How To ... Add Copper in MCADX

January 20, 2024



Desktop EDA Brian Watson

1

January 2024

Revision 1.0

Table of Contents

1: Overview	3
How to	3
Add Copper in MCADX	3
Prerequisites	3
Note	3
2: Adding a Copper Shape in MCADX	4
2: Reading in a Copper Shape into Allegro	17

2

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.

3

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.

💦 SOLIDWORKS 🔸 🗥 🗋 • 🕅 • 🔚 • 🚔 • 🖄 • 🍘	- 🕞 - 🕄 🖽	} -		+ 6
Import Export Collaborate Preferences Preferences Preferences Place Place Outline Part Place Place As Bottom P Part D	ssign Clear Clear All Part Selected Parts Data Parts	Show Check Part Assembly Data	Getting About Guide	
Assembly Layout Sketch Markup Evaluate SOLIDWORKS Add-Ins M	ICADX			
•				
🍕 📰 🖹 🕁 🕎				
₹.				
Oemo_1 (Default) < Display State-1>				
History				
Sensors				
Annotations				
Front Plane				
Dop Plane				
[j] Right Plane				
L→ Origin				
(f) Demo_1_Outline<1> (Default) < Display State-1>				
Sancorr				
 Annotations 				
T Front Plane				
[] Top Plane				
[] Right Plane				
🔔 Origin				
(f) TOP_Demo_1<1> (Default) < <default>_Display State 1></default>				
(f) DIELECTRIC_P2_Demo_1<1> (Default) < <default>_Display State 1></default>				
(f) BOTTOM_Demo_1<1> (Default) < <default>_Display State 1></default>				
00 Mates				
(-) MTGH200C125D_PL<1> (Default) << Default>_Display State 1> {MTG4}				
(-) MTGH200C125D_PL<2> (Default) << Default>_Display State 1> {MTG3}				u l
(-) MTGH200C125D_PL<3> (Default) << Default>_Display State 1> {MTG2}	4			
(-) MIGH200C125D_PL<4> (Default) << Default>_Display State 1> {MTG1}	•			
(-) K040-7499010211A<1> (Default) << Default>_Display State 1> {P5}				
UrUP Mates	-			

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.



January 2024

Revision 1.0

<i>3</i> s so∟	.IDWORKS	►			- 🆻	7 - 🖫	- 📇	• 19 •	@ •	3	- 8		ŝ	•	
	<	Ζ	- 🖸	- N	-	24		C	$\langle\!\!\!\!\!\!\rangle$	더	Mirror	Entities			
Sketch	Smart Dimension		•	• 💿	·A	Entities	Conver Entitie	t Offset s Entities	Offset On	СС СС	Linear 9	sketch	Pattern	•	Display/De Relation
-	•		Corne	Recta	ngle				Surface	: <u>7</u> 0	Move E	ntities		~	~
Features	Sketch		Center	Recta	ngle		S	OLIDWORI	KS Add-l	ns N	MCADX				
	0	\diamond	3 Poin	t Corne	er Recta	angle				_					
% [3	3 Poin	t Cente	r Recta	ingle	Be	pardOutlin	e 📃						
7		□	Paralle	logran	ו			Ľ» 🗌	BoardOu	tlineSl	ketch				
	CTRIC_P2_D	emo	1 (Defa	ult) < <	D										

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.

Features Sketc	h Markup	Evaluate	MBD Dim
	0		🕨 🌯 die
🌯 📰 🖡		۸	
🔞 Boss-Extrude	e	7 7	
🗸 🗙 👁			
From		^	
Sketch Plan	e	~	
Direction 1			
Blind		~	
7			
€D1 0.10in		•	
Merge res	sult		
	ward	-	
Direction 2		~	
Thin Feature		~	
Selected Contours		~	

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



January 2024

Revision 1.0

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.

Define IDX Copper	Area	-		×
Conner Area Feat	ire Na Feature Type			
Boss-Extrude1				\sim
ОК	Cancel			
CADX	Click in a row, then click on dow	wn arrow to assign	a sketch	as a (🔡
	11			Revis

19. Click on the down arrow at the right end of the **Boss-Extrude1** entry and select the **IdxCopperArea** entry.

Define IDX Copper Area	_		×
Copper Area Feature Na Feature Type			
Boss-Extrude1			~
IdxCopperArea			
OK Cancel			
MCADX Click in a row, then click on down arro	w to assign	a sketch	as a (🔡

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.

Assign Feature as Copper Area : B	oss-Extrude1	×
Enter the Netname NoNet		
	OK	Cancel

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.

Assign Feature as Copper Area : Boss-Extre	ide1 ×
Enter the Netname GND	
c	K Cancel

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

January 2024

Revision 1.0

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.

 Image: BoardOutline 	
$\begin{array}{c c} & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ &$	
▼ Q ConnerArea GND	_
reature (copperArea_GND)	
Comment	►
<u>P</u> arent/Child	
Configure Feature	
🗙 De <u>l</u> ete	
Add to Favorites	
Save Selection	•
Add <u>t</u> o New Folder	
Eeature Properties	
🛞 C <u>h</u> ange Transparency	
Feature <u>W</u> orks	•
<u>G</u> o To	
Create New Folder	
Collapse Items	
Rename tree item	
Hide/Show Tree Items	
*	

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.

