Getting FEA Into the Design Process: Rapid Analysis of Tubes

Raymond N. Frick, Ph.D., P.E.

Nacelle Components
Pratt & Whitney
400 Main St., M/S 161–15
East Hartford, CT 06108
(203) 565–0717

Abstract

This paper shows how Pratt & Whitney increased engineering productivity by incorporating finite element analysis (FEA) with MSC/NASTRAN into the design cycle. In particular, Pratt & Whitney has developed a fast, efficient process to analyze external tubes of jet engines called the Preliminary Tube Analysis (PTA) System. This process was originally intended to be a preliminary analysis prior to a more formal analysis. However, it has been proven to be very accurate and is usually the only analysis. This process uses a commercial CAD program, MSC/NASTRAN, and custom pre and post processors. This process enables all tubes for new designs to be analyzed before manufacture and testing. While relating some of the details of the custom programs, the emphasis of this paper will be on the process development; i.e. how finite element analysis can be put to effective use within the design environment. This required understanding the design process, creating custom programs to interface with commercial codes, and applying engineering judgment.

Introduction

Pratt & Whitney has developed a fast, efficient process to analyze external tubes of jet engines called the Preliminary Tube Analysis (PTA) System. This process was originally intended to be a preliminary analysis prior to a more formal analysis. However, it has been proven to be very accurate and is usually the only analysis. This process uses a commercial CAD program, MSC/NASTRAN, and custom pre and post processors. This process enables all tubes for new designs to be analyzed before manufacture and testing. While relating some of the details of the custom programs, the emphasis of this paper will be on the process development; i.e. how finite element analysis can be put to effective use within the design environment. The major components of this workstation based system are:

1. EDS UNIGRAPHICS®: This is the CAD program used by the designers.

2. Tube Analysis Pre-Processor (TUBA): This Pratt & Whitney pre-processor creates a run-ready Version 68 MSC/NASTRAN input file for static, normal modes, bracket optimization, and frequency response analyses.

3. MSC/NASTRAN

4. PTA Post Processor: This custom program reads the resulting OUTPUT2 results file and with the input file creates a summary of the analysis.

5. MSC/PATRAN3: This commercial CAE program is used for more detailed pre and post processing.

Problem Definition

With the popularity of large twin engine aircraft and the corresponding emphasis on engine dependability for ETOPS (Extended Twin OPerationS) approval, Pratt & Whitney has committed to reducing the In-Flight Shut-Down (IFSD) rate to zero. A large cause of IFSD's are failure of external tubes. The failure of a relatively low cost tube effects fleet IFSD rates as severely as a major component failure. Thus, Pratt & Whitney made a commitment to improve its external tube design process in order to reduce the IFSD rate.

In 1989, the P&W Structures Group, which was responsible for the analysis of tubes after field problems occurred, decided to upgrade the tube analysis system to analyze tubes during the design process. Prior to this, tubes were designed, manufactured, and then vibration tested. It was expensive to redesign tubes that failed either the testing or in the field. A team was formed to improve the process. It was decided to use MSC/NASTRAN as the finite analysis program, an old internal tube pre-processor (TUBA), and PATRAN as a post processor. TUBA was substantially upgraded to satisfy the (changing) needs of the designers. A custom built translator was written to go from tube definition files to TUBA while allowing necessary additional information required for analysis to be created. In three years, a system was developed which allowed the designer to accurately analyze tubes. It consisted of:

1. *UNIGRAPHICS®:* This is the CAD program from EDS which is the primary designer tool. First the tube was designed and routed with this program. Then a tube definition file was created. This would be done for every tube in the system.

2. *Tube Definition File Translator:* The designer next used this program to take the geometric definition of the tube, add required analysis information like temperatures and bracket stiffness (which are not known in the initial tube design), and create a TUBA input file.

3. *Tube Analysis Pre-Processor (TUBA):* The designer would then run this program to create the MSC/NASTRAN input file for either a normal modes or static analysis. A beam or shell model could be created.

4. *MSC/NASTRAN:* The designer would then run the finite element analysis. He would read the print file to get the modal frequencies or error messages if he was unlucky.

5. *PATRAN:* This was used for graphical post-processing the results (mode shapes and stresses).

