

By Eric Miller

For decades, ANSYS software has been the dominant tool for people who make machines that spin.

Turbo-machinery, motors, shafting, all of these industries have relied upon ANSYS for simulating various aspects of their devices. But through all of this, one aspect of simulating rotating systems was missing needed capability - Rotor Dynamics. The last two releases of ANSYS (10.0 and 11.0) have largely filled in this important area.

Before we get into the detail of the ANSYS implementation, you may notice that this article does not really go into detail on the history, theory or details of the math. This is intentional because others do a much better job at explaining it. We recommend that you first read a nice paper by Swanson, Powell and Weissman called "A Practical Review of Rotating Machinery Critical Speeds and Modes"

www.sandv.com/downloads/0505swan.pdf

Next, read section 15.4 in the ANSYS The-

Spinning up Rotor Dynamics

ory manual and Chapter 8 of the ANSYS Advanced Guide. If you want some history, I recommend "A Brief History of Early Rotor Dynamics" by Nelson findarticles.com/p/articles/mi_qa4075/is_2_00306/ai_n9296359

Reading these should give you a good foundation. In the following sections we will talk very briefly about some of the key things that PADT has learned about these tools that anyone looking to use them should know. But remember that this is a huge topic and before you do a RD analysis you need to do more research and some test cases.

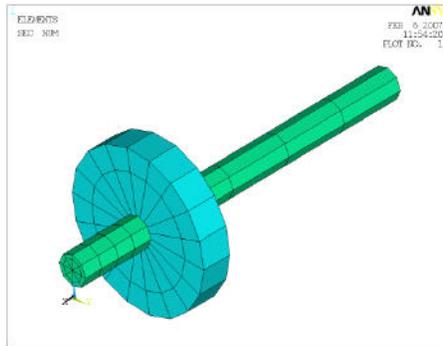


Figure 1: Simple Model

A very simple test case is shown in the Macro ROTDYN1.MAC. Use it to explore

$$[M]\{\ddot{u}\} + ([G] + [C])\{\dot{u}\} + ([K] - [K_c])\{u\} = \{F\}$$

some of the topics discussed below. Figure 1 shows the model.

Supported Elements

After doing the homework above you will see how you can add RD simulation by calculating the Coriolis Terms [G] and adding them to your damping matrix [C] as is shown by the equation of motion that ANSYS uses:

Adding those terms for a simple pipe and mass element is not so hard and this was done years ago. The last two releases have added far more general elements that allow for much more accurate modeling, which has been a lot of work for the programmers in Pittsburgh. A summary is shown in Table 1.

The CORIOLIS Command and Changing the Solver

The basic step that you need to do to include the Coriolis terms in your solution is execute the CORIOLIS, Option,-,-RefFrame command. When Option = 1, ANSYS cal-

(Cont. on Pg. 2.)



By: Doug Oatis

As many people have found, working with arrays and tables is nothing like riding a bike. One day you are writing a complicated macro that scales tables according to a quadratic function dependent on the table indices, then one week later you're relearning how to use the *dim (well, maybe not that bad).

The new year has brought an influx of table questions. At first glance, it seems like a daunting task to go through and scale an

Table Operations - A Lost Art

entire table (i.e. your film coefficient values for a transient thermal analysis). The *m commands are limited in how you can modify a table. One tidbit that people seem surprised to hear is that you can use all of the *vxxx commands to modify tables (*voper, *vfun, etc.). The only issue you have to deal with is that the *vxxx commands only operate on a single column.

You can get around this limitation using a *get and *do loop. Using the *get, you can pull the number of columns (note, that the value returned does not include the 0-th column in the total column count). You can then use the returned scalar as the limit in your *do loop to operate over an entire table.

Using this method, you can further improve it by having your script only operate on certain columns, rather than the entire table. If you look closer at the *v commands, you can also add controls on which rows you want to operate on. The *v commands require you to specify the starting point for the operation, and then they

(Cont. on Pg. 5.)

Contents	
Rotor Dynamics.....	1
Table Operations	1
Tension/Comp Mat Props.....	5
Awesome APDL.....	6
Advertising	7

(Rotordynamics, Cont.)

