Created in COMSOL Multiphysics 5.6

# In-Plane and Space Truss

## *Introduction*

In the following example you first build and solve a simple 2D truss model using the 2D Truss interface. Later on, you analyze a 3D variant of the same problem using the 3D Truss interface. This model calculates the deformation and forces of a simple geometry. The example is based on problem 11.1 in *Aircraft Structures for Engineering Students* by T.H.G Megson (Ref. 1). The results are compared with the analytical results given in Ref. 1.

## *Model Definition*

The 2D geometry consists of a square symmetrical truss built up by five members. All members have the same cross-sectional area $A$. The side length is $L$, and the Young's modulus is $E$.

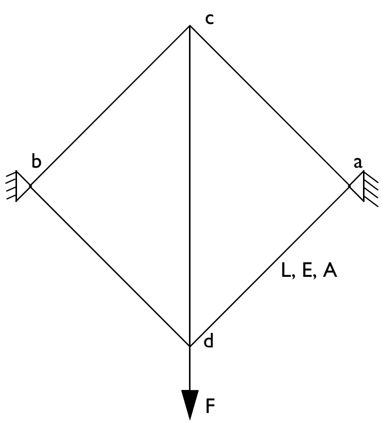


*Figure 1: The truss geometry.*

In the 3D case, another copy of the diagonal bars are rotated 90° around the vertical axis so that a cube with one space diagonal is generated. The figure above is thus applicable to a view in the $zy$-plane as well as in the $xy$-plane. The central bar is then given twice the area of the other members. In this way, a space truss with exactly the same type of symmetry, but twice the vertical stiffness is generated.

#### GEOMETRY

- Truss side length, $L$ = 2 m
- The truss members have a circular cross section with a radius of 0.05 m. In the 3D case, the area of the central bar is doubled.

#### MATERIAL

Aluminum: Young's modulus, $E$ = 70 GPa, Poisson's ration $\nu$ = 0.3.

#### CONSTRAINTS

In the 2D case, displacements in both directions are constrained at vertices a and b. In the 3D case, the two new points are constrained in the same way.

#### LOAD

In the 2D case, a vertical force $F$ of 50 kN is applied at the bottom corner. In the 3D case, the value 100 kN is used instead in order to get the same displacements.

## *Results and Discussion*

The following table shows a comparison between the results calculated with the Structural Mechanics Module and the analytical results from Ref. 1.

| RESULT | COMSOL MULTIPHYSICS | Ref. 1 |
|---|---|---|
| Displacement at d | $-5.14 \cdot 10^{-4}$ m | $-5.15 \cdot 10^{-4}$ m |
| Displacement at c | $-2.13 \cdot 10^{-4}$ m | $-2.13 \cdot 10^{-4}$ m |
| Axial force in member ac=bc | -10.4 kN | -10.4 kN |
| Axial force in member ad=bd | 25.0 kN | 25.0 kN |
| Axial force in member cd | 14.6 kN | 14.6 kN |

The results are in nearly perfect agreement.

Figure 2 and Figure 3 show plots visualizing the deformed geometry together with the axial forces in the truss members.

