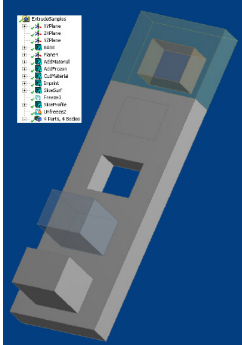


Understanding DM: Extrusion Options

By Eric Miller



This article is the next in a series of articles looking into some of the unique capabilities in the ANSYS, Inc. tool DesignModeler, which the lazy amongst us refer to as DM. This tool

has been written from the ground up with a powerful set of features that address the unique needs of analysts and what they need to do to their geometry. In this article we will discuss some of the unique and useful options in the Extrude operation.

Of all the 3D geometry creation operations in any modern CAD tool extruding is the most common, especially if the system combines adding, slicing and removing material into one command as DM does. It therefore pays to really understand all the options in the command and especially those that are put there by our friends in development for the express use of FE analysts. Figure 1 gives a good overview of all the options available in the detail view for the extrude command. As with all detail

views, the various displayed options change based upon which ones you choose.

As the table shows, most of these options are pretty standard. But some are more important than others and are the subject of this article.

Special Operations: Frozen, Imprint and Slice
Previous articles in this series have focused on the ideas of multiple bodies that can share surfaces where they meet and the concept of frozen geometry. The unique operations of Add Frozen, Imprint Faces and Slice Material all fit into this special category because they create geometry that most CAD packages will not allow. Figure 2 shows the results of the same extrude command for all three operations.

Add Frozen does just what the name implies; it creates a new body and labels it as frozen. No Boolean operations are done. If you need a refresher on what frozen bodies are and why you need them, take a look back at last month's [article](#).

Imprint Faces is the most Finite Element-ish option in the group. Who else would want to take a perfectly continuous surface and break it up? One way to visualize this command is to imagine what the program does. When you generate an imprint, the program extrudes the edges (Cont. on Pg. 2.)

Detail	Choices	Description
Extrude	Text	Enter the Name you want the extrusion to have
Base Object	Object Name	Select sketch to make extrusion from
Operation	Choice	
	Add Material	Extrude sketch and boolean add to existing geometry or as stand along solid
	Add Frozen	Extrude but don't boolean add
	Cut Material	Extrude then boolean subtract from existing geometry. Requires existing unfrozen geometry to cut
	Imprint Faces	Extrude and do a boolean intersection on the surfaces the split faces with the resulting curves
	Slice Material	Extrude and do a boolean subtract and fill void with boolean intersection. Requires existing frozen geometry to slice
Direction Vector	Object Count	Select edge to define a non-normal extrusion direction. If none is given, direction is normal to Base Object
Direction	Choice	
	Normal	Normal to Base Object or Along Vector
	Reversed	Opposite of Normal
	Both-Symmetric	Both sides of Base Object, same extent on each side
	Both-Asymmetric	Both sides of Base Object but user specifies extent on each side.
Type	Choice	
	Fixed	User gives a distance
	Through All	Go past all existing geometry
	To Next	Go till intersection with a surface
	To Faces	Go to a user selected planer face
	To Surface	Go to a user selected surface
Depth	Length	Supply distance for extrusion
Target Faces	Object Count	Select face or surface to extrude to for "To Faces" or "to Surface"
Type 2	Choice	Same choices as Type but for second side of Asymmetric extrusion
Depth 2	Length	Distance if Asymmetric
Target Faces 2	Object Count	Face or surface for Asymmetric
Target Bodies	Object Count	Select entities that the extrusion effects. Only active for Cut, Imprint and Slice
As Thin Surface?	Yes/No	Turns on creation of thin solid or surface geometry
Inward Thickness	Length	If Thin Surface is yes, then the inside wall thickness.
Outward Thickness	Length	If Thin Surface is yes, then the outside wall thickness. If both are zero, then make a surface body
Merge Topology	Yes/No	Should DM merge any patches that are created?

