

xPSD For ANSYS 8.1 and 9.0

Status of xPSD:

As of the beginning of 2003, PADT has stopped officially supporting the distribution and development of xPSD. However, as long as the ANSYS community may derive some benefit from this software, PADT will continue to offer the “frozen” version (Release 1.1b) for Windows platforms only for free download on its website. Currently, this download can be found on:

<http://www.padtinc.com/support/freesw/xpsd/default.htm>, or by pointing your browser to www.padtinc.com home page and clicking on Support->Free Downloads->xPSD.

What You Get:

Once one clicks on the link above, a zipped archive file containing all the necessary files for running xPSD on a Windows platform is downloaded. The files are:

Filename	Description
mk_tabwrite.mac	macro for exporting results in spreadsheet form
mk_writecomp.mac	macro for solving only for a select component
PSDTildeCommand.dll	shared library for creating xPSD modal database
Setup.exe	install script
xspd_demo.mac	Getting Started demo ANSYS macro
xPSD_Manual_V1_1_0-b.pdf	xPSD v. 1.1 documentation
xPSDDemo.tcl	Getting started demo TCL script
xpsolve.exe	xPSD executable

Installation Instructions:

If one has ANSYS versions 5.7 thru 7.1, they may open the zip file and simply double-click on the file setup.exe, which will automatically install the software. However, since development and support of xPSD has ceased, the installation script (setup.exe) has not been updated for versions of ANSYS later than 7.1. So, for later versions, the users must install the software “manually”. The following procedure will work ANSYS versions 8.1 and 9.0.

Here’s how to install xPSD:

Step 1: Find out where ANSYS is installed. On a particular computer, this can be accomplished by either doing a Windows Explorer Search for “Ansys Inc”, or by going to the Control Panel->System->Advanced->Environment Variables .

Scroll through the System Variables list. For Ansys8.1, the variable name is ANSYS81_DIR. For Ansys 9.0, the variable is ANSYS90_DIR. In the right-hand column next to this variable is the directory path, and it usually looks something like:

```
C:\Program Files\Ansys Inc\v90\ANSYS
```

For a typical ANSYS installation, everything to the right of Ansys Inc is standard. That is to say the subdirectory structure is identical for all Windows installations. So the only variable is what's to the left of "Ansys Inc". Thus, we will refer to the variable installation location above as:

```
Install_Dir\Ansys Inc\v90\ANSYS
```

Step 2: Once the installation directory has been found, copy the file "xPSDDemo.tcl" from the zip archive to:

```
Install_Dir\Ansys Inc\vXX\ANSYS\gui\scripts
```

where vXX is either v8.1 for ANSYS8.1 or v90 for ANSYS9.0. In that same directory, you will find a file called "tclIndex". Open that file in a text editor (use Wordpad rather than Notepad because Notepad does not translate Unix newline characters correctly). Go to the very end of the file and add the line:

```
set auto_index(::::runXPSDDemo) [list source [file join $dir  
xPSDDemo.tcl]]
```

Save the file.

Step 3: Copy the file "PSDTildeCommand.dll" from the zip archive to the location:

```
Install_Dir\Ansys Inc\vXX\ANSYS\lib\intel
```

In that same directory, you will find a file called "ANS_EXT.TBL". Open the file in a text editor, go the very end of the file and add the line:

```
Install_Dir\Ansys Inc\v90\ANSYS\Lib\Intel\PSDTildeCommand.dll ~xpw  
PSDWrite
```

Making sure to replace the word "Install_Dir" with your correct top level installation directory location (for example, C:\Program Files). Also, there should be space between ~xpw and PSDWrite.

Save the file.

Step 4: Copy the files "mk_tabwrite.mac", "mk_writecomp.mac", and "xpsd_demo.mac" to the location:

```
Install_Dir\Ansys Inc\v90\ANSYS\apdl
```

Step 5: Copy the file "xpsolve.exe" to the location:

Install_Dir\Ansys Inc\v90\ANSYS\bin\intel

Step 6: For the final step, copy the file “xPSD_Manual_V1_1_0-b.pdf” to any location you want. This is the documentation for xPSD.

Getting Started:

It is highly recommended that the new user run the demo first by issuing the command “xpsd_demo” at the ANSYS command prompt. The demo is interactive and between each step, the user is encouraged to review the model being created. The demo works by passing ANSYS commands to ANSYS via a higher level script (TCL). Because of this, the commands may be reviewed in the log file at the end of each step, and all of the model’s characteristics (real constants, material properties, etc.) may be queried. In particular, the user should note that the large mass method is being used. To use xPSD, the user must attach a large mass to the “base” of the structure – the demo demonstrates how this may be done for a simple flat plate example. The user should also pay careful attention to how the modes are extracted, how the xpsolve executable is solved, and what input files this solver uses. Again, all this is demonstrated in the demo. For questions, the user should consult the documentation (file “xPSD_Manual_V1_1_0-b.pdf”).