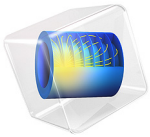


Inductance Matrix Calculation of PCB Coils

Introduction

PCB (printed-circuit board) coils are widely used in a variety of industrial applications such as micromotor and microelectronic devices. For an array of PCB coils, it is generally of interest to know the inductive coupling represented by the inductance matrix. This model demonstrates how to use the Magnetic Fields, Currents Only interface to compute the inductance matrix of an array of coils in a multilayer PCB, as shown in Figure 1.

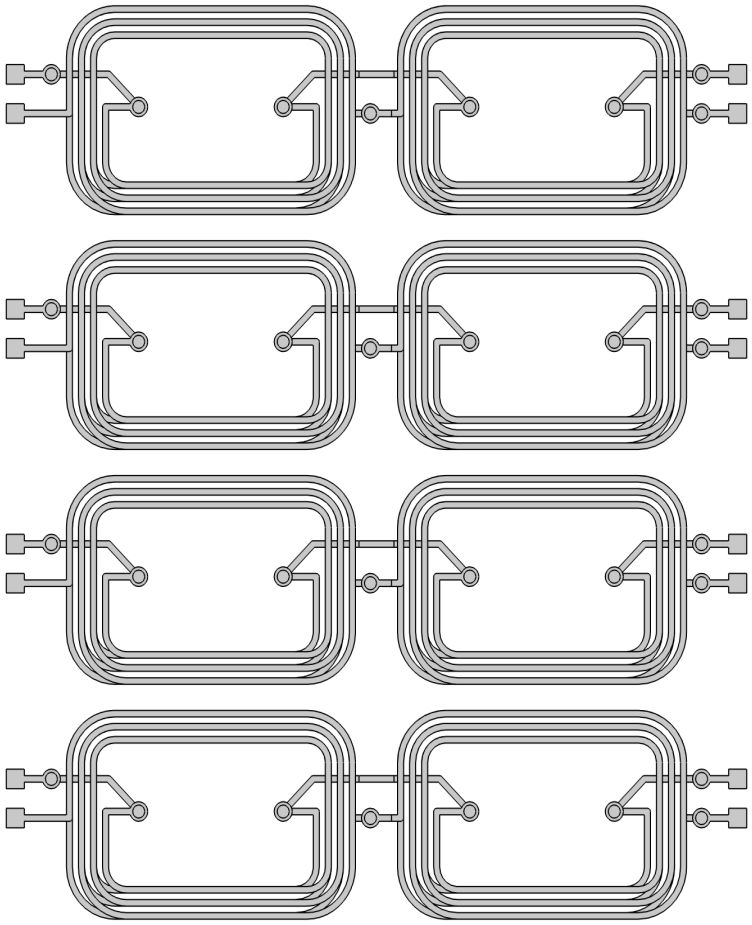


Figure 1: The geometry of PCB coils.

Model Definition

The geometry of the coils is usually 'open'; the modeling of a closed current loop is not of interest and is also not necessary. The Magnetic Fields, Currents Only interface can model such nondivergence-free currents. For more details, see *Theory for the Magnetic Fields, Currents Only Interface* in the *AC/DC Module User's Guide*.

The PCB coils are modeled by the Conductor feature, which can be used to easily set up the current sources. The model is solved with a Stationary Source Sweep with Initialization study, which is dedicated to computing the lumped inductance matrix.

Results and Discussion

Figure 2 shows the distribution of the magnetic flux density norm around the PCB coils when the 11th conductor is activated. Figure 3 illustrates the inductance matrix of the PCB coils.

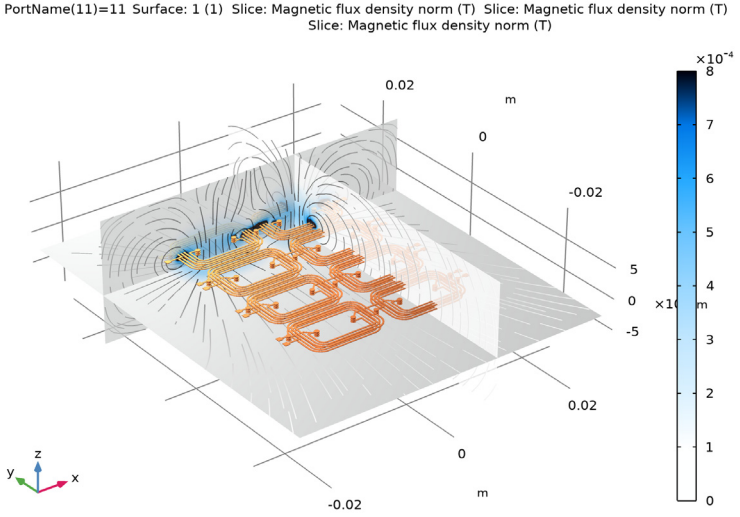


Figure 2: The distribution of the magnetic flux density norm around PCB coils.

