## 1. Introduction:

At ANSYS 6.0, the modal cyclic symmetry procedure has been significantly improved. New commands, CYCLIC and CYCOPT, replace the older CYCGEN and CYCSOL macros and provide more flexibility and ease-of-use.

Cyclic symmetry analysis has been changed to allow for a more generalized method which, in the future, should allow for other types of analyses for cyclically symmetric structures.

2. Background Discussion:

The analysis of cyclically symmetric structures requires that phase information be stored to determine the displacement (and stress, strain) fields for the full 360° model, based on a given sector. In other words, in modal analyses, the stored information is complex (real and imaginary). To accomplish this, ANSYS actually uses a duplicate sector and writes constraint equations for each nodal diameter solved for — thus, the complex solution can be calculated in this manner. The symmetry planes for each sector need to have matching node patterns, and these symmetry planes are referred to as "high/low" edges, referring to low-angle or high-angle components, as defined in the global cylindrical coordinate system.
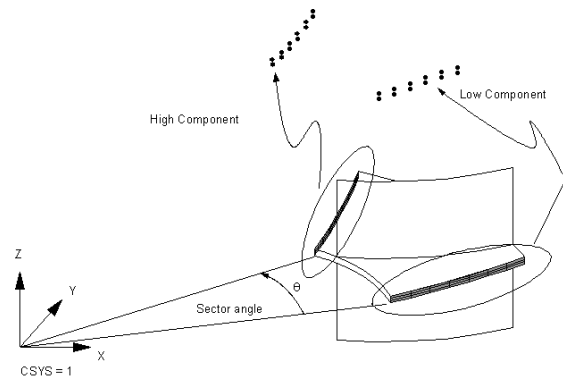
Prior to 6.0, CYCGEN and CYCSOL macros were used to generate the additional sector, then to solve a range of nodal diameters. Postprocessing was performed using either EXPAND or /EXPAND commands. These commands had various limitations and were cumbersome to use. Also, the user had to ensure that the symmetry planes had matching node patterns (for the constraint equations) — this was usually done through the use of the MSHCOPY command.

3. New Cyclic Symmetry Procedure:

Although the old CYCGEN and CYCSOL methods are still available in 6.0, the new CYCLIC and CYCOPT commands provide more flexibility in analyzing cyclically symmetric models.

The procedure for a 'stress-free' modal cyclic symmetry is similar to any other analysis, with a few exceptions:[1]

1. Define basic sector of the model
2. Specify cyclic symmetry (CYCLIC), then mesh [or vice-versa]
3. Define a modal analysis (ANTYPE,MODAL) and eigenvalue extraction options (MODOPT)
4. Define cyclic options, if necessary (CYCOPT)
5. Solve model (SOLVE)
6. Read results set in postprocessing (SET)
7. Calculate min/max displacements/stresses/strains and phase angle (CYCPHASE)
8. Display results on full 360° model (/CYCEXPAND)

The procedures specific to cyclic symmetry models are highlighted in dark red.

After generating the sector of a model, the use should issue the CYCLIC command without any arguments prior to meshing.[2] ANSYS will automatically determine the sector angle and place "high/low" areas (3D) or lines (2D) in components. If CYCLIC is issued prior to meshing, this ensures that the subsequent area-only-mesh (AMESH) or volume-only-mesh (VMESH[3]) will have matching nodes at the "high/low" edges. Currently, CYCLIC only can be used for volume-only or area-only meshes. If the user has a combination of shell and solid elements or line elements, the user must *manually* create the "high/low" edges and define the options for CYCLIC.

---

[1] The procedure for prestressed modal cyclic symmetry analysis is covered in the Release Notes for ANSYS 6.0
[2] The equivalent menu item is: Main Menu > Preprocessor > Cyclic Sector > Auto Defined
[3] Currently, VSWEEP is not supported for CYCLIC-generated meshes.

