DESIGN THROUGH MANUFACTURING:
THE SOLID MODEL - FINITE ELEMENT ANALYSIS INTERFACE

Carol Rubin
Vanderbilt University
EM3
August 4, 2000

Raymond Aronoff
Manufacturing Integration and Technology Branch
Manufacturing, Materials, and Process Technology Division
Engineering Directorate

Carol Rubin

Raymond Aronoff

DESIGN THROUGH MANUFACTURING:
THE SOLID MODEL - FINITE ELEMENT ANALYSIS INTERFACE

Final Report

NASA/ASEE Summer Faculty Fellowship Program - 2000

Johnson Space Center

Prepared by:	Carol Rubin, Ph.D. P.E.
Academic Rank:	Professor
University & Department:	Vanderbilt University Mechanical Engineering Nashville, TN 37235

NASA/JSC

Directorate:	Engineering
Division:	Manufacturing, Materials, and Process Technology
Branch:	Manufacturing Integration and Technology
JSC Colleague:	Raymond Aronoff
Date Submitted:	August 4, 2000
Contract Number:	NAG 9-867

ABSTRACT

State-of-the-art computer aided design (CAD) presently affords engineers the opportunity to create solid models of machine parts which reflect every detail of the finished product. Ideally, these models should fulfill two very important functions: (1) they must provide numerical control information for automated manufacturing of precision parts, and (2) they must enable analysts to easily evaluate the stress levels (using finite element analysis - FEA) for all structurally significant parts used in space missions. Today's state-of-the-art CAD programs perform function (1) very well, providing an excellent model for precision manufacturing. But they do not provide a straightforward and simple means of automating the translation from CAD to FEA models, especially for aircraft-type structures.

The research performed during the fellowship period investigated the transition process from the solid CAD model to the FEA stress analysis model with the final goal of creating an automatic interface between the two. During the period of the fellowship a detailed multi-year program for the development of such an interface was created. The ultimate goal of this program will be the development of a fully parameterized automatic ProE/FEA translator for parts and assemblies, with the incorporation of data base management into the solution, and ultimately including computational fluid dynamics and thermal modeling in the interface.

INTRODUCTION

State-of-the-art computer aided design (CAD) presently affords engineers the opportunity to create solid models of machine parts which reflect every detail of the finished product. Ideally, these models should fulfill two very important functions:

(1) they must provide numerical control information for automated manufacturing of precision parts, and

(2) they must enable analysts to easily evaluate the stress levels (using finite element analysis - FEA) for all structurally significant parts used in space missions.

Today's state-of-the-art CAD programs perform function (1) very well, providing an excellent model for precision manufacturing. But they do not provide a straightforward and simple means of automating the translation from CAD to FEA models, especially for aircraft-type structures. Presently, the process of preparing CAD models for FEA consumes a great deal of the analyst's time.

The aim of the research performed during the Summer Faculty Fellowship Program period was to explore the transition from the solid CAD model to the FEA stress analysis model with the aim of making it uncomplicated and automatic. The ultimate goal of this work will be the development of an Automatic CAD/FEA Interface (ACFI) for parts and assemblies. ACFI will be able to (a) extract a fully parameterized part or assembly of parts, (b) identify and test its individual features for possible suppression, (c) suppress the appropriate features, (d) rework the part geometries to prepare the model for finite element meshing, (e) export the revised geometries to a preprocessor, (f) identify element type to be associated with each feature geometry, (g) rerun the solution based on any design changes made, (h) import the part/assembly back to the CAD program after analysis, (i) update any geometries which have been changed as a result of the analysis, and (j) resume all previously suppressed features on the updated design.

This project is consistent with the Intelligent Synthesis Environment (ISE) initiative (http://www.ise.nasa.gov/) which NASA has recently set in motion. ISE is an Agency objective which seeks to place NASA operations on the leading edge of technology. This effort includes automation of many manual processes, interactive-collaborative design through manufacturing efforts, hologram visualization of designs, and automatic assessment and modification analysis based on changes in requirements and/or design.

