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**USING Pro/ENGINEER'S®
INTERFACE MODULE.**

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Sandia National Laboratories

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ABSTRACT.

When the ACCORD¹ Process was introduced Pro/ENGINEER to Sandians several years ago, a new process for design/definition was implemented. Prior to ACCORD, engineers and draftsmen worked in the 2-D mode with a program called ANVIL®, which had limited capabilities. Although the transition from 2-D modeling to 3-D modeling met with some resistance from some of the older workers, most engineers have embraced this new concept with enthusiasm. They are now able to work in the 3-D mode and at increased levels of productivity with appropriate time savings never achieved before.

One area that Pro/ENGINEER is noted for that this report will concentrate on, is the powerful interface module with its wide selection of transfer file configurations. This allows the engineer to create parts or assemblies and transfer them to many different second party software packages whose vendors can provide the capability for stress analysis, rapid prototypes, virtual reality environments, or many other forms of advanced manufacturing modes of communication.

The ACCORD Program has at its core, the Pro/ENGINEER program from Parametric Technology Inc.. Included in the ACCORD program, are several supporting programs from other vendors to make this cooperation between software packages a reality. It is possible to create parts in Pro/ENGINEER, transfer those parts to another package that has the capability to analyze the parts for deficiencies, then optimize those parts, and allow for changes to be made. All this in a fraction of the time formerly required to accomplish the same tasks just a few years ago.

Also included in this report, are other packages closely tied to Pro/ENGINEER, but not necessarily supported under the ACCORD program. Some of these packages allow you to create very impressive video productions, or allow you to meander through a virtual reality scenario. All of these new software packages will give you a new perspective on performance.

This report will show how some of these interfaces work, and how you can improve your productivity if you utilize the ACCORD program as it is implemented here at Sandia.

¹ The ACCORD Process was initiated as a plan for upgrading Sandia's design/definition capabilities going into the 90's.

Acknowledgments.

This is one of those subjects where no matter how much one knows, information is needed from other experts to complete the amount of necessary information that must be published. Without the input from people like John Bell, Gerald McCarty, and Dave Rakestraw from Parametric Technology, this report would not be published. Thanks is also directed toward the software vendors, who I have cajoled and pushed over the years to make their products compatible with Pro/ENGINEER. The results are software packages that more easily integrate with Pro/ENGINEER thus allowing mechanical design engineers more freedom in their design and analysis tools.

I must also thank Robert Alvis, my immediate supervisor, and Bill Nickell, my director, for being very tolerant of the time required to do this report.

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Part I.

INTERFACING TO OTHER ANALYSIS PACKAGES THROUGH Pro/ENGINEER.

INTRODUCTION.

One of the most powerful features of Pro/ENGINEER is the Interface module, which allows for the transfer of your part and assembly files to a myriad of other software programs. Many engineers commonly use other software analysis packages to do stress and thermal analysis, or to create a rapid prototype, and the best way to utilize these second party vendors is through the interface module. Most of the software vendors who utilize this approach, use a different command other than "**pro**", to run their software from within Pro/ENGINEER. This advantage is gained by utilizing the Pro/DEVELOP module, with its ability to interface directly to the vendor's software. The result is the vendors particular menu picks now show up in Pro/ENGINEER's window scheme. This ability to interact lies in the many different type of file transfer methods incorporated into the interface module.

Before interfacing to some of the other software packages, a meshed part will have to be created. Thus, this report will also detail the use of Pro/FEA, and its fine automatic tetrahedral mesher contained in the Finite Element Meshing module.

DIRECT GEOMETRY ACCESS WITH P3/PATRAN®

PDA Engineering was one of the first vendors to utilize the .prt file directly from Pro/ENGINEER, rather than use a secondary method of file transfer. PDA was also one of the vendors chosen under the ACCORD program to compliment Pro/ENGINEER with stress and thermal analysis modules that interface easily in a workstation environment. Parametric Technology makes available a module called Pro/DEVELOP which allows this direct access. In the case of P3/PATRAN, it is possible to transfer the part file to PATRAN, thus allowing the 3D part geometry to be used by the solver.

Starting PATRAN from Pro/ENGINEER.

It is very easy to access PATRAN from within Pro/ENGINEER through the use of a special Unix script provided with P3/PATRAN release 1.3. This script is called **p3_proe**, and it is used in place of the standard "**pro**" command to execute Pro/ENGINEER. When you purchase P3/PATRAN from PDA, you will receive direct access to one CAD system geometry at no additional cost. In this case, the interface to Pro/ENGINEER was chosen. By running Pro/ENGINEER in this way, a new menu

pick appears under the **Interface, Export** menu within Pro/ENGINEER. This menu pick is called **P3/PATRAN**. Selecting this pick brings up another menu with the following choices:

Run PATRAN. This command runs Patran from within Pro/ENGINEER. It creates the .geo file for the part or assembly that is being worked on in Pro/ENGINEER, and then loads the information into P3/PATRAN's database. You can then view and manipulate the part in both Pro/ENGINEER and P3/PATRAN simultaneously.

Create .db. This command creates the .geo file, and reads the part file information into a PATRAN database, but does not run PATRAN.

Create .geo. This command just creates the .geo file.

This menu within Pro/ENGINEER allows users who are not comfortable with running P3/PATRAN to create either .geo files or PATRAN databases to pass on to their colleagues who will be using the information within PATRAN. It also allows users who are comfortable with both programs, to move back and forth between them and insure that the geometry data created in Pro/ENGINEER is optimal for the desired analysis within P3/PATRAN.

You may also want to utilize the Pro/MESH module instead of transferring part geometry directly. PATRAN will use the triangulated mesh in its solvers just as well.