The user can always manually create the mesh first, then issue the CYCLIC command without any arguments. As noted above, if the model has solid-elements only or shell/planar-elements only, the sector angle and high/low components will be automatically defined, as shown in the status listing below:

```
LIST CYCLIC SYMMETRY STATUS


*******************************************************
****** CYCLIC SYMMETRY STORED QUANTITIES *************
*******************************************************
*   NUMBER OF SECTORS =              12               *
*   SECTOR ANGLE =               30.000               *
*   CYCLIC COORDINATE SYSTEM =        1               *
*   COMPONENT NAME ROOT = CYCLIC                      *
*   LOW  EDGE COMPONENT = CYCLIC_M01L       MATCHED   *
*   HIGH EDGE COMPONENT = CYCLIC_M01H                 *
*   LOW  EDGE COMPONENT = CYCLIC_M02L       MATCHED   *
*   HIGH EDGE COMPONENT = CYCLIC_M02H                 *
*   LOW  EDGE COMPONENT = CYCLIC_M03L       MATCHED   *
*   HIGH EDGE COMPONENT = CYCLIC_M03H                 *
*   LOW  EDGE COMPONENT = CYCLIC_M04L       MATCHED   *
*   HIGH EDGE COMPONENT = CYCLIC_M04H                 *
*   LOW  EDGE COMPONENT = CYCLIC_M05L       MATCHED   *
*   HIGH EDGE COMPONENT = CYCLIC_M05H                 *
*   LOW  EDGE COMPONENT = CYCLIC_M06L       MATCHED   *
*   HIGH EDGE COMPONENT = CYCLIC_M06H                 *
*   DUPLICATE SECTOR NOT DEFINED                      *
*******************************************************
```

Note that ANSYS is often 'smart' enough to determine the coordinate system (e.g., CSYS=1),[4] sector angle (e.g., 30°), and automatically creates components of the high/low edges. The duplicate sector is not defined until solution.[5] If a previous solution has already been performed, the last line will note that a duplicate sector has already been defined.

If, for any reason, the user needs to manually define the sector and low/high components, this can be done with the CYCLIC command, too, by supplying the appropriate arguments.

After specifying modal analysis options (one would select the Block Lanczos eigenvalue extraction method, which is the default), the user can optionally specify which nodal diameters to solve with the CYCOPT command.[6] By default, ANSYS will solve for all nodal diameters if CYCOPT is not issued. CYCOPT behaves similar to OUTRES, where the user can issue it multiple times to specify any number of nodal diameters. CYCOPT,STATUS will list the nodal diameters to be solved for. CYCOPT can also be used to define what degrees of freedom to include in constraint equations in the high/low edges (by default, all DOF are included).

The user then issues a regular SOLVE command instead of using the CYCSOL macro. The solution status will look like the following:

```
            S O L U T I O N   O P T I O N S


    PROBLEM DIMENSIONALITY. . . . . . . . . . . .3-D
    CYCLIC SYMMETRY SECTOR ANGLE. . . . . . . . .  30.000    DEGREES
    DEGREES OF FREEDOM. . . . . UX   UY   UZ
    ANALYSIS TYPE . . . . . . . . . . . . . . . .MODAL
        EXTRACTION METHOD. . . . . . . . . . . . .BLOCK LANCZOS
    EQUATION SOLVER OPTION. . . . . . . . . . . .SPARSE
    NUMBER OF MODES TO EXTRACT. . . . . . . . . .  10
    NUMBER OF MODES TO EXPAND . . . . . . . . . .  10
    ELEMENT RESULTS CALCULATION . . . . . . . . .OFF
```

---

[4] The older CYCGEN/CYCSOL method required that the model be set up with Z-axis as the axis of revolution. The newer CYCLIC/CYCOPT procedure can be used for any coordinate system, including CSYS,5 (Y-axis as axis of rotation).
[5] This is different behavior from the older CYCGEN method, where the duplicate sector is automatically defined in /PREP7 once the command is issued.
[6] The CYCOPT command can be accessed in the menus via: "Main Menu > Solution > Cyclic Options"

Please note a couple of changes in this /STATUS listing. The cyclic symmetry sector angle is now shown in the listing. Moreover, there is no longer a *Cekey* argument to MODOPT for regular Block Lanczos method because of the enhancements to the sparse solver at 6.0 which include symbolic assembly.[7] Hence, one will no longer see the following line in /STATUS:

```
        CEs PROCESSED WITH LAGRANGE MULTIPLIERS . . . .ACCURATE SOLN.
```

because of the fact that symbolic assembly does not have the same issues frontal assembly did with the processing of constraint equations. Symbolic assembly processes constraint equations after the stiffness matrix is assembled, so the wavefront will not drastically increase for large number of CEs. *Symbolic sparse assembly results in faster solution times, especially for modal cyclic symmetry analyses.*

After the solution is complete, the user can use CYCPHASE,[8] which was introduced at 5.7.1, to determine the min/max displacement/stress/strain values and the phase shift at which they occur:

```
LISTING OF THE DATA ON FILE __cycphase.lis

 Summary of Modal Cyclic Symmetry Phase Angle Sweep
 Load Step = 3   Sub Step = 5

                    MINIMUM                              MAXIMUM
          Node      Value      Phase Ang (DEG)   Node     Value      Phase Ang
 (Deg)
     UX:    43    -12.327259    62.0000          43    12.327259    242.0000
     UY:    51    -19.436973   242.0000          51    19.436973     62.0000
     UZ:     2    -12.016844   152.0000           2    12.016844    332.0000
   USUM:  3878     0.013534133 311.0000        2104    19.442685    217.0000
```

