Resin Transfer Molding of a Wind Turbine Blade

Introduction

Resin Transfer Molding (RTM) is a manufacturing process for composite structures. The resin is injected under pressure into the mold cavity. Reinforcement materials like fiber structures can be placed into the empty mold before the resin is injected, which makes it easier to use any kind of reinforcement material and orientation. After the resin has been injected the curing takes place.

To avoid air bubbles that might be trapped within the resin several vents are placed at certain positions of the mold. Simulation can be used to optimize the vent positions.

In this example the RTM process of a wind turbine blade is investigated. The model geometry and setup was inspired by Ref. 1.

Model Definition

The blade consists of five parts of different composites, which have different anisotropic permeabilities. The colors in Figure 1 indicate the different permeabilities. The blade has a thickness of 1 cm. The geometry was constructed using COMSOL Multiphysics and the Design Module. However, in this example we just import the ready-made geometry file.

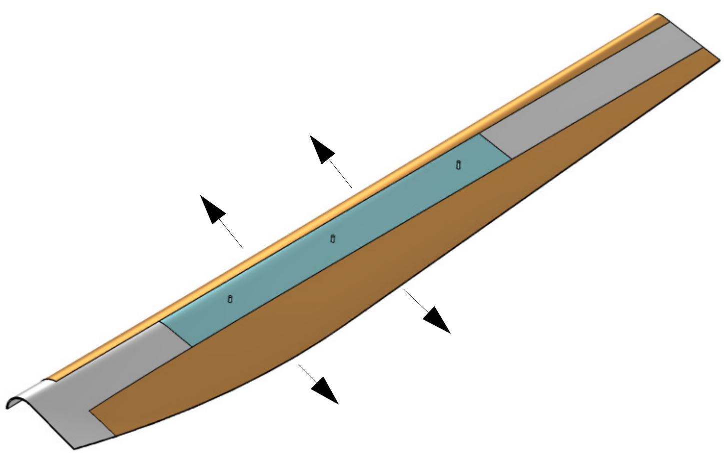

Figure 1: Geometry of the wind turbine blade. The colors indicate the different porous materials and different permeabilities.

Initially, the blade is filled with air. Resin is injected via the cylindrical inlet pipes at the top of the blade with a pressure of 800 kPa. The air can escape through the outlets which are defined at both short ends of the blade and at four additional vents along the blade rim which are marked by the outgoing arrows in Figure 1. All other boundaries are assumed as solid walls and the No Slip wall condition applies.

The model is set up using the combined multiphysics interface Two Phase Flow, Level Set, Brinkman Equations.

MODEL EQUATIONS

The flow through the porous media within the mold (due to the reinforcement material) is described by the Brinkman Equations for incompressible flow:

$$\frac{\rho}{\varepsilon_{\mathrm{p}}}\frac{\partial\mathbf{u}}{\partial t} = \nabla\cdot\left[-p\mathbf{I}+\frac{\mu}{\varepsilon_{\mathrm{p}}}(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)\right]-\left(\frac{\mu}{\kappa}+\beta\rho|\mathbf{u}|\right)\mathbf{u} \qquad (1)$$

$$\rho\nabla\cdot\mathbf{u} = 0$$

In these equations, where the inertial term has been neglected, μ (SI unit: kg/(m·s)) is the dynamic viscosity of the fluid, $\mathbf{u}$ (SI unit: m/s) is the velocity vector, ρ (SI unit: kg/m^3) is the density of the fluid, p (SI unit: Pa) is the pressure, ε_{p} is the porosity, and κ (SI unit: m^2) the permeability of the porous medium.

For the two-phase flow level-set method, an equation for the level-set function ϕ, which describes the interface between the two phases, is solved:

$$\varepsilon_{\mathrm{p}}\frac{\partial\phi}{\partial t}+\mathbf{u}\cdot\nabla\phi = \gamma\nabla\cdot\left(\varepsilon_{\mathrm{ls}}\nabla\phi-\phi(1-\phi)\frac{\nabla\phi}{|\nabla\phi|}\right) \qquad (2)$$

ϕ is a smoothed step function which is zero within one phase and one within the other, ε denotes the porosity, γ determines the amount of reinitialization, and $\varepsilon_{\mathrm{ls}}$ describes the thickness of the interface.