Example.

We will create a small part in Pro/ENGINEER's part mode and export it to P3/PATRAN to demonstrate the different methods you can use to analyze the part. The part will be call 'Bracket' and is shown below.

After creating this part, you will have one of two options before analyzing your part. One, you will need to mesh the part in Pro/MESH before storing the part as a Patran Neutral File, or Two, you can store the part directly as a .prt file, exit Pro/ENGINEER, start P3/PATRAN, and then recall the .prt file. This latter method allows you more freedom to manipulate the part, but requires that you use the meshing tools inside of P3/PATRAN to create the meshed part for analysis.

After doing the analysis in P3/PATRAN, the results are shown below. In this case, the part was meshed in PATRAN, rather than using the Tetrahedral mesher from Pro.

NEUTRAL FILES & RASNA'S MECHANICA®

Although MECHANICA was not an ACCORD participant, I include its use here due to the popularity of the program among users at Sandia Laboratories.

Neutral files are formatted files in ASCII text format containing information about parts and assemblies created in Pro/ENGINEER. Since Pro uses feature-based design in their

modeling system, those features are retained by providing topology, geometry, and symbolic features. Part geometry is formatted so that the data can be read by other software systems such as Rasna's MECHANICA software. MECHANICA creates its own neutral file, directly accessing the Pro/ENGINEER database. MECHANICA's neutral file, ".mnf", is similar in appearance to Pro/ENGINEER's neutral file in operation. Since most applications involving the neutral file format are used in transferring the data to MECHANICA for stress, thermal, and motion analysis, we will concentrate on MECHANICA.

Before you can run MECHANICA directly from Pro/ENGINEER, you must install interface files to integrate Pro/ENGINEER directly to the MECHANICA product. When you install the MECHANICA software, the interface files are also installed, but you must purchase their interface license (an extra cost item) in order to activate the interface. This interface installs the necessary files to create menu picks within Pro/ENGINEER that allow you to transfer data files in a smooth transition to the MECHANICA solvers. Rasna does not use the Pro/ENGINEER window motif for their operations, but for you who are familiar with Rasna's software, this should not be a problem.

Starting the integrated version of Pro/ENGINEER.

To start the integrated version of Pro/ENGINEER, enter the following command at the operating system command prompt: **ptcmech**. This brings up the Pro/ENGINEER program with a small difference, that being an additional MECHANICA menu item. After you have created your part, you will pick **MECHANICA** from the interface menu. When the **MECHANICA** menu appears, you will notice the menu contains the **Filter-All**, **FilterGeom**, **Motion**, **Structure**, and **Thermal** commands. If you are in assembly mode, notice the Structure and Thermal commands are inactive. When you are in the part mode, the filter commands are inactive. The Filter commands are used if you are transferring an assembly, but do not want all of the parts to transfer. This allows you to transfer only the needed geometry to Applied Motion that you need.

You then have the choice of selecting the proper module in the **MECHANICA** menu, after which you will be prompted for some defaults. Follow the prompts if you do not want to make changes, and you will see a message that indicates that the .mnf file has been created and Mechanica has been started. The Pro/ENGINEER window is exited and the Mechanica window replaces it with your model appearing after a few seconds. At this point Mechanica has created from your original part, a neutral file with the .mnf designator. You are now in a totally new environment, that being the Mechanica window motif. Had you selected a stress analysis, you would be in the Applied Structure module. A thermal analysis would bring up Applied Thermal.

You should also keep in mind the following factors when you create a part that you plan to transfer to Mechanica Version 5.0:

Only geometric entities, mass properties, and part coordinate frames transfer to MECHANICA. However, mass properties and part coordinate frames are visible only in Applied Motion. Some Pro/ENGINEER geometric entities do not transfer to MECHANICA. For more detailed information, see the MECHANICA Interface Guide.²

² Mechanica Interface Guide, June 1994, Release 6.0.

Using the Latest Version.

With the release of Version 6.0, several new features were made available. A new Direct Geometry Access allows better transfer of curves from Pro/ENGINEER than before. This local area tolerancing allows for better association of data entries with different geometric accuracy ranges. Also, the Automesher on Version 6.0 is much more robust and faster. Since Rasna has never used the Pro/MESH module, this feature will enhance their product.

Since we are dealing with interfaces that look and feel like Pro/ENGINEER, we will not dwell on the capabilities of MECHANICA at this point, other than to say that the MECHANICA software is some of the best to be had in the stress and thermal analysis field. Their use of geometric elements map directly to the part geometry without any approximations except as noted. For more information about running MECHANICA, contact the Rasna Corporation.

SLA FILES AND ALGOR®

Of all the methods used by second party vendors to interface to their products, ALGOR has chosen the most unique. After installing ALGOR's interface to Pro/ENGINEER, you will notice that their menu box shows up in the Pro/ENGINEER menu system. Algor has chosen to utilize the .slp file to transfer the triangulated surface data to the ALGOR menu system. The reason for this file transfer method has to do with the automatic meshing capabilities ALGOR uses in creating its tetrahedral mesh. This operation is completely transparent and after following the menu prompts you will be in the traditional ALGOR system which has been adapted to X-windows and runs under the UNIX environment quite well. We will not dwell on this analysis package more than this.

For more information, contact Algor Inc., 150 Beta Drive, Pittsburgh, PA 15238-2932.

COSMOS/M ®ENGINEER.

Since COSMOS/M is a program that operates and has the look of Pro/ENGINEER, I include it as an alternative FEA solver that is also in common use here.

