License Notice

IDT's *Inductive Coil Design Tool* software (the Software) is licensed for use solely in designs that will employ an IDT inductive position sensor (IPS) IC, and any other use will constitute a material breach of the license. Further, the licensee shall not himself/herself, and shall not authorize or permit any other third party to, (i) copy or use the Software in any manner not expressly authorized herein; (ii) distribute, sell, license, provide or otherwise make available the Software to third parties; (iii) use the Software to perform services for, or otherwise on behalf of, third parties, whether on a service bureau, SaaS, time sharing basis or otherwise; (iv) reverse engineer or decompile, decrypt, disassemble or otherwise reduce the Software or any portion thereof; (v) modify or create derivative works based upon the Software; (vi) use the Software or any portion thereof in any way that is in violation of any applicable laws; (vii) defeat or circumvent any access or security controls of the Software; or (viii) use any documentation provided with and related to the Software, except in support of authorized use of the Software.

Contents

List of Figures

List of Tables

---

1. Introduction

This document describes how to use IDT's *Inductive Coil Design Tool* software to design coils used in applications for the ZMID520x Inductive Position Sensor IC. There are three design options available: linear, arc and rotary.

Important: This design tool produces designs that are not optimized, and the results could have an approximate FS error up to 5% without linearization.

The linearization can improve the FS error by a factor of up to 0.5; therefore the expected FS error is approximately 2.5%. For linearization procedures, refer to the *ZMID520x Datasheet* and manuals for the ZMID520x evaluation kits.

The main features of the tool are visual representation of the coil designs, help diagrams, and Gerber files.

Recommendation: Read the *ZMID520x Datasheet* first.

2. Software Setup

2.1 System Requirements

In order to use the *Inductive Coil Design Tool* software, the following software is required:

- Microsoft Windows Vista® / Microsoft Windows® 7 / Microsoft Windows® 8 / Microsoft Windows® 10
- Microsoft .NET 4.5.1

2.2 Software Installation

The *Inductive Coil Design Tool* software is provided as a zip file on the www.IDT.com product page for the specific ZMID520x product:

- www.IDT.com/zmid5201
- www.IDT.com/zmid5202
- www.IDT.com/zmid5203

Follow these steps to install the *Inductive Coil Design Tool* software on the user's computer:

1. Navigate to the product page and download this file: *ZMID520x_ICDT_[YYYYMMDD].zip,* where YYYYMMDD is the date of the current version of the file. Extract the contents.

2. Activate the installation by clicking on the *setup.exe* file. Installation is automatic. If Microsoft .NET 4.5.1 is not installed on the user's computer, the setup wizard displays a message to install it from the Microsoft website. The setup wizard will automatically create a desktop icon that can be used to activate the *Inductive Coil Design Tool* software.

