



Model created in COMSOL Multiphysics 6.4

Radial Pump with Mixing Planes

Introduction

Pumps are some of the most ubiquitous and essential turbomachines driving modern society. They are employed in a variety of applications, mainly for transporting and compressing fluids. Centrifugal or radial pumps work by adding kinetic energy to the fluid in the radial direction, which is later recovered as elevated pressure in the diffuser.

The flow field inside a pump at any instant depends on, among many factors, the spatial position of the rotating components, such as rotor blades, relative to the stationary components, such as guide vanes and stators. A mixing plane methodology averages flow quantities across the interface between rotating and stationary domains along the circumferential direction, thereby, providing a representative average value of the flow field irrespective of the spatial configuration.

The present example computes an averaged flow field in a radial pump model using the mixing plane methodology with a frozen rotor study. The solution approximates an overall flow resulting from various possible positions of the rotor blade. This circumvents the need to perform costly, time-dependent simulations.

Model Definition

The radial pump geometry in the current model is a modified version of the experimental setup in [Ref. 1](#). The pump receives flow through a central cylindrical pipe along the axial direction. The flow is redirected by the pump casing and pushed in the radial direction by six rotating blades. The flow moves into a circular diffuser and exits the device via twelve outlet pipes.

GEOMETRY

Due to the rotational symmetry, only one-sixth of the full pump geometry is modeled, as shown in [Figure 1](#). Circular arcs are used in the construction of the blade profile. See the

[Geometry Modeling Instructions](#) section for the step-by-step instructions on constructing the model geometry.

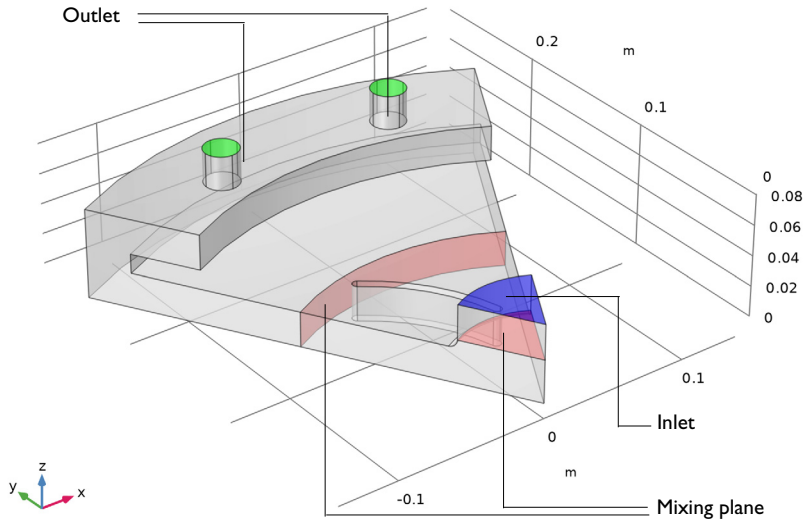


Figure 1: The geometry setup of the model shows one-sixth of the radial pump. Some of the boundary conditions for the flow problem are annotated.

A rotating domain containing the rotor blade is connected to two stationary domains via mixing plane boundaries, one to the tubular inlet at the upstream end, and the other to the circular diffuser at the downstream end.

PHYSICS INTERFACE SETTINGS

The model uses a Turbulent Flow, k - ϵ physics interface. The fluid properties are taken from predefined properties for water available in the material library. A Wall feature with the slip wall condition is chosen to simplify the model, while a Periodic Flow Condition accounts for rotational periodicity. At the inlet boundary, an average normal velocity is specified. The outlet is open to atmosphere at sea level. Two Mixing Plane feature nodes with axial and radial flow directions, respectively, are added to the model. A suitable edge is chosen for each instance of the Mixing Plane feature where mixing is consolidated. The rotational speed is 600 rotations per minute and the inlet mass flow rate is 13.2 kg/s.

MESHING

A physics-controlled mesh with “normal” element size is built automatically. The mesh is composed of unstructured tetrahedral elements, as shown in [Figure 2](#).

