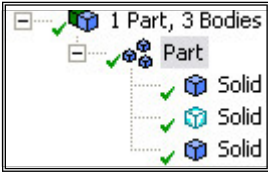


## Understanding DM: Parts and Bodies



By Eric Miller & Rod Scholl

This article is the first 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 first article we will discuss an important concept in DM - Parts and Bodies.

If you have used any modern CAD tool you know that they generally allow you to create single solids. Then you can use some sort

of assembly tool to combine the solids into an assembly. That is great for a designer, but for analysis you often have things such as curves for beams, surfaces for thin structures (shells) or solids that share areas between them (VGLUE/AGLUE/LADD). The folks who developed DM came up with a clever way to support the traditional CAD part-assembly paradigm while allowing for the flexibility that analysts need.

In DM, they call all geometric objects that can be lines, surfaces or solids "bodies", and then allow you to group a collection of bodies as "parts." This concept carries from DM into the Workbench FE pre/post processor, Simulation. In most cases a given part will have only one body and this is what you would see if you attached geometry from

your CAD. So let's learn about this Part vs. Bodies business:

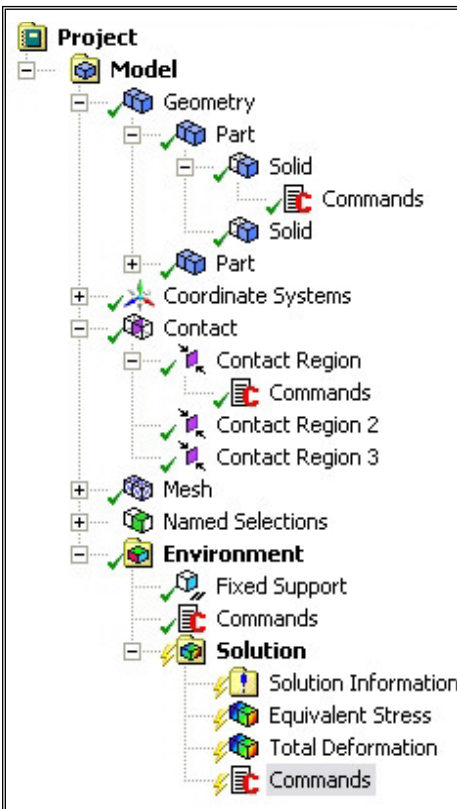
Generally an analyst has 1 of the following desires for interfacing two bodies:

- A) Bodies are distinct, and related by contact, or constraint equations.
- B) Bodies share a surface(s) or edge(s) where one wants mesh continuity (shared nodes).
- C) Wants to perform Boolean operations on the two bodies.
  - 1) Add, so that internal or shared surfaces, and edges are eliminated
  - 2) Subtract, removing one body from another.
  - 3) Intersect, retaining some or all bodies, but creating shared surfaces where boundaries of one part intersect the other.

(Continued on Page 2)

## Become a Snippet Master: APDL in Workbench

By Doug Oatis



You're working in Workbench when you suddenly realize you need to write a constraint equation, define orthotropic coefficients of thermal expansion, modify a contact key option, or pull nodal data from a certain location. What's an analyst to do? Lucky for you, Workbench allows you to easily perform all those tasks using command snippets, a.k.a. APDL for all you old-schoolers.

Inserting a command snippet is extremely easy. Simply right click on the branch in the Simulation tree, insert, and select Commands. You can insert these into just about every branch, as shown in Figure 1.

Depending on where you insert the command snippet, you can perform different actions, as listed in Table 1.

The header of each command, shown in the graphics window, will tell you where the commands are inserted into the ANSYS batch file. These headers can be helpful when you are using APDL that is processor sensitive (i.e. cerig only works in /prep7).

