

ANSYS AIM 3D Static Force Computation - Physics Setup

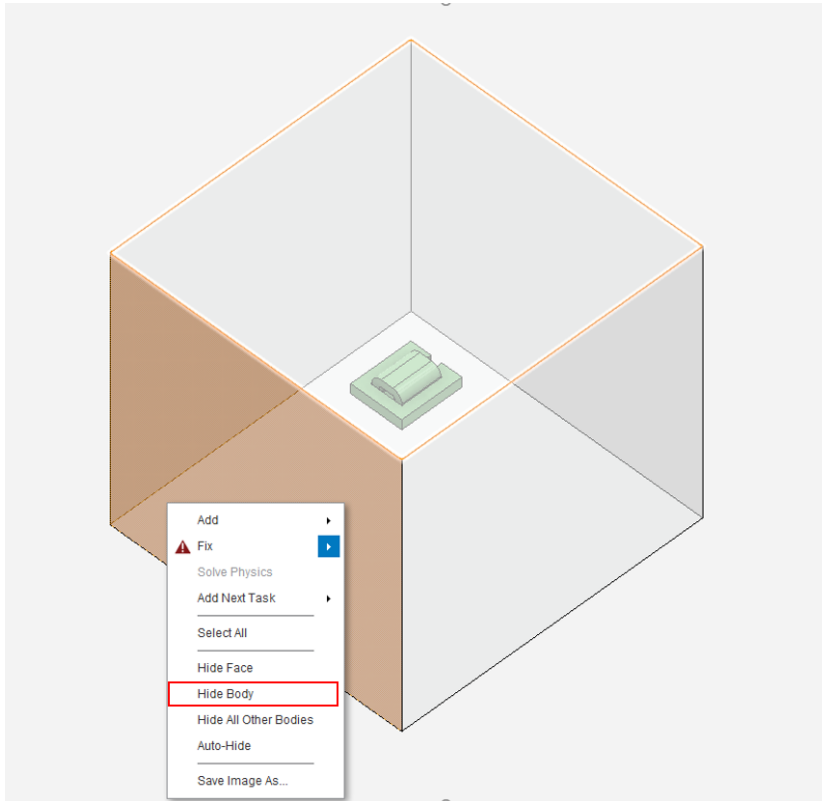
Author(s): Joshua Wallace & Steve Scampoli, ANSYS Inc.

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Physics Setup
4. Solution/Results

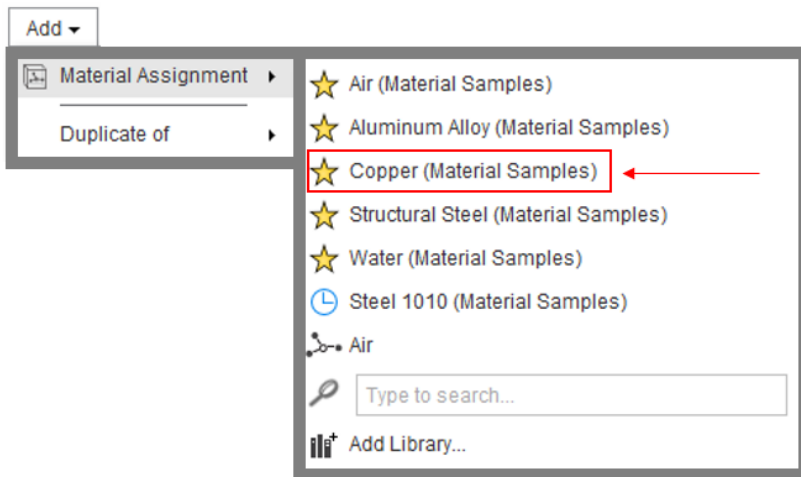
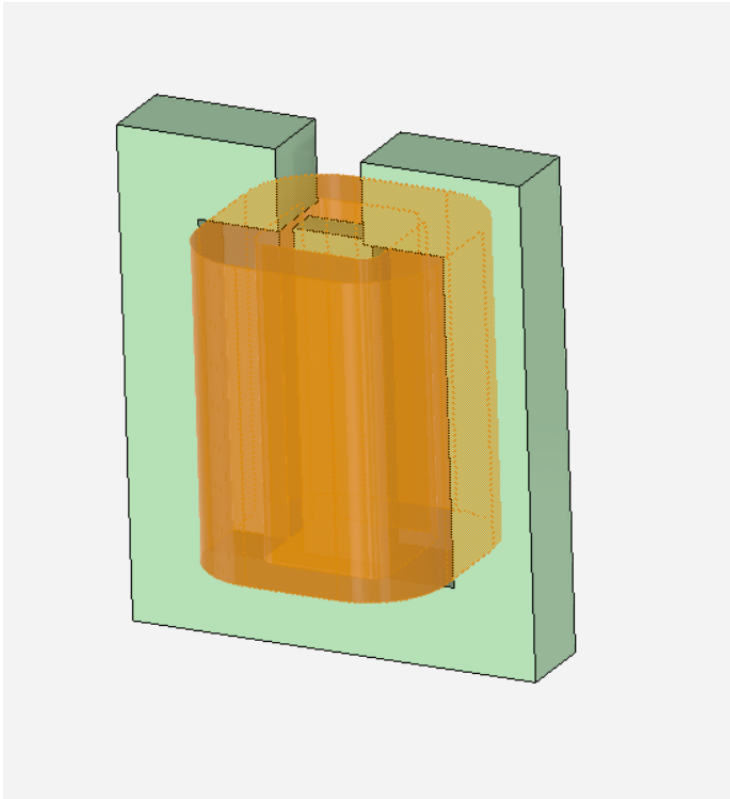
Physics Setup