Figure 1: Table of Detail Options for Extrude

Workbench Bolt Loads with Split Cylinders

By Doug Oatis

Applying a bolt load in ANSYS Workbench is one of many examples of how Workbench has simplified a cumbersome analysis. However, one of the requirements is that if you apply your bolt load to the face of a part, it must be a full 360° cylindrical surface. This creates a problem with most CAD imports, since they divide the area into two (i.e. Pro/E, SolidWorks, etc.). Don't fret! There is a work-around for this phenomenon.

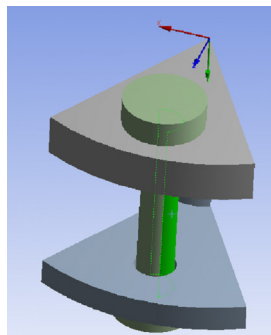


Figure 1: Split Cylindrical Surface on Imported CAD

When you create a cylinder in SolidWorks or Pro/E, it comes across as two "clam-shell" faces, seen in Figure 1. If you try to apply a bolt load to these two surfaces, Workbench will discard your selection and ask you to select again. Instead of applying the bolt load to the surface (and forcing Workbench to do the leg-work as where to perform the pretension section calculations), you can apply the bolt load to a body.

To do this, set your selection

filter to be bodies, and hit apply next to the Geometry selection in the details window of the Bolt Load. Next, you'll see that the bolt preload icon in the graphics window is shown at the Global Origin, with the preload acting in the z-direction.

(Cont. on Pg. 3.)

Contents

DM: Extrusion Options	1
WB Split Cylinder Bolt Loading	1
Grok'ing Harmonic Index	3
Animation Macro.....	5

("DM: Extrude" cont.)

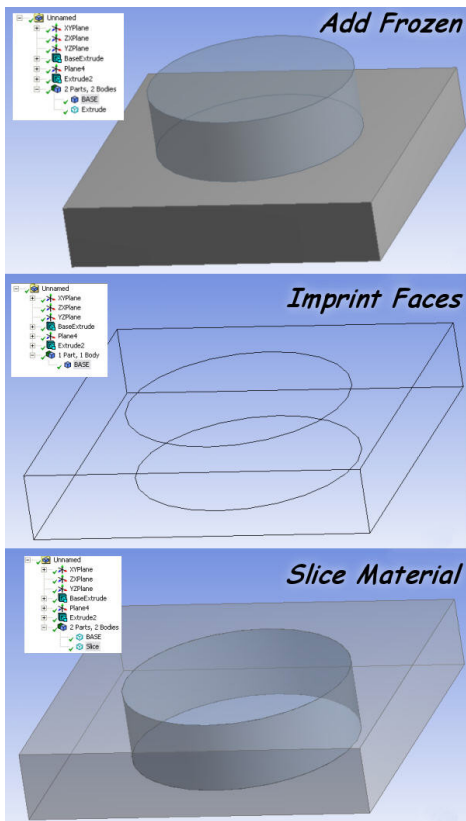


Figure 2: Results of Add, Imprint and Slice Operations

of your profile through the bodies you are operating on and calculates the curves that result as the intersection of those surfaces and the existing body surfaces. The program then goes in and splits the original surfaces with those curves, leaving patches of surfaces behind that are perfect for applying loads to.

The final special operation, Slice, is another one of those tools that make you say "how did I survive without this?" What it does is extrude the profile as a cut, then fill the void created with a new body. If you think about how DM works with frozen geometry, you realize that for this to work you need to make sure that the geometry that you are operating on is frozen first for the option to even show up in the details.

Extrusion Types (Extent)

When you create an extrusion you must specify how far the extrusion goes. This is done with the "Type" detail, which most

CAD tools refer to as Extent or Distance. The default option, and most commonly used is Fixed. With this option you simply specify a distance value. But often you want the distance of the extrusion to be based on some other geometric characteristic of your model, and this is where the other options come in. Figure 3 gives examples of each.

