Model created in COMSOL Multiphysics 6.4

Mixing of Water in a Flat Bottom Mixer

Introduction

This model simulates the flow in a flat bottom mixer agitated by a pitched four blade impeller. The mixed fluid is water, and the flow is assumed turbulent due to the significant mixer Reynolds number ($5.33 \cdot 10^4$). The flow is modeled using the k-ε turbulence model, and a time-dependent simulation corresponding to 34 revolutions of the impeller is performed in order to reach the operating conditions of the mixer.

When postprocessing the results, the self-similarity of the axial flow along the baffles is analyzed. In agreement with Ref. 1, the normalized velocity profiles at different axial positions are found to be self-similar indicating that the flow in this region resembles a three dimensional wall jet.

Model Definition

The mixer simulated consists of a baffled flat-bottom vessel and a pitched four-blade impeller. The mixer geometry and the simulation conditions used in this model correspond to the ones used in the experiments of Bittorf and Kresta (Ref. 1). The height (H) and diameter (T) of the mixer vessel is 0.24 m. The diameter of the impeller, D_a, is $0.33T$, and the clearance, C, between the impeller and the bottom is $0.40D_a$. The pitch of the impeller blades is 45°. The geometry of the mixer simulated is shown in Figure 1. The fluid contained in the mixer is water, which is mixed using an impeller rotational frequency of N = 8.58 turns per second. The resulting impeller Reynolds number is:

$$N_{\mathrm{Re}} = \frac{N D_a^2 \rho}{\mu} = 5.33 \cdot 10^4 \qquad (1)$$

where μ and ρ are the dynamic viscosity (SI unit: kg/(m·s)) and density (SI unit: kg/m^3) of water, respectively. The impeller Reynolds number is significantly higher than one, and the resulting flow will be treated as turbulent. The turbulent flow is modeled using the k-ε turbulence model, and the time-dependent problem is solved for 34 revolutions of the impeller.

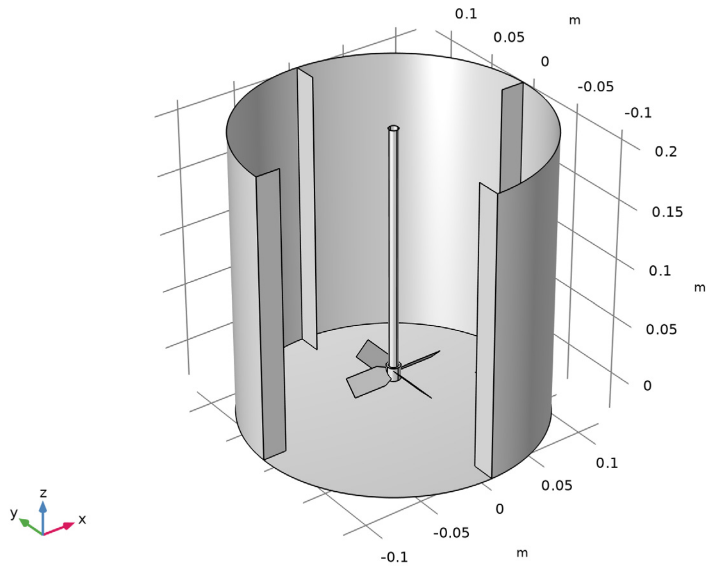


Figure 1: Mixer geometry showing the vessel, baffles, and impeller.

WALL JET FLOW

Bittorf and Kresta (Ref. 1) compared the flow at the wall of a stirred tank to that of a turbulent wall jet. They found that the flow along the baffles of the stirred tank compares well to a three-dimensional wall jet. In order to assess whether the flow in this region of the mixer has the characteristics of a wall jet, the self-similarity of the axial velocity profiles can be examined. Self-similarity implies that the mean velocity profiles can be collapsed onto a single profile when scaled with properly chosen length and velocity scales. The existence of self-similarity indicates that the turbulent time scale is small enough for the turbulence to adjust locally to the development of the jet.

