



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.

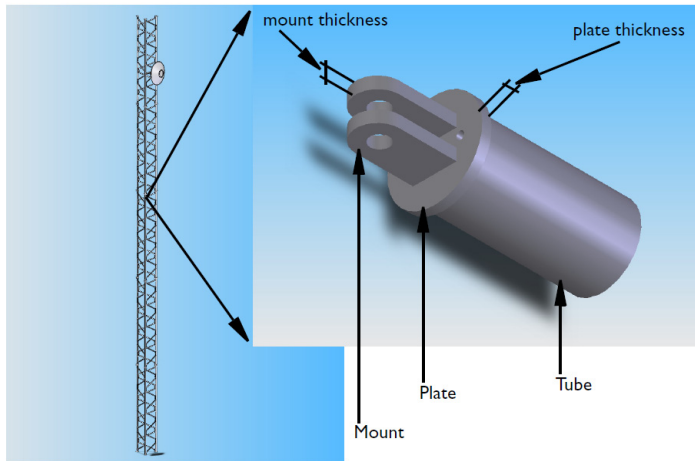


Figure 1: Mounting detail of a mast diagonal.

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, this the entire assembly id 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 $2.0 \cdot 10^{11} \text{ N/m}^2$ 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, $F = 30 \text{ kN}$, which is transferred through the bolt to the mounting.

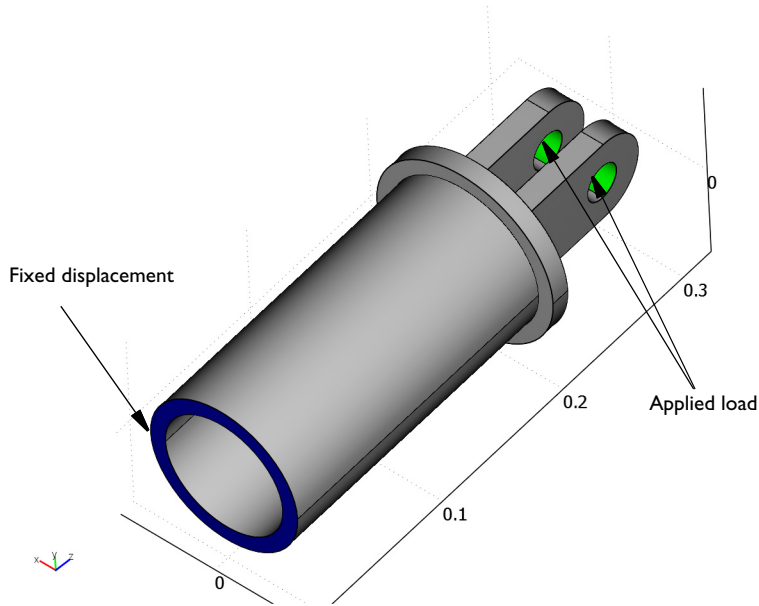


Figure 2: Boundaries with constrained displacements and applied loads.

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

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

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

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

In 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 δ_{mh} , the stiffness of the assembly is given by

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

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 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

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

Since the stiffness is defined as applied force divided by resulting displacement, which is the left hand side in the above expression, the right hand side can be seen as an ideal stiffness, S_{I} , of a body with a constant cross section. In current example that would be a tube stretching as far as the mounting hole, to be welded together with the rest of the mast. The ideal stiffness can thus be expressed with

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

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

where L_{t} is the equivalent tube length if the entire assembly was made of a tube only, h_{t} is the tube height in the assembly with a mount, 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 tube height is used to relate the tube used in the mount assembly and tube length is used to define the tube in an assembly consisting of a tube only.

The variable used as the evaluation parameter is the stiffness ratio between the real stiffness of the assembly with a mount and the ideal stiffness.

$$S_R = \frac{S}{S_I}$$

If this value equals one, then the stiffness of the assembly with a mount is exactly the same as the stiffness 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 0.38. Then the plate thickness is increased to 12 mm and the mount thickness to 15 mm, the stiffness ratio increases to 0.53. Figure 3 shows the z-component of the displacement when loading the stiffer geometry.

