# How To …
# Add Copper in MCADX

# January 20, 2024



## Desktop EDA
Brian Watson

Table of Contents

# 1: Overview

## How to ….

This document, and others like it, will demonstrate and show users various "how to" procedures using the MCADX plug-in for SolidWorks with Cadence PCB Editors.

## Add Copper in MCADX

**Note: The screenshots in this document show MCADX for SolidWorks 2023. The instructions in this document apply for all versions of supported SolidWorks, though some of the menus, dialogs or icon designs may be slightly different.**

This document will demonstrate and show the user how to add a copper shape in MCADX for SolidWorks and then send information about that copper shape to Cadence PCB Editors.

## Prerequisites

Note: It is assumed that both MCADX for SolidWorks and a Cadence PCB Editor have already been installed and are working properly.

The following is also assumed:
- That both MCAD and ECAD designers are familiar with the MCADX co-design process between SolidWorks and Cadence PCB Editors
- A baseline of the co-design project has been exchanged between the two design teams.
- That the net that is to be assigned to the copper shape in MCADX is a valid net in the PCB design on the ECAD side.

## Note

Copper objects can be transferred from ECAD to MCAD and MCAD to ECAD (as this document outlines), but copper objects are not collaborative. This means that while copper objects can be sent between domains, once the object is received, it cannot be edited or changed and have that change co back to the original source.

# 2: Adding a Copper Shape in MCADX

## Adding a Copper Shape in MCADX

Instructions for adding a copper shape in MCADX are as follows:

1. An assumption is made that a baseline file has been received by MCAD from ECAD and read in.
2. Zoom out the design in SolidWorks.
3. Tap on the keyboard **spacebar** to invoke the cubed view of the design as shown below.
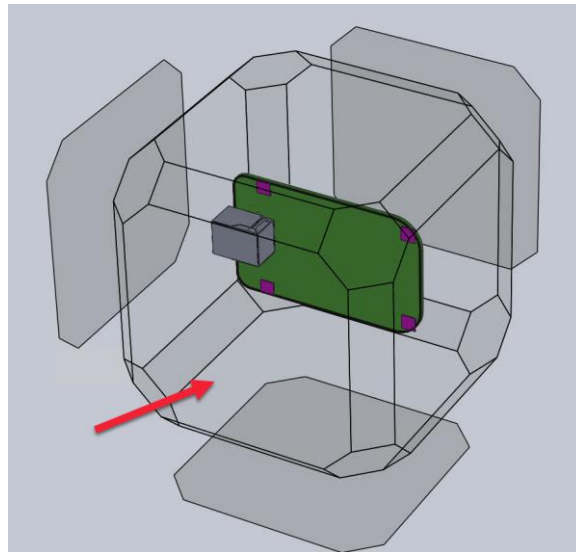


4. Click on the **Front Face** as shown by the red arrow in the screen shot above. The front face will turn blue when the mouse hovers over it.
5. The design will now change from an isometric view to a top view as shown below.
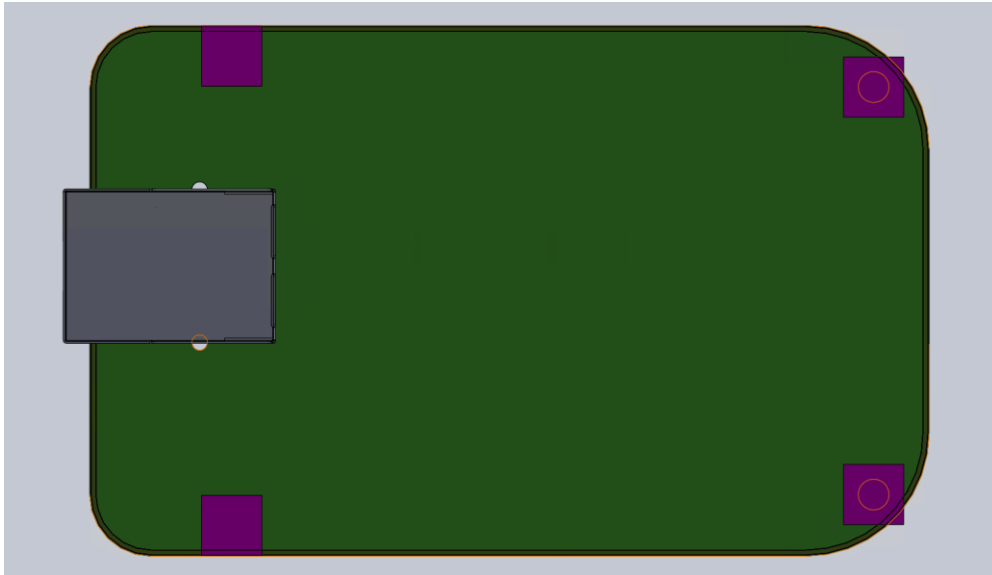
6. In SolidWorks, make sure the **MCADX** tab (shown below) is in focus. If it is not, then click on it to bring it into focus.
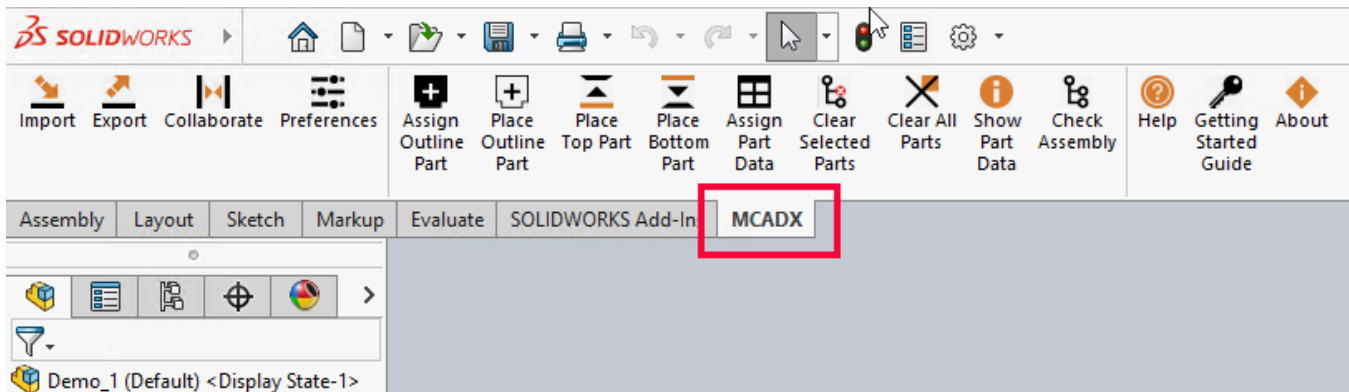