Through all a great command because it tells the program to slice/cut/imprint through everything along the extrusions path, regardless of how big or small that base geometry will become when you change upstream parameters. It basically tells DM to extrude the profile past any geometry that it might intersect with, then carry out the specified operation. Imagine if you are putting a hole through a one inch plate and you use Cut Material/Fixed/1.25 in options. What happens if you change the plate thickness to two inches. Now the hole is blind. By using Through All you capture what you really want: a through hole.

A very handy option is To Next. This extrudes the profile until it intersects some surface beyond the one you start from. This has the same advantages as Through All in that you are capturing your intent of making a feature that goes till it hits another surface.

To Faces and To Surface work the same way, except that instead of going until they hit the first surface, they go until they hit a Face or a Surface that the user specifies. Looking at these two commands, many people ask: "What is the difference between To Faces and To Surface?" The best way to understand the difference is to look at Figure 3 and see. In To Faces the extrusion must actually intersect with the faces within the boundary of the Faces. In To Surface, it intersects with an extrusion of the Surface.

Specifying Thin Walls

The last important Details to really understand are those that deal with the Thin Surface option. When this is set to Yes, the user must then specify an Inside and Out-

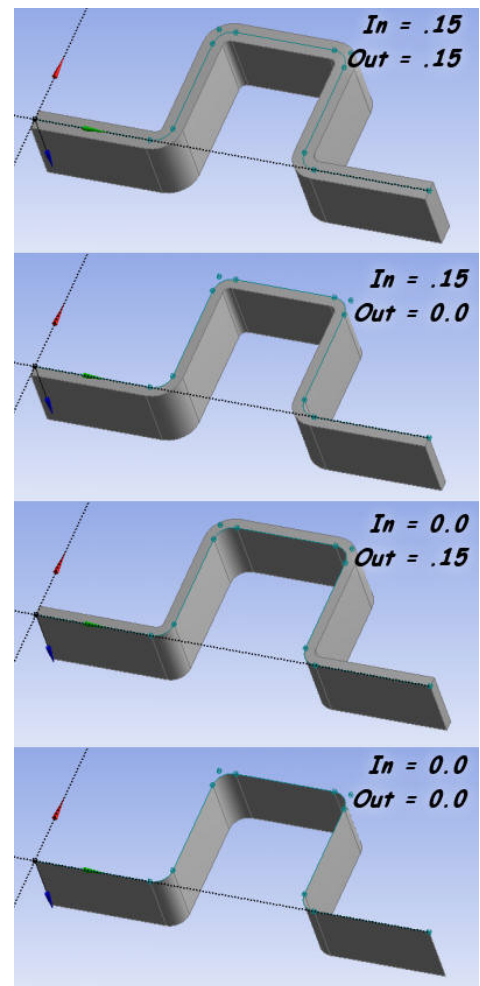


Figure 4: Thin Wall Distance Specifications

side wall thickness. The program then extrudes your profile to create a surface, and then expands inward or outward by the distance specified. If you give a zero distance for both, it simply makes a surface body. Figure 4 shows the results of different distance specifications.

Doing More with Extrude

Play around with these options on some test geometry and get to know them well. Most of us are used to using extrude to create a base feature, a boss or a hole. With the power of DM you can do much more and do it in a way that prepares your model for the mesh you want, not the mesh you are forced to live with

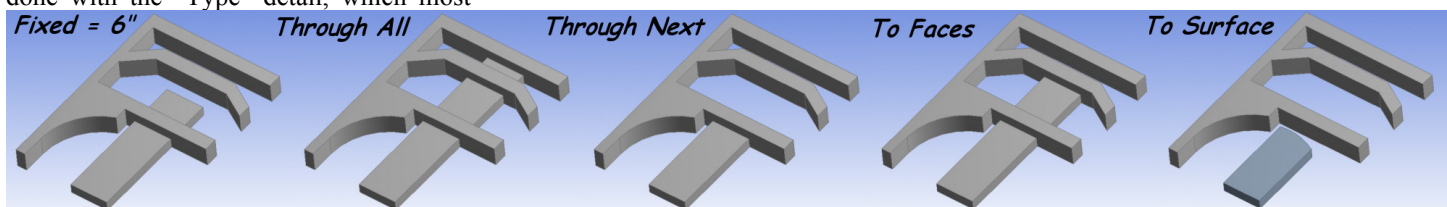


Figure 3: Effect of Extent Options

("Bolt Load" cont.)