Now you're probably asking, "Well that's just great, but how can I really use this?" This is where it comes in handy if you understand what different objects do in terms of ANSYS Classic. A named selection becomes either a nodal or element component. Coordinate systems are translated as user-defined coordinate systems. The first coordinate system represents  $csys=0$ , the next represents  $csys=11$ , the following  $csys=12$ , and so on. Of course, you can manually specify the coordinate system numbers in the details window, but that can become complex. It's best to consistently and intelligently define your coordinate systems and just remember how the counting works. With these two features, you now have the ability to jump to different parts of the model, select objects, and do

(Continued on Page 3)

### Contents

DM: Parts & Bodies.....	1
APDL in Workbench.....	1
FLUENT Purchase.....	3
Rigid Body Rotation.....	4

Figure 1: Sample Tree with Command Snippets

Operation	DM Method	ANSYS Methods
A) Distinct Bodies	N/A	CPINTF, CERIG, Contact, Etc...
B) Share Boundaries	Form New Part	VGLUE, AGLUE, LADD, NUMMERGE
C1) Boolean Add	Freeze/Unfreeze Body Operations	VADD, AADD, LCOMB
C2) Boolean Subtract	Form New Part & Suppress Body Operations	VSBV, ASBA, LSBL
C3) Boolean Intersect	Form New Part & Suppress Body Operation	VOVLAP, AOV LAP, LOVLAP, VPTN, APTN, LPTN

("DM" cont.)

Table 1: Body Operations

Forming a new part is pretty simple. Select the bodies you want to either share boundaries choose "Form New Part" command in the Tools menu or in the RMB menu. You will see a new part appear in the tree, which is comprised of the original bodies. Visually in DM you won't see much difference, but if you are a doubter, do some fancy scoping ('selection' in ANSYS-speak) and you will see that they now share boundaries. Don't expect to get a BTOL message. Or one about keypoints not lying on lines? Or one about improper "topology"? It really is amazingly robust. I always check it in the tree, because I can't quite believe it worked that simply (and quickly).

Using 'Form new Part' in combination with 'Suppress' in Simulation allows one to achieve the subtract and intersection Booleans from the table above (C2, C3). After the new part is formed, in Simulation choose which ones you want to eliminate, and suppress it using the RMB menu.

This all may be easier to see in some examples. If you look at Figure 1 you will see a simple model that consists of three boxes. The middle box sits on the top surface of the bottom box. The third box is offset and sits in the air.

#### Method A

If you take this into Simulation and mesh it you get three discrete volumes with meshes that do not line up, and contact defined between the bottom and middle boxes, as shown in Figure 2.

#### Method B

If we wanted to save computation time and get a more accurate answer, we would want

to merge the bottom and middle boxes to form one continuous mesh. We do this with the "Form new Part" Command in DM. Figure 3 shows that the volumes on the screen look the same, but the tree has grouped the bodies into a part. Moving over to Simulation and meshing, we now see that the bottom and middle box are merged and the mesh is continuous (Figure 4).

#### Method C1

The simplest operation to do is add all of these parts together as you build them, or to boolean add them with a body operation. This is so simple we cut out that picture to save space.

#### Method C2 and C3

Now, to make things interesting, what if we took that top box and moved it down so it sits inside the middle box? In a CAD system you would model this as an assembly with interference. But in DM you simply move the boxes to overlap, as shown in Figure 5, 'Form new Part', and when you bring it into Simulation a new volume is created that is the intersection of the top and middle boxes, and they merge to share faces so that you get the continuous mesh shown in Figure 6 (this is equivalent to ANSYS's VOVLAP). In Simulation if desired, eliminate which portion of the new parts are unwanted and suppress them, as exemplified in Figure 7.

The tools on solids also apply to surfaces. So if you import or create surfaces it identifies them as bodies and lets you do body operations such as adding, imprinting, etc. An example of a surface intersection is shown in Figure 8. Using "Form New Part" it will mesh with shared nodes in Simulation (Figure 9).

