. In the **z** text field, type **0.014**.
- 6 Click Build All Objects.

Rectangle 3 (r3)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type 0.001.
- 4 In the **Height** text field, type 0.023.
- **5** Locate the **Position** section. In the **z** text field, type **0.017**.
- 6 Click Build All Objects.

Difference I (dif1)

- I On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 2 Select the object rl only.
- 3 In the Settings window for Difference, locate the Difference section.
- 4 Find the Objects to subtract subsection. Select the Active toggle button.
- 5 Select the objects r2 and r3 only.
- 6 Clear the Keep interior boundaries check box.
- 7 Click Build All Objects.
- 8 Click the Zoom Extents button on the Graphics toolbar.

MATERIALS

In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.

Material I (mat1)

- I In the Settings window for Material, locate the Material Contents section.
- **2** In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	1e3	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 In the Settings window for Laminar Flow, locate the Physical Model section.
- **3** Select the **Swirl flow** check box.
- **4** In the **Model Builder** window's toolbar, click the **Show** button and select **Discretization** in the menu.

5 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P1**.

This setting gives quadratic elements for the velocity field.

Wall 2

- I On the Physics toolbar, click Boundaries and choose Wall.
- **2** Select Boundaries 3–5 and 7 only.
- 3 In the Settings window for Wall, locate the Boundary Condition section.
- 4 From the Boundary condition list, choose Sliding wall.
- **5** In the $v_{\rm w}$ text field, type omega*r.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- **2** Select Boundary 6 only.

Pressure Point Constraint I

- I On the Physics toolbar, click Points and choose Pressure Point Constraint.
- 2 Select Point 8 only.

STUDY I

Step 1: Stationary

- I In the Model Builder window, expand the Study I node, then click Step I: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study extensions section.
- **3** Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 4 Click Add.
- **5** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega	0.25*pi 0.5*pi 2*pi 4*pi	

6 On the Home toolbar, click Compute.

RESULTS

Velocity (spf) To create Figure 3 do the following steps:

I In the Model Builder window, under Results click Velocity (spf).

- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Parameter value (omega) list, choose 0.7854.
- **4** On the **Velocity (spf)** toolbar, click **Plot**.
- 5 In the Model Builder window, expand the Velocity (spf) node, then click Surface.
- 6 In the Settings window for Surface, locate the Expression section.
- 7 From the **Unit** list, choose **mm/s**.
- 8 In the Model Builder window, right-click Velocity (spf) and choose Streamline.
- 9 In the Settings window for Streamline, locate the Streamline Positioning section.
- **IO** From the **Positioning** list, choose **Uniform density**.
- **II** In the **Separating distance** text field, type 0.02.
- 12 Locate the Coloring and Style section. From the Color list, choose White.
- **I3** On the **Velocity (spf)** toolbar, click **Plot**.

To produce the series of snapshots of the velocity and streamlines of the velocity field shown in Figure 4, proceed with the following steps:

- 14 In the Model Builder window, under Results>Velocity (spf) click Surface.
- **I5** In the **Settings** window for Surface, locate the **Coloring and Style** section.
- **I6** Clear the **Color legend** check box.
- **17** In the **Model Builder** window, click **Velocity (spf)**.
- **18** In the **Settings** window for 2D Plot Group, locate the **Data** section.
- 19 From the Parameter value (omega) list, choose 1.571.
- **20** On the **Velocity (spf)** toolbar, click **Plot**.
- 21 From the Parameter value (omega) list, choose 6.283.
- **22** On the **Velocity (spf)** toolbar, click **Plot**.
- 23 From the Parameter value (omega) list, choose 12.57.
- 24 On the Velocity (spf) toolbar, click Plot.

To plot the isocontours for the azimuthal velocity component Figure 5, proceed with the following steps.

2D Plot Group 4

- I On the Home toolbar, click Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Parameter value (omega) list, choose 0.7854.

- 4 Right-click 2D Plot Group 4 and choose Surface.
- 5 Right-click 2D Plot Group 4 and choose Contour.
- 6 In the Settings window for Contour, click to expand the Quality section.
- 7 On the 2D Plot Group 4 toolbar, click Plot.
- 8 Click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Laminar Flow>Velocity and pressure>Velocity field>v Velocity field, phi component.
- 9 Locate the Levels section. In the Total levels text field, type 15.
- 10 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- II From the **Color** list, choose **White**.
- 12 Locate the Quality section. From the Resolution list, choose Finer.
- **I3** From the **Recover** list, choose **Within domains**.

To reproduce Figure 6 do the following steps.

- I4 In the Model Builder window, under Results>2D Plot Group 4 click Surface I.
- 15 In the Settings window for Surface, locate the Coloring and Style section.
- **I6** Clear the **Color legend** check box.
- **I7** In the **Model Builder** window, click **2D Plot Group 4**.
- **18** In the **Settings** window for 2D Plot Group, locate the **Data** section.
- 19 From the Parameter value (omega) list, choose 1.571.
- 20 On the 2D Plot Group 4 toolbar, click Plot.
- 21 From the Parameter value (omega) list, choose 6.283.
- 22 On the 2D Plot Group 4 toolbar, click Plot.
- 23 From the Parameter value (omega) list, choose 12.57.

24 On the 2D Plot Group 4 toolbar, click Plot.

Now, modify the model to simulate turbulent swirl flow. **Pseudo time stepping** enhances the stability of stationary turbulence simulations. In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 In the Settings window for Laminar Flow, locate the Physical Model section.
- **3** From the **Turbulence model type** list, choose **RANS**.
4 Click to expand the Advanced settings section. Locate the Advanced Settings section. Find the Pseudo time stepping subsection. Select the Use pseudo time stepping for stationary equation form check box.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- **3** From the **Element size** list, choose **Fine**.

STUDY I

Step 1: Stationary

- I In the Model Builder window, under Study I click Step I: Stationary.
- 2 In the Settings window for Stationary, locate the Study Extensions section.
- **3** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
omega	range(pi*100,pi*200,pi*500)	

4 On the Home toolbar, click Compute.

RESULTS

Velocity (spf)

Next, define a surface plot visualizing the turbulent viscosity and the streamlines of the velocity field (Figure 7).

Velocity (spf) 2

- I In the Model Builder window, under Results right-click Velocity (spf) and choose Duplicate.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Parameter value (omega) list, choose 1571.
- 4 In the Model Builder window, expand the Velocity (spf) 2 node, then click Surface 1.
- 5 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>Turbulent Flow, k-ε> Turbulence variables>spf.muT - Turbulent dynamic viscosity.
- 6 Locate the Coloring and Style section. Select the Color legend check box.

14 | SWIRL FLOW AROUND A ROTATING DISK



Syngas Combustion in a Round-Jet Burner

Introduction

This model simulates turbulent combustion of syngas (synthesis gas) in a simple round jet burner. Syngas is a gas mixture, primarily composed of hydrogen, carbon monoxide and carbon dioxide. The name syngas relates to its use in creating synthetic natural gas.

The model set up corresponds to the one studied by Couci et al. in Ref. 1. The temperature and composition resulting from the nonpremixed combustion in the burner setup have also been experimentally investigated by Barlow and coworkers (Ref. 2 and Ref. 3) as a part of the International Workshop on Measurement and Computation of Turbulent Nonpremixed Flames (Ref. 4). The model is solved in COMSOL Multiphysics by combining a Reacting Flow and a Heat Transfer in Fluids interface.

Model Definition

The burner studied in this model consists of a straight pipe placed in a slight co-flow. The gas phase fuel is fed through the pipe using an inlet velocity of 76 m/s, while the co-flow velocity outside of pipe is 0.7 m/s. At the pipe exit, the fuel gas mixes with the co-flow, creating an unconfined circular jet. The gas fed through the tube consists of three compounds typical of syngas; carbon monoxide (CO), hydrogen (H_2) and nitrogen (N_2) . The co-flow gas consists of air. At the pipe exit, the fuel is ignited. Since the fuel and oxidizer enter the reaction zone separately, the resulting combustion is of the non-premixed type. A continuous reaction requires that the reactants and the oxidizer are mixed to stoichiometric conditions. In this set-up the turbulent flow of the jet effectively mixes the fuel from the pipe with the co-flowing oxygen. Furthermore the mixture needs to be continuously ignited. In this burner the small recirculation zones generated by the pipe wall thickness provide the means to decelerate hot product gas. The recirculation zones hereby promote continuous ignition of the oncoming mixture and stabilizes the flame at the pipe orifice. In experiments (Ref. 4) no lift-off or localized extinction of the flame has been observed.

In the current model, the syngas combustion is modeled using two irreversible reactions:

$$CO + 0.5O_2 \rightarrow CO_2$$
$$H_2 + 0.5O_2 \rightarrow H_2O \tag{1}$$

~~

This assumption of a complete oxidation of the fuel corresponds to one of the approaches used in Ref. 1. The mass transport in the reacting jet is modeled by solving for the mass fractions of six species; the five species participating in the reactions and nitrogen N_2 originating in the co-flowing air.

The Reynolds number for the jet, based on the inlet velocity and the inner diameter of the pipe, is approximately 16700, indicating that the jet is fully turbulent. Under these circumstances, both the mixing and the reactions processes in the jet are significantly influenced by the turbulent nature of the flow. To account for the turbulence when solving for the flow field, the k- ω turbulence model is applied.

Taking advantage of the symmetry, a two-dimensional model using a cylindrical coordinate system is solved.

TURBULENT REACTION RATE

When using a turbulence model in a Reacting Flow interface, the production rate (SI unit: $kg/(m^3 \cdot s)$) of species *i* resulting from reaction *j* is modeled as the minimum of the mean-value-closure reaction rate and the eddy-dissipation-model rate:

$$R_{ij} = v_{ij}M_i \cdot \min[r_{\text{MVC},j}, r_{\text{ED},j}]$$

The mean-value-closure rate is the kinetic reaction rate expressed using the mean mass fractions. This corresponds to the characteristic reaction rate for reactions which are slow compared to the turbulent mixing, or the reaction rate in regions with negligible turbulence levels. This can be quantified through the Damköhler number, which compares the turbulent time scale (τ_T) to the chemical time scale (τ_c). The mean-value-closure is appropriate for low Damköhler numbers:

$$Da = \frac{\tau_T}{\tau_c} \ll 1$$

The reaction rate defined by the eddy-dissipation model (Ref. 5) is:

$$r_{\text{ED},j} = \frac{\alpha_j}{\tau_{\text{T}}} \rho \cdot \min\left[\min\left(\frac{\omega_r}{v_{rj}M_r}\right), \beta \sum_p \left(\frac{\omega_p}{v_{pj}M_p}\right)\right]$$
(2)

where τ_T (SI unit: s) is the mixing time scale of the turbulence, ρ is the mixture density (SI unit: kg/m³), ω is the species mass fraction, ν denotes the stoichiometric coefficients, and M is the molar mass (SI unit: kg/mol). Properties of reactants of the reaction are indicated using a subscript r, while product properties are denoted by a subscript p.

The eddy-dissipation model assumes that both the Reynolds and Damköhler numbers are sufficiently high for the reaction rate to be limited by the turbulent mixing time scale τ_{T} . A global reaction can then at most progress at the rate at which fresh reactants are mixed, at the molecular level, by the turbulence present. The reaction rate is also assumed to be

limited by the deficient reactant; the reactant with the lowest local concentration. The model parameter β specifies that product species is required for reaction, modeling the activation energy. For gaseous non-premixed combustion the model parameters have been found to be (Ref. 5):

$$\alpha = 4, \beta = 0.5$$

In the current model the molecular reaction rate of the reactions is assumed to be infinitely fast. This is achieved in the model by prescribing unrealistically high rate constants for the reactions. This implies that the production rate is given solely by the turbulent mixing in Equation 2.

It should be noted that the eddy-dissipation model is a robust but simple model for turbulent reactions. The reaction rate is governed by a single time scale, the turbulent mixing time-scale. For this reason, the reactions studied should be limited to global one step (as in Equation 1), or two step reactions.

HEAT OF REACTION

The heat of reaction, or change in enthalpy, following each reaction is defined from the heat of formation of the products and reactants:

$$\Delta H_{\rm r} = \sum_{\rm products} \Delta H_{\rm f} - \sum_{\rm reactants} \Delta H_{\rm f}$$

The heat of formations for each species is given in Table 1 (based on Ref. 6). Since the heat of formation of the products is lower than that of the reactants, both reactions are exothermic and release heat. The heat release is included in the model by adding a Heat Source feature to the Heat Transfer in Fluids interface. The heat source (SI unit: W/m^3) applied is defined as:

$$q = r_{\text{ED},1} \Delta H_{\text{r1}} + r_{\text{ED},2} \Delta H_{\text{r2}}$$

SPECIES	$\Delta H_{ m f}$ (cal/mol) T = 298 K	$C_{\rm p}~{ m (cal/(mol·K)}$ T = 300 K	$C_{ m p}~({ m cal/(mol\cdot K)})$ T = 1000 K	C _p (cal/(mol·K) T = 2000 К
N_2	0	6.949	7.830	8.601
H_2	0	6.902	7.209	8.183
O_2	0	7.010	8.350	9.032
H_2O	-57.80	7.999	9.875	12.224

TABLE I: SPECIES ENTHALPY OF FORMATION AND HEAT CAPACITY

4 SYNGAS COMBUSTION IN A ROUND-JET BURNER

SPECIES	$\Delta H_{\rm f}$ (cal/mol) T = 298 K	C _p (cal/(mol·K) Т = 300 К	C _p (cal/(mol·K) T = 1000 К	$C_{\rm p}$ (cal/(mol·K) T = 2000 K
со	-26.420	47.259	6.950	7.948
CO_2	-94.061	51.140	8.910	12.993

TABLE I: SPECIES ENTHALPY OF FORMATION AND HEAT CAPACITY

HEAT CAPACITY

The temperature in the jet increases significantly due to the heat release following the reactions, this is one of the defining features of combustion. For an accurate prediction of the temperature it is important to account for the temperature dependence of the species heat capacities. In the model, interpolation functions for the heat capacity at constant pressure, $C_{\mathbf{p},i}$ (SI unit: cal/(mol·K)), for each species are defined using the values at three different temperatures given in Table 1. The heat capacity of the mixture, $c_{\mathbf{p},\min}$ (SI unit: J/(kg·K)), is computed as a mass fraction weighted mean of the individual heat capacities:

$$c_{\rm p, mix} = \sum_{i} \frac{\omega_i C_{\rm p, i}}{M_i}$$

SOLUTION PROCEDURE

The syngas combustion model is solved in three steps.

- I Use an initial submodel to solve for isothermal turbulent flow in a straight pipe with the same diameter as the burner. The fully developed flow at the pipe outlet is then used as inlet condition for the burner.
- **2** Solve for the turbulent and reacting, but isothermal, flow in the round jet burner configuration.
- **3** Include the heat transfer and solve for the fully coupled reacting flow, using the previous solution as initial condition.

Using several solution steps is vital for a robust solution procedure when solving models with a high degree of coupling. This is the case for turbulent reacting flow including heat transfer.

Results and Discussion

The resulting velocity field in the non-isothermal reacting jet is visualized in Figure 1. The expansion and development of the hot free jet is clearly seen. The turbulent mixing in the outer parts of the jet acts to accelerate fluid originating in the co-flow, and incorporate it

in the jet. This is commonly referred to as entrainment and can be observed in the co-flow streamlines which bend towards the jet downstream of the orifice.



Figure 1: The velocity magnitude and flow paths (streamlines) of the reacting jet.

The temperature in the jet is shown in Figure 2 where a revolved data set has been used to emphasize the structure of the round jet. The maximum temperature in the jet is seen to be approximately 1960 K. The carbon dioxide mass fraction in the reacting jet is plotted in Figure 3. The formation of CO_2 takes place in the outer shear layer of the jet. This is where the fuel from the pipe encounters oxygen in the co-flow and reacts. The reactions are promoted by the turbulent mixing in the jet shear layer. It is also seen that the CO_2 formation starts just outside of the pipe. This is also the case for the temperature increase in Figure 2. This implies that there is no lift-off and the flame is attached to the pipe.

In Figure 4, Figure 5, and Figure 6 the results reached in the model are compared with the experimental results of Barlow and coworkers (Ref. 2, Ref. 3, and Ref. 4). In Figure 4 the jet temperature is further examined and compared with the experiments. In the left panel the temperature along the centerline is plotted. It is seen that the maximum temperature predicted in the model is close to that in the experiment. However in the model the temperature profile is shifted in the downstream direction. This is most likely due to the fact that radiation has not been included in the model.

In the right panel of Figure 4 temperature profiles at 20 and 50 pipe diameters downstream of the pipe exit are compared with the experiments. The axial velocity of the jet is compared with the experimental results in Figure 5, using the same down stream positions. The axial velocity is found to compare well with the experimental values at both positions.

In Figure 6 the species concentration along the jet centerline is analyzed and compared with the experimental results. For some species, N_2 , and CO_2 , the axial mass fraction development agrees well with the experimental results. For the fuel species CO and H_2 a fair agreement is observed. For the remaining species, O_2 and H_2O , the trend appears correct but the profiles are shifted downstream, as was the case with the temperature. The reason for the discrepancy in the mass fractions can in part be attributed to the fact that radiation is not included, but the accuracy is probably also significantly influenced by the simplified reaction scheme and the eddy-dissipation model.



Figure 2: Jet temperature shown using a revolved data set.



Figure 3: CO_2 mass fraction in the reacting jet.



Figure 4: Jet temperature along the centerline (left), and radially at two different positions downstream of the pipe exit (right) scaled by the inlet temperature. The centerline and radial distance is scaled by the inner diameter of the pipe. Model results are plotted using lines, while experimental results are indicated using symbols. The downstream positions are defined in terms of the inner diameter of the pipe (d).



Figure 5: Axial velocity at two different positions downstream of the pipe exit, scaled by the inlet velocity. The radial distance is scaled by the inner diameter of the pipe. Model results are plotted using lines, while experimental results are indicated using symbols.



Figure 6: Species mass fractions along the jet centerline. The centerline distance is scaled by the

inner diameter of the pipe. Model results are plotted using lines, while experimental results are indicated using symbols.

References

1. A. Cuoci, A. Frassoldati, G. Buzzi Ferraris, T. Faravelli, E. Ranzi, "The ignition, combustion and flame structure of carbon monoxide/hydrogen mixtures. Note 2: Fluid dynamics and kinetic aspects of syngas combustion," *Int. J. Hydrogen Energy*, vol. 32, pp. 3486–3500, 2007

2. R. S. Barlow, G. J. Fiechtner, C. D. Carter, and J.-Y. Chen, "Experiments on the Scalar Structure of Turbulent CO/H2/N2 Jet Flames," *Comb. and Flame*, vol. 120, pp. 549–569, 2000.

3. M. Flury, *Experimentelle Analyse der Mischungstruktur in turbulenten nicht vorgemischten Flammen*, Ph.D. Thesis, ETH Zurich, 1998.

4. R. S. Barlow et al., "Sandia/ETH-Zurich CO/H2/N2 Flame Data - Release 1.1," http://www.sandia.gov/TNF/DataArch/SANDchnWeb/SANDchnDocl1.pdf, 2002.

5. B.F. Magnussen and B.H. Hjertager, "On Mathematical Modeling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion," *16th Symp.* (*Int.*) on Combustion. Comb. Inst., Pittsburg, Pennsylvania, pp.719–729, 1976.

6. A. Frassoldati, T. Faravelli, and E. Ranzi, "The Ignition, Combustion and Flame Structure of Carbon Monoxide/Hydrogen Mixtures. Note 1: Detailed Kinetic Modeling of Syngas Combustion Also in Presence of Nitrogen Compounds," *Int. J. Hydrogen Energy*, vol. 32, pp. 3471–3485, 2007.

Application Library path: CFD_Module/Non-Isothermal_Flow/round_jet_burner

Modeling Instructions

From the File menu, choose New.

NEW

I In the New window, click Model Wizard.

MODEL WIZARD

I In the Model Wizard window, click 2D Axisymmetric.

- 2 In the Select physics tree, select Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ω (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select study tree, select Preset Studies>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Click Load from File.
- **4** Browse to the application's Application Library folder and double-click the file round_jet_burner_params.txt.

GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type Di/2.
- 4 In the Height text field, type Di*200.
- 5 Right-click Rectangle I (rI) and choose Build Selected.
- 6 Click the Zoom Extents button on the Graphics toolbar.

For easier visualization of the slender geometry, disable preserve aspect ratio for the view.

DEFINITIONS

View I

In the Model Builder window, expand the Component I (compl)>Definitions node.

Axis

- I In the Model Builder window, expand the View I node, then click Axis.
- 2 In the Settings window for Axis, locate the Axis section.
- **3** From the **View scale** list, choose **Automatic**.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Apply fluid properties for the pipe simulation. An approximate density can be used.

Turbulent Flow, k-@ (spf)

Fluid Properties 1

- I In the Model Builder window, under Component I (compl)>Turbulent Flow, k-ω (spf) click Fluid Properties I.
- 2 In the Settings window for Fluid Properties, locate the Fluid Properties section.
- **3** From the ρ list, choose **User defined**. In the associated text field, type 1.
- 4 From the μ list, choose User defined. In the associated text field, type mu_mix.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 2 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- **4** In the $L_{\rm T}$ text field, type 0.07*Di.
- **5** Locate the **Velocity** section. In the U_0 text field, type Ujet.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 3 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

MESH I

Mapped I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Distribution I

- I In the Model Builder window, under Component I (compl)>Mesh I right-click Mapped I and choose Distribution.
- 2 Select Boundaries 2 and 3 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- **4** From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the Number of elements text field, type 25.

- 6 In the Element ratio text field, type 5.
- 7 Click the **Build Selected** button.

