

# The Focus



A Publication for ANSYS Users

## Contents

### Feature Articles

- [Modal Cyclic Symmetry and You, Version 6.0](#)
- [File Jobname.anf Does Not Exist](#)
- [Do Loop Techniques](#)

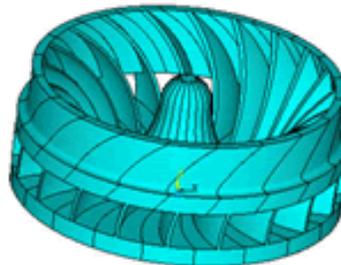
### On the Web

- [ANSYS 6.1 Release](#)

### Resources

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

### The New Cyclo-Symmetry



... Also

- [Pro/E Connection Setup](#)
- [Implied Do Loops](#)



**ANSYS**  
Support Distributor

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

# The Focus



A Publication for ANSYS Users

## Modal Cyclic Symmetry and You, Version 6.0

by [Jeff Psycho-Symmetry Strain](#), Structural Analyst and ANSYS Support Guy

With the release of ANSYS 6.0 came a new procedure for performing modal cyclic symmetry analysis. The new method is the preferred method, although the help system does not document the procedure in much detail. In this article, I will guide you through the new modal cyclic symmetry analysis procedure available in version 6.0 of ANSYS and compare it to the method used in previous versions. I will cover the most common usage of the commands involved although I recommend reading about them further since each one has a number of options available.

### A Brief Definition

Cyclic symmetry is characterized by a sector that repeats itself  $360^\circ$  about an axis. Examples include turbine wheels, gears, and the sword from Krull. This feature lends itself to a simplified analysis where only one sector of the part has to be modeled. As with any form of symmetry modeling, this allows for a finer mesh and/or a lower amount of CPU time to solve compared to that of a full model.

### Preprocessing

The first thing you'll want to do is build the geometry for a single sector of the part, either through importing or creating it in ANSYS. Although the geometry needn't be created in the global cylindrical coordinate system, it does have to lie in a cylindrical coordinate system, be it local or global.

After the geometry is modeled, and typically before meshing, issue the command CYCLIC (accept defaults). Assuming the geometry lends itself to cyclic symmetry, the CYCLIC command detects the coordinate system the geometry was built in, detects the symmetry planes and creates components from them, configures the symmetry planes so that the nodes match up with one another (this is necessary in generating the constraint equations that enforce cyclic symmetry), detects the sector angle, and calculates the number of sectors necessary to complete the full  $360^\circ$ . Prior to ANSYS 6.0, this all had to be done manually, and still can be controlled manually through the CYCLIC command arguments. You can also execute this command after meshing the geometry; just make sure the nodes on the symmetry planes match up with one another

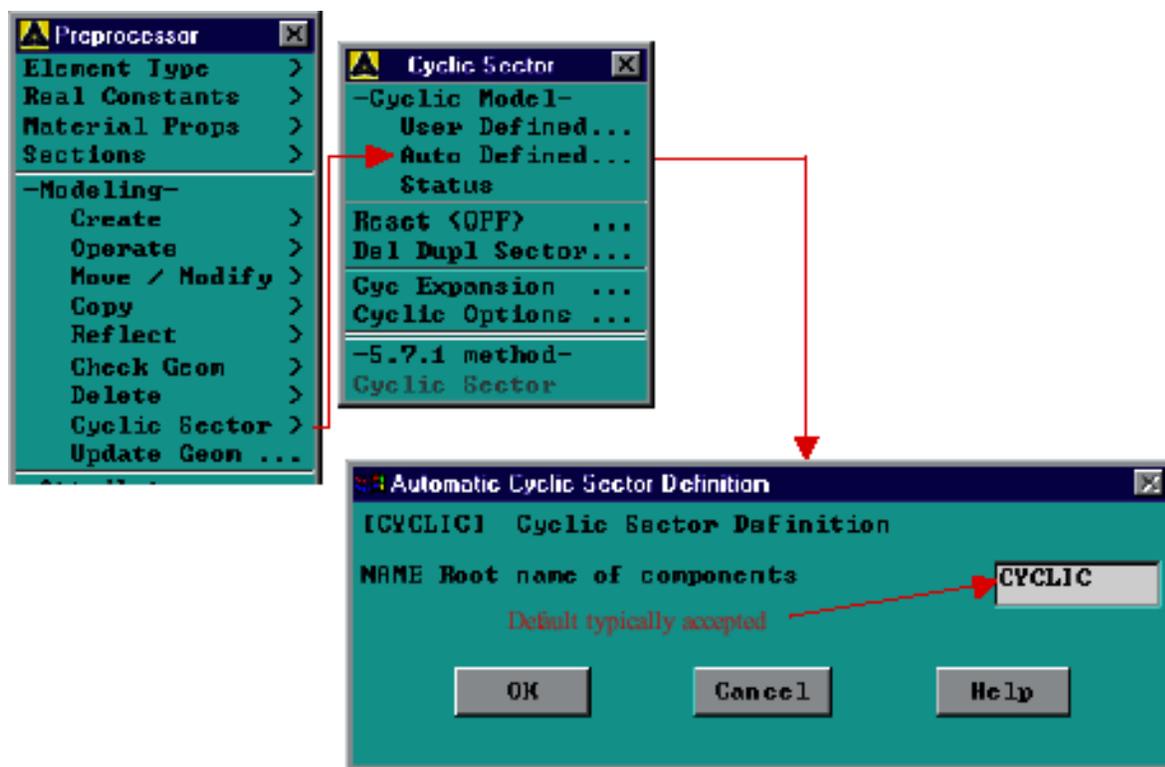
# The Focus



A Publication for ANSYS Users

and you can ignore the warning that ANSYS gives. Once the CYCLIC command is executed, ANSYS will echo in the Output window the number of sectors needed to complete the full 360°, the components created in defining the symmetry plane, and whether the component pairs are matched or unmatched. My experience has been that unmatched components do not necessarily indicate impending failure of the solution; as long as there is at least one set of matched components, the solution usually progresses smoothly. Please note the use of the word *usually* in the previous sentence.