Unfortunately, the above system had very little influence on the design cycle. While the Structures engineer could do a much more rapid analysis after tube failure, the system was too cumbersome for the designer. It required the designer to go outside his primary design tool, UNIGRAPHICS®, and use and learn four other programs. There was no communication between UNIGRAPHICS® and the analytical process. Analysis results were uneven due to assumptions made about unknowns like bracket stiffness. However, this system did provide good analyses while identifying the primary tube design requirement.

Solution

Tubes must satisfy a minimum frequency requirement so that vibration does not occur in operation. (Static loads cause few failures.) Once the tube has been routed, the major influence on the frequency is the bracket location. Any reasonably stiff bracket placed at the maximum modal displacement location will effectively increase the modal frequency. Following this logic, reasonable stiffness can be used at the bracket location to properly design the tube before the bracket is designed. Bracket stiffness cross-coupling terms can be ignored.

The goal became to design a tube analysis system in which the analysis tools are integrated into the CAD program. The system would show the location of the maximum modal displacements in the CAD program to allow the designer to easily locate a bracket in the most effective location. A new team formed in 1992 accomplished this goal by:

1. *Eliminating the tube definition file translator.* Custom UNIGRAPHICS® routines were written to add the necessary information required for a finite element analysis. Since the designer is familiar with the format of the interface, minimal training was required.

2. *Making reasonable assumptions.* Bracket stiffness, clip stiffness, boundary conditions, finite element model type (beam or shell), etc. are made in the preliminary design phase. As the design becomes finalized, the proper data is substituted.

3. *Structural analysis programs put into a black box.* The designer initiates the analysis within UNIGRAPHICS®. The MSC/NASTRAN input file is created, submitted, and post-processed without user intervention.

4. *Post processing is done within UNIGRAPHICS®.* The results are graphically displayed within UNIGRAPHICS®. This enables the designer to locate brackets where required to increase frequency, increase stiffnesses, or otherwise modify the design. A special post processor was created to do this. Beam models with CBEND elements require special techniques to get pressure and ovalization stress (Ref. 1). A short summary is provided for documentation.

The designer begins the tube design within UNIGRAPHICS®. When the tube is routed and initial bracket locations are selected, the preliminary tube analysis (PTA) system will create a TUBA input file with assumptions made for any unknown data such as bracket spring rates. The PTA system will run TUBA to create a MSC/NASTRAN input file and then submit the MSC/NASTRAN job without user intervention. Messages will be displayed to a