7. In the **Feature Manager Design Tree** view, expand the top-level assembly entry – by clicking on the small triangle to the left of the entry, locate the *Dielectric_P2_Demo_1* layer **(Note: In this example, the design name is Demo_1, hence why it is part of the layer name. In your design, this layer name will reflect the name of your design).** Click on the layer and **RMB** and select **Open Part** – the top left entry of the **RMB** context menu.
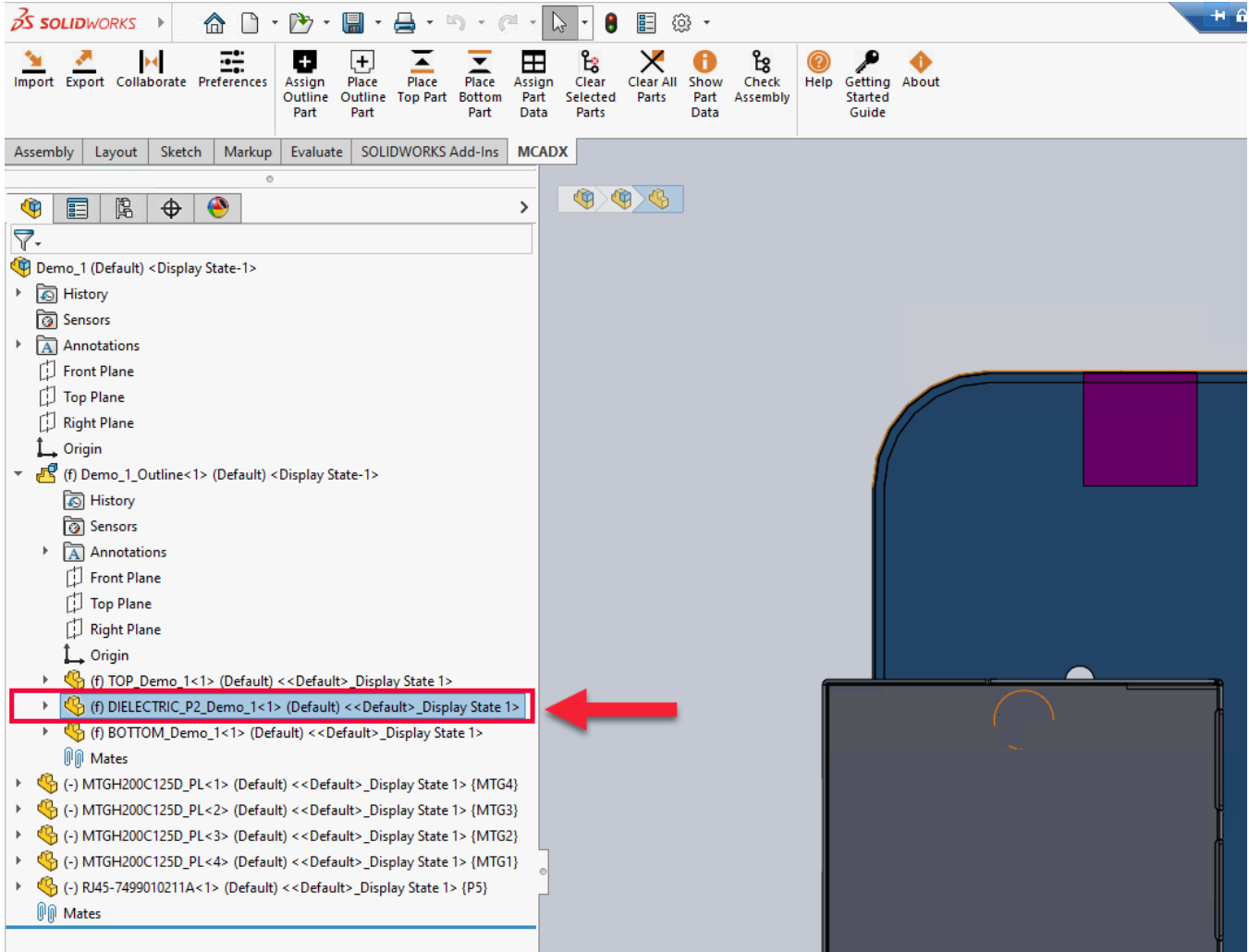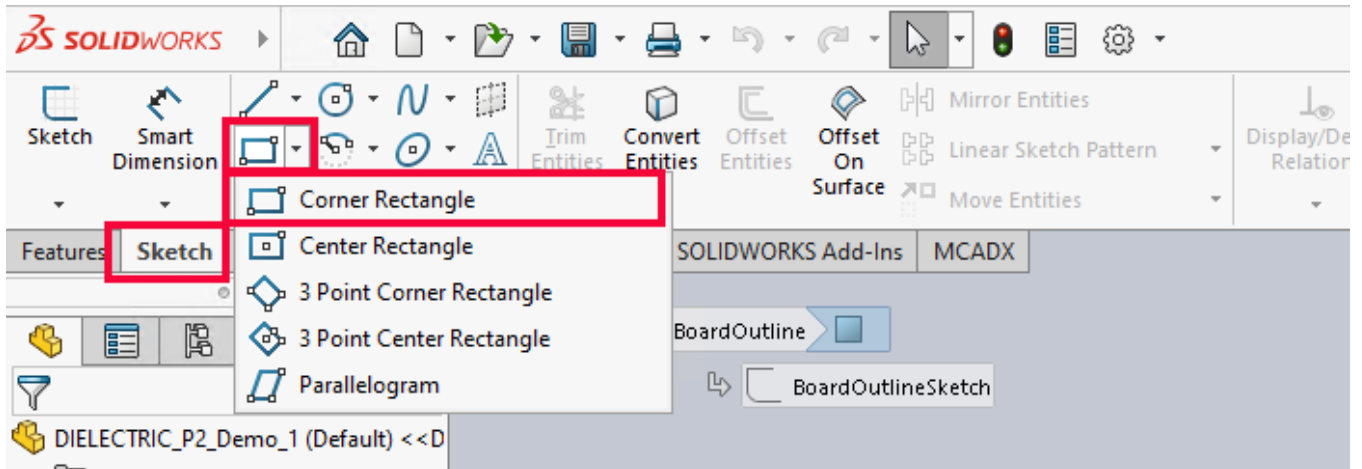
8. The canvas will now change to the dielectric part view as shown below.
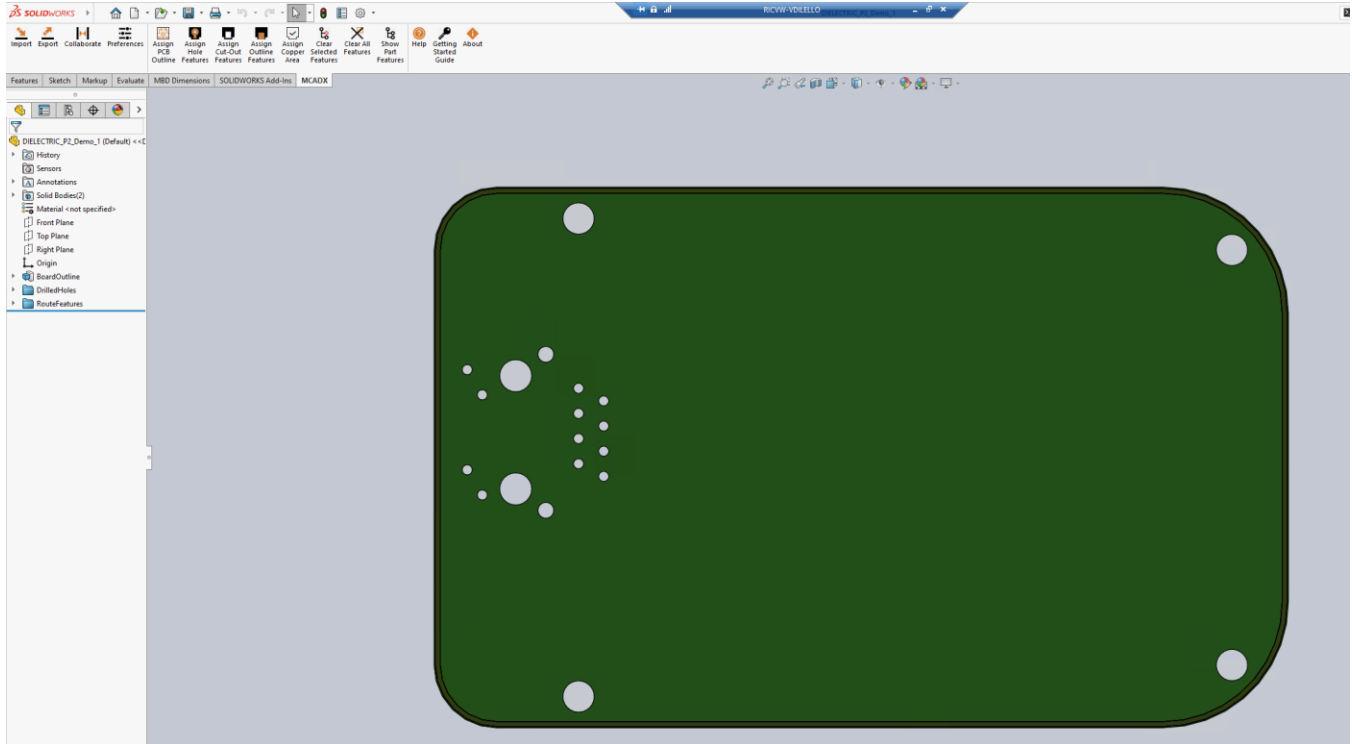
9. **LMB** click anywhere on the top face of the PCB design. I clicked where the red arrow is pointing to in the screen shot below. The top face of the PCB will turn in a different color – in this example – blue.
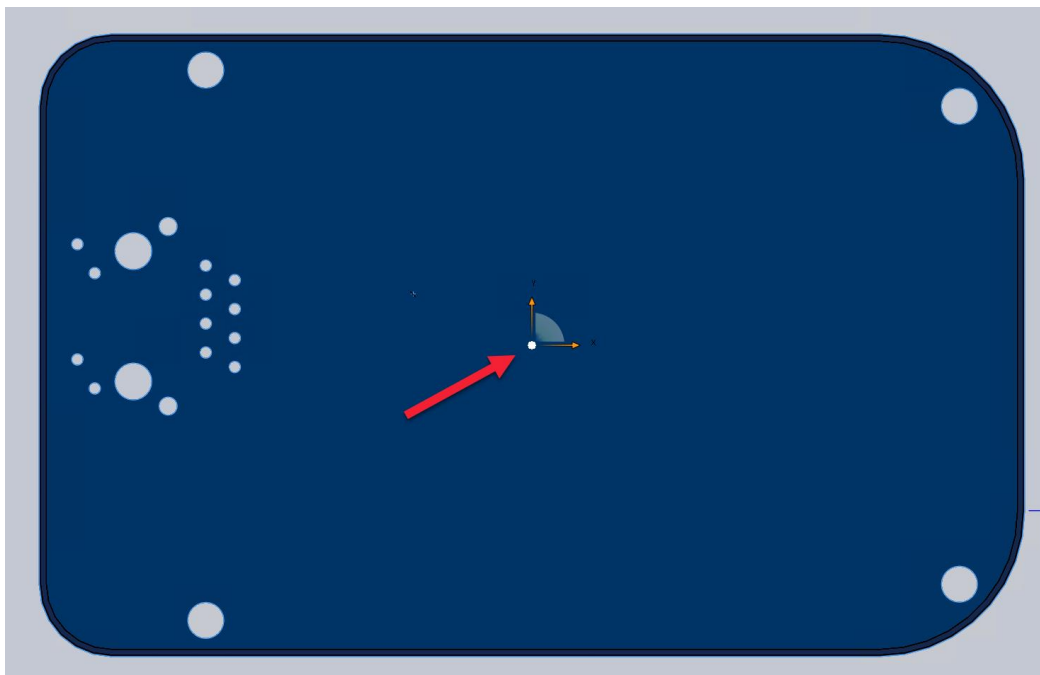
10. Next, select the **Sketch** menu and select a shape. In this example the **"Corner Rectangle"** was chosen.



