Created in COMSOL Multiphysics 5.6

Modular Mixer

Introduction

This model uses the Part Libraries to build a variety of mixers by combining two common types of vessels with two types of impellers. The mixers are baffled flat and dished bottom vessels with either pitched blade impellers or Rushton turbines. By changing the parameters in the model, you can modify the geometry to fit your applications. In case your mixer components cannot be built from the Geometry sequences available in the Part Libraries, you can add your own ones. The model includes three examples using the Rotating Machinery, Fluid Flow branch with the frozen-rotor study type. The first example solves a laminar mixing problem in a flat bottom mixer with a Rushton turbine. The second and third examples solve for turbulent mixing in a dished bottom mixer with a pitched blade impeller using the k-ε and k-ω turbulence models.

Model Definition

A typical mixer principally consists of two physical components — a vessel with or without baffles and an impeller structure. Baffles are used to suppress the main vortex formation in the bulk and thereby improve the mixing. The radial baffles, consisting of flat vertical solid strips set radially along the vessel wall, are evenly distributed around the perimeter of the vessel wall. Figure 1 shows the two combinations of vessels and impellers used in the simulations.

Figure 1: A baffled flat bottom mixer with a Rushton turbine (left) and a baffled dished bottom mixer with a pitched four blade impeller (right).

The Part Libraries contain geometry subsequences for building the mixer components shown in the figure. The types of components and their respective properties can be specified and varied by changing their parameters. The vessel used for mixers and reactors typically consists of a vertical cylinder with a dish-shaped or flat bottom. The vessel dimensions are defined by prescribing the vessel height H and diameter T. For the dished bottom vessel, the bottom minor radius R_d must also be specified. The mixer geometries and dimensions are shown in Figure 2.

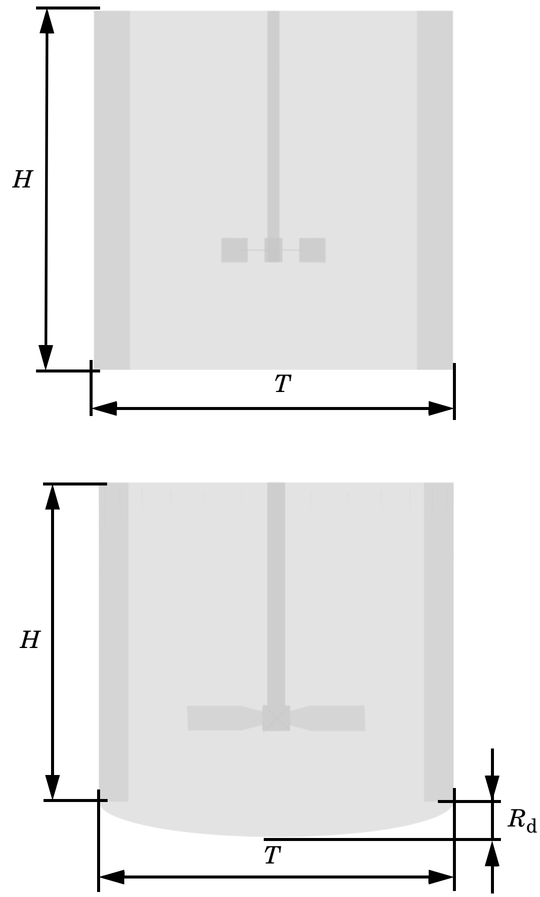


Figure 2: Side view of a flat bottom mixer (top) and a dished bottom mixer (bottom). The specified geometric properties are the vessel height (H), diameter (T), and for the dished bottom vessel, the bottom minor radius (R_d).

The rotation of the impeller drives the mixing of the fluid in the vessel. Due to the rotation and the design of the impeller, the liquid is discharged in the axial and radial directions by the impeller. Many types of impellers, with a variety of purposes, are used in industrial applications. Impellers used for mixing are commonly classified as radial or axial impellers corresponding to the predominant discharge direction. The impellers included in this example are the pitched blade impeller and the six-blade Rushton disc turbine. The pitched blade impeller is a so-called general purpose impeller discharging the fluid both radially and axially whereas the Rushton turbine is a radial impeller used for high-shear mixing of liquid solutions. The impellers are shown in Figure 3. Other types of impellers can be found in the Part Libraries.

Figure 3: A four-blade pitched impeller (left) and a six-blade Rushton disc turbine (right).

The geometry subsequences to build the impellers and vessels are imported from the Part Libraries: Rushton Impeller, Pitched Blade Impeller, Impeller Shaft, Flat Bottom Tank and Dished Bottom Tank. An assembly for the mixer geometry is formed under the main geometry node. The subsequences also define selections, such as Rotating Wall, Rotating Interior Wall, Interior Wall, or Symmetry to facilitate setting up the physics features.

You can modify the geometry to fit your own application. In case your mixer components cannot be built from the supplied geometry subsequences, you can add your own subsequences to the geometry file using the existing ones as templates.