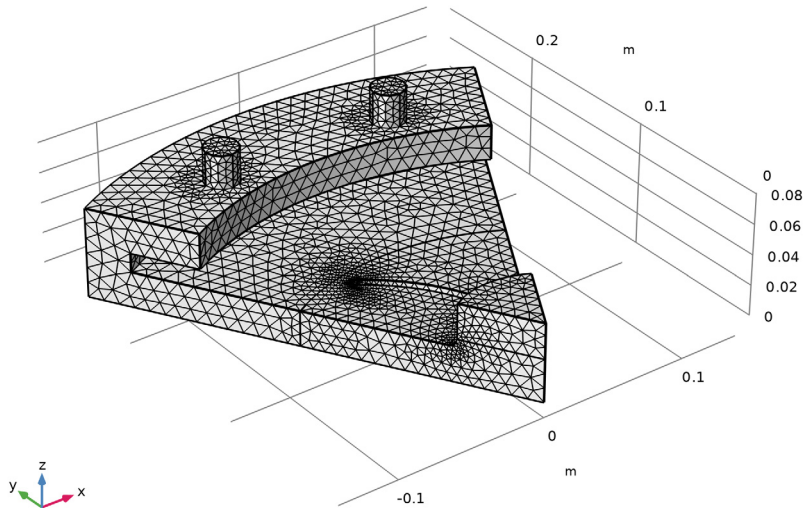


Figure 2: Unstructured tetrahedral mesh used in the model.

STUDY

The steady-state, frozen rotor solution is obtained in two stages. Two Frozen Rotor study steps are added. The solution to the first step corresponds to ignoring the Mixing Plane features. This solution serves as a good initial solution for the second Frozen Rotor study step with Mixing Plane features enabled. Pseudo time stepping is turned on for accelerating the solution for both Frozen Rotor study steps.

Results and Discussion

The frozen rotor solution with mixing plane condition is shown in [Figure 3](#) and [Figure 4](#). The surface plot of velocity in [Figure 3](#) help in visualizing the averaging of flow quantities performed at the mixing plane boundaries. In addition, the surface arrow plots show the velocity field immediately upstream (in red) and downstream (in black) of the mixing plane boundaries. The high velocity jet of fluid ejected from the rotating domain at the upstream

end of the mixing plane boundary with radial flow is observed along its downstream end as averaged along the circumferential direction.

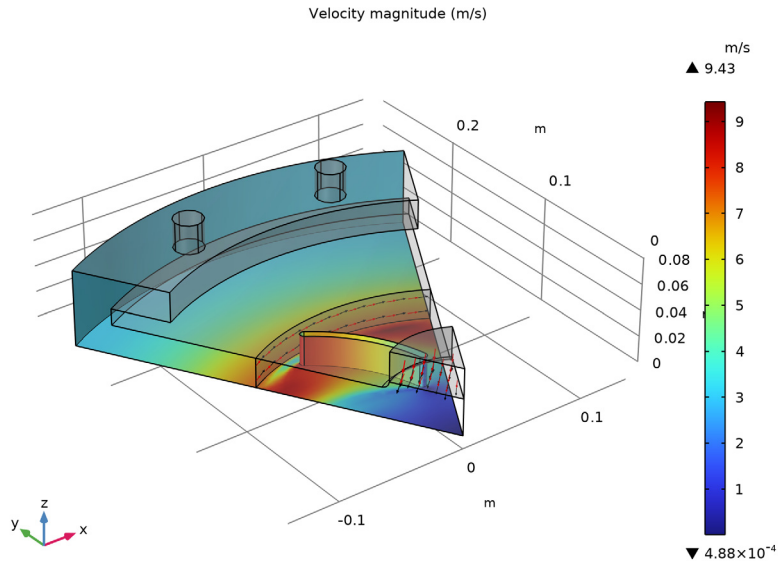


Figure 3: Frozen rotor solution of the radial pump model. Surface plot of fluid velocity show the averaging of flow quantities at the mixing plane boundaries. Surface arrow plots show fluid velocity upstream (in red) and downstream (in black) at both mixing plane boundaries.

The increase in the fluid pressure developed by the pump's rotation may be observed in [Figure 4](#). Moreover, streamlines help visualize the flow field.

