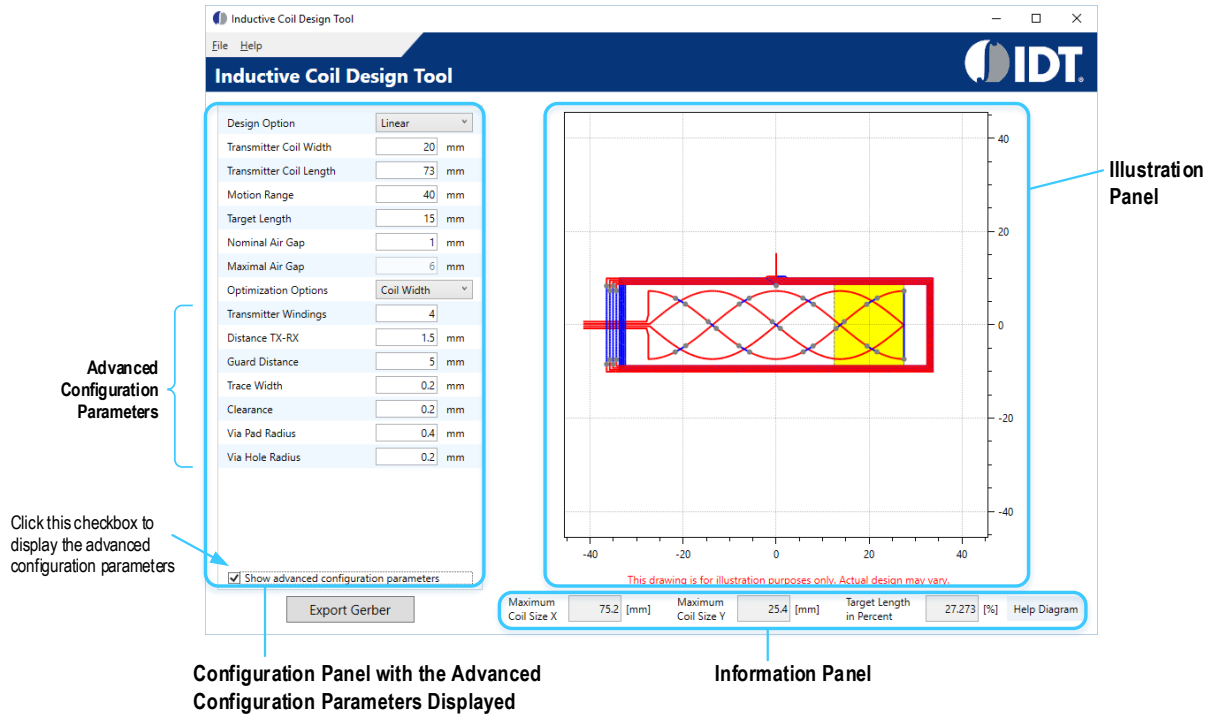




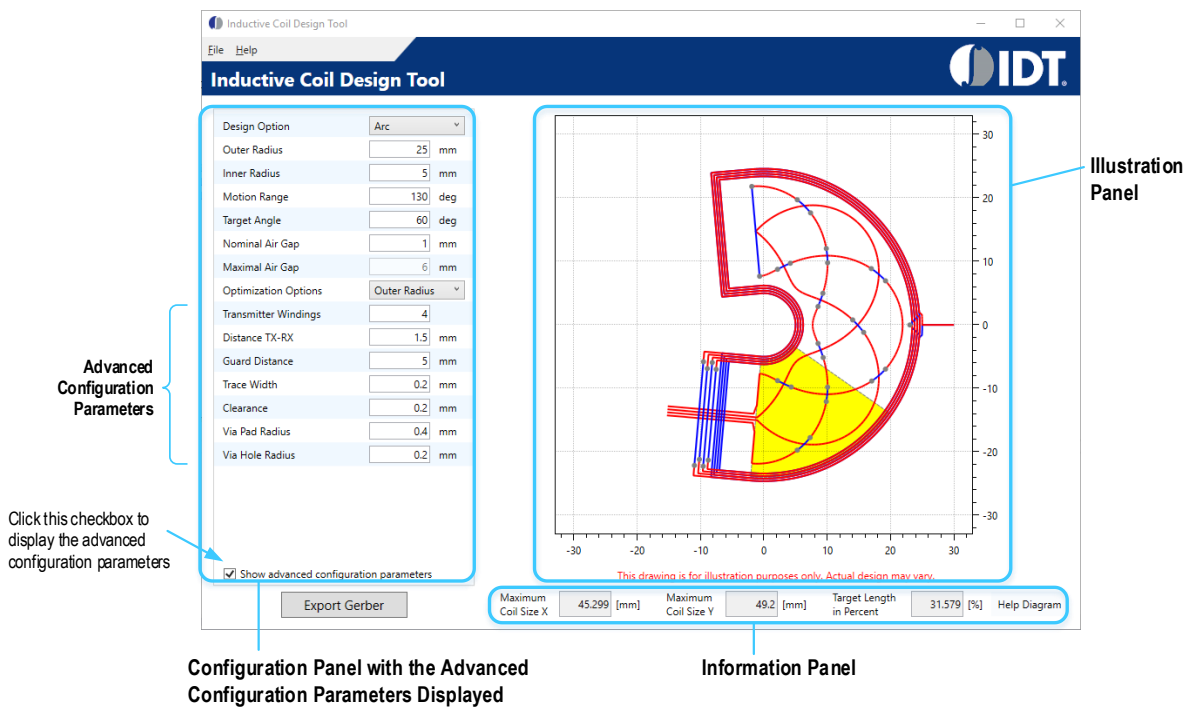


See Figure 1 for an example showing the main window for a linear position sensing coil design. Figure 2 is an example