At this point, we must assume that the reader is familiar with the Pro/ENGINEER design program which includes the Pro/MESH and Pro/INTERFACE modules. The installation of this product is covered in the Interface Users Guide³ from SRAC.

COSMOS/M ENGINEER is a seamless interface that combines the modeling strengths of Pro/ENGINEER with the analysis capabilities of COSMOS/M finite element analysis system. The analysis capabilities of the interface are selected to maintain simplicity and cover most of the design engineers' needs which include linear structural analysis, stress, thermal and frequency analysis.

The program is simple to use and has the feel and look of the Pro/ENGINEER environment. Design concept engineers may use it to do design optimization by

³ Interface Users Guide, Version 1.70, SRAC. February, 1994.

evaluating alternative designs. Version 1.70, is compatible with release 12 of Pro/ENGINEER.

We will assume that your installation has been completed successfully, and will not dwell on it here. The COSMOS installation is one of the easier ones (if there is such a thing in UNIX installations), and is covered quite well in the User's interface guide.

Starting COSMOS.

After a successful installation, COSMOS/M ENGINEER may be started by typing **cospro**, or **runcosmos** from the operating system level of any sub directory. After the program starts, the top level startup menu is displayed. From here, you can start Pro/ENGINEER.

Example.

We are going to continue through the rest of the interface by creating an example called **ex1.prt**. Note that the drawing shown of the example does not contain any dimensions. Adding all of the dimensions clutters up the figure so we have left applying dimensions to you.

ENGINE MOUNT

This part is similar to an engine block mount and can be handled in one of two ways: In one procedure, you can handle all pre-processing in Pro/ENGINEER. The second

method allows you to complete all pre-processing, other than modeling, in COSMOS/M. We will select this second method initially, and later go back through the first method of using Pro/MESH while in another chapter dealing with Pro/MESH.

After you have created the part **ex1.prt**, the next step is to take it back to COSMOS, and this is done by selecting the **Cosmos Geom** from the **Export** menu of **Interface**. You will get a message that the cosmos trimmed file has been created. Exit Pro/ENGINEER, and you will be returned to COSMOS/M ENGINEER. Notice that you are switching between two programs, but they look very much alike. The model will automatically be loaded into COSMOS/M ENGINEER under the same name.

Property sets.

You will need to select **properties**, **element groups** and **define group** at this point. Here is the first time you get to select the type of elements you are going to create and the choice is either four node tetrahedral or ten node tetrahedral. If you want accuracy and have the computer power to dictate speed, choose the ten node elements. Next you need to choose a material, again from the properties list. For this example, we are going to use steel, which comes up as the default.

Mesh Generation.

We are now ready to generate the mesh. The program calculates and assigns an average element size to be used. You can modify this if you wish, but we have found that in most cases the default element size will work just fine. The menus detail the default selections automatically, and if changes are to be made, merely picking a new value from the menus will give you different direction in which to proceed. Choosing either linear or quadratic elements is your method of selecting either four node or ten node elements. This part is a good example for meshing, containing different features that will challenge most meshers from other companies. This mesher from COSMOS is one of the best in the world. It is not the fastest, but on small parts it work very well. You can watch the elements being created and this gives you an idea of where the meshing process is at all times. The elements created are very symmetrical with a good aspect ratio. Once you have chosen the element size and tolerance, you can mesh the part.

Loads and Boundary Conditions.

By following the menu picks, you should be able to assign boundary conditions to the two holes in our model. You can turn the labels on by selecting **Status** from the **Utilities** menu and turn on label plotting for **RG**. This allows you to pick areas with the mouse without having to type in the regions that you want to select. Next we will apply a pressure of 2000 lb. to the top surface. This action is straight forward and doesn't need elaboration.

Analysis.

One of the nice features of working within the interface, is that you can apply temperature conditions, stress conditions, frequency analysis, and other environments without having to quit one program for another. The interface automatically calls up the proper program when you do the analysis for the condition in which you want answers. For instance, when you want an analysis for the stress condition we just applied to the support part, you select **Analysis, Static**, and the other defaults, and the STAR module is displayed replacing the interface screen. STAR starts to display information as the analysis proceeds. Once the deformations are calculated, control is given to the STRESS module to calculate the corresponding stresses. After calculating stresses, control is once again given back to COSMOS/M ENGINEER. Had you called out for more than one type of analysis, the same procedure of menu picks would be used to select each different type of analysis, and the corresponding solvers would have been displayed. For example, if you had added a thermal condition to the model, you would have selected the proper menu items to invoke HSTAR in the same manner as you picked STAR, and the screen would show the thermal solver.

Post Processing.

Results of your analysis are now available in the database. To activate post processing for the static analysis on your model, select **Postprocessing** from the main menu. Follow the menu picks, which are fairly straight forward, remembering that "act" is short for "activate". This section is one of the few where you must tell the post processor every step to take. If you remember this, then displaying the information is simple.

PRT FILES AND ANSYS®

This section describes the installation prerequisites you will need to run the ANSYS/Pro/FEA interface. Regardless of the computer you are using, these requirements will remain common from machine to machine. Thus, we will not dwell on machine specifics other than some mention of graphics later in this section. Some of the hardware requirements are; 32 MB minimum memory, 300 MB minimum disc space, 60 MB swap space, and a 3-D graphics monitor. You can get by with a lessor graphics device, but the enhancement from a better graphics device will pay off many times over since analysis results tend to be best displayed in livid color.

Starting ANSYS.

ANSYS/Pro/FEA 5.0 runs entirely within the Pro/ENGINEER session. Once the ANSYS/Pro/FEA interface has been properly installed (a more difficult process than most other analysis programs exhibit), you activate the interface session by issuing the standard system call for Pro/ENGINEER. On most systems, this involves just typing in the word **pro**.