For wall jets, the spreading is typically found to be characterized by the maximum jet velocity, U_m, and the distance from the wall to the position where the local velocity is equal to half of the maximum velocity value. The latter is often referred to as the jet half width and denoted by $y_{1/2}$. Using the velocity data in Ref. 1, a similarity profile for a wall jet in a recirculating flow was suggested by Bhattacharya and Kresta in Ref. 2:

$$\frac{U}{U_m} = 1 - 1.5\tanh[0.78(\eta - 0.15)]^2 \qquad (2)$$

Here η denotes the distance from the wall normalized by the half width, $\eta = y/y_{1/2}$. When analyzing the results, the self-similarity will be assessed by plotting the normalized axial velocity along the line shown in Figure 2. This corresponds to the position used in Ref. 1.

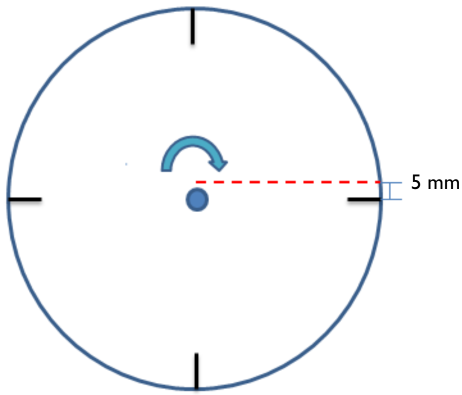


Figure 2: Measurement position for analyzing the self-similarity of the axial velocity profiles.

Solution Procedure

In order to minimize the time required to reach a fully developed, yet transient, flow state, the model is solved in two steps. First, a Frozen Rotor study is used to reach a good initial solution without having to solve the startup of the problem. In order to converge this step, a parametric sweep is used to first solve the model with a low Reynolds number and increased dynamic viscosity, and then compute it again with the actual dynamic viscosity of the fluid.

The frozen rotor solution is then used as initial condition for the transient simulation. During the transient simulation the model is run for 4.0 s corresponding to 34 full revolutions of the impeller.

TIME AVERAGING

An Identity Mapping nonlocal coupling is used when computing time-averaged velocity profiles. This is done to make sure that the average is computed along the measurement positions shown Figure 2, regardless of the position of the rotating impeller domain. The averages are computed using the time average operator in the manner of

```
timeavg(3.5,4,idmap1(w),'nointerp')
```

where the axial velocity is averaged between $t = 3.5$ and 4 s. The `nointerp` flag is used to compute the average without applying interpolation between the stored time steps.

Results and Discussion

The resulting flow field in the mixer after 4.0 s of transient simulation is shown in Figure 3. The streamlines show how the impeller pumps fluid downward along the shaft and ejects it toward the bottom of the vessel. The maximum velocity magnitude occurs at the tip of the impeller blades.

The velocity magnitude in a single cross section including the velocity vectors is shown in Figure 4. From this figure the main flow structure of the mixer can be further examined. The fluid flows with high speed along the bottom of the mixer. Upon reaching the wall, vertical high speed streaks form along the outer walls, and the fluid velocity decreases toward the top of the vessel. Fluid at the top of the vessel travel toward the impeller shaft prior to becoming pumped down toward the impeller.

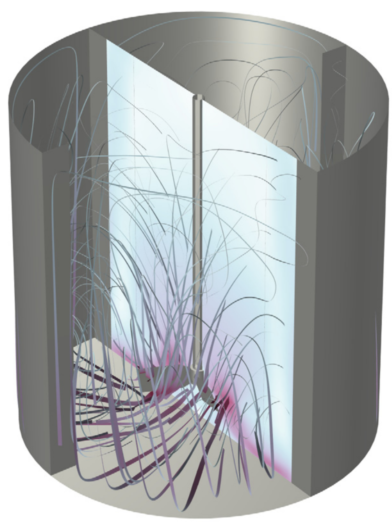


Figure 3: Fluid flow at $t = 4.0$ s. The color of the slice plots and the width of the streamlines indicate the velocity magnitude.

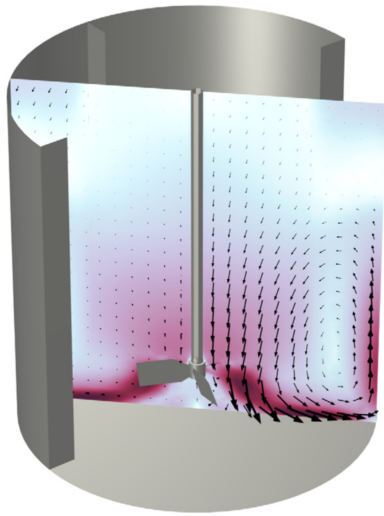


Figure 4: Fluid flow magnitude and direction, at t = 4.0 s, in a cross section of the mixer.