After this, mesh the model as you normally would. As dictated about 500 times throughout the [Dynamics](#) course, Remember density!



## Solution Processing

After your model is meshed, enter the Solution processor. Specify modal analysis, extraction method (hint: Block-Lanczos), number of modes to extract and expand, and frequency range as you would for any other modal analysis. Constrain your model appropriately. Then issue the command `CYCOPT,NODDIA,ndlow,ndhigh`, where `ndlow` and `ndhigh` are the lower and upper bounds of the nodal diameter range for which you want to solve. Again, `CYCOPT` has many other functions as well, although its most common use is to specify nodal diameters.

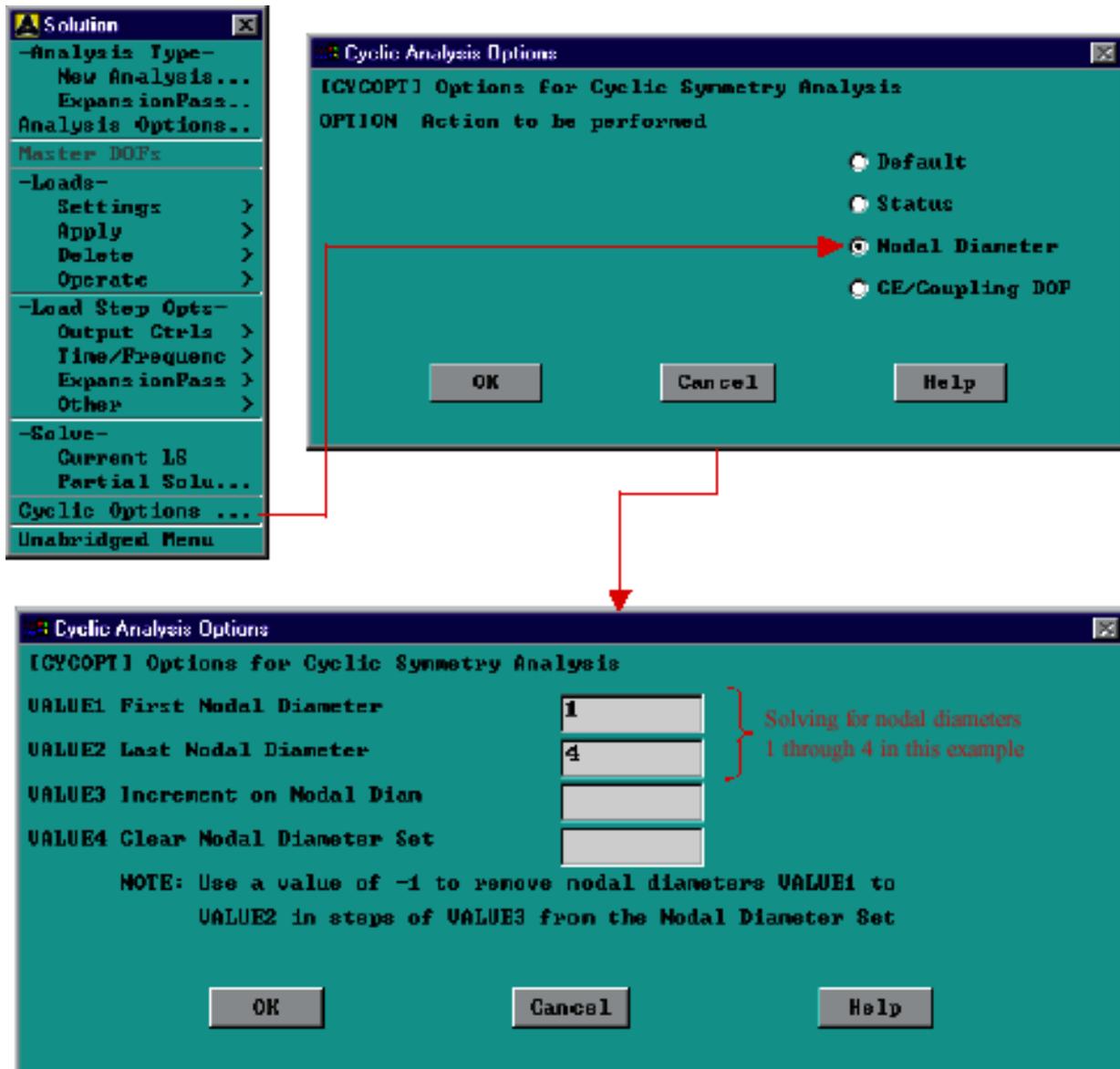
Execute the `SOLVE` command. One thing your humble narrator likes about the new

# The Focus



A Publication for ANSYS Users

procedure is that you can check the solution status before allowing the modal cyclic symmetry analysis to continue. Once the SOLVE command is executed, ANSYS will automatically generate a duplicate sector and copy the boundary conditions to it (you used to have to perform both of these operations manually), generate constraint equations between the symmetry planes, and proceed with the solution.