ELEMENT	Stationary Reference Frame	Rotating Reference Frame
BEAM4		
PIPE16		
MASS21		
SOLID45		
SOLID95		
SHELL181		
PLANE182		
PLANE183		
SOLID185		
SOLID186		
SOLID187		
BEAM188		
BEAM189		
SOLSH190		
SHELL281		

Table 1: Summary of Rotor Dynamics Elements at 11.0

calculates the Coriolis terms for the appropriate elements. You then use the RefFrame argument to specify if you are doing a stationary reference frame (see below).

The only down side of applying the Coriolis terms in the damping matrix is that damping matrices are non-symmetric, and therefore require a non-symmetric eigenvalue solver for eigenvalue problems. Fortunately, the QRDAMP solver has had a lot of work done on it and it is very fast and robust at 11.0. So don't forget the MODOPT, QRDAMP,,ON to activate the solver and tell it to do a complex solution.

Typical RD Simulations

You can include the Coriolis terms in AN-

TYPE of STATIC, MODAL, HARMONIC and TRANS, and HARMONIC and TRANS include full and modal superposition solutions. The most common type of RD simulation is a natural frequency simulation (modal) analysis. When the geometry is not spinning a standard modal analysis works just fine, but when there is spinning the Coriolis term adds those non-symmetric terms that introduce forces to the system, which cause the natural frequencies to split and shift up and down (see Bearings and Damping along with Whirling below and the "Practical Review" article.) So in order to design a rotating machine, you need to know where your natural frequencies are at a given speed, usually so you can avoid a nasty excitation at some common speed. As is shown in example one, you get a feel for how speed effects frequency by running at 0 RPM and then several speeds up to the maximum rotational velocity that the system will see.

The next most common simulation is a harmonic analysis. Here you sweep through a range of excitations where the excitation frequency is also applied as the rotating frequency. Again, the Coriolis terms shift the frequencies, and damping plays a bigger role, making such a simulation very valuable. If your excitation is different from your rotating frequency, you can use the SYNCHRO command to scale it up or down.

Finally, many people want to know the loads that are exerted on structures, joints and bearings when a rotating structure rotates (think jet engine spinning on a wing then the plane turns). This can be done as a static (apply IC commands to specify velocities) or transient dynamic simulation where the Coriolis effects are included.

Campbell Diagrams

The primary post processing tool for people doing RD work is the Campbell Diagram. This workhorse graph for turbomachinery designers is also very powerful when applied to RD. The big difference from a traditional Campbell Diagram is that you can see the modes split because of

whirling. Sometimes, the deviation can be significant and you may end up right at an excitation frequency.

Figure 2 shows the Campbell Diagram produced by ANSYS with the PLCAMP command. There is also a PRCAMP command. For simple systems these commands work great and are very quick to generate. If you have a more complex system, you may want to take the time to do a set, all to a file, massage in Excel, and make your own diagram.

Stationary vs. Rotating Frames

Another thing you should have learned from the above references is the difference between solving in a rotating vs. a stationary reference frame. If you are having a tough time with the difference, you may find the Wikipedia article "Rotating reference frame" useful from a math perspective.

From a practical standpoint, if the parts are attached to a fixed structure, then you need to solve in the stationary frame and if it is not, use the rotating frame. A key thing to note when using the rotating reference frame is that spin softening is automatically included. This can then cause solution problems because the stiffness matrix becomes negative when the the rotational velocity is higher than the natural frequency (see equations 3-77 to 3-79 in the Theory Manual.) If you run into this problem apply a negative shift to a modal run or use large deflection for transient dynamic or static runs.

Bearings, Damping and Whirling

In the real world, rotating structures are attached to static structures through some sort of bearing. Bearings are not infinitely stiff, and the friction and lubricant in them introduce damping. So now you can visualize your system rotating on a set of springs, and often springs that have stiffness that varies with speed and direction. The same goes for damping.

You can capture this in ANSYS with a spring-damper element like COMBI14, which has been around for a while, or with the newer COMBI214 which was purpose built for modeling bearings in rotor dynamics. The 214 allows you to specify stiffness and damping ratio at 0, 45, 90 and 135 if you have non-symmetric bearings (just 0 if it is symmetric) and you can give ANSYS a table for those values. What this means is that you can make the con-

(Cont. on Pg. 3.)