The characteristics of the upward flow along the outer walls is studied in Figure 5 to analyze the wall jet. The averaged axial velocity, measured along the line shown in Figure 2, is plotted at different heights from the vessel bottom. The lines correspond to z/T equal to 0.458, 0.563, 0.667, and 0.771. The fluid velocity profiles are scaled by their individual maximum velocity, and the distance from the wall is normalized by the jet half width as defined in the section Wall Jet Flow. The similarity solution of Bhattacharya and Kresta (Equation 2) is also plotted.

The scaled velocity profiles are found to collapse on top of the similarity solution, at least for the first three positions from the bottom. The collapse occurs for $\eta < 1.5$; further out from the wall, the velocity is directed in the opposite direction, corresponding to the downward flow along the impeller shaft as seen in Figure 3 and Figure 4. The collapse in the region with upward flow indicates that the flow along the outer walls does indeed correspond to a wall jet flow. The results in Figure 5 compare well with the experimental results in Ref. 1.

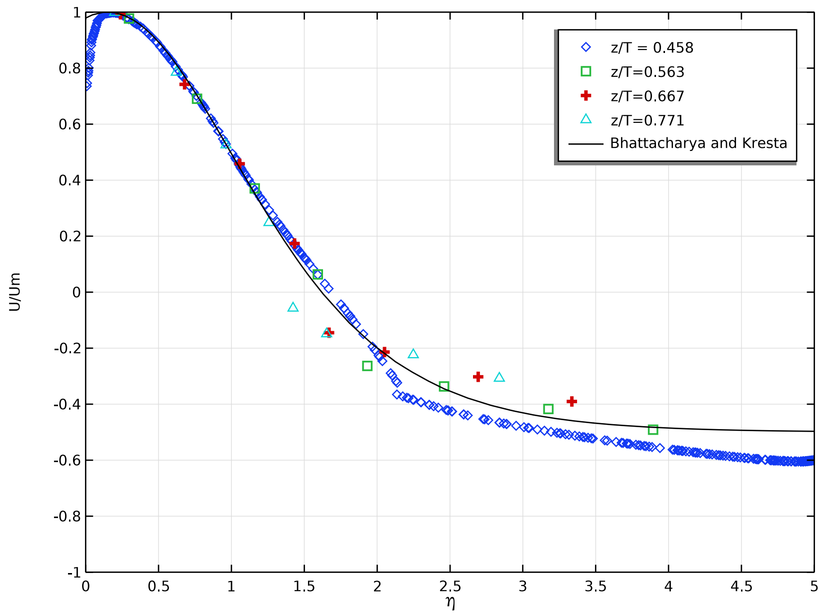


Figure 5: Normalized axial velocity profiles measured at increasing mixer heights.

References

1. K.J. Bittorf and S.M. Kresta, "Three-dimensional wall jets: axial flow in a stirred tank," *AIChE Journal*, vol. 47, no. 6, pp. 1277–1284, 2001.

2. S. Bhattacharya and S.M. Kresta, "CFD Simulations of Three-dimensional Wall Jets in a Stirred Tank," *Can. J. Chem. Eng.*, vol. 80, no. 4, pp. 1–15, 2002.

Application Library path: `Mixer_Module/Benchmarks/wall_jet_4PB_mixer_flat`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Rotating Machinery, Fluid Flow** > **Turbulent Flow** > **Turbulent Flow, k-ε**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Frozen Rotor**.
6 Click **Done**.

