

# The Focus



A Publication for ANSYS Users

## Contents

### Feature Articles

- [DO Loops – Program Flow](#)
- [xPSD – Speed up your PSD runs!](#)

### On the Web

- [ANSYS Users' Conference 2002](#)
- [PADT: 3D Printing with Dimension](#)

### Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About \*The Focus\*](#)
  - *The Focus* Library
  - Contributor Information
  - Subscribe / Unsubscribe
  - Legal Disclaimer

### Do Loops

Time	DO	DO	DO	DO
1.0000	0.000000	0.000000	0.000000	0.000000
2.0000	0.000000	0.000000	0.000000	0.000000
3.0000	0.000000	0.000000	0.000000	0.000000
4.0000	0.000000	0.000000	0.000000	0.000000
5.0000	0.000000	0.000000	0.000000	0.000000
6.0000	0.000000	0.000000	0.000000	0.000000
7.0000	0.000000	0.000000	0.000000	0.000000
8.0000	0.000000	0.000000	0.000000	0.000000
9.0000	0.000000	0.000000	0.000000	0.000000
10.0000	0.000000	0.000000	0.000000	0.000000
11.0000	0.000000	0.000000	0.000000	0.000000
12.0000	0.000000	0.000000	0.000000	0.000000
13.0000	0.000000	0.000000	0.000000	0.000000
14.0000	0.000000	0.000000	0.000000	0.000000
15.0000	0.000000	0.000000	0.000000	0.000000
16.0000	0.000000	0.000000	0.000000	0.000000
17.0000	0.000000	0.000000	0.000000	0.000000
18.0000	0.000000	0.000000	0.000000	0.000000
19.0000	0.000000	0.000000	0.000000	0.000000
20.0000	0.000000	0.000000	0.000000	0.000000



... Also

- Speed Up your PSD Runs
- 3-D Printing



**ANSYS**  
Support Distributor

© 2002, by Phoenix Analysis & Design Technologies, Inc. All rights reserved.

# The Focus



A Publication for ANSYS Users

## DO Loops Program Flow

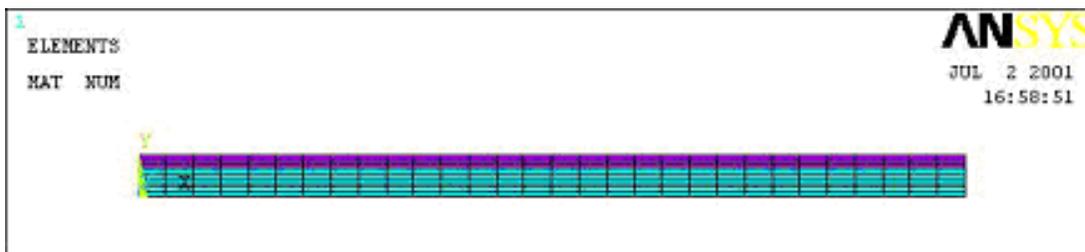
For those who use APDL, the ANSYS command-line language, DO loops and IF-THEN logic is essential for controlling program flow. If you only use the GUI (Menu Picks) consider adding DO Loops to begin learning ANSYS macro-writing. If you have a repetitive task, using a DO loop can result in a huge time savings:

### Case Study

Apply many different uniform temperatures to a bi-metallic strip and store each solution.

#### Geometry

Use either the [input file](#) or the [equivalent menu picks](#) to generate the geometry shown below. Save as bistrip.db.



Geometry

#### Boundary Conditions

Use either the [input file](#) or the [equivalent menu picks](#) to create the following boundary conditions.

- Apply uniform temperatures from 100, 110, 120, ..., 200.
- The two strips are bonded together by the geometry, so they share nodes.
- Fix the lines at the left end.

### Approach

- A. Develop model for single temperature using GUI
- B. Build DO Loop Macro
- C. Run Macro
- D. Review Results

# The Focus



A Publication for ANSYS Users

## A. Develop working model for single temperature using GUI

1. Build your model using provided input files ([1](#) and [2](#)) or menu picks ([1](#) and [2](#)).
2. Verify that your model solves correctly.