AIM will automatically create an enclosure around the geometry. In order to see the inside geometry, click on **Physics** > click on the box > right click > **Hide Body**. The internal geometry can now be seen. A material will now be assigned to the different components of the internal geometry.



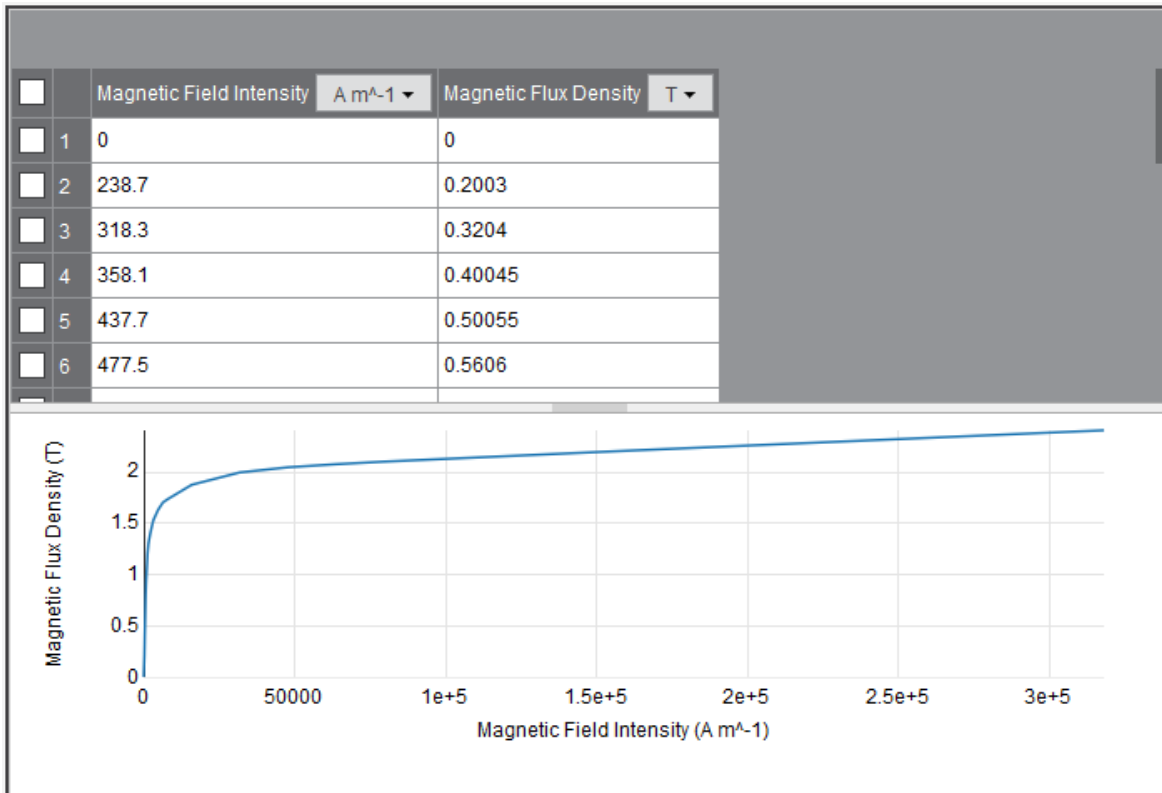
Specify Material

Click on **Add**, next to **Material Assignments** > **Material Assignment** > click **Copper (Material Assignment)**. Next, click on body selection in top right corner. Select the coil and click **Add**. This assigns copper to the coil.



Then click on next step and repeat the process, assigning **Steel 1010 (Material Assignment)** to the yoke and center pole by control selecting both and click on the "+". (You can use "Type to Search" to find **Steel 1010** in the material library.)

