Created in COMSOL Multiphysics 6.3

Postbuckling Analysis of an Aircraft Fuselage

Introduction

In this model, an aircraft fuselage structure made of a skin–stringer assembly connected by rivets is studied. Frames and stringers reinforce the structure in the hoop and axial directions, respectively. In the case of compressive axial forces, such a lightweight structure can be subject to failure due to local buckling.

A postbuckling analysis of the structure is performed. An important part of this example is to show how to use the **Fasteners** feature to connect boundaries in the Shell interface.

Model Definition

The full geometry consists of C-shape frames connected with U-shape stringers and surrounded by a 1 mm thick skin sheet. All parts of the assembly are made of aluminum. The diameter of the fuselage is 4 m.

The frames have a 0.5 m spacing in the longitudinal direction and there are 60 stringers in the circumferential direction. The beams, made of aluminum, have dimensions 60 mm by 20 mm for the frames, and 30 mm by 20 mm for the stringers. Both have the thickness 1.3 mm.

All parts of the assembly are joined using steel rivets with a diameter of 6 mm. For the skin–stringer connection, the rivet pitch is chosen to be 50 mm to avoid inter-rivet buckling. The rivet holes are represented in the geometry.

A 15 mm high L-shape clip is used for the connection between the frame and the stringers.

The model geometry is a section of the full geometry including two frame segments and two stringers. As shown in Figure 1, the section cut is 1 m long and $1/30^{\text{th}}$ in the circumferential direction.

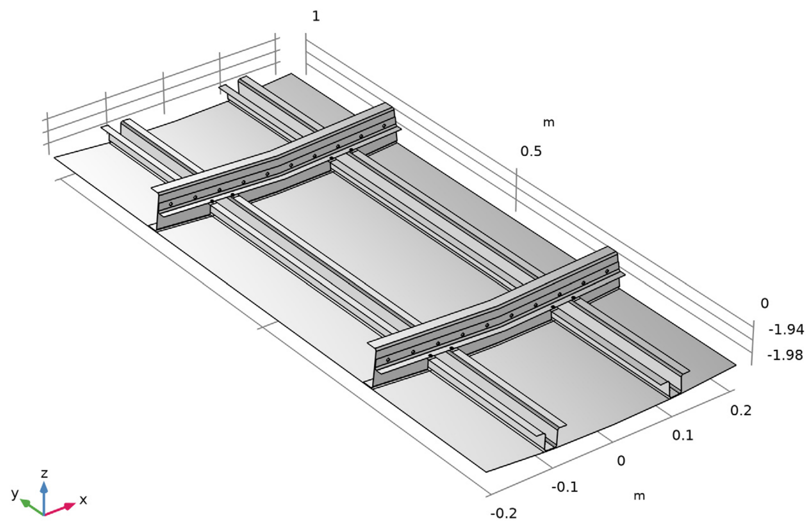

Figure 1: Model geometry of an aircraft fuselage panel.

The rivets connect all parts by transmitting normal and tangential forces. When buckling occurs, it is important to also include contact forces to avoid that parts overlap away from the rivet holes.

The fuselage skin is pressurized at p = 0.5 atm. In an ideal cylinder with radius R and thickness t, this pressure generates an axial stress (σ_a) and a hoop stress (σ_h) with the relation

$$\sigma_h = 2\sigma_a = \frac{pR}{t} \tag{1}$$

In the current geometry, the force is shared between the skin and the stiffening elements, but the same relation applies between the resultant circumferential and axial forces.

In the axial direction, an additional compressive load is added to study the structure when buckling occurs. The relation between the axial force, F_a, and the force in the hoop direction, F_h, is then

$$:F_a = -kF_h\frac{A_a}{A_h} \tag{2}$$

where A_a and A_h are the areas in the axial and hoop directions, respectively, and k is a biaxial loading parameter varying from –0.5 (nominal pressure condition) to 1 (compressive loading condition).

The edges in the plane normal to the hoop direction are constrained using symmetry conditions.

The edges in the plane normal to the axial direction are also given symmetry conditions. On one side, a standard symmetry condition is used, while on the opposite side the normal displacement is constant but nonzero. Instead, the axial force is prescribed, using the axial force F_a from Equation 2.

The following safety criterion is evaluated to check whether or not the rivets can sustain the transmitted force:

$$\left(\frac{F_{\mathrm{n}}}{F_{\mathrm{n,\,max}}}\right)^2 + \left(\frac{2F_{\mathrm{s}}}{F_{\mathrm{n,\,max}}}\right)^2 = 1 \qquad (3)$$

Here, $F_{n,max}$ is the critical normal force defined as function of the rivet yield stress σ_0 and rivet diameter d as

$$F_{\mathrm{n,\,max}} = \frac{\sigma_0 \pi d^2}{4} \qquad (4)$$

The form of Equation 3 essentially implies a Tresca yield criterion for the rivets.

The mesh size is set with a maximum element size of 10 mm, and a minimum element size of 3 mm, giving the mesh shown in Figure 2.

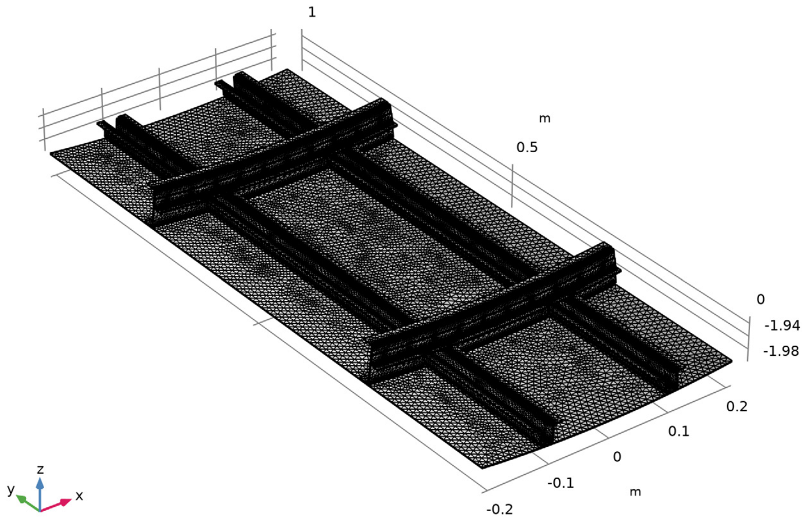

Figure 2: Model mesh.

