

# How To ... Add Components in MCAD<sub>X</sub>

January 17, 2024



Desktop EDA

Brian Watson

Table of Contents

<b>1: Overview .....</b>	<b>3</b>
How to .....	3
Add Components in MCADX.....	3
Prerequisites .....	3
<b>2: Adding a Component in MCADX.....</b>	<b>4</b>

---

# 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 Components 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 component (model) in MCADX for SolidWorks and then send information about that component/model 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.
- The component/model, in an “.SLDPRT” format is available in the MCAD library.
- The component/model with the correct Reference Designator (see next point below) has been added to the PCB design schematic and the schematic has been synchronized with the PCB design and the component is in the PCB Editor Placement buffer.
- The MCAD designer and ECAD designer have communicated, and both understand/know what the Reference Designator will be for the new component/model.

---

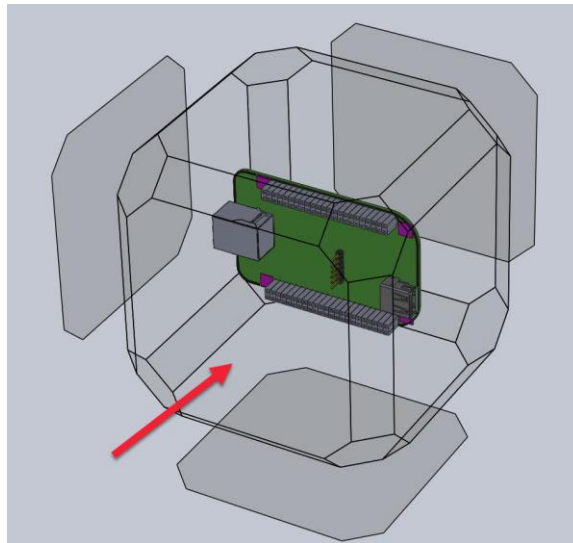
## 2: Adding a Component in MCADX

---

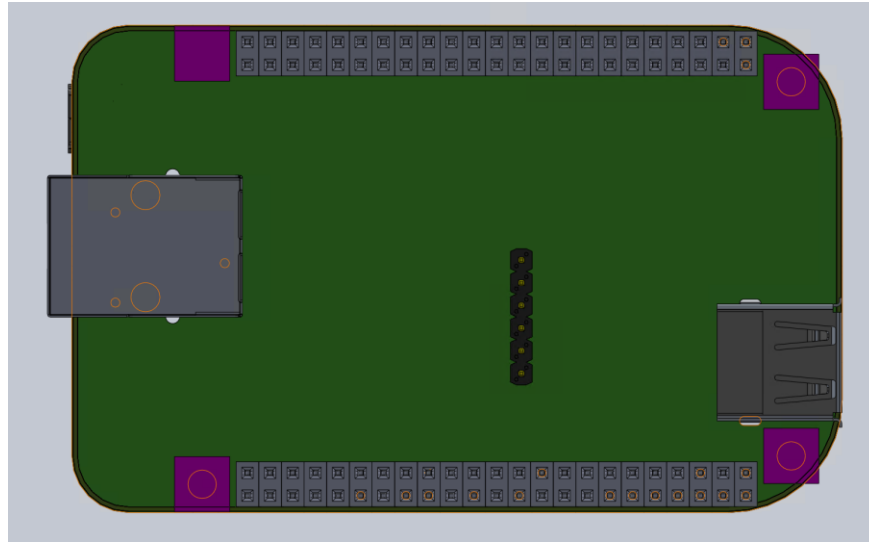
### Adding a Component in MCADX

Instructions for adding a component in MCADX are as follows:

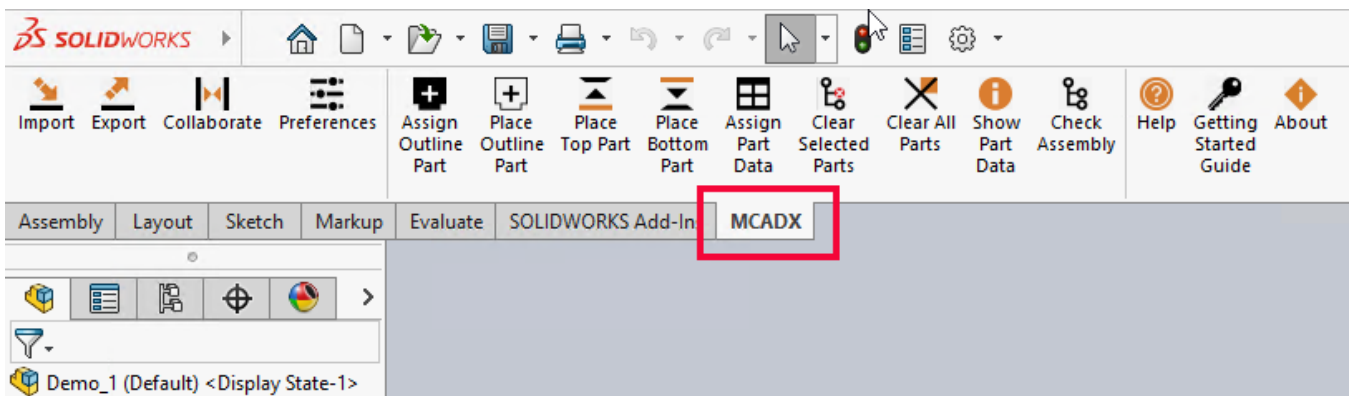
1. Zoom out the design in SolidWorks.
2. Tap on the keyboard **spacebar** to invoke the cubed view of the design as shown below.