This is where knowledge of the preload method in ANSYS Classic comes in handy. If we want to apply the bolt preload to an imported body, we need to setup a coordinate system that does the following:

1. Z-axis is oriented along with the bolt axis
2. XY-Plane intersects the body. This is where the preload is going to be applied (where the psmesh command operates)

If you need to, you can easily create your own coordinate system that meets both of these criteria. To do this, make sure you highlight the 'Model' level in the analysis

tree. Reposition and reorient until the coordinate system is in the correct spot

Applying a bolt load to a body also allows you to control where the loading is applied. By default, if you select a single cylindrical surface, it will apply the preload in the middle. If, for some reason, you need to apply your preload to a location other than the middle (i.e. your thread length is long for a relatively short bolt), you can position your coordinate system in the right location and proceed as usual.

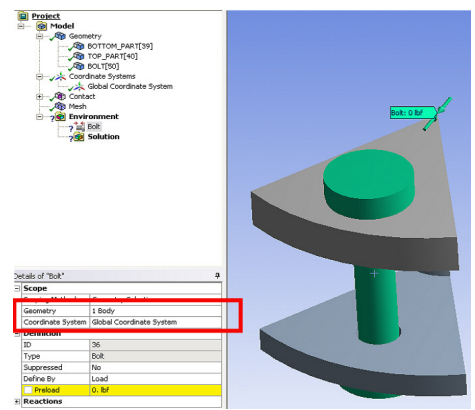


Figure 2: Completed Specification

Grok'ing the Harmonic Index

By Rod Scholl

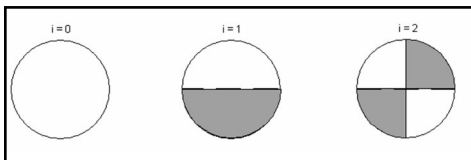


When it comes to modal cyclic analysis, I figure something like the term "Harmonic Index" is the kind of thing that builds job security for the ANSYS analyst. A simple, but hard to grasp at first concept, will provide hours of fun as you banter it about in technical reviews. Watch as folks nod uncomfortably fearing to admit they have no idea what you are talking about.

Prior to 7.0, one would only need to specify the number of Nodal Diameters to extract in a modal run. Although the ideas are closely linked, there's a deeper understanding required to graduate from understanding Nodal Diameter to understanding Harmonic Index.

Nodal Diameter

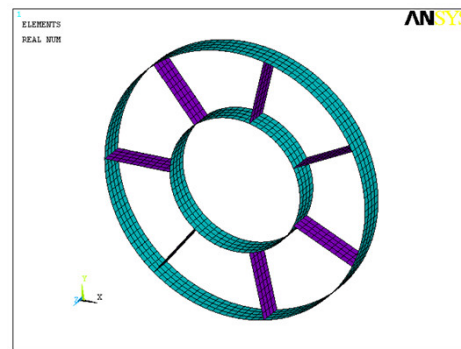
Pictures do the trick here:



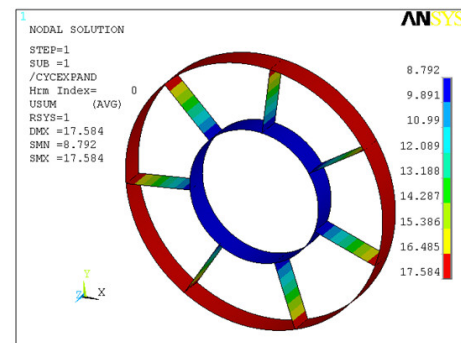
A nodal diameter is a region (line in this case) of zero displacement for a mode shape. In 2-D, they are the unmoving nodes of a vibrating string. In ANSYS 6.1, one would specify the nodal diameter range to extract using the CYCLIC command. A disk with N sectors would only need N/2 or (N-1)/2 nodal diameters extracted to capture all possible modes in the frequency range.

