![Texas Instruments logo]

Simulate Inductive Sensors Using FEMM (Finite Element Method Magnetics)

ABSTRACT

When determining the resolution of an inductive sensing system, it can be especially helpful to simulate the response of the system before building the sensor. Finite Element Method Magnetics (FEMM) is open-source software that can simulate electromagnetic and electrostatic problems. This application note is a step-by-step guide for using FEMM to simulate inductive sensing solutions using LDC devices and estimate system resolution using the simulation results. This application note is applicable to the LDC161x, LDC131x, and LDC1101 devices.

Contents

List of Figures

Trademarks

All trademarks are the property of their respective owners.

1 Introduction

Simulation tools like SPICE are an important part of the design process for many systems because they reveal design problems before prototypes are built. SPICE, however, is not suitable to simulate inductive sensing systems because it does not include target interactions with the sensor coil. The open-source magnetics simulation software FEMM (Finite Element Method Magnetics) can provide information about these target interactions and the robustness of the system before production. Before using FEMM, we recommend using the LDC Tools Excel calculator to design the sensor coil, as well as reading LDC Sensor Design and LDC Target Design. For smooth targets that are at least as large as the sensor coil, TI also recommends using the Excel_FEMM tool (included in LDC Tools), which will automatically simulate custom sensors in FEMM. For smaller or irregularly shaped targets, the following guide will give the reader working knowledge of FEMM for inductive sensing simulations. The program can be downloaded from the FEMM homepage.

2 Inductive Sensor Simulation

1. Design a coil using the LDC Tools Excel calculator's "Spiral Inductor Designer." The dimensions of this coil will be used to create the FEMM simulation. Instructions are embedded in the tool.
2. Create a new problem
   a. Open FEMM
   b. Click **File → New**
   c. In the **Create a new problem** dialog box, select **Magnetics Problem** and click **OK**.
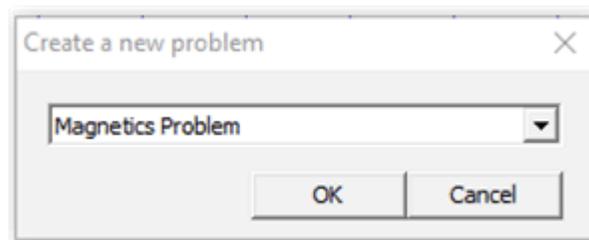
Figure 1. Create a New Problem Dialog Box

3. Make the new problem a planar problem or an axisymmetric problem.

   When choosing between the two problem types, consider the geometry of the coil. For both planar and axisymmetric problems, the FEMM workspace forms a cross section of the coil and target. The workspace is a slice of the PCB coil such that all layers and traces of the PCB are visible. Keep this in mind while constructing the coil. It is important to construct the cross section such that the simulation will create a coil shape. See how each problem type turns the 2D workspace into a 3D solution:

a. Planar problems:

Planar problems expand along the axis perpendicular to the plane. The user must specify the distance of this expansion. This type of problem is most often used to simulate racetrack coils. However, planar problems cut off the rounded edges of the coils, leaving only the long parallel section. This makes the simulation much less accurate than spiral coil simulations using axisymmetric problems.
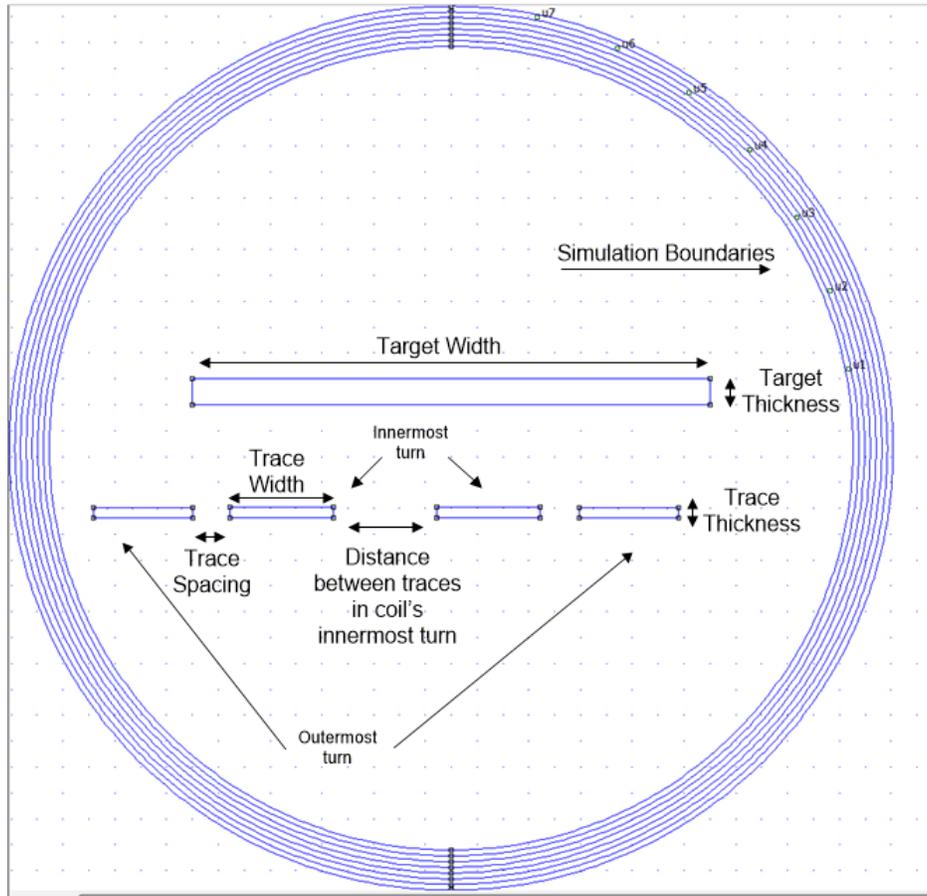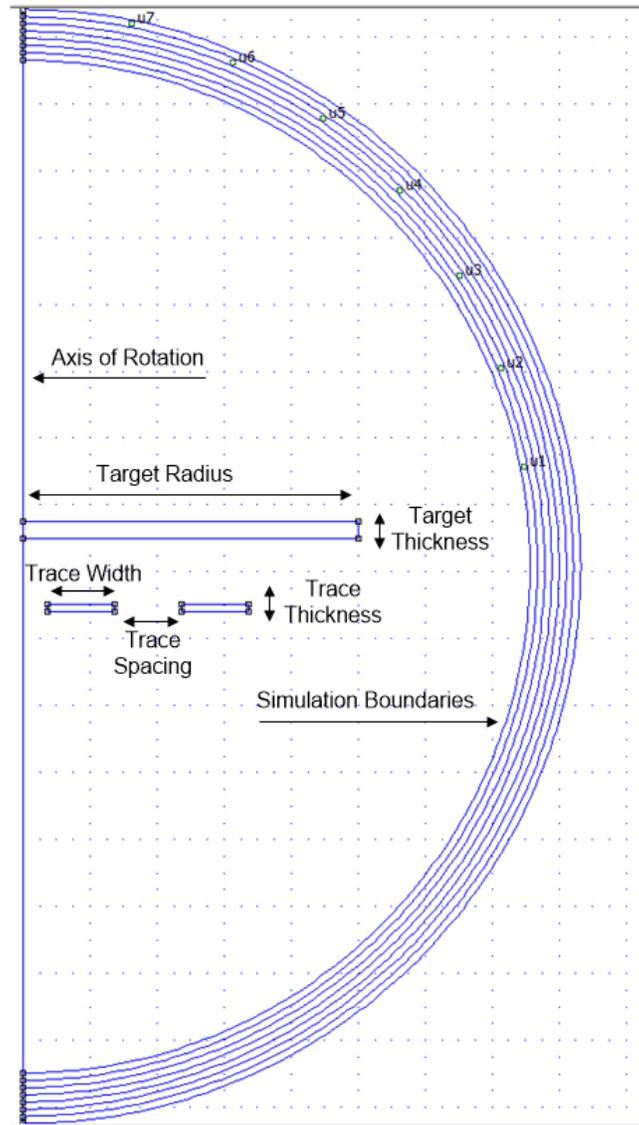

