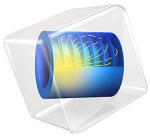


Laminar Modular Mixer — Template File

This model is a template MPH-file used by the laminar case of the Modular Mixer models. The geometry is a combination of a Rushton impeller and a flat bottom tank. The geometry subsequences to build the impeller and vessel are imported from the Part Libraries.

Application Library path: `Mixer_Module/Tutorials/modular_mixer_geom`

Modeling Instructions

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **3D**.

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
H	0.0805[m]	0.0805 m	Vessel height
T	H	0.0805 m	Vessel diameter
C	1/3*H	0.026833 m	Clearance
B	4	4	Number of baffles
bw	T/10	0.00805 m	Baffle width
Da	1/3*T	0.026833 m	Impeller diameter
shaft_diameter	1/10*Da	0.0026833 m	Shaft diameter
blade_length	Da/4	0.0067083 m	Blade length for Rushton turbine
blade_width	Da/5	0.0053667 m	Width of impeller blade

PART LIBRARIES

1 In the **Home** toolbar, click **Windows** and choose **Part Libraries**.

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

3 In the **Part Libraries** window, select **Mixer Module>Impellers, Radial>rushton_impeller** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Rushton Impeller 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Rushton Impeller 1 (pi1)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
d_hu	`shaft_diameter+Da/20`	0.004025 m	Diameter of impeller hub
l_ib	`blade_length`	0.0067083 m	Length of the impeller blades
w_ib	`blade_width`	0.0053667 m	Width of the impeller blade
d_id	`Da-2*(blade_length*3/4)`	0.016771 m	Disk diameter for Rushton and Smith turbines
d_im	`Da`	0.026833 m	Impeller diameter
hp_im	`-blade_width/2`	-0.0026833 m	Position of the lowest part of the impeller hub or impeller shaft along the z-axis
d_is	`shaft_diameter`	0.0026833 m	Diameter of impeller shaft

4 Click to expand the **Object Selections** section. Click **New Cumulative Selection**.

5 In the **New Cumulative Selection** dialog box, type `Impeller Domains` in the **Name** text field.

6 Click **OK**.

7 In the table, enter the following settings:

Name	Contribute to
Impeller	`Impeller Domains`
Tank	`None`

8 In the **Settings** window for **Part Instance**, click to expand the **Boundary Selections** section.

9 Click **New Cumulative Selection**.

10 In the **New Cumulative Selection** dialog box, type `Rotating Interior Wall` in the **Name** text field.

11 Click **OK**.

12 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

13 Click **New Cumulative Selection**.

14 In the **New Cumulative Selection** dialog box, type `Rotating Wall` in the **Name** text field.

15 Click **OK**.

16 In the table, enter the following settings:

Name	Contribute to
Impeller blades	`Rotating Interior Wall`
Impeller disk	`Rotating Interior Wall`
Impeller hub	`Rotating Wall`
Surfaces to hide	`None`
Control surface uppper	`None`
Control surface lower	`None`
Control surface side	`None`

17 In the **Settings** window for **Part Instance**, click to expand the **Edge Selections** section.

18 Click **New Cumulative Selection**.

19 In the **New Cumulative Selection** dialog box, type `Remove Edges` in the **Name** text field.

20 Click **OK**.

21 In the table, enter the following settings:

Name	Contribute to
Edges to remove	`Remove Edges`

22 In the **Settings** window for **Part Instance**, locate the **Selection Settings** section.

23 Select the **Keep noncontributing selections** check box.

PART LIBRARIES

1 In the **Home** toolbar, click  **Windows** and choose **Part Libraries**.

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

3 In the **Part Libraries** window, select **Mixer Module>Shafts>impeller_shaft** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Impeller Shaft 1 (pi2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Impeller Shaft 1 (pi2)**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
hp_im	`-blade_width/2`	-0.0026833 m	Position of the lowest part of the impeller hub or impeller shaft along the z-axis
d_is	`shaft_diameter`	0.0026833 m	Impeller shaft diameter
l_is	`H-C+blade_width`	0.059033 m	Impeller shaft length

4 Click to expand the **Object selections** section. In the table, enter the following settings:

Name	Contribute to
Impeller	`Impeller Domains`

5 Click to expand the **Boundary selections** section. In the table, enter the following settings:

Name	Contribute to
Exterior	`None`
Impeller	`Rotating Wall`

6 Click to expand the **Edge selections** section. In the table, enter the following settings:

Name	Contribute to
Edges to remove	`Remove Edges`
Impeller	`None`

7 Locate the **Selection Settings** section. Select the **Keep noncontributing selections** check box.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

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

4 From the **Repair tolerance** list, choose **Relative**.

PART LIBRARIES

1 In the **Geometry** toolbar, click **Parts** and choose **Part Libraries**.

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

3 In the **Part Libraries** window, select **Mixer Module>Tanks>flat_bottom_tank** in the tree.

4 Click **Add to Geometry**.

GEOMETRY 1

Flat Bottom Tank 1 (pi3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Flat Bottom Tank 1 (pi3)**.