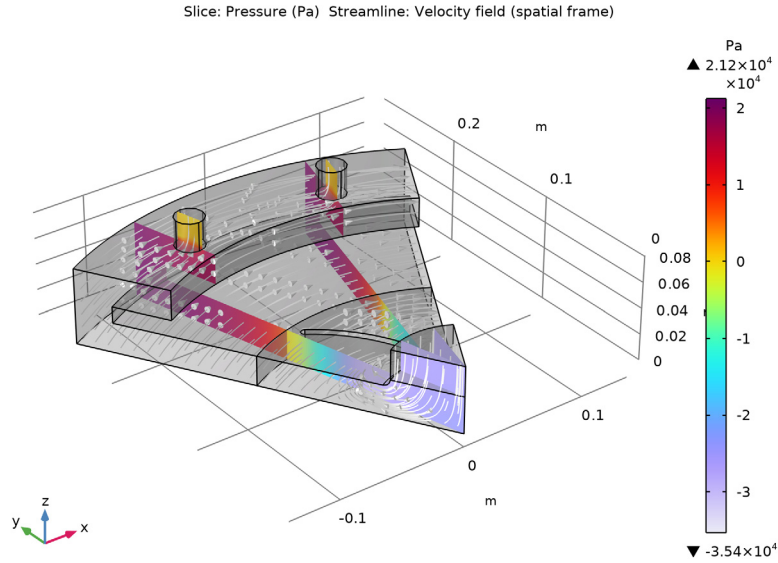


Figure 4: Frozen rotor solution of the radial pump model. Slice plots of fluid pressure show the averaging of flow quantities at the mixing plane boundaries. Streamline plot helps visualize the flow field.

Moreover, the solution with mixing plane preserves normal mass flux across various boundaries, as computed in the model and tabulated in [Table 1](#). Additionally, the total head created by the pump, computed in the current model, shows good agreement with that computed from a time-averaged, full-model solution (which is left as an exercise for the user), thereby, highlighting the advantage of using the mixing plane methodology.

TABLE 1: NORMAL MASS FLUX [KG/M³] ACROSS VARIOUS BOUNDARIES IN THE MODEL.

Inlet	Outlet	Mixing plane 1, upside	Mixing plane 1, downside	Mixing plane 2, upside	Mixing plane 2, downside
2.1950	2.1951	2.1966	2.1933	2.1969	2.1933

Reference


I. N. Krause, K. Zahringer, and E. Pap, “Time-resolved particle imaging velocimetry for the investigation of rotating stall in a radial pump,” *Experiments in Fluids*, vol. 39, pp. 192–201, 2005.

Application Library path: Mixer_Module/Tutorials/radial_pump


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 Click  **Done**.

GEOMETRY I

Import the radial pump geometry from the geometry sequence file.


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 `radial_pump_geom_sequence.mph`.

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

DEFINITIONS

Explicit 1

In the **Definitions** toolbar, click  **Explicit**.


GLOBAL DEFINITIONS

Import flow parameters from file.

Parameters 2

1 In the **Home** toolbar, click  **Parameters** and choose **Add > Parameters**.

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

- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `radial_pump_parameters_2.txt`.


DEFINITIONS

Create explicit selections for important boundaries and domains.

Inlet Boundary


- 1 In the **Model Builder** window, under **Component 1 (comp1) > Definitions > Selections** click **Explicit 1**.
- 2 Select Domain 4 only.
- 3 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 4 From the **Geometric entity level** list, choose **Boundary**.
- 5 Select Boundary 26 only.
- 6 Locate the **Color** section. From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.
- 7 Click **Define custom colors**.
- 8 Set the RGB values to 0, 0, and 255, respectively.
- 9 Click **Add to custom colors**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.
- 11 In the **Label** text field, type Inlet Boundary.

Outlet Boundary


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 16 and 33 only.
- 5 Locate the **Color** section. From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.
- 6 Click **Define custom colors**.
- 7 Set the RGB values to 0, 255, and 0, respectively.
- 8 Click **Add to custom colors**.
- 9 Click **Show color palette only** or **OK** on the cross-platform desktop.

