Model created in COMSOL Multiphysics 6.4

Centrifugal Pump

Introduction

Centrifugal pumps are widely used in the industry and can be found in various applications. These pumps belong to the axisymmetric work-absorbing turbomachinery category for which fluid is transported through the conversion of rotational kinetic energy into hydrodynamic energy. In most applications, fluid enters the pump along the rotating axis and is accelerated by the impeller. The flow is expelled radially outward into a diffuser, or volute chamber, from where it exits. The rotational kinetic energy of the pump is typically supplied by an engine or a motor.

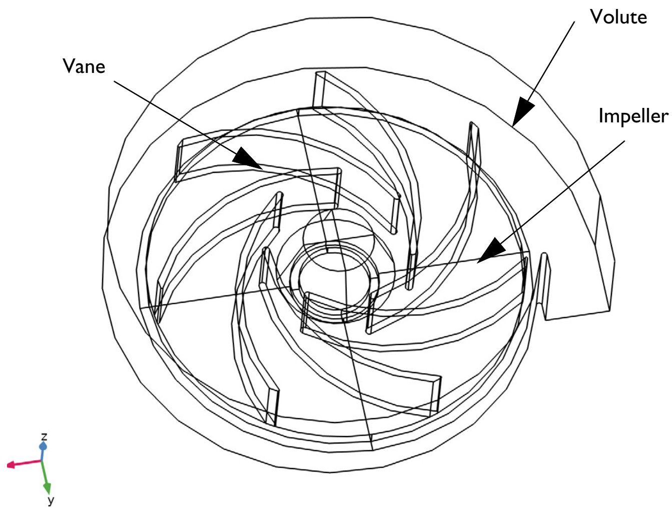


Figure 1: Geometry of the semiopen centrifugal pump.

The current model represents a semiopen centrifugal pump with seven vanes. For the semiopen impeller, the vanes are attached to the hub with a shroud on one side of the impeller. The volute has a spiral shape and the outer radius of the impeller is 10 cm. The size of the modeled pump is typical for automotive applications. The geometry in this work is highly parameterized, allowing straightforward modifications of the geometry to study different configurations of the centrifugal pump if needed.

Model Definition

This model shows how to set up rotating machinery simulations with the frozen rotor approach for centrifugal pumps. The equations that govern the physics are the Navier-Stokes equations and the continuity equation.

A frozen rotor is a cost and time efficient steady-state approximation where individual zones are assigned rotational different speeds. The flow in each of these zones is solved using the moving reference frame equations. In a sense, this approach can be described as freezing the motion of the moving part in a given position and then observing the resulting flow field with the rotor in that fixed position.

Turbulence is modeled with the k-ω model. This is a widely used model for turbomachinery simulations, with good performance for swirling flows and in the near-wall region.

The pressure condition at the inlet and outlet is set up using the `aveop` operator:

$$p_{inlet} = p_{\mathrm{tot}} - 0.5\rho \cdot \mathrm{aveop}(|\mathbf{u}|^2)$$

and

$$p_{outlet} = 0.5\rho \cdot \mathrm{aveop}(|\mathbf{u}|^2)$$

The problem is solved for different total pressure values, p_{tot}, at the inlet in order to obtain a pump curve for the specific geometry considered here.

Results and Discussion

The mass flow is monitored by two probe plots, one at the inlet and one at the outlet. Figure 2 shows that the mass flow at the inlet and the outlet are the equal, which means that mass conservation is achieved.