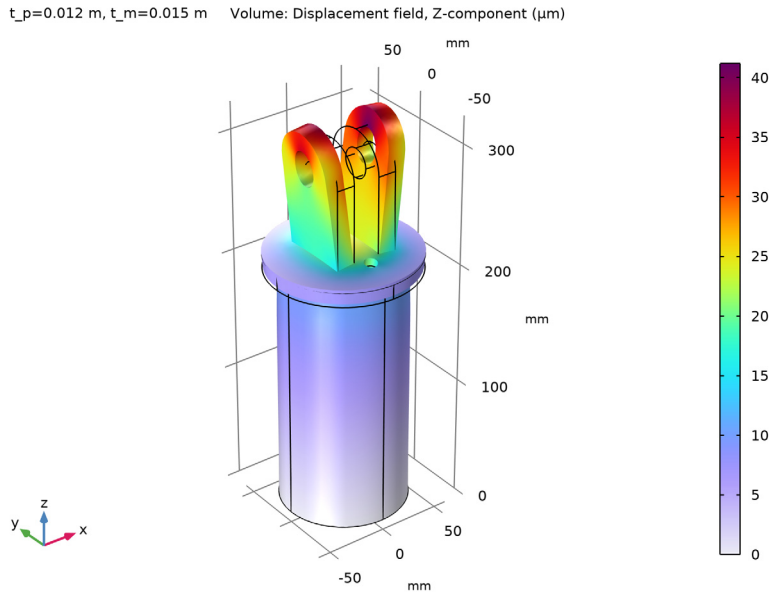



Figure 3: Deformed shape and plot of the axial displacement for the mounting assembly with an end-plate thickness of 12 mm and mount thickness of 15 mm.

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

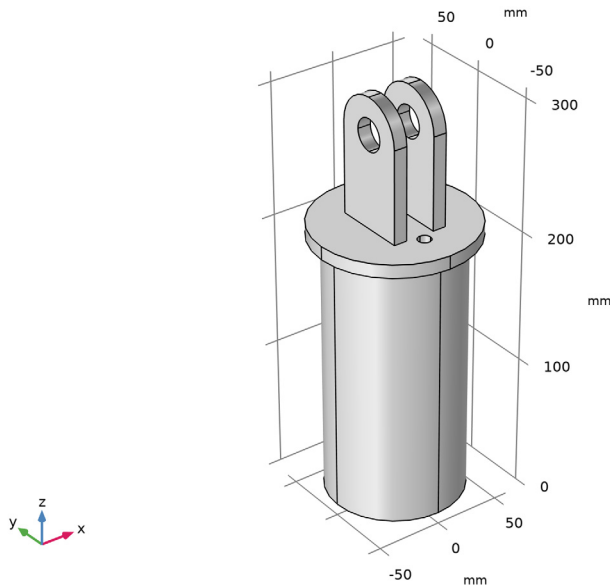
- 1 In the **Model Wizard** window, click  **3D**.
- 2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Stationary**.
- 6 Click  **Done**.

GEOMETRY I

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

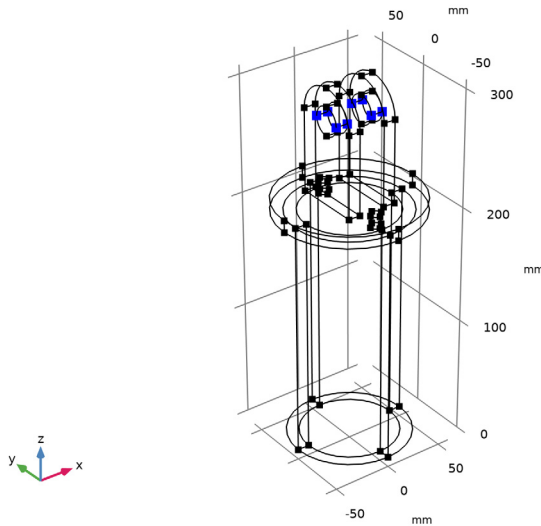
Create a nonlocal average coupling for evaluation of variables across the mid plane of both mount holes.

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 box, type Mount, mid_level in the **Selection name** text field.

- 7 Click **OK**.