Distribution 2

- I Right-click Mapped I and choose Distribution.
- 2 Select Boundaries 1 and 4 only.
- 3 In the Settings window for Distribution, locate the Distribution section.
- **4** From the **Distribution properties** list, choose **Predefined distribution type**.
- 5 In the Number of elements text field, type 200.
- 6 In the Element ratio text field, type 20.
- 7 Click the **Build Selected** button.

Now add a second model for the round reacting jet simulation.

I On the Home toolbar, click Add Component and choose 2D Axisymmetric.

ADD PHYSICS

- I On the Home toolbar, click Add Physics to open the Add Physics window.
- 2 Go to the Add Physics window.
- 3 In the Add physics tree, select Chemical Species Transport>Reacting Flow>Turbulent Flow> Turbulent Flow, k-ω (rspf).
- 4 Find the Physics interfaces in study subsection. In the table, enter the following settings:

Studies	Solve
Study I	

- 5 Click to expand the Dependent variables section. Locate the Dependent Variables section. In the Number of species text field, type 6.
- 6 In the Mass fractions table, enter the following settings:

7 Click Add to Component in the window toolbar.

- 8 Go to the Add Physics window.
- 9 In the Add physics tree, select Heat Transfer>Heat Transfer in Fluids (ht).

10 Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Studies	Solve
<u> </u>	

Study I

II Click Add to Component in the window toolbar.

12 On the Home toolbar, click Add Physics to close the Add Physics window.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select study tree, select Preset Studies>Stationary.
- 4 Click Add Study in the window toolbar.
- 5 On the Home toolbar, click Add Study to close the Add Study window.

DEFINITIONS

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.
- **3** Click Load from File.
- 4 Browse to the application's Application Library folder and double-click the file round_jet_burner_vars.txt.

GEOMETRY 2

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type GeomW.
- 4 In the **Height** text field, type GeomH.

Rectangle 2 (r2)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Rectangle.

- 3 In the Settings window for Rectangle, locate the Size and Shape section.
- 4 In the Width text field, type Pth.
- **5** In the **Height** text field, type P1.
- 6 Locate the **Position** section. In the **r** text field, type Di/2.

Chamfer 1 (cha1)

- I Right-click Rectangle 2 (r2) and choose Build Selected.
- 2 On the Geometry toolbar, click Chamfer.
- 3 On the object r2, select Points 3 and 4 only.
- 4 In the Settings window for Chamfer, locate the Distance section.
- 5 In the Distance from vertex text field, type Pth*0.15.
- 6 Right-click Chamfer I (chal) and choose Build Selected.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Bézier Polygon I (b1)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 4 Find the Added segments subsection. Click Add Linear.
- 5 Find the Control points subsection. In row I, set r to GeomW.
- 6 In row 2, set r to GeomW*1.5.
- 7 In row 2, set z to GeomH.
- 8 Find the Added segments subsection. Click Add Linear.
- 9 Find the Control points subsection. In row 2, set r to GeomW.

Union I (unil)

- I Right-click Bézier Polygon I (bI) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Union.
- **3** Select the objects **b1** and **r1** only.
- 4 In the Settings window for Union, locate the Union section.
- **5** Clear the **Keep interior boundaries** check box.

Difference I (dif1)

- I Right-click Union I (unil) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Difference.

- 3 Select the object unil only.
- 4 In the Settings window for Difference, locate the Difference section.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the object chal only.

Bézier Polygon 2 (b2)

- I Right-click Difference I (difl) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 4 Find the Added segments subsection. Click Add Linear.
- **5** Find the **Control points** subsection. In row **I**, set **r** to Di/2.
- 6 In row I, set z to P1+0.3e-3.
- **7** In row **2**, set **r** to **Di**.
- 8 In row 2, set z to GeomH.

Bézier Polygon 3 (b3)

- I Right-click Bézier Polygon 2 (b2) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 4 Find the Added segments subsection. Click Add Linear.
- 5 Find the Control points subsection. In row I, set r to Di/2+Pth.
- 6 In row I, set z to P1+0.3e-3.
- 7 In row 2, set r to 0.04.
- 8 In row 2, set z to GeomH.

Form Union (fin)

- I Right-click Bézier Polygon 3 (b3) and choose Build Selected.
- 2 In the Model Builder window, under Component 2 (comp2)>Geometry 2 right-click Form Union (fin) and choose Build Selected.
- **3** In the **Settings** window for Form Union/Assembly, locate the **Form Union/Assembly** section.
- 4 In the Relative repair tolerance text field, type 1e-6.

Mesh Control Edges 1 (mcel)

I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Edges.

- 2 On the object fin, select Boundaries 6 and 11 only.
- 3 Right-click Mesh Control Edges I (mcel) and choose Build Selected.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.

Form Composite Edges 1 (cme1)

- I On the Geometry toolbar, click Virtual Operations and choose Form Composite Edges.
- 2 On the object mcel, select Boundaries 3 and 11 only.
- 3 Right-click Form Composite Edges I (cmel) and choose Build Selected.

That concludes the geometry for the reacting jet. Now define a coupling variable that can be used to apply the outlet conditions from the previous model to the inlet of the current.

DEFINITIONS

In the Model Builder window, under Component I (compl) click Definitions.

Linear Extrusion 1 (linext1)

- I On the Definitions toolbar, click Component Couplings and choose Linear Extrusion.
- 2 Select Domain 1 only.
- 3 In the Settings window for Linear Extrusion, locate the Source Vertices section.
- **4** Select the **Active** toggle button.
- 5 Select Point 2 only.
- 6 Select the Active toggle button.
- **7** Select Point 4 only.
- 8 Click to expand the **Destination** section. From the **Destination geometry** list, choose **Geometry 2**.
- 9 Locate the **Destination Vertices** section. Select the **Active** toggle button.
- **IO** Select Point 1 only.
- II Select the Active toggle button.
- **12** Select Point 3 only.

REACTING FLOW (RSPF)

- I In the Model Builder window, under Component 2 (comp2) click Reacting Flow (rspf).
- 2 In the Settings window for Reacting Flow, locate the Species section.
- 3 From the From mass constraint list, choose wN2.
- **4** Locate the **Physical Model** section. Find the **Mass transport mechanisms** subsection. From the **Diffusion model** list, choose **Fick's law**.

Fluid I

Apply the temperature from the heat transfer interface.

- I In the Model Builder window, under Component 2 (comp2)>Reacting Flow (rspf) click Fluid
 I.
- 2 In the Settings window for Fluid, locate the Model Inputs section.
- **3** From the *T* list, choose **Temperature (ht)**.
- 4 Locate the Fluid Properties section. From the ρ list, choose Ideal gas.
- **5** From the μ list, choose **User defined**. In the associated text field, type mu_mix.
- **6** In the $M_{\rm wCO}$ text field, type M_CO.
- 7 In the $M_{\rm wO2}$ text field, type M_02.
- 8 In the $M_{\rm wCO2}$ text field, type M_CO2.
- **9** In the $M_{\rm wH2}$ text field, type M_H2.
- **IO** In the $M_{\rm wH2O}$ text field, type M_H2O.
- II In the $M_{\rm wN2}$ text field, type M_N2.
- 12 Locate the Mixing Length Limit section. From the list, choose Manual.
- **I3** In the $l_{\text{mix},lim}$ text field, type Di*10.

Initial Values 1

- I In the Model Builder window, under Component 2 (comp2)>Reacting Flow (rspf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the $w_{0,wCO}$ text field, type 0.
- **4** In the $w_{0,wO2}$ text field, type wcf_02.
- **5** In the $w_{0,wCO2}$ text field, type **0**.
- 6 In the $w_{0,wH2}$ text field, type 0.
- 7 In the $w_{0,wH2O}$ text field, type 0.
- 8 In the Model Builder window, click Reacting Flow (rspf).

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundary 2 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- 4 Click the Specify turbulence variables button.

- **5** In the k_0 text field, type comp1.linext1(k).
- **6** In the ω_0 text field, type comp1.linext1(om).
- 7 Locate the Velocity section. In the U_0 text field, type comp1.linext1(w).

Inlet 2

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundaries 9 and 10 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- **4** In the $I_{\rm T}$ text field, type 0.01.
- **5** In the $L_{\rm T}$ text field, type 0.1*Di.
- 6 Locate the Velocity section. Click the Velocity field button.
- **7** Specify the **u**₀ vector as

0 r Ucf z

Inflow I

- I On the Physics toolbar, click Boundaries and choose Inflow.
- **2** Select Boundary 2 only.
- 3 In the Settings window for Inflow, locate the Inflow section.
- 4 From the Mixture specification list, choose Mole fractions.
- **5** In the $x_{0,wCO}$ text field, type x0_CO.
- **6** In the $x_{0,wO2}$ text field, type x0_02.
- 7 In the $x_{0,wCO2}$ text field, type x0_CO2.
- **8** In the $x_{0,wH2}$ text field, type x0_H2.
- **9** In the $x_{0,wH2O}$ text field, type x0_H2O.

Inflow 2

- I On the Physics toolbar, click Boundaries and choose Inflow.
- **2** Select Boundaries 9 and 10 only.
- 3 In the Settings window for Inflow, locate the Inflow section.
- **4** In the $w_{0,wCO}$ text field, type 1e-5.
- **5** In the $w_{0,wO2}$ text field, type wcf_02.
- 6 In the $w_{0,wCO2}$ text field, type 1e-5.
- 7 In the $w_{0,wH2}$ text field, type 1e-5.

8 In the $w_{0.wH2O}$ text field, type 1e-5.

Outflow I

- I On the Physics toolbar, click Boundaries and choose Outflow.
- 2 Select Boundary 3 only.

Outlet 2

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 3 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

Reactions I

- I On the Physics toolbar, click Domains and choose Reactions.
- 2 Select Domain 1 only.
- 3 In the Settings window for Reactions, locate the Reaction Rate section.
- **4** In the v_{wCO} text field, type -1.
- **5** In the v_{wO2} text field, type -0.5.
- **6** In the v_{wCO2} text field, type 1.
- 7 Locate the Rate Constants section. Clear the Use Arrhenius expressions check box.

Apply an unrealistically high reaction rate to model the reactions as infinitely fast. In this case the reaction rate will be given by the turbulent mixing.

8 In the k^{f} text field, type 1e100.

Reactions 2

- I On the Physics toolbar, click Domains and choose Reactions.
- 2 Select Domain 1 only.
- 3 In the Settings window for Reactions, locate the Reaction Rate section.
- **4** In the v_{wO2} text field, type -0.5.
- **5** In the v_{wH2} text field, type -1.
- **6** In the v_{wH2O} text field, type **1**.
- 7 Locate the Rate Constants section. Clear the Use Arrhenius expressions check box.
- 8 In the k^{f} text field, type 1e100.

Use the tabulated heat capacities to create interpolation functions, one for each species.

DEFINITIONS

Interpolation 1 (int1)

- I On the Home toolbar, click Functions and choose Local>Interpolation.
- 2 In the Settings window for Interpolation, locate the Definition section.
- **3** In the **Function name** text field, type Cp_CO.
- **4** In the table, enter the following settings:

t	f(t)
300	47.259
1000	6.950
2000	7.948

- 5 Locate the Interpolation and Extrapolation section. From the Interpolation list, choose Piecewise cubic.
- 6 Locate the Units section. In the Arguments text field, type K.
- 7 In the Function text field, type cal/mol/K.

Plot the resulting interpolation function.

8 Click the **Plot** button.

Interpolation 2 (int2)

- I Right-click Interpolation I (intl) and choose Duplicate.
- 2 In the Settings window for Interpolation, locate the Definition section.
- 3 In the Function name text field, type Cp_C02.
- **4** In the table, enter the following settings:

t	f(t)
300	51.140
1000	8.910
2000	12.993

Interpolation 3 (int3)

- I Right-click Component 2 (comp2)>Definitions>Interpolation 2 (int2) and choose Duplicate.
- 2 In the Settings window for Interpolation, locate the Definition section.
- **3** In the **Function name** text field, type Cp_H2.

4 In the table, enter the following settings:

t	f(t)
300	6.902
1000	7.209
2000	8.183

Interpolation 4 (int4)

- I Right-click Component 2 (comp2)>Definitions>Interpolation 3 (int3) and choose Duplicate.
- 2 In the Settings window for Interpolation, locate the Definition section.
- **3** In the **Function name** text field, type Cp_H20.
- **4** In the table, enter the following settings:

t	f(t)
300	7.999
1000	9.875
2000	12.224

Interpolation 5 (int5)

I Right-click Component 2 (comp2)>Definitions>Interpolation 4 (int4) and choose Duplicate.

- 2 In the Settings window for Interpolation, locate the Definition section.
- **3** In the **Function name** text field, type Cp_N2.
- **4** In the table, enter the following settings:

t	f(t)
300	6.949
1000	7.830
2000	8.601

Interpolation 6 (int6)

- I Right-click Component 2 (comp2)>Definitions>Interpolation 5 (int5) and choose Duplicate.
- 2 In the Settings window for Interpolation, locate the Definition section.
- **3** In the **Function name** text field, type Cp_02.

4 In the table, enter the following settings:

t	f(t)
300	7.010
1000	8.350
2000	9.032

Define the mixture heat capacity. It is computed as the mass average of the species capacities. Also define the enthalpy change for each of the reactions included.

Variables I

I In the Model Builder window, under Component 2 (comp2)>Definitions click Variables I.

2 In the Settings window for Variables, locate the Variables section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Cp_mix	<pre>wC0*Cp_C0(T)/M_C0+ wC02*Cp_C02(T)/ M_C02+wH2*Cp_H2(T)/ M_H2+wH20* Cp_H20(T)/M_H20+ wN2*Cp_N2(T)/M_N2+ w02*Cp_02(T)/M_02</pre>	J/(kg·K)	Heat capacity, mixture
dH_R1	dH_CO2-(dH_CO+0.5* dH_O2)	J/mol	Enthalpy change reaction 1
dH_R2	dH_H2O-(dH_H2+0.5* dH_O2)	J/mol	Enthalpy change reaction 2

Now setup the heat transfer interface.

HEAT TRANSFER IN FLUIDS (HT)

On the Physics toolbar, click Reacting Flow (rspf) and choose Heat Transfer in Fluids (ht).

Heat Transfer in Fluids 1

- I In the Model Builder window, under Component 2 (comp2)>Heat Transfer in Fluids (ht) click Heat Transfer in Fluids I.
- 2 In the Settings window for Heat Transfer in Fluids, locate the Model Inputs section.
- **3** From the p_A list, choose **Absolute pressure (rspf)**.
- 4 From the **u** list, choose **Velocity field (rspf/rfluid I)**.
- 5 Locate the Heat Conduction, Fluid section. From the k list, choose User defined. In the associated text field, type k_mix+Cp_mix*rspf.muT/0.72.

- 6 Locate the Thermodynamics, Fluid section. From the Fluid type list, choose Ideal gas.
- 7 From the Gas constant type list, choose Mean molar mass.
- 8 From the M_n list, choose User defined. In the associated text field, type rspf.Mn.
- **9** From the C_p list, choose User defined. In the associated text field, type Cp_mix.

Add a pressure work feature.

Pressure Work I

On the Physics toolbar, click Attributes and choose Pressure Work.

Initial Values 1

- I In the Settings window for Initial Values, locate the Initial Values section.
- **2** In the T text field, type T0.

Temperature 1

- I On the Physics toolbar, click Boundaries and choose Temperature.
- **2** Select Boundaries 2, 9, and 10 only.
- 3 In the Settings window for Temperature, locate the Temperature section.
- **4** In the T_0 text field, type T0.

Outflow I

- I On the Physics toolbar, click Boundaries and choose Outflow.
- 2 Select Boundary 3 only.

Heat Source 1

- I On the Physics toolbar, click Domains and choose Heat Source.
- 2 Select Domain 1 only.
- 3 In the Settings window for Heat Source, locate the Heat Source section.
- **4** In the Q_0 text field, type (dH_R1*rspf.r_treac1+dH_R2*rspf.r_treac2).

MESH 2

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

Size

- I In the Model Builder window, under Component 2 (comp2) right-click Mesh 2 and choose Edit Physics-Induced Sequence.
- 2 In the Model Builder window, under Component 2 (comp2)>Mesh 2 click Size.
- 3 In the Settings window for Size, locate the Element Size section.

- 4 Click the **Custom** button.
- 5 Locate the Element Size Parameters section. In the Maximum element size text field, type 0.05.
- 6 In the Maximum element growth rate text field, type 1.12.
- 7 In the Resolution of narrow regions text field, type 5.

Size 1

- I In the Model Builder window, under Component 2 (comp2)>Mesh 2 click Size I.
- 2 In the Settings window for Size, locate the Element Size section.
- **3** Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. Select the **Resolution of narrow regions** check box.
- **5** In the associated text field, type **5**.
- 6 Click the **Build Selected** button.

Size 2

- I In the Model Builder window, right-click Mesh 2 and choose Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 Click the **Custom** button.
- 4 Locate the Geometric Entity Selection section. From the Geometric entity level list, choose Boundary.
- **5** Select Boundaries 1, 13, and 14 only.
- 6 Locate the Element Size Parameters section. Select the Maximum element size check box.
- 7 In the associated text field, type 0.01.
- 8 Select the Maximum element growth rate check box.
- 9 In the associated text field, type 1.04.

Free Triangular 1

- I In the Model Builder window, under Component 2 (comp2)>Mesh 2 click Free Triangular I.
- 2 In the Settings window for Free Triangular, click to expand the Scale geometry section.
- 3 Locate the Scale Geometry section. In the z-direction scale text field, type 0.5.

Boundary Layer Properties 1

I In the Model Builder window, expand the Boundary Layers I node, then click Boundary Layer Properties I.

- 2 In the Settings window for Boundary Layer Properties, locate the Boundary Layer Properties section.
- 3 In the Number of boundary layers text field, type 6.
- 4 Click the **Build All** button.

Solve the fully developed turbulent pipe flow set up in Component I.

STUDY I

Solution 1 (soll)

- I On the Study toolbar, click Show Default Solver.
- 2 In the Model Builder window, expand the Solution I (soll) node.
- 3 Right-click Solution I (soll) and choose Compute.

Use the second study to solve the axially symmetric jet flow in **Component 2**. This study solves a reacting isothermal jet using the fully developed turbulent outlet profiles as inlet conditions for the pipe.

STUDY 2

Step 1: Stationary

I In the Model Builder window, under Study 2 click Step I: Stationary.

2 In the Settings window for Stationary, locate the Physics and Variables Selection section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Turbulent Flow, k-ω {spf}		physics
Heat Transfer in Fluids {ht}		physics

- 4 Click to expand the Values of dependent variables section. Locate the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 5 From the Method list, choose Solution.
- 6 From the Study list, choose Study I, Stationary.

Now solve the reacting isothermal jet. The complicated reactions require adjustment of the CFL-number controller parameters.

Solution 2 (sol2)

I On the Study toolbar, click Show Default Solver.

- 2 In the Model Builder window, expand the Solution 2 (sol2) node.
- 3 In the Model Builder window, expand the Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver I node, then click Segregated I.
- 4 In the Settings window for Segregated, locate the General section.
- 5 In the Initial CFL number text field, type 1.
- 6 In the PID regulator-Proportional text field, type 0.5.
- 7 In the PID regulator-Derivative text field, type 0.01.
- 8 In the Target error estimate text field, type 0.05.
- 9 In the Model Builder window, right-click Solution 2 (sol2) and choose Compute.

Add a third study to include the temperature and solve for a non-isothermal reacting jet. For a robust and efficient solution, use two study steps. Start by solving for the temperature using the velocity and mass fractions from the isothermal jet solution. Then in the second step, solve for the fully coupled non-isothermal reacting flow with the previous solution as initial condition.

ADD STUDY

- I On the Study toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select study tree, select Preset Studies>Stationary.
- 4 Find the Physics interfaces in study subsection. In the table, enter the following settings:

Physics	Solve
Turbulent Flow, k- ω (spf)	
Reacting Flow (rspf)	

- 5 Click Add Study in the window toolbar.
- 6 On the Study toolbar, click Add Study to close the Add Study window.

STUDY 3

Step 1: Stationary

- I In the Model Builder window, under Study 3 click Step I: Stationary.
- **2** In the **Settings** window for Stationary, click to expand the **Values of dependent variables** section.
- **3** Locate the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.

- 4 From the Method list, choose Solution.
- 5 From the Study list, choose Study 2, Stationary.

Stationary 2

On the Study toolbar, click Study Steps and choose Stationary>Stationary.

Step 2: Stationary 2

- I In the Settings window for Stationary, locate the Physics and Variables Selection section.
- 2 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Turbulent Flow, k-ω {spf}		physics

- 3 Click to expand the Values of dependent variables section. Locate the Values of Dependent Variables section. Find the Initial values of variables solved for subsection. From the Settings list, choose User controlled.
- 4 From the Method list, choose Solution.
- 5 From the Study list, choose Study 3, Stationary.
- 6 Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 7 From the Method list, choose Solution.
- 8 From the Study list, choose Study 3, Stationary.

Solution 3 (sol3)

I On the Study toolbar, click Show Default Solver.

Before solving, disable the automatic generation of default plot groups. The existing ones will be used to post process the non-isothermal results.

- 2 In the Settings window for Study, locate the Study Settings section.
- **3** Clear the **Generate convergence plots** check box.
- 4 In the Model Builder window, click Study 3.
- 5 In the Settings window for Study, locate the Study Settings section.
- 6 Clear the Generate default plots check box.

Now solve the non-isothermal reacting case.

7 On the Study toolbar, click Compute.

Now move on to post process the result from the non-isothermal jet. Start by creating a mirrored 2D data set as well as a revolved 3D data set.

RESULTS

Mirror 2D I

On the Results toolbar, click More Data Sets and choose Mirror 2D.