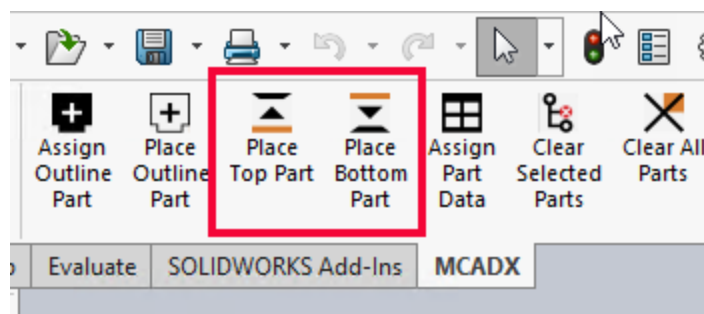
3. Click on the **Front Face** as shown by the red arrow in the screen shot above. The front face will turn blue when the mouse hovers over it.
4. The design will now change from an isometric view to a front view as shown below.



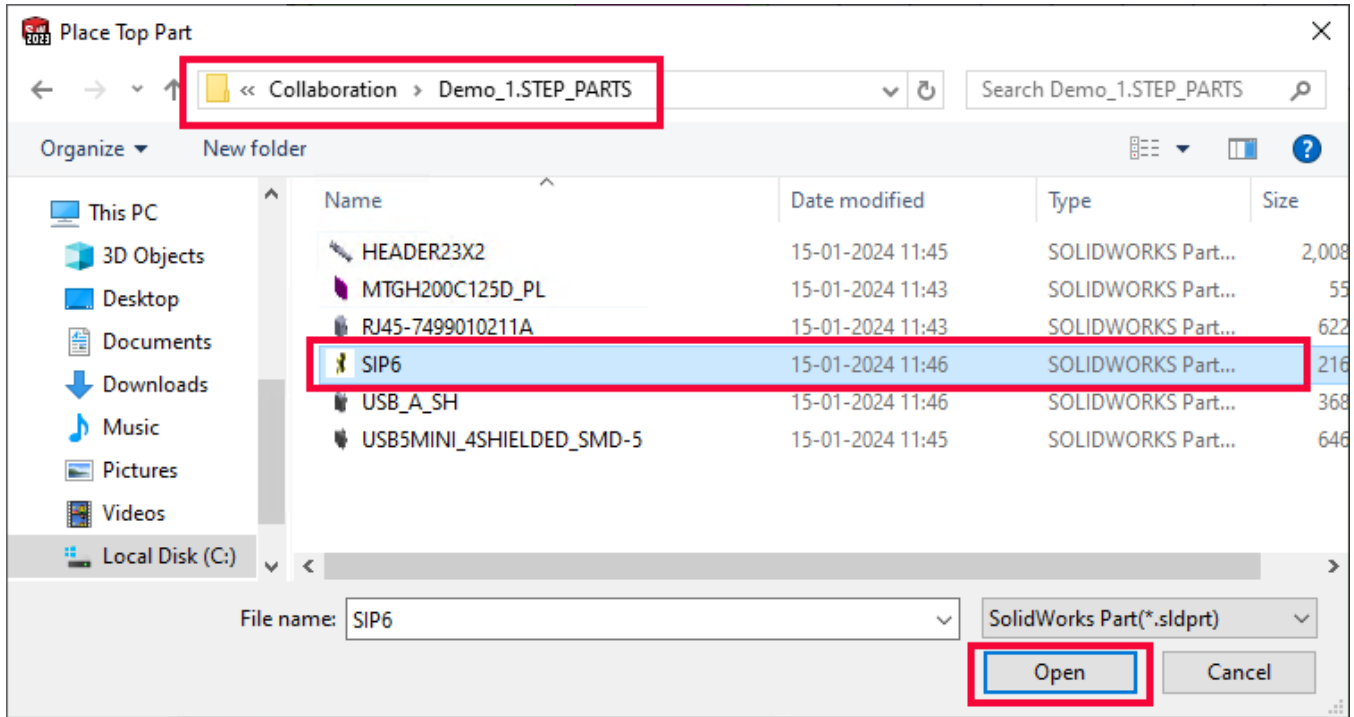
5. In SolidWorks, make sure the **MCADX** tab (shown below) is in focus. If it is not, then click on it to bring it into focus.



