Created in COMSOL Multiphysics 6.1

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.

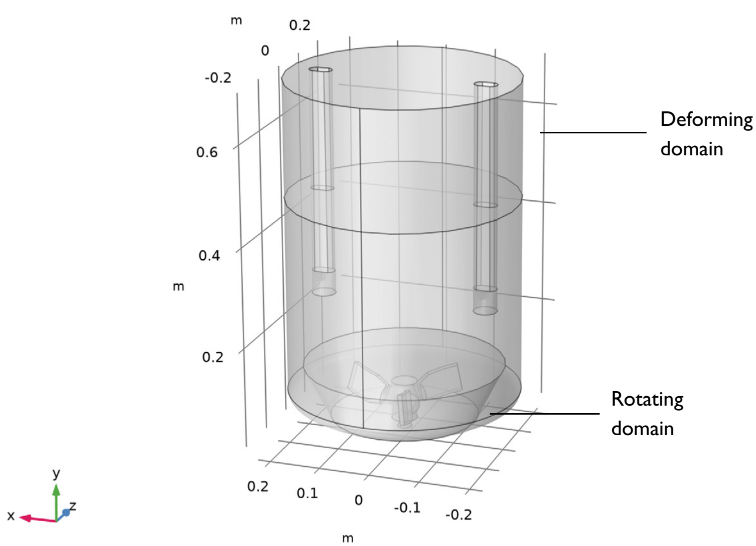


Figure 1: Geometry of the mixer with two rods and a three-bladed impeller.

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.

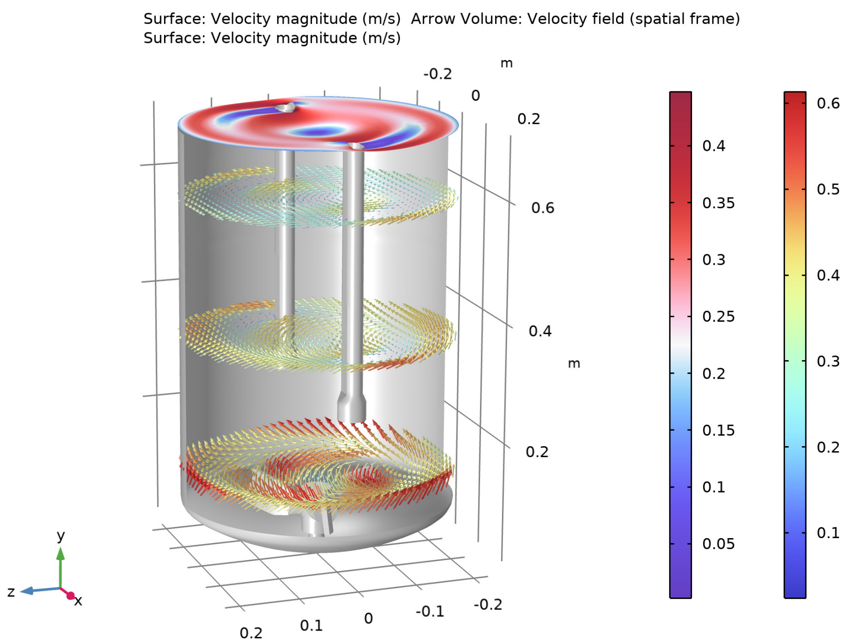


Figure 2: Velocity field and free surface position for the frozen rotor study.