Data Sets

- I In the Settings window for Mirror 2D, locate the Data section.
- 2 From the Data set list, choose Study 3/Solution 3 (6) (sol3).
- 3 In the Model Builder window, under Results>Data Sets click Revolution 2D 2.
- 4 In the Settings window for Revolution 2D, click to expand the Revolution layers section.
- 5 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type 180.
- 6 Right-click Results>Data Sets>Revolution 2D 2 and choose Duplicate.
- 7 In the Settings window for Revolution 2D, locate the Data section.
- 8 From the Data set list, choose Study 3/Solution 3 (6) (sol3).

Create two cut lines at fixed heights from the pipe exit.

Cut Line 2D I

On the **Results** toolbar, click **Cut Line 2D**.

Data Sets

- I In the Settings window for Cut Line 2D, locate the Data section.
- 2 From the Data set list, choose Mirror 2D I.
- **3** Locate the Line Data section. From the Line entry method list, choose Point and direction.
- 4 Find the **Point** subsection. In the **y** text field, type P1+20*Di.
- 5 Click to expand the Advanced section. Find the Space variable subsection. In the x text field, type r_mirr20.
- 6 Right-click Cut Line 2D I and choose Duplicate.
- 7 In the Settings window for Cut Line 2D, locate the Line Data section.
- 8 Find the **Point** subsection. In the **y** text field, type P1+50*Di.
- 9 Locate the Advanced section. Find the Space variable subsection. In the x text field, type r_mirr50.

Now apply the mirror data set to the existing plot groups.

Velocity (rspf)

- I In the Model Builder window, under Results click Velocity (rspf).
- 2 In the Settings window for 2D Plot Group, locate the Data section.

- 3 From the Data set list, choose Mirror 2D I.
- 4 On the Velocity (rspf) toolbar, click Plot.
- 5 In the Model Builder window, expand the Velocity (rspf) node.
- 6 Right-click Results>Velocity (rspf) and choose Streamline.
- 7 In the Settings window for Streamline, locate the Streamline Positioning section.
- 8 From the Positioning list, choose Uniform density.
- **9** In the **Separating distance** text field, type **0.035**.
- 10 Locate the Coloring and Style section. From the Color list, choose Gray.
- II On the Velocity (rspf) toolbar, click Plot.
- 12 Click the Zoom Extents button on the Graphics toolbar.

Pressure (rspf)

- I In the Model Builder window, under Results click Pressure (rspf).
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Data set list, choose Mirror 2D I.
- 4 On the Pressure (rspf) toolbar, click Plot.

Wall Resolution (rspf)

- I In the Model Builder window, under Results click Wall Resolution (rspf).
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Data set list, choose Mirror 2D I.
- 4 On the Wall Resolution (rspf) toolbar, click Plot.

Mass Fraction (rspf)

- I In the Model Builder window, under Results click Mass Fraction (rspf).
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Data set list, choose Mirror 2D I.
- 4 Locate the Plot Settings section. From the Color list, choose White.
- 5 On the Mass Fraction (rspf) toolbar, click Plot.
- 6 In the Model Builder window, expand the Mass Fraction (rspf) node, then click Surface I.
- 7 In the Settings window for Surface, locate the Expression section.
- 8 In the **Expression** text field, type wC02.
- 9 On the Mass Fraction (rspf) toolbar, click Plot.
- **IO** Click the **Zoom Extents** button on the **Graphics** toolbar.

Velocity, 3D (rspf)

- I In the Model Builder window, under Results click Velocity, 3D (rspf).
- 2 In the Settings window for 3D Plot Group, locate the Plot Settings section.
- 3 Clear the **Plot data set edges** check box.

Velocity, 3D (rspf) I

- I Right-click Results>Velocity, 3D (rspf) and choose Duplicate.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Data set list, choose Revolution 2D 3.
- 4 Right-click Results>Velocity, 3D (rspf) I and choose Rename.
- 5 In the Rename 3D Plot Group dialog box, type Temperature, 3D (rspf) in the New label text field.
- 6 Click OK.

Temperature, 3D (rspf)

- I In the Model Builder window, expand the Results>Temperature, 3D (rspf) node, then click Surface 1.
- 2 In the Settings window for Surface, locate the Expression section.
- **3** In the **Expression** text field, type T.
- 4 Locate the Coloring and Style section. From the Color table list, choose RainbowLight.
- 5 In the Model Builder window, under Results click Temperature, 3D (rspf).
- 6 On the Temperature, 3D (rspf) toolbar, click Plot.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Import the experimental data files. The files corresponds to the ones published online (Ref. 4) by R. Barlow and co-workers. The name of the model round_jet_burner has been prepended to the file names.

Table I

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- **4** Browse to the application's Application Library folder and double-click the file round_jet_burner_chnAclY.fav.

TABLE

I Go to the Table window.

- 2 Right-click Table I and choose Rename.
- 3 In the Rename Table dialog box, type Centerline data in the New label text field.
- 4 Click OK.

RESULTS

Table 2

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- 4 Browse to the application's Application Library folder and double-click the file round_jet_burner_chnAd20Y.fav.

TABLE

- I Go to the Table window.
- 2 Right-click Table 2 and choose Rename.
- **3** In the **Rename Table** dialog box, type z/Di = 20, radial data in the **New label** text field.
- 4 Click OK.

RESULTS

Table 3

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- 4 Browse to the application's Application Library folder and double-click the file round_jet_burner_chnAd50Y.fav.

TABLE

- I Go to the Table window.
- 2 Right-click Table 3 and choose Rename.
- **3** In the **Rename Table** dialog box, type z/Di = 50, radial data in the **New label** text field.
- 4 Click OK.

RESULTS

Table 4

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- 4 Browse to the application's Application Library folder and double-click the file round_jet_burner_seq1420.dat.

TABLE

- I Go to the Table window.
- 2 Right-click Table 4 and choose Rename.
- 3 In the **Rename Table** dialog box, type z/Di = 20, radial velocity data in the **New** label text field.
- 4 Click OK.

RESULTS

Table 5

- I On the **Results** toolbar, click **Table**.
- 2 In the Settings window for Table, locate the Data section.
- 3 Click Import.
- 4 Browse to the application's Application Library folder and double-click the file round_jet_burner_seq1450.dat.

TABLE

- I Go to the Table window.
- 2 Right-click Table 5 and choose Rename.
- 3 In the **Rename Table** dialog box, type z/Di = 50, radial velocity data in the **New label** text field.
- 4 Click OK.

RESULTS

ID Plot Group II

- I On the **Results** toolbar, click **ID Plot Group**.
- 2 In the Settings window for 1D Plot Group, locate the Data section.

3 From the Data set list, choose Study 3/Solution 3 (6) (sol3).

Line Graph 1

On the ID Plot Group II toolbar, click Line Graph.

ID Plot Group II

- I Select Boundary 1 only.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type T/T0.
- 4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- **5** In the **Expression** text field, type (z-P1)/Di.
- 6 Click to expand the Coloring and style section. Locate the Coloring and Style section. From the Color list, choose Black.
- 7 Click to expand the Legends section. Select the Show legends check box.
- 8 From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

Legends

Model

IO In the **Model Builder** window, click **ID Plot Group 11**.

Table Graph 1

On the ID Plot Group II toolbar, click Table Graph.

ID Plot Group II

- I In the Settings window for Table Graph, locate the Data section.
- 2 From the x-axis data list, choose r(mm).
- 3 From the Plot columns list, choose Manual.
- 4 In the Columns list, select T(K).
- 5 Click to expand the **Preprocessing** section. Find the **x-axis column** subsection. From the **Preprocessing** list, choose **Linear**.
- 6 In the Scaling text field, type 1/(Di*1000).
- 7 Find the y-axis columns subsection. From the Preprocessing list, choose Linear.
- 8 In the Scaling text field, type 1/T0.
- **9** Locate the Coloring and Style section. Find the Line style subsection. From the Line list, choose None.
IO From the **Color** list, choose **Black**.

II Find the Line markers subsection. From the Marker list, choose Square.

12 From the **Positioning** list, choose **In data points**.

I3 Click to expand the **Legends** section. Select the **Show legends** check box.

14 From the Legends list, choose Manual.

I5 In the table, enter the following settings:

Legends

Exp.

- 16 Right-click 1D Plot Group 11 and choose Rename.
- 17 In the Rename ID Plot Group dialog box, type T @ centerline in the New label text field.

I8 Click **OK**.

- 19 In the Settings window for 1D Plot Group, locate the Plot Settings section.
- **20** Select the **x-axis label** check box.
- **2** In the associated text field, type (z-P1)/Di.
- **22** Select the **y-axis label** check box.
- **23** In the associated text field, type T/TO.
- 24 Locate the Axis section. Select the Manual axis limits check box.
- **25** In the **x minimum** text field, type 10.
- **26** In the **x maximum** text field, type 120.
- **27** In the **y minimum** text field, type 0.5.
- **28** In the **y maximum** text field, type 8.
- **29** Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- **30** In the **Title** text area, type Temperature along the centerline.
- **3I** On the **T @** centerline toolbar, click **Plot**.

ID Plot Group 12

- I On the Home toolbar, click Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose None.

Line Graph 1

On the ID Plot Group 12 toolbar, click Line Graph.

ID Plot Group 12

- I In the Settings window for Line Graph, locate the Data section.
- 2 From the Data set list, choose Cut Line 2D I.
- 3 Locate the y-Axis Data section. In the Expression text field, type T/T0.
- 4 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- **5** In the **Expression** text field, type r_mirr20/Di.
- 6 Click to expand the Coloring and style section. Locate the Coloring and Style section. From the Color list, choose Black.
- 7 Click to expand the Legends section. Select the Show legends check box.
- 8 From the Legends list, choose Manual.
- **9** In the table, enter the following settings:

Legends

z/Di = 20, Model

IO Right-click **Line Graph I** and choose **Duplicate**.

II In the Settings window for Line Graph, locate the Data section.

12 From the Data set list, choose Cut Line 2D 2.

- **I3** Locate the **x-Axis Data** section. In the **Expression** text field, type r_mirr50/Di.
- 14 Locate the Coloring and Style section. Find the Line style subsection. From the Line list, choose Dashed.
- 15 Locate the Legends section. In the table, enter the following settings:

Legends

z/Di = 50, Model

16 On the 1D Plot Group 12 toolbar, click Plot.

I7 In the Model Builder window, click **ID** Plot Group **I2**.

Table Graph 1

On the ID Plot Group I2 toolbar, click Table Graph.

ID Plot Group 12

- I In the Settings window for Table Graph, locate the Data section.
- 2 From the Table list, choose z/Di = 20, radial data.
- 3 From the x-axis data list, choose r(mm).
- 4 From the Plot columns list, choose Manual.

- 5 In the Columns list, select T(K).
- 6 Click to expand the **Preprocessing** section. Find the **x-axis column** subsection. From the **Preprocessing** list, choose **Linear**.
- 7 In the Scaling text field, type 1/(Di*1000).
- 8 Find the y-axis columns subsection. From the Preprocessing list, choose Linear.
- 9 In the Scaling text field, type 1/T0.
- **10** Click to expand the **Coloring and style** section. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- II From the Color list, choose Black.
- 12 Find the Line markers subsection. From the Marker list, choose Square.
- **I3** From the **Positioning** list, choose **In data points**.
- 14 Locate the Legends section. Select the Show legends check box.
- **I5** From the Legends list, choose Manual.
- **I6** In the table, enter the following settings:

Legends

z/Di = 20, Exp

I7 Right-click **Table Graph I** and choose **Duplicate**.

- 18 In the Settings window for Table Graph, locate the Data section.
- **I9** From the **Table** list, choose **z/Di = 50**, **radial data**.
- **20** Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.
- 21 Locate the Legends section. In the table, enter the following settings:

Legends

z/Di = 50, Exp

- **22** In the Model Builder window, right-click **ID Plot Group 12** and choose Rename.
- **23** In the **Rename ID Plot Group** dialog box, type T @ z/Di = 20, 50 in the **New label** text field.

24 Click OK.

- 25 In the Settings window for 1D Plot Group, locate the Title section.
- **26** From the **Title type** list, choose **Manual**.
- 27 In the Title text area, type Temperature downstream of the pipe exit.

- 28 Locate the Plot Settings section. Select the x-axis label check box.
- **29** In the associated text field, type r/Di.
- **30** Select the **y-axis label** check box.
- **3I** In the associated text field, type T/T0.
- **32** Locate the **Axis** section. Select the **Manual axis limits** check box.
- **33** In the **x minimum** text field, type 10.
- **34** In the **x maximum** text field, type 10.
- **35** In the **y minimum** text field, type 0.5.
- **36** In the **y maximum** text field, type 8.
- **37** On the **T** @ **z/Di = 20, 50** toolbar, click **Plot**.
- T @ z/Di = 20, 50.1
- I In the Model Builder window, right-click T @ z/Di = 20, 50 and choose Duplicate.
- 2 Right-click T @ z/Di = 20, 50.1 and choose Rename.
- 3 In the Rename ID Plot Group dialog box, type uz @ z/Di = 20, 50 in the New label text field.
- 4 Click OK.
- uz @ z/Di = 20, 50
- I In the Model Builder window, expand the Results>uz @ z/Di = 20, 50 node, then click Line Graph I.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type u21z/Ujet.
- 4 In the Model Builder window, under Results>uz @ z/Di = 20, 50 click Line Graph 2.
- 5 In the Settings window for Line Graph, locate the y-Axis Data section.
- 6 In the Expression text field, type u21z/Ujet.
- 7 In the Model Builder window, under Results>uz @ z/Di = 20, 50 click Table Graph I.
- 8 In the Settings window for Table Graph, locate the Data section.
- 9 From the x-axis data list, choose Fblgr.
- **IO** From the **Table** list, choose **z/Di = 20**, **radial velocity data**.
- II In the Columns list, select uz.
- 12 Locate the Preprocessing section. Find the y-axis columns subsection. From the Preprocessing list, choose Linear.
- **I3** In the **Scaling** text field, type 1/Ujet.

- I4 In the Model Builder window, under Results>uz @ z/Di = 20, 50 click Table Graph 2.
- 15 In the Settings window for Table Graph, locate the Data section.
- **I6** From the **x-axis data** list, choose **Fblgr**.
- **I7** From the **Table** list, choose **z/Di = 50**, **radial velocity data**.
- **18** In the **Columns** list, select **uz**.
- 19 Locate the Preprocessing section. Find the y-axis columns subsection. From the Preprocessing list, choose Linear.
- 20 In the Scaling text field, type 1/Ujet.
- 2I In the Model Builder window, click uz @ z/Di = 20, 50.
- **2** In the **Settings** window for 1D Plot Group, locate the **Title** section.
- **23** In the **Title** text area, type Axial velocity downstream of the pipe exit.
- 24 Locate the Plot Settings section. In the y-axis label text field, type uz/Ujet.
- **25** Locate the **Axis** section. In the **x minimum** text field, type -10.
- **26** In the **x maximum** text field, type **10**.
- **27** In the **y minimum** text field, type -0.25.
- **28** In the **y maximum** text field, type **1.25**.
- **29** On the **uz @ z/Di = 20, 50** toolbar, click **Plot**.
- T @ centerline 1
- I In the Model Builder window, under Results right-click T @ centerline and choose Duplicate.
- 2 Right-click T @ centerline I and choose Rename.
- 3 In the Rename ID Plot Group dialog box, type CO, N2 @ centerline in the New label text field.
- 4 Click OK.
- 5 In the Settings window for 1D Plot Group, locate the Title section.
- 6 In the Title text area, type Mass fraction along the centerline.
- 7 Locate the Plot Settings section. In the y-axis label text field, type wCO, wN2.
- 8 Locate the Axis section. In the y minimum text field, type -0.05.
- 9 In the y maximum text field, type 1.
- IO On the CO, N2 @ centerline toolbar, click Plot.
- II Click to expand the Legend section. From the Position list, choose Middle right.

CO, N2 @ centerline

- I In the Model Builder window, expand the Results>CO, N2 @ centerline node, then click Line Graph I.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- 3 In the Expression text field, type wCO.
- **4** Locate the **Legends** section. In the table, enter the following settings:

Legends

CO, Model

- 5 In the Model Builder window, under Results>CO, N2 @ centerline click Table Graph I.
- 6 In the Settings window for Table Graph, locate the Data section.
- 7 In the Columns list, select YCO.
- 8 Locate the **Preprocessing** section. Find the **y-axis columns** subsection. In the **Scaling** text field, type 1.
- 9 Locate the Legends section. In the table, enter the following settings:

Legends

CO, Exp.

- IO On the CO, N2 @ centerline toolbar, click Plot.
- II In the Model Builder window, under Results>CO, N2 @ centerline right-click Line Graph I and choose Duplicate.
- 12 In the Settings window for Line Graph, locate the y-Axis Data section.
- **I3** In the **Expression** text field, type wN2.
- 14 Click to expand the Coloring and style section. Locate the Coloring and Style section. Find the Line style subsection. From the Line list, choose Dashed.
- **I5** Locate the **Legends** section. In the table, enter the following settings:

Legends

N2, Model

- I6 In the Model Builder window, under Results>CO, N2 @ centerline right-click Table GraphI and choose Duplicate.
- 17 In the Settings window for Table Graph, locate the Data section.
- **I8** In the **Columns** list, select **YN2**.

- **19** Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Triangle**.
- **20** Locate the **Legends** section. In the table, enter the following settings:

Legends

N2, Exp

21 On the CO, N2 @ centerline toolbar, click Plot.

- CO, N2 @ centerline 1
- I In the Model Builder window, right-click CO, N2 @ centerline and choose Duplicate.
- 2 Right-click CO, N2 @ centerline I and choose Rename.
- 3 In the Rename ID Plot Group dialog box, type H2, H20 @ centerline in the New label text field.
- 4 Click OK.

H2, H2O @ centerline

- I In the Model Builder window, expand the Results>H2, H20 @ centerline node, then click Line Graph I.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type wH2.
- 4 Locate the Legends section. In the table, enter the following settings:

Legends

H2, Model

- 5 In the Model Builder window, under Results>H2, H20 @ centerline click Table Graph I.
- 6 In the Settings window for Table Graph, locate the Data section.
- 7 In the Columns list, select YH2.
- 8 Locate the Legends section. In the table, enter the following settings:

Legends

H2, Exp.

9 In the Model Builder window, under Results>H2, H20 @ centerline click Line Graph 2.

- 10 In the Settings window for Line Graph, locate the y-Axis Data section.
- II In the **Expression** text field, type wH20.

12 Locate the **Legends** section. In the table, enter the following settings:

Legends

H2O, Model

13 In the Model Builder window, under Results>H2, H20 @ centerline click Table Graph 2.

14 In the Settings window for Table Graph, locate the Data section.

I5 In the Columns list, select YH2O.

I6 Locate the **Legends** section. In the table, enter the following settings:

Legends

H2O, Exp

17 In the Model Builder window, click H2, H20 @ centerline.

18 In the Settings window for 1D Plot Group, locate the Plot Settings section.

- **I9** In the **y-axis label** text field, type wH2, wH20.
- **20** Locate the **Axis** section. In the **y maximum** text field, type **0.15**.
- **2I** In the **y minimum** text field, type -0.02.
- **2** Locate the Legend section. From the Position list, choose Upper right.
- 23 On the H2, H20 @ centerline toolbar, click Plot.

H2, H2O @ centerline I

- I In the Model Builder window, right-click H2, H20 @ centerline and choose Duplicate.
- 2 Right-click H2, H20 @ centerline I and choose Rename.
- 3 In the Rename ID Plot Group dialog box, type 02, CO2 @ centerline in the New label text field.
- 4 Click OK.

02, CO2 @ centerline

- I In the Model Builder window, expand the Results>02, CO2 @ centerline node, then click Line Graph I.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type w02.
- **4** Locate the **Legends** section. In the table, enter the following settings:

Legends

02, Model

5 In the Model Builder window, under Results>02, CO2 @ centerline click Table Graph 1.

6 In the Settings window for Table Graph, locate the Data section.

7 In the Columns list, select YO2.

8 Locate the Legends section. In the table, enter the following settings:

Legends

02, Exp.

9 In the Model Builder window, under Results>02, CO2 @ centerline click Line Graph 2.

10 In the Settings window for Line Graph, locate the y-Axis Data section.

II In the **Expression** text field, type wC02.

12 Locate the Legends section. In the table, enter the following settings:

Legends

CO2, Model

I3 In the Model Builder window, under Results>02, CO2 @ centerline click Table Graph 2.

14 In the Settings window for Table Graph, locate the Data section.

I5 In the **Columns** list, select **YCO2**.

I6 Locate the **Legends** section. In the table, enter the following settings:

Legends

CO2, Exp

- 18 In the Settings window for 1D Plot Group, locate the Plot Settings section.
- **19** In the **y-axis label** text field, type w02, wC02.
- **20** Locate the **Axis** section. In the **y minimum** text field, type -0.05.
- **2I** In the **y maximum** text field, type 0.4.
- 22 On the 02, CO2 @ centerline toolbar, click Plot.

¹⁷ In the Model Builder window, click 02, CO2 @ centerline.

44 | SYNGAS COMBUSTION IN A ROUND-JET BURNER



Transonic Flow in a Sajben Diffuser

Introduction

In the present example the high speed turbulent gas flow in a converging and diverging nozzle is modeled using the High Mach Number Flow interface. The diffuser is transonic in the sense that the flow at the inlet is subsonic, but due to the contraction and the low outlet pressure, the flow accelerates and becomes sonic (Ma = 1) in the throat of the nozzle. After a short region of supersonic flow, a normal shock wave brings the flow back to subsonic flow. This setup has been studied in a number of experiments and numerical simulations by M. Sajben and co-workers (see for example Ref. 1, Ref. 2, Ref. 3, and Ref. 4), in an effort to study unsteady fluctuations in supersonic inlets with applications in supersonic aircraft propulsion systems. The geometry and setup is often referred to as a Sajben diffuser and constitutes a common test case for the simulation of high Mach number internal flows. In this example, the time-averaged transonic flow through a Sajben diffuser is solved for using two different exit pressures. The flow in the diffuser is fully turbulent with a inlet Reynolds number of 7×10^5 based on the inlet fluid properties and the channel height. The model uses the Spalart-Allmaras turbulence model to compute the turbulent viscosity. For the first outlet pressure value a normal shock is present, but the flow remains attached throughout the diverging part. For the second, lower, outlet pressure value, the shock is strong enough to cause a shock-induced separation in the diverging part. Based on the ability to induce flow separation, the shock in the first case is referred to as weak, while in the second case it is termed strong following the definition in Ref. 2.