10 In the **Label** text field, type Outlet Boundary.


Mixing Plane: Axial Flow

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 25 only.
- 5 In the **Label** text field, type Mixing Plane: Axial Flow.
- 6 Locate the **Color** section. From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.
- 7 Click **Define custom colors**.
- 8 Set the RGB values to 255, 0, and 0, respectively.
- 9 Click **Add to custom colors**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Mixing Plane: Radial Flow

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 10 only.
- 5 In the **Label** text field, type Mixing Plane: Radial Flow.
- 6 Locate the **Color** section. From the **Color** list, choose **None** or — if you are running the cross-platform desktop — **Custom**. On the cross-platform desktop, click the **Color** button.
- 7 Click **Define custom colors**.
- 8 Set the RGB values to 255, 0, and 0, respectively.
- 9 Click **Add to custom colors**.
- 10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Periodic Boundary

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 1, 9, 23, 27, 28, and 36 only.

5 In the **Label** text field, type **Periodic Boundary**.

Rotating Domain

1 In the **Definitions** toolbar, click  **Explicit**.

2 Select **Domain 2** only.

3 In the **Settings** window for **Explicit**, type **Rotating Domain** in the **Label** text field.

View 1

Turn on transparency temporarily to view geometry selections, specially the mixing plane boundaries.

1 In the **Model Builder** window, under **Component 1 (comp1) > Definitions** click **View 1**.

2 In the **Settings** window for **View**, click to expand the **Transparency** section.

3 Select the **Transparency** checkbox.

4 Clear the **Transparency** checkbox.

Average At Inlet

Add average operators at inlet and outlet boundaries.

1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type **Average At Inlet** in the **Label** text field.

3 In the **Operator name** text field, type **avein**.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Inlet Boundary**.

Average At Outlet

1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type **Average At Outlet** in the **Label** text field.

3 In the **Operator name** text field, type **aveout**.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 From the **Selection** list, choose **Outlet Boundary**.

COMPONENT 1 (COMP1)

Add a rotating domain to the moving mesh interface.

Rotating Domain 1



1 In the **Physics** toolbar, click  **Moving Mesh** and choose **Rotating Domain**.

- 2 In the **Settings** window for **Rotating Domain**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **Rotating Domain**.
- 4 Locate the **Rotation** section. From the **Rotation type** list, choose **Specified rotational velocity**.
- 5 In the ω text field, type -omega.

MATERIALS

Add predefined material properties for water from library.



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 Click the **Add to Component** button in the window toolbar.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

COMPONENT 1 (COMP1)

Add a Turbulent Flow, $k-\epsilon$ physics interface.

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 > Turbulent Flow > Turbulent Flow, k- ϵ (spf)**.
- 4 Click the **Add to Component 1** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.

TURBULENT FLOW, K- ϵ (SPF)

Turn on pseudo time stepping.

- 1 In the **Settings** window for **Turbulent Flow, k- ϵ** , click to expand the **Advanced Settings** section.
- 2 Find the **Pseudo time stepping** subsection. From the **Use pseudo time stepping for stationary equation form** list, choose **On**.


Wall 1

Choose slip condition for all wall boundaries.

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Turbulent Flow, k-ε (spf)** click **Wall 1**.
- 2 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.
- 3 From the **Wall condition** list, choose **Slip**.


Inlet 1

Add a velocity inlet boundary condition.

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Inlet Boundary**.
- 4 Locate the **Velocity** section. In the U_0 text field, type `mfr/material.def.rho/(pi*R_in^2)`.


Outlet 1

Add a zero pressure outlet condition.

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Outlet**.
- 2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Outlet Boundary**.

Mixing Plane 1

Add an axial flow mixing plane boundary condition.


- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Mixing Plane**.
- 2 In the **Settings** window for **Mixing Plane**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Mixing Plane: Axial Flow**.
- 4 Locate the **Edge Where Mixing Is Consolidated** section. Click to select the **Activate Selection** toggle button.
- 5 Select Edge 42 only.
- 6 Locate the **Axis Definition** section. Specify the \mathbf{e}_r vector as

0	x
1	y

Mixing Plane 2

Add a radial flow mixing plane boundary condition.

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Mixing Plane**.


- 2 In the **Settings** window for **Mixing Plane**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Mixing Plane: Radial Flow**.
- 4 Locate the **Edge Where Mixing Is Consolidated** section. Click to select the  **Activate Selection** toggle button.
- 5 Select Edge 14 only.
- 6 Locate the **Axis Definition** section. Specify the \mathbf{e}_r vector as

0	x
1	y

- 7 Locate the **Flow Direction** section. From the **Flow direction** list, choose **Radial**.




