Model created in COMSOL Multiphysics 6.4

Free Surface Mixer

Introduction

This example of turbulent flow in a partially baffled mixer shows how to set up The Rotating Machinery, Turbulent Flow interfaces from the Mixer Module with free surface and stationary free surface features. The mixed fluid is water, and the flow is modeled using the k-ε turbulence model. Both frozen-rotor and time-dependent simulations are performed and compared. The mixer geometry used in this model corresponds to the one used by J.-P. Torré and others (Ref. 1).

Model Definition

The mixer is a glass-lined, under-baffled stirred vessel. Figure 1 shows the model geometry. It includes a three-bladed impeller and two beavertail baffles supported at the top of the vessel. The tank bottom is curved, allowing the three-bladed impeller to be placed close to the bottom. The cylindrical part of the tank has a diameter of 450 mm, and the initial water level is 700 mm. The two contoured baffles hang from the top of the mixer, and their lower part is 256 mm above the bottom of the vessel.

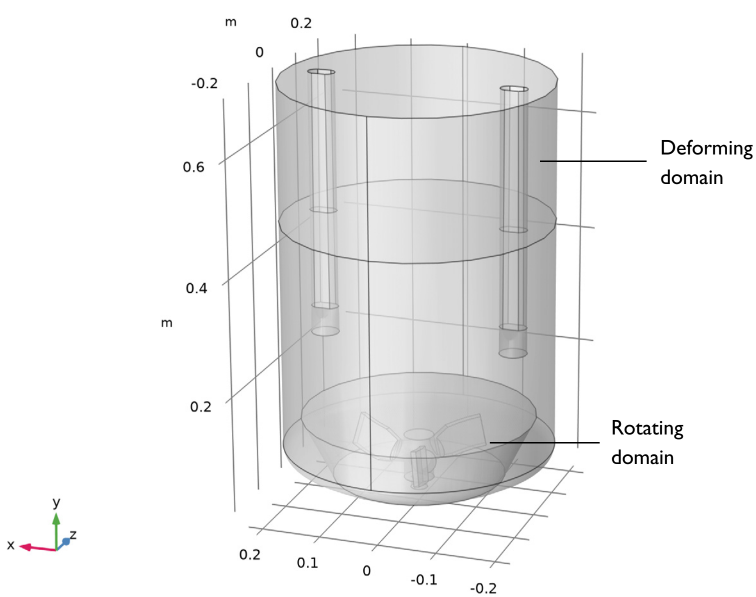


Figure 1: Geometry of the mixer with two rods and a three-bladed impeller.

The fluid contained in the mixer, water, is mixed using an impeller with a rotational velocity of 100 rpm. The flow in the mixer is modeled using the k-ε turbulence model.

The model is solved in two steps. First, a Frozen Rotor study is used to reach a good initial solution for the time-dependent study without having to solve the transient startup of the problem. In order to converge this step, a parametric sweep is used to first solve the model with a lower Reynolds number by using a higher value of the dynamic viscosity, and then computed again with the actual dynamic viscosity of the fluid. Then, a Time Dependent study is solved for a period of 1.8 seconds. This corresponds to 3 revolutions of the impeller.

FREE SURFACE

The top boundary corresponds to the two-phase interface separating the modeled water from the external air. In the transient case, this is modeled using the free surface feature and a deforming domain. Assuming that the dynamic viscosity and density of the air are negligible compared to the corresponding values for the water, so that the velocity and viscous stresses of the air do not affect the flow in the vessel, the following condition is applied for the normal stress:

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

Here $p_{\text{ext}} = 0$ Pa is the pressure outside the free surface domain, and σ is the water/air surface tension coefficient. The kinematic free surface condition sets the mesh to follow the motion of the fluid in the normal direction to the surface:

$$\mathbf{u}_{\text{mesh}} \cdot \mathbf{n} = \mathbf{u} \cdot \mathbf{n}$$

The free surface feature is not available in the frozen rotor study since the deformation of the mesh cannot be tracked. In this case, the Stationary Free Surface feature can be used to approximate the deformation of the free surface from the pressure distribution on the boundary. This boundary condition applies a slip condition together with an external pressure. After computing the frozen rotor simulation, a Stationary Free Surface study step can be used to evaluate the free surface deformation η_{FS} from the pressure obtained in the frozen rotor study using a linearized free surface condition:

$$p(\mathbf{x}_0) - p_{\text{ext}} + \hat{\mathbf{n}} \cdot \nabla p\big|_{\mathbf{x} = \mathbf{x}_0}\eta_{\text{FS}} = -\sigma\nabla_{\text{S}}^2\eta_{\text{FS}}$$

Here $\mathbf{x} = \mathbf{x}_0$ represents the position of the undisturbed surface.