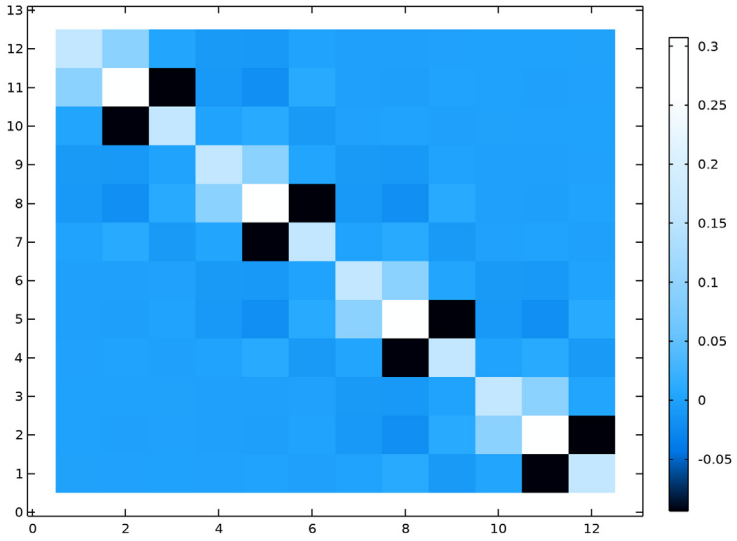


Figure 3: Visualization of the inductance matrix of PCB coils.

Application Library path: `ACDC_Module/Inductive_Devices_and_Coils/inductance_matrix_pcb_coils`

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 **AC/DC>Electromagnetic Fields>Vector Formulations>Magnetic Fields, Currents Only (mfco)**.
3. Click **Add**.
4. Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces> Stationary Source Sweep with Initialization**.

6 Click **Done**.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Advanced** section.

3 From the **Geometry representation** list, choose **CAD kernel**.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

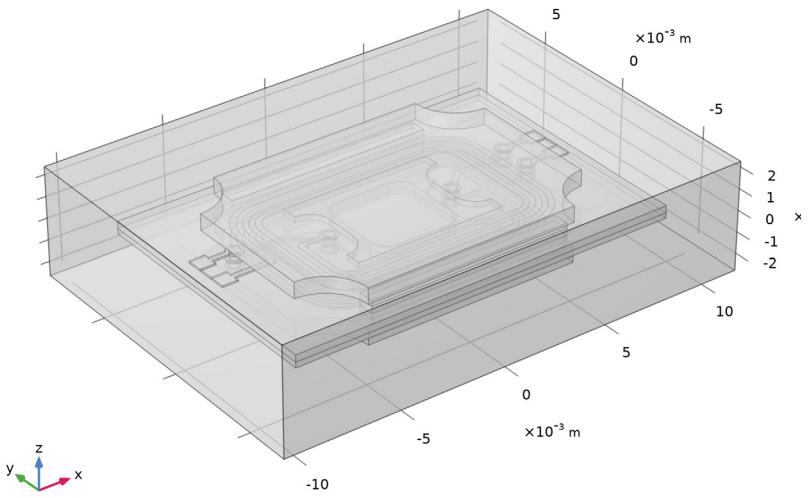
2 In the **Settings** window for **Import**, locate the **Import** section.

3 From the **Source** list, choose **COMSOL Multiphysics file**.

4 In the **Filename** text field, type `inductance_matrix_pcb_coils.mphbin`.

5 Click **Import**.

6 Click the **Transparency** button in the **Graphics** toolbar.



Explicit Selection 1 (sel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 On the object **imp1**, select Domains 4, 5, 8–15, 26–31, and 34–54 only.