With the chosen element size, there are at least eight elements around the each rivet hole.

Results and Discussion

Figure 3 below shows the von Mises stress for the fully compressive loading case. The pattern clearly shows the buckling in the skin. Note that high stress values occur around the rivet holes. This is because the forces between the joined plates are transmitted by the hole edges. The stress at some distance away from a rivet hole is, however, not affected by this local singularity.

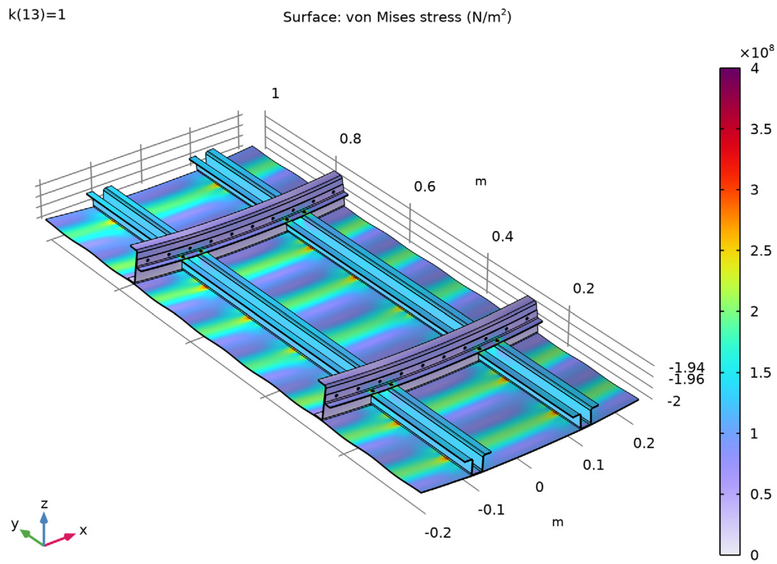

Figure 3: von Mises Stress under compressive loading.

Figure 4 shows the displacement in the radial direction for the nominal pressure loading condition. The stiffening effect of the stringers is clearly visible.

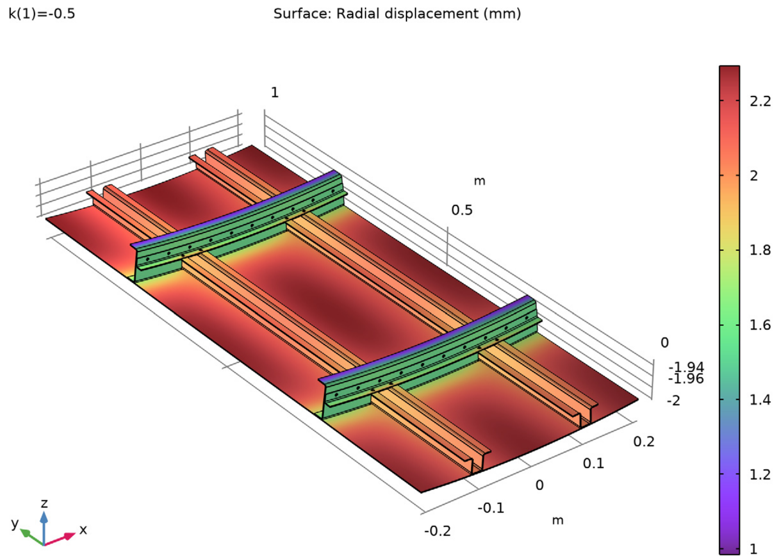

Figure 4: Radial displacement at nominal pressure.

Figure 5 shows the radial displacement at maximum compressive load when the skin has started to buckle.

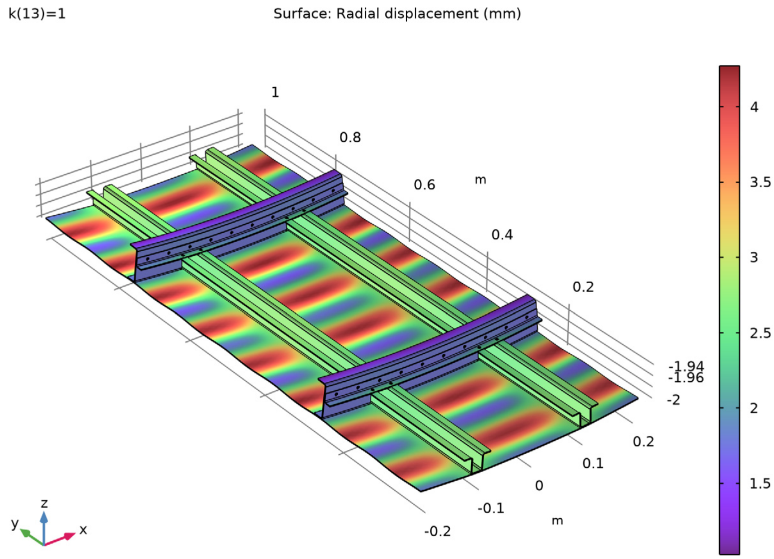

Figure 5: Radial displacement after buckling.

Figure 6 shows the axial displacement variation with respect to the biaxial loading parameter. For the nominal pressure condition (on the left), the axial displacement is positive and slightly decrease as the compressive axial force increase. Around 0.6 the slope of the curve changes since buckling occurs.