## Postprocessing

After your model has run successfully, enter the General Postprocessor and read in the results for the desired frequency. As before, the nodal diameters are grouped in load steps and the frequencies are grouped in substeps. My preference is to click on Results

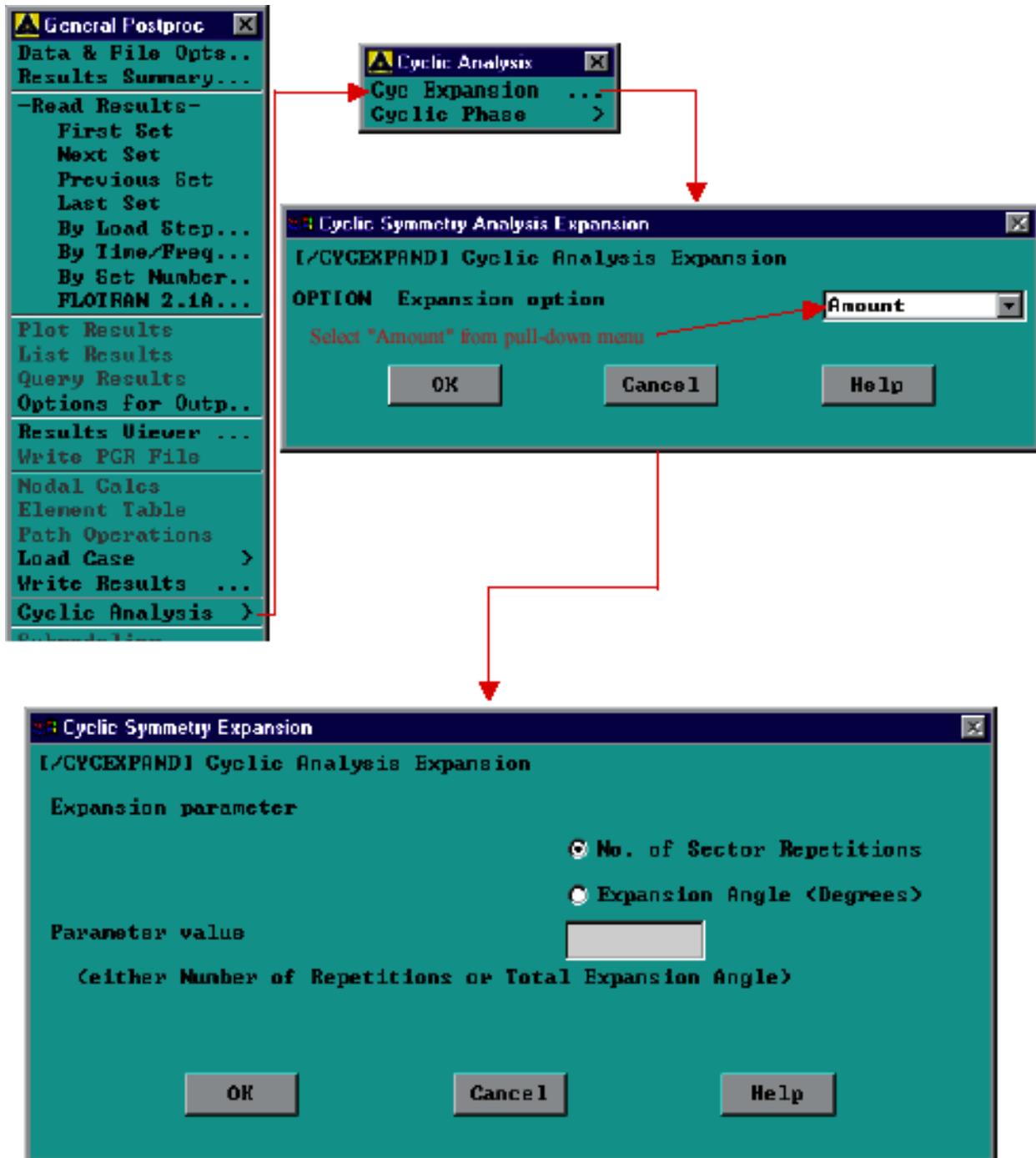
# The Focus



A Publication for ANSYS Users

Summary and select the frequency of interest from the list.

Expand the results through the full 360°. Do this by executing the command /CYCEXPAND,,AMOUNT,ANGLE. The default argument after ANGLE is 360° but you have the option of specifying a different angle to which to expand.