Model Definition

Figure 1 shows the physical geometry of the converging and diverging nozzle model. The nozzle dimensions correspond to those used in the experiments in Ref. 2 and in the benchmarks simulation of Ref. 5 and Ref. 6. In the central contraction part, the minimum vertical height separating the lower and upper walls, the throat height $h_{\rm th}$, is 1.7322 in (44 mm). The channel height at the inlet is $1.4h_{\rm th}$, and the outlet height is $1.5h_{\rm th}$.



Figure 1: Geometry and dimensions of the Sajben diffuser model.

FLUID PROPERTIES

The fluid occupying the channel is assumed to be air, by specifying a specific gas constant of 287 J/(kg·K) and a ratio of specific heats of 1.4. The dynamic viscosity is computed using Sutherland's Law:

$$\mu = \mu_{\text{ref}} \left(\frac{T}{T_{\text{ref}}}\right)^{\frac{3}{2}} \frac{T_{\text{ref}} + S_{\mu}}{T + S_{\mu}}$$

where the T_{ref} = 500 R (about 278 K) corresponds to the inlet total temperature, and the Sutherland constant is S_{μ} = 111 K. The reference viscosity μ_{ref} is defined from the inlet Reynolds number:

$$\operatorname{Re}_{\operatorname{in}} = \frac{\rho_{\operatorname{in}} U_{\operatorname{in}} h_{\operatorname{in}}}{\mu_{\operatorname{ref}}}$$

which in turn is used in a parametric sweep, where the model is solved for increasing Reynolds number. The final inlet Reynolds number to be computed is 7×10^5 ,

corresponding to the one used in the simulation in Ref. 5. The thermal conductivity of the gas is defined using the definition of the Prandtl number

$$\Pr = \frac{C_p \mu}{k}$$

which is assumed to be 0.71 in this case. This is a typical number for air around 293 K.

BOUNDARY CONDITIONS

Inlet Condition

The flow at the inlet is subsonic with a flow speed corresponding to a Mach number of 0.46. The inlet conditions are specified in terms of total properties, where the total pressure is defined as 19.58 psi and the total temperature is 500 R. The inlet conditions are applied using an Inlet feature, where the Flow condition is specified to be Characteristics based. This provides a numerically consistent boundary condition by evaluating the flow characteristics at the inlet (for more background on this boundary condition, see the *CFD Module User's Guide*).

Outlet Condition

At the outlet the static pressure is specified. The model is solved for the two outlet pressure values specified in Table 1 below.

TABLE	1:	OUTLET	PRESSURE

PRESSURE	FRACTION OF INLET TOTAL PRESSURE	DESCRIPTION
16.05 psi	0.82	Case I: weak shock
14.10 psi	0.72	Case 2: strong shock

These outlet pressure values are known from experiments and simulations to be low enough to produce sonic conditions at the throat of the nozzle. However, they are not low enough for the flow to stay supersonic throughout the diverging part. The supersonic flow in the divergent part is terminated by a normal shock wave, so that the flow in the following part including the outlet becomes subsonic. The pressure is specified in the model using an Outlet node with the Flow condition set to Subsonic.

Results and Discussion

Below, some of the results from the transonic diffuser model computed in COMSOL Multiphysics are shown and discussed. The results are compared to experimental data from Ref. 2 (strong shock case) and Ref. 4 (weak shock case). The experimental data was extracted as tabulated data from Ref. 5 and Ref. 6 and plotted in COMSOL using

interpolation functions.

Figure 2 shows the Mach numbers and velocity streamlines resulting from applying the first outlet pressure, 16.05 psi. It can be seen that the flow accelerates in the converging part, reaches sonic conditions at the throat, after which a region of supersonic flow follows in the diverging part. The supersonic region is terminated by a normal shock wave, which brings the flow back to subsonic conditions. In the remaining part of the channel the flow decelerates subsonically toward the outlet. The zero contour of the *x*-component velocity is also plotted in the figure, but this is only present on the walls and not visible inside the domain. Hence no separation zone is present, and the flow remains attached throughout the divergent part of the channel. The shock is not able to cause flow separation and is therefore termed weak. These results correlate well with those in Ref. 2 and Ref. 5.



Figure 2: Mach number, flow streamlines, and zero x-component velocity contour resulting from the weak shock case.

Figure 3 shows the same quantities as Figure 2, but uses the results from the second outlet pressure case, $p_{out} = 14.10$ psi. Due to the lower outlet pressure, the normal shock wave is positioned further downstream in the divergent channel part. More importantly, a flow separation zone can be seen behind the shock, as indicated by the zero *x*-component velocity contour. The shock wave in this case is apparently strong enough to induce flow



separation. This result is in accordance with those presented in Ref. 2 and Ref. 6.

Figure 3: Mach number, flow streamlines, and zero x-component velocity contour resulting from the strong shock case.

Figure 4 shows the development of the static pressure on the upper wall normalized by the inlet total pressure. Results from both the weak and strong cases are plotted and compared with the experimental data of Ref. 1 and Ref. 2. The results from both outlet cases are in general in very good agreement with the experimental results in the diffuser. Note however that the shock positions in the model are slightly shifted in the downstream direction in comparison with the experiments.

For analysis of the results in the interior of the channel, Figure 5 plots the streamwise velocity profiles from the strong shock case at two different positions in the divergent part of the channel together with experimental results. The velocity profile at the first position, $x = 4.611h_{\text{th}}$, compares very well with the experimental results. Both the velocity magnitude and the size of the separation zone, including reversed flow, are accurately reproduced. Further downstream, at the $x = 6.340h_{\text{th}}$ position, the velocity magnitude in the central part of the channel is also in good agreement with the experimental results. Closer to the upper wall, the model results include flow reversal at this position. This is not found in the experimental result, indicating that the separation zone in the model extends further downstream than that in the experiment.



Figure 4: Top wall static pressure normalized by the inlet total pressure. Model results (lines) and experimental results (diamonds) are shown.



Figure 5: Mean x-component velocity at two positions downstream of the strong shock. Model results (lines) and experimental results (diamonds) are shown.

Notes About the COMSOL Implementation

The present model is highly nonlinear and sensitive to the solution procedure. The sensitivity is accentuated by the fact that the model seeks to determine the equilibrium position of a normal shock wave, positioned in a channel with smoothly varying channel height.

You solve the model in two steps. First, apply the higher outlet pressure to simulate the weak shock case. To solve this case, use a parametric sweep where the inlet Reynolds number increases stepwise by decreasing the dynamic viscosity. When you have obtained a converged result for the highest Reynolds number, use this solution as the initial condition

for the second case with the lower outlet pressure value. In both cases, use pseudo-time stepping with manually defined CFL number expressions to compute stationary solutions.

References

1. M. Sajben, J.C. Kroutil, and C.P. Chen, "A High-Speed Schlieren Investigation of Diffuser Flows with Dynamic Distortion", *AIAA Paper 77-875*, 1977.

2. T.J. Bogar, M. Sajben, and J.C. Kroutil, "Characteristic Frequencies of Transonic Diffuser Flow Oscillations," *AIAA Journal*, vol. 21, no. 9, pp. 1232–1240, 1983.

3. J.T. Salmon, T.J. Bogar, and M. Sajben, "Laser Doppler Velocimetry in Unsteady, Separated, Transonic Flow," *AIAA Journal*, vol. 21, no. 12, pp. 1690–1697, 1983.

4. T. Hsieh, A.B. Wardlaw Jr., T.J. Bogar, P. Collins, and T. Coakley, "Numerical Investigation of Unsteady Inlet Flowfields," *AIAA Journal*, vol. 25, no. 1, pp. 75–81, 1987.

5. NPARC Alliance Validation Archive, "Sajben Transonic Diffuser: Study #1,", 2008, http://www.grc.nasa.gov/WWW/wind/valid/transdif/transdif01/transdif01.html

6. NPARC Alliance Validation Archive, "Sajben Transonic Diffuser: Study #2,", 2008, http://www.grc.nasa.gov/WWW/wind/valid/transdif/transdif02/transdif02.html

Application Library path: CFD_Module/High_Mach_Number_Flow/sajben_diffuser

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 2 In the Select Physics tree, select Fluid Flow>High Mach Number Flow>Turbulent Flow>High Mach Number Flow, Spalart-Allmaras (hmnf).
- 3 Click Add.
- 4 Click Study.

5 In the Select Study tree, select Preset Studies>Stationary with Initialization.

6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
Rein	7e5	7E5	Inlet Reynolds number
case	1	1	Case number; 1 = weak shock, 2 = strong shock
x0	-6.99809[in]	-0.1778 m	Inlet x-position
xEnd	14.98353[in]	0.3806 m	Outlet x-position
h_in	2.44483[in]	0.0621 m	Diffuser inlet height
h_out	2.59830[in]	0.066 m	Diffuser outlet height
h_th	1.732[in]	0.04399 m	Throat height

Interpolation 1 (int1)

I On the Home toolbar, click Functions and choose Global>Interpolation.

2 In the Settings window for Interpolation, locate the Definition section.

- 3 From the Data source list, choose File.
- 4 Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file
top_pos	1

- 5 Click Browse.
- 6 Browse to the application's Application Libraries folder and double-click the file sajben_diffuser_upper_wall.txt.
- 7 Click Import.
- 8 Locate the Units section. In the Function text field, type in.

Interpolation 2 (int2)

I On the Home toolbar, click Functions and choose Global>Interpolation.

- 2 In the Settings window for Interpolation, locate the Definition section.
- 3 From the Data source list, choose File.
- 4 Find the Functions subsection. In the table, enter the following settings:

Function name	Position in file
ptop_weak	1

- 5 Click Browse.
- 6 Browse to the application's Application Libraries folder and double-click the file sajben_diffuser_ptop_weak.txt.
- 7 Click Import.

Interpolation 3 (int3)

- I On the Home toolbar, click Functions and choose Global>Interpolation.
- 2 In the Settings window for Interpolation, locate the Definition section.
- 3 From the Data source list, choose File.
- 4 Find the Functions subsection. In the table, enter the following settings:

Function name	Position in file	
ptop_strong	1	

- 5 Click Browse.
- 6 Browse to the application's Application Libraries folder and double-click the file sajben_diffuser_ptop_strong.txt.
- 7 Click Import.

Interpolation 4 (int4)

- I On the Home toolbar, click Functions and choose Global>Interpolation.
- 2 In the Settings window for Interpolation, locate the Definition section.
- 3 From the Data source list, choose File.
- 4 Find the Functions subsection. In the table, enter the following settings:

Function name	Position in file
u_at4611	1

- 5 Click Browse.
- 6 Browse to the application's Application Libraries folder and double-click the file sajben_diffuser_u-xh4611.txt.

7 Click Import.

Interpolation 5 (int5)

- I On the Home toolbar, click Functions and choose Global>Interpolation.
- 2 In the Settings window for Interpolation, locate the Definition section.
- 3 From the Data source list, choose File.
- **4** Find the **Functions** subsection. In the table, enter the following settings:

Function name	Position in file	
u_at6340	1	

- 5 Click Browse.
- 6 Browse to the application's Application Libraries folder and double-click the file sajben_diffuser_u-xh6340.txt.
- 7 Click Import.

DEFINITIONS

Variables I

- I On the Home toolbar, click Variables and choose Local Variables.
- 2 In the Settings window for Variables, locate the Variables section.

Name	Expression	Unit	Description
Min	0.46		Inlet Mach number
rhoin	pin_stat/Rs/Tin_stat	kg/m³	Inlet density
Tin_tot	500[R]	К	Inlet total temperature
Tin_stat	Tin_tot/(1+0.5*Min^2* (-1+gamma))	к	Inlet static temperature
pin_tot	19.58[psi]	Pa	Inlet total pressure
pin_stat	<pre>pin_tot/(1+0.5*Min^2* (-1+gamma))^(gamma/ (-1+gamma))</pre>	Pa	Inlet static pressure
mu_ref	rhoin*u_in*h_in/Rein	kg/(m·s)	Reference dynamic viscosity
u_in	Min*sqrt(gamma*Rs* Tin_stat+eps)	m/s	Inlet velocity
pOut	if(case==1,16.05, 0)[psi]+if(case==2, 14.1,0)[psi]	Pa	Outlet pressure
CFLnum	if(case==1,CFLweak, 0)+if(case==2, CFLstrong,0)		CFL number for pseudo time stepping
CFLweak	1.3^min(niterCMP-1, 9)+if(niterCMP>25,5* 1.2^min(niterCMP-26, 12),0)		CFL number, weak case
CFLstrong	<pre>1+if(niterCMP>10, 1.2^min(niterCMP-10, 12),0)+if(niterCMP> 120, 1.3^min(niterCMP-120, 9),0)+if(niterCMP> 220, 1.3^min(niterCMP-220, 9),0)</pre>		CFL number, strong case
Rs	287[J/kg/K]	J/(kg·K)	Specific gas constant
gamma	1.4		Ratio of specific heats
Pr	0.72		Prandtl number

3 In the table, enter the following settings:

The manual CFL number expression for the strong shock corresponds to the implemented automatic expression for turbulent flows. In this case the solution already contains a shock that will move due to the change in the outlet pressure, and a cautious increase of the CFL number is needed. In the weak case simulation, a shock is not yet formed, and the simulation time can be reduced by using a more aggressive ramping of the CFL number.

GEOMETRY I

Parametric Curve 1 (pc1)

- I On the Geometry toolbar, click Primitives and choose Parametric Curve.
- 2 In the Settings window for Parametric Curve, locate the Parameter section.
- **3** In the **Minimum** text field, type x0[1/in].
- **4** In the **Maximum** text field, type xEnd[1/in].
- **5** Locate the **Expressions** section. In the **x** text field, type **s**[in].
- 6 In the y text field, type top_pos(s).

Bézier Polygon I (b1)

- I Right-click Parametric Curve I (pcl) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 4 Find the Added segments subsection. Click Add Linear.
- 5 Find the Control points subsection. In row I, set x to x0 and y to h_in.
- 6 In row 2, set x to x0.
- 7 Right-click Bézier Polygon I (bI) and choose Build Selected.
- 8 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 9 Find the Added segments subsection. Click Add Linear.
- **IO** Find the **Control points** subsection. In row **2**, set **x** to **xEnd**.
- II Find the Added segments subsection. Click Add Linear.
- 12 Find the Control points subsection. In row 2, set y to h_out.
- **I3** Locate the **General** section. From the **Type** list, choose **Open curve**.

Convert to Solid 1 (csol1)

- I Right-click Bézier Polygon I (b1) and choose Build Selected.
- 2 On the Geometry toolbar, click Conversions and choose Convert to Solid.

3 Click in the Graphics window and then press Ctrl+A to select both objects.

4 Right-click Convert to Solid I (csoll) and choose Build Selected.

Add a rectangular domain in the divergent part of the nozzle. This will be used to increase the resolution in the region where the shock is located.

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type 0.16.
- 4 In the **Height** text field, type 0.1.
- 5 Locate the **Position** section. In the **x** text field, type 0.025.

Compose I (col)

- I Right-click Rectangle I (rI) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Compose.
- 3 Click in the Graphics window and then press Ctrl+A to select both objects.
- 4 In the Settings window for Compose, locate the Compose section.
- 5 In the Set formula text field, type r1*csol1+csol1.
- 6 Right-click Compose I (col) and choose Build Selected.

Add a Mesh Control Edges feature to specify the internal boundaries as mesh control entities. In this manner these entities can be used to control the mesh, but at the same time they will automatically be omitted when defining the physics and when postprocessing results.

Mesh Control Edges 1 (mcel)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Edges.
- 2 On the object fin, select Boundaries 3 and 5 only.
- 3 On the Geometry toolbar, click Build All.

HIGH MACH NUMBER FLOW, SPALART-ALLMARAS (HMNF)

Fluid I

- I In the Model Builder window, expand the High Mach Number Flow, Spalart-Allmaras (hmnf) node, then click Fluid I.
- 2 In the Settings window for Fluid, locate the Heat Conduction section.

3 From the k list, choose User defined. In the associated text field, type hmnf.Cp* hmnf.mu/Pr.

Here the conductivity is defined using a constant Prandtl number.

- 4 Locate the Thermodynamics section. From the R_s list, choose User defined. In the associated text field, type Rs.
- 5 From the Specify Cp or γ list, choose Ratio of specific heats.
- **6** From the γ list, choose **User defined**. In the associated text field, type gamma.
- 7 Locate the Dynamic Viscosity section. In the μ_{ref} text field, type mu_ref.
- 8 In the $T_{\mu, ref}$ text field, type Tin_stat.

Initial Values 1

- I In the Model Builder window, under Component I (compl)>High Mach Number Flow, Spalart-Allmaras (hmnf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** Specify the **u** vector as

u_in x 0 y

- 4 In the *p* text field, type pin_stat.
- 5 In the *nutilde* text field, type subst(hmnf.nutildeinit,p,pin_stat).

This ensures that when evaluating the initial condition for nutilde, the pressure used corresponds to pin_stat.

6 In the T text field, type Tin_stat.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Flow Properties section.
- 4 From the Input state list, choose Total.
- **5** In the *p*_{0.tot} text field, type pin_tot.
- **6** In the $T_{0,tot}$ text field, type Tin_tot.
- 7 In the Ma_0 text field, type Min.

Outlet I

I On the Physics toolbar, click Boundaries and choose Outlet.

- **2** Select Boundary **3** only.
- 3 In the Settings window for Outlet, locate the Flow Condition section.
- 4 From the Flow condition list, choose Subsonic.
- 5 Locate the Flow Properties section. From the Boundary condition list, choose Pressure.
- **6** In the p_0 text field, type pOut.

CFL number

To apply the manually defined CFL number, first enable the Advanced Physics Options.

- I In the Model Builder window's toolbar, click the Show button and select Advanced Physics Options in the menu.
- 2 In the Model Builder window, click High Mach Number Flow, Spalart-Allmaras (hmnf).
- **3** In the **Settings** window for High Mach Number Flow, Spalart-Allmaras, click to expand the **Advanced settings** section.
- **4** Locate the **Advanced Settings** section. Find the **Pseudo time stepping** subsection. From the **CFL number expression** list, choose **Manual**.
- **5** In the CFL_{loc} text field, type CFLnum.

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Mapped I

In the Model Builder window, under Component I (compl)>Mesh I right-click Mapped I and choose Distribution.

Distribution I

- I Select Boundaries 4 and 6 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the Distribution properties list, choose Predefined distribution type.
- **4** In the **Number of elements** text field, type **40**.
- 5 In the Element ratio text field, type 1/4.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 2

- I Select Boundaries 5 and 7 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 90.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 3

- I Select Boundaries 2 and 8 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 50.
- 5 In the Element ratio text field, type 3.
- 6 Select the **Reverse direction** check box.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 4

- I Select Boundaries 1 and 3 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 25.
- 5 In the Element ratio text field, type 2.5.
- 6 Click Build All.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Boundary Layers 1

I In the Model Builder window, right-click Mesh I and choose Boundary Layers.

In this case the mesh transition region, between the boundary layer and the interior mesh, is explicitly controlled by the specified distributions. The default mesh smoothing of the transition region can hence be disabled.

- 2 In the Settings window for Boundary Layers, click to expand the Transition section.
- **3** Clear the **Smooth transition to interior mesh** check box.

Boundary Layer Properties

- I In the Model Builder window, under Component I (comp1)>Mesh I>Boundary Layers I click Boundary Layer Properties.
- 2 Select Boundaries 2, 4, 6, and 8–10 only.
- **3** In the **Settings** window for Boundary Layer Properties, locate the **Boundary Layer Properties** section.
- 4 In the Number of boundary layers text field, type 20.
- 5 In the Thickness adjustment factor text field, type 0.11.
- 6 Click Build All.

STUDY I

Parametric Sweep

- I On the Study toolbar, click Parametric Sweep.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click Add.
- **4** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
case	1	

Step 2: Stationary

Set up an auxiliary continuation sweep for the 'Rein' parameter.

- I In the Model Builder window, under Study I click Step 2: Stationary.
- 2 In the Settings window for Stationary, click to expand the Study extensions section.
- 3 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 4 Click Add.
- **5** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Rein	5e3 5e4 2e5 7e5	

6 On the Study toolbar, click Compute.

RESULTS

Velocity (hmnf)

To reproduce the plot in Figure 2 perform the steps below.

- I In the Model Builder window, expand the Velocity (hmnf) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 In the **Expression** text field, type hmnf.Ma.
- 4 In the Model Builder window, right-click Velocity (hmnf) and choose Streamline.
- 5 In the Settings window for Streamline, locate the Expression section.
- 6 In the **x** component text field, type u.
- 7 In the y component text field, type v.
- 8 Select Boundary 1 only.
- 9 Locate the Streamline Positioning section. In the Number text field, type 9.
- **IO** Right-click **Velocity (hmnf)** and choose **Contour**.
- II In the Settings window for Contour, locate the Expression section.
- **12** In the **Expression** text field, type u.
- **I3** Locate the **Levels** section. From the **Entry method** list, choose **Levels**.
- 14 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- **I5** From the **Color** list, choose **White**.
- **I6** Clear the **Color legend** check box.
- **I7** On the **Velocity (hmnf)** toolbar, click **Plot**.
- **I8** Click the **Zoom Extents** button on the **Graphics** toolbar.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- **3** Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies**>**Stationary with Initialization**.
- 4 Click Add Study in the window toolbar.
- 5 On the Home toolbar, click Add Study to close the Add Study window.

