NASA Conference Publication 3142

Computational Structures Technology for Airframes and Propulsion Systems

(NASA-CP-3142) COMPUTATIONAL STRUCTURES TECHNOLOGY FOR AIRFRAMES AND PROPULSION SYSTEMS (NASA) 516 p

N92-25911
--THRU--
N92-25935
Unclas
H1/39 0088975

Proceedings of two workshops held at Lewis Research Center, Cleveland, Ohio, July 29–31, 1991, and at Langley Research Center, Hampton, Virginia, September 4–5, 1991

NASA

NASA Conference Publication 3142

Computational Structures Technology for Airframes and Propulsion Systems

Compiled by
Ahmed K. Noor
Center for Computational Structures Technology
University of Virginia
Hampton, Virginia

Jerrold M. Housner
and James H. Starnes, Jr.
NASA Langley Research Center
Hampton, Virginia

Dale A. Hopkins
and Christos C. Chamis
NASA Lewis Research Center
Cleveland, Ohio

Proceedings of two workshops sponsored by the National Aeronautics and Space Administration, Washington, D.C., and the Center for Computational Structures Technology, University of Virginia, Hampton, Virginia, and held at Lewis Research Center, Cleveland, Ohio, June 26–27, 1991, and at Langley Research Center, Hampton, Virginia, September 4-5, 1991

NASA
National Aeronautics and
Space Administration
Office of Management
Scientific and Technical
Information Program
1992

PREFACE

This document contains the proceedings of the two Workshops on Computational Structures Technology for Airframes and Propulsion Systems. The Workshops were jointly sponsored by the Center for Computational Structures Technology of the University of Virginia and NASA. The first workshop was held on June 26-27, 1991 at NASA Lewis Research Center and focused on computational technology for advanced propulsion systems. The second workshop was held on September 4-5, 1991 at NASA Langley Research Center and focused on computational technology for airframes. The attendees of the workshops came from government agencies, airframe and engine manufacturers and universities. The objectives of the workshops were to assess the status of CST in the aerospace industry, to identify the technical needs in the CST area, and to provide guidelines for focused future research leading to an enhanced capability for future national programs, such as the High-Speed Civil Transport and the National Aerospace Plane.

Certain materials and products are identified in this publication in order to specify adequately the materials and products that were investigated in the research effort. In no case does such identification imply recommendations or endorsement of products by NASA nor does it imply that the materials and products are the only ones or the best ones available for the purpose. In many cases equivalent materials and products are available and would probably produce equivalent results.

Ahmed K. Noor
Center for Computational Structures Technology
University of Virginia

Jerrold M. Housner and James H. Starnes, Jr.
NASA Langley Research Center
Hampton, Virginia

and

Dale A. Hopkins and Christos C. Chamis
NASA Lewis Research Center
Cleveland, Ohio

PRECEDING PAGE BLANK NOT FILMED

CONTENTS

PREFACE iii

ATTENDEES vii

INTRODUCTION 1

COMPUTATIONAL STRUCTURES TECHNOLOGY AND UVA CENTER FOR CST 5
Ahmed K. Noor

COMPUTER CODES DEVELOPED AND UNDER DEVELOPMENT AT NASA LEWIS . . . 43
Christos C. Chamis

PROGRESS IN INTEGRATED ANALYSIS WITH ADAPTIVE UNSTRUCTURED MESHING 59
Pramote Dechaumphai

A BRIEF OVERVIEW OF COMPUTATIONAL STRUCTURES TECHNOLOGY RELATED ACTIVITIES AT NASA LEWIS RESEARCH CENTER 81
Dale A. Hopkins

CSM ACTIVITIES AT THE NASA LANGLEY RESEARCH CENTER 91
Jerrold M. Housner

OVERVIEW OF MECHANICS OF MATERIALS BRANCH ACTIVITIES IN THE COMPUTATIONAL STRUCTURES AREA 121
C. C. Poe, Jr.

ANALYSIS AND DESIGN TECHNOLOGY FOR HIGH-SPEED AIRCRAFT STRUCTURES . 137
James H. Starnes, Jr. and Charles J. Camarda

HIGH SPEED CIVIL TRANSPORT 173
R. L. McKnight

STRUCTURES TECHNOLOGY APPLICATIONS FOR THE NATIONAL AEROSPACE PLANE 189
T. E. Little

LARGE SCALE OPTIMIZATION USING ASTROS: AN OVERVIEW 209
Vipperla B. Venkayya

LIGHT THERMAL STRUCTURES AND MATERIALS FOR HIGH SPEED FLIGHT 231
Earl A. Thornton

MODELING "BRITTLE" HIGH-TEMPERATURE COMPOSITE STRUCTURES 253
K. T. Kedward, R. M. McMeeking and S. Janson

ALGORITHMIC DEVELOPMENT IN STRUCTURES TECHNOLOGY 269
Frank Sagendorph

PRECEDING PAGE BLANK NOT FILMED

STRUCTURAL MECHANICS SIMULATIONS 303
Johnny H. Biffle

STRUCTURAL ANALYSIS FOR PRELIMINARY DESIGN OF HIGH SPEED CIVIL TRANSPORT (HSCT) 321
Kumar G. Bhatia

APPLICATION OF INTEGRATED STRUCTURAL ANALYSIS TO THE HIGH SPEED CIVIL TRANSPORT 335
C. R. Saff

OVERVIEW OF COMPUTATIONAL STRUCTURAL METHODS FOR MODERN MILITARY AIRCRAFT 395
J. N. Kudva

A PERSPECTIVE ON TECHNICAL NEEDS IN COMPUTATIONAL STRUCTURES TECHNOLOGY 375
Frank F. Abdi, Gregory L. Savoni and Kenneth J. Newell

COMPUTATIONAL STRUCTURES TECHNOLOGY AT GRUMMAN-CURRENT PRACTICE/FUTURE NEEDS 395
Allan B. Pifko and Harvey Eidinoff

AIRFRAME LIFE PREDICTION 431
G. P. Sendeckyj

PROBABILISTIC DESIGN APPLICATIONS FOR THE SPACE TRANSPORTATION MAIN ENGINES 447
Kevin O'Hara

COMPUTATIONAL STRUCTURAL ANALYSIS AND ADVANCED COMMERCIAL ENGINES 473
R. B. Wilson

MILITARY ENGINE COMPUTATIONAL STRUCTURES TECHNOLOGY 491
Daniel E. Thomson

COMPUTATIONAL STRUCTURES TECHNOLOGY ENGINE/AIRFRAME COUPLING 507
Bruce C. McClintick

Specialty Workshops on Computational Structures Technology for Airframes and Propulsion Systems

Attendees

Mr. Frank Abdi
D/734-11-GB13
Rockwell International
201 North Douglas Street
El Segundo, CA 90245
(213) 922-0990; Fax (213) 414-0430

Dr. Kumar Bhatia
Mail Stop 3T-AF
Boeing Corporation
P.O. Box 3707
Seattle, WA 98124
(206) 393-6993; Fax (206) 477-2345

Dr. Johnny H. Biffle
Applied Mechanics III, Div. 1523
Sandia National Laboratory
P.O. Box 5800
Albuquerque, NM 87185
(505) 844-5385; Fax (505) 846-9833

Dr. Christos C. Chamis
Mail Stop 49-8
NASA Lewis Research Center
21000 Brookpark Road
Cleveland, OH 44135
(216) 433-3252; Fax (216) 433-8011

Dr. Pramote Dechaumphai
Mail Stop 395
NASA Langley Research Center
Hampton, VA 23665
(804) 864-1357

Mr. Dale A. Hopkins
Mail Stop 49-8
NASA Lewis Research Center
21000 Brookpark Road
Cleveland, OH 44135
(216) 433-3260; Fax (216) 433-8011

Dr. Jerrold M. Housner
Mail Stop 240
NASA Langley Research Center
Hampton, VA 23665
(804) 864-2906

Prof. Keith T. Kedward
Mechanical Engineering Department
University of California
Santa Barbara, CA 93106
(805) 893-3381; Fax (805) 893-8651

Dr. N. J. Kudva
MS Dept. 3852-MF
Northrop Aircraft Division
Northrop Corporation
One Northrop Avenue
Hawthorne, CA 90250-3277
(213) 332-8300; Fax (213) 332-5853

Mr. Bruce C. McClintick
Mail Drop A33A
General Electric Company
Highway 75
Cincinnati, OH 45215
(513) 583-5146; Fax (513) 583-5170

Mr. R. L. McKnight
Mail Drop A333
General Electric Aircraft Engines
P.O. Box 156301
Cincinnati, OH 45215
(513) 583-5068; Fax (513) 583-5170

Mr. James E. Newell
Mail Code D/545-055-JB11
Rockwell International
6633 Canoga Park
Canoga Park, CA 91303
(818) 773-5505; Fax (818) 773-5542

Prof. Ahmed K. Noor
Center for Computational Structures Technology
Mail Stop 210
NASA Langley Research Center
Hampton, VA 23665
(804) 864-1978; Fax (804) 864-8089

Mr. Kevin T. O'Hara
Mail Code D/611-055-RA02
Rockwell International
2227 Drake Avenue, Suite 45
Huntsville, AL 35805
(205) 880-4519; Fax (205) 880-4596

Dr. Allan B. Pifko
Mail Stop A08-35
Research Department
Grumman Corporation
Bethpage, NY 11714
(516) 575-1965; Fax (516) 575-7716

Mr. Clarence C. Poe, Jr.
Mail Stop 188E
NASA Langley Research Center
Hampton, VA 23665
(804) 864-3467

Dr. Charles R. Saff
Mail Code 1021322
McDonnell Douglas
P.O. Box 516
St. Louis, MO 63166
(314) 233-8623; Fax (314) 777-1171

Mr. Frank E. Sagendorph IV
Mail Drop A333
General Electric Aircraft Engines
P.O. Box 156301
Cincinnati, OH 45215
(513) 583-5001; Fax (513) 583-5170

Dr. George P. Sendeckyj
WL/FIBEC
Wright Patterson Air Force Base, OH 45433
(513) 255-6104; Fax (513) 255-3717

Dr. James H. Starnes, Jr.
Mail Stop 190
NASA Langley Research Center
Hampton, VA 23665
(804) 864-3168; Fax (804) 864-7791

Mr. Daniel E. Thomson
WL/POTC
Wright Patterson Air Force Base, OH 45433
(513) 255-2081; Fax (513) 476-4531

Prof. Earl A. Thornton
Light Thermal Structures Center
School of Engineering and Applied Science
Thornton Hall
University of Virginia
Charlottesville, VA 22903
(804) 924-6291; Fax (804) 982-2037

Dr. V. B. Venkayya
WL/FIBR
Wright Patterson Air Force Base, OH 45433
(513) 255-7191; Fax (513) 255-3740

Mr. Samuel L. Venneri
Code RM
NASA Headquarters
Washington, D.C. 20546
(202) 453-2760; Fax (202) 755-4068

Mr. Raymond B. Wilson
Mail Stop 163-10
Engineering Division
Pratt & Whitney
400 Main Street
East Hartford, CT 06108
(203) 565-2901; Fax (203) 565-9615

INTRODUCTION

Performance requirements for future flight vehicles are rapidly increasing due to ambitious objectives of the U.S. civil and military aerospace programs. In aeronautics, future goals include higher cruising speeds, altitudes and thrust-to-weight ratios. The technology drivers for future aircraft include reduction in material, fabrication and maintenance costs; reduction in weight; extended life; higher operating temperature; and signature reduction. In space, future goals include lower transportation costs to space; long-duration space flights; planetary missions; and extraterrestrial bases.

To successfully achieve the performance requirements for planned and future aeronautical and space systems major advances are needed in: 1) computational structures technology (CST); 2) advanced and engineered materials such as high-temperature composites and advanced metallics; 3) formulation and solution of coupled multidiscipline problems; 4) computational simulation of concurrent engineering; and 5) accurate quantification of risk. The timely development and deployment of these technologies is essential to insure U.S. superiority in the aerospace field.

Several national programs such as High-Speed Civil Transport (HSCT), National Aerospace Plane (NASP), National Launch System (NLS), and Integrated High-Performance Turbine Engine Technology (IHPTET) need major advances in a number of key areas of computational structures technology. To this end, there are a number of primary pacing items and related tasks that must be addressed by the research community.

The joint NASA/University of Virginia Workshops held at NASA Lewis Research Center, June 26-27, 1991 and at NASA Langley Research Center, September 4-5, 1991 focused on the status of computational structures technology and the pacing items of this technology. The list of pacing items given in this introduction was compiled from a number of participants. It is anticipated that the items in the list can impact the design and operation of future flight vehicles in the following four ways: 1) by providing better understanding of the phenomena associated with response, failure and life, thereby identifying the desirable structural design attributes; 2) by improving the productivity of the design team, and reducing the response time to resolve operability problems; 3) by verifying and certifying designs, and making low-cost design modifications during the design process; and 4) by allowing major improvements and innovations in the design process so as to achieve a fully integrated design in a concurrent engineering environment. In such an environment, computer simulation is made of the entire life cycle of the flight vehicle including material selection and processing, multidisciplinary design, automated manufacturing and fabrications, quality assurance, certification, operation, health monitoring and control (e.g., maintenance and repairs), retirement and disposal. The ultimate aim of CST research is to impact the fully integrated design process.

Primary Pacing Items

The primary pacing items identified by the participants can be grouped into the following six headings: 1) computational material modeling; 2) failure and life prediction methodologies; 3) hierarchical, integrated multiple methods and adaptive modeling techniques; 4) probabilistic analysis, stochastic modeling and risk assessment; 5) validation of numerical simulations; and 6) multidisciplinary analysis and design optimization. For each of the aforementioned items attempts should be made to exploit the major characteristics of high-performance computing technologies, as well as the future computing environment. The six primary pacing items are described subsequently. Note that some of the tasks within the pacing items are of generic nature, others are specific to either propulsion systems or airframes.

1. Computational Material Modeling. The reliability of the predictions of response, failure and life of structures is critically dependent on the accurate characterization and modeling of material behavior. The simple material models used to date are inadequate for many of the future applications, especially those involving severe environment (e.g., high temperatures). Needed work on material modeling can be grouped in two general areas: a) modeling the response and damage of advanced material systems in the actual operating environment of future flight vehicles; and b) numerical simulation of manufacturing (fabrication) processes.

Advanced material systems include new polymer composites, metal matrix composites, ceramic composites, carbon/carbon and advanced metallics. The length scale selected in the model must be adequate for capturing the response phenomena of interest (e.g., micromechanics, mesomechanics, macromechanics). For materials used in propulsion systems, work is needed on the modeling of damage accumulation and propagation to fracture; modeling of thermoviscoplastic response, thermal-mechanical cycling and ratcheting; and prediction of long-term material behavior from short-term data, which are particularly important.

2. Failure and Life Prediction Methodologies. Practical numerical techniques are needed for predicting the life, as well as the failure initiation and propagation in structural components made of new, high-performance materials in terms of measurable and controllable parameters. Examples of these materials are high-temperature materials for hypersonic vehicles; piezoelectric composites; and electronic, optical, and smart materials for space applications. For some of the materials, accurate constitutive descriptions, failure criteria, damage theories, and fatigue data are needed, along with more realistic characterization of interface phenomena (such as contact and friction). The constitutive descriptions may require investigations at the microstructure level or even the atomic level, as well as carefully designed and conducted experiments. Failure and life prediction of structures made of these materials is difficult and numerical models often constructed under restricting assumptions may not capture the dominant and underlying physical failure mechanisms. Moreover, material failure and structural response (such as instability) often couple in the failure mechanism.

3. Hierarchical, Integrated Multiple Methods and Adaptive Modeling Techniques. The effective use of numerical simulations for predicting the response, life, performance and failure of future flight vehicles requires strategies for treating phenomena occurring at disparate spatial and time scales, using reasonable computer resources. The strategies are based on using multiple mathematical models in different regions of the structure to take advantage of efficiencies gained by matching the model to the expected response in each region. To achieve the full potential of hierarchical modeling, there should be minimal reliance on *a priori* assumptions about the response. This is accomplished by adding adaptivity to the strategy. The key tasks of the research in this area are the following:

1) simple design-oriented models for use in the early stages of the design process

2) rational selection of a set of nested mathematical models for different regions, and discretization techniques for use in conjunction with the mathematical models. This, in turn, requires the availability of a capability for holistic modeling from micro to structural response with varying degrees of accuracy.

3) simulation of local phenomena through global/local methodologies

4) automated (or semiautomated) coupling of different mathematical/discrete models

5) error estimation and adaptive modeling strategies

6) high fidelity modeling of details (such as damping, material nonlinearities, joints). For propulsion systems, this may require, among other things, efficient full three-dimensional multi-load analyses;

7) efficient methods for engine airframe and rotor/engine-frame coupling.

8) sensitivity analysis to assess the sensitivity of the response to each of the parameters neglected in the current mathematical model.

4. Probabilistic Analysis, Stochastic Modeling and Risk Assessment. The new methodology developed for treating general forms of uncertainties in geometry, material properties, boundary conditions, loading, and operational environment in the structural analysis formulation of structural

components needs to be extended to probabilistic design/risk assessment of full flight vehicles. The ability to quantify inherent uncertainties in the response of flight vehicles is obviously of great advantage. However, the principal benefit of using any stochastic method consists of the insights into engineering, safety, and economics that are gained in the process of arriving at those quantitative results and carrying out reliability analyses. As future flight-vehicle structures become more complicated, failure mechanisms will be probabilistically modeled from the beginning of the design process, and potential design improvements will be evaluated to assess their effects on reducing overall risk. The results, combined with economic considerations, will be used in systematic cost-benefit analyses (perhaps also done on a probabilistic basis) to determine the structural design with the most acceptable balance of cost and risk.

5. Validation of Numerical Simulations. In addition to selecting a benchmark set of flight-vehicle structures for assessing new computational strategies and numerical algorithms, a high degree of interaction and communication is needed between computational modelers and experimentalists. This is done on four different levels, namely, 1) laboratory tests on small specimens to obtain material data; 2) component tests to validate computational models; 3) full-scale tests to validate the modeling of details; and 4) flight tests to validate the entire modeling process.

6. Multidisciplinary Analysis and Design Optimization. The realization of new complex aerospace vehicles requires integration between the structures discipline and other traditionally separate disciplines such as aerodynamics, propulsion and control. This is mandated by significant interdisciplinary interactions and couplings which need to be accounted for in predicting response, as well as in optimal design of these vehicles. Examples are the couplings between the aerodynamic flow field, structural heat transfer, and structural response of high-speed aircraft and propulsion systems; and the couplings between the control system and structural response in control-configured aircraft and spacecraft. This activity also includes design optimization with multi-objective functions (e.g., performance, durability, integrity, reliability and cost), and multi-scale structural tailoring (micro, local, and global levels). For propulsion systems it also includes design with damping for high-cycle fatigue, low-cycle-fatigue, and acoustic fatigue.

Typically, in the design process questions arise regarding influence of design variable changes on system behavior. Answers to these questions, quantified by the derivatives of behavior with respect to the design variables or by parametric studies, guide design improvements toward a better overall system. In large applications this improvement process is executed by numerical optimization, combined with symbolic/AI techniques, and human judgement aided by data visualization. Efficiency of the computations that provide data for such a process, is decisive for the depth, breadth, and rate of progress achievable, and hence, ultimately, is critical for the final product quality.

Related Tasks

For CST to impact the design process, the following three tasks need to be addressed by the research community: 1) development of automated or semi-automated model (and mesh) generation facilities; 2) pre- and postdata processing and use of advanced visualization technology; 3) adaptation of AI tools (knowledge-based/expert systems and neural networks) to CST.

N92-25912

Computational Structures Technology and UVA Center for CST

Ahmed K. Noor
University of Virginia

PRECEDING PAGE BLANK NOT FILMED

OUTLINE

Rapid advances in computer hardware have had a profound effect on various engineering and mechanics disciplines, including the materials, structures and dynamics disciplines. A new technology, computational structures technology (CST), has recently emerged as an insightful blend between material modeling, structural and dynamic analysis and synthesis on the one hand, and other disciplines such as computer science, numerical analysis and approximation theory, on the other hand (see Ref. 1). CST is an outgrowth of finite element methods developed over the last three decades. The focus of this presentation is on some aspects of CST which can impact future airframes and propulsion systems, as well as on the newly established UVA Center for CST. The outline is shown in Fig. 1. *First*, the background and goals for CST are described along with the motivations for developing CST. *Second*, a brief discussion is made on computational material modeling. *Third*, we look at the future in terms of technical needs, computing environment and research directions. *Fourth*, the newly established UVA Center for CST is described. *Fifth*, one of the research projects of the Center is described, and *finally* a brief summary of the presentation is given.

- **Background, goals for CST and motivations for development of CST**
- **Computational material modeling**
- **A look at the future**
  - **Technical needs**
  - **Computing environment**
  - **Research directions**
- **UVA Center for CST**
- **Description of one research project**
- **Summary**

Figure 1

DIFFERENT ASPECTS OF CST WORK

Current CST work includes the following facets (Fig. 2):

• computational material modeling

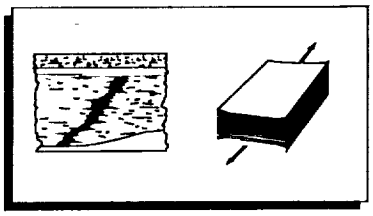
• computational methods for predicting the response, performance, failure and life of structures and components thereof. Some of these activities have been labeled computational structural mechanics - CSM (Refs. 2 and 3).

• automated methods of structural synthesis and optimization. In addition, CST is an important component in multidisciplinary analysis and design of many engineering fields.

- **Computational material modeling**
- **CSM - Computational methods for predicting**
  - **Response**
  - **Performance**
  - **Failure, and**
  - **Life**

  } **of structures and components thereof**

- **Automated methods of structural synthesis and optimization. Also, CST is an important component in the multidisciplinary analysis and design of many engineering systems.**



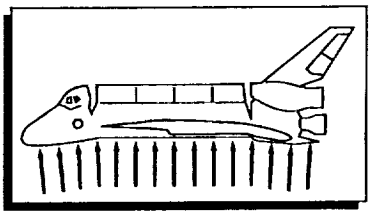


Figure 2

GOALS FOR CST

Within the aeronautics and space fields, the following four major goals of CST can be identified (Fig. 3):

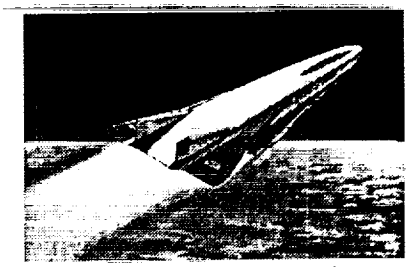
First. To predict, in a reasonable amount of time, the response, failure and life of flight-vehicle structures and their components at actual operating conditions.

Second. To complement and supplement experiments and flight testing, and to help in the design of experiments.

Third. To explore new phenomena, which are difficult to understand by means other than numerical simulations.

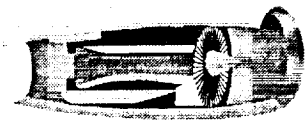
Fourth. To aid in the design process of flight vehicles.

First
To predict response, performance, failure and life of flight-vehicle structures at actual operating conditions



Second
To design, complement and supplement experiments and other tests for flight-vehicle structures

Third
To study phenomena which are difficult to understand by means other than numerical simulations (e.g., damage and failure mechanisms of high-performance materials at high temperatures)



Fourth
To aid in the design of flight-vehicle structures

Figure 3

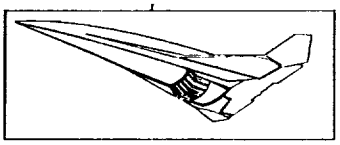
MOTIVATIONS FOR DEVELOPING CST

There are three compelling motivations for vigorously developing CST (Fig. 4). The *first compelling motivation* is that a number of unsolved practical problems awaits experiment and/or numerical solutions. Some of these problems involve numerical simulations so large and complex that they overtax the capacity of even present-day large computers. Examples of these problems are the simulation of aircraft response to multidirectional crash impact forces, and the study of thermoviscoplastic response of structural components used in advanced propulsion systems. In other structural problems, the fundamental mechanics concepts are still being explored (e.g., damage initiation and propagation in structural components made of new material systems).

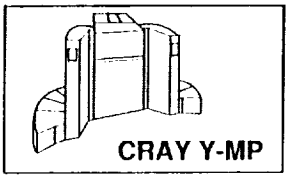
A *second compelling reason* is that numerical simulation is needed to reduce the dependence on extensive and expensive testing, which is frequently component- or mission-oriented. Moreover, in some mission-critical areas in space, computer modeling may, of necessity, replace tests. This is because future large aerospacecrafts and space structures are likely to exceed the limits of ground-test technology.

A *third major motivation* for developing CST relates to the anticipated power and potential of emerging and future high-performance computing systems in solving large-scale structural problems. The potential of these high-performance computers can be realized only by developing new formulations, computational strategies and numerical algorithms that exploit the capabilities of the new machines.

- *Practical problems awaiting solutions*
  - Very large problems (e.g., due to high-degree of integration required and/or high-degree of sophistication needed in modeling)
  - Problems for which fundamental mechanics concepts are still being explored (e.g., failure mechanisms of structural components made from new material systems)



- *Reduce dependence on testing*
  - Reliability of testing large space structures in 1-G environment is questionable
- *Exploit new and emerging computing technology*



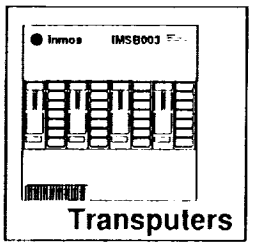


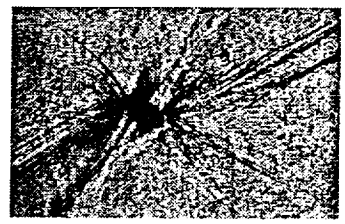
Figure 4

COMPUTATIONAL MATERIAL MODELLING
Objective and Status

Considerable attention has recently been devoted to computational material modeling. In the context of CST, the objective and status of the activity are summarized in Fig. 5. The overall objective is to develop a hierarchy of material models (multilevel/multiscale) to describe the different phenomena associated with response, life and failure of structures. This has resulted in a gradual evolution from empirical models to physics-based models that predict the processing response and properties. However, many gaps still exist in the hierarchy of models.

Objective:

- **Development of a hierarchy (multilevel/ multiscale) of material models to describe the different phenomena associated with response, life and failure of structures**



Status:

- **Gradual evolution from empirical models to physics-based models that predict processing response and properties.**
- **Many gaps still exist in the hierarchy of models**

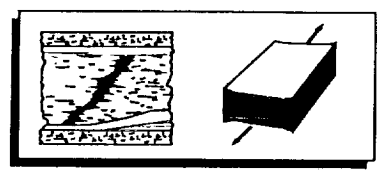


Figure 5

COMPUTATIONAL MATERIAL MODELING
Hierarchy of models

A hierarchy of material models are shown in Fig. 6. The models are arranged according to the phenomena they describe and the length scale at which this phenomena is studied (from 10^{-10}m to one meter). The disciplines involved include computational chemistry, which covers molecular dynamics and quantum mechanics; computational material science; and computational structural mechanics. The models used range from atomistic to single crystals to polycrystals to micromechanical and macromechanical models.

Models	Discipline		Length scale, m	Phenomena
Phenomenological/ macromechanical	Computational structural mechanics		10^{0} 10^{-2}	• Structural response • Metal forming
Micromechanical	Computational material science		10^{-4}	• Plastic strain localization • Crack tip fields • Indentation fracture
Polycrystals (homogenized models)			10^{-6}	• Void growth • Polycrystalline slip • Microstructural effects
Single crystals			10^{-8}	• Dislocations • Particles and interfaces
Atomistic models	Molecular dynamics	Computa-tional chemistry	10^{-10}	• Creep diffusion • Cleavage • Discrete defects
	Quantum mechanics			• Basic transport properties • Phase transformation

Figure 6

A LOOK AT THE FUTURE

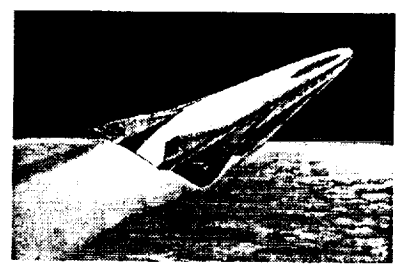
CST is likely to play a significant role in the future development of structures technology as well as in the multidisciplinary design and certification of future flight vehicles. For this to happen major advances and computational tools are needed in a number of key areas of CST. To this end, there are a number of primary and secondary pacing items that must be addressed by the research community (see Ref. 1). The following three factors are taken into account in identifying the pacing items (Fig. 7):

1) characteristics of future flight vehicles, their technical needs and implications for CST;

2) future computing environment; and

3) recent and projected developments in other fields of computational technology which can be adapted to CST.

- **Computational technology will play a significant role in the development of structures technology and in the multi-disciplinary design and certification of future flight vehicles**
- **Major and secondary pacing items**
- **Basis for determining the pacing items**
  - **Characteristics of future flight vehicles, their technical needs and the implications for CST**
  - **Future computing environment**
  - **Recent and projected developments in other fields of computational technology which can be adapted to CST**



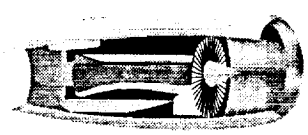


Figure 7

TECHNICAL NEEDS FOR FUTURE FLIGHT-VEHICLE STRUCTURES AND THEIR COMPUTATIONAL IMPLICATIONS

The technical needs for future flight vehicles can be grouped into three major areas (Fig. 8), namely:

1) *new high-performance material systems.* These include new composite materials, advanced metallics as well as intelligent/smart material systems;

2) *novel structural concepts* which include structural tailoring and smart/adaptive structural concepts;

3) *expanding the scope of engineering problems* considered to include investigation of more complex phenomena, such as damage tolerance of new material systems; and study of interdisciplinary couplings (e.g., structure/fluid/thermal/control interaction problems).

The implications of the aforementioned technical needs for CST include:

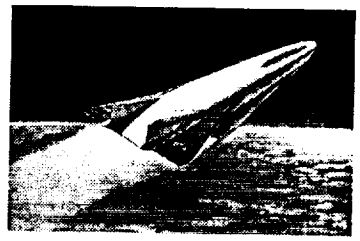
1) *development of computational models* for new material systems over the entire stress/strain/temperature range of interest;

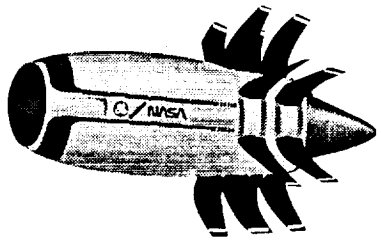
2) *high fidelity representation of details* (e.g., material response, joints and damping); and

3) *development of effective computational strategies* for large-scale coupled problems.

Needs

- New high-performance material systems (including intelligent/smart materials)
- Novel structural concepts (e.g., structural tailoring and smart/adaptive structures)
- Investigation of complex phenomena and interdisciplinary couplings





Implications

- Computational material models
- High fidelity representation of details
- Strategies for large-scale coupled problems

Figure 8

TRENDS IN HIGH-PERFORMANCE COMPUTING

The trends in high-performance computing is shown in Fig. 9. The increase in speed of computers in the 1950's and early 1960's was a result of advances in device technology. In the late 1960's pipelining and vectorization was introduced. Development of computers with homogeneous parallelism began in the late 1970's. Recent trend is moving towards *distributed heterogeneous supercomputing*. It is anticipated that before the end of the century teraflop computing will be achieved (speeds reaching trillion floating-point operations per second) .

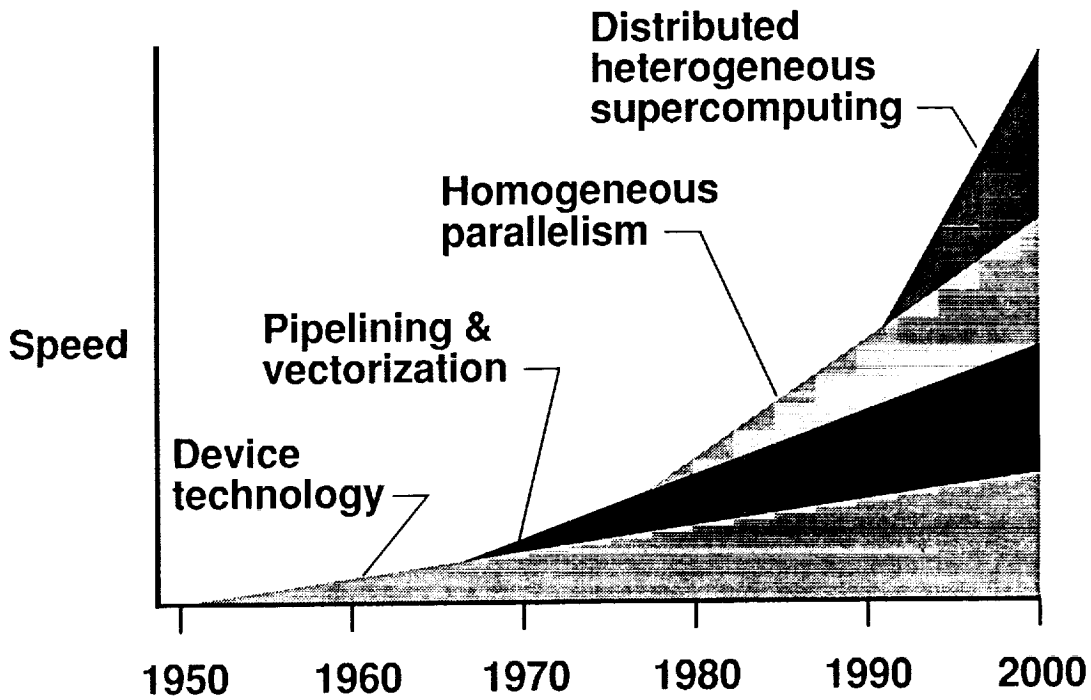


Figure 9

DISTRIBUTED HETEROGENEOUS MULTICOMPUTERS

The basic concept of *distributed heterogeneous multicomputers (DHM)* is highlighted in Fig. 9a. The concept refers to an integrated computing environment consisting of networked computer systems in which the network is the computer. The use of DHM can greatly alleviate the limitations on the size of numerical simulations imposed by current supercomputer speeds and memory capacities. This is accomplished by combining the resources of disparate supercomputing platforms through high-speed networks. New buzzwords like "META Computer" are currently used to refer to this concept.

The hardware can include a plethora of architectures such as: large-grain vector supercomputers (e.g., CRAY 3, C-90, SSI), minisuper computers (e.g., Convex C-380); clustered processors (Cedarlike); massively parallel systems (e.g., Intel Touchstone); application-specific computer systems; advanced workstations (IBM RISC, SGI, SUN, ...); and HDTV hardware and software with video facilities.

Description: An integrated computing environment consisting of networked computer systems - Garden (or Plethora) of new architectures - META Computer.

Hardware includes:

- **Large-grain vector supercomputers (e.g., CRAY 3, C-90, SSI)**
- **Minisuper computers (Convex C380)**
- **Clustered processors (Cedarlike)**
- **Massively-parallel systems (Touchstone - gamma, delta or sigma)**
- **Application-specific computer systems**
- **Advanced workstations (IBM RISC, SGI, SUN, . . .)**
- **HDTV hardware and software with video facilities**

Figure 9a

DISTRIBUTED HETEROGENEOUS MULTICOMPUTERS (CONT'D.)

The effectiveness of DHM is strongly dependent on the availability of high-speed local and wide area networks for data transfer between the different computers.

For local area networks (LAN), FDDI (Fiber distributed data interface), HiPPI (high-performance parallel interface) and UltraNet links are currently being used. For wide area networks (WAN) the NSF net T-3 internet will be upgraded to the T-5 internet (Fig. 10).

Because of the very large volumes of data, significant improvement in mass storage facilities will be needed. Future mass storage systems will have high bandwidth (IEEE reference model), and include the following key elements: large disk arrays directly connected through HiPPI channels; mass robotic media, and a massive high-speed file transfer network.

Networking:

LAN	FDDI	100 Mb/sec.
	HiPPI	1 Gb/sec.
WAN	T3-Internet	60 Mb/sec.
	T5-Internet	

Mass Storage:

- High bandwidth mass storage (IEEE reference model)
- Large disk arrays directly connected through HiPPI (> 100 G Bytes)
- Mass robotic media
- Massive file transfer network (> 200 M Bytes/sec.)

Figure 10

WHY DISTRIBUTED HETEROGENEOUS MULTIPROCESSORS?

The primary motivations for using DHM are to achieve higher speeds and higher levels of parallelism than can be done on a single architecture. These are referred to as *hypercomputing* and *superconcurrency* (Fig. 11).

Experience with different vector and parallel architectures has shown that different architectures can be relatively effective (or ineffective) on different sections of the computational process. Therefore, for most of the practical applications the sustained speed is a fraction of the peak performance of the machine. In distributed heterogeneous multiprocessors, the physical characteristics of the problem, and the numerical algorithms used in each module, will form the basis for identifying the most suitable platform for that module.

Hypercomputing and superconcurrency

- **Optimal support for algorithmically diverse parts of an application program on architectually diverse machines.**
  - **Different architectures can be relatively effective or ineffective on different sections of the computational process (code profiling and machine matching).**
- **Parameters and data lengths can affect choice of architecture.**

Figure 11

CROSS OVER POINTS FOR SAXPY

As an example, the speed of performing the operation Z(I) = R*X(I) + Y(I) on four different computers as a function of the vector length is depicted in Fig. 12. As can be seen from this figure for each range of vector lengths, higher speed is achieved by a different machine.

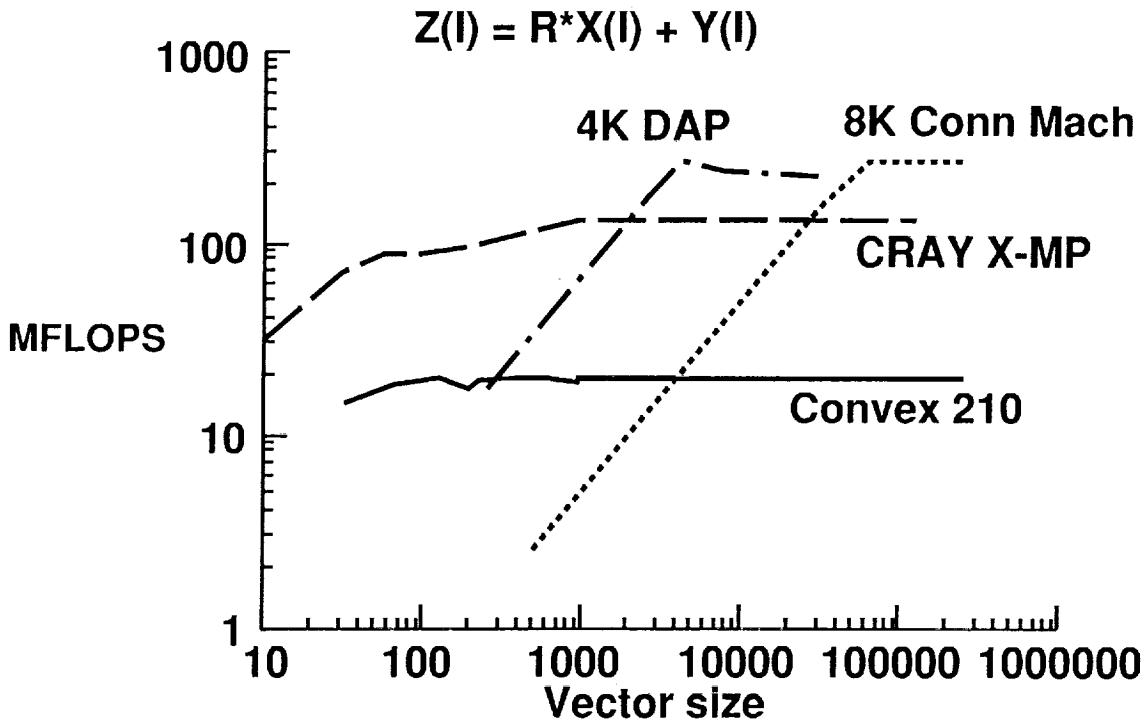


Figure 12

PROFILES OF PROCESSOR TYPES

The results of the previous figure are extrapolated to the entire computation spectrum of a finite element analysis on a network of n supercomputer platforms. For each module, the most suitable platform is identified as the one which results in the highest speed for that module (Fig. 13).