Figure 2. Example Planar Problem Workspace With Labeled Traces and Target

b. Axisymmetric problems:

Axisymmetric problems rotate around a user-specified axis. This type of problem is the most useful for inductive sensing simulations and should be used for all circular coils. Note that coils simulated with this problem type will not be true coils, but will be a series of concentric circles. Even so, the coils are fairly accurate simulations.

Figure 3. Example Axisymmetric Problem Workspace With Labeled Traces and Target

4. Define the problem.

a. Navigate to the **Problem** menu shown in Figure 4.

b. In the **Problem Definition** dialog box, choose planar or axisymmetric for the **Problem Type**. If "planar" is selected, enter the depth of the problem expansion in the **Depth** field.

c. Enter the appropriate length units for the coil design in the **Length Units** field. Typically mm or mils are most useful.

d. Define a Frequency (Hz). All magnetic problems must have a defined frequency. For inductive sensing problems, this is comparable to choosing a capacitor for a known inductor.

Use the **Spiral Inductor Designer** in **LDC Tools** to calculate a frequency and enter that frequency in the problem **Frequency (Hz)** field. If the FEMM simulation includes a target, use the sensor frequency with target interaction.

e.   In the **AC Solver** field, select **Succ. Approx**. Ignore the other fields and click **OK**.
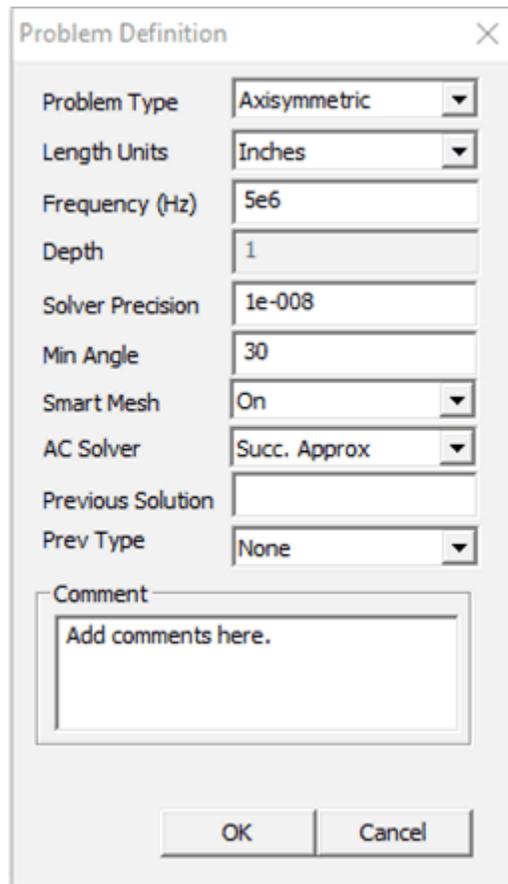

Figure 4. Problem Definition Dialogue Box

5.   Place nodes

a.   Use this button to place nodes. Use either segments or arcs to connect them. At least two nodes must be placed before placing segments. These will form the boundaries of your shape(s).

Use the trace width, trace spacing, and trace thickness from the **Spiral Inductor Designer** to create a cross section of each trace. Here are key tips for placing objects:

i.   It can be challenging to place nodes exactly in-line with each other. It may be easier to place one node, copy it, and specify X- and Y- translations for the new nodes. Note that the units are specified in the **Problem Definition** menu.

ii.   Click close to the desired nodes to place segments or arcs on the node.

iii.   Though the coils in this tutorial will use rectangular traces, the best approximation of real traces is a trapezoidal shape.

b.   Other helpful tips when moving objects:

i.   Right-click near an object to select it.

ii.   Double-right-click the object to show the X,Y position of the object.

iii.   Navigate with keyboard arrows.

iv.   Click the Page Up and Page Down keys to zoom in and out.

v.   Undo only works ONCE.

vi.   Click the button to move objects.

      vii. Click the [icon] button to copy objects.

      viii. Click the [icon] button to select a group of objects. Then click the [icon] button to highlight objects in a rectangular area or the [icon] button to highlight objects in a circular area. Hit the spacebar to join the objects into a group.

      ix. Click the [icon] button to scale

   c. For axisymmetric problems only: Mouse over the leftmost segment or arc or double-right-click the arc and look at the bottom left corner of the screen to determine the X,Y location of the object. Select all drawn objects and translate them left so that the leftmost segment or arc now has an x-value equal to the inner radius of the sensor coil. The axis of rotation will be the y-axis, so offsets in the x-direction will form gaps or holes in the object.

6. Define boundaries

   a. Click the [icon] button to open the **Boundary Builder**.

   b. The **Boundary Builder** builds the model, shows the axis of symmetry, and defines several boundaries that form a gradient to infinity.