Results and Discussion

The resulting velocity and free surface deformation in the mixer for the frozen-rotor and time-dependent studies are shown in Figure 2 and Figure 3, respectively. The turbulent viscosities are depicted in Figure 4 and Figure 5. It can be observed that the velocity field and turbulent viscosity obtained in the frozen-rotor simulation agree well with the time-dependent results. The stationary free surface feature is also able to provide a good prediction of the vortex induced deformation that develops on the free surface. However, the time-dependent study also provides the transient evolution of the free surface deformation and its instabilities.

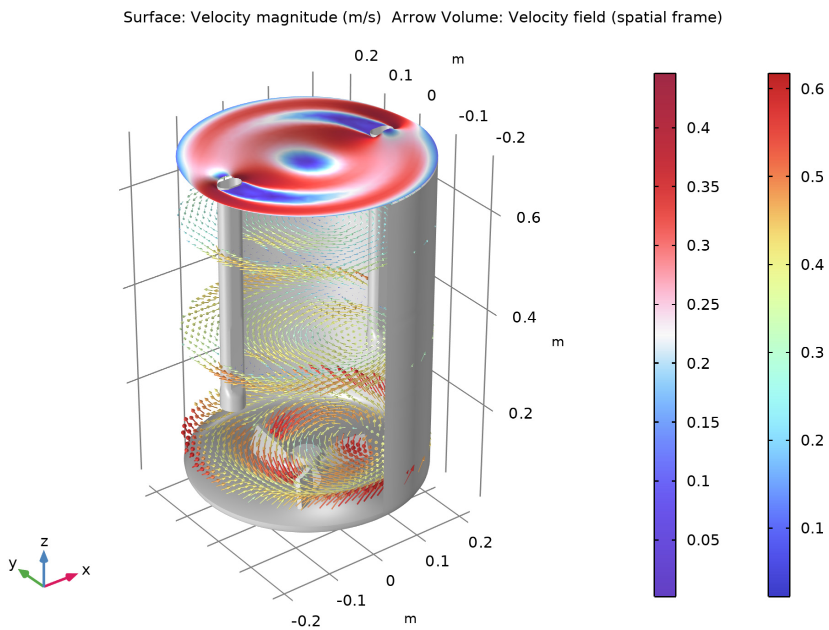


Figure 2: Velocity field and free surface position for the frozen rotor study.

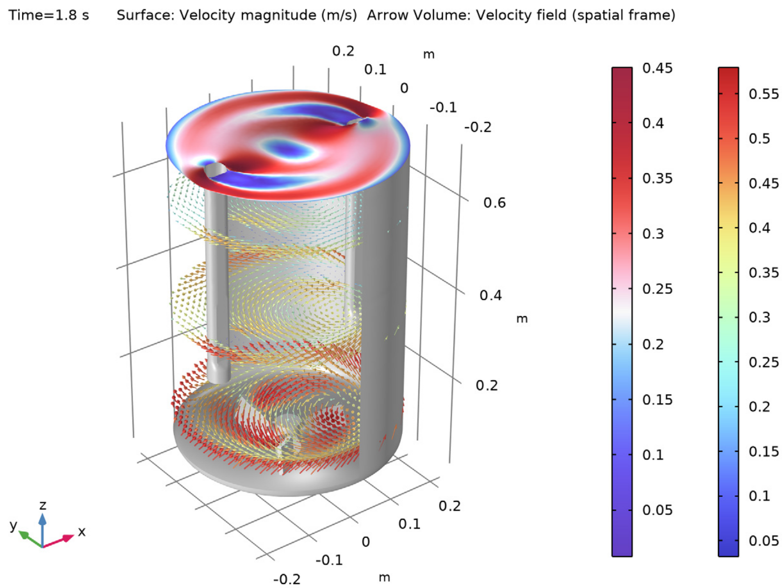


Figure 3: Velocity field and free surface position at t = 1.8 s.