The Johnson Space Center (JSC), specifically, has focused on the areas of design-through-delivery of hardware, including data mining and repository issues. The goals of design-through-delivery for JSC are to define the tools required for the design through manufacturing process as well as automate the interaction among these tools.

The project described herein will play a vitally important part in this process by providing a seamless link between the design and analysis processes.

BACKGROUND

This project examines one aspect of the design-through-manufacturing process, that is, the process by which computer aided design (CAD) models are translated into finite element analysis (FEA) models. Ideally, this process should be an automatic and parameterized two-way street: After the part has been designed on the computer it is moved to the FEA program for analysis, it is analyzed, optimized in some way, and then seamlessly moved back to the CAD program which sends it to the numerical control (NC) program for manufacture (see Figure 1).

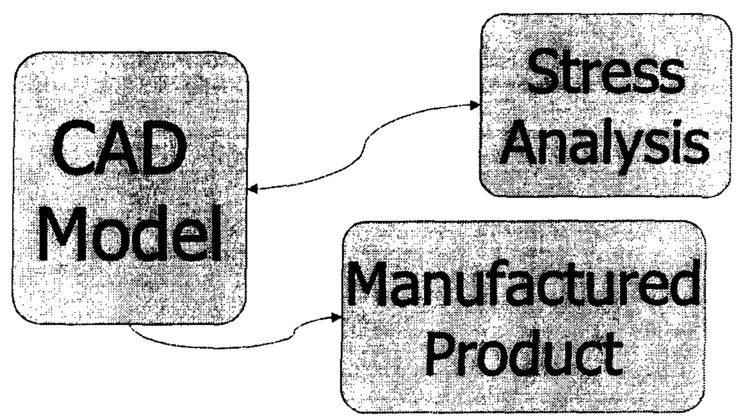

Figure 1. The Idealized Design-Through-Manufacturing Model

While the programs which translate from the CAD model to the NC model, in fact, are automatic and simple, that is not the case for the interface between the CAD model and the FEA stress analysis model. This process is extremely software and model dependent. We shall examine some of the tools that are used for both CAD and FEA at NASA/Johnson Space Center (JSC) and describe the development of a program for an Automatic CAD to FEA Interface (ACFI).

Figure 2 summarizes the most common design/analysis processes used at JSC on parts and assemblies for the International Space Station, the X38, and other systems. The design tool used most widely at JSC is PTC's ProEngineer; this is a state-of-the-art CAD package which is used by the major aerospace companies worldwide. Stress analysis packages consist of a combined pre- and post-processor, and a processing program. The packages most widely available at JSC are (pre- and post-processor/processor):

- Mechanica/Mechanica
- PATRAN/NASTRAN
- I-DEAS/NASTRAN
- ProMesh/NASTRAN

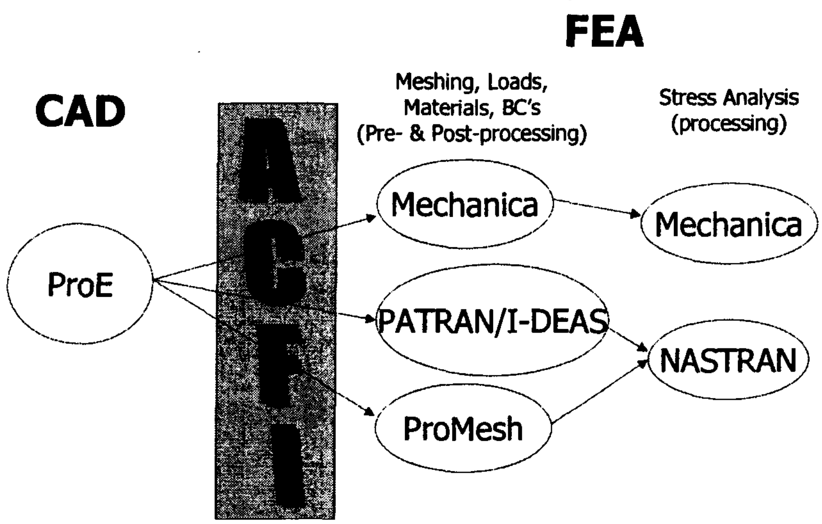

Figure 2. Tools for the CAD-to-FEA Process at NASA/JSC

There are a few other programs in use (e.g. Stress-Check), but the list above covers the major packages; in addition, it is hoped that whatever interface tools are developed will be extended to work with other platforms as well.

