Model created in COMSOL Multiphysics 6.4

Stiffness Analysis of a Communication Mast's Diagonal Mounting

Introduction

Communication masts usually have a framework with a bolted triangular lattice design as illustrated in Figure 1. The diagonals of the framework are assembled from several parts and welded together.

When operating under a given wind load at a specific location, the antenna's total rotation angle should stay below a certain limit to ensure uninterrupted communications. For the type of mast used in this example, the engineers have determined that its torsional stiffness is too low, and this effect is due to the geometry of the diagonal mountings. The goal is to increase the stiffness of such a diagonal mounting by first analyzing a parameterized geometry followed by an update of the geometry and a new analysis.

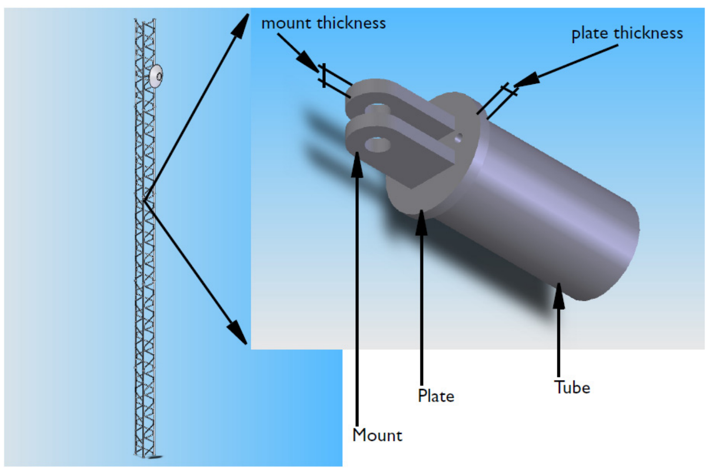


Figure 1: Mounting details of a mast diagonal.

Model Definition

The model geometry includes only a short section of the diagonal tubing together with the other parts of the mounting as illustrated in Figure 1. Although a symmetry exists in both the geometry and load for this problem, the entire assembly is modeled for illustrative purposes.

After obtaining the original stiffness of the diagonal mounting, it is assumed that the geometry has been updated to improve the stiffness. Originally 10 mm, the plate thickness and mount thickness (see Figure 1) have been changed to 12 mm and 15 mm, respectively.

MATERIAL PROPERTIES

The material is a structural steel, having a Young's modulus of 200 GPa and a Poisson's ratio of 0.33.

BOUNDARY CONDITIONS

Figure 2 shows the boundaries with an applied load and constrained displacements. Assume that the diagonal is loaded in tension by a force, F = 30 kN, which is transferred through the bolt to the mounting.

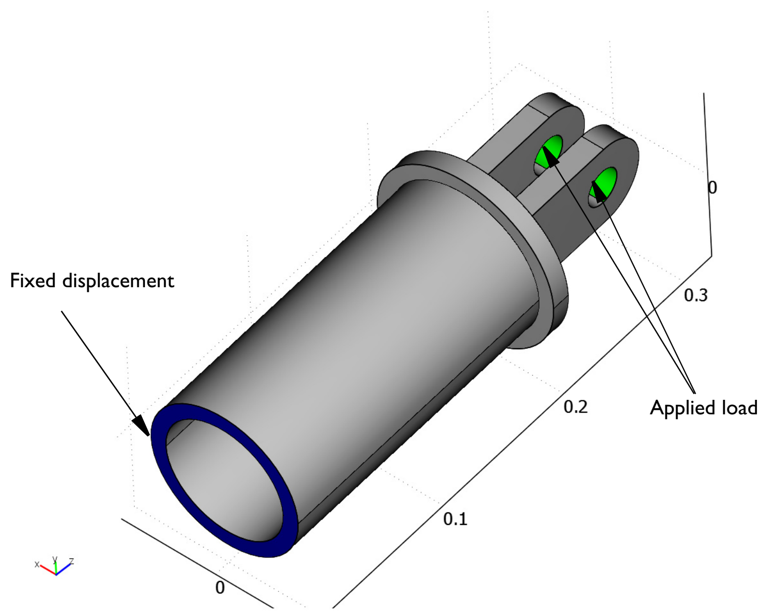


Figure 2: Boundaries with constrained displacements and applied loads.

