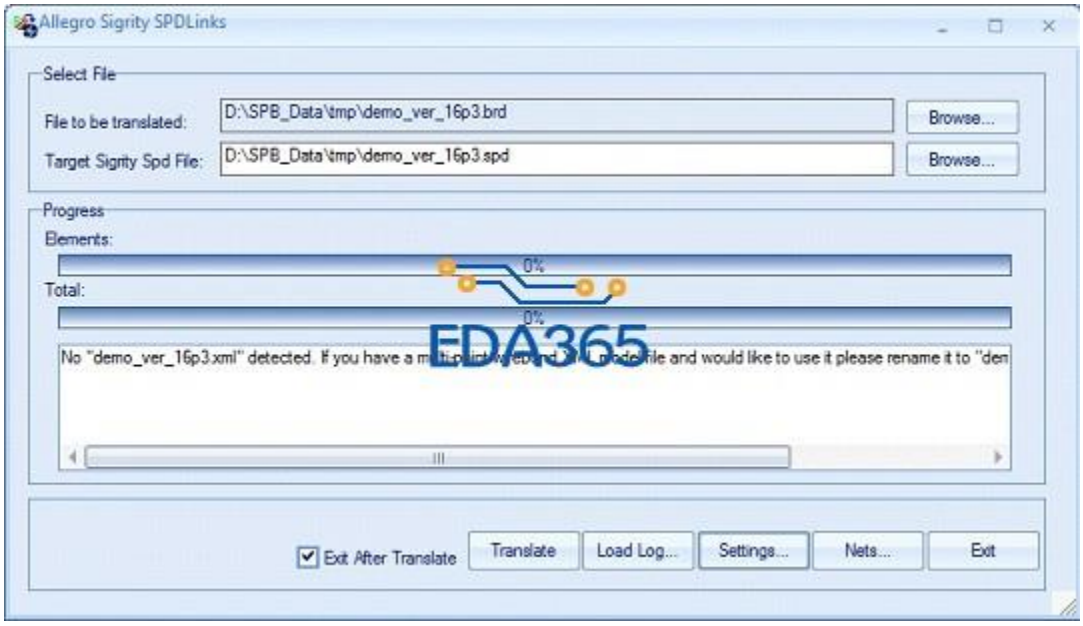
Allegro *.brd to *.spd Sigrity Conversion Procedure

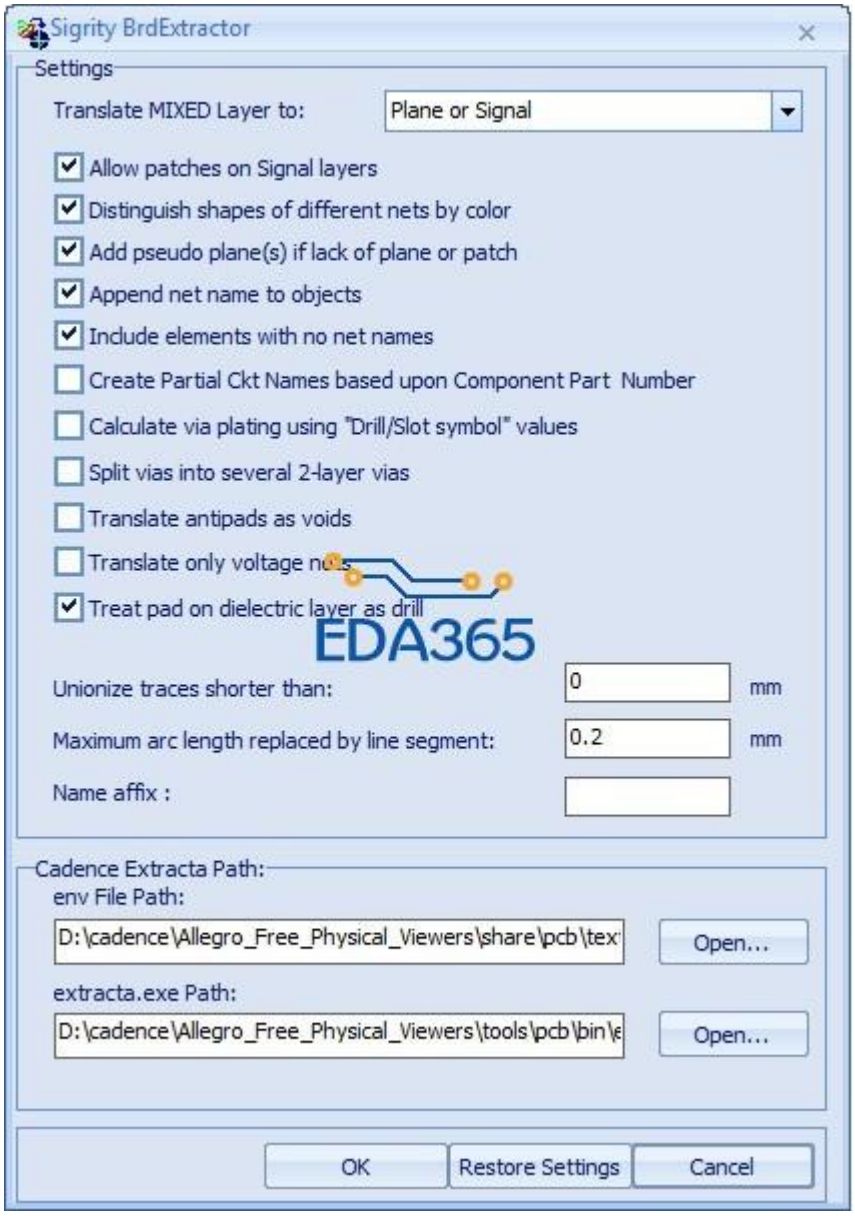
There are three general methods for how to convert Allegro/SIP design files to Sgrity's spd files:

1. The Brd/sip file is converted directly by the spdlinks tool, which is mentioned in this call. This method can also be used without the license of Allegro/SIP. The specific approach is:

A. Install Allegro Free Physical Viewer.
B. Copy extracta.exe and cdsCommon.dll from other computers that have installed Allegro/SIP. The two files are located at:
<your_spb_install_dir>\tools\bin\cdsCommon.dll
<your_spb_install_dir>\tools\pcb\bin\extracta.exe
C. Copy these two files to the tools\pcb\bin directory of the directory where Allegro Free Physical Viewer is installed, replacing the original cdsCommon.dll.
D. Open spdlinks and select the brd/sip file to be converted.



E. Click Setting in the spdlinks window and output the AFV env and extracta.exe directory in the pop-up Sigrity BrdExtractor window.
<your_AFPV_install_dir>\share\pcb\text\env
<your_AFPV_install_dir>\share\pcb\tools\bin\extracta.exe



F. OK can be converted

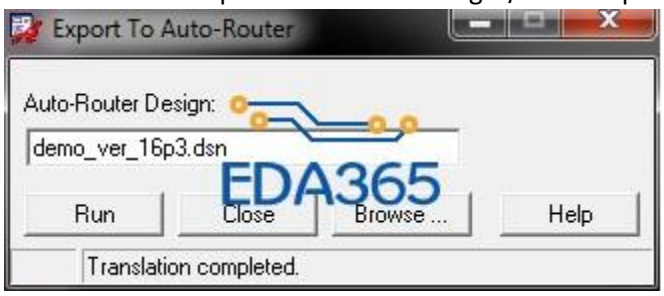
---

2. In the Allegro/SIP, first generate the ODB++ file from the design file and then convert it with the spdlinks tool.

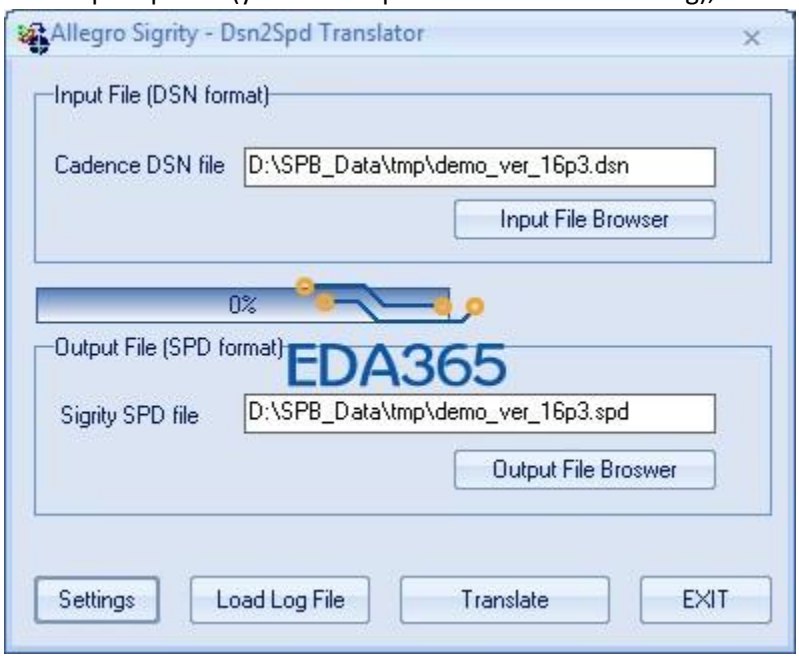
---

3. Export the design file to the dsn file in Allegro/SIP and convert it with the DSN2SPD tool. The specific method is as follows:

A. Select File -> export -> Router in Allegro/SIP to export the dsn file



B. Open the DSN2SPD tool in Allegro Sigrity Suite Manager, then select the input dsn file and the output spd file (you can keep the default in the setting), click Translate.



<u>Recommended to use ODB++, there may be data loss with DSN2SPD.</u>