Laminar Mixing

The first example is a simulation of silicone oil in a flat bottom mixer with a six-blade Rushton turbine rotating at 40 rps. The simulation is set up using a Rotating Machinery, Laminar Flow interface with the Frozen Rotor study type. The geometry is built from the supplied geometry file: `modular_mixer_geom.mph` using the following parameters:

TABLE 1: :LAMINAR MIXER GEOMETRY PARAMETERS

H	0.0805[m]	Vessel height
T	H	Vessel diameter
C	1/3*T	Clearance
B	4	Number of baffles
bw	T/10	Baffle width
Boz	0	Baffle offset from bottom
Rd	T/10	Minor radius of dished bottom
Da	T/3	Impeller diameter
shaft_diameter	Da/10	Shaft diameter
blade_length	Da/4	Blade length for Rushton turbine
blade_width	Da/5	Width of impeller blade

The fluid in the mixer is Silicone oil *Si1000* with a density of $972\ \mathrm{kg/m^3}$ and a dynamic viscosity of $1.0\ \mathrm{Pa{\cdot}s}$.

The example is taken from Ref. 1, which also includes comparisons with experiments from Ref. 2. In Ref. 1, results are presented for three different rotation rates. The mixer simulation in this example is for the highest of the three rotation rates. You can adjust the rotation rate to simulate the other two cases. This example uses the frozen-rotor study type whereas time-dependent (sliding-mesh) studies are performed in Ref. 1. Due to the topology variations resulting from the relative motion between the rotating impeller and baffled vessel, the flow field is really time dependent. The frozen-rotor solution should in this case be viewed as a quasi-steady approximation to the flow field. The solution gives a general idea of the circulation pattern set up in the mixing vessel, and may produce good approximations of certain averaged flow quantities, such as the power draw. You can use the results of the frozen-rotor simulation as initial conditions for a time-dependent study. If you want to perform a time dependent study, you will need to use an assembly on the geometry level so that the inner rotating volume as a continuity pair with sliding mesh toward the static outer volume. Then you can set the initial conditions to the frozen-rotor solution.

Results and Discussion

Figure 4 shows the velocity magnitude in the xz-plane and a projection of the velocity vectors on the yz-plane for the laminar-mixing simulation.

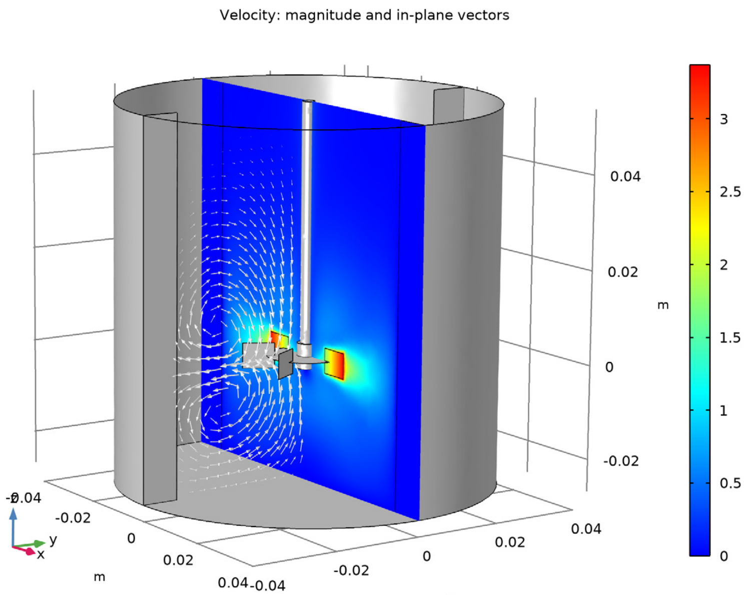


Figure 4: Velocity magnitude and in-plane velocity vectors (yz-plane) for laminar mixing of Silicon oil in a flat bottom mixer with a Rushton turbine.

The Rushton turbine discharges the fluid radially outward, whereby two zonal vortices are created. Mixing occurs between the top and bottom vortices but less intensely than within each vortex. This phenomenon is referred to as compartmentalization and is a characteristic feature of radial impellers.

The torque on the impeller, given by

$$M = \left| \hat{\mathbf{z}} \cdot \int_A \mathbf{r} \times \mathbf{T}\, dA \right|$$

where A is the surface area of the impeller and $\mathbf{T}$ is the total stress, is 0.011 Nm. The power draw is obtained from

$$P = \left|\Omega \cdot \int_A \mathbf{r} \times \mathbf{T}\, dA\right| = |\Omega| M$$

where Ω is the angular-velocity vector. The power draw is 2.86 W in this case. In Ref. 1, a flow number is defined as

$$N_\mathrm{q} = \frac{Q}{ND_\mathrm{a}^3}$$

where Q is the radial flux through a cylindrical surface with a radius of $0.186T$, extending axially from the bottom to the top of the impeller blades. The surface defined for the evaluation of the flow number is shown in Figure 5.

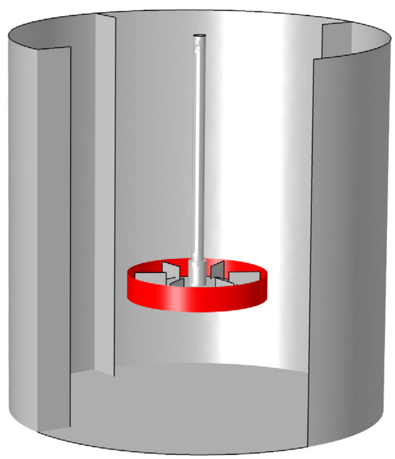


Figure 5: The surface defined for the evaluation of the flow number.