Harmonic Index

Let's back into this definition. Take the case of a 7-sector disk - created and solved using this script adapted from the ANSYS 10.0 help manual.

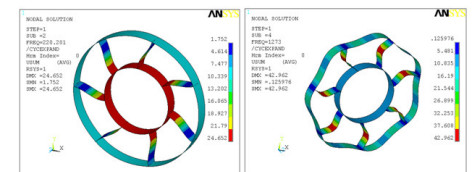


For the case of 0 nodal diameters we of course expect the breathing mode. This model could have been solved by modeling a single sector with cyclo-symmetric boundary conditions. Note it is referred to as Harmonic Index 0.



What about the case of 7 nodal diameters? It too could have been solved with a single sector and cyclo-symmetric boundary con-

ditions. This is the first torsional mode. So although it would have been given in ANSYS 6.1 by specifying 0 nodal diameters... its kinda a misnomer, since its actually the 7th. And also shown is the 14th nodal diameter - with two zero-displacement lines per sector. Note that all three solutions can be obtained by merely coupling the opposite edges in XYZ.



So getting back to our example, all three of these modes are of Harmonic Index = 0. More generally, for Harmonic Index = 0:

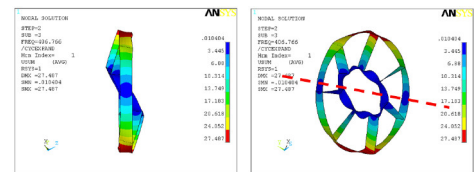
$$d = m \cdot N$$

where $m = 0, 1, 2, 3, \dots, \infty$

d is the Nodal Diameter
 N is the number of sectors.

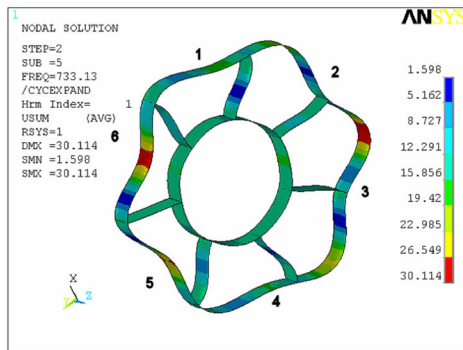
Okay, easy enough - now for the next twist. What about the 1st Nodal Diameter and 1st Harmonic Index?

The 1st nodal diameter looks like this:



But for the 6th nodal diameter, we are still at the Harmonic Index = 1! You can count 6 locations on the outer ring that are unmoving - 6 nodal diameters:

("Grok'ing" cont.)



And the 8th nodal diameter is also of Harmonic Index =1, as is the 13th and 15th. More generally we find that:

$$d = m \cdot N \pm k$$

where $m = 0, 1, 2, 3, \dots, \infty$

$$6 = 1 \cdot 7 - 1$$

$$8 = 1 \cdot 7 + 1$$

$$13 = 2 \cdot 7 - 1$$

$$15 = 2 \cdot 7 + 1$$

d is the Nodal Diameter
 N is the number of sectors
 k is harmonic index

So rather than tracking an infinite number of possible nodal diameters, the Harmonic Index allows us a way to classify them finitely. In fact, there are only $N/2$ or $(N-1)/2$ Harmonic Indices. For our 7-sector case, that means only 3 Harmonic Indices, each with infinite number of possible nodal diameters associated.

In practice of course, the frequency range extracted often only captures the 1st Nodal diameter for Harmonic Index 1, with the 8th, 13th, 15th, etc. being too high frequency (and energy) to worry about.