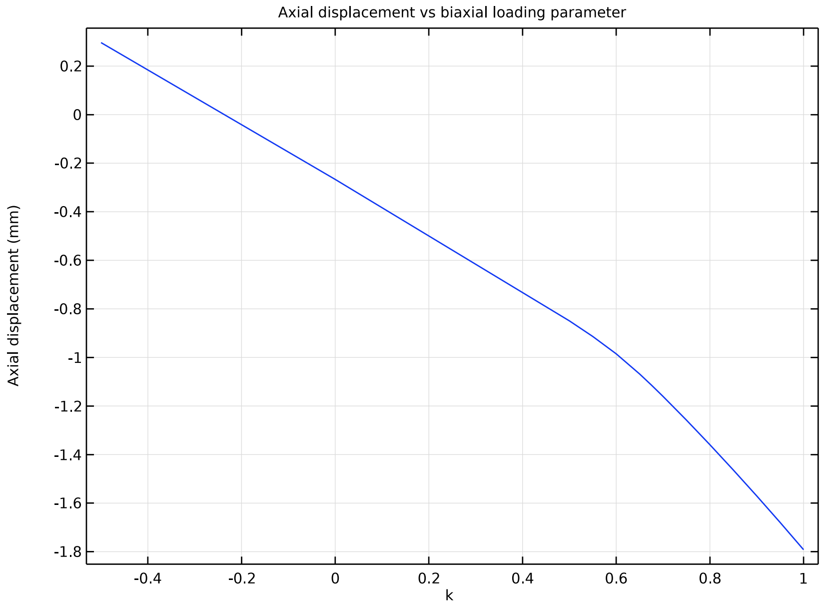

Figure 6: Axial displacement versus biaxial loading parameter.

Figure 7 shows the contact pressure at the frame–stringer intersection under maximum compressive loading.

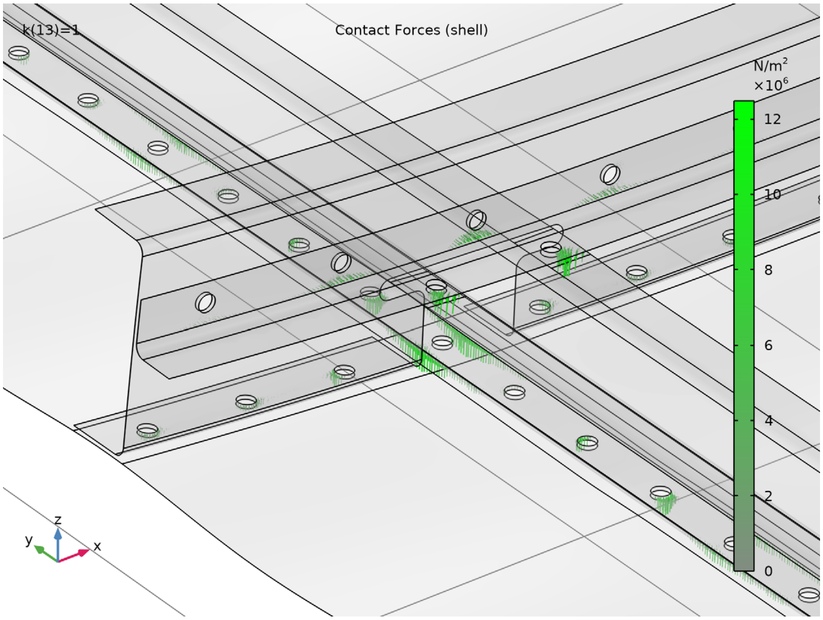

Figure 7: Contact pressure.

Notes About the COMSOL Implementation

The **Fasteners** feature is intended to model ideal local joint connection between two shell boundaries. The axial, shear, and bending forces transmitted by the fasteners are directly applied at the edges of the connected fastener holes. By default, the fastener flexibilities are evaluated based on a beam idealization. They can be tuned by the user. Alternatively, you can provide user-defined normal and tangential stiffness values.

Once the solution is available, you can visualize and evaluate the fastener forces by using result templates for plots and evaluation groups. Figure 8 shows some of the forces acting on the rivets. The tangential forces are shown in blue and the axial forces in green. An annotation is included for each fastener to make them easily identifiable. In Figure 8, the rivets with the highest tangential forces are marked with `5 (fst2)` and `7 (fst2)`. This should be interpreted as the fasteners with number 5 and 7, respectively, defined in the **Fasteners** node with the tag `fst2`. In the **Fastener Forces** evaluation group you can find the values, which are about 860 N.

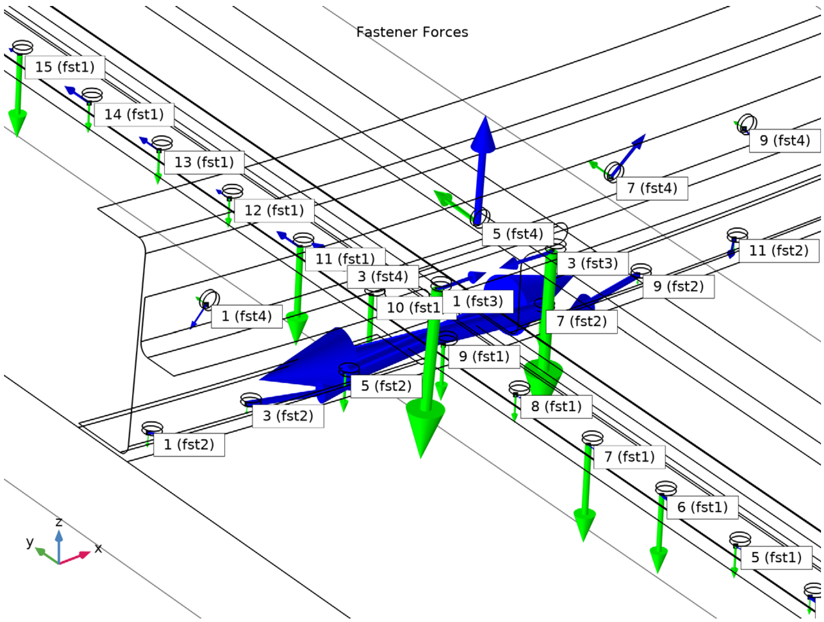

Figure 8: Forces acting on rivets for the compressive loading case.

