

AUTODESK FUSION - SET UP DESIGN RULES

Fusion released an updated version that looks a little different in the PCB Design Rules setup. The DRC rules must be set up at the start of switching into PCB mode, from Schematic mode.

- 1) Check the installed version of Fusion on your computer. On the top right corner, click on the question mark symbol (?) then click on **About**.

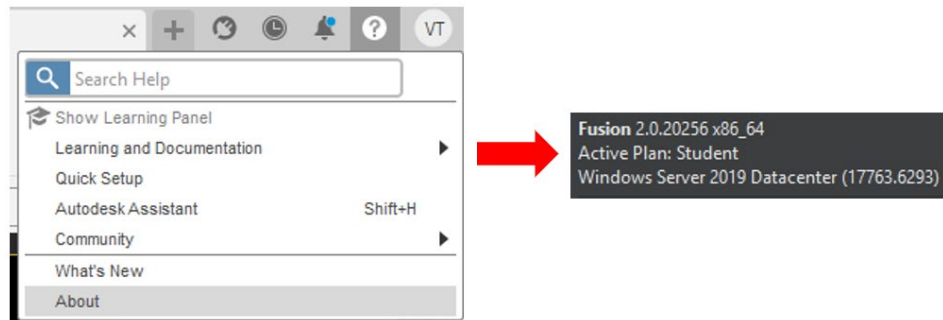


Figure 1. Checking Fusion installed version.

- a. If the version is **2.0.20256 x86_64** or **newer**, **skip** Step 2 and **continue** to Step 3 (pg.3).
- b. If the version is **older** than mentioned above, **continue** with Step 2 and **skip** Step 3.

- 2) On the menu bar, select **RULES DRC/ERC > DRC**. Note: If the symbols are not the same as Figure 2, skip this step and go to Step 3 on page 3.



Figure 2. DRC option.

- a. Click on tab **Clearance** then set the rules as Figure 3 below.

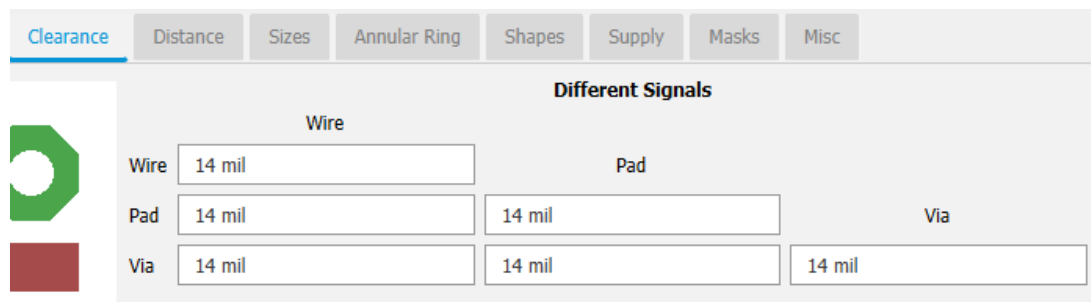
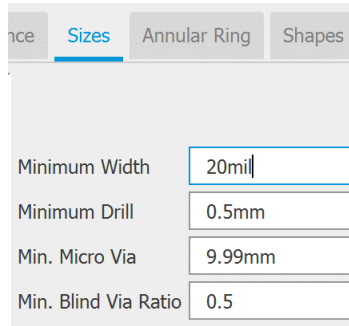


Figure 3. Clearance tab in DRC rules.

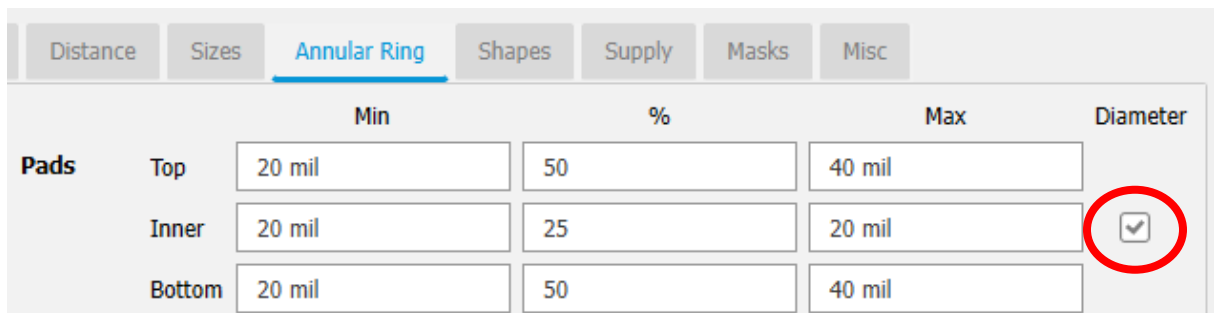
- b. Click on tab **Sizes** then set the rules as Figure 4 below.



