Created in COMSOL Multiphysics 5.6

Solar Panel in Periodic Flow

Introduction

This example is a study of a solar panel structure placed in a periodic flow field. The solar panel in question is located inside a regularly spaced array of panels subjected to an oncoming strong wind; see Figure 1. The model solves for the flow around the panel and the structural displacement due to the fluid load. It is assumed that enough panels are positioned both upstream and downstream of the panel for periodic flow conditions to be applicable in the streamwise direction. Seen from a position high above the ground, the array of solar panels acts as a rough boundary for the atmospheric flow.

Note: This application requires the CFD Module and the Structural Mechanics Module.

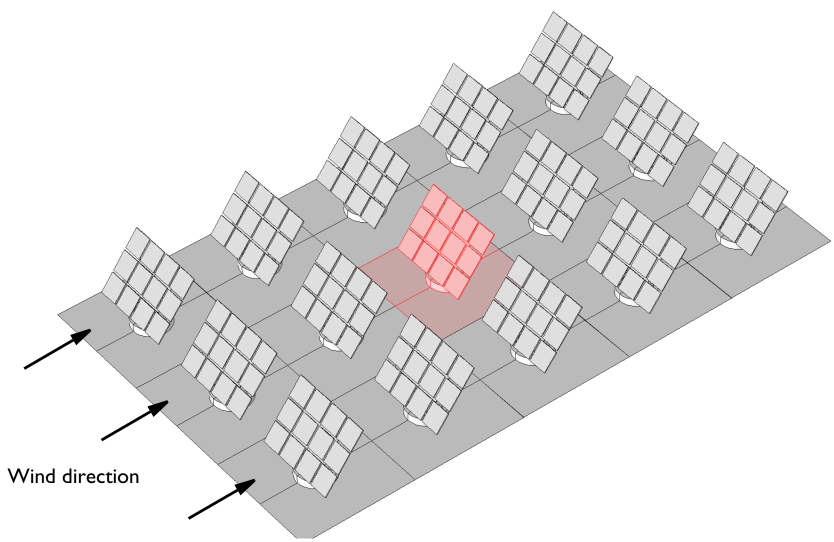


Figure 1: The modeled solar panel (in red) located in a regular array of identical panels.

Model Definition

The current investigation consists of two main parts:

- Solving for the fluid flow around the solar panel with a free stream velocity above the solar panel of 25 m/s (90 km/h).
- Studying the deformation of the solar panel caused by the fluid load.

MODEL GEOMETRY

This model uses the same geometry for both the fluid-flow modeling and the subsequent structural-mechanics simulation. Figure 2 and Figure 3 show the solar panel geometry. More specifically, Figure 2 shows the solar panel's front, which faces the flow, while Figure 3 shows its rear..

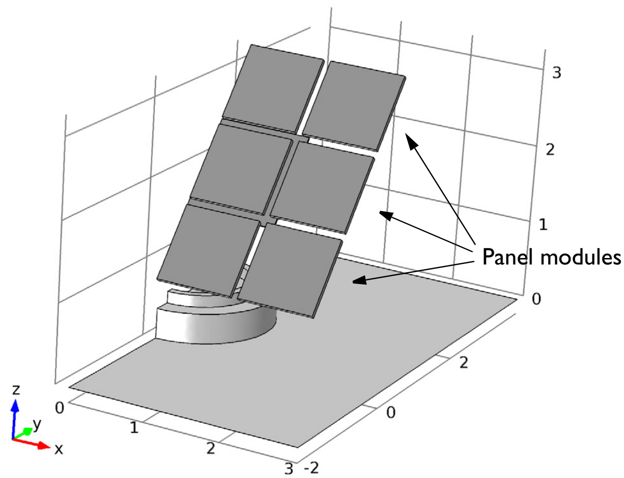


Figure 2: Front view of the solar panel geometry.

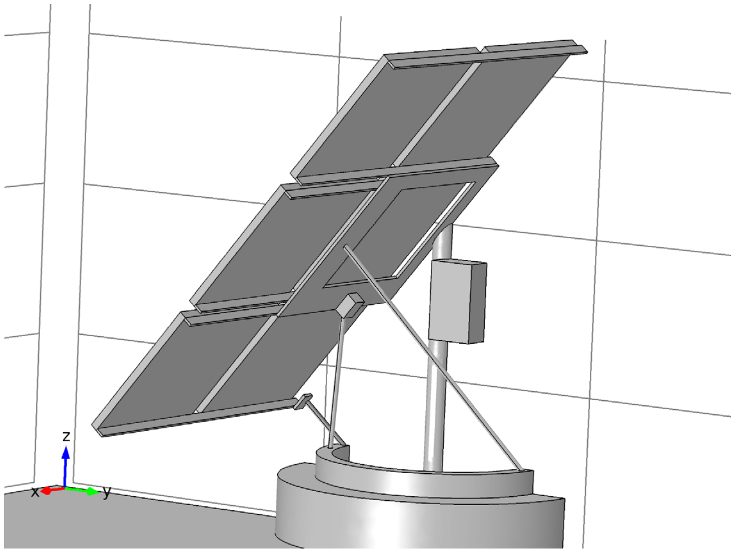


Figure 3: Rear view of the solar panel geometry.