GEOMETRY 1

Load the parameterized geometry sequence from 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 `wall_jet_4PB_mixer_flat_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
N0	8.58[Hz]	8.58 Hz	Rotational speed of impeller
visc_fact	1	1	Viscosity factor

MOVING MESH

Rotating Domain 1

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

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

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

4 In the list box, select **1**.

5 Click **Remove from Selection**.

6 Select Domain 2 only.

7 Locate the **Rotation** section. In the *f* text field, type `-N0`.

DEFINITIONS

Identity Mapping 1 (idmap1)

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

2 In the **Settings** window for **Identity Mapping**, locate the **Source Selection** section.

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

4 Locate the **Frames** section. From the **Destination frame** list, choose **Material (X, Y, Z)**.

MATERIALS

Add water from the Material 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 Right-click and choose **Add to Component 1 (comp1)**.

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

MATERIALS

Water, liquid (mat1)

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

2 In the table, enter the following settings:

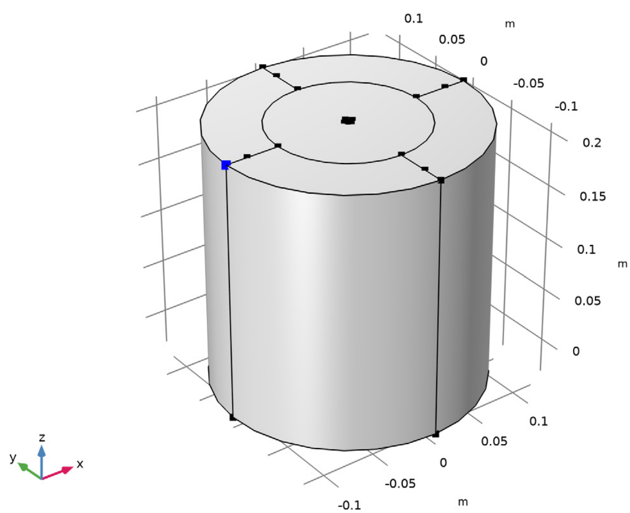
Property	**Variable**	**Value**	**Unit**	**Property group**
Dynamic viscosity	mu	`eta(T[1/K])[Pa*s]*visc_fact`	Pa·s	Basic

TURBULENT FLOW, K-ε (SPF)

Pressure Point Constraint 1

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

2 Select Point 2 only.



Symmetry 1

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

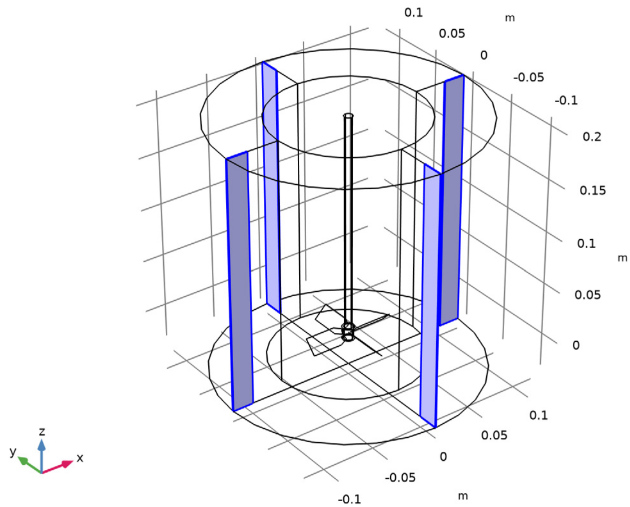
2 Select Boundaries 6, 7, 14, 18, and 25 only (top boundaries).

Interior Wall 1

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

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

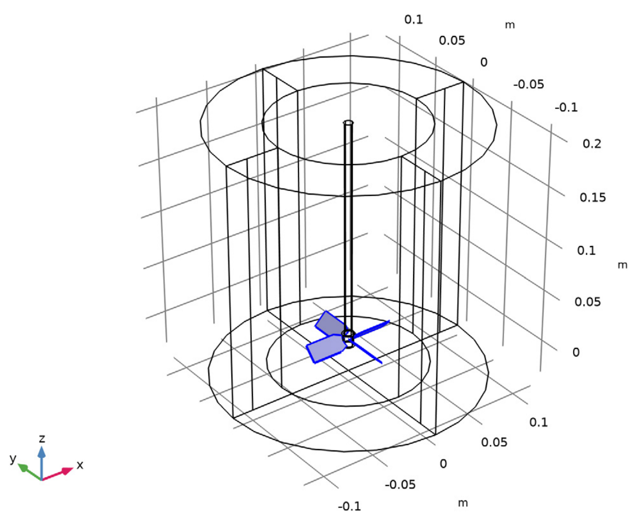
3 Select Boundaries 1, 11, 19, and 21 only.