Minimum Width	20mil
Minimum Drill	0.5mm
Min. Micro Via	9.99mm
Min. Blind Via Ratio	0.5

Figure 4. Sizes tab in DRC rules.

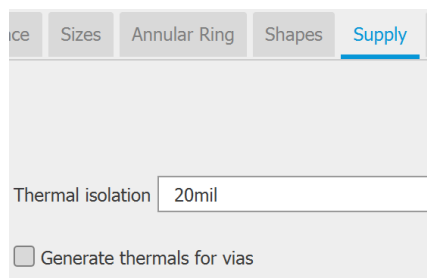
- c. Click on tab **Annular Ring** then set the rules as Figure 5 below then check on the **Diameter** box.



		Min	%	Max	Diameter
Pads	Top	20 mil	50	40 mil	<input checked="" type="checkbox"/>
	Inner	20 mil	25	20 mil	<input checked="" type="checkbox"/>
	Bottom	20 mil	50	40 mil	<input checked="" type="checkbox"/>

Figure 5. Annular Ring tab in DRC rules.

- d. Click on tab **Supply** and set the rules as Figure 6 below.



Thermal isolation	20mil
<input type="checkbox"/> Generate thermals for vias	

Figure 6. Supply tab in DRC rules.

- e. Click **Apply** then close the window.

In the latest version, you will need to set the Design Rules and the Design Preferences.

3) On the menu bar, click **RULES > Design Rules**.

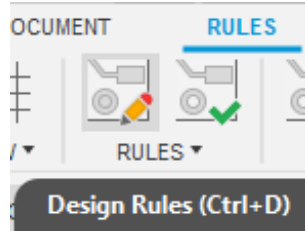


Figure 7. Design Rules option.

a. Change the rules as the figure below.

General Rules Custom Rules		
Search		
Name	Rule	Enabled
Wire - Wire Clearance	14 mil	<input checked="" type="checkbox"/>
Wire - Pad Clearance	14 mil	<input checked="" type="checkbox"/>
Wire - Via Clearance	14 mil	<input checked="" type="checkbox"/>
Pad - Pad Clearance	14 mil	<input checked="" type="checkbox"/>
Pad - Via Clearance	14 mil	<input checked="" type="checkbox"/>
Via - Via Clearance	6 mil	<input checked="" type="checkbox"/>
SMD - Pad Clearance	6 mil	<input checked="" type="checkbox"/>
SMD - Via Clearance	6 mil	<input checked="" type="checkbox"/>
SMD - SMD Clearance	6 mil	<input checked="" type="checkbox"/>
Board Outline Clearance	40 mil	<input checked="" type="checkbox"/>
Copper - Restrict Clearance	0 mil	<input checked="" type="checkbox"/>
Via - Restrict Clearance	0 mil	<input checked="" type="checkbox"/>
Soldermask - Silkscreen Clearance	0 mil	<input type="checkbox"/>
Drill Clearance	6 mil	<input checked="" type="checkbox"/>
Component Exclude Clearance	0 mil	<input checked="" type="checkbox"/>
Unconnected Lines	N/A	<input checked="" type="checkbox"/>
Copper Width	20 mil	<input checked="" type="checkbox"/>
Drill Size	19.69 mil	<input checked="" type="checkbox"/>
Acute Angle	N/A	<input type="checkbox"/>
Wire Stub	N/A	<input checked="" type="checkbox"/>
Air Wire	N/A	<input checked="" type="checkbox"/>

Enter "0.5 mm". It will convert to "mil" automatically.

Figure 8. General Rules tab in Design Rules.

b. Click **OK**.

- Design Preferences**

d. Select tab **Annular Ring** and set the rules for **Pads** as the figure below then check on the **Diameter** box.

		Distance	Sizes	Annular Ring	Shapes	Supply	Masks	Misc
Pads		Min	%	Max	Diameter			
	Top	20 mil	50	40 mil	<input checked="" type="checkbox"/>			
	Inner	20 mil	25	20 mil				
	Bottom	20 mil	50	40 mil				

e. Click on tab **Supply** and set the rule for **Thermal isolation** to **20 mil.**

Thermal isolation 20 mil

f. Click OK.