The processing programs Mechanica and NASTRAN are theoretically quite different. Mechanica is a P-version FEA code, and NASTRAN is an H-version code; the basic difference between them lies in the way the analysis elements are formulated. But there are other differences as well. NASTRAN is a state-of-the-art code which includes sophisticated material capabilities and advanced elements, loading and constraints which permit treatment of much more sophisticated problems than Mechanica. It is felt that the initial implementation of a CAD/FEA interface will be from ProE to PATRAN and I-DEAS, although there is another possibility which includes the use of ProMesh, a preprocessing code which comes with ProE and is meant to be used as an interface to other preprocessors and processors. ProMesh has certain capabilities that PATRAN and I-DEAS presently lack (although they are under development), namely the ability to extract midsurface planes from thin features, and beam axes from long, thin features in CAD models. This is an enhancement that can be used together with PATRAN/I-DEAS to simplify the interfacing process (see Figure 3).

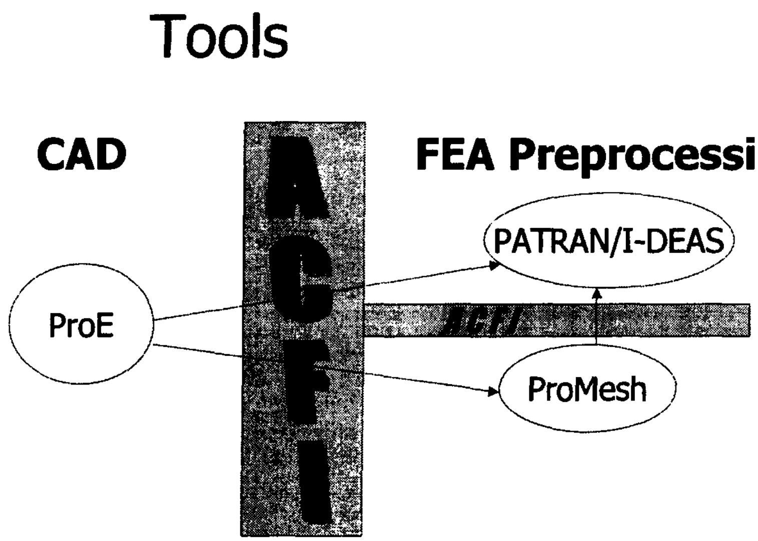

Figure 3. ProMesh and PATRAN/I-DEAS Used in Combination

What are the problems that the analyst faces when importing a CAD model? We will look at several examples. The first is a longeron from the X38 (See Figure 4). If this part were imported from ProE, without first making any changes to it, using the NASTRAN automatic import facility, it would be imported as over 150 geometric surfaces. These surfaces would need to be altered dramatically by the analyst in order to obtain an FEA mesh. It has been found that removing some of the ProE features will yield a geometry that will be imported directly into PATRAN as a single solid. Figure 5 shows the longeron with all the fillets, holes and bosses removed. Removal of certain features (e.g. fillets) will increase the stresses in a part, so that the results of the FEA analysis will be conservative. Removal of other parts (e.g. holes) will result in an analysis showing lower stresses than the actual case; this must somehow be taken into account so that the final design will have sufficient strength. When the simplified model is exported to PATRAN, it consists of a single solid which can be easily meshed for FEA; however, using PATRAN automatic meshing with this model will produce an inferior mesh with many solid elements, not the appropriate type of element for this type of shell structure. The best solution, in this case, is to first use ProMesh to create midsurface planes which represent the shell nature of this part, and then extract them (in this case 9 planes) to PATRAN. In PATRAN the planes can be quickly meshed using shell elements, yielding a very accurate mesh with a minimal number of excellent elements. Once proper loading, boundary conditions, and material behavior are inserted into the preprocessor, the finite element analysis can be performed yielding accurate results using a minimum amount of computer time.