Most engineers that are using Pro already should feel right at home using the ANSYS interface since it has the look and feel of the Pro/ENGINEER environment through all phases of your analysis and optimization sessions. This familiar environment is only shared by one or two other analysis programs and has distinct advantages if you are doing Finite Element Analysis on a part time basis or infrequently.

Example.

After starting Pro/ENGINEER, create a part called BRACKET. One difference when running the ANSYS interface to Pro, and different from other analysis programs is that you need to mesh the part in Pro/MESH in order to utilize the interface fully. Since meshing is covered in another chapter, we will not spend any more time describing it now. Shown below is the model of the BRACKET that you need to make.

BRACKET

To define a study name for this example, choose **ANSYS** from the **PART** menu. Accept the defaults which give you a default name **Study 1**. To generate the FE model, choose **Mesh/Model**. This starts the Pro/MESH module. After meshing, choose **Output** from the **AFTER MESH** menu and **Accept** from the **OUTPUT MESH** menu. You will notice a message being displayed on file creation. Choose **Done** and exit Pro/MESH. Once you have built a part and created a FEM model using Pro/MESH, you will discover that you need make only a few additional picks to analyze and optimize your design using this program.

Analysis.

You may want to run an optimization study but normally you will just want a design verification and a simple analysis will more than likely be sufficient. For more detailed results, refer to the ANSYS/ Pro/FEA Training Guide⁴. Choose **Run Study**, **2D**, and **Analysis** and accept the defaults. ANSYS detects that you have already generated a meshed file for this part and asks you to verify it.

You have now submitted the analysis study to run in the background. Note that when you choose to run the analysis in the background, you still have control over the Pro/ENGINEER menus. After the run is complete, choose **Run Status** to verify the end of the run.

Results.

To view the results, choose **Results** from the ANSYS menu, **ANSYS View**, and **View Angle**. The first thing you will notice is a change in the screen. A new screen appears with a small coordinate system showing in the middle of the window. There are quite a few variations on viewing the model, picking the appropriate results and whether you want to view deformed or non-deformed results.

To view the deformed shape of the bracket, choose **Deform Shape** from the ON MODEL menu. You should typically view the deformed shape for two reasons. First, it gives you a visual representation (greatly exaggerated) of how the model is deforming under its applied load. Second, it gives you the SEPC value of the FE mesh. This value indicates the accuracy of the mesh based on a single solution.

After viewing the results, you should then perform optimization studies if you want to vary your design parameters for efficiency of design.

⁴ ANSYS Pro/FEA Training Guide, Swanson Analysis Systems, Inc, DN-P515:50A
11/93.

PART II.

Pro/MESH: Preparing the model for FEA.

Introduction.

This section provides guidelines for preparing your model in Pro/MESH for a Finite Element Analysis in other software packages. These guidelines will highlight the more important steps that you should follow. We assume that if you are using Pro/MESH, that you have a good idea of its basic operation. However, we will include enough details so you can follow along if you are not totally familiar with the module. We will assume at this point, that you have created your part model and you are ready to start preparing it for finite element generation.

What is Finite Element Analysis?

You are well aware of the fact that any part, when put in use, is subject to various forces, temperature variations, and vibrations. It is your job as the design engineer, to make sure parts and assemblies that you conceive, perform satisfactorily. In other words, you do not want to be responsible for a part or assembly that you have designed, breaking, sagging, overheating, or suffering some general failure that could have been prevented with initial stress and thermal analysis.

You have probably used proven mathematical formulas from other sources, along with known experience to assess the adequacy of your models in the past. Unfortunately, only the simplest of parts can be analyzed in this manner. With the advent of CAD workstations, allowing much more complicate designs, the old analysis methods become less and less accurate. This is where Finite Element Analysis comes into play.

The Finite Element method divides a continuous part into many discrete mathematical components, known as **finite elements**. Most software analysis programs available today solve for the primary responses of the system at every point of connection between elements. In general, the accuracy of a finite element analysis improves with the decrease in element size, but this contributes to a larger number of elements that need analysis, thus driving computer requirements up in terms of speed, memory, and storage.

Preparing a Design For Analysis.

Most part models require some modification and other preparation to make a model more suitable for analysis.

Simplifying Your Model.

You can often speed up analysis by simplifying your model. For instance, your part model may contain certain features such as small holes, fillets, and chamfers that become insignificant to the analysis due to size or placement. Including these features will more than likely increase the number or the density of elements generated, thus increasing the solution time unnecessarily.

Consider simplifying your part model by removing details that are either not needed, or insignificant for the analysis. This is easily done in Pro/ENGINEER by using the **suppress** feature. If you are not sure which features should be suppressed, leave the small features in the model, but don't apply any localized mesh control around these features, as Pro/MESH will generate a reasonable, yet coarser mesh around these features. If you see after the first analysis that better definition is required, simply go back and refine the mesh for better results.

Many parts have some kind of symmetry, whether it be repetitive, reflective, or axisymmetric. You can often take advantage of these facts by reducing the size and scope of your FEM model. This can be accomplished by either "halving" or "quartering" your model. When the analysis is complete, the "cut" feature can be suppressed to regenerate the full part. Significant savings in analysis time can be obtained using this technique.

Creating a Coordinate System.

You will need to create a coordinate system while you are in part mode. Pro/MESH uses this coordinate system to define the vector components of many of the loads and constraints that you apply to your model.