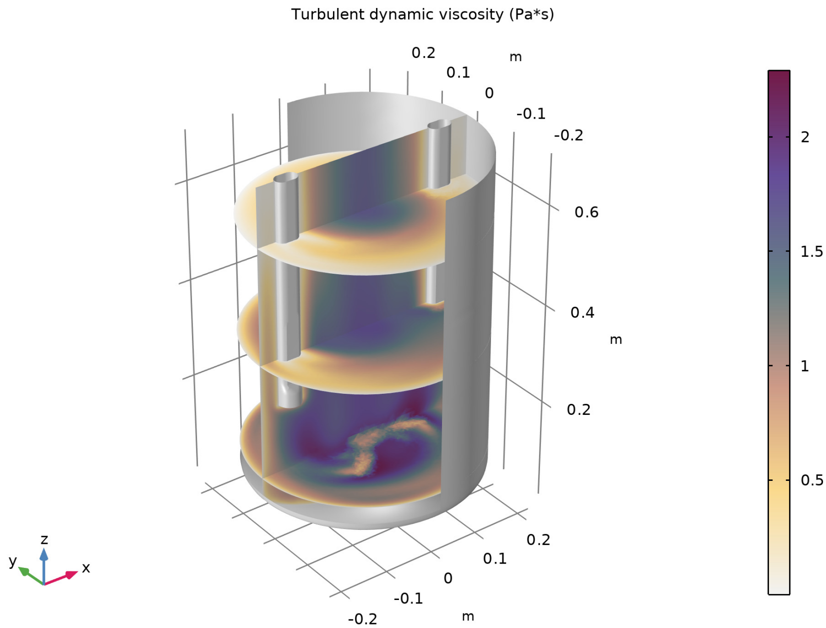


Figure 4: Turbulent viscosity for the frozen rotor case.

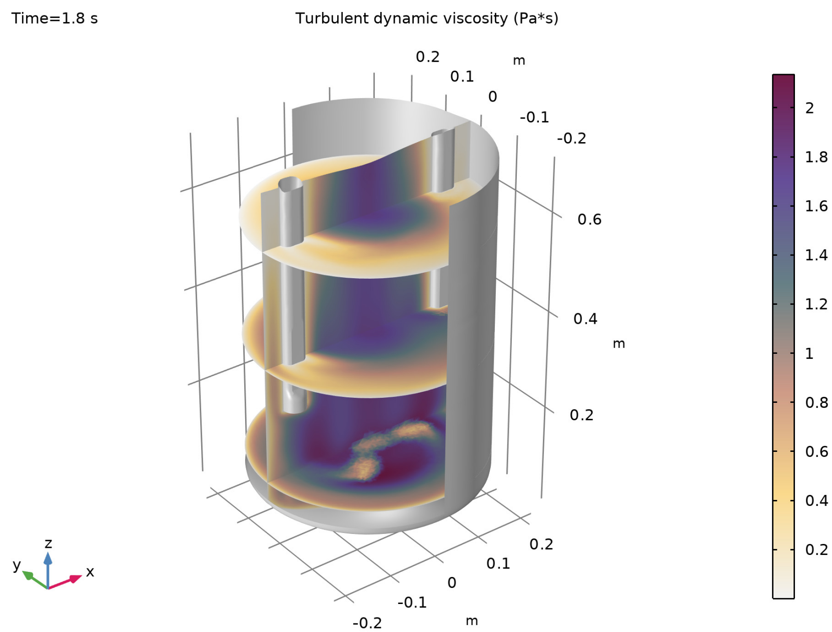


Figure 5: Turbulent viscosity at t = 1.8 s.

Figure 6 and Figure 7 show the velocity distribution in one vertical and five horizontal cross sections. The larger velocities are observed near the rotating blades, which induce a flow toward the walls. Upon reaching the wall, vertical high speed streaks form along the outer walls, and the fluid velocity decreases toward the top of the vessel. A swirl flow pattern can be observed at the upper part of the tank, resulting in a small central vortex adjacent to the free surface.