STUDY 2

Parametric Sweep

- I On the Study toolbar, click Parametric Sweep.
- 2 In the Settings window for Parametric Sweep, locate the Study Settings section.
- 3 Click Add.
- **4** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
case	2	

Solution 5 (sol5)

I On the Study toolbar, click Show Default Solver.

Before computing the solution for the strong shock case, apply the last solution from the weak shock case as initial value.

- 2 In the Model Builder window, expand the Study 2>Solver Configurations node.
- **3** In the Model Builder window, expand the Solution **5** (sol5) node, then click Dependent Variables **2**.
- 4 In the Settings window for Dependent Variables, locate the General section.
- **5** From the **Defined by study step** list, choose **User defined**.
- 6 Locate the Initial Values of Variables Solved For section. From the Solution list, choose Solution 1 (soll).
- 7 From the Parameter value (Rein) list, choose 7E5.
- 8 On the Study toolbar, click Compute.

RESULTS

Velocity (hmnf) 1

To reproduce the plot in Figure 3 perform the steps below.

- I In the Model Builder window, expand the Velocity (hmnf) I node, then click Surface.
- 2 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>High Mach Number Flow, Spalart-Allmaras>Velocity and pressure>hmnf.Ma - Mach number.
- 3 On the Velocity (hmnf) I toolbar, click Plot.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.
- 5 In the Model Builder window, right-click Velocity (hmnf) I and choose Streamline.

- 6 In the Settings window for Streamline, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>High Mach Number Flow, Spalart-Allmaras (Turbulent Flow, Spalart-Allmaras)>Velocity and pressure>u,v Velocity field.
- 7 Select Boundary 1 only.
- 8 Locate the Streamline Positioning section. In the Number text field, type 9.
- 9 Right-click Velocity (hmnf) I and choose Contour.
- In the Settings window for Contour, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I>High Mach Number Flow, Spalart-Allmaras (Turbulent Flow, Spalart-Allmaras)>Velocity and pressure>Velocity field> u Velocity field, x component.
- II Locate the Levels section. From the Entry method list, choose Levels.
- 12 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- **I3** From the **Color** list, choose **White**.
- 14 Clear the Color legend check box.
- IS On the Velocity (hmnf) I toolbar, click Plot.
- I6 Click the Zoom Extents button on the Graphics toolbar.

Data Sets

Create cut line data sets to plot results at two downstream positions in the diverging part of the nozzle.

Cut Line 2D I

On the **Results** toolbar, click **Cut Line 2D**.

Data Sets

- I In the Settings window for Cut Line 2D, locate the Data section.
- 2 From the Data set list, choose Study 2/Solution 5 (sol5).
- 3 Locate the Line Data section. In row Point I, set x to 4.611*h_th.
- 4 In row **Point 2**, set **x** to 4.611*h_th and **y** to 2*h_th.
- 5 Click Plot.

The position of the cut line is indicated with a red line.

Cut Line 2D 2

On the **Results** toolbar, click **Cut Line 2D**.

Data Sets

- I In the Settings window for Cut Line 2D, locate the Data section.
- 2 From the Data set list, choose Study 2/Solution 5 (sol5).
- 3 Locate the Line Data section. In row Point 1, set x to 6.340*h_th.
- 4 In row **Point 2**, set **x** to 6.340*h_th and **y** to 2*h_th.
- 5 Click Plot.

The following steps reproduce the normalized static pressure plots in Figure 4.

ID Plot Group II

- I On the **Results** toolbar, click **ID Plot Group**.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Parameter selection (Rein) list, choose Last.

Line Graph 1

On the ID Plot Group II toolbar, click Line Graph.

ID Plot Group II

- I Select Boundary 4 only.
- 2 In the Settings window for Line Graph, locate the y-Axis Data section.
- **3** In the **Expression** text field, type p/pin_tot.
- 4 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Component I>Geometry>Coordinate>x x-coordinate**.
- 5 Right-click Line Graph I and choose Duplicate.
- 6 In the Settings window for Line Graph, locate the y-Axis Data section.
- 7 In the Expression text field, type ptop_weak(x/h_th).
- 8 Click to expand the Coloring and style section. Locate the Coloring and Style section. Find the Line style subsection. From the Line list, choose None.
- 9 From the Color list, choose Black.
- 10 Find the Line markers subsection. From the Marker list, choose Diamond.
- II In the **Number** text field, type **30**.
- 12 In the Model Builder window, click ID Plot Group II.
- **I3** In the **Settings** window for 1D Plot Group, click to expand the **Title** section.
- **I4** From the **Title type** list, choose **Manual**.
- **I5** In the **Title** text area, type Weak shock.
- **I6** Click to expand the **Axis** section. Select the **Manual axis limits** check box.

- **I7** In the **x minimum** text field, type -0.2.
- **I8** In the **x maximum** text field, type **0.4**.
- **I9** In the **y minimum** text field, type **0.25**.
- **20** In the **y maximum** text field, type **1**.
- 21 Click to expand the Grid section. Select the Manual spacing check box.
- **22** In the **x spacing** text field, type 0.05.
- **23** In the **y spacing** text field, type **0.1**.
- 24 On the ID Plot Group II toolbar, click Plot.

Compare the result with that in the left panel of Figure 4.

To reproduce the plot in the right panel, use the plot you just created as the starting point.

ID Plot Group 12

- I Right-click ID Plot Group II and choose Duplicate.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Study 2/Solution 5 (sol5).
- 4 Locate the **Title** section. In the **Title** text area, type **Strong** shock.
- 5 In the Model Builder window, expand the ID Plot Group 12 node, then click Line Graph 2.
- 6 In the Settings window for Line Graph, locate the y-Axis Data section.
- 7 In the **Expression** text field, type ptop_strong(x/h_th).
- 8 On the ID Plot Group 12 toolbar, click Plot.
- 9 In the Model Builder window, click ID Plot Group 12.
- IO Click Plot.

Compare with the right panel of Figure 4.

Similarly, reproduce the x-velocity plots in Figure 5.

ID Plot Group 13

- I On the Home toolbar, click Add Plot Group and choose ID Plot Group.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Cut Line 2D I.

Line Graph 1

On the ID Plot Group 13 toolbar, click Line Graph.

ID Plot Group 13

- In the Settings window for Line Graph, click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component I>High Mach Number Flow, Spalart-Allmaras (Turbulent Flow, Spalart-Allmaras)>Velocity and pressure>Velocity field> u Velocity field, x component.
- 2 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 3 In the **Expression** text field, type y/0.0617.
- 4 In the Model Builder window, click ID Plot Group 13.

Line Graph 2

On the ID Plot Group 13 toolbar, click Line Graph.

ID Plot Group 13

- I In the Settings window for Line Graph, locate the y-Axis Data section.
- **2** In the **Expression** text field, type u_at4611(y/0.0617).
- 3 Locate the x-Axis Data section. From the Parameter list, choose Expression.
- 4 In the **Expression** text field, type y/0.0617.
- **5** Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 6 From the Color list, choose Black.
- 7 Find the Line markers subsection. From the Marker list, choose Diamond.
- 8 In the Number text field, type 30.
- 9 In the Model Builder window, click ID Plot Group 13.

10 In the Settings window for 1D Plot Group, locate the Title section.

- II From the **Title type** list, choose **Manual**.
- 12 In the Title text area, type $x/h_t = 4.611$.
- **I3** Locate the **Plot Settings** section. Select the **y-axis label** check box.
- **I4** In the associated text field, type u (m/s).
- 15 Locate the Axis section. Select the Manual axis limits check box.
- **I6** In the **x minimum** text field, type -0.1.
- **I7** In the **x maximum** text field, type 1.1.
- **18** In the **y minimum** text field, type 80.
- **I9** In the **y maximum** text field, type **320**.
- 20 Locate the Grid section. Select the Manual spacing check box.

- **2I** In the **x spacing** text field, type 0.05.
- **22** In the **y spacing** text field, type 20.
- 23 On the ID Plot Group I3 toolbar, click Plot.

ID Plot Group 14

- I Right-click ID Plot Group I3 and choose Duplicate.
- 2 In the Settings window for 1D Plot Group, locate the Data section.
- 3 From the Data set list, choose Cut Line 2D 2.
- 4 Locate the **Title** section. In the **Title** text area, type $x/h_t = 6.340$.
- 5 Locate the Plot Settings section. In the y-axis label text field, type u (m/s).
- 6 In the Model Builder window, expand the ID Plot Group 14 node, then click Line Graph 1.
- 7 In the Settings window for Line Graph, locate the x-Axis Data section.
- 8 In the **Expression** text field, type y/0.066.
- 9 In the Model Builder window, under Results>ID Plot Group 14 click Line Graph 2.
- 10 In the Settings window for Line Graph, locate the y-Axis Data section.
- **II** In the **Expression** text field, type u_at6340(y/0.066).
- 12 Locate the x-Axis Data section. In the Expression text field, type y/0.066.
- **I3** On the **ID Plot Group 14** toolbar, click **Plot**.


Contaminant-Removal from Wastewater in a Secondary Clarifier by Sedimentation

This model is licensed under the COMSOL Software License Agreement 5.2a. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Wastewater treatment is a several-step process for removing contaminants. First, large, solid particles are removed through sedimentation, flotation, and filtration. In a second step, biological treatment causes the smaller particles to aggregate, forming so-called flocs. These flocs can more easily be removed, for instance through sedimentation. The present example studies the separation of flocs from water in a circular secondary clarifier. To model the turbulent multiphase flow in the tank, this example uses the Mixture Model, Turbulent Flow interface.

Model Definition



The model geometry is shown in Figure 1.

Figure 1: Cross section of the circular clarifier.

The clarifier has a diameter of 24 m and a gently sloping bottom, which makes the dept vary between 3.3 and 4 m, attached to a funnel at the center of the tank. The incoming sludge, consisting of a mixture of solid flocs and water, enters through the inlet in the middle of the tank. In the clarifier, gravity causes the flocs to settle to the bottom of the tank. The tank contains two outlets. One is located at the center, at the bottom of the funnel. The purpose of this outlet is to remove the sedimented flocs from the tank. There



is also a peripheral outlet for the purified water as shown in the figure. Figure 2 shows the corresponding 2D axisymmetric model.

Figure 2: Axisymmetric representation of the clarifier geometry.

The mixture enters the clarifier in the form of a jet. The Reynolds number based on the inlet velocity and diameter is 5×10^5 , which means that the flow is turbulent. Upon impact with the free surface, the mixture spreads out, causing the turbulence production to diminish with radial distance from the inlet. The turbulent flow in the tank tends to mix the phases together, and thus has a negative effect on the separation process. The of this example is to study the turbulent multiphase flow within the circular secondary clarifier.

For simplicity, you can model the flocs as spherical solid particles of equal size. To solve for the mixture velocity, pressure and the phase volume fractions, you can use the Mixture Model, Turbulent Flow interface. The mixture model is a multiphase-flow model, particularly well suited for suspensions of solid particles in liquid at low particle volume fractions. For the slip velocity, you can use the Hadamard-Rybczynski drag law. See the theory in the *CFD Module User's Guide* for more information.

BOUNDARY CONDITIONS

At the inlet, the velocity is fixed to 1.25 m/s and the dispersed phase volume fraction is 0.003. The turbulence intensity and length scale are set to 5% and $0.07*r_{in}$ (see Inlet

Values for the Turbulence Length Scale and Turbulent Intensity in the *CFD Module User's Guide*), where r_{in} =0.2 m is the radius of the inlet. The velocity at the bottom outlet is set to 0.05 m/s, while a constant pressure is set at the peripheral outlet. A slip condition is applied on the free surface and an axial symmetry condition on the centerline.

INITIAL CONDITIONS

Initially, the velocity as well as the solid phase volume fraction is zero in the entire clarifier.

Results and Discussion

Figure 3 shows streamlines of the mixture velocity and the dispersed phase volume fraction after 12 hours, at which time the solution is close to steady state. Opposing effects of gravity settling and turbulence-induced particle dispersion produce volume-fraction gradients in the interior. The magnitude of the mixture strain rate (and hence the turbulence production) decreases with radial distance from the centerline, and at the peripheral outlet settling dominates over turbulent dispersion. This allows for a relatively clear efflux.



Figure 3: Mixture-velocity streamlines and solid phase volume fraction after 12 hours, when the flow has reached a steady state solution.

4 | CONTAMINANT-REMOVAL FROM WASTEWATER IN A SECONDARY CLARIFIER BY

As you can see in Figure 3, the maximum volume fraction is only about 4 %. Hence, the Mixture Model should be sufficiently accurate. See Two-Phase Flow Modeling of a Dense Suspension in the Application Libraries on how to modify the mixture model for high volume fractions or use the Euler-Euler model for such cases.

Figure 4 shows the dispersed phase mass flux at the inlet and the two outlets.



Figure 4: Mass flux of the dispersed phase at the inlet (blue), peripheral outlet (green) and central outlet (red).

The mass flux of the dispersed phase is given by

$$M_{\rm d} = \int \rho_{\rm d} \phi_{\rm d} \mathbf{u}_{\rm d} \cdot \mathbf{n} dS$$

Computing the removal rate from the results shows that the clarifier removes 0.52 - 0.10 = 0.42 kg solid particles per second. The separation efficiency is about 81%.

Figure 5 shows a cut through the swept-out volume of the clarifier with streamlines for each phase after 12 hours.



Figure 5: Volume fraction of the dispersed phase and streamlines for the dispersed (black) and continuous (white) velocity fields.

To further examine the performance of the clarifier, you can easily modify the model in several ways. You can, for instance, modify the geometry by adding baffles, changing the inlet and outlet velocities, increasing the dispersed-phase volume fraction in the incoming sludge, or changing the density and size of the dispersed particles.

Notes About the COMSOL Implementation

It is straightforward to set up a multiphase flow model with the Mixture Model, Turbulent Flow interface. To simplify the startup of the transient calculation, you can gradually increase the inlet and outlet velocities from zero to their constant values. For this purpose, use a Step function feature to implement a smooth step function that gradually increases from zero to one.

Application Library path: CFD_Module/Multiphase_Tutorials/sedimentation

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Mixture Model>Mixture Model, Laminar Flow (mm).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Time Dependent.
- 6 Click Done.

GEOMETRY I

Bézier Polygon I (b1)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Click Add Linear.
- 4 In row 2, set r to 12.
- 5 Click Add Linear.
- 6 In row 2, set z to -3.3.
- 7 Click Add Linear.
- 8 In row 2, set r to 2 and z to -4.
- 9 Click Add Linear.
- **IO** In row **2**, set **r** to **0.5** and **z** to **-7**.
- II Click Add Linear.
- **12** In row **2**, set **z** to -7.4.
- 13 Click Add Linear.
- **I4** In row **2**, set **r** to **0**.
- 15 Click Add Linear.
- I6 Click Close Curve.

Bézier Polygon 2 (b2)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Click Add Linear.
- 4 In row I, set z to -5.4.
- 5 In row 2, set r to 0.4 and z to -5.4.
- 6 Click Add Linear.
- 7 In row 2, set z to -3.4.
- 8 Click Add Quadratic.
- 9 In row 2, set z to -2.2.
- **10** In row **3**, set **r** to **1.6** and **z** to **-2.2**.
- II Click Add Linear.
- **12** In row **2**, set **z** to -2.
- **I3** Click **Add Quadratic**.
- **I4** In row **2**, set **r** to **0.2**.
- **I5** In row **3**, set **r** to **0.2** and **z** to **-3.4**.
- I6 Click Add Linear.
- **I7** In row **2**, set **z** to -5.2.
- **18** Click Add Linear.
- **I9** In row **2**, set **r** to **0**.
- 20 Click Add Linear.
- 21 Click Close Curve.

Circle I (cI)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.05.
- 4 Locate the **Position** section. In the **z** text field, type -3.4.

Square 1 (sq1)

- I Right-click Circle I (cl) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Square.
- 3 In the Settings window for Square, locate the Size section.

- 4 In the Side length text field, type 0.4.
- 5 Locate the Position section. From the Base list, choose Center.
- 6 In the r text field, type 11.4.
- 7 In the z text field, type -0.2.

Difference I (dif1)

- I Right-click Square I (sqI) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- **3** Select the object **b1** only to add it to the **Objects to add** list.
- 4 In the Settings window for Difference, locate the Difference section.
- 5 Find the Objects to subtract subsection. Select the Active toggle button.
- 6 Select the objects sql, cl, and b2 only.

Bézier Polygon 3 (b3)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Click Add Linear.
- 4 In row I, set r to 0.19 and z to -3.22.
- 5 In row 2, set r to 0.2 and z to -2.9.

Bézier Polygon 4 (b4)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Click Add Linear.
- 4 In row I, set r to 0.2 and z to -2.9.
- 5 In row 2, set r to 0.35 and z to -0.25.

Bézier Polygon 5 (b5)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Click Add Linear.
- 4 In row I, set r to 0.2 and z to -0.1.
- 5 In row 2, set r to 7.6 and z to -0.6.

Use Mesh Control Edges to obtain a mesh which is aligned with the turbulent shear.

Mesh Control Edges 1 (mcel)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Edges.
- 2 On the object fin, select Boundaries 8, 10, and 11 only.
- 3 On the Geometry toolbar, click Build All.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
rho_c	1000[kg/m^3]	1000 kg/m ³	Continuous phase density
mu_c	1e-3[Pa*s]	0.001 Pa·s	Continuous phase viscosity
rho_d	1100[kg/m^3]	1100 kg/m³	Dispersed phase density
d_d	0.2[mm]	2E-4 m	Dispersed phase particle diameter

DEFINITIONS

Add a **Step** function feature to use for implementing a gradual increase of the inlet and outlet velocities from zero to their constant values.

Step I (step I)

- I On the Home toolbar, click Functions and choose Local>Step.
- 2 In the Settings window for Step, click to expand the Smoothing section.
- 3 Locate the Parameters section. In the Location text field, type 0.1.
- **4** Locate the **Smoothing** section. In the **Size of transition zone** text field, type **0.2**.

5 Click Plot.



Variables 1



2 In the Settings window for Variables, locate the Variables section.

Name	Expression	Unit	Description
v_in	1.25*step1(t[1/ s])[m/s]	m/s	Inlet velocity
v_out	0.05*step1(t[1/ s])[m/s]	m/s	Outlet velocity
phid_in	0.003		Inlet dispersed phase volume fraction
qd_out	2*pi*r*(mm.udr*nr+ mm.udz*nz)*phid* mm.rhod	kg/(m·s)	Dispersed phase mass-outflux

3 In the table, enter the following settings:

MIXTURE MODEL, LAMINAR FLOW (MM)

- I In the Model Builder window, under Component I (compl) click Mixture Model, Laminar Flow (mm).
- **2** In the **Settings** window for Mixture Model, Laminar Flow, locate the **Physical Model** section.

- 3 From the Slip model list, choose Hadamard-Rybczynski.
- 4 From the Turbulence model type list, choose RANS, k-ε.

MIXTURE MODEL, TURBULENT FLOW (MM)

On the Physics toolbar, click Mixture Model, Laminar Flow (mm) and choose Mixture Model, Turbulent Flow (mm).

Mixture Properties I

- In the Model Builder window, expand the Component I (comp1)>Mixture Model, Turbulent
 Flow (mm) node, then click Mixture Properties 1.
- **2** In the **Settings** window for Mixture Properties, locate the **Continuous Phase Properties** section.
- **3** From the ρ_c list, choose **User defined**. In the associated text field, type rho_c.
- **4** From the μ_c list, choose **User defined**. In the associated text field, type mu_c.
- 5 Locate the Dispersed Phase Properties section. From the ρ_d list, choose User defined. In the associated text field, type rho_d.
- **6** In the d_d text field, type d_d.

Gravity I

- I On the Physics toolbar, click Domains and choose Gravity.
- 2 Select Domain 1 only.

Wall 2

- I On the Physics toolbar, click Boundaries and choose Wall.
- 2 Select Boundary 7 only.
- 3 In the Settings window for Wall, locate the Mixture Boundary Condition section.
- 4 From the Mixture boundary condition list, choose Slip.

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 5 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the U_0 text field, type v_in.
- **5** Locate the **Turbulence Properties** section. In the $L_{\rm T}$ text field, type 0.2*0.07.
- **6** Locate the **Dispersed Phase Boundary Condition** section. In the $\phi_{d, 0}$ text field, type phid_in.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- **2** Select Boundary 2 only.
- 3 In the Settings window for Outlet, locate the Mixture Boundary Condition section.
- 4 From the Mixture boundary condition list, choose Velocity.
- **5** Locate the **Velocity** section. In the U_0 text field, type v_out.

Outlet 2

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 17 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- 3 From the Element size list, choose Extra fine.
- 4 From the Sequence type list, choose User-controlled mesh.

Size 1

I In the Model Builder window, under Component I (compl)>Mesh I click Size I.

2 In the Settings window for Size, click Build Selected.

Size 2

- I In the Model Builder window, right-click Mesh I and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- 4 Select Boundaries 23–25 only.
- 5 Locate the Element Size section. From the Calibrate for list, choose Fluid dynamics.
- 6 From the Predefined list, choose Extremely fine.

Corner Refinement I

- I In the Model Builder window, under Component I (comp1)>Mesh I right-click Corner Refinement I and choose Disable.
- 2 In the Model Builder window, click Mesh I.
- 3 In the Settings window for Mesh, click Build All.