of the coil design for arc position sensing, and Figure 3 shows a coil design for rotary position sensing.

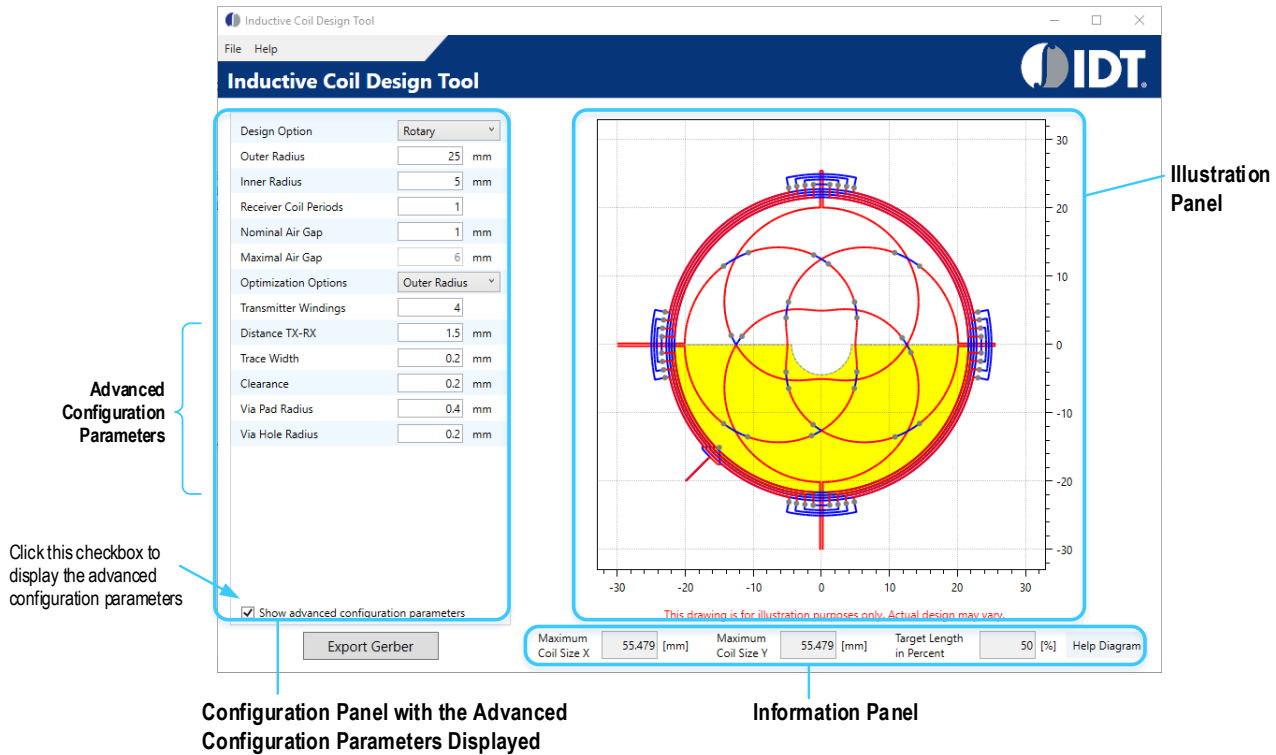
**Figure 1. Main Window – Linear Design Example**



