UNITED TECHNOLOGIES
HAMILTON STANDARD

AUTOMATED STRESS ANALYSIS

REDUCING STRESS ANALYSIS TIME
BY AN ORDER OF MAGNITUDE

John C. Lambert/Brent J. Merritt

United Technologies Hamilton Standard Division
One Hamilton Road
Windsor Locks, CT 06096-1010

ABSTRACT

In an environment of ever increasing global competition, pressure on engineering organizations to produce high quality products quickly, has become extreme. As a result, fast and accurate stress analysis has become a critical issue. Analysts and Designers must conduct analyses in much shorter time spans. This paper discusses a new rule-based approach that automates finite element analysis and the entire pre- and post-processing tasks. Hamilton Standard has achieved order of magnitude reductions in analysis turnaround times as a result of rule-based automation.

INTRODUCTION

As part of an effort to reduce design cycle time and development testing of Turbomachinery, Hamilton Standard has developed a rule-based blade analysis system using ICAD[1]. This rule based system, working in conjunction with I-DEAS[2] and MSC/NASTRAN, automates the complete modal analysis process of Turbomachinery rotor blades. This system has been in production for about one year. Its extensive use has established it as the standard tool for Turbomachinery blade analysis at Hamilton Standard.

PROBLEM DEFINITION

During the development process of Turbomachinery rotor blades, geometry is modified numerous times until an optimized blade shape is obtained. All geometry modifications during this iterative process require a 3-D structural and modal analysis utilizing a MSC/NASTRAN finite element model. During this geometry optimization process, the corresponding finite element models are continuously modified to represent the revised geometry. Previous procedures utilized to create these finite element models consisted of numerous in-house codes that required large amounts of data manipulation and finite element training. Due to the complexity of this process and the amount of data manipulation required to produce an accurate model, large time intervals resulted between iterations. With the current environment of reduced cycle time, a procedure was required to allow for faster iterations which yield accurate results.

1. ICAD is a registered trademark of ICAD INC., Cambridge Ma.
2. I-DEAS is a registered trademark of SDRC INC., Milford OH.

ANALYSIS

Blade modal analysis is conducted using a CQUAD4 element model with mapped nodal thicknesses. This model is fixed at the blade/hub interface as shown in Figure 1. The blade is then subjected to a rotational load. This is accomplished by using a solution sequence developed by United Technologies Corporation. This solution sequence then extracts modal data and conducts a frequency response analysis, as well as a complete structural analysis. Results from this analysis are retrieved and adjustments are made to the blade geometry in order to achieve both optimized frequency response characteristics and structural integrity.

DISCUSSION

The entire analytical procedure described above has been automated using the ICAD rule-based software system. An example of the user interface is shown in Figure 2. A brief description of the system follows.

The initial blade definition (points) is supplied by the H.S. Aerodynamics Group and read directly into ICAD. The blade geometry, wire frame and surfaces, are then constructed directly within the ICAD system. In addition, finite element controls, such as global and local element lengths, plate element thickness maps and boundary condition application areas, are defined via rules within ICAD. Information such as element types, loads and I-DEAS grouping definitions, are also specified utilizing the rules in the ICAD system.

This high level FEA data is then transferred to SDRC/I-DEAS along with a custom generated I-DEAS program file. ICAD then starts an I-DEAS session to process the FEA data. The result of this step is a completed MSC/NASTRAN input file. At the end of this process, ICAD allows the user to preview and/or edit the autogenerated I-DEAS finite element model prior to submission of the MSC/NASTRAN analysis. It should be stressed that no manual intervention is required. A user may modify the FEA model, without breaking the automated loop, if desired.

Upon completion of the finished FEA model, ICAD conducts final preparation of the bulk MSC/NASTRAN input. Editing is automatic and rule-based. The finished MSC/NASTRAN deck is custom-built and ready for submission.