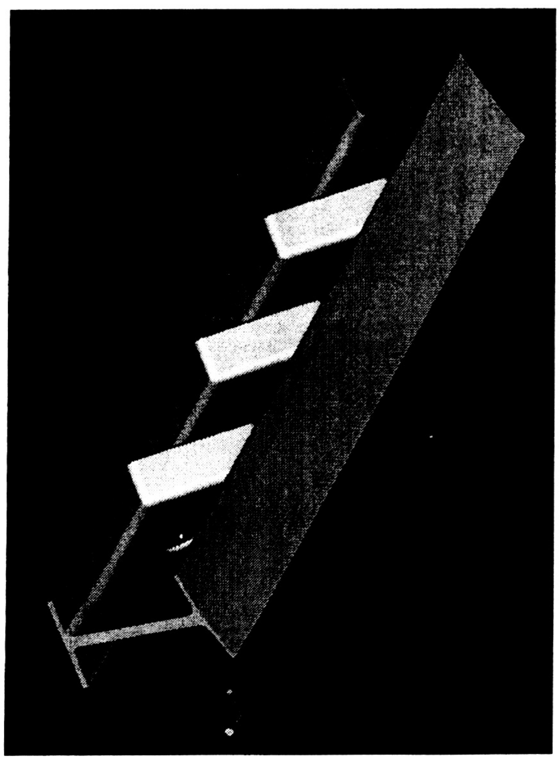

Figure 4. Longeron Part from the X38

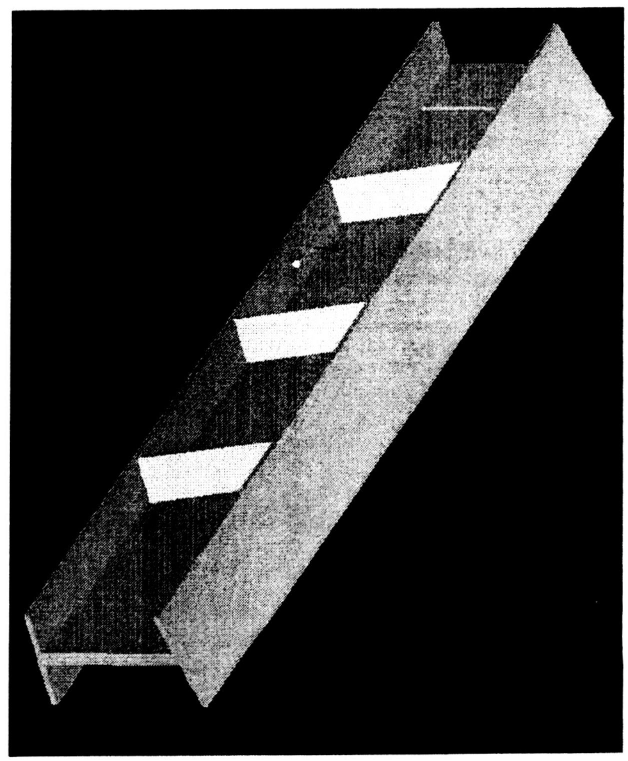

Figure 5. Longeron with Features Removed

Another example is a waffle part from the skid bracket on the X38 (see Figure 6). This part has also been simplified with fillets, rounds and holes removed (see Figure 7). Once the features have been removed, all thin features are replaced by midsurfaces using ProMesh, the results are exported to PATRAN and then the end bars are modeled in PATRAN ultimately resulting in an efficient and accurate mesh, which can be rapidly created, containing a combination of shell and beam elements.