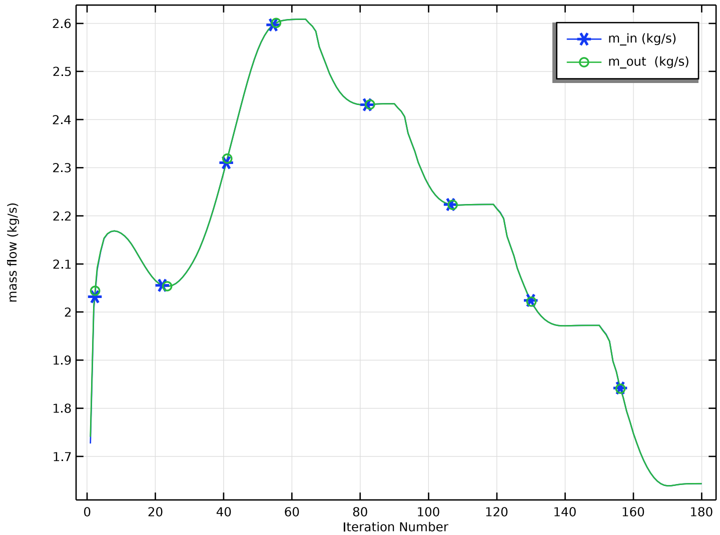


Figure 2: Mass flow probes at the inlet and the outlet.

Note that the five jumps in the curve represent a change in the given total pressure value at the inlet.

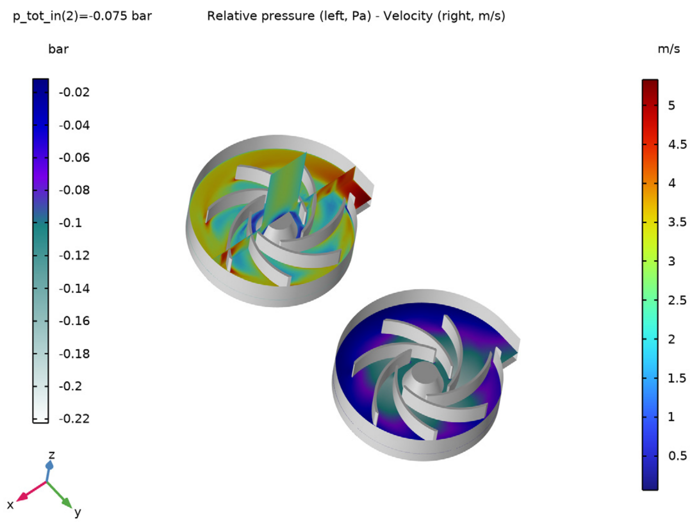


Figure 3: Distribution of the pressure and the velocity magnitude.

Examples of the pressure and velocity magnitude distributions are given in Figure 3. The solution clearly shows a rise in pressure and the corresponding change in velocity from the incoming (inlet) flow, radially toward the volute.

Finally, Figure 4 shows the pump performance curve. The total pressure at the inlet is expressed in terms of the pressure head, H, which is equal to

$$H = \frac{\Delta p_{\mathrm{tot}}}{\rho \cdot g}$$

This curve is central when designing a pump for a given application. Choosing the right pump configuration maximizes the pump and system efficiency, prolongs the life of the system and reduces operational costs.

Table 1 shows the relation between shaft power consumption and pump efficiency.

TABLE 1: PERFORMANCE DATA.

p_tot_in (bar)	**Shaft power consumption (Nm/s)**	**Pump efficiency**
-0.050	54.7	0.276
-0.075	52.6	0.382
-0.100	49.8	0.473

TABLE 1: PERFORMANCE DATA.

p_tot_in (bar)	Shaft power consumption (Nm/s)	Pump efficiency
-0.125	45.7	0.555
-0.150	39.6	0.620