11. Draw a shape on the PCB canvas. ***Note: While a rectangle is being shown as an example in this App Note, any shape can be drawn on the PCB using any of the Sketch drawing objects.*** A rectangle was drawn by clicking on spot **#1** and dragging the mouse to spot **#2** and releasing the mouse.
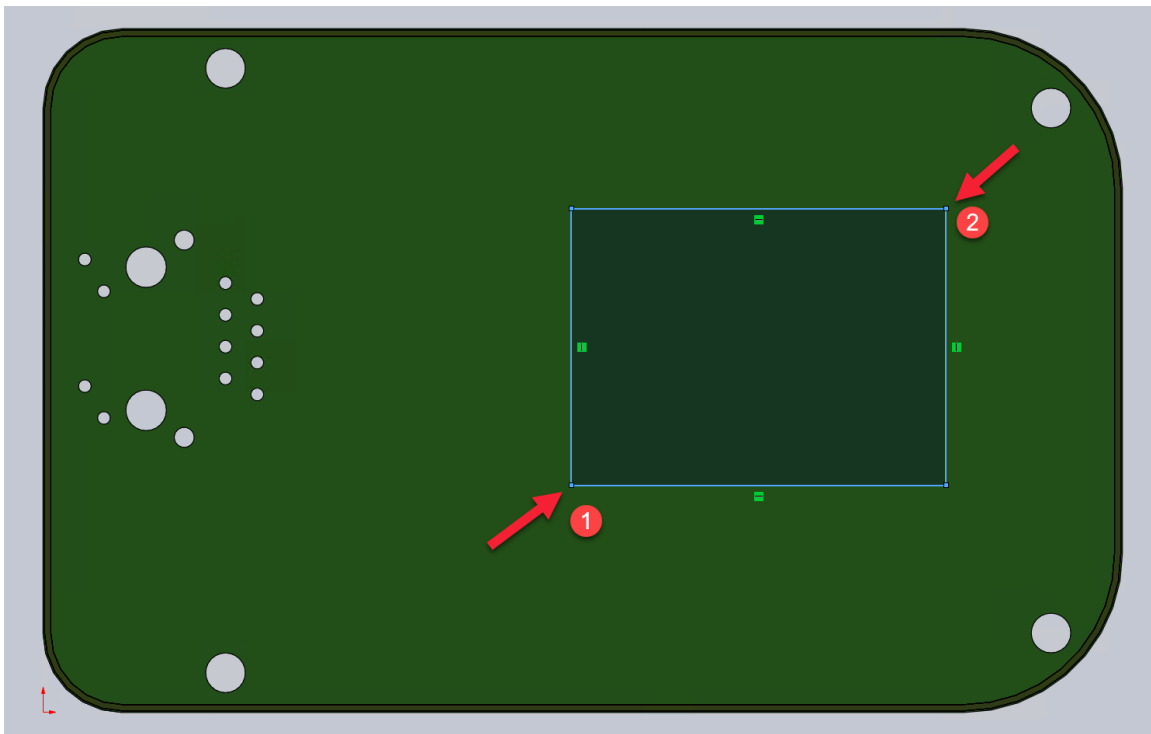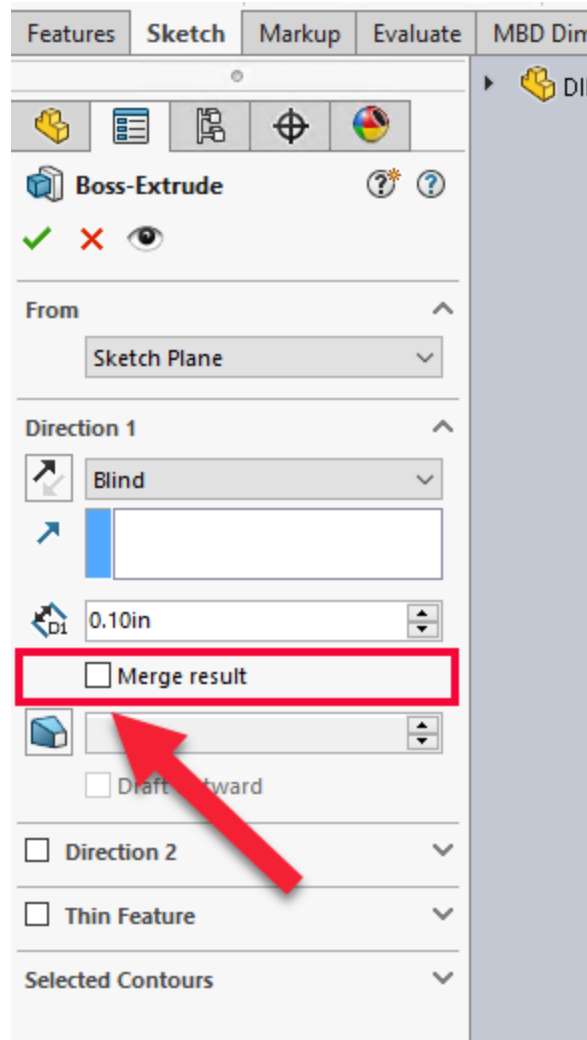
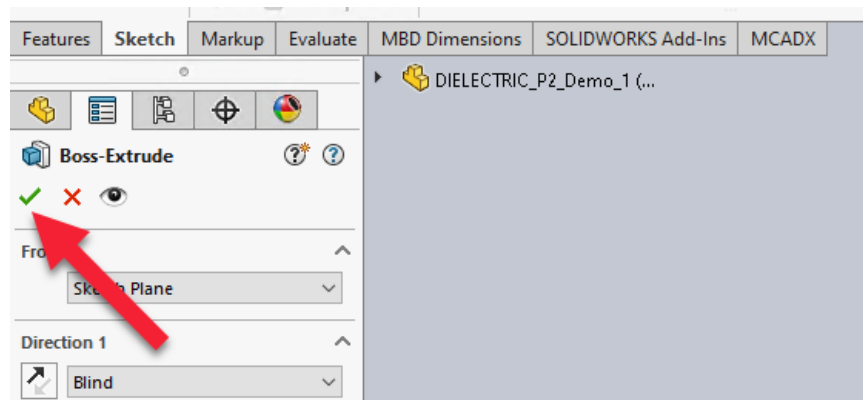12. Next, in the menu, select **Insert > Boss/Base > Extrude**.
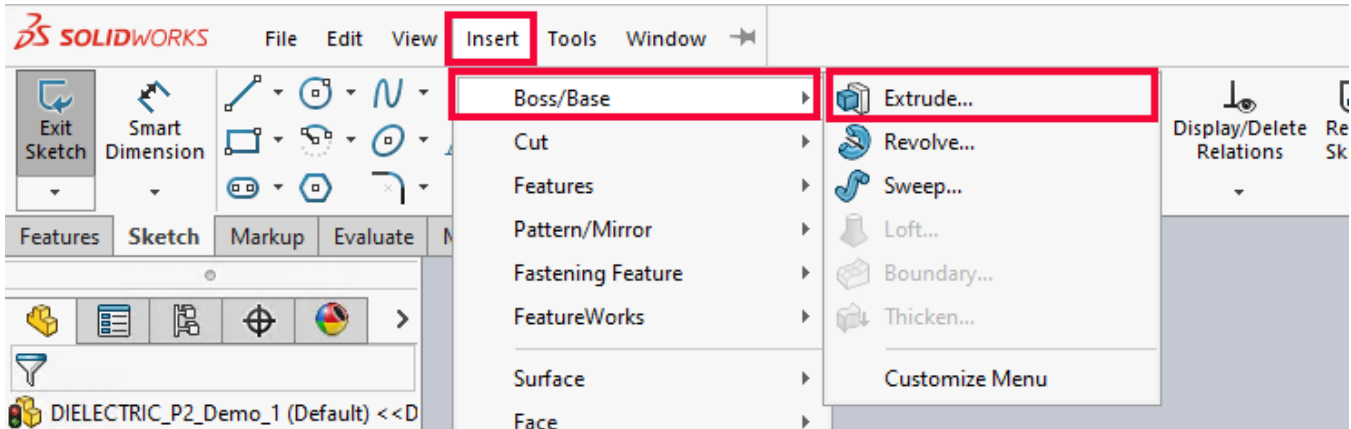