In order to see the B-H curve of Steel 1010, click on the arrow next to the blue lettered Steel 1010. Then click on the chart symbol for the B-H Curve. Below is the nonlinear B-H for Steel 1010. The chart below shows that the yoke and center pole are made out of a saturable material. This means as we increase the current the Steel 1010 will saturate, which results in the inability of the Steel 1010 material to carry any more magnetic flux. Click on the x in the top right corner when finished viewing. Boundary conditions must now be added.

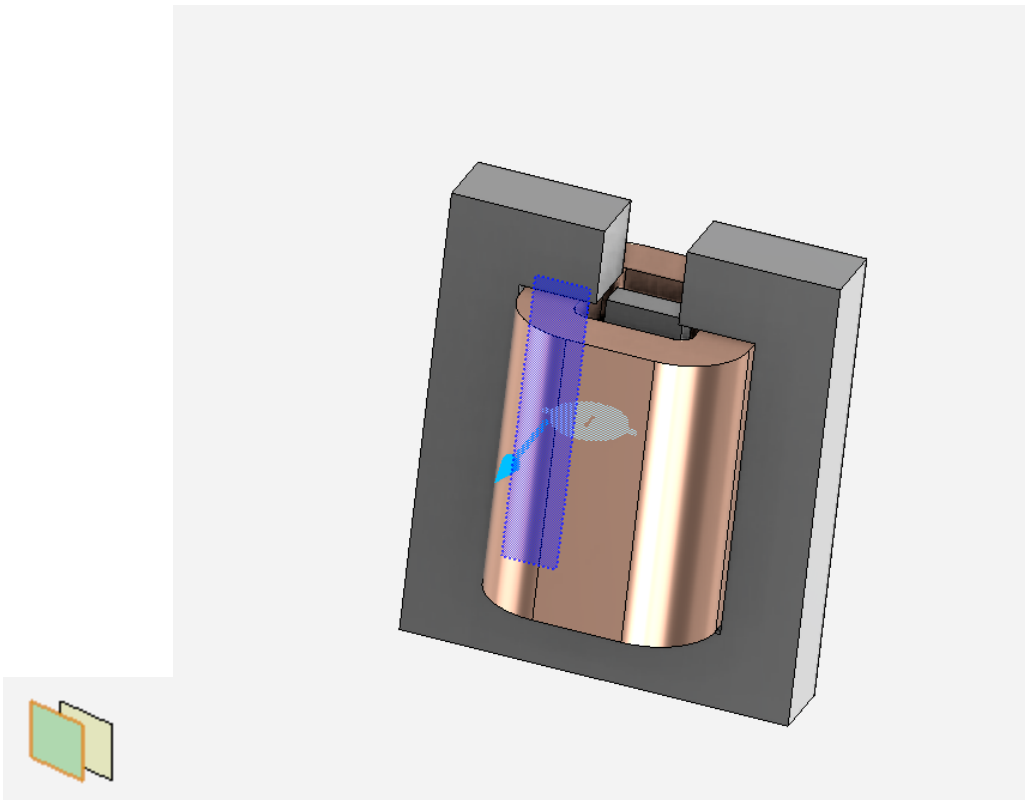


Boundary Conditions / Forces

First, we need to assign the center pole to the **Force computation**. Click **Next Step > Fix > Force Computation**. Select the center pole and click the **+**. Leave all other setting in default condition.