SOLUTION STRATEGY

The model uses two different studies: one solving for the turbulent flow around the solar panel using a Turbulent Flow, k-ε physics interface, and the other solving for the structural stresses and displacements using a Solid Mechanics physics interface. The Multiphysics coupling feature Fluid-Structure Interaction, Fixed Geometry, is used to couple the fluid flow boundary load from the fluid domains to the solid domains. This corresponds to a one-way fluid-structure interaction analysis.

FLUID-FLOW SIMULATION

In the fluid-flow part, the goal is to simulate the conditions when the free-stream flow above the solar panel is about 25 m/s. This corresponds to 10 on the Beaufort scale and the land conditions "trees broken or uprooted; considerable structure damage" (Ref. 1). The Reynolds number, based on the height of the panel structure and the free stream velocity, is about $6.6 \cdot 10^6$. These conditions generate a very complex flow field. To reach a converged flow solution, the following modeling steps are applied:

1. The fluid-flow geometry is defined tall enough to capture a boundary layer type flow reaching above the array of solar panels; see Figure 4. A mapped mesh is used in the solar panels, and the fluid domain is refined to resolve the flow field. The definition of the geometry and the mesh are very important for this model, but they are rather long

and complex. They are imported from the template MPH-file `solar_panel_geom.mph` included in the model's Application Libraries folder. The step-by-step instructions for the geometry and mesh can be found in the documentation of the *Solar Panel in Periodic Flow Template* model.

2. A periodic flow condition is used to prescribe a pressure difference in the streamwise direction that ensures an average velocity of 25 m/s at the top. An ODE is used to compute the pressure drop necessary to obtain the desired free stream velocity. The top boundary is modeled with a boundary stress condition.
3. Convergence in the first iterations of the nonlinear solver is ensured adding isotropic diffusion inversely proportional to the CFL number used for pseudo-time stepping. The CFL number starts at a low value (often 3 or 5), and increases as the solution progress until it reaches CFL>1000. The isotropic diffusion will have an important effect on the first iterations of the nonlinear solver, but will tend to zero as the CFL parameter increases. This strategy provides convergence without polluting the solution.

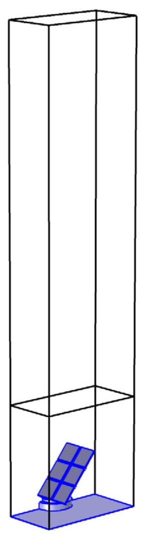


Figure 4: Computational box used in the fluid-flow simulation.

FLUID-FLOW BOUNDARY CONDITIONS

Streamwise Conditions

A periodic flow condition is applied in the streamwise directions with a prescribed pressure difference.

Lateral Conditions

The solar panel structure is assumed to be symmetric about a streamwise plane through the center of the structure. Because identical solar panels are positioned on both sides of the panel in a regular pattern, you only need to include one half of the geometry and apply symmetry boundary conditions in the lateral directions.

Top Boundary Condition

You can apply normal stress conditions for the momentum equations to prescribe a flow of a boundary layer type at the top. Assuming that the viscous stresses are small at this position, the stress can be prescribed in accordance with a pressure decreasing linearly from the inlet to the outlet.

STRUCTURAL-MECHANICS SIMULATION

In the structural mechanics simulation, the fluid load computed from the fluid-flow simulation is applied to the solar panel structure, and then the model is solved for the resulting structural stresses and displacements. The Multiphysics coupling feature Fluid-Structure Interaction, Fixed Geometry, is used to couple the total fluid stress on the panel surfaces from the flow simulation to the structural one. Safety subnodes are added to check the risk of failure of the different materials conforming the solar panel.

Solar Panel Material

The solar panel supports — the main cylindrical base and the thinner solid supports — are made of structural steel, as is the plate on the back of the panel connecting the panel modules. The square cross-sectional bars connecting the panel modules are made of aluminum. Concerning the design of solar panels, the panel modules consist of assemblies of small solar cells. These are typically covered with glass and mounted in an aluminum frame. For simplicity, model the individual solar panel modules as made of a material with a stiffness of 3.5 GPa (5 percent the stiffness of solid aluminum), and a Poisson's ratio of 0.33.

Structural Mechanics Boundary Conditions

The panel supports are assumed to be fixed at the ground, so that fixed constraints apply at these positions. A symmetry condition is applied to the plane where the structure has been cut in half. On the panel boundaries subjected to flow in the fluid-flow simulation, boundary loads defined from the computed total stresses are imposed.