3. Activate the software. The main window is displayed as described in section 3.

3. Main Window

The main window is divided into three main sections: the configuration panel, illustration panel, and information panel.
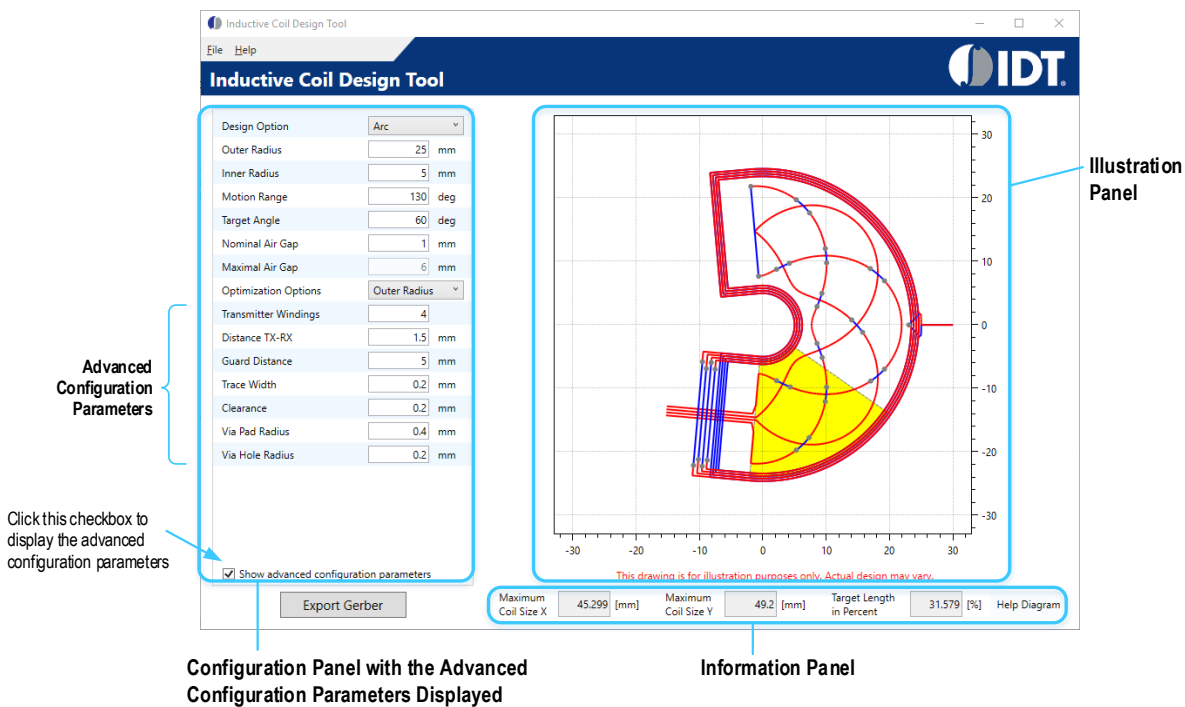
- The configuration panel contains the design parameter options. The "Design Option" drop-down menu in this section is used to select the coil design type.
- The illustration panel shows a drawing of the design for illustration purposes only.
- The information panel shows some of the basic design characteristics.

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

Configuration Panel with the Advanced Configuration Parameters Displayed

Information Panel
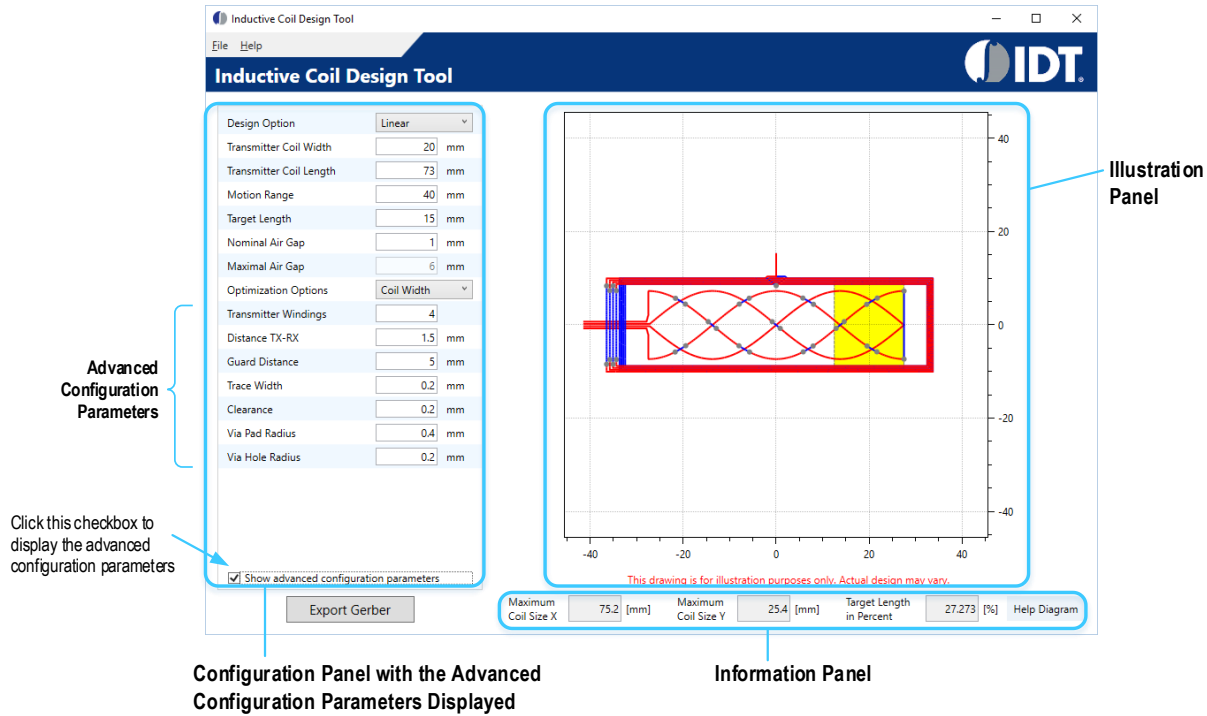
Figure 2.   Main Window – Arc Design Example