Beside defining the interface between the two phases, the level-set function is used in the multiphysics coupling feature node **Two-Phase Flow, Level Set** to smooth the density and viscosity jumps across the interface through the definitions

$$\begin{aligned}\rho &= \rho_{\mathrm{air}}+(\rho_{\mathrm{resin}}-\rho_{\mathrm{air}})\phi\\ \mu &= \mu_{\mathrm{air}}+(\mu_{\mathrm{resin}}-\mu_{\mathrm{air}})\phi\end{aligned} \qquad (3)$$

Table 1 lists the parameters that are used in the model.

TABLE 1: MATERIAL PROPERTIES.

Material property	Value
Air Density: ρ_{air}	1 kg/m^3
Dynamic Viscosity of Air: μ_{air}	10^{-5} Pa·s
Resin Density: ρ_{resin}	1250 kg/m^3
Dynamic Viscosity of Resin: μ_{resin}	0.195 Pa·s
Porosity, Material 1: ε_1	0.45
Porosity, Material 2: ε_2	0.5
Porosity Material 3: ε_3	0.5
Permeability, Material 1: $\kappa_{mat1,ii}$	$2.9\cdot10^{-10}$, $8\cdot10^{-11}$, $2.9\cdot10^{-10}$ m^2
Permeability, Material 2: $\kappa_{mat2,ii}$	$2.5\cdot10^{-10}$, $7\cdot10^{-11}$, $2.5\cdot10^{-10}$ m^2
Permeability, Material 3: $\kappa_{mat3,ii}$	$1.7\cdot10^{-10}$, $8\cdot10^{-11}$, $1.7\cdot10^{-10}$ m^2

Results and Discussion

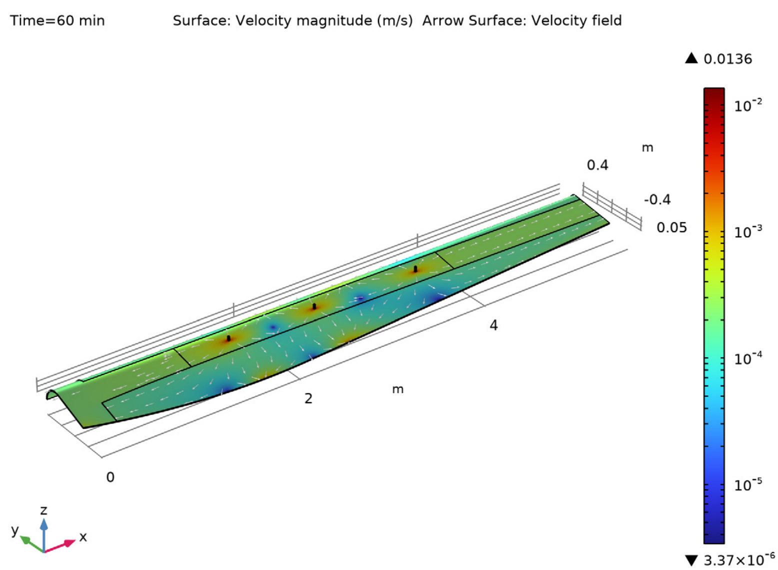

Figure 2: Velocity magnitude in logarithmic scale plotted on a cross section in the middle of the mold.

After 1 hour of simulated time, the mold is filled with resin. Figure 2 shows the velocity magnitude in the middle of the mold. The highest velocity values appear near the inlets and the small vents, where the flow channel narrows.

Figure 3 shows the pressure at the external surfaces of the blade. The highest pressure appears at the inlets.

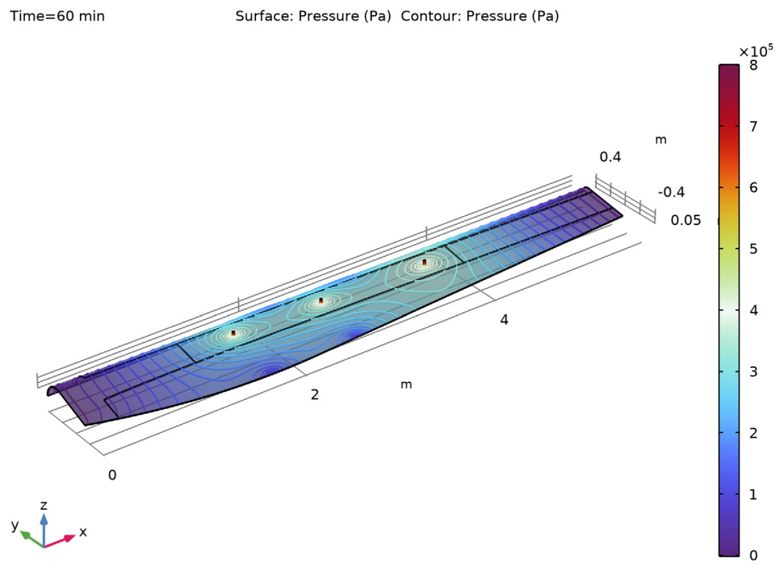

Figure 3: Pressure plotted on the external surfaces of the wind turbine blade.