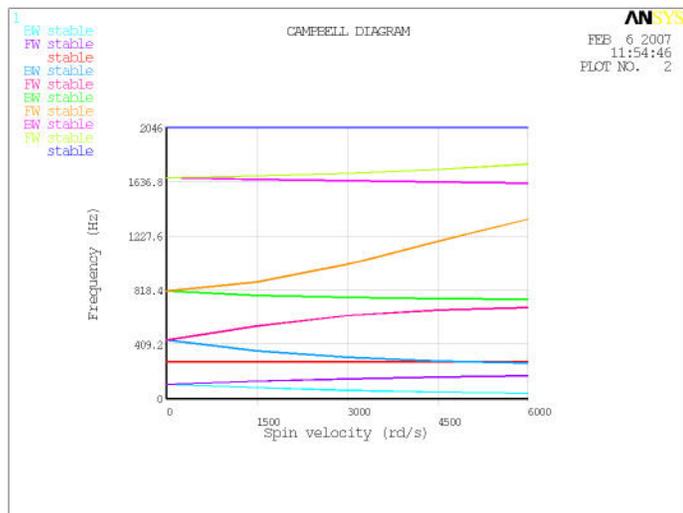


Figure 2: Campbell Diagram for Simple Model

(Rotordynamics, Cont.)

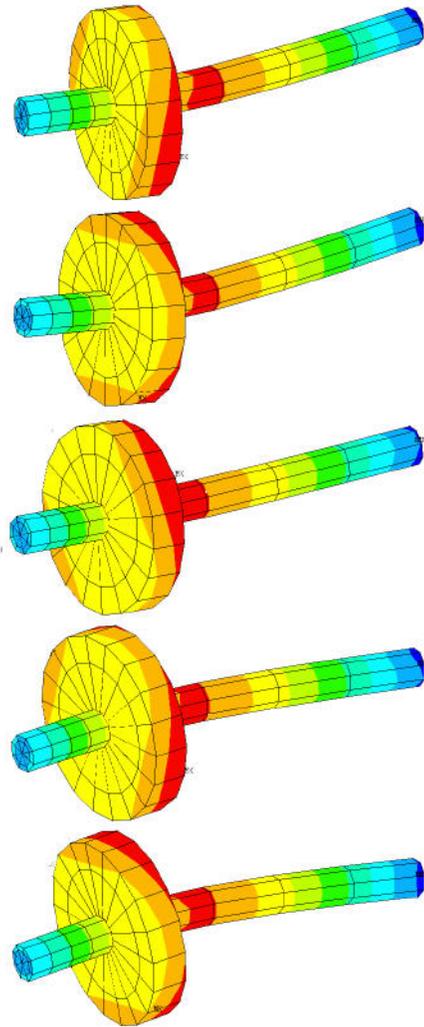


Figure 3: Whirl Model

tribution dependent on other calculated values like time, speed or temperature. See the doc on the element for more information.

One of the improvements to the software for RD was the inclusion of the PLORB and PRORB commands in POST1 for plotting the whirling (or orbiting) of beam/mass RD models. Figure 3 shows an example for ROTDYN1.mac. In addition, you can use the ANHARM macro to create an animation of the whirl for beam/mass and solid RD models. This has been a very useful tool because some modes whirl “forward” and other “backward” The ANHARM macro is a great way to visualize this.

Full 3D Models

For many decades people have been doing rotor dynamics with in-house and commercial codes that used beams and masses. And

for most rotor assemblies, this is still not only the most efficient, but the most accurate method. But sometimes a system does not lend itself to this type of approximation and ANSYS has provided a unique solution in the industry to address it.

A good example is the geometry shown in Figure 4. This is an approximation of a turbo-molecular pump that PADT worked on a while ago. The design was never viable because we could never get through a critical speed do to size/bearing constraints. But one thing we found was that spin stiffening and spin softening played an important role along with bearing damping and stiffness. There is no way to model that type of geometry for rotor dynamics without using a full 3D model. ROTDYN2.MAC is a simplified sample problem with stress stiffening turned on. Figure 5 shows typical results for this geometry.