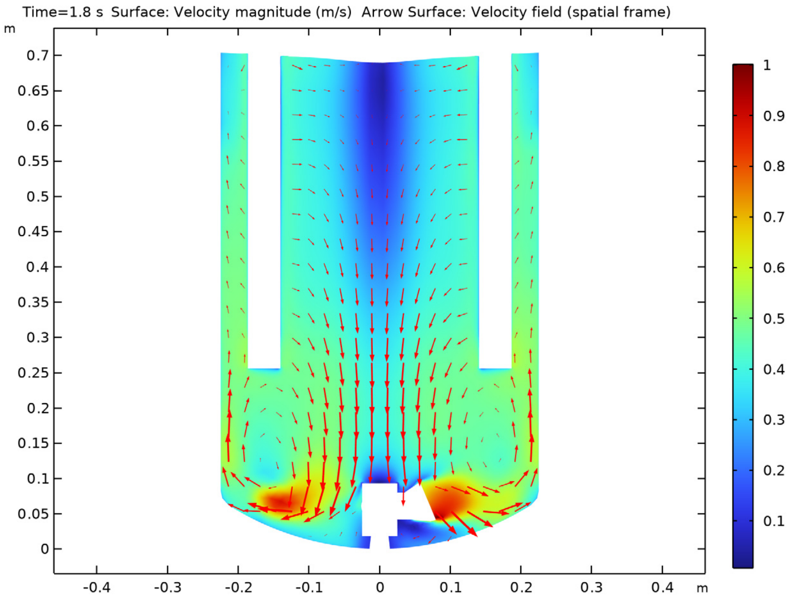


Figure 6: Velocity field in the middle xz-plane at $t = 1.8$ s.

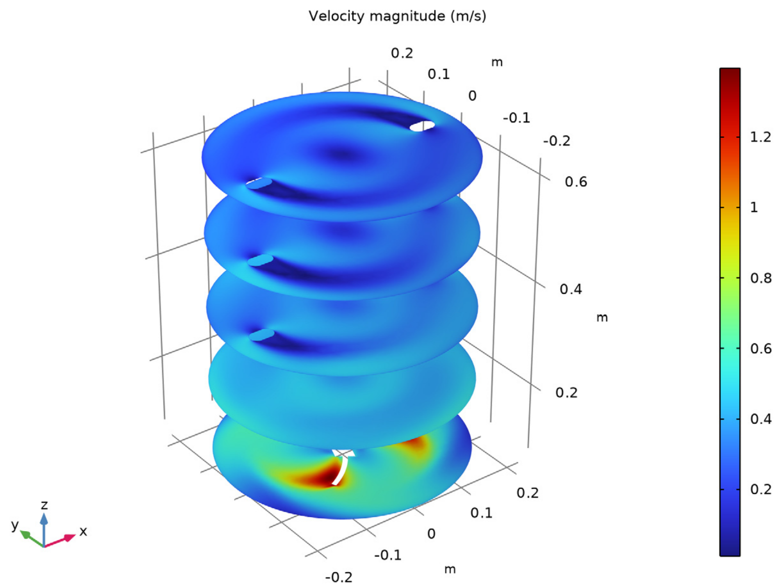


Figure 7: Velocity field in 5 xy-planes.

Notes About the COMSOL Implementation

This model has a large number of degrees of freedom and requires an iterative solver for optimal performance. The Algebraic Multigrid Solver suggested per default is optimal for the Time Dependent study. A Geometric Multigrid Solver should be used in the frozen rotor study step since the Algebraic Multigrid Solver is not suited for the Stationary Free Surface feature.

Reference

1. J.-P. Torré and others, "An Experimental and Computational Study of the Vortex Shape in a Partially Baffled Agitated Vessel," *Chem. Eng. Sci.*, vol. 62, no. 7, pp. 1915–1926, 2007.

Application Library path: `Mixer_Module/Tutorials/free_surface_mixer`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Rotating Machinery, Fluid Flow** > **Turbulent Flow** > **Turbulent Flow, k-ε**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Frozen Rotor with Stationary Free Surface**.

6 Click **Done**.

GEOMETRY 1

Load the model geometry from a geometry sequence file. It imports the mixer geometry and uses virtual operations to simplify the geometry for efficient meshing. It also contains selections that are used throughout the model setup and results processing.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `free_surface_mixer_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
viscFact	1	1	Viscosity factor

Step 1 (step1)

1 In the **Home** toolbar, click **f(X) Functions** and choose **Global** > **Step**.

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

3 In the **Location** text field, type `0.5`.

4 Click to expand the **Smoothing** section. From the **Number of continuous derivatives** list, choose **1**.

5 Click **Plot**.

MOVING MESH

Rotating Domain 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Rotating Domain 1**.

2 Select Domain 3 only.

3 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

4 In the *f* text field, type `100[rpm]`.

5 Locate the **Axis** section. Specify the $\mathbf{u}_{\mathrm{rot}}$ vector as

0	X
0	Y
1	Z

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Moving Mesh** toolbar, click **Deforming Domain**.