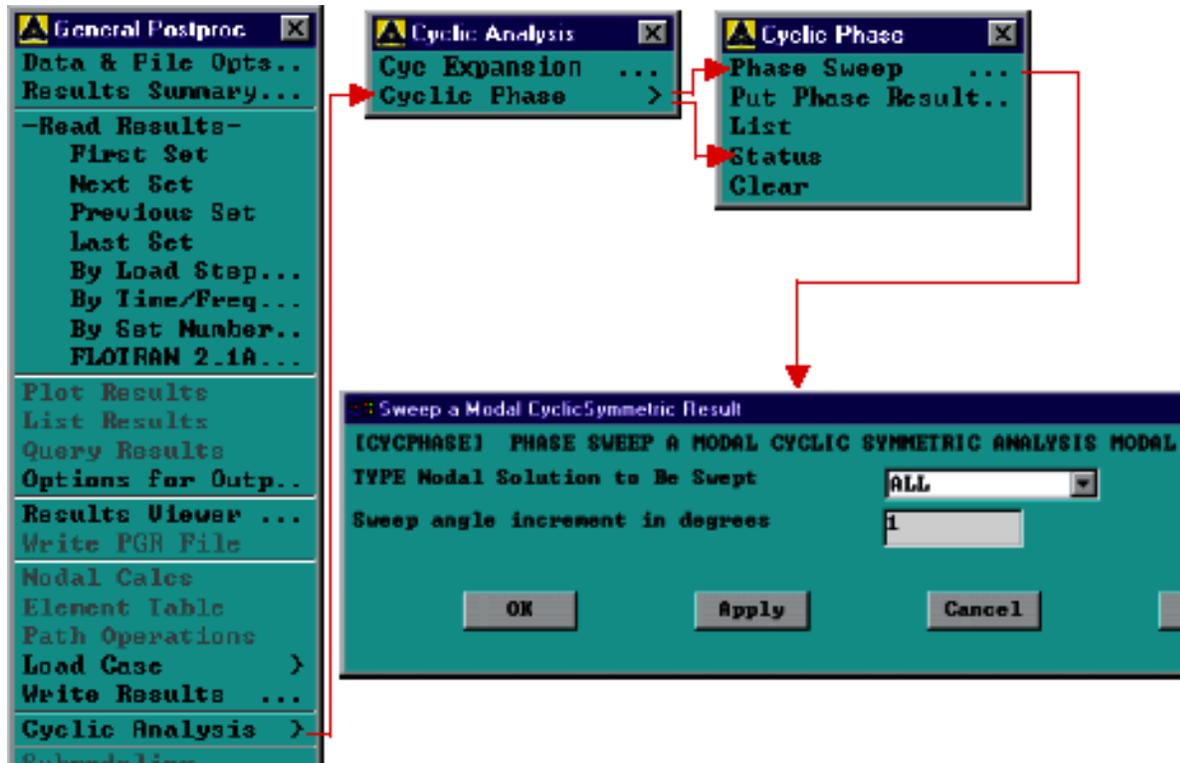


# The Focus



A Publication for ANSYS Users

Calculate the phase angles at which the maximum strains, displacements, and stresses occur. To do this, issue CYCPHASE,ALL. This process may take a few minutes. After this is complete, you can list the maximum results values and the phase angles at which they occur by entering CYCPHASE,STAT.

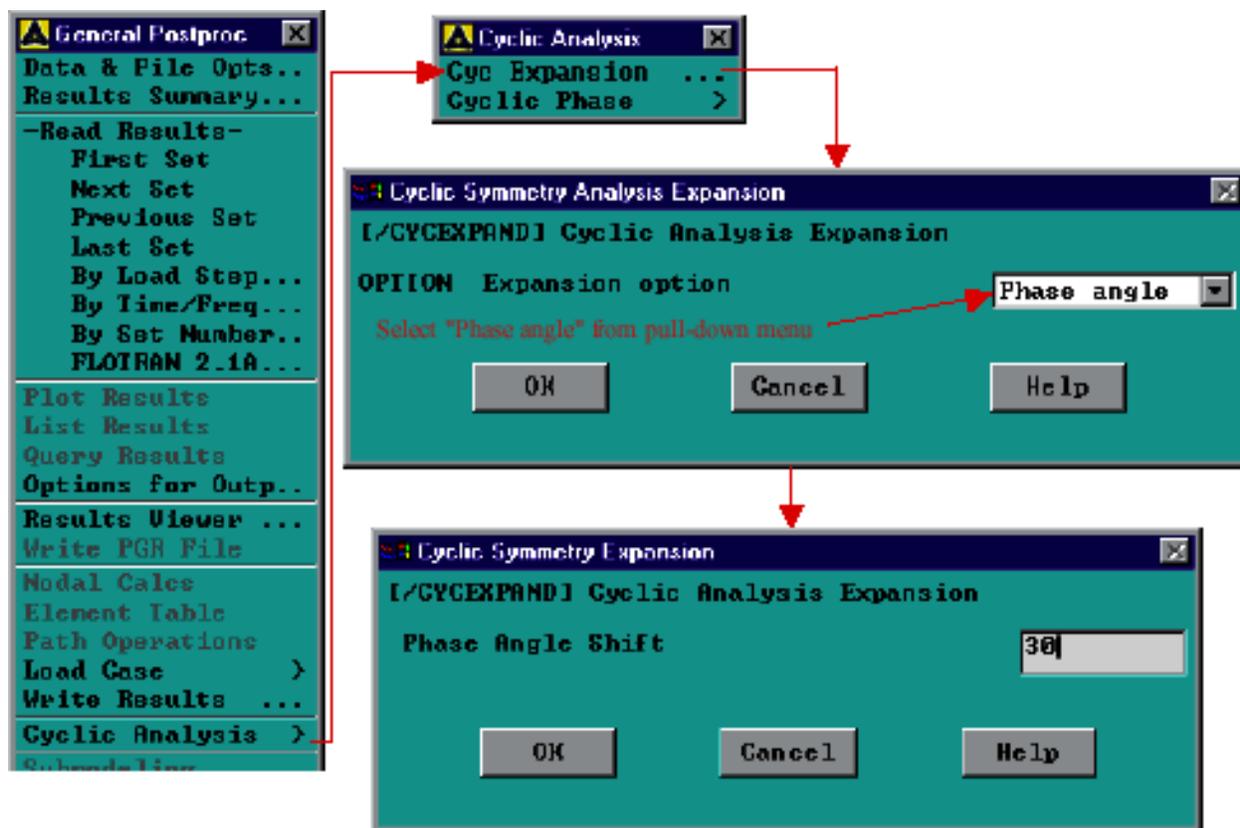


Note the phase angle at which the maximum quantity of interest occurs and expand to it using the PHASEANG argument in the /CYCEXPAND command. For example, if you were interested in plotting the critical first principle stresses and noted that the maximum value occurred at a phase angle of  $30^\circ$  from the CYCPHASE,STAT listing, you would issue /CYCEXPAND,,PHASEANG,30 followed by PLNSOL,S,1.

# The Focus



A Publication for ANSYS Users



## Conclusion

I encourage you to read up on the commands mentioned in this article since I only touched on the full capability of each. In 6.1, more modal cyclic analysis enhancements have been added, so stay tuned to the continuing saga. If you have any questions about performing modal cyclic symmetry analysis, or any other questions about ANSYS, please call PADT at 480-813-4884 or email us at [techsupport@padtinc.com](mailto:techsupport@padtinc.com).