If a **Safety** subnode has been added to a **Fasteners** node, the arrows are colored red for fasteners that do not satisfy the safety criterion.

As shell elements are defined on embedded faces in 3D, there is no geometric distinction between the physical top and bottom surfaces. When connecting two shells, for instance using contact or fasteners, you need to specify the physical surfaces in order to evaluate the connecting forces accurately. In the **Contact** node, the selection of the physical contact surface is done manually and one can use the **Shell Geometry** plot from results templates to determine the orientation of the shell. In Figure 9, the top surface is colored in red while the bottom surface is colored in blue.

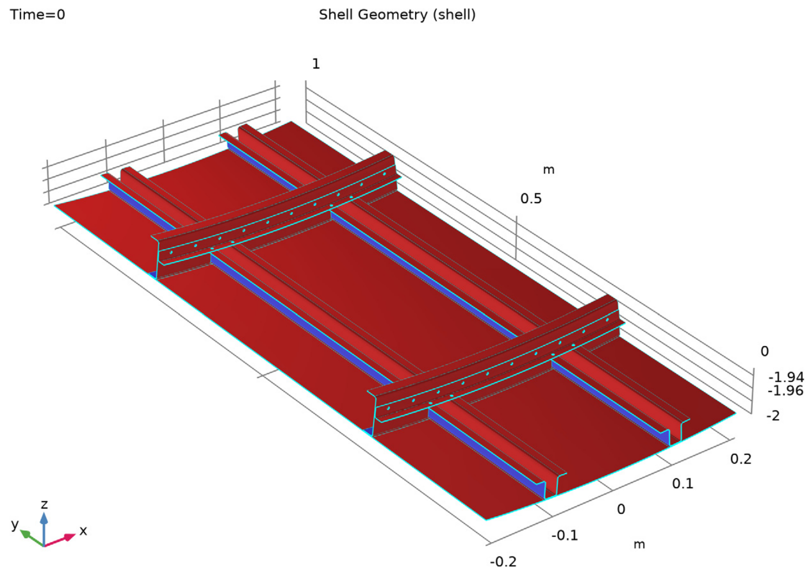

Figure 9: Shell geometry and orientation.

The **Fasteners** node includes an automatic detection of the connected location because it is assumed that there are no topology changes during the analysis. For a complex geometry with a large number of boundaries, this can induce an extra cost in terms of computational time. The automatic detection algorithm may also fail in cases where the same boundary is both a top and bottom surface for the same **Fasteners** node. In this example, the manual definition of the connected location is chosen as it anyway has to be done for the contact feature.

In addition, the **Fasteners** feature includes an automatic detection of the fasteners location based on the holes in the geometry. It automatically pairs source and destination holes that, within a tolerance, have the same radius, the same axis orientation, and centers on the same axis. The fasteners forces are then evaluated and applied to the adjacent edges of the holes. To visualize where the fasteners are active, you can use the **Connected Region Indicator** plot from the result templates. This plot is shown in Figure 9 for the current model.

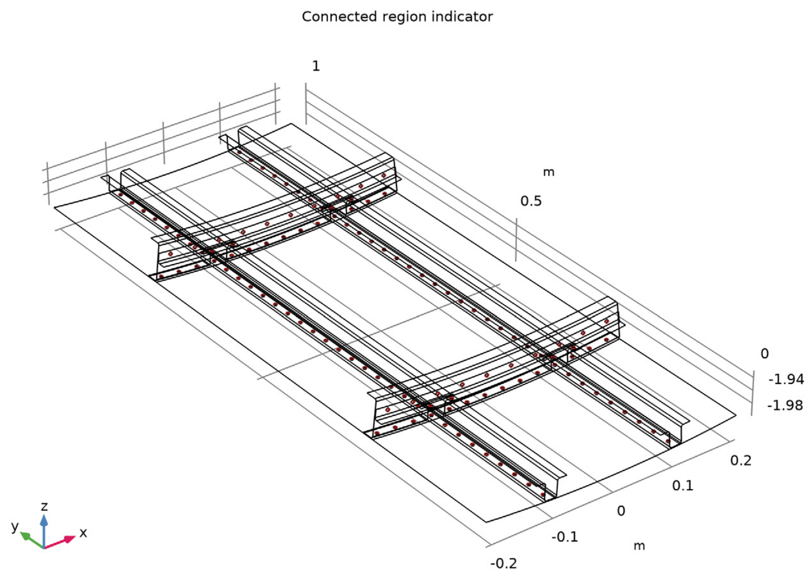

Figure 10: Active fasteners in the model.

Application Library path: `Structural_Mechanics_Module/Beams_and_Shells/fuselage_buckling`

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 **Structural Mechanics** > **Shell (shell)**.
3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

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

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

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

GLOBAL DEFINITIONS

Parameters 2

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `fuselage_buckling_parameters.txt`.

ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click **Windows** and choose **Add Material from Library**.

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Aluminum 6063-T83**.

3 Click the **Add to Component** button in the window toolbar.

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

SHELL (SHELL)

Thickness and Offset 1

1 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

2 In the d_0 text field, type `th2`.

Thickness and Offset 2

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

2 In the **Settings** window for **Thickness and Offset**, locate the **Boundary Selection** section.

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

4 Locate the **Thickness and Offset** section. In the d_0 text field, type `th1`.

Face Load 1

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

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

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

4 Locate the **Through-Thickness Location** section. From the list, choose **Bottom surface**.

5 Locate the **Force** section. From the **Load type** list, choose **Pressure**.

6 In the p text field, type `p0`.

Symmetry 1

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

2 In the **Settings** window for **Symmetry**, in the **Graphics** window toolbar, click ▾ next to **Select Edges**, then choose **Group by Continuous Tangent**.