Configuration Panel with the Advanced Configuration Parameters Displayed

Information Panel

February 23, 2018

Figure 3. Main Window – Rotary Design Example
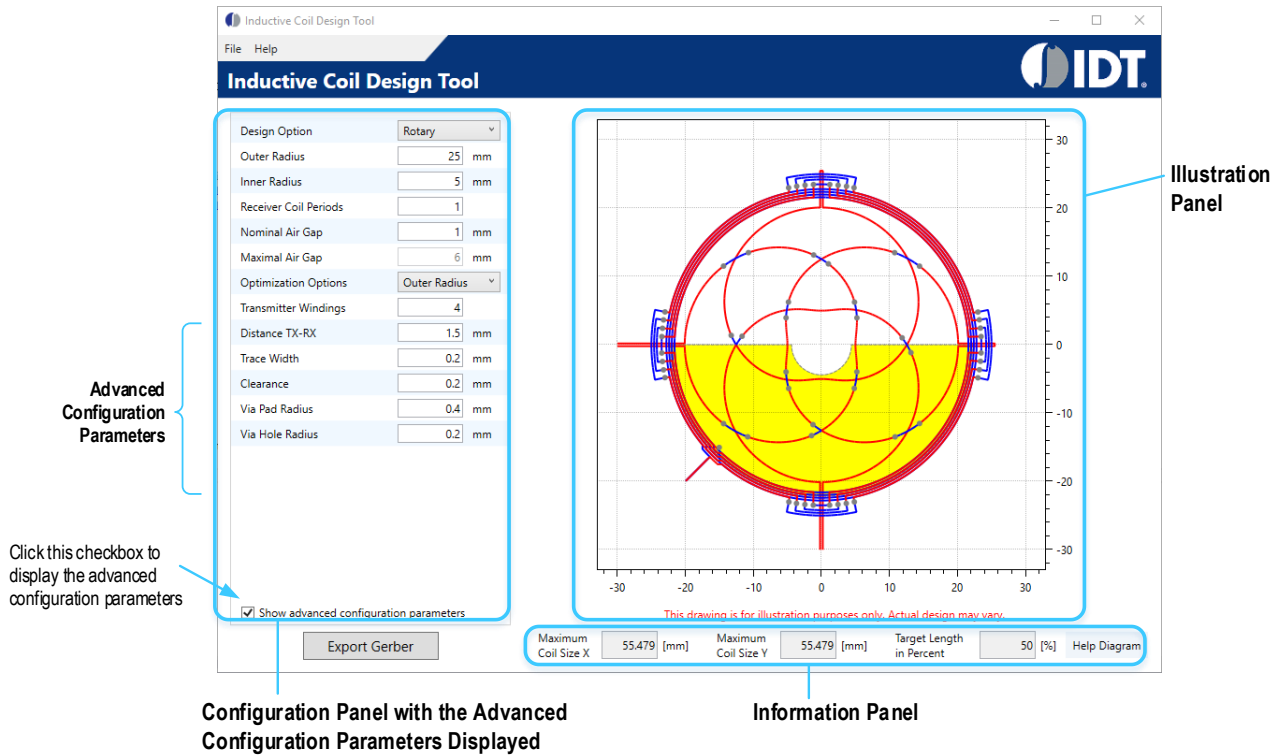
3.1 The Configuration Panel