Stress and strain values, the max/min values, and the nodes at which they occur will also be listed.[9]

/CYCEXPAND[10] can then be used to display the full 360° model at any given phase shift. /CYCEXPAND can also be used to expand a portion of the mesh only or to revolve the model partially (e.g., by number of sectors or by angle). /CYCEXPAND, similar to /EXPAND, is a graphical way to expand the elements, unlike EXPAND which modifies the database.

---
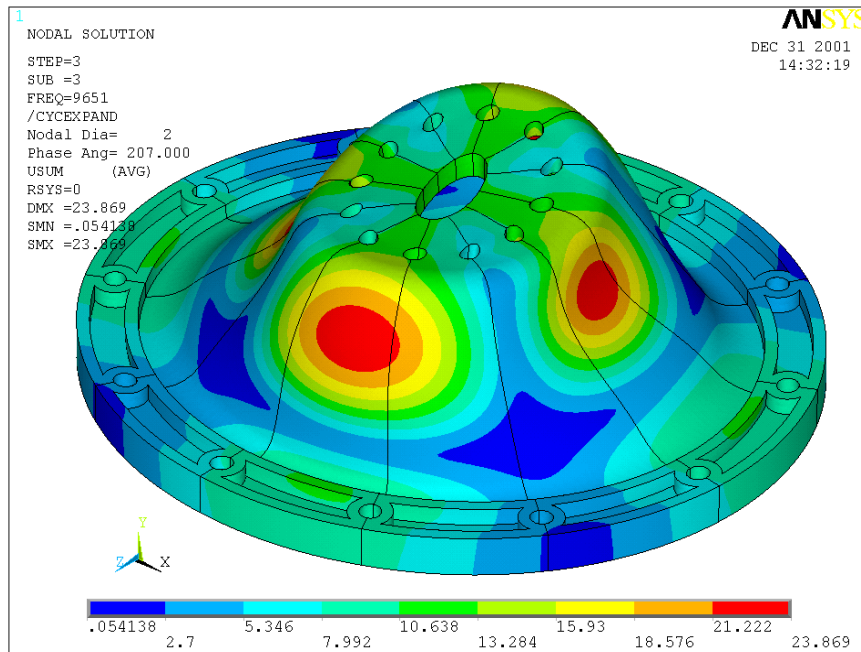
[7] For details, please refer to the ANSYS.NET Tips and Tricks "Sparse Solver Improvements at 6.0" document.

[8] To access CYCPHASE in menus: "Main Menu > General Postproc > Cyclic Analysis > Cyclic Phase"

[9] For components such as SX or UY, the min/max may not be true unless they lie in the first sector. Scalar values such as USUM, SEQV and principal values will contain correct min/max values. See CYCPHASE online help for more information.

4. Example of Performance Increase of Sparse Solver/Block Lanczos Eigenvalue Extraction:

A sample model, provided by Chris Distaso at Wilden Pump & Engineering Co., was imported into ANSYS and meshed. 10 modes for 7 nodal diameters (70 modes in all) were extracted for the model, consisting of 28,000 nodes and 17,400 SOLID92 elements. The runtimes for 5.7 and 6.0 are compared below:

Method	Version	CPU Time	Total Time
CYCGEN	5.7	10388	13696
CYCGEN	6.0	6425	9543
CYCLIC	6.0	6450	9820
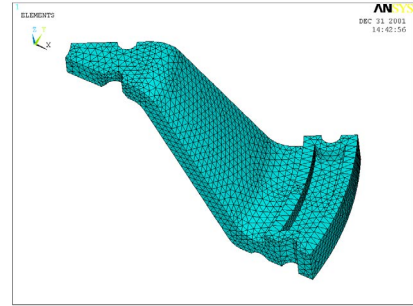
This analysis was performed on a 866 MHz PIII PC with 1 GB of RAM. The solver ran out-of-core, so that explains why the total elapsed time is greater than the CPU time. Also, because the solution method is basically the same for CYCGEN/CYCSOL (old) and CYCLIC/CYCOPT (new) methods, the CPU and total elapsed times at 6.0 were similar. The advantage of the CYCLIC (new) method is related to ease-of-use and flexibility which the new procedure provides, as discussed above.