The value obtained from the simulation is 0.57, which is very close to the experimental value presented in Ref. 2. This illustrates how the frozen-rotor simulation, in addition to giving a general idea of the circulation pattern set up in the mixing vessel, can give good approximations of averaged flow quantities.

References

1. M.J. Rice, *High Resolution Simulation of Laminar and Transitional Flows in a Mixing Vessel*, PhD thesis, Virginia Polytechnic Institute and State University, 2011.

2. J. Hall, *Study of Viscous and Visco-elastic Flows with Reference to Laminar Stirred Vessels*, PhD thesis, Department of Mechanical Engineering, King's College London, 2005.

Application Library path: `Mixer_Module/Tutorials/modular_mixer`

Modeling Instructions

Begin by loading the geometry file.

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

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

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Fluid Flow>Single-Phase Flow>Rotating Machinery, Fluid Flow>Laminar Flow**.

4 Click **Add to Component 1** in the window toolbar.

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

MATERIALS

Add the values for Silicone oil Si1000.

Silicone oil Si1000

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Density	rho	972	kg/m³	Basic
Dynamic viscosity	mu	1	Pa·s	Basic

4 In the **Label** text field, type `Silicone oil Si1000`.

Set up the rotating domain conditions in Definitions.

DEFINITIONS

Rotating Domain 1

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

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

3 From the **Selection** list, choose **Rotating Fluid Domain**.

4 Locate the **Rotation** section. In the *f* text field, type `40`.

Use the predefined selections to set up the physics features.

LAMINAR FLOW (SPF)

Interior Wall 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Interior Wall**.

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

3 From the **Selection** list, choose **Interior Wall**.

Interior Wall 2

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

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

3 From the **Selection** list, choose **Rotating Interior Wall**.

As alternative the union selection for **Interior Wall 1** and **Interior Wall 2** can be done.

Symmetry 1

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

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

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

Pressure Point Constraint 1

1. In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2. In the **Settings** window for **Pressure Point Constraint**, locate the **Point Selection** section.
3. From the **Selection** list, choose **Pressure Point Constraint**.

DEFINITIONS

Define the postprocessing variables for the torque and power draw.

Variables 1

1. In 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
tau_riw	x*(spf.T_stress_uy+ spf.T_stress_dy)-y* (spf.T_stress_ux+ spf.T_stress_dx)	N/m	Torque per area (interior walls)
tau_rw	x*(spf.T_stressy)-y* (spf.T_stressx)	N/m	Torque per area (rotating walls)
P_riw	tau_riw*rot1.alphat	W/m²	Power draw per area (rotating interior walls)
P_rw	tau_rw*rot1.alphat	W/m²	Power draw per area (rotating walls)

Integration 1 (intop1)

Define a nonlocal integration coupling on **Rotating Interior Wall** to evaluate its contributions to the torque and power draw.

1. In the **Definitions** toolbar, click **Nonlocal 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 **Rotating Interior Wall**.

Integration 2 (intop2)

Define a nonlocal integration coupling on **Rotating Wall** to evaluate its contributions to the torque and power draw.

1. In the **Definitions** toolbar, click **Nonlocal 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 **Rotating Wall**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

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

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 **Preset Studies for Selected Physics Interfaces>Frozen Rotor**.

4 Click **Add Study** in the window toolbar.

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

STUDY 1

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

RESULTS

Cut Plane 1

1 In 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 **xz-planes**.

Cut Plane 2

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

Define the surface used to calculate the flow number.

Parameterized Surface 1

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

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

3 Find the **First parameter** subsection. In the **Maximum** text field, type `2*pi`.

4 Find the **Second parameter** subsection. In the **Minimum** text field, type `-Da/10`.

5 In the **Maximum** text field, type `Da/10`.

6 Locate the **Expressions** section. In the **x** text field, type `0.186*T*cos(s1)`.

7 In the **y** text field, type `0.186*T*sin(s1)`.

8 In the **z** text field, type `s2`.

9 Click **Plot**.

The surface for the calculation of the flow number is now visualized in the **Graphics** window.

Velocity: Magnitude and Vectors

Use the vessel and impeller surface plots set up under the **Pressure (spf)** results node to create a plot of the velocity magnitude and in-plane velocity vectors.

1 In the **Model Builder** window, expand the **Results>Pressure (spf)** node, then click **Pressure (spf)**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **All Walls**.

Pressure

1 In the **Model Builder** window, right-click **Pressure** and choose **Delete**.

2 Click **Yes** to confirm.

Upside Pressure

1 In the **Model Builder** window, right-click **Upside Pressure** and choose **Delete**.

2 Click **Yes** to confirm.

Downside Pressure

1 In the **Model Builder** window, right-click **Downside Pressure** and choose **Delete**.

2 Click **Yes** to confirm.

Surface 2

1 In the **Model Builder** window, right-click **Velocity: Magnitude and Vectors** and choose **Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 1**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Arrow Surface 1

1 Right-click **Velocity: Magnitude and Vectors** and choose **Arrow Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 2**.