Most of the examples where you would use this geometry include rotating structures where stress stiffening is important or where the axisymmetric static modes in the part may interact with whirl modes.

Random RD Thoughts

This article is already too long. But there are a few more key things users should know when diving into this great capability:

- Models in the stationary reference frame must be axisymmetric. No bladed disks!
- When you do Campbell Diagram plot/list or a orbit plot/list POST11 goes into a special mode and some post processing commands don't work as expected. Always do a new SET command when you are done with

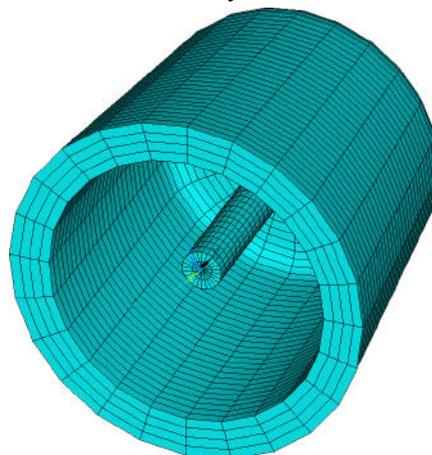


Figure 4: 3D Representative TMP Model

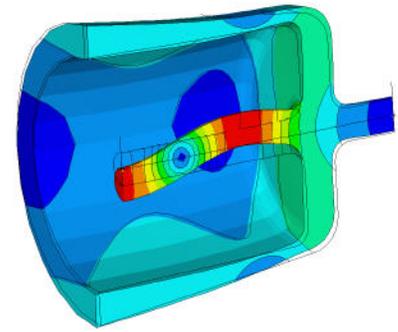


Figure 5: Mixed Cup and Shaft Mode

Campbell and Orbit post processing. This is especially true for the ANHARM command. It won't work unless you do the SET.

- If you have more than one rotating group, not a problem. Use the CMO-MEGA command to specify the rotating axis and magnitude. PLORB, ANHARM recognize the different groups automatically and you can specify a group to plot in the Campbell diagram with PLCAMP
- If you are going to turn on stress stiffening, you need to tell ANSYS to not save the stress run in the results file with the CAMPBELL,ON command. Otherwise on each run in your loop that steps through speed, your result file will get overwritten.
- It can sometimes be tricky to constrain out the rotational degree of freedom on a stationary reference frame 3D model. A good way is to create a massless mass element, constrain its axial rotational DOF, then use RBE3's to tie that DOF to the ends of you shafts (See ROTDYN2.MAC for an example). On beams, just constrain the rotational DOF.
- If you are building a beam/mass model and you want to plot results on the 3D geometry (/ESHAPE,1) then you must save the element results by setting the 4th argument in MXPAND to yes: MXPAND, ,,,YES

Animations of the whirl solutions along with the example macros can be downloaded from :

ftp.padtinc.com/public/downloads/roto_dyn_focus.zip

ROTDYN1.MAC

```

finish $/clear
/file,rotdyn1
/prep7
! Define geometry params
shftlen = 12
shftdia = 1
dskoff = shftlen*.75
dskdia = 5
dskthk = 1
bstf = 60000 ! Brng Stiffness
nmd = 20 ! Number of modes
nmspn = 5 ! Number of speeds
mxspn = 6000 ! Max Omega
! Build model nodes
n,1
n,5,dskoff-(dskthk/2)
n,6,dskoff+(dskthk/2)
n,10,shftlen
fill,1,5
fill,6,10
! Bearing ground nodes
n,101,
n,201,
n,110,nx(4)
n,210,nx(4)
! Use 188 elements, solid
et,1,188,,2
sectype,11,beam,csolid
secdata,shftdia/2
sectyp,12,beam,csolid
secdata,dskdia/2,18
! Make elements
et,11,14,,2
et,12,14,,3
r,10,bstf

secnum,11
type,1 $mat,1
real,1
e,1,2 $e,2,3
e,3,4 $e,4,5
e,6,7 $e,7,8
e,8,9 $e,9,10
secnum,12 $e,5,6
type,11 $real,10
e,1,101 $e,4,110

type,12 $real,10
e,1,201 $e,4,210

! Fix axial and rotational DOF at ends
d,1,ux $d,1,rotx
d,4,ux $d,4,rotx
! Fix Brng ground nodes
d,101,all $d,201,all
d,110,all $d,210,all
! Plot the model
/view,1,1,1,1 $/vup,1,z

/eshape,1,1 $/pnum,sect,1
/num,1 $epplot
! Dummy mat props
ex,1,10e6 $nuxy,1,.23 $dens,1,.001

finish $/solu
! Setup modal run
antype,modal
coriolis,on,,,on
modopt,qrdamp,nmd,,,on
! Loop on speeds
*do,i,1,nmspn
  spn = (i-1)*(mxspn/(nmspn-1))
  omega,spn
  mxpand,nmd,,,yes
  solve
*enddo
finish
/post1
! Plot/List Campbell Diagrams
plcamp,,1,rpm
prcamp,,1,rpm
! Plot orbit
set,5,6
plorb
! Animate whirl
set,5,6
plnsol,u,sum
anharm