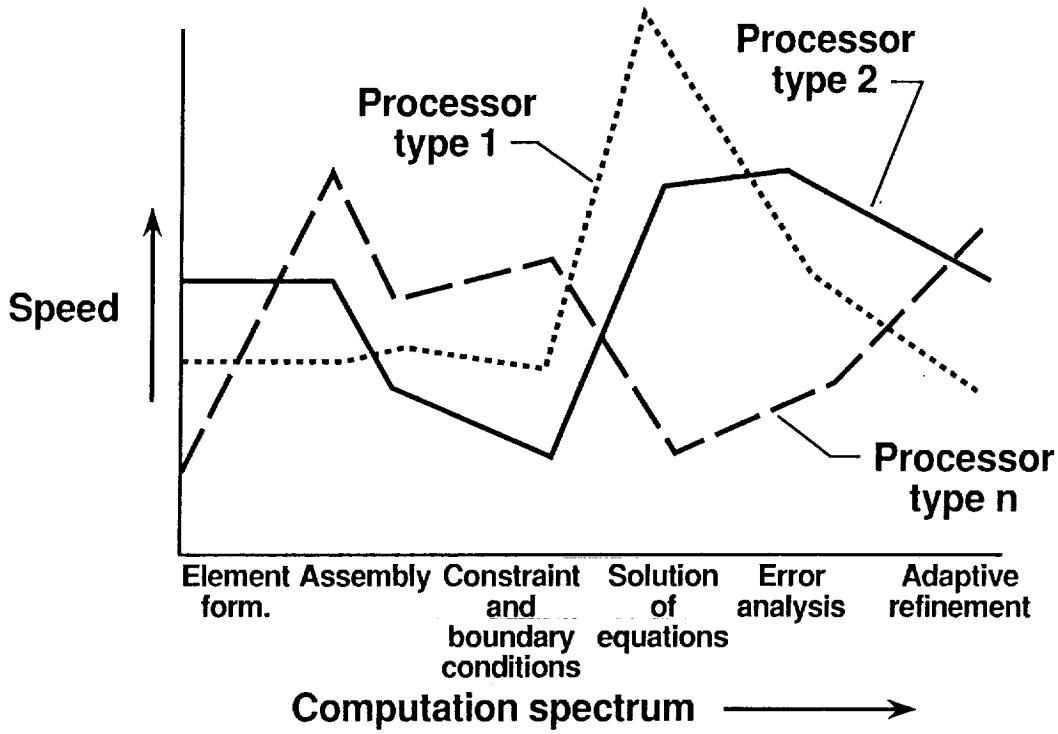


Figure 13

FORMS OF DISTRIBUTED HETEROGENEOUS MULTICOMPUTERS (DHM)

Three different forms of DHM are shown in Fig. 14. The first is *micro DHM*, in which a number of different workstations are connected. An example of this is the *PASM project* at Purdue University. The second is the *site DHM*, in which different architectures are connected at the same site. Examples of site DHM are the NSF Centers at Illinois, Pittsburgh and San Diego, and NOSC. The third is the *global DHM*, in which supercomputing platforms at different sites are connected. In each of the different DHM sites, the application program is executed on different platforms. The proposed National Metacenter concept, connecting the four NSF Supercomputing Centers is an example of global DHM.

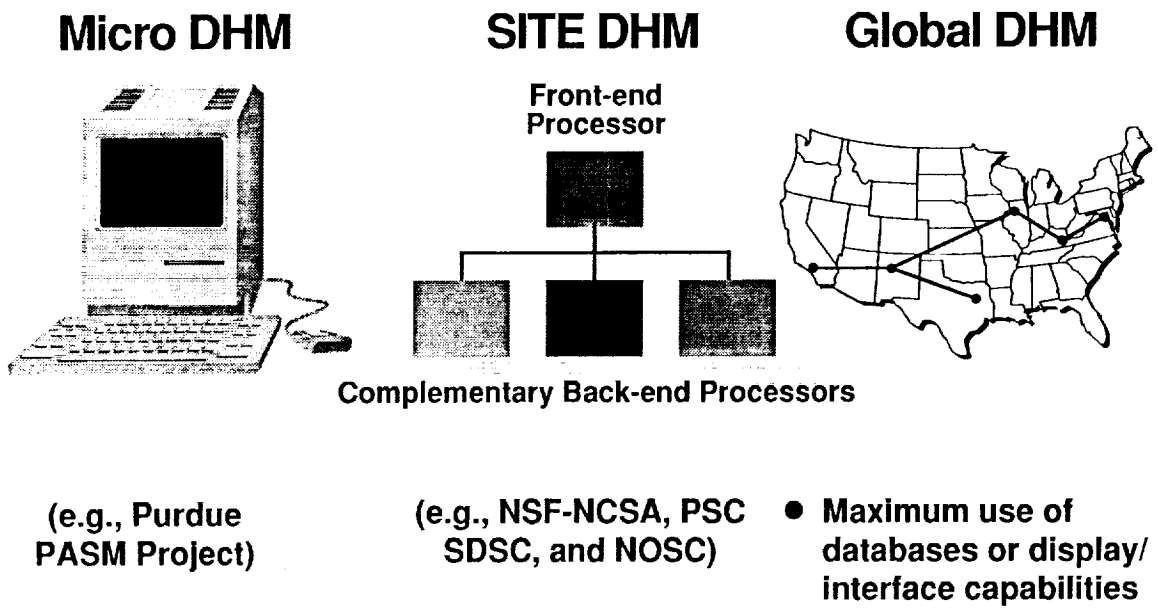


Figure 14

SUPPORT FACILITIES

Despite its potential, DHM also has a number of potential pitfalls. In particular, networks must be capable of adequate transfer rates in order to sustain high computing speeds on the supercomputers. In addition, the data format of one computer is not likely to be identical to that of another. Thus, the amount of time spent transforming or translating data must be measured. For DHM to be viable, this should not be a significant fraction, since it figures into the overall effective bandwidth. If transfers between two computers must cross several networks, such mismatches as packet size and transfer rate differences could affect the overall transfer rate.

In addition, a number of support facilities are needed to realize the full potential of DHM, (Fig. 15). These include: distributed operating system (e.g., MARK or KRONOS); network language (e.g., Linda, express); high-level programming abstractions and object-oriented tools; expert systems and user interfaces to aid in the partitioning of the program into tasks and scheduling the tasks on processors of different types.

- **Distributed operating system (e.g., MARK, KRONOS)**
- **Network language (e.g., Linda, Express)**
- **Object oriented tools and user interfaces**

Figure 15

FUTURE DIRECTIONS FOR RESEARCH

The future directions for research in CST are listed in Fig. 16. For convenience, they are divided into major pacing items and related tasks. The *major pacing items* include the following four:

1) high fidelity modeling of material response, structural, geometrical and topological details as well as the environmental effects;

2) life prediction and analysis of failure of structural components made of new materials;

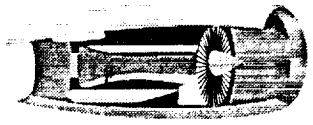
3) effective computational strategies for large-scale problems which include: integrated fluid/thermal/structural/control analysis; sensitivity analysis; and multidisciplinary analysis and optimization of large systems; and

4) validation and assessment of the reliability of numerical simulations.

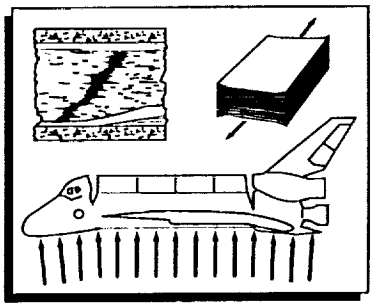
The *related tasks* include predata, postdata processing and effective use of visualization technology; and integration of analysis programs into CAD/CAE and concurrent engineering systems.

Major Pacing Items

- **High fidelity modeling of material response, structural, geometrical and topological details, environmental effects (e.g., boundary-layer transition, interference heating)**
- **Life prediction and analysis of failure for structural components and structures made of new materials**



- **Effective computational strategies for large-scale problems**
  - **Integrated fluid/thermal/structural analysis**
  - **Sensitivity analysis**
  - **Multidisciplinary design and optimization**
- **Validation and assessment of reliability of aerodynamic/thermal/structural response predictions.**



Secondary Pacing Items

- **Predata, postdata processing and effective use of visualization technology**
- **Integration of analysis programs into CAD/CAE and concurrent engineering systems**

Figure 16

UVA CENTER FOR CST

The background and objectives of the UVA Center for CST are highlighted in Fig. 17. The Center was established in July 1990. It is located at NASA Langley. The primary funding source is NASA Headquarters. Research grants have also been obtained from NASA Langley and AFOSR.

The *overall goal* of the Center is to serve as a focal point for the diverse CST activities including modeling, analysis, sensitivity studies, optimization and use of AI methods. The Center has the following four specific objectives:

1) to *conduct innovative research* on advanced topics of CST;

2) to *act as pathfinder*, by demonstrating to the research community what can be done (high-potential, high-risk research);

3) to *help in identifying future directions of research* in support of the aeronautical and space missions of the twenty-first century; and,

4) to *help in the rapid transfer of research results* and in broadening awareness among researchers and engineers of the state-of-the-art in CST as well as in other areas of computational technology which can impact CST (notably CFD and computational mathematics).

BACKGROUND:

- Established in July 1990
- Located at NASA Langley in Hampton
- Funded by NASA Headquarters, NASA Langley, AFOSR,...

OVERALL GOAL:

Serve as focal point for CST development (including modeling, analysis, sensitivity studies, optimization and use of AI methods)

SPECIFIC OBJECTIVES:

- *Conduct innovative reseach* on advanced topics of CST
- *Act as pathfinder*, by demonstrating what can be done (high-potential, high-risk research)
- *Help in identifying future directions* for research
- *Help in the rapid transfer of research results and broaden awareness of the state-of-the-art* in CST as well as other areas of computational technology that can impact CST (serve as central clearing house for information)

Figure 17

FUTURE DIRECTIONS FOR RESEARCH

To accomplish its mission the Center will carry out the following four major activities (Fig. 18):

1) *research.* This will be done in strong collaboration with NASA (Langley and Lewis), UVA faculty, industry, and university researchers;

2) *organize a series of seminars*, workshops and national symposia;

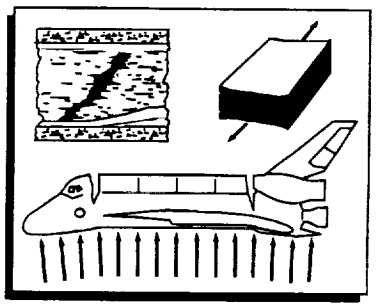
3) *write survey papers*, special publications and state-of-the-art monographs on timely topics; and

4) *publish quarterly newsletter* listing CST research activities at various government laboratories, research centers and universities, as well as recent contributions on selected topics.

Major Pacing Items

- **High fidelity modeling of material response, structural, geometrical and topological details, environmental effects (e.g., boundary-layer transition, interference heating)**
- **Life prediction and analysis of failure for structural components and structures made of new materials**

- **Effective computational strategies for large-scale problems**
  - **Integrated fluid/thermal/structural analysis**
  - **Sensitivity analysis**
  - **Multidisciplinary design and optimization**
- **Validation and assessment of reliability of aerodynamic/thermal/structural response predictions.**



Related Tasks

- **Predata, postdata processing and effective use of visualization technology**
- **Integration of analysis programs into CAD/CAE and concurrent engineering systems**

Figure 18

UVA CENTER FOR CST

The initial research projects selected for the Center include (see Fig. 19):

• *Design-oriented CST* with application to high-speed transport and large flexible spacecrafts. This activity will focus on effective technique for evaluating the sensitivity derivatives, optimization techniques and AI methods for large systems.

• *Innovative computational strategies* for large-scale and coupled problems. This activity will include hierarchical adaptive modeling strategies, hybrid analysis techniques, novel partitioning strategies, effective use of artificial neural networks, and development of intelligent/smart computational modules.

INITIAL RESEARCH PROJECTS:

- ***Design-oriented CST*** **(with application to high-speed transport and large flexible spacecrafts)**
  - **Optimization, sensitivity analysis and AI methods**
- ***Innovative computational strategies*** **for large-scale structural and coupled problems.**
  - **Hierarchical adaptive modeling**
  - **Hybrid techniques**
  - **Novel partitioning strategies**
  - **Neural networks**
  - **Intelligent computational modules**

Figure 19

UVA CENTER FOR CST (Cont'd.)

The initial research projects also include the following four (Fig. 20):

• *Computational modeling of high-temperature multilayered composites.* This includes development of micromechanical models for thermoviscoplastic analysis of composites, thermal lamination theories for heat transfer analysis (Refs. 4 and 5), analytic three-dimensional thermoelastic models for thermal buckling (Refs. 6, 7 and 8), and effective predictor-corrector procedures for the accurate prediction of the thermal buckling and postbuckling responses (Refs. 9, 10 and 11).

• *High fidelity modeling* of flight-vehicle and engine structures. This includes: hierarchical (multilevel/multiscale) computational models, effective coupling of numerical simulations and experiments, and modeling of joints and damping.

• *Failure analysis* and mechanisms of failure of structural components made of new materials.

• *Quality assessment*, adaptive control and validation of numerical simulations.

INITIAL RESEARCH PROJECTS:

- *Computational modeling* of high-temperature multilayered composites
  - Micromechanical models
  - Thermal lamination models for heat transfer
  - Computational models for thermal buckling and postbuckling
- *High-fidelity modeling* of flight-vehicle and engine structures
  - Hierarchical (multilevel/multiscale) computational material models
  - Effective coupling of numerical simulations and experiments
  - Modeling of joints and damping
- *Failure analysis and mechanisms of failure* of structural components made of new materials
- *Quality assessment and adaptive control* of numerical solutions; and validation of numerical simulations

Figure 20

CHARACTERISTICS OF AN EFFECTIVE COMPUTATIONAL STRATEGY FOR LARGE STRUCTURAL SYSTEMS

The remainder of the presentation is devoted to a description of one of the research projects at the Center, viz., development of effective computational strategies for large-scale and complex structural systems.

The three major characteristics of an effective computational strategy are listed in Fig. 21.

First, the strategy should give physical insight about the response. This is accomplished by using hierarchical adaptive modeling - in the sense of starting from a simpler model and increasing the level of sophistication, as needed, to model the actual structure. As was suggested by Einstein, "The model used must be the simplest possible one, but not simpler."

The *second characteristic* of the strategy is that it should help in assessing the adequacy of the computational model. This is accomplished by obtaining sensitivity information about the modeling details as part of the analysis.

The *third characteristic* of the strategy is that it should be highly efficient, which is accomplished by linking the degrees of freedom used in the initial discretization, and by exploiting the major features of new computing systems (vector, parallel and AI capabilities).

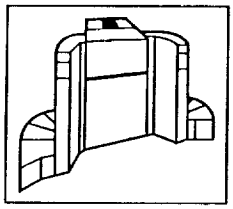
Characteristics	Accomplished by
• Gives physical insight about response	• Hierarchical adaptive modeling - start from a simpler model and increase the level of sophistication, as needed, to model the actual structure *"The model used must be the simplest possible one, but not simpler."*
• Helps in assessing adequacy of computational model	• Obtain sensitivity information about modeling details
• High computational efficiency 	• Reduce number of degrees of freedom used in original discretization • Exploit major features of new computing systems (vector, parallel and AI capabilities)

Figure 21

BASIC IDEA OF PROPOSED COMPUTATIONAL STRATEGIES

The basic idea of the strategies, which appear to satisfy the three criteria, is to generate the response of a complex system using *large perturbations* from that of a simpler model associated with either the simpler system or a simpler mathematical/discrete model of the original system (Fig. 22). The discrete equations of the simpler system are then embedded into those of the original complex system. Sensitivity derivatives of the response of the system with respect to modeling details neglected, are directly available at very small computational cost. The response of the simpler model is then used as a predictor and an iterative process (e.g., preconditioned conjugate gradient-PCG or multigrid-MG) is applied to generate the response of the original model.

- **Response of a complex system is generated using large perturbations from that of a simpler model associated with either:**
  - **Simpler system**
  - **Simpler mathematical/discrete model of original system**
- **Discrete equations of simpler system are embedded into those of original complex system**
- **Sensitivity derivatives of the response of the system with respect to modeling details neglected, are directly available**
- **Response of simpler model used as a predictor and an iterative process (PCG or MG) is applied to generate response of original model**

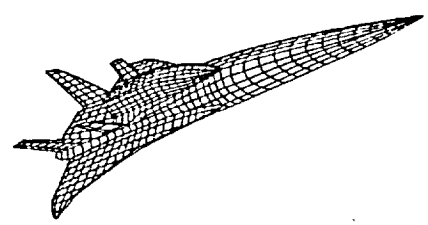


Figure 22

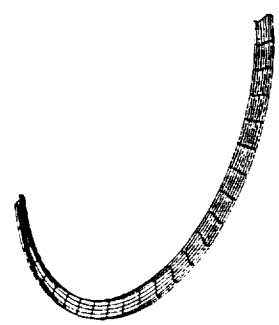
APPROACHES FOR SELECTING SIMPLER MODEL

Two general approaches for selecting the simpler model are outlined in Fig. 23 (Ref. 12).

The first is *hierarchical modeling* which includes the classical multigrid technique and the physical multigrid approach based on selecting a simpler model, associated with a mathematical model of lower dimensionality. The interpolation and restriction operators reflect the physical assumptions of that mathematical model.

The second approach is the *decomposition or partitioning strategy* which can be thought of as physical domain decomposition. It is based on either uncoupling of different fields in coupled problems, or uncoupling of the load-carrying mechanisms in structural problems, or using symmetry transformations. The two approaches are briefly discussed in the succeeding figures.

- ***Hierarchical modeling (multimodel or multigrid)***
  - **Mathematical model of a lower dimensionality (multimodel or physical multigrid-PMG)**
  - **Coarse finite element grid (classical multigrid)**



- ***Decomposition or partitioning (physical domain decomposition-PDD)***
  - **Uncoupling of different fields in coupled problems (e.g., aerodynamics, thermal and mechanical fields)**
    - **Uncoupling of load-carrying mechanisms in structural problems (e.g., extensional and bending components)**
  - **Symmetry transformations**

Figure 23

HIERARCHICAL MODELING STRATEGY

The application of the hierarchical modeling strategy to a composite airframe is outlined in Fig. 24. The structure is modeled by using two-dimensional plate and shell elements. The resulting discrete model is referred to as the actual structure. The simpler structure corresponds to a one-dimensional thin-walled beam model. Although the governing discrete equations for both the actual and simpler structures are similar, those of the simpler structure are much smaller in number.

The degrees of freedom of the actual structure are related to those of the simpler model by the interpolation operator $[\Gamma]$, which reflects the basic assumptions used in the dimensionality reduction (from two-dimensional shell model to one-dimensional thin-walled beam model). As usual the restriction operation is taken to be $[\Gamma]^t$. The relations between the stiffness and load vectors of the actual structure and the simpler model are given in Fig. 24.

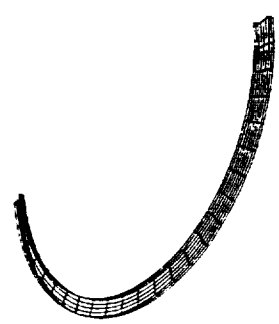
Governing equations:

Actual structure $[K]\{Z\} = \{Q\}$

Simpler structure $[k]\{z\} = \{q\}$

Relationship between actual structure and simpler model:

$\{Z\} = [\Gamma]\{z\}$



Interpolation $[\Gamma]$ reflects basic assumptions in dimensionality reduction

$[\Gamma]^t$ = restriction operator

$[k] = [\Gamma]^t[K][\Gamma]$

$\{q\} = [\Gamma]^t\{Q\}$

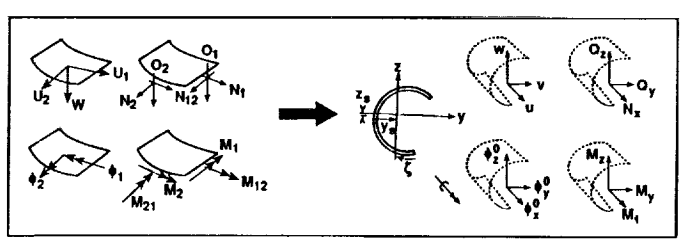


Figure 24

GEOMETRIC INTERPRETATION OF SYMMETRY TRANSFORMATION APPROACH

The geometric interpretation for the symmetry transformation approach, as applied to an unsymmetric domain, is given in Fig. 25. The response vector $\{Z\}$ is partitioned into two equal length subvectors, $\{Z\}_1$ and $\{Z\}_2$, and can be written in the form shown in Fig. 25. The equation in Fig. 25 is a statement of the fact that each unsymmetric vector can be written as the sum of a symmetric and an antisymmetric component. Each of the two vectors can be obtained by applying a matrix transformation to the original vector. The transformation matrices are shown in the figure.

It is important to note that each of the symmetric and antisymmetric components of the response vector requires only half the model for their determination. For an unsymmetric domain the equations governing $\{Z\}_s$ and $\{Z\}_{as}$ are coupled. The simpler model corresponds to the uncoupled set of equations. The symmetry transformation process can be repeated to effect further reduction in the size of the model (further partitioning of the domain). The strategy has several advantages including its suitability for parallelism.

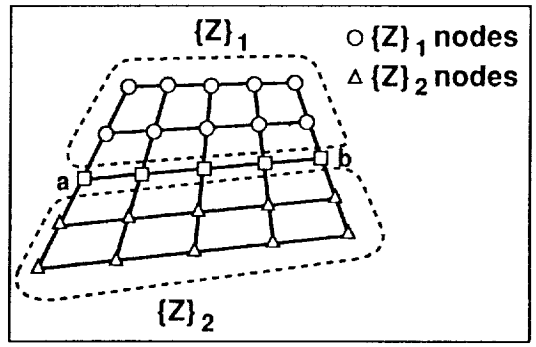
Decomposition and Transformation matrices

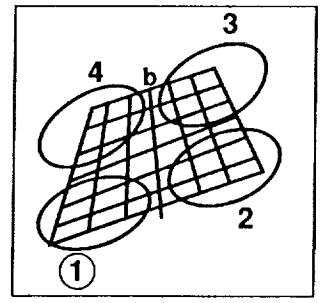
$$\begin{Bmatrix} Z_1 \\ Z_2 \end{Bmatrix} = \begin{Bmatrix} \frac{1}{2}(Z_1 + Z_2) \\ \frac{1}{2}(Z_1 + Z_2) \end{Bmatrix} + \begin{Bmatrix} \frac{1}{2}(Z_1 - Z_2) \\ -\frac{1}{2}(Z_1 - Z_2) \end{Bmatrix}$$

$$= \{Z\}_s + \{Z\}_{as}$$

$$= \left[[T]_s + [T]_{as}\right]\{Z\}$$

$$[T]_s = \begin{bmatrix} \frac{1}{2}[I] & \frac{1}{2}[I] \\ \frac{1}{2}[I] & \frac{1}{2}[I] \end{bmatrix}, \quad [T]_{as} = \begin{bmatrix} \frac{1}{2}[I] & -\frac{1}{2}[I] \\ -\frac{1}{2}[I] & \frac{1}{2}[I] \end{bmatrix}$$





Comments on computational procedure

- Each of $\{Z\}_s$ and $\{Z\}_{as}$ can be determined by using a smaller (reduced-size) model or subdomain
- Simpler model corresponds to uncoupled set of equations in $\{Z\}_s$ and $\{Z\}_{as}$

Figure 25

OPERATOR SPLITTING AND ITERATIVE SOLUTION PROCESS

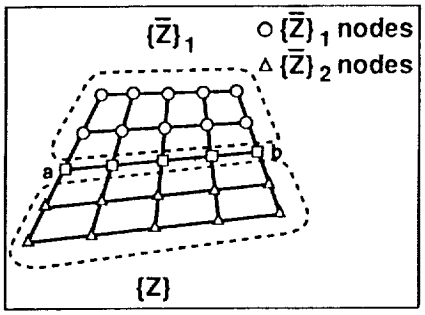
The application of operator splitting and iterative solution process, in conjunction with the partitioning strategy, is outlined in Fig. 26.

The vector of fundamental unknowns is partitioned into two subvectors $\{\bar{Z}_1\}$ and $\{\bar{Z}_2\}$. These correspond to the degrees of freedom associated with either different load-carrying mechanisms, or symmetric/antisymmetric components of response. The discrete equations are partitioned accordingly. The coupling terms are identified by the parameter λ. The simpler structure corresponds to the case λ=0 (uncoupled equations). The PCG iterative process is used for generating the response of the actual structure.

Partitioned Equations and Unknowns

$$\left(\begin{bmatrix} K_{11} & \cdot \\ \cdot & K_{22} \end{bmatrix} + \lambda \begin{bmatrix} \cdot & K_{12} \\ K_{21} & \cdot \end{bmatrix}\right) \begin{Bmatrix} \bar{Z}_1 \\ \bar{Z}_2 \end{Bmatrix} = \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix}$$

where λ is a tracing parameter.
$\lambda = 1 \rightarrow$ original equations
$\lambda = 0 \rightarrow$ simpler-structure equations



	Uncoupling of load-carrying mechanisms	Symmetry transformation
$\{\bar{Z}_1\}$, $\{\bar{Z}_2\}$ are associated with	different load-carrying mechanisms (e.g., membrane and bending)	symmetric/antisymmetric components of response vector

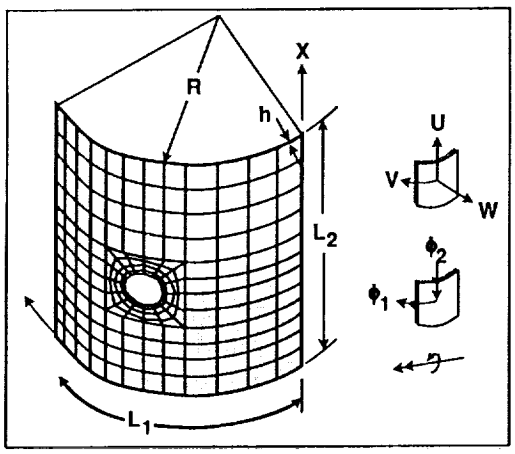
Iterative Solution Process

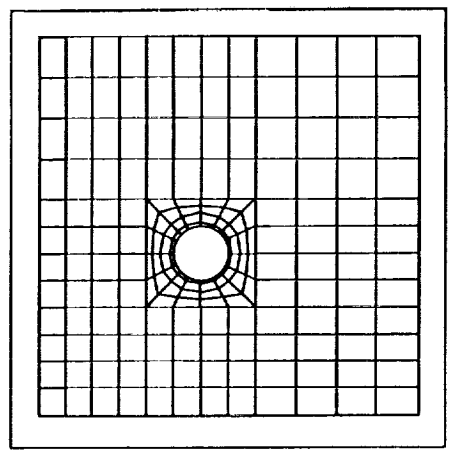
- **PCG used for solution**
- **Left-hand-side matrix corresponding to $\lambda = 0$ used as a preconditioner**

Figure 26

APPLICATION TO NONLINEAR DYNAMIC ANALYSIS
Cylindrical panel with cutout

The strategy was applied to a nonlinear dynamic problem of a composite cylindrical panel with an off-center circular cutout subjected to uniform pressure loading (Fig. 26a).





3818 nonzero displacement degrees of freedom
6144 stress degrees of freedom

Loading

Uniform normal loading with intensity p_o

Boundary conditions

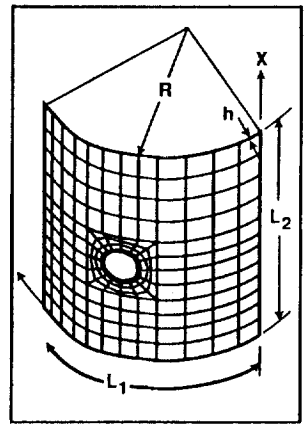
At $x = 0, L_1$:
$u = v = w = \phi_1 = \phi_2 = 0$
At $y = 0, L_2$:
$w = \phi_1 = 0$

Figure 26a

PERFORMANCE EVALUATION OF STRATEGY ON CRAY-Y MP4/432 (MENDOTA HEIGHTS)

The performance of the proposed strategy on the CRAY-YMP4/432 at Mendota Heights is shown in Fig. 27. The speedup resulting from the use of four processors was over an order of magnitude. The details of the study are given in Ref. 12.

	Full Structure (Optimized Code) (278 MFLOPS)	Partitioned Structure (Nearly Optimized Code) (246 MFLOPS)
Number of degrees of freedom	3818 displacements 6144 stress	971 displacements 1536 stresses
Semibandwidth of equations	700	315
Wall clock time (sec.) (first ten steps)	171	58.6 one processor 29.7 two processors 16.4 four processors
Speedup	1.0	2.92 one processor 5.76 two processors 10.43 four processors



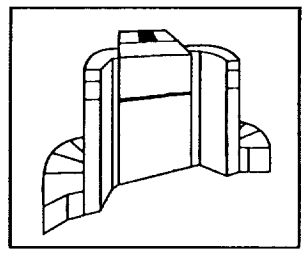


Figure 27

SUMMARY

In summary (Fig. 28), the goals of CST have been described. Future high performance computing environment, technical needs, and future directions for research in support of planned and projected aeronautical and space systems have been identified. The mission and activities of the newly established UVA CST Center have been described, along with the details of one of the research projects.

CST has greatly enhanced our understanding of physical phenomena associated with the response and failure of structures. CST provides a valuable complement to experimental and analytical methods for flight-vehicle structures. Its future development requires strong interaction with researchers in other fields (for example, computational mathematics, CFD, and computational electromagnetics).

The future of CST is bright; advanced computational material models, smart/intelligent computational tools for structural analysis and synthesis and future high performance computers should contribute significantly to the development of structures technology, as well as to improving the design of future flight vehicles.

- **Goals of CST described**
- **Future high-performance computing environment; technical needs and future directions for research, in support of aeronautical and space systems, identified**
- **Mission and activities of UVA CST Center described**
- **CST technology has greatly enhanced our understanding of physical phenomena associated with high temperatures**
- **Future development of CST requires strong interaction with researchers in other fields (e.g., computational mathematics, CFD, and computational electromagnetics).**

Figure 28

REFERENCES

1. Noor, A. K. and Venneri, S. L. "Advances and Trends in Computational Structures Technology," Computing Systems in Engineering, Vol. 1, No. 1, 1990, pp. 23-36.
2. Noor, A. K. and Atluri, S. N., "Advances and Trends in Computational Structural Mechanics," AIAA Journal, Vol. 25, 1987, pp. 977-995.
3. Grandhi, R. V., Stroud, W. J. and Venkayya, V. B. (eds), Computational Structural Mechanics and Multidisciplinary Optimization, AD Vol. 16, American Society of Mechanical Engineers, New York, 1989.
4. Noor, A. K. and Burton, W. S., "Steady-State Heat Conduction in Multilayered Composite Plates and Shells," Computers and Structures, Vol. 39, No. 1/2, March 1991, pp. 185-193.
5. Noor, A. K. and Tenek, L.H., "Steady-State Nonlinear Heat Transfer in Multilayered Composite Panels," Journal of Engineering Mechanics, ASCE 1992.
6. Noor, A. K. and Burton, W. S., "Three-Dimensional Solutions for Thermal Buckling of Multilayered Anisotropic Plates," Journal of Engineering Mechanics, ASCE 1992.
7. Noor, A. K. and Burton, W. S., "Three-Dimensional Solutions for the Free Vibrations and Buckling of Thermally Stressed Multilayered Angle-Ply Composite Plates," Journal of Applied Mechanics 1992.
8. Noor, A. K. and Burton, W. S., "Three-Dimensional Solutions for the Thermal Buckling and Sensitivity Derivatives of Temperature-Sensitive Multilayered Angle-Ply Plates," Journal of Applied Mechanics 1992.
9. Noor, A. K. and Burton, W. S., "Predictor-Corrector Procedures for Thermal Buckling of Multilayered Composite Plates," Computers and Structures, Vol. 40, No. 5, 1991, pp. 1071-1084.
10. Noor, A. K. and Peters, J. M., "Postbuckling of Multilayered Composite Plates Subjected to Combined Axial and Thermal Loads," Finite Elements in Analysis and Design 1992.
11. Noor, A. K. and Peters, J. M., "Thermal Postbuckling of Thin-Walled Composite Stiffeners," Computing Systems in Engineering, Vol. 2, No. 1, 1991, pp. 1-16.
12. Noor, A. K. and Peters, J. M., "Strategies for Large-Scale Structural Problems on High-Performance Computers," Communications in Applied Numerical Methods, Vol. 7, 1991, pp. 465-478.

N92-25913

Computer Codes Developed and Under Development at Lewis

Christos C. Chamis
NASA Lewis Research Center
Cleveland, Ohio

PRECEDING PAGE BLANK NOT FILMED

The objective of this summary is to provide a brief description of: (1) Codes developed or under development at Lewis and (2) the development status of IPACS with some typical early results.

The computer codes that have been developed and/or are under development at Lewis Research Center are listed in the accompanying Charts 1 - 5. This list includes (1) the code acronym, (2) select physics descriptors, (3) current enhancements and (4) present (9/91) code status with respect to its availability and documentation. The computer codes list is grouped by related functions such as: (1) Composite Mechanics, (2) Composite Structures, (3) Integrated and 3-D Analysis, (4) Structural Tailoring, and (5) Probabilistic Structural Analysis. These codes provide a broad computational simulation infrastructure (technology base-readiness) for assessing the structural integrity/durability/reliability of propulsion systems. These codes serve two other very important functions: (1) They provide an effective means of technology transfer and (2) they constitute a depository of corporate memory.