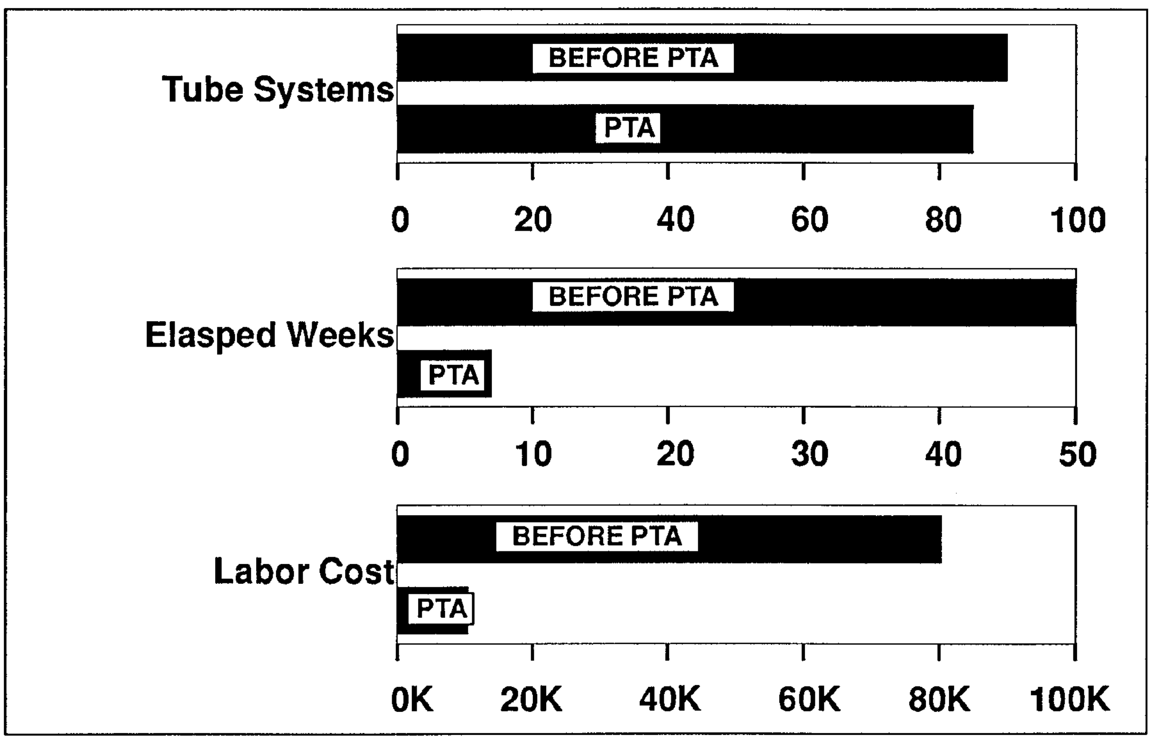

FIGURE 1: EFFECT OF THE PTA SYSTEM

separate window in case errors occur. After the analysis, the PTA post-processor will create a summary file of the results. A UNIGRAPHICS® UFUNC program plots the analysis results on the screen. The designer can now add and modify bracket locations and stiffness, as well as other variables, and repeat the cycle until the tube meets design requirements. As brackets become designed, the actual bracket stiffness is used.

The result is that tubes are now analyzed as part of the design process. Figure 1 shows how the PTA system improved productivity in the design of tubes in two engine designs. Using the original system. specialists took 50 weeks to analyze 90 tube systems. Using the new system, designers took eight weeks to analyze 84 tube systems. Table 1 compares the two systems.

	BEFORE PTA	PTA
USER	SPECIALISTS	DESIGNERS
TRAINING REQ'D	HIGH	MINIMAL
USER INTERFACE	5 PROGRAMS	1 PROGRAM
ANALYSIS TIME PER SYSTEM	16 HOURS	2 HOURS
LABOR COST PER SYSTEM	$946	$118
QUALITY	INCONSISTENT (POOR TO GOOD)	CONSISTENT (GOOD)
REWORK	EXTENSIVE	MINIMAL
CUSTOMER SATISFACTION	☹	☺

TABLE 1: PTA BENEFITS

Discussion

Very brief descriptions of the programs which create and interpret MSC/NASTRAN input and output follow. This will give the user an idea of the amount and type of information the designer sees. A short discussion on how bracket models are created follows.

Tube Analysis Finite Element Pre Processor (TUBA)

TUBA is the finite element preprocessor which converts a geometric description of the tube into a MSC/NASTRAN finite element model. Clamps, consisting of clips and brackets, are simulated with spring rates. (Clips connect the tube to the bracket.) TUBA outputs MSC/NASTRAN input files for both static, natural frequency, frequency response, and bracket stiffness optimization analyses. For circular tubes under .75 inch diameter, the user may choose beam type elements (CBEND's) to model the tube. For larger tubes, or tubes