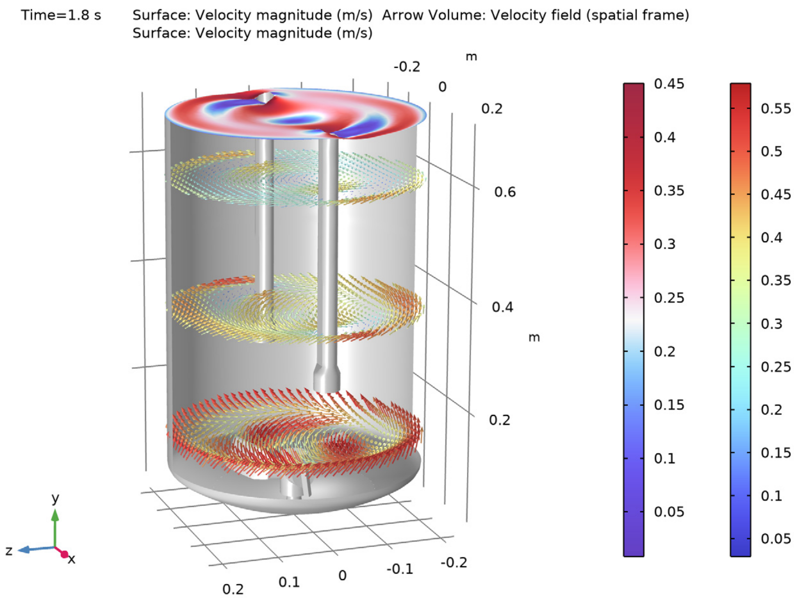


Figure 3: Velocity field and free surface position at t = 1.8 s.

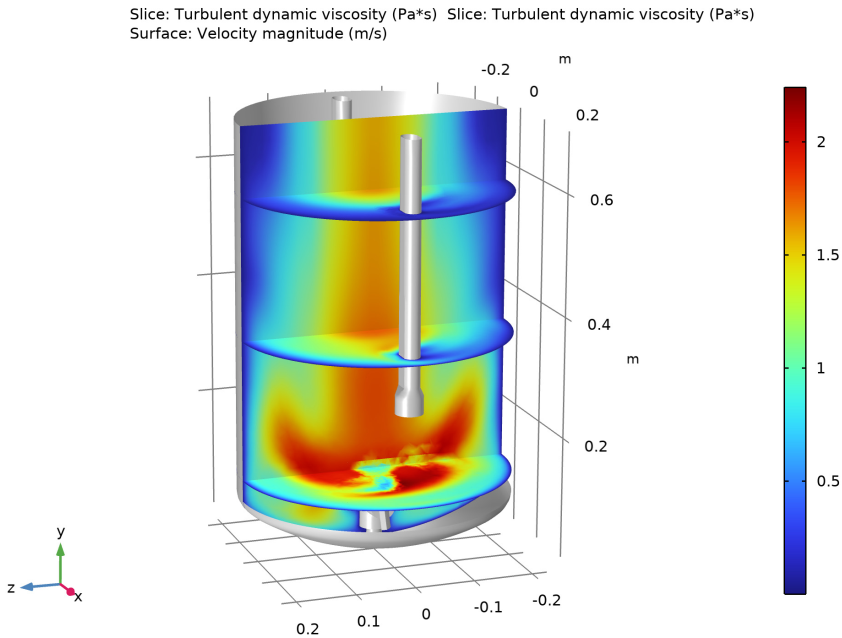


Figure 4: Turbulent viscosity for the frozen rotor case.

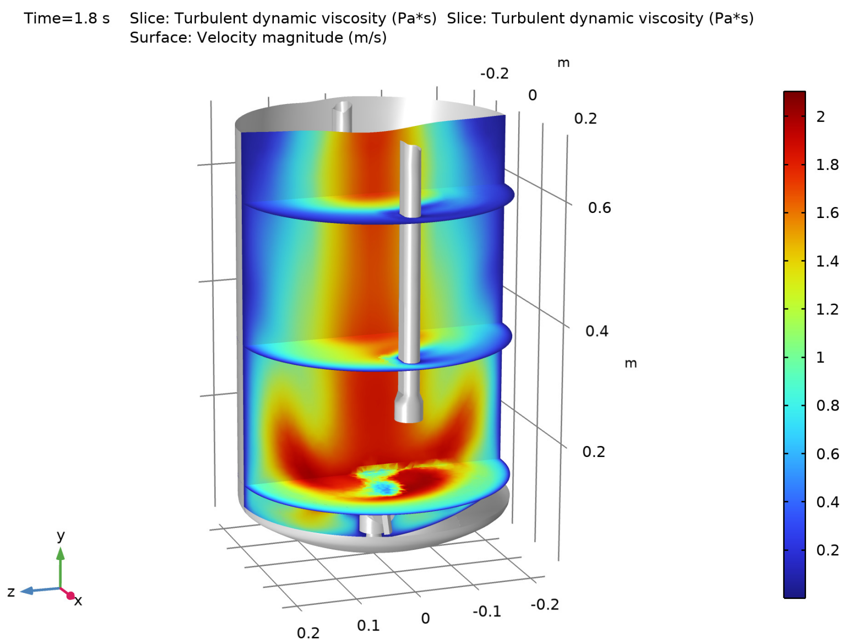


Figure 5: Turbulent viscosity at t = 1.8 s.