Complement Selection 1 (comsel1)

1 In the **Geometry** toolbar, click **Selections** and choose **Complement Selection**.

2 In the **Settings** window for **Complement Selection**, locate the **Input Entities** section.

3 Click **Add**.

4 In the **Add** dialog box, select **Explicit Selection 1** in the **Selections to invert** list.

5 Click **OK**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **imp1**, select Domains 1–3, 6, 7, 16–25, 32, and 33 only.

3 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

5 From the **Selection** list, choose **Complement Selection 1**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **del1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `2`.

5 Locate the **Displacement** section. In the **x** text field, type `16.25[mm]`.

Block 1 (blk1)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.01-0.00625+0.0002`.

4 In the **Depth** text field, type `0.001375`.

5 In the **Height** text field, type `0.01`.

6 Locate the **Position** section. In the **x** text field, type `0.00625-0.0001`.

7 In the **y** text field, type `0.001375/2`.

8 In the **z** text field, type `-0.005`.

Block 2 (blk2)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `0.0025`.

4 In the **Depth** text field, type `0.001375`.

5 In the **Height** text field, type `0.01`.

6 Locate the **Position** section. In the **x** text field, type `0.0075`.

7 In the **y** text field, type `-0.001375/2`.

8 In the **z** text field, type `-0.005`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.

2 Select the objects **arr1(1,1,1)** and **arr1(2,1,1)** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the  **Activate Selection** toggle button.

5 Select the objects **blk1** and **blk2** only.

Split 1 (spl1)

1 In the **Geometry** toolbar, click  **Conversions** and choose **Split**.

2 Select the object **dif1** only.

Move 1 (mov1)

1 In the **Geometry** toolbar, click  **Transforms** and choose **Move**.

2 Select the objects **spl1(15)**, **spl1(16)**, **spl1(17)**, **spl1(18)**, **spl1(19)**, **spl1(20)**, **spl1(21)**, **spl1(22)**, **spl1(23)**, **spl1(24)**, **spl1(25)**, **spl1(26)**, **spl1(27)**, **spl1(28)**, **spl1(29)**, **spl1(30)**, **spl1(31)**, **spl1(32)**, **spl1(33)**, **spl1(34)**, **spl1(35)**, **spl1(50)**, **spl1(51)**, **spl1(52)**, **spl1(53)**, **spl1(54)**, **spl1(55)**, **spl1(56)**, **spl1(57)**, and **spl1(59)** only.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **x** text field, type `-0.0025`.

5 Click  **Build Selected**.

Extrude 1 (ext1)

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

2 On the object **mov1(26)**, select Boundary 1 only.

3 In the **Settings** window for **Extrude**, locate the **Distances** section.

4 In the table, enter the following settings:

Distances (m)
`(0.01-0.00625+0.0002)-0.0025`

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **spl1(1)**, **spl1(10)**, **spl1(11)**, **spl1(12)**, **spl1(13)**, **spl1(14)**, **spl1(2)**, **spl1(3)**, **spl1(4)**, **spl1(5)**, **spl1(6)**, **spl1(60)**, **spl1(7)**, **spl1(8)**, and **spl1(9)** only.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Select the **Resulting objects selection** check box.

Union 2 (uni2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **ext1**, **mov1(22)**, **mov1(23)**, **mov1(25)**, **mov1(27)**, **mov1(28)**, **mov1(29)**, **spl1(36)**, **spl1(37)**, **spl1(38)**, **spl1(39)**, **spl1(40)**, **spl1(41)**, **spl1(42)**, **spl1(43)**, **spl1(44)**, **spl1(45)**, **spl1(46)**, **spl1(47)**, **spl1(48)**, **spl1(49)**, and **spl1(58)** only.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Select the **Resulting objects selection** check box.

Union 3 (uni3)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **mov1(1)**, **mov1(10)**, **mov1(11)**, **mov1(12)**, **mov1(13)**, **mov1(14)**, **mov1(15)**, **mov1(16)**, **mov1(17)**, **mov1(18)**, **mov1(19)**, **mov1(2)**, **mov1(20)**, **mov1(21)**, **mov1(24)**, **mov1(3)**, **mov1(30)**, **mov1(4)**, **mov1(5)**, **mov1(6)**, **mov1(7)**, **mov1(8)**, and **mov1(9)** only.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Select the **Resulting objects selection** check box.