CODES DEVELOPED BY AND ARE AVAILABLE FROM THE STRUCTURAL MECHANICS BRANCH, 2 Oct 91

Probabilistic Structural Analysis

Code Name	Description/Current Enhancements	Status
NESSUS	Probabilistic Structural Analysis includes MHOST as the analysis module/component risk and reliability	Available with documentation
CLS	Probabilistic Loads Simulation for Space Shuttle Main Engine components/loads simulation based on deterministic models	Available with documentation
IPACS	Probabilistic Structural Analysis of Composites/ Couple PCAN with NESSUS	Development

Chart 1

CODES DEVELOPED BY AND ARE AVAILABLE FROM THE STRUCTURAL MECHANICS BRANCH, 2 Oct 91

Structural Tailoring

Code Name	Description/Current Enhancements	Status
STAHYC	Structural Tailoring of Hypersonic Structures/ Computational efficiency, documentation	Operational
STAEBL	Structural Tailoring of Engine Blades	Completed & documented
STAEBL/ GENCOMP	Structural Tailoring of General Composites Structures/Improved finite element and documentation	Operational no documentation
STAEBL/AERO	Structural Tailoring of Engine Blades for Aero-dynamic Performance and Flutter	Completed and documented
STAEBL/TURBINE	Structural Tailoring of Turbine Blades	Completed & documented
STAT	Structural Tailoring of Swept Turboprops	Completed & documented
CSTEM	Coupled Structural/Thermal/Electromagnetic Tailoring/Gain familiarity	Available with documentation

Chart 2

CODES DEVELOPED BY AND ARE AVAILABLE FROM THE STRUCTURAL MECHANICS BRANCH, 2 Oct 91

Integrated and 3-D Inelastic Analyses

Code Name	Description/Current Enhancements	Status
MHOST	Lewis-owned Finite Element Analysis Computer Code - based on mixed iterative scheme/add mix-element and large problem capabilities	Available with documentation
3-D INAN	A compilation of 9 different 3-D finite element codes for nonlinear structural and stress analysis with progressive level of sophistication, nonlinear simulation models/Dormant - needs code user familiarity	Available with documentation
BEST3D	Boundary Element Structural Analysis code with heat transfer	Available with documentation
ESMOSS	Engine Structures Modeling Software System/ Dormant - needs code user familiarity	Available with documentation
COSMO	Component Specific Modular for Hot Engine Structures/Dormant - needs code user familiarity	Available with documentation

Chart 3

CODES DEVELOPED BY AND ARE AVAILABLE FROM THE STRUCTURAL MECHANICS BRANCH, 2 Oct 91

Composite Structures

Code Name	Description/Current Enhancements	Status
COBSTRAN	Composite Blade Structural Analysis/Add MHOST, add updated ICAN	Operational
CODSTRAN	Composite Durability Structural Analysis/ Extension to complete structures	Development
HITCAN	High Temperature Composite Structural Analysis/ Computational efficiency, documentation	Operational

Chart 4

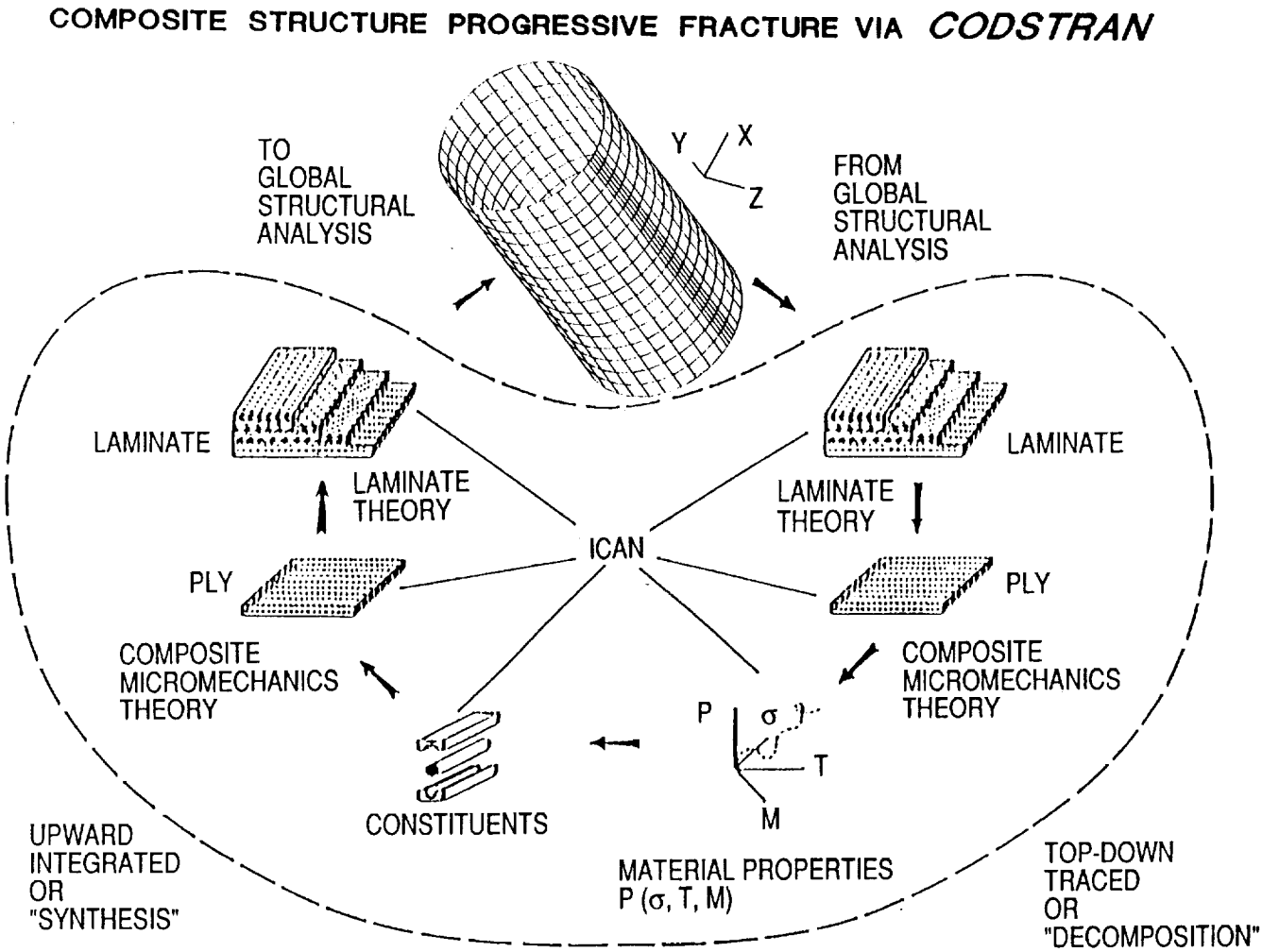
CODES DEVELOPED BY AND ARE AVAILABLE FROM THE STRUCTURAL MECHANICS BRANCH, 2 Oct 91

Composite Mechanics

Code Name	Description/Current Enhancements	Status
ICAN	Integrated Composite Analyzer - PMC/ Incorporate damping capability	Available from COSMIC
PICAN	Probabilistic Integrated Composite Analyzer/ Probabilistic capability to ICAN	Development
METCAN	Metal Matrix Composite Analyzer/Time and unloading features; documentation	Operational sketchy documentation
CEMCAN	Ceramic Matrix Composite Analyzer - based on progressive ply/fiber substructuring	Development
ICAN/SCS	Integrate Composite Analysis for Structural Composite Sandwiches/Dormant - needs stress recovery capability	Available with documentation
MMLT	Metal Matrix Laminate Tailoring/Capability to tailor for specific life	Development

Chart 5

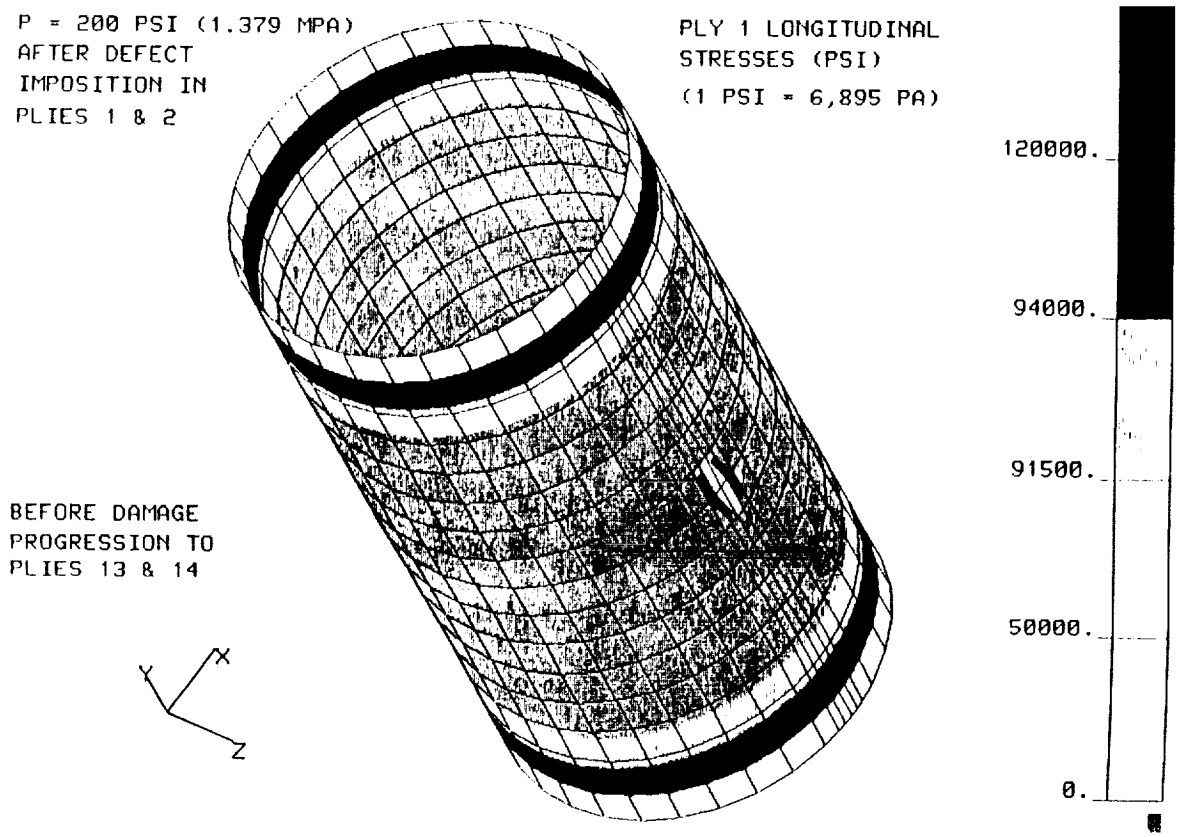
The schematic in Figure 1 represents an excellent illustration of the physics embodied in an integrated computer code for the computational simulation of progressive fracture in composite structures. The code synthesizes the effects of the local constituent materials behavior to the global structural response through the various scales inherent in composites. And at the same time it progressively decomposes the global structural response to the corresponding local constituent materials state which permits continuous feedback for local damage initiation accumulation and growth. Thus, the computational simulation closes the loop between local material capability and global structural performance where component/system specific design concepts are evaluated. COBSTRAN and HITCAN have structures similar to CODSTRAN.



Simulation of Composite Damage and Fracture Propagation via CODSTRAN

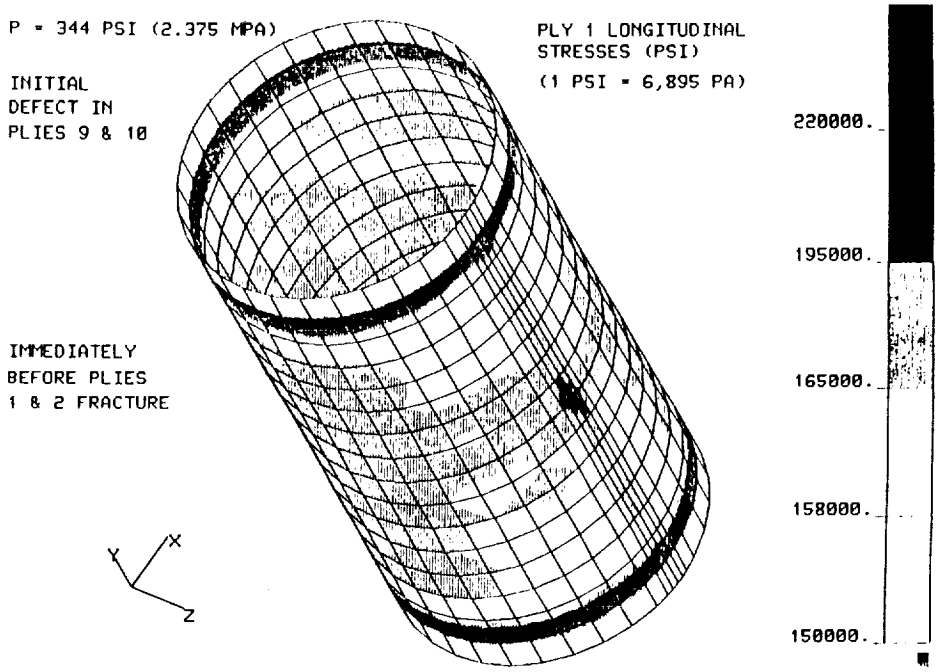
Figure 1

Typical results obtained by using CODSTRAN to assess the deformation evolution during progressive fracture in this composite shell with a longitudinal surface slit and subjected to internal pressure are shown in Figure 2. The deformation is highly localized exemplifying the difficulty of locating sensors to capture this deformation evolution in either verification tests or in service health monitoring. The corresponding deformation pattern for an embedded slit is shown in Figure 3. The deformation evolution is even more localized in this case and remains so until the shell fractures catastrophically. The difference in the damage tolerance, with respect to internal pressure and to some inspection plan, between the location of the damage is shown in Figure 4. The shell with the surface defect exhibits a limited damage growth and higher internal pressure (damage tolerance) prior to structural fracture. However, that with the embedded defect exhibits "brittle-fracture-like" behavior. Obviously, these two rather simplistic examples demonstrate the usefulness of computational simulation in all aspects of the structural component/system development process.



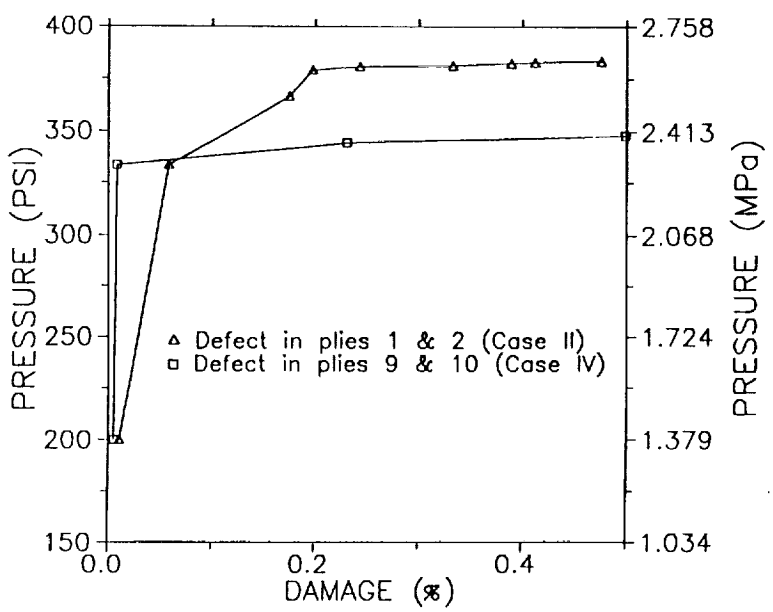
Ply 1 Longitudinal Stresses after Initial Fracture of Plies 1 and 2
Composite Shell T300/Epoxy[$90_2/\pm15/90_2/\pm15/90_2/\mp15/90_2$]

Figure 2



Ply 1 Longitudinal Stresses at 2.375 MPa; before Ply 1 Fractures
Composite Shell T300/Epoxy[$90_2/\pm15/90_2/\pm15/90_2/\mp15/90_2$]

Figure 3



Damage Propagation with Pressure
Composite Shell T300/Epoxy[$90_2/\pm15/90_2/\pm15/90_2/\mp15/90_2$]

Figure 4

Modules from the various codes in the list can be stacked up (assembled) to develop computational simulation capability for specific structural response. The schematic in Figure 5 illustrates a combination to evaluate acoustic fatigue in composite panels in a hygrothermal environment. The combination of codes include: (1) CSTEM for acoustic source, (2) ICAN for composite properties synthesis, (3) MHOST for structural dynamic analysis due to acoustic excitations, (4) ICAN for ply stresses and strengths and (5) ICAN for cyclic load effects.

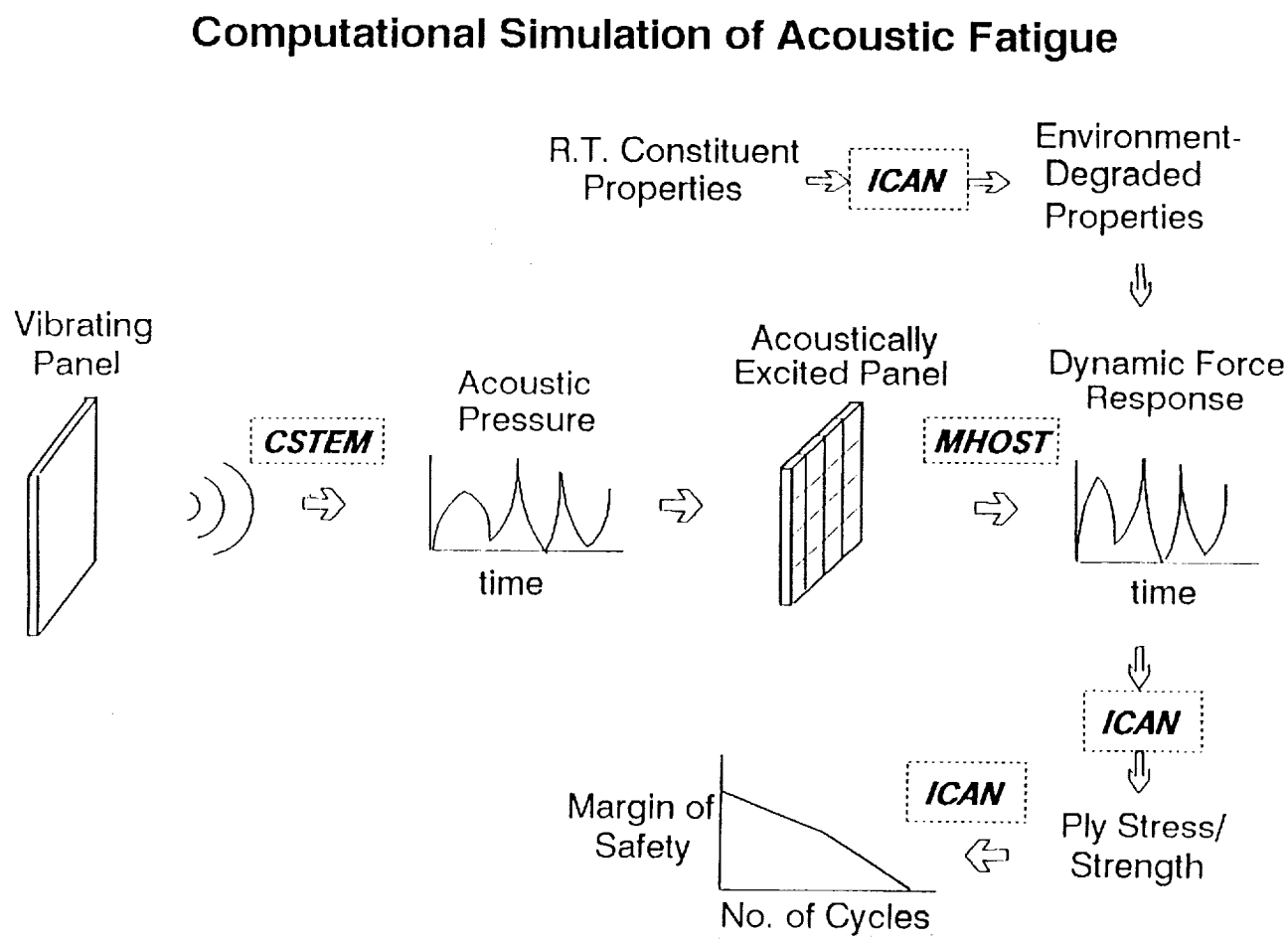


Figure 5

Typical results from the computational simulation of acoustic fatigue on composites are shown in Figure 6 where the remaining strength in terms of margin of safety is plotted versus the number of cycles. Three different curves are shown: (1) The bottom curve is the base case. It is for a $[(0/\pm 45/90_2)_3]$ AS/E laminate at room temperature conditions. (2) The middle curve is for the base laminate at (200°F, 1% moisture by weight) environmental conditions. The top curve is the base laminate with rearranged plies. Environmental effects enhance composite fatigue because they tend to soften the composite. Ply stacking sequences can be selected to significantly increase acoustic fatigue provided that other design requirements are not violated. The significant observation is that the infrastructure available in the computer codes listed in Charts 1 - 5 provide a base to tackle a variety of problems as they arise.

Demonstration: acoustic fatigue

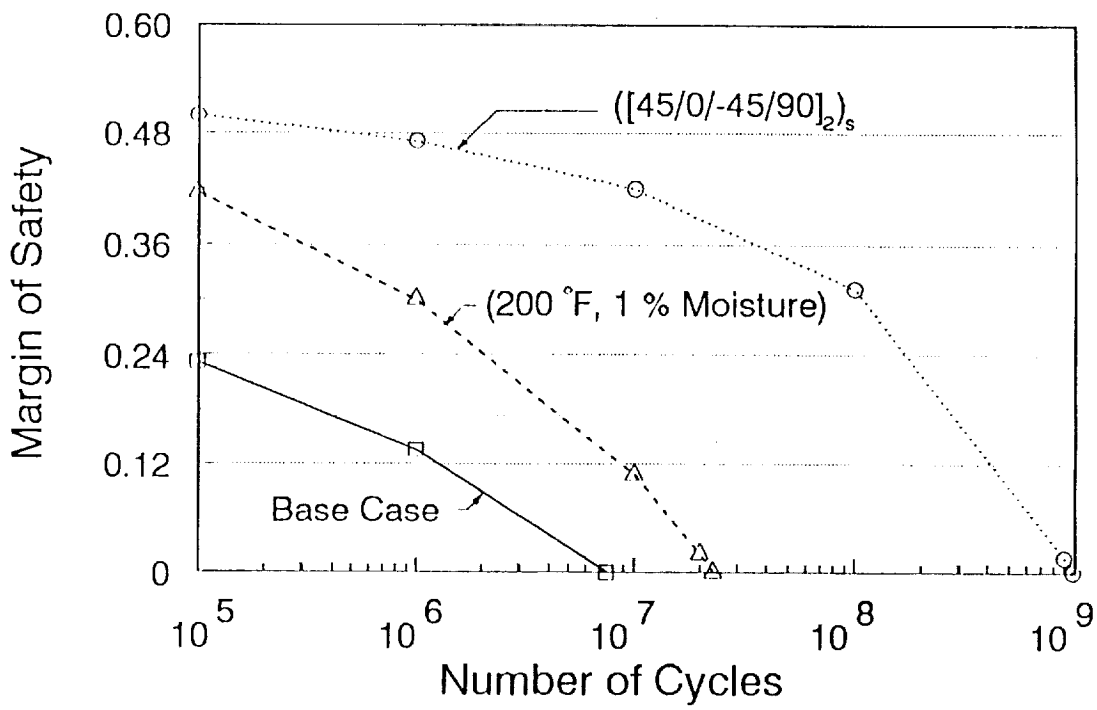


Figure 6

The major elements of the Advanced Composites Technology Program are depicted within the ellipse in Figure 7. These elements include activities in all major aspects of advanced composites technology: (1) Constituent materials development and characterization (top). (2) Demonstration of these constituent materials in simple structural elements (right). (3) Structural components made from these simple elements (bottom right). (4) Analysis methods development (bottom left). (5) Design and fabrication process (left). Another small element of this relatively inclusive program is the development of probabilistic methods to incorporate the uncertainties associated with the major technology elements. These uncertainties are depicted schematically around the ellipse in Figure 7 next to their respective program element.

ADVANCED COMPOSITE TECHNOLOGY
(A Bold New Program in Composites Research & Technology)

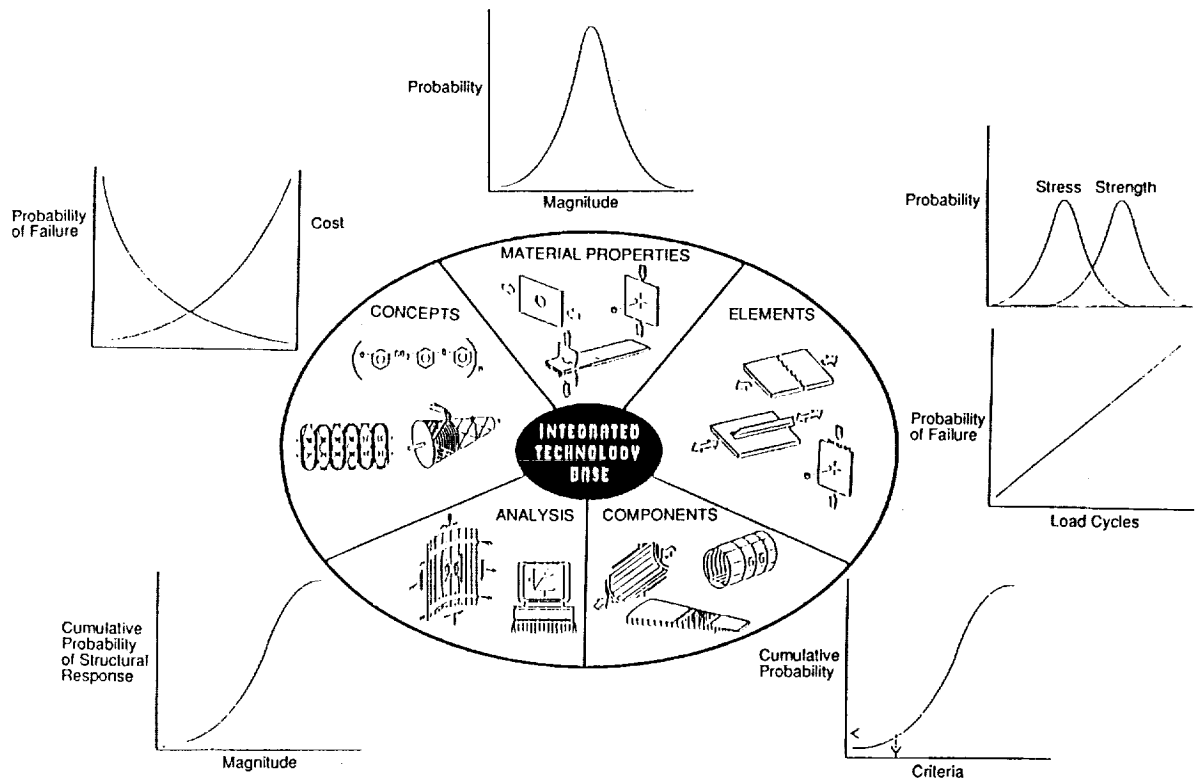
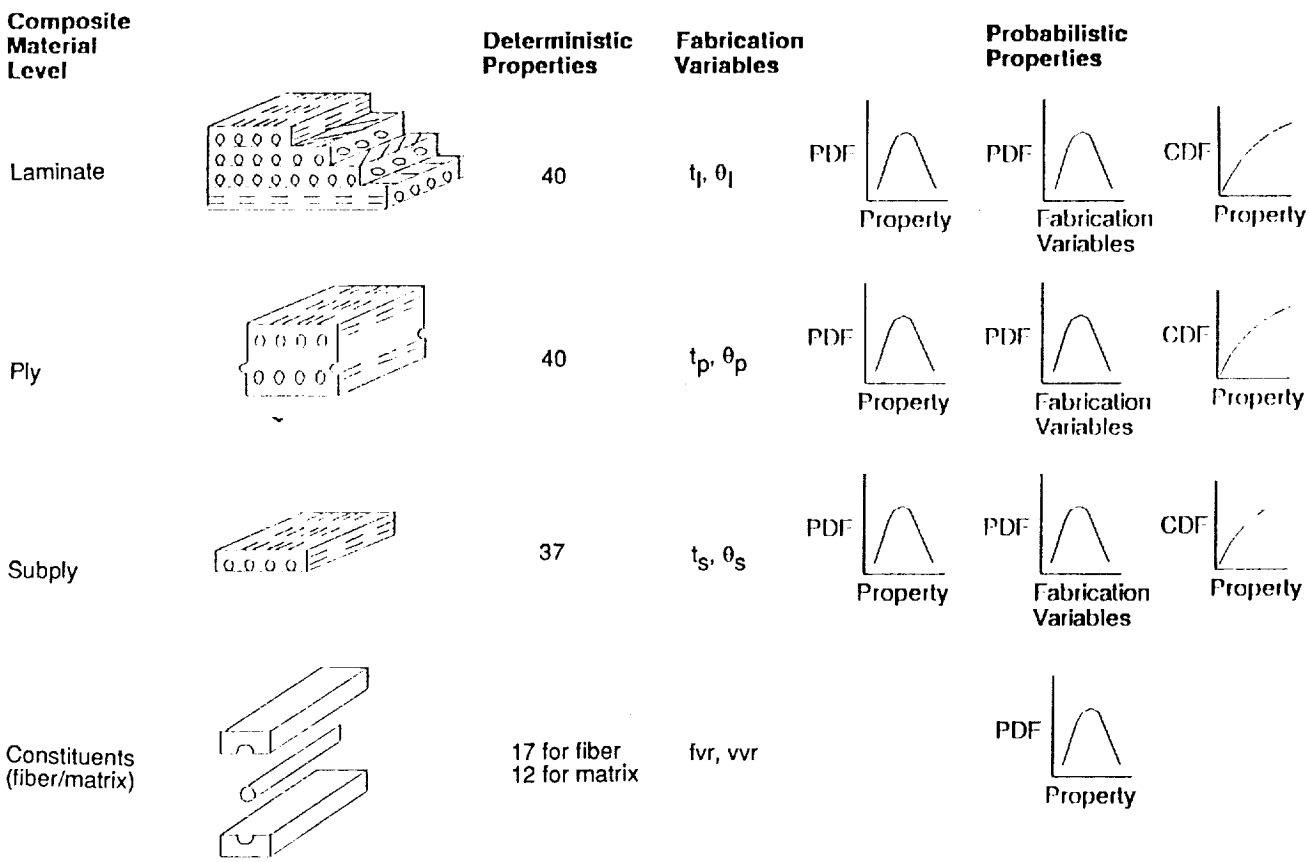


Figure 7

The probabilistic methods to be developed will be embodied into two different computer codes. The first of these is identified as PICAN for Probabilistic Integrated Composite Analyzer. The computational simulation capability in PICAN is schematically illustrated in Figure 8. The code accepts probabilistic uncertainties for 29 constituent material properties and uncertainties for two fabrication process variables (bottom). These uncertainties are integrated upward through the different inherent composite scales using composite mechanics available in ICAN. Additional uncertainties, for respective composite descriptors are introduced as the integration progresses from scale to scale. The result is the probabilistic description of the uncertainties of 40 - different composite properties which are required to fully characterize the composite.

Probabilistic Simulation of Composite Mechanics with PCAN



NOTATION: t = thickness, θ = misalignment, fvr = fiber volume ratio, vvr = void volume ratio, PDF = probability density function
CDF = Cumulative Distribution function, Subscripts l = laminate, p = ply, s = subply

Figure 8

Probabilistically described results for stiffnesses of three different laminates are tabulated in Figure 9. Probabilistic values are given for the mean and for two standard deviations on either side of the mean. Average experimental values are included for comparisons. All the experimental values, except the major Poisson's ratio, (last line) are within the two standard deviations. The results in Figure 9, though preliminary, verify PICAN predictions for laminate stiffness.

PICAN VERIFICATION FOR LAMINATE STIFFNESS

Laminate	lower bound (mean-2σ)	mean	experimental value	upper bound (mean+2σ)
$[0/\pm45_2/0/\pm45]_s$				
Long. modulus (MSI)	5.40	6.19	6.30	6.98
Trans. modulus (MSI)	2.46	3.07	3.08	3.68
Shear modulus (MSI)	3.33	3.84	3.21	4.35
Major Poisson's ratio	0.690	0.806	0.803	0.922
$[0_2/\pm45/0_2/90/0)]_s$				
Long. modulus (MSI)	11.41	13.30	13.00	15.09
Trans. modulus (MSI)	3.69	4.30	4.20	4.96
Shear modulus (MSI)	1.40	1.59	1.50	1.78
Major Poisson's ratio	0.276	0.313	0.325	0.350
$[(0/\pm45/90)_2]_s$				
Long. modulus (MSI)	6.12	7.15	6.68	8.18
Trans. modulus (MSI)	6.12	7.15	6.62	8.18
Shear modulus (MSI)	2.37	2.72	2.34	3.07
Major Poisson's ratio	0.290	0.317	0.350	0.344

Figure 9

The other computer code under development as a part of the ACT program is IPACS, Integrated Probabilistic Assessment of Composite Structures. This consists of PICAN and NESSUS with modules from COBSTRAN for automatic finite element model generation and composite configuration description. A schematic of the computational simulation capability in IPACS is depicted in Figure 10. Preliminary probabilistic results obtained from IPACS of a composite panel, loaded in compression (Figure 11), are tabulated in Figure 12. Inclusion of probabilistic boundary conditions for two panels appear to verify IPACS' predictions for buckling loads. A few summary remarks are listed in Chart 6.

IPACS : Integrated Probabilistic Assessment of Composite Structures

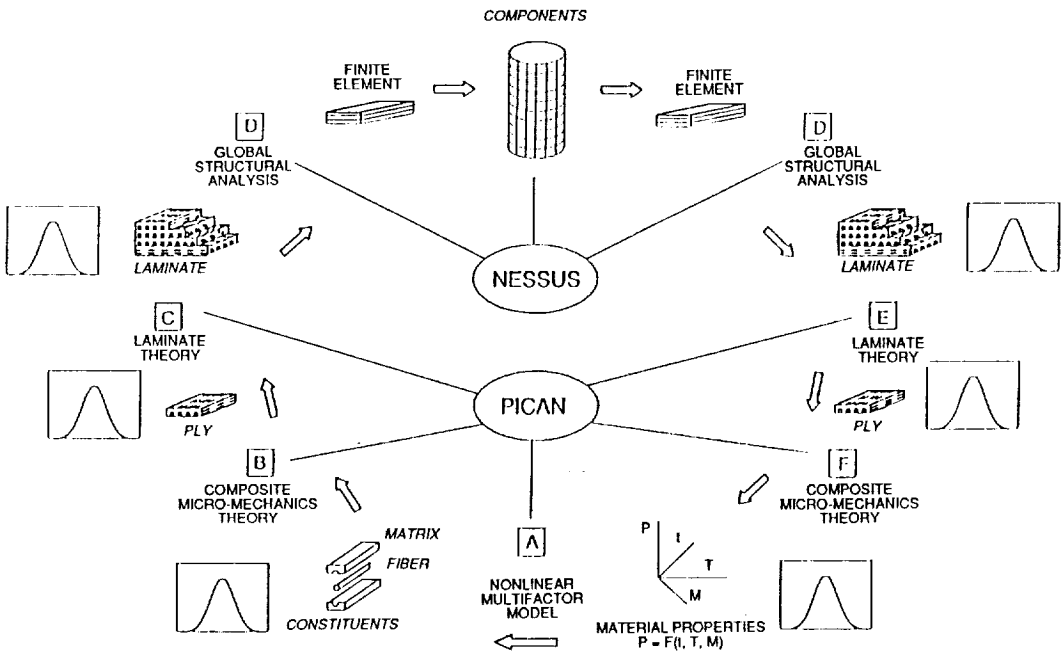


Figure 10

GEOMETRY OF THE PLATE

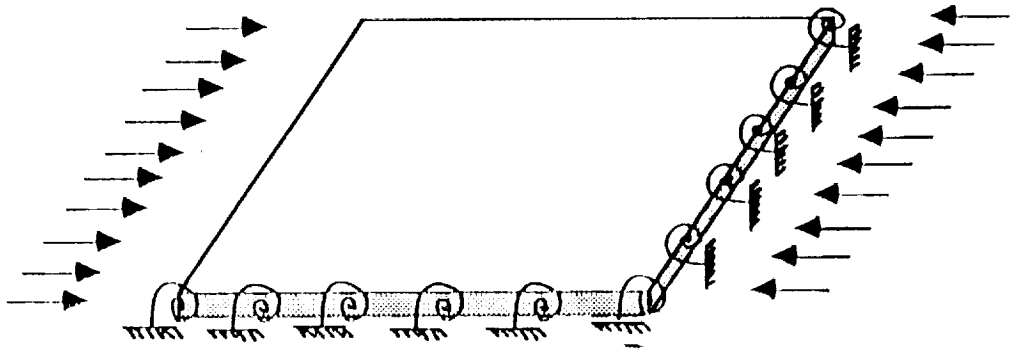


Figure 11

IPACS VERIFICATION FOR BUCKLING LOADS

Laminate	lower bound (mean 2σ)	mean	experimental value	upper bound *(mean $+2\sigma$)
20(0) buckling load	247	284	271	322
20(90) *buckling load	173	195	251	293
10(±30), 10(∓30) *buckling load	513	567	662	688
10(±45), 10(∓45) buckling load	555	609	592	663
10(±60), 10(∓60) buckling load	562	623	661	684

* with uncertainties in the boundary conditions

Figure 12

SUMMARY

- o A variety of computer codes have been developed or are under development at Lewis.

- o These codes were mainly developed for propulsion structures.

- o Current focus is on:
  - High temperature composites
  - Progressive fracture
  - Integrated probabilistic assessment of composite structures

Chart 6

N92-25914

Progress in Integrated Analysis with Adaptive Unstructured Meshing

Pramote Dechaumphai
NASA Langley Research Center
Hampton, Virginia

PRECEDING PAGE BLANK NOT FILMED

INTRODUCTION

Design of lightweight structures and thermal protection systems for hypersonic vehicles depends on accurate prediction of aerothermal loads, structural temperatures and their gradients, and structural deformations and stresses. Several adaptive methods have been investigated at the Aerothermal Loads Branch, NASA Langley Research Center, to improve the fluid, thermal, and structural analysis solution accuracy. These methods include: (1) the h-method where elements in the initial mesh are refined into smaller elements or derefined into larger elements, (2) the p-method where the element geometries are unchanged but the element interpolation polynomials are increased (or decreased), and (3) the r-method where the number of the elements and the element connectivities remain the same but the nodes are relocated. This paper will concentrate on an alternative meshing technique (ref. 1) which generates an entirely new adaptive unstructured mesh based on the solution obtained from the earlier mesh.

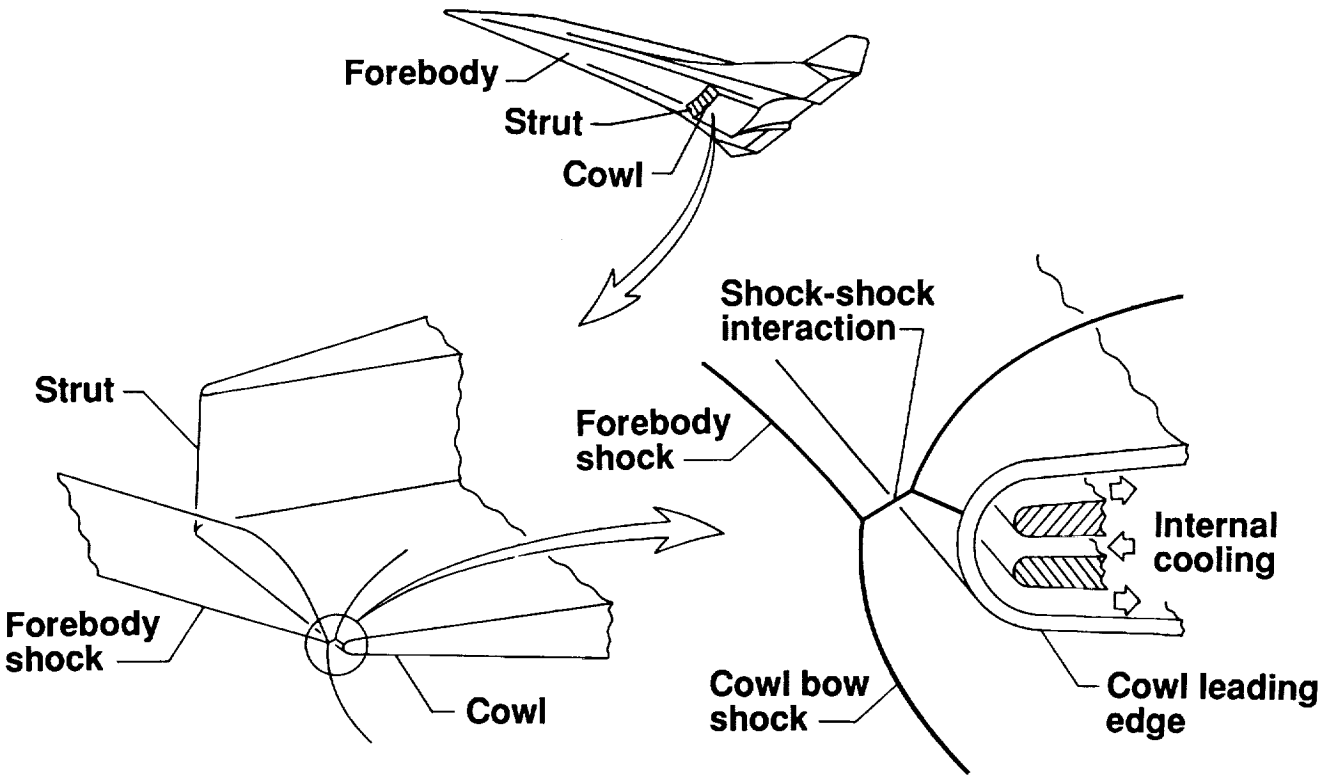
The technique combined with the finite element method has been shown to significantly improve the efficiency and accuracy of the fluid, thermal, and structural analyses (ref. 2). Current capability of the adaptive unstructured meshing technique for the integrated fluid-thermal-structural analysis will be described first. The technique has been extended to transient thermal analysis of structures with time-dependent adaptive meshing to capture the detailed temperature response with a minimum number of unknowns and computational cost. Both linear and higher-order finite elements are implemented to demonstrate the generality of the technique and to investigate their solution accuracy. Currently, the adaptive meshing technique is being developed for plane structures that can be modelled with membrane elements and built-up structures modelled with membrane and bending elements. The capability of the technique to these different disciplinary problems will be demonstrated by several examples in this paper.