# The Focus



A Publication for ANSYS Users

## File Jobname.anf Does Not Exist

by [Jeff Strain](#), ANSYS Tech Support/Training Guy

A common question making frequent appearances in the ANSYS Technical Support Database as of late concerns the issuance of the File jobname.anf does not exist error when the user is attempting to import a Pro/E part or assembly file into ANSYS. As you will see in this article, any number of errors occurring during the Pro/E file import process can lead to ANSYS failure to find the elusive jobname.anf file. In order to help you more effectively use the ANSYS Connection for Pro/E product, I will explore three main topics in detail:

- The process ANSYS uses to import a Pro/E file
- How to set up the Connection for Pro/E product
- Tips for troubleshooting should you encounter problems with the Pro/E Connection

### Pro/E Part Import Process

In order to gain an understanding of the errors that may be encountered when ANSYS is importing a Pro/E file, it is important to take a look at what goes on behind the scenes after the ~PROEIN command is executed via the Input window or Utility Menu > File > Import > Pro/E.

1. First, and probably most importantly, ANSYS will open Pro/E in batch mode and load the part you've specified. The success of this is indicated by a long, cryptic command echoed in the output window, as seen below. Also, a DOS or X11 window will open indicating that Pro/E has been started and the geometry is being transferred, as seen at the top of the next page. Since ANSYS must run Pro/E to import a file, it is imperative that (1) Pro/E be installed on the user's machine and (2) the Pro/E Command entered in the ANSYS Connection for Pro/E must be valid.
2. The connection product reads the geometry data from Pro/E and writes it into a jobname.anf file. The jobname.anf file is an ASCII file, similar to a jobname.cdb file, containing commands that are recognizable by ANSYS. Another ASCII file, jobname.pro is created during this process as well. This file contains information about the resulting ANSYS geometry.
3. Once the translation is complete, the connection product reads the jobname.anf file into ANSYS, essentially executing the command /INPUT,jobname,ANF. An important thing to note is that the connection product will attempt to input the jobname.anf file from the working directory, regardless of its success in performing any of the previously mentioned steps. Therefore, any errors encountered during the translation process will ultimately result in a File jobname.anf does not exist error. Thus it is necessary for the user to trace back through all error and warning messages issued while the connection product was running to better pinpoint the cause of the problem.

# The Focus



A Publication for ANSYS Users

```

ANSYS 6.0 Output Window
ACTIVATING THE GRAPHICAL USER INTERFACE (GUI). PLEASE WAIT...

ANSYS Connection for Pro/ENGINEER

Pro Part:      proepart.prt
Pro/E Command: proe2001
File Type:     ANF
Model Type:    Part

QUERY = COMMON,,IGES_TOL,,INT,5

E:\proepartdir>"proe2001" pro_wait -g:no_graphics -FILE:proepart -TYP
pro_wait -g:no_graphics -FILE:proepart -TYPE:prt -PATH= -GT:0 1>

/INPUT FILE= proepart.anf  LINE=      0

*** ANSYS GEOMETRY NEUTRAL FILE ***

ANSYS60 Connection for Pro/ENGINEER

D:\Program Files\Ansys Inc\ANSYS60\ac4\bin\pro\intel\ac4pro.exe

Checking out a connection license...

*** NOTICE ***
Masif: The license for product ANSYS Connection for Pro/ENGINEER
(feature 'conproe') will expire in 25 day(s)!
Please contact your ANSYS/DesignSpace Sales Representative
or ANSYS, Inc.

Begin transfer of Pro/E geometry to ANSYS
-
  
```

## Setting up ANSYS Connection for Pro/E

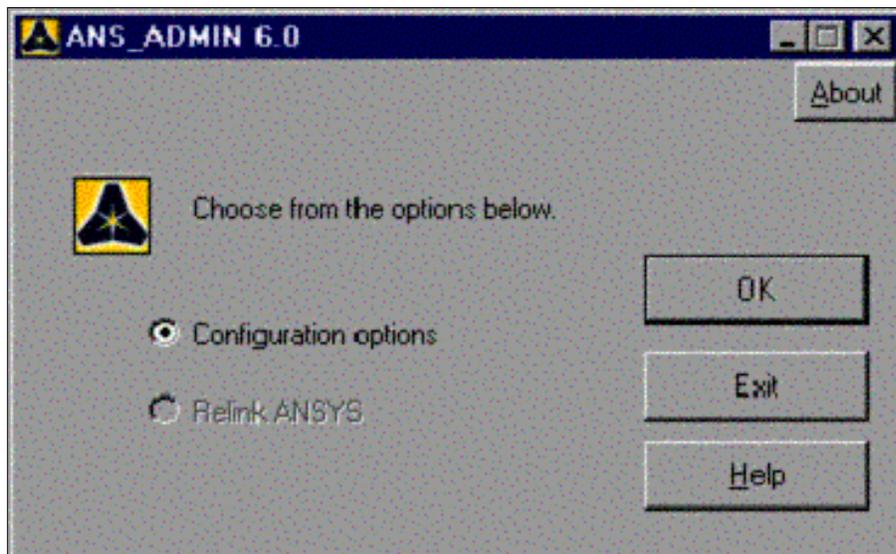
order to get the Pro/E connection product working in the first place, it is important to configure it after installation so that it knows where to find the Pro/E command file. The configuration procedure is as follows.

1. Open the ANS\_ADMIN Utility. In Windows, click on the Start Button and select ANSYS 6.0 (or ANSYS 5.7 or ANSYS 5.6, depending on what version you have) > ANS\_ADMIN Utility. In UNIX, type /ansys\_inc/ansys60/bin/ans\_admin60 and hit enter (directory and command obviously will vary depending on your version of ANSYS).
2. In the ANS\_ADMIN Utility window that comes up, select Configuration options.