As always, the best way to get your mind about these concepts is to fire up DM and play with it yourself. If you don't have a license, contact your salesperson and ask for a temp key to try it

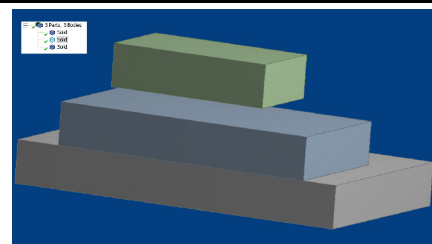


Figure 1: 3 Separate Blocks

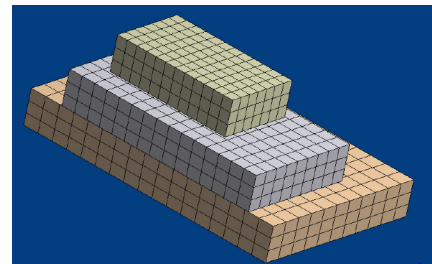


Figure 2: 3 Separate Blocks Meshed

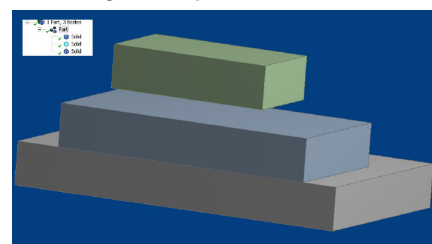


Figure 3: Bottom and Middle Blocks Glued

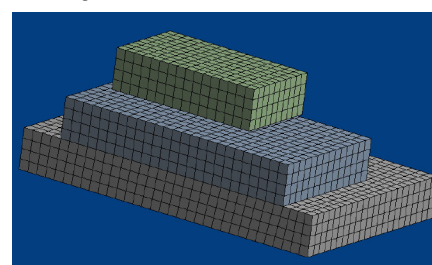


Figure 4: Bottom and Middle Blocks Glued &amp; Meshed

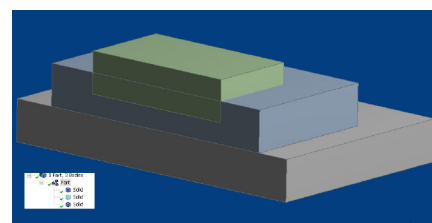


Figure 5: Top Block Imbedded

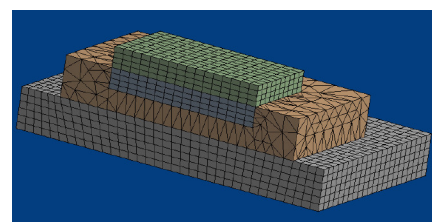


Figure 6: Top Block Imbedded &amp; Meshed

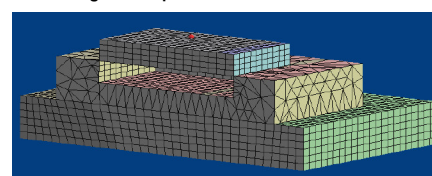


Figure 7: Imbedded &amp; Meshed with Subtraction



Figure 8: Surface Bodies

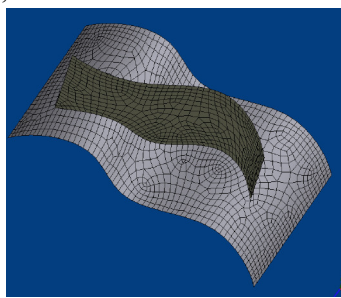


Figure 9: Meshed Surface Bodies



("Snippet Master" cont.)

just about anything you want. Just realize, however, that any elements you create through APDL will not be displayed in the Simulation post processor.

If you have Commands in the 'Environment' and 'Solution', you have additional features available in the "Details" window. In 'Environment', you can specify input arguments, just like defining a macro in ANSYS classic (arg1, arg2...up to arg9). In 'Solution', you can specify input and output parameters. Simply set your output prefix correctly, and you can pull back nodal loads on user-defined elements. You can also read and create files using any of the \*read/writes, just make sure to include your C/Fortran formatting statements.

As a tip, If you need help with your APDL, you can always launch ANSYS Classic through the project tab Workbench (make sure your Simulation is highlighted), do your work, then copy out the log file and paste it into one of the 'Commands'

As you gain experience in using APDL, you will quickly find that you have developed some fairly intricate command snippets. My fanciest macro does the following.

