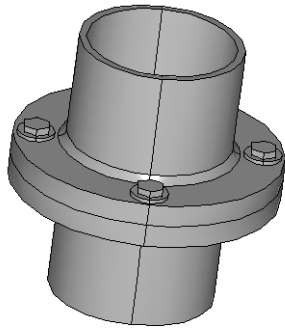# Tube Connection

A tube connection consisting of a flange with four prestressed bolts is subjected to tensile forces. A sketch of the connection is shown below.

Model Definition

The tube is made of steel and has an outer diameter of 220 millimeters and an inner diameter of 200 millimeters. The flange has a diameter of 360 millimeters and it is 30 millimeters thick. The connection consists of four prestressed M24 bolts. The bolts are prestressed to 80% of the yield strength. The tensile force in the tube varies from 0 to 500 kN.

To compute the influence of the tensile force on the stress level in the bolt, the model includes a parametric analysis. Because of symmetry in both load and geometry, you only need to analyze one eighth of one of the flanges. The geometry has been created in the CAD software SolidWorks and is available both as an IGES file and as a SolidWorks file. The model exists in two different versions depending how the geometry is imported. The IGES version (`tube_connection_igs.mph`) is created from the IGES file through the IGES file import. Read more about IGES import in the section Importing 3D IGES Files in the *CAD Import Module User's guide*. The SolidWorks version (`tube_connection_sw.mph`) is created from the SolidWorks live connection. Read more about the SolidWorks interface in the section "SolidWorks Live Connection" on page 30 in the *CAD Import Module User's guide*.

A contact pair between the bottom surface of the flange and the top surface of an additional fixed solid models the contact. To represent the bolt, the model uses a distributed linear boundary condition on a surface of the size of the washer.

Note: This model requires both the Structural Mechanics Module and the CAD Import Module.

Modeling in COMSOL Multiphysics

Two application modes for 3D solid stress-strain analysis are used to solve the problem. One application mode models the initial prestress problem of calculating the deformation from the prestressed bolts. The other is used for solving the parametric problem including the tensile force, using the solution from the prestress application mode to set up the correct boundary conditions.
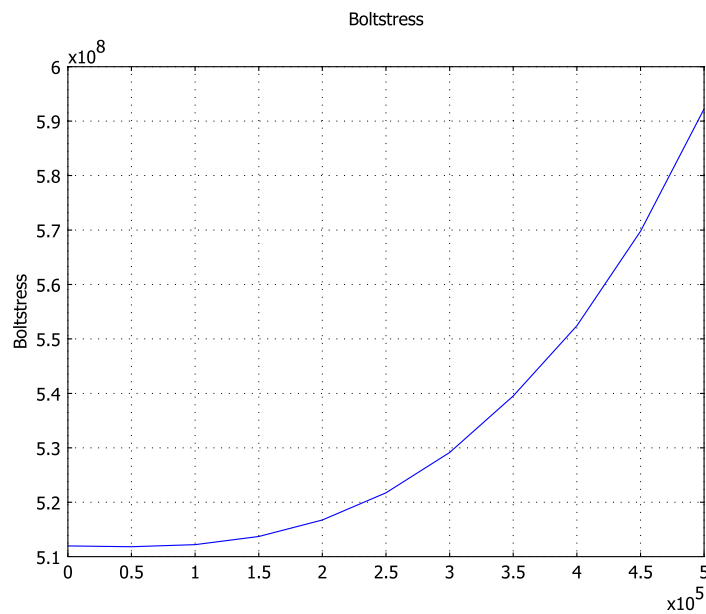
Results and Discussion

Figure 7-8: The bolt stress as a function of the tensile force.

For a tensile force below 250 kN the prestressed connection works well, the bolt stress increasing less than 2%. In the region 250 kN–500 kN the bolt stress increases by more than 13% due to the nonlinear effects in the connection.