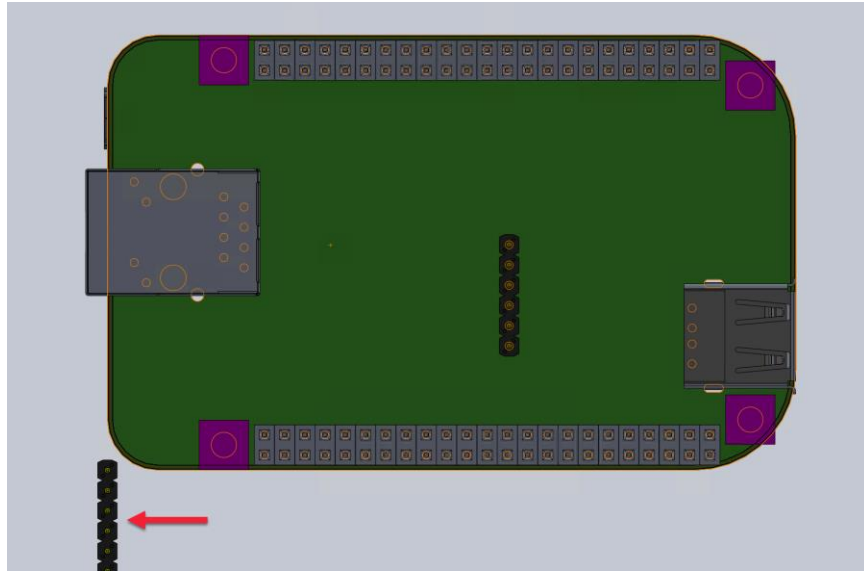
6. Click on the **Place Top Part** if the part to be added will be placed on the top side of the design or click on the **Place Bottom Part** if the part to be added will be placed on the bottom side of the design. For this example, click on the **Place Top Part**.



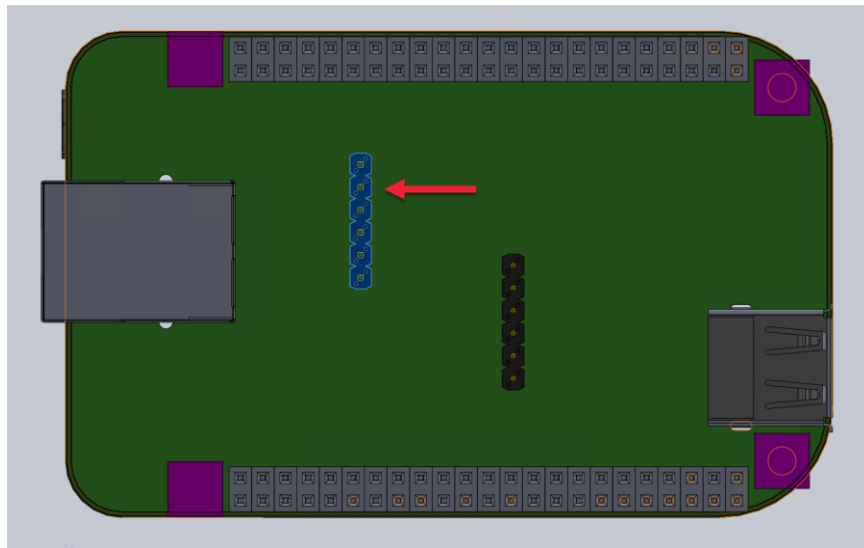
7. When the **Place Top Part** dialog appears, located the folder with the SolidWorks parts, select the part/model to be added and click on the **Open** button at the bottom of the dialog as shown below.