Neglect contact conditions between the bolt and the mounting hole, and also neglect the constraint imposed on the mount by the bolt. Assume that the bolt fills out the entire hole volume. The load is distributed on the appropriate halves of the hole surfaces by applying a pressure, p, according to

$$p = \frac{F}{2 \cdot A_{\mathrm{mh}}^{\mathrm{xy}}} \cdot \frac{3}{2}\left(1 - \left(\frac{y}{r_{\mathrm{mh}}}\right)^2\right)$$

$$A_{\mathrm{mh}}^{\mathrm{xy}} = 2 \cdot r_{\mathrm{mh}} t_{\mathrm{m}}$$

where r_{mh} is the mount hole radius, t_{m} is the thickness of the mount, y is the y-coordinate and $A_{\mathrm{mh}}^{\mathrm{xy}}$ is the mount hole cross section area projected on the xy-plane.

In the current analysis, the purpose is to increase the stiffness of the assembly. Since the load is transferred through the mount holes, the displacement of the holes under the given external load is sought. If the average z-displacement of the middle plane of the mount holes is denoted by δ_{mh}, the stiffness of the assembly is given by

$$S = \frac{F}{\delta_{\mathrm{mh}}}$$

In a bar with a constant cross section, the relation between the applied force and resulting displacement is given by

$$\frac{F}{A} = E\frac{\delta}{L}$$

where E is the Young's modulus, δ is the displacement, A is the cross section area, and L is the total length. This relation can be rearranged to

$$\frac{F}{\delta} = E\frac{A}{L}$$

It is an expression for the effective stiffness of a bar with a constant cross section. Here, it is considered as the ideal stiffness, S_{I}, against which the actual stiffness of the diagonal mounting is compared. The ideal stiffness can thus be expressed as

$$S_{\mathrm{I}} = E\frac{A_{\mathrm{t}}}{L_{\mathrm{t}}}$$

$$L_{\mathrm{t}} = h_{\mathrm{t}} + t_{\mathrm{p}} + o_{\mathrm{mh}}$$

where L_{t} is the equivalent tube length if the entire assembly was made of a tube only, h_{t} is the actual tube height in the present assembly, t_{p} is the plate thickness and o_{mh} is the mount hole center offset, measured from the plate surface. Note that the tube height and the equivalent tube length are both measured along the same dimension of the tube. The difference is the fact that the tube height is used to relate the tube used in the mount assembly and the tube length is used to define the tube in an assembly consisting of a tube only.

For evaluation purposes, the ratio between the real stiffness of the assembly with a mount and the ideal stiffness is introduced as

$$S_\mathrm{R} = \frac{S}{S_\mathrm{I}}$$

If this value equals one, the stiffness of the assembly with a mount is exactly the same as the stiffens of a single tube.

Results and Discussion

In the original geometry, where both the plate thickness and the mount thickness are 10 mm, the stiffness ratio is evaluated to 0.38. For the setup with the increased plate thickness to 12 mm and the increased mount thickness to 15 mm, the stiffness ratio is evaluated to 0.53. Figure 3 shows the z-component of the displacement for the geometry with the thickened components.