2 Select Domain 2 only.

TURBULENT FLOW, K-ε (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.

2 In the **Settings** window for **Turbulent Flow, k-ε**, locate the **Physical Model** section.

3 Select the **Include gravity** checkbox.

Wall 2

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

2 Select Boundary 32 only.

3 In the **Settings** window for **Wall**, click to expand the **Wall Movement** section.

4 From the **Translational velocity** list, choose **Zero (Fixed wall)**.

The **Translational velocity** is set to **Zero (Fixed Wall)** to ensure that the wall is not moving. If **Automatic from frame** is selected, the wall will rotate due to the angular velocity of the **Rotating Domain**.

Stationary Free Surface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Stationary Free Surface**.

2 In the **Settings** window for **Stationary Free Surface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Free Surface**.

Free Surface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Free Surface**.

2 In the **Settings** window for **Free Surface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Free Surface**.

The **Stationary Free Surface** will be used in the **Frozen Rotor** study, while **Free Surface** will override it in the **Time Dependent** study.

ADD MATERIAL

1 In the **Materials** 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 Right-click and choose **Add to Component 1 (comp1)**.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Water, liquid (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Dynamic viscosity	mu	eta(T)*viscFact	Pa·s	Basic

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3. In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.
4. Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** 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 `0.02`.
5. In the **Minimum element size** text field, type `0.008`.
6. In the **Maximum element growth rate** text field, type `1.12`.

Size 1

1. In the **Model Builder** window, click **Size 1**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.
4. Click the **Custom** button.
5. Locate the **Element Size Parameters** section.
6. Select the **Maximum element size** checkbox. In the associated text field, type `0.02`.
7. Select the **Minimum element size** checkbox. In the associated text field, type `0.004`.
8. Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.12`.
9. Select the **Curvature factor** checkbox.
10. Select the **Resolution of narrow regions** checkbox.

Free Tetrahedral 1

1. In the **Model Builder** window, click **Free Tetrahedral 1**.
2. In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domains 1 and 3 only.
5. Click the **Wireframe Rendering** button in the **Graphics** toolbar to be able to locate the entities inside the vessel.

Size 1

1. Right-click **Free Tetrahedral 1** 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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `8, 20, 30, 31, 32` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 From the **Predefined** list, choose **Extra fine**.

9 Click the **Custom** button.

10 Locate the **Element Size Parameters** section.

11 Select the **Maximum element size** checkbox. In the associated text field, type `0.013`.

12 Select the **Curvature factor** checkbox. In the associated text field, type `0.35`.

Size 2

1 Right-click **Free Tetrahedral 1** 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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `34, 38, 68` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 From the **Predefined** list, choose **Extremely fine**.

Free Tetrahedral 1

Right-click **Free Tetrahedral 1** and choose **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **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 4 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `16`.

5 In the **Element ratio** text field, type 1.6.

Swept 2

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type 12.

5 In the **Element ratio** text field, type 2.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

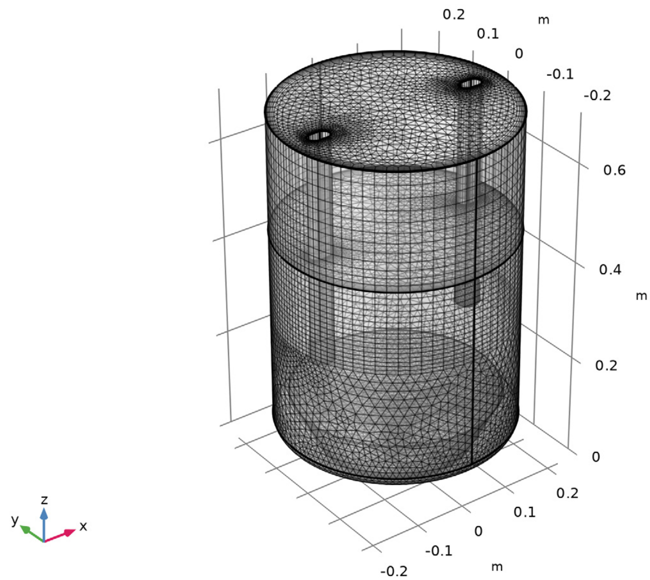
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

3 In the **Number of layers** text field, type 6.

4 In the **Thickness adjustment factor** text field, type 2.

5 Click **Build All**.

6 Click the **Wireframe Rendering** button in the **Graphics** toolbar to go back to the default view.



Build manual multigrid levels for optimal performance. Start by referencing the initial mesh, then scale it. To prevent elements from becoming too small, set a minimum element size manually for the new meshes.

MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

Reference 1

1 In the **Mesh** toolbar, click **Modify** and choose **Reference**.

2 In the **Settings** window for **Reference**, locate the **Reference** section.

3 From the **Mesh** list, choose **Mesh 1**.

Scale 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Scale**.

2 In the **Settings** window for **Scale**, locate the **Scale** section.

3 In the **Element size scale** text field, type 2.

Reference 1

In the **Model Builder** window, right-click **Reference 1** and choose **Expand**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Meshes** > **Mesh 2** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Minimum element size** text field, type `0.009*2`.

Size 1

1 In the **Model Builder** window, click **Size 1**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Minimum element size** text field, type `0.004*2`.

4 Click **Build All**.

MESH 3

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

Reference 1

1 In the **Mesh** toolbar, click **Modify** and choose **Reference**.

2 In the **Settings** window for **Reference**, locate the **Reference** section.

3 From the **Mesh** list, choose **Mesh 2**.

Scale 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Scale**.

2 In the **Settings** window for **Scale**, locate the **Scale** section.

3 In the **Element size scale** text field, type `2`.

Reference 1

In the **Model Builder** window, right-click **Reference 1** and choose **Expand**.

Size 1

1 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

2 In the **Minimum element size** text field, type `0.004*4`.

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size Parameters** section.

3 In the **Minimum element size** text field, type `0.009*4`.

4 Click **Build All**.

Disable **Deforming Domain** in the frozen rotor study.

STUDY 1

Step 1: Frozen Rotor

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frozen Rotor**.
2 In the **Settings** window for **Frozen Rotor**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration for study step** checkbox.
4 In the tree, select **Component 1 (comp1)** > **Moving Mesh, Controls spatial frame** > **Deforming Domain 1**.
5 Click **Disable**.
6 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Geometry 1	Mesh 1

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
8 Click **Add**.
9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
viscFact (Viscosity factor)	50 1	

10 From the **Run continuation for** list, choose **No parameter**.
11 From the **Reuse solution from previous step** list, choose **Yes**.
12 Click the **Show More Options** button in the **Model Builder** toolbar.
13 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Study** > **Multigrid Level**.
14 Click **OK**.
15 Right-click **Study 1** > **Step 1: Frozen Rotor** and choose **Multigrid Level**.
16 In the **Settings** window for **Multigrid Level**, locate the **Mesh Selection** section.
17 In the table, enter the following settings:

Component	Mesh
Geometry 1	Mesh 2

18 In the **Model Builder** window, right-click **Step 1: Frozen Rotor** and choose **Multigrid Level**.

Step 2: Stationary Free Surface

1 In the **Settings** window for **Stationary Free Surface**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 In the tree, select **Component 1 (comp1)** > **Moving Mesh, Controls spatial frame** > **Deforming Domain 1**.

4 Right-click and choose **Disable**.

5 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Geometry 1	Mesh 1

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Segregated 1** node.

4 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **AMG, fluid flow variables (spf_spatial1)** node, then click **Multigrid 1**.

5 In the **Settings** window for **Multigrid**, locate the **General** section.

6 From the **Hierarchy generation method** list, choose **Manual**.

7 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf)

Before working with plots, disable the automatic update of plots. Due to the size of the datasets created, it is more convenient to generate plots actively.

1 In the **Model Builder** window, click **Results**.

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** checkbox.

4 In the **Model Builder** window, click **Velocity (spf)**.

5 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

6 Clear the **Plot dataset edges** checkbox.

Multislice 1

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

2 Right-click **Multislice 1** and choose **Delete**.

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **WaveLight**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Free Surface**.

Deformation 1

1 In the **Model Builder** window, right-click **Surface 1** 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 `0`.

5 In the **z-component** text field, type `spf.etaFS`.

6 Locate the **Scale** section. Select the **Scale factor** checkbox.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Velocity (spf)** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `30`.

4 Find the **y grid points** subsection. In the **Points** text field, type `30`.

5 Find the **z grid points** subsection. In the **Points** text field, type `3`.

6 Locate the **Coloring and Style** section.

7 Select the **Scale factor** checkbox. In the associated text field, type `0.07`.

Color Expression 1

1 Right-click **Arrow Volume 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **RainbowLight**.

Surface 2

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Walls**.