Variables I

- 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	$-(3/2) * F / (2 * A_{xy_mh}) * (1 - (Y/r_{mh})^2)$	N/m ²	Mount hole stress
fy_mh	p*nY	N/m ²	Mount hole force, y-component
fz_mh	p*nZ	N/m ²	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

Name	Expression	Unit	Description
Axy_mh	$2 \cdot r_{mh} \cdot t_m$	m ²	Mount hole xy projected area
A_t	$\pi \cdot (r_t^2 - (r_t - t_t)^2)$	m ²	Tube cross section area
S	F/d_{mh}	N/m	Stiffness of the assembly
S_i	$200e9[\text{Pa}] \cdot A_t/L_t$	N/m	Ideal stiffness
S_R	S/S_i		Stiffness ratio

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 **Home** 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

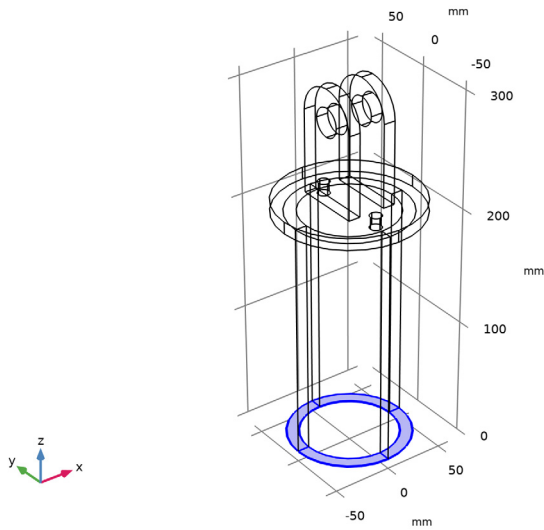
By default, the physics interface takes the required material model properties from the domain material.

Next, define the boundary conditions.

Fixed Constraint 1

- 1 In the **Model Builder** window, right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

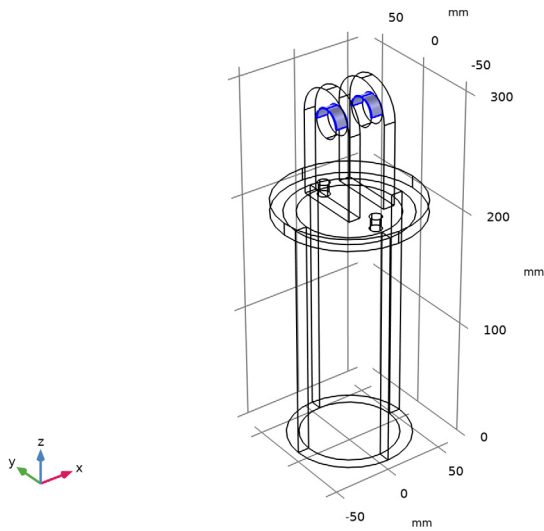
2 Select Boundaries 8, 9, 33, and 42 only.



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 I

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

Free Tetrahedral I

- 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 I

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

Free Tetrahedral I

In the **Model Builder** window, right-click **Free Tetrahedral I** 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 I

- 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

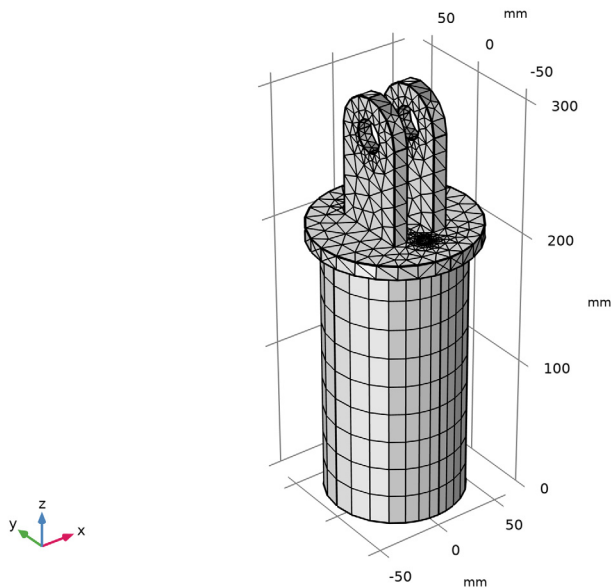
- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, click to expand the **Source Faces** section.
- 3 Select Boundary 4 only.
- 4 Click to expand the **Destination Faces** section. Select Boundary 3 only.

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 **Home** 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** check box.

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

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

Observe that the stiffness ratio obtained now is 0.53.