You have three different types of coordinate systems at your disposal: Cartesian, Cylindrical, and Spherical. These terms are fairly self explanatory. You should keep in mind that if you use the default to create the coordinate system, its placement will be defined by the first point of creation in your model. This may not cause you any problems in analysis, but might cause a problem when you go to place the part in an assembly.

Datum Points.

Datum points can be used to position point loads or constraints at a point on your surface. It is the only way to apply a point load when no other implicitly defined features exist at that particular location. For instance, when you have a part defined in the part mode, you can apply one or many datum points when you pick the **Create, Datum, Point**, commands. Notice that the spot you pick is directly connected to the mouse control and merely represents a spot on the screen. The default location may not be the spot you want to locate your datum point exactly, and you may have to adjust it with different dimensions as you go through the menu.

Defining Material Properties.

Before the part can be submitted for analysis, the material properties must be assigned. Failure to assign properties will cause a default to an isotropic status in material. You will need to set up a material file while in part mode. Following the menu will tell you what is needed, and you can then assign that material file to selected parts of your part model when you are in the FEM menu. Remember; you do not need to fill in all of the blanks from the material file, only the ones pertinent to the desired analysis.

Surface Mesh Vs Solid Mesh.

You need to determine whether your part is a solid or is it a thinner structure like a pressure vessel. If the aspect ratio of the elements created during the meshing process begin to stretch, then going to shell elements will give you a better answer. Shell elements are a little harder to create as the surfaces must be declared.

Sometimes it is much simpler to use solid meshes because visualizing how to apply loads and constraints to a solid model is more straight forward. Unfortunately, many more elements are generated in curves and radii of solid models. If you have the computer power to handle to the increased number of elements generated in 3-D models, you may have more confidence in the solutions obtained.

Mixed meshes may offer you a good compromise solution for parts and assemblies when you have both solid and thin sections. For instance, if your part has a thick base leading to a thin wall, then you should investigate the creation of mixed elements.

Bar elements are created from datum points and are used to constrain other elements. More information is available from the User's Guide⁵

Tetrahedral Elements.

Pro/MESH creates its mesh using Tetrahedral elements. The advantage of tetrahedral elements is that any shape can be meshed since tetrahedral elements can be made to conform to any shape. It is much more difficult to solid mesh with brick type elements and the automatic meshers are just not perfected yet to create realistic brick elements at this time.

Unfortunately, tetrahedral elements have some severe drawbacks in that they lack the degrees of freedom that brick elements display and thus become "stiff". If your analysis is not dependent upon whether an element will be stiff or not, then tetrahedral elements should be sufficient for your analysis. This is one point to keep in mind.

⁵ Bar Elements, Section 2 of the Pro/MESH Users Guide, Parametric Technology Corp.

Preparing a Bracket For Analysis.

Example:

We are going to create a model from start to finish and hopefully show when and how to use Pro/mesh to this point. In this case, we will create a model called BRACKET, and is defined in the next figure.

Datum Point for Point Load.

Before you send the model to the FEM module, you will want to place a datum point on the tip approximately 1" from the end, and .5" from either side. This will be used to apply a point load to a portion of the surface rather than to the edge, which in this case is not realistic. You could have used the **pressure** command to apply a load to the entire surface, but the results are not exactly what we are looking for in this case. Note that the point position is shown in the previous figure.

Starting the FEM Module.

If you are using version 12, you will notice the menu picks have changed from earlier versions. Now the FEM pick is under the Part menu. After selecting, you should have the model on the screen in front of you at this point. Pick **FEM**, **Define model**, and **Loads/BC**. You need to give your model a case name at this point. Call it "example".

NOTE: Though you can mesh parts in either part or assembly mode, loads and boundary conditions can only be applied in part mode.

Applying Constraints.

The next thing we will do is apply a constraint case. This is done by picking **Displacement**. Note that a series of defaults are automatically displayed, one of which is constrain. In this case, we will accept the defaults and then use **Query Select** to pick the back surface of the plate. Once that is done, you will notice that the vertical back surface ends up with a series of small stars in a different color indicating that this surface is now immovable. If you want to adjust the constraint for axial movement, now is the time to do that.

Applying Loads.

Now we will apply the load. Select **Force** and when asked, select the datum point that was installed earlier when you made the model. In this case, you want to apply a more realistic force to a surface rather than using the edge for applying a load. You cannot manipulate the direct application of a point load to a surface without the use of a datum point, thus the creation of a datum point at the point of loading. Since Parametric Technology has incorporated this method of point loads, you will find many uses for it when you are applying loads and constraints. This method more closely approximates a real loading situation. Loads on edges cause undue shear and that solution will not give you the right answer, at least not in this case.

Give the model a point load of 500 pounds in the -y direction. This is done by following the prompts on the screen. Make sure you think about the directional application of your load.

Now you see the need for the coordinate system before you passed the model to the FEM module. Tip: A good rule of thumb is to always create a coordinate system in any model that you plan on exporting, or if you plan on using that part in the building of an assembly.

Next, pick **Material**, the **Whole Part**, and **Assign**. A default material may come up if you have any material files stored. If not, then you will need to create a material file, and in this case, call it "steel". Use the **Setup** pick under the **Part** menu to do this. If you are not sure of the procedure to do this, refer to the Modeling User's Guide from Parametric Technology, chapter 24.

Normally we would apply a mesh control to the part, especially if it was not symmetrical. But in this case, we will bypass that feature and allow Pro/MESH to control the mesh.

Select **Make Model**, and **Tet Mesh**. The program takes control and the best thing you can do is sit back and watch the proceedings. Pro/MESH is one of the best meshers that can be bought and is well worth the price. If you look at the mesh on the screen now, you will notice a finer mesh concentrated around radii, giving a higher resolution in your solution. The last operation to take place is the writing of the file to the output under the name BRACKET.bde. Since we did not specify a change in the mesh control, notice that the mesh control fixed a higher density mesh to the curved areas of the bracket.