Periodic Flow Condition 1

Add a periodic flow boundary condition.

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Periodic Flow Condition**.
- 2 In the **Settings** window for **Periodic Flow Condition**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Periodic Boundary**.

MESH 1


Build a physics-controlled predefined mesh with a normal element size option.

- 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 From the **Element size** list, choose **Normal**.
- 4 Click  **Build All**.
- 5 In the **Graphics** window toolbar, click  next to  **Colors**, then choose **Show Selection Colors**.

ROOT

Add two Frozen Rotor study steps. Disable Mixing Plane features for the first study step.

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 the **Add Study** button in the window toolbar.


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

STUDY 1




Step 2: Frozen Rotor 1

In the **Model Builder** window, under **Study 1** right-click **Step 1: Frozen Rotor** and choose **Duplicate**.

Step 1: Frozen Rotor

- 1 In the **Settings** window for **Frozen Rotor**, 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) > Turbulent Flow, k-ε (spf) > Mixing Plane 1** and **Component 1 (comp1) > Turbulent Flow, k-ε (spf) > Mixing Plane 2**.
- 4 Click  **Disable**.
- 5 In the **Model Builder** window, click **Study 1**.
- 6 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 7 Clear the **Generate default plots** checkbox.


Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 Click  **Show Default Plots**.
Solve both study steps.
- 3 Click  **Compute**.

RESULTS

Add surface and arrow plots to visualize velocity values.

Velocity


- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type **Velocity** in the **Label** text field.
- 3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.
- 4 Select the **Show titles** checkbox.
- 5 Select the **Show units** checkbox.

Arrow Surface 1

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

- 2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
- 3 In the **x-component** text field, type up (u).
- 4 In the **y-component** text field, type up (v).
- 5 In the **z-component** text field, type up (w).
- 6 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 7 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type 20.
- 8 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.
- 9 Select the **Scale factor** checkbox. In the associated text field, type $1e-3$.

Selection 1

- 1 Right-click **Arrow Surface 1** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 10 in the **Selection** text field.
- 5 Click **OK**.

Arrow Surface 2



- 1 In the **Model Builder** window, under **Results > Velocity** right-click **Arrow Surface 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
- 3 In the **x-component** text field, type down (u).
- 4 In the **y-component** text field, type down (v).
- 5 In the **z-component** text field, type down (w).
- 6 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Tail**.
- 7 From the **Color** list, choose **Black**.

Arrow Surface 3

- 1 Right-click **Arrow Surface 1** and choose **Duplicate**.
- 2 In the **Model Builder** window, click **Arrow Surface 3**.
- 3 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.
- 4 In the **Scale factor** text field, type $5e-3$.

Selection 1

- 1 In the **Model Builder** window, click **Selection 1**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.

- 3 Click  **Clear Selection**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 25 in the **Selection** text field.
- 6 Click **OK**.

Arrow Surface 4

- 1 In the **Model Builder** window, under **Results > Velocity** right-click **Arrow Surface 3** and choose **Duplicate**.
- 2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
- 3 In the **x-component** text field, type $\text{down}(u)$.
- 4 In the **y-component** text field, type $\text{down}(v)$.
- 5 In the **z-component** text field, type $\text{down}(w)$.
- 6 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Tail**.
- 7 From the **Color** list, choose **Black**.

Surface 1

- 1 In the **Model Builder** window, right-click **Velocity** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type 1.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Material Appearance 1


- 1 Right-click **Surface 1** and choose **Material Appearance**.
- 2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
- 3 From the **Appearance** list, choose **Custom**.
- 4 From the **Material type** list, choose **Steel (anodized)**.
- 5 From the **Color** list, choose **Gray**.

Transparency 1

- 1 In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.
- 2 In the **Settings** window for **Transparency**, locate the **Transparency** section.
- 3 Find the **Transparency** subsection. In the **Transparency** text field, type 0.7.

Selection 1


- 1 Right-click **Surface 1** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.

- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 2-8, 10-22, 24-26, 29-35 in the **Selection** text field.
- 5 Click **OK**.




Surface 2

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

Selection 1


- 1 In the **Model Builder** window, right-click **Surface 2** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 2, 3, 11, 19, 20, 22, 29 in the **Selection** text field.
- 5 Click **OK**.