4 Locate the **Expression** section. In the **x component** text field, type `0`.

5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

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

7 In the associated text field, type `0.005`.

8 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `1000`.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

Velocity: Magnitude and Vectors

1 In the **Model Builder** window, click **Velocity: Magnitude and Vectors**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Velocity: magnitude and in-plane vectors`.

5 In the **Velocity: Magnitude and Vectors** toolbar, click **Plot**.

Evaluate the torque on the impeller and the power draw using the postprocessing variables you defined.

Torque

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 Right-click **Global Evaluation 1** and choose **Rename**.

3 In the **Rename Global Evaluation** dialog box, type `Torque` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

6 In the table, enter the following settings:

Expression	Unit	Description
`abs(intop1(tau_riw)+intop2(tau_rw))`	`N*m`	

7 Click **Evaluate**.

Power Draw

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 Right-click **Global Evaluation 2** and choose **Rename**.

3 In the **Rename Global Evaluation** dialog box, type `Power Draw` in the **New label** text field.

4 Click **OK**.

5 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

6 In the table, enter the following settings:

Expression	Unit	Description
`abs(intop1(P_riw)+intop2(P_rw))`	W	

7 Click **Evaluate**.

Evaluate the flow number.

Flow Number

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration>Surface Integration**.

2 Right-click **Surface Integration 1** and choose **Rename**.

3 In the **Rename Surface Integration** dialog box, type `Flow Number` in the **New label** text field.

4 Click **OK**.

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

6 From the **Dataset** list, choose **Parameterized Surface 1**.

7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`(u*cos(s1)+v*sin(s1))/40[1/s]/Da^3`	1	

8 Click **Evaluate**.

Turbulent Mixing — k-ε Model

In the second example, turbulent mixing of water is simulated in a dished bottom mixer with a pitched four blade impeller rotating at 20 rpm. This simulation is set up using a Rotating Machinery, Turbulent Flow, k-ε interface with the frozen-rotor study type. The geometry is built from the supplied geometry file: `modular_mixer_turbulent_geom.mph` using the parameters in Table 2 below. Periodicity is utilized to reduce the computational time and hence a quarter of the domain is simulated in this case.

TABLE 2: TURBULENT MIXER GEOMETRY PARAMETERS

H	0.5[m]	Vessel height
T	H	Vessel diameter
C	1/4*H	Clearance
B	4	Number of baffles
bw	T/12	Baffle width
Boz	0	Baffle offset from bottom
Rd	T/10	Minor radius of dished bottom
Da	1/2*T	Impeller diameter
shaft_diameter	1/10*Da	Shaft diameter
blade_width	Da/5	Width of impeller blade
alpha	-45[deg]	Pitch angle
N_blades	4	Number of blades for pitched blade impeller

This example uses the frozen-rotor study type. Due to the topology variations resulting from the relative motion between the rotating impeller and baffled vessel, the flow field is really time dependent. The frozen-rotor solution should in this case be viewed as a quasi-steady approximation to the flow field. The solution gives a general idea of the circulation pattern set up in the mixing vessel, and may produce good approximations of certain averaged flow quantities, such as the power draw.

Results

Figure 6 shows a surface plot of the velocity magnitude and a projection of the velocity vectors on the yz-plane for the turbulent simulation with the k-ε turbulence model.

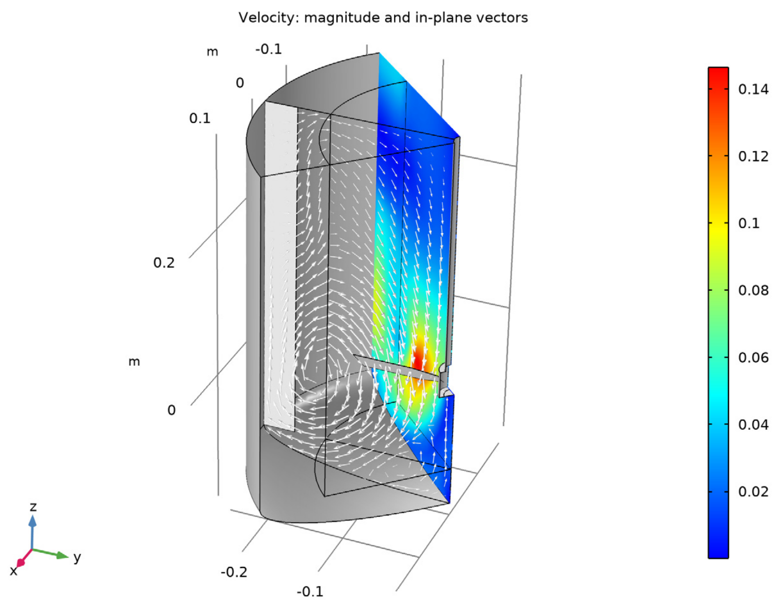


Figure 6: Velocity magnitude and in-plane velocity vectors (yz-plane) for turbulent mixing in a dished bottom mixer with a pitched blade impeller using the k*-ε turbulence model.*

The pitched blade impeller expels the fluid axially as well as radially, and a large zonal vortex, extending from the bottom to the top of the vessel, is formed. A smaller zonal vortex forms below the impeller and may in certain cases cause aggregation of heavy dispersed particles at the bottom center of the vessel.