Once the output file is generated, it is available for exporting to one of many analysis formats. These packages will be discussed in later chapters.

Summary.

In this section we have discussed and demonstrated methods to prepare your part for Finite Element Meshing. Some of the details discussed were:

Use of Datum points.

Suppression of non-essential details in your part.

Discussion of mesh densities.

Importance of the Coordinate system.

Application of loads and constraints.

Pro/MESH offers a very convenient method of FEM for further use by several analysis packages. In this era of design verification, this module leads to and lends considerable credence to the adage of "Analysis on the fly"!

PART III.

Using Pro/ENGINEER in Rapid Prototyping.

Introduction.

With the changing atmosphere in the world today, there is a corresponding change to the funding of the weapons business with a focus on cutting the cost of doing business and a smaller output of new designs for the weapons of the future. This new way of doing business has mandated the use of advanced manufacturing techniques with an emphasis on rapid prototyping and production of models rather than real parts and assemblies.

SLA FILES AND THEIR USE IN STEREOLITHOGRAPHY.⁶

When Pro/ENGINEER decided to include the capability to output .stl files, the rapid prototyping field became the major beneficiary. The .stl file is a valuable tool and has become the standard for the leaders in rapid prototyping, such as the 3D Systems stereolithography machines. (It should be noted here that the designation of 'SLA' is a broad representation standing for "StereoLithography Apparatus", and is a term commonly used throughout the industry. The actual file designator is '.stl').

Stereolithography is a process which reproduces a solid plastic like model from a CAD model using a .stl file format. This process typically uses a helium cadmium laser and a photo polymer that is photo curable when exposed to a laser beam. In other words, the polymer hardens when exposed to a laser light. The parts are drawn with the laser on the surface of the resin which is contained in a vat. These sequentially hardening layers are attached to a platform by a support structure (described later) and as each layer is created, the platform lowers in the vat. Thus, parts are built from the bottom up along the Z axis curing each layer to the previous layer until a completed solid model is produced.

It is extremely important that the engineer producing the necessary part files in Pro/ENGINEER understand the sequences needed to produce a stereolithographic model. The operator of the SLA machine needs to converse with the engineer designing the part in order to obtain specific information such as part accuracy, surface finish, shrinkage, fit, and other details.

Support Structures.

Support structures are the foundations needed to build the part on and are created in the Pro/ENGINEER assembly mode. You can draw as many supports as you need to support your part and they can be combined under one file name and output as a single .stl file or as many .stl files. You need to attach a single coordinate system to orient both parts and support structure in the SLA machine. Once the file has been transferred to the SLA machine software, a slicing operation takes place that "honeycombs" the support structure files to create a different structure from the actual part structure. The difference being that the support structure is cut away from the part after the model is created and cured and this different texture allows you to more easily cut away the supports.

Some tips useful in the creation of stereolithographic parts:

1. Supports should be kept to a minimum since all supports will need to be removed from the part later.
2. All horizontal surfaces will need support.

⁶ This section on Stereolithography was augmented with the help of Gerald D. McCarty, a fellow Sandian who has been instrumental in the implementation of Rapid Prototyping at Sandia National Labs.

3. Angles, radii, holes, and curves are self supporting.
4. Do not attach supports to vertical walls. They are not needed.
5. Do not run supports through holes or other structure. Removal becomes difficult.
6. Do not build supports where a good surface finish is required.
7. Avoid trapped volumes in your structure if fluid will accumulate.
8. Walls of less than 30 mils in thickness are not recommended since the laser beam is usually only 10 mils wide.

Creating Video Productions From Pro/ENGINEER.

One of the more interesting things a person can do with Pro/ENGINEER is to create stunning graphics, either in the still form, or video form. Most engineers are familiar with the program's ability to create highly colorful and exact computer generated 3-D models and they know how to print these images to one of the many fine graphics printers that are available. However, not many engineers know that they can also output these images directly to another software package for the computer generation of some very fine animated video. This method of animation is accomplished in a program called "The Advanced Visualizer", from Wavefront Technologies.

For many years, Sandia has had the ability to create video reproductions, but only with the recent implementation of Pro/ENGINEER as the CAD package of preference has Sandia had the ability to create video productions directly from the engineer's design in Pro/ENGINEER.

One of the real experts in this field, is John Bell⁷ from the Art Department here at Sandia. He is responsible for most of the information in this section on video productions.

Concurrent Engineering.

Engineers do not normally associate video production with concurrent engineering, simply because there has been little relationship between the two in the past. Computer animation is a relatively new art to the engineering community and while the more established processes of rapid prototyping have generally taken the spotlight, video animation resided in a world of its own. It is only when the advantages of computer aided engineering are applied to video animation, that a relationship becomes apparent.

This new, exciting process of engineering videos, while still in its infancy, has more potential for advancing the art of visual applications than do most other prototyping processes.

⁷John Bell, a visual arts expert, is employed by Sandia National Laboratories in Dept. 7155.

Consider for a moment, the impact of watching a process as it proceeds from individually designed parts to full assembly with a smooth motion. For instance, an assembly being put together or dismantled, in sequence and in motion similar to an actual detailed procedure, all while the viewer manipulates the modeled parts in their hands. That is visual impact!

Pro/ENGINEER has always had the ability to make assemblies from individual parts, but the program has never attached a smooth, frame by frame motion to their assembly. It has either been exploded or un-exploded, with no interpolated action.

The Old Way.