**Figure 2. Main Window – Arc Design Example**



**Figure 3. Main Window – Rotary Design Example**



### 3.1 The Configuration Panel

The configuration parameters in the configuration panel are divided into two groups:

- Basic:
  - Transmitter Coil Width (linear coil designs only; see Figure 1)
  - Transmitter Coil Length (linear coil designs only; see Figure 1)
  - Outer Radius (arc and rotary coil designs only; see Figure 2 and Figure 3)
  - Inner Radius (arc and rotary coil designs only; see Figure 2 and Figure 3)
  - Motion Range (linear and arc coil designs only; see Figure 1 and Figure 2)
  - Target Length (linear coil designs only; see Figure 1)
  - Target Angle (arc coil designs only; see Figure 2)
  - Receiver Coil Periods (rotary coil designs only; see Figure 3)
  - Nominal Air Gap
  - Maximal Air Gap
  - Optimization Options

- Advanced (to view these options, check the “Show advanced configuration parameters” box):
  - Transmitter Windings
  - Distance TX-RX
  - Guard Distance (linear and arc designs only)
  - Trace Width
  - Clearance
  - Via Pad Radius
  - Via Hole Radius

### 3.2 The Information Panel

The maximum size of the design is calculated and shown in the fields “Maximum Coil Size X” and “Maximum Coil Size Y” in the information panel. These dimensions should be taken just as a general indication.

The “Target Length in Percent” field shows the size of the target as a percent of the length of one period of the receiver coil. Recommendation: Select a target length in the range of 20% to 80% of the length of one period of the receiver coil.

The “Help Diagram” button displays a diagram where most of the configuration parameters are defined. Examples of the different help diagrams for the three coil design types are shown in Figure 4, Figure 5, and Figure 6.

If the resulting design is not feasible, an error message might appear. Table 1 lists possible error messages.

**Table 1. List of Possible Errors**

Error Code	Error Text	Steps to Solve
11, 33, 55	"The design is too dense!"	Increase the size of the design.
31	"The whole structure cannot be bigger than 360 degrees!"	Reduce motion range or target angle of the design.
32, 51	"Receiver coil width too small! This is not a valid design."	Increase the size of the design.
34, 52, 56	"Outer radius too small! This is not a valid design."	Increase the outer radius.
35, 57	"Receiver coils overlap transmitter coils! This is not a valid design."	Increase the size of the design.
36	"Transmitter wires are too close! This is not a valid design."	Reduce the motion range or target angle of the design.
54	"Not enough space for connections for receiver coils!"	Increase the size of the design.
58	"Connections not properly placed."	Increase the size of the design.
71	"There are no transmitter coils!"	An error has occurred during generation; change the parameters of the design.