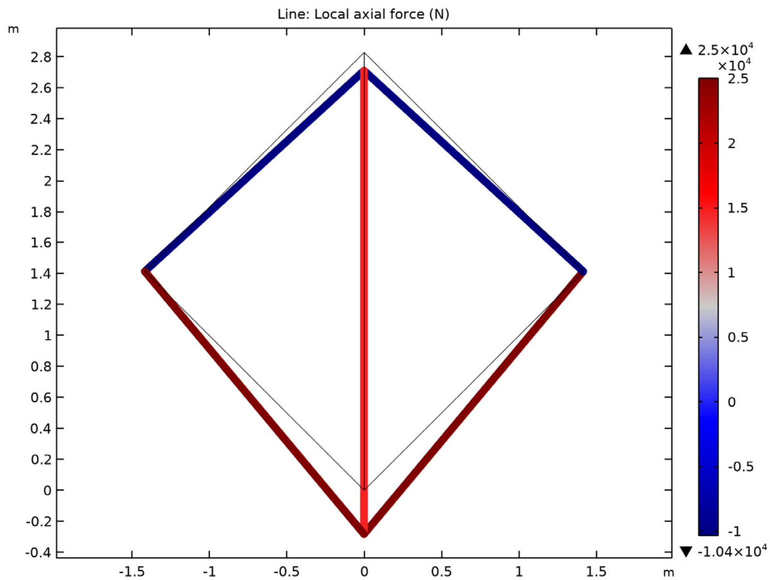


*Figure 2: Deformed geometry and axial forces for the 2D case.*

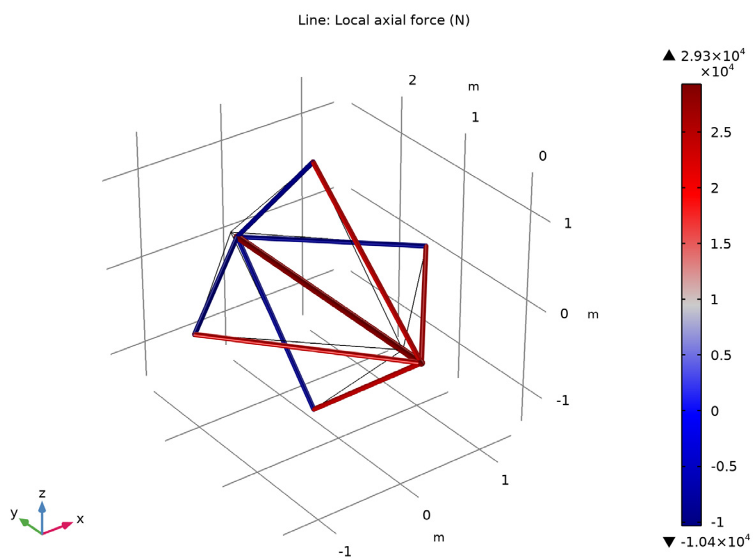


*Figure 3: Deformed geometry and axial forces for the 3D case.*

## *Notes About the COMSOL Implementation*

In this example you build the 2D and the 3D truss as two different components within the same MPH file. This is not essential, you could equally well choose to create the components in separate MPH files.

## *Reference*

1. T.H.G. Megson, *Aircraft Structures for Engineering Students*, Edward Arnold, p. 404, 1985

**Application Library path:** `Structural_Mechanics_Module/Verification_Examples/inplane_and_space_truss`

## *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 **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Truss (truss)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

### GEOMETRY 1

*Square 1 (sq1)*

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

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

3 In the **Side length** text field, type 2.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type 45.

5 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

6 Click **Build All Objects**.

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

*Line Segment 1 (ls1)*

1 In the **Geometry** 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 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **y** text field, type `sqrt(8)`.

6 Click **Build All Objects**.

## TRUSS (TRUSS)

*Cross-Section Data 1*

1 In the **Model Builder** window, under **Component 1 (comp1)>Truss (truss)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, locate the **Cross-Section Data** section.

3 In the $A$ text field, type `pi/4*0.05^2`.

*Pinned 1*

1 In the **Physics** toolbar, click **Points** and choose **Pinned**.

2 Select Points 1 and 4 only.

*Point Load 1*

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 2 only.

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

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

| | |
|---|---|
| 0 | x |
| -50e3 | y |

## GLOBAL DEFINITIONS

In this example, the same material data will be referenced from two different components, so it is convenient to define a global material.

*Material 1 (mat1)*

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

3 In the **Material properties** tree, select **Basic Properties>Density**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material> Young's modulus and Poisson's ratio**.

6 Click **Add to Material**.

7 Locate the **Material Contents** section. In the table, enter the following settings:

| **Property** | **Variable** | **Value** | **Unit** | **Property group** |
|---|---|---|---|---|
| Density | rho | 2900 | kg/m³ | Basic |
| Young's modulus | E | 70e9 | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | 0.3 | 1 | Young's modulus and Poisson's ratio |

## MATERIALS

*Material Link 1 (matlnk1)*

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

## STUDY 1

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

## RESULTS

*Force (truss)*

1 In the **Model Builder** window, expand the **Results>Force (truss)** node, then click **Force (truss)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.

