**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:

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

Note: Minimum of PADS V9.3.1 must be installed to pass the following files to the machine where Allegro is installed.

1. Copy the contents of the <PADS install dir>\SDD\_HOME\translators\skill\_scripts folder on the PADS machine to the $HOME\pcbenv (e.g.. C:\SPB\_Data\pcbenv) folder on the Allegro machine.
2. Copy <PADS install dir>\SDD\_HOME\translators\win32\bin\tech\_translator.exe on the PADS machine to the $HOME\pcbenv folder on the Allegro machine. This is the Allegro to Expedition translator which prepares the output folders and files under the design folder (e.g. C:\SPB\_Data\convert\_1).
3. Verify these **“System”** Variables are set prior to running “skill load dfl\_main.il” otherwise translator executable files do not properly generate correct output files and folders in the background.
* AEX\_BIN\_ROOT=$HOME\pcbenv (e.g. C:\SPB\_Data\pcbenv)
* AEX\_ENABLE\_JOBPREFS\_LAYER\_FIX=1

Note: Some environments remove the above Variables when The MGC SDD Configurator is run. If this happens, add them to **“User”** variables.

* **For each design you want to translate:**
1. Create a new folder (e.g. C:\SPB\_Data\convert\_1), and add the Allegro design file (.brd) you want to migrate.
2. Open the (.brd) file, and in the Allegro command prompt window enter these command lines:

Command> skill load “dfl\_main.il” (include quotes)

Command> main out

1. The “main out” opens the Allegro to Expedition Translator dialog box, click the Start One Way Translation button.
2. After the SKILL script has completed, any errors reported must be fixed, then rerun the SKILL script. The migration will not complete correctly if all errors are not fixed. When completed numerous folders and files are generated under design folder (e.g. C:\SPB\_Data\convert\_1).
3. Zip-up and transfer the entire design folder (e.g. C:\SPB\_Data\convert\_1) to the PADS machine.
4. On the PADS machine, unzip to any location to migrate the prepared Allegro design(s) to PADS Layout.
* Migrate the prepared Allegro design(s) (e.g. C:\SPB\_Data\convert\_1) to PADS Layout, two choices:
1. A single design, use the File > Import command in PADS Layout this procedure automatically includes attributes. When migration has completed, it is automatically opened in PADS Layout.
2. Multiple designs, or want to choose whether to translate attributes or not, perform the following steps:

1)           From the Start Menu, run the Allegro to PADS Translator.

2)           In the Allegro Designs Translator dialog box:

3)           Identify the location where you want the translated files placed.

4)           Use the Add button to specify the files you want to translate.

5)           Specify whether or not to translate attributes.

6)           Click the Translate button. The output filename(s) will be in the format design\_##########.pcb.