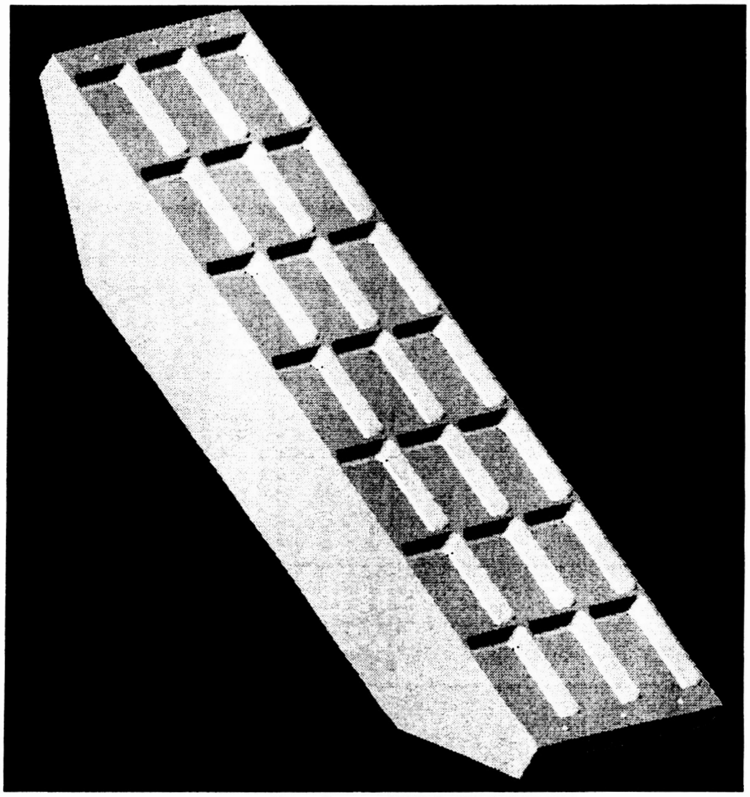

Figure 6. Waffle from X38 Skid Bracket

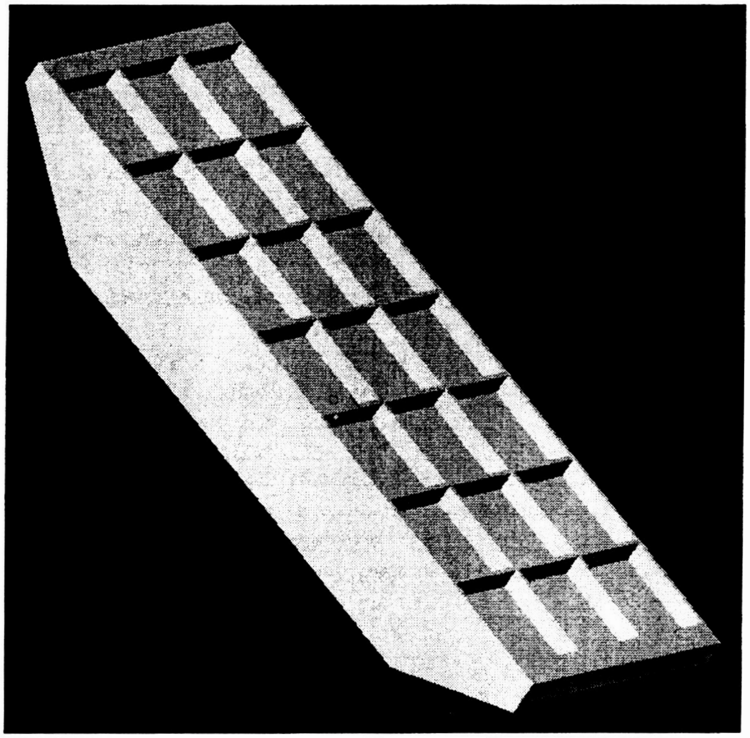

Figure 7. Waffle with Features Removed

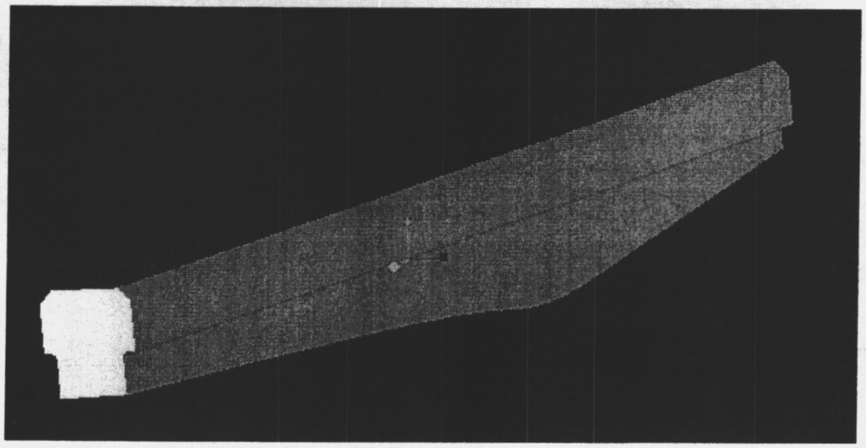

Figure 8. Bulky X38 Part

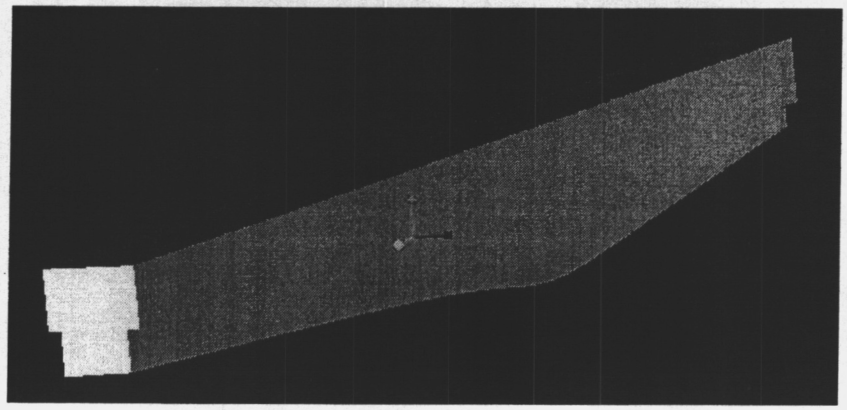

Figure 9. Bulky Part with Features Removed

In general, parts that are bulky, rather than thin, are easier to translate from the CAD program for FEA meshing. However, an automatic interface would help even with these. Figures 8 and 9 show a bulky X38 part fully featured, and with chamfers and rounds removed. When translated directly to PATRAN, the part with features removed will be easier to mesh with better shaped elements, and the resulting stress analysis will be conservative.

The ACFI program, developed at NASA/JSC this summer, is based on the concept that automating the process of translation from CAD model to FEA model will result in tremendous improvements in the design-through-manufacturing process at NASA/JSC. It will increase analyst productivity; at the present time it is not uncommon for analysts to spend days or weeks modifying the solid model to prepare it for analysis, and they often just use the dimensions from the CAD model to rebuild the

FEA model from scratch. ACFI, when fully implemented, will also enable analysts to rapidly transfer design improvements back to the original model for manufacturing. Smoother interfacing between programs will also enable designers and analysts to concentrate their efforts on what they do best, designing and analyzing, respectively.

ACFI is based on the current capabilities of the common CAD and FEA programs presently in use at JSC. The resources used in the above examples are shown in Table 1. These tools are not automatic, but will be automated as part of the proposed program. Additional development tools available within the software packages are listed in Table 2. It is expected that these tools will provide the resources for developing the capabilities described in the program.