Figure 4. Linear Design Help Diagram

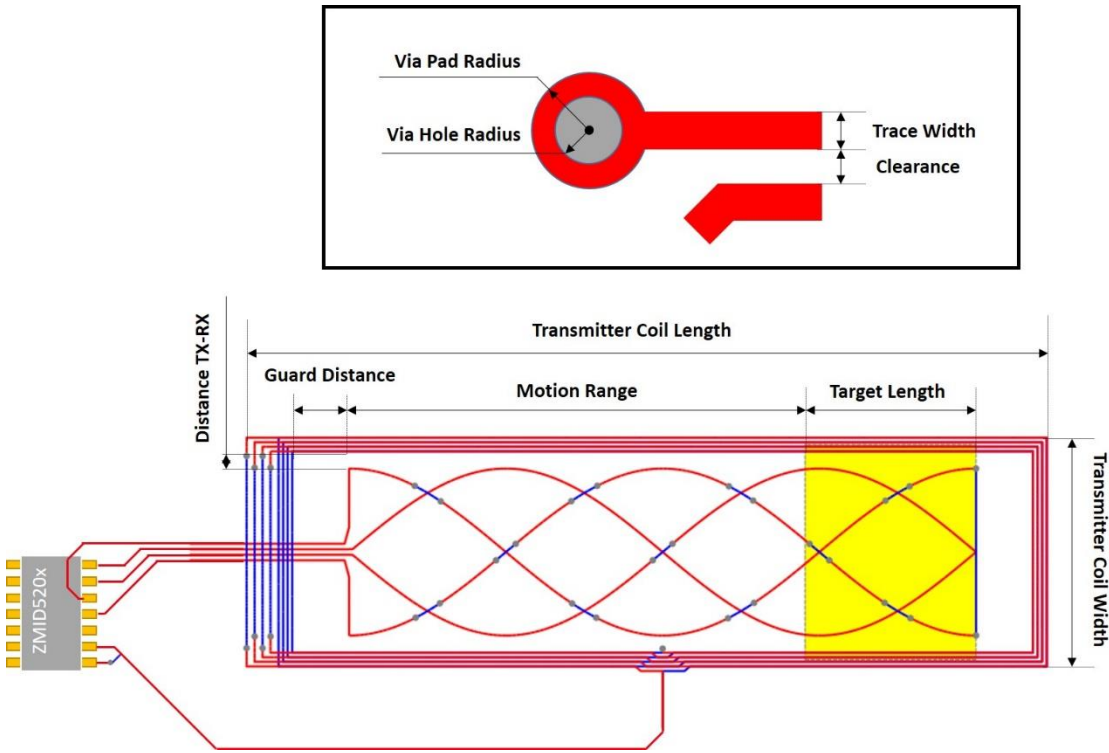
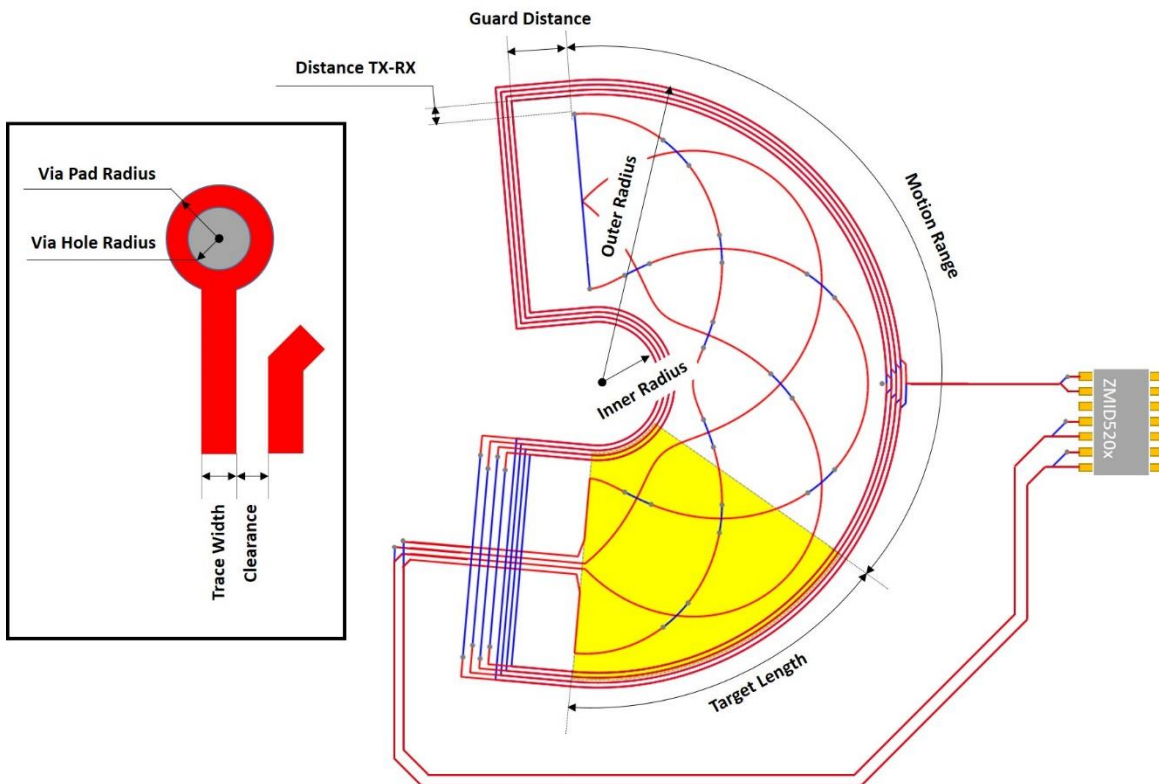
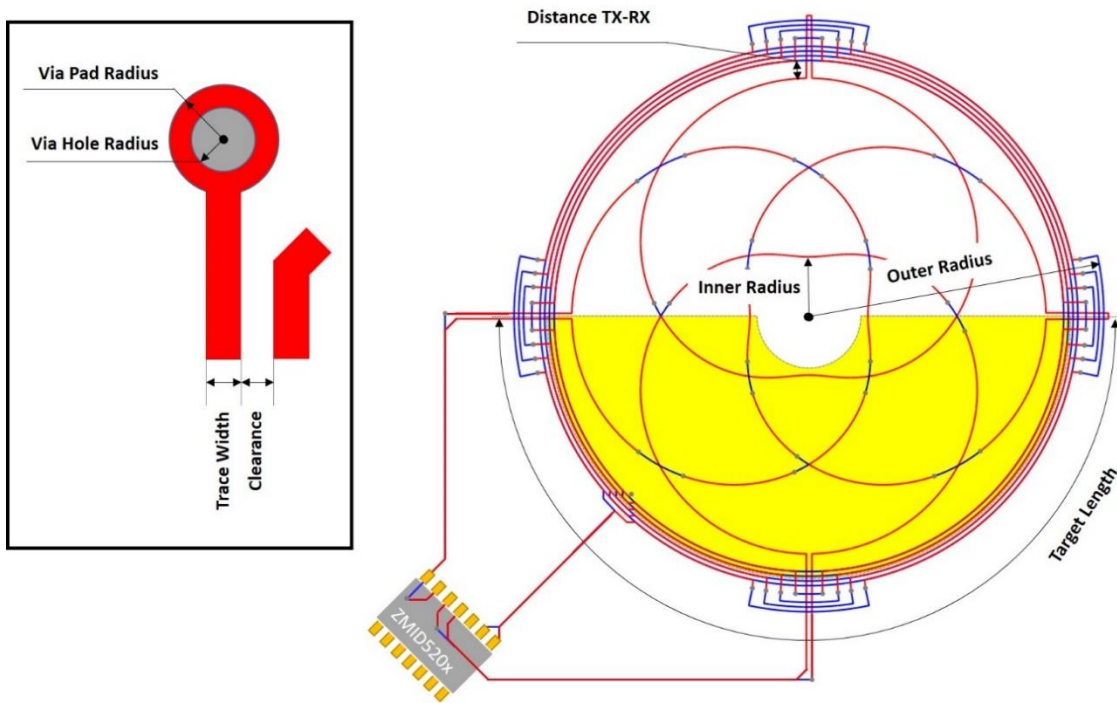