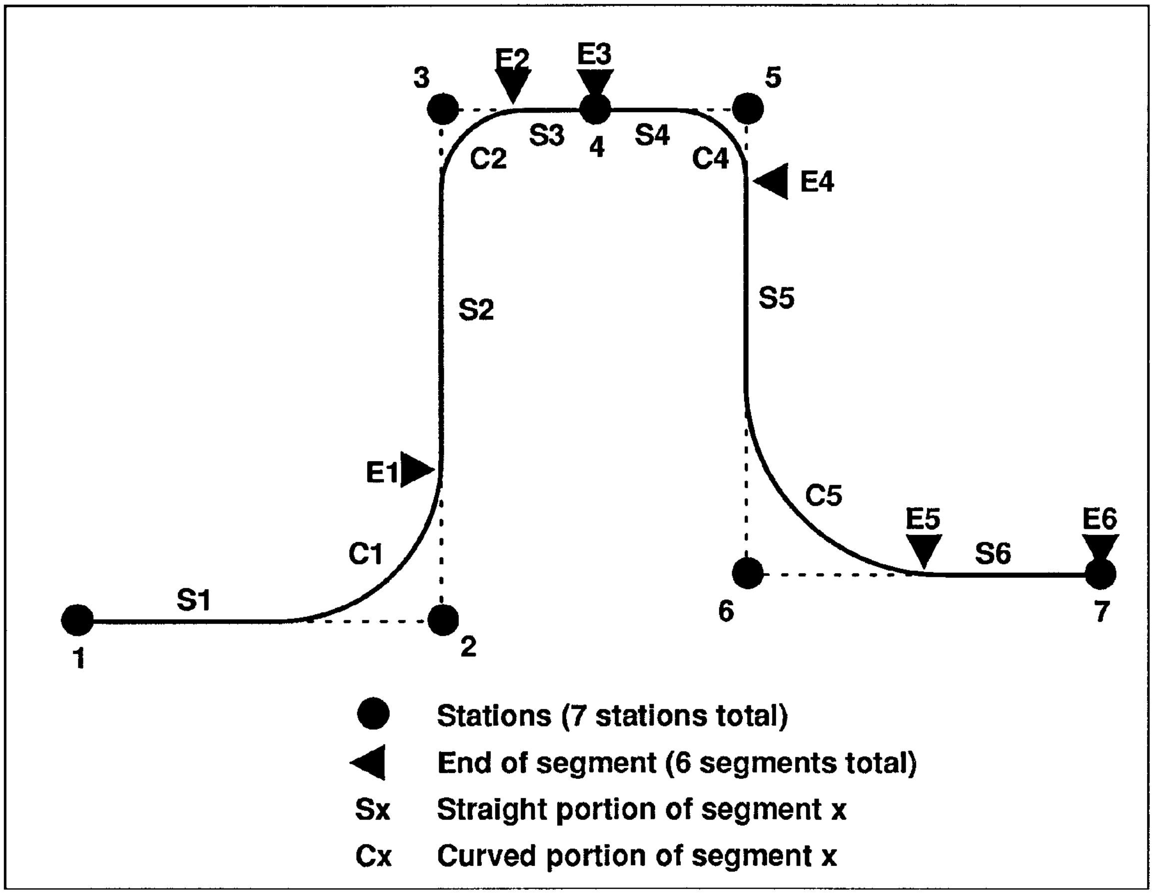

FIGURE 2: SCHEMATIC OF TUBE DESCRIPTION FOR TUBA

with elliptical cross sections, the tube is modeled with plate elements (CQUAD4's). This diametral limit is increased to 1.5 inches for frequency response and bracket optimization analyses. While uniform thickness circular tubes are generally modeled, TUBA will properly model tapered thickness tubes and tubes with elliptical cross sections. Temperature dependent material properties are obtained from a database. Static loads may include thermal loads, internal pressure, and end condition and bracket displacements (case thermal growth). Concentrated masses may be added to model the effect of valves. Also, the effect of the fluid mass within the tube may be modeled.

Figure 2 is a schematic of a tube showing the required TUBA input requirements. For the tube in the figure, seven stations are required to define the tube which results in six tube segments. Notice that stations **2**, **3**, **5**, and **6** are located at the intersection of the tangents to the curve ends. Each segment is divided into a straight portion first (identified with a **S**), and then a curved portion (identified with a **C**). Note that station **4** is not required to describe the tube geometry, but is defined to locate a clamp. Clamps are always located at the beginning of a segment. For example, if a clamp was defined at station **2**, it would be