So if you stare at the above equation and study the table below you can easily determine what Harmonic Index a particular Nodal Diameter belongs to:

Harmonic Index (k)	Nodal Diameter (d)					
0	0	N	N	$2N$	$2N$...
1	1	$N-1$	$N+1$	$2N-1$	$2N+1$...
2	2	$N-2$	$N+2$	$2N-2$	$2N+2$...
3	3	$N-3$	$N+3$	$2N-3$	$2N+3$...
4	4	$N-4$	$N+4$	$2N-4$	$2N+4$...
...
$N/2$ (N is even)	$N/2$	$N/2$	$3N/2$	$3N/2$	$5N/2$...
$(N-1)/2$ (N is odd)	$(N-1)/2$	$(N+1)/2$	$(3N-1)/2$	$(3N+1)/2$	$(5N-1)/2$...

But what *IS* the Harmonic Index you ask? And why are there only $N/2$ (even case)? Just can't leave well enough alone? Okay...

When building the duplicate sector and coupling the outer edges, ANSYS uses the following boundary conditions:

Equation 15.183.

$$\begin{Bmatrix} u'_A \\ u'_B \end{Bmatrix} = \begin{bmatrix} \cos k\alpha & \sin k\alpha \\ -\sin k\alpha & \cos k\alpha \end{bmatrix} \begin{Bmatrix} u_A \\ u_B \end{Bmatrix}$$

where:

u_A, u_B = calculated displacements on lower angle side of basic and duplicated sectors (A and B, respectively)

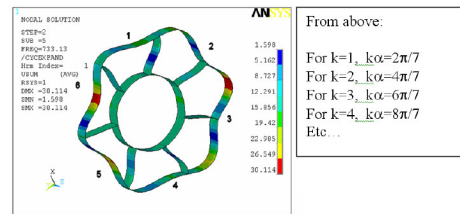
u'_A, u'_B = displacements on higher angle side of basic and duplicated sectors (A and B, respectively) determined from constraint relationships

$$k = \text{harmonic index} = \begin{cases} N/2 & \text{if } N \text{ is even} \\ N-1 & \text{if } N \text{ is odd} \end{cases}$$

$\alpha = 2\pi/N$ = sector angle

N = number of sectors in 360°

For case of the 1st Harmonic Index, 6th Nodal Diameter of our 7 sector disk above:



Note that there are unique magnitudes for $\cos(k\alpha)$ and $\sin(k\alpha)$ only for $k = 1, 2$, and 3. All other values of k are repeats! So we see that a Harmonic Index of k refers to the coupling angles on the edges of our two sectors (duplicate sector created during cyclo-symmetry analysis).

$$\begin{Bmatrix} u'_A \\ u'_B \end{Bmatrix} = \begin{bmatrix} \cos k\alpha & \sin k\alpha \\ -\sin k\alpha & \cos k\alpha \end{bmatrix} \begin{Bmatrix} u_A \\ u_B \end{Bmatrix}$$

Combine this knowledge with the understanding of why 360 degree cyclo-symmetric mode extraction can be accomplished on a 2-sector model using fourier transformation process, and you now grok the Harmonic Index.

15.15.2. Complete Mode Shape Derivation

The mode shape in each sector is obtained from the eigenvector solution. The displacement components (x, y , or z) at any node in sector j for harmonic index k , in the full structure is given by:

$$u = u_A \cos(j-1)k\alpha - u_B \sin(j-1)k\alpha \quad (15-188)$$

where:

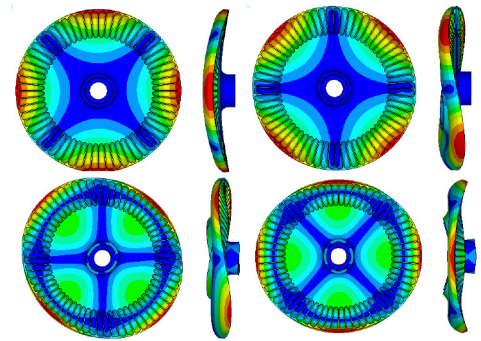
j = sector number, varies from 1 to N

u_A = basic sector displacement

u_B = duplicate sector displacement

From the 10.0 ANSYS Manual:

Thus it is left as an exercise for the reader to prove that the two-term fourier series of 15-188 perfectly matches the full 360 degree result.