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.

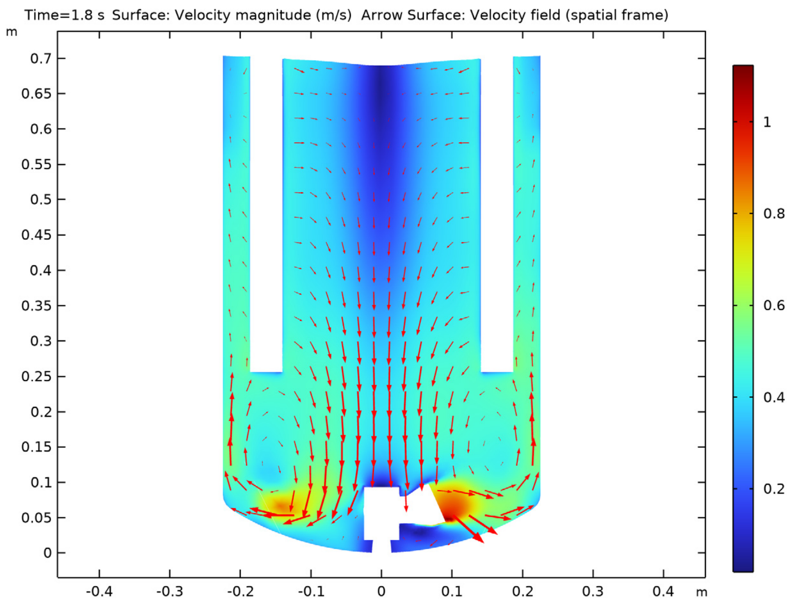


Figure 6: Velocity field in the middle xy-plane at t = 1.8 s.

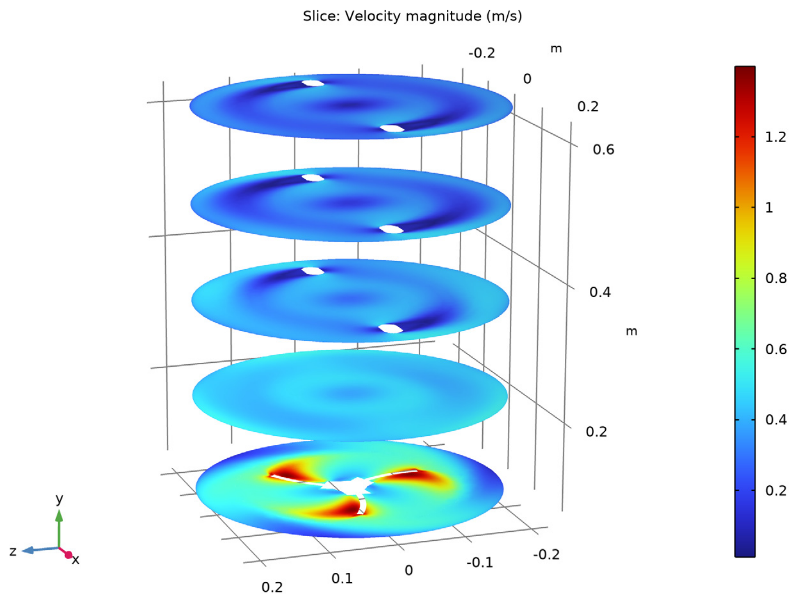


Figure 7: Velocity field in 5 xz-planes.

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 et al., "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**.

Load the model geometry from a geometry sequence.

GEOMETRY 1

1 In the **Geometry** toolbar, click **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**.

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 **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
1	Y
0	Z

COMPONENT 1 (COMP1)

Deforming Domain 1

1 In the **Definitions** toolbar, click **Moving Mesh** and choose **Domains>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** check box.

Gravity 1

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

2 In the **Settings** window for **Gravity**, locate the **Acceleration of Gravity** section.

3 Specify the **g** vector as

0	x
-g_const	y
0	z

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 Select Boundary 12 only.

Free Surface 1

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

2 Select Boundary 12 only.

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

5 In the **Home** 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

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Edit Physics-Induced Sequence**.