STUDY I

Step 1: Time Dependent

- I In the Model Builder window, expand the Study I node, then click Step I: Time Dependent.
- 2 In the Settings window for Time Dependent, locate the Study Settings section.
- 3 In the Times text field, type range(0,0.01,0.1) range(1,10) 100*range(1,10) 1800*range(1,24).
- 4 Select the **Relative tolerance** check box.
- **5** In the associated text field, type **0.01**.
- 6 Click to expand the **Results while solving** section. Locate the **Results While Solving** section. Select the **Plot** check box.

Solution I (soll)

I On the Study toolbar, click Show Default Solver.

BDF is the default time-dependent solver for CFD models. When solving a model with a large number of time steps, it can sometimes be more efficient to use the Generalized-alpha solver. However, the parameters have to be adjusted to obtain a stable setup for CFD.

- 2 In the Model Builder window, expand the Solution I (soll) node, then click Time-Dependent Solver I.
- **3** In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
- 4 Locate the Time Stepping section. From the Method list, choose Generalized alpha.
- **5** In the **Amplification for high frequency** text field, type **0.5**.
- 6 From the Predictor list, choose Constant.
- 7 Select the Initial step check box.
- 8 In the associated text field, type 0.001.

RESULTS

2D Plot Group 1

On the Home toolbar, click Add Plot Group and choose 2D Plot Group.

Surface 1

I In the Model Builder window, right-click 2D Plot Group I and choose Surface.

- In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (comp1)>Mixture Model, Turbulent Flow>phid Volume fraction, dispersed phase.
- 3 Click to expand the Range section. Select the Manual color range check box.
- **4** In the **Minimum** text field, type 0.
- 5 In the Maximum text field, type 0.006.
- 6 Click the Zoom Extents button on the Graphics toolbar.

2D Plot Group I

In the Model Builder window, under Results right-click 2D Plot Group I and choose Streamline.

Streamline 1

- I In the Settings window for Streamline, locate the Streamline Positioning section.
- 2 From the Positioning list, choose Uniform density.
- **3** In the **Separating distance** text field, type **0.02**.

MIXTURE MODEL, TURBULENT FLOW (MM)

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Mixture Model, Turbulent Flow (mm) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *p* text field, type -g_const*z*rho_c.

RESULTS

2D Plot Group 1

- I In the Model Builder window, under Results click 2D Plot Group I.
- 2 In the Settings window for 2D Plot Group, locate the Plot Settings section.
- 3 Click Go to Source.

STUDY I

On the Home toolbar, click Compute.

RESULTS

Derived Values

Calculate the inflow and outflow rates of the dispersed phase. Start with the inflow rate.

Line Integration 1

- I On the **Results** toolbar, click **More Derived Values** and choose **Integration>Line Integration**.
- **2** Select Boundary 5 only.
- 3 In the Settings window for Line Integration, locate the Expressions section.
- **4** In the table, enter the following settings:

Expression	Unit	Description
-qd_out	kg/s	

The negative sign is used since qd_out is defined as the outward flux.

5 Click **Evaluate**.

The result shown in **Table I**should be close to 0.52 kg/s.

Line Integration 2

- I On the Results toolbar, click More Derived Values and choose Integration>Line Integration.
- **2** Select Boundary 17 only.
- 3 In the Settings window for Line Integration, locate the Expressions section.
- **4** In the table, enter the following settings:

Expression	Unit	Description
qd_out	kg/s	Dispersed phase mass-outflux

5 Click Table I - Line Integration I (-qd_out).

The result should be close to 0.075 kg/s.

Line Integration 3

- I On the Results toolbar, click More Derived Values and choose Integration>Line Integration.
- **2** Select Boundary 2 only.
- 3 In the Settings window for Line Integration, locate the Expressions section.
- **4** In the table, enter the following settings:

Expression	Unit	Description
qd_out	kg/s	Dispersed phase mass-outflux

5 Click Table I - Line Integration I (-qd_out).

TABLE

- I Go to the Table window.
- 2 Click Table Graph in the window toolbar.

RESULTS

Study I/Solution I (soll)

In the Model Builder window, expand the Data Sets node, then click Study I/Solution I (soll).

Selection

- I On the Results toolbar, click Selection.
- 2 In the Settings window for Selection, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Domain.
- 4 Select Domain 1 only.

Revolution 2D 3

- I On the Results toolbar, click More Data Sets and choose Revolution 2D.
- 2 In the Settings window for Revolution 2D, locate the Revolution Layers section.
- 3 In the **Revolution angle** text field, type 270.

Edge 2D 2

- I On the Results toolbar, click More Data Sets and choose Edge 2D.
- **2** Select Boundaries 3, 5, 8–16, and 18–22 only.

Revolution 2D 4

- I On the Results toolbar, click More Data Sets and choose Revolution 2D.
- 2 In the Settings window for Revolution 2D, locate the Data section.
- 3 From the Data set list, choose Edge 2D 2.
- 4 Locate the **Revolution Layers** section. In the **Revolution angle** text field, type 270.
- 5 Click Plot.

3D Plot Group 3

- I On the Results toolbar, click 3D Plot Group.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Data set list, choose Revolution 2D 3.

Surface 1

I Right-click **3D Plot Group 3** and choose **Surface**.

- 2 In the Settings window for Surface, locate the Data section.
- 3 From the Data set list, choose Revolution 2D 4.
- **4** Locate the **Expression** section. In the **Expression** text field, type **1**.
- **5** Select the **Description** check box.
- 6 Clear the associated text field.
- 7 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 8 From the Color list, choose Gray.

3D Plot Group 3

In the Model Builder window, under Results right-click 3D Plot Group 3 and choose Volume.

Volume 1

- I In the Settings window for Volume, locate the Data section.
- 2 From the Data set list, choose Revolution 2D 3.
- **3** Locate the **Expression** section. Select the **Description** check box.
- 4 Click to expand the Range section. Select the Manual color range check box.
- **5** In the **Minimum** text field, type **0**.
- 6 In the Maximum text field, type 0.006.

3D Plot Group 3

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

Streamline 1

- I In the Settings window for Streamline, locate the Expression section.
- 2 In the x component text field, type mm.udr.
- **3** In the **y** component text field, type **0**.
- **4** In the **z** component text field, type mm.udz.
- **5** Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.
- 6 In the x text field, type range(0.01,0.02,0.19).
- 7 In the y text field, type 0.
- 8 In the z text field, type -1*1^range(1,10).
- 9 Locate the Coloring and Style section. From the Color list, choose Black.
- **IO** Locate the **Expression** section. Select the **Description** check box.
- II In the associated text field, type dispersed phase (black).

3D Plot Group 3

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

Streamline 2

- I In the Settings window for Streamline, locate the Expression section.
- 2 In the x component text field, type mm.ucr.
- **3** In the **y** component text field, type **0**.
- 4 In the z component text field, type mm.ucz.
- 5 Locate the Streamline Positioning section. From the Entry method list, choose Coordinates.
- 6 In the x text field, type range(0,0.02,0.2) range(0.5,0.5,12).
- 7 In the y text field, type 0.
- 8 In the z text field, type -1^range(1,35).
- 9 Locate the Coloring and Style section. From the Color list, choose White.
- IO On the 3D Plot Group 3 toolbar, click Plot.
- II Locate the **Expression** section. Select the **Description** check box.
- **12** In the associated text field, type continuous phase (white).

View 3D 2

- I In the Model Builder window, expand the Results>Views node, then click View 3D 2.
- 2 In the Settings window for View 3D, locate the View section.
- **3** Clear the **Show grid** check box.
- 4 Clear the Show axis orientation check box.

3D Plot Group 3

- I Click the Scene Light button on the Graphics toolbar.
- 2 In the Model Builder window, under Results click 3D Plot Group 3.
- 3 On the 3D Plot Group 3 toolbar, click Plot.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.



Bubble-induced Entrainment Between Stratified Liquid Layers

This model is licensed under the COMSOL Software License Agreement 5.2a. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Three-phase flows occur in numerous industrial applications, in free and porous media (nuclear safety, petroleum engineering, etc). Many biomedical and chemical processes involve mixtures of three or more fluids, for example double emulsions, which play critical roles in applications such as prolonged drug delivery systems, entrapment of vitamins and flavor encapsulation for food additives. Considering a configuration of two immiscible liquids in stratified layers, it is known that a gas bubble of sufficient size rising through the stratified layers can entrain some of the heavier liquid from the lower layer into its wake and transport it to the upper layer of lighter liquid. This entrainment phenomenon has significant effects on both heat and mass transfer, and is encountered in a number of industrial applications. For example, in chemical processing, gas bubbles are sometimes released into pools of stratified liquids to induce mixing and phase transport. In nuclear-reactor safety applications, bubbles of non-condensable gas rise through stratified liquid pools of molten fuel and materials. In metallurgical processing, bubbles of oxygen and other gases may bubble through pools of molten metal with overlying pools of molten silica slag (Ref. 1).

However, computational models for direct simulation of three-phase flow are rather rare in comparison with the large body of research on two-phase flow. The current model, based on the predefined multiphysics coupling between the Laminar flow and Ternary phase field interfaces, simulates three-phase flow (gas and two liquids) with large density and viscosity differences.

This benchmark model simulates a single gas bubble which rises through the interface between two stratified liquid layers thereby inducing entertainment, and validates the results against literature (Ref. 2).

Model Setup

When a gas bubble rises in a two-stratified-liquid-layers configuration, the bubble can either become captured at the interface, or penetrate into the light phase, possibly leading to entrainment of the heavy phase. The criterion is based on a macroscopic balance between buoyancy and surface tension forces, and states that the bubble will penetrate the liquid-liquid interface, if its volume, *V*, satisfies (Ref. 2),

$$V < V^{c} = \left[\frac{2\pi \left(\frac{3}{4\pi}\right)^{1/3}}{g(\rho_{3} - \rho_{1})}\right]^{\frac{3}{2}}$$
(1)

where 1, 2, 3 are the suffixes for gas, heavy liquid, and light liquid respectively. This criterion has been validated both experimentally and numerically (Ref. 1, Ref. 2).

This model is one of the case considered in Ref. 2. The physical properties of the fluids used in the simulation are listed in Table 1 and Table 2.

TABLE I: SURFACE TENSION BETWEEN THREE PHASES.

Parameters			Surface tension (N/m)
σ_{12} , σ_{13} (between the gas and the two liquids)			0.07
σ_{23} (between the two liquids)			0.05
TABLE 2: PHYSICAL PROPERTIES OF THREE PHASES.			
	Density (kg/m ³)	Vis	cosity (Pa·s)
Bubble	1	1e	- 4
Heavy liquid	1200	0.	15
Light liquid	1000	0.	1

Inserting the physical parameters above in Equation 1 gives $V^c = 8.8726 \cdot 10^{-8} \text{ kg/m}^3$. Because the volume of a sphere is given by $V = 4\pi r^3/3$, the critical bubble radius can be estimated to be $r^c = 2.778$ mm.

The radius of the bubble in the simulation is, r = 8 mm. Since $r > r^c$, the gas bubble will penetrate the liquid-liquid interface. Due to its relatively large volume, it will furthermore entrain some of the heavy liquid into its wake and transport it into the light liquid forming small droplets of heavy liquid in the upper layer. The mobility M_0 , which is taken as a function of the phase field variables, is defined in such a way that it becomes zero in each pure phase, $M_0 = M_{\text{const}}(\phi_A^2 \phi_B^2 + \phi_A^2 \phi_C^2 + \phi_A^2 \phi_B^2)$, where $M_{\text{const}} = 2 \cdot 10^{-5} \text{ m}^3/\text{s}$.

The model is axisymmetric; Figure 1 shows its geometry. The mesh contains 36155 elements with linear shape functions. The simulation time is 0.65 s.



Figure 1: Geometry of the computation domain.

Results and Discussion

The simulation results are shown in Figure 2. Initially, a gas bubble is placed underneath the surface between a heavy liquid and a light liquid, see Figure 2(a). The gas bubble then passes into the upper pool and a volume of heavy liquid is clearly seen to be entrained in its wake, having been pulled through the interfaces between the two liquid layers, see Figure 2(b). If the bubble were of insufficient size, the levitated column would fall back to the lower pool and the gas bubble would continue to rise through the upper pool without having caused any entrainment. This is not the case in current model. Here, the size of the bubble is sufficiently large that the levitated liquid column rises to a height such that it eventually pinches off from the lower layer, see Figure 2(c). As the column elongates, it becomes hydro-dynamically unstable and pinches off into a series of small droplets. A small amount of the heavy liquid adheres to the bubble, see Figure 2(d), and can be considered to have been successfully entrained into the upper fluid layer.



Figure 2: Processing of bubble-induced entrainment between stratified liquid layers. The silver and pink surfaces represent the interfaces between gas and light liquid and between light liquid and heavy liquid, respectively. The colored surface half-enclosing the domain represents the velocity field magnitude.



Figure 3: History of mass changes.

It is important that the mass of each phase is conserved to within a tolerance given by the numerical method and the computational mesh. The mass conservation history is shown in Figure 3. It is observed that the mass loss for the two liquid phases is about 0.1%, and that of the gas phase is about 1.5%. The mass loss of gas is the largest because its volume fraction is much smaller than the other two phases, whereby its relative change becomes most noticeable.

References

1. Bubble induced entrainment between stratified liquid layers, G. A. Green, J. C. Chen, and M. T. Conlin, *International Journal of Heat and Mass Transfer*, vol. 34, pp 149–157, 1991.

2. Cahn-Hilliard/Navier-Stokes Model for the Simulation of Three-Phase Flows, F. Boyer, C. Lapuerta, S. Minjeaud, B. Piar, and M. Quintard, *Transport in Porous Media*, vol. 82, pp 463–483, 2010.

Application Library path: CFD_Module/Multiphase_Benchmarks/ three_phase_bubble

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D Axisymmetric.
- 2 In the Select Physics tree, select Fluid Flow>Multiphase Flow>Three-Phase Flow, Phase Field>Laminar Three-Phase Flow, Phase Field.
- 3 Click Add.
- 4 Click Done.

GEOMETRY I

- I In the Model Builder window, click Geometry I.
- 2 In the Settings window for Geometry, locate the Units section.
- 3 From the Length unit list, choose cm.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** Click Load from File.
- 4 Browse to the application's Application Libraries folder and double-click the file three_phase_bubble_parameters.txt.

DEFINITIONS

In the Model Builder window, under Component I (compl) right-click Definitions and choose Variables.

Variables I

- I In the Settings window for Variables, locate the Variables section.
- 2 Click Load from File.
- **3** Browse to the application's Application Libraries folder and double-click the file three_phase_bubble_variables.txt.

GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- 3 In the Width text field, type width.
- 4 In the **Height** text field, type height.
- 5 Locate the Position section. From the Base list, choose Center.
- 6 In the z text field, type center_rec.

Circle I (c1)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type radius.
- 4 Locate the **Position** section. In the z text field, type center_bub.

Polygon I (poll)

- I On the Geometry toolbar, click Primitives and choose Polygon.
- 2 In the Settings window for Polygon, locate the Coordinates section.
- 3 In the r text field, type 0 radius*2.
- 4 In the z text field, type line_z line_z.

Form Union (fin)

On the Geometry toolbar, click Build All.

TERNARY PHASE FIELD (TERPF)

Mixture Properties 1

- I In the Model Builder window, under Component I (compl)>Ternary Phase Field (terpf) click Mixture Properties I.
- 2 In the Settings window for Mixture Properties, locate the Phase Field Parameters section.

- **3** In the M_0 text field, type MO.
- 4 Locate the Surface Tension section. From the Surface tension coefficient of interface between phase A and phase B list, choose User defined. In the $\sigma_{A, B}$ text field, type sigma_AB.
- 5 From the Surface tension coefficient of interface between phase B and phase C list, choose User defined. In the $\sigma_{B, C}$ text field, type sigma_BC.
- $\label{eq:constraint} \begin{array}{l} \textbf{6} \quad \mbox{From the Surface tension coefficient of interface between phase A and phase C list, choose} \\ \textbf{User defined. In the } \sigma_{A,\ C} \ text field, type \ \texttt{sigma_AC}. \end{array}$
- 7 In the Model Builder window, click Ternary Phase Field (terpf).

Initial Values 2

- I On the Physics toolbar, click Domains and choose Initial Values.
- 2 Select Domain 1 only.
- 3 In the Settings window for Initial Values, locate the Initial Values section.
- 4 In the *phiB* text field, type 1.

Initial Values 3

On the Physics toolbar, click Domains and choose Initial Values.

Initial Values 3

- I In the Model Builder window, under Component I (compl)>Ternary Phase Field (terpf) click Initial Values 3.
- 2 Select Domain 2 only.

Initial Values 1

- I In the Model Builder window, under Component I (compl)>Ternary Phase Field (terpf) click Initial Values I.
- 2 In the Settings window for Initial Values, locate the Initial Values section.
- **3** In the *phiA* text field, type 1.

MULTIPHYSICS

- I In the Model Builder window, expand the Component I (compl)>Multiphysics node, then click Three-Phase Flow, Phase Field I (tfpfl).
- **2** In the **Settings** window for Three-Phase Flow, Phase Field, locate the **Fluid A Properties** section.
- 3 From the ρ_A list, choose User defined. In the associated text field, type rho_A.
- **4** From the μ_A list, choose **User defined**. In the associated text field, type mu_A.

- 5 Locate the Fluid B Properties section. From the ρ_B list, choose User defined. In the associated text field, type rho_B.
- **6** From the μ_B list, choose **User defined**. In the associated text field, type mu_B.
- 7 Locate the Fluid C Properties section. From the ρ_C list, choose User defined. In the associated text field, type rho_C.
- **8** From the μ_C list, choose **User defined**. In the associated text field, type mu_C.

LAMINAR FLOW (SPF)

On the Physics toolbar, click Ternary Phase Field (terpf) and choose Laminar Flow (spf).

In the Model Builder window, expand the Component I (compl)>Laminar Flow (spf) node.

DEFINITIONS

Ramp I (rm I)

- I On the Home toolbar, click Functions and choose Global>Ramp.
- 2 In the Settings window for Ramp, locate the Parameters section.
- 3 In the Slope text field, type 200.
- 4 Select the **Cutoff** check box.
- 5 Click Plot.

LAMINAR FLOW (SPF)

- I In the Model Builder window, under Component I (compl) click Laminar Flow (spf).
- 2 In the Settings window for Laminar Flow, locate the Physical Model section.
- 3 Select the Include gravity check box.

Gravity I

- I In the Model Builder window, under Component I (compl)>Laminar Flow (spf) click Gravity I.
- 2 In the Settings window for Gravity, locate the Acceleration of Gravity section.
- **3** Specify the **g** vector as

0 r -g_const*rm1(t[1/s]) z

4 In the Model Builder window, click Laminar Flow (spf).

Pressure Point Constraint I

- I On the Physics toolbar, click Points and choose Pressure Point Constraint.
- 2 Select Point 7 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- **3** From the Sequence type list, choose User-controlled mesh.

Size

- I In the Model Builder window, under Component I (compl)>Mesh I click Size.
- 2 In the Settings window for Size, locate the Element Size section.
- 3 Click the **Custom** button.
- **4** Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type **0.04**.
- 5 Click Build All.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select Preset Studies>Time Dependent.
- 4 Click Add Study in the window toolbar.

STUDY I

Step 1: Time Dependent

- I In the Settings window for Time Dependent, locate the Study Settings section.
- 2 In the **Times** text field, type range(0,0.025,1)*0.65.

Solution 1 (soll)

On the Study toolbar, click Show Default Solver.

Step 1: Time Dependent

- I In the **Settings** window for Time Dependent, click to expand the **Results while solving** section.
- 2 Locate the **Results While Solving** section. Select the **Plot** check box.

- 3 From the Update at list, choose Time steps taken by solver.
- **4** Click to expand the **Values of dependent variables** section. Click to collapse the **Physics and variables selection** section.

Solution 1 (soll)

- I In the Model Builder window, expand the Study I>Solver Configurations>Solution I (soll) node, then click Time-Dependent Solver I.
- **2** In the **Settings** window for Time-Dependent Solver, click to expand the **Time stepping** section.
- 3 Locate the Time Stepping section. Select the Maximum step check box.
- 4 In the associated text field, type 0.0005.
- 5 In the Model Builder window, expand the Study I node.

RESULTS

Velocity (spf)

On the Study toolbar, click Get Initial Value.

STUDY I

Step 1: Time Dependent

- I In the **Settings** window for Time Dependent, click to expand the **Results while solving** section.
- 2 Locate the Results While Solving section. From the Plot group list, choose 2D Plot Group: Three Phases (terpf).
- 3 From the Update at list, choose Time steps taken by solver.
- 4 On the Study toolbar, click Compute.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Air.
- 4 Click Add to Component in the window toolbar.

RESULTS

Revolution 2D 2

On the Results toolbar, click More Data Sets and choose Revolution 2D.

3D Plot Group 8

- I On the **Results** toolbar, click **3D Plot Group**.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- 3 From the Data set list, choose Revolution 2D 2.

Isosurface 1

- I Right-click **3D Plot Group 8** and choose Isosurface.
- 2 In the Settings window for Isosurface, locate the Expression section.
- **3** In the **Expression** text field, type phiB.
- 4 Locate the Levels section. From the Entry method list, choose Levels.
- **5** In the **Levels** text field, type **0.5**.
- 6 Locate the Title section. From the Title type list, choose None.
- 7 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 8 Clear the **Color legend** check box.
- 9 From the Color list, choose Magenta.

3D Plot Group 8

In the Model Builder window, under Results right-click 3D Plot Group 8 and choose Isosurface.

Isosurface 2

- I In the Settings window for Isosurface, locate the Expression section.
- 2 In the **Expression** text field, type terpf.phiC.
- 3 Locate the Levels section. From the Entry method list, choose Levels.
- 4 In the Levels text field, type 0.5.
- 5 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- 6 Locate the Title section. From the Title type list, choose None.
- 7 Locate the Coloring and Style section. Clear the Color legend check box.
- 8 From the Color list, choose Gray.
- 9 On the 3D Plot Group 8 toolbar, click Plot.