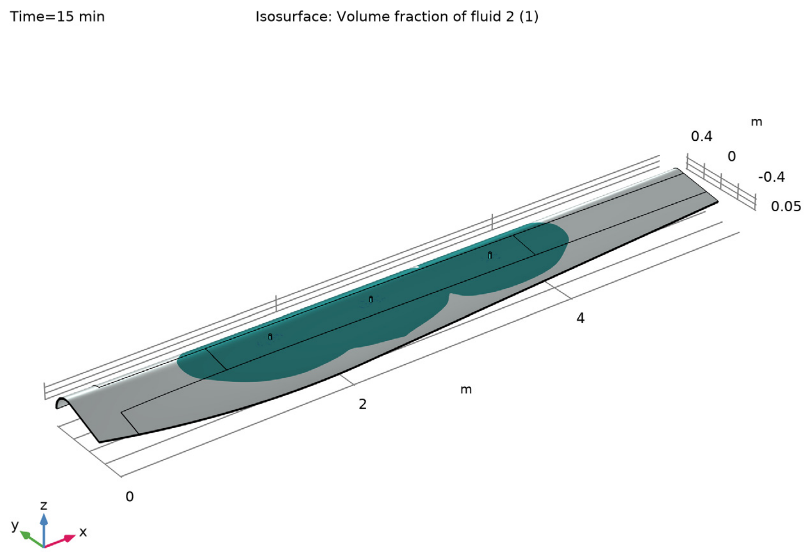

Figure 4: Volume fraction of resin after 15 minutes simulated time.

The resin front advances as shown in Figure 4 and Figure 5. In Figure 4, the resin front is shown after 15 minutes, in Figure 5, the resin interface is plotted after 5, 10, 20, and 30 minutes of simulated time. After 1 hour, the mold is completely filled with resin. The position of the vents could be optimized to avoid air bubbles within the molding process.

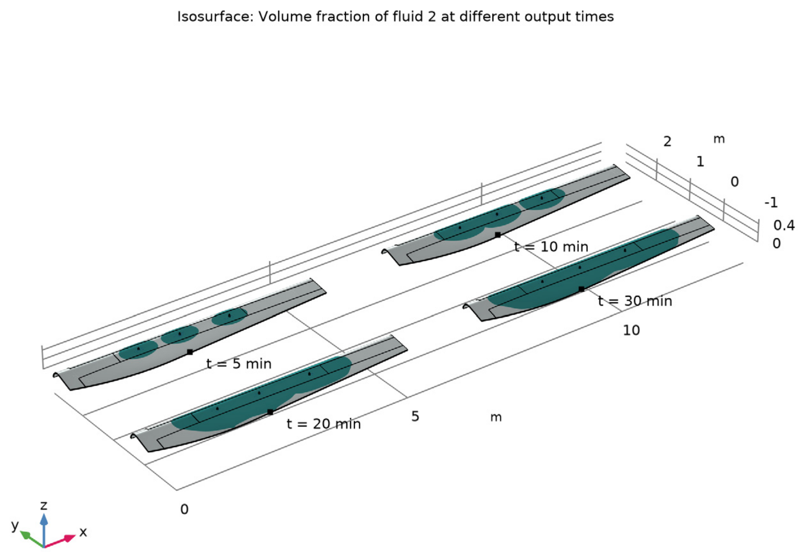

Figure 5: The Resin front advancing during the filling process. The figures show the resin interface position after 5, 10, 20, and 30 minutes.

Reference

1. Y. Jung. *An Efficient Analysis of Resin Transfer Molding Process using Extended Finite Element Method*. Other. Ecole Nationale Superieure des Mines de Saint- Etienne; Seoul National University, 2013. English. (NNT: 2013EMSE0701). (tel-00937556, https://tel.archives-ouvertes.fr/tel-00937556/)

Application Library path: `Polymer_Flow_Module/Tutorials/rtm_wind_turbine_blade`

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>Multiphase Flow>Two-Phase Flow, Level Set>Brinkman Equations**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Time Dependent with Phase Initialization**.
6 Click **Done**.