Interior Wall 2

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

2 Select Boundaries 26, 27, 32, and 42 only.



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 From the **Element size** list, choose **Coarser**.

5 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

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 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. Select the **Maximum element size** checkbox.

5 Select the **Minimum element size** checkbox.

6 In the **Maximum element size** text field, type `0.01`.

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 **Domain**.

4 From the **Selection** list, choose **Tank (Pitched Blade Impeller 1)**.

5 Select Domain 3 only.

6 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

7 Click the **Custom** button.

8 Locate the **Element Size Parameters** section.

9 Select the **Maximum element size** checkbox. In the associated text field, type `0.008`.

10 Select the **Minimum element size** checkbox. In the associated text field, type `0.001`.

11 Right-click **Size 2** and choose **Move Up** three times.

Free Tetrahedral 1

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

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

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

4 Select Domain 2 only.

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

STUDY 1

Step 1: Frozen Rotor

Set up an auxiliary continuation sweep for the `visc_fact` parameter.

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

2 In the **Settings** window for **Frozen Rotor**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
visc_fact (Viscosity factor)	10 1	

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

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

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

9 In the **Settings** window for **Study**, locate the **Study Settings** section.

10 Clear the **Generate default plots** checkbox.

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

Now run a time-dependent simulation using the results from the frozen rotor study as initial values.

DEFINITIONS

Identity Boundary Pair 1 (ap1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **Identity Boundary Pair 1 (ap1)**.

2 In the **Settings** window for **Pair**, locate the **Advanced** section.

3 From the **Elementwise mapping for compatible meshes** list, choose **Off**.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

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

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(0,0.2,3), range(3, 0.05, 4)`.

3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Method** list, choose **Solution**.

5 From the **Study** list, choose **Study 1, Frozen Rotor**.

6 From the **Parameter value (visc_fact)** list, choose **Last**.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Absolute Tolerance** section.

4 Click to expand the **Time Stepping** section. Select the **Initial step** checkbox.

5 From the **Minimum BDF order** list, choose **2**.

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

RESULTS

When working with large 3D models it is often convenient to disable automatic plot updates.

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

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** checkbox.

The following instructions recreate Figure 3. Start by creating a selection for all walls in the **Geometry** node. The geometry sequence already contains some selections that you can use.

GEOMETRY 1

Union Selection 1 (unisel1)

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

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

3 From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog, in the **Selections to add** list, choose **Rotating Interior Wall**, **Rotating Wall**, **Interior Wall**, and **Tank walls (Flat Bottom Tank 1)**.

6 Click **OK**.

RESULTS

Multislice 1

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

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

3 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **z-planes** subsection. In the **Planes** text field, type `0`.

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

Streamline 1

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

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

3 From the **Positioning** list, choose **Uniform density**.

4 In the **Density level** text field, type `8`.

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

6 In the **Width expression** text field, type `spf.U`.

7 Select the **Width scale factor** checkbox. In the associated text field, type `2e-3`.

8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Multislice 1**.

Color Expression 1

Right-click **Streamline 1** and choose **Color Expression**.

Surface 1

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

Material Appearance 1

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

Selection 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Union Selection 1**.
4 In the list box, select **2**.
5 Click **Remove from Selection**.
6 Select Boundaries 1, 3–5, 11–13, 16, 19–21, 26–35, 37–40, and 42 only.

Velocity (spf)

1 In the **Model Builder** window, under **Results** 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 **None**.
4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
5 Locate the **Color Legend** section. Select the **Show units** checkbox.
6 In the **Velocity (spf)** toolbar, click **Plot**.

The following instructions recreate Figure 4.

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 2/Solution 2 (sol2)**.
4 Locate the **Plane Data** section. From the **Plane type** list, choose **General**.
5 In row **Point 2**, set **y** to -1.
6 In row **Point 3**, set **y** to 0 and **z** to 1.
7 Click **Plot**.