3 Select the **Show maximum and minimum values** check box.

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

Next, compute the displacements at d (Vertex 2) and c (Vertex 3).

*Displacement of Vertices (2D)*

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

**2** In the **Settings** window for **Evaluation Group**, type `Displacement of Vertices (2D)` in the **Label** text field.

*Point Evaluation 1*

**1** Right-click **Displacement of Vertices (2D)** and choose **Point Evaluation**.

**2** Select Points 2 and 3 only.

**3** In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

**4** In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `v` | `m` | `Displacement field, Y component` |

**5** In the **Displacement of Vertices (2D)** toolbar, click **Evaluate**.

Although you can read off the values of the local axial force in the members ac and ad from the max and min values for the color legend for the plot in the **Graphics** window, it is instructive to see how you can compute such values more generally.

## DEFINITIONS

Add nonlocal average couplings for the members ac, ad, and cd. You will use these for defining variables that evaluate the axial forces in these members.

*Average 1 (aveop1)*

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

**2** In the **Settings** window for **Average**, type `aveop_ac` in the **Operator name** text field.

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

**4** Select Boundary 5 only.

*Average 2 (aveop2)*

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

**2** In the **Settings** window for **Average**, type `aveop_ad` in the **Operator name** text field.

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

**4** Select Boundary 4 only.

*Average 3 (aveop3)*

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

2 In the **Settings** window for **Average**, type `aveop_cd` in the **Operator name** text field.

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

4 Select Boundary 3 only.

*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 |
|---|---|---|---|
| F_ac | aveop_ac(truss.Nxl) | N | Axial force, member ac |
| F_ad | aveop_ad(truss.Nxl) | N | Axial force, member ad |
| F_cd | aveop_cd(truss.Nxl) | N | Axial force, member cd |

## STUDY 1

Update the solution to evaluate the variables you just defined.

*Solution 1 (sol1)*

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

2 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)** and choose **Solution>Update**.

## RESULTS

*Axial Force in Members (2D)*

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

2 In the **Settings** window for **Evaluation Group**, type `Axial Force in Members (2D)` in the **Label** text field.

*Global Evaluation 1*

1 Right-click **Axial Force in Members (2D)** and choose **Global Evaluation**.

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

3 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| F_ac | N | Axial force, member ac |
| F_ad | N | Axial force, member ad |
| F_cd | N | Axial force, member cd |

4 In the **Axial Force in Members (2D)** toolbar, click **Evaluate**.

The values in the evaluation group agree with those of the analytical reference solution.

Now create the 3D truss as a new model.

## ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

## ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Recently Used>Truss (truss)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 2** in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

## ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

Switch off the 2D truss physics in this study.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Truss (truss)**.

5 Click **Add Study** in the window toolbar.

6 In the **Model Builder** window, click the root node.

7 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

## GEOMETRY 2

In the **Model Builder** window, under **Component 2 (comp2)** click **Geometry 2**.

### *Work Plane 1 (wp1)*

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

2 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

### *Work Plane 1 (wp1)>Plane Geometry*

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

### *Work Plane 1 (wp1)>Square 1 (sq1)*

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

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

3 In the **Side length** text field, type `2`.

4 Locate the **Rotation Angle** section. In the **Rotation** text field, type `45`.

5 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

6 In the **Work Plane** toolbar, click **Build All**.

### *Rotate 1 (rot1)*

1 In the **Model Builder** window, right-click **Geometry 2** and choose **Transforms>Rotate**.

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

3 Select the **Keep input objects** check box.

4 Select the object **wp1** only.

5 Locate the **Rotation** section. From the **Axis type** list, choose **Cartesian**.

6 In the **y** text field, type `1`.

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

8 In the **Angle** text field, type `90`.

9 Click **Build All Objects**.

### *Line Segment 1 (ls1)*

1 In the **Geometry** 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 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 In the **y** text field, type `sqrt(8)`.