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/tube_connection_igs`

Model Library path: `Structural_Mechanics_Module/Contact_and_Friction/tube_connection_sw`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Click the **New** button or start COMSOL Multiphysics to open the **Model Navigator**.

2 Select **3D** in the **Space dimension** list.

3 In the list of application modes, open the **Structural Mechanics Module** folder and then **Solid, Stress-Strain**. Select the **Static analysis**.

4 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, select **Constants**.

2 Enter the following constant names, expressions, and (optionally) descriptions. When finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
Abolt	pi/4*24e-3^2[m^2]	Bolt area
Atube	pi/4*(0.22^2-0.2^2)[m^2]	Tube area
Force	0[N]	Tensile force in the tube
Sboltini	800*0.8*0.8*1e6[N/m^2]	Prestress in the bolt
Awash	pi/4*(0.05^2-0.025^2)[m^2]	Washer area
thickness	0.03[m]	Flange thickness

| NAME	EXPRESSION	DESCRIPTION
Emod	2e11[N/m^2]	Modulus of elasticity of the bolt
Kbolt	Emod*Abolt/thickness/Awash	Distributed stiffness of the bolt (N/m^3)
Fbolt	Sboltini*Abolt/Awash	Distributed prestress from the bolt (N/m^2)

GEOMETRY MODELING

There are two different ways to create the geometry:

- From an IGES file, through the IGES file import.
- From a SolidWorks file, through the SolidWorks live connection.

IGES File Import

1 On the **File** menu, select **Import>CAD Data From File** to open the **Import CAD Data From File** dialog box.

2 Open the IGES file `tube_connection.igs` located in the `models\Structural_Mechanics_Module\Contact_and_Friction` folder in the COMSOL installation directory.

3 Click **Import**.
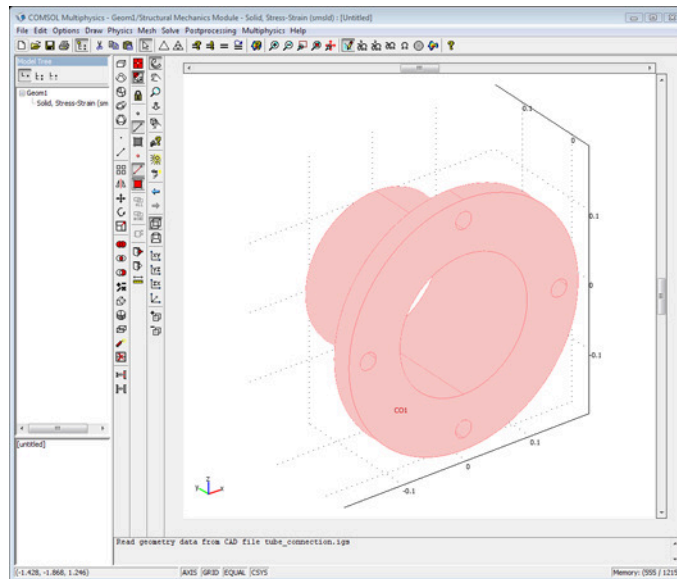
Due to symmetry, you can reduce the geometry to one eighth of the full geometry. To do so, create a 45-degree wedge and then intersect it with the imported geometry:

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click **OK** in the **Work-Plane Settings** dialog box as you will use the default settings.

3 Click the **Zoom Extents** toolbar button.

4 Draw a rectangle with opposite corners at $(0, -0.1)$ and $(0.2, 0.2)$.

5 Select **Revolve** from the **Draw** menu and enter 45 as α**2** in the **Revolve** dialog box. Click **OK**.

6 Press Ctrl+A to select all objects and then click the **Intersection** button on the Draw toolbar to create an eighth.

Model a washer by creating a boundary around the hole. The distributed forces from the bolt are applied on this boundary.

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click the **Add** button, select the **z-x** option in the **Plane** area and click **OK**.