13. The just added shape will be selected on the canvas, and the pane on the left will change to the **Property Manager** pane.
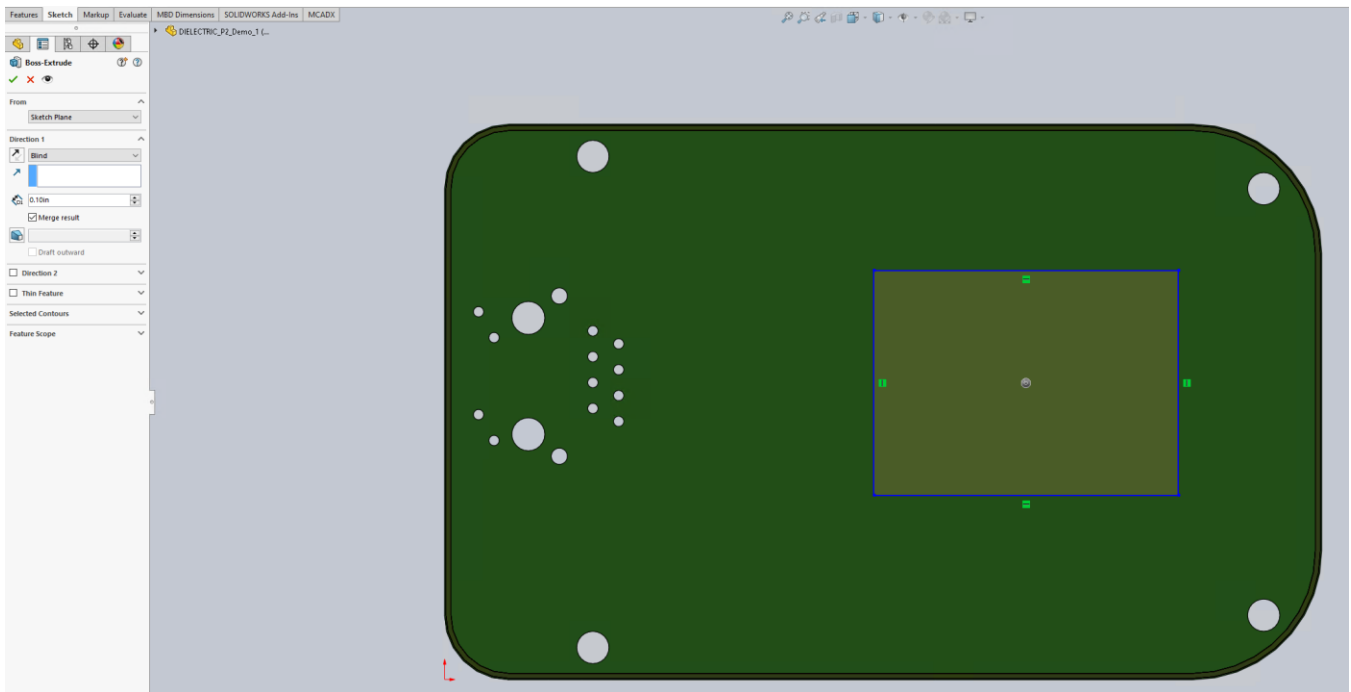


14. First, uncheck the **Merge Results** setting as shown below.

15. Click on the **green checkmark** at the top left of the **Boss-Extrude** pane.

16. NOTE: While a thickness for the added shape can be specified during the Boss-Extrude process, in an upcoming step, when this shape is specified as a top layer copper shape, then the thickness for the shape (copper) will be changed to what is setup in the Preferences.
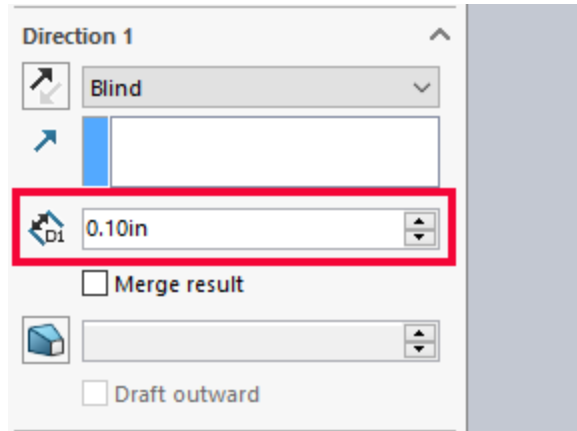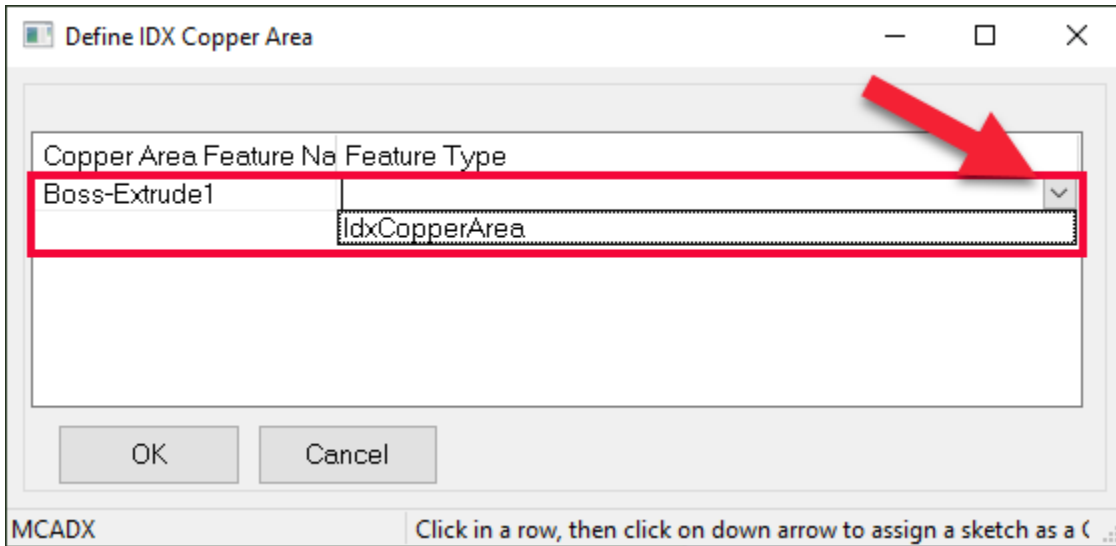


17. In the MCADX ribbon, select the Assign Copper Area tool bar icon.
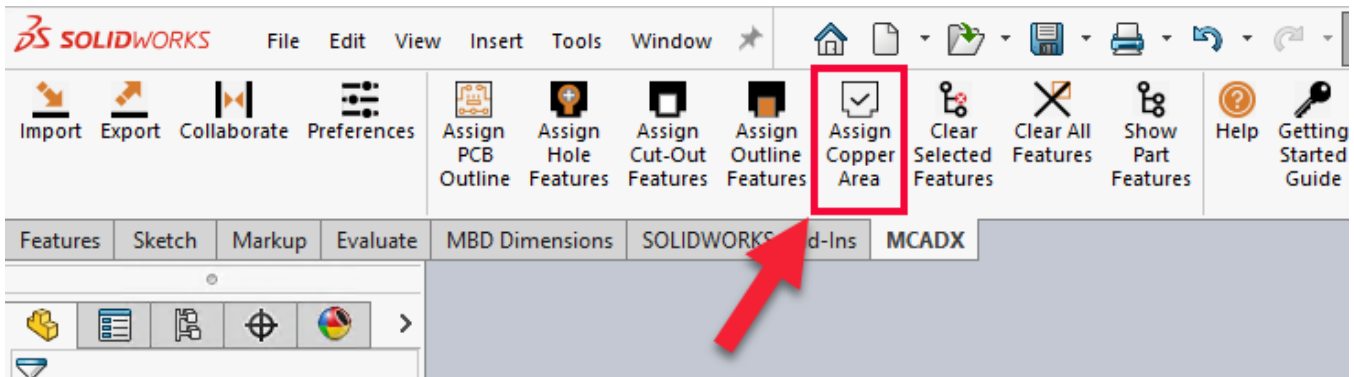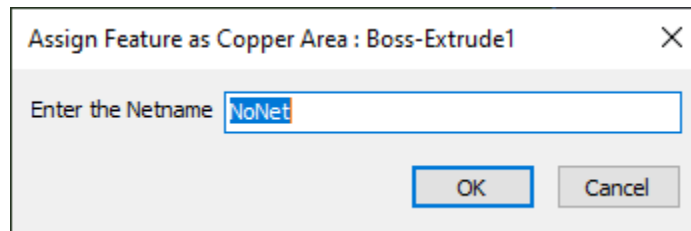


18. The Define IDX Copper Area dialog will appear as shown below.
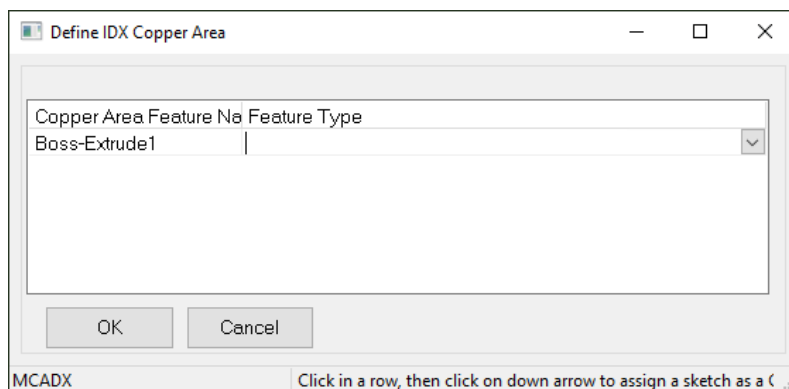
19. Click on the down arrow at the right end of the **Boss-Extrude1** entry and select the **IdxCopperArea** entry.



20. Click **OK** to close the **Define IDX Copper Area** dialog. An **Assign Feature as Copper Area : Boss-Extrude1** dialog will appear asking you to enter a **Net** name to be assigned to the selected copper shape.