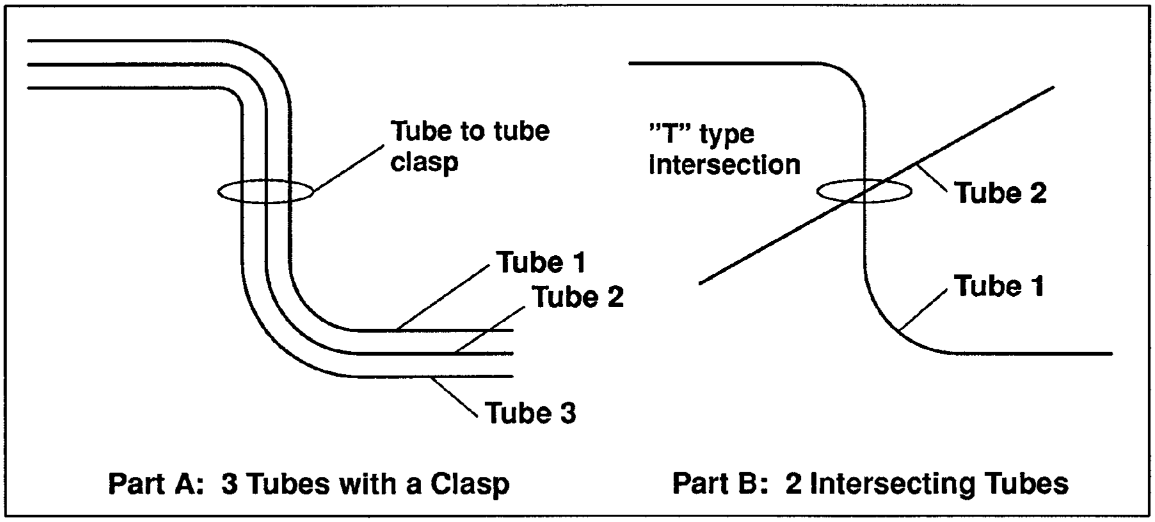

FIGURE 3: MULTIPLE TUBES WITH INTERSECTIONS

located at the beginning of segment **2**, or **E1**. Therefore, the only way to define a clamp midway between stations **3** and **5** is to define station **4**.

The tube in Figure 2 would be described with one *branch.* TUBA allows up to 50 branches in order to model a series of tubes joined with clasps, or tubes joined with T type intersections. For example, in Figure 3 Part A, three branches would be used with the proper intersection input to describe the tube to tube clasps. Two branches with a T type intersection card would be used to describe the tubes in Figure 3 Part B. Rigid body elements are used to define these intersections. Intersecting tubes are not geometrically modeled.

Preliminary Tube Analysis Post Processor

The PTA post-processor produces a summary file which is used by UNIGRAPHICS® for post-processing. This is the key link in the system as this provides feedback to the designer while he is still in the CAD program so that changes can be made. It also provides an analysis summary which is used as documentation. This file contains:

1. *Heading:* The heading contains the date of analysis, date the summary file was created, the analyst name, and the title and subtitle of the analysis.

2. *Displacement Summary:* This section contains the location, value, and direction of the maximum location for each subcase or mode. The modal frequency is included for a vibration analysis. For a vibration analysis, displacements are normalized to 1 inch. This data is plotted in UNIGRAPHICS®.

3. *Maximum Stress Summary:* This optional table presents the maximum Von Mises, maximum principal, minimum principal, maximum axial, and maximum hoop stress in the tube system for each mode or subcase. In all the stress tables, for a vibration analysis, stress values are normalized to either a 1 inch displacement or via

the "Q" factor. (The "Q" factor is an experimentally derived value to scale the normalized modal stresses to predict actual vibratory stress.)

4. *Tube Stress Summary:* The maximum Von Mises stress and location is given for each tube of the system for each subcase or mode.

5. *Fitting Stress Summary:* The Von Mises, maximum principal, and axial stresses are given for each fitting and branch end. The location of the fitting or end is also given.

6. *Model Summary:* The clamp type and stiffness as well as the bracket part number and stiffness are shown at each clamp point. The fitting weights and branch materials and titles are also listed. The PTA group comments which contain the pressure are listed at the end.

Figure 4 shows a summary file for a vibratory analysis in which the maximum stress summary was requested. In this case, four modes were determined. The small *e* after the location number in the branch stress summary table means that the location number is an shell element. This means that this summary is for a shell model. Note that the "Q" factor calculation was also done. Figure 5 shows a summary file for a static analysis in which the fitting stress was requested. Instead of modes, the subcases are each summarized. Note that subcase 1, the pressure loading, is missing. This means that this summary is for a beam model since there is no pressure loading for a beam. The small *n* after the location number in the branch stress summary table means that the location number is a beam node.

The results from the PTA summary file, as shown in Figure 4, can be plotted within UNIGRAPHICS®. The summary file is required for plotting. Figure 6 shows an example of such a plot. (This is not the tube system in the previous examples.) For each mode in the summary file, a vector at the location of maximum deflection is displayed pointing in the direction of maximum deflection. The corresponding frequency is displayed near the head.

Rapid Bracket Analysis

When the bracket is finally designed, a Rapid Bracket Analysis (RBA) program creates a MSC/NASTRAN data deck by using custom UNIGRAPHICS® and MSC/PATRAN3 routines. These routines will fold a bracket pattern and provide ties at gusset welds. A relatively crude model will give sufficiently accurate stiffness values. The bracket can be included in the tube analysis by using the INCLUDE statement with TUBA connecting the tube to a bracket grid. Otherwise, a series of static load cases are run to determine the bracket springrates and to create a GENEL element. Either can be used to input the complete bracket stiffness without the added cost of the full bracket model.