The torque on the impeller, given by

$$M = \left| \hat{\mathbf{z}} \cdot \int_A \mathbf{r} \times \mathbf{T} \, dA \right|$$

where A is the surface area of the impeller and $\mathbf{T}$ is the total stress, is 0.016 Nm. The power draw is obtained from

$$P = \left| \Omega \cdot \int_A \mathbf{r} \times \mathbf{T}\, dA \right| = |\Omega| M$$

where Ω is the angular-velocity vector. The power draw is 0.033 W in this case.

The process of setting up a time-dependent study with the results of the frozen-rotor simulation as initial conditions is a bit more involved in this case because of the geometry reduction. You can either use general extrusions to define the solution on the remaining three quarters of the domain, or, you can recompute the frozen-rotor simulation on the full domain.

Modeling Instructions

ROOT

Begin by loading the geometry file.

1 From the **File** menu, choose **Open**.

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

GEOMETRY 1

Mesh Control Faces 1 (mcf1)

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

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

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

4 Click **Add to Component 1** in the window toolbar.

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

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 Click **Add to Component** in the window toolbar.

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

TURBULENT FLOW, K-ε (SPF)

Use the predefined selections to set up the physics features.

Interior Wall 1

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

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

3 From the **Selection** list, choose **Rotating Interior Wall**.

Interior Wall 2

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

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

3 From the **Selection** list, choose **Interior Wall**.

Symmetry 1

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

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

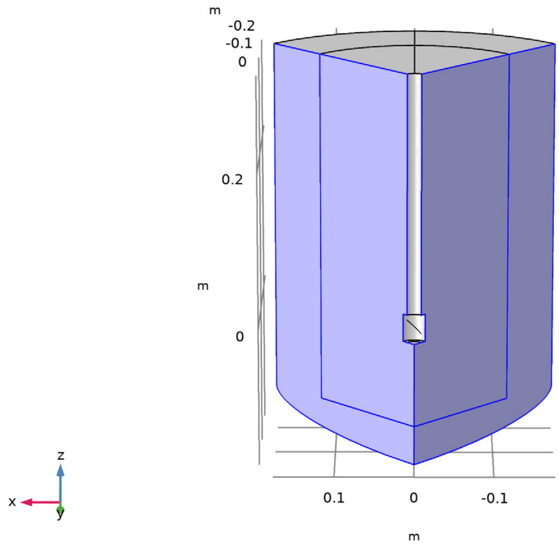
3 From the **Selection** list, choose **Symmetry**.

Use a **Periodic Flow Condition** to account for the excluded domain.

Periodic Flow Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Flow Condition**.

2 Select Boundaries 2, 6, 22, and 23 only.



Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

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

3 From the **Selection** list, choose **Pressure Point Constraint**.

DEFINITIONS

Rotating Domain 1

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

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

3 From the **Selection** list, choose **Rotating domain (Dished Bottom Tank 1)**.

4 Locate the **Rotation** section. In the *f* text field, type `20[rpm]`.

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

Add postprocessing variables for the torque and power draw.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
tau_riw	x*(spf.T_stress_uy+spf.T_stress_dy)-y*(spf.T_stress_ux+spf.T_stress_dx)	N/m	Torque per area (interior walls)
tau_rw	x*(spf.T_stressy)-y*(spf.T_stressx)	N/m	Torque per area (rotating walls)
P_riw	tau_riw*rot1.alphat	W/m²	Power draw per area (rotating interior walls)
P_rw	tau_rw*rot1.alphat	W/m²	Power draw per area (rotating walls)

Integration 1 (intop1)

Define a nonlocal integration coupling on **Rotating Interior Wall** to evaluate its contributions to the torque and power draw.

1. In the **Definitions** toolbar, click **Nonlocal 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 **Rotating Interior Wall**.

Integration 2 (intop2)

Define a nonlocal integration coupling on **Rotating Wall** to evaluate its contributions to the torque and power draw.

1. In the **Definitions** toolbar, click **Nonlocal 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 **Rotating Wall**.

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.

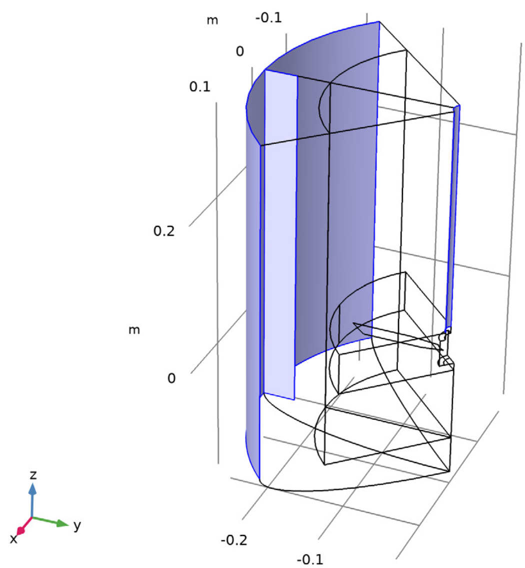
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.02`.