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
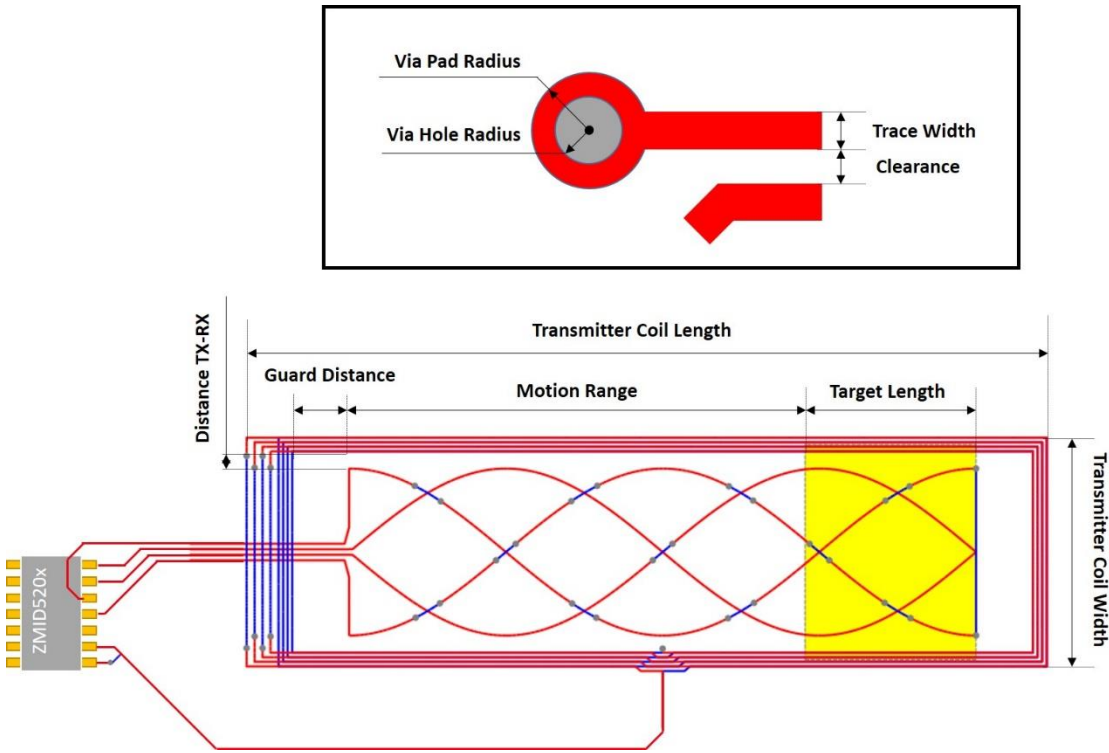