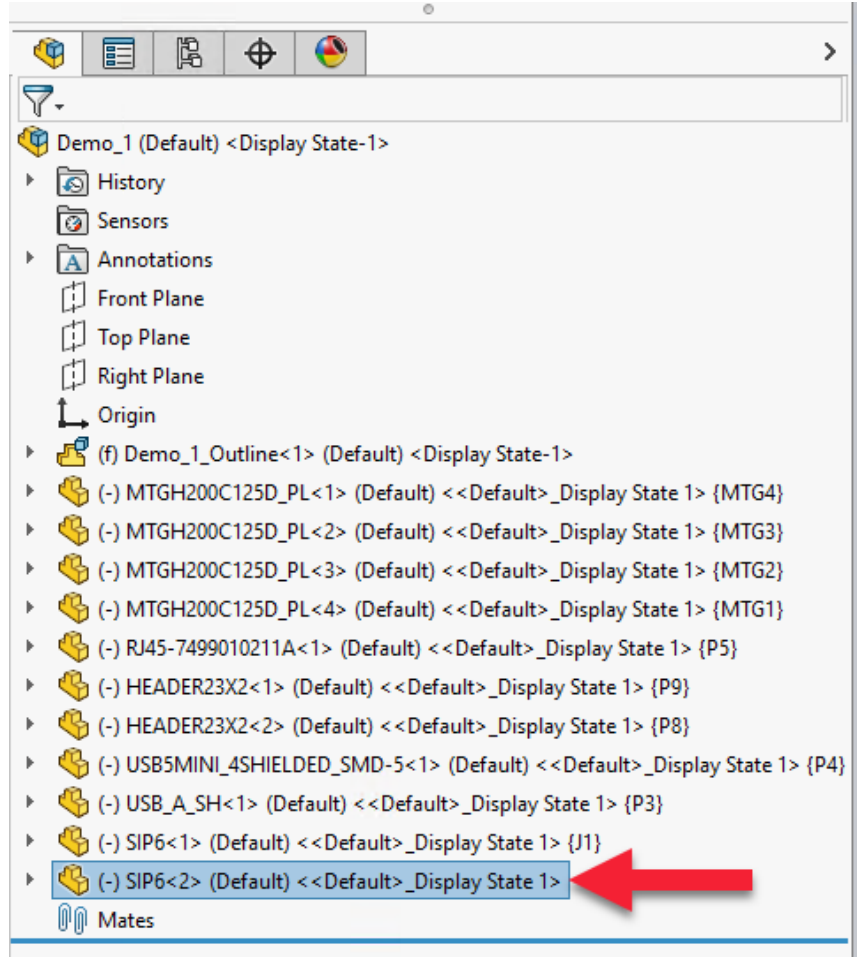
8. The select component/model will be placed into the MCAD design. The datum (0,0) of the part, will be placed on the datum (0,0) of the MCAD design.



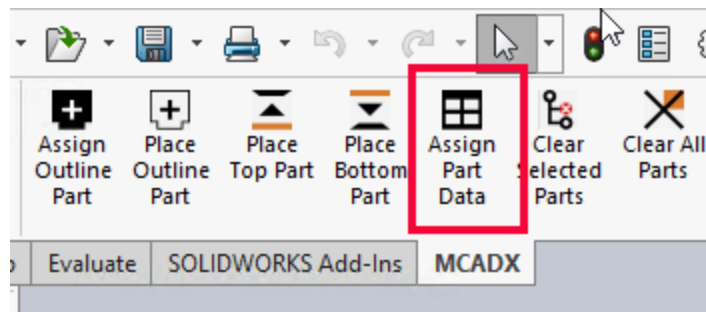
9. LMB+Hold (Left Mouse Button click + hold) on the just added component/model and move to the desired location in the MCAD design. Release the LMB when the component/model is in the desired location. Additionally, other SolidWorks functions/procedures can be used to precisely located the added component/model in its preferred desired location.



10. Notice in the **Top Level Assembly** tree view, the new component/model has been added to the tree. Notice though, that this new component/model does **NOT** have a reference designator – such as all the other components have.