Size 1

1 Click the **Wireframe Rendering** button in the **Graphics** toolbar to see the interior of the geometry.

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

3 Select Boundaries 1, 15, 16, and 25 only.



4 In the **Settings** window for **Size**, click **Build Selected**.

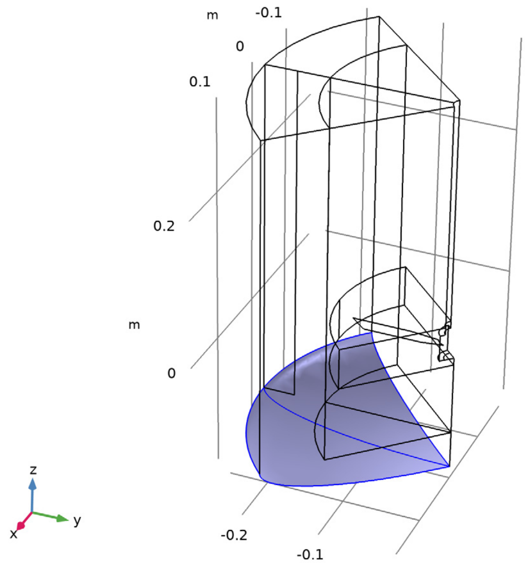
Size 2

1 In the **Model Builder** window, right-click **Mesh 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 Select Boundaries 3 and 17 only.



5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Fine**.

7 Click the **Custom** button.

8 Click **Build Selected**.

Refine the mesh in the domain defined by the control surfaces.

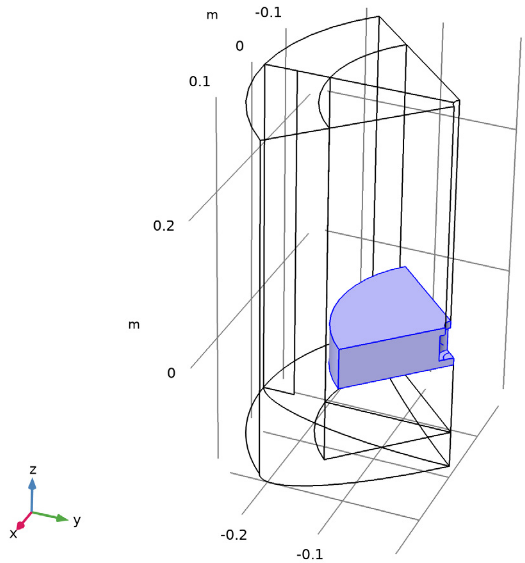
Size 3

1 Right-click **Mesh 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 **Domain**.

4 Select Domain 3 only.



5 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

6 From the **Predefined** list, choose **Finer**.

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

9 In the associated text field, type `0.0114`.

10 Select the **Minimum element size** check box.

11 In the associated text field, type `0.00123`.

12 Select the **Maximum element growth rate** check box.

13 In the associated text field, type `1.08`.

Refine the mesh further near the blade's surface.

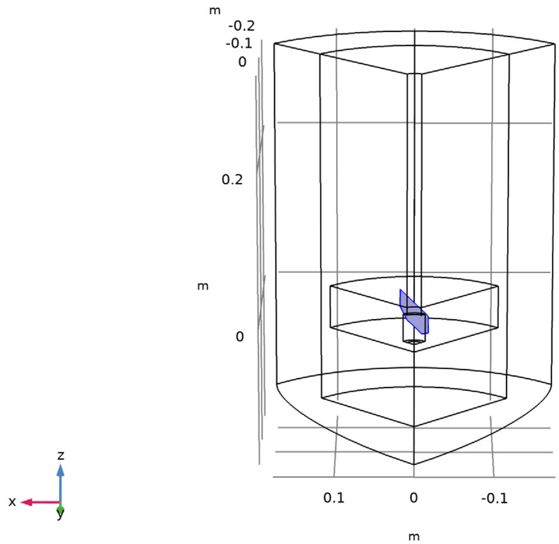
Size 4

1 Right-click **Mesh 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 Select Boundary 9 only.



5 Click the **Wireframe Rendering** button in the **Graphics** toolbar to disable wireframe rendering.

6 Locate the **Element Size** section. From the **Calibrate for** list, choose **Fluid dynamics**.

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

8 Click **Build All**.

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

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 **Preset Studies for Selected Physics Interfaces>Frozen Rotor**.

4 Click **Add Study** in the window toolbar.

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

STUDY 1

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

RESULTS

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.
2 In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
3 From the **Plane type** list, choose **General**.
4 In row **Point 1**, set **x** to `-0.2` and **y** to `-0.2`.
5 In row **Point 2**, set **x** to `-0.2`, **y** to `-0.2`, and **z** to `0.2`.
6 In row **Point 3**, set **y** to `0`.
7 Click **Plot**.

Cut Plane 2

1 In the **Results** toolbar, click **Cut Plane**.
2 In the **Settings** window for **Cut Plane**, click **Plot**.

Use the vessel and impeller surface plots set up under the **Pressure (spf)** results node to create a plot of the velocity magnitude and in-plane velocity vectors.