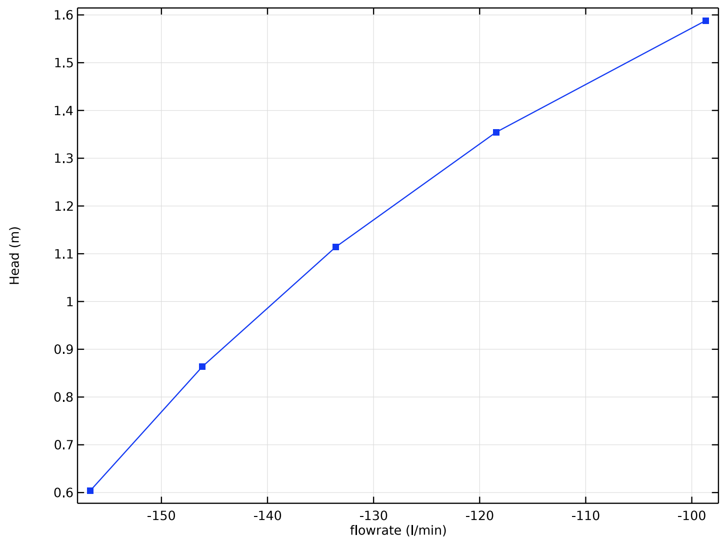


Figure 4: Pump curve.

Application Library path: `Mixer_Module/Tutorials/centrifugal_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 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Rotating Machinery, Fluid Flow** > **Turbulent Flow** > **Turbulent Flow, k-ω**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

Load the parameterized geometry sequence from file. Note that parameters used in the geometry are included with the sequence.

GEOMETRY 1

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 `centrifugal_pump_geom_sequence.mph`.

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

Partition Domains 1 (pard1)

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

2 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

3 From the **Partition with** list, choose **Extended faces**.

4 On the object **cmf2**, select Boundaries 13, 14, 73, and 94 only.

5 Click to select the **Activate Selection** toggle button for **Domains to partition**.

6 On the object **cmf2**, select Domain 1 only.

7 Click **Build Selected**.

Disable the analysis of the geometry as the remaining small geometric details are needed.

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

9 In the **Settings** window for **Geometry**, locate the **Cleanup** section.

10 Clear the **Automatic detection of small details** checkbox.

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

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
p_tot_in	-0.05[bar]	-5000 Pa	Total pressure at the inlet
rot_rpm	1000[rpm]	16.667 1/s	Rotational speed
T_ref	20[degC]	293.15 K	Reference temperature

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.

DEFINITIONS

Boundary Probe 1 (bnd1)

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `m_in` in the **Variable name** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Manual**.

5 Click **Clear Selection**.

6 Select Boundary 58 only.

7 Locate the **Expression** section. In the **Expression** text field, type `-rhoRef*(u*nx+v*ny+w*nz)`.

8 Click to expand the **Table and Window Settings** section. Click **Add Plot Window**.

Boundary Probe 2 (bnd2)

1 In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.

2 In the **Settings** window for **Boundary Probe**, type `m_out` in the **Variable name** text field.

3 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Manual**.

5 Click **Clear Selection**.

6 Select Boundary 9 only.

7 Locate the **Expression** section. In the **Expression** text field, type `rhoRef*(u*nx+v*ny+w*nz)`.

8 Locate the **Table and Window Settings** section. From the **Plot window** list, choose **Probe Plot 1**.

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, type `int_rot` in the **Operator name** text field.

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

4 From the **Selection** list, choose **Walls 2**.

Integration 2 (intop2)

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

2 In the **Settings** window for **Integration**, type `int_in` in the **Operator name** text field.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `57` in the **Selection** text field.

6 Click **OK**.

Integration 3 (intop3)

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

2 In the **Settings** window for **Integration**, type `int_out` in the **Operator name** text field.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `6` in the **Selection** text field.