21. Enter any valid net name into the **Enter the Netname** box. *Note: In the example below, the net name GND was entered. The MCAD designer will need to confirm with the ECAD (PCB) designer as what the net name should be.*
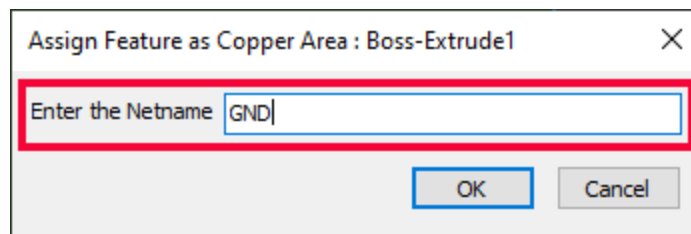


22. After entering the net name, click **OK** to close the dialog.

23. Note in the **Feature Manager Design Tree**, that there is now an entry at the bottom called **StaticCopperAreas**, and if it is expanded (using the triangle to the left of the entry), that the **CopperArea_GND** is now listed. This is the copper area that was just added using the above steps.
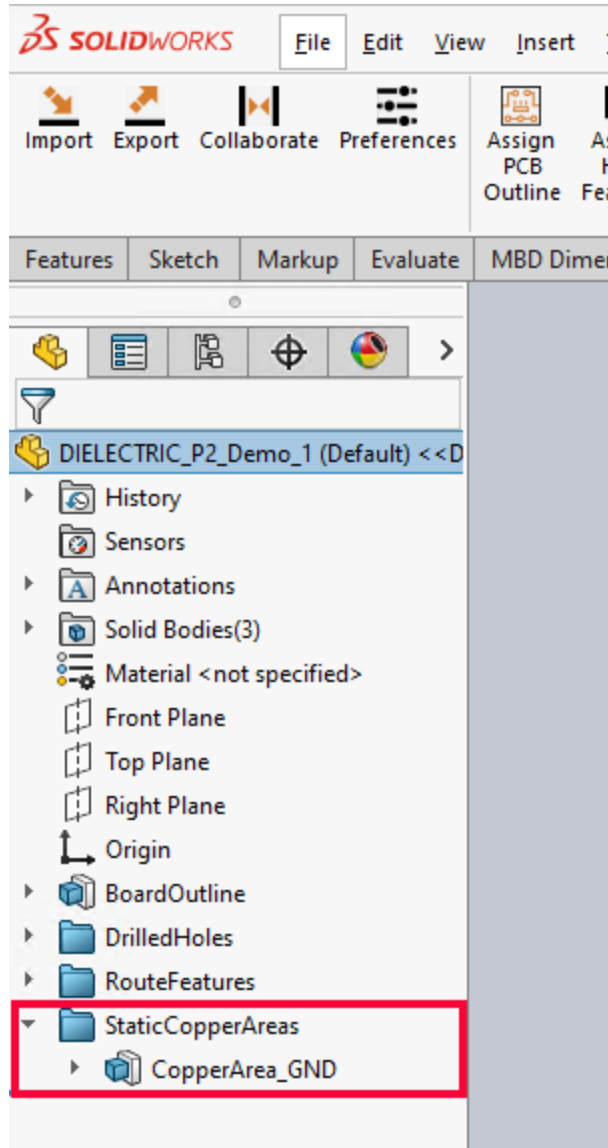


24. Also note, this entry can be renamed by performing a **RMB** on *CopperArea_GND* and selecting **Rename tree item** as shown below. This function can be used to give the newly added copper shape a name – such as **RF_Shield_GND** – as an example.
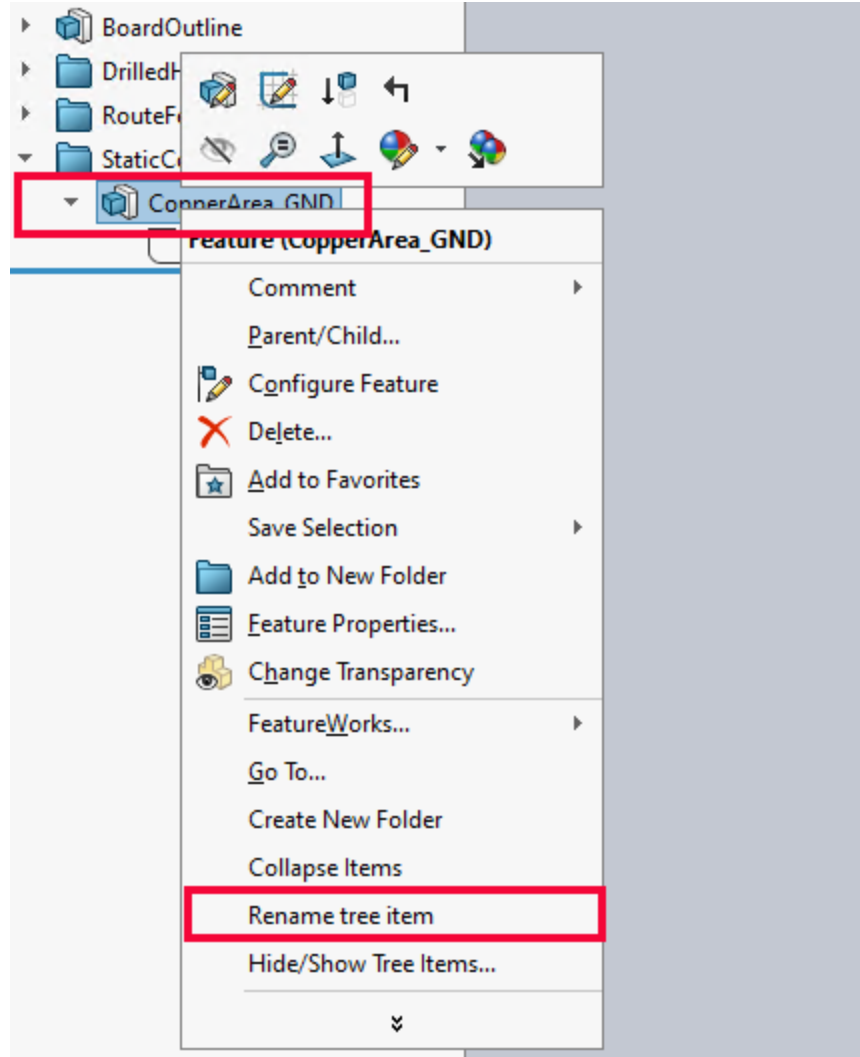
25. After the rename is complete, click anywhere on the canvas for the rename to take effect.
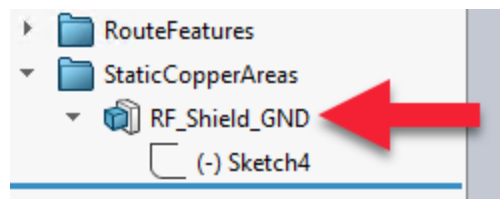


26. To conclude the addition of the copper, click on the **"X"** at the top right of the part canvas as shown below.
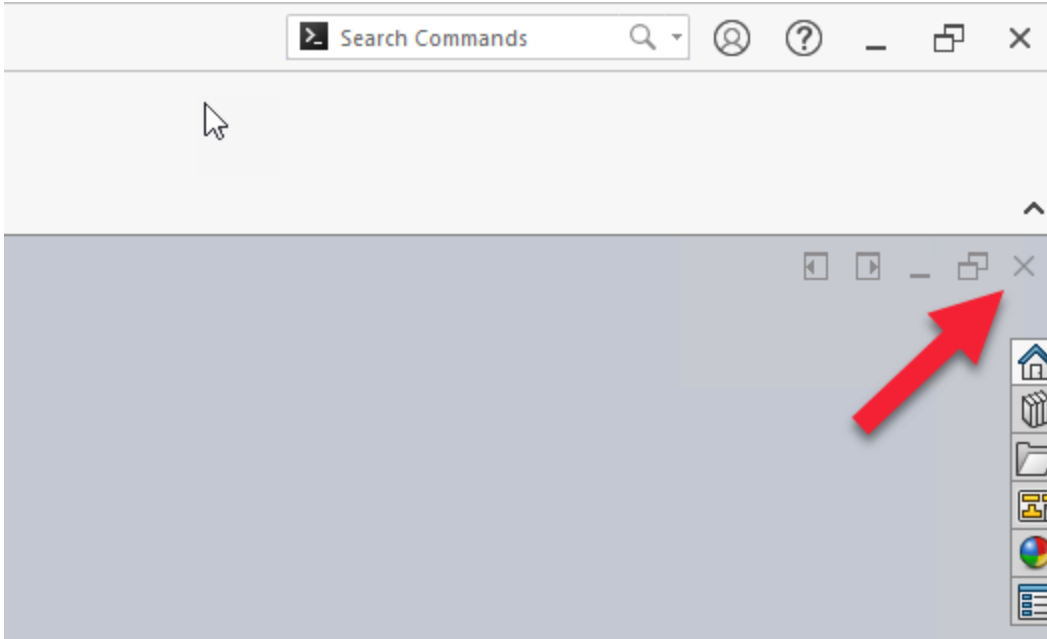
27. When the **Save Part** dialogs appears, click **OK** to proceed.
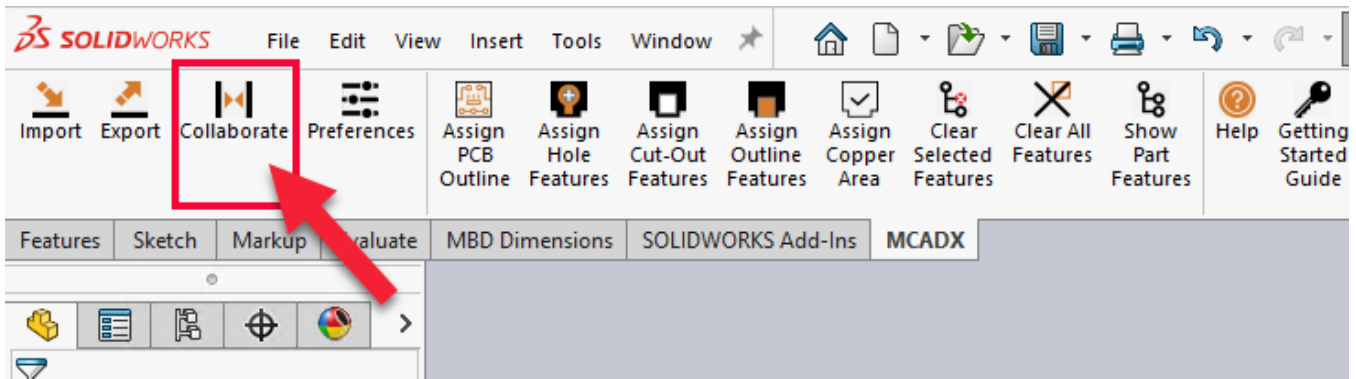28. To send the changes of the newly added copper shape to the PCB editor, click on the **MCADX Collaboration** toolbar icon.