- CURRENT CAPABILITY OF INTEGRATED FLUID-THERMAL-STRUCTURAL ANALYSIS WITH ADAPTIVE METHODS
- TIME-DEPENDENT ADAPTIVE UNSTRUCTURED MESHING FOR TRANSIENT THERMAL PROBLEMS WITH:
  - LINEAR ELEMENTS
  - HIGHER-ORDER ELEMENTS
- ADAPTIVE UNSTRUCTURED MESHING FOR STRUCTURAL PROBLEMS:
  - PLANE STRUCTURES WITH MEMBRANE ELEMENTS
  - BUILT-UP STRUCTURES WITH MEMBRANE AND BENDING ELEMENTS
- CAPABILITY DEMONSTRATED BY APPLICATIONS

AEROSPACE PLANE SCRAMJET ENGINE STRUCTURE

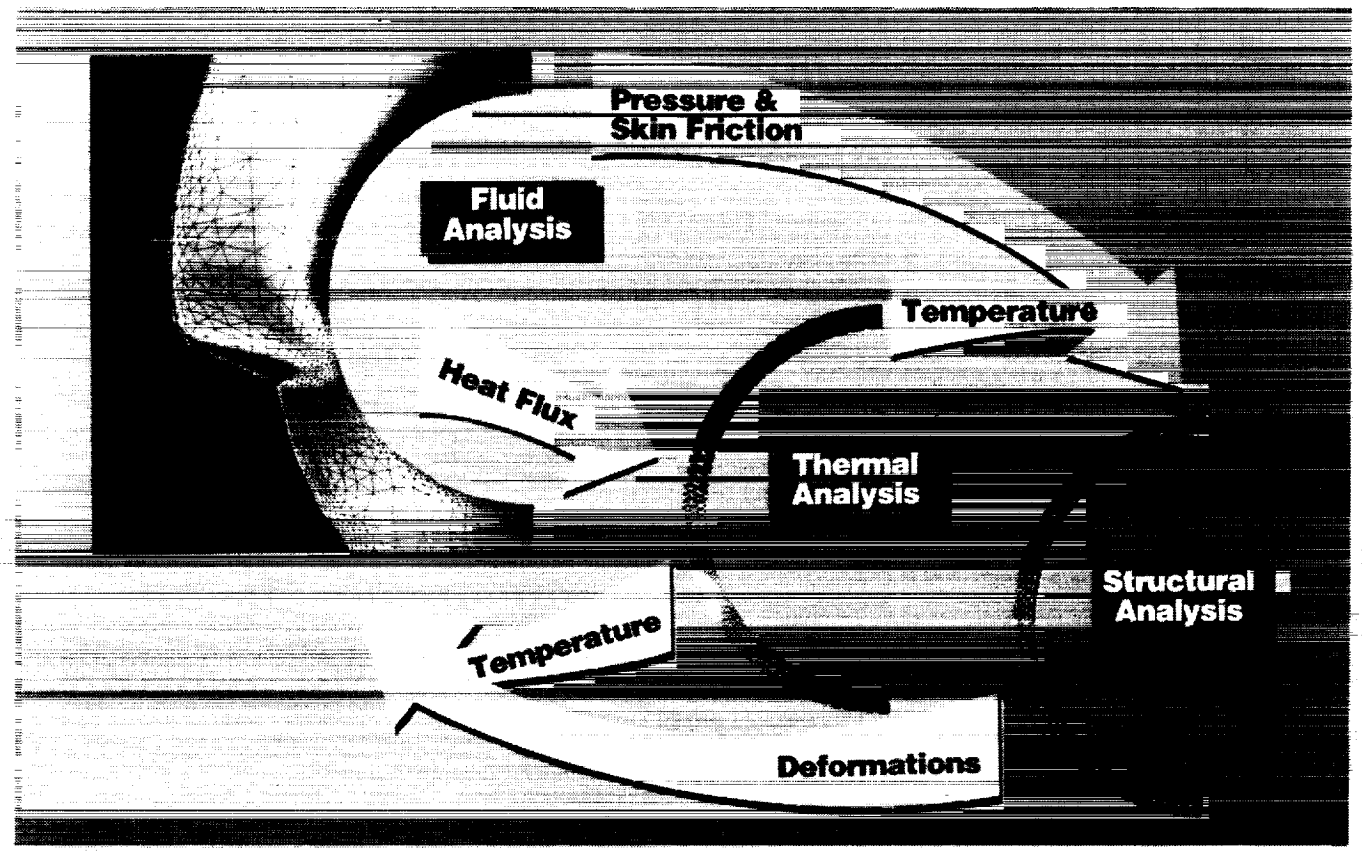
The current capability will be illustrated by a fluid-thermal-structural analysis of a shock-shock interaction on the cowl leading edge of a scramjet engine shown in the figure. As the vehicle accelerates at a high Mach number, the forebody shock sweeps across the engine cowl leading edge (lower left figure) resulting in the shock-shock interaction phenomena shown schematically in the lower right figure. The integrated approach uses the finite element method to solve: (1) the Navier-Stokes equations for the aerodynamic flow field over the leading edge and the aerothermal loads; (2) the structural energy equation for heat transfer and temperature distribution of the cowl leading edge; and (3) the structural equilibrium equations for deformation and stresses of the cowl leading edge. Details of the equations and the finite element analysis solution algorithms are given in Ref. 2.

The problem statement described by the lower right figure will be used to demonstrate the adaptive meshing capability for fluid, thermal, and structural analyses and to assess a new transient adaptive meshing capability for time-dependent thermal analysis. The problem statement described by the lower left figure will be used to describe a new adaptive meshing capability under development for three-dimensional built-up structures. The current integrated fluid-thermal-structural analysis capability with adaptive unstructured meshes for the shock-shock interaction on the cowl leading edge is highlighted in the next page.



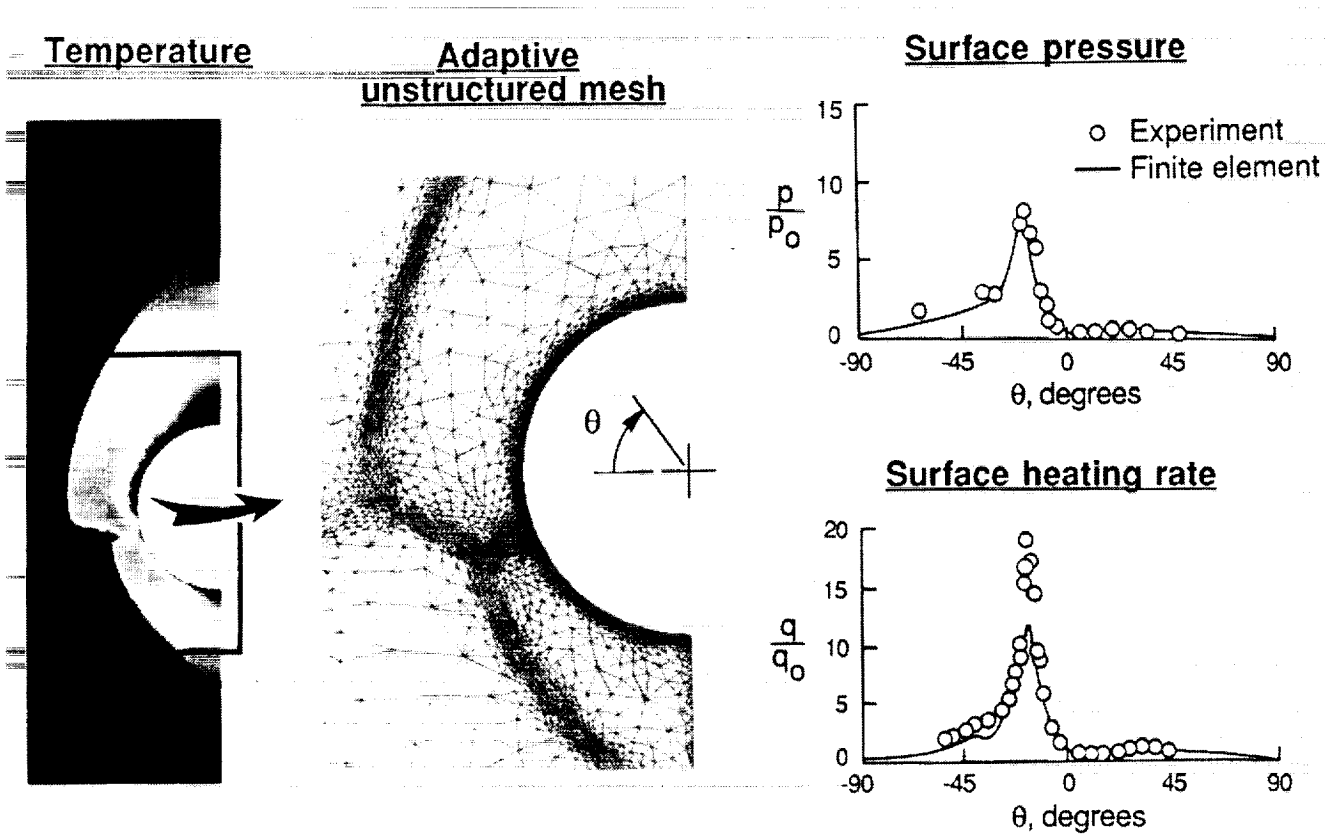
INTEGRATED FLUID-THERMAL-STRUCTURAL ANALYSIS WITH ADAPTIVE UNSTRUCTURED MESHES

Results from the integrated fluid-thermal-structural analysis using adaptive unstructured meshes for high-speed flow over the cowl leading edge is highlighted in the figure. The integrated analysis predicts: (1) complex flow behavior consisting of the shock-shock interaction phenomena from the fluid analysis, (2) the temperature response from the thermal analysis, and (3) the deformation and thermal stresses from the structural analysis. Adaptive unstructured meshes (superimposed on the fluid, thermal, and structural solutions as shown in the figure) are used in the three analysis disciplines to increase the analysis solution accuracy and to minimize the problem size thus the analysis computational time. The figure also highlights the coupling and interaction between the three disciplines. The aerodynamic heat flux causes the surface temperature to change which influences the aerodynamic flow. The temperature distribution within the leading edge and the aerodynamic pressure and skin friction result in structural deformations and stresses. These deformations may affect the flow field. The next three pages describe the use of adaptive unstructured meshes for each of these disciplines.



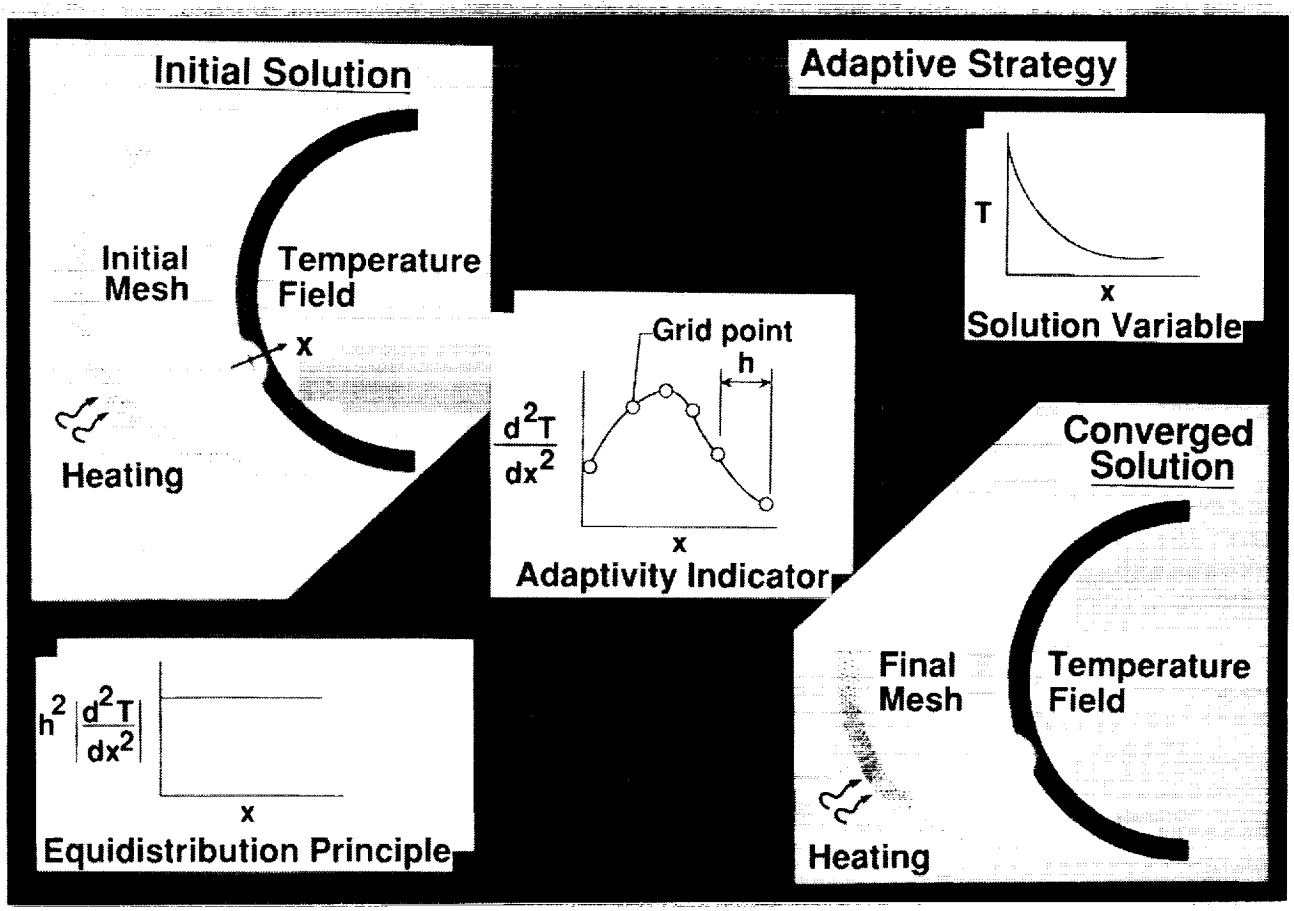
MACH 8 SHOCK INTERFERENCE

For the fluid analysis, the left figure shows the predicted fluid temperature contours obtained from the adaptive unstructured mesh shown in the center figure. The adaptive meshing procedure starts from a fairly uniform mesh with elements that have a size about a half of those located at the lower left corner of the center figure. During the meshing process, which will be described in detail in the next page, the mesh adapts itself with finer elements to capture the shocks. The fluid density was used as the key parameter for mesh adaptivity. Note that the adaptive meshing technique does not require a priori knowledge of the flow solution. To obtain accurate aerothermal loads, a refined structured mesh of quadrilateral elements is used in the boundary layer. The analytically predicted surface pressure and heating rate are normalized by their undisturbed stagnation quantities and are compared with the experimental data as shown in the right figures. The analytically predicted pressure agrees well with the experimental data. However, the peak predicted heating rate is only 60% of the peak experimental heating rate. The higher experimental heating rate is attributed to the free stream and supersonic jet shear layer turbulences which are not taken into account in the fluid analysis. Both the predicted pressure and heating rate will be used in the thermal and structural analyses of the leading edge as described in the next two pages.



ADAPTIVE UNSTRUCTURED MESH IMPROVES ACCURACY AND EFFICIENCY OF THERMAL ANALYSIS

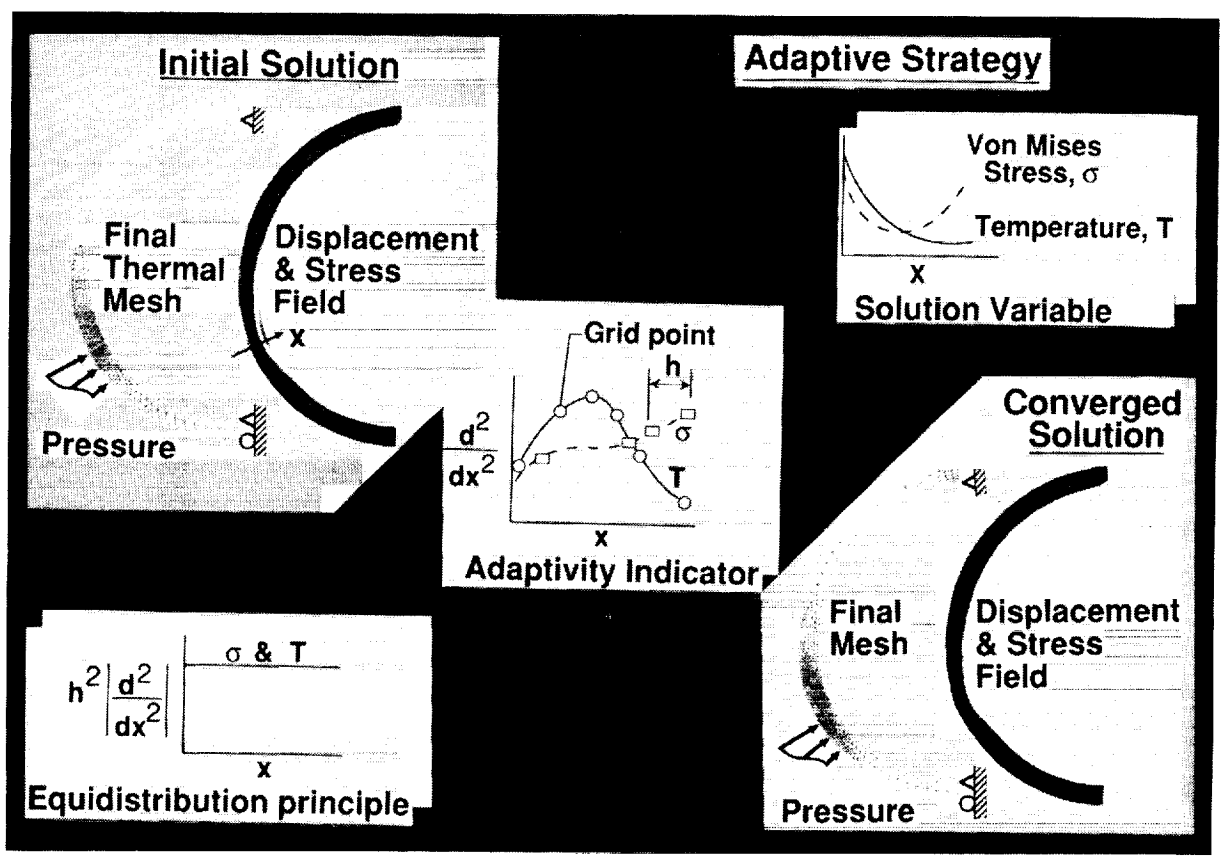
The adaptive unstructured meshing technique is illustrated with the thermal analysis of the engine cowl leading edge shown schematically in the figure. An initial solution for the temperature field is obtained on an initial (fairly uniform) mesh. To refine the solution, the adaptive strategy shown by the three diagonal figures is used. From the temperature distribution, T(x), second derivatives (d^2T/dx^2) at grid points are computed. These second derivatives are then used to determine the proper element lengths, h, based on the equidistribution principle ($h^2|d^2T/dx^2|$ = constant). The computed element lengths, h, are then used to locate new grid points for constructing a new mesh as shown in the lower right figure. The procedure places small elements in areas of high gradient changes and larger elements elsewhere. The thermal analysis is then repeated to predict a new temperature solution that has a higher solution accuracy. The final mesh and the temperature solution obtained can be used directly in the structural analysis for predicting the leading edge deformation and thermal stresses as will be described next.



ADAPTIVE UNSTRUCTURED MESH IMPROVES ACCURACY AND EFFICIENCY OF STRUCTURAL ANALYSIS

The final thermal mesh is used as the initial mesh for the structural analysis. The mesh has known nodal temperatures thus the structural analysis can be performed directly by applying the additional aerodynamic and internal coolant pressure loads. To highlight the capability of the adaptive meshing technique for combined mechanical and thermal stress problems, the lower and upper sections of the leading edge are constrained differently as shown in the upper left figure. The constraint at the upper section will result in a high mechanical stress concentration which requires refined elements in that region. To construct a new mesh, both the temperature and the Von Mises stress are used simultaneously as the key parameters for adaptive meshing. Temperature is used as the key parameter to capture thermal stress and Von Mises stress is used for mechanical stress. Their second derivatives are computed and the higher quantities are selected for calculating proper element lengths, h. The lower right figure shows the final mesh with refined elements in the regions of high thermal stress and high stress concentration. Note that two elements are required to predict bending loads even though one element was sufficient for the thermal analysis. For subsequent analyses, the final structural mesh would serve as the initial mesh.

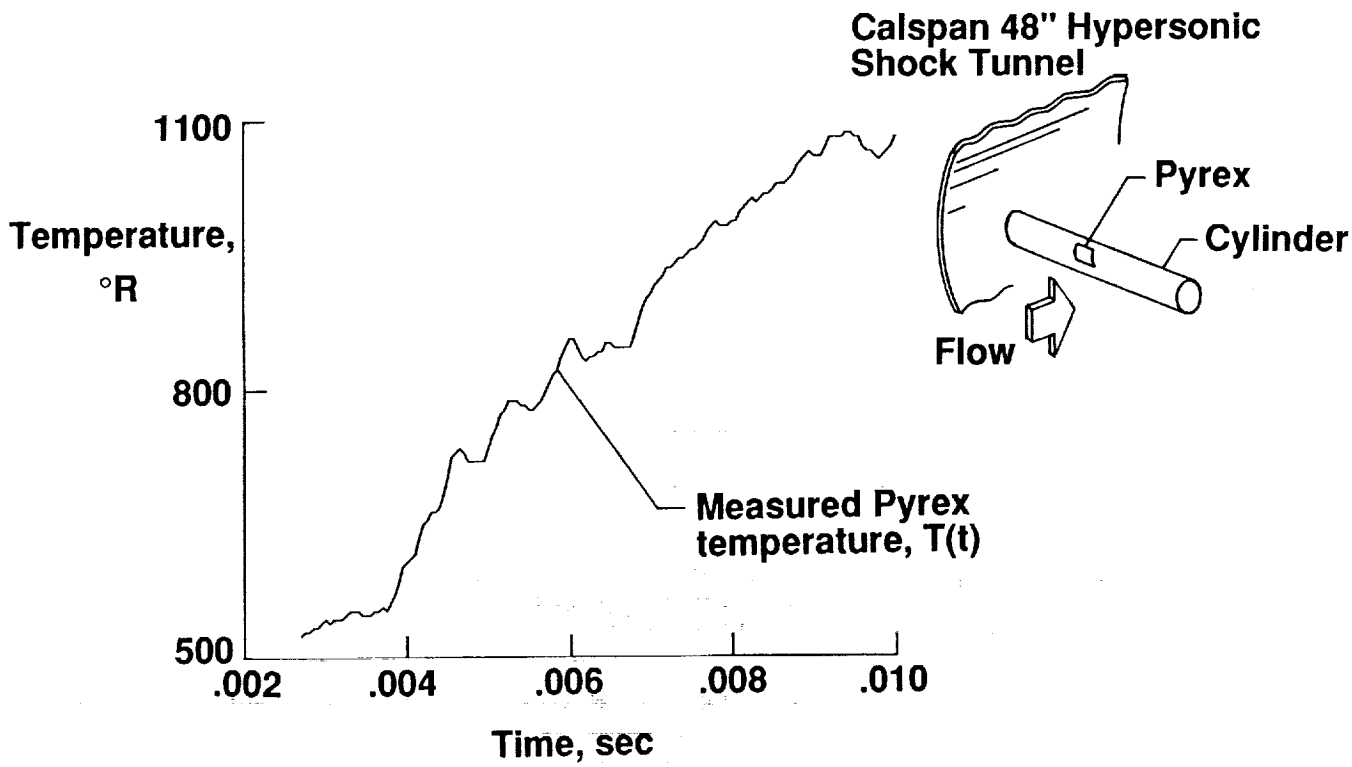
The examples shown so far demonstrate the capability of the adaptive meshing technique for steady-state problems. The next example will be used to assess the capability of the technique for a time-dependent problem.



PREDICTION OF HEATING RATE USING TRANSIENT ADAPTIVE MESHING WITH HIGHER-ORDER ELEMENTS

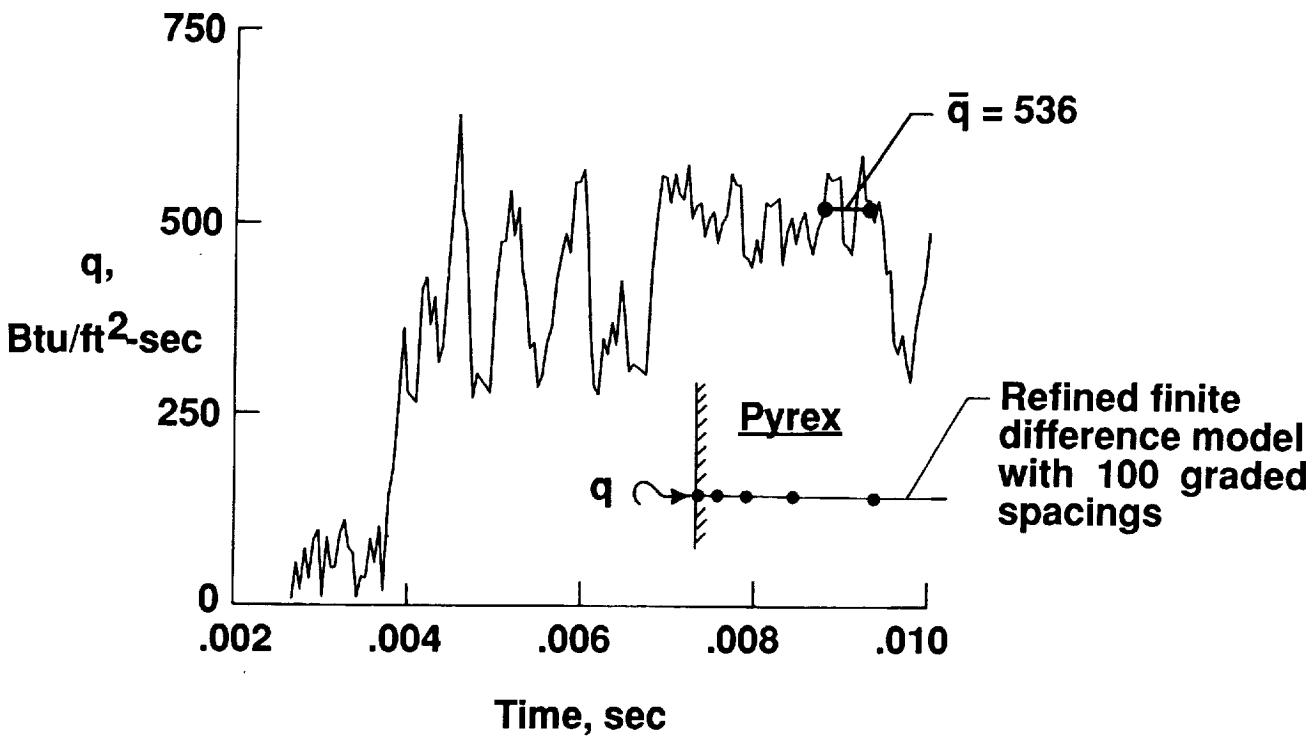
The experimental heating rate shown previously was obtained by the procedure illustrated in the figure. A thin film platinum resistance thermometer was mounted on the surface of a Pyrex substrate which was embedded into a cylindrical model (upper right insert). During a normal test period of about 10 milliseconds at Mach 8 in the Calspan 48" Hypersonic Shock Tunnel, the Pyrex time-dependent surface temperature was recorded. As shown in the temperature versus time plot, the Pyrex surface temperature increased from a room temperature of 530°R (at .00265 sec) to approximately 1100°R (at .010 sec). With the known Pyrex surface temperature history and the Pyrex material properties, the detailed transient temperature distribution through the thickness of the Pyrex (exponential decay shape) was predicted by numerical methods assuming that the substrate behaves as a semi-infinite slab. The amount of conduction heat flux at any point through the thickness of the Pyrex can then be calculated. The conduction heat flux at the Pyrex surface represents the amount of the aerodynamic heating rate to the cylinder.

Note that the ideal temperature response on the Pyrex surface should be parabolic. The oscillation of the surface temperature shown in the figure is caused by the unsteadiness of the supersonic jet from the shock-shock interaction. This surface temperature oscillation causes the predicted surface heating rate to oscillate in time as will be shown in the next page.



CALSPAN HEATING RATE PREDICTION

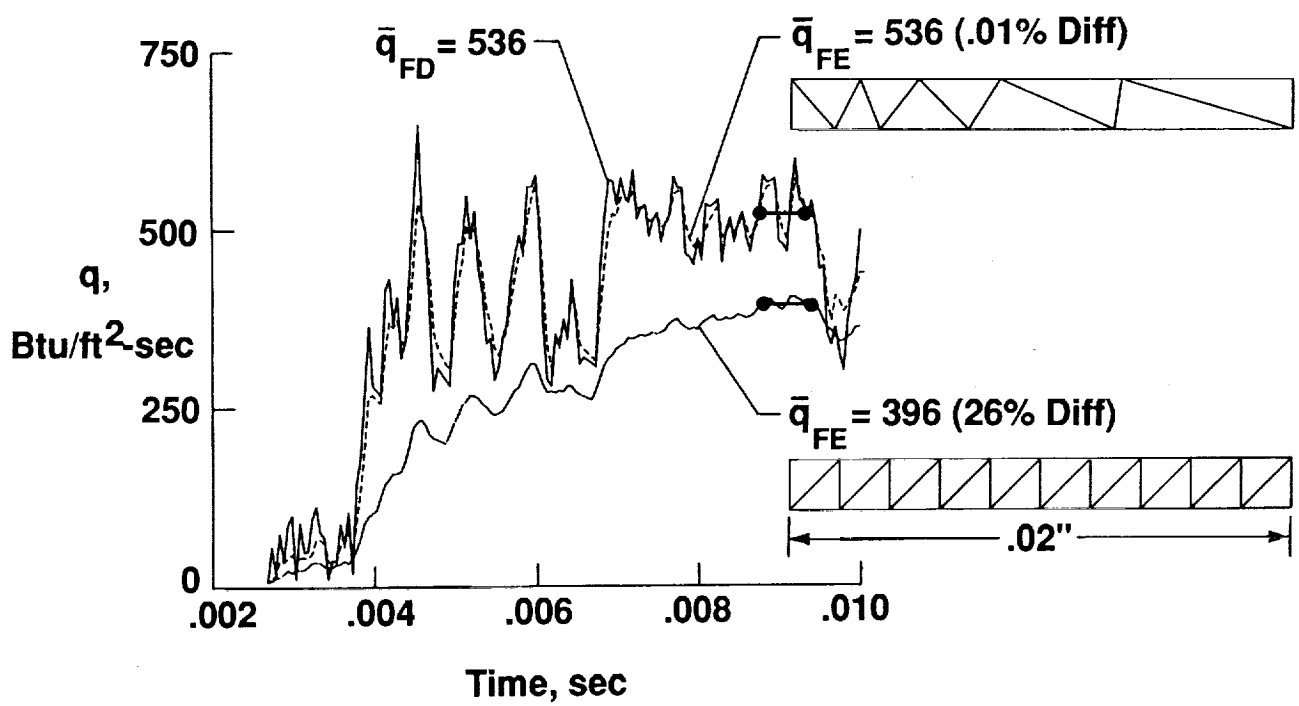
To predict the transient temperature distribution through the thickness of the Pyrex and the aerodynamic heating rate at the Pyrex surface, a finite difference technique can be used. The insert in the figure highlights a refined finite difference model employed by Calspan to predict the surface heating rate. The model consists of 101 grid points (100 graded spacings) which move (expand) with time. At an early time, for example, the grid points are clustered near the surface to capture the steep temperature gradient and detailed temperature distribution. The finite difference formulation and the analysis procedure are given in Ref. 3. The predicted heating rate obtained from the technique is plotted with time as shown in the figure. This heating rate will be used as a reference for comparison with the results from the finite element technique which will be presented next. For this test case, the aerodynamic heating rate of 536 Btu/ft^2-sec is registered which is averaged from the time of .00875 to .00925 seconds.



COMPARISON OF PREDICTED HEATING RATES

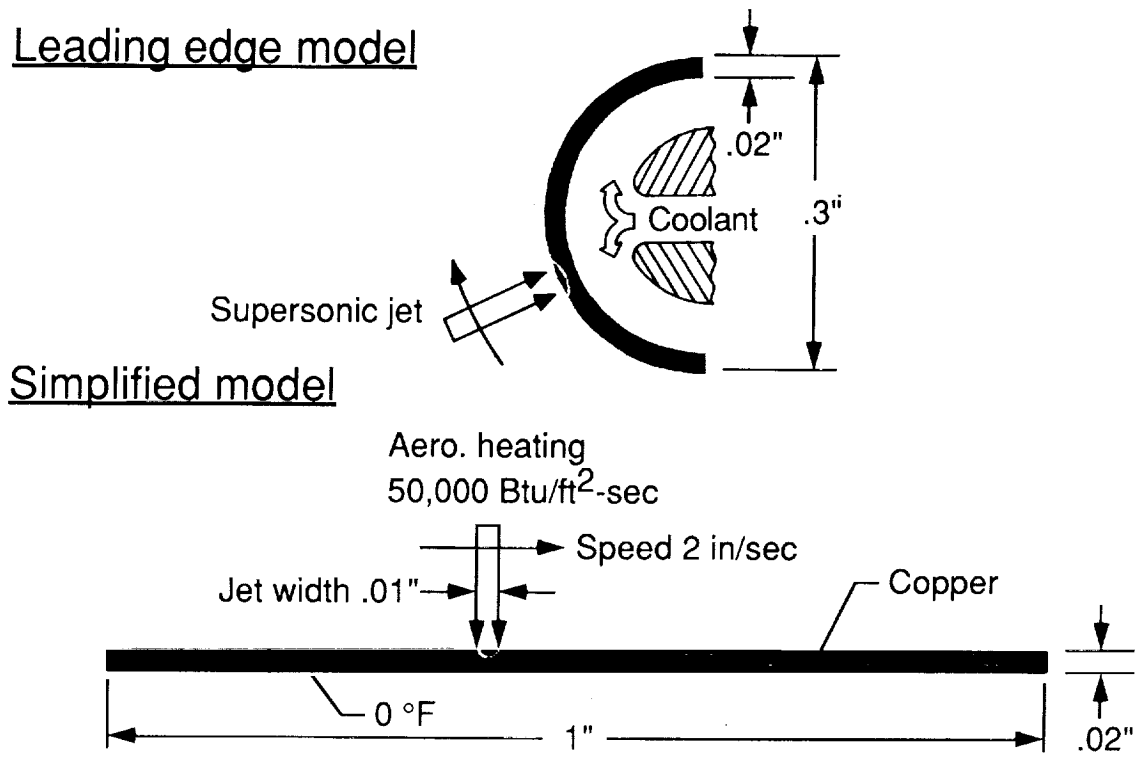
To assess the capability of the transient adaptive meshing technique, two finite element models were used for predicting the Pyrex time-dependent temperature response and the aerodynamic heating rate. The first model consisted of 20 triangles with assumed linear temperature distribution over the elements. The mesh was fixed in time as shown in the lower right insert. The predicted heating rate history obtained from this model is shown by the lower line with the predicted heating rate of 396 Btu/ft^2-sec (26% lower than the heating rate from the refined finite difference model). The second model was a transient adaptive mesh model with a variable number of triangles. These triangles assume quadratic temperature distribution over the elements. The initial mesh of this second model is identical to the first model. During the transient process, the temperature gradient changes at all the nodes are computed. The mesh is reconstructed with different element sizes according to these computed temperature gradient changes. Finer elements are placed in the regions of high temperature gradient changes to capture accurate temperature distribution and coarser elements are used elsewhere. A typical mesh during the transient process is shown in the upper right insert (another illustration of adaptive meshes at different times is shown in the next example). This adaptive model provides a very accurate heating rate prediction with a difference of .01% from the finite difference solution. The model also predicts a heating rate history that closely resembles that obtained from the refined finite difference model with 101 grid points.

This example demonstrates the capability of the transient adaptive meshing technique with a variable number of elements during the solution process for a one-dimensional heat transfer problem. The next example will demonstrate this transient adaptive meshing capability for a two-dimensional heat transfer problem.



LEADING EDGE WITH MOVING HEAT SOURCE

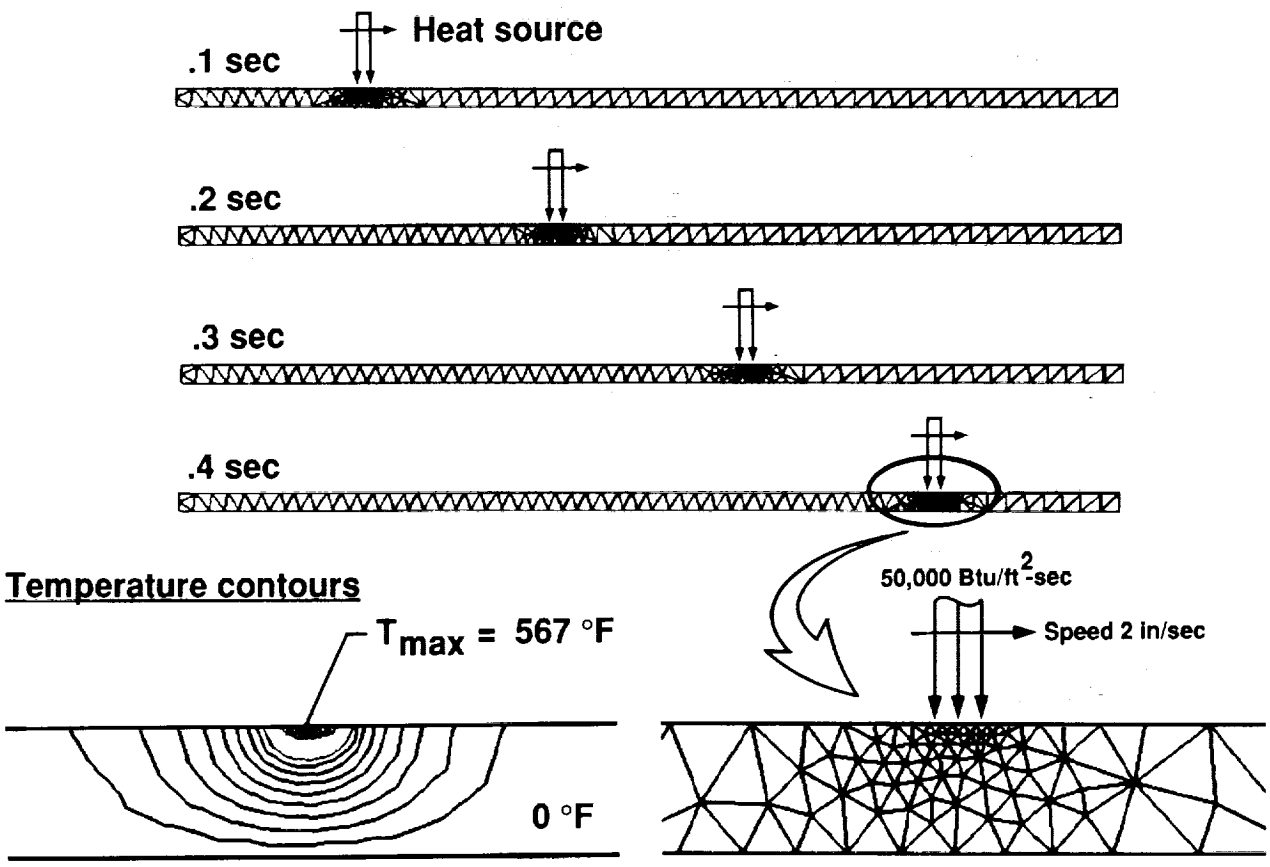
As the NASP vehicle accelerates at a high Mach number, the forebody shock sweeps across the leading edge as illustrated in the figure. High temperature and temperature gradients are concentrated in the region near the supersonic jet impingement location. In other regions, the temperatures are nearly uniform due to the internal convective cooling. For the purpose of demonstrating the transient adaptive meshing technique, the simplified model shown in the lower figure is adopted. The model offers a known exact solution (ref. 4) for the transient temperature response and thus the analysis solution accuracy from different finite element meshes (adaptive unstructured meshes and standard nonadaptive meshes) can be determined. A peak supersonic jet heat transfer rate of 50,000 Btu/ft^2-sec is simulated as a square pulse of 0.01-inches-wide. The pulse is assumed to move at a speed of 2 in/sec. This heating level is representative of a Mach 16 flight condition at a dynamic pressure of 2,000 lb/ft^2. The model material is assumed to be copper and the temperature along the back surface is maintained at zero degree Fahrenheit.



Note: Simplified model has known exact transient temperature response.