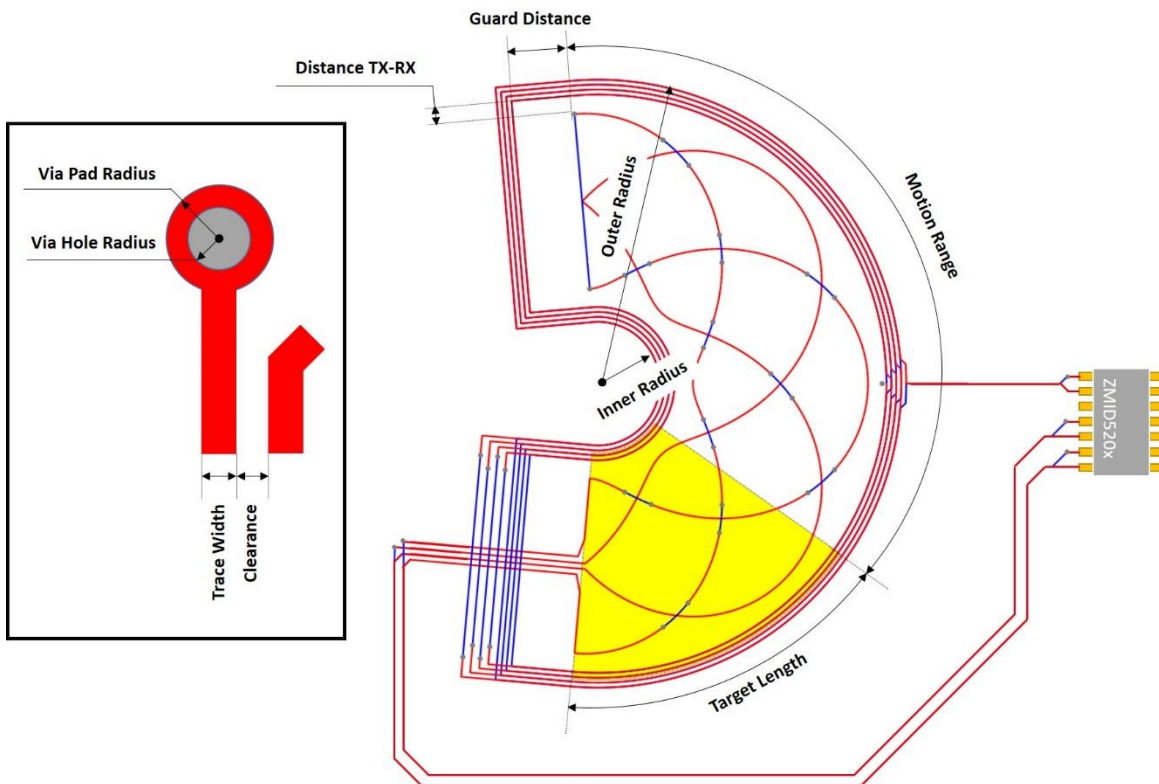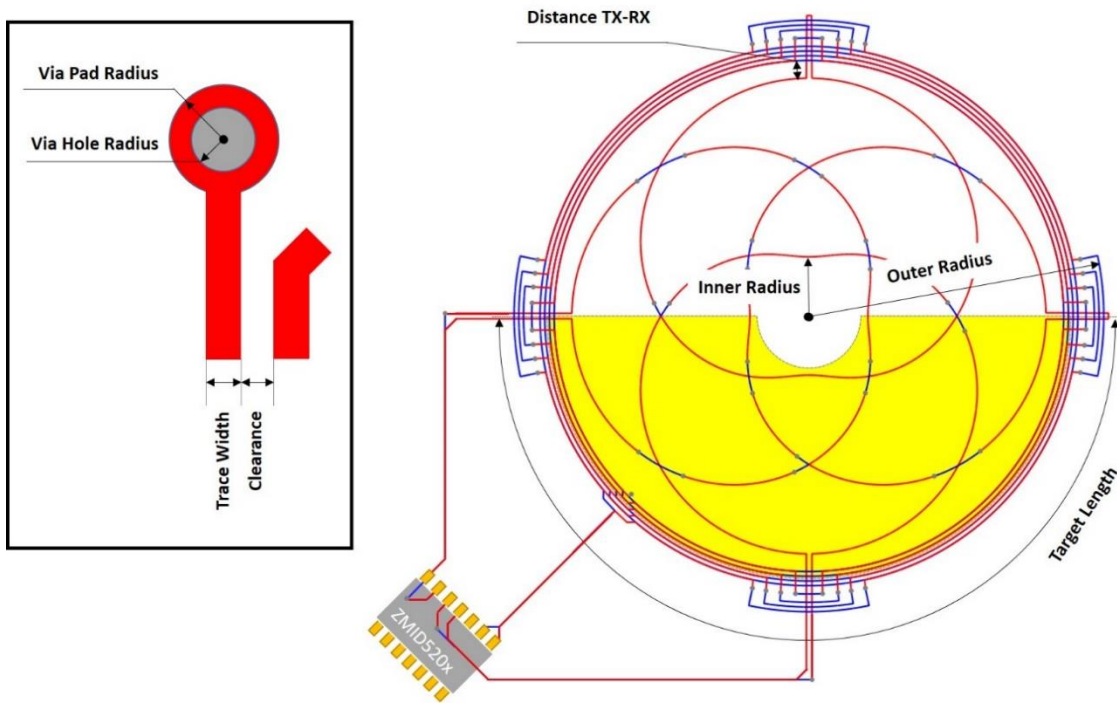