Preliminary Tube Analysis — Date of analysis: 7/20/93
Analyst: Ray Frick — Time: Tue Sep 7 17:04:23 EDT 1993

PTA TEST CASE 4 - SHELL MODEL WITH INTERSECTIONS

Mode	Freq.	Grid	Locations of Max. Deflection X	Y	Z	Dir. of Max. Deflection X	Y	Z
1	22.	121	0.000	5.857	10.000	1.000	0.000	0.000
2	25.	121	0.000	5.857	10.000	0.000	1.000	-0.966
3	25.	101	0.000	3.000	10.000	1.000	0.000	0.000
4	31.	141	0.000	7.354	8.409	-1.000	0.000	0.000

*** Stress is adjusted with a Q factor of 35. for the x190 ***

Mode	Stress Summary Von Mises	Max. Prin.	Min. Prin.	Max. Axial	Max. Hoop
1	0.122E+08	0.934E+07	-0.934E+07	0.929E+07	0.895E+07
2	0.162E+08	0.182E+08	-0.179E+08	0.182E+08	0.126E+08
3	0.145E+08	0.155E+08	-0.155E+08	0.154E+08	0.136E+08
4	0.295E+08	0.334E+08	-0.334E+08	0.334E+08	0.222E+08

Maximum Von Mises Stress in Branch

Mode	Tube	Loc.	Von Mises	X	Y	Z
1	1	156e	0.122E+08	-0.520	7.216	7.030
	2	280e	0.338E+05	0.000	3.000	5.736
2	1	141e	0.162E+08	0.736	7.354	8.409
	2	283e	0.546E+05	0.521	3.000	4.480
3	1	142e	0.145E+08	0.520	6.877	8.438
	2	300e	0.429E+05	0.000	5.000	5.736
4	1	142e	0.295E+08	0.520	6.877	8.438
	2	290e	0.123E+06	0.000	4.000	5.736

Tube Bracket Num. Station	Clamp Type Bracket #	Spring Stiffness Kx	Ky	Kz	Krx	Kry	Krz
2 1-a	ST1435-10	3000.0	1400.0	1000.0			
		1000.0	1000.0				

Fitting Weights:

Int./Stat.	Weight
bend2	0.3330
1p01-1p02	0.3450

Material Data:

Tube	Material	Tube Title
1.	pwa1010	tube num 1
2.	pwa1010	tube num 2

PTA Group Comments:
Pressure - 200.000
Circum Nodes- 8
Fluid Mass - Fuel
PTA Test case 4
Shell models with intersections.