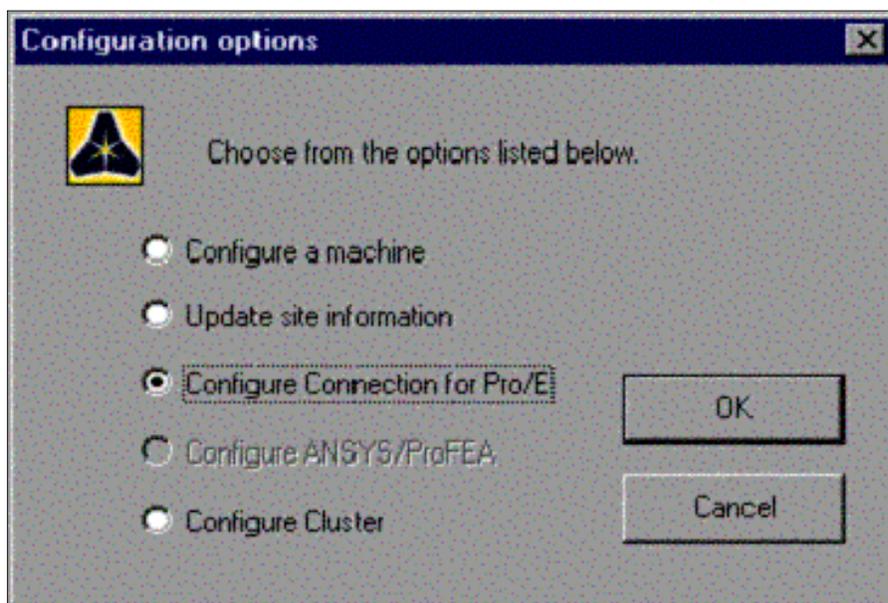
# The Focus



A Publication for ANSYS Users



3. In the next window, choose Configure Connection for Pro/E.

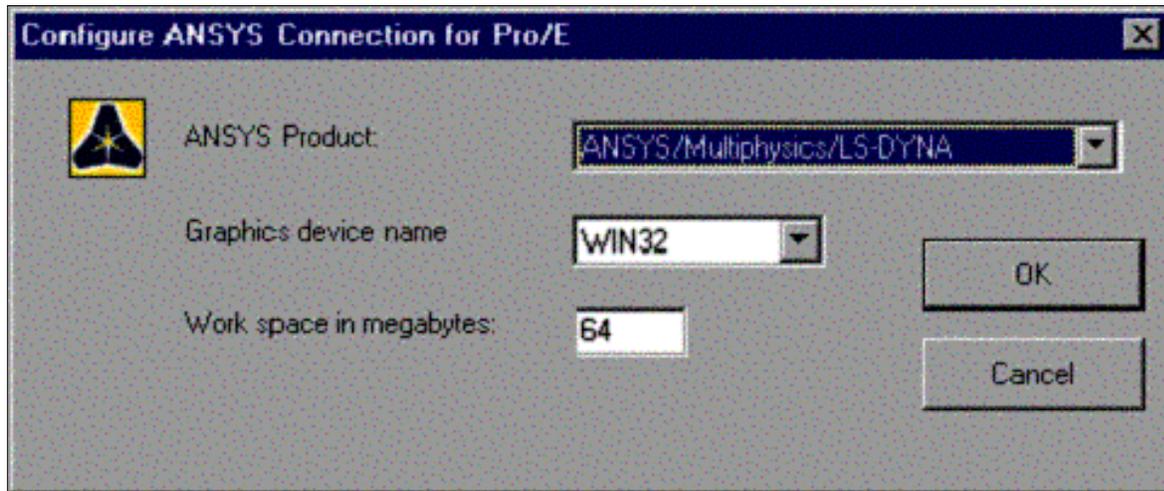


4. In the following window, select the appropriate product if necessary and click the OK button.

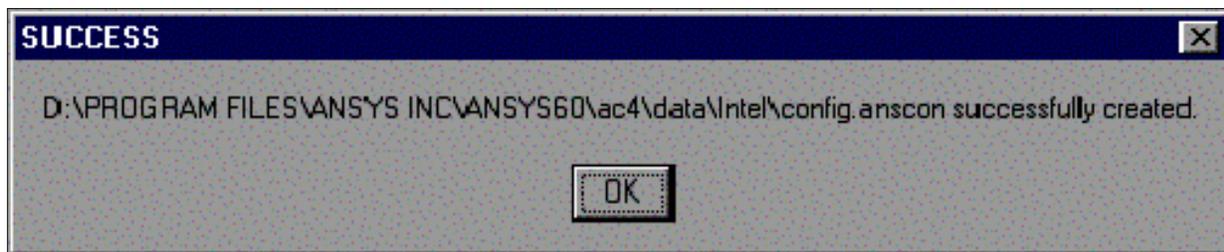
# The Focus



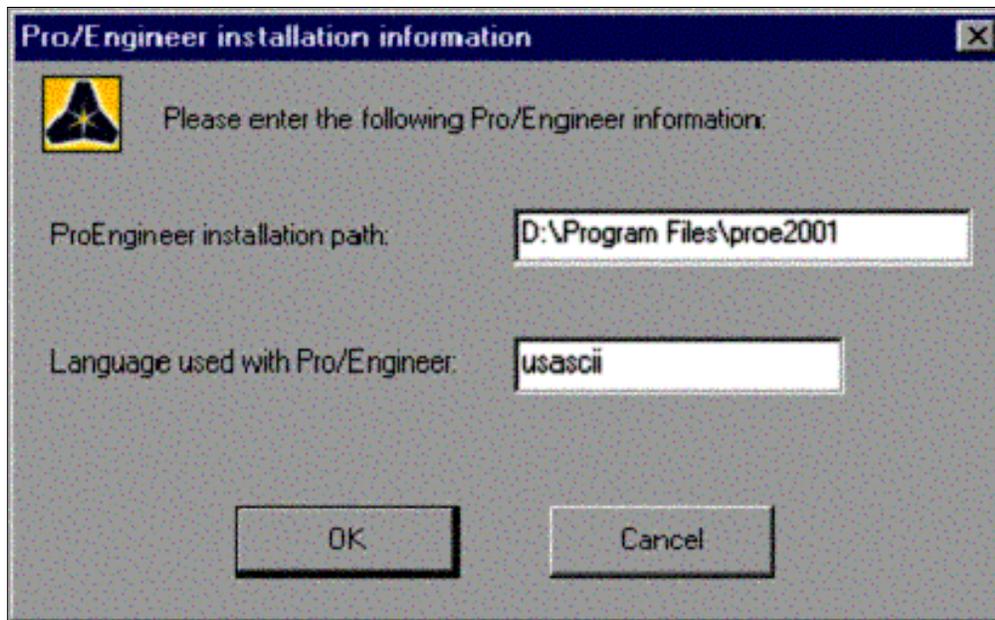
A Publication for ANSYS Users



5. You will get a message stating that the config.anscon file has been successfully created.



6. When prompted, enter the Pro/E installation path, for example D:\Program Files\proe2001. Be sure not to include the bin subdirectory.