Figure 5.   Arc Design Help Diagram

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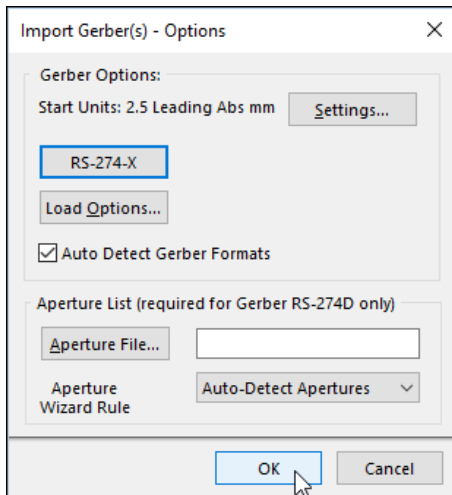
If the file is not imported properly, change the file format to RS-274 or Fire9000 and check the options selected under the "Load Options" button as shown in Figure 8.

Figure 8.   Import Gerber Options Window
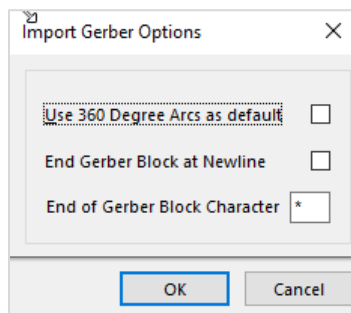
- Click the "OK" button in the "Import Gerber(s) Options" window. This shows the log of the file loading. Check for any errors or warnings and then close the window.

- 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

- 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

- 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

- 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

IMPORTANT NOTICE AND DISCLAIMER

RENESAS ELECTRONICS CORPORATION AND ITS SUBSIDIARIES ("RENESAS") PROVIDES TECHNICAL SPECIFICATIONS AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, WITHOUT LIMITATION, ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT OF THIRD-PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for developers who are designing with Renesas products. You are solely responsible for (1) selecting the appropriate products for your application, (2) designing, validating, and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. Renesas grants you permission to use these resources only to develop an application that uses Renesas products. Other reproduction or use of these resources is strictly prohibited. No license is granted to any other Renesas intellectual property or to any third-party intellectual property. Renesas disclaims responsibility for, and you will fully indemnify Renesas and its representatives against, any claims, damages, costs, losses, or liabilities arising from your use of these resources. Renesas' products are provided only subject to Renesas' Terms and Conditions of Sale or other applicable terms agreed to in writing. No use of any Renesas resources expands or otherwise alters any applicable warranties or warranty disclaimers for these products.

(Disclaimer Rev.1.01)

Corporate Headquarters

TOYOSU FORESIA, 3-2-24 Toyosu,
Koto-ku, Tokyo 135-0061, Japan
www.renesas.com

Trademarks

Renesas and the Renesas logo are trademarks of Renesas Electronics Corporation. All trademarks and registered trademarks are the property of their respective owners.

Contact Information

For further information on a product, technology, the most up-to-date version of a document, or your nearest sales office, please visit www.renesas.com/contact-us/.