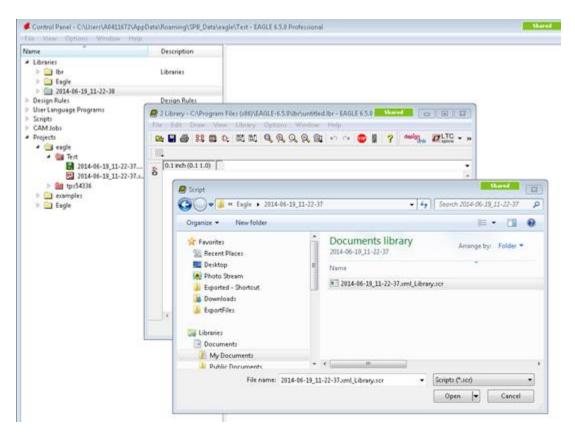
Below are the steps to integrating the Eagle library for Schematic and convert it to board netlist:

1. Open a new library window and go to File and select Execute Script. Select the downloaded library.scr file.



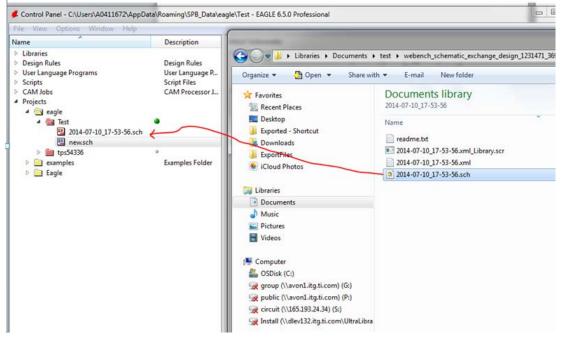
2. Yes, at this point, the library conversion starts and there will be some errors as you can see here but just ignore the errors and select OK for all.

Sheet Sheet	a first of	dolion he DESCRIPTION command to ente	+ r a description of this object.		
Error				0%	Shared In

3. Once the library conversion is completed, please save the library as "AcceleratedDesigns\_Lib.lbr" so the schematic can link to the default name of the library into all Eagle library directory. Here is a sample of the imported components from the design.

nloads 🖉 2 🖉 Edit	Library - C:\Program	Files 0.86/VEAGLE-6.5.87/bir/unbtiled.lbr (TPISSA336DDAR.dev) - EAGLE 6.5.8 Pr Library Options Window Help III III III Q, Q, Q, Q, III Pr Co 100 1 2 Control 10 Cont	
Device CC0805JRNP09BN20 CC0805JRN07R888271 CRCW040232K4FKED CRCW040247K5FKED CRCW0402100K7KED GRM21BR51E475MA121 GRM031R51A103KA01D SRR1260-270M TPS54336DDAR New: Dev Pac	E Sym Cancel	-CAL COMP B COMP	* Secure Secure 18112 Secure 181
I (\\dlev132.itg.ti.com\\ k 411672B	Use the	Echoniogics Attributes DATASHEETURL FREQUENCY IMAX MANUFAC PS54336COAR http://www.b.com/product/tps54336_340000.0_3.0_Texas Inst III. +	Volue Off On

4. Create a project folder "test" and drag the schematic file into the project folder using windows explorer



5. Go to the schematic and select Library and select Update; find the newly imported AcceleratedDesigns\_Lib.lbr. It will ask you update each component in the schematic (Note: Match the part # as shown above the library list) and make sure it's done for all components.

	new package malanti	is not present in the new versi	on of device set CC0805JRNP09BN12	20.	
Name		Description		(2x Scale)	
C0805 CC08 CC08	220MW 6C104K5RACTU 05JRNP09BN120 05KRX7R9BB271 V040232K4FKED			= F <sub>57</sub> 143d0 ≫∰≣E E	
CRCV CRCV GRM2 GRM0 SRR12	200023101051620 20002100656620 218861E475MA12L 233861A103KA01D 260-270M 43360DAR		Package: 0805		
CRCV CRCV GRM2 GRM0 SRR12	V04024NGFKED V0402100KFKED 21BR61E475MA12L J33R61A103KA01D 260-270M		Package: 0805	Value	
CRCV CRCV GRM2 GRM0 SRR12	V04024NGFKED V0402100KFKED 21BR61E475MA12L J33R61A103KA01D 260-270M		Attribute	Value 1.2E-11	
CRCV CRCV GRM2 GRM0 SRR12	V04024NGFKED V0402100KFKED 21BR61E475MA12L J33R61A103KA01D 260-270M	☑ Description ☑ Preview	Attribute CAP ESR	Value 1.2E-11 0.0	m 1
CRCV CRCV GRM2 GRM0 SRR12	V0402100KFKED V0402100KFKED V1BR61E475MA12L V33R61A103KA01D V60-270M V336DDAR	Description V Preview	Attribute	Value 1.2E-11	

6. Save the schematic, select File  $\rightarrow$  Switch to Board. It should generate the pcb board with the component netlist (ratsnest) with footprints showing for all the components.