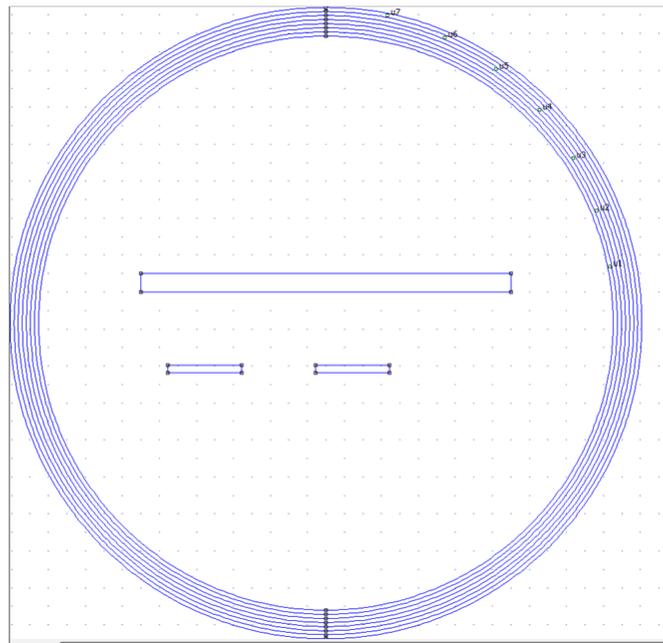   c. Leave the values in the **Open Boundary Builder** dialog box unchanged and click **OK**.
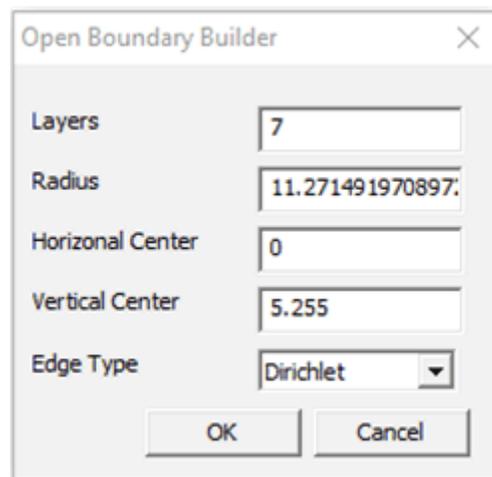
Figure 5. Boundary Builder Dialogue Box

   d. The workspace should now look similar to Figure 6 for axisymmetric problems or similar to Figure 7 for planar problems:
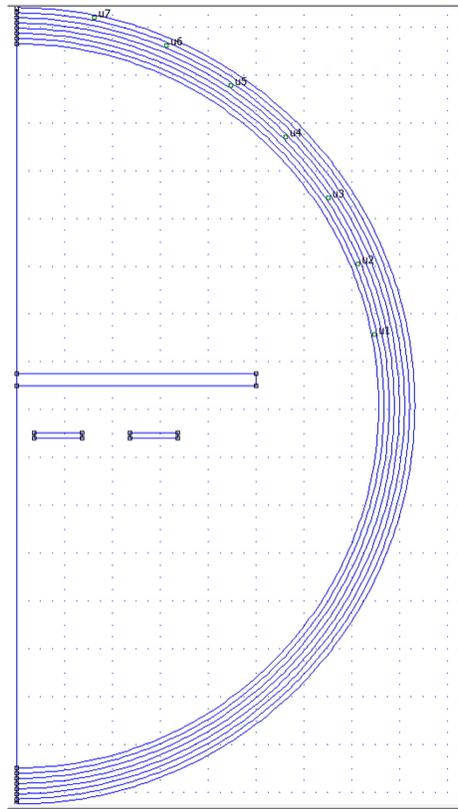
Figure 6. Axisymmetric Problem With Boundaries

Figure 7. Planar Problem With Boundaries

7. Define materials

a. Select the circular green button ⊚ to place block labels. Place a block label inside each boundary area, including the white space inside the outer boundary. The result should look something like Figure 8:

Figure 8. Axisymmetric Problem Workspace With Block Labels

b. Each block label must be changed from <None> to a defined material. First, the materials must be added to the Materials Library. Navigate to **Properties → Materials Library.**
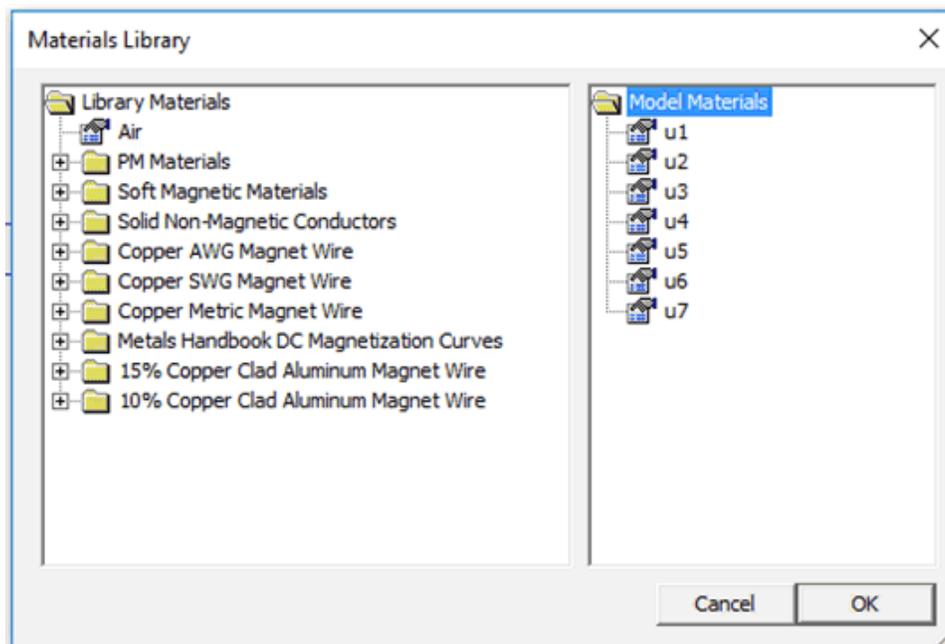
Figure 9. Default Materials Library