Check the Risk of Failure

Two Safety subnodes are added to set up variables which can be used to check the risk of failure according to various criteria: Von Mises for the ductile materials (aluminum and structural steel), and Rankine for the glass. The tensile strengths of the aluminum, structural steel, and glass are assumed to be 270 MPa, 250 MPa, and 45 MPa, respectively. For the sake of security, the compressive strength of the glass is assumed to be equal to its tensile strength.

Results and Discussion

Figure 5 shows the flow-field solution around the solar panel. The figure shows the flow field in the center plane of the solar panel using a surface plot combined with velocity vectors at the inlet and outlet. The flow field above the panel is of boundary layer type with a maximum velocity of 25 m/s in the free stream. The velocity magnitude in the vicinity of the solar panel is also significantly lower due to shielding from the panels positioned upstream and downstream. The flow field can therefore be said to correspond to a rough wall turbulent boundary layer, where the solar panels act as roughness elements.

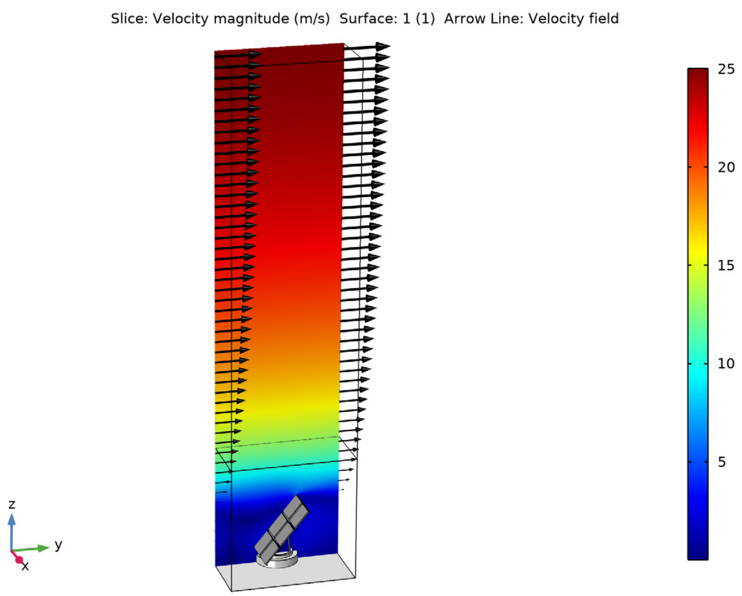


Figure 5: Final flow field in the center plane of the solar panel.

Figure 6 shows the pressure on the solar panel structure due to the surrounding flow. The maximum relative pressure, about 70 Pa, occurs in the panel's upper-right corner. By also plotting the in-plane velocity components in a plane perpendicular to the flow direction, it is possible to find the reason for the pressure maximum at the upper-right corner. A large streamwise vortex is generated behind the panel, the center of which is aligned with the panel's outer side. At the upper-right corner, the high pressure correlates with a high degree of deflection of the flow and formation of the streamwise vortex.

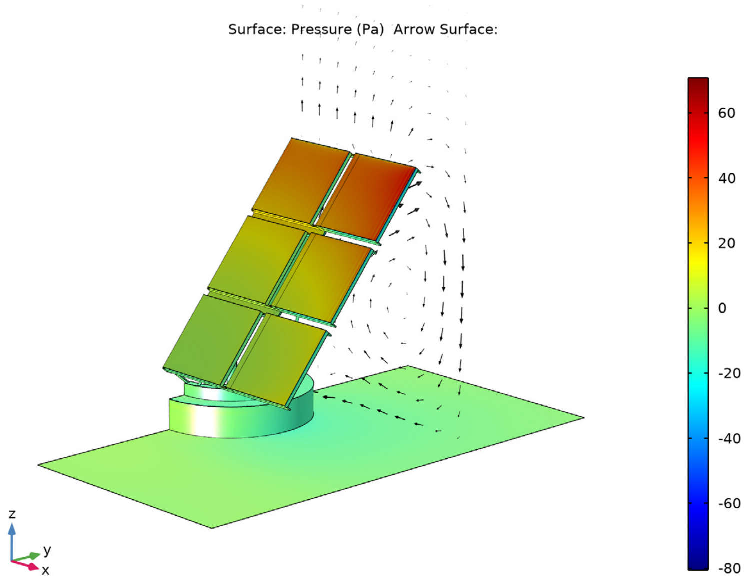


Figure 6: Surface fluid pressure contours present and in-plane velocity components 6 cm behind the panel.