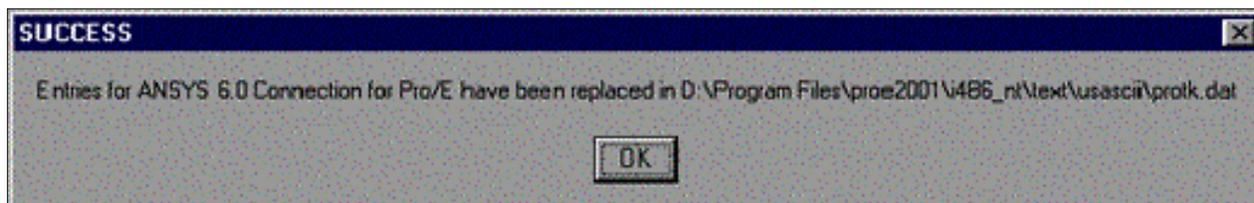
7. Upon successful completion of the Connection for Pro/E configuration, you will get a message saying, Entries for ANSYS 6.0 Connection have been replaced in

# The Focus

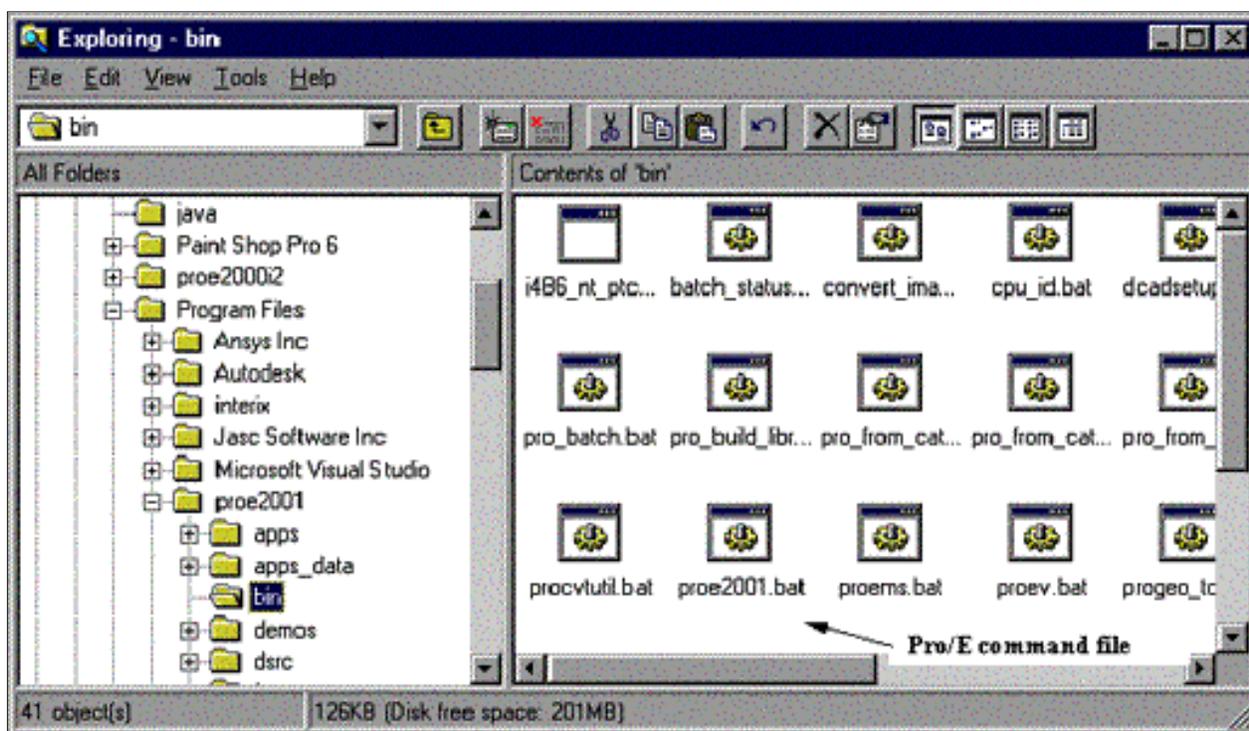


A Publication for ANSYS Users

D:\Program Files\proe2001\i486\_nt\text\usascii\protk.dat. The installation of the protk.dat file into an appropriate directory is essential to the functioning of the Connection for Pro/E.