6 Click **OK**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Unit**	**Description**
rhoRef	spf.rhoref	kg/m³	Reference density
delta_p	int_out(p)/int_out(1)-int_in(p)/int_in(1)	N/m²	Static pressure increase
delta_p_tot	((int_out(p+1/2*rhoRef*spf.U^2)/int_out(1)-int_in(p+1/2*rhoRef*spf.U^2)/int_in(1)))	N/m²	Total pressure increase
Torque	int_rot(+spf.T_tracx*y-spf.T_tracy*x)	N·m	Torque
Power	abs(int_rot(rot1.alphat)*Torque/int_rot(1))	N·m/s	Shaft power consumption
flowrate	int_in(u*nx+v*ny+w*nz)	m³/s	Flow rate
massflow	rhoRef*flowrate	kg/s	Mass flow
H_power	abs(massflow*delta_p_tot/rhoRef)	N·m/s	Power given to fluid
H	delta_p_tot/(rhoRef*g_const)	m	Head
eta	H_power/Power		Pump efficiency

MOVING MESH

Rotating Domain 1

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

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

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

0	X
0	Y
-1	Z

TURBULENT FLOW, K-ω (SPF)

Inlet 1

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

2 Select Boundary 58 only.

3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. From the **Pressure** list, choose **Total**.

6 Select the **Average** checkbox.

7 In the p_0 text field, type `p_tot_in`.

8 Locate the **Turbulence Conditions** section. In the U_{ref} text field, type `3[m/s]`.

Outlet 1

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

2 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

3 From the **Pressure** list, choose **Total**.

4 Select Boundary 9 only.

Wall 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `64, 65, 87, 93` in the **Selection** text field.

5 Click **OK**.

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

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

The **Translational velocity** is set to **Zero (Fixed Wall)** to ensure zero velocity at the lower wall. If set to **Automatic from frame**, it will rotate since it is adjacent to the **Rotating Domain**.

MESH 1

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

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

3 In the table, clear the **Use** checkbox for **Geometric Analysis, Detail Size**.

4 Right-click **Component 1 (comp1)** > **Mesh 1** and choose **Edit Physics-Induced Sequence**.

Size 1

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

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

3 From the **Predefined** list, choose **Normal**.

Free Tetrahedral 1

1 In the **Model Builder** window, click **Free Tetrahedral 1**.

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

3 From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `1, 3, 4, 5` in the **Selection** text field.

6 Click **OK**.

Boundary Layers 1

1 In the **Model Builder** window, click **Boundary Layers 1**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `1, 3, 4, 5` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Boundary Layers**, click to expand the **Corner Settings** section.

8 In the **Trim for angles greater than** text field, type `280`.

Boundary Layer Properties 1

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

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

3 In the **Number of layers** text field, type `5`.

4 From the **Thickness specification** list, choose **First layer**.

5 In the **Thickness** text field, type `2.5e-4`.

Boundary Layer Properties 2

1 Right-click **Component 1 (comp1)** > **Mesh 1** > **Boundary Layers 1** > **Boundary Layer Properties 1** and choose **Duplicate**.

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

3 In the **Thickness** text field, type `6e-5`.

4 Select Boundaries 24, 27, 31, 36, 42, 45, 48, 75, and 105 only.

Boundary Layer Properties 3

1 Right-click **Boundary Layer Properties 2** and choose **Duplicate**.

2 Select Boundary 4 only.

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

4 In the **Thickness** text field, type `1.2e-4`.

Boundary Layer Properties 4

1 Right-click **Boundary Layer Properties 3** and choose **Duplicate**.

2 Select Boundaries 15, 64–69, 87–90, 92, and 93 only.

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

4 In the **Thickness** text field, type `2e-4`.

Swept 1

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

Distribution 1

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

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

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

4 Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `6` in the **Selection** text field.

7 Click **OK**.

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

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

10 In the **Number of elements** text field, type `10`.

11 In the **Element ratio** text field, type `4`.

Distribution 2

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 2 in the **Selection** text field.

6 Click **OK**.

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

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

9 In the **Number of elements** text field, type 20.

10 In the **Element ratio** text field, type 4.

Use mapped mesh to improve the mesh quality.

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Drag and drop below **Size**.

3 Select Boundary 11 only.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edge 36 only.

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

4 In the **Number of elements** text field, type 20.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edge 16 only.

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

4 In the **Number of elements** text field, type 3.

Use mapped mesh to improve the mesh quality.

