Step 1

Open Capture CIS 17.4 and create a new design (or open an existing design)

Step 2

In the left panel, click on the 'Design Resources' folder to highlight it



With this folder highlighted, go to tools -> generate part

Step 3

Specify the source file to the EAB450M12XM3.cir on your computer. Copy the settings below.

Generate Part			×			
Netlist/source file:			ОК			
C:\Pspice_Models\CAB650M17HN	13.cir	Browse	Cancel			
Netlist/source file type:			Canoca			
PSpice Model Library			Help			
Part name:			FPGA Setup			
<all></all>						
Destination part library:						
C:\Pspice_Models\CAB650M17HM	13.OLB		Browse			
Create new part		n existing part in libi				
Pick symbols manually						
Sort pins	Additional pins					
	☐ Specify the number of additional ☐ pins on part					
 Descending order 						
Retain alpha-numeric pin-numbers. Device is pin grid array type package.						
Implementation type:	Implementation n	ame:				
PSpice Model 🔹	<all></all>					
Implementation file:						
		Brow				

You should see something like the below in the left column. Note that I'm using a CAB650M12HM3 model for my example.



Step 4

Now, go to Pspice -> new simulation profile (or edit an existing one if you have it already). Go to Configuration Files -> Library and add the .cir file to the library as a global.

Simulation Settings - WS_Sim				×
General	Category:	Filename:		Browne
Analysis Configuration Files	Stimulus Library Include	Configured Files	× + +	biowse
Options		C:\Pspice_Models\CAB650M17HM3.cir*		Add as Global
Data Collection Probe Window Probe Window Probe Window Probe Window Probe Window			Add to Design Add to Profile	
			Edit	
			Change	
	Library Path "C:\Cadence\SPB_17.4\tools\PSpice\Library"		Browse	
		OK Cancel	Apply Res	iet Help

Step 5

On the left-panel, under the 'Library' folder, double-click on the CAB650M17HM3 model. You should see a default symbol appear. You can either keep this symbol, or edit the symbol to something more appropriate.