The flow around the solar panel is examined in Figure 7, which shows streamlines originating at the inlet. Note that some streamlines near the center of the solar panel enter the domain and then turn around and exit, indicating that a circulation zone is present between the panels in the array. The streamlines are colored using the local turbulent kinetic energy, normalized by the kinetic energy in the free stream. The kinetic energy is low in the circulation zone in front and in the flow wake behind the panel. This is as expected because they are part of the same flow structure. The turbulent kinetic energy increases with the distance from the ground, especially when the flow passes the top of the panel. The high kinetic energy at this position correlates with a large gradient of the streamwise mean-flow velocity, as can be seen in Figure 5.

Figure 8 shows the resulting displacement of the detailed structure, and the velocity streamlines around the whole panel. In correspondence with the pressure plot in Figure 6, the largest displacement occurs in the upper-right corner. However, the maximum displacement is small, about 1 mm. This indicates that the surrounding solar panels effectively shield the solar panel from oncoming flow. It also points in the direction that for a solar panel positioned inside a large array, the fluid load on the structure at the current free stream velocity is not significant enough to dictate the design of the structure. This conclusion is supported by the minimum safety factor obtained, which is much larger than one.

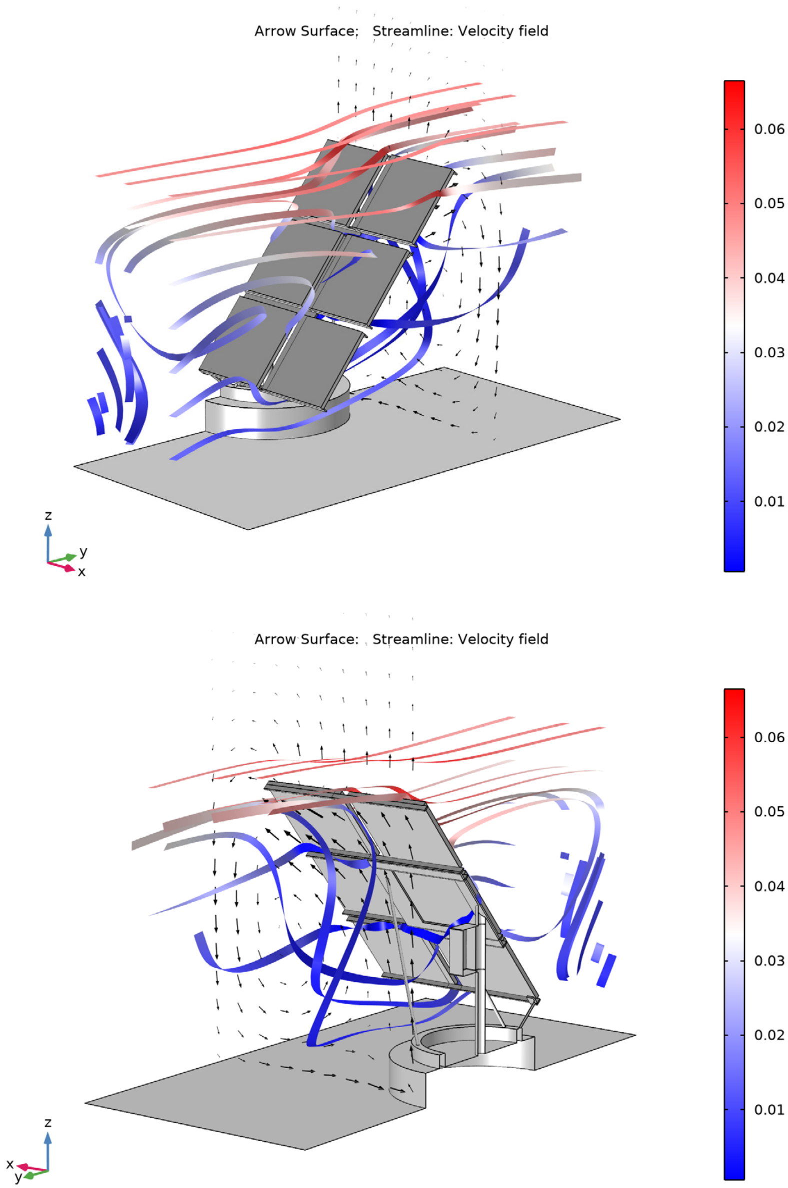


Figure 7: Velocity streamlines, colored by the turbulent kinetic energy normalized by the kinetic energy in the free stream, and in-plane velocity components 6 cm behind the panel.