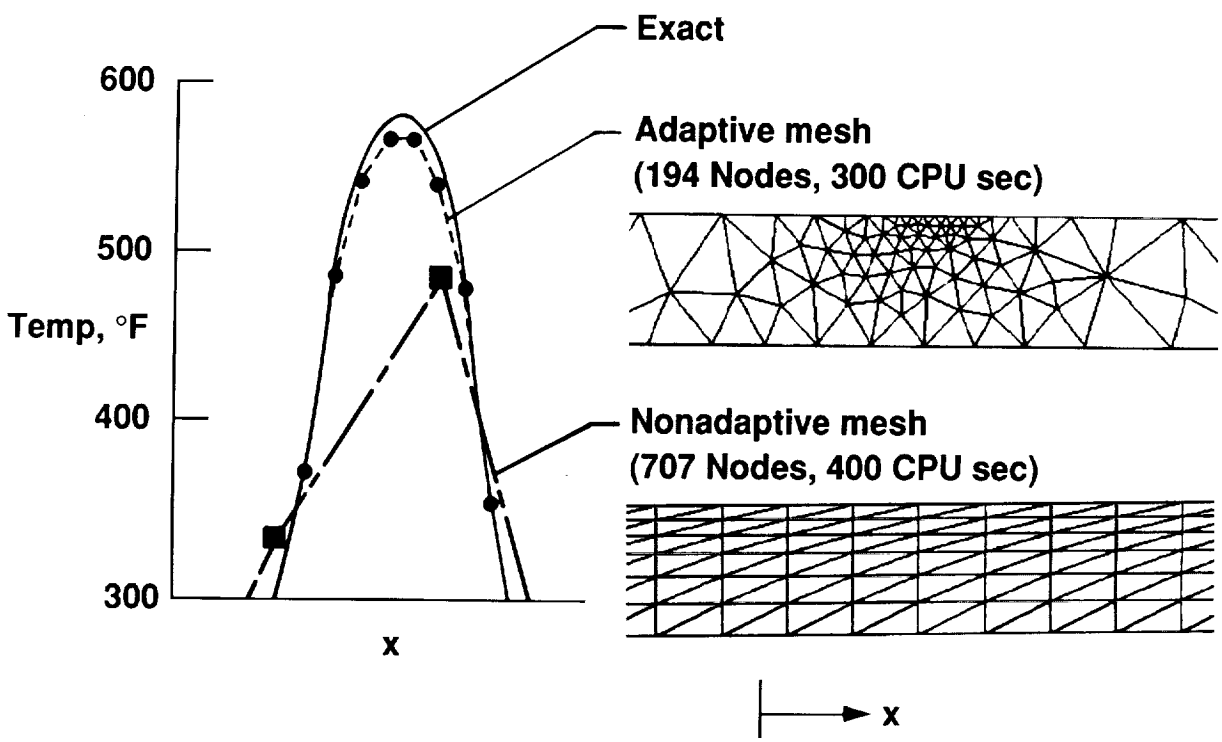
TRANSIENT ADAPTIVE MESHING

The same transient meshing procedure described in the preceding example was used to construct the meshes as the supersonic jet moved across the leading edge. Typical meshes at the four different times are shown in the figure. During the transient process, the second derivatives of the temperature at the nodes throughout the model are computed. The regions with the second derivatives that are higher than a specified threshold are identified for remeshing. The meshes in these regions are reconstructed with finer or coarser elements according to the computed second derivatives. In the other regions, the meshes are untouched. Because the technique places refined elements near the jet impingement location (lower right figure), accurate temperatures (lower left figure) are obtained with the peak temperature about 2% lower than the exact solution. It is important to note that the refined mesh at the jet impingement location also allows a more accurate aerodynamic heating rate distribution to be input into the analysis model, and hence a better temperature distribution is obtained.



TOP SURFACE TEMPERATURES

The predicted temperatures obtained from the adaptive unstructured mesh (The mesh consists of approximately 245 triangles at a typical time.) and a nonadaptive structured mesh (fixed mesh with 1,200 triangles) are compared with the exact solution along the top surface near the jet impingement location as shown in the figure. The adaptive unstructured mesh provides a more realistic detailed temperature distribution compared to that from the nonadaptive structured mesh (2% versus 15% error for the peak temperatures) with a fewer number of unknowns (194 versus 707 nodes) and lower analysis computational time (300 versus 400 CRAY-2 CPU seconds). For a nonadaptive structured mesh to provide an equivalent detailed temperature distribution, the mesh will require a number of nodes at least an order of magnitude higher than the current model and a much larger computational time. Note that detailed temperature distribution is necessary for the prediction of accurate thermal stress response from the structural analysis. Thus the temperature solution obtained from the adaptive unstructured mesh will further improve the thermal stress solution accuracy for the structural analysis.

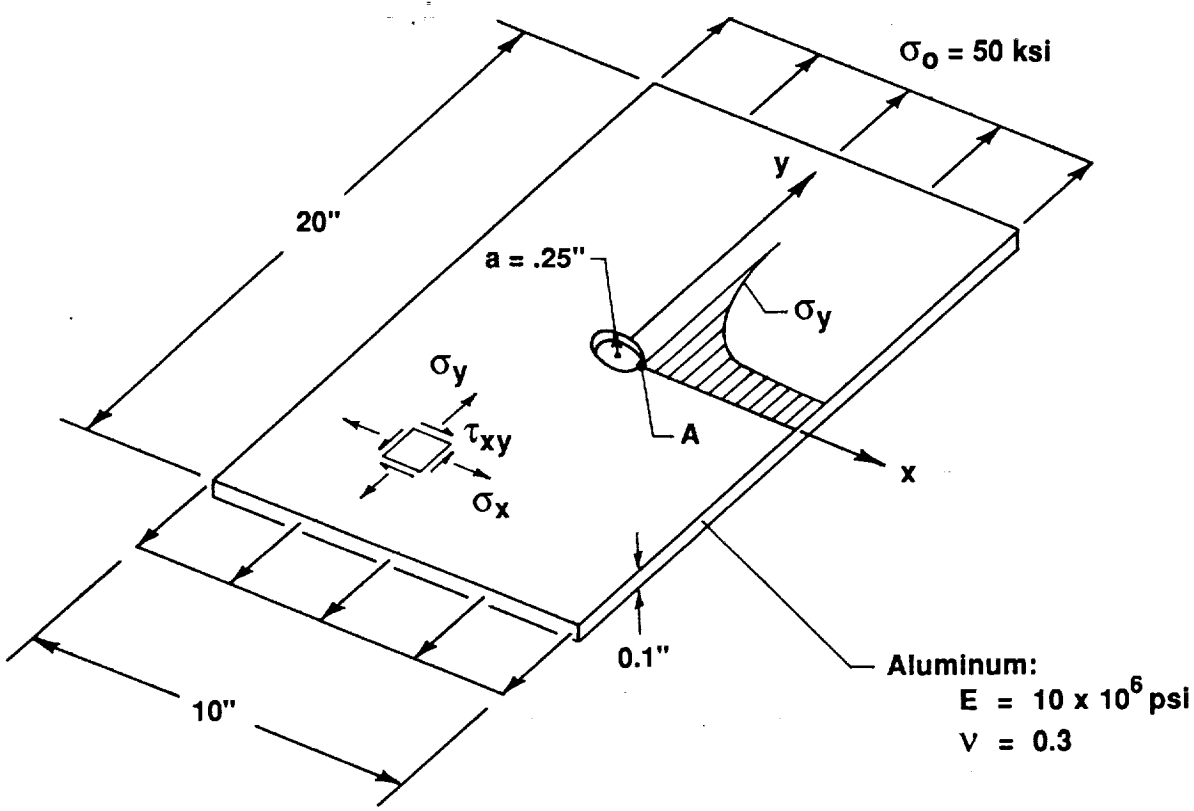


PANEL WITH CIRCULAR CUTOUT

The adaptive meshing technique has been extended to structural analysis. Before applying the technique to complex structural problems, the technique was evaluated on a two-dimensional plane stress problem. A panel with a circular cutout, shown in the figure, is an ideal structure to demonstrate the capability of the technique because closed-form solutions (ref. 5) are available. The panel is subjected to an applied uniform stress (σ_o) of 50 ksi in the longitudinal y-direction (see figure). The exact normal stress (σ_y) distribution along the x-direction at y=0 is given by,

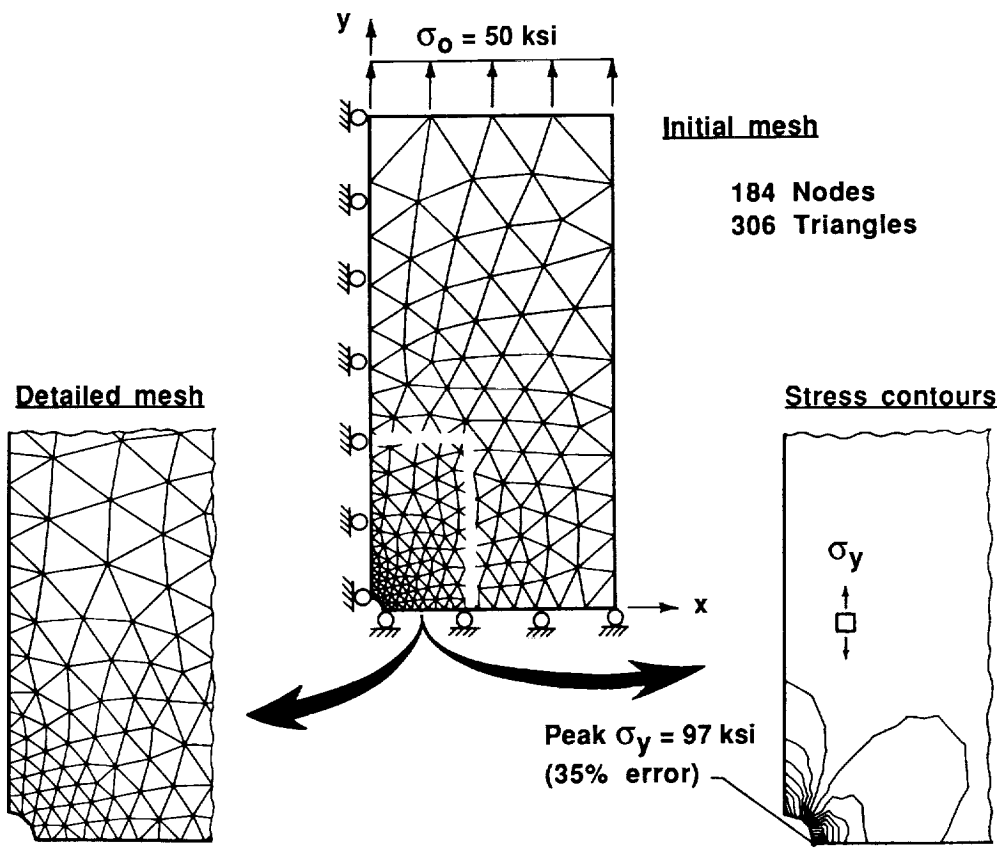
$$\sigma_y = \frac{\sigma_o}{2}\left(2 + \frac{a^2}{x^2} + 3\frac{a^4}{x^4}\right)$$

where a is the radius of the circular cutout. The peak stress ($3\sigma_o$) of 150 ksi occurs at the edge of the cutout (point A in the figure).



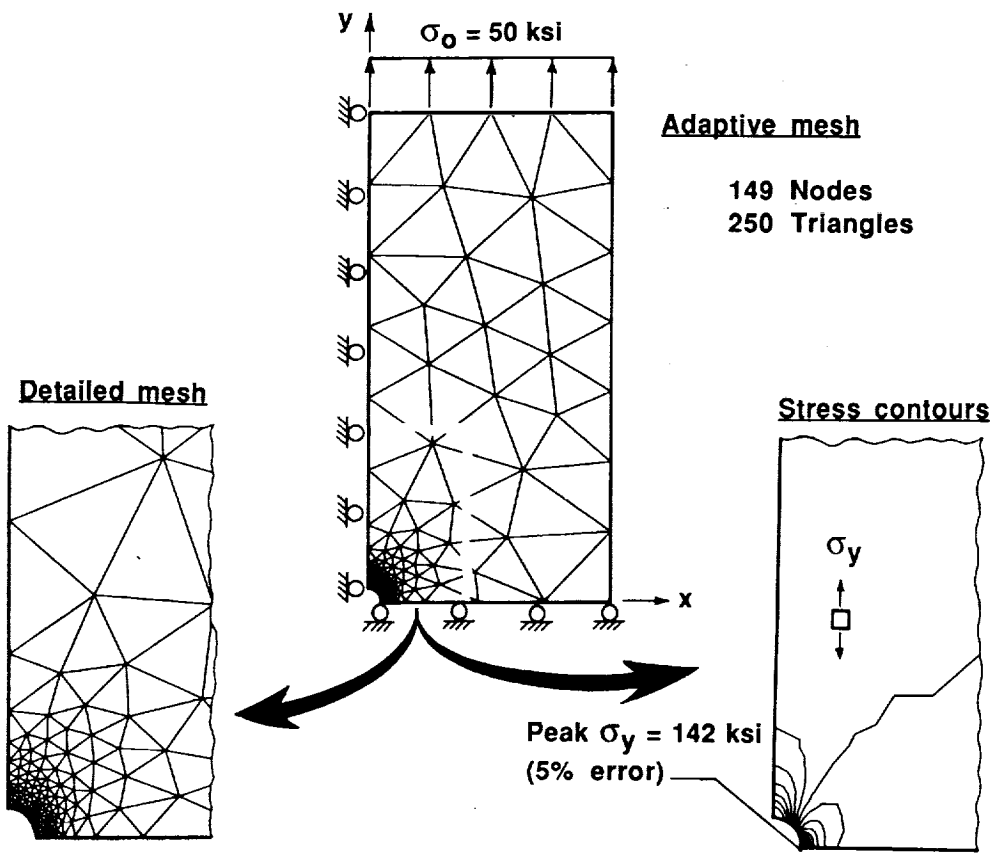
INITIAL MESH FOR PANEL WITH CIRCULAR CUTOUT

Due to symmetry, a quarter of the panel can be used in the analysis. An initial mesh was constructed which consisted of 184 nodes and 306 triangles as shown in the center of the figure. Detail of the mesh near the cutout is shown in the lower left figure. With this mesh, the structural analysis (plane stress) was performed to predict the panel deformation and the stress distributions. The contours of the normal stress (σ_y) distribution in the longitudinal y-direction near the cutout are shown in the lower right figure. The peak predicted stress (σ_y) at the edge of the cutout obtained from this initial mesh is 97 ksi compared to the exact peak stress of 150 ksi. The 35% error in the peak stress is due to the coarse elements used near the cutout region.



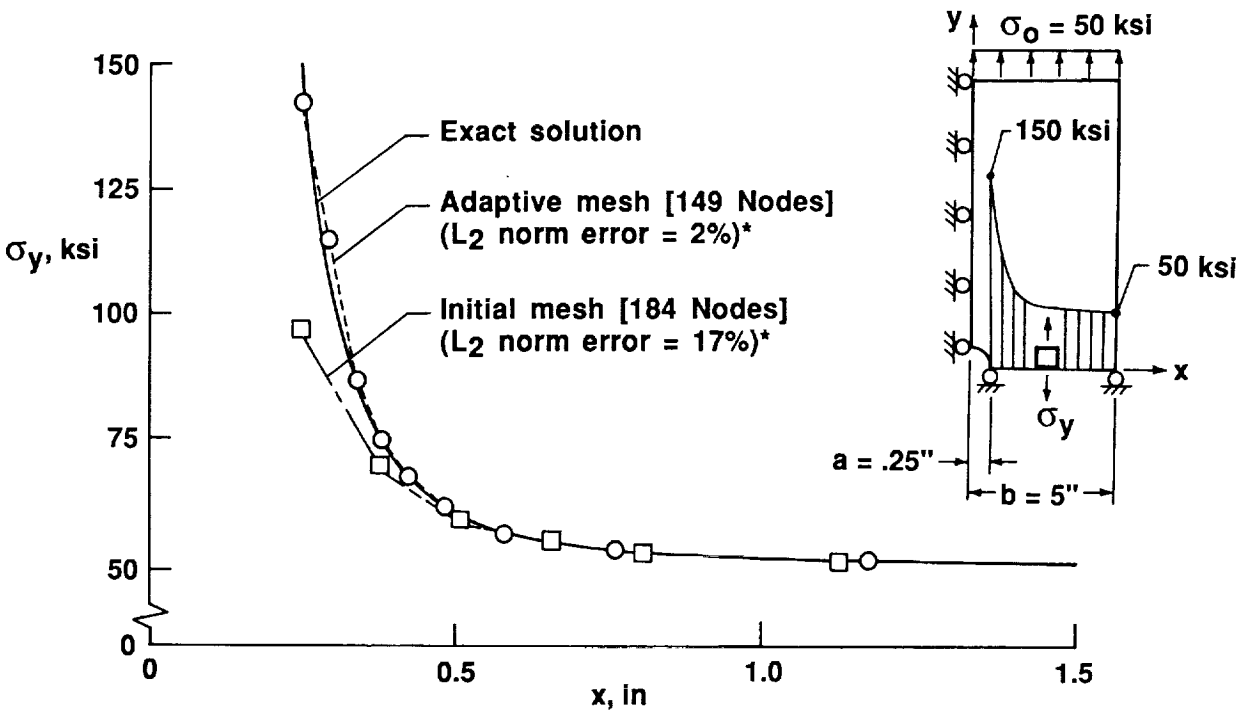
ADAPTIVE MESH FOR PANEL WITH CIRCULAR CUTOUT

The stress distribution (in form of the Von Mises stress) obtained from the initial mesh is used as the meshing parameter to construct a new adaptive mesh. The new adaptive mesh, shown in the center of the figure, has fewer nodes and elements (149 nodes and 250 triangles) than the initial mesh (184 nodes and 306 triangles). However, finer elements are concentrated in the region of high stress gradients near the cutout (shown in the lower left figure) to provide a more accurate stress solution. The structural analysis is then performed and the contours of the normal stress (σ_y) distribution near the cutout are shown in the lower right figure. The peak predicted stress (σ_y) at the edge of the cutout is now 142 ksi which is only 5% lower than the exact level of 150 ksi. It is important to note that the adaptive meshing technique automatically generates refined elements in the regions of high stresses. A priori knowledge of the solution to the problem (e.g. high stress regions that require refined elements) is not needed before performing the analysis. The technique thus provides an advantage over the standard finite element procedure especially for more complex problems or larger structures (such as the structure which will be shown in the last example) where a priori knowledge of the solution is not known.



COMPARATIVE STRESS DISTRIBUTIONS

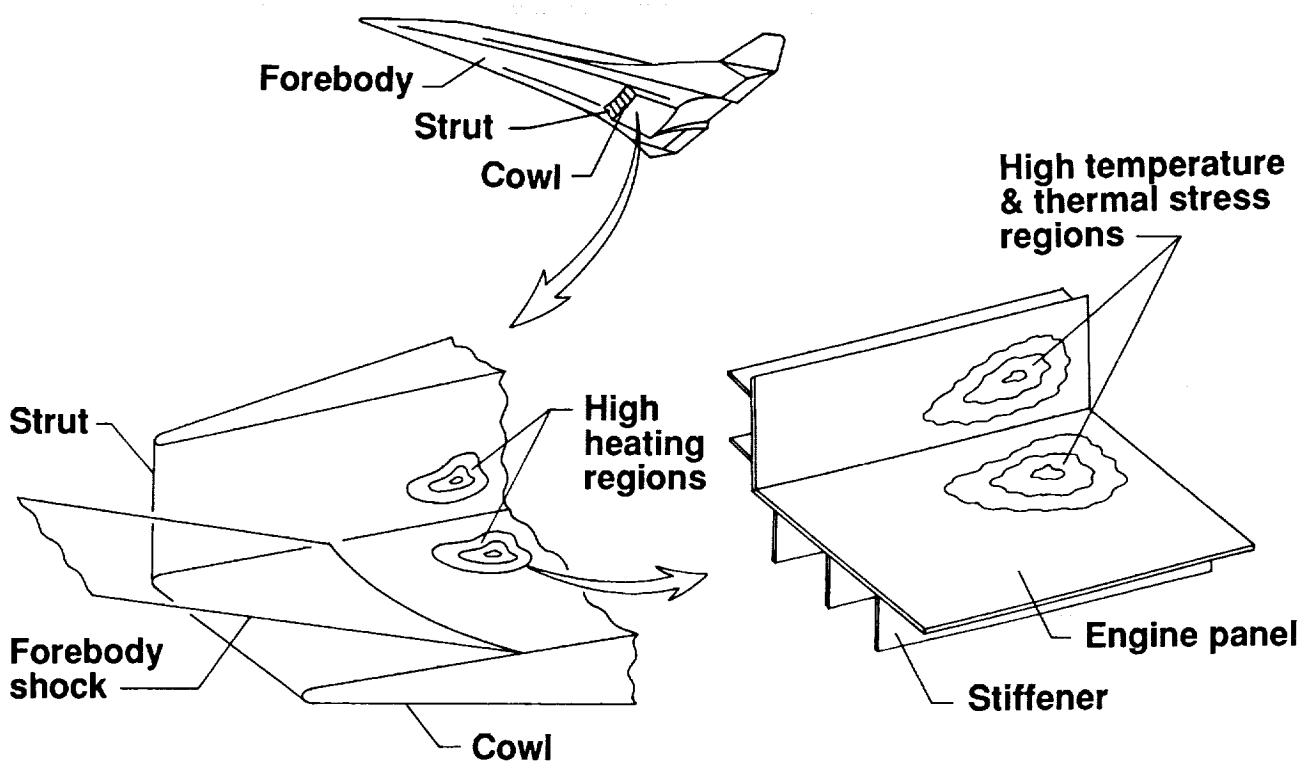
The normal stress (σ_y) distributions at y=0 along the x-direction obtained from the initial and the adaptive mesh are compared with the exact stress distribution in the figure. The L_2 norm error defined at the bottom of the figure represents the finite element solution error relative to the exact solution. The solution error was reduced (from 17% to 2%) with one adaptive remesh. In addition, as the solution accuracy increased, the number of unknowns decreased, because finer elements were generated in the regions of high solution gradients near the cutout whereas larger elements are used in other regions. This example demonstrates the capability of the adaptive meshing technique for a two-dimensional plane stress problem. The next example will highlight the planned application of the technique to a three-dimensional built-up structure.



$$\text{*Definition: } L_2 \text{ norm error} = \sqrt{\int_a^b \left[(\sigma_y)_{Exact} - (\sigma_y)_{F.E.} \right]^2 dx}$$

SHOCK IMPINGEMENT ON ENGINE PANEL

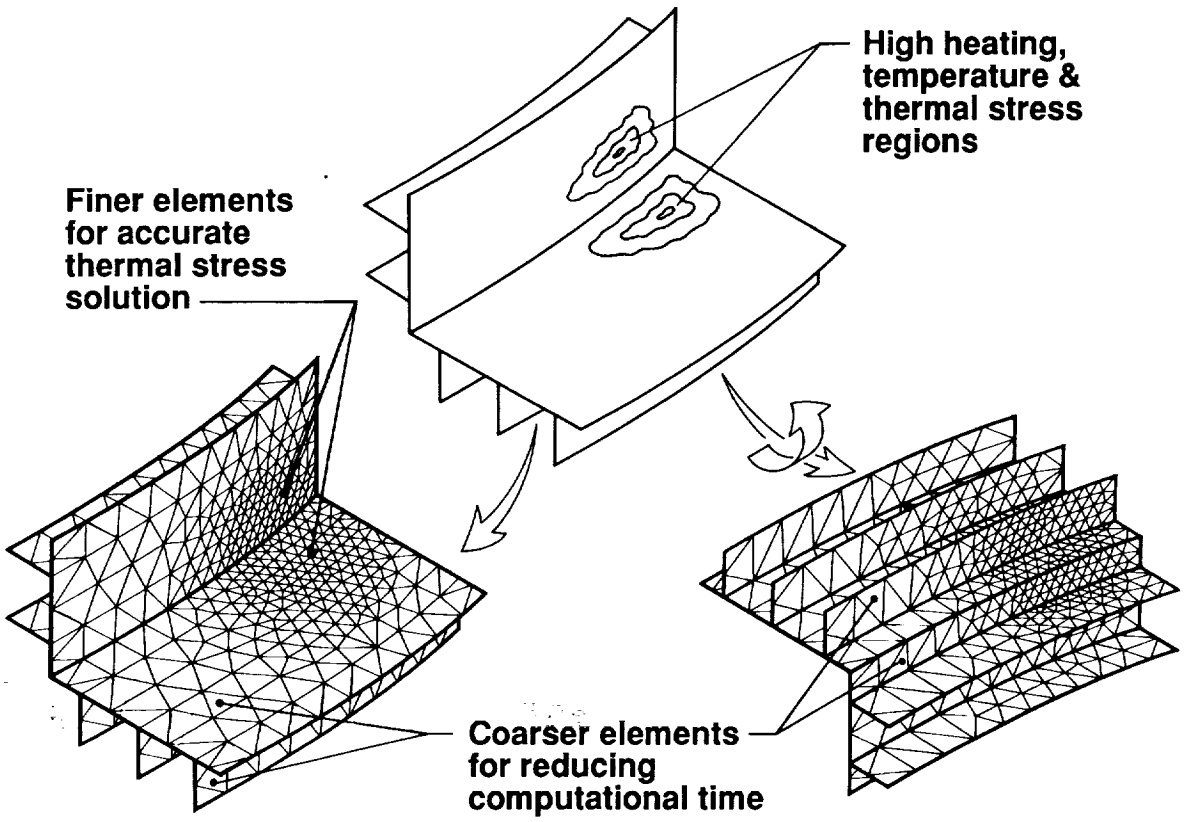
To further demonstrate the capability of the adaptive unstructured meshing technique, a more complex three-dimensional structure which represents a scramjet engine inlet is considered. The structure experiences high aerodynamic heating from the impingement of an oblique shock from the vehicle forebody as illustrated in the lower left figure. The localized heating regions result in high temperature gradients and their attendant thermal stresses in the engine panel. A typical "built-up" engine inlet structure, shown in the lower right figure, may consist of panels and stiffeners and may also be actively cooled. To predict the thermal stress response of such a structure accurately, a finite element model consisting of plate bending and membrane elements for the panels and stiffeners is required. The same nodal discretization is also required along the interface between the stiffeners and the panels. In the standard finite element analysis procedure, high thermal stress regions need to be identified. Construction of a finite element model, with refined elements in these regions while maintaining the same nodal discretization along the interface between different structural components, is tedious. Furthermore, several modelling iterations are normally needed before achieving the desired finite element model for solution accuracy. These difficulties can be alleviated by the use of the adaptive unstructured meshing technique as will be illustrated on the next page.



ADAPTIVE THERMAL-STRUCTURAL PLATE BENDING ELEMENTS

An adaptive meshing capability for "built-up" structures is currently being developed. The technique has been tested on this engine panel problem and a typical adaptive mesh is shown in the figure. Finer elements are concentrated in the high thermal stress regions to provide solution accuracy. Coarser elements are generated in other regions for reducing the problem size and thus the computational time. The same nodal discretization is maintained along the interface between the panels and the stiffeners. The figure shows that the model can be constructed easily using triangles. These triangles also provide a smooth transition of the element sizes from refined to coarse mesh regions. The smooth element transition will enhance the thermal stress prediction with a more realistic distribution. The adaptive unstructured mesh shown in the figure can be used in the structural analysis to obtain accurate deformations and thermal stresses without a priori knowledge of high stress regions.

To further increase the analysis solution accuracy from a given mesh, the Discrete Kirchoff Triangle (DKT) plate bending element (ref. 6) subjected to both the mechanical and thermal loads is also being investigated. This DKT plate bending element, under applied mechanical load alone (ref. 7), has been shown to provide higher solution accuracy compared to other triangular bending elements. The finite element thermal stress formulation for this DKT element, the solution algorithm, and its capability for improving thermal stress solution for a built-up structure problem (including this convectively-cooled engine panel problem) will be presented in detail.* Combination of a more efficient triangular bending element and a proper adaptive mesh model will provide higher solution accuracy and require less computational time for the structural analysis of three-dimensional built-up structures.



*Dechaumphai, P. "Adaptive Meshing for Thermal Stress Analysis of Built-Up Structures," To be presented at the AIAA/ASME/ASCE/AHS/ASC 33rd Structures, Structural Dynamics, and Materials Conference, Dallas, Texas, April 13-15, 1992.

CONCLUDING REMARKS

Progress in the integrated fluid-thermal-structural analysis with adaptive unstructured meshes was presented. The adaptive unstructured meshing technique for the fluid, thermal and structural analyses was described. The technique was applied to the three different analysis disciplines for improving the solution accuracy, reducing the problem size and the computational time. Coupling and interaction between the three different analysis disciplines were highlighted using the problem of shock-shock interference on a scramjet engine convectively-cooled leading edge. The examples presented in this paper reflect some of the current adaptive unstructured meshing capability which exists at the Aerothermal Loads Branch, NASA Langley Research Center. These adaptive meshing capabilities are for: (1) two- and three-dimensional fluid analyses; (2) thermal analysis with transient adaptive meshing using linear and higher-order elements; and (3) thermal stress analysis for two- and three-dimensional continuum structures with an extension to built-up structures. The results from the examples presented in this paper have demonstrated the viability of the adaptive meshing technique combined with the finite element method to provide efficient accurate solutions to complex flow-thermal-structural behavior.

- ADAPTIVE UNSTRUCTURED MESHING TECHNIQUE DESCRIBED FOR FLUID, THERMAL AND STRUCTURAL ANALYSES
- THERMAL AND STRUCTURAL ANALYSES ARE INTEGRATED
- ADAPTIVE UNSTRUCTURED MESHING CAPABILITIES:
  - 2 & 3-D FLUID ANALYSES
  - 2-D THERMAL ANALYSIS WITH TRANSIENT ADAPTIVE MESHING
  - 2 & 3-D STRUCTURAL ANALYSES FOR CONTINUUM STRUCTURES CURRENTLY BEING EXTENDED TO BUILT-UP STRUCTURES
- APPLICATIONS SHOW ADAPTIVE UNSTRUCTURED MESHING:
  - REDUCES COMPUTATIONAL TIME
  - INCREASES SOLUTION ACCURACY

REFERENCES

1. Peraire, J., Vahdati, M., Morgan, K. and Zienkiewicz, O. C., "Adaptive Remeshing for Compressible Flow Computations," Journal of Computational Physics, Vol. 72, 1987, pp. 449-466.

2. Dechaumphai, P., "Integrated Fluid-Thermal-Structural Analysis Using Adaptive Unstructured Meshes," Proceedings of the 17th Congress of the International Council of the Aeronautical Sciences, Stockholm, Sweden, September 1990.

3. Cook, W. J., "Determination of Heat Transfer Rates from Transient Surface Temperature Measurements," AIAA Journal, Vol. 8, No. 7, 1970, pp. 1366-1368.

4. McGowan, D. M., Camarda, C. J. and Scotti, S. J., "A Simplified Method for Thermal Analysis of a Cowl Leading Edge Subject to Intense Localized Heating," NASA TP-16505, November 1990.

5. Timoshenko, S. P. and Goodier, J. N., Theory of Elasticity, Third Edition, McGraw-Hill Book Company, New York, 1934, pp. 90-97.

6. Batoz, J. L., Bathe, K. J. and Ho, L. W., "A Study of Three-Node Triangular Plate Bending Elements," International Journal for Numerical Methods in Engineering, Vol. 5, 1980, pp. 1771-1812.

7. Batoz, J. L., "An Explicit Formulation for an Efficient Triangular Plate-Bending Element," International Journal for Numerical Methods in Engineering, Vol. 18, 1982, pp. 1077-1089.

N92-25915

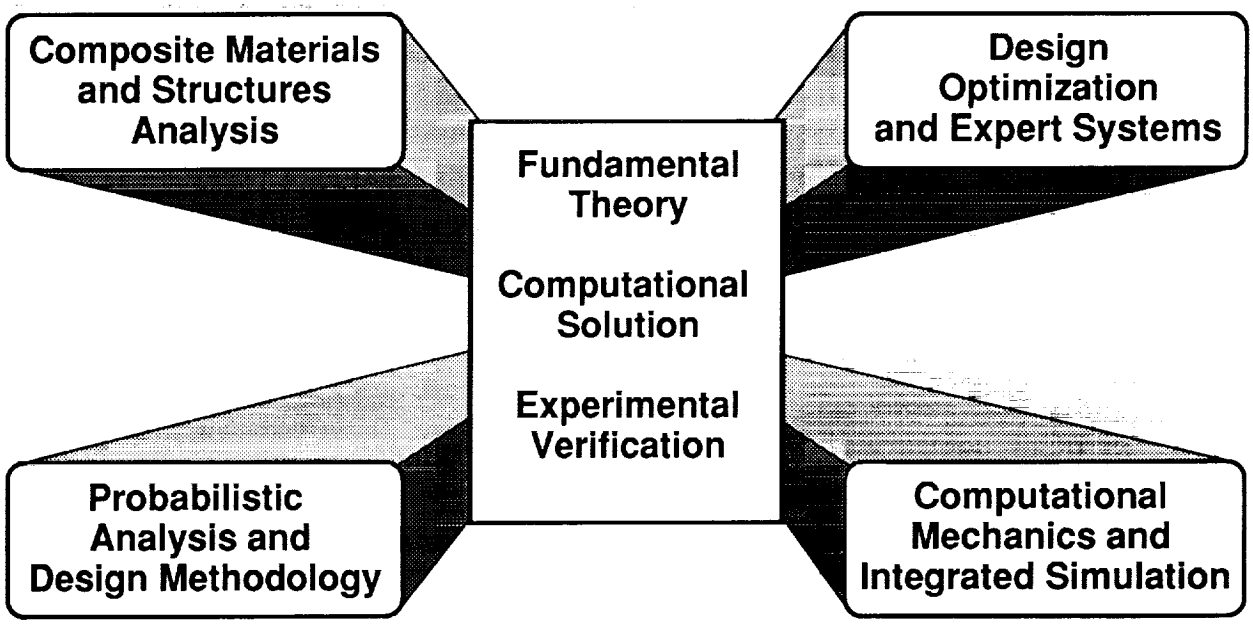
A Brief Overview of Computational Structures Technology Related Activities at NASA Lewis Research Center

Dale A. Hopkins
NASA Lewis Research Center
Cleveland, Ohio

PRECEDING PAGE BLANK NOT FILMED

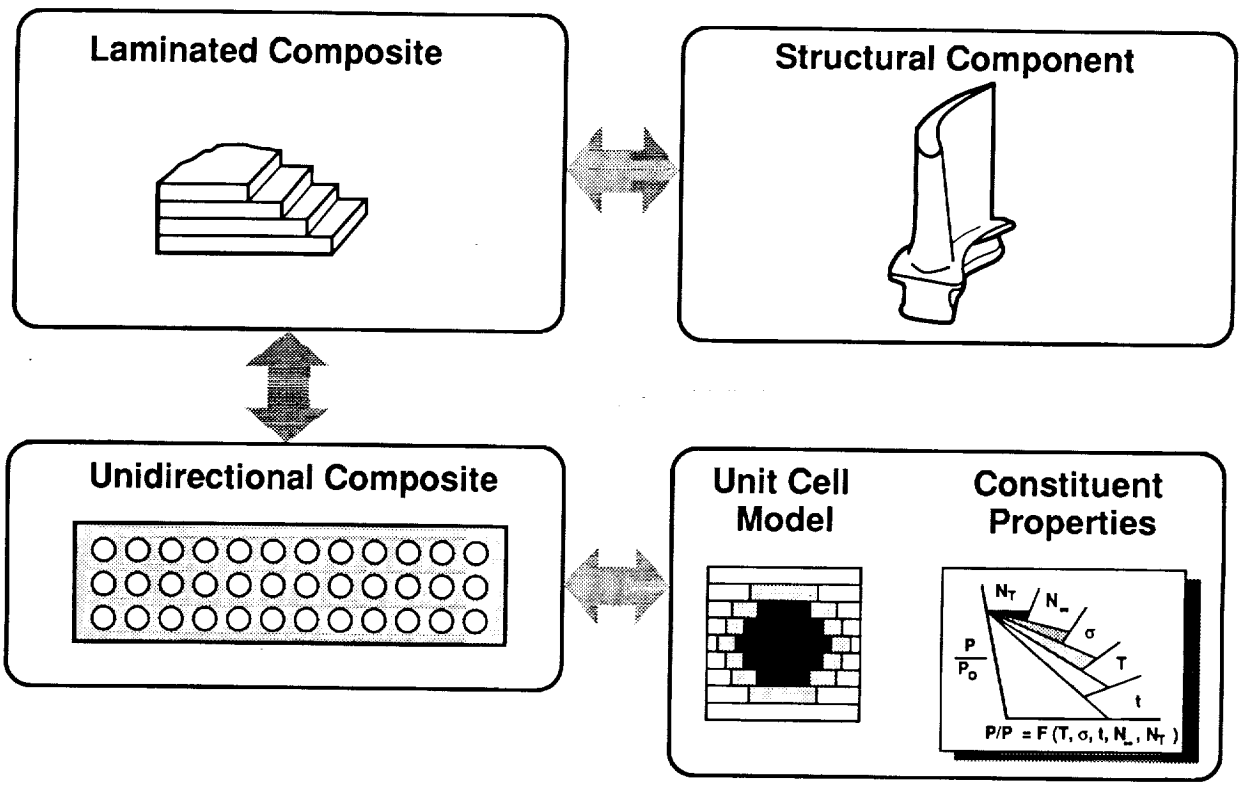
COMPUTATIONAL STRUCTURES TECHNOLOGY DEVELOPMENT IS DIVERSE AND COMPREHENSIVE

The presentation gives a partial overview of research and development underway in the Structures Division of Lewis Research Center, which collectively is referred to as the computational structures technology program. The activities in the program are diverse and encompass four major categories as shown in the figure, namely (1) composite materials and structures, (2) probabilistic analysis and reliability, (3) design optimization and expert systems, and (4) computational methods and simulation. The approach of the program is comprehensive and entails exploration of fundamental theories of structural mechanics to accurately represent the complex physics governing engine structural performance, formulation and implementation of computational techniques and integrated simulation strategies to provide accurate and efficient solutions of the governing theoretical models by exploiting the emerging advances in computer technology, and validation and verification through numerical and experimental tests to establish confidence and define the qualities and limitations of the resulting theoretical models and computational solutions. The program comprises both in-house and sponsored research activities. The remainder of the presentation provides a sample of activities to illustrate the breadth and depth of the program and to demonstrate the accomplishments and benefits that have resulted.



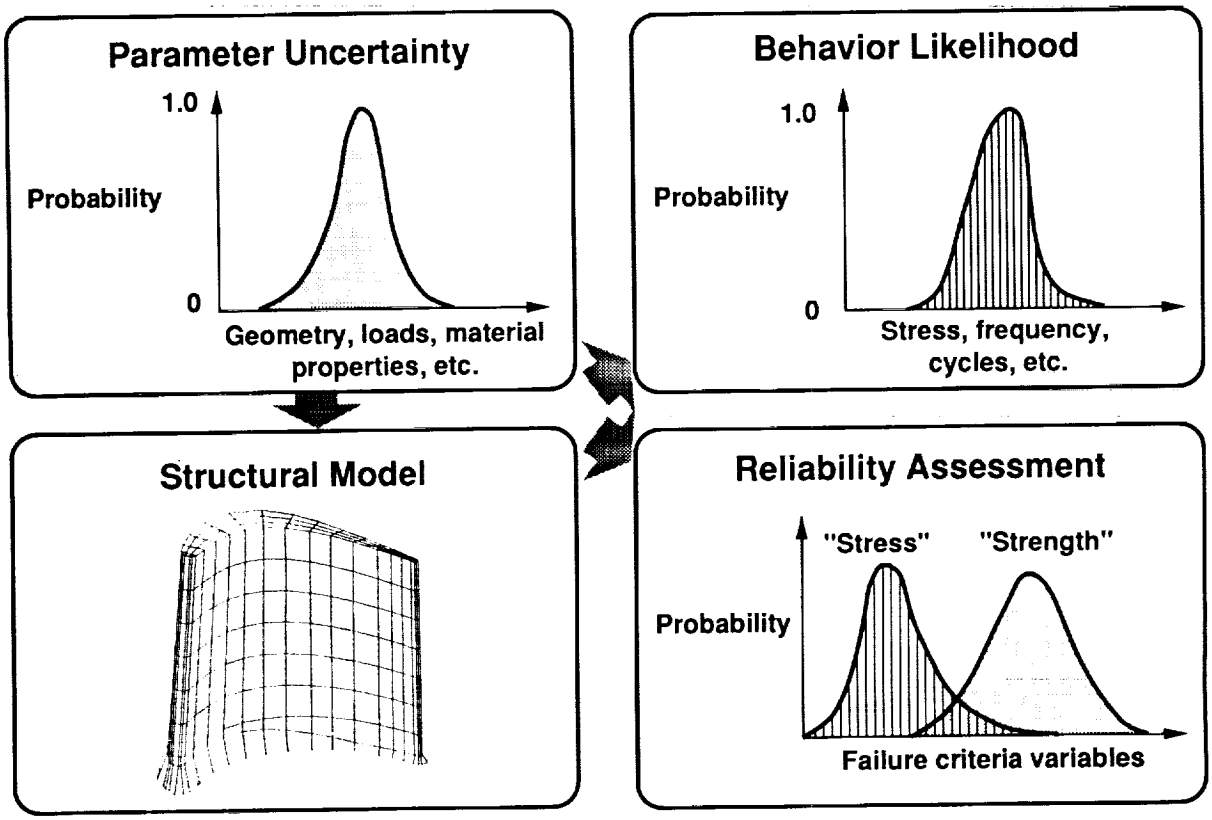
MULTI-SCALE APPROACH RELATES LOCAL EFFECTS
TO GLOBAL RESPONSE OF COMPOSITE MATERIALS AND STRUCTURES

The thermomechanical performance and structural integrity of advanced composite materials in high temperature engine applications is governed by local (constituent scale) phenomena. Local influences which affect global (component scale) behavior include fiber-matrix interface bonding, progressive damage and failure processes, and constituent material behavior which exhibits cyclic nonlinear history dependence. For the analysis and design of engine components it is desirable to account for local influences and to relate their effect on global structural performance. With this purpose, an integrated multi-scale approach has been developed which incorporates nonlinear constituent material models and failure models, composite mechanics models, and finite element global structural analysis models. The unique unit cell which provides the basis of the composite micromechanics model allows an arbitrary resolution of locally nonuniform or discontinuous behavior of material properties, stress and strain, temperature, and other critical field variables. Even with the capability to capture local details, the multi-scale approach maintains a practical computational efficiency for realistic engine component analyses.