3 Select Edges 1, 3, 5, 7, 9, 12, 13, 16, 17, 20, 22, 24, 26, 28, 30, 32, 34, 38, 40, 42, 45, and 1228–1248 only.

Symmetry 2

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

2 Click the **Go to Default View** button in the **Graphics** toolbar.

3 Select Edges 11, 144, 177, 188, 192, 193, 197, 488, 491, 502, 505, 528, 734, 763, 770, 781, 784, 787, 1078, 1082, 1083, 1095, and 1106 only.

DEFINITIONS

Integration 1 (intop1)

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

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

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

4 Select Edges 2, 179, 493, 772, and 1086 only.

Integration 2 (intop2)

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

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

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

4 Select Edges 135, 172, 190, 191, 196, 487, 490, 504, and 519 only.

Integration 3 (intop3)

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

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

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

4 Select Edges 1, 3, 5, 7, and 9 only.

Integration 4 (intop4)

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

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

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

4 Select Edges 16, 17, 22, 28, 30, 34, 40, and 45 only.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Aa	intop1(th1)+2*intop2(th2)	m²	Area in axial direction
Ah	intop3(th1)+2*intop4(th2)	m²	Area in hoop direction

SHELL (SHELL)

Symmetry 3

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

2 Select Edges 2, 135, 172, 179, 190, 191, 196, 487, 490, 493, 504, 519, 725, 758, 769, 772, 783, 786, 1077, 1080, 1081, 1086, and 1097 only.

3 In the **Settings** window for **Symmetry**, click to expand the **Normal Direction Condition** section.

4 From the list, choose **Prescribed force**.

5 In the F_n text field, type `k*Fh*Aa/Ah`.

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

7 In the **Show More Options** dialog, select **Physics** > **Advanced Physics Options** in the tree.

8 In the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

9 Click **OK**.

10 In the **Settings** window for **Symmetry**, click to expand the **Constraint Settings** section.

11 From the **Constraint** list, choose **Weak constraints**.

DEFINITIONS

Contact Pair: Skin/Stringer

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `Contact Pair: Skin/Stringer` in the **Label** text field.

3 In the **Graphics** window toolbar, click ▾ next to **Select Boundaries**, then choose **Group by Continuous Tangent**.

4 Select Boundaries 28–32 and 60–64 only.

5 Locate the **Source Boundaries** section. Click **Create Selection**.

6 In the **Create Selection** dialog, type `Skin_Stringer_Src` in the **Selection name** text field.

7 Click **OK**.

8 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

9 Click to select the **Activate Selection** toggle button.

10 Select Boundaries 35 and 66 only.

11 Click **Create Selection**.

12 In the **Create Selection** dialog, type `Skin_Stringer_Dst` in the **Selection name** text field.

13 Click **OK**.

Contact Pair: Skin/Frame

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `Contact Pair: Skin/Frame` in the **Label** text field.

3 Select Boundaries 2, 4, 39, 41, 71, and 73 only.

4 Locate the **Source Boundaries** section. Click **Create Selection**.

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

6 Click **OK**.

7 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

8 Click to select the **Activate Selection** toggle button.

9 Select Boundaries 6, 8, 44, 46, 82, and 83 only.

10 Click **Create Selection**.

11 In the **Create Selection** dialog, type `Skin_Frame_Dst` in the **Selection name** text field.

12 Click **OK**.

Contact Pair: Stringer/Clip

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `Contact Pair: Stringer/Clip` in the **Label** text field.

3 Select Boundaries 23, 25, 49, 51, 54, 56, 76, and 78 only.

4 Locate the **Source Boundaries** section. Click **Create Selection**.

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

6 Click **OK**.

7 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

8 Click to select the **Activate Selection** toggle button.

9 Select Boundaries 12 and 14 only.

10 Click **Create Selection**.

11 In the **Create Selection** dialog, type `Stringer_Clip_Dst` in the **Selection name** text field.

12 Click **OK**.

Contact Pair: Frame/Clip

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, type `Contact Pair: Frame/Clip` in the **Label** text field.

3 Select Boundaries 10 and 11 only.

4 Locate the **Source Boundaries** section. Click **Create Selection**.

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

6 Click **OK**.

7 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

8 Click to select the **Activate Selection** toggle button.

9 Select Boundaries 16 and 17 only.

10 Click **Create Selection**.

11 In the **Create Selection** dialog, type `Frame_Clip_Dst` in the **Selection name** text field.

12 Click **OK**.

SHELL (SHELL)

Contact 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Contact 1**.

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

3 From the **Contact surface, destination** list, choose **Bottom**.

4 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Automatic, soft**.

Contact 1a

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

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

3 Click **Add**.

4 In the **Add** dialog, select **Contact Pair: Skin/Frame (p2)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.

7 From the **Penalty factor control** list, choose **Automatic, soft**.

Fasteners 1

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

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

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

4 Locate the **Boundary Selection, Destination** section. From the **Selection** list, choose **Skin_Stringer_Dst**.

5 Locate the **Location** section. From the **Connected location** list, choose **Manual**.

6 Locate the **Fastener Stiffness** section. In the d_f text field, type `d_r`.

Safety 1

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

2 In the **Settings** window for **Safety**, click to expand the **Equation** section.

3 Locate the **Safety** section. In the $f_{\mathrm{n},}max$ text field, type `fnmax`.

4 In the α_n text field, type `2`.

5 In the $f_{\mathrm{s},}max$ text field, type `fsmax`.

6 In the α_s text field, type `2`.

Fasteners 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Fasteners**.
2 In the **Settings** window for **Fasteners**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Skin_Frame_Src**.
4 Locate the **Boundary Selection, Destination** section. From the **Selection** list, choose **Skin_Frame_Dst**.
5 Locate the **Location** section. From the **Connected location** list, choose **Manual**.
6 From the **Source** list, choose **Top surface**.
7 Locate the **Fastener Stiffness** section. In the d_f text field, type `d_r`.