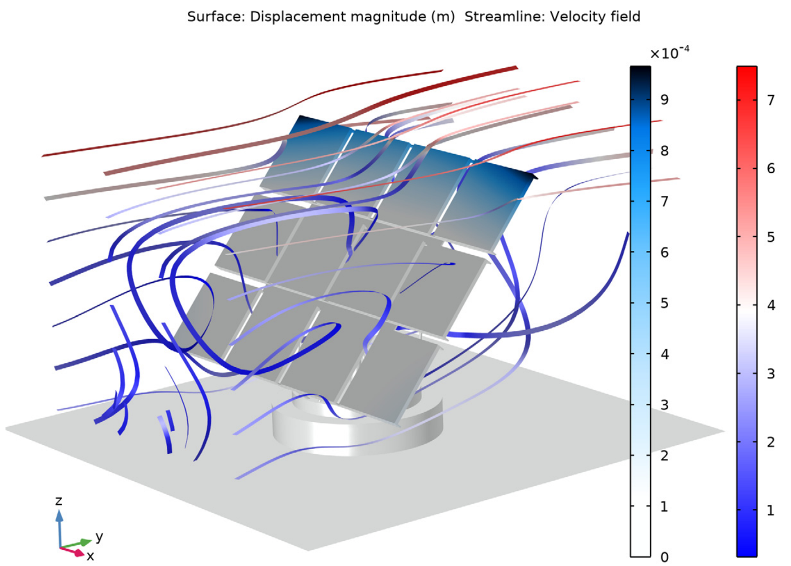


Figure 8: Structural displacement of the solar panel due to the fluid-flow load, and velocity streamlines colored by the velocity magnitude.

Reference

1. http://en.wikipedia.org/wiki/Beaufort_scale

Application Library path: `CFD_Module/Fluid-Structure_Interaction/solar_panel`

Modeling Instructions

Begin by loading the file containing the geometry and the mesh.

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

Browse to the model's Application Libraries folder and double-click the file `solar_panel_geom.mph`.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Air**.

4 Click **Add to Component** in the window toolbar.

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

DEFINITIONS

Add a nonlocal coupling that can calculate averages on the top surface.

Average 1 (aveop1)

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

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

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

4 Select Boundary 7 only (top boundary).

TURBULENT FLOW, K-ε (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ε (spf)**.

2 Select Domains 1 and 2 only (fluid domains).

Add isotropic diffusion to improve convergence. Specify the coefficient to be inversely proportional to the pseudo time-stepping parameter CFLCMP so that the isotropic diffusion added tends to zero when CFLCMP increases.

3 Click the **Show More Options** button in the **Model Builder** toolbar.

4 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Stabilization**.

5 Click **OK**.

6 In the **Settings** window for **Turbulent Flow, k-ε**, click to expand the **Inconsistent Stabilization** section.

7 Find the **Navier-Stokes equations** subsection. Select the **Isotropic diffusion** check box.

8 In the δ_{id} text field, type `0.5/CFLCMP`.

Prescribe an approximate streamwise velocity profile and a streamwise pressure drop to go with it.

Initial Values 1

1 In the **Model Builder** window, expand the **Turbulent Flow, k-ε (spf)** node, then click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

`0`	x
`flc1hs(z[1/m]-5,5)*Utop`	y
`0`	z

4 In the p text field, type `0.5[Pa]*(yEnd-y)/yLen`.

Periodic Flow Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Flow Condition**.

2 Select Boundaries 2, 5, 134, and 135 only (inflow and outflow boundaries).

3 In the **Settings** window for **Periodic Flow Condition**, locate the **Pressure Difference** section.

4 In the Δp text field, type `pdiff`.

Symmetry 1

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

2 Select Boundaries 1, 4, 319, and 320 only (lateral boundaries).

Open Boundary 1

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

2 Select Boundary 7 only.

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

4 In the f_0 text field, type `pdiff*(yEnd-y)/yLen`.

5 Locate the **Turbulence Conditions** section. In the U_{ref} text field, type `spf.U`.

Add an ODE that calculates the pressure drop necessary to obtain a free stream velocity of 25 m/s.

6 Click the **Show More Options** button in the **Model Builder** toolbar.

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

8 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Description
`pdiff`	`-aveop1(v)+Utop`	`0.5[Pa]`	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `pressure` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Pressure (Pa)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `velocity` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Velocity (m/s)**.

14 Click **OK**.

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

STUDY 1

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 1>AMG, fluid flow variables (spf)>Multigrid 1** node.

The equation for the pressure difference is a DAE, that is, its system matrix has zeros on the diagonal and must therefore be treated with a Vanka-smoother in the iterative solver. This is already taken care of when working with the default solver for fluid flow.

4 In the **Model Builder** window, right-click **Solution 1 (sol1)** and choose **Compute**.

Before working with plots, disable plot when selected (**Automatic update of plots**). Due to the size of the datasets created, it is more convenient to generate plots actively.