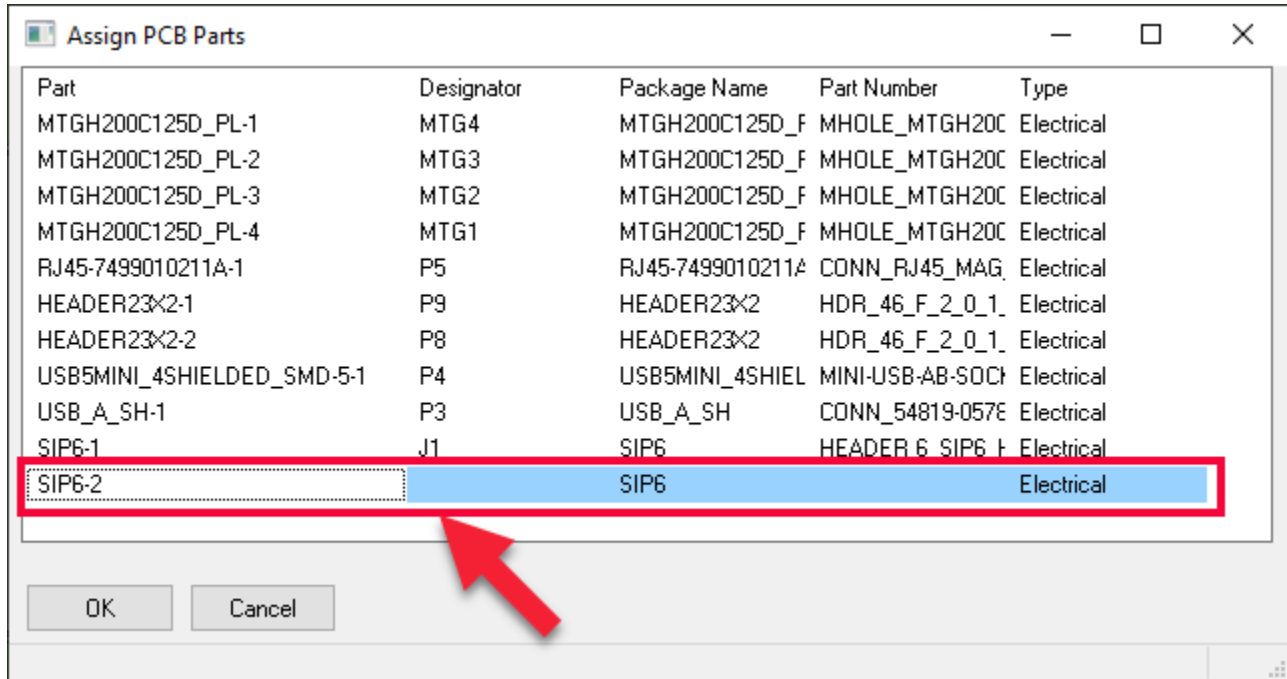


11. By clicking once, select the new component/model in the tree view or select the component/model in the design and then click on the Assign Part Data on the MCADX toolbar as shown below.

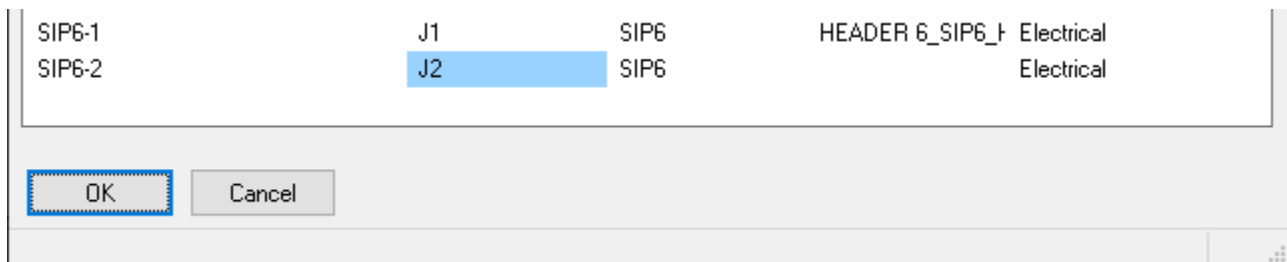


12. The Assign PCB Parts dialog will appear. Notice that all the component/models in the design are listed. Also notice, the just added component does **NOT** have a **Reference Designator** as shown below.