29. When the **MCADX Collaboration** dialog appears, click on the **To PCB** mode by clicking on the radio button at the top left. The canvas will show all the changes made to the MCAD design since reading in the last file from ECAD or since the last time an incremental file was sent from MCAD. In the example below, the addition of the copper area is shown. To send this change to ECAD, click on (check mark on) the check box to the left of the change. **Comments** for this change can be added as well as a **Global Comment** can also be added as indicated below. Once the selections are made and the comments added, click on the **Push Changes to PCB** button at the bottom left.
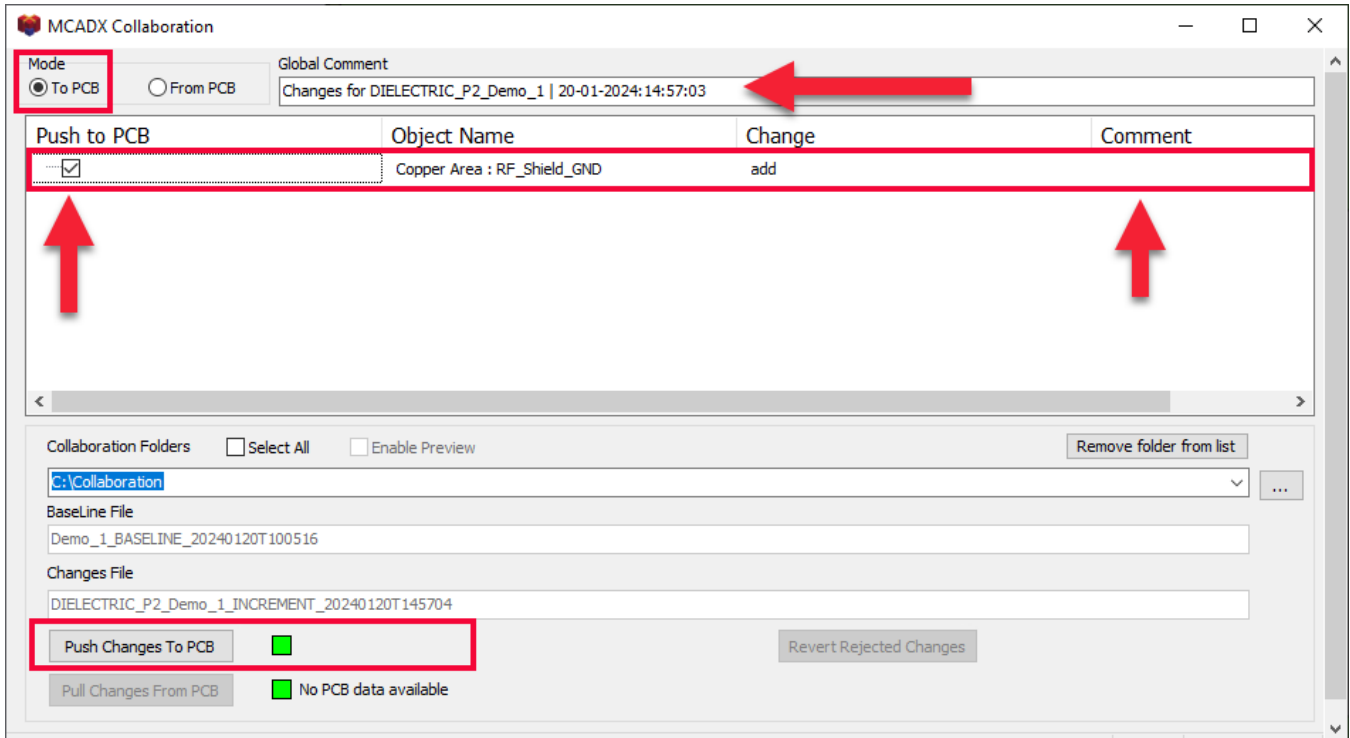
30. As can be seen in the Collaboration folder, an Incremental file with the selected change(s) has been created.
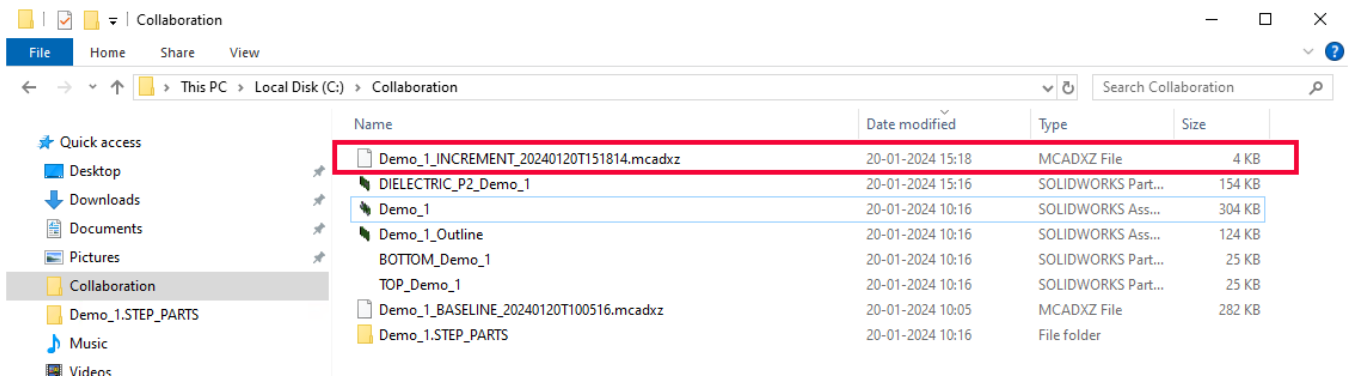


31. If the **File Based** flow is being used for the ECAD-MCAD co-design flow, then send, copy, or make the file available to the **ECAD** designer in some manner.
32. If the **Collaboration Flow** (common folder) flow is being used for the ECAD-MCAD co-design flow, then the file will be deposited into the common folder and the ECAD designer will be notified the next time the collaboration function will check the common folder for the addition of a new file.

# 2: Reading in a Copper Shape into Allegro

## Reading in a Copper Shape into Allegro

Instructions for reading in a copper shape into Allegro are as follows:

1. The Incremental file created in the prior steps can be made available to the ECAD designer either by a manual means (email, DropBox, OneDrive, etc.) Or, if the **Collaboration Flow** is being used, then **MCADX** will automatically deposit the created Incremental file into the common folder.
2. Assuming that the **Collaboration Flow** is being employed, the ECAD designer will be notified of the existence of a new file when the collaboration flow functionality checks the common folder for the existence of a new file, or if the ECAD designer is made aware that a new file was sent, he/she can click on the **Update Status** button as shown below. If there is a new file, the color box to the right of **Pull Updates** will turn **yellow** and the text to the right of the yellow box will change to **New MCADX available** as shown below.
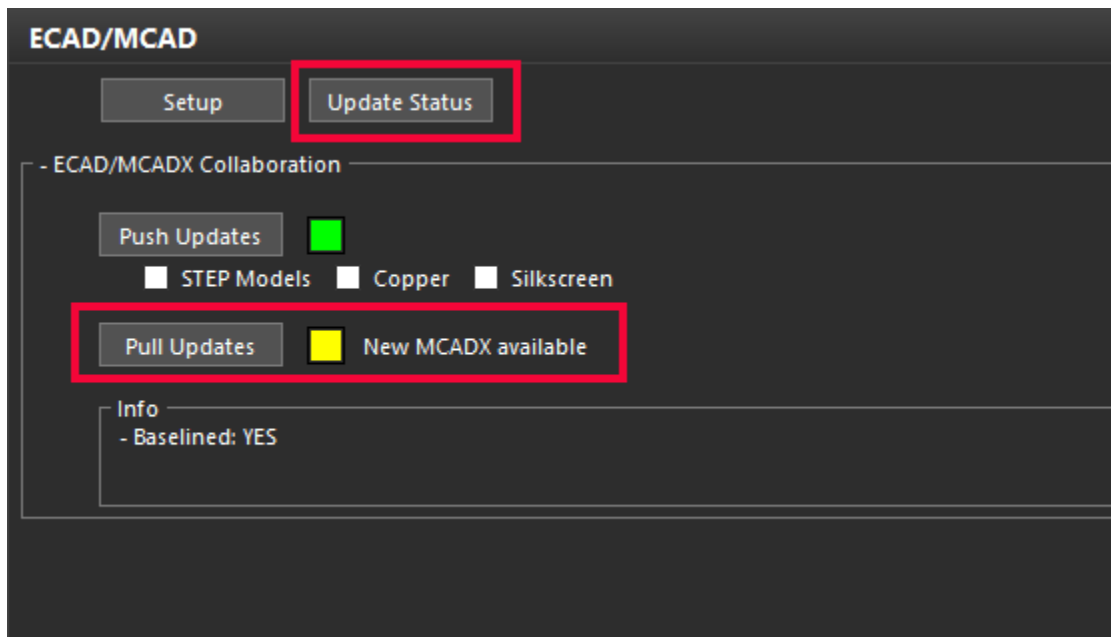