Now the current condition for the stranded coil needs to be selected. First click **Next Step > Add > Electromagnetic Conditions > Current**. Click on the left side of the coil. Go down to the bottom left corner and click on the plane behind the front one. If the plane does not show up click on the other side of the coil. The selected plane is seen below.



Click the "+" next to cross-sectional face or body. Input 5 A for the **Operating Current** and 200 for the **Number of Conductors**.

I

Current 1

✓

Up-to-date

⚡

Define path by

☒ Cross-section
 ☐ Source and return
 ☐ Multiple terminals

Location1

1 volume

+

↔

Operating current

5 A

+P

▶

Number of conductors

200

▶

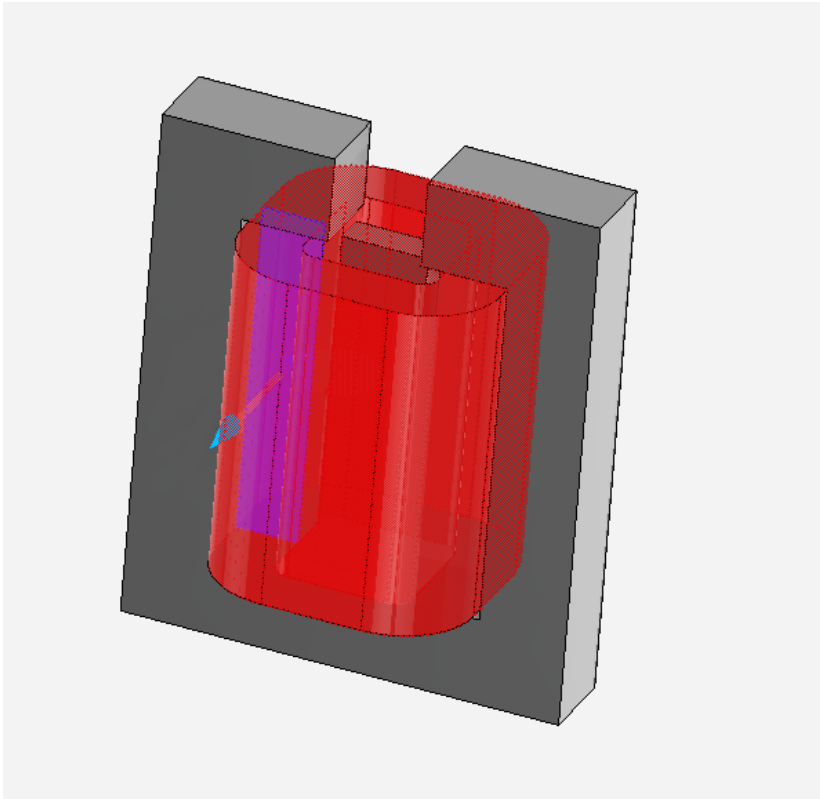
☐ Reverse polarity
 ☐ Connected in parallel
 ☒ Show conduction path

Generate Conduction Paths

Next Step ▾

> Related Objects and Tasks

Click on the small arrow next to the operating current box. Select the **P** to parametrise current. Then click on **Generate Conduction Paths**, this will show the solver interpretation of the coil. If the solver's interpretation of the coil is incorrect (current is flowing outside of the coil) insulating boundary conditions must be added to ensure that the current stays within the coil. For this model an insulating condition is not required.



[Go to Step 4: Solution/Results](#)

[Go to all ANSYS AIM Learning Modules](#)