3 Set grid settings in Geom 3 according to the following table:

GRID	
x spacing	5e-3
Extra x	
y spacing	5e-3
Extra y	

4 Click the **Zoom Extents** button on the Main toolbar to see the geometry.

5 Press the Shift key and click the **Ellipse/Circle (Centered)** button on the Draw toolbar to create a centered circle with radius 0.025 and center at $(0, 0.15)$.

6 Draw another centered circle with radius 0.0125 and center at $(0, 0.15)$.

7 Select both circles and click the **Difference** button to create a washer.

8 Draw a rectangle with opposite corners at $(0, 0.12)$ and $(0.03, 0.18)$.

9 Click the **Create Composite Object** button on the Draw toolbar, type CO1-R1 in the **Set formula** edit field, and click **OK** to close the dialog box.

10 Select **Embed** from the **Draw** menu, then click **OK**.
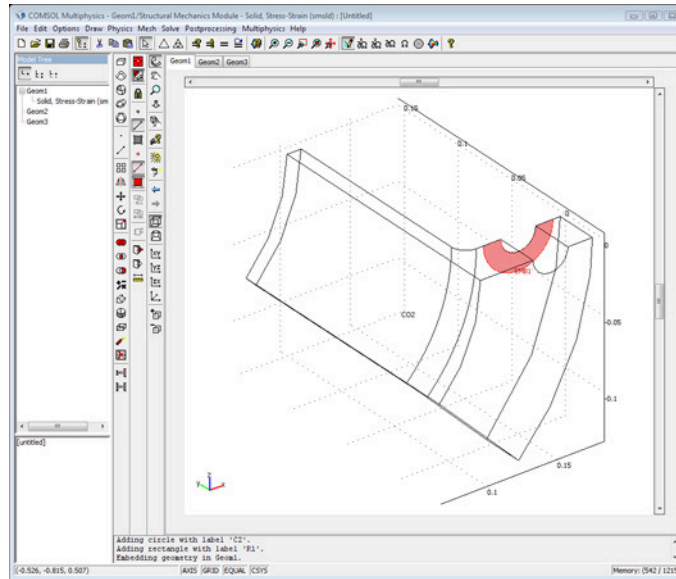
Create an additional solid below the bottom flange (this solid is needed for the contact modeling).

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click the **Add** button, select the **x-y** option in the **Plane** area and click **OK**.

3 Set grid settings in Geom4 according to the following table:

GRID	
x spacing	0.01
Extra x	
y spacing	0.01
Extra y	

4 Draw a rectangle with opposite corners at (0.09, −0.04) and (0.19, −0.03). To do so, press Shift and click the **Rectangle/Square** button on the Draw toolbar. In the **Rectangle** dialog box, enter (0.09, −0.04) as the coordinates for lower-left corner and a width and height of 0.10 and 0.01, respectively.

5 Select **Revolve** from the **Draw** menu and enter 45 as α**2** in the **Revolve** dialog box. Click **OK**.

You need to work in assembly mode to model the contact between the two initially connecting solids. The embedded washer surface must be a part of a solid to be included in the assembly between solids.

1 Select the upper solid part, CO2, and the embedded surface, EMB1.

2 Click the **Coerce to Solid** button on the Draw toolbar to make the washer a part of the solid.

3 Select **Use Assembly** from the **Draw** menu to leave draw mode in assembly mode.

SolidWorks Live Connection
Another way to create your geometry is to import it directly from SolidWorks. To follow these instructions you need to have SolidWorks installed on your computer.

1 Start SolidWorks and open the SolidWorks file `tube_connection.SLDPRT` located in the `models/Structural_Mechanics_Module/Contact_and_Friction` in the COMSOL installation directory.

2 Switch to COMSOL and select **File>SolidWorks Connection>Initialize**.

Create an additional solid below the bottom flange, this solid is needed for the contact modeling.