4 In the list, choose **1** and **3**.

5 Click **Remove from Selection**.

6 Select Boundaries 2, 8–29, and 32–69 only.

7 In the **Velocity (spf)** toolbar, click **Plot**.

Next, create a plot for the turbulent viscosity.

Turbulent Viscosity

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Turbulent Viscosity` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Multislice 1

1 In the **Turbulent Viscosity** toolbar, click **More Plots** and choose **Multislice**.

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Turbulent Flow, k-ε** > **Turbulence variables** > **spf.muT - Turbulent dynamic viscosity - Pa·s**.

3 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **z-planes** subsection. In the **Planes** text field, type `3`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **ConopiformisZero**.

Surface 1

1 In the **Model Builder** window, right-click **Turbulent Viscosity** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Walls**.

4 In the list, choose **1** and **3**.

5 Click **Remove from Selection**.

6 Select Boundaries 2, 8–29, and 32–69 only.

7 In the **Turbulent Viscosity** toolbar, click **Plot**.

Velocity xy

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Velocity xy` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Slice 1

1 Right-click **Velocity xy** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **xy-planes**.

4 In the **Velocity xy** toolbar, click **Plot**.

ADD STUDY

1 In 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 **General Studies** > **Time Dependent**.

4 Right-click and choose **Add Study**.

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,0.05,1.8)`.

3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 1, Stationary Free Surface**.

6 From the **Selection** list, choose **Last**.

7 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Geometry 1	Mesh 1

8 In the **Model Builder** window, click **Study 2**.

9 In the **Settings** window for **Study**, locate the **Study Settings** section.

10 Clear the **Generate default plots** checkbox because you can duplicate the previous plots and only need top update the dataset.

11 In the **Study** toolbar, click $\overset{U}{t=0}$ **Get Initial Value**.

First, group the plots to make the results section clearer and easier to follow. Then, duplicate the group and assign the correct dataset. This way, during the time-dependent study, you can view the results while they are being computed.

RESULTS

Pressure (spf), Turbulent Viscosity, Velocity (spf), Velocity xy, Wall Resolution (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf)**, **Pressure (spf)**, **Wall Resolution (spf)**, **Turbulent Viscosity**, and **Velocity xy**.

2 Right-click and choose **Group**.

Frozen Rotor

In the **Settings** window for **Group**, type `Frozen Rotor` in the **Label** text field.

Time Dependent

1 Right-click **Frozen Rotor** and choose **Duplicate**.

2 In the **Settings** window for **Group**, type `Time Dependent` in the **Label** text field.

Velocity (spf) 1

1 In the **Model Builder** window, expand the **Time Dependent** node, then click **Velocity (spf) 1**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

4 In the **Model Builder** window, expand the **Velocity (spf) 1** node.

Deformation 1

1 In the **Model Builder** window, expand the **Results** > **Time Dependent** > **Velocity (spf) 1** > **Surface 1** node.

2 Right-click **Deformation 1** and choose **Delete**.

Update the dataset for all remaining plots in this group.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.

3 Select the **Plot** checkbox.

4 In the table, enter the following settings:

Plot group	Plot window
Velocity (spf) 1	Graphics

Solver Configurations

In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** node.

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 3 (sol3)** node.

2 In the **Model Builder** window, click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Minimum BDF order** list, choose **2**.

The BDF order must be larger than 1 to accurately capture the surface movement.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Velocity (spf) 1

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Time (s)** list, choose **Last (1.8)**.

3 In the **Velocity (spf) 1** toolbar, click **Plot**.

Turbulent Viscosity 1

1 In the **Model Builder** window, click **Turbulent Viscosity 1**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **Last (1.8)**.

4 In the **Turbulent Viscosity 1** toolbar, click **Plot**.

xz-Plane, Time Dependent

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, type `xz-Plane, Time Dependent` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 3 (sol3)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

Velocity xz

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Velocity xz` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

Right-click **Velocity xz** and choose **Surface**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Velocity xz** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `20`.

4 Find the **y grid points** subsection. In the **Points** text field, type `20`.

5 Locate the **Coloring and Style** section.

6 Select the **Scale factor** checkbox. In the associated text field, type `0.1`.

7 In the **Velocity xz** toolbar, click **Plot**.