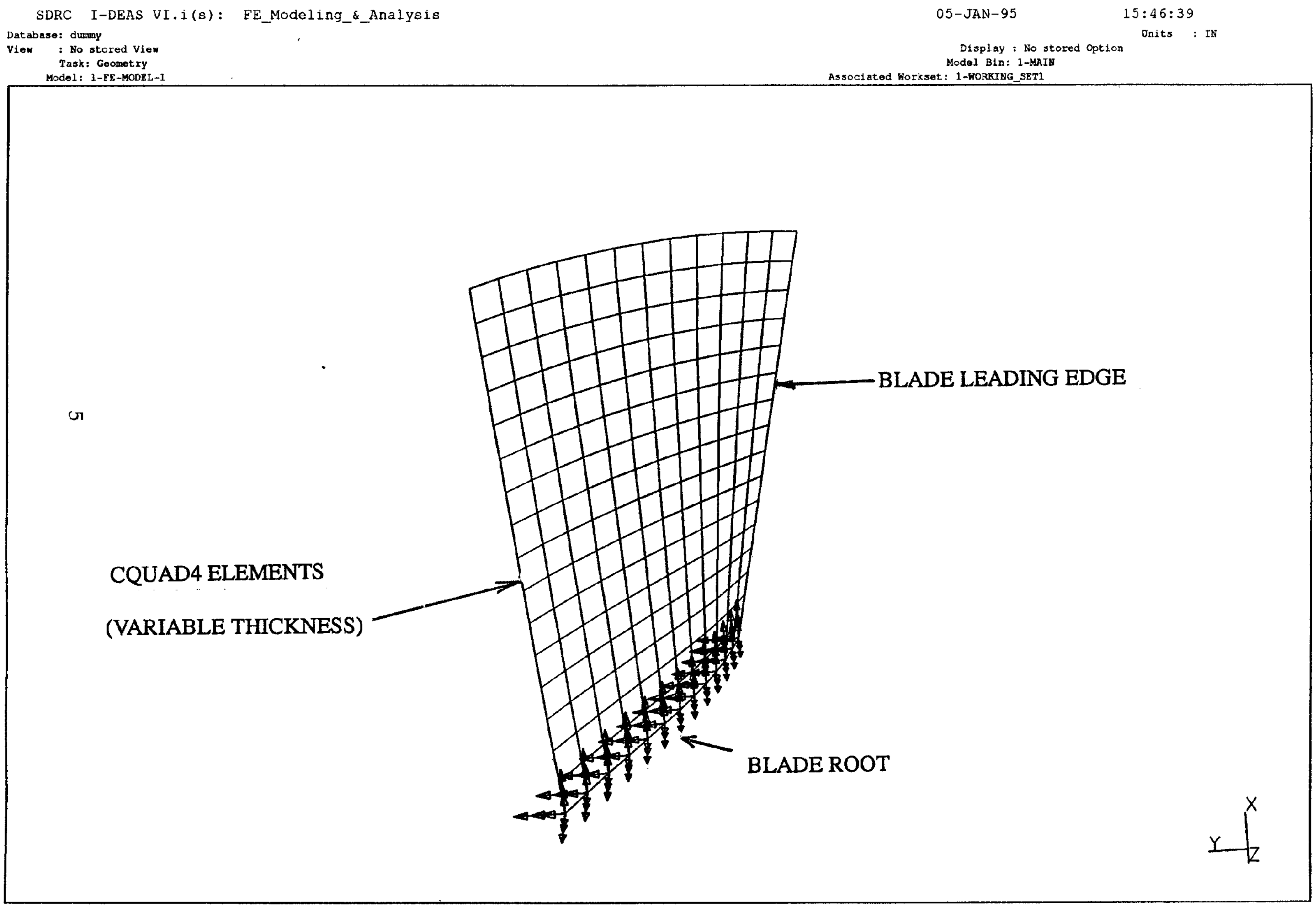
The FEA job is then submitted, via ICAD, to an IBM 590. ICAD monitors the job and upon completion translates the .op2 file and loads the results back into I-DEAS. At this point, all the post-processing is conducted via a custom-generated I-DEAS program file.

CONCLUSION

The Rule Based Blade Analysis System allows the user to quickly create finite element models with minimal data manipulation. By utilizing a systematic procedure to accurately model geometry and boundary conditions, the engineer can focus on the functional requirements of the geometry, not the mechanics of finite element modelling. This allows a user with minimal finite element related training to perform complex analysis quickly and accurately.