Downside Pressure

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

Upside Pressure

In the **Model Builder** window, right-click **Upside Pressure** and choose **Delete**.

Pressure

In the **Model Builder** window, right-click **Pressure** and choose **Delete**.

Velocity: Magnitude and Vectors

1 In the **Model Builder** window, under **Results** click **Pressure (spf)**.
2 In the **Settings** window for **3D Plot Group**, type `Velocity: Magnitude and Vectors` in the **Label** text field.

Surface 2

1 Right-click **Velocity: Magnitude and Vectors** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Cut Plane 1**.
4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Arrow Surface 1

1 Right-click **Velocity: Magnitude and Vectors** and choose **Arrow Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 2**.

4 Locate the **Expression** section. In the **x component** text field, type `0`.

5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

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

7 In the associated text field, type `0.26`.

8 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `500`.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

Velocity: Magnitude and Vectors

1 In the **Model Builder** window, click **Velocity: Magnitude and Vectors**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Velocity: magnitude and in-plane vectors`.

5 Locate the **Plot Settings** section. From the **View** list, choose **View 6**.

6 In the **Velocity: Magnitude and Vectors** toolbar, click **Plot**.

Evaluate the torque on the impeller and the power draw using the imported postprocessing variables.

Torque

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Torque` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`4*abs(intop1(tau_riw)+intop2(tau_rw))`	`N*m`	

Due to the periodicity conditions, multiply by 4 to get the torque on the whole impeller.

4 Click **Evaluate**.

Power Draw

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Power Draw` in the **Label** text field.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
4*abs(intop1(P_riw)+intop2(P_rw))	W	

Multiply by 4 to get the full power draw.

4 Click **Evaluate**.

Turbulent Mixing — k-ω Model

In the third example, similarly to the second example, turbulent mixing of water is simulated in a dished bottom mixer with a pitched four blade impeller rotating at 20 rpm. Here, the simulation is set up using a Rotating Machinery, Turbulent Flow, k-ω interface with the frozen-rotor study type. The k-ω model is better suited for flows with recirculation regions but, due to the strong nonlinearity in the turbulence coefficients, takes longer to converge than the k-ε model. By using the previous k-ε model as initial condition for the flow and turbulence variables, a faster convergence can be obtained. In the interfaces for turbulent fluid flow, there is a global option for creating a new turbulent flow interface. This option will add a new turbulent flow interface with the desired turbulence model, and automatically add settings for the initial values so that the previous solution is used in a new study.

Results

Figure 7 shows a surface plot of the velocity magnitude and a projection of the velocity vectors on the yz-plane for the turbulent simulation with the k-ω turbulence model.

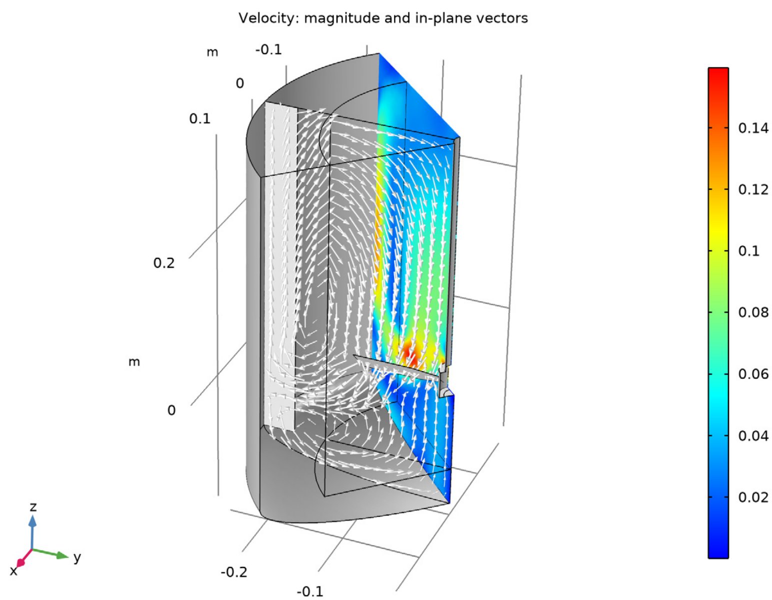


Figure 7: Velocity magnitude and in-plane velocity vectors (yz-plane) for turbulent mixing in a dished bottom mixer with a pitched blade impeller using the k-ω turbulence model.

Similarly to the k-ε simulation, the flow pattern is dominated by a large zonal vortex but its core appears to be more stretched in the vertical direction. The smaller zonal vortex below the impeller appears to be stretched in the radial direction.

The torque on the impeller, given by

$$M = \left| \hat{\mathbf{z}} \cdot \int_A \mathbf{r} \times \mathbf{T}\, dA \right|$$

where A is the surface area of the impeller and $\mathbf{T}$ is the total stress, is 0.025 Nm. The power draw is obtained from

$$P = \left| \Omega \cdot \int_A \mathbf{r} \times \mathbf{T}\, dA \right| = |\Omega| M$$

where Ω is the angular-velocity vector. The power draw is 0.053 W in this case. These values are higher than the corresponding results from the k-ε simulation. Although the k-ω model is better suited for these types of flows, experimental results are needed to determine whether these results are indeed more accurate.