Array 2 (arr2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

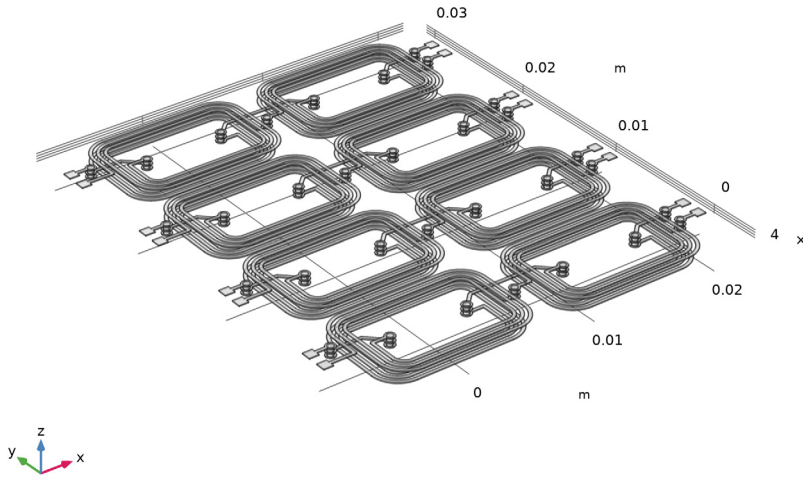
2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **y size** text field, type `4`.

5 Locate the **Displacement** section. In the **y** text field, type `9[mm]`.

6 Click **Build Selected**.



Coils

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Move**, locate the **Displacement** section.

4 In the **x** text field, type `-(0.02225-0.0085)/2`.

5 In the **y** text field, type `-(0.031-0.004)/2`.

6 In the **z** text field, type `-(6.500803969E-4-7.508789687E-5)/2`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 From the **Show in physics** list, choose **All levels**.

9 In the **Label** text field, type `Coils`.

Block 3 (blk3)

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `5[cm]`.

4 In the **Depth** text field, type `6[cm]`.

5 In the **Height** text field, type `2[cm]`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `-2[mm]`.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **blk3** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Work Plane 2 (wp2)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `7[mm]`.

Partition Objects 2 (par2)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **par1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Work Plane 3 (wp3)

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

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

4 In the **y-coordinate** text field, type `14.25[mm]`.

Partition Objects 3 (par3)

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **par2** only.

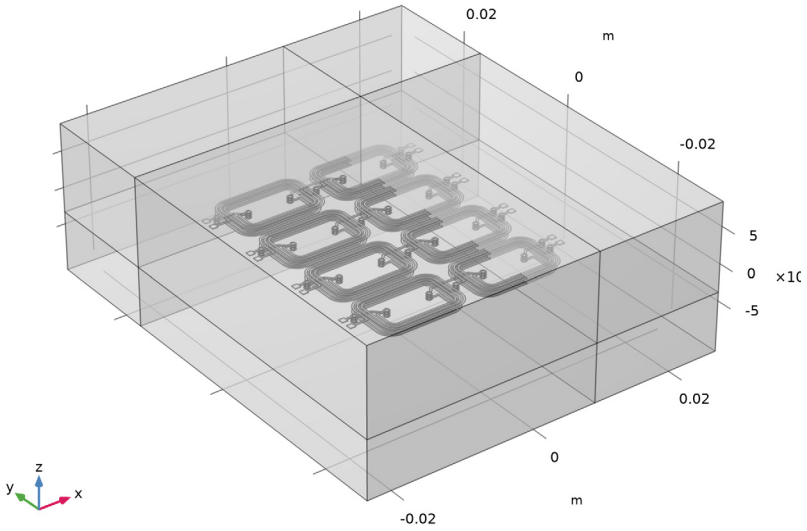
3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 From the **Partition with** list, choose **Work plane**.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

2 Click the **Go to Default View** button in the **Graphics** toolbar.



DEFINITIONS

The geometry is now complete. Next, define coil selections for the physics interface to use.

Box 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Component 1 (comp1)>Definitions>Selections** and choose **Box**.

3 In the **Settings** window for **Box**, locate the **Input Entities** section.

4 From the **Entities** list, choose **From selections**.

5 Under **Selections**, click **Add**.

6 In the **Add** dialog box, select **Union 1** in the **Selections** list.

7 Click **OK**.

8 In the **Settings** window for **Box**, locate the **Box Limits** section.

9 In the **x minimum** text field, type `-10[cm]`.

10 In the **x maximum** text field, type `10[cm]`.

11 In the **y maximum** text field, type `-9[mm]`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 2

1 In the **Definitions** toolbar, click **Box**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click **Add**.

5 In the **Add** dialog box, select **Union 2** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y maximum** text field, type `-9[mm]`.