Figure 5. Arc Design Help Diagram



**Figure 6. Rotary Design Help Diagram**



## 4. Initial Set up

Follow these steps to configure the tool depending on the type of coil design:

1. Under the “Design Option” drop-down menu in the configuration panel, select the design option for the type of the position measurement: linear, rotary, or arc. The display changes according to the selected design option.
2. In the configuration panel, enter the design parameters according to the descriptions given in section 3.1. Ensure that the design requirements have been met.

## 5. Generation of the Design Files

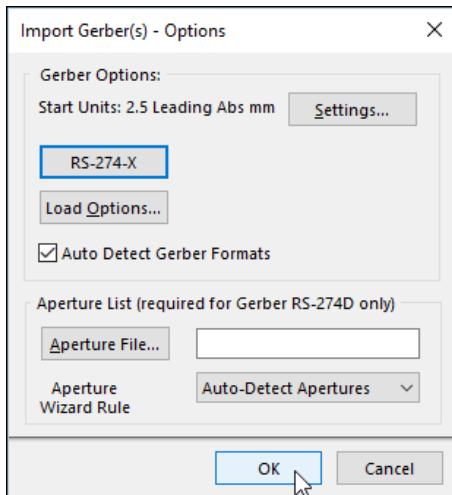
The design can be exported in two ways:

- Click the **File->Save Parameters** menu option. The values for all fields are exported as design parameters that can be imported later in the software by choosing “Import Design” from the “File” menu.
- Click the “Export Gerber” button. The list of coordinates is exported as Gerber files that can be imported in other software.