Creates nodes in center of each bolt hole > CERIGs node to flange area > Creates a single pre-strained beam through the bolt hole to apply a preload > Solves for bolt load > Write out reactions to a file and copies it to my c:\ to be used for calibration purposes

Then, if I set a single variable to 1, it will go and read the file created > Calculate the ratio between defined and actual bolt load > Scale up or down my pre-strained value > Then solve.

This creates a pre-load model that I can use, and reduces the number of elements because I don't have to model the bolt. You can go even further and perform a PSD analysis, which isn't currently directly supported in the Workbench GUI. Now get out there and write some snippets!

Details of "Environment" Snippet

<b>File</b>	
File Name	
File Status	File not found
<b>Definition</b>	
Suppressed	No
<b>Input Arguments</b>	
<input type="checkbox"/> ARG1	
<input type="checkbox"/> ARG2	
<input type="checkbox"/> ARG3	
<input type="checkbox"/> ARG4	
<input type="checkbox"/> ARG5	
<input type="checkbox"/> ARG6	
<input type="checkbox"/> ARG7	
<input type="checkbox"/> ARG8	
<input type="checkbox"/> ARG9	

Details of "Solution" Snippet

<b>File</b>	
File Name	
File Status	File not found
<b>Definition</b>	
Suppressed	No
Output Search Prefix	my_
<b>Input Arguments</b>	
<b>Results</b>	
<input checked="" type="checkbox"/> my_test	0.25

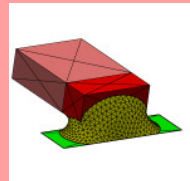
Command Region	Operation Available
Geometry > Part	Modify material properties using 'MATID' as material number
Contact > Contact Region	Modify contact keyopts using 'TID' and 'CID' as type or real numbers
Environment	Create loads and/or geometry, change solution settings
Solution	Post process and pull parameters back into WB



## Resources

Some folks on XANSYS pointed us to this very useful tool for calculating the shape of liquids due to surface tension. Called Surface Evolver, it is perfect for simulating solder balls and such. It is from the Susquehanna University and is free.

<http://www.susqu.edu/brakke/evolver/>



# Commentary: FLUENT Purchase Success!

By Eric Miller

As we were rushing to get the February issue of The Focus out the door, just like we are rushing to get the April issue out, we talked about the fact that ANSYS, Inc. was attempting to purchase FLUENT. Well, a recent press release announced that the Federal Trade Commission had approved the transaction and that they will officially announce the purchase on May 1, 2006.

To say the least, that went very quickly and we all hope that the speed in which the deal was approved is indicative of how the integration of FLUENT's very talented staff and large customer base will go. We know that lots of folks at ANSYS, Inc. have been working very

hard in planning things, but they have not been allowed to discuss details until after everything is approved. The hope is that we will learn a lot more at the User Conference next week in Pittsburgh. We will certainly share what we can when we learn more.

In talking to our users here in the Southwest, it seems that everyone is over the shock of such a big deal and are looking forward to the technical integration of the two companies. Adding that many talented developers and support people will certainly help everyone in the ANSYS family.

Because we have so many electronics customers we are especially interested in learning more about the ICEPAK product as well as their FLOWizard tool.

Right now we are waiting to see how things end up, and will share what we learn with all of you as we go. If the stock price is any indication Wall Street shares our enthusiasm as well.

Maybe in 12 months we will need to add another page to The Focus for a monthly FLUENT article!



# Determining Equivalent Rigid Body Rotation

By Carlos Shultz

This analysis demonstrates an unusual solution to an atypical requirement...just another way you can use ANSYS to get answers to life's difficult questions.

**Problem:** Given a mesh, determine the amount of rigid body rotation which best matches a deformed shape of that mesh.

A test case of SHELL93 elements was created. The left side has a row of fixed displacements. The right side was connected to a MASS21 element through RBE3 constraints. Figure 1 shows the deformed shape with boundary conditions shown.