11 In the **z minimum** text field, type `-1[cm]`.

12 In the **z maximum** text field, type `1[cm]`.

13 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 3

1 In the **Definitions** toolbar, click **Box**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click **Add**.

5 In the **Add** dialog box, select **Union 3** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y maximum** text field, type `-9[mm]`.

11 In the **z minimum** text field, type `-1[cm]`.

12 In the **z maximum** text field, type `1[cm]`.

13 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 4

1 In the **Definitions** toolbar, click  **Box**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 1** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `-9[mm]`.

11 In the **y maximum** text field, type `0`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 5

1 In the **Definitions** toolbar, click  **Box**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 2** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `-9[mm]`.

11 In the **y maximum** text field, type `0`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 6

1 In the **Definitions** toolbar, click  **Box**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 3** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `-9[mm]`.

11 In the **y maximum** text field, type `0`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 7

In the **Definitions** toolbar, click  **Box**.

Box 8

In the **Definitions** toolbar, click  **Box**.

Box 9

In the **Definitions** toolbar, click  **Box**.

Box 7

1 In the **Model Builder** window, click **Box 7**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 1** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `0`.

11 In the **y maximum** text field, type `9[mm]`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 8

1 In the **Model Builder** window, click **Box 8**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 2** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `0`.

11 In the **y maximum** text field, type `9[mm]`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 9

1 In the **Model Builder** window, click **Box 9**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 3** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `0`.

11 In the **y maximum** text field, type `9[mm]`.

12 In the **z minimum** text field, type `-1[cm]`.

13 In the **z maximum** text field, type `1[cm]`.

14 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 10

In the **Definitions** toolbar, click  **Box**.

Box 11

In the **Definitions** toolbar, click  **Box**.

Box 12

In the **Definitions** toolbar, click  **Box**.

Box 10

1 In the **Model Builder** window, click **Box 10**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 1** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `9[mm]`.

11 In the **z minimum** text field, type `-1[cm]`.

12 In the **z maximum** text field, type `1[cm]`.

13 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Box 11

1 In the **Model Builder** window, click **Box 11**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 2** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `9[mm]`.

11 In the **z minimum** text field, type `-1[cm]`.

12 In the **z maximum** text field, type `1[cm]`.

Box 12

1 In the **Model Builder** window, click **Box 12**.

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

3 From the **Entities** list, choose **From selections**.

4 Under **Selections**, click + **Add**.

5 In the **Add** dialog box, select **Union 3** in the **Selections** list.

6 Click **OK**.

7 In the **Settings** window for **Box**, locate the **Box Limits** section.

8 In the **x minimum** text field, type `-10[cm]`.

9 In the **x maximum** text field, type `10[cm]`.

10 In the **y minimum** text field, type `9[mm]`.

11 In the **z minimum** text field, type `-1[cm]`.

12 In the **z maximum** text field, type `1[cm]`.

13 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

MAGNETIC FIELDS, CURRENTS ONLY (MFCO)

Conductor 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic Fields, Currents Only (mfco)** and choose **Conductor**.
2. In the **Settings** window for **Conductor**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Box 1**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1. In the **Model Builder** window, click **Terminal 1**.
2. Select Boundary 17 only.

Ground 1

1. In the **Model Builder** window, click **Ground 1**.
2. Select Boundary 22 only.

Conductor 2

1. In the **Physics** toolbar, click **Domains** and choose **Conductor**.
2. In the **Settings** window for **Conductor**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Box 2**.

Terminal 1

1. In the **Model Builder** window, expand the **Conductor 2** node, then click **Terminal 1**.
2. Select Boundary 1347 only.

Conductor 3

1. In the **Physics** toolbar, click **Domains** and choose **Conductor**.
2. In the **Settings** window for **Conductor**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Box 3**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1. In the **Model Builder** window, click **Terminal 1**.
2. Select Boundary 2913 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 2914 only.

Conductor 4

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 4**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1 In the **Model Builder** window, click **Terminal 1**.

2 Select Boundary 27 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 32 only.

Conductor 5

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 5**.

Terminal 1

1 In the **Model Builder** window, expand the **Conductor 5** node, then click **Terminal 1**.

2 Select Boundary 1352 only.

Conductor 6

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 6**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1 In the **Model Builder** window, click **Terminal 1**.