Safety 1

1 In the **Physics** toolbar, click **Attributes** and choose **Safety**.
2 In the **Settings** window for **Safety**, locate the **Safety** section.
3 In the $f_{n,}max$ text field, type `fnmax`.
4 In the α_n text field, type `2`.
5 In the $f_{s,}max$ text field, type `fsmax`.
6 In the α_s text field, type `2`.

Fasteners 3

1 In the **Physics** toolbar, click **Boundaries** and choose **Fasteners**.
2 In the **Settings** window for **Fasteners**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Stringer_Clip_Src**.
4 Locate the **Boundary Selection, Destination** section. From the **Selection** list, choose **Stringer_Clip_Dst**.
5 Locate the **Location** section. From the **Connected location** list, choose **Manual**.
6 Locate the **Fastener Stiffness** section. In the d_f text field, type `d_r`.

Safety 1

1 In the **Physics** toolbar, click **Attributes** and choose **Safety**.
2 In the **Settings** window for **Safety**, locate the **Safety** section.
3 In the $f_{n,}max$ text field, type `fnmax`.
4 In the α_n text field, type `2`.
5 In the $f_{s,}max$ text field, type `fsmax`.
6 In the α_s text field, type `2`.

Fasteners 4

1 In the **Physics** toolbar, click **Boundaries** and choose **Fasteners**.
2 In the **Settings** window for **Fasteners**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Frame_Clip_Src**.
4 Locate the **Boundary Selection, Destination** section. From the **Selection** list, choose **Frame_Clip_Dst**.
5 Locate the **Location** section. From the **Connected location** list, choose **Manual**.
6 Locate the **Fastener Stiffness** section. In the d_f text field, type `d_r`.

Safety 1

1 In the **Physics** toolbar, click **Attributes** and choose **Safety**.
2 In the **Settings** window for **Safety**, locate the **Safety** section.
3 In the f_n,*max* text field, type `fnmax`.
4 In the α_n text field, type `2`.
5 In the f_s,*max* text field, type `fsmax`.
6 In the α_s text field, type `2`.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.
2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.

Size

1 In the **Model Builder** window, click **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. In the **Maximum element size** text field, type `1e-2`.
5 In the **Minimum element size** text field, type `3e-3`.
6 Click **Build All**.

DEFINITIONS

Cylindrical System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Cylindrical System**.

2 In the **Settings** window for **Cylindrical System**, locate the **Settings** section.

3 Find the **Origin** subsection. In the table, enter the following settings:

x (m)	y (m)	z (m)
0	0	R

4 Find the **Longitudinal axis** subsection. In the table, enter the following settings:

x	y	z
0	1	0

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

6 In the **Show More Options** dialog, select **General** > **Variable Utilities** in the tree.

7 In the tree, select the checkbox for the node **General** > **Variable Utilities**.

8 Click **OK**.

Vector Transform 1 (vectr1)

1 In the **Definitions** toolbar, click **Variable Utilities** and choose **Vector Transform**.

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

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

4 From the **Selection** list, choose **All boundaries**.

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

x	y	z
`shell.u`	`shell.v`	`shell.w`

6 Locate the **Output** section. From the **Coordinate system** list, choose **Cylindrical System 2 (sys2)**.

STUDY 1

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, 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
k (Biaxial loading parameter)	`-0.5 0 range(0.5,5e-2,1)`	

6 In the **Study** toolbar, click **Get Initial Value**.

RESULT TEMPLATES

1 In the **Home** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Shell Geometry (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Connected Region Indicator (shell)**.

3 Click the **Add Result Template** button in the window toolbar.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Displacement (shell)**.

3 Click the **Add Result Template** button in the window toolbar.

4 In the **Home** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Displacement (shell)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `vectr1.vr`.

4 From the **Unit** list, choose **mm**.

5 Select the **Description** checkbox. In the associated text field, type `Radial displacement`.

Radial Displacement (shell)

1 In the **Model Builder** window, under **Results** click **Displacement (shell)**.

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

DEFINITIONS

Global Variable Probe 1 (var1)

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

2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.

3 In the **Expression** text field, type `-shell.sym3.un`.

4 From the **Table and plot unit** list, choose **mm**.

5 Select the **Description** checkbox. In the associated text field, type `Axial displacement`.

STUDY 1

Step 1: Stationary

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

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

3 Select the **Plot** checkbox.

4 From the **Plot group** list, choose **Radial Displacement (shell)**.

5 From the **Update at** list, choose **Steps taken by solver**.

Solver Configurations

In the **Model Builder** window, expand the **Study 1 > Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1 > Solver Configurations > Solution 1 (sol1) > Dependent Variables 1** node, then click **Displacement Field (comp1.u)**.

2 In the **Settings** window for **Field**, locate the **Scaling** section.

3 In the **Scale** text field, type `1e-3`.

4 In the **Model Builder** window, click **Normal Displacement (comp1.shell.sym3.un)**.

5 In the **Settings** window for **State**, locate the **Scaling** section.

6 In the **Scale** text field, type `1e-3`.

Step 1: Stationary

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

Results

Surface 1

1 In the **Model Builder** window, expand the **Stress (shell)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Range** section.

3 Select the **Manual color range** checkbox.

4 Set the **Maximum** value to **4E8**.

5 In the **Stress (shell)** toolbar, click **Plot**.

Radial Displacement (shell)

1 In the **Model Builder** window, under **Results** click **Radial Displacement (shell)**.

2 In the **Settings** window for **3D Plot Group**, click **Plot First**.

3 In the **Radial Displacement (shell)** toolbar, click **Plot**.

4 Click **Plot Last**.

5 In the **Radial Displacement (shell)** toolbar, click **Plot**.

Axial Displacement

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 5**.

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

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Axial displacement vs biaxial loading parameter`.

5 Locate the **Legend** section. Clear the **Show legends** checkbox.

Result Templates

1 In the **Home** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Fastener Forces (shell)**.

4 Click the **Add Result Template** button in the window toolbar.

Result Templates

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Fastener Forces (Study 1) (shell)**.