c. Typically, the engineer only uses air, copper, and the target material if the material is not copper. Drag **Air** from the left side to the right side of the menu. Open the **Solid Non-Magnetic Conductors** folder and drag **Copper** to the right side of the menu. Click **OK**.
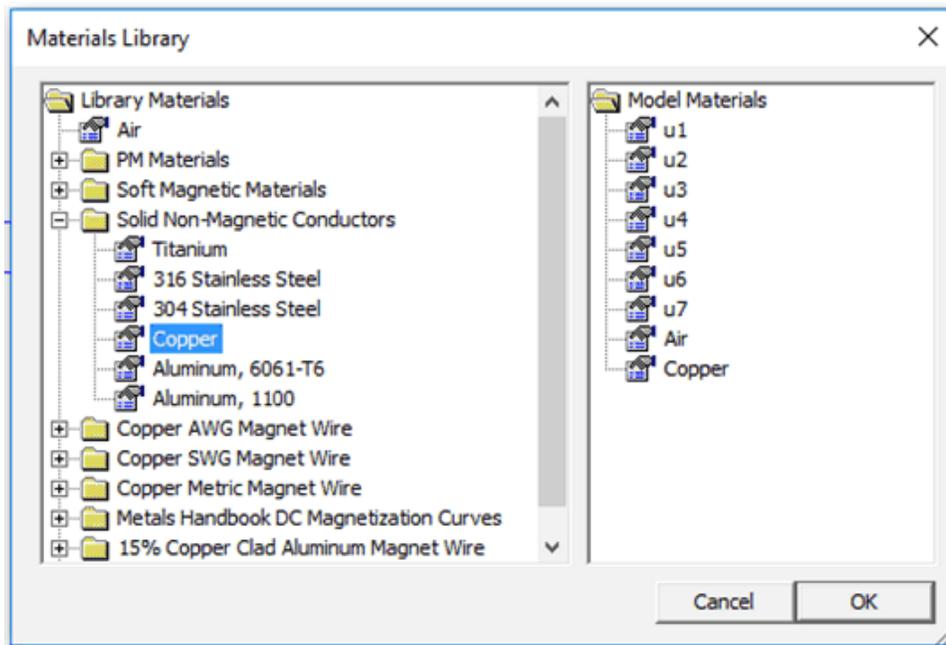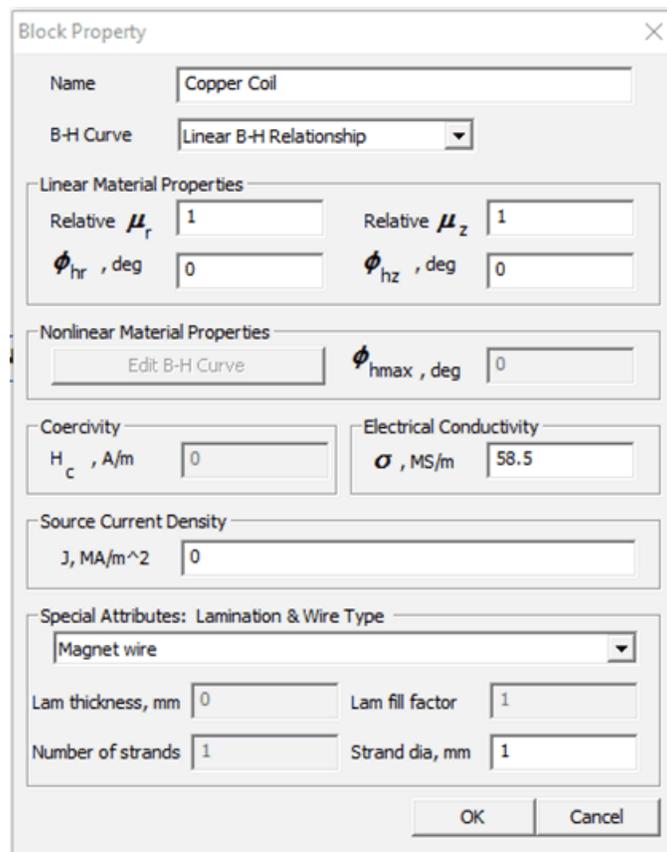

Figure 10. Materials Library With Air and Copper Added to Model Materials

d. The copper coil requires a special material type. Navigate to **Properties → Materials**. In the **Property Definition** menu, click **Add Property** to open the **Block Property** dialog box.

Figure 11. Block Properties Menu

e.

   i.   In the **Name** field, enter the material name **Copper Coil**.

   ii. In the **B-H Curve** drop-down menu, select **Linear B-H Relationship**.

   iii. The **Linear Material Properties** do not need to be modified. Copper is a conductor, therefore the material will have a relative permeability of 1.

   iv. Change the **Electrical Conductivity** field to **58.5 MS/m**.

   v. Change the **Special Attributes: Lamination & Wire Type** to **Magnet wire**.

   vi. In the **Strand dia, mm** field, enter the strand diameter based on the trace width and copper thickness of the real coil. Use Equation 1 to calculate the equivalent strand diameter:

     i.   The trace height and width form the cross section of the trace. The area of the equivalent copper wire must equal the cross sectional area of the PCB trace.

     ii. Solve for r in mm. Use 2r for the strand diameter and enter this value in the **Strand dia, mm** field.

     iii. This calculated diameter does not affect the size of the inductor nor the spacing. This is determined by the boundaries in the workspace. The strand diameter does affect the coil's series resistance.

$$TW = \pi r^2$$

- T is the trace thickness in mm
- W is the trace width in mm
- r is the strand radius in mm   (1)

f. Right-click near the material labels to highlight the names. Hit the spacebar to open the **Properties for selected block** menu. Change the Block type to the appropriate material for the area. Repeat this for all material labels.
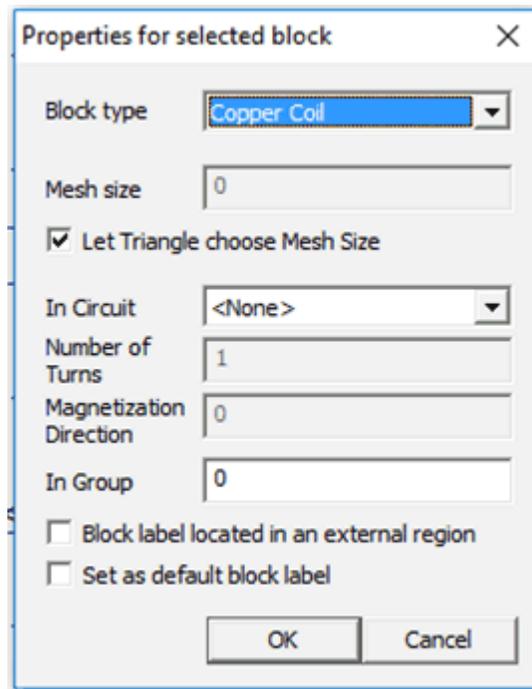
Figure 12. Properties for Selected Block Menu