The ICAD based analysis procedure can be easily expanded to include other tasks associated with Turbomachinery rotor design. Combining aerodynamic definition, drawing and machining definition, and inspection into one design / analysis system, results in a complete automated procedure for the manufacture of Turbomachinery rotors.

FIGURE 1: CQUAD4 FEA MODEL



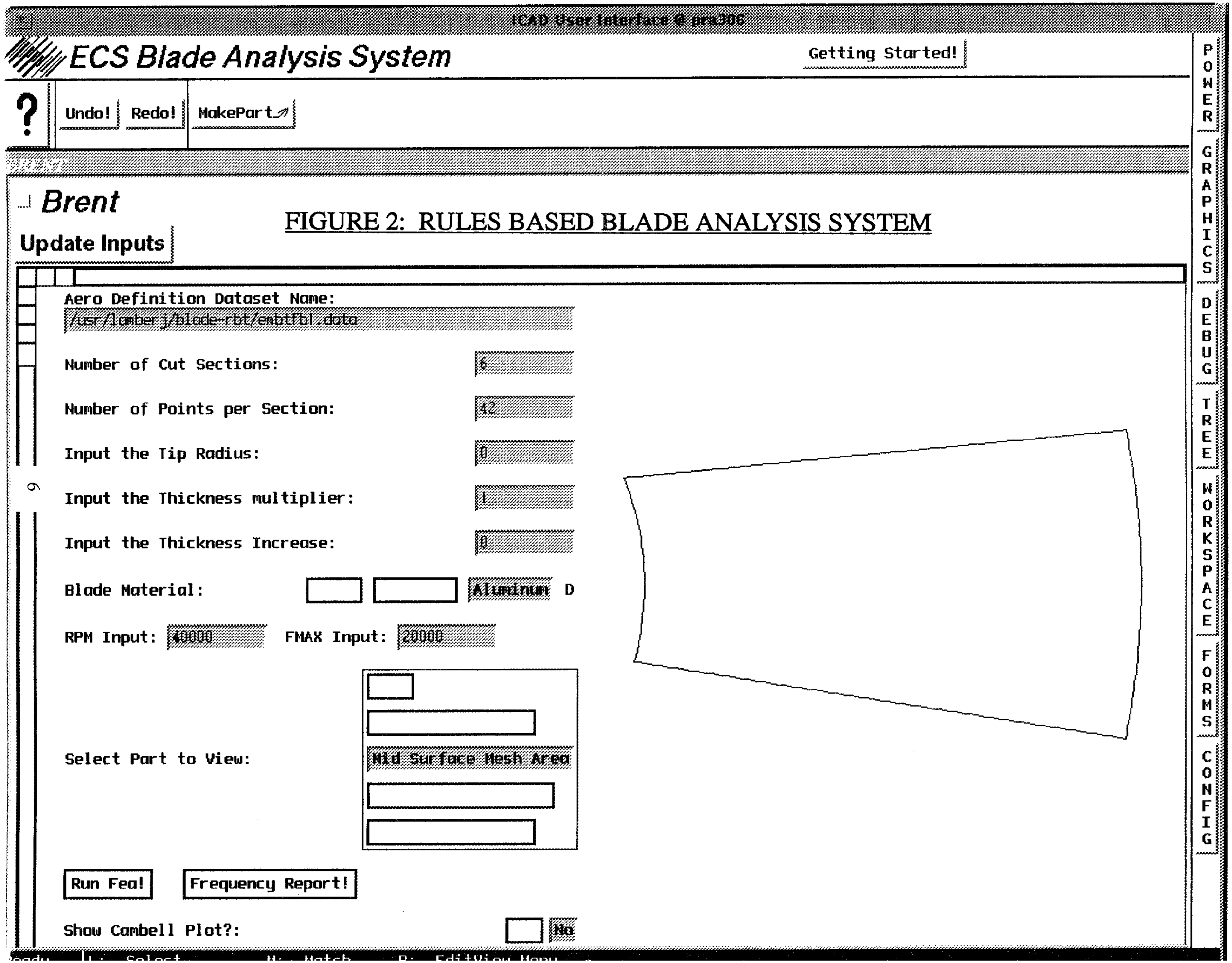


FIGURE 2: RULES BASED BLADE ANALYSIS SYSTEM