FIGURE 4: PTA SUMMARY FILE WITH MAXIMUM STRESS

```
Preliminary Tube Analysis                               Date of analysis:  5/10/93
Analyst:  Ray Frick                             Time: Tue Sep 7 16:54:28 EDT 1993

90 DEGREE BEND .75 DIAMETER, .028 THICK, 10 INCH TUBE.
TEST CASE FOR MANUAL
--------------------------------------------------------------------------------

Subcase   Disp.   Grid    Locations of Max. Deflection    Dir. of Max. Deflection
                             X         Y         Z           X       Y       Z
    2  0.021869    21      0.000     1.974     9.819       0.000   0.258   1.000
    3  0.010350    53      0.000     2.286     5.000       1.000   0.000   0.000
    4  0.023543    20      0.000     1.500     9.598       0.407   0.179   1.000
--------------------------------------------------------------------------------

     Subcase                               Stress Summary
                Von Mises    Max. Prin.   Min. Prin.    Max. Axial     Max. Hoop
   Temp.    2   0.199E+06     0.931E+04   -0.198E+06     0.499E+04     0.931E+04
   Displ.   3   0.519E+05     0.529E+05   -0.505E+05     0.529E+05     0.937E+04
   Total    4   0.210E+06     0.125E+05   -0.208E+06     0.875E+04     0.120E+05
--------------------------------------------------------------------------------

                         Maximum Von Mises Stress in Branch
     Subcase   Tube    Loc.    Von Mises         X          Y          Z
   Temp.    2    1       9n    0.137E+05       0.000      0.000      4.444
                 2      43n    0.199E+06       0.000     -3.000      5.000
   Displ.   3    1       1n    0.989E+04       0.000      0.000      0.000
                 2      58n    0.519E+05       0.000      5.000      5.000
   Total    4    1       1n    0.172E+05       0.000      0.000      0.000
                 2      58n    0.210E+06       0.000      5.000      5.000
--------------------------------------------------------------------------------

                Locations of Fitting Stress              Stress Summary
Fitting    Tube     X         Y         Z        Von Mises  Max. Prin.  Max. Axial
**Subcase   2 (Temperature):
Beginning    1     0.000     0.000     0.000    0.107E+05    0.258E+04  -0.263E+04
      num1   1     0.000     0.000     5.000    0.940E+04    0.334E+04   0.285E+04
 1p01-1p02   1     0.000     4.000     5.000    0.260E+04    0.258E+04   0.218E+04
Beginning    2     0.000    -3.000     5.000    0.199E+06    0.258E+04  -0.104E+06
      num1   2     0.000     0.000     5.000    0.187E+06    0.258E+04  -0.116E+06
 1p01-1p02   2     0.000     4.000     5.000    0.166E+06    0.258E+04  -0.136E+06
Ending       2     0.000     5.000     5.000    0.178E+06    0.258E+04  -0.125E+06
**Subcase   3 (Displacement):
Beginning    1     0.000     0.000     0.000    0.989E+04    0.106E+05   0.102E+05
      num1   1     0.000     0.000     5.000    0.541E+04    0.575E+04   0.436E+04
 1p01-1p02   1     0.000     4.000     5.000    0.380E+04    0.414E+04   0.407E+04
Beginning    2     0.000    -3.000     5.000    0.184E+05    0.193E+05   0.192E+05
      num1   2     0.000     0.000     5.000    0.478E+04    0.529E+04   0.498E+04
 1p01-1p02   2     0.000     4.000     5.000    0.404E+05    0.414E+05   0.414E+05
Ending       2     0.000     5.000     5.000    0.519E+05    0.529E+05   0.529E+05
**Subcase   4 (Total Loads):
Beginning    1     0.000     0.000     0.000    0.172E+05    0.497E+04   0.372E+04
      num1   1     0.000     0.000     5.000    0.111E+05    0.516E+04   0.375E+04
 1p01-1p02   1     0.000     4.000     5.000    0.396E+04    0.434E+04   0.428E+04
Beginning    2     0.000    -3.000     5.000    0.202E+06    0.259E+04  -0.101E+06
      num1   2     0.000     0.000     5.000    0.187E+06    0.259E+04  -0.115E+06
 1p01-1p02   2     0.000     4.000     5.000    0.194E+06    0.259E+04  -0.108E+06
Ending       2     0.000     5.000     5.000    0.210E+06    0.260E+04  -0.927E+05
--------------------------------------------------------------------------------
Tube Bracket    Clamp Type                       Spring Stiffness
Num. Station    Bracket #      Kx        Ky        Kz        Krx       Kry       Krz
  2 1-a         ST1435-10    3000.0    1400.0    1000.0
                             1000.0    1000.0
--------------------------------------------------------------------------------
         Fitting Weights:            |           Material Data:
     Int./Stat.          Weight      | Tube   Material       Tube Title
              4          0.3330      |  1.   pwa1010    tube num 1
      1p01-1p02      0.3450          |  2.   pwa1010    tube num 2
--------------------------------------------------------------------------------

  PTA Group Comments:
 Pressure      -     500.000
 Temperature -    1000.000
 Circum Nodes-           6
 Fluid Mass   - Air
```