```

ROTDYN2.MAC

```

finish $/clear
/file,rotdyn2
/prep7

! Set Geometry Parameters
shftrad = .25
shftlen = 3
webthk = .5
conir1 = 1.75
conir2 = 1.65
conor1 = 2.25
conor2 = 1.75
conoff = -.5
stblen = 1
! Set Speed and Bearing Stiffness
mxspn = 10000*2*3.14159/60
bstf = 20000
nmd = 20 !number of modes
nmspn = 5 ! Number of speeds

! Build model
k,1,0, $k,2,shftlen
k,3,shftlen+webthk

k,4,0,shftrad
k,5,shftlen,shftrad
k,6,shftlen+webthk,shftrad
k,7,0+conoff,conir1
k,8,shftlen,conir2
k,9,shflen+webthk,conir2
k,10,0+conoff,conor1
k,12,shftlen+webthk,conor2
k,13,kx(3)+stblen
k,14,kx(3)+stblen,shftrad

a,1,13,14,6,12,10,7,8,5,4
adel,all
lfill,8,9,.25 $lfill,7,8,.25
lfill,3,4,.25 $lfill,4,5,.5
al,all

et,200,200,6 !Mesh 2D Area
esize,.125
amesh,all

et,1,185 ! Setup solid element and
revolve
type,1
esize,,6
vrotat,all,,,,,1,2,360

asel,s,loc,x,0 !Fix ends in axial DOF
nsla,s,1
d,all,ux
cm,nfx1,node
asel,s,loc,x,kx(13)
nsla,s,1
d,all,ux
cm,nfx2,node
nall

et,10,214,,1,0 ! Make the Brng Element
r,10,bstf,bstf
! find node on shaft
nn1 = node(kx(1),ky(1),kz(1))
nn2 = node(kx(13),ky(13),kz(13))
! Ground node
n,30000,nx(nn1),ny(nn1)+.5,nz(nn1)
n,30001,nx(nn2),ny(nn2)+.5,nz(nn2)
n,40000,nx(nn1),ny(nn1),nz(nn1)
n,40001,nx(nn2),ny(nn2),nz(nn2)

type,10 $real,10 $mat,10
e,nn1,30000 $e,nn2,30001
d,30000,all $d,30001,all
! Fake mass elements for fixing ROTX
et,201,21 $r,200
type,201 $real,200
e,40000 $e,40001
d,40000,rotx $d,40001,rotx
! Tie masses to ends of shaft
cmsel,s,nfx1

nset,a,,,40000
rbe3,40000,rotx,all
cmsel,s,nfx2
nset,a,,,40001
rbe3,40001,rotx,all
nall
! Dummy mat props
ex,1,10e6 $nuxy,1,.23
dens,1,.001

finish
/solu
coriolis,on,,,on ! Turn on RD
campbell,on !tell ANSYS to save rst
*do,i,1,nmspn !loop on speeds
  finish
  /solu
  antype,static !Prestress run
  pstres,on
  spn = (i-1)*(mxspn/(nmspn-1))
  omega,spn
  solve
  finish
  /solu
  antype,modal !Modal run
  pstres,on
  modopt,qrdamp,nmd,,,on
  mxpand,nmd
  solve
*enddo

save
finish
/post1
/view,1,-1,1,1
/vup,1,z
/dist,1,2.937
/focus,1,1.25,.43,0
set,nmspn,3
plnsol,u,sum,1 ! Plot mode shape
anharm ! Animate whirling

```

(Table, cont.)

want you to modify the first 3 rows or your table, set your starting point to be the fourth row (i.e. *voper,tablename(4,i),... where i is the column number).