Velocity

- 1 In the **Model Builder** window, under **Results** click **Velocity**.
- 2 In the **Velocity** toolbar, click  **Plot**.
- 3 Click the  **Go to Default View** button in the **Graphics** toolbar.
- 4 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Pressure

Add slice plots to visualize pressure values.

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Pressure in the **Label** text field.

Surface 1

- 1 Right-click **Pressure** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type 1.
- 4 Locate the **Title** section. From the **Title type** list, choose **None**.

Material Appearance 1


- 1 Right-click **Surface 1** and choose **Material Appearance**.
- 2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
- 3 From the **Appearance** list, choose **Custom**.
- 4 From the **Material type** list, choose **Steel (anodized)**.

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

Transparency 1

- 1 In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.
- 2 In the **Settings** window for **Transparency**, locate the **Transparency** section.
- 3 Find the **Transparency** subsection. In the **Transparency** text field, type 0.6.

Selection 1

- 1 Right-click **Surface 1** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 Click  **Paste Selection**.
- 4 In the **Paste Selection** dialog, type 2-8, 10-22, 24-26, 29-35 in the **Selection** text field.
- 5 Click **OK**.

Slice 1


- 1 In the **Model Builder** window, right-click **Pressure** and choose **Slice**.
- 2 In the **Settings** window for **Slice**, locate the **Expression** section.
- 3 In the **Expression** text field, type p.
- 4 Locate the **Plane Data** section. From the **Plane type** list, choose **General**.
- 5 In row **Point 2**, set **x** to 0.
- 6 In row **Point 2**, set **z** to 1.
- 7 In row **Point 3**, set **x** to $\cos(75[\text{deg}])$.
- 8 In row **Point 3**, set **y** to $\sin(75[\text{deg}])$.
- 9 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Slice 2



- 1 Right-click **Slice 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Slice**, locate the **Plane Data** section.
- 3 In row **Point 3**, set **x** to $\cos(105[\text{deg}])$.
- 4 In row **Point 3**, set **y** to $\sin(105[\text{deg}])$.
- 5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.
- 6 Locate the **Title** section. From the **Title type** list, choose **None**.

Streamline 1



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

- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 3 In the **Number** text field, type 100.
- 4 Locate the **Selection** section. Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 1, 9 in the **Selection** text field.
- 6 Click **OK**.
- 7 In the **Settings** window for **Streamline**, locate the **Coloring and Style** section.
- 8 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.
- 9 From the **Color** list, choose **White**.

Streamline 2




- 1 Right-click **Streamline 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
- 3 In the **Number** text field, type 300.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Locate the **Streamline Positioning** section. In the **Number** text field, type 30.
- 6 Locate the **Selection** section. Click  **Clear Selection**.
- 7 Click  **Paste Selection**.
- 8 In the **Paste Selection** dialog, type 23, 26, 28 in the **Selection** text field.
- 9 Click **OK**.
- 10 In the **Settings** window for **Streamline**, click to expand the **Inherit Style** section.
- 11 From the **Plot** list, choose **Streamline 1**.

Streamline 3

- 1 Right-click **Streamline 2** and choose **Duplicate**.
- 2 In the **Settings** window for **Streamline**, locate the **Selection** section.
- 3 Click  **Clear Selection**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog, type 19, 22, 29 in the **Selection** text field.
- 6 Click **OK**.

Pressure

- 1 In the **Model Builder** window, click **Pressure**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.
- 3 Select the **Show maximum and minimum values** checkbox.

- 4 Select the **Show units** checkbox.
- 5 Select the **Show titles** checkbox.
- 6 In the **Pressure** toolbar, click  **Plot**.
- 7 Click the  **Go to Default View** button in the **Graphics** toolbar.
- 8 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Inlet Mass Flux

Evaluate normal mass flux across inlet, outlet and mixing plane boundaries. Also compute the pump head.