RESULTS

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

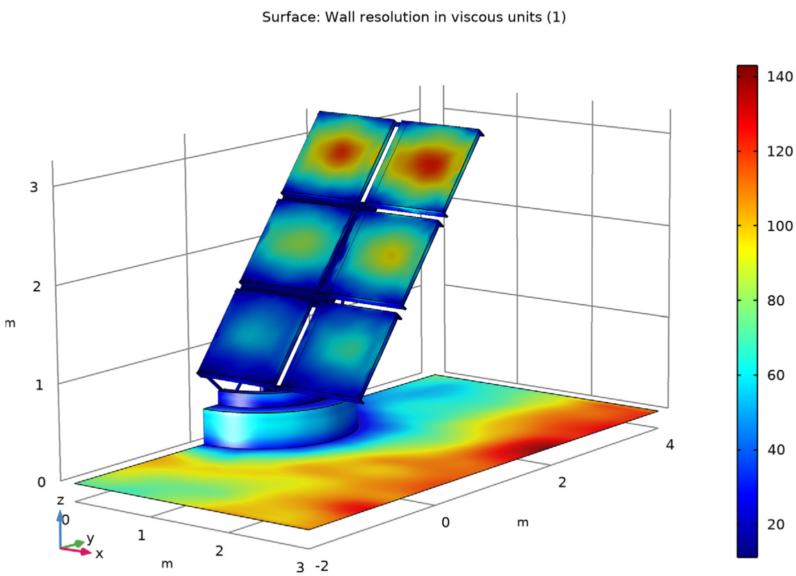
2 Select the **Only plot when requested** check box.

Check the wall resolution to see if the wall distance is such that the computational domain is in the logarithmic layer.

Wall Resolution (spf)

1 In the **Model Builder** window, under **Results** click **Wall Resolution (spf)**.

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



The maximum wall resolution in viscous units is around 160. Since the Reynolds number is approximately $6.6 \cdot 10^6$, the wall distance is within the logarithmic layer.

To plot Figure 5 and inspect the velocity field follow the steps below.

Study 1/Solution 1 (2) (sol1)

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Datasets** and choose **Solution**.

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 **Edge**.

4 Select Edges 1, 4, 198, and 200 only.

Slice

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

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

3 From the **Entry method** list, choose **Coordinates**.

4 In the **x-coordinates** text field, type `1e-3`.

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 **Exterior Walls**.

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

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

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

Arrow Line 1

1 Right-click **Velocity (spf)** and choose **Arrow Line**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 Locate the **Coloring and Style** section. Select the **Scale factor** check box.

5 In the associated text field, type `0.09`.

6 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `100`.

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

8 In the **Velocity (spf)** toolbar, click **Plot**.

To plot the pressure as done in Figure 6 perform the following steps

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 (2) (sol1)**.

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

5 In the **y-coordinate** text field, type `2.0`.

Pressure

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

2 Right-click **Pressure** and choose **Delete**.

Surface 2

1 In the **Pressure (spf)** toolbar, click **Surface**.

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

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

Arrow Surface 1

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

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

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

4 Locate the **Expression** section. In the **y component** text field, type `0`.

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

6 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `800`.

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

Follow the steps below to create Figure 7, which plots fluid flow streamlines colored by the turbulent kinetic energy.

Streamline plot

1 Right-click **Pressure (spf)** and choose **Duplicate**.

2 Right-click **Pressure (spf) 1** and choose **Rename**.

3 In the **Rename 3D Plot Group** dialog box, type `Streamline plot` in the **New label** text field.

4 Click **OK**.

Surface 2

1 In the **Model Builder** window, expand the **Streamline plot** node.

2 Right-click **Surface 2** and choose **Delete**.

Streamline plot

Click **Yes** to confirm.

Streamline 1

1 Right-click **Streamline plot** and choose **Streamline**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.

5 In the **x** text field, type `0.5*1^range(1,7), 1.0*1^range(1,7), 1.75*1^range(1,7)`.

6 In the **y** text field, type `-2`.

7 In the **z** text field, type `range(0.5,3/6,3.5), range(0.5,3/6,3.5), range(0.5,3/6,3.5)`.

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

9 Select the **Width scale factor** check box.

10 In the associated text field, type `0.05`.

Color Expression 1

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

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

3 In the **Expression** text field, type `k/(0.5*25^2)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 In the **Streamline plot** toolbar, click **Plot**.

This concludes the study of the flow field. The next step is to add the structural analysis and couple it with the flow field.

Add new selections to be used when setting up the Solid Mechanics physics interface and in the plots.

DEFINITIONS

Solid Domains

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

2 In the **Settings** window for **Union**, type `Solid Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Structural steel**, **Aluminum**, and **Glass**.

5 Click **OK**.

Solid Domains except Glass

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

2 In the **Settings** window for **Union**, type `Solid Domains except Glass` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **Structural steel**, **Aluminum**, and **Glass**.

5 Click **OK**.

Exterior Boundaries to Solid Domains

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

2 In the **Settings** window for **Adjacent**, type `Exterior Boundaries to Solid Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Solid Domains** in the **Input selections** list.