The use of Video tapes for staff training of people has been used at Sandia and other large companies with increasing frequency over the last few years, but the cost associated with graphic arts has precluded the average small company from enjoying these tools in the training of their personnel. For instance, if one wanted visual creations to show the assembly procedure for a mechanical device that contained a number of small parts, one either had to film or video tape the actual assembly procedure, and then spend considerable money and time with a commercial supplier that specialized in editing and formalizing film productions. An alternative was that the entire assembly procedure was created by artists in a frame by frame mode until enough frames were drawn to depict the procedure. Many frames were required that would give enough detail to the observer watching this tape or film to learn how to assemble the device. Letting the computer generate the needed interpolated frames is certainly a better alternative as is shown in the previous figure.

An Internal Presentation Tool.

With the advent of computer generated animation, as is evidenced in the Star Wars series of movies, it is easy to bypass the tedious recreation of events by using a computer to manage the manipulation of parts. Contemplate for a minute the potential of this computer generated art as a new tool Taken from the Star Wars movies. By adapting this new concept to the engineering process, one can integrate this process into rapid prototyping as a concurrent process. If a presentation to the management team associated with a particular project is presented with a strikingly animated video depicting the new project in motion, along with a hands on, "touch and feel" from stereo lithography produced parts, it would seem likely that their attitude would be far more favorable towards implementing the project with fewer questions and less uncertainty felt by the team. This now becomes a very strong presentation tool. In Pro/ENGINEER, this ability comes about due to a couple of interesting concepts. One, a very robust interface package which includes the capability to produce a render, or as commonly known, the .SLP file. Two, Wavefront Technologies, the company that produces "The Advanced Visualizer", developed a translator, called "The Visualizer Translators", to read the render file produced in Pro/ENGINEER.

Background.

Wavefront is an animation software company that was formed in 1984. Their goal was to develop broadcast quality 3-D animation software to run on commonly used computer hardware. 3-D animation is now well accepted, not only in broadcasting, but also in film production and Engineering visualization.

The determining factor of what filters or translators are developed by Wavefront is customer driven. Fortunately there were enough Pro/ENGINEER users wanting to generate advanced graphics that Wavefront developed such a filter. The work that was involved in creating a filter was not as difficult as they thought. They discovered that the stereo lithography object file generated by Pro/ENGINEER had similar requirements that were needed to render a Wavefront object, so in effect, Pro/ENGINEER had already met Wavefront halfway.

From Design to Translation to Production.

The production of a video tape now becomes a fairly simple process. First the object, or 3-D part geometry, is created in Pro/ENGINEER and saved as a render or SLP file. The object in the render file is defined much like the stereolithography file defines its models and both appear to be identical. Once the render file is created, it is available for import to the translator. Using this filter is simple and straight forward. The file created is an ASCII file with a ".obj" extension. The translation time is quite fast and runs in verbose, such as indicating the output number of vertex points, normals and polygons to the observer. Once the translation is complete, that file is brought into "The Advanced Visualizer's" modeler. The next Figure shows a part in place.

In the modeler, the object is treated the same as any other object that one either creates in The Advanced Visualizer, or imports from some other source. At this point, the next operation is to check to see if the object is correct and that all elements are facing the right way. If an element is incorrect, direction wise, a simple auto smooth may be necessary. If all entities are correct, the next step is to apply properties such as color, texture, shadow casting, ray tracing, etc., to the object.

Animation.

The object or part is now ready to be animated. This is accomplished by entering The Advanced Visualizer's preview module, which is like entering a stage and all the actors

are the 3-d objects. A motion channel is assigned to the object and there is one motion property per channel, thus motion in the X, Y, and Z directions require 3 motion channels. Applying object motion is achieved by moving the desired channels to an editor box. This is where selection of the correct motion channel is made and it is accomplished by either dragging the mouse or using the keyboard arrows. Once the correct starting position is established, this frame is saved and represents the first frame of motion. Next, the ending frame position in this sequence is made and that frame is stored. Now with the first and last frame in place, all that is required is to generate the interpolation command. Remember, there are 30 frames/second, so many positional interpolations will be created automatically by the computer. One has a choice of either linear, spline, ease in, ease out, etc., type interpolations. This process is known as "Key frame animation", and it is applied not only to the objects, but to the lighting and camera view.

The above figure shows a preview window with key frames 1, 11, 21, and 30 with the ghost tool feature turned on to show actual positions in the key frame sequence. This is one of many parts created by Pro/ENGINEER and sent into the animator. To check the motion of the animation, one can play it back frame by frame in either a wireframe or a shaded mode. The shaded image represents the original position.

Saving the Rendered Files.

Once the animation sequences are satisfactory, it is time to "render" each frame and save the rendered files either to a large storage device or send them directly to video recording equipment. Anti-aliasing is also needed to give the image a smoothed look when the part shown is off the vertical or horizontal axis. This eliminates that "stair step" look that most computer screens produce. Unfortunately, it uses lots of storage per frame.

The above figure shows a rendered frame from a recent video produced at Sandia Labs.

Conclusion.

It is not often that any one process becomes such a stand out in the rapid prototyping arena, but this process of computer aided animation adapted to CAD/CAE certainly qualifies. The amount of time spent to create these videos today, is a small percentage of the time required in the past and if one has the need for high caliber, high power presentations, this may be what the engineer needs.

Virtual Reality.

Introduction.

Every where you turn, you see some form of Virtual Reality these days. Down at the video arcade, some of the most sophisticated games in town depend on the realism that virtual reality brings to the computer screen. Even the military has opted to enter the playing field with SIMNET®, a simulated combat training system, used to teach everything from tank manipulation to flying a fighter plane. SIMNET®, as it exists today, is among the more successful real-world applications using computer simulation to replace actual real life experience.