3. Click on the **Pull Updates** button to read in the new **Incremental** file.
4. The **ECAD-MCAD Collaboration** dialog will appear listing all the changes that were part of the Incremental file. As can be seen below, only the added copper shape is

listed in this example. The change(s) can be previewed in the Allegro canvas, by clicking **On** and **Off** the checkbox under the Import header (red arrow below).
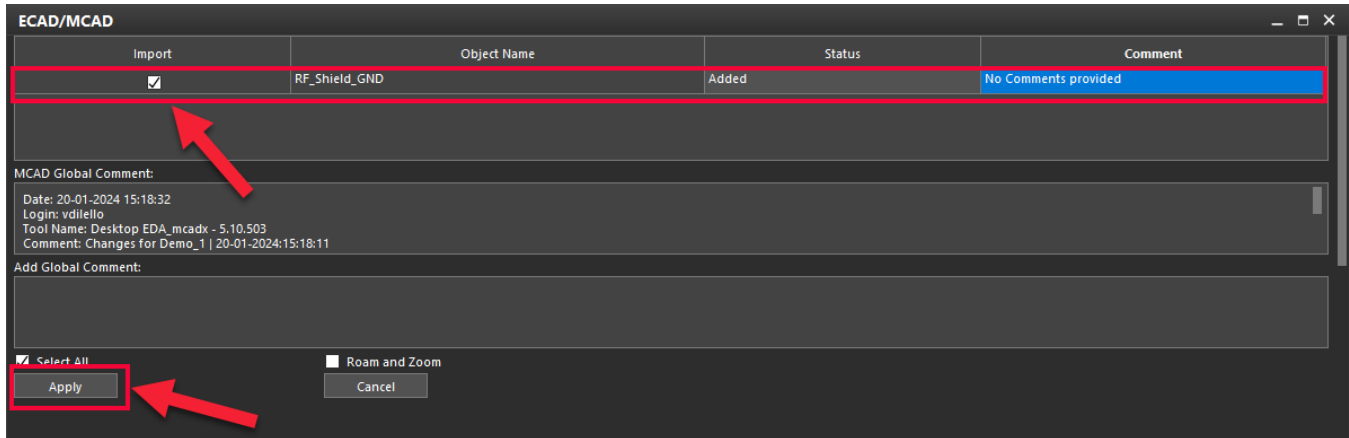


5. Leave the Import checkbox of the change to **On** and then click on the **Apply** button as shown below. ***Note: If the Apply button cannot be seen, the dialog may need resizing or you may need to scroll down to see it.***

6. Once the import of the **Incremental** file is complete, the copper shape that was added in MCAD is now part of the PCB design on the ECAD side as shown below.
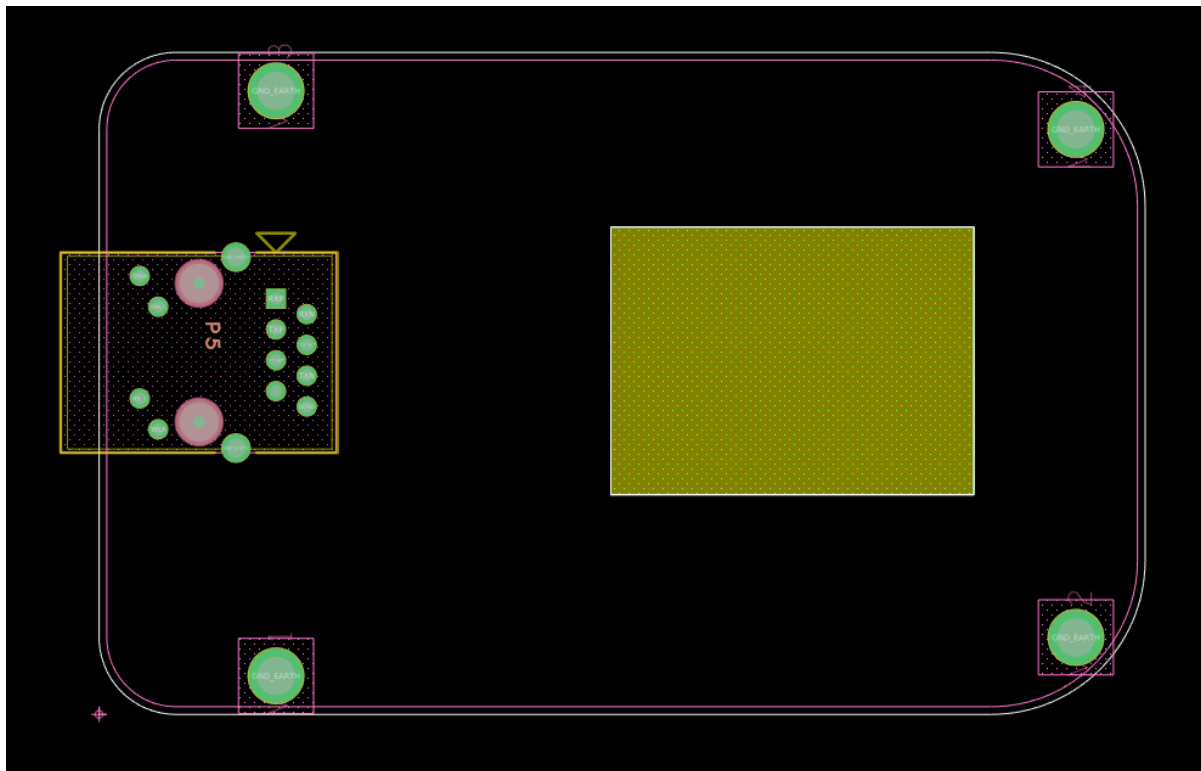
7.  Selecting the just added copper shape in Allegro and doing a **Show Element,** shows the shape is **Etch**, is on the **Top** layer, is **NOT** part of a net – hence why it indicates the *"element is on a dummy net".* Further reviewing the **Show Element** dialog shows the **IDX_OBJID** as **RF_Shield_GND** as was specified on the MCAD side.
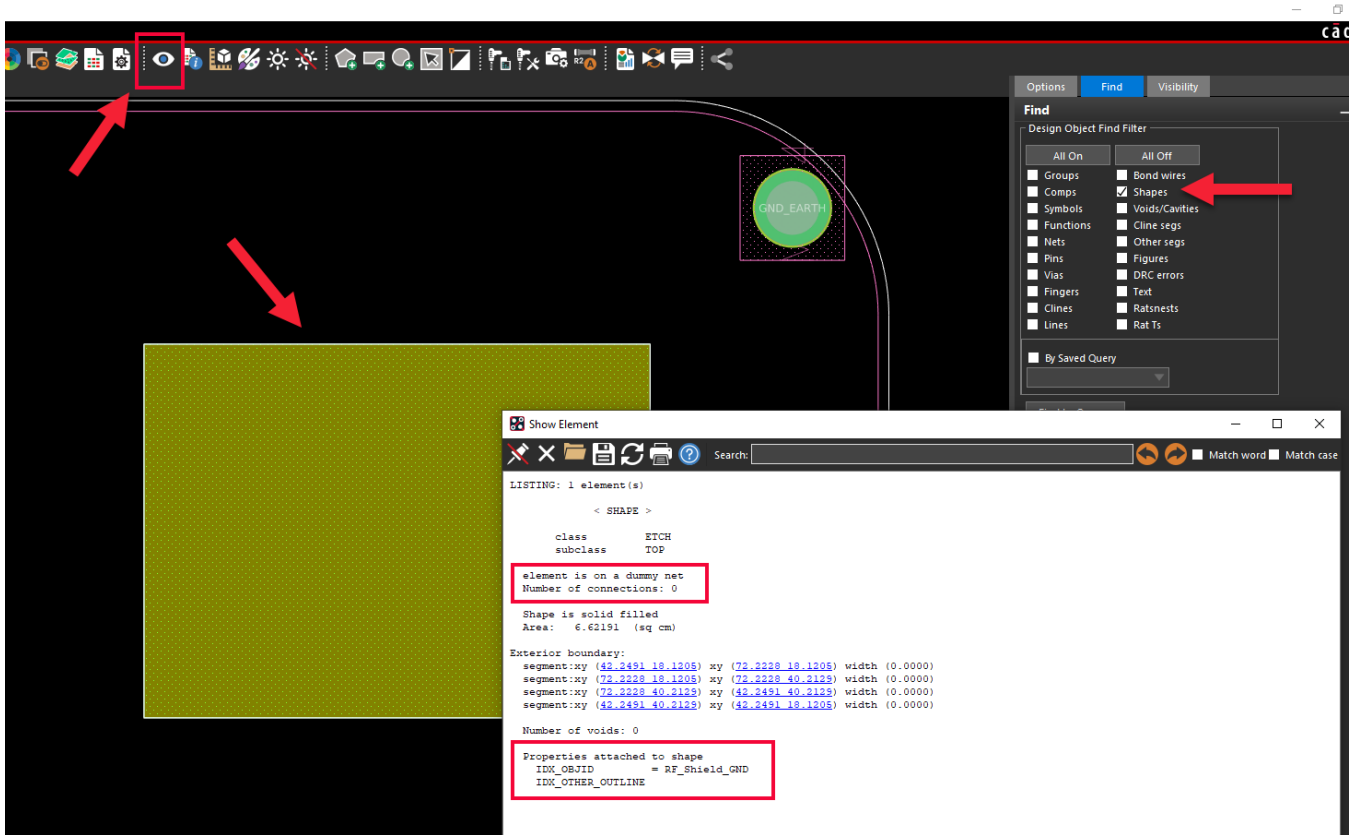