Velocity Magnitude and Flow Direction

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

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

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

4 From the **Time (s)** list, choose **Last (4)**.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

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

7 Locate the **Color Legend** section. Select the **Show units** checkbox.

Arrow Surface 1

1 Right-click **Velocity Magnitude and Flow Direction** and choose **Arrow Surface**.

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

3 In the **Number of arrows** text field, type `500`.

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

5 Select the **Scale factor** checkbox. In the associated text field, type `0.03`.

6 From the **Color** list, choose **Black**.

Surface 1

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

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

3 From the **Color table** list, choose **Passiflora**.

Surface 2

Right-click **Velocity Magnitude and Flow Direction** and choose **Surface**.

Material Appearance 1

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

Surface 2

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

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

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

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Union Selection 1**.

4 In the list, choose **2**, **11**, and **12**.

5 Click **Remove from Selection**.

6 Select Boundaries 1, 3–5, 13, 16, 19–21, 26–35, 37–40, and 42 only.

7 In the **Velocity Magnitude and Flow Direction** toolbar, click **Plot**.

Create the plot in Figure 5. Start by creating **Cut Line** datasets for the different heights.

Cut Line 3D 1

1 In the **Results** toolbar, click **Cut Line 3D**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **x** to `-0.005`, **y** to `T/2-0.12`, and **z** to `0.458*H-C`.

4 In row **Point 2**, set **x** to `-0.005`, **y** to `T/2`, and **z** to `0.458*H-C`.

5 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

6 Click **Plot**.

Cut Line 3D 2

1 Right-click **Cut Line 3D 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **z** to `0.563*H-C`.

4 In row **Point 2**, set **z** to `0.563*H-C`.

Cut Line 3D 3

1 Right-click **Cut Line 3D 2** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **z** to `0.667*H-C`.

4 In row **Point 2**, set **z** to `0.667*H-C`.

Cut Line 3D 4

1 Right-click **Cut Line 3D 3** and choose **Duplicate**.

2 In the **Settings** window for **Cut Line 3D**, locate the **Line Data** section.

3 In row **Point 1**, set **z** to `0.771*H-C`.

4 In row **Point 2**, set **z** to `0.771*H-C`.

Proceed to plot the axial velocity profiles.

Unscaled profiles

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

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

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `y`.

6 Select the **y-axis label** checkbox. In the associated text field, type `U`.

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

8 In the **x minimum** text field, type `0`.

9 In the **x maximum** text field, type `0.12`.

10 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Line Graph 1

1 Right-click **Unscaled profiles** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Cut Line 3D 1**.

4 From the **Time selection** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `timeavg(3.5,4, idmap1(w),'nointerp')`. This is the expression discussed in the Time averaging section.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `y`.

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

9 From the **Positioning** list, choose **Interpolated**.

10 In the **Number** text field, type `25`.

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

12 Select the **Show legends** checkbox.

13 In the table, enter the following settings:

Legends
z/T = 0.458

Line Graph 2

1 Right-click **Line Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Line 3D 2**.

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

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

Legends
z/T=0.563

Line Graph 3

1 Right-click **Line Graph 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Line 3D 3**.

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

Legends
z/T=0.667

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

Line Graph 4

1 Right-click **Line Graph 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Line 3D 4**.

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

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

Legends
z/T=0.771

6 In the **Unscaled profiles** toolbar, click **Plot**It takes some time to evaluate.

For the normalized axial velocity profile, determine the maximum velocity and the distance at which the velocity equals half of the maximum. With the help of **Graph Markers**, these values can be extracted from the results.

Graph Marker 1

1. In the **Model Builder** window, right-click **Line Graph 1** and choose **Graph Marker**.
2. In the **Settings** window for **Graph Marker**, locate the **Display** section.
3. From the **Display** list, choose **Max**.

   Add a **Graph Marker** to the remaining line graphs.

   The values are displayed and saved to result tables. To evaluate the position of half the maximum velocity, the value for the respective maximum velocity is needed.

Parameters

1. In the **Model Builder** window, expand the **Results** > **Tables** node.
2. Right-click **Results** and choose **Parameters**.

   For convenience, these parameters along with additional ones to be used later, are available in a file.

3. In the **Settings** window for **Parameters**, locate the **Parameters** section.
4. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `wall_jet_4PB_mixer_flat_parameters_results.txt`.