This built-in looping of the *v commands can cause problems if you want to apply different operations to different rows of your table. For example, say you wanted to modify an existing table that is graphically shown as DougTable.

The fix is to copy DougTable to another temporary table (created using *dim). Then, simply set the “stored to location” parameter of the *voper command to be DougTable, as shown below:

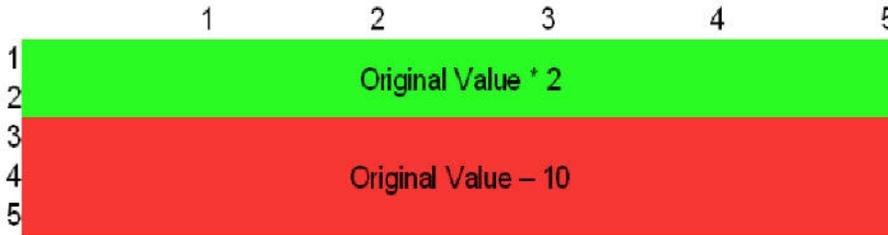
```
*dim,d_temp,table,5,5
*mfun,d_temp(0,0),copy,DougTable(0,0)
*get,d_column,parm,'dougtable',dim,
*do,i,1,d_column
```

```
*voper,DougTable(1,i),d_temp(1,i),mult,2
```

```
*voper,DougTable(3,i),d_temp(3,i),sub,10
*enddo
```

A little practice and creativity can help you to further expand on this macro. Ultimately, you can move your get to the front of the macro to pull the x (row) and y (columns) dimensions of your table and use those to define your temporary table. To really show off to your fellow analysts (because if your spouse isn't an engineer, they definitely won't appreciate it – just ask my wife), you can use Jeff Strain's “*Varying your *Vwrite” (July 2004) to make your script more flexible in handling table names and different scaling operations.

DougTable



Unequal Tension/Compression Mat. Prop's

By: Rod Scholl

Back in release 9.0, some significant improvements were made to the Cast-Iron material model. The most remarkable feature of this material model is the ability to implement different tensile vs. compressive plasticity curves. However, this capability can be expanded (with a little effort) to model a different compressive vs. tensile modulus by tweaking the plasticity curve.

The expected usage of the cast-iron input looks like the following:

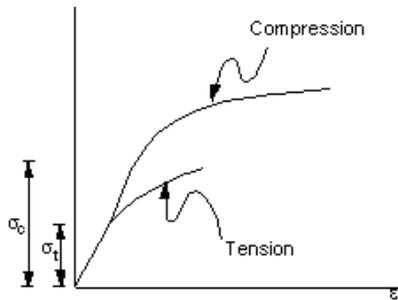
```
TB,UNIAXIAL,1,1,3,COMPRESSION
TBTEMP,10
TBPT,,0.203E-02,0.300E+05
TBPT,,0.500E-02,0.500E+05
TBPT,,...
```

To extend this power further, one can essentially model materials with unequal tensile and compressive modulus. Of course the problem is non-linear, so there is a penalty here, but it is a fairly cheap resolution to the case where the moduli are unequal.

By making the linear elastic portion of the material model negligibly small, one can accomplish two separate moduli curves.

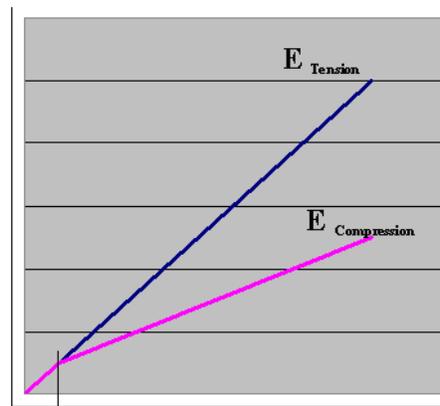
higher, the corresponding modulus is applied. This, of course requires a non-linear solution, which may or may not be an added expense depending if other non-linearities are already present at their relative convergence rate. Test cases on a simple cube showed very quick convergence with this model, with 3 iterations required when forced to a single substep via

(Cont. on Pg. 6.)