- 1 In the **Results** toolbar, click $\frac{8.85}{e-12}$ **More Derived Values** and choose **Integration > Surface Integration**.
- 2 In the **Settings** window for **Surface Integration**, type Inlet Mass Flux in the **Label** text field.
- 3 Locate the **Selection** section. From the **Selection** list, choose **Inlet Boundary**.
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
$spf.rho*(u*nx+v*ny+w*nz)$	kg/s	Inlet Mass Flux

Outlet Mass Flux

- 1 In the **Results** toolbar, click $\frac{8.85}{e-12}$ **More Derived Values** and choose **Integration > Surface Integration**.
- 2 In the **Settings** window for **Surface Integration**, type Outlet Mass Flux in the **Label** text field.
- 3 Locate the **Selection** section. From the **Selection** list, choose **Outlet Boundary**.
- 4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
$spf.rho*(u*nx+v*ny+w*nz)$	kg/s	Outlet Mass Flux

Mixing Plane 1 Mass Flux

- 1 In the **Results** toolbar, click $\frac{8.85}{e-12}$ **More Derived Values** and choose **Integration > Surface Integration**.
- 2 In the **Settings** window for **Surface Integration**, type Mixing Plane 1 Mass Flux in the **Label** text field.
- 3 Locate the **Selection** section. From the **Selection** list, choose **Mixing Plane: Axial Flow**.


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

Expression	Unit	Description
$\text{spf} \cdot \rho \cdot (\text{up}(u) \cdot n_x + \text{up}(v) \cdot n_y + \text{up}(w) \cdot n_z)$	kg/s	Upside
$\text{spf} \cdot \rho \cdot (\text{down}(u) \cdot n_x + \text{down}(v) \cdot n_y + \text{down}(w) \cdot n_z)$	kg/s	Downside

Mixing Plane 2 Mass Flux

- 1 Right-click **Mixing Plane 1 Mass Flux** and choose **Duplicate**.
- 2 In the **Settings** window for **Surface Integration**, type **Mixing Plane 2 Mass Flux** in the **Label** text field.
- 3 Locate the **Selection** section. From the **Selection** list, choose **Mixing Plane: Radial Flow**.

Total Head

- 1 In the **Results** toolbar, click  **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, type **Total Head** in the **Label** text field.
- 3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
$(\text{aveout}(p/\text{spf} \cdot \rho) - \text{avein}(p/\text{spf} \cdot \rho)) / g_const + (\text{aveout}(\text{spf} \cdot U)^2 - \text{avein}(\text{spf} \cdot U)^2) / 2 / g_const$	m	


- 4 In the **Results** toolbar, click  **Evaluate** and choose **Evaluate All**.

Geometry Modeling Instructions



Follow these steps below to generate the geometry.

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 Click  **Done**.

GLOBAL DEFINITIONS


Import parameters that define the geometry of the radial pump.

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 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `radial_pump_parameters_1.txt`.

GEOMETRY 1


Work Plane 1 (wp1)

In the **Geometry** toolbar, click  **Work Plane**.


Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Circular Arc 1 (ca1)

- 1 In the **Work Plane** toolbar, click  **More Primitives** and choose **Circular Arc**.
- 2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.
- 3 From the **Specify** list, choose **Endpoints and radius**.
- 4 Locate the **Starting Point** section. In the **xw** text field, type $R_LE \cdot \cos(0[\text{deg}])$.
- 5 In the **yw** text field, type $R_LE \cdot \sin(0[\text{deg}])$.
- 6 Locate the **Endpoint** section. In the **xw** text field, type $R_TE \cdot \cos(\text{delta_theta} - 0[\text{deg}])$.
- 7 In the **yw** text field, type $R_TE \cdot \sin(\text{delta_theta} - 0[\text{deg}])$.
- 8 Locate the **Radius** section. In the **Radius** text field, type `r_b`.

Work Plane 1 (wp1) > Partition Edges 1 (pare1)

- 1 In the **Work Plane** toolbar, click  **Booleans and Partitions** and choose **Partition Edges**.
- 2 On the object **ca1**, select Boundary 1 only.
- 3 In the **Settings** window for **Partition Edges**, locate the **Positions** section.
- 4 In the table, enter the following settings:

Relative arc length parameters
0.35
0.85

Work Plane 1 (wp1) > Delete Entities 1 (dell)

- 1 Right-click **Plane Geometry** and choose **Delete Entities**.