Add new **Graph Markers** using the `Ymax` values.

Graph Marker 2

1. In the **Model Builder** window, right-click **Line Graph 1** and choose **Graph Marker**.
2. In the **Settings** window for **Graph Marker**, locate the **Display** section.
3. From the **Display mode** list, choose **Line intersection**.
4. From the **Line type** list, choose **Horizontal**.
5. In the **y-coordinates** text field, type `Ymax1/2`.

   Repeat for the remaining line graphs.

Line Graph 2

1. In the **Model Builder** window, under **Results** > **Unscaled profiles** click **Line Graph 2**.

2 In the **Unscaled profiles** toolbar, click **Plot**.

The values of the intersections are again stored in their respective tables. The first value corresponds to the `Xhalf` values in the **Parameters** list.

With this, the normalized profile can be plotted. First, write the data for each graph into a table.

Line Graph 1

1 In the **Model Builder** window, right-click **Line Graph 1** and choose **Copy Plot Data to Table**.

Repeat for the remaining line graphs.

Now, the tables can be used to create Figure 5 by scaling them using the parameters evaluated before.

Scaled profiles

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

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

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `\eta`.

6 Select the **y-axis label** checkbox. In the associated text field, type `U/Um`.

7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

8 In the **x minimum** text field, type `0`.

9 In the **x maximum** text field, type `5`.

Leave the y-axis limits at their default values.

Table Graph 1

1 Right-click **Scaled profiles** and choose **Table Graph**.

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

3 From the **Table** list, choose **Table 1**.

4 Click to expand the **Preprocessing** section. Find the **x-axis column** subsection. From the **Transformation** list, choose **Linear**.

5 In the **Scaling** text field, type `-scaleX1`.

6 In the **Shift** text field, type `T/2*scaleX1`.

7 Find the **y-axis columns** subsection. From the **Transformation** list, choose **Linear**.

8 In the **Scaling** text field, type `1/Ymax1`.

9 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

10 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

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

12 Select the **Show legends** checkbox.

13 In the table, enter the following settings:

Legends
`z/T = 0.458`

Table Graph 2

1 Right-click **Table Graph 1** and choose **Duplicate**.

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

3 From the **Table** list, choose **Table 2**.

4 Locate the **Preprocessing** section. Find the **x-axis column** subsection. In the **Scaling** text field, type `-scaleX2`.

5 In the **Shift** text field, type `T/2*scaleX2`.

6 Find the **y-axis columns** subsection. In the **Scaling** text field, type `1/Ymax2`.

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

8 From the **Positioning** list, choose **Interpolated**.

9 Locate the **Legends** section. In the table, enter the following settings:

Legends
`z/T=0.563`

Table Graph 3

1 Right-click **Table Graph 2** and choose **Duplicate**.

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

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

4 Locate the **Preprocessing** section. Find the **x-axis column** subsection. In the **Scaling** text field, type `-scaleX3`.

5 In the **Shift** text field, type `T/2*scaleX3`.

6 Find the **y-axis columns** subsection. In the **Scaling** text field, type `1/Ymax3`.

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

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
z/T=0.667

Table Graph 4

1 Right-click **Table Graph 3** and choose **Duplicate**.

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

3 From the **Table** list, choose **Table 4**.

4 Locate the **Preprocessing** section. Find the **x-axis column** subsection. In the **Scaling** text field, type `-scaleX4`.

5 In the **Shift** text field, type `T/2*scaleX4`.

6 Find the **y-axis columns** subsection. In the **Scaling** text field, type `1/Ymax4`.

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

8 Locate the **Legends** section. In the table, enter the following settings:

Legends
z/T=0.771

Scaled profiles

Finally, add the plot for Equation 2.

Line Graph 1

1 In the **Model Builder** window, right-click **Scaled profiles** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Cut Line 3D 1**.

4 From the **Time selection** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `1-1.5*tanh(0.78*(y*50-0.15))^2`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `y*50`.

8 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

9 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

10 Select the **Show legends** checkbox.

11 In the table, enter the following settings:

Legends
`Bhattacharya and Kresta`

Scaled profiles

1 In the **Model Builder** window, click **Scaled profiles**.

2 In the **Scaled profiles** toolbar, click **Plot**.