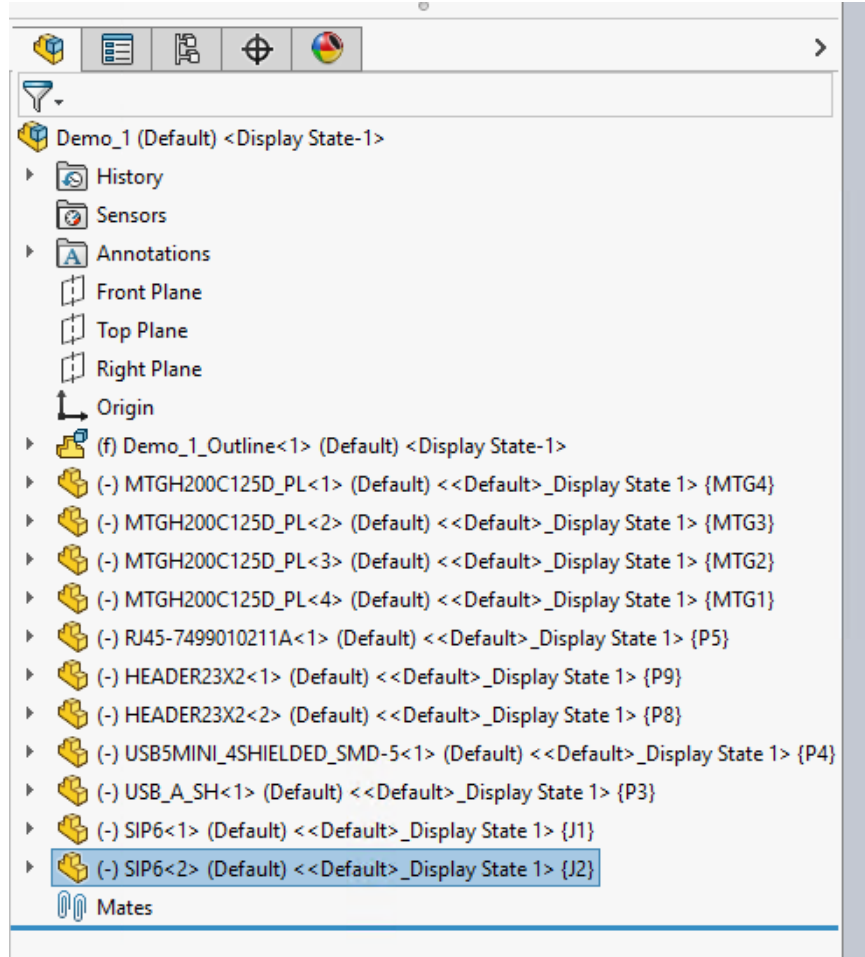




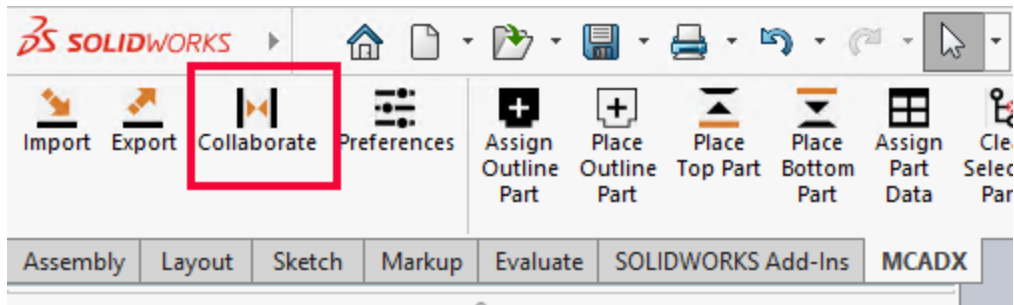
13. **LMB** click on the empty **Designator** cell for the just added component/model and enter the desired and agreed upon **Reference Designator**. Click **OK** to close the dialog.



14. Note: The **Part Number** for the newly added component/model may **NOT** be desired or correct **Part Number**. When we completed the above steps, the **Part Number** for the new component/model read “**SIP6**”, the same as the **Package Name**. To correct the issue, click the cell and change the **Part Number** to the correct **Part Number**, or in this example, since **J2** was the same component/model type as **J1**, we clicked in the **Part Number** cell for **J1**, selected the text, and copied it using **CTRL+C** and then we clicked on the **Part Number** cell for **J2** and copied the text using **CTRL+V**.
15. The newly added component/model in the tree view, will get updated with the new Reference Designator as shown below.