1 Select **Work-Plane Settings** from the **Draw** menu.

2 Click **OK** in the **Work-Plane Settings** dialog box as you will use the default settings.

3 Set grid settings in Geom2 according to the following table:

GRID	
x spacing	0.01
Extra x	
y spacing	0.01
Extra y	

4 Draw a rectangle with opposite corners at $(0.09, -0.04)$ and $(0.19, -0.03)$.

5 Select **Revolve** from the **Draw** menu and enter 45 as $\alpha2$ in the **Revolve** dialog box. Click **OK**.

You need to work in assembly mode to model the contact between the two initially connecting solids.

6 Select **Use Assembly** from the **Draw** menu to leave draw mode in assembly mode.

Boundary Settings

In the first application mode you compute the initial deformation from the prestressed bolt. You then use a second Solid Stress-Strain application mode for the parametric analysis where the tensile force in the tube is increased. You need the solution from the first application mode to set up the boundary conditions for the parametric analysis.

1 Select **Model Navigator** from the **Multiphysics** menu to open the **Model Navigator**.

2 Select **Structural Mechanics Module>Solid, Stress-Strain** from the application mode tree and click the **Add** button to add a second solid stress-strain application mode to your 3D geometry. Click **OK** to close the **Model Navigator**.

To model the contact, use contact pairs. Start by creating two contact pairs, one for each application mode. Make the weaker boundary the slave and the stiffer one the master boundary.

3 Select **Physics>Contact Pairs** to open the **Contact Pairs** dialog box.

4 Click the **New** button to create a contact pair.

5 Select Boundary 4 in the **Master boundaries** list.

6 Select the check box in front of **4** in the **Master boundaries** list.

7 Select Boundary 9 in the **Slave boundaries** list.

8 Select the check box in front of **9** in the **Slave boundaries** list.

9 Click the **New** button to create an additional contact pair.

10 Select the same boundaries as master and slave boundaries as you made for the first pair.

11 Click **OK** to close the dialog box.

12 Select **1 Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

13 Select **Physics>Boundary Settings** and clear the **Active pair** check box for **Pair 2**.

14 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the **Multiphysics** menu.

15 Select **Physics>Boundary Settings** and clear the **Active pair** check box for **Pair 1**.

16 Select **1 Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu.

17 Select **Physics>Boundary Settings** and specify the boundary and contact-pair settings according to the following tables:

	BOUNDARIES 8, 14, 18	BOUNDARY 15	
Page	Constraint	Load	
	Symmetry plane	F_y	-Fbolt

	PAIR 1		PAIR 2
Page	Contact		
	P_n	E_smsld/h*min(1e-3*5^auglagiter,100)	
Page	Contact, Initial		
	T_n	1e7	

The `auglagiter` variable is the iteration number in the augmented Lagrange solver. It makes the penalty parameter soft at the beginning (to help the solver get started) and then gradually makes it stiffer (to speed up convergence).

18 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the **Multiphysics** menu.

19 Select **Physics>Boundary Settings** and specify the boundary and contact pair settings according to the following tables:

	BOUNDARIES 8, 14, 18	BOUNDARY 10		BOUNDARY 15	
Page	Constraint	Load		Load	
	Symmetry plane	F_y	Force/Atube	F_y	-Fbolt-Kbolt*(v2-v)

	PAIR 2		PAIR 1
Page	Contact		
	P_n	E_smsld2/h*min(1e-3*5^auglagiter,100)	
Page	Contact, Initial		
	T_n	Tn_cp1_smsld	

20 For enabling plotting of the bolt stress as a function of the applied tensile force an integration coupling variable is created, representing the integral of the force on the washer boundary.

21 Select **Options>Integration Coupling Variables>Boundary Variables** to open the **Boundary Integration Variables** dialog box.

22 Select Boundary 15 in the **Boundary selection** list.

23 In the **Name** column type `Boltstress`.

24 In the **Expression** column type `(Fbolt+Kbolt*(v2-v))/(Abolt/2)`.

25 Leave **Global destination** enabled and click **OK** to close the dialog box.

Subdomain Settings

The material properties will be defined using the materials library and are identical for both application modes. Specify the extra solid (added for the contact modeling) to be fixed.