8.  Note: The current MCADX ECAD-MCAD co-design flow, while it is possible to transfer copper from MCAD to ECAD, the process does **NOT** (yet) transfer the **Net** name of the copper shape from MCAD to ECAD. The name of the object on the MCAD side was created with the net name embedded so that the ECAD designer will be made aware of what net this copper shape should be part of. Assigning a net to this copper shape will be as follows.
9.  In Allegro, select the **Shape Select** toolbar icon as shown below.
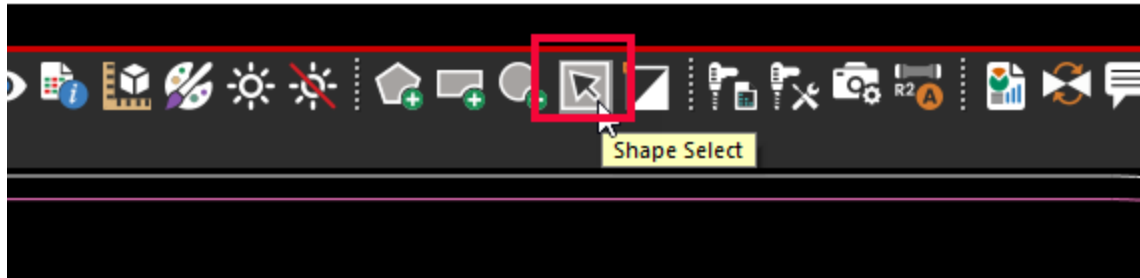
10. Next, select the just added copper shape on the Allegro 2D canvas by clicking on one of the edges of the copper shape.
11. Bring the **Options** pane into focus if it is not the top-level pane.
12. In the Options pane, click the Browse (three dots) of the Assign net name: entry to the right of the Dummy Net entry.
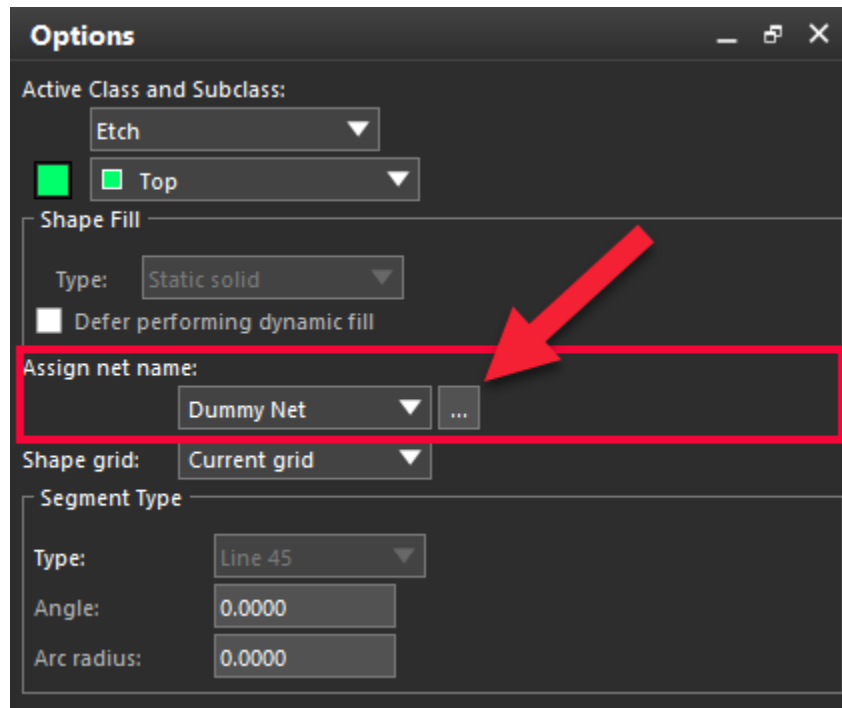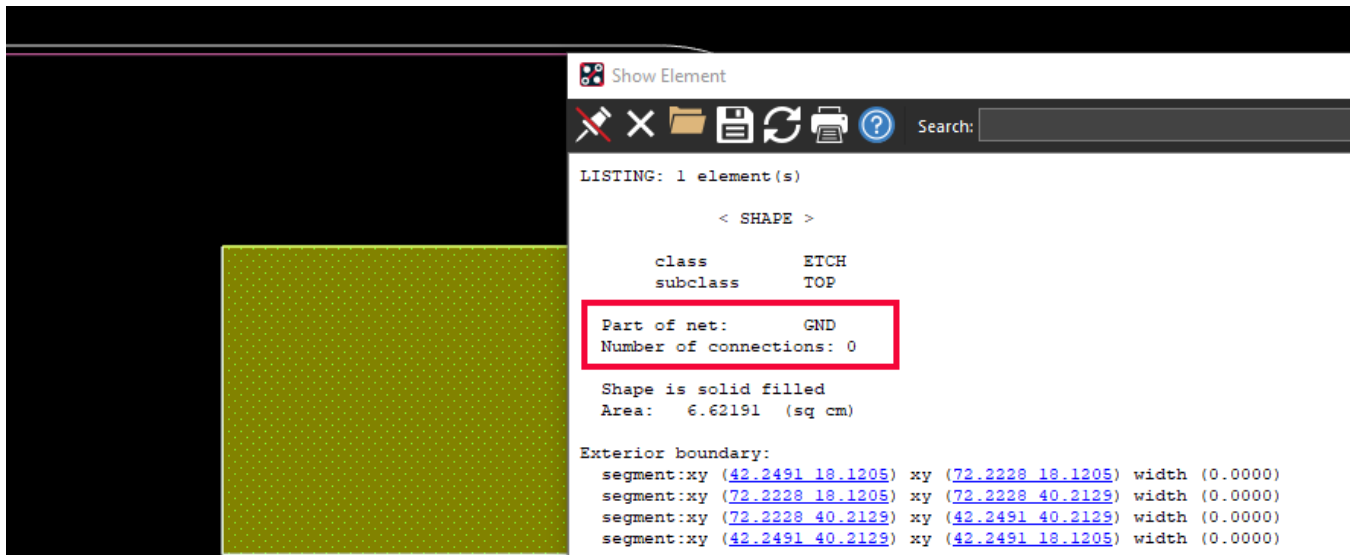


13. Locate the net **GND** and select it. Click OK to close the dialog.
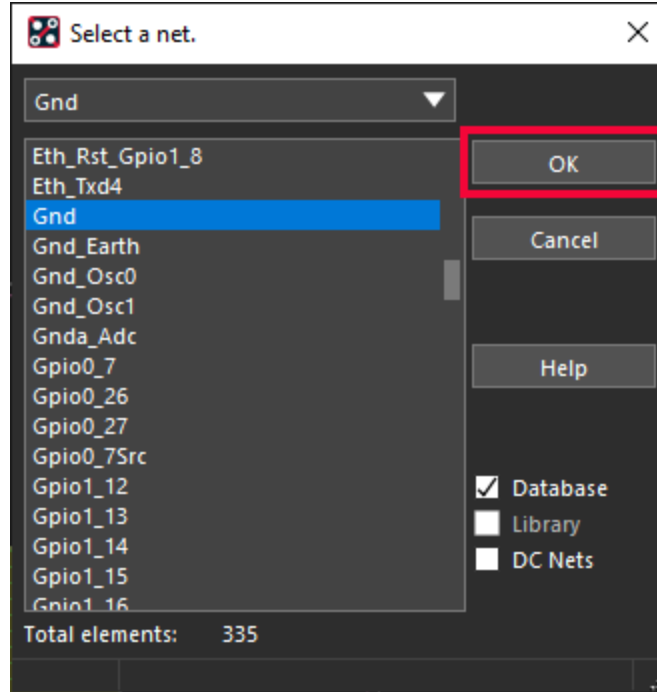
14. **RMB** anywhere on the 2D canvas in a blank area and select **Done**.
15. Performing a Show Element on the copper shape, now shows that the shape is part of the assigned net GND.



16. This concludes this **How To** document.