Input Commands Menu Picks

/SOLU /Solution > Solve - Current LS

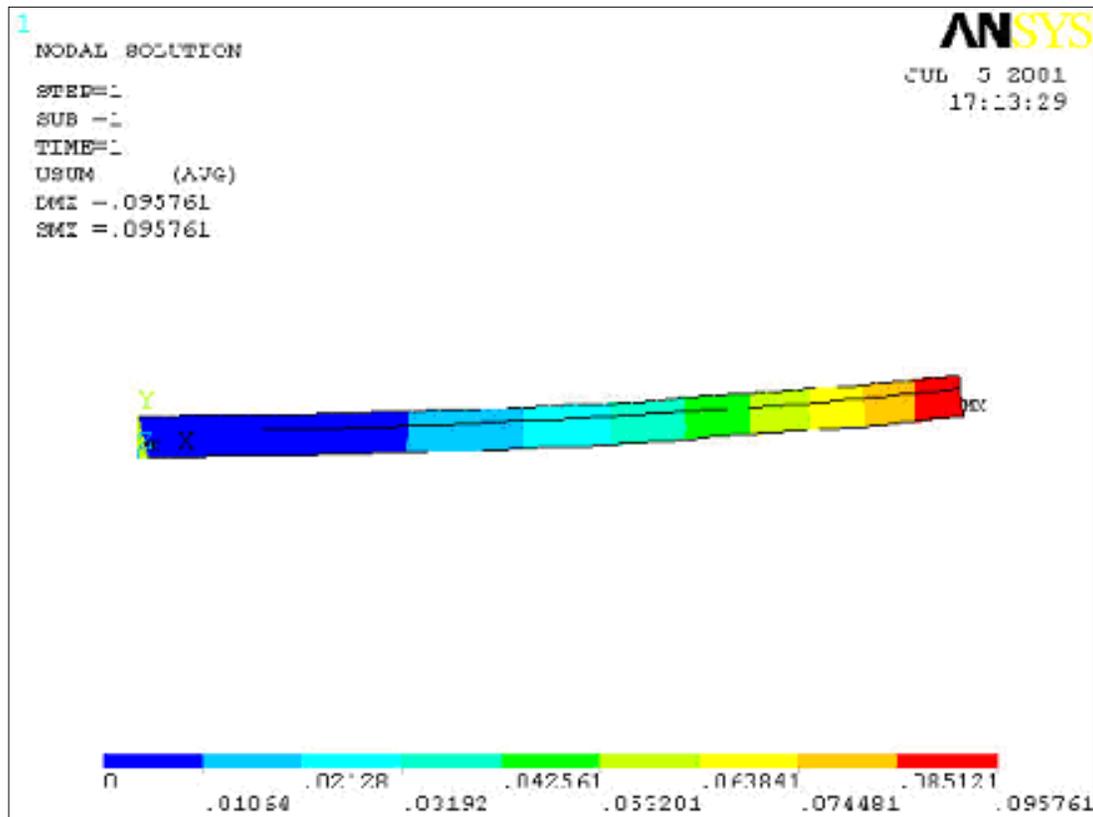
SOLVE OK

/POST1

PLNSOL,U,SUM Postprocessor > Plot Results > Contour Plot - Nodal Solution

Translation USUM

OK



3. Save the model to bistrup.db.

# The Focus



A Publication for ANSYS Users

## B. Build DO Loop Macro

To utilize DO loops you must use APDL (ANSYS Parametric Design Language), a.k.a. macros, a.k.a. scripts. We will take advantage of one of the simplest of macros:

- Resume from a .db file
- Perform a single operation
- Solve

Although you can type \*DO commands into the ANSYS input window, here we will use a small macro to implement the DO loop. This DO loop will repeat the TUNIF and SOLVE commands.

We will now build the "macro" needed to implement \*DO.

1. Open a text editor. Select one that doesn't store hidden formatting characters.
2. Copy the following template:

```

fini
/clear
RESUME,bistrip,db !resumes saved database
*DO, icount,1,10,1 !counts on variable icount from 1 to 10 times

/Solu          !switches to solution processor
TUNIF,100+icount*10 !GUTS: the action looped on sets new uniform temp.
TIME,icount     !Note how it uses the variable icount
Solve          !Solve with the new temperature settings
*ENDDO         !Go back and begin next loop on variable icount
  
```

3. Save as bistrip.mac. Note that saving any text file with extension .mac automatically creates a macro such that by simply typing the filename, such as bistrip at the ANSYS command line will cause ANSYS to check for the bistrip.mac file and read it as a series of input commands. ANSYS checks the working directory, the installation/docu directory and any directory specified by the /PSEARCH command for .mac files.