A few plots from the 2nd harmonic index of an actual an Impeller in a pump made by PADT

What is a Grok?

While doing the layout of this issue of the focus I pasted in Rod's article and came across his title: "Grok'ing the Harmonic Index." I had never heard that term and wanted to make sure it was not some cold war trigger word that would activate sleeper cells in Wichita.

Going to Wikipedia I found:

- It comes from the Heinlein Novel "Stranger in a Strange Land"
- It is a Martian word and literally means "to drink".
- Grok is explained in the novel as:

"...to understand so thoroughly that the observer becomes a part of the observed—to merge, blend, intermarry, lose identity in group experience. It means almost everything that we mean by religion, philosophy, and science.."

- It's use is primarily amongst science-fiction fans, geeks and some pagans, particularly those belonging to the Church of All Worlds

Which leaves a couple of unanswered questions:

What is the Martian word for "Unconverged Solution" and is it a swear word on Mars?

Can one truly Grok something like Harmonic Index? Or is it Un-Grok'able?

Rod Scholl: SciFi fan, geek or high-priest of the Church of All Worlds?

- EM

Awesome APDL: Custom Animation Macro

No surprise, the customer called late in the afternoon and let us know that we needed to make animations on the results of over 25 runs that we completed last week so they can take them with them to their design review with their customer.

What we wanted was an animation of the static solution that went from zero stress and zero deflection to 100% stress and deflection. And we wanted to set our own contour and use it for the animation. Oh yea, and we wanted to do this on various machines in our Linux Cluster.

Simple:

PlotCtrls->Animate->Over Results...

Nothing is ever simple. First off we could not get the anim files to convert to AVI reliably. Second the macro that ANSYS calls does the stress scaling by changing the contour, and therefore hides the contour in the animation. So we took that ANSYS

macro, stripped it to its bare minimum and came up with the macro on the right.

It does a /show,png to make PNG files instead of an AVI. We then use MEncoder from the mplayer package (www.mplayerhq.hu) to convert the PNG's to an very compact and smooth AVI file.

If you look closely at the macro you will see that we are using the undocumented /SSS command. This command multiplies whatever value you are plotting by the scale factor given as the first argument. So as you can see the macro scales displacement and stresses from 0 to 1 over the *DO loop.

The goal of creating an animated version of the plot on the screen is achieved with a /cont,1,,user and /repl commands.

In a fancier version of the macro, we have imbedded a /sys command to run Mencoder and clean up the PNG files afterwards.

```
_nmfrm = 5          ! Default frames
*if, arg1, ne, 0, then
  _nmfrm = arg1 ! Set frames to arg1
*endif
! Get /DSCALE scale factor
*get, _dmlt, graph, 1, dscale, dmult
/user, all ! Freeze plot settings
_dsc=0.0 ! Start scale at 0
! Calc Scale increment
*if, _nmfrm, eq, 1, then
  _ddd=0.0
*else
  _ddd=1.0/(_nmfrm-1)
*endif
/cont, 1,,user ! Freeze contour
/show,png      ! Turn on PNG save
*do, _I, 1, _nmfrm ! Loop on frames
! Calc displacement scale factor
_dsf=_dmlt*_dsc
*if, _dsf, eq, 0, then
  _dsf=_dmlt*.0000001
*endif
/dscal, all, _dsf ! Scale disp
/sss, _dsc      ! Scale stress
/repl          ! Plot
_dsc=_dsc+_ddd ! Incrmnt Scale
*enddo
! Close out
/show,close
/show,term
/go
```



Resources

The Inquire commands are a great resource for extracting information from the ANSYS database. A handy summary sheet of inquiry functions can be downloaded at: www.padtinc.com/docs/lit/inquiry_sheet.pdf

The best resource for Workbench users out there is the ANSYS Workbench Community. Files, discussion groups and related resources help WB users share with each other and the developers at ANSYS, Inc. It just passed 1400 members: www.ansys.com/customer