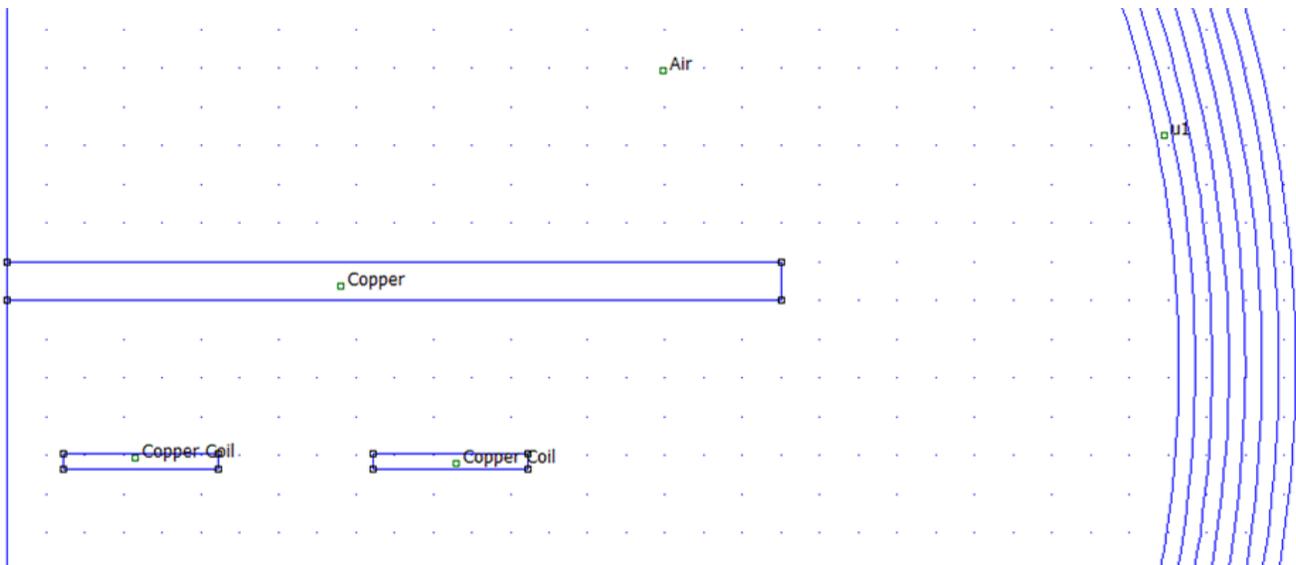   g. After these steps, the workspace should look something like Figure 13:

Figure 13. Axisymmetric Problem Workspace With Defined Materials

8. Add circuits to the problem
    a. Navigate to the **Properties → Circuits** menu. In the **Property Definition** dialog dox, click **Add Property**.
    b. In the **Circuit Property** dialog box, name the circuit and specify an operating current.
       i. The current value is not especially important. A value anywhere from 1 mA to 1 A is acceptable.
    c. Make sure the **Series** option is checked and click **OK**.
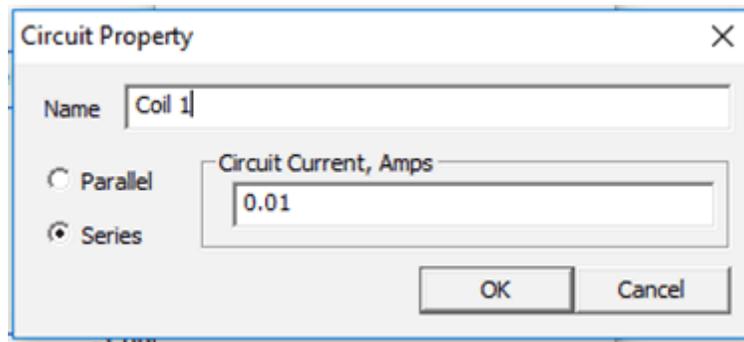
Figure 14. Circuit Property Menu

d. For planar problems only:

    i. For racetrack coils and similar circuits, the parallel traces are not automatically connected by the planar expansion. At least two separate circuits are required to correctly model the current flow.

    ii. To model the current flow, define two circuits with the same current magnitude. One should be positive and the other should be negative. Assign one to each side of the racetrack coil. Your workspace should look similar to Figure 15:
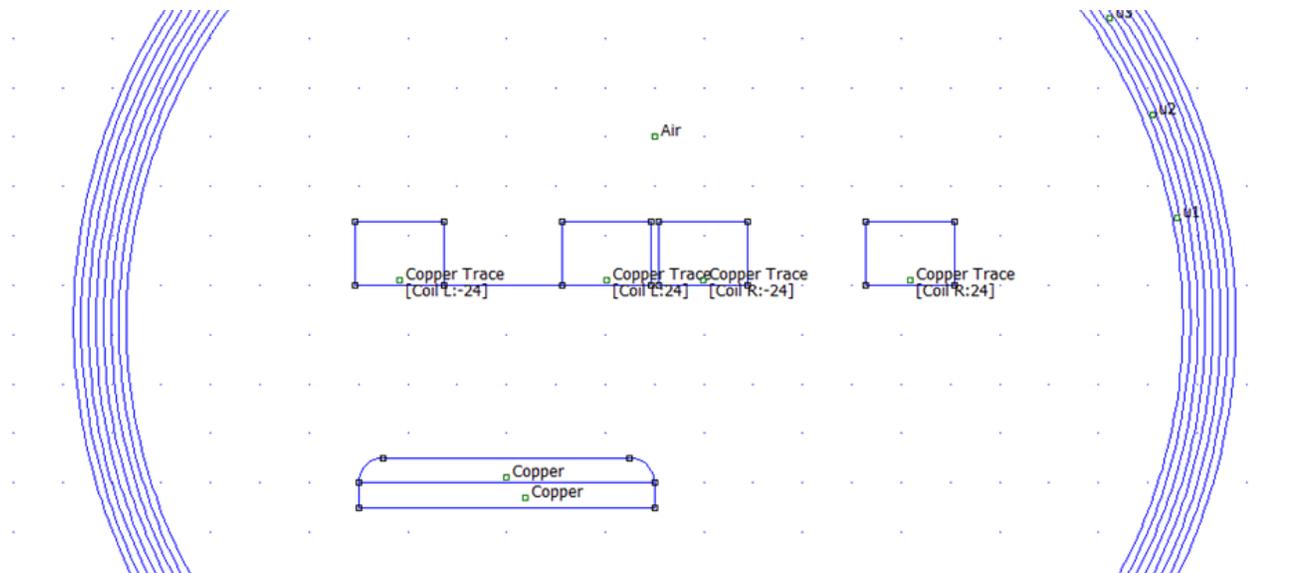
Figure 15. Example Planar Problem Workspace With Defined Circuits for Racetrack Coil