3D Plot Group 8

Right-click **3D Plot Group 8** and choose Slice.

Slice 1

I In the Settings window for Slice, locate the Plane Data section.

- **2** In the **Planes** text field, type **1**.
- 3 From the Plane list, choose zx-planes.
- **4** In the **Planes** text field, type 1.

Deformation 1

- I Right-click Results>3D Plot Group 8>Slice I and choose Deformation.
- 2 In the Settings window for Deformation, locate the Expression section.
- **3** In the **x component** text field, type **0**.
- 4 In the y component text field, type sqrt(0.016^2-r^2).
- **5** In the **z** component text field, type **0**.
- 6 Locate the Scale section. Select the Scale factor check box.
- 7 In the associated text field, type 1.
- 8 On the 3D Plot Group 8 toolbar, click Plot.

3D Plot Group 8

- I In the Model Builder window, under Results click 3D Plot Group 8.
- 2 In the Settings window for 3D Plot Group, locate the Data section.
- **3** From the **Time (s)** list, choose **0**.
- 4 Locate the Color Legend section. Clear the Show legends check box.
- 5 Click to expand the Title section. From the Title type list, choose None.
- 6 On the 3D Plot Group 8 toolbar, click Plot.
- 7 In the Settings window for 3D Plot Group, locate the Data section.
- 8 From the Time (s) list, choose 0.2113.
- 9 On the 3D Plot Group 8 toolbar, click Plot.
- 10 In the Settings window for 3D Plot Group, locate the Data section.
- II From the Time (s) list, choose 0.4063.
- 12 On the 3D Plot Group 8 toolbar, click Plot.
- **I3** In the **Settings** window for 3D Plot Group, locate the **Data** section.
- I4 From the Time (s) list, choose 0.6013.
- **I5** On the **3D Plot Group 8** toolbar, click **Plot**.

Surface Integration 1

I On the Results toolbar, click More Derived Values and choose Integration>Surface Integration.

- 2 In the Settings window for Surface Integration, locate the Selection section.
- 3 From the Selection list, choose All domains.
- 4 Locate the Integration Settings section. Select the Compute volume integral check box.
- 5 In the Settings window for Surface Integration, locate the Expressions section.
- 6 In the table, enter the following settings:

Expression	Unit	Description
tfpf1.VfA/8.447E-5	m^3	
tfpf1.VfB/2.603e-5	m^3	
tfpf1.VfC/2.1451e-6	m^3	

7 Click Evaluate.

TABLE

- I Go to the Table window.
- 2 Click Table Graph in the window toolbar.

RESULTS

Table Graph 1

- I In the Model Builder window, under Results>ID Plot Group 9 click Table Graph I.
- 2 In the Settings window for Table Graph, locate the Legends section.
- 3 Select the Show legends check box.
- 4 From the Legends list, choose Manual.
- **5** In the table, enter the following settings:

Legends

Mass of light fluid Mass of heavy fluid Mass of gas


Turbulent Flow Over a Backward Facing Step

Introduction

0 -0.1 -0.2 -0.3 -0.4 -0.5 -0.6



The backward facing step has long been a central benchmark case in computational fluid

Figure 1: Backstep geometry. Dimensions in SI units.

0.2

0

-0.2

Fully developed channel flow enters at the domain from the left. When the flow reaches the step, it detaches and a recirculation zone is formed behind the step. Because of the expansion of the channel, the flow slows down and eventually reattaches. The flow field is

0.6

0.8

1

1.2

0.4

displayed in Figure 2.



Figure 2: Resulting flow field.

Though seemingly simple, the flow field is a challenge for turbulence models that utilize wall functions. The reason is that wall functions are derived by invoking equilibrium assumptions. Separation and reattachment do not adhere to these assumptions and it must therefore be asserted by numerical experiments that the wall functions can give accurate results even if the underlying theoretical assumptions are not strictly satisfied. The experiment is motivated by the fact that flow with separation and subsequent reattachment are of central importance in many engineering applications.

The model data is taken from Ref. 1. The parameters are given in Table 1. The Reynolds number based on V_{inl} and the step height, S, is $4.8 \cdot 10^4$ and the flow is therefore clearly turbulent.

Property	Value	Description
S	0.0381 m	Step height
h _c	2·S	Inlet channel height
Н	3·S	Outlet channel height
LI	0.3048 m	Inlet channel length
L2	1.3335 m	Outlet channel length
V _{inl}	18.2 m/s	Velocity at center of upstream channel
ρ	1.23 kg/m ³	Density
μ	1.79·10 ⁻⁵	Dynamic viscosity

TABLE I: MODEL PARAMETERS

You build the model in two steps:

- I Simulate flow in a long channel of the same height as the inlet to give inlet boundary conditions for the actual geometry.
- **2** Simulate the flow over the backward facing step using the inlet boundary condition from Step 1.

THE INLET CHANNEL

Ref. 1 suggests to simulate a channel that is $100 \cdot h_c$ in length. Because the channel is symmetric around the midplane, the geometry is taken to be a rectangle with lower left corner at (x, y) = (0, 0) and upper right corner at $(x, y) = (100 \cdot h_c, 0.5 \cdot h_c)$. The upper boundary at $y = 0.5 \cdot h_c$ is a symmetry plane and the lower boundary at y = 0 is the wall.

Inlet Boundary Conditions

At the inlet x = 0, a plug flow boundary condition with 3% turbulent intensity and a turbulent length scale according to the Theory for the Turbulent Flow Interfaces in the *CFD Module User's Guide* is prescribed. The inflow velocity cannot be set directly to 18.2 m/s since the resulting centerline velocity at the outlet then becomes too high. While it is possible to set up an ODE that automatically computes the appropriate inlet velocity, it is far easier for small models like this one to find it by trial and error. A few iterations reveal that an inlet velocity of 16.58 m/s gives a centerline value at the outlet very close to 18.2 m/s.

Outlet Boundary Conditions

Outlet boundary conditions can give local artifacts at the outlet. One possible strategy is to elongate the channel and extract data some distance before the outlet. That is however not necessary since fully developed flow in a channel only has a velocity component tangential to the wall, that is normal to the outflow. By prescribing that the outflow must have no tangential component, the outlet artifacts can be removed.

THE BACKWARD FACING STEP

There are two aspects of the backward facing step that need special consideration.

Mesh Generation

It is important to apply a fine enough mesh at the separation point to accurately capture the creation of the shear layer. It must also be remembered that both the flow field and turbulence variables can feature strong gradients close to the walls and that the mesh must be fine enough there to represent these gradients.

Solver Settings

The balance between the turbulence transport equations and the Navier-Stokes equations is rather delicate. If an iteration brings the flow into a state with unphysically large gradients, there is a considerable risk that the simulation diverges. It is therefore advisable to use the parametric solver to gradually increase the Reynolds number of the flow. The most robust way is to decrease the viscosity which is done in this model.

Results and Discussion

As shown in Figure 3, the recirculation length normalized by the step height becomes 6.73. Ref. 2 gives an experimental result of 7.1. The result provided by COMSOL is well within the range shown by other investigations (see Ref. 1 and Ref. 3). The separation lengths in Ref. 1 ranges between 6.12 and 7.24. In Ref. 3, recirculation lengths between 5.4 and 7.1 are obtained. Furthermore, Ref. 3 shows that the recirculation length can





Figure 3: Contour plot of streamwise velocity equal to zero, colored by x/S where S is the step height.

Finally, note that the recirculation length can shift quite significantly with the mesh resolution. The current result does not shift much if the mesh is refined, but coarser meshes can yield very different recirculation lengths. This emphasizes the need to ensure that the mesh is fine enough.

References

1. Ist NAFEMS Workbook of CFD Examples. Laminar and Turbulent Two-Dimensional Internal Flows, NAFEMS, 2000.

2. J. Kim, S.J. Kline, and J.P. Johnston, "Investigation of a Reattaching Turbulent Shear Layer: Flow Over a Backward Facing Step," *Transactions of the ASME*, vol. 102, p. 302, 1980.

3. D. Kuzmin, O. Mierka, and S. Turek, "On the Implementation of the k- ε Turbulence Model in Incompressible Flow Solvers Based on a Finite Element Discretization," *Int'l J Computing Science and Mathematics*, vol. 1, no. 2–4, pp. 193–206, 2007.

Application Library path: CFD_Module/Single-Phase_Benchmarks/ turbulent_backstep

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 2D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ε (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

Name	Expression	Value	Description
S	0.0381[m]	0.0381 m	Step height
hc	0.0762[m]	0.0762 m	Inlet channel height
Н	0.1143[m]	0.1143 m	Outlet channel height
L1	0.3048[m]	0.3048 m	Inlet channel length
L2	1.3335[m]	1.334 m	Outlet channel length

Name	Expression	Value	Description
Vinl	16.58[m/s]	16.58 m/s	Centerline inlet velocity
rhof	1.23[kg/m^3]	1.23 kg/m ³	Density
muf	1.79e-5[Pa*s]*fact	1.79E-5 Pa·s	Dynamic viscosity
fact	1.0	1	Viscosity scaling factor

GEOMETRY I

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type 100*L1.
- 4 In the **Height** text field, type hc/2.
- 5 On the Geometry toolbar, click Build All.

MATERIALS

In the Model Builder window, under Component I (compl) right-click Materials and choose Blank Material.

Material I (mat1)

- I In the Settings window for Material, locate the Material Contents section.
- 2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	rhof	kg/m³	Basic
Dynamic viscosity	mu	muf	Pa∙s	Basic

TURBULENT FLOW, K- ϵ (SPF)

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** In the U_0 text field, type Vin1.
- **5** Locate the **Turbulence Conditions** section. In the $L_{\rm T}$ text field, type 0.07*hc.
- **6** In the $I_{\rm T}$ text field, type 0.03.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- 2 Select Boundary 3 only.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 4 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

MESH I

In the Model Builder window, under Component I (compl) right-click Mesh I and choose Mapped.

Mapped I

In the Model Builder window, under Component I (compl)>Mesh I right-click Mapped I and choose Distribution.

Distribution I

- I Select Boundaries 1 and 4 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- **4** From the **Distribution method** list, choose **Geometric sequence**.
- **5** In the **Element ratio** text field, type **10**.
- 6 In the Number of elements text field, type 40.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 2

- I Select Boundary 2 only.
- 2 In the Settings window for Distribution, locate the Distribution section.
- **3** From the **Distribution properties** list, choose **Predefined distribution type**.
- 4 In the Number of elements text field, type 250.
- 5 In the Element ratio text field, type 2.
- 6 Select the Reverse direction check box.

Mapped I

Right-click Mapped I and choose Distribution.

Distribution 3

- I In the Settings window for Distribution, locate the Distribution section.
- **2** From the **Distribution properties** list, choose **Predefined distribution type**.
- **3** Select Boundary **3** only.
- 4 In the Number of elements text field, type 250.
- 5 In the Element ratio text field, type 2.
- 6 In the Model Builder window, click Mesh I.
- 7 In the Settings window for Mesh, click Build All.

STUDY I

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

RESULTS

Velocity (spf)

Check that the flow is fully developed by plotting the turbulent dynamic viscosity along the centerline.

ID Plot Group 4

On the Home toolbar, click Add Plot Group and choose ID Plot Group.

Line Graph I

On the ID Plot Group 4 toolbar, click Line Graph.

ID Plot Group 4

I Select Boundary 3 only.

This is the top surface.

- 2 In the Settings window for Line Graph, click Replace Expression in the upper-right corner of the y-axis data section. From the menu, choose Component I>Turbulent Flow, k-ε> Turbulence variables>spf.muT Turbulent dynamic viscosity.
- 3 In the Model Builder window, right-click ID Plot Group 4 and choose Rename.
- **4** In the **Rename ID Plot Group** dialog box, type **Turbulent** viscosity in the **New label** text field.
- 5 Click OK.

6 On the Turbulent viscosity toolbar, click Plot.

As can be seen in the resulting plot, the turbulent viscosity has obtained a constant value well before the outlet.



With the initial simulation step completed, create the backstep model.

ROOT

On the Home toolbar, click Add Component and choose 2D.

GEOMETRY 2

In the Model Builder window, under Component 2 (comp2) click Geometry 2.

ADD PHYSICS

- I On the Home toolbar, click Add Physics to open the Add Physics window.
- 2 Go to the Add Physics window.
- **3** In the tree, select **Recently Used>Turbulent Flow, k-ε (spf)**.
- 4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Study 1.
- 5 Click Add to Component in the window toolbar.
- 6 On the Home toolbar, click Add Physics to close the Add Physics window.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select Preset Studies>Stationary.
- **4** Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Turbulent Flow**, **k**-ε (**spf**) interface.
- 5 Click Add Study in the window toolbar.
- 6 On the Home toolbar, click Add Study to close the Add Study window.

GEOMETRY 2

Rectangle 1 (r1)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type L1+L2.
- 4 In the **Height** text field, type hc.
- **5** Locate the **Position** section. In the **x** text field, type -L1.
- 6 In the y text field, type S.

Point I (ptl)

- I On the Geometry toolbar, click Primitives and choose Point.
- 2 In the Settings window for Point, locate the Point section.
- **3** In the **x** text field, type -L1.
- 4 In the y text field, type S+hc/2.

Union I (unil)

- I On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 2 Click in the Graphics window and then press Ctrl+A to select both objects.

Rectangle 2 (r2)

- I On the Geometry toolbar, click Primitives and choose Rectangle.
- 2 In the Settings window for Rectangle, locate the Size and Shape section.
- **3** In the **Width** text field, type L2.
- 4 In the **Height** text field, type S.

Union 2 (uni2)

I Right-click Rectangle 2 (r2) and choose Build Selected.

- 2 On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 3 Click in the Graphics window and then press Ctrl+A to select both objects.

Mesh Control Edges 1 (mcel)

- I On the Geometry toolbar, click Virtual Operations and choose Mesh Control Edges.
- 2 On the object fin, select Boundary 7 only.
- 3 On the Geometry toolbar, click Build All.
- 4 Click the **Zoom Extents** button on the **Graphics** toolbar.

DEFINITIONS

In the Model Builder window, under Component I (compl) click Definitions.

Linear Extrusion 1 (linext1)

- I On the Definitions toolbar, click Component Couplings and choose Linear Extrusion.
- 2 In the Settings window for Linear Extrusion, locate the Source Selection section.
- **3** From the Geometric entity level list, choose Boundary.
- **4** Select Boundary 4 only.
- 5 Locate the Source Vertices section. Select the Active toggle button.
- **6** Select Point **3** only.
- 7 Select the Active toggle button.
- 8 Select Point 4 only.
- Click to expand the Destination section. From the Destination geometry list, choose Geometry 2.
- **IO** Locate the **Destination Vertices** section. Select the **Active** toggle button.
- II Select Point 1 only.
- **12** Select the **Active** toggle button.
- **I3** Select Point 2 only.

Linear Extrusion 2 (linext2)

- I On the Definitions toolbar, click Component Couplings and choose Linear Extrusion.
- 2 In the Settings window for Linear Extrusion, locate the Source Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- **4** Select Boundary 4 only.
- 5 Locate the Source Vertices section. Select the Active toggle button.
- 6 Select Point 3 only.

- 7 Select the **Active** toggle button.
- 8 Select Point 4 only.
- 9 Click to expand the Destination section. From the Destination geometry list, choose Geometry 2.
- **IO** Locate the **Destination Vertices** section. Select the **Active** toggle button.
- II Select Point 3 only.
- **12** Select the **Active** toggle button.
- **I3** Select Point 2 only.

MATERIALS

In the Model Builder window, under Component 2 (comp2) right-click Materials and choose Blank Material.

Material 2 (mat2)

- I In the Settings window for Material, locate the Material Contents section.
- 2 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Density	rho	rhof	kg/m³	Basic
Dynamic viscosity	mu	muf	Pa·s	Basic

TURBULENT FLOW, $K-\epsilon 2$ (SPF2)

Inlet 1

- I On the Physics toolbar, click Boundaries and choose Inlet.
- **2** Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** Click the **Velocity field** button.
- **5** Specify the **u**₀ vector as

<pre>comp1.linext1(comp1.u)</pre>	x
0	у

6 Locate the Turbulence Conditions section. Click the Specify turbulence variables button.

- 7 In the k_0 text field, type comp1.linext1(comp1.k).
- **8** In the ε_0 text field, type comp1.linext1(comp1.ep).

Inlet 2

I On the Physics toolbar, click Boundaries and choose Inlet.

- 2 Select Boundary 3 only.
- 3 In the Settings window for Inlet, locate the Velocity section.
- **4** Click the **Velocity field** button.
- **5** Specify the **u**₀ vector as

<pre>comp1.linext2(comp1.u)</pre>	x
0	у

- 6 Locate the Turbulence Conditions section. Click the Specify turbulence variables button.
- 7 In the k_0 text field, type comp1.linext2(comp1.k).
- **8** In the ε_0 text field, type comp1.linext2(comp1.ep).

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- **2** Select Boundary 7 only.
- 3 In the Settings window for Outlet, locate the Pressure Conditions section.
- 4 Select the Normal flow check box.

MESH 2

- I In the Model Builder window, under Component 2 (comp2) click Mesh 2.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- **3** From the **Element size** list, choose **Coarse**.

Size 1

- I Right-click Component 2 (comp2)>Mesh 2 and choose Edit Physics-Induced Sequence.
- 2 In the Model Builder window, under Component 2 (comp2)>Mesh 2 click Size 1.
- 3 In the Settings window for Size, click Build Selected.

Size 2

- I In the Model Builder window, right-click Mesh 2 and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- 3 From the Geometric entity level list, choose Boundary.
- **4** Select Boundary 9 only.
- 5 Locate the Element Size section. Click the Custom button.

- **6** Locate the **Element Size Parameters** section. Select the **Maximum element growth rate** check box.
- 7 In the associated text field, type 1.03.

Size 3

- I Right-click Mesh 2 and choose Size.
- 2 In the Settings window for Size, locate the Geometric Entity Selection section.
- **3** From the **Geometric entity level** list, choose **Point**.
- 4 Locate the **Element Size** section. Click the **Custom** button.
- 5 Locate the Element Size Parameters section. Select the Maximum element size check box.
- **6** Select Point 5 only.
- 7 In the associated text field, type 5e-4.

Boundary Layer Properties 1

- In the Model Builder window, expand the Component 2 (comp2)>Mesh 2>Boundary Layers
 I node, then click Boundary Layer Properties 1.
- 2 In the Settings window for Boundary Layer Properties, locate the Boundary Layer Properties section.
- **3** In the **Thickness adjustment factor** text field, type **2**.
- 4 In the Number of boundary layers text field, type 6.
- 5 Click Build All.

STUDY 2

Step 1: Stationary

- I In the Settings window for Stationary, click to expand the Study extensions section.
- 2 Locate the Study Extensions section. Select the Auxiliary sweep check box.
- 3 Click Add.
- **4** In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
fact	5 1	

- 5 Click to expand the Values of dependent variables section. Locate the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 6 From the Method list, choose Solution.

- 7 From the Study list, choose Study I, Stationary.
- 8 On the Home toolbar, click Compute.

RESULTS

Velocity (spf2)

Check that the wall lift-off is 11.06 almost everywhere by selecting the **Wall Resolution** (spf2) plot group.

Wall Resolution (spf2) Next, reproduce the flow-field plot with the following steps:

Velocity (spf2)

- I In the Model Builder window, under Results right-click Velocity (spf2) 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 0.007.
- 5 Locate the Coloring and Style section. From the Color list, choose White.
- 6 On the Velocity (spf2) toolbar, click Plot.

Finally, visualize the recirculation length.

2D Plot Group 8

- I On the Home toolbar, click Add Plot Group and choose 2D Plot Group.
- 2 In the Settings window for 2D Plot Group, locate the Data section.
- 3 From the Data set list, choose Study 2/Solution 2 (3) (sol2).
- 4 Right-click 2D Plot Group 8 and choose Contour.
- 5 In the Settings window for Contour, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component 2>Turbulent Flow, k-ε 2> Velocity and pressure>Velocity field>u2 Velocity field, x component.
- 6 Locate the Levels section. From the Entry method list, choose Levels.
- 7 Right-click Results>2D Plot Group 8>Contour I and choose Color Expression.
- 8 In the Settings window for Color Expression, locate the Expression section.
- **9** In the **Expression** text field, type x/S.
- IO In the Model Builder window, right-click 2D Plot Group 8 and choose Rename.

- II In the **Rename 2D Plot Group** dialog box, type **Recirculation length** in the **New label** text field.
- I2 Click OK.
- **I3** On the **Recirculation length** toolbar, click **Plot**.



Turbulent Mixing of a Trace Species

Introduction

This tutorial demonstrates how mixing can be simulated in a stirred vessel by seeding a trace species from a point. The flow is modeled using the Rotating Machinery, Fluid Flow interface, which solves the Navier-Stokes equations on geometries with rotating parts such as impellers. The transport of the trace species is modeled using the Transport of Diluted Species interface.

Model Definition

MODEL GEOMETRY

Figure 1 shows the model geometry which is s a schematic cross section of a tank with a four-blade impeller. The tank has four baffles attached to the wall to enhance mixing. The mixer blades and the impellers are approximated to be infinitely thin. The seeding of the trace species is done at the marked point.

The circle between the impeller and the tank wall is the assembly boundary where the mesh is allowed to slide when the impeller rotates.



Figure 1: Model geometry.

DOMAIN EQUATION AND BOUNDARY CONDITIONS

The mixer fluid is water, and a rotational speed of 20 rpm is prescribed for the impeller. This rotation is achieved by prescribing the inner domain to be a rotating domain. The Rotating Machinery, Fluid Flow interface then rotates the inner domain with the prescribed rotational speed. The boundary between the inner and the outer domain is prescribed to be a continuity boundary that transfers momentum to the fluid in the outer domain.