2 In the **Settings** window for **Part Instance**, locate the **Position and Orientation of Output** section.

3 Find the **Rotation** subsection. In the **Rotation angle** text field, type `90`.

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

Name	Expression	Value	Description
n_ba	B	4	Number of baffles
w_ba	bw	0.00805 m	Baffle width
d_im	Da	0.026833 m	Impeller diameter
d_ta	T	0.0805 m	Tank diameter
h_ta	H	0.0805 m	Tank height
hp_ta	-C	-0.026833 m	Height position, cylindrical surface
rf_ta	0	0 m	Fillet radius of lower tank edge

5 Locate the **Boundary Selections** section. Click **New Cumulative Selection**.

6 In the **New Cumulative Selection** dialog box, type `Symmetry` in the **Name** text field.

7 Click **OK**.

8 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

9 Click **New Cumulative Selection**.

10 In the **New Cumulative Selection** dialog box, type `Interior Wall` in the **Name** text field.

11 Click **OK**.

12 In the **Settings** window for **Part Instance**, locate the **Boundary Selections** section.

13 Click **New Cumulative Selection**.

14 In the **New Cumulative Selection** dialog box, type `View suppression` in the **Name** text field.

15 Click **OK**.

16 Click to expand the **Boundary selections** section. In the table, enter the following settings:

Name	Contribute to
Exterior	None
Fluid surface	Symmetry
Tank walls	None
Baffles	Interior Wall
Tank	None
Surfaces to hide	View suppression

17 In the **Settings** window for **Part Instance**, locate the **Selection Settings** section.

18 Select the **Keep noncontributing selections** check box.

Fluid Domain

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.

2 In the **Settings** window for **Difference**, type `Fluid Domain` in the **Label** text field.

3 Locate the **Difference** section. From the **Objects to add** list, choose **Tank (Flat Bottom Tank 1)**.

4 From the **Objects to subtract** list, choose **Impeller Domains**.

5 From the **Repair tolerance** list, choose **Relative**.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

7 From the **Show in physics** list, choose **All levels**.

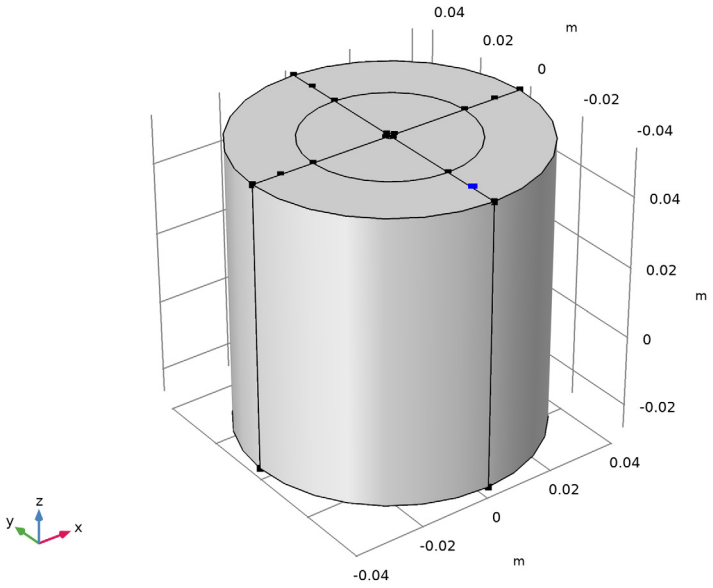
Flat Pressure Point

1 In the **Geometry** toolbar, click  **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Flat Pressure Point` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Point**.

4 On the object **dif1**, select Point 34 only.



5 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Pressure Point Constraint` in the **Name** text field.

7 Click **OK**.

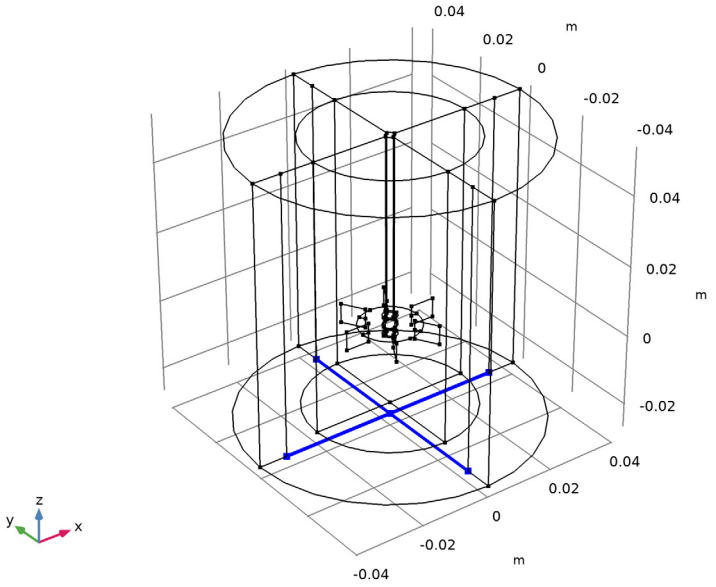
Edges to Remove

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Edges to Remove` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Edge**.

4 On the object **dif1**, select Edges 9, 61, 78, and 79 only.



5 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Remove Edges**.

Rotating Fluid Domain

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

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

3 On the object **dif1**, select Domain 2 only.

4 In the **Settings** window for **Explicit Selection**, type `Rotating Fluid Domain` in the **Label** text field.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 In the **Settings** window for **Ignore Edges**, locate the **Input** section.

3 From the **Edges to ignore** list, choose **Remove Edges**.

4 Clear the **Ignore adjacent vertices** check box.

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

The model geometry is now complete.