Mapped 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Drag and drop below **Mapped 1**.

3 Select Boundaries 39, 54, 55, 83, 84, and 97 only.

Distribution 1

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 66 and 159 only.

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

4 In the **Number of elements** text field, type 20.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 158, 189, and 190 only.

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

4 In the **Number of elements** text field, type 3.

Convert the mapped mesh to a triangular mesh.

Convert 1

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

2 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 2: Frozen Rotor

1 In the **Model Builder** window, under **Study 1** click **Step 2: Frozen Rotor**.

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

3 From the **Probes** list, choose **None**.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p_tot_in (Total pressure at the inlet)	range(-0.05,-0.1/4,-0.15)	bar

The continuation solver works best for models with linear dependence on the parameter. A more robust alternative for nonlinear applications is to start from the solution for the previous parameter value.

7 From the **Run continuation for** list, choose **No parameter**.

8 From the **Reuse solution from previous step** list, choose **Yes**.

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** node, then click **Segregated 1**.

4 In the **Settings** window for **Segregated**, click to expand the **Results While Solving** section.

5 From the **Probes** list, choose **All**.

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

RESULTS

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results > Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

3 From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `1 3 4 5 6` in the **Selection** text field.

6 Click **OK**.

Walls

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

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

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

4 In the **Label** text field, type `Walls`.

Performance data

1 In the **Results** toolbar, click **More Derived Values** and choose **Average > Volume Average**.

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

3 From the **Selection** list, choose **All domains**.

4 In the **Label** text field, type `Performance data`.

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

Expression	Unit	Description
delta_p	N/m^2	static pressure increase
delta_p_tot	N/m^2	total pressure increase
Torque	N*m	torque
Power	N*m/s	shaft power consumption

Expression	Unit	Description
H_power	N*m/s	power given to fluid
eta	1	pump efficiency
H	1	Head
flowrate	l/min	flowrate

6 Click next to **Evaluate**, then choose **New Table**.

Performance data

1 In the **Model Builder** window, expand the **Results** > **Tables** node, then click **Table 2**.

2 In the **Settings** window for **Table**, type `Performance data` in the **Label** text field.

Pump Curve

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

2 In the **Settings** window for **1D Plot Group**, type `Pump Curve` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **None**.

Table Graph 1

1 Right-click **Pump Curve** and choose **Table Graph**.

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

3 From the **Table** list, choose **Performance data**.

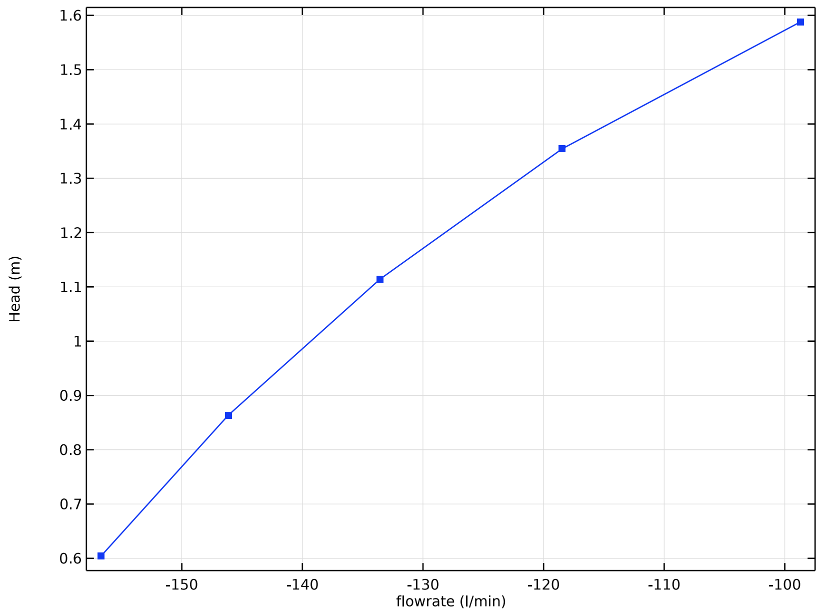
4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list box, select **Head (m)**.