Upcoming Training Classes

Month	Start	End	#	Title	Location
Jul '06	10-Jul	12-Jul	104	ANSYS WB Simulation - Intro	Albq, NM
	13-Jul	13-Jul	105	ANSYS WB Sim - Struct NL	Albq, NM
	20-Jul	21-Jul	801	ANSYS Customization APDL	Tempe, AZ
	24-Jul	25-Jul	203	Dynamics	Albq, NM
	26-Jul	28-Jul	101	Introduction to ANSYS, Part 1	Irvine, CA
Aug '06	01-Aug	03-Aug	104	ANSYS WB Simulation - Intro	LV, NV
	04-Aug	04-Aug	105	ANSYS WB Sim Struct NL	LV, NV
	07-Aug	09-Aug	152	AI*Environment	Tempe, AZ
	10-Aug	11-Aug	202	Advanced Structural NL	Tempe, AZ
	14-Aug	15-Aug	502	ANSYS/LS-DYNA	Tempe, AZ
	17-Aug	18-Aug	604	Introduction to CFX	Tempe, AZ
	23-Aug	25-Aug	502	High Frequency EMAG	Tempe, AZ
Sep '06	7-Sep	8-Sep	301	Heat Transfer	Albq, NM
	11-Sep	13-Sep	101	Introduction to ANSYS, Part 1	Tempe, AZ
	18-Sep	21-Sep	802	Advanced APDL & Custom.	Tempe, AZ
	25-Sep	26-Sep	201	Basic Structural Nonlinearities	Tempe, AZ
	27-Sep	28-Sep	204	Adv. Contact and Bolt Pret.	Tempe, AZ



Links

Want to learn more about FLUENT? Download their latest newsletter at: www.fluent.com/about/news/newsletters/06v15i1/06v15i1.pdf

Want to see some COOL analysis! Check out the AUTODYN animations page. Lots of crashes and explosions:

www.century-dynamics.com/cgi-bin/animater.exe/grp?prod=autodyn



News

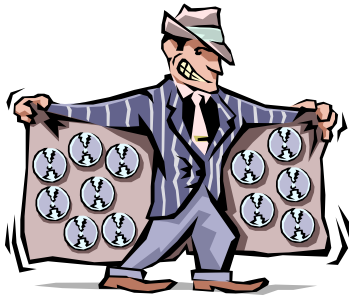
- ANSYS, Inc. Makes the 'B2 100' list for 5th "Year [link](#)

- Granta and ANSYS Collaborate on Material Data Tools [link](#)

- ANSYS, Inc. Joins European @neurist Project [link](#)

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

Outsourcing your Analysis to PADT Puts Food in this Child's Mouth



Call 1-800-293-PADT or e-mail info@padtinc.com Today



PADT is on a Pace to Sell over 20 Dimension 3D Printers in Arizona for 2006

Have you had your Demo yet?

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 Injeciton Molding

Let PADT Quote your Next RP Job

rp@padtinc.com



It's Making '55 Famous—For Fun!

For sheer driving pleasure, Chevrolet's stealing the thunder from the high-priced cars.

Let's be frank. Up to now, maybe there were reasons for wanting one of the higher priced cars. If you desired something really special in the way of driving fun, you simply had to pay a premium to get it. Usually, a big one!

Not any more! The Motorama Chevrolet is taking the play away from the high-priced cars with pure excitement on wheels! Chevrolet's new "Turbo-Fire V8" puts a

heaping bowlful of fun under your foot—162 h.p.! (Special to adventure lovers: 180 h.p. "Super Turbo-Fire V8" is optional at extra cost.) You can also choose from the two highest powered 6's in the low-price field.

As for drives, just name it. Chevrolet offers Overdrive, super-smooth Powerglide (extra-cost option) and a new and finer Synchromesh transmission.

Find out how the Motorama Chevrolet puts new fun in your driving life! Take the key at your Chevrolet dealer's—and you'll want it for keeps! . . . Chevrolet Division of General Motors, Detroit 2, Michigan.

Motorama 

See and drive it at your Chevrolet dealer's



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

ANSYS Product TRAINING

Instruction by Engineers for Engineers