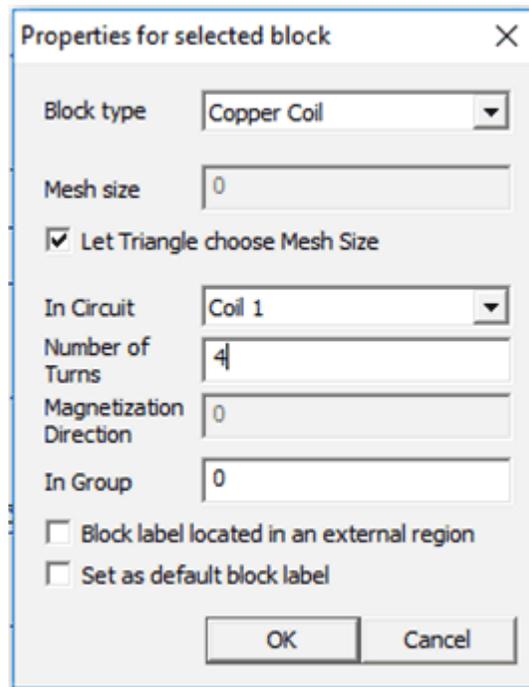e. Right-click on the material nodes again and hit spacebar to return to the **Properties for selected block** menu.

Figure 16. Properties for Selected Block Menu

f.
    i.   Select the circuit that you just created in the **In Circuit** field.
    ii. Specify the number of turns in the coil. This will not change the size of the coil and only
         change the generated magnetic field. The simulation will yield the most realistic results if only
         one turn is used per coil boundary and the number of coil boundaries is equal to the number of
         turns in the real coil.
    iii. Leave the **Let Triangle choose Mesh Size** box checked.

g. Repeat this process for all boundaries in the coil. The workspace should now look similar to
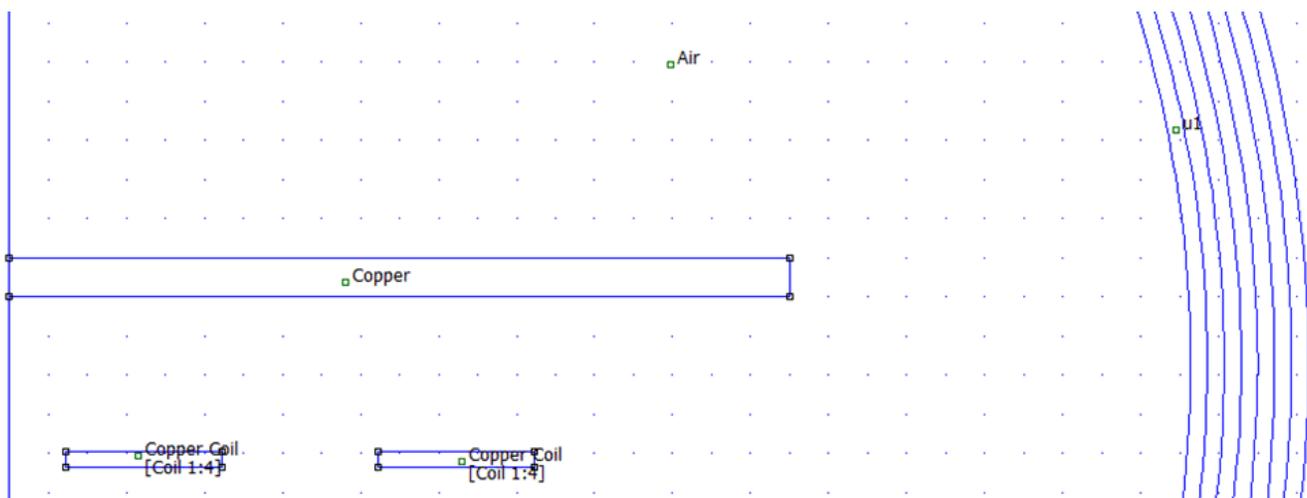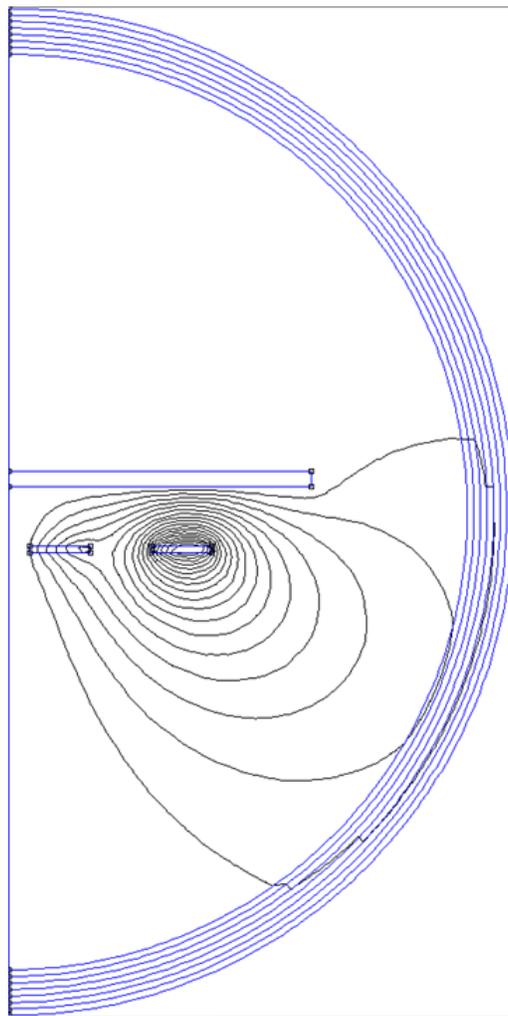   Figure 17:

Figure 17. Axisymmetric Problem Workspace With Defined Circuits

9. Generate the answer file

a. Click the **Solve** button [icon] to generate the answer file.

b. Answer files will be saved in the .ans file format in the current working directory.