6 From the **x-axis data** list, choose **flowrate (l/min)**.

7 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

8 In the **Pump Curve** toolbar, click **Plot**.



Velocity (spf)

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

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

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

4 Locate the **Data** section. From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

5 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

6 From the **Array axis** list, choose **y**.

Multislice 1

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

2 In the **Settings** window for **Multislice**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

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

5 Find the **y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type 0.

7 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

8 In the **Coordinates** text field, type `0.01`.

9 Locate the **Coloring and Style** section. Select the **Color legend** checkbox.

Surface 1

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

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

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

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

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

8 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Multislice 1, Surface 1

1 In the **Model Builder** window, under **Results** > **Velocity (spf)**, Ctrl-click to select **Multislice 1** and **Surface 1**.

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

Multislice 2

1 In the **Settings** window for **Multislice**, locate the **Expression** section.

2 In the **Expression** text field, type `p`.

3 From the **Unit** list, choose **bar**.

4 Locate the **Multiplane Data** section. Find the **y-planes** subsection. From the **Entry method** list, choose **Number of planes**.

5 In the **Planes** text field, type `0`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralis**.

7 Select the **Color legend** checkbox.

8 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

9 In the **Index** text field, type `1`.

Surface 2

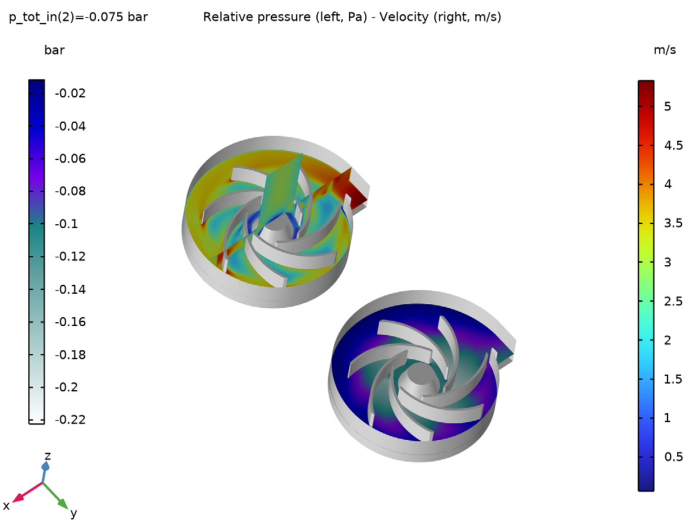
1 In the **Model Builder** window, click **Surface 2**.

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

3 In the **Index** text field, type `1`.

Velocity (spf)

1. In the **Model Builder** window, click **Velocity (spf)**.
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 `Relative pressure (left, Pa) - Velocity (right, m/s)`.
5. Locate the **Color Legend** section. From the **Position** list, choose **Alternating**.
6. Select the **Show units** checkbox.
7. In the **Velocity (spf)** toolbar, click **Plot**.
8. Click the **Zoom Extents** button in the **Graphics** toolbar.



Pressure (spf)

1. In the **Model Builder** window, click **Pressure (spf)**.
2. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3. From the **Parameter value (p_tot_in (bar))** list, choose **-0.15**.

Surface

1. In the **Model Builder** window, expand the **Pressure (spf)** node, then click **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. From the **Unit** list, choose **bar**.

4 In the **Pressure (spf)** toolbar, click **Plot**.

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

Surface 1

1 In the **Model Builder** window, expand the **Wall Resolution (spf)** node, then click **Surface 1**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type `spf.d_w_plus`.