The process of setting up a time-dependent study with the results of the frozen-rotor simulation as initial conditions is a bit more involved because of the geometry reduction. You can either use general extrusions to define the solution on the remaining three quarters of the domain, or, you can recompute the frozen-rotor simulation on the full domain.

Modeling Instructions

ROOT

Begin by loading the model file with the k-ε-model that contains the solution.

1 From the **File** menu, choose **Open**.

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

COMPONENT 1 (COMP1)

Add a new k-ω turbulent flow interface and a study that use the solution from the k-ε-model as initial values.

TURBULENT FLOW, K-ε (SPF)

Generate New Turbulence Model Interface 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** node.

2 Right-click **Component 1 (comp1)>Turbulent Flow, k-ε (spf)** and choose **Generate New Turbulence Model Interface**.

3 In the **Settings** window for **Generate New Turbulence Model Interface**, locate the **Turbulence Model Interface** section.

4 From the list, choose **Turbulent Flow, k-ω**.

5 Locate the **Model Generation** section. Click **Create**.

DEFINITIONS

Since the new interface has different variable names for the stress components, duplicate the current variables from the previous model and replace the respective interface tag spf with spf2 to create new variables for torque and power for the k-ω-model.

Variables 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Component 1 (comp1)>Definitions>Variables 1** and choose **Duplicate**.

3 In the **Settings** window for **Variables**, locate the **Variables** section.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
tau_riw2	x*(spf2.T_stress_uy+ spf2.T_stress_dy)-y* (spf2.T_stress_ux+ spf2.T_stress_dx)	N/m	Torque per area (interior walls)
tau_rw2	x*(spf2.T_stressy)-y* (spf2.T_stressx)	N/m	Torque per area (rotating walls)
P_riw2	tau_riw2*rot1.alphat	W/m²	Power draw per area (rotating interior walls)
P_rw2	tau_rw2*rot1.alphat	W/m²	Power draw per area (rotating walls)

STUDY 2

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

RESULTS

Velocity (spf2)

Duplicate the cut plane datasets from the k-ε-model to make similar plots for the k-ω-model.

Cut Plane 3

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Cut Plane 1** and choose **Duplicate**.

3 In the **Settings** window for **Cut Plane**, locate the **Data** section.

4 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Cut Plane 4

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Plane 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Duplicate the Velocity: Magnitude and Vectors-plot and change the data sources to be from k-ω-solution.

Velocity: Magnitude and Vectors 1

1 In the **Model Builder** window, right-click **Velocity: Magnitude and Vectors** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 2

1 In the **Model Builder** window, expand the **Velocity: Magnitude and Vectors 1** node, then click **Surface 2**.

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

3 From the **Dataset** list, choose **Cut Plane 3**.

4 Locate the **Expression** section. In the **Expression** text field, type `spf2.U`.

Arrow Surface 1

1 In the **Model Builder** window, click **Arrow Surface 1**.

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

3 From the **Dataset** list, choose **Cut Plane 4**.

4 Locate the **Expression** section. In the **y component** text field, type `v2`.

5 In the **z component** text field, type w2.

6 In the **Velocity: Magnitude and Vectors 1** toolbar, click **Plot**.

Group the plots for the k-ω-model

Pressure (spf2), Velocity (spf2), Velocity: Magnitude and Vectors 1, Wall Resolution (spf2)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf2)**, **Pressure (spf2)**, **Wall Resolution (spf2)**, and **Velocity: Magnitude and Vectors 1**.

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

k-omega

In the **Settings** window for **Group**, type `k-omega` in the **Label** text field.

Put the plots from the k-ε-model into a group to make it easier to know which plot that belong to the different solutions.

Velocity (spf), Velocity: Magnitude and Vectors, Wall Resolution (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf)**, **Velocity: Magnitude and Vectors**, and **Wall Resolution (spf)**.

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

k-epsilon

In the **Settings** window for **Group**, type `k-epsilon` in the **Label** text field.

Duplicate the global evaluations for torque and power. Replace the old variable names with new variables from the k-ω-solution.

Torque 1

1 In the **Model Builder** window, expand the **Results>Derived Values** node.

2 Right-click **Torque** and choose **Duplicate**.

3 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

4 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
4*abs(intop1(tau_riw2)+intop2(tau_rw2))	N*m	

6 Click **Evaluate**.

Power Draw 1

1 In the **Model Builder** window, right-click **Power Draw** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
4*abs(intop1(P_riw2)+intop2(P_rw2))	W	

5 Click **Evaluate**.

Power Draw 1, Torque 1

1 In the **Model Builder** window, under **Results>Derived Values**, Ctrl-click to select **Torque 1** and **Power Draw 1**.

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

Power/Torque k-omega

In the **Settings** window for **Group**, type `Power/Torque k-omega` in the **Label** text field.

Power Draw, Torque

1 In the **Model Builder** window, under **Results>Derived Values**, Ctrl-click to select **Torque** and **Power Draw**.

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

Power/Torque k-epsilon

In the **Settings** window for **Group**, type `Power/Torque k-epsilon` in the **Label** text field.