## C. Run Macro

1. Inside a fresh session of ANSYS, type bistrip at the command line.

# The Focus



A Publication for ANSYS Users

## D. Review Results

1. General Postprocessor > Results Summary. You should see a listing similar to this:

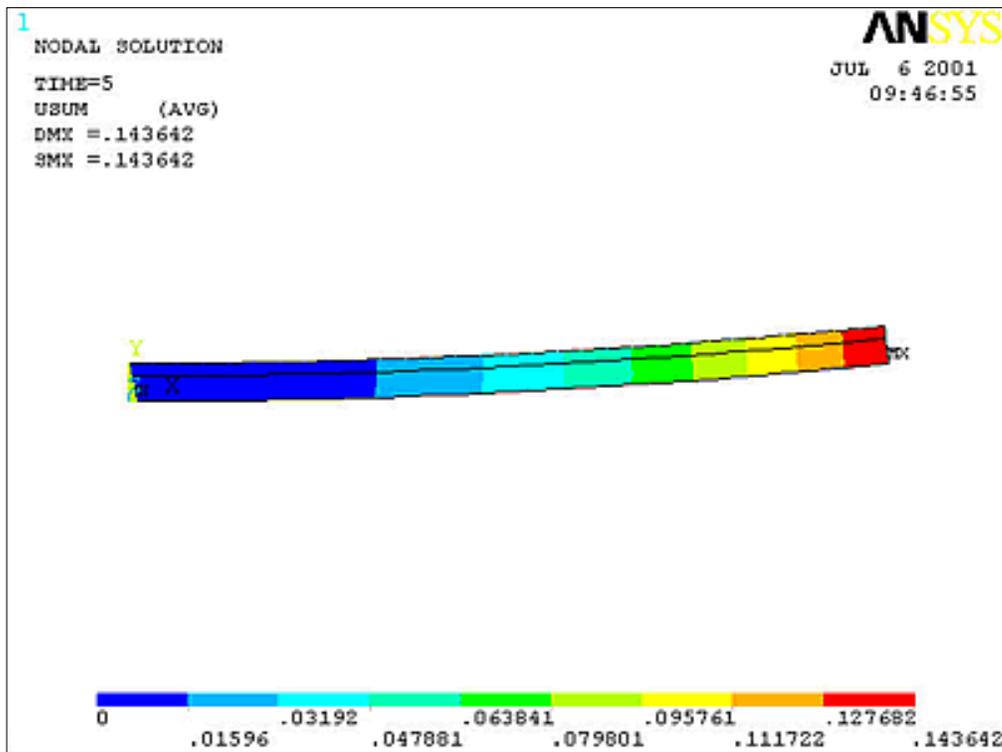
Set	Time	Load Step	Substep	Cumulative
1	1.0000	1	1	1
2	2.0000	2	1	2
3	3.0000	3	1	3
4	4.0000	4	1	4
5	5.0000	5	1	5
6	6.0000	6	1	6
7	7.0000	7	1	7
8	8.0000	8	1	8
9	9.0000	9	1	9
10	10.0000	10	1	10

2. General Postprocessor > Read Results - By Time/Freq. Select Time=5 which corresponds to the 150°F case.
3. General Postprocessor > Plot Results > Contour Plots - Nodal Solution > Deflection > USUM. It should look like this:

# The Focus



A Publication for ANSYS Users



4. Review each of the 10 results in the results set, or use Time History Post-Processing to summarize the results!

# The Focus



A Publication for ANSYS Users

The screenshot shows a window titled 'PRVAR Command' with a 'File' menu. The main content area displays the following text:

```
***** ANSYS POST26 VARIABLE LISTING *****
```