GEOMETRY 1

Start creating this model by importing the model geometry.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Advanced** section.
3 From the **Geometry representation** list, choose **CAD kernel** to make sure that the geometry, which was created using the COMSOL Multiphysics Design Module, can be imported properly.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.
2 In the **Settings** window for **Import**, locate the **Import** section.
3 Click **Browse**.
4 Browse to the model's Application Libraries folder and double-click the file `rtm_wind_turbine_blade.mphbin`.
5 Click **Import**.

GLOBAL DEFINITIONS

The parameters needed for this model are stored in an external file. You can load them as follows:

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 `rtm_wind_turbine_blade_parameters.txt`.

MATERIALS

Define the materials in the next step. Introduce them as empty material nodes first; as soon as the physics has been defined, the material node menu will show you which properties are needed for the simulation. You can then just fill in the values defined in the **Parameters** list.

Air

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

2 In the **Settings** window for **Material**, type `Air` in the **Label** text field.

Resin

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Resin` in the **Label** text field.

Porous Material 1 (pmat1)

1 Right-click **Materials** and choose **More Materials>Porous Material**.

2 Select Domains 1, 2, 12, and 13 only.

3 In the **Settings** window for **Porous Material**, locate the **Porosity** section.

4 In the ε_p text field, type `epsilon_1`.

Porous Material 2 (pmat2)

1 Right-click **Materials** and choose **More Materials>Porous Material**.

2 Select Domains 7–11 only.

3 In the **Settings** window for **Porous Material**, locate the **Porosity** section.

4 In the ε_p text field, type `epsilon_2`.

Porous Material 3 (pmat3)

1 Right-click **Materials** and choose **More Materials>Porous Material**.

2 Select Domains 3–6 only.

3 In the **Settings** window for **Porous Material**, locate the **Porosity** section.

4 In the ε_p text field, type `epsilon_3`.

BRINKMAN EQUATIONS (BR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Brinkman Equations (br)**.

2 In the **Settings** window for **Brinkman Equations**, locate the **Physical Model** section.

3 Find the **Porous treatment of no slip condition** subsection. From the list, choose **Porous slip**. Using the new porous-slip wall treatment accounts for porous walls without resolving the full flow profile in the boundary layer. Instead, a stress condition is applied at the porous surfaces by utilizing an asymptotic solution.

Inlet 1

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

2 Select Boundaries 44, 60, and 72 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. In the p_0 text field, type `800[kPa]`.

6 Locate the **Boundary Selection** section. Click **Create Selection**.

7 In the **Create Selection** dialog box, type `Inlet` in the **Selection name** text field.

8 Click **OK**.

Create selections as you will need the selected groups of boundaries again when defining the boundary settings for the **Level-Set in Porous Media** interface.

Outlet 1

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

2 Select Boundaries 1, 4, 5, 8, 40, 51, 52, 56, 65, 66, and 84–91 only.

3 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Outlet` in the **Selection name** text field.

6 Click **OK**.

LEVEL SET IN POROUS MEDIA (LS)

Level Set Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Level Set in Porous Media (ls)>Porous Medium 1** click **Level Set Model 1**.

2 In the **Settings** window for **Level Set Model**, locate the **Level Set Model** section.

3 In the γ text field, type `2e-4`.

4 In the ε_{ls} text field, type `2*d_i`.

Initial Values, Fluid 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Level Set in Porous Media (ls)** click **Initial Values, Fluid 2**.

2 Select Domains 9–11 only.

Inlet 1

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**.

4 Locate the **Level Set Condition** section. From the list, choose **Fluid 2 (φ = 1)**.

Outlet 1

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**.

MULTIPHYSICS

Two-Phase Flow, Level Set 1 (tpf1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Two-Phase Flow, Level Set 1 (tpf1)**.

2 In the **Settings** window for **Two-Phase Flow, Level Set**, locate the **Fluid 1 Properties** section.

3 From the **Fluid 1** list, choose **Air (mat1)**.

4 Locate the **Fluid 2 Properties** section. From the **Fluid 2** list, choose **Resin (mat2)**.

MATERIALS

Now fill in the material properties.