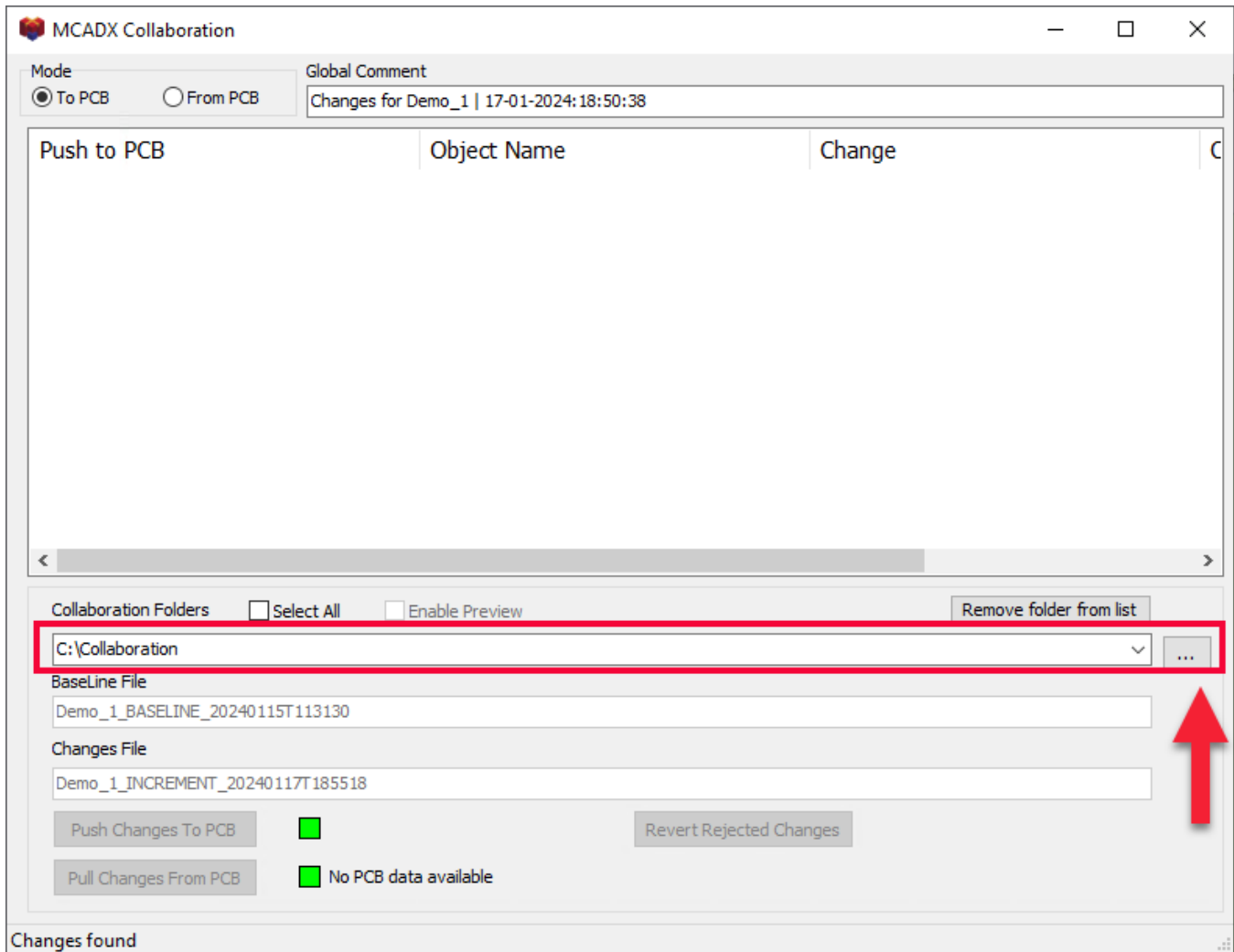


16. Now that recently added component/model has a designator, it is time to send this new change to the Cadence PCB Editor. To do, click on the Collaborate button in the MCADX ribbon toolbar.