3   Using the Electromagnetic Problem Answer File

1. Open the .ans file

    a. The .ans file will generate in the same directory as the .FEM file. Open the .ans file.

    b. Note that if any changes are made to the .FEM file, solving the problem again will overwrite the .ans file.

    c. The .ans file should show the magnetic field lines, which should bend around the target. The outer boundaries represent a gradient to infinity, so the magnetic field lines may contort in this region. Only the region contained by the innermost boundary line is of interest.

Figure 18. Axisymmetric Answer File Showing the Contour Plot of the Magnetic Field Around the Coil and the Target

2. Change the contour plot options

    a. Navigate to **View → Contour Plot**, or click on the **Contour Plot** button [icon]

    b. In the **Contour Plot Options**, change the number of contour lines, as well as their upper and lower bounds, and click **OK**.
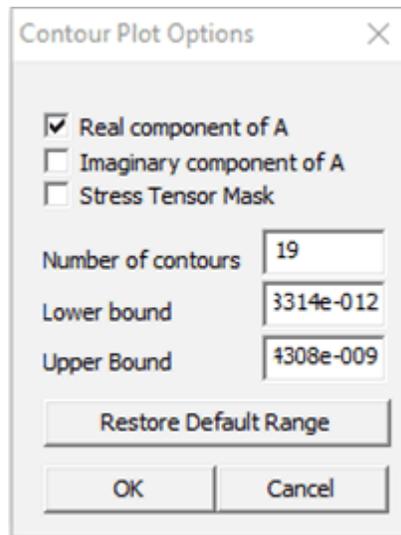
Figure 19. Contour Plot Options Menu

3. Add various density plots

    a. Navigate to **View → Density Plot**, or click on the **Density Plot** button

    b. In the **Dialog** menu, check the **Show Density Plot** box.

    c. Select the desired **Plotted Value**, generally |Flux Density| (T) or |Current Density| (MA/m^2). Flux density is useful for determining optimal target placement, because we want the target in the area of highest flux density. Current density is useful for displaying eddy currents.
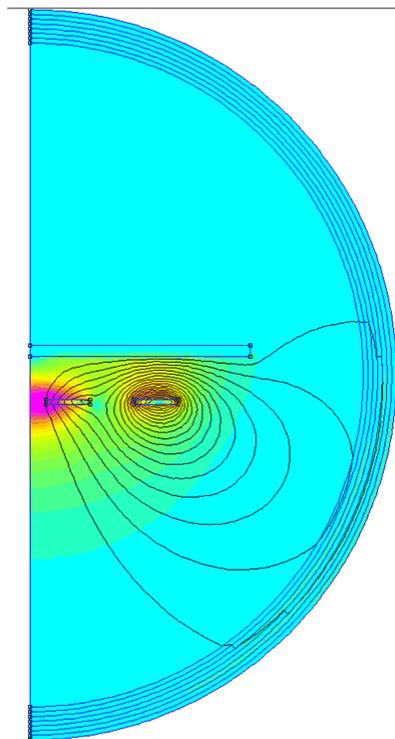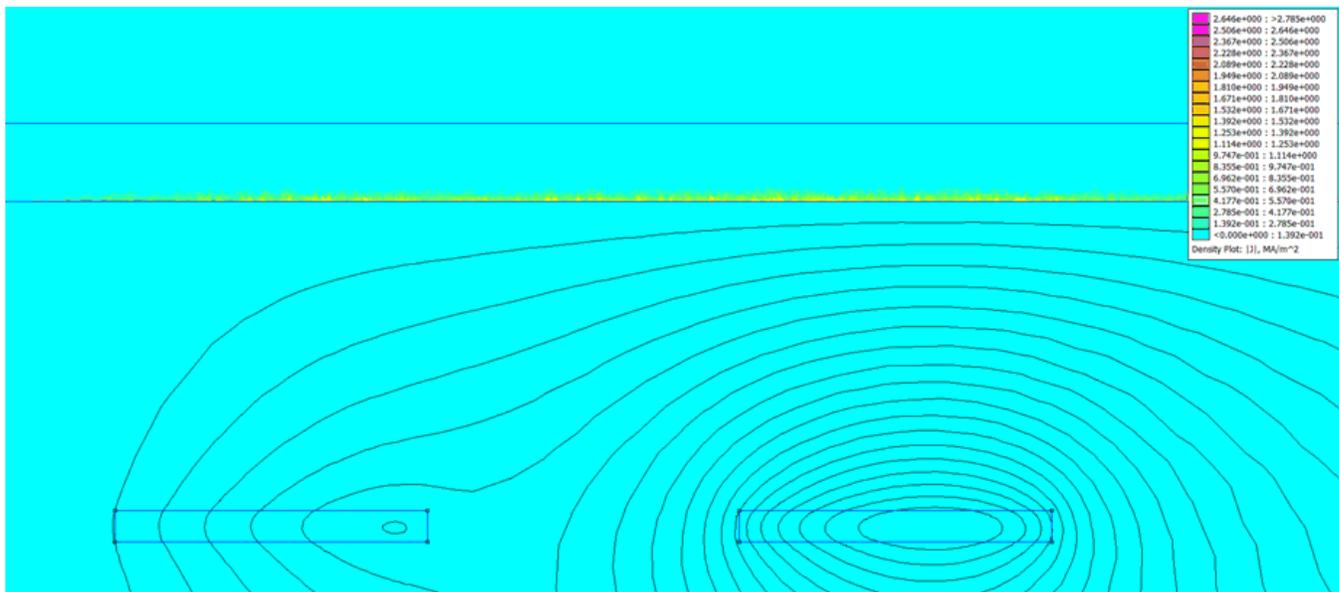

Figure 20. Axisymmetric Answer File Showing Flux Density Plot

Figure 21. Axisymmetric Answer File Showing Current Density Plot That Shows Eddy Currents on the Target

4. Change the mesh

   The mesh is one of the most important aspects in generating an accurate answer file. Generally, it is best to choose a small mesh around the coils, the target, and the axis of symmetry (for axisymmetric problems).

   a. To display the mesh, navigate to **View → Show Mesh**, or click the **Show Mesh** button

   b. To change the mesh, return to the .FEM file.

   c. To select the desired material, click the green materials button then right-click near the desired material node and hit the spacebar. The **Properties for selected block** menu should open.

   d. Uncheck the **Let Triangle choose Mesh Size** box

   e. Enter a new Mesh size. A mesh size around 0.01 is a good starting point. Note that the problem must be solved again to reflect any changes.