The next step was to select which nodes to

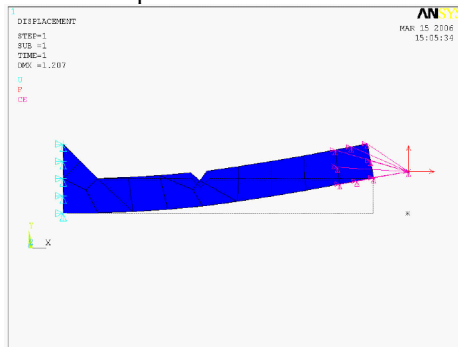


Figure 1

fit. For this example, the nodes on the top and bottom surface were selected. Figure 2 shows the 2 rows of selected nodes. These nodes were connected to COMB14 springs with coincident nodes. The coincident nodes then have displacements applied from the solution shown in Figure 1. All other boundary conditions are then removed

from the model. The springs are weak and therefore cannot deform the mesh; the mesh was allowed to float on the springs.

The static solution to this loading minimizes the strain energy in the springs. This was

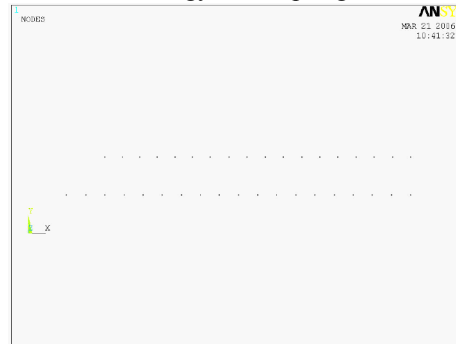


Figure 2

equivalent to finding the minimum least squares difference in position between the deformed mesh and the rigidly rotated mesh. The results of this solution are shown in Figure 3.

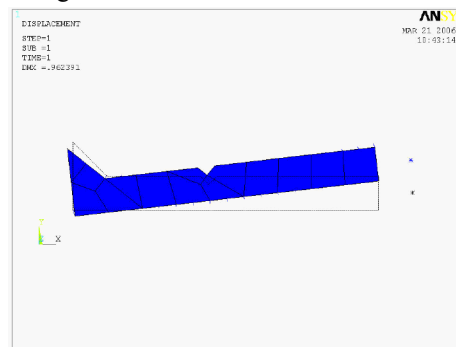


Figure 3

The final step was to determine how much rotation resulted. This was done by calculating the difference between the average cylindrical nodal coordinates, before and after the final solution. Those calculations included a lot of APDL with \*vmasks, \*vgets, \*vopers, and \*vscfuns which are beyond the scope of this article and left as an exercise for the reader (or you can email carlos.shultz@padtinc.com and I'll send it to you).

## RBE3's, A Visitor from the NASTRAN World

Way back, when you talked to a NASTRAN user, they would complain that ANSYS doesn't have RBE3's and therefore sucked. If you pointed out that RBE3's were just a class of constraint equations that you could create with APDL very easily it fell on deaf ears.

To solve this blood feud, ANSYS added an RBE3 command to generate this type of constraint equation. It uses a value supplied by the user in an array that defines the weighting factor between the master to the slaves. Pretty useful for connecting a point to a model. To learn more, read up on the RBE3 command in the manual.

## Upcoming Training Classes

Month	Start	End	#	Title	Location
May '06	8-May	9-May	100	Engineering with FE Analysis	Tempe, AZ
	11-May	12-May	803	Tcl/Tk for ANSYS	Tempe, AZ
	15-May	16-May	604	Introduction to CFX	Irvine, CA
	18-May	19-May	203	Dynamics	Irvine, CA
	25-May	25-May	106	ANSYS WB DesignXplorer	Tempe, AZ
Jun '06	31-May	31-May	108	Paramesh	Tempe, AZ
	12-Jun	14-Jun	101	Introduction to ANSYS, Part 1	Tempe, AZ
	19-Jun	20-Jun	201	Basic Structural Nonlinearities	Albq., NM
	21-Jun	22-Jun	204	Advanced Contact & Bolts	Albq, NM
	26-Jun	27-Jun	301	Heat Transfer	Irvine, CA
Jul '06	28-Jun	29-Jun	107	ANSYS WB DesignModeler	Irvine, CA
	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