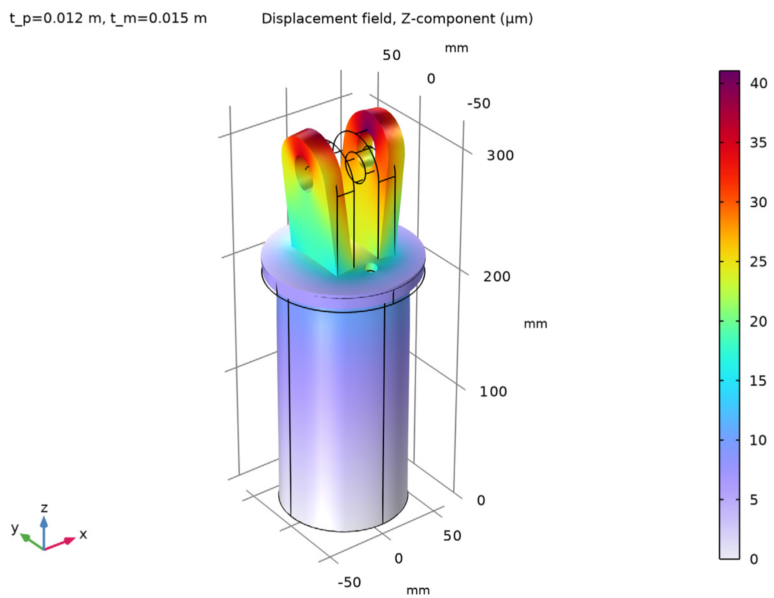


Figure 3: Deformed shape and plot of the axial displacement for the mounting assembly with an end-plate thickness of 12 mm and mount thickness of 15 mm.

Application Library path: `COMSOL_Multiphysics/Structural_Mechanics/mast_diagonal_mounting`

Modeling Instructions

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

NEW

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

MODEL WIZARD

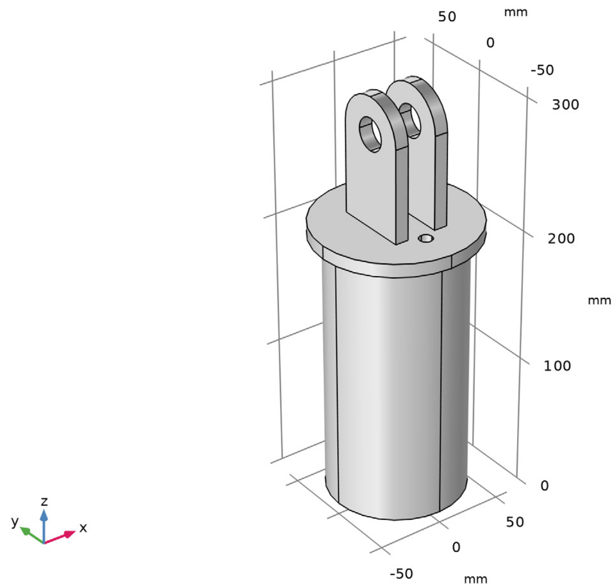
1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Stationary**.
6 Click **Done**.

GEOMETRY 1

The model geometry is available as a parameterized geometry sequence in a separate MPH file.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.
2 Browse to the model's Application Libraries folder and double-click the file `mast_diagonal_mounting_geom_sequence.mph`.
3 In the **Geometry** toolbar, click **Build All**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.

5 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.



GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`F`	`30[kN]`	30000 N	`Applied force`

DEFINITIONS

Create a nonlocal average coupling for evaluation of variables across the mid plane of both mount holes.

Mount, mid level

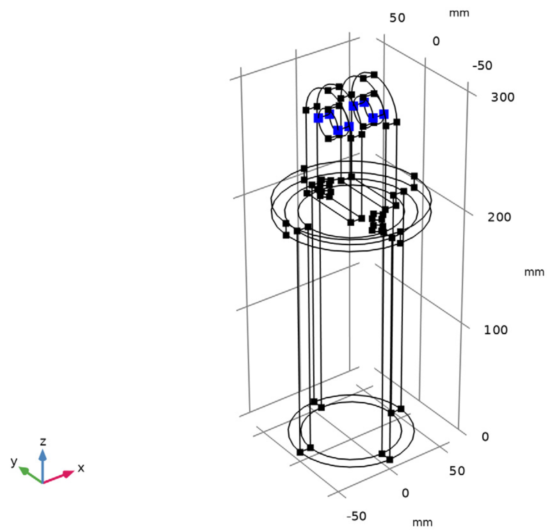
1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type `Mount, mid level` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Points 9, 13, 18, 22, 55, 59, 64, and 68 only.