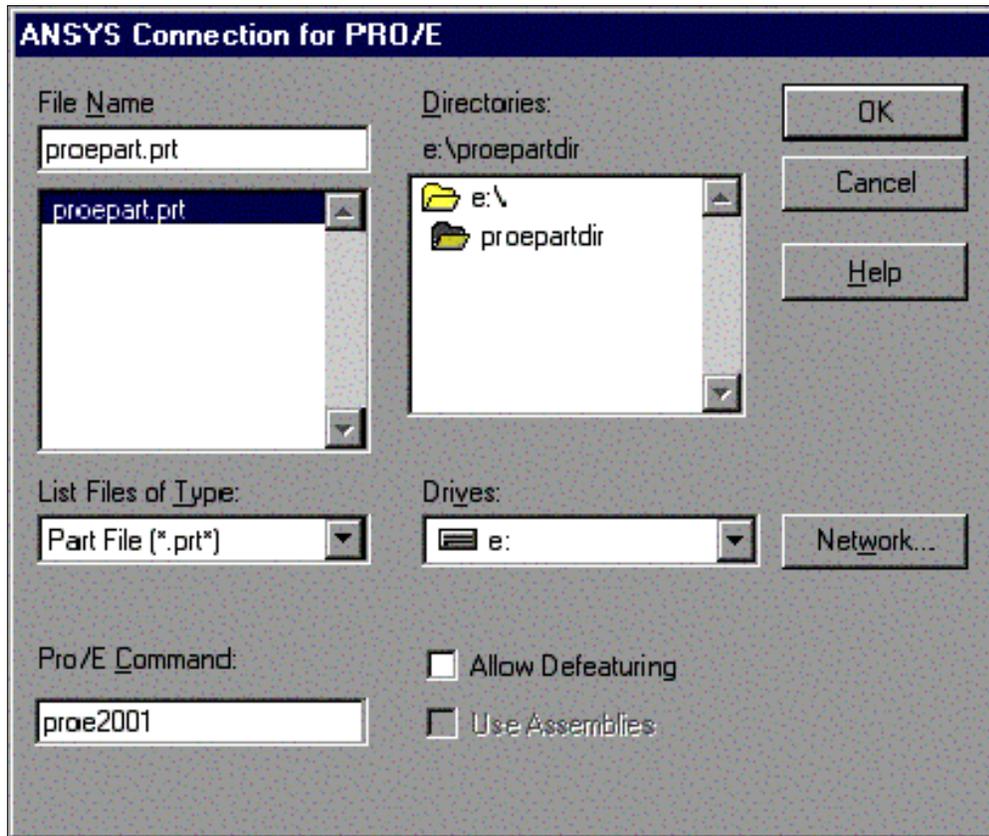
8. Importing a file. To import a Pro/E file into ANSYS, click on Utility Menu > File > Import > Pro/E. In the dialog window that comes up, select the appropriate Pro/E part or assembly file and enter the Pro/E execution command next to "Pro/E command".  
*Note:* In versions 5.7 and earlier, the Pro/E command must have eight characters or less or it will not work.



# The Focus



A Publication for ANSYS Users



There is a good chance the default Pro/E command will not be to your liking. This can be changed by setting the ANSYS\_PROCMD environment variable.

## Troubleshooting

I'll start off basic and work up to more complex issues. The first two things you want to make sure of are as follows.

1. The Connection for Pro/E is a separate product requiring a separate license. If the Connection for Pro/E is not installed, the Pro/E selection under File > Import will be grayed out.
2. Make sure the Connection for Pro/E is configured. See the steps above for configuring the connection product.

Beyond this point, things get more difficult.

3. In the Pro/E import dialog window, be sure you are using the correct command to start Pro/E. For ANSYS versions 5.7.1 and earlier, the command must be less than eight characters. You can change the default Pro/E command by setting the ANSYS\_PROCMD environment variable.
4. Make sure the protk.dat file exists in the i486\_nt\text\usascii, the i486\_nt\text, or the text subdirectory of the Pro/E installation directory. It should be automatically placed

# The Focus



A Publication for ANSYS Users

there when Connection for Pro/E is configured, but if you are still having problems importing than it s something you want to check. If it exists and any or all of these directories, then try copying the protk.dat file to your working directory and see if that works. Granted, copying protk.dat to your working directory would be a temporary work-around; we would want to come up with a permanent solution.

5. Check the contents of the protk.dat file to make sure they are correct for the version of Pro/E being used.
6. Despite the configuration, you may still need to create a pointer to the Pro/E installation directory. This is set through the Path environment variable. A good check to see if the Pro/E command is valid and if it is pointing to the Pro/E installation directory is to try to execute Pro/E from the command line, using the Pro/E command. If this does not start Pro/E, then ANSYS is not going to be able to start it either.

If all else fails, or if you have any questions along the way, give us a call at 480-813-4884 or email us at [techsupport@padtinc.com](mailto:techsupport@padtinc.com).

# The Focus



A Publication for ANSYS Users

## Do Loop Techniques

By [Rod Scholl](#) & [Eric Miller](#) of PADT

Ever wanted to perform an loop operation on a selected set of nodes in your model and the numbering was not sequential?

We will explore 3 choices:

- Standard Do Loop
- Next Do Loop
- Implied Do Loop



Compare computation time to assign nodal temperature BF's based on node number of a set of 4000 non-sequentially numbered nodes. I can't dream up a reason why one would do this but we'll run with it...

**16 Seconds. Standard Do Loop** gets the selection status on each node, then \*IF's on that status to see if the operation is performed:

```
*GET, mxnd, node, , num, max
*DO, icount, 1, mxnd
  *GET, issel, node, icount, nsel
  *IF, issel, eq, 1, then
    BF, icount, temp, icount
  *ENDIF
*ENDDO
```

**5 Seconds. Next Do Loop** uses the NDNEXT get function to avoid the selection status check:

```
*GET, mxnd, node, , num, max
*GET, nmnd, node, , count
nmin1 = -1
*do, icount, 1, nmnd
  nmin1 = NDNEXT(nmin1)
  BF, nmin1, temp, nmin1
*enddo
```

**<1 Second. Implied Do Loop** uses undocumented Implied Loop, VGET, and VMASK:

# *The Focus*



A Publication for ANSYS Users

```
*GET, mxnd, node, , num, max
*GET, nmnd, node, , count
*VGET, nslct(1), node, 1, nsel
*VMASK, nslct(1)
BF, (1:MXND), temp, (1:MXND)
```

Once you use implied do looping, you won't want to ever go back!

# 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.