3 Click the **Add Result Template** button in the window toolbar.

RESULTS

Fastener Forces (Study 1) (shell)

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

2 From the **Parameter selection (k)** list, choose **Last**.

3 In the **Fastener Forces (Study 1) (shell)** toolbar, click **Evaluate**.

RESULT TEMPLATES

1 Go to the **Result Templates** window.

2 In the tree, select **Study 1/Solution 1 (sol1)** > **Shell** > **Contact Forces (shell)**.

3 Click the **Add Result Template** button in the window toolbar.

4 In the **Home** toolbar, click **Result Templates** to close the **Result Templates** window.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 Click **Done**.

GLOBAL DEFINITIONS

Geometry

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

2 In the **Settings** window for **Parameters**, type `Geometry` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `fuselage_buckling_geom_parameters.txt`.

GEOMETRY 1

Work Plane 1 (wp1)

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

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

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

4 Click to expand the **Local Coordinate System** section. In the **yw-displacement** text field, type -R.

Work Plane 1 (wp1) > Plane Geometry

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

Work Plane 1 (wp1) > Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

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

xw (m)	yw (m)
-5/4*l_s	th1-th2+h_s
-l_s/2	th1-th2+h_s
-l_s/2	th1+th2/2
l_s/2	th1+th2/2
l_s/2	th1-th2+h_s
5/4*l_s	th1-th2+h_s

Work Plane 1 (wp1) > Fillet 1 (fil1)

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

2 On the object **pol1**, select Points 2–5 only.

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

4 In the **Radius** text field, type fillet.

5 Click **Build Selected**.

Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (m)
L

4 Select the **Reverse direction** checkbox.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog, type `Stringer` in the **Name** text field.

7 Click **OK**.

8 In the **Settings** window for **Extrude**, click **Build Selected**.

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

Work Plane 2 (wp2)

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

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

3 From the **Plane type** list, choose **Face parallel**.

4 On the object **ext1**, select Boundary 9 only.

5 Locate the **Local Coordinate System** section. From the **Origin** list, choose **Vertex projection**.

6 On the object **ext1**, select Point 1 only.

Work Plane 2 (wp2) > Plane Geometry

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

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

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

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

3 In the **Width** text field, type `5/2*l_s`.

4 In the **Height** text field, type `l_clip`.

5 Locate the **Position** section. In the **yw** text field, type `L/4-l_f/2-l_clip`.

Work Plane 2 (wp2) > Move 1 (mov1)

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

2 Select the object **r1** only.

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

4 In the **yw** text field, type `L/2`.

5 Locate the **Input** section. Select the **Keep input objects** checkbox.

6 Click **Build Selected**.

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

Partition Objects 1 (par1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Booleans and Partitions** > **Partition Objects**.
2 Select the object **ext1** only.
3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
4 Click to select the **Activate Selection** toggle button for **Tool objects**.
5 Select the object **wp2** only.

Work Plane 3 (wp3)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **yz-plane**.
4 Locate the **Local Coordinate System** section. In the **yw-displacement** text field, type `-R`.

Work Plane 3 (wp3) > Plane Geometry

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

Work Plane 3 (wp3) > Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Coordinates** section. In the table, enter the following settings:

xw (m)	**yw (m)**
`L/4+l_f/2`	`th1+th2/2`
`L/4-l_f/2`	`th1+th2/2`
`L/4-l_f/2`	`th1-th2/2+h_f`
`L/4+l_f/2`	`th1-th2/2+h_f`

Work Plane 3 (wp3) > Polygon 2 (pol2)

1 In the **Work Plane** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.

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

xw (m)	yw (m)
L/4-l_f/2-l_clip	th1-th2+h_s+th2
L/4-l_f/2-th2	th1-th2+h_s+th2
L/4-l_f/2-th2	th1-th2+h_s+th2+l_clip

Work Plane 3 (wp3) > Fillet 1 (fil1)

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

2 On the object **pol1**, select Points 2 and 3 only.

3 On the object **pol2**, select Point 2 only.

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

5 In the **Radius** text field, type `fillet`.

Work Plane 3 (wp3) > Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **yw** text field, type `th1/2`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **xw** text field, type `L`.

7 In the **yw** text field, type `th1/2`.

Work Plane 3 (wp3) > Line Segment 2 (ls2)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **xw** text field, type `L/4-l_f/2`.

5 In the **yw** text field, type `th1/2`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **xw** text field, type `L/4+l_f/2`.

8 In the **yw** text field, type `th1/2`.

Work Plane 3 (wp3) > Move 1 (mov1)

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

2 Select the objects **fil1(1)**, **fil1(2)**, and **ls2** only.

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

4 In the **xw** text field, type `L/2`.

5 Locate the **Input** section. Select the **Keep input objects** checkbox.

Work Plane 3 (wp3) > Union 1 (uni1)

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

2 Select the objects **ls1**, **ls2**, and **mov1(3)** only.

Work Plane 3 (wp3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Work Plane 3 (wp3)**.

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

3 Clear the **Unite objects** checkbox.

Revolve 1 (rev1)

1 In the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **General** section.

3 In the list, select **wp3.uni1**.

4 Click the **Remove from Selection** button for **Input objects**.

5 Locate the **Revolution Angles** section. Click the **Angles** button.

6 In the **Start angle** text field, type `-alpha`.

7 In the **End angle** text field, type `alpha`.

8 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

9 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

10 In the **New Cumulative Selection** dialog, type `Frame` in the **Name** text field.

11 Click **OK**.

Revolve 2 (rev2)

1 In the **Geometry** toolbar, click **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **General** section.

3 From the **Input object handling** list, choose **Keep**.

4 Locate the **Revolution Angles** section. Click the **Angles** button.

5 In the **Start angle** text field, type `-alpha`.

6 In the **End angle** text field, type `alpha`.

7 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

8 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

9 In the **New Cumulative Selection** dialog, type `Skin` in the **Name** text field.

10 Click **OK**.

11 In the **Settings** window for **Revolve**, click **Build Selected**.

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

Revolve 3 (rev3)

1 Right-click **Revolve 2 (rev2)** and choose **Duplicate**.

2 In the **Settings** window for **Revolve**, locate the **General** section.

3 From the **Input object handling** list, choose **Unite with**.

4 Locate the **Revolution Angles** section. In the **Start angle** text field, type `-alpha/20`.

5 In the **End angle** text field, type `alpha/20`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Stringer**.

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `d_r/2`.

