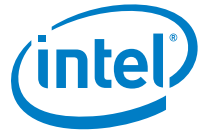


ISCF Export Process for Cadence OrCAD 17.2

March 2018
Version 1.1



Revision History

Revision Number	Description	Revision Date
1.0	<ul style="list-style-type: none">• Initial release	January 2017
1.1	<ul style="list-style-type: none">• Corrected spelling and changed title	March 2018

Purpose:

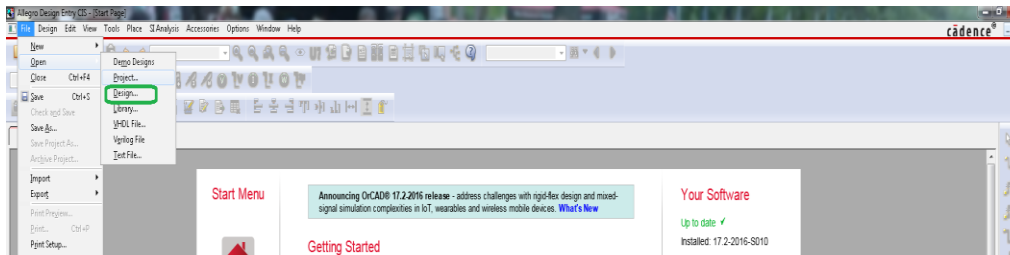
This process is applicable to DSN files created using Cadence allegro 17.2

The purpose is to create an output in the Intel Schematic Checking Format (ISCF) that would be compatible and readable with the Schematic Checking Tool.



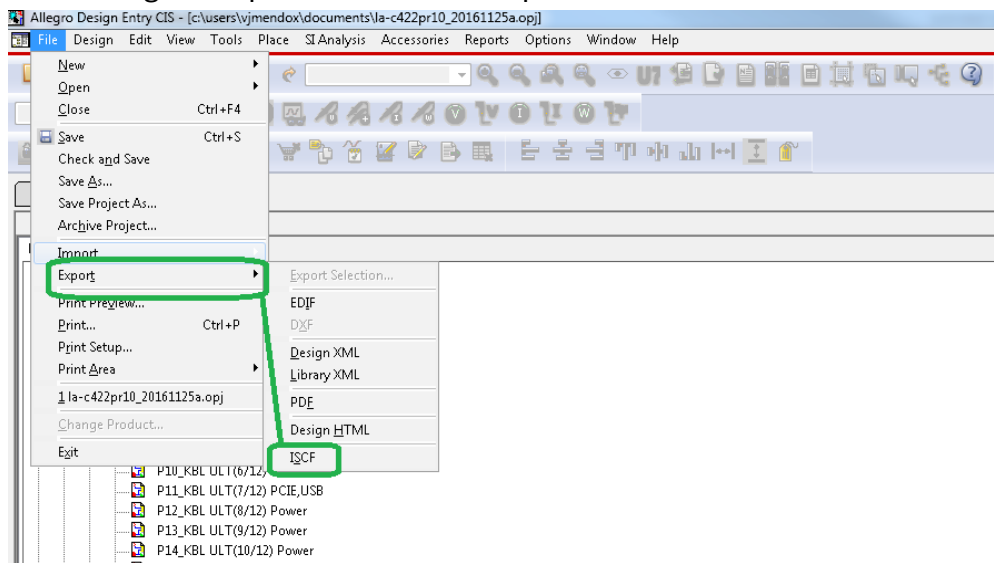
Steps to export ISCF

1. Open the design DNS file using Cadence Allegro Design Entry CIS, then select the file:

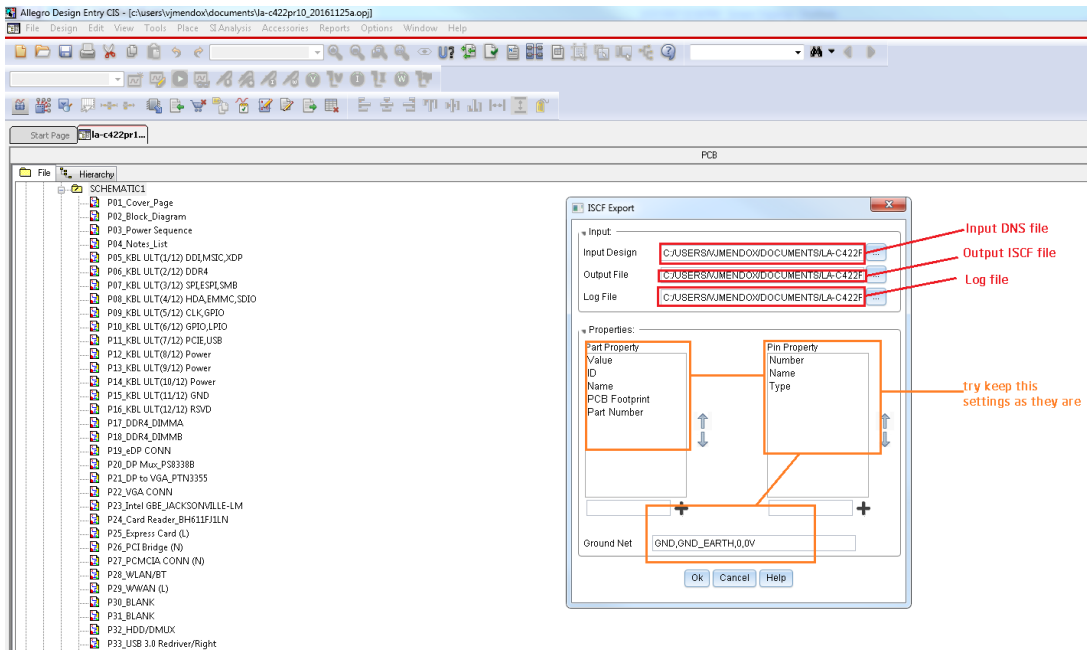


2. Then select the option for schematic (in the case that appears).

3. In the file tab go to export and select the option for ISCF:



4. The ISCF export will automatically fill in the files, then just press OK:



5. Let the converter run until a notification appears in the command line box in Cadence:



6. The files are ready, go ahead and upload the .iscf file to your ticket.