5 Click **OK**.

Floor and Solar Panel Base

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

2 In the **Settings** window for **Difference**, type `Floor and Solar Panel Base` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **Walls** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **Add**.

9 In the **Add** dialog box, select **Exterior Boundaries to Solid Domains** in the **Selections to subtract** list.

10 Click **OK**.

ADD PHYSICS

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

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

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component 1** in the window toolbar.

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

SOLID MECHANICS (SOLID)

1 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Solid Domains**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Safety 1

1 In the **Physics** toolbar, click **Attributes** and choose **Safety**.

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

3 From the **Selection** list, choose **Solid Domains except Glass**.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Safety 2

1 In the **Physics** toolbar, click **Attributes** and choose **Safety**.

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

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

4 Locate the **Failure Model** section. From the **Failure criterion** list, choose **Rankine**.

ADD MULTIPHYSICS

1 In the **Physics** toolbar, click **Add Multiphysics** to open the **Add Multiphysics** window.

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

3 In the tree, select **Fluid Flow>Fluid-Structure Interaction>Fluid-Solid Interaction, Fixed Geometry**.

4 Click **Add to Component** in the window toolbar.

5 In the **Physics** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

MULTIPHYSICS

Fluid-Structure Interaction 1 (fsi1)

1 In the **Settings** window for **Fluid-Structure Interaction**, locate the **Fixed Geometry** section.

2 From the **Fixed geometry coupling type** list, choose **Fluid loading on structure**.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in>Structural steel**.
4. Click **Add to Component** in the window toolbar.
5. In the tree, select **Built-in>Aluminum**.
6. Click **Add to Component** in the window toolbar.
7. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

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 **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Manual**.
4. Click **Clear Selection**.
5. Select Domains 1 and 2 only.

Structural steel (mat2)

1. In the **Model Builder** window, click **Structural steel (mat2)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Structural steel**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Tensile strength	sigmat	250[MPa]	Pa	Isotropic strength parameters

Aluminum (mat3)

1. In the **Model Builder** window, click **Aluminum (mat3)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Aluminum**.

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

Property	Variable	Value	Unit	Property group
Tensile strength	sigmat	270[MPa]	Pa	Isotropic strength parameters

Material 4 (mat4)

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

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

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

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	35e8	Pa	Basic
Poisson's ratio	nu	0.33	1	Basic
Density	rho	1000	kg/m³	Basic
Tensile strength	sigmat	45[MPa]	Pa	Isotropic strength parameters
Compressive strength	sigmac	45[MPa]	Pa	Isotropic strength parameters

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

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

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

Symmetry 1

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

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

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

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>Stationary**.

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

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

STUDY 2

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

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

3 Clear the **Generate default plots** check box.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Turbulent Flow, k-ε (spf)**.

4 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

6 From the **Study** list, choose **Study 1, Stationary**.

Use an iterative solver to reduce the memory requirements.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node.

4 Right-click **Suggested Iterative Solver (solid)** and choose **Enable**.

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

RESULTS

Study 2/Solution 2, Solid Domains

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

2 In the **Settings** window for **Solution**, type `Study 2/Solution 2, Solid Domains` in the **Label** text field.

3 Locate the **Solution** section. From the **Solution** list, choose **Solution 2 (sol2)**.

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 From the **Selection** list, choose **Solid Domains**.

Mirror 3D, Solution 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D, Solution 1` in the **Label** text field.

Mirror 3D, Solid Domains

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D, Solid Domains` in the **Label** text field.

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

Floor and Solar Panel Base

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

2 In the **Settings** window for **Surface**, type `Floor and Solar Panel Base` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **Floor and Solar Panel Base**.

Mirror 3D, Floor and Solar Panel Base

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, type `Mirror 3D, Floor and Solar Panel Base` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Floor and Solar Panel Base**.

Stress (solid)

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

2 In the **Settings** window for **3D Plot Group**, type `Stress (solid)` in the **Label** text field.

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

Surface 1

1 Right-click **Stress (solid)** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.mises`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Stress (solid)** toolbar, click **Plot**.

Displacement magnitude and streamlines

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement magnitude and streamlines` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 Right-click **Displacement magnitude and streamlines** and choose **Surface**.

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

3 From the **Dataset** list, choose **Mirror 3D, Solid Domains**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>solid.disp - Displacement magnitude - m**.

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

6 Select the **Reverse color table** check box.

Displacement magnitude and streamlines

In the **Model Builder** window, click **Displacement magnitude and streamlines**.