6 Click **Build All Objects**.

## DEFINITIONS (COMP2)

Add nonlocal average couplings for the members ac, ad, and cd and corresponding axial force variables.

### *Average 4 (aveop4)*

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

2 In the **Settings** window for **Average**, type `aveop_ac` in the **Operator name** text field.

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

4 Select Edge 8 only.

### *Average 5 (aveop5)*

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

2 In the **Settings** window for **Average**, type `aveop_ad` in the **Operator name** text field.

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

4 Select Edge 4 only.

### *Average 6 (aveop6)*

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

2 In the **Settings** window for **Average**, type `aveop_cd` in the **Operator name** text field.

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

4 Select Edge 5 only.

### *Variables 2*

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 |
|---|---|---|---|
| F_ac | aveop_ac(truss2.Nxl) | N | Axial force, member ac |
| F_ad | aveop_ad(truss2.Nxl) | N | Axial force, member ad |
| F_cd | aveop_cd(truss2.Nxl) | N | Axial force, member cd |

## TRUSS 2 (TRUSS2)

### *Cross-Section Data 1*

1 In the **Model Builder** window, under **Component 2 (comp2)>Truss 2 (truss2)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, locate the **Cross-Section Data** section.

**3** In the $A$ text field, type `pi/4*0.05^2`.

*Cross-Section Data 2*

**1** In the **Physics** toolbar, click **Edges** and choose **Cross-Section Data**.

**2** Select Edge 5 only.

**3** In the **Settings** window for **Cross-Section Data**, locate the **Cross-Section Data** section.

**4** In the $A$ text field, type `2*pi/4*0.05^2`.

*Pinned 1*

**1** In the **Physics** toolbar, click **Points** and choose **Pinned**.

**2** Select Points 1, 3, 4, and 6 only.

*Point Load 1*

**1** In the **Physics** toolbar, click **Points** and choose **Point Load**.

**2** Select Point 2 only.

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

**4** Specify the $\mathbf{F}_\mathrm{P}$ vector as

| | |
|---|---|
| 0 | x |
| -100e3 | y |
| 0 | z |

## MATERIALS

*Material Link 2 (matlnk2)*

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

## STUDY 2

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

## RESULTS

*Force (truss2)*

**1** In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

**2** Select the **Show maximum and minimum values** check box.

Proceed to compute the displacements at d (Vertex 2) and c (Vertex 5).

### *Displacement of Vertices (3D)*

1 In the **Model Builder** window, right-click **Displacement of Vertices (2D)** and choose **Duplicate**.
2 In the **Settings** window for **Evaluation Group**, type `Displacement of Vertices (3D)` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

### *Point Evaluation 1*

1 In the **Model Builder** window, expand the **Displacement of Vertices (3D)** node, then click **Point Evaluation 1**.
2 Select Points 2 and 5 only.
3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.
4 In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| `v2` | `m` | `Displacement field, Y component` |

5 In the **Displacement of Vertices (3D)** toolbar, click **Evaluate**.

The results are nearly identical to those of the 2D case.

Finally, compute the axial force values.

### *Axial Force in Members (3D)*

1 In the **Model Builder** window, right-click **Axial Force in Members (2D)** and choose **Duplicate**.
2 In the **Settings** window for **Evaluation Group**, type `Axial Force in Members (3D)` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

### *Global Evaluation 1*

Because the applied force was doubled to get the same displacement as, in the 2D case, you need to divide the value of the axial force in member cd by 2 to get a value comparable to that of the 2D case.

1 In the **Model Builder** window, expand the **Axial Force in Members (3D)** node, then click **Global Evaluation 1**.
2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

**3** In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| F_cd/2 | N | |

**4** In the **Axial Force in Members (3D)** toolbar, click **Evaluate**.

Again, the values in the evaluation group agree very well with the reference solution.