2 Select Boundary 2915 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 2916 only.

Conductor 7

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 7**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1 In the **Model Builder** window, click **Terminal 1**.

2 Select Boundary 37 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 42 only.

Conductor 8

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 8**.

Terminal 1

1 In the **Model Builder** window, expand the **Conductor 8** node, then click **Terminal 1**.

2 Select Boundary 1357 only.

Conductor 9

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 9**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1 In the **Model Builder** window, click **Terminal 1**.

2 Select Boundary 2917 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 2918 only.

Conductor 10

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 10**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1 In the **Model Builder** window, click **Terminal 1**.

2 Select Boundary 47 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 52 only.

Conductor 11

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 11**.

Terminal 1

1 In the **Model Builder** window, expand the **Conductor 11** node, then click **Terminal 1**.

2 Select Boundary 1362 only.

Conductor 12

1 In the **Physics** toolbar, click **Domains** and choose **Conductor**.

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

3 From the **Selection** list, choose **Box 12**.

Ground 1

In the **Physics** toolbar, click **Attributes** and choose **Ground**.

Terminal 1

1 In the **Model Builder** window, click **Terminal 1**.

2 Select Boundary 2919 only.

Ground 1

1 In the **Model Builder** window, click **Ground 1**.

2 Select Boundary 2920 only.

DEFINITIONS

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 4, 5, and 7 only.

MESH 1

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

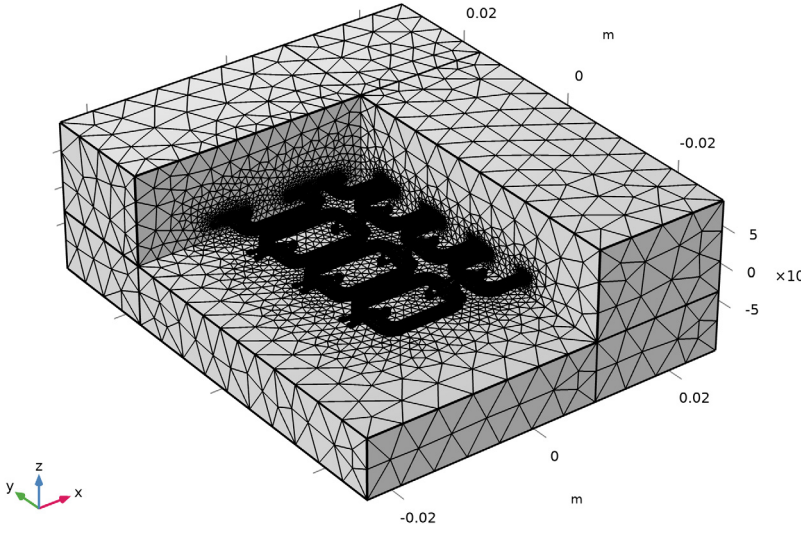
2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Fine**.

4 Click the **Go to Default View** button in the **Graphics** toolbar.

5 Click the **Transparency** button in the **Graphics** toolbar.

6 Click **Build All**.



MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Coils**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	6e7	S/m	Basic

STUDY 1

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

RESULTS

Cut Plane 1

1. In the **Model Builder** window, expand the **Results>Datasets** node, then click **Cut Plane 1**.
2. In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
3. In the **z-coordinate** text field, type `-2[mm]`.

Cut Plane 2

1. In the **Model Builder** window, click **Cut Plane 2**.
2. In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
3. In the **x-coordinate** text field, type `7[mm]`.

Cut Plane 3

1. In the **Model Builder** window, click **Cut Plane 3**.
2. In the **Settings** window for **Cut Plane**, locate the **Plane Data** section.
3. In the **y-coordinate** text field, type `14.25[mm]`.

Multislice 1

1. In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mfco)** node.
2. Right-click **Multislice 1** and choose **Delete**.

Surface 1

1. In the **Model Builder** window, right-click **Magnetic Flux Density Norm (mfco)** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `1`.

Selection 1

1. Right-click **Surface 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Coils**.

Material Appearance 1

1. In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.
2. In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
3. From the **Appearance** list, choose **Custom**.
4. From the **Material type** list, choose **Copper**.