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1	kg/m³	Basic
Dynamic viscosity	mu	1e-5	Pa·s	Basic

Resin (mat2)

1 In the **Model Builder** window, click **Resin (mat2)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	1250	kg/m³	Basic
Dynamic viscosity	mu	0.195	Pa·s	Basic

Porous Material 1 (pmat1)

1 In the **Model Builder** window, click **Porous Material 1 (pmat1)**.

2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	{kappa11, kappa22, kappa33} ; kappaij = 0	{2.9e-10, 8e-11, 2.9e-10}	m²	Basic

Porous Material 2 (pmat2)

1 In the **Model Builder** window, click **Porous Material 2 (pmat2)**.

2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	{kappa11, kappa22, kappa33} ; kappaij = 0	{2.5e-10, 7e-11, 2.5e-10}	m²	Basic

Porous Material 3 (pmat3)

1. In the **Model Builder** window, click **Porous Material 3 (pmat3)**.
2. In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	{kappa11, kappa22, kappa33} ; kappaij = 0	`{1.7e-10, 8e-11, 1.7e-10}`	m²	Basic

DEFINITIONS

Create a few more selections that are useful for meshing the geometry.

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

Upside Boundaries

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>Selections** node.
2. Right-click **Definitions** and choose **Selections>Explicit**.
3. In the **Settings** window for **Explicit**, locate the **Input Entities** section.
4. From the **Geometric entity level** list, choose **Boundary**.
5. Select the **Group by continuous tangent** check box.
6. Select Boundaries 2, 14, 27, 36, 43, 59, 71, and 81 only.
7. In the **Label** text field, type `Upside Boundaries`.

Vent Edges

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 **Edge**.
4. Select Edges 59, 64, 65, 69, 72, 88, 96, 101, 102, 106, 109, 121, 128, 129, 133, and 136 only.
5. In the **Label** text field, type `Vent Edges`.

MESH 1

Now mesh the geometry. This example uses a swept mesh. For thin geometries a swept mesh offers a good mesh quality with a moderate number of mesh elements.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

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

3 From the **Selection** list, choose **Upside Boundaries**.

Size 1

1 Right-click **Free Triangular 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 **Edge**.

4 From the **Selection** list, choose **Vent Edges**.

5 Locate the **Element Size** section. Click the **Custom** button.

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

7 Select the **Maximum element size** check box. In the associated text field, type `d_i/4`.

8 Select the **Minimum element size** check box. In the associated text field, type `d_i/5`.

Size 2

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** check box. In the associated text field, type `1.5*d_i`.

6 Click **Build Selected**.

Swept 1

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

First, mesh the blade leaving out the cylindrical inlet ports.

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

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

4 Select Domains 1–8, 12, and 13 only. You can use the **Zoom Box** and **Zoom Extents** buttons to zoom into and out of the domain to select small boundaries.

Distribution 1

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

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

3 In the **Number of elements** text field, type `2`.

4 Click **Build Selected**.

Now mesh the cylindrical inlet ports separately.

Swept 2

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

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

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

4 Select Domains 9–11 only.

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.

2 Right-click **Distribution 1** and choose **Build Selected**.

STUDY 1

Before solving the model, specify the output times and make sure that the time step is sufficiently small to catch the advancement of the resin front properly.

Step 2: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 2: Time Dependent**.

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

3 From the **Time unit** list, choose **min**.

4 In the **Output times** text field, type `range(0,1,60)`.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

4 From the **Steps taken by solver** list, choose **Strict**. This way the maximum time step is limited to the output interval of 1 minute.

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

DEFINITIONS

To get Figure 2 you have to modify the default plot as described below. First, however, define another selection to facilitate plotting the velocity on the midsurface of the geometry.

Midsurface

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 the **Group by continuous tangent** check box.

5 Select Boundaries 6, 17, 26, 35, and 80 only.

6 In the **Label** text field, type `Midsurface`.

RESULTS

Slice

1 In the **Model Builder** window, expand the **Velocity (br)** node.

2 Right-click **Slice** and choose **Disable**.

Velocity (br)

1 In the **Model Builder** window, click **Velocity (br)**.

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

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

4 From the **Selection** list, choose **Midsurface**.

Surface 1

1 Right-click **Velocity (br)** and choose **Surface**.

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

3 From the **Scale** list, choose **Logarithmic**.

Arrow Surface 1

1 Right-click **Velocity (br)** and choose **Arrow Surface**.

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

3 From the **Components to plot** list, choose **Tangential**.

4 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Normalized**.

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

6 Select the **Scale factor** check box. In the associated text field, type `80`.

Velocity (br)

1 In the **Model Builder** window, click **Velocity (br)**.

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

3 Select the **Show maximum and minimum values** check box.

4 In the **Velocity (br)** toolbar, click **Plot**.

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

Pressure (br)

Change the default pressure plot to get Figure 3 by following the steps below.