1 Select **1 Geom1: Solid, Stress-Strain (smsld)** from the **Multiphysics** menu to set the subdomain parameters for the first application mode.

2 Select **Physics>Subdomain Settings**.

3 Select Subdomain 2 from the subdomain list. Click the **Load** button on the **Material** page to open the **Materials/Coefficients Library** dialog box.

4 Select **Structural Steel** from the **Basic Material Properties** folder and click **OK** to close the **Materials/Coefficients Library** dialog box.

5 Select Subdomain 1 from the subdomain list. Click the **Constraint** page and select **Fixed** from the **Constraint condition** list.

6 Click **OK** to close the **Subdomain Settings** dialog box.

Do the same for the other application mode:

7 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the **Multiphysics** menu to set the subdomain parameters for the second application mode.

8 Select **Physics>Subdomain Settings**.

9 Select Subdomain 2 from the subdomain list.

10 Select **Structural Steel** from the **Library material** list.

11 Select Subdomain 1 from the subdomain list. Click the **Constraint** page and select **Fixed** from the **Constraint condition** list.

12 Click **OK** to close the **Subdomain Settings** dialog box.

MESH GENERATION

The default mesh settings are used.

COMPUTING THE SOLUTION

1 Select **Solver Parameters** from the **Solve** menu.

2 Click the **Stationary** tab and find the **Augmented Lagrangian solver** area.

3 In the **Tolerance** edit field type 1e-5.

4 Click **OK** to close the **Solver Parameters** dialog box

First solve the initial problem (no tensile force, only prestress) to determine the initial displacements.

1 Select **Solver Manager** from the **Solve** menu to open the **Solver Manager** dialog box.

2 On the **Solve For** page select **Geom1: Solid, Stress-Strain (smsld)** from the **Solve for variables** list, then click **OK** to close the dialog box.

3 Click the **Solve** button on the Main toolbar to compute the solution.

You have now solved the prestress problem without any external load.

POSTPROCESSING AND VISUALIZATION

Visualize the displacement in the *y* direction and the deformation as follows:

1 Select **Plot Parameters** from the **Postprocessing** menu.

2 On the **General** page select **Frame (ref)** from the **Frame** list.

3 Clear **Slice** plot, then select **Deformed shape** plot and **Subdomain** plot.

4 Click the **Subdomain** tab.

5 From the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>y displacement**.

6 Click **OK**.
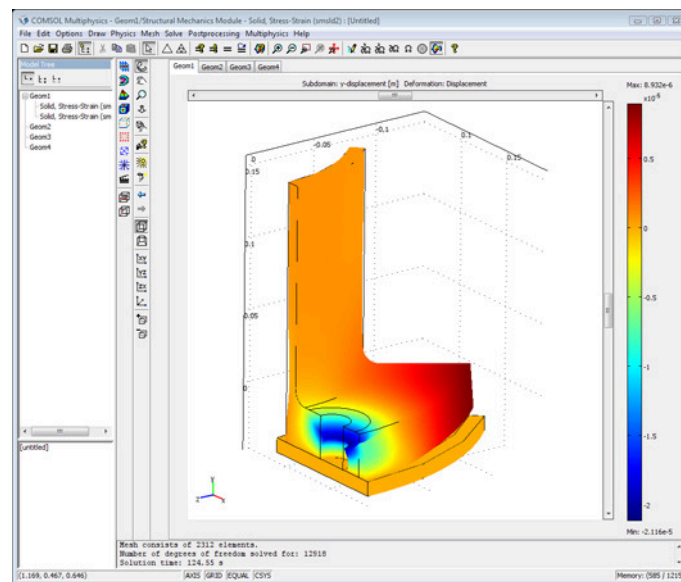
Now consider the externally applied tensile force.

COMPUTING THE SOLUTION

Switch to the parametric solver and solve for the displacements in the second application mode using the solution from the first application mode in the boundary condition and as initial condition.