The points along the mid plane are shown in figure below.



5 Click **Create Selection**.

6 In the **Create Selection** dialog, type `Mount, mid level` in the **Selection name** text field.

7 Click **OK**.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
p	-(3/2)*F/(2*Axy_mh)*(1-(Y/r_mh)^2)	N/m²	Mount hole stress
fy_mh	p*nY	N/m²	Mount hole force, y-component
fz_mh	p*nZ	N/m²	Mount hole force, z-component
d_mh	aveop1(w)	m	Mount hole displacement
L_t	h_t+t_p+o_mh	m	Equivalent tube length
Axy_mh	2*r_mh*t_m	m²	Mount hole xy projected area

Name	Expression	Unit	Description
A_t	pi*(r_t^2-(r_t-t_t)^2)	m²	Tube cross section area
S	F/d_mh	N/m	Stiffness of the assembly
S_i	200e9[Pa]*A_t/L_t	N/m	Ideal stiffness
S_R	S/S_i		Stiffness ratio

ADD MATERIAL

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

MATERIALS

Structural steel (mat1)

By default, the first material you add applies on all domains so you need not alter any settings.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

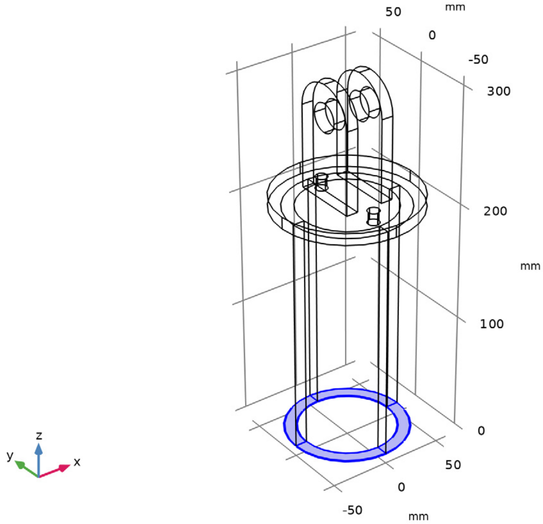
By default, the physics interface takes the required material model properties from the domain material.

Next, define the boundary conditions.

Fixed Constraint 1

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

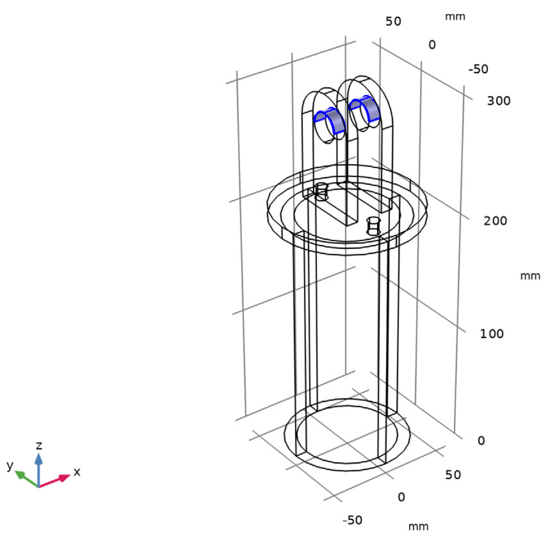
2 Select Boundaries 8, 9, 33, and 42 only.