The creation of true synthetic environments, or virtual reality as it is popularly termed, depends on the integration of such technologies as stereo displays (with wide fields of view), head tracking, auto generation, and force feedback. Success will also depend on being able to replace conventional methods of training and testing at a considerable savings in time and cost (not to mention its application to entertainment). Time now becomes a major factor in the development of virtual reality to become more than an oddity to the rest of the scientific community. The Military has shown that real time simulations in training can be successful. Now the rest of us need to explore the potential of full immersive environments.

MUSE

In FY91 Sandia Labs began a project to develop a new approach for the interactive multi-dimensional representation of complex information. The project, named MUSE⁸ (Multi-dimensional User-oriented Synthetic Environment) developed by Dr. Creve Maples of Sandia Labs, involved designing a modular, device independent system, which would take users data or models, and provide a highly interactive environment to examine, explore and manipulate the information. As a result of this effort, an information-rich, immersive environment has been created which permits users greater flexibility to explore complex multi-dimensional information. The system was implemented in an open and modular fashion, which permits new software capabilities and hardware devices to be added without impacting existing software models.

In the National Laboratories, one area where computer modeling and specifically, the synthetic environment might become a factor, is in the training of personnel for dismantlement of the weapon stockpile. If models of weapon systems could be transferred from the CAD systems in use today, to the computer environment needed to drive the virtual reality scheme, then real life models of weapons could be eliminated, and computer models substituted.

⁸MUSE PROJECT, "A FUNCTIONALLY BASED, HUMAN-COMPUTER INTERFACE for the INTERACTIVE PRESENTATION and EXPLORATION of INFORMATION". Author Dr. Creve Maples.

Using MUSE with Pro/ENGINEER

Since Pro/ENGINEER is the CAD system of choice at Sandia Labs, ways of transmitting the computer models to the MUSE software were needed. To test the concept, an assembly consisting of 19 parts and representing a storage container was modeled in Pro/ENGINEER, and transferred to the MUSE system via the "render", or SLA, file format. This format allows for the surface information of a solid model to be represented as groups of small polygons.

These polygons are written to an ASCII text file in set format. The quality of the SLA file may be controlled before the SLA file is generated. The quality depends on the deviation of the tessellated surface from the actual surface; this being controlled by Chord Height and Angle Control. In this case, lack of the correct chord height led to voids being visible in the case due to the thin shell and large diameter. By decreasing

the chord height, these voids then disappear. Craig Peterson⁹, one of the computer engineers responsible for operation of the MUSE system, had the responsibility to import the rendered files to the MUSE interactive atmosphere. Since the .SLA file format was used, Craig had to write a translator in "C" to interpret the .SLA (render) file and create a description of the components that MUSE could use. Initially, there were some problems determining the relative location of the components (with respect to each other). The co-ordinate systems of each part appeared to be out of alignment or in the wrong position, since exact orientation became very difficult to achieve.

Back in Pro/ENGINEER, the old coordinate systems were removed and new ones were created so that when in the assembly mode, the parts would line up along a common axis. To eliminate this problem from the start, all that needs to be done is to make sure the first point created in each part is started in the same location part to part. This is especially true if the parts are to line up along a common axis. This eliminated the location problem when the parts were re-introduced to MUSE as can be seen in the next two figures.

⁹Craig Peterson, a computer engineer, is employed by Alliance Technology, and is on contract to Sandia National Labs.

It becomes obvious at this point though, that the correct method of part introduction would be for the actual .prt file be used since its information does not have to be regenerated in a polygonal format for file transfer. In order to accomplish this, the Pro/ENGINEER development kit would probably have to be purchased so that access could be made to the .prt file.

What is Next?

The future of this technology holds both promise and questions. For instance, how does one incorporate tactile feedback to the user from the computer that would provide the information necessary to indicate the weight, or the contact position of any two parts. Currently that type of information can be synthesized in a different form to a different sense. A changing audio tone could indicate a closing distance of two parts with an abrupt end of the tone to show interference of the parts. But how do you indicate mass of an object? Answers to these and other questions are currently under investigation, and finding them will provide an exciting and challenging undertaking.

Addendum

Update:

Since beginning this report, some significant changes to software have been reported by the different vendors that are mentioned in this report, and the updated software has become available and should be commented on. The most wide sweeping change by far, was announced recently by Parametric Technology and refers to the introduction of a new module in release 13 to handle the post processing from different second party stress analysis vendors. Rather than handling all of the post processing within the second party software, the option now exists to bring the results back into Pro/ENGINEER and display the results in a different environment. Several companies have signed on to this feature as an option to their software. However, this mode of operation is not without its faults. You must purchase the Fem-Post module from Parametric at additional expense, and knowing Parametric Technology, it will not be inexpensive. Second, some of the features that PTC will offer, are already included in the cost of the software you would need to buy from the analysis software vendor in the first place. Third, as it stands now, you would still need to purchase Pro/MESH in order to use the Fem-Post module. This statement has not been verified by Parametric Technology, but by the vendors involved in using Parametric's new module.

Another significant change comes from ALGOR, in the form of an automatic hex generator that will work on a workstation in conjunction with Pro/ENGINEER. It is not clear at this time if this new brick generator is completely automatic, or if it merely works from an existing surface mesh with the creation of the surface mesh left up to the engineer doing the design. Their advertisements lead one to believe that the brick mesh will be created from the .STL file, which creates a triangular mesh on the surface for export to other software packages. This software is being evaluated at this time and results should be available in the near future.

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