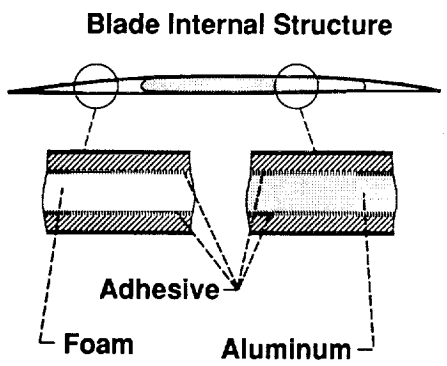
PROBABILISTIC ANALYSIS METHODS QUANTIFY RISK
FOR A BETTER DESIGN DECISION BASIS

The variables of the structural design process including geometry, material properties, loads, and boundary conditions, exist only with a degree of variability. The uncertainty contributes to an associated degree of risk that the performance of a structural component will deviate from its design expectation. The risk in a design is not directly quantified by the traditional deterministic design methodology. Rather, the deterministic methodology relies on the safety factor concept as a qualitative indicator of design risk. This approach is inherently conservative and provides no basis to attain the desired balance between safety and efficiency of the design. A probabilistic analysis and design methodology, on the other hand, provides a formalism to quantify design uncertainty. As the figure depicts, a probabilistic methodology is being developed as a rational basis to assess risk and to make risk management decisions. A probabilistic analysis and design methodology is especially pertinent in the design of high performance, high energy propulsion systems where mission economy and safety are the overwhelming (and competing) design objectives. The ability to accurately quantify reliability and risk is essential to achieve an acceptable balance between performance and safety.



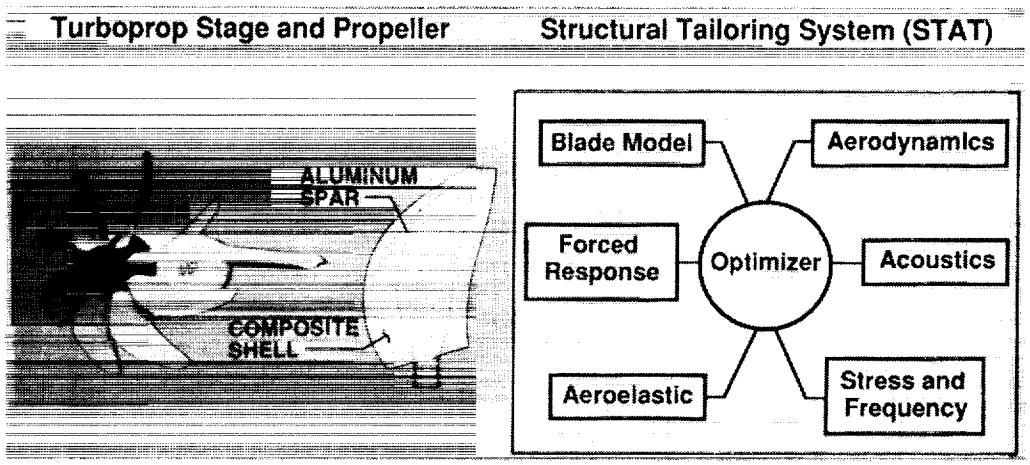
STRUCTURAL TAILORING RESOLVES COMPLEX REQUIREMENTS
FOR COMPONENT-SPECIFIC OPTIMUM DESIGN

The traditional approach to engine component design has been to satisfy competing multidisciplinary requirements independently through manual design iterations. This process, which is usually conducted between several disciplinary specialist groups, is inherently time consuming, cumbersome, error-prone, and subjective. The typical process, therefore, is carried out only to the point where a satisfactory design is achieved. The luxury of continuing the process to find the best design is virtually never afforded. A relevant example of this design scenario occurs for the case of engine blades. Recent development of the advanced turboprop propulsion concept presents a consummate example of the unwieldy task of satisfying multidisciplinary design requirements. The difficult challenge to design advanced propfan blades presented an opportunity to demonstrate an alternative methodology. The approach taken was to streamline, automate and formalize the propfan design process by incorporating the multidisciplinary analyses together with numerical optimization techniques into a computationally effective design tailoring system. As summarized in the figure, the design tailoring strategy proved to be highly successful for the propfan application. The concept of component-specific design tailoring has been successfully extended to cooled turbine blade applications and actively-cooled panel structure.



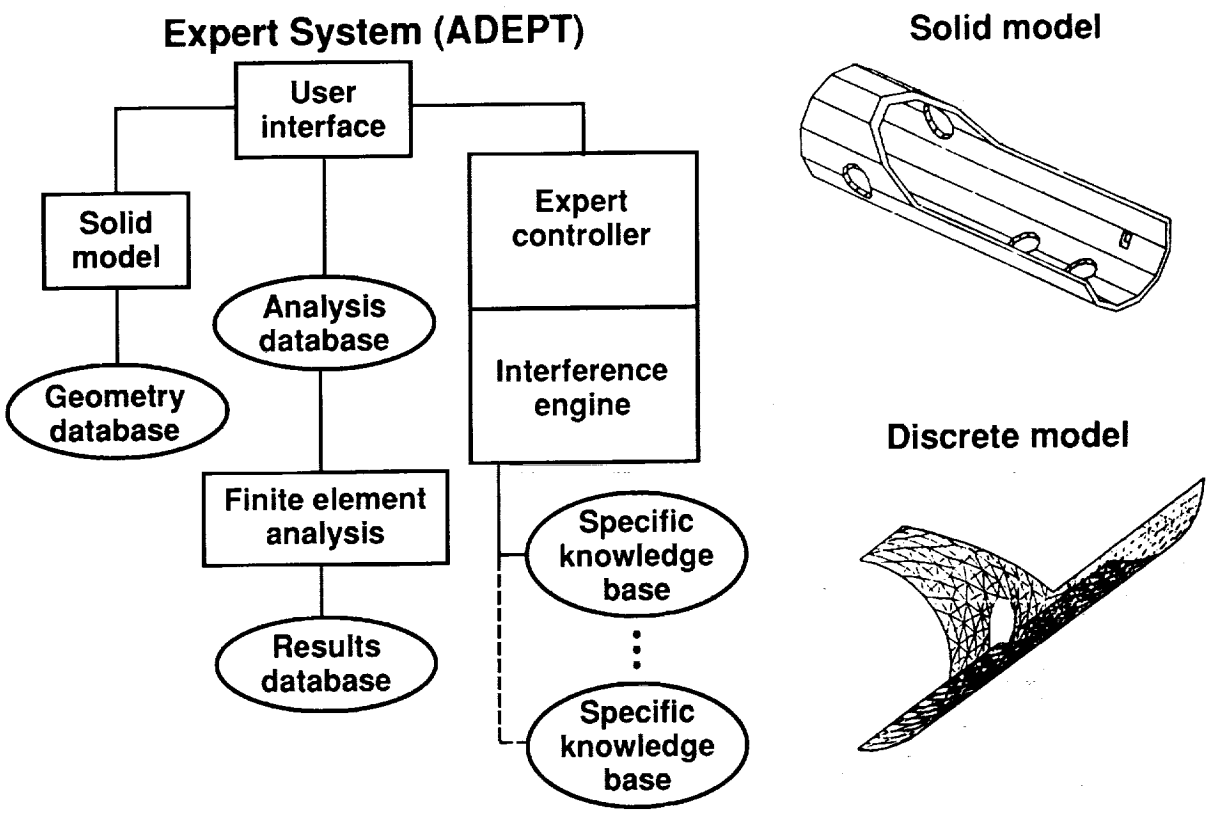
Propfan SR7 Optimization

	Initial design	Final design
Efficiency, %	82.86	83.17
Noise, dB	143.8	137.3
Weight, lb	41.1	41.2
Cost reduction, %		3.32



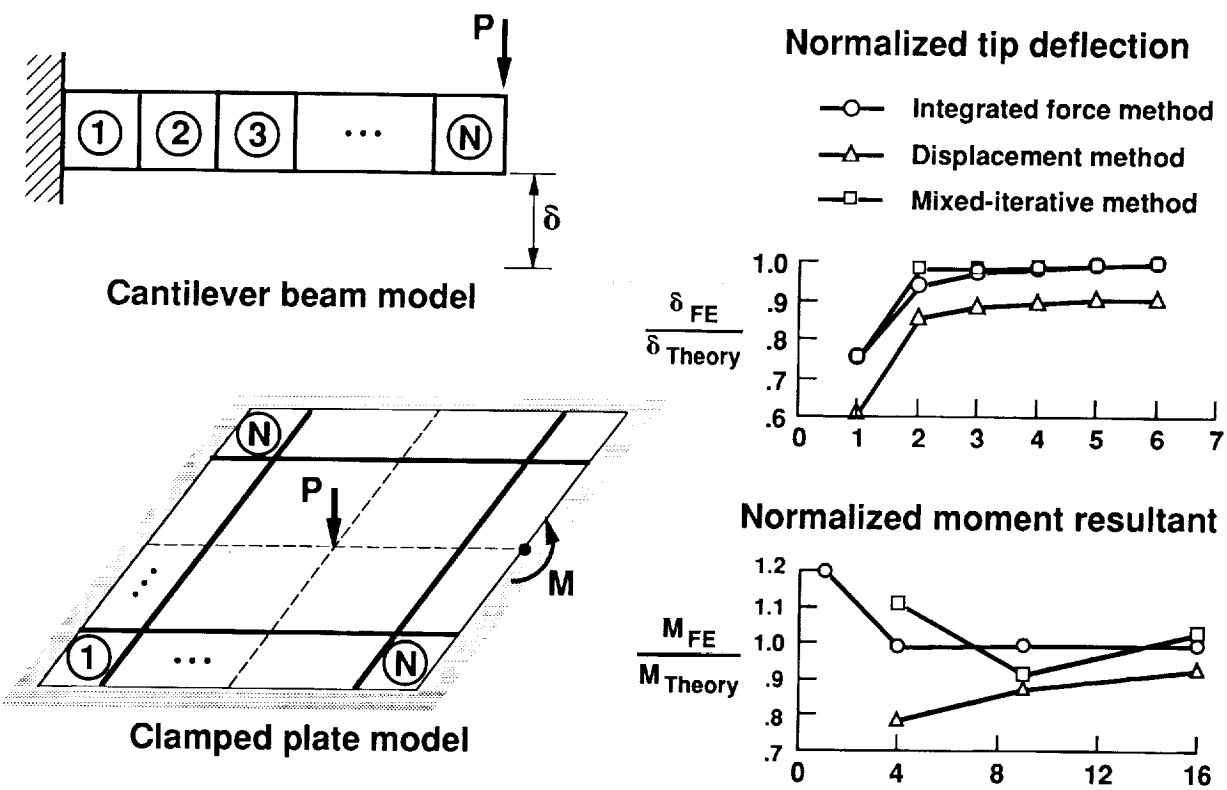
EXPERT SYSTEMS CAPTURE HEURISTIC KNOWLEDGE
TO GUIDE STRUCTURAL MODELING AND ANALYSIS

The creation of geometric and discrete models of structural components for analysis by widely used finite element methods remains a subjective process that relies to a great extent on the experience and judgement of the structural engineer. Of considerable interest is the notion of capturing the heuristic reasoning and knowledge that constitutes the structural design process. The potential benefit of such a concept is to enable less experienced engineers to consistently create more effective models and achieve more reliable analyses. The ability to configure an "advisor" for structural modeling and analysis has been demonstrated with the development of the automated design expert (ADEPT) system. ADEPT combines solid and discrete model creation facilities with an expert system that embodies knowledge pertaining to the assumptions and methodology of finite element structural analysis. ADEPT guides the engineer through an examination of various features of the component model including geometric attributes, loading, and boundary conditions. ADEPT makes recommendations for creating the appropriate and most effective discrete model for subsequent solution using specific finite element analysis application programs. As the figure illustrates, the feasibility of a specialized expert system such as ADEPT for assisting engineers in the structural modeling and analysis process has been demonstrated for complex configurations.



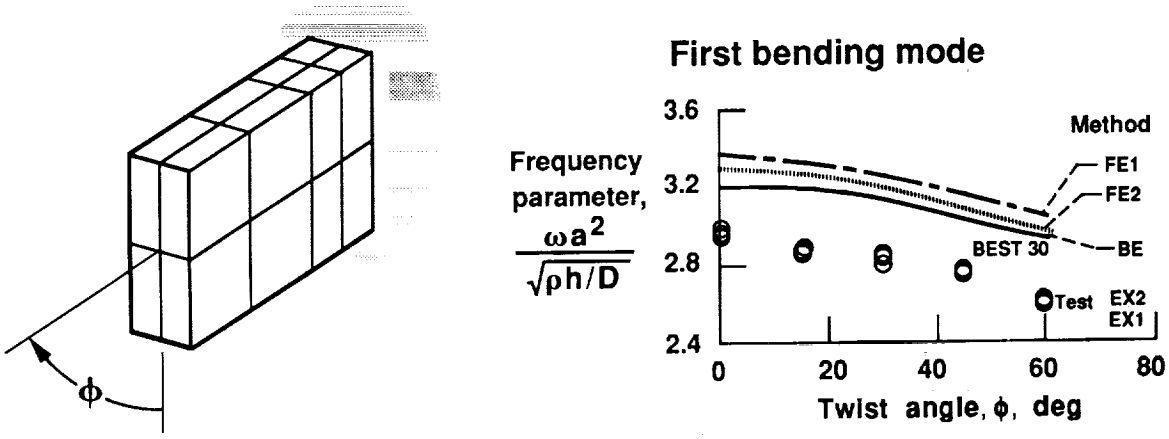
ADVANCED FINITE ELEMENT METHODS PROVIDE
MORE EFFECTIVE STRUCTURAL ANALYSES

Reliable and cost-effective engine component design requires accurate and efficient structural analysis tools. The finite element method is clearly the predominant tool utilized today to analyze complex structures. Furthermore, the majority of production level finite element analyses use programs based on the conventional displacement (stiffness) method concept which originated in the late 1950's. Despite this prominence, the displacement based finite element method exhibits deficiencies, especially in its ability to resolve internal force and stress fields. These limitations generally require the utilization of very dense finite element models with many degrees-of-freedom to adequately resolve important field quantities. The penalty of this is manifested in the person-time to create models, the computational resources to conduct analyses, and the difficulty of assessing the results. Two new formulations for finite element analysis are being developed, known as the mixed-iterative method and the integrated force method, to alleviate shortcomings of the displacement method. The results presented in the figure are for simple test cases that have been investigated to determine the potential benefits of the new methods. As seen, the new methods appear to provide more accurate representations of both displacement and stress fields with sparser models.



ADVANCED BOUNDARY ELEMENT METHODS ARE
A VIABLE ALTERNATIVE FOR STRUCTURAL ANALYSES

Although the boundary element method has been understood for nearly as long, compared to the finite element method it has received comparatively little attention and therefore has had limited utility as a structural analysis tool. The fundamental advantage of the boundary element method, in its numerical implementation, is that it requires discretization of only the surface of a structure and not its complete volume. This fundamental advantage, unfortunately, existed only for a limited class of problems for which the fundamental integral representation contained no volumetric components. Because the early perception was that boundary element methods would never be as generally useful as finite element methods, it received little development until only very recently. Concerted efforts have been applied in the last few years to extend the boundary element method to be viable for a broader class of problems. The focus has been to contend with the difficulties presented by engine component analyses including complex geometry, anisotropic and history dependent material behavior, cyclic loading, and heterogeneous boundary conditions. The recent efforts have culminated in comprehensive new boundary element structural analysis capabilities. The new capabilities provide a significant alternative to supplement the well-developed finite element capabilities.



Result designator	Description
FE1	Finite-element shell model (294 DOF)
FE2	Finite-element solid model (660 DOF)
BE	Boundary-element model (288 DOF)
EX1	NASA experimental data
EX2	AF experimental data

COMPUTATIONAL STRUCTURES TECHNOLOGY CAN ENHANCE FUTURE PROPULSION SYSTEMS

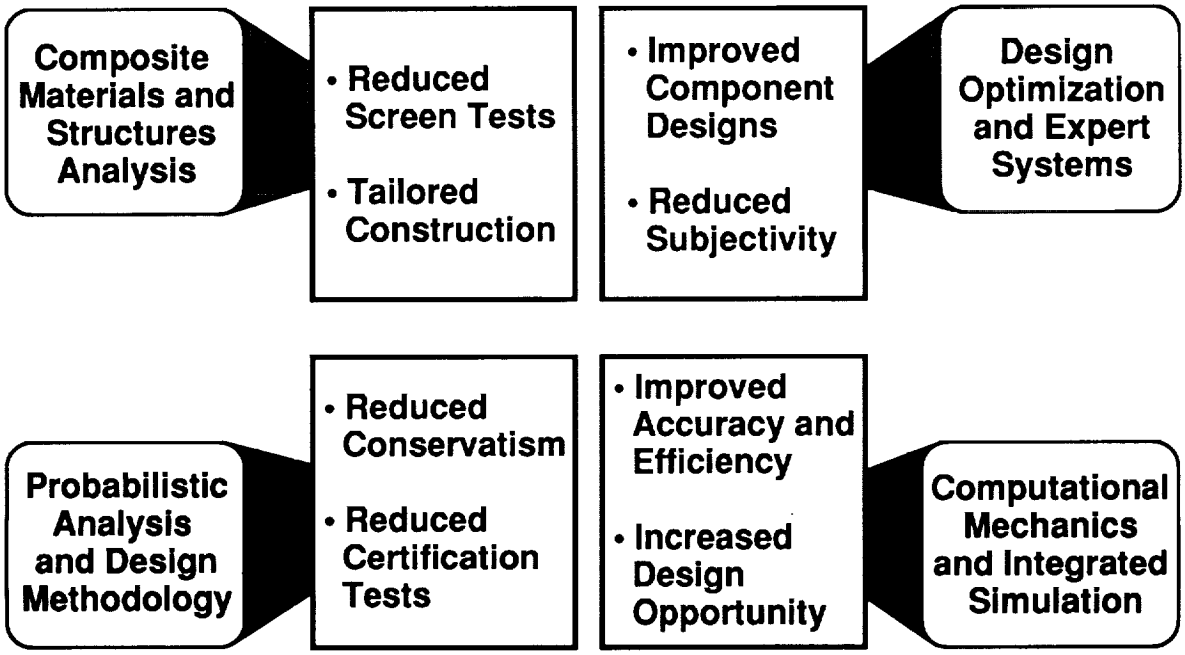
This presentation has given an overview of research and development efforts underway in the Structures Division of Lewis Research Center, which collectively can be referred to as the computational structures technology program. The benefits anticipated from this program will generally expand the role of computation far beyond the capability of today's analysis and design practice. To be more specific, some benefits expected in each of the four major categories discussed are:

(1) composite materials and structures - reduced requirements for candidate composite material screening tests and new opportunities for tailored material and structural design

(2) probabilistic analysis and reliability - reduced design conservatism and reduced requirements for hardware certification tests

(3) design optimization and expert systems -improved component designs and reduced subjectivity of the design process

(4) computational methods and simulation - improved accuracy and efficiency for structural analysis and expanded design opportunities to examine alternative concepts and to address other design issues



N92-25916

CSM Activities at the NASA Langley Research Center

Jerrold M. Housner
NASA Langley Research Center
Hampton, Virginia

PRECEDING PAGE BLANK NOT FILMED

C-2

CSM OBJECTIVE

AND

COMPUTATIONAL STRUCTURAL MECHANICS 5-YEAR PLAN

The objective and goals of the Computational Structural Mechanics (CSM) program as applied to airframe structures are given in the figure. It is recognized that the rapid evolution of computer hardware has opened up new opportunities for solving more complex and larger structural analysis problems than was hitherto imagined. To utilize these computers, new methods of computational structural mechanics are required. Methods are now being developed, assessed and validated to meet the goals shown in the figure. Each of these goals is addressed in subsequent figures. Plans and approaches are shown and highlights of results achieved in meeting these goals are given.

Three research thrusts are shown in the five-year plan: advanced robust CSM methods, large-scale solutions and validation/demonstration studies. The areas of research activity reflect the CSM goals. The mapping of developed methods onto high-performance and massively parallel computers is an integral part of the CSM five-year plan.

CSM OBJECTIVE

DEVELOP ADVANCED METHODS OF COMPUTATIONAL STRUCTURAL MECHANICS WHICH UTILIZE EVOLVING HIGH-PERFORMANCE COMPUTERS OF THE 1990'S

CSM PROGRAM GOALS

- o DEVELOP METHODS ENABLING INTEGRATED AND MORE COMPREHENSIVE STRUCTURAL SYSTEM ANALYSIS AND DESIGN
  - GLOBAL/LOCAL LINEAR AND NONLINEAR ANALYSES
  - INTEGRATED MULTIPLE METHODS
  - RAPID ANALYSIS/DESIGN METHODS FOR STRUCTURAL AND THERMAL RESPONSE
- o DEVELOP METHODS FOR ANALYSIS ERROR DETECTION AND CONTROL TO IMPROVE SOLUTION ACCURACY
- o DEVELOP METHODS FOR PROGRESSIVE FAILURE AND PROBABILISTIC ANALYSIS
- o DEMONSTRATE DEVELOPED CAPABILITIES ON APPLICATION PROBLEMS AND PERFORM VALIDATION STUDIES

COMPUTATIONAL STRUCTURAL MECHANICS

MAJOR ELEMENTS	FY 91	FY 92	FY 93	FY 94	FY 95	FY 96	EXPECTED RESULTS
ADVANCED ROBUST COMPUTATIONAL METHODS	ROBUST LINEAR AND NONLINEAR SOLVERS AND STRATEGIES						VALIDATED COMPUTATIONAL METHODS FOR STRUCTURAL ANALYSIS AND DESIGN
	ADAPTIVE MODELING AND ERROR CONTROL						
	INTEGRATED MULTIPLE METHODS						
	NONLINEAR ANALYSIS						
	THERMAL ANALYSIS						
	PROGRESSIVE FAILURE AND PROBABILISTIC METHODS						
	RAPID APPROXIMATE ANALYSIS/DESIGN						
	INTERDISCIPLINARY METHODS						
LARGE-SCALE SOLUTIONS	DEVELOPMENT AND DEMONSTRATION OF ADVANCED METHODS ON HIGH PERFORMANCE COMPUTERS						
AEROSPACE STRUCTURES VALIDATION AND DEMONSTRATION	V22 COMPONENT						
	HIGH-SPEED TRANSPORT						
	ADVANCED COMPOSITE TECHNOLOGY PROGRAM FOCUS COMPONENTS						
	SPACE, LAUNCH AND RE-ENTRY VEHICLES						

HIERARCHIC METHODS FOR AIRCRAFT

This figure and the one which follows it entitled, CSM METHODS FOR INTEGRATED STRUCTURAL SYSTEM ANALYSIS AND DESIGN, describe the general conventional design process which can be viewed as hierarchic in nature. Normally, the design process is broken down into many steps which are loosely coupled to one another. These steps run from global full-vehicle models all the way down to localized models of specific components and joints. Both special purpose and general purpose codes are used in the analysis and design process. Indeed, many special purpose codes reside in any particular airframe company to handle the component and detail designs. Interaction between the parts or steps of the process is basically the transfer of loads. Internal loads are generated at each step and are used as input to the more detailed analysis of the succeeding step. Although there may be some exceptions, the process is basically sequential and operates in one direction. Relatively late in the design, more complete and somewhat integrated models may be assembled.

There are certain shortcomings to this process as outlined in the figure. High-performance computers in conjunction with new methodologies can remedy some of these shortcomings. These methods will need to permit interaction between the steps of the process.

CSM METHODS FOR INTEGRATED STRUCTURAL SYSTEM ANALYSIS AND DESIGN

CONVENTIONAL ANALYSIS AND DESIGN PRACTICE

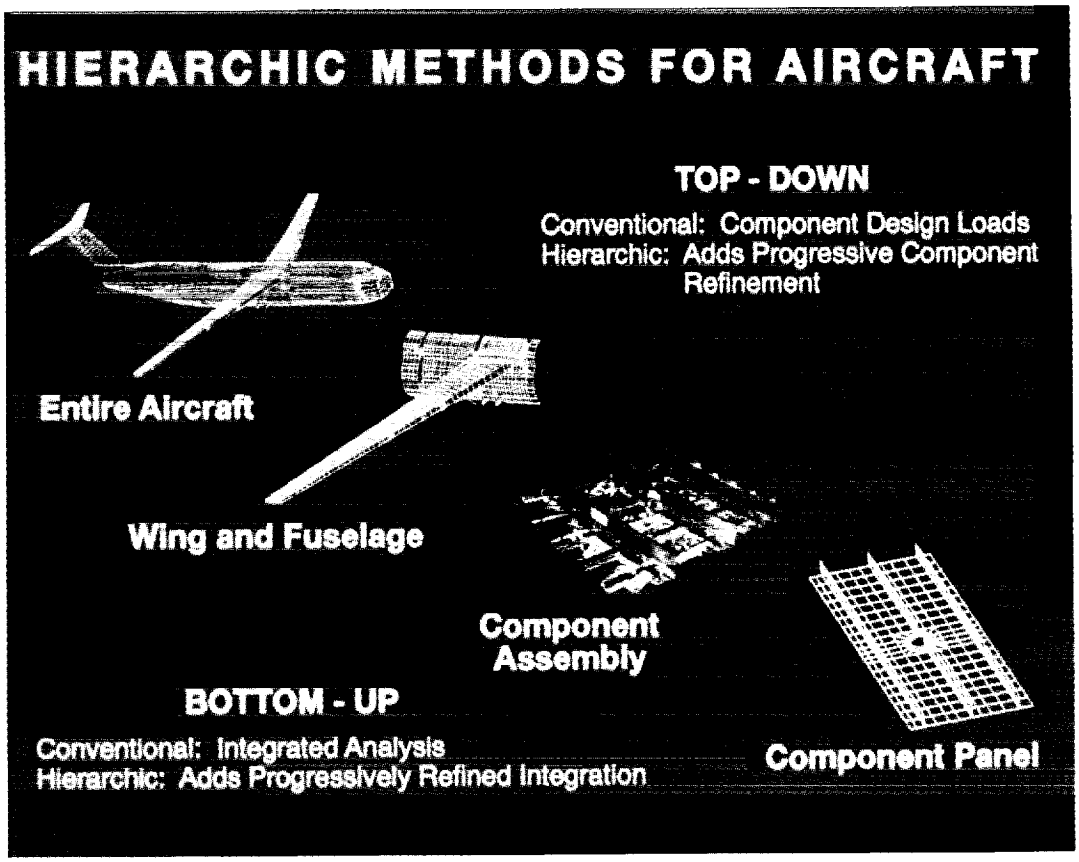
- o PROCESS BROKEN UP INTO MANY STEPS, FROM FULL-VEHICLE TO DETAIL BOLT HOLE DESIGN
  - MANY SEPARATE SPECIAL PURPOSE SOFTWARE PACKAGES
  - INTERACTION BETWEEN STEPS THROUGH INTERNAL LOADS

SHORTCOMINGS OF CONVENTIONAL PRACTICE

- o PROCESS IS BASICALLY SEQUENTIAL FROM TOP TO BOTTOM
- o DESIGNS AT EACH STEP ONLY AFFECT THOSE AT SUBSEQUENT STEPS
- o PROPAGATION OF DESIGN CHANGE TO PRECEDING STEPS COMES LATE
- o DETAILS ARE NOT INCORPORATED EARLY INTO THE DESIGN PROCESS
- o DIFFICULT TO INSERT RE-DESIGN INTO THIS PROCESS
- o COMPONENT BEHAVIOR WHICH AFFECTS ITS OWN LOAD IS NEGLECTED (e.g., NONLINEAR BEHAVIOR, COMPONENT STIFFNESS)
- o NOT AMENABLE TO PARALLEL COMPUTING SINCE SEQUENTIAL PROCESS

METHODS NEEDED WHICH

- o EMPLOY CONCURRENT ENGINEERING PHILOSOPHY IN STRUCTURAL DESIGN
- o PERMIT MORE INTERACTION BETWEEN THE STEPS OF THE PROCESS
- o ACCOUNT FOR MORE DETAIL EARLIER IN DESIGN PROCESS


HIERARCHIC METHODS FOR AIRCRAFT
TOP - DOWN
Conventional: Component Design Loads
Hierarchic: Adds Progressive Component Refinement
Entire Aircraft
Wing and Fuselage
Component Assembly
Component Panel
BOTTOM - UP
Conventional: Integrated Analysis
Hierarchic: Adds Progressively Refined Integration

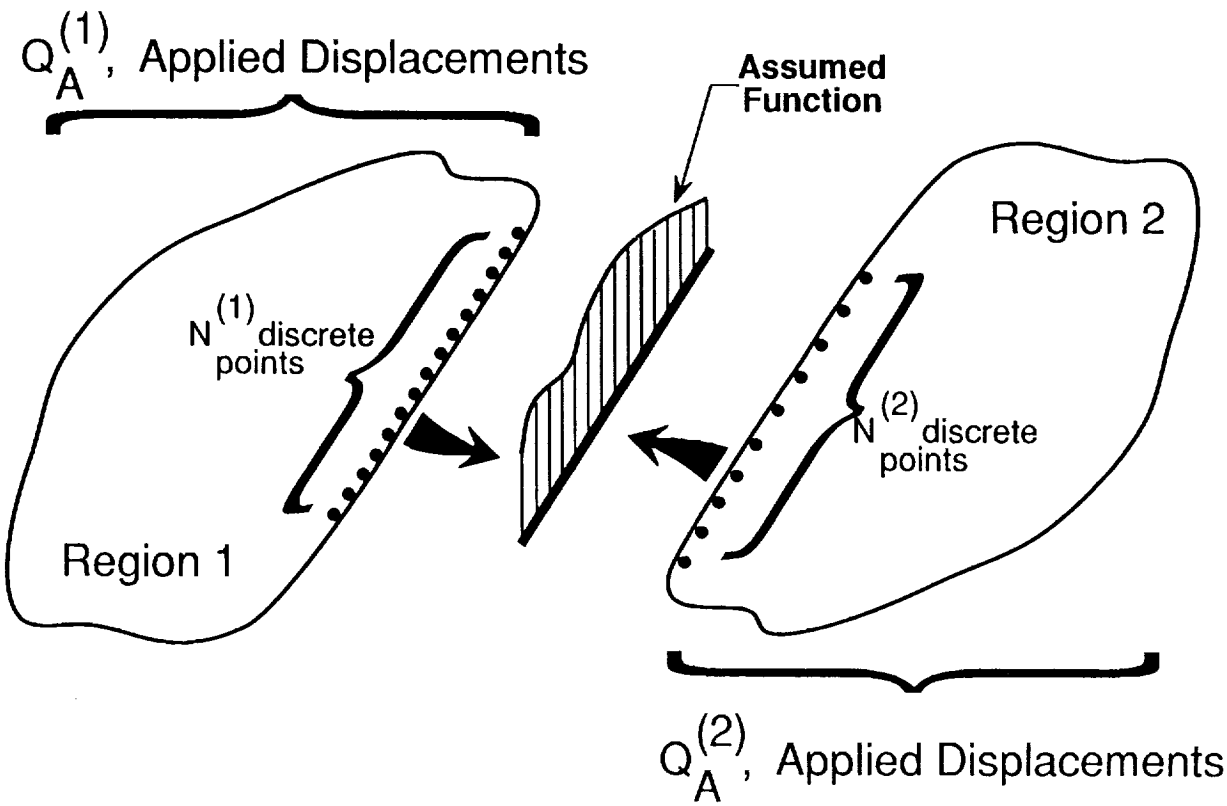
NEEDED METHODS DEVELOPMENTS FOR INTEGRATED ANALYSIS AND DESIGN

In order to achieve the integrated analysis capability discussed in conjunction with previous figures, it is necessary to develop new interface methods which will synthesize component models which may have been generated independently from one another and by different engineers or industrial organizations. These new interface methods should also allow local models to be embedded within global models and solved as a unified whole. Since some components or structural regions are better modeled using diverse approaches (e.g., finite elements, boundary elements, finite differences and Rayleigh-Ritz methods), the interface procedures must be developed to handle this variety of methodologies.

- o NEW INTERFACE METHODS FOR INTEGRATING STRUCTURAL COMPONENTS AND GLOBAL/LOCAL REGIONS
  - SYNTHESIZE STRUCTURAL COMPONENTS
  - EMBED DETAIL LOCAL MODELS IN GLOBAL MODELS
  - PERFORM SUB-STRUCTURING IN PARALLEL COMPUTING ENVIRONMENT
- o INTEGRATED MULTIPLE METHODS AND REGIONS
  - e.g., FEM, BEM, FD, RAYLEIGH-RITZ
- o IDENTIFICATION OF STRUCTURAL "HOT SPOTS" FOR SPECIAL DETAIL MODELING

INTERFACING TWO CONNECTED REGIONS VIA ASSUMED INTERFACE DEFORMATION FUNCTION

For a variety of reasons, structural models of different structural regions or components often do not have coinciding grid-points or in some other way are inconsistent with one another at their common interface. Some of these reasons are: the models use different discretization techniques, such as finite elements, finite differences, boundary elements or Rayleigh-Ritz approximations; the models may have been developed by different individuals, organizations or companies; or the models may have been developed for different purposes, such as one being a global relatively coarse model, while the other a local more refined model. Indeed, for global/local analysis, it is desirable to be able to easily change-out different models capable of performing different functions and modeling different levels of structural detail. To accomplish this, it is necessary to remove the need for complex transitional regions which link up the refined local model with the global model. This chart illustrates an effective way of accomplishing this using an interface function which contains free parameters and approximates the deformation or stress state at the interface (see Ref. 1). The degrees-of-freedom associated with the grid points of each region are appropriately tied to the function via either slaving, least squares or some other technique. Some degrees-of-freedom will be eliminated through this process and the free parameters of the interface function will become new variables of the problem. Thus, the modeling of each region becomes independent of the other and the transition between the regions is accomplished via the interface function. The next chart addresses one particular formulation for deriving an interface procedure.



VARIATIONAL FORMULATION: LAGRANGE MULTIPLIER

As discussed in conjunction with the previous chart, degrees-of-freedom associated with the models of each region at the interface are appropriately tied to the assumed interface function so that the appropriate compatibility is achieved. The necessary compatibility equations which are consistent with the assumed function and with the method employed to tie to the interface function may be derived using a variational approach. In the example of this chart, a Lagrange multiplier approach is employed. Of course, it is not necessary to use Lagrange multipliers, direct application of the interfacing constraints can be performed or a penalty method may be used. All of these are present areas of research. In the figure, u represents vector containing the degrees-of-freedom of the regional model and v represents the assumed function which has free parameters denoted by "a". The Lagrange multiplier can also be expanded as a series of assumed functions. The stationary value of the energy-like functional provides an approximation to the Lagrange multipliers, equilibrium at the interface and compatibility consistent with the assumptions. The resulting coupled equations of motion which come from the stationary value of the functional are shown. With such formulations it is easy to change-out the regional models (diagonal stiffness matrix blocks) or the interface function (the M and G matrices). Some succeeding charts address the application of these interface methods.

Define: $\overline{\Pi} = \Pi + \int_{S_i} \boldsymbol{\lambda}^T (\mathbf{v} - \mathbf{u}) ds.$ **Then,** $\delta\overline{\Pi} = 0 \Rightarrow \begin{cases} \boldsymbol{\lambda} = \mathbf{n}^T \boldsymbol{\sigma} \\ \boldsymbol{\lambda}^1 + \boldsymbol{\lambda}^2 = 0 \quad \text{on } S_i. \\ \mathbf{u} = \mathbf{v} \end{cases}$

Assume: $\mathbf{u} = \mathbf{N}\mathbf{q}, \quad \mathbf{v} = \mathbf{T}\mathbf{a}, \quad \boldsymbol{\lambda} = \mathbf{R}\boldsymbol{\alpha}.$ **and** $\overline{\Pi} = \Pi + \boldsymbol{\alpha}^T \mathbf{G}^T \mathbf{a} + \boldsymbol{\alpha}^T \mathbf{M}^T \mathbf{q}$

$\delta\overline{\Pi} = 0$ gives:

$$\begin{bmatrix} \mathbf{K}^1_{ii} & \mathbf{K}^1_{io} & 0 & 0 & 0 & \mathbf{M}_1 & 0 \\ \mathbf{K}^1_{oi} & \mathbf{K}^1_{oo} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & \mathbf{K}^2_{ii} & \mathbf{K}^2_{io} & 0 & 0 & \mathbf{M}_2 \\ 0 & 0 & \mathbf{K}^2_{oi} & \mathbf{K}^2_{oo} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & \mathbf{G}_1 & \mathbf{G}_2 \\ \mathbf{M}^T_1 & 0 & 0 & 0 & \mathbf{G}^T_1 & 0 & 0 \\ 0 & 0 & \mathbf{M}^T_2 & 0 & \mathbf{G}^T_2 & 0 & 0 \end{bmatrix} \begin{bmatrix} \mathbf{q}^1_i \\ \mathbf{q}^1_o \\ \mathbf{q}^2_i \\ \mathbf{q}^2_o \\ \mathbf{a} \\ \boldsymbol{\alpha}_1 \\ \boldsymbol{\alpha}_2 \end{bmatrix} = \begin{bmatrix} \mathbf{F}^1_i \\ \mathbf{F}^1_o \\ \mathbf{F}^2_i \\ \mathbf{F}^2_o \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

FOCUS PROBLEMS FOR INTEGRATED MULTIPLE METHODS AND INTERFACING PROCEDURES

This chart summarizes the present research path being pursued to assess the performance of integrated multiple methods and the interfacing procedures. Presently, fundamental validation studies are being carried out on classical problems involving beams, plates and shells where localized stress concentrations or singularities exist. Simultaneously, practical problems involving advanced composite designs in aircraft fuselage and wings are being pursued to assess the handling of cracks, cutouts and discontinuous stiffeners in built-up composite components. Following this, it is planned to demonstrate integrated multiple methods on a high speed civil transport configuration. The wing box can be modeled using equivalent plate methods (see Ref. 2), and the leading/trailing edges by finite elements. Equivalent plate models work well on the wing box, but are more cumbersome and less efficient on the curved surfaces of the airfoil. Adaptive mesh refinement techniques, discussed on subsequent charts can be used to identify stress concentrations and then local models can be embedded into the multiple method wing model to resolve the highly-stressed regions.