S SOLIDWORKS File Edit View	w Insert Tools	Window 🖈	🏠 🗋 - 🕑	• 🗐 • 📇 •	ŋ • @ •
Import Export Collaborate Preferences	Assign Assign PCB Hole Outline Features	Assign Assign Cut-Out Outline Features Features	Assign Clear Copper Selected Area Features	Clear All Features Feature	Help Getting Started Guide
Features Sketch Markup valuate	MBD Dimensions	SOLIDWORKS Ad	d-Ins MCADX		
0					
🌯 🗐 🛱 🔶 ゝ					

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.

e lo PCB O From PCB	Global Comment Changes for DIELECTRIC P2 Demo 1 20-01-2024	14:57:03	
ish to PCB	Object Name	Change	Comment
	Copper Area : RF_Shield_GND	add	comment
-			
ollaboration Folders	Select All Enable Preview		Remove folder from list
ollaboration Folders	Select All Enable Preview		Remove folder from list
ollaboration Folders	Select All Enable Preview		Remove folder from list
ollaboration Folders	Select All Enable Preview 20T100516		Remove folder from list
iollaboration Folders	Select All Enable Preview 20T100516		Remove folder from list
iollaboration Folders	Select All Enable Preview 207100516 CREMENT_20240120T145704		Remove folder from list
iollaboration Folders	Select All Enable Preview 20T100516 CREMENT_20240120T145704	Revert Rejected Changes	Remove folder from list

30. As can be seen in the Collaboration folder, an Incremental file with the selected change(s) has been created.

- □ X				×		
File Home Share View					~ ?	
\leftarrow \rightarrow \checkmark \uparrow \square \rightarrow This PC \rightarrow Loc	al Disk (C) > Collaboration		✓ ট Search Col	laboration	Q
		Name	Date modified	Туре	Size	
Quick access		Demo_1_INCREMENT_20240120T151814.mcadxz	20-01-2024 15:18	MCADXZ File	4 KB	1
Desktop 🖈	*	DIELECTRIC_P2_Demo_1	20-01-2024 15:16	SOLIDWORKS Part	154 KB	-
🕂 Downloads	*	N Demo_1	20-01-2024 10:16	SOLIDWORKS Ass	304 KB	
Documents	*	Demo_1_Outline	20-01-2024 10:16	SOLIDWORKS Ass	124 KB	
Pictures	*	BOTTOM_Demo_1	20-01-2024 10:16	SOLIDWORKS Part	25 KB	
Collaboration		TOP_Demo_1	20-01-2024 10:16	SOLIDWORKS Part	25 KB	
Demo 1.STEP PARTS		Demo_1_BASELINE_20240120T100516.mcadxz	20-01-2024 10:05	MCADXZ File	282 KB	
Music		Demo_1.STEP_PARTS	20-01-2024 10:16	File folder		
Videos						

- 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:

- 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.

ECAD/MCAD
Setup Update Status
- ECAD/MCADX Collaboration
Push Updates STEP Models Copper Silkscreen
Pull Updates New MCADX available
- Baselined: YES

- 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).

ECAD/MCAD			_ = ×
Import	Object Name	Status	Comment
∠	RF_Shield_GND	Added	No Comments provided
MCAD Global Comment:			
Date: 20-01-2024 15:18:32 Login: vdilello Tool Name: Desktop EDA_mcadx - 5.10.503 Comment: Changes for Demo_1 20-01-2024:1	5:18:11		'
Add Global Comment:			·
Apply	Cancel		

- 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 <u>NOT</u> 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.

		cā
🕽 🕞 🚔 📓 💿 🤚 🔛 % 🔆 🔆 🖙 🔍 🔽 🎵	s 🗫 🗟 📷 📸 🗢 루 <	
	Image: Second	C à l Options Find Visibility Find Pesign Object Find Filter All On All Off Groups Bond wires Symbols Claretes Symbols Claretes Symbols Claretes Fingers Figures Nets Other seas Fingers Text Unes Rat Ts ins Saved Query Compose Match word Match case

- 8. Note: The current MCADX ECAD-MCAD co-design flow, while it is possible to transfer copper from MCAD to ECAD, the process does <u>NOT</u> (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.

Options		-	8	×
Active Class and Subclass:				
Type: Static	Type: Static solid			
Defer perform	Defer performing dynamic fill			
Assign net name:				
Du	mmy Net 🔻			
Shape grid: Cu	rrent grid 🔍 🔍			
Segment Type				
Туре:				
Angle:	0.0000			
Arc radius:	0.0000			

13. Locate the net **GND** and select it. Click OK to close the dialog.

20

🚼 Select a net.	×
Gnd 🔻	
Eth_Rst_Gpio1_8 Eth_Txd4	ОК
Gnd Gnd_Earth	Cancel
Gnd_Osc1 Gnda_Adc	
Gpio0_7 Gpio0_26	Help
Gpio0_27 Gpio0_7Src	
Gpio1_12 Gpio1_13	✓ Database Library
Gpio1_14 Gpio1_15 Gpio1_16	DC Nets
Total elements: 335	

- 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.

Show Element
🗙 🗙 📛 🖹 💭 🖶 🞯 🛛 Search:
LISTING: 1 element(s)
< SHAPE >
class ETCH subclass TOP
Part of net: GND Number of connections: 0
Shape is solid filled Area: 6.62191 (sq cm)
Exterior boundary:
segment:xy (<u>42.2491 18.1205</u>) xy (<u>72.2228 18.1205</u>) width (0.0000) segment:xy (<u>72.2228 18.1205</u>) xy (<u>72.2228 40.2129</u>) width (0.0000) segment:xy (<u>72.2228 40.2129</u>) xy (<u>42.2491 40.2129</u>) width (0.0000)
segment:xy (<u>42.2491 40.2129</u>) xy (<u>42.2491 18.1205</u>) width (0.0000)

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