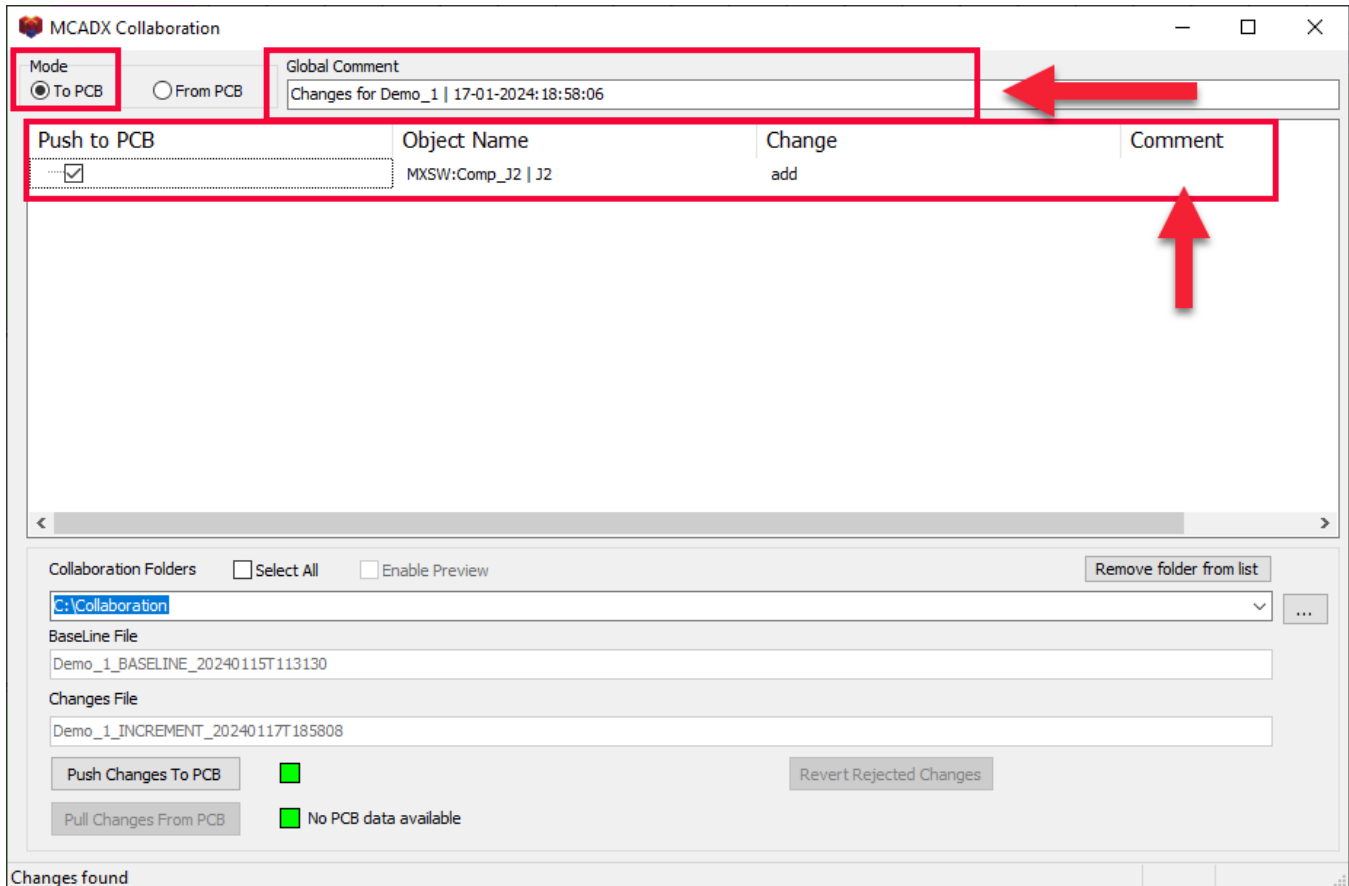
17. The **Collaboration** dialog will appear. Check the **Collaboration Folders** entry and make sure that it is the correct folder – that being the agreed upon folder location for this co-design project. If it is not the correct folder, then click on the **Browse** button

(three dots button at the end of the entry), locate the proper folder, and click **Select**.



18. Click on the **“To PCB”** radio button and the just added component/model addition will be shown in the pane (as shown below) along with any other changes made to the design since the last baseline or incremental file was received/sent from the opposite design team. As can be see below, **J2** is listed as an **“add”**. To send this change over to the PCB designer, click the check box on the left side of the change (right under **Push to PCB**) to the **ON** position (small box has a check mark in it). If you wish to add a **Comment** regarding this particular change, click on the **Comment** cell (red arrow below) and add an appropriate comment for this change. When sending files with lots of changes, **Comments** can be added to each individual change, as well as a **Global Comment** can be added (second red arrow below). When done with selections and comments, click on the **Push Changes to PCB** button at the bottom left. The light beside it will turn from green to red – indicating

that the **Collaboration** functionality is awaiting a **Response** file from the PCB designer indicating whether the change was **Accepted** or **Rejected**. For explanation on this, refer to the MCADX User Guide.



19. The **Incremental** file will be received and consumed by the PCB designer using a Cadence PCB Editor. As we noted in the **Prerequisites** at the top of this document, if the two designers spoke ahead of the change and agree on the change, then the PCB designer could go ahead and add the **J2** component to the schematic and then synchronize with the PCB, **J2** will be ready and waiting in the placement buffer. The PCB designer can then use the **MCADX Collaboration** dialog to read in and consume this just sent **Incremental** file. If he/she accepts the change, then the Cadence PCB Editor will automatically pull the **J2** component from the placement buffer and add it to the PCB design in the location matching the MCAD design.
20. As part of the process, a **Response** file will be generated and then when that **Response** file is deposited into the **Collaboration** folder, the **MCADX Collaboration** dialog will notify the MCAD designer that a **Response** file is available (the green light to the left of **Pull Changes from PCB** will turn yellow). Once the MCAD designer sees this yellow light, he/she simply clicks on the **From PCB** radio

button at the top left of the dialog and the pane will indicate the change just sent and whether the PCB designer **Accepted** or **Rejected** the change. If the change was **Accepted**, then there is nothing else for the MCAD design to do. Both tools will be back in sync. If the change was **Rejected**, then the MCAD designer will need to click on the **Revert Rejected Changes** button at the bottom right of the dialog. Clicking on this button will revert the change in the MCAD design.

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