4 In the **Height** text field, type `th1+th2`.

5 Locate the **Position** section. In the **y** text field, type `L/4`.

6 In the **z** text field, type `-R`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog, type `Rivet` in the **Name** text field.

9 Click **OK**.

Move 1 (mov1)

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

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Rivet**.

4 Locate the **Displacement** section. In the **y** text field, type `range(-8,1,8)*L/2/17`.

Ball Selection 1 (ballsel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Ball Selection**.
2 In the **Settings** window for **Ball Selection**, locate the **Geometric Entity Level** section.
3 From the **Level** list, choose **Object**.
4 Locate the **Ball Center** section. In the **y** text field, type `L/4`.
5 In the **z** text field, type `-R`.
6 Locate the **Ball Radius** section. In the **Radius** text field, type `2*d_r`.
7 Click **Build Selected**.
8 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside ball**.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
2 In the **Settings** window for **Rotate**, locate the **Input** section.
3 From the **Input objects** list, choose **Ball Selection 1**.
4 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.
5 In the **Angle** text field, type `range(-3,1,3)/7*alpha`.
6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Rivet**.

Cylinder 2 (cyl2)

1 In the **Geometry** toolbar, click **Cylinder**.
2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `d_r/2`.
4 In the **Height** text field, type `th1+th2`.
5 Locate the **Position** section. In the **x** text field, type `l_s*7/8`.
6 In the **y** text field, type `L/4-l_f/2-l_clip/2-th2`.
7 In the **z** text field, type `-R+th1/2-th2+h_s`.
8 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Rivet**.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.
2 Select the object **cyl2** only.

3 In the **Settings** window for **Mirror**, locate the **Normal Vector to Plane of Reflection** section.

4 In the **x** text field, type `1`.

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

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Rivet**.

7 Locate the **Input** section. Select the **Keep input objects** checkbox.

Cylinder 3 (cyl3)

1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `d_r/2`.

4 In the **Height** text field, type `th1+th2`.

5 Locate the **Position** section. In the **y** text field, type `L/4-l_f/2-th1/2-th2`.

6 In the **z** text field, type `-R+th1+h_s+th2+l_clip/2`.

7 Locate the **Axis** section. From the **Axis type** list, choose **y-axis**.

Rotate 2 (rot2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **cyl3** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 From the **Axis type** list, choose **y-axis**.

5 In the **Angle** text field, type `range(-2,1,2)*alpha/5`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Rivet**.

Move 2 (mov2)

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

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Rivet**.

4 Select the **Keep input objects** checkbox.

5 Locate the **Displacement** section. In the **y** text field, type `L/2`.

Work Plane 4 (wp4)

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

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

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

4 In the **y-coordinate** text field, type `L/4-l_f/2-l_clip`.

5 Locate the **Local Coordinate System** section. In the **yw-displacement** text field, type `-R`.

Work Plane 4 (wp4) > Plane Geometry

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

Work Plane 4 (wp4) > Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `l_c`.

4 In the **Height** text field, type `h_c`.

5 Locate the **Position** section. In the **xw** text field, type `-l_c/2`.

6 In the **yw** text field, type `th1`.

Work Plane 4 (wp4) > Rectangle 2 (r2)

1 Right-click **Component 1 (comp1)** > **Geometry 1** > **Work Plane 4 (wp4)** > **Plane Geometry** > **Rectangle 1 (r1)** and choose **Duplicate**.

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

3 In the **Width** text field, type `l_c/4`.

4 In the **Height** text field, type `h_c-2*fillet`.

Work Plane 4 (wp4) > Mirror 1 (mir1)

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

2 Select the object **r2** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

Work Plane 4 (wp4) > Difference 1 (dif1)

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

2 Select the object **r1** only.

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

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

5 Select the objects **mir1** and **r2** only.

Work Plane 4 (wp4) > Fillet 1 (fil1)

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

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

3 Click to select the **Activate Selection** toggle button for **Vertices to fillet**.

4 On the object **dif1**, select Points 1, 2, 4, and 6–8 only.

5 Locate the **Radius** section. In the **Radius** text field, type `fillet`.

6 Click **Build Selected**.

Extrude 2 (ext2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 4 (wp4)** and choose **Extrude**.

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

3 In the table, enter the following settings:

Distances (m)
`l_f+l_clip`

4 Select the **Reverse direction** checkbox.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog, type `Stringer cut` in the **Name** text field.

7 Click **OK**.

Move 3 (mov3)

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

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Stringer cut**.

4 Select the **Keep input objects** checkbox.

5 Locate the **Displacement** section. In the **y** text field, type `L/2`.

Rotate 3 (rot3)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Stringer**.

4 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.

5 In the **Angle** text field, type `range(-1/2,1,1/2)*alpha`.

Rotate 4 (rot4)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Rivet**.

4 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.

5 In the **Angle** text field, type `range(-1/2,1,1/2)*alpha`.

Rotate 5 (rot5)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

3 From the **Input objects** list, choose **Stringer cut**.

4 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.

5 In the **Angle** text field, type `range(-1/2,1,1/2)*alpha`.

Difference 1 (dif1)

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

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

3 From the **Objects to add** list, choose **Frame**.

4 From the **Objects to subtract** list, choose **Stringer cut**.

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 **Object**.

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

5 In the **Add** dialog, in the **Selections to add** list, choose **Stringer**, **Frame**, and **Skin**.

6 Click **OK**.

Difference 2 (dif2)

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

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

3 From the **Objects to add** list, choose **Union Selection 1**.

4 From the **Objects to subtract** list, choose **Rivet**.

5 Click **Build Selected**.