Table 1. Existing Software Features

ProE	ProMesh	PATRAN/ I-DEAS
Manual feature suppression	Manual midsurface, beam extraction	Automatic geometry interpretation

Table 2. Development Tools within Existing Software

ProE	ProMesh	PATRAN/ I-DEAS
▪ Mapkey ▪ ProProgram ▪ J-Link ▪ Pro/TOOLKIT ▪ Config.pro, Win.pro ▪ UDF ▪ IGES and PATRAN export	▪ Mapkey ▪ Pro/TOOLKIT ▪ IGES export	▪ IGES & PATRAN import ▪ PCL - PATRAN Control Language

PROGRAM FOR THE DEVELOPMENT OF AN AUTOMATIC CAD/FEA INTERFACE

Task 1 - Establishment of program at Vanderbilt

- Installation of computers, software
- Training of graduate/undergraduate students, thorough investigation of software
- package capabilities: ProE, ProMesh, PATRAN/NASTRAN, I-DEAS, Mechanica

Task 2 - Collect information on typically removed features from CAD parts

Task 3 - Development of strategy for treatment (removal/extraction and export) of each class of feature prior from CAD software

- Thin features - midsurface extraction
- Solid 3-D features – export
- Fillets, rounds, chamfers - possible removal
- Holes - possible removal
- Beam type features - midline/cross-section extraction
- Two-force features - midline extraction
- Other features (springs, contact, etc.)

Task 4 - Development of routine for feature evaluation

- Incorporation of on-the-fly FEA analysis of individual features
- Evaluation and assessment of feature importance
- Incorporation of stress concentration factors of removed features for inclusion with analysis results

Task 5 - Development of automatic capability for CAD geometry export and FEA import

- Automatically remove features which do not impact design
  fillets, rounds, holes
- Automatically evaluate importance (via stress concentration factor, SCF) of features which do impact design, but should be removed for analysis
- Automatically replace removed features which impact design with their appropriate SCF's
- Automatically export revised CAD geometry
  thin, solid features
  beam type, two-force, others
- Automatically import revised CAD geometry to FEA program

- Automatically create report describing alterations in the model

Task 6 - Implement model export so that loading, constraints, and material properties automatically translate across the interface and are permanently attached to the model

Task 7 - Address data base management issues

- Develop method for management of part versions with removed/altered features
- Develop method for maintenance of revision status for original CAD models

Task 8 – Development of routine for automatic specification of element type required for each feature

Task 9 - Parameterization of Interface

- Development of strategy for parameterization of geometry by interface
- Implementation of automatic parameterization

Task 10 - Web implementation of interface

Task 11 - Extension of interface to include assemblies

Task 12 - Development of platform-independent interface

Task 13 - Expansion of interface to include CFD and thermal analyses

The total time required for completion of the program will depend on the level of funding. It will be performed at Vanderbilt University by Dr. Carol Rubin and graduate/undergraduate students maintaining close contact with EM and ES Design and Analysis colleagues at NASA/JSC.

SUMMARY OF SUMMER, 2000 ACTIVITIES

Consistent with the tasks listed above, the work performed at NASA/JSC this summer included:

- Investigation of the capabilities of most of the CAD and FEA tools used at JSC
- Establishment of the **ACFI** program requirements with advice from EM & ES designers and analysts

- Creation of a detailed development plan for **ACFI**
- Initial development of specific tools for **ACFI**
- Collection of parts which will be useful for testing **ACFI** during development
- Preparation of a Director's Research Grant proposal

ACKNOWLEDGMENTS

I would like to thank the EM3 Branch for their help and encouragement this summer; especially Raymond Aronoff my project colleague, Hector Saenz who provided prompt and efficient IT support, and Carolyn Krumrey EM3 Branch Chief.

I would also like to thank the following people who shared ideas with me during my work at JSC, and have expressed their interest in seeing this project through to completion. I expect to maintain extensive continued contact with them:

Steve Caperton
John Edgecombe
Brent Evernden
Chris Hansen
Chris Lupo
Galen Overstreet
Brandan Robertson
James Smith
Ted Tsai
Dave Wade