For this model, one can see that the solution time decreases by about 30% at 6.0, compared with 5.7. This is due to the improvements to the sparse direct solver, most notably the use of symbolic sparse assembly (the Block Lanczos eigenvalue extraction method uses the sparse solver).

5. Conclusion:

The enhancements to the sparse solver decrease solution times noticeably for modal cyclic symmetry analyses when using the Block Lanczos eigenvalue extraction method.

Also, 6.0 introduces a new procedure for analyzing cyclically symmetric models. The new CYCLIC procedure has many more benefits compared with the older CYCGEN/CYCSOL macros:

- The user can issue CYCLIC prior to xMESH commands. The CYCLIC command will automatically find the high/low components and ensure that AMESH or VMESH will create duplicate nodes at those boundaries. This is much easier than manually matching nodes (such as with the MSHCOPY command) and manually creating nodal components.
- CYCGEN defines a duplicate sector immediately. CYCLIC does not create the duplicate sector until solution. With CYCLIC, the duplicate sector can easily be removed, or cyclic options can be turned off altogether.
- Coordinate systems other than CSYS=1 (global cylindrical) can be used with CYCLIC. For example, CSYS=5 (y-axis is axis of rotation) can be used. Please note, however, that nodes in the symmetry planes will be rotated to be consistent with a cylindrical coordinate system (nodal z-axis is up), so please keep this in mind when applying boundary conditions.
- A regular "SOLVE" command can be issued instead of needing to run CYCSOL with its arguments. CYCOPT can also be used in the new method to define given nodal diameters to solve, although, by default, ANSYS will solve for all nodal diameters.
- CYCPHASE allows the user to sweep to obtain min/max values of displacements, stresses, and strains for the model.
- /CYCEXPAND is a more convenient method than EXPAND to postprocess the 'expanded' model. It is graphical only (like /EXPAND), and does not modify the database.

The main reasoning behind this new procedure, besides ease-of-use, is to provide a consistent framework for other types of analyses of cyclically symmetric structure in the future, so the user will not be limited to modal analyses.

_____

Sheldon Imaoka
ANSYS, Inc.

This document is not being provided in my capacity as an ANSYS employee. I am solely responsible for the content.

---

[10] To access /CYCEXPAND in menus: "Main Menu > General Postproc > Cyclic Analysis > Cyc Expansion"

Sheldon's ANSYS.NET Tips and Tricks

Sheldon's ANSYS.NET Tips and Tricks will be emailed to subscribers about once a week (or whenever I have time to write one of these up). You can subscribe by visiting the following URL:

http://ansys.net/ansys/ansys_tips.html

Archives will be posted on that page with password access (which will be mailed with each new issue). General ANSYS Tips and Tricks can also be found at the above URL.

These tips are written with the latest version of ANSYS in mind (version 6.0, as of the time of this writing). Please remember that, with each new release of ANSYS, new features and methods may be introduced, so please refer to the online help as well as your local ANSYS support distributor to verify that these tips are the most efficient way of doing things.

ANSYS Training

ANSYS, Inc. as well as the ANSYS Support Distributors (ASDs) provide training classes in many different areas of ANSYS, ranging from Introductory classes to CFD to Structural Nonlinearities to Electromagnetics. Information on training classes and schedules at ANSYS, Inc. can be found at:

http://www.ansys.com/services/training/index.htm

New classes current offered include *High-Frequency Electromagnetics* and *Advanced Structural Nonlinearities*. Please contact ANSYS, Inc. or your local ASD for their training schedules and offerings.

XANSYS Mailing List

The XANSYS mailing list has more than 2300 subscribers (as of 9/25/01) with about 40 postings per day. This is a forum for exchanging ideas, providing/receiving assistance from other users, and general discussions related to ANSYS. (Note that it is recommended to contact your local ASD for issues related to technical support) You can sign up by visiting the following URL:

http://groups.yahoo.com/group/xansys

Otherwise, you can also subscribe/unsubscribe by sending an email to the following address:

```
Post message:    xansys@yahoogroups.com
Subscribe:       xansys-subscribe@yahoogroups.com
Unsubscribe:     xansys-unsubscribe@yahoogroups.com
List owner:      xansys-owner@yahoogroups.com
```

Because the amount of emails is very large, you can also subscribe in "digest mode" or access the postings via a web browser instead:

```
Digest mode:     xansys-digest@yahoogroups.com
Web-based:       xansys-nomail@yahoogroups.com
                 http://groups.yahoo.com/group/xansys/messages
```