

Migrating Allegro Designs to PADS Layout

To perform the migration, you must have access both to PADS Layout and to the Cadence Allegro PCB Editor in at least the XL version. If both applications reside on the same machine, use Procedure 1 to migrate the design. If they are on different machines, use Procedure 2.

Procedure 1— PADS and Allegro on the Same Machine

Perform the following steps to migrate an Allegro design to PADS Layout when PADS Layout and Allegro PCB Editor reside on the same machine:

1. Prepare the Allegro design(s) for migration, as follows:
 - a. Copy the contents of the *<PADS install dir>\SDD_HOME\translators\skill_scripts* folder to the *\$HOME\pcbenv* folder.
 - b. For each design you want to translate:
 - i. Create a new folder (...myboards_<boardID>), for example), and copy into it the Allegro design (.brd) file you want to migrate.
 - ii. Open the design file, and in the Allegro command prompt window enter these command lines:

Command> skill load "dfl_main" (include quotes)
Command> main out
 - iii. In the Allegro to Expedition Translator dialog box, click the **Start One Way Translation** button.
 - iv. When the SKILL script has completed, if any errors were found, fix them and rerun the SKILL script. The migration will not complete correctly if all errors are not fixed.
2. Migrate the prepared Allegro design(s) to PADS Layout.

Tip: If you're migrating only a single design including attributes, use the **File > Import** command in PADS Layout instead of the following procedure. After the design is migrated, it is automatically opened in PADS Layout. If you are migrating multiple designs, or want to choose whether to translate attributes or not, perform the following steps:

- a. From the Start Menu, run the Allegro to PADS Translator.
- b. In the Allegro Designs Translator dialog box:
 - i. Identify the location where you want the translated files placed.
 - ii. Use the Add button to specify the files you want to translate.
 - iii. Specify whether or not to translate attributes.
 - iv. Click the **Translate** button. The output filename(s) will be in the format
design_#####.pcb.

Procedure 2— PADS and Allegro on Different Machines

Perform the following steps to migrate an Allegro design to PADS Layout when PADS Layout and Allegro PCB Editor reside on different machines:

¹ On the Allegro machine, prepare the Allegro design(s) for migration as follows:

1. On the Allegro machine, prepare the Allegro design(s) for migration, as follows:
 - a. Copy the contents of the *<PADS install dir>\SDD_HOME\translators\skill_scripts* folder on the PADS machine to the *\$HOME\pcbenv* folder.
 - b. Copy *<PADS install dir>\SDD_HOME\translators\win32\bin\tech_translator.exe* on the PADS machine to the *\$HOME\pcbenv* folder.
 - c. Set the *AEX_BIN_ROOT* environment variable to point to *\$HOME\pcbenv*.
 - d. For each design you want to translate:
 - i. Create a new folder (...myboards_<boardID>), for example), and copy into it the Allegro design (.brd) file you want to migrate.
 - ii. Open the design file, and in the Allegro command prompt window enter these command lines:

```
Command> skill load "dfl_main" (include quotes)
Command> main out
```
 - iii. In the Allegro to Expedition Translator dialog box, click the **Start One Way Translation** button.
 - iv. When the SKILL script has completed, if any errors were found, fix them and rerun the SKILL script. The migration will not complete correctly if all errors are not fixed.
 - e. Copy all the prepared design folders (...myboards_<board*>) to the PADS machine.
2. On the PADS machine, migrate the prepared Allegro design(s) to PADS Layout.

Tip: If you're migrating only a single design and including attributes, use the [File > Import](#) command in PADS Layout instead of the following procedure. After the design is migrated, it is automatically opened in PADS Layout.
If you are migrating multiple designs, or want to choose whether to translate attributes or not, perform the following steps:

- a. From the Start Menu, run the Allegro to PADS Translator.
- b. In the Allegro Designs Translator dialog box:
 - i. Identify the location where you want the translated files placed.
 - ii. Use the Add button to specify the files you want to translate.
 - iii. Specify whether or not to translate attributes.
 - iv. Click the **Translate** button. The output filename(s) will be in the format *design_#####.pcb*.

Related Topics

[Error Codes and Log Messages](#)

Allegro to PADS Layout Translator User's Guide, PADS 9.3.1
© 1987-2011 Mentor Graphics Corporation. All rights reserved.