FIGURE 5: PTA SUMMARY FILE WITH FITTING STRESS

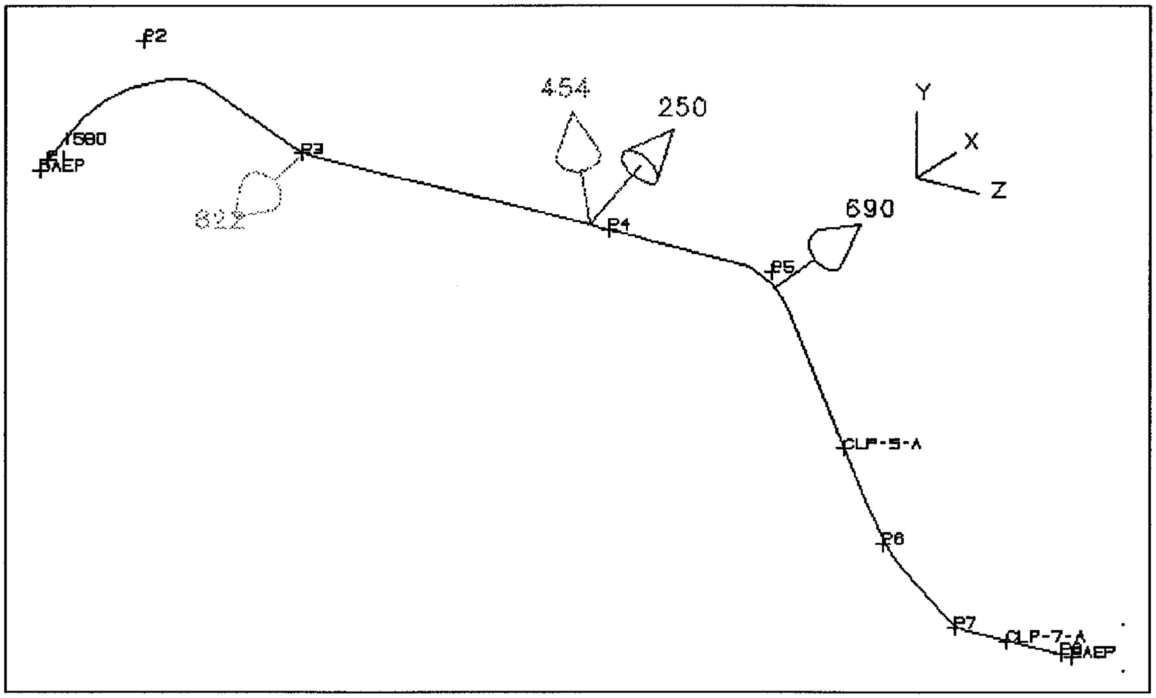

FIGURE 6: UNIGRAPHICS PLOT OF A PTA SUMMARY FILE

Conclusions

The PTA system has brought the power of MSC/NASTRAN finite element analysis into the hands of the designers. The PTA system has reduced the analysis time for a tube system from 16 hours to 2 hours. The tube analysis is now being done by designers during the design phase instead of by finite element specialists after hardware procurement. The successful implementation of this system has substantially increased the number of tube systems passing proof testing. The net result is a substantial increase in designer productivity, reduced rework costs, and a more reliable product.

The success of this effort was due to the following:

1. *Understanding the process:* The initial system was designed to fully analyze tubes and was based upon information and requests from the designers. It was slow and cumbersome and did not effect the design. Only after understanding the design process and fitting the system into it did it become successful.

2. *Getting a champion:* One designer decided to make it his responsibility to get the analysis into the design process. He defined the system requirements.

3. *Using a team:* The cross-functional team had members from Structures, Structures Technology, Design, CAD programming, Scientific Programming, and

Design Methods. Recognition of the need came from the team. The team convinced management to provide resources, established required intergroup communications, and provided user support and training.

4. *Selling management:* Management must buy into the objectives in order to provide the necessary resources.

5. *Avoiding over–design:* Proper engineering assumptions were made to provide a rapid, reliable, and sufficiently accurate analysis. This enables the designer to begin analyzing before the system is completely designed. This is where the expert is required to "engineer" the process.

6. *Providing the right tools:* The use of MSC/NASTRAN, UNIGRAPHICS®, and very well tested internal programs provide the robustness and quality demanded by the users.

As a result of this successful implementation, more powerful features of MSC/NASTRAN are being introduced. For tubes with frequencies which may be excited, a frequency response analysis is being developed. Since a stiffer bracket usually weighs more, to reduce engine weight MSC/NASTRAN will be used to optimize the bracket stiffnesses.

This paper showed how Pratt & Whitney increased engineering productivity by incorporating finite element analysis using MSC/NASTRAN into the design cycle. This required understanding the design process, creating custom programs to interface with commercial codes, and applying engineering judgment.

Acknowledgements

Michael O'Brien was the designer who championed the PTA system and provided Figure 6. Raymond Matheis was the key Structures support engineer. Both graciously reviewed this paper.

References

1. Frick, Raymond N. ***Determining Tube Stress from CBEND Element Forces and Moments***, The MSC 1993 World Users' Conf. Proc., Paper No. 64, May, 1993.