## Links

Wonder what the rest of the FE world is up to? NAFEMS is the international organization for Engineering Analysis

[www.nafems.org](http://www.nafems.org)

Master ANSYS Post Processing by reading old The Focus articles on the subject by searching on "post"

[www.padtinc.com/epubs/focus](http://www.padtinc.com/epubs/focus)

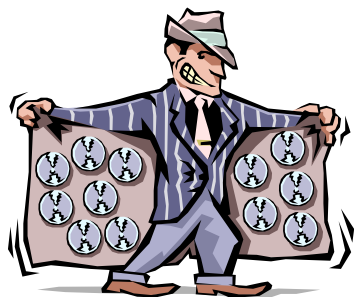


## News

- ANSYS CEO asked to ring NASDAQ bell in recognition of outstanding performance in 2005 [link](#)
- Sec. of Tres. Snow hosted a roundtable event at ANSYS HQ to highlight high-tech industries successes and needs. [Link](#)
- ANSYS, Inc. posts another record Q1 financial performance. [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***

# AMAZING ILLUSORY X-RAY VISION INSTANTLY!

A MILDLY, LAUGHINGLY FRIGHT ILLUSION!  
As though finger-thrust-out-in-front-of-eye-seen  
lead in pearls, many, many amazing, astonishing "X-Ray"  
"X-Ray" views yours to see ALMAY - when YOU wear  
"X-Ray" "X-Ray" Vision. Bring, bring to justice for real  
FUN - GUARANTEED - They give you a 3-dimensional  
illusion of "X-Ray" Vision - the instant you put them on.  
When you look at your friends you will most likely  
laughingly (and) amazing things! See the instant you are  
asked, but you get an illusion of "X-Ray" vision as a  
leading you will hardly believe your eyes. Fast instructions  
on How to Enjoy Them To The Fullest Last For Years -  
Furniture - Requires No Electricity or Batteries - Comes  
Complete - Permanent - Bright - Fun - Fun - Fun - Fun -  
Send \$1. plus 25¢ for postage and handling or send \$2. for  
the De Luxe Model. Money Back If Not 100% Satisfied.

**REMBRANDT CO. "X-Ray" Dept. 16**  
285 Market Street, Newark, New Jersey

1 envelope \$1 plus 25¢ for postage and handling Total \$1.25.  
Send on the Silliman - X-Rays, or \$2 for the De Luxe Model.  
My money will be refunded if I am not 100% Satisfied.

NAME \_\_\_\_\_  
ADDRESS \_\_\_\_\_  
CITY & STATE \_\_\_\_\_




**Final Exam-Water Tower Macro**

Using APDL, create a macro program for the final exam shown on the right.

Global material properties are as follows:

- E = 3.0E6
- $\nu$  = 0.3
- $\alpha$  = 1E-6
- $\mu$  = 0.5
- Mod = 1.0E+09
- $\nu$  = 0.3333
- $\alpha$  = 0.000001



ANSYS APDL Macro Program

## Affordable, effective Dimension 3D Printers



Nothing communicates design ideas faster than a three-dimensional part or model. With a Dimension 3D Printer, you can bring CAD files and design ideas to life – right from your desktop. Test form, fit and function – and as many design iterations as you like – with functional ABS parts. For added hands-free convenience, consider the Dimension SST with a water soluble support removal system. Starting at \$24,900,\* Dimension 3D printers represent a major price breakthrough – with leading 3D printing technology.

Simply connect a Dimension 3D printer to your network. Load Catalyst software. Insert the self-loading material cartridges and follow the display panel prompts. That's it. 3D printing with Dimension or Dimension SST is as easy as clicking print. See why a Dimension 3D Printer is the fastest selling office-friendly machine of its kind – and start shortening your design and development cycle.



**ANSYS Product  
TRAINING**  
*Instruction by Engineers for Engineers*