1 Select **Solver Parameters** from the **Solve** menu to open the **Solver Parameters** dialog box.

2 Select **Parametric** in the **Analysis** list.

3 On the **General** page type Force in the **Parameter name** edit field and type 0:50e3:500e3 in the **Parameter values** edit field. Click **OK**.

4 Choose **Solve>Solver Manager**, then click the **Initial Value** tab.

5 Click the **Initial value expression evaluated using stored solution** option button in the **Initial value** area.

6 Click the **Stored solution** option button in the **Values of variables not solved for and linearization point** area.

7 Click the **Store Solution** button to save the initial solution.

8 Select **Geom1: Solid, Stress-Strain (smsld2)** from the **Solve for variables** list on the **Solve For** page, then click **OK** to close the dialog box.

Set the initial values in the second application mode to the result from the first application mode.

1 Select **2 Geom1: Solid, Stress-Strain (smsld2)** from the multiphysics menu.

2 From the **Physics** menu, choose **Subdomain Settings**.

**3** Click the **Init** tab and specify the initial values according to the following table.

SUBDOMAIN	1, 2	
Page	Init	
	$u2(t_0)$	u
	$v2(t_0)$	v
	$w2(t_0)$	w

4 Click **OK**.

5 Click the **Solve** button on the Main toolbar to start the parametric solver.

POSTPROCESSING AND VISUALIZATION

To visualize the result of the parametric analysis, you can use a plot of the von Mises stress combined with a deformation plot:

1 From the **Postprocessing** menu, select **Plot Parameters**.

2 Select **5e5** in the **Parameter value** list on the **General** page.

3 Click the **Deform** tab.

4 In the **Deformation data** area, select **Solid, Stress-Strain (smsld2)>Displacement** in the **Predefined quantities** list.

5 Click the **Subdomain** tab.

6 Select **Solid, Stress-Strain (smsld2)>von Mises stress** from the **Predefined quantities** list.

7 Click **OK**.
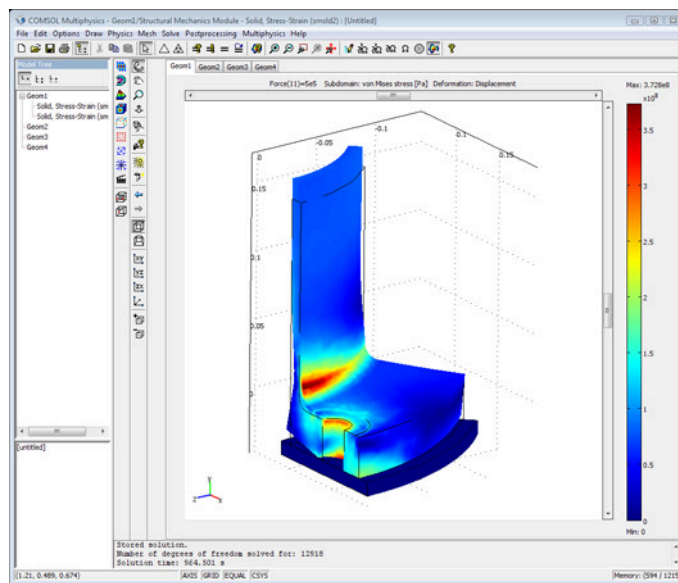
To plot the stress in the bolt as a function of the external tensile force, do the following steps:

1 From the **Postprocessing** menu, choose **Global Variables Plot Parameters**.

2 Make sure that all parameter values are selected in the **Solutions to use** list.

3 Enter `Boltstress` in the **Expression** edit field and click the **>** button.

4 Select **Boltstress** from the **Quantities to plot** list.

5 Click **OK** to create the plot.

Boltstress