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** check box. In the associated text field, type `0.02`.

7 Select the **Minimum element size** check box. In the associated text field, type `0.004`.

8 Select the **Maximum element growth rate** check box. In the associated text field, type `1.12`.

9 Select the **Curvature factor** check box.

10 Select the **Resolution of narrow regions** check box.

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.

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 box, type `2, 3, 5, 21, 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** check box. In the associated text field, type `0.013`.

12 Select the **Curvature factor** check box. 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 box, type `34, 38, 56` 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**.

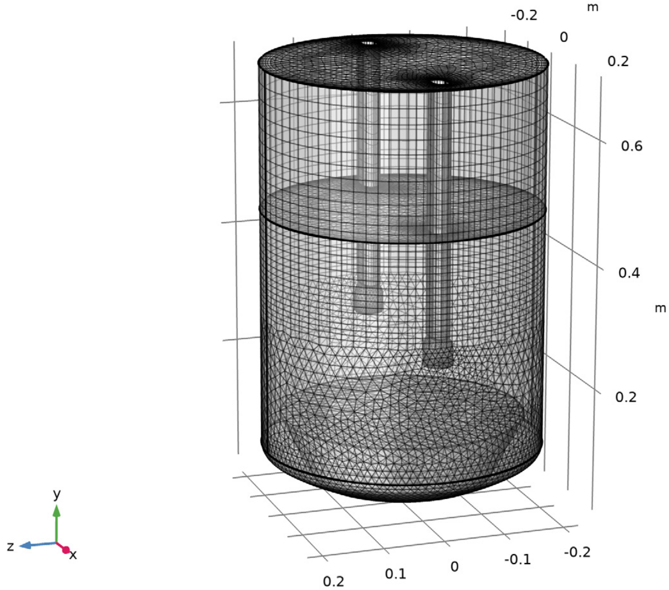
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Layers** section.

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 **Transparency** button in the **Graphics** toolbar.



7 Click the **Transparency** button in the **Graphics** toolbar.

Build manual multigrid levels for optimal performance.

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 **Model Builder** window, under **Component 1 (comp1)>Meshes>Mesh 3** 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*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** check box.
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
Component 1	Mesh 1

7. Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
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 box, in the tree, select the check box 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
Component 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** check box.

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
Component 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 ()** 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**.

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.

RESULTS

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** check box.

Exterior Walls, Frozen Rotor

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Exterior Walls**.

2 In the **Settings** window for **Surface**, type `Exterior Walls, Frozen Rotor` in the **Label** text field.

3 Locate the **Selection** section. In the list, choose **17** and **19**.

4 Click **Remove from Selection**.

5 Select Boundaries 1, 4–8, 10, 11, 13–16, 18, 20–29, and 32–57 only.

Free Surface, Frozen Rotor

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

3 Select Boundary 12 only.

Velocity, Frozen Rotor

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice

1 In the **Model Builder** window, expand the **Velocity, Frozen Rotor** node.

2 Right-click **Results>Velocity, Frozen Rotor>Slice** and choose **Delete**.

Surface 1

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

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

3 From the **Dataset** list, choose **Free Surface, Frozen Rotor**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

6 Click **OK**.

Deformation 1

1 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 `spf.etaFS`.

5 In the **z component** text field, type `0`.

6 Locate the **Scale** section. Select the **Scale factor** check box.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Velocity, Frozen Rotor** 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 `3`.

5 Find the **z grid points** subsection. In the **Points** text field, type `30`.

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

7 Select the **Scale factor** check box. 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 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

5 Click **OK**.

Surface 2

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

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

3 From the **Dataset** list, choose **Exterior Walls, Frozen Rotor**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

6 In the **Velocity, Frozen Rotor** toolbar, click **Plot**.

Pressure, Frozen Rotor

1 In the **Model Builder** window, under **Results** click **Pressure (spf)**.

2 In the **Settings** window for **3D Plot Group**, type `Pressure, Frozen Rotor` in the **Label** text field.

3 In the **Pressure, Frozen Rotor** toolbar, click **Plot**.

Wall Resolution, Frozen Rotor

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

2 In the **Settings** window for **3D Plot Group**, type `Wall Resolution, Frozen Rotor` in the **Label** text field.

3 In the **Wall Resolution, Frozen Rotor** toolbar, click **Plot**.

Turbulent Viscosity, Frozen Rotor

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice 1

1 Right-click **Turbulent Viscosity, Frozen Rotor** and choose **Slice**.

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

3 In the **Expression** text field, type `spf.muT`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

Slice 2

1 Right-click **Slice 1** and choose **Duplicate**.

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

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

4 In the **Planes** text field, type `3`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Surface 1

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

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

3 From the **Dataset** list, choose **Exterior Walls, Frozen Rotor**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