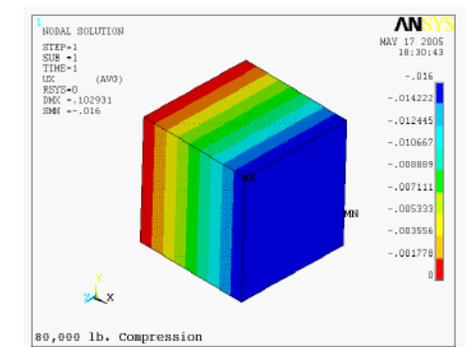
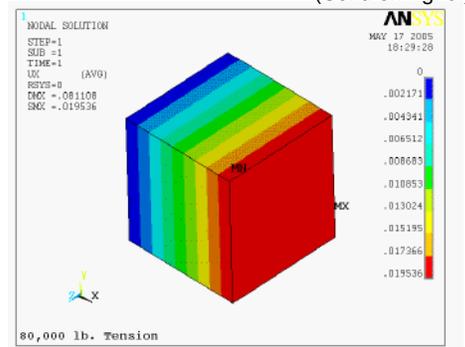


A Typical input looks like:

```
TB,CAST,1,,ISOTROPIC
TBDATA,1,0.04
TB,UNIAXIAL,1,1,5,TENSION
TBTEMP,10
TBPT,,0.550E-03,0.813E+04
TBPT,,0.100E-02,0.131E+05
TBPT,,...
```



Thus up to some small value of strain, such as 1e-6 in/in the compression and tension will use the higher modulus. But for strains



Awesome APDL: Customizing Animation Macros

By Rod Scholl

Several times now I've posted emails looking for some special graphics script, such as a custom animation, and each time someone has to remind me that I can edit the ANSYS scripts themselves. Not every command is a script, but the animation actions are. There is a collection of them in the installation directory. ANMODE.MAC is a good example for mode shapes.

One thing I've never liked is that it only shows the 1st half of the modal cycle in the .avi file. I want it to show the expansion AND contraction cycle. That way I can put the created .avi file loop-play in a media player and stare ad infinitum at the gracefully moving image. Ahhh, peace at last... Of

course this is how the ANSYS gui displays it, but I wanted to be able to easily send these mesmerizing videos to the customer.

It was a simple change to make the 2nd half of the capture loop backward. The first part of the code to the right shows the original APDL chunk that sets the distortion scale factor (DSC) and does the replot. In the second half, an *if statement is added that scales forward on the first half and backwards on the second.

It usually only takes a few minutes of poking around, making changes, and seeing what happens to figure out some minor modifications. So take advantage of these "open source" goodies from ANSYS Inc!

```
!--- Original Script ----
!COM *****
!COM CAPTURE FOR JavaScript
!COM *****

~tcl,'ansys::report::animImage'
*ELSE ! } {
  /REPLOT
*ENDIF ! }
  _DSC=_DSC-_DDD
*ENDDO

!--- Modified Script ----
!COM *****
!COM CAPTURE FOR JavaScript
!COM *****

~tcl,'ansys::report::animImage'
*ELSE ! } {
  /REPLOT
*ENDIF ! }
  *if, _i, lt, AR11/2, then
    _DSC=_DSC-_DDD*2
  *else
    _DSC=_DSC+_DDD*2
  *endif
*ENDDO
```

(tension/compression, cont.)

NSUBST,1,1,1. This method shouldn't be a path-dependent (irreversible) solution if you are simulating linear modulus with this method.

And finally, one last trick (which took quite some time to sort out) is that the curve can't quite be linear... it is allowed in KINH for example, but not in the case of cast-iron. Thus you must make the curve minimally decreasing at each point.