Contour 1

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

2 Right-click **Pressure (br)** and choose **Contour**.

3 In the **Settings** window for **Contour**, locate the **Expression** section.

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

5 Locate the **Levels** section. In the **Total levels** text field, type `40`.

6 Clear the **Round the levels** check box.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface**.

8 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

9 In the **Pressure (br)** toolbar, click **Plot**.

Volume Fraction of Fluid 1 (ls)

The default plot shows the volume fraction of fluid 1. Do the following changes to get Figure 4.

Slice 1

1 In the **Model Builder** window, expand the **Results>Volume Fraction of Fluid 1 (ls)** node.

2 Right-click **Slice 1** and choose **Disable**.

Isosurface 1

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

2 In the **Settings** window for **Isosurface**, locate the **Expression** section.

3 In the **Expression** text field, type `ls.Vf2`.

4 Locate the **Levels** section. In the **Levels** text field, type `0 0.5 1`.

5 Locate the **Coloring and Style** section. From the **Isosurface type** list, choose **Filled**.

6 From the **Coloring** list, choose **Color table**.

7 Click **Change Color Table**.

8 In the **Color Table** dialog box, select **Aurora>AuroraAustralisDark** in the tree.

9 Click **OK**.

Volume Fraction of Fluid 2 (ls)

1 In the **Model Builder** window, click **Volume Fraction of Fluid 1 (ls)**.

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

3 From the **Time (min)** list, choose **15**.

4 In the **Label** text field, type `Volume Fraction of Fluid 2 (ls)`.

5 In the **Volume Fraction of Fluid 2 (ls)** toolbar, click **Plot**.

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

Volume Fraction of Fluid 2 - Array

Finally, to produce Figure 5 copy this plot and do the following.

1 Right-click **Volume Fraction of Fluid 2 (ls)** and choose **Duplicate**.

2 In the **Model Builder** window, click **Volume Fraction of Fluid 2 (ls) 1**.

3 In the **Settings** window for **3D Plot Group**, type `Volume Fraction of Fluid 2 - Array` in the **Label** text field.

4 Click to expand the **Plot Array** section. Select the **Enable** check box.

5 From the **Array shape** list, choose **Square**.

Isosurface 1

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

2 In the **Settings** window for **Isosurface**, click to expand the **Plot Array** section.

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

4 From the **Time (min)** list, choose **5**.

5 Locate the **Plot Array** section. Select the **Manual indexing** check box.

6 In the **Row index** text field, type `1`.

Isosurface 2

1 Right-click **Results>Volume Fraction of Fluid 2 - Array>Isosurface 1** and choose **Duplicate**.

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

3 From the **Time (min)** list, choose **10**.

4 Locate the **Plot Array** section. In the **Column index** text field, type `1`.

Isosurface 1, Isosurface 2

1 In the **Model Builder** window, under **Results>Volume Fraction of Fluid 2 - Array**, Ctrl-click to select **Isosurface 1** and **Isosurface 2**.

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

Isosurface 3

1 In the **Settings** window for **Isosurface**, locate the **Data** section.

2 From the **Time (min)** list, choose **20**.

3 Locate the **Plot Array** section. In the **Row index** text field, type 0.

Isosurface 4

1 In the **Model Builder** window, click **Isosurface 4**.

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

3 From the **Time (min)** list, choose **30**.

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

5 Locate the **Plot Array** section. In the **Row index** text field, type 0.

6 In the **Volume Fraction of Fluid 2 - Array** toolbar, click **Plot**.

Now add the annotations to the different plots within the plot array.

Volume Fraction of Fluid 2 - Array

In the **Model Builder** window, click **Volume Fraction of Fluid 2 - Array**.

Table Annotation 1

1 In the **Volume Fraction of Fluid 2 - Array** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
2	1.5	0.5	t = 5 min
9	1.5	0.5	t = 10 min
2	-1	0.5	t = 20 min
9	-1	0.5	t = 30 min

Volume Fraction of Fluid 2 - Array

1 In the **Model Builder** window, click **Volume Fraction of Fluid 2 - Array**.

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 `Isosurface: Volume fraction of fluid 2 at different output times`.

5 Clear the **Parameter indicator** text field.

6 In the **Volume Fraction of Fluid 2 - Array** toolbar, click **Plot**.

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