Slice 1

1. In the **Model Builder** window, right-click **Magnetic Flux Density Norm (mfco)** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Plane Data** section.
3. From the **Plane** list, choose **xy-planes**.
4. In the **Planes** text field, type `-2[mm]`.
5. Locate the **Coloring and Style** section. From the **Color table** list, choose **JupiterAuroraBorealis**.
6. Select the **Reverse color table** check box.
7. Locate the **Plane Data** section. From the **Entry method** list, choose **Coordinates**.
8. In the **z-coordinates** text field, type `-2[mm]`.
9. Click to expand the **Range** section. Select the **Manual color range** check box.
10. In the **Maximum** text field, type `8.0E-4`.
11. Click the  **Transparency** button in the **Graphics** toolbar.

Slice 2

1. Right-click **Magnetic Flux Density Norm (mfco)** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Plane Data** section.
3. From the **Entry method** list, choose **Coordinates**.

4 In the **x-coordinates** text field, type `7[mm]`.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Transparency 1

1 Right-click **Slice 2** and choose **Transparency**.

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

3 In the **Transparency** text field, type `0.2`.

Slice 3

1 In the **Model Builder** window, right-click **Magnetic Flux Density Norm (mfco)** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **zx-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 In the **y-coordinates** text field, type `14.25[mm]`.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Slice 2**.

Transparency 1

1 Right-click **Slice 3** and choose **Transparency**.

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

3 In the **Transparency** text field, type `0.2`.

Magnetic Flux Density Norm (mfco)

1 In the **Model Builder** window, click **Magnetic Flux Density Norm (mfco)**.

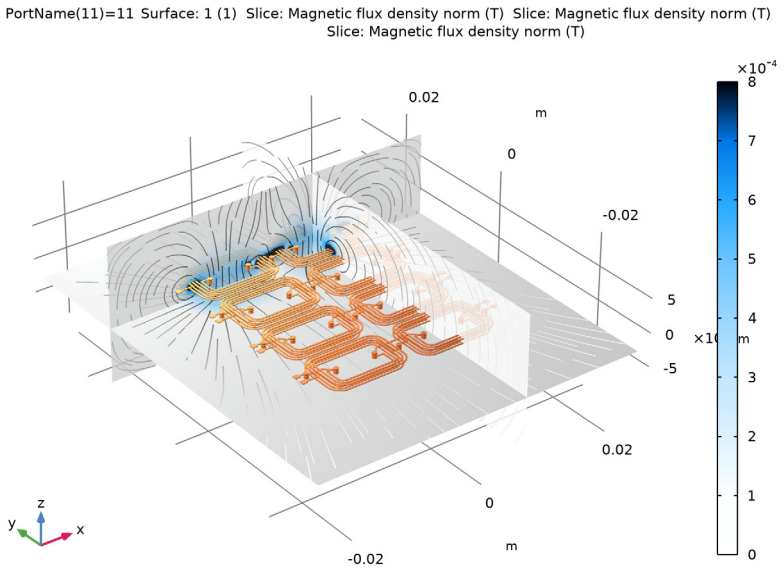
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (PortName)** list, choose **11**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Locate the **Color Legend** section. Clear the **Show maximum and minimum values** check box.

6 In the **Magnetic Flux Density Norm (mfco)** toolbar, click **Plot**.



Inductance (mfco, dset1)

1 In the **Model Builder** window, expand the **Results>Derived Values** node, then click **Inductance (mfco, dset1)**.

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

3 From the **Parameter selection (PortName)** list, choose **First**.

4 Locate the **Expression** section. From the **Unit** list, choose **µH**.

5 Click **Evaluate**.

The inductance matrix is shown in the table.

TABLE

1 Go to the **Table** window.

2 Click **Table Surface** in the window toolbar.

RESULTS

Table Surface 1

1 In the **Model Builder** window, under **Results>2D Plot Group 2** click **Table Surface 1**.

2 In the **Settings** window for **Table Surface**, locate the **Data** section.

3 From the **Data format** list, choose **Cells**.

4 Locate the **Coloring and Style** section. From the **Function** list, choose **Discrete**.

5 In the **2D Plot Group 2** toolbar, click **Plot**.

6 From the **Color table** list, choose **JupiterAuroraBorealis**.

7 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Inductance Matrix

1 In the **Model Builder** window, under **Results** click **2D Plot Group 2**.

2 In the **Settings** window for **2D Plot Group**, type `Inductance Matrix` in the **Label** text field.

3 In the **Inductance Matrix** toolbar, click **Plot**.