The whole script can be downloaded here,

but the interesting part is as follows:

```
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,10e6
MPDATA,PRXY,1,,.3
```

```
TB,CAST,1,,,ISOTROPIC
TBDATA,1,0.04
```

```
TB,UNIAXIAL,1,1,3,TENSION
TBTEMP,10
TBPT,,1e-6,10e6*1e-6
!NOTE the -1 for slightly
! decreasing slope
tbpt,,1e-5,10e6*1e-5-1
```

```
!NOTE the -2 for slightly
! decreasing slope
tbpt,,1e-1,10e6*1e-1-2

TB,UNIAXIAL,1,1,3,COMPRESSION
TBTEMP,10
TBPT,,1e-6,10e6*1e-6
!NOTE the -1 for slightly
! decreasing slope
tbpt,,1e-5,5e6*1e-5-1
!NOTE the -2 for slightly
! decreasing slope
TBPT,,1e-1,5e6*1e-1-2

nsubst,1,1,1
solve
```

Upcoming Training Classes						
Month	Start	End	#	Title	Location	
Feb '07	26-Feb	28-Feb	104	ANSYS Workbench Sim - Intro	Tempe, AZ	
Mar '07	8-Mar	9-Mar	105	ANSYS WB NL Structural	Tempe, AZ	
	12-Mar	13-Mar	203	Dynamics	LV., NV	
	19-Mar	20-Mar	501	ANSYS/LS-DYNA	Tempe, AZ	
	28-Mar	30-Mar	902	Multiphysics Sim for MEMS	Tempe, AZ	
	Apr '07	2-Apr	4-Apr	101	Introduction to ANSYS, Part I	Tempe, AZ
	5-Apr	6-Apr	107	ANSYS WB DesignModeler	Tempe, AZ	
	9-Apr	11-Apr	401	Low Freq. Electromagnetics	Tempe, AZ	
	12-Apr	13-Apr	604	Introduction to CFX	Tempe, AZ	
	16-Apr	18-Apr	201	Basic Structural Nonlinearities	Tempe, AZ	
	18-Apr	19-Apr	204	Adv. Contact and Fasteners	Tempe, AZ	
	25-Apr	27-Apr	152	ICEM CFD/AI*Environment	Tempe, AZ	
	May '07	2-May	4-May	104	ANSYS WB Simulation - Intro	LV, NV
	7-May	8-May	100	Engineering with FEA	Tempe, AZ	



ANSYS, Inc. Is doing some great regional user group meetings in North America. These are highly technical and well attended: www.converge.ansys.com

Feel the need for making high quality graphs in batch mode outside of ANSYS. GNUPlot has been around for years and keeps getting better: www.gnuplot.info

Links



- ANSYS 11.0 Products Released
Visit www.ansys.com for details in the near future. PADT will cover the 11.0 release heavily at our on-line seminar and in future issues of The Focus

News

The Focus is a periodic publication of Phoenix Analysis & Design Technologies (PADT). Its goal is to educate and entertain the worldwide ANSYS user community. More information on this publication can be found at: <http://www.padtinc.com/epubs/focus/about>

XANSYS
A Mailing List for the
ANSYS User Community



The Shameless Advertising Page

Converge

converge.ansys.com

2007 ANSYS U.S. Regional Conference Series

Texas, Feb 20-22
 California, Apr 3-5
 Pennsylvania, week of May 14
 Michigan, Sept 10-11

- ANSYS executive greeting
- Keynote presentations from key customers
- Technology roadmap overview
- Product integration update
- Management roundtable
- ANSYS Workbench Application Showcase
- Partner exhibits
- Product demos
- Product updates
- User presentations
- Special industry content



Learn APDL the "Home Schooling" Way
 Purchase PADT's **Guide to ANSYS Customization with APDL**

Do You Read the Focus and Find it Useful?

Does Your Company Outsource their RP Jobs?

Have you Considered PADT as your RP Source?

PADT is a Leader in High Quality SLS, SLA, FDM and Injection Molding

Let us Quote your Next RP Job

rp@padtinc.com

2007 The 2007 PADT CAE Wall Calendars are here!

One year, at a glance. And this year, no froo-froo pastel colors!

Visit: www.padtinc.com/docs/lit/padt_2007_1.pdf to download your copy before your wall fills up!

ANSYS Product TRAINING

Instruction by Engineers for Engineers

CFD Simulation Services
PADT Knows Flow

Outsource your next CFD Job to the PADT's CFD Experts
www.padtinc.com 1-800-293-PADT