5. View the inductance of the coil

   a. Click the **Inductor** button at the top.

   b. The **Circuit Properties** menu should open. The inductance will be listed as Flux/Current.
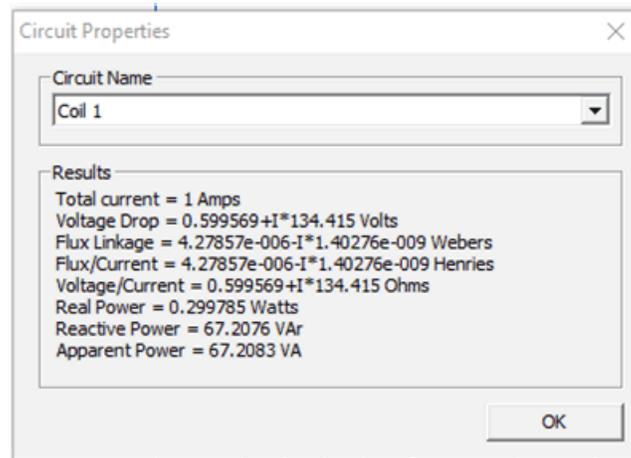
Figure 22. Circuit Properties Menu

4 Calculating Resolution

Most LDC devices measure shifts in inductance by measuring the resonant frequency of an LC tank. Their output codes are digital representations of the ratio between the LC tank sensor frequency and a reference frequency. It is most useful to calculate resolution in units of Hz for this reason. If the user has a desired resolution in units of distance, this distance should be the change in target distance during simulations. Ultimately, users want to determine the SNR for the frequency shift associated with the minimum detectable target movement.

1. Calculate the Frequency:
    a. A real inductive sensor will have a known capacitance, which should stay constant. Although FEMM accurately calculates the magnetic fields of the sensor (and therefore the change in inductance) at the given frequency, it does not have the option to incorporate a capacitor. By holding the frequency constant, it is essentially changing the capacitance value of the sensor. Calculate the change in frequency:
        i. For the first iteration, pick a frequency and a target distance and complete Part 1. Use the .ans file to calculate the inductance.
        ii. Use the known capacitance of the sensor and the newly calculated inductance to calculate the corresponding resonant frequency as shown in Equation 2:

$$f = \frac{1}{2\pi\sqrt{LC}}$$

(2)

        iii. Use the new frequency and the original target distance in FEMM to calculate a new inductance.
        iv. Repeat steps 2 and 3 until the inductance does not change significantly between iterations.
        v. Move the target by the minimum desired distance resolution and repeat steps 1-4. Determine the difference in calculated frequencies between the two target positions.
2. Calculate Resolution Using Simulation Results:
    a. Measure the Noise Floor:

        For inductive sensing applications, the first step is determining the expected noise floor for the system. Basic measurements can be done using the LDC evaluation modules, which can be purchased on ti.com. The accuracy of these measurements can be improved by exposing the evaluation module to the expected thermal range of the real-world application. The measurement accuracy can also be improved by removing the default sensor of the evaluation module and replacing the default sensor with a sensor designed for the real-world application. The default evaluation module, however, will still provide a reasonable noise floor estimate.

To measure the noise floor using an evaluation module, first download and open the Sensing Solutions GUI. For more detailed installation and use instructions, see the user's guide associated with the EVM.

   i. Open the GUI and navigate to the **Configuration tab**.

   ii. Configure the RCOUNT setting so that the LDC device will sample the sensor frequency at the desired rate.

   - For non-default sensors, the user must use an oscilloscope now to verify that the sensor oscillation amplitude and the sensor frequency are within the acceptable range for the specific device, as specified in the datasheet.

   - If the sensor oscillation amplitude is outside of the recommended range, the sensor drive current will need to be modified. Please see Setting LDC1312/4, LDC1612/4, and LDC1101 Sensor Drive Configuration for instructions.

   - If the sensor oscillation frequency is outside of the frequency range supported by the device, either the inductor or the capacitor in the LC tank will need to be modified.

   iii. Next, navigate to the **Data Streaming** tab.

   iv. Click the **Start Streaming** button to begin the noise floor measurement.

   v. Ensure that the graph displays **Detected Sensor Frequency (MHz)**, which is controlled by the drop-down menu in the upper left corner.

   vi. Click on the **Show Statistics** button to display the average and standard deviation of the data. Increase the number of decimals of the standard deviation until the last digit is rapidly changing. Use three times the standard deviation as the noise floor.

The noise floor measurements are more accurate when measured across the full temperature range that the final system will see. The measured data can be logged in a .csv file during temperature testing, and then the standard deviation can be calculated from the whole dataset. In general, the noise floor measurements will be more accurate the longer they are measured.

   b. Calculate the Final Resolution (SNR):

   To calculate the SNR, use Equation 3.

   - Remember that the SNR should be at least 10 for the desired resolution to be achievable in the real-world application.

   - Note that resolution significantly increases when the target is closer to the sensor. The FEMM simulations should test the resolution at both the minimum and maximum target distances.

   - Also note that very fine resolutions may not be achievable at high sample rates. For more information about how sample rate affects resolution, see the *Optimizing L Measurement Resolution for the LDC161x and the LDC1101* application report.

   - If the FEMM simulations show that the desired resolution is not viable, then the sensor design or the target design must be modified. LDC Sensor Design and LDC Target Design are good resources.

$$SNR \ = \ \frac{\Delta f}{3\sigma_N}$$

   - SNR is the signal-to-noise ratio of the inductive sensing system
   - Δf is the calculated frequency shift from
   - σ_N is the measured standard deviation of the noise from   (3)

5   Other Resources

For inductive sensing simulations, the Excel_FEMM tool can be especially helpful to speed up the simulation process.

For basic target shapes, the tool can automate the simulation process and will output the simulation results. It can also be useful to automate the creation of the coil in FEMM, and the resulting .fem file can be modified for custom target shapes. This tool is included with the LDC Tools Excel calculator.

For more general instructions and help using FEMM, please see the Finite Element Method Magnetics : Homepage.

6 Summary

This application note provides the step-by-step instructions for simulating inductive sensors using FEMM. The simulation results can be especially helpful to estimate system resolution and determine whether sensor design changes are needed before creating a prototype. Because the simulations are approximations of the real sensors, some differences between the real prototype and the simulation results are expected.

7 References

- D. C. Meeker, Finite Element Method Magnetics, Version 4.2 (28 Feb 2018 Build), http://www.femm.info

IMPORTANT NOTICE AND DISCLAIMER