Streamline 1

1 In the **Displacement magnitude and streamlines** toolbar, click **Streamline**.

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

3 From the **Dataset** list, choose **Mirror 3D, Solution 1**.

4 Locate the **Streamline Positioning** section. From the **Entry method** list, choose **Coordinates**.

5 In the **x** text field, type `-1.75*1^range(1,7), -0.5*1^range(1,7), 0.5*1^range(1,7), 1.75*1^range(1,7)`.

6 In the **y** text field, type `-2`.

7 In the **z** text field, type `range(0.5,3/6,3.5), range(0.5,3/6,3.5), range(0.5,3/6,3.5), range(0.5,3/6,3.5)`.

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

9 In the **Width expression** text field, type `0.03`.

10 Select the **Width scale factor** check box.

Color Expression 1

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

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

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

Surface 2

1 In the **Model Builder** window, right-click **Displacement magnitude and streamlines** and choose **Surface**.

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

3 From the **Dataset** list, choose **Mirror 3D, Floor and Solar Panel Base**.

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

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

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 In the **Displacement magnitude and streamlines** toolbar, click **Plot**.

Plot the safety factor to check that the structure is far from failure.

3D Plot Group 7

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Volume 1

1 Right-click **3D Plot Group 7** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Safety>von Mises>solid.lemm1.sf1.s_f - von Mises safety factor**.

3 Click to expand the **Range** section. Select the **Manual color range** check box.

4 In the **Maximum** text field, type `2000`.

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

6 Select the **Reverse color table** check box.

3D Plot Group 7

In the **Model Builder** window, click **3D Plot Group 7**.

Max/Min Volume 1

1 In the **3D Plot Group 7** toolbar, click **More Plots** and choose **Max/Min Volume**.

2 In the **Settings** window for **Max/Min Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Safety>von Mises>solid.lemm1.sf1.s_f - von Mises safety factor**.

3 Click to expand the **Advanced** section. Locate the **Display** section. From the **Display** list, choose **Min**.

4 Locate the **Coloring and Style** section. From the **Background color** list, choose **White**.

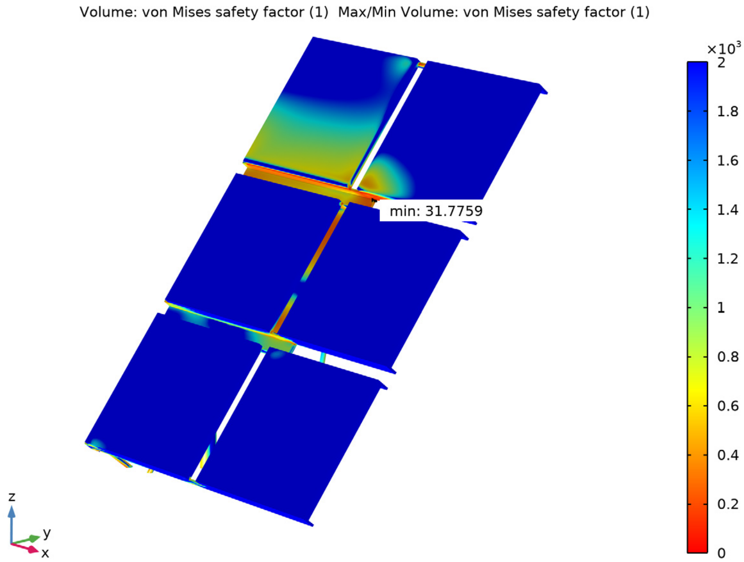
Safety factor, ductile materials

1 Right-click **3D Plot Group 7** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `Safety factor, ductile materials` in the **New label** text field.

3 Click **OK**.

4 In the **Safety factor, ductile materials** toolbar, click **Plot**.



The minimum safety factor for the ductile materials (steel and aluminum) is much larger than one, and they work within the elastic regime.

3D Plot Group 8

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Volume 1

1 Right-click **3D Plot Group 8** and choose **Volume**.

2 In the **Settings** window for **Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Safety>Rankine>solid.lemm1.sf2.s_f - Rankine safety factor**.

3 Locate the **Range** section. Select the **Manual color range** check box.

4 In the **Maximum** text field, type 2000.

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

6 Select the **Reverse color table** check box.

3D Plot Group 8

In the **Model Builder** window, click **3D Plot Group 8**.

Max/Min Volume 1

1 In the **3D Plot Group 8** toolbar, click **More Plots** and choose **Max/Min Volume**.

2 In the **Settings** window for **Max/Min Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Safety>Rankine>solid.lemm1.sf2.s_f - Rankine safety factor**.

3 Locate the **Display** section. From the **Display** list, choose **Min**.

4 Locate the **Coloring and Style** section. From the **Background color** list, choose **White**.

Safety factor, glass

1 Right-click **3D Plot Group 8** and choose **Rename**.

2 In the **Rename 3D Plot Group** dialog box, type `Safety factor, glass` in the **New label** text field.

3 Click **OK**.

4 In the **Safety factor, glass** toolbar, click **Plot**.

The minimum safety factor for the glass is large enough to consider that the structure is far from failure.