TIME	218 UY	270 S	EQV	270 EPTHY
	UY	SX		EPTHY
1.0000	0.104994	4800.69		0.880000E-03
2.0000	0.114539	5237.11		0.960000E-03
3.0000	0.124084	5673.54		0.104000E-02
4.0000	0.133629	6109.96		0.112000E-02
5.0000	0.143174	6546.39		0.120000E-02
6.0000	0.152719	6982.81		0.128000E-02
7.0000	0.162264	7419.24		0.136000E-02
8.0000	0.171809	7855.67		0.144000E-02
9.0000	0.181354	8292.09		0.152000E-02
10.000	0.190899	8728.52		0.160000E-02

Now that you have BISTRIP.MAC, use it as a template to write other macros needing DO loops or to automate any task. As you build a library of macros, your ANSYS productivity will greatly increase!

## Bonus Tip

Variable values are stored within a database. Therefore, if you resume a database within a loop, the variable that is being looped on (i or icount, for example) will no longer be defined! This will cause errors. If you ever need to resume a database without changing current variable values, utilize the PARSAV and PARRES commands.

# The Focus

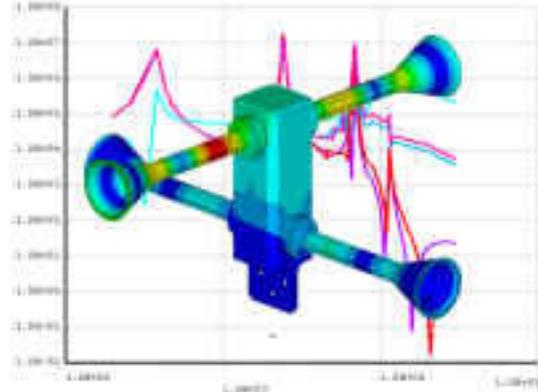


A Publication for ANSYS Users

## xPSD Speed up your PSD runs!

xPSD for ANSYS BETA TESTING NOW!

One of the most important types of mechanical simulation is the prediction of a system's behavior due to random vibration. Almost every type of machinery, from computer circuit cards to satellite sub-assemblies, undergoes some type of random vibration, and it is the job of the analyst to determine if the system can stand up to that loading. xPSD is a tool from PADT that allows a user to take the results from a standard modal analysis and conduct fast and flexible studies on the behavior of the structure under just such loading.



xPSD uses the large mass method to capture the modal response of a system to random vibration expressed in terms of a Power Spectral Density (PSD) input. The user can quickly specify many different PSD input curves and extract the response of the system. The goal of the tool is to provide a simple but fast and accurate way for users to carry out PSD analysis. In fact, on a typical model (300,000 DOFs, 200+ modes) nodal response and 1-sigma values can be calculated in a matter of minutes.

The first version of the tool, xPSD for ANSYS, works with the latest version of the ANSYS general purpose FE code and has the following features:

- Supports Single Base Excitation
- Utilizes a Standard Modal Analysis (with large mass) as Input
- Stores modal information in efficient binary file that is an order of magnitude smaller than the equivalent \*.RST modal result file
- Solves in minutes or seconds
- User can request displacement, velocity, acceleration, or component stress response at any node
- User can request 1-sigma displacement, velocity, acceleration, or component stress information for the entire model or for sub-components of the model

# The Focus



A Publication for ANSYS Users

- Response curves can be output for display in ANSYS or a spreadsheet
- 1-sigma results can be output for contour display in ANSYS
- Entire process can be run from within ANSYS, either interactively or in batch
- Input PSD and output request controlled by simple text file

This initial release was developed specifically to address the immediate needs of one of PADT's large aerospace customers. It is aimed at delivering to the user community the capabilities listed above with minimal development time, and therefore does not have a lot of bells and whistles. Future releases will include:

- Graphical User Interface
- Direct output to an ANSYS \*.RST file
- Multiple point forced excitation
- One or more statistical approximations for the 1-sigma invariant stress values (principal and Von Mises stresses)
- Support for other FE solvers and generic modal results
- User requested enhancements

PADT has found this tool to be very useful, resulting in significant productivity gains on our own internal and external analysis projects.

For more information on xPSD, please see the xPSD portion of the PADT web site.

# The Focus



A Publication for ANSYS Users

## About *The Focus*

*The Focus* is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

## *The Focus* Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

## Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

## Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

## Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.