- o FUNDAMENTAL VALIDATION CASES

  CLASSICAL BEAM, PLATE AND SHELL CASES WITH LOCAL STRESS CONCENTRATIONS

  FEM-to-FEM, BEM-to-FEM, Rayleigh-Ritz-to-FEM

- o ADVANCED COMPOSITES TECHNOLOGY (ACT) FUSELAGE AND WING PANELS WITH CRACKS, CUTOUTS AND DISCONTINUOUS STIFFENERS

- o HIGH SPEED CIVIL TRANSPORT

  INTERFACE EQUIVALENT PLATE WING BOX TO FEM LEADING AND TRAILING EDGES

  EXERCISE ADAPTIVE MESHING TO IDENTIFY STRESS CONCENTRATIONS

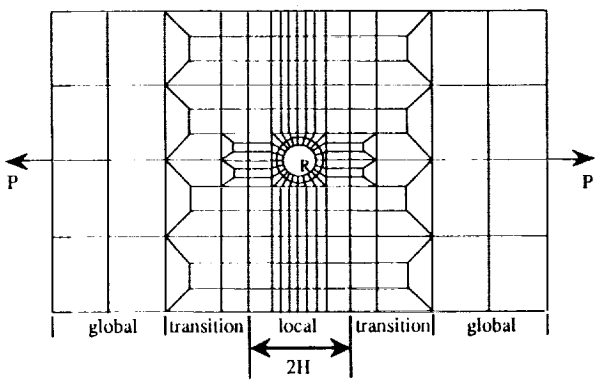
  EMBED LOCAL MODELS AT IDENTIFIED HIGHLY-STRESSED REGIONS

FUNCTIONAL INTERFACE METHOD SIMPLIFIES FINITE ELEMENT GLOBAL/LOCAL MODELING

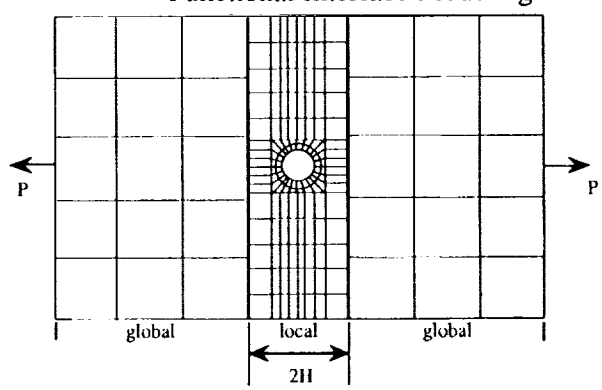
Conventional finite element global/local modeling typically involves the development of a transition region between the relatively coarse global mesh and the finer local mesh. The use of a transition mesh has three shortcomings. First, the development of such a mesh is time consuming; second, the engineer is never sure where to place the transition mesh and how far it should extend (i.e., the appropriate extent of the global and local models is not known a priori); and third, changing the location and extent of the transition is tedious and thus not usually done. The functional interface modeling described on previous charts eliminates the need for a transition region. Indeed, the functional interface becomes the transition region. However, unlike the transition region model, its accuracy is easily controlled.

The chart illustrates the classical case of an isotropic plate with a circular hole under uniaxial tension. The conventional and functional interface models are shown, and the accuracy of the functional interface approach is assessed by examining the predicted hole stress concentration as the position of the functional interface or size of the local model is varied for different interface polynomials. (A slaving procedure is used here to tie the finite element degrees-of-freedom to the interface functional.) Notice that very good results are derived even when the global/local interface is two radii from the hole center (i.e., one radii from the hole edge). The functional interface modeling makes it easy to conduct this study since modeling of a transition region has been eliminated.

Conventional Modeling

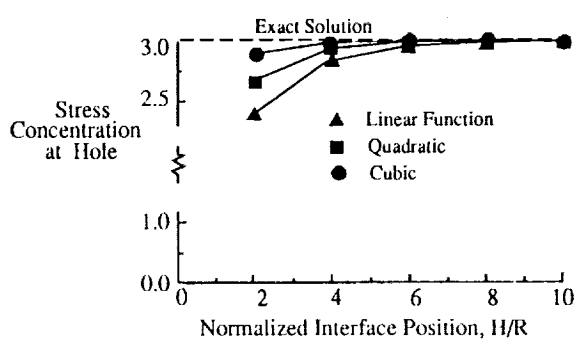


Functional Interface Modeling



- No Tedious Transition Region Modeling
- Grid Points Along Interface Need Not Coincide
- Requires Fewer Degrees of Freedom
- Retains Accuracy

Accuracy of Functional Interface Method



INTEGRATED MULTIPLE METHODS AND POTENTIAL BENEFITS OF INTEGRATED MULTIPLE METHODS

As shown in the previous figures, research is being carried out to efficiently interface regions or components whose models have been independently developed. With these interface methods, the analysis techniques used in each component can be different. For example, finite elements can be used in one region and boundary elements in another. Thus, this area of research has been referred to as integrated multiple methods. Quite naturally, three categories of these methods can be defined on the basis of the type of interface required. In the first category, the interface is defined on the basis of the physical geometry. Examples in this category are component substructuring, modal synthesis and interdisciplinary analyses. In each of these the interface is well-defined. Since the interface is a physical juncture, a tight and detailed interfacing approach is required to capture the critical load state which will arise at the interface, for it is here that likely failure will initiate. In the second category fall those interfaces which, while physical in nature, are user selected rather than set by geometry. Examples of these are global/local modeling, 2D-to-3D modeling and use of different transient algorithms in different regions. The interfacing method used for this second category need not be as detailed as the first, since the interface tends to lie in a less highly stressed area. In the third category are the non-physical interfaces; that is, the interface is strictly mathematical. Examples in this category are predictor-corrector methods, hybrid methods, re-analysis processes, multigrid methods and hierarchic methods. Having defined the three categories, the next chart summarizes the benefits of integrated multiple methods. [The chart is fairly self-sufficient and no narrative is presente for it.]

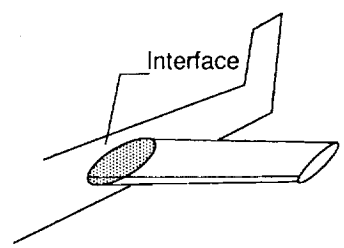
INTEGRATED MULTIPLE METHODS

Definition: The joining together of two or more different modeling methods which operate in an interactive fashion

Categories

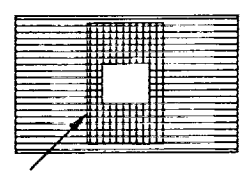
1. PHYSICAL INTERFACE WELL-DEFINED

- COMPONENT SUBSTRUCTURING
- MODAL SYNTHESIS
- INTERDISCIPLINARY METHODS



2. PHYSICAL INTERFACE NOT WELL-DEFINED (USER SELECTED)

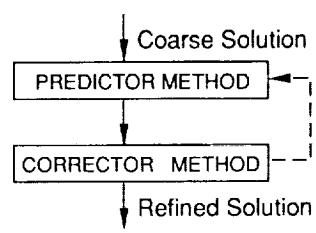
- GLOBAL-LOCAL MODELING
- MULTIPLE TRANSIENT ANALYSIS REGIONS
- 2D-3D MODELING



3. MATHEMATICAL INTERFACE

- INTERFACE IS TRANSFORMATION BETWEEN MULTIPLE ANALYSIS PHASES

  e.g., PREDICTOR/CORRECTOR METHODS
  HYBRID METHODS
  RE-ANALYSIS PROCESSES
  MULTI-GRID METHODS
  HIERARCHIC METHODS



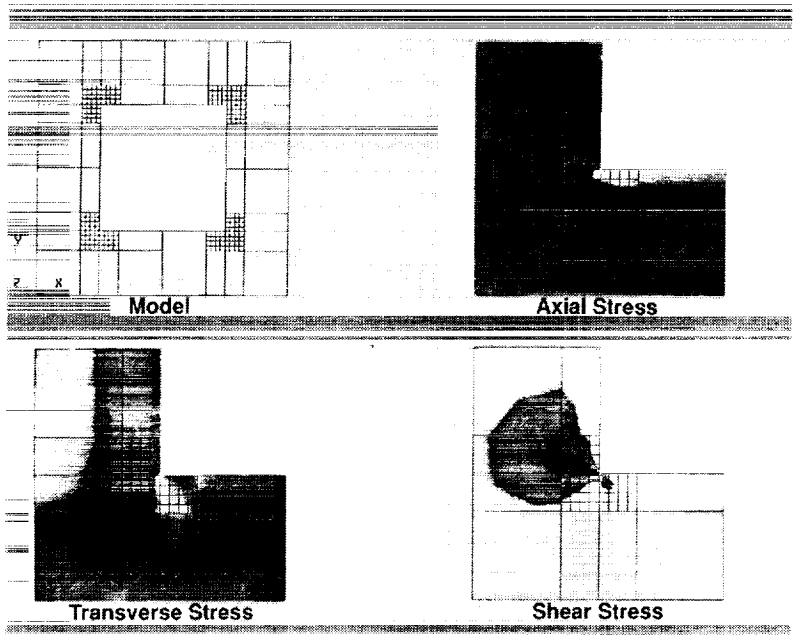
POTENTIAL BENEFITS OF INTEGRATED MULTIPLE METHODS

- UTILIZES PREFERRED ANALYSIS METHOD IN THE MOST APPROPRIATE REGION
- ALLOWS INDEPENDENT REFINEMENT OF COMPONENTS AND REGIONAL MODELS
- TAKES ADVANTAGE OF PARALLEL COMPUTING BY SPLITTING UP COMPUTATIONAL EFFORT BY REGION
- ENHANCES EFFICIENCY OF FINITE ELEMENT METHOD WITHOUT COMPROMISING ACCURACY
- MAINTAINS OR ESTABLISHES MODULARITY
- REDUCES DESIGN CYCLE TIME BY PROVIDING EASIER MEANS OF MODEL AND COMPONENT INTEGRATION

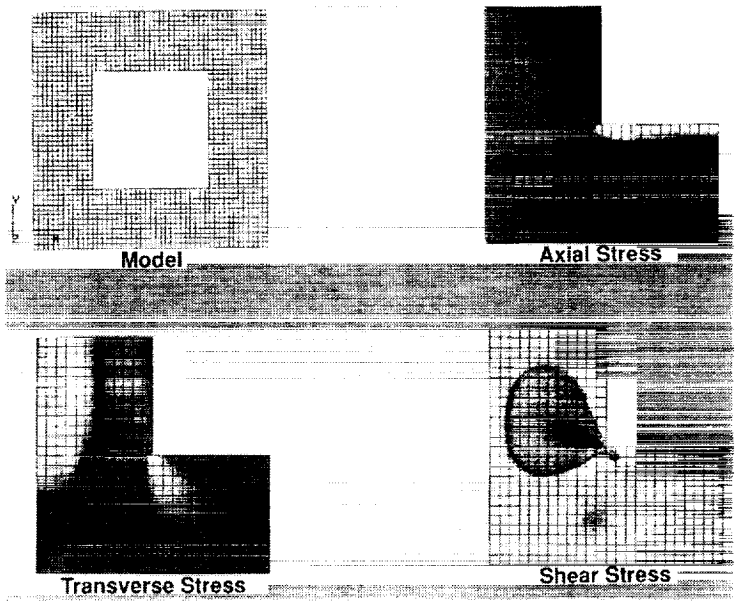
ACCURACY OF CORNER STRESSES USING FUNCTIONAL INTERFACE GLOBAL/LOCAL MODELING

A square plate with a square cutout under uniform tension is the subject of these two figures which are entitled TENSILE LOAD ON SQUARE PLATE WITH CUTOUT: INTERFACE METHOD. The axial, shear and transverse stress resultants as predicted using a fine uniform mesh are presented on one figure and a global/local model using a coarse global and a fine uniform mesh tied together with a piecewise cubic spline functional interface are shown on the other figure. A comparison of the two figures indicates that the functional interface method provides accurate stress predictions when compared with the fine uniform mesh. Moreover, and very significantly, the discontinuity in the modeling of the global and local meshes does not produce a discontinuous stress state. Results such as those shown in these figures and the preceding one, indicate the potential of the functional interface method for global/local modeling.

TENSILE LOAD ON SQUARE PLATE WITH CUTOUT: INTERFACE METHOD



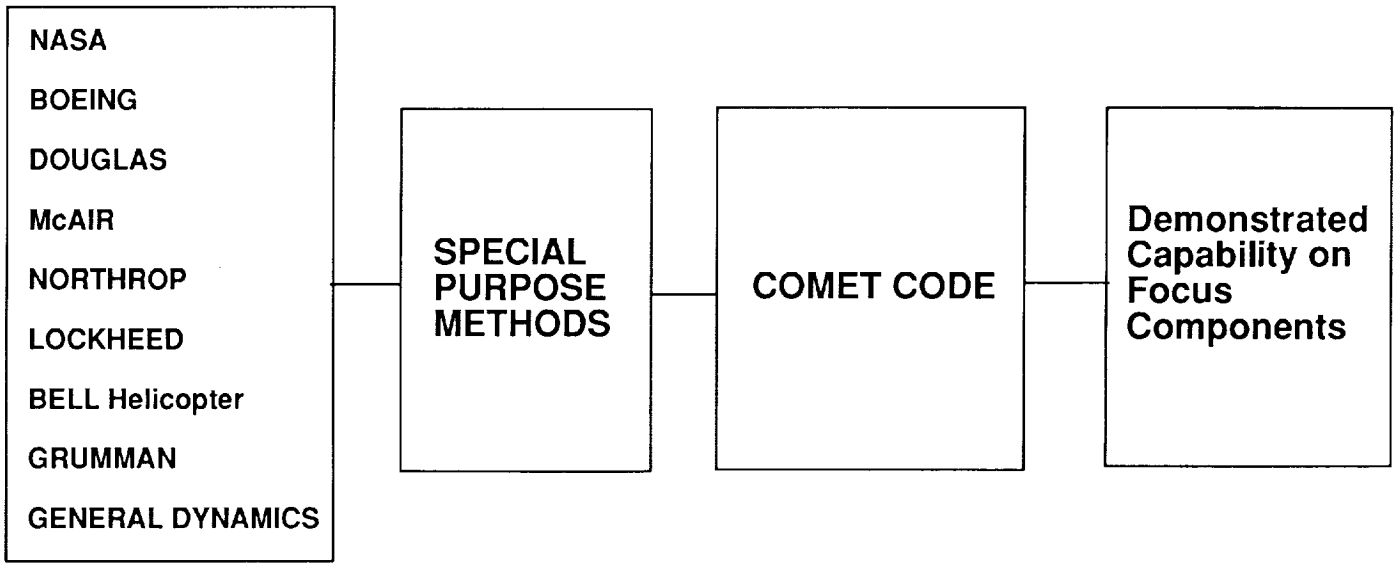
TENSILE LOAD ON SQUARE PLATE WITH CUTOUT: UNIFORM MESH



NASA/INDUSTRY DEVELOPMENT PARTNERSHIP

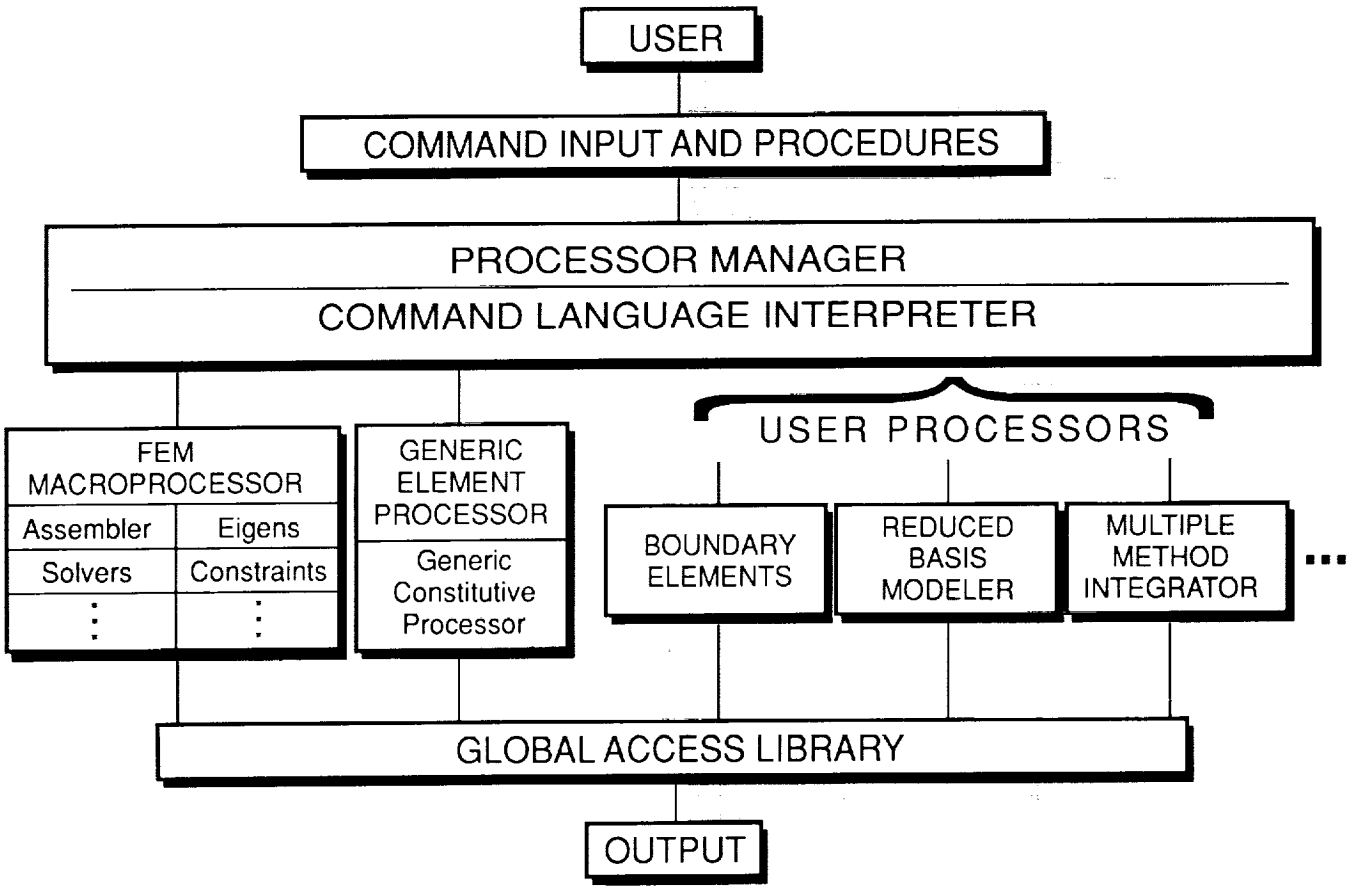
To advance the development of computational structural mechanics methods as needed by the U.S. aircraft industry, a partnership between NASA and the aircraft industry companies has been formed. NASA and their partners are preparing to place special purpose methods software into NASA's COMET code which is described in the next figure. Some of this special purpose software treats localized structural response and can be tied to the finite element capability of the COMET code through the functional methods described in previous figures. The merging of the special purpose and global finite element capabilities permits more detail modeling to be captured in global models earlier in the design process. It is believed that a more comprehensive early analysis should help to defray costly fixes arising from detail design late in the design process. Some other special purpose software being incorporated into COMET provides material constitutive modeling for variable temperature and moisture environments.

OBJECTIVE: To advance the development of CSM methods which will enable more cost-effective designs of aerospace vehicles with special emphasis on composite aircraft.



COMET RESEARCH CODE

The COMET (COmputational MEchanics Testbed) code is a research code which permits new methods of computational structural mechanics to be assessed and enhanced within a general purpose nonlinear finite element structural analysis framework (see Ref. 2). The user communicates with the code through a command language and may use PATRAN or some other commercial graphics modeling code as a front-end to COMET. New methods may be inserted as processors. For installing special purpose localized analysis, the multiple methods integrator is employed to tie the localized analysis to methods operating in different structural regions. The multiple methods integrator uses the functional interface methods described in preceding figures.

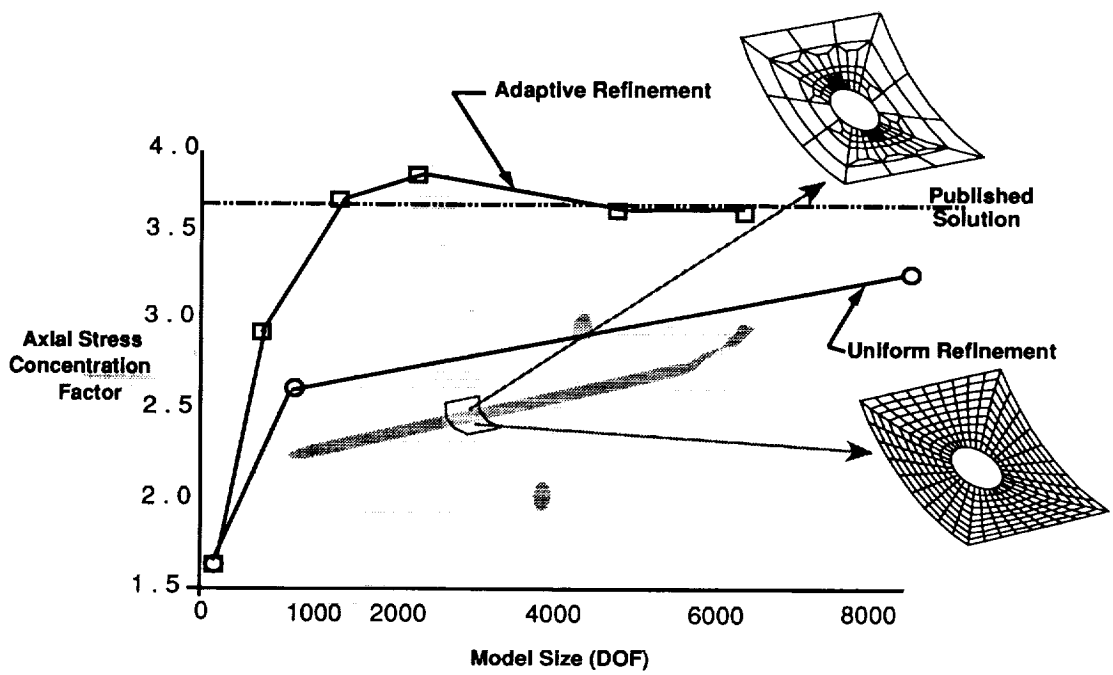


PERFORMANCE OF ADAPTIVE MESH REFINEMENT DEMONSTRATED ON COMPOSITE FUSELAGE-LIKE COMPRESSION PANEL

Considerable research and development of automated mesh generation techniques guided by refinement indicators has been performed for in-plane structural problems and to a lesser degree for three-dimensional models composed of brick elements. However, very little has been done for shell type structures, typical of aerospace vehicle construction.

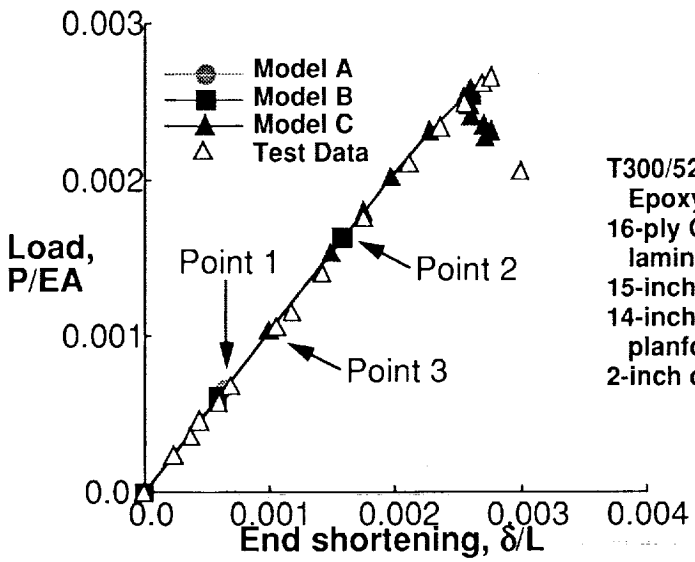
In this figure, a comparison is made of two automatic finite element mesh refinement techniques on a graphite epoxy curved fuselage-like compression panel containing a cutout. In one technique, the mesh is graded uniformly while in the other, it is graded adaptively at strategic locations. Each technique uses only quadrilateral elements because for shell structures, these are usually more accurate and robust than triangular elements. The initial mesh contains only four finite elements. The response is predicted at each progressive step of mesh refinement and refinement indicators based on stress intensity are used to determine when to remesh and for the adaptive refinement, where to remesh by identifying high stress gradient regions. A user selected tolerance is employed to terminate refinement.

A comparison of the performance of the two techniques for predicting the stress concentration factor at the cutout is shown. The model size of degrees-of-freedom increases with automatic refinement. The adaptive refinement technique requires less than 1000 degrees-of-freedom for accuracy because the meshing tends to zoom in on the high-stressed region around the cutout, whereas the uniform refinement is still inaccurate even at 6000 degrees-of-freedom. The published solution is accepted as the true stress concentration and adaptive technique does overshoot this answer. This is apparently due to element distortion which naturally occurs in the transition region between the fine mesh at the cutout and the coarse mesh away from the cutout. Distortion of quadrilateral elements significantly degrades their accuracy. The research shown on previous figures which allows independent modeling of structural regions should eliminate the need for transition modeling with its inherent inaccuracies.

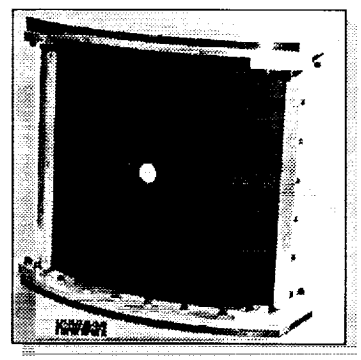


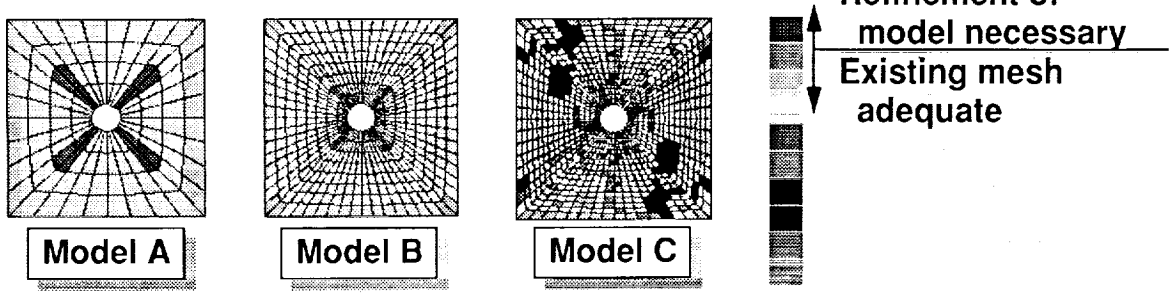
ADAPTIVE ANALYSIS PROCEDURE DEMONSTRATED ON NONLINEAR RESPONSE OF COMPOSITE PANEL

In this figure, results for a nonlinear analysis using the automated uniform mesh technique are provided for the same example problem of the previous figure. When applied to a nonlinear analysis, the automatic uniform mesh becomes adaptive in the sense that the mesh must be updated as the load is increased and the response becomes more nonlinear. Four different converged meshes are used up to failure (three of these are shown). When the refinement indicators trigger a mesh refinement, the load is decreased back to the last converged solution. The mesh is then refined and a new solution using the new mesh is sought before increasing the load again. Load-end shortening results are in excellent agreement with experiment right up to failure.



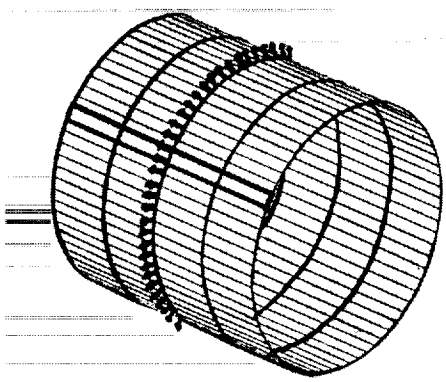
T300/5208 Graphite-Epoxy Tape
16-ply Quasi-isotropic laminate
15-inch panel radius
14-inch square planform
2-inch circular cutout



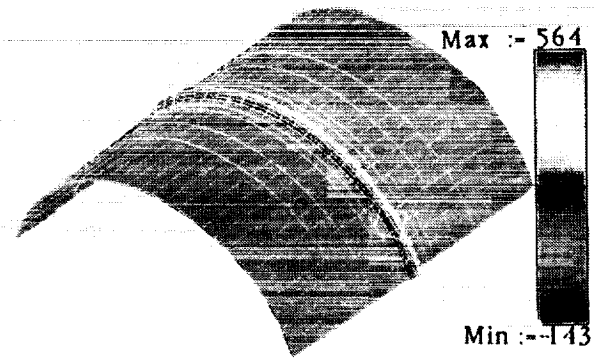


PERFORMANCE OF ADAPTIVE MESH REFINEMENT DEMONSTRATED ON FUSELAGE-LIKE STRUCTURE

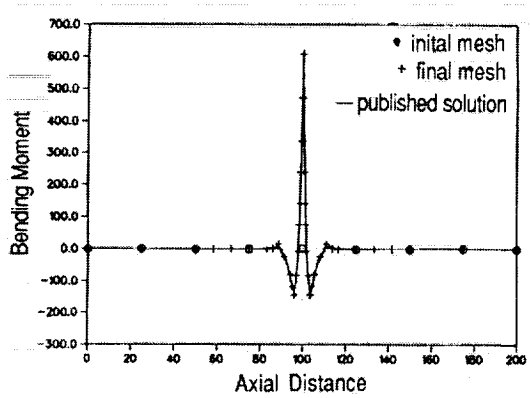
In this figure, the adaptive meshing discussed in connection with previous figures is examined on a cylindrical shell structure subject to a ring load. This application is of special interest since its response will have characteristics similar to that of a ring-stiffened fuselage under internal pressure. Because a high gradient bending moment arises in the vicinity of the ring load, the refinement indicators lead the adaptive mesh to zoom in on this region. Comparison with a closed-form solution by C. Steele at Stanford University demonstrates that the adaptive refinement captures the response quite well.



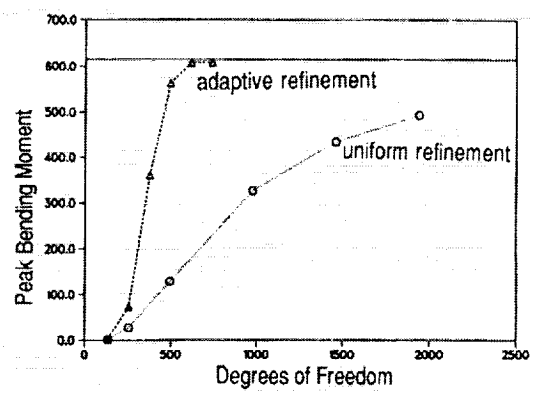
Cylinder with Radial Ring Load



Bending Moment Distribution



Accuracy of Bending Moment



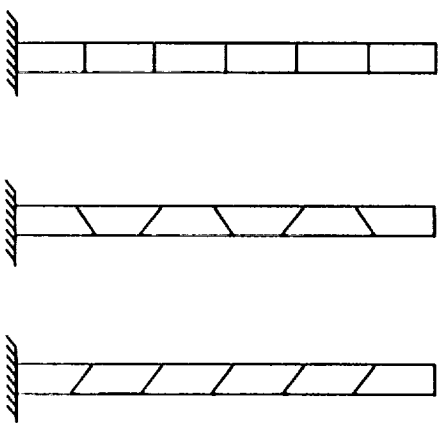
Convergence of Adaptive and Uniform Refinement

MODELING PERFORMANCE OF PLATE ELEMENTS ON CANTILEVER BEAM

As alluded to in discussing previous figures, the distortion of commonly-used quadrilateral finite elements tends to rapidly deteriorate their performance. This has led several researchers to develop finite elements which are more robust with respect to element distortion. This table presents a performance comparison of typically used finite elements as well as a newly developed more robust element (see Refs. 3 and 4 for more information concerning these elements). A cantilever beam modeled by quadrilateral finite elements and subject to either an in-plane tip load, an out-of-plane tip load or a tip twist is considered in this figure. Three quadrilateral element shapes are examined, namely rectangular, trapezoidal (with 45 degree angles) and parallelogram (with 45 degree angles). Tip displacement in the direction of the loading normalized to the exact value is provided as a performance measure. Thus, a value of unity represents perfect performance. Note that some elements perform extremely poorly and models built of these elements and subjected to the corresponding behavior require a very refined mesh for accurate modeling. In transition regions produced by adaptive refinement, distorted elements naturally appear and the mesh will of necessity be somewhat coarse. Fortunately, the adaptive meshing does tend to push the transition region out beyond the high stress gradient region, but unless robust elements are used, such will occur with an over-refined mesh. The new element appears to have an improved robust character.

$L = 6.0,\ h = 0.2,\ t = 0.1$

$E = 10^7,\ \nu = 0.3$



Tip Loading Direction	Normalized Tip Displacement in Direction of Loads			
	Quad4 (Nastran)	ANS (Lockheed)	STAGS/410 (Lockheed)	New Element
In-Plane Shear	.904	.904	.915	.993
Out-of-Plane Shear	.986	.980	.986	.981
Twist	.941	.856	.680	1.011
In-Plane Shear	.071	.305	.813	.986
Out-of-Plane Shear	.968	.763	-	.965
Twist	.951	.843	-	1.009
In-Plane Shear	.080	.324	.794	.972
Out-of-Plane Shear	.977	.939	.991	.980
Twist	.945	.798	.677	1.010

PROGRESSIVE FAILURE OF COMPOSITE STRUCTURES

These two charts describe the scope of the research now underway at the NASA Langley Research Center on progressive failure of composite structural components. Progressive failure prediction goes beyond failure initiation (often referred to as first-ply failure) and attempts to predict failure growth out to final component failure. As indicated there are many sources of failure initiation. Cut-outs, impact damage and stiffener run-out are presently receiving significant attention. Attempts are being made to model all failure modes.

Various failure analysis methods are being developed. One class of such approaches is based on the relatively simple ply discount theories which eliminate or reduce the contribution of plies to the composite stiffness when the plies have been predicted to fail. All methods rely on detail (often global/local) modeling for accurate stress calculation. Thus, the integrated multiple methods and global/local research discussed in connection with previous figures is fundamental to progressive failure prediction. Also, the progressive failure methods are being implemented in the COMET code discussed earlier.

APPROACH

- Developing progressive failure analysis methodologies
  - Ply discount schemes
  - Continuum damage mechanics model
  - Delamination analysis methodologies
  - Nonlinear analysis
- Implement progressive failure methodologies in COMET
- Apply progressive failure analysis methodologies to aircraft structures
- Initiate development of advanced local detail stress analysis methodologies
  - Stiffener/skin interface analysis
  - Interlaminar stress analysis
  - Joint analysis
- Develop integrated multiple methods for efficient and accurate analysis
  - BEM/FE, Finite Strips/FE
  - 2D/3D Global/Local

PROGRESSIVE FAILURE OF COMPOSITE STRUCTURES

WHAT IS PROGRESSIVE FAILURE?

FAILURE INITIATION → **FAILURE GROWTH** → **FINAL FAILURE**

Sources:
- Cut-out
- Impact damage
- Manufacturing defects
- Out-of-plane stresses
- Eccentric load path (stiffener run-out)
- Structural instability

Modes:
- Matrix cracking/crushing
- Fiber breakage or buckling/kinking
- Delamination
- Structural buckling induced skin-stiffener separation

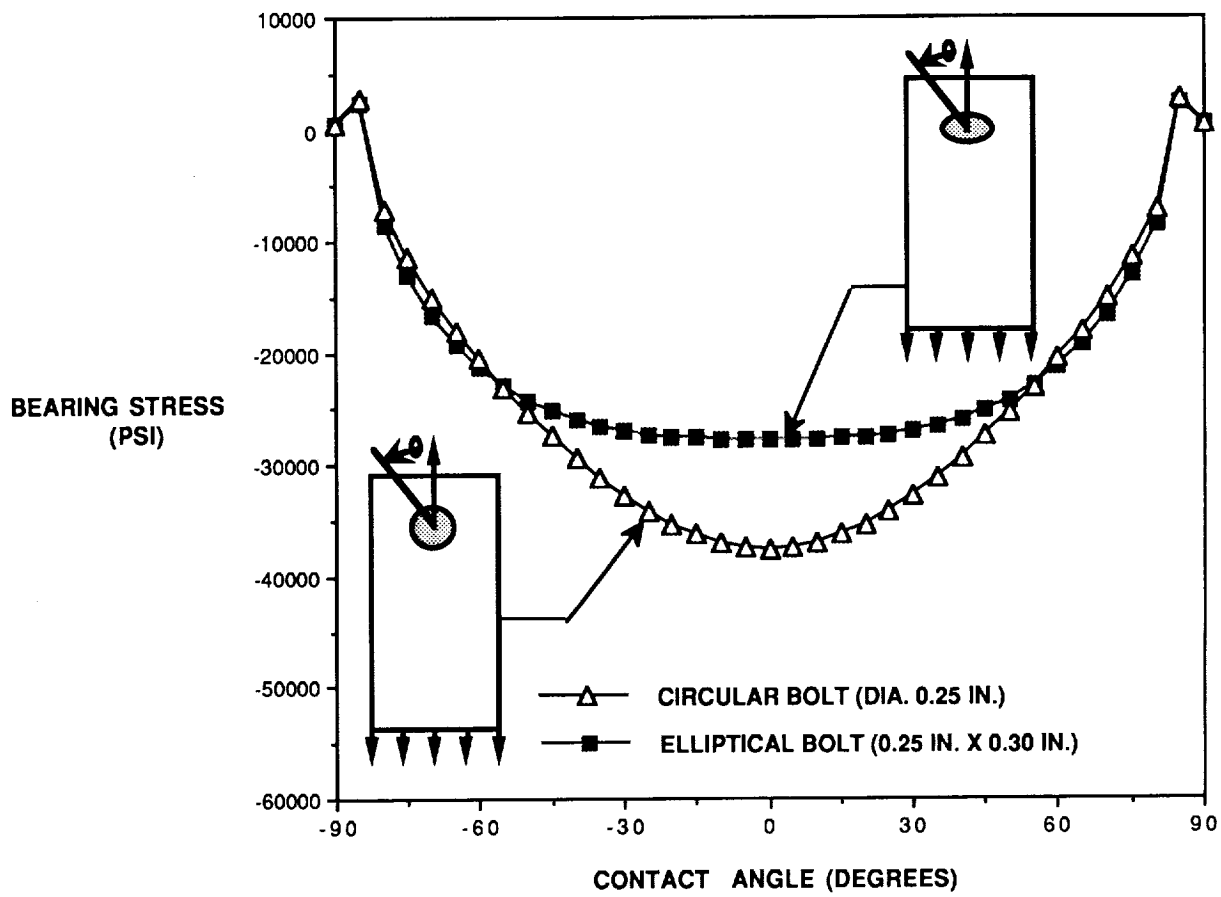
Loadings:
- Static
- Impact
- Fatigue

Design Requirements:
- Strength critical
- Stiffness critical

ANALYSIS REVEALS ADVANTAGES OF ELLIPTICAL SHAPED BOLTS IN LAMINATED COMPOSITE JOINTS

The objective of this research is to provide rapid design tools. Here, a rapid analysis has been developed for predicting the strength of elliptic bolts and pins in composite joints. (Elliptic bolts may be fastened with an attached circular shaft and nut, and elliptic holes may be placed in composites using water jet technology.) A closed-form solution for bolt-loaded elliptical holes has been formulated based on laminate theory and anisotropic elasticity. The normal load distribution on the edge of the elliptical hole is represented by a cosine series. Unknown coefficients of the cosine series are determined by a boundary collocation procedure in which the bolt is assumed to be rigid. Bearing and hoop stresses along the loaded edge of the hole are obtained from this very efficient analysis procedure and compared with finite element solutions. Finally, a modified Tsai-Wu failure criterion is used to predict failure.