5 In the **Wall Resolution (spf)** toolbar, click **Plot**.

Probe Plot Group 1

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

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

3 Select the **y-axis label** checkbox. In the associated text field, type `mass flow (kg/s)`.

Probe Table Graph 1

1 In the **Model Builder** window, expand the **Probe Plot Group 1** node, then click **Probe Table Graph 1**.

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

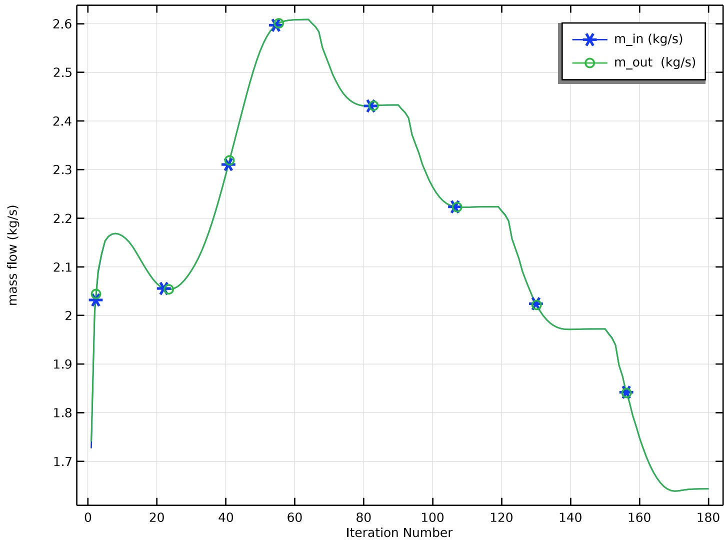
3 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`m_in (kg/s)`
`m_out  (kg/s)`

6 In the **Probe Plot Group 1** toolbar, click **Plot**.



Exterior Walls 2

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

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

3 Select Boundaries 1–3, 5, 7, 8, 10, 11, 15, 25–38, 40–53, 56, 59–61, 66–78, 82, 85, 88–90, 92, 94, 99, and 101–115 only.

Study 1/Solution 1 (4) (sol1)

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

Cut Plane 1

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

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

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

5 In the **z-coordinate** text field, type `0.0125`.

Velocity Streamlines

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Solution 1 (4) (sol1)**.

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

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

Surface 1

1 Right-click **Velocity Streamlines** and choose **Surface**.

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

3 From the **Dataset** list, choose **Exterior Walls 2**.

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

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

Surface 2

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

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

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

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

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

Velocity Streamlines

In the **Model Builder** window, click **Velocity Streamlines**.

Streamline Surface 1

1 In the **Velocity Streamlines** toolbar, click **More Plots** and choose **Streamline Surface**.

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

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

4 From the **Parameter value (p_tot_in (bar))** list, choose **-0.075**.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

6 In the **Density level** text field, type `9.4`.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

8 In the **Tube radius expression** text field, type `0.05`.

9 Select the **Radius scale factor** checkbox. In the associated text field, type `0.005`.

10 Find the **Point style** subsection. From the **Color** list, choose **Custom**.

11 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

12 Click **Define custom colors**.

13 Set the RGB values to 105, 105, and 105, respectively.

14 Click **Add to custom colors**.

15 Click **Show color palette only** or **OK** on the cross-platform desktop.

Streamline 2

1 Right-click **Velocity Streamlines** and choose **Streamline**.

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

3 In the **Number** text field, type `14`.

4 Select Boundary 58 only.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

6 In the **Tube radius expression** text field, type `0.05`.

7 Select the **Radius scale factor** checkbox. In the associated text field, type `0.005`.

8 Find the **Point style** subsection. From the **Color** list, choose **Custom**.

9 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

10 Click **Define custom colors**.

11 Set the RGB values to 105, 105, and 105, respectively.

12 Click **Add to custom colors**.

13 Click **Show color palette only** or **OK** on the cross-platform desktop.

14 In the **Velocity Streamlines** toolbar, click **Plot**.

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