

Stiffness Analysis of a Communication Mast's Diagonal Mounting

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 $2.0 \cdot 10^{11}$ N/m² 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

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

where $r_{\rm mh}$ is the mount hole radius, $t_{\rm m}$ is the thickness of the mount, y is the y-coordinate and $A_{\rm mh}^{\rm xy}$ 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

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

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, $S_{\rm I}$, 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

$$S_{\mathrm{I}} = E \frac{A_{\mathrm{t}}}{L_{\mathrm{t}}}$$
$$L_{\mathrm{t}} = h_{\mathrm{t}} + t_{\mathrm{p}} + o_{\mathrm{mh}}$$

where L_t is the equivalent tube length if the entire assembly was made of a tube only, h_t is the 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.

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

If this value equals one, then 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 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

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Structural Mechanics>Solid Mechanics (solid).
- 3 Click Add.
- 4 Click \bigcirc 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.

- I 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 **Comextents** button in the **Graphics** toolbar.



y z x

GLOBAL DEFINITIONS

Parameters 1

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

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

- I 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 here are a create Selection.
- 6 In the Create Selection dialog box, type Mount, mid level in the Selection name text field.
- 7 Click OK.

Variables I

- I In the **Definitions** toolbar, click **a**= **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
р	-(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

Name	Expression	Unit	Description
Axy_mh	2*r_mh*t_m	m²	Mount hole xy projected area
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

- I 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 I

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

Next, define the boundary conditions.

Fixed Constraint I

I 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 I

- I 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	у
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

- I In the Mesh toolbar, click \land 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

- I 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

- I In the Mesh toolbar, click \land 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

- I 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 I

- I 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

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

Swept 1

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



STUDY I In the **Home** toolbar, click **= Compute**.

RESULTS

Displacement

I 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

- I In the Model Builder window, expand the Displacement node, then click Volume I.
- 2 In the Settings window for Volume, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>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 **O Plot**.
- **5** Click the \longleftrightarrow **Zoom Extents** button in the **Graphics** toolbar.

Global Evaluation 1

- I In the **Results** toolbar, click (8.5) **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 I (compl)> Definitions>Variables>S_R Stiffness ratio.
- 3 Click **= Evaluate**.

TABLE

I Go to the Table window.

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

STUDY I

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

Parametric Sweep

- I 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.

- I 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 I/Parametric Solutions I (sol2).

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

- **4** In the **Displacement** toolbar, click **O** Plot.
- **5** Click the \longleftrightarrow **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

- I In the Results toolbar, click (8.5) Global Evaluation.
- 2 In the Settings window for Global Evaluation, locate the Data section.
- 3 From the Dataset list, choose Study I/Parametric Solutions I (sol2).
- 4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component I (compl)>Definitions>Variables>S_R Stiffness ratio**.
- 5 Click **=** Evaluate.

TABLE

I Go to the **Table** window.

Observe that the stiffness ratio obtained now is 0.53.

16 | STIFFNESS ANALYSIS OF A COMMUNICATION MAST'S DIAGONAL MOUNTING