The closed-form solution for the bearing and hoop stresses along the loaded edge of the hole agree very well with converged finite element predictions. As is demonstrated in the figure, both analyses predict that the maximum bearing stress along the hole loaded edge is substantially reduced when an elliptic-shaped bolt is used instead of a circular one. For the case shown, the reduction is 26%. Failure analysis results for two selected joint configurations indicate that a joint designed for a bearing failure critical mode exhibits a 35% strength improvement while a joint designed for a shearing failure critical mode exhibits a 12.9% strength improvement.



ADVANCED PARALLEL-VECTOR ALGORITHMS

Research is being carried out to significantly improve the efficiency of algorithms which form the underpinnings of structural analysis, such as static analysis, eigenvalue analysis, time integration and reduced order modeling, flutter analysis and optimization including sensitivity and design search algorithms. Algorithms in all of these areas require efficient and reliable equation solvers and matrix assembly routines. Advances in these areas is coming from innovative utilization of vector and more recently, parallel computing. The next two charts highlight some of the progress in these developments.

Static	Eigenvalue	Dynamics-Control	Flutter	Optimization
$\mathbf{K}\mathbf{z} = \mathbf{F}$	$\mathbf{K}\varphi = \lambda\mathbf{M}\varphi$	$\mathbf{M}\mathbf{z} + \mathbf{C}\mathbf{z} + \mathbf{K}\mathbf{z} = \mathbf{F}(t)$	$\mathbf{K}\Phi = \lambda\mathbf{M}\Phi$	$b^{i+1} = b^{i} + \alpha_i P_i$
Substructuring NL Algorithms & Strategies	Subspace Lanczos	Time Integration Reduced-Order Simulate Multibody	Unsymmetric Choleski and Lanczos	Search methods Sensitivity

Matrix Assemblers
- Finite Element based
- Degree-of-Freedom based

Equation Solvers
- Direct
- Iterative
- Sparse

Parallel-Vector Structures Algorithms

Static	Eigenvalue	Dynamics-Control	Flutter	Optimization
$Ku = f$	$K\varphi = \lambda M\varphi$	$M\ddot{u} + C\dot{u} + Ku = f(t)$	$K\Phi = \lambda M\Phi$	$b_{k+1} = b_k + s_k d_k$
Substructuring NL Algorithms & Strategies	Subspace Lanczos	Time Integration Reduced-Order Simulate Multibody	Unsymmetric Choleski and Lanczos	Search methods Sensitivity

Matrix Assemblers

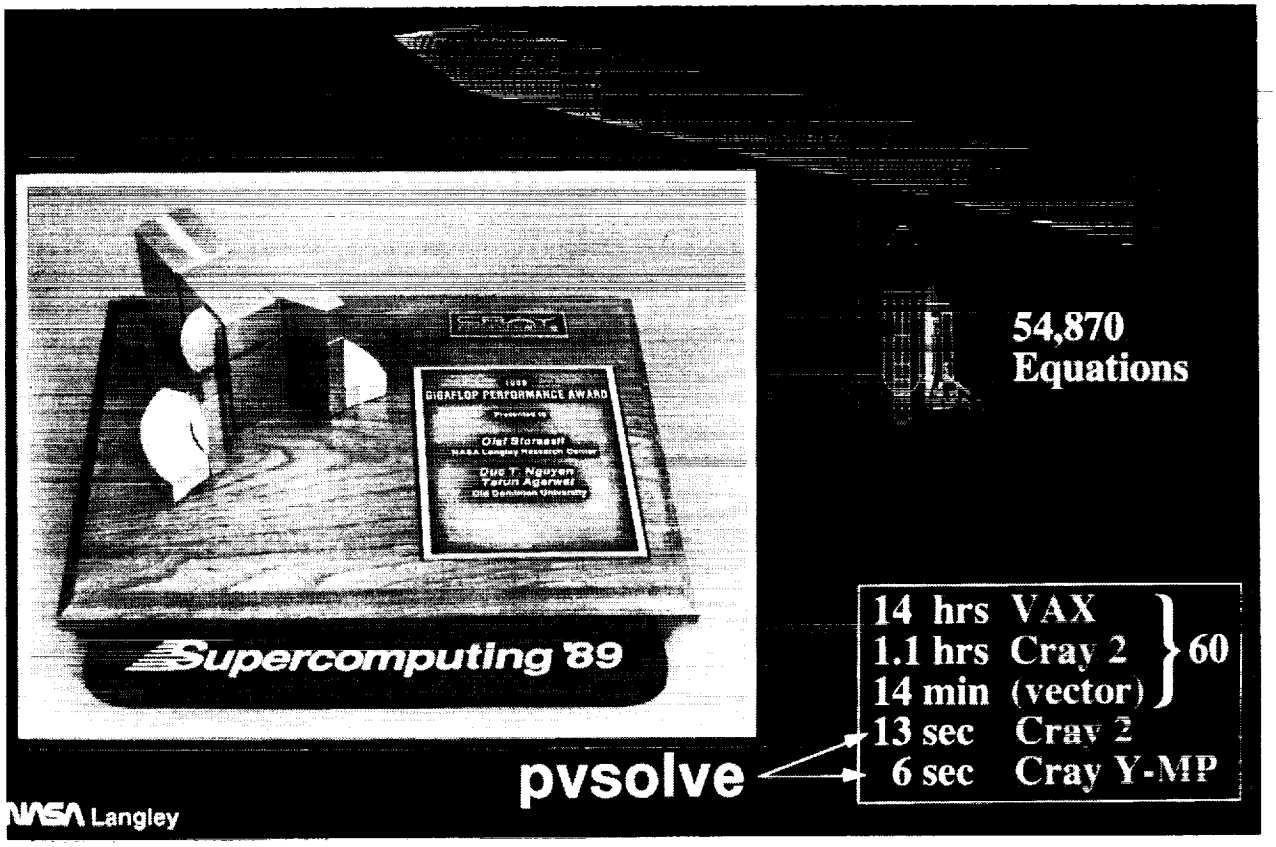
- Finite Element based
- Degree-of-Freedom based

Equation Solvers

- Direct
- Iterative
- Sparse
- SVD

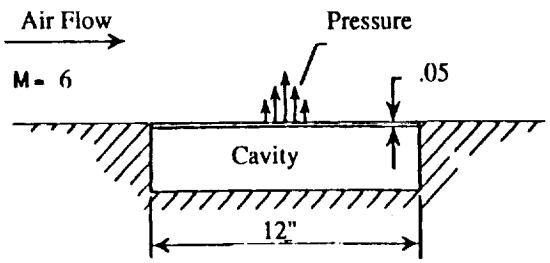
SHUTTLE SRB ANALYSIS WINS CRAY GIGAFLOP AWARD

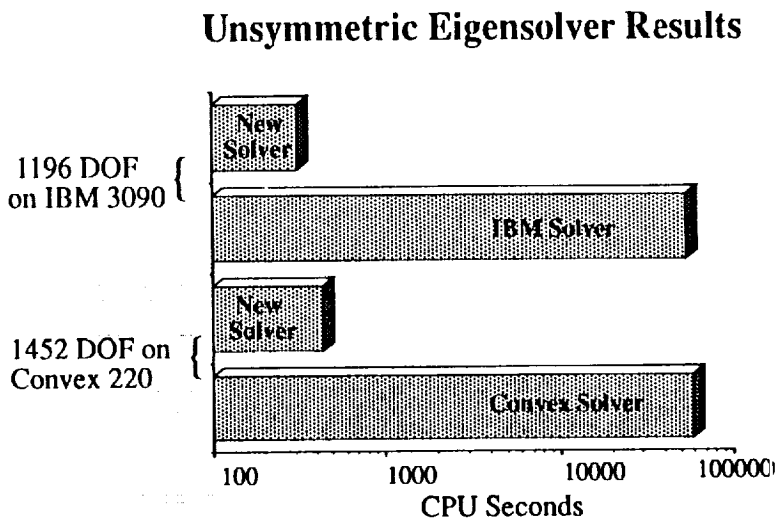
After the Challenger accident, extensive structural analyses were carried out on the Shuttle Rocket Booster (SRB). As part of this analytical investigation, dramatic advances were made in reducing computing time for the large-scale analyses required for accurate stress and deflection predictions. The figure shows the impact of hardware and software in reducing the computing time associated with a nearly 55,000 degree-of-freedom analysis. Moving from the VAX computers to Cray computers resulted in a 60-fold speedup and introducing a newly developed parallel vector equation solver called pvsolve resulted in an additional 140-fold speedup.

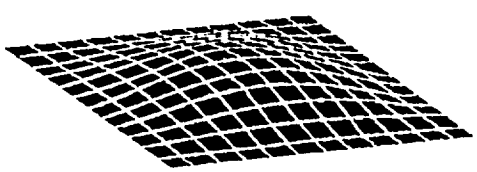


NEW EIGENSOLVER SPEEDS SUPERSONIC-HYPERSONIC FLUTTER ANALYSIS BY TWO ORDERS OF MAGNITUDE

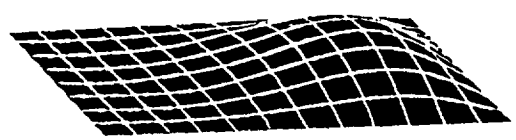
This figure demonstrates the efficiency of a new eigenvalue solver for unsymmetric matrices such as arise in supersonic flutter. The aeroelastic effects of this problem were modeled using unsteady, third-order aerodynamic piston theory. Comparisons are made with existing unsymmetric eigensolvers in both the IBM and CONVEX mathematical solver libraries. The new unsymmetric eigensolver is 134 times faster than existing ones.







1st Mode (no flutter)

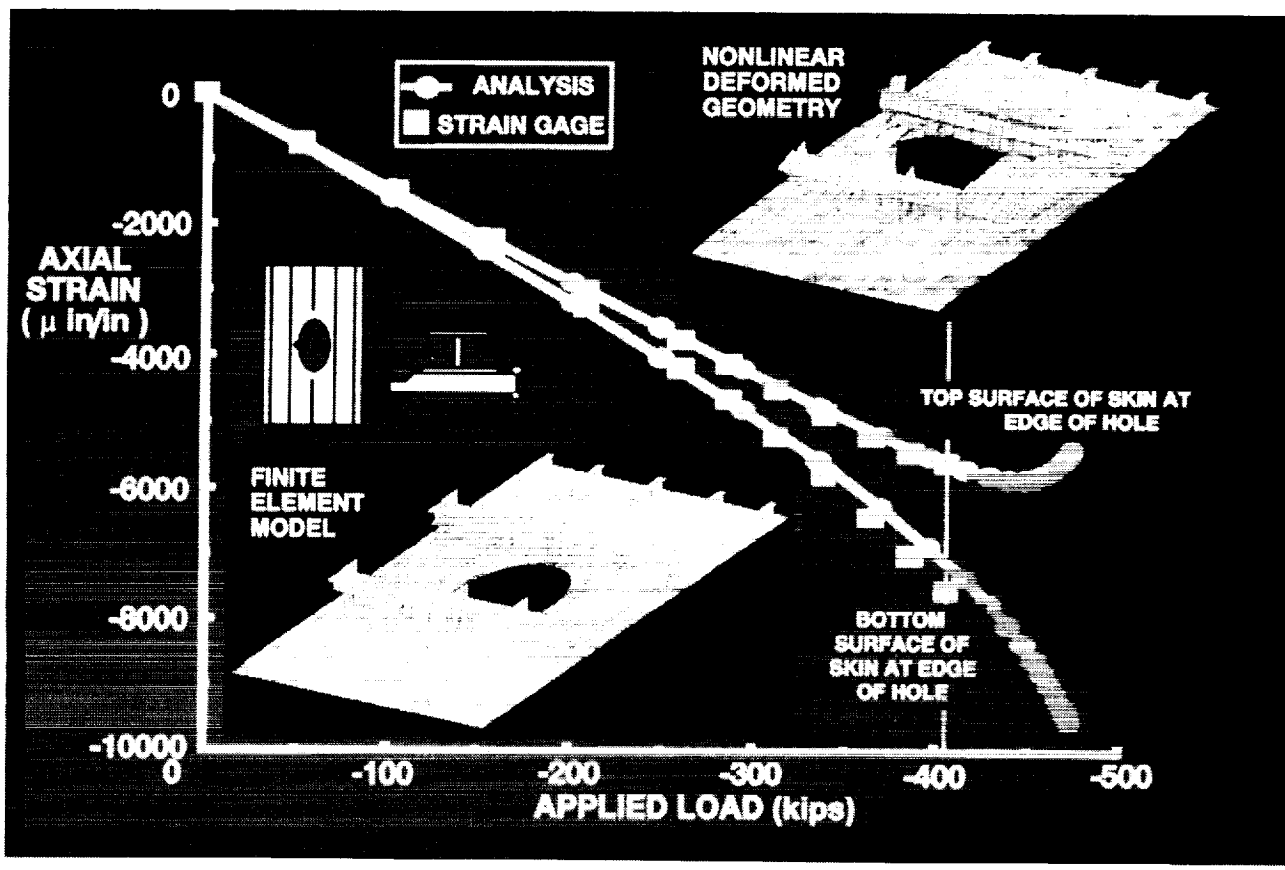


1stMode (flutter)

NONLINEAR ANALYSIS ACCURATELY PREDICTS STRAIN IN STIFFENED COMPOSITE WING PANEL

Nonlinear structural analysis techniques were applied to a panel from the lower skin of the V-22 tiltrotor aircraft. The panel is made totally of graphite-epoxy composite material and contains design features such as ply drops, ply interleaves, axial stiffeners, transverse ribs, clips, brackets, and a large central elliptical access hole, all of which greatly complicates modeling and analysis. Linear and nonlinear analyses were performed using state-of-the-art finite element technology and nonlinear solution strategies. The finite elements include first-order nonlinear effects and higher-order nonlinear effects due to large deformation are handled by a co-rotational finite element approach. First-ply failure techniques were applied to the results of the stress analyses and linear and nonlinear buckling analyses were also performed to gain insight into the failure mechanism of the panel.

As is demonstrated in the figure, excellent agreement with experimental strain gage readings was obtained. The nonlinear analysis accurately predicted the highly nonlinear response of the panel whereas a traditional linear analysis did not. Moreover, the nonlinear analysis predicted considerably more damage at the site of failure initiation, at the access hole, than did the linear analysis.



SUMMARY

Advanced CSM methods are presently being developed to enable more comprehensive analysis of proposed aerospace structure design earlier in the design process. Adaptive refinement methods will shorten modeling time and will identify thermal and stress "hot spots" in global preliminary design models. Through the use of integrated methods and the new interfacial techniques now being developed, local models of identified "hot spots" will readily be incorporated into the global preliminary design model so that such detail can be factored into the design. It is anticipated that this will reduce the amount of redesigning which usually occurs due to overlooked details. Several aircraft companies are presently participating in a methods development partnership with NASA to advance the development and assessment of special purpose methods which can be employed in localized structural regions.

In addition, advanced solution algorithms are being developed for structural analysis and optimization, and comprehensive application studies are being carried out to assess and validate method developments.

- o METHODS BEING DEVELOPED TO ENABLE MORE COMPREHENSIVE AND INTEGRATED ANALYSIS/DESIGN
- o ADAPTIVE REFINEMENT CAPABILITY BEING DEVELOPED FOR PLATE AND SHELL TYPE AEROSPACE COMPONENTS
- o ADVANCED SOLUTION ALGORITHMS BEING DEVELOPED FOR STRUCTURAL ANALYSIS AND OPTIMIZATION
- o COMPREHENSIVE APPLICATION STUDIES BEING CARRIED OUT

REFERENCES

1. Housner, J. M. and Aminpour, M. A.: Multiple Methods Integration for Structural Mechanics Analysis and Design. NASA CP-3104, Nov. 1990.

2. Knight, N. F., Jr.; Gillian, R. E.; McCleary, S. L.; Lotts, C. G.; Poole, E. L.; Overman, A. L. and Macy, S. C.: CSM Testbed Development and Large-Scale Structural Applications. NASA TM-4072, April 1989.

3. Housner, J. M.; Aminpour, M. A. and McCleary, S. L.: Some Recent Developments in Computational Structural Mechanics. Proceedings of the 1991 International Conference on Computational Engineering Science, Melbourne, Australia, August 1991.

4. Aminpour, M. A.: Direct Formulation of a Four-Node Hybrid Shell Element with Rotational Degrees of Freedom. NASA CR-4282, 1990.

N92-25917

Overview of Mechanics of Materials Branch Activities in the Computational Structures Area

C. C. Poe, Jr.
NASA Langley Research Center
Hampton, Virginia

MECHANICS OF MATERIALS BRANCH

Theoretical mechanics are used to develop prediction methodologies, which in turn can be used to improve materials. Experiments are conducted to guide and verify the development. The methodologies involve constitutive relationships, strength models, and fatigue life models. Fracture mechanics and microstructural mechanics are used in developing the strength and fatigue life models. Notches, defects, and interfaces in polymeric and metal matrix composites and advanced metals must be addressed; computational mechanics are used extensively.

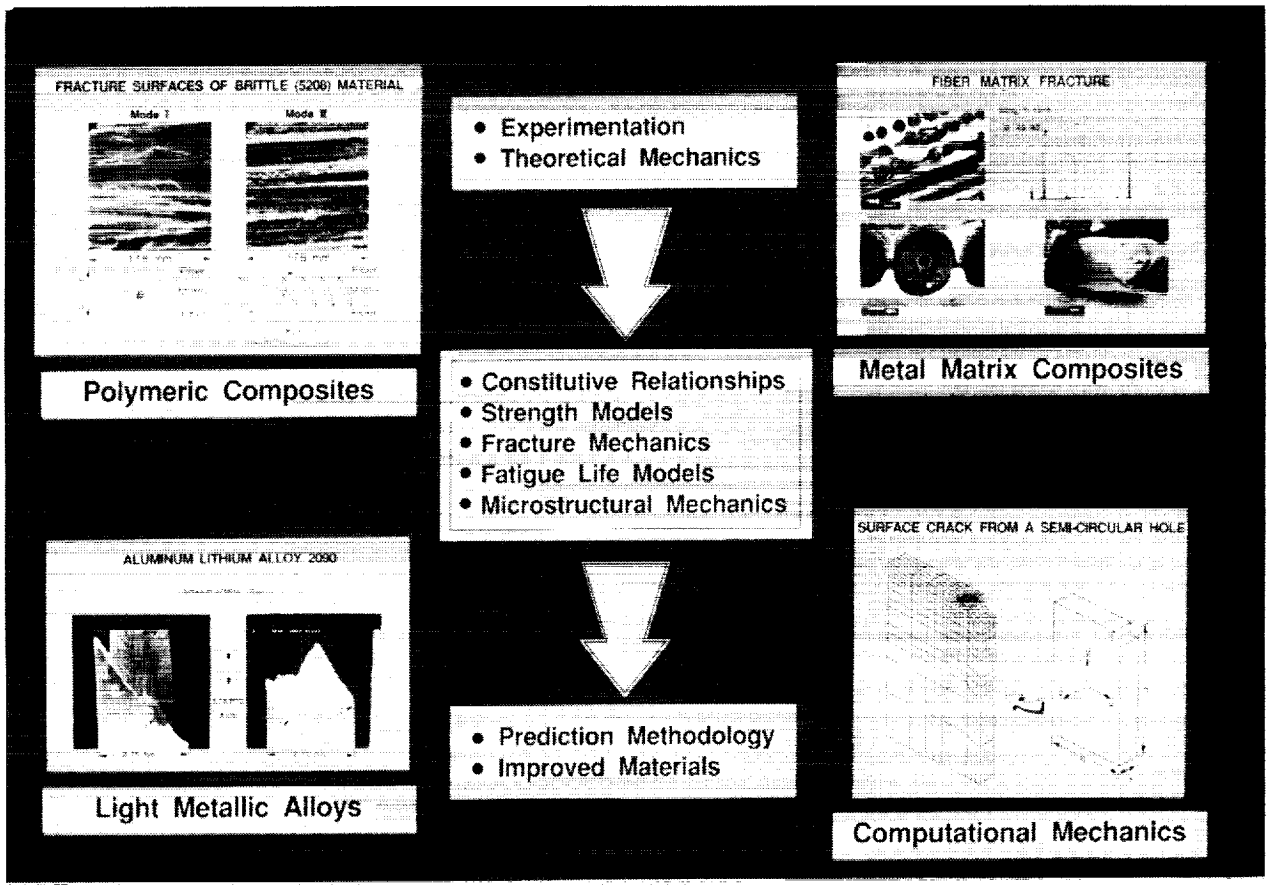


Figure 1

PROGRAMS OF MECHANICS OF MATERIALS BRANCH

Base programs and system programs are supported. The base programs include fundamental research of composites and metals for airframes leading to characterization of advanced materials, models of behavior, and methods for predicting damage tolerance. Results from the base programs support the system programs, which change as NASA's missions change. The National Aerospace Plane (NASP), Advanced Composites Technology (ACT), Airframe Structural Integrity Program (Aging Aircraft), and High Speed Research (HSR) programs are currently being supported. Airframe durability is one of the key issues in each of these system programs. The base program has four major thrusts, which will be reviewed subsequently. Also, several technical highlights will be reviewed for each thrust.

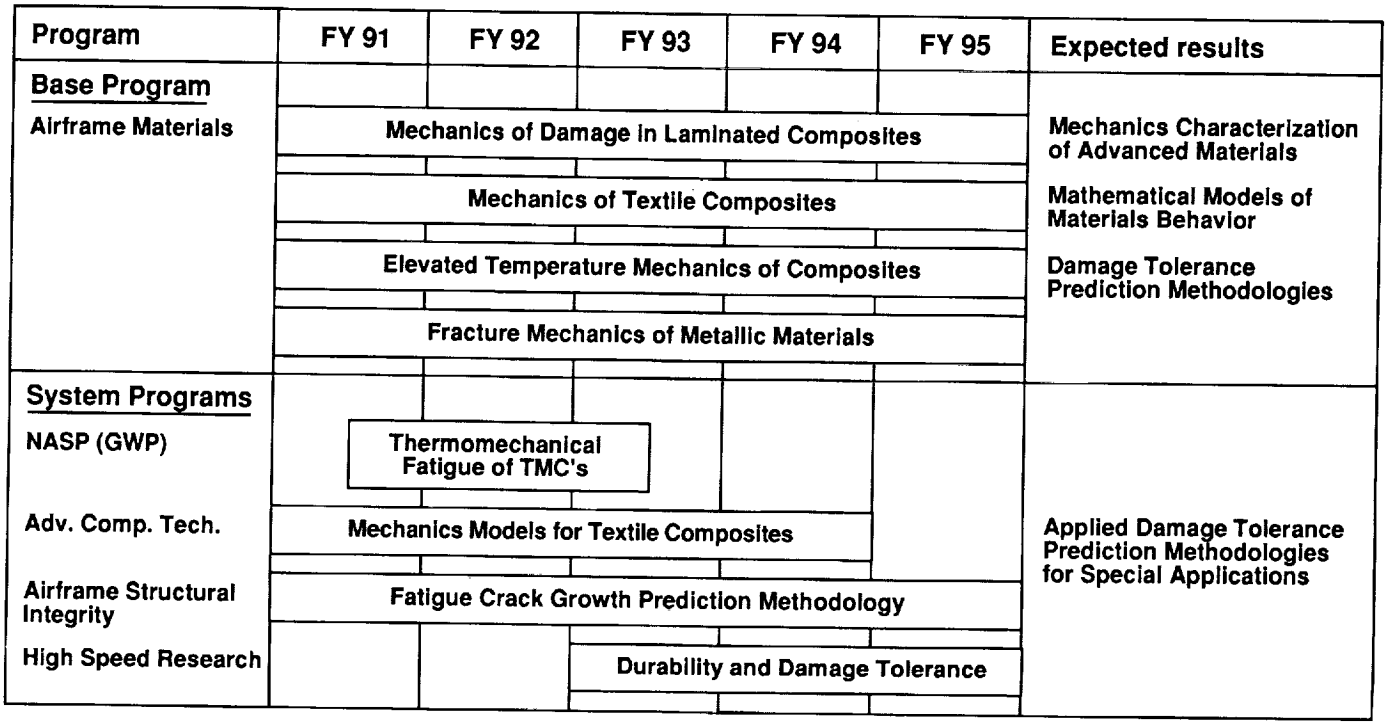


Figure 2

Mechanics of Damage in Laminated Composites is the first thrust shown in the base program in Figure 2. The goal is to develop residual strength and life prediction methodology for laminated composites for rotorcraft and subsonic transport aircraft. The in-house activities and supporting researchers are shown in the center oval. The activities involve delamination mechanics, fracture toughness, progressive failure methodology, and fiber/matrix interface micromechanics. These activities also support the *Advanced Rotorcraft Technology* system program. The researchers include NASA and Army civil servants and nonpersonal services contractors co-located at Langley Research Center (LaRC). The in-house work is supported by grants with the universities and principal investigators shown in the outer ovals.

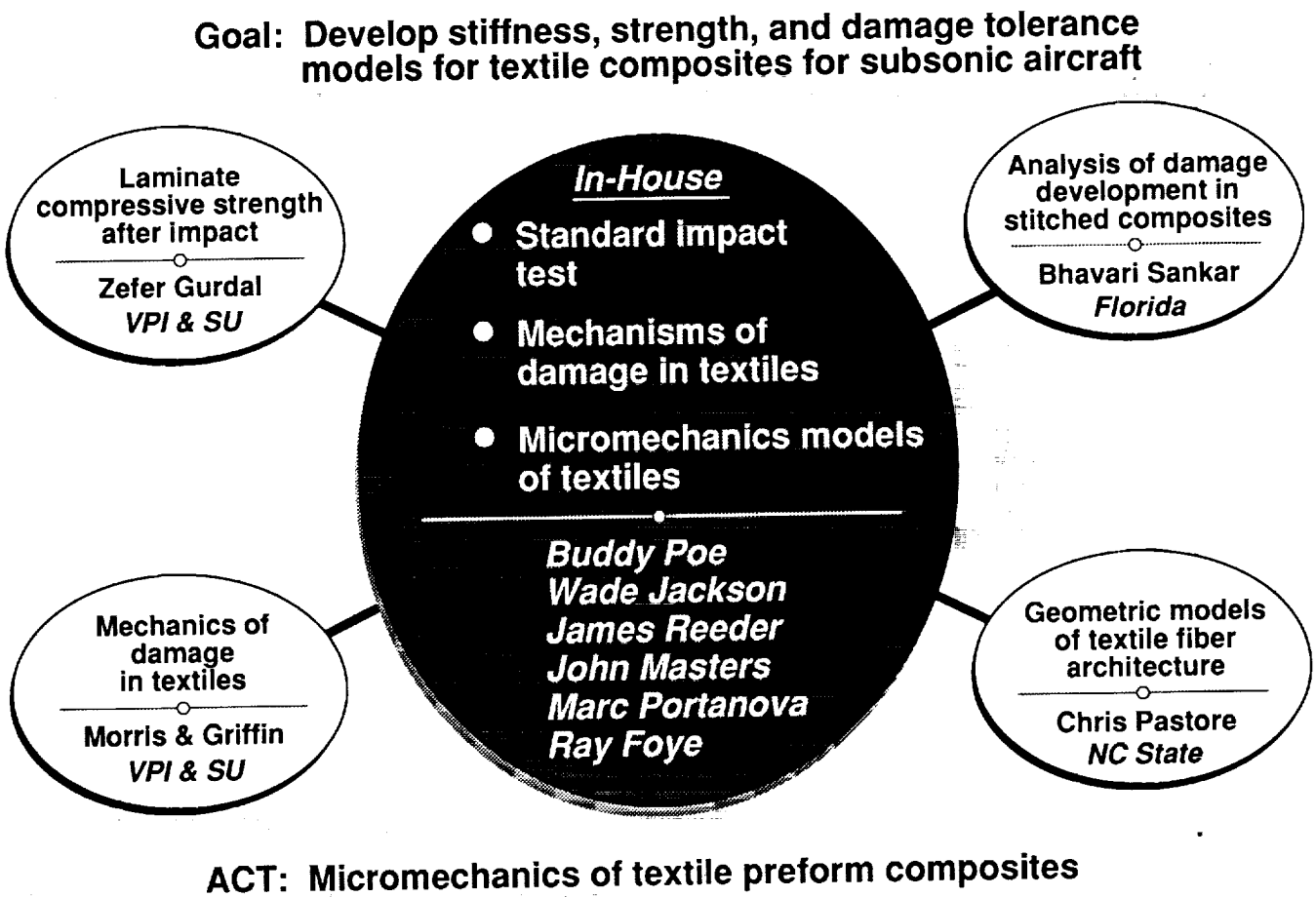


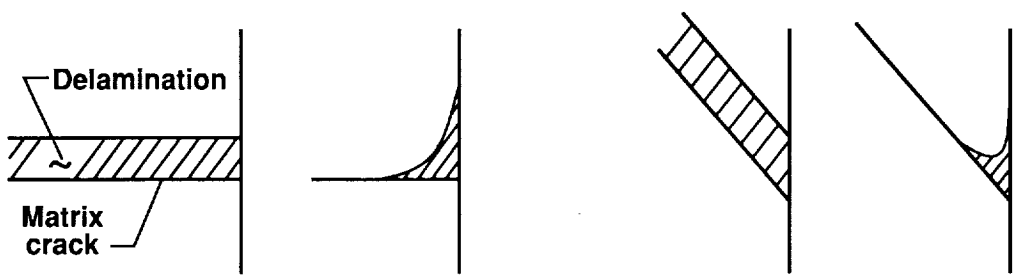
Figure 3

Composite structures often delaminate at structural discontinuities and free edges. It has been observed that, when matrix cracks exits, delaminations initiate where the matrix cracks intersect a free edge. These intraply matrix cracks occur in a plane that is parallel to the fibers and can be caused by thermal or mechanical stresses. Delaminations initiating at a matrix crack in a 90° ply and in a ply with angle θ are illustrated in planar views of laminates in sketches I and II, respectively. The delaminations initiate at free edges as shown in the right parts of sketches I and II and subsequently become approximately uniform in length away from the free edge as shown in the left parts. The propensity for delaminations to propagate is expressed by the magnitude of total strain energy release rate, G_T, which depends upon the configuration, the material properties, and the applied load squared. Simple closed form expressions for G_T have been developed for delaminations of uniform length shown in the left parts of sketches I and II. These solutions have been used to characterize the formation of local delaminations in fatigue.

Finite element analyses of matrix crack-delamination interactions

I. 90° ply matrix crack intersecting straight edge

II. θ° ply matrix crack intersecting straight edge



Results for $\theta = 90°$:

- **Delamination initiates at corner**
- **High localized G_I**
- **Validity of G_T equation**

Results for $15° \leq \theta \leq 45°$:

- **High tensile σ_z near corner**
- **G_I exists near corner**
- **Curved cracks due to principal tension**

Future direction:

- **Matrix cracks intersecting curved boundaries**
- **Delamination growth under combined loads**

Figure 4

A $(\pm 45/90_4)_S$ glass epoxy laminate containing an intraply matrix crack in the central group of 90° plies was analyzed using a 3-D finite element method. The laminate is uniformly strained normal to the matrix crack. The total strain energy release rate G for delaminations initiating from the matrix crack was calculated for various delamination lengths. The delaminations are assumed to occur on the -45/90 interfaces and have a uniform length across the laminate. The graph on the left indicates that G increases with increasing delamination length a/h, where h is ply thickness, and approaches the closed form equation for a delamination of uniform length. The value of G is greater near the free edge ($y = 5h/8$) than in the interior. In this case, G is the algebraic sum of an opening component G_I and a shearing component G_{II}. The G_I component is most often associated with out-of-plane loads. The graph on the right indicates that G_I is also large during the onset of delaminations that initiate from matrix cracks caused by in-plane loads. As the delaminations lengthen, $G_I \to 0$ and $G \to G_{II}$. As in the case of G, G_I and G_{II} are also less in the interior than near the free edge, $y/h = 0.375$. This result is consistent with observations that delaminations initiate at the free edge and propagate across the laminate.

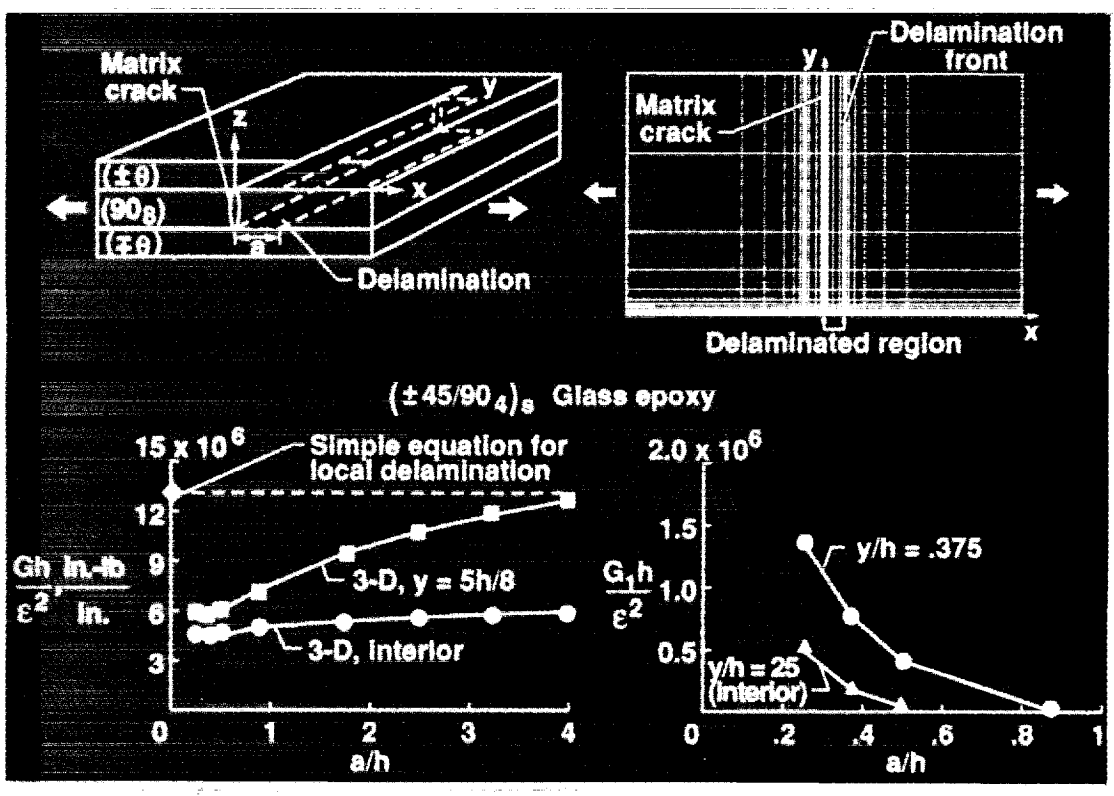
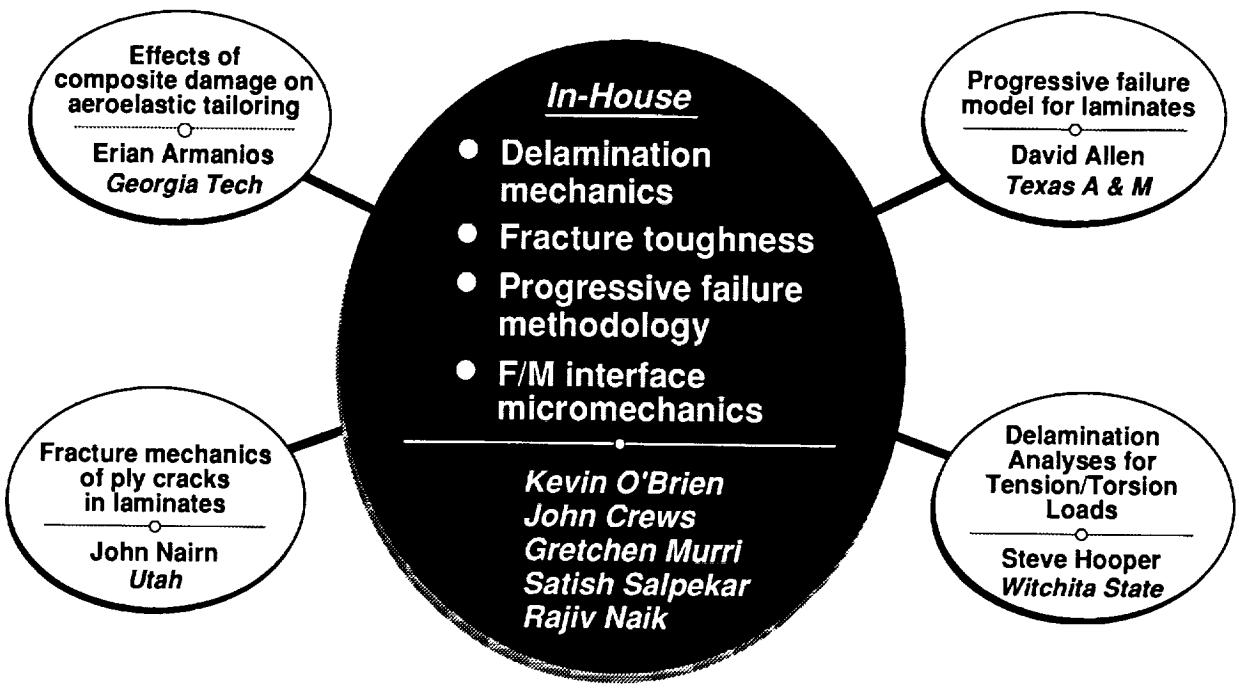


Figure 5

Mechanics of Textile Composites is the second thrust in the base program. The goal is to develop stiffness, strength, and damage tolerance models for textile composites for subsonic transport aircraft. As before, the in-house activities and supporting researchers are shown in the center oval. The activities involve standard impact tests, mechanisms of damage development, and micromechanics models. These activities also support the *Advanced Composites Technology* system program. The co-located researchers include NASA and Army civil servants and nonpersonal services contractors. The in-house work is supported by grants with the universities and principal investigators shown in the outer ovals.

Goal: Develop residual strength and life prediction methodology for laminated composites for rotorcraft and subsonic transport aircraft



Army: Advanced rotorcraft technology

Figure 6

Lamination theory is not generally applicable to textile composites. Thus, unit cell methods will be developed to calculate global compliance. A unit cell is the smallest repeating element of the textile composite; it is analogous to a ply in a laminated composite. A single subcell can be translated and rotated to generate a unit cell. Fiber architecture will be predicted in terms of textile processing parameters by computer codes. Representing the yarn and matrix discretely, stress-strain relationships will be calculated for subcells, and then the compliance of the unit cell will be calculated from that of the subcells using finite element theory. Continuity conditions are imposed on the cell boundaries. Continuum level strength models analogous to those based on lamination theory will be developed for the unit cells. Also, strength models that account explicitly for matrix cracking and yarn debonding will be developed.