2 On the object **pare1**, select Boundaries 1 and 3 only.

Work Plane 1 (wp1) > Thicken 1 (th1)

1 In the **Work Plane** toolbar, click  **Conversions** and choose **Thicken**.

2 Select the object **dell** only.

3 In the **Settings** window for **Thicken**, locate the **Options** section.

4 In the **Total thickness** text field, type T_b.

5 From the **Ends** list, choose **Circular**.

Work Plane 1 (wp1) > Move 1 (mov1)

1 In the **Work Plane** toolbar, click  **Transforms** and choose **Move**.

2 Select the object **th1** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **xw** text field, type -0.05.

5 In the **yw** text field, type 0.005.

Extrude 1 (ext1)


1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
W_b

Work Plane 2 (wp2)

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


2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

Work Plane 2 (wp2) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2) > Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click  **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type (R_TE+7.5[mm]).

4 In the **Height** text field, type 2*W_b+r_p.


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

Layer name	Thickness (m)
Layer 1	W_b

6 Clear the **Layers on bottom** checkbox.

7 Select the **Layers on top** checkbox.

Work Plane 2 (wp2) > Plane Geometry

In the **Work Plane** toolbar, click  **Rectangle**.

Work Plane 2 (wp2) > Rectangle 2 (r2)

1 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

2 In the **Width** text field, type $(R_{TE} + 7.5[\text{mm}]) - R_{in}$.

3 In the **Height** text field, type $W_b + r_p$.

4 Locate the **Position** section. In the **xw** text field, type R_{in} .

5 In the **yw** text field, type W_b .

Work Plane 2 (wp2) > Plane Geometry

In the **Work Plane** toolbar, click  **Booleans and Partitions** and choose **Difference**.

Work Plane 2 (wp2) > Difference 1 (dif1)

1 Select the object **r1** only.

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

3 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.

4 Select the object **r2** only.

Work Plane 2 (wp2) > Plane Geometry

In the **Work Plane** toolbar, click  **Fillet**.


Work Plane 2 (wp2) > Fillet 1 (fil1)

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

2 In the **Settings** window for **Fillet**, locate the **Radius** section.

3 In the **Radius** text field, type r_p .

Work Plane 2 (wp2) > Plane Geometry

In the **Work Plane** toolbar, click  **Polygon**.

Work Plane 2 (wp2) > Polygon 1 (pol1)

1 In the **Settings** window for **Polygon**, locate the **Coordinates** section.



2 In the table, enter the following settings:

xw (m)	yw (m)
R_TE+7.5 [mm]	0
R_op+d_op+W_b+5 [mm]	0
R_op+d_op+W_b+5 [mm]	$W_b*7/3+5$ [mm]
R_op-d_op	$W_b*7/3+5$ [mm]
R_op-d_op	$W_b*4/3+5$ [mm]
R_op+d_op	$W_b*4/3+5$ [mm]
R_op+d_op	W_b
R_TE+7.5 [mm]	W_b


Revolve 1 (rev1)

- 1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Revolve**.
- 2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.
- 3 Click the **Angles** button.
- 4 In the **Start angle** text field, type 60.
- 5 In the **End angle** text field, type 120.
- 6 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.
- 7 Find the **Direction of revolution axis** subsection. In the **y** text field, type 0.
- 8 In the **z** text field, type 1.

Difference 1 (dif1)


- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **rev1** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 5 Select the object **ext1** only.

Cylinder 1 (cyl1)

- 1 In the **Geometry** toolbar, click  **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type $d_{op}/2$.
- 4 In the **Height** text field, type d_{op} .
- 5 Locate the **Position** section. In the **x** text field, type $R_{op}*\cos(75[\text{deg}])$.

- 6 In the **y** text field, type $R_{op} \cdot \sin(75[\text{deg}])$.
- 7 In the **z** text field, type $W_b + 5[\text{mm}] + (W_b/3) + W_b$.

Cylinder 2 (cyl2)

- 1 In the **Geometry** toolbar, click  **Cylinder**.
- 2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type $d_{op}/2$.
- 4 In the **Height** text field, type d_{op} .
- 5 Locate the **Position** section. In the **x** text field, type $R_{op} \cdot \cos(105[\text{deg}])$.
- 6 In the **y** text field, type $R_{op} \cdot \sin(105[\text{deg}])$.
- 7 In the **z** text field, type $W_b + 5[\text{mm}] + (W_b/3) + W_b$.

Form Union (fin)

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