The Reynolds number based on the impeller radius and the impeller tip speed is approximately 1.9×10^6 which means that the flow is turbulent. The Rotating Machinery, Fluid Flow interface supports the k- ϵ turbulence model which is applied in this example.

There are two methods to reach operating conditions. One is to accelerate the impeller up to full speed and wait for the flow to reach a quasi steady-state. This approach is simple, but can be time consuming. A computationally more efficient method is to first simulate the flow using the frozen rotor approach. Frozen rotor means that the impeller, or rotor, is frozen in position. 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 Coriolis and centrifugal forces. This solution couples to the nonrotating parts where the flow is also assumed to be stationary, but in a nonrotating coordinate system. See the *CFD Module User's Guide* for more information about frozen rotor.

The result of a frozen rotor simulation is an approximation to the flow at operating conditions. The result depends on the angular position of the impeller and cannot represent transient effects. It is still a very good starting condition to reach operating conditions. Quasi steady-state from a frozen rotor simulation is typically reach within a few revolutions, while starting from zero velocity requires tens of revolutions to reach operating conditions.

A trace species is a species introduced in very small quantities. It is often of a sharp color to be clearly visible even in small amounts. A trace species is not supposed to affect the flow, and hence, the flow can be solved for first and then the trace species transport solved for subsequently. Because it is the mixing at operating condition that is interesting, the trace species is introduced only once the flow is closed to fully developed, which it is after approximately six seconds starting from the frozen rotor simulation.

The seeding is modeled as a point source with normal distribution around the release time, t = 7.0 seconds. The absolute value of the pulse is arbitrary since the trace species does not affect the flow.

Results and Discussion

Figure 2 shows the frozen rotor velocity field. As expected, the highest velocity magnitude is found at the tip of the mixer blades. There are also clearly visible recirculation zones both before and behind the baffles.



Figure 2: The velocity field obtained from the frozen rotor simulation.

Figure 3 shows the velocity field at t = 30 s. The rotor position is the same as in Figure 2, and the results in the figures are similar. There are however differences. The most notably difference is the recirculation zones before the baffles that are smaller in Figure 3 than in Figure 2. The size and shape of the recirculation zones for the time-dependent simulation also vary with the position of the impeller.



Figure 3: A snapshot of the time-dependent velocity field at t=30 s.

Figure 4 shows four snapshots of the mixing process, time running from top left to bottom right picture. Since the velocity is rather slow at the seeding point (see also Figure 3), the initial transport is almost isotropic from the seeding point (t = 9 s). Some trace species is however entrained in the faster velocity field in the center of the mixer and becomes thereby spread in the azimuthal direction (t = 14.7 s). It only takes a few seconds more for the trace species to be almost homogeneous distributed in the mixer. The slowest spreading is to the regions in between the impeller blades. This is a well known

phenomenon and is the reason to why chemical substances are commonly added as close to the impeller axis as possible.



Figure 4: The mixing process. Surface plots of the trace species at t = 9, 14.7, 20.4 and 26.1 seconds.

Application Library path: CFD_Module/Single-Phase_Tutorials/

turbulent_mixing

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

I In the Model Wizard window, click 2D.

- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Rotating Machinery, Fluid Flow>Rotating Machinery, Turbulent Flow, k-ε (rmspf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Frozen Rotor.
- 6 Click Done.

GEOMETRY I

Circle I (c1)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.5.

Circle 2 (c2)

- I On the Geometry toolbar, click Primitives and choose Circle.
- 2 In the Settings window for Circle, locate the Size and Shape section.
- 3 In the Radius text field, type 0.35.

Difference I (dif1)

- I Right-click Circle 2 (c2) and choose Build Selected.
- 2 On the Geometry toolbar, click Booleans and Partitions and choose Difference.
- 3 In the Settings window for Difference, locate the Difference section.
- 4 Select the Keep input objects check box.
- **5** Select the object **cl** only.
- 6 Find the Objects to subtract subsection. Select the Active toggle button.
- 7 Select the object **c2** only.

Delete Entities I (del I)

- I In the Model Builder window, right-click Geometry I and choose Delete Entities.
- **2** In the **Settings** window for Delete Entities, locate the **Entities or Objects to Delete** section.
- 3 From the Geometric entity level list, choose Object.
- **4** Select the object **cl** only.

Bézier Polygon I (b1)

- I Right-click Component I (comp1)>Geometry 1>Delete Entities I (del1) and choose Build Selected.
- 2 On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 3 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 4 Find the Added segments subsection. Click Add Linear.
- 5 Find the Control points subsection. In row 1, set x to -0.5.
- 6 In row 2, set x to -0.4.

Bézier Polygon 2 (b2)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the **Control points** subsection. In row I, set y to 0.5.
- **5** In row **2**, set **y** to **0.4**.

Bézier Polygon 3 (b3)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the **Control points** subsection. In row I, set x to 0.5.
- **5** In row **2**, set **x** to **0.4**.

Bézier Polygon 4 (b4)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the Control points subsection. In row 1, set y to -0.5.
- 5 In row 2, set y to -0.4.

Point I (ptl)

- I On the Geometry toolbar, click Primitives and choose Point.
- 2 In the Settings window for Point, locate the Point section.
- **3** In the **x** text field, type **0.3**.
- 4 In the y text field, type 0.3.

Union I (uniI)

I On the Geometry toolbar, click Booleans and Partitions and choose Union.

The final operation will later be set to **Form an assembly**. Union operations are therefore necessary to merge the domains and the lines.

2 Select the objects difl, ptl, bl, b3, b2, and b4 only.

Bézier Polygon 5 (b5)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the Control points subsection. In row I, set x to -0.25.
- 5 In row 2, set x to 0.25.

Bézier Polygon 6 (b6)

- I On the Geometry toolbar, click Primitives and choose Bézier Polygon.
- 2 In the Settings window for Bézier Polygon, locate the Polygon Segments section.
- 3 Find the Added segments subsection. Click Add Linear.
- 4 Find the Control points subsection. In row I, set y to -0.25.
- 5 In row 2, set y to 0.25.

Union 2 (uni2)

- I On the Geometry toolbar, click Booleans and Partitions and choose Union.
- 2 Select the objects **b6**, **b5**, and **c2** only.

Form Union (fin)

The boundary between the rotating and non-rotating domain must be an assembly boundary so that the parts can move relative to each other.

- I In the Model Builder window, under Component I (compl)>Geometry I click Form Union (fin).
- 2 In the Settings window for Form Union/Assembly, locate the Form Union/Assembly section.
- 3 From the Action list, choose Form an assembly.
- **4** On the **Geometry** toolbar, click **Build All**.
- 5 Click the Zoom Extents button on the Graphics toolbar.

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Water, liquid.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

ROTATING MACHINERY, TURBULENT FLOW, K- ϵ (RMSPF)

Rotating Domain I

- I On the Physics toolbar, click Domains and choose Rotating Domain.
- 2 Select Domain 2 only.
- 3 In the Settings window for Rotating Domain, locate the Rotating Domain section.
- **4** In the *f* text field, type 20[1/min].

Rotating Interior Wall I

- I On the Physics toolbar, click Boundaries and choose Rotating Interior Wall.
- 2 Select Boundaries 13–16 only.

Interior Wall I

- I On the Physics toolbar, click Boundaries and choose Interior Wall.
- **2** Select Boundaries 1–4 only.

Flow Continuity I

- I On 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 Pair I (apl).

Pressure Point Constraint 1

I On the Physics toolbar, click Points and choose Pressure Point Constraint.

The pressure level must be specified since water is an incompressible liquid.

2 Select Point 10 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- 3 From the Element size list, choose Fine.

STUDY I

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

RESULTS

Velocity (rmspf)

Re-create Figure 2 using the following steps.

Streamline 1

- I In the Model Builder window, under Results right-click Velocity (rmspf) 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. From the Color list, choose White.
- 6 On the Velocity (rmspf) toolbar, click Plot.

Add a Time Dependent study.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select Preset Studies>Time Dependent.
- 4 Click Add Study in the window toolbar.
- 5 On the Home toolbar, click Add Study to close the Add Study window.

STUDY 2

Step 1: Time Dependent

It is important to specify frequent enough output times for the flow field. The subsequent species simulation might otherwise be effected by interpolation errors.

- I 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 0 range(6,0.15,30).

Start from the frozen rotor solution.

- 4 Click to expand the Values of dependent variables section. Locate 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 I, Frozen Rotor.
- 7 On the Home toolbar, click Compute.

RESULTS

Velocity (rmspf) 1

Re-create Figure 3 using the following steps.

Streamline 1

- I In the Model Builder window, under Results right-click Velocity (rmspf) I 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. From the Color list, choose White.
- 6 On the Velocity (rmspf) I toolbar, click Plot.

DEFINITIONS

Gaussian Pulse 1 (gp1)

- I On the Home toolbar, click Functions and choose Local>Gaussian Pulse.
- 2 In the Settings window for Gaussian Pulse, locate the Parameters section.
- **3** In the **Location** text field, type **7**.
- 4 In the Standard deviation text field, type 0.25.

ADD PHYSICS

- I On the Home toolbar, click Add Physics to open the Add Physics window.
- 2 Go to the Add Physics window.
- 3 In the tree, select Chemical Species Transport>Transport of Diluted Species (tds).
- 4 Find the Physics interfaces in study subsection. In the table, clear the Solve check box for Study 1 and Study 2.
- 5 Click Add to Component in the window toolbar.

6 On the Home toolbar, click Add Physics to close the Add Physics window.

ADD STUDY

- I On the Home toolbar, click Add Study to open the Add Study window.
- 2 Go to the Add Study window.
- 3 Find the Studies subsection. In the Select Study tree, select Preset Studies>Time Dependent.
- **4** Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for the **Rotating Machinery, Turbulent Flow, k-ε (rmspf)** interface.
- 5 Click Add Study in the window toolbar.
- 6 On the Home toolbar, click Add Study to close the Add Study window.

TRANSPORT OF DILUTED SPECIES (TDS)

On the Physics toolbar, click Rotating Machinery, Turbulent Flow, k- ε (rmspf) and choose Transport of Diluted Species (tds).

Transport Properties 1

In the Model Builder window, expand the Component I (compl)>Transport of Diluted Species (tds) node, then click Transport Properties I.

Turbulent Mixing I

- I On the Physics toolbar, click Attributes and choose Turbulent Mixing.
- **2** In the **Settings** window for Turbulent Mixing, locate the **Turbulent Mixing Parameters** section.
- 3 From the v_T list, choose Turbulent kinematic viscosity (rmspf/fpl).
- **4** In the $Sc_{\rm T}$ text field, type 1.

Transport Properties 1

- In the Model Builder window, under Component I (compl)>Transport of Diluted Species (tds) click Transport Properties I.
- 2 In the Settings window for Transport Properties, locate the Model Inputs section.
- **3** From the **u** list, choose **Velocity field (rmspf)**.

Continuity I

- I On 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 Pair I (apl).

Thin Impermeable Barrier I

- I On the Physics toolbar, click Boundaries and choose Thin Impermeable Barrier.
- **2** Select Boundaries 1–4 and 13–16 only.

Line Mass Source 1

- I On the Physics toolbar, click Points and choose Line Mass Source.
- **2** Select Point 10 only.
- 3 In the Settings window for Line Mass Source, locate the Species Source section.
- 4 In the $\dot{q}_{1,c}$ text field, type gp1(t[1/s])/10.

STUDY 3

Step 1: Time Dependent

- I In the Model Builder window, expand the Study 3 node, then 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 range(6,0.15,30).
- 4 Click to expand the Values of dependent variables section. Locate the Values of Dependent Variables section. Find the Values of variables not solved for subsection. From the Settings list, choose User controlled.
- 5 From the Method list, choose Solution.
- 6 From the Study list, choose Study 2, Time Dependent.

The Automatic Time setting would only use the last solution from Study 2. Change to All

- 7 From the Time (s) list, choose All.
- 8 On the Home toolbar, click Compute.

RESULTS

Concentration (tds)

The following steps creates an animation that contains the plots in Figure 4.

Plot the data set edges on the spatial frame to make them follow the rotation.

- I In the Model Builder window, under Results click Concentration (tds).
- 2 In the Settings window for 2D Plot Group, locate the Plot Settings section.
- **3** From the Frame list, choose Spatial (x, y, z).

Surface

- I In the Model Builder window, expand the Concentration (tds) node, then click Surface.
- 2 In the Settings window for Surface, click to expand the Range section.
- 3 Select the Manual color range check box.
- **4** In the **Minimum** text field, type 0.
- 5 In the Maximum text field, type 1.

Animation I

- I On 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 Concentration (tds).Set the frames to be displayed for as long as the time between the saved solutions.
- 5 Locate the Playing section. In the Display each frame for text field, type 0.15.
- 6 Locate the Frames section. From the Frame selection list, choose All.
- 7 Right-click Animation I and choose Play.



Water Purification Reactor

Introduction

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

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

Model Definition

MODEL GEOMETRY

The model geometry along with some boundary conditions are shown in Figure 1. The full reactor has a symmetry plane which is utilized to reduce the size of the model.


Figure 1: Model geometry. All boundaries except the inlet, outlet, and symmetry plane are walls.

DOMAIN EQUATIONS AND BOUNDARY CONDITIONS

Based on the inlet velocity and diameter, which in this case correspond to 0.1 m/s and 0.4 m respectively, the Reynolds number is

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

Here v is the kinematic viscosity. The high Reynolds number clearly indicates that the flow is turbulent. This means that the flow must be modeled using a turbulence model. In this case, the k- ε turbulence model is used, as it is often done in industrial applications, much because it is both relatively robust and computationally inexpensive compared to more advanced turbulence models. One major reason to why the k- ε model is inexpensive is that it makes use of wall functions to describe the flow close to walls instead of resolving the very steep gradients there. All boundaries in Figure 1, except the inlet, the outlet, and the symmetry plane, are walls.

The inlet velocity is prescribed as a plug flow profile. The turbulent intensity is set to 5% and the turbulent length scale is specified according to in the theory section for the

Turbulent Flow interfaces in the *CFD Module User's Guide*. A constant pressure is prescribed on the outlet.

Notes About the COMSOL Implementation

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

Results and Discussion

Figure 2 shows the velocity field in the symmetry plane. The jet from the inlet hits the top of the first baffle which splits the jet. One half creates a strong recirculation zone in the first "chamber." The other half continues down into the reactor and gradually spreads out. The velocity magnitude decreases as more fluid is entrained into the jet.



Figure 2: Velocity field in the symmetry plane.

Figure 3 gives a more complete picture of the mixing process in the reactor. The streamlines are colored by the velocity magnitude, and their width are proportional to the turbulent viscosity. Wide lines hence indicate high degree of mixing. The turbulence in this example is mainly produced in the shear layers between the central jet and the recirculation zones. The mixing can be seen to be relatively weak in the beginning of the reactor. The plot also shows that it increases further downstream.



Figure 3: Streamlines colored by velocity. The width of the streamlines are proportional to the turbulent viscosity.

Reference

1. J. Hofman, D. Wind, B. Wols, W. Uijttewaal, H. van Dijk, and G. Stelling, "The use of CFD Modeling to determine the influence of residence time distribution on the disinfection of drinking water in ozone contactors," COMSOL Conference 2007, Grenoble, 2007.

Application Library path: CFD_Module/Single-Phase_Tutorials/ water_purification_reactor

Modeling Instructions

From the File menu, choose New.

NEW

In the New window, click Model Wizard.

MODEL WIZARD

- I In the Model Wizard window, click 3D.
- 2 In the Select Physics tree, select Fluid Flow>Single-Phase Flow>Turbulent Flow>Turbulent Flow, k-ε (spf).
- 3 Click Add.
- 4 Click Study.
- 5 In the Select Study tree, select Preset Studies>Stationary.
- 6 Click Done.

GLOBAL DEFINITIONS

Parameters

- I On the Home toolbar, click Parameters.
- 2 In the Settings window for Parameters, locate the Parameters section.
- **3** In the table, enter the following settings:

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

GEOMETRY I

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

I On the Geometry toolbar, click Insert Sequence.

6 | WATER PURIFICATION REACTOR

- 2 Browse to the application's Application Libraries folder and double-click the file water_purification_reactor_geom_sequence.mph.
- 3 On the Geometry toolbar, click Build All.

The model geometry is now complete (Figure 1).

ADD MATERIAL

- I On the Home toolbar, click Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-In>Water, liquid.
- 4 Click Add to Component in the window toolbar.
- 5 On the Home toolbar, click Add Material to close the Add Material window.

TURBULENT FLOW, $K-\epsilon$ (SPF)

Inlet I

- I On the Physics toolbar, click Boundaries and choose Inlet.
- 2 Select Boundary 1 only.
- 3 In the Settings window for Inlet, locate the Turbulence Conditions section.
- **4** In the $L_{\rm T}$ text field, type 0.07*0.4[m].
- **5** Locate the **Velocity** section. In the U_0 text field, type u_in.

Symmetry I

- I On the Physics toolbar, click Boundaries and choose Symmetry.
- 2 Select Boundary 3 only.

Outlet I

- I On the Physics toolbar, click Boundaries and choose Outlet.
- 2 Select Boundary 28 only.

MESH I

- I In the Model Builder window, under Component I (compl) click Mesh I.
- 2 In the Settings window for Mesh, locate the Mesh Settings section.
- **3** From the **Element size** list, choose **Coarser**.
- 4 Right-click Component I (comp1)>Mesh I and choose Edit Physics-Induced Sequence.

Size 1

In the Model Builder window, under Component I (compl)>Mesh I right-click Size I and choose Disable.

Boundary Layer Properties 1

- I In the Model Builder window, expand the Component I (comp1)>Mesh 1>Boundary Layers
 I node, then click Boundary Layer Properties 1.
- 2 In the Settings window for Boundary Layer Properties, locate the Boundary Layer Properties section.
- 3 In the Number of boundary layers text field, type 2.
- 4 In the Thickness adjustment factor text field, type 6.
- 5 In the Model Builder window, collapse the Mesh I node.
- 6 In the Settings window for Mesh, click Build All.
- 7 Click the **Zoom Extents** button on the **Graphics** toolbar.

Confirm that the mesh matches the figure bellow.



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

STUDY I

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

RESULTS

The following steps reproduce Figure 2.

First, create a data set that corresponds to the inlet, outlet, and symmetry plane.

Surface 2

On the Results toolbar, click More Data Sets and choose Surface.

Data Sets Select Boundaries 1, 3, and 28 only.

Velocity (spf)

- I In the Model Builder window, expand the Results>Velocity (spf) node.
- 2 Right-click Slice and choose Disable.
- 3 In the Model Builder window, click Velocity (spf).
- 4 In the Settings window for 3D Plot Group, locate the Data section.
- 5 From the Data set list, choose Surface 2.
- 6 Right-click Velocity (spf) and choose Surface.
- 7 In the Settings window for Surface, locate the Data section.
- 8 From the Data set list, choose Exterior Walls.
- 9 Locate the Coloring and Style section. From the Coloring list, choose Uniform.
- **IO** From the **Color** list, choose **Gray**.
- II Right-click Velocity (spf) and choose Surface.
- 12 Right-click Velocity (spf) and choose Arrow Surface.
- 13 In the Settings window for Arrow Surface, locate the Coloring and Style section.
- **I4** From the **Arrow length** list, choose **Logarithmic**.
- **I5** Select the **Scale factor** check box.
- **I6** In the associated text field, type **1.4**.
- **I7** In the Number of arrows text field, type 300.
- 18 From the Color list, choose White.
- **19** In the **Model Builder** window, click **Velocity (spf)**.
- 20 In the Settings window for 3D Plot Group, click to expand the Title section.
- 21 From the Title type list, choose Manual.

- 22 In the Title text area, type Velocity field.
- **23** On the **Velocity (spf)** toolbar, click **Plot**.
- **24** Click the **Zoom Extents** button on the **Graphics** toolbar.

Proceed to reproduce Figure 3 as follows.

3D Plot Group 4

- I In the Model Builder window, under Results>Velocity (spf) right-click Surface I and choose Copy.
- 2 On the Home toolbar, click Add Plot Group and choose 3D Plot Group.
- **3** In the Model Builder window, right-click **3D Plot Group 4** and choose Paste Surface.
- 4 Right-click **3D Plot Group 4** and choose **Streamline**.
- **5** Select Boundary 1 only.
- 6 In the Settings window for Streamline, locate the Streamline Positioning section.
- 7 In the **Number** text field, type 45.
- 8 Locate the Coloring and Style section. From the Line type list, choose Ribbon.
- **9** In the **Width expression** text field, type **spf.nuT*1[s/m]**.
- **IO** Select the **Width scale factor** check box.
- II In the associated text field, type 100.
- 12 Right-click Results>3D Plot Group 4>Streamline 1 and choose Color Expression.
- **I3** In the **Settings** window for Color Expression, click to expand the **Range** section.
- **I4** Select the **Manual color range** check box.
- **I5** In the **Minimum** text field, type 0.
- **I6** In the **Maximum** text field, type **0.1**.
- **I7** In the **Model Builder** window, click **3D Plot Group 4**.
- **18** In the **Settings** window for 3D Plot Group, locate the **Title** section.
- **19** From the **Title type** list, choose **Manual**.
- **20** In the **Title** text area, type Streamlines colored by velocity. Width proportional to turbulent viscosity..
- 21 On the 3D Plot Group 4 toolbar, click Plot.
- 22 Click the Zoom Extents button on the Graphics toolbar.
- 23 Right-click 3D Plot Group 4 and choose Rename.
- 24 In the Rename 3D Plot Group dialog box, type Streamlines in the New label text field.
- 25 Click OK.