Boundary Load 1

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

2 Select Boundaries 20, 22, 51, and 53 only.



Use the force components you defined earlier.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{f}_A$ vector as

0	x
fy_mh	y
fz_mh	z

MESH 1

This section illustrates how you can mesh different parts of the model individually to get a suitable mesh.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

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

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

4 Select Domains 4 and 7 only.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Fine**.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Build Selected**.

Free Tetrahedral 2

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

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

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

4 Select Domain 1 only.

Size 1

1 Right-click **Free Tetrahedral 2** and choose **Size**.

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

3 From the **Predefined** list, choose **Finer**.

Free Tetrahedral 2

In the **Model Builder** window, right-click **Free Tetrahedral 2** and choose **Build Selected**.

Swept 1

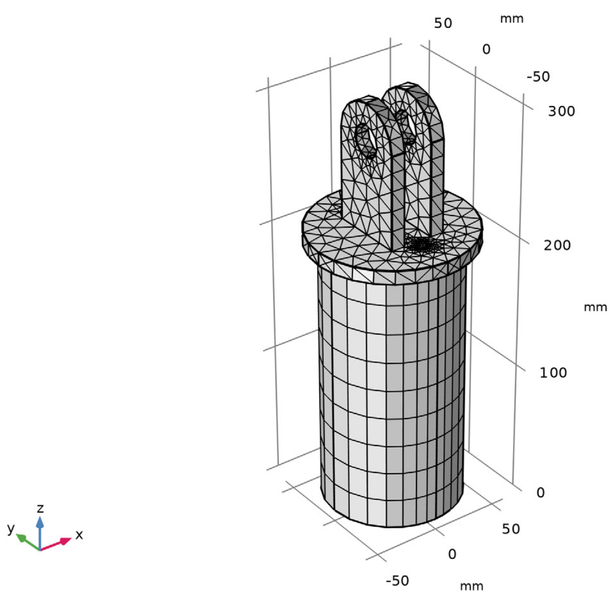
1. In the **Mesh** toolbar, click **Swept**.
2. In the **Settings** window for **Swept**, click to expand the **Source Faces** section.
3. Select Boundary 4 only.
4. Click to expand the **Destination Faces** section. Select Boundary 3 only.

Size 1

1. Right-click **Swept 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.

Swept 1

1. In the **Model Builder** window, right-click **Swept 1** and choose **Build Selected**.
2. Click the **Wireframe Rendering** button in the **Graphics** toolbar to restore the default state.
3. In the **Model Builder** window, click **Mesh 1**.



STUDY 1

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

RESULTS

Displacement

1 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

Now, plot the z-displacement and compare the stiffness ratio for the current model geometry dimensions with the ideal one.

Volume 1

1 In the **Model Builder** window, expand the **Displacement** node, then click **Volume 1**.
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** > **Displacement** > **Displacement field - m** > **w - Displacement field, Z-component**.
3 Locate the **Expression** section. From the **Unit** list, choose **µm**.
4 In the **Displacement** toolbar, click **Plot**.
5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.
2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **S_R - Stiffness ratio - 1**.
3 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The stiffness ratio obtained is 0.38, which is less than the desired value.

STUDY 1

Proceed to add a parametric sweep feature that varies the mount thickness and plate thickness.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t_p (Plate thickness)	10[mm] 12[mm]	m

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t_m (Mount thickness)	10[mm] 15[mm]	m

7 Locate the **Output While Solving** section. Select the **Plot** checkbox.

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

RESULTS

Displacement

To display the results from the parametric sweep, change the dataset.

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

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

The default parameter values correspond to those for the sweep's last parameter pair.

4 In the **Displacement** toolbar, click **Plot**.

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

Finally, compare the updated stiffness value for the updated model geometry dimensions with the ideal one.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **Variables** > **S_R - Stiffness ratio - 1**.

5 Click **Evaluate**.

TABLE 2

1 Go to the **Table 2** window.

Observe that the stiffness ratio obtained now is 0.53.