6 In the **Turbulent Viscosity, Frozen Rotor** toolbar, click **Plot**.

Velocity xz, Frozen Rotor

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice 1

1 Right-click **Velocity xz, Frozen Rotor** and choose **Slice**.

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

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

4 In the **Velocity xz, Frozen Rotor** 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
Component 1	Mesh 1

8 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Exterior Walls, Time Dependent

1 In the **Model Builder** window, under **Results>Datasets** click **Exterior Walls**.

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

3 Locate the **Selection** section. In the list, choose **17** and **19**.

4 Click **Remove from Selection**.

5 Select Boundaries 1, 4–8, 10, 11, 13–16, 18, 20–29, and 32–57 only.

Free Surface, Time Dependent

1 In the **Model Builder** window, right-click **Free Surface, Frozen Rotor** and choose **Duplicate**.

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

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

xy-Plane, Time Dependent

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

2 In the **Settings** window for **Cut Plane**, type `xy-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 **xy-planes**.

Velocity, Time Dependent

1 In the **Model Builder** window, under **Results** click **Velocity (spf)**.

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice

1 In the **Model Builder** window, expand the **Velocity, Time Dependent** node.

2 Right-click **Results>Velocity, Time Dependent>Slice** and choose **Delete**.

Surface 1

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

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

3 From the **Dataset** list, choose **Free Surface, Time Dependent**.

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Wave>WaveLight** in the tree.

6 Click **OK**.

Arrow Volume 1

1 Right-click **Velocity, Time Dependent** 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 `3`.

5 Find the **z grid points** subsection. In the **Points** text field, type `30`.

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

7 Select the **Scale factor** check box. 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 Click  **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.

5 Click **OK**.

Surface 2

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

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

3 From the **Dataset** list, choose **Exterior Walls, Time Dependent**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

Pressure, Time Dependent

1 In the **Model Builder** window, under **Results** click **Pressure (spf)**.

2 In the **Settings** window for **3D Plot Group**, type `Pressure, Time Dependent` in the **Label** text field.

Wall Resolution, Time Dependent

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

2 In the **Settings** window for **3D Plot Group**, type `Wall Resolution, Time Dependent` in the **Label** text field.

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** check box.

4 From the **Plot group** list, choose **Velocity, Time Dependent**.

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 one to accurately capture the surface movement.

5 Find the **Algebraic variable settings** subsection. In the **Fraction of initial step for Backward Euler** text field, type `1`.

The initial time-step is already rather small, so the initialization time-step can be of the same order of magnitude. This will make the initialization easier.

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

RESULTS

Velocity, Time Dependent

In the **Velocity, Time Dependent** toolbar, click **Plot**.

Turbulent Viscosity, Time Dependent

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Turbulent Viscosity, 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 **Plot Settings** section. Clear the **Plot dataset edges** check box.

Slice 1

1 Right-click **Turbulent Viscosity, Time Dependent** and choose **Slice**.

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

3 In the **Expression** text field, type `spf.muT`.

4 Locate the **Plane Data** section. In the **Planes** text field, type `1`.

Slice 2

1 Right-click **Slice 1** and choose **Duplicate**.

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

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

4 In the **Planes** text field, type `3`.

5 Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Surface 1

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

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

3 From the **Dataset** list, choose **Exterior Walls, Time Dependent**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

6 In the **Turbulent Viscosity, Time Dependent** toolbar, click **Plot**.

Velocity xy, Time Dependent

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **xy-Plane, Time Dependent**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

Right-click **Velocity xy, Time Dependent** and choose **Surface**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Velocity xy, Time Dependent** 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** check box. In the associated text field, type `0.1`.

7 In the **Velocity xy, Time Dependent** toolbar, click **Plot**.