## 6. Converting the Gerber File to a \*.PcbDoc File in Altium Designer®

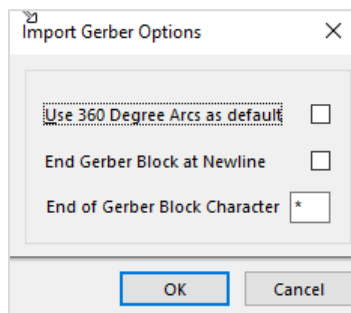
- Open Altium Designer® (trademark of Altium Limited) and create a new CAM document. Select **File -> New -> CAM Document**. Right click on the document and select **Save** to name and save it.
- Import the Gerber files in the CAM document: **File -> Import -> Gerber**.
- Go to the exported files directory and ensure that all file formats are displayed by selecting *All Files(\*.\*)* on the bottom right corner of the window.
- Select all files to be exported to the printed circuit board (PCB) including any Gerber \*.txt drill files.
- The “Import Gerber(s) Options” dialog window opens. Usually no additional changes are needed. An example for the “Start Units” value is given for reference in case previous settings have been made or the defaults are set differently. The Gerber file format should be RS-274-X. The “Start units” value and the options in the menu accessed by clicking the “Settings” button might differ in each case.
- After checking settings, click the “OK” button.

**Figure 7. Import Gerber(s) – Options Window**



If the file is not imported properly, change the file format to RS-274 or Fire9000 and check the options selected under the “Load Options” button as shown in Figure 8.

**Figure 8. Import Gerber Options Window**



- Click the “OK” button in the “Import Gerber(s) Options” window. This shows the log of the file loading. Check for any errors or warnings and then close the window.



- Open the CAM document and select **Tables -> Layers**. In the “Type” column, the Gerber drill \*.txt file will be a “Temporary” layer. Change it to a “Drill Top” layer.

**Figure 9. Example of the “Layers Table” Window**

#	Name	Type	Dr
1	linear_optimized_design.gbl	Bottom	
2	linear_optimized_design.gtl	Top	
3	linear_optimized_design.txt	Temporary	

- Click the “OK” button to close the “Layers Table” window. Select **Tables -> Layers Order**. Set the layer’s physical order in the “Layer Physical Order” column, and click the “OK” button.

**Figure 10. Example of the “Create / Update Layers Order” Window**

#	Layer Name	Layer Logical Order	Layer Physical Order
1	L1: gbl	1	1
2	L2: gtl	2	2

- Go to **Tables -> NC Tools** to open the “Tool Table” window (see Figure 11). In the “Size” column, manually enter the size in millimeters for the first tool and press the “Enter” button or click on another cell to update the table. The code in the “Ref. DCode” column is generated automatically. Click on the *Ref. DCode* cell to select the exact tool. Check the “Plated” box if it is not checked. Click the “OK” button to close and save the table.

**Figure 11. Tool Table Window**

#	Size (mm.)	Comp. (mm.)	Feed (mm/Sec.)	Speed (RPM)	Z-Axis (mm)	Plated	Ref. DCode	Comments
1	0.1500	0.1500				<input checked="" type="checkbox"/>	9500 Round	
2						<input checked="" type="checkbox"/>	9500 Round 0.1503	0.1503
3						<input checked="" type="checkbox"/>		

- Generate a netlist by selecting **Tools -> Netlist -> Extract**.
- If a problem occurs with the drill layer, try building a drill layer. First, select **View -> NC Editor**. Build the layer by selecting **Place -> Drill -> Build Drill Layer (PTH)**. If this is done, the hole size should be set from *PCB inspector* in the generated PCB.
- When the netlist is extracted, the file is ready to be exported to a PCB. Select **File -> Export -> Export to PCB**. The new \*.PcbDoc file is generated in the same directory as the CAM document, and its name is the same as the CAM file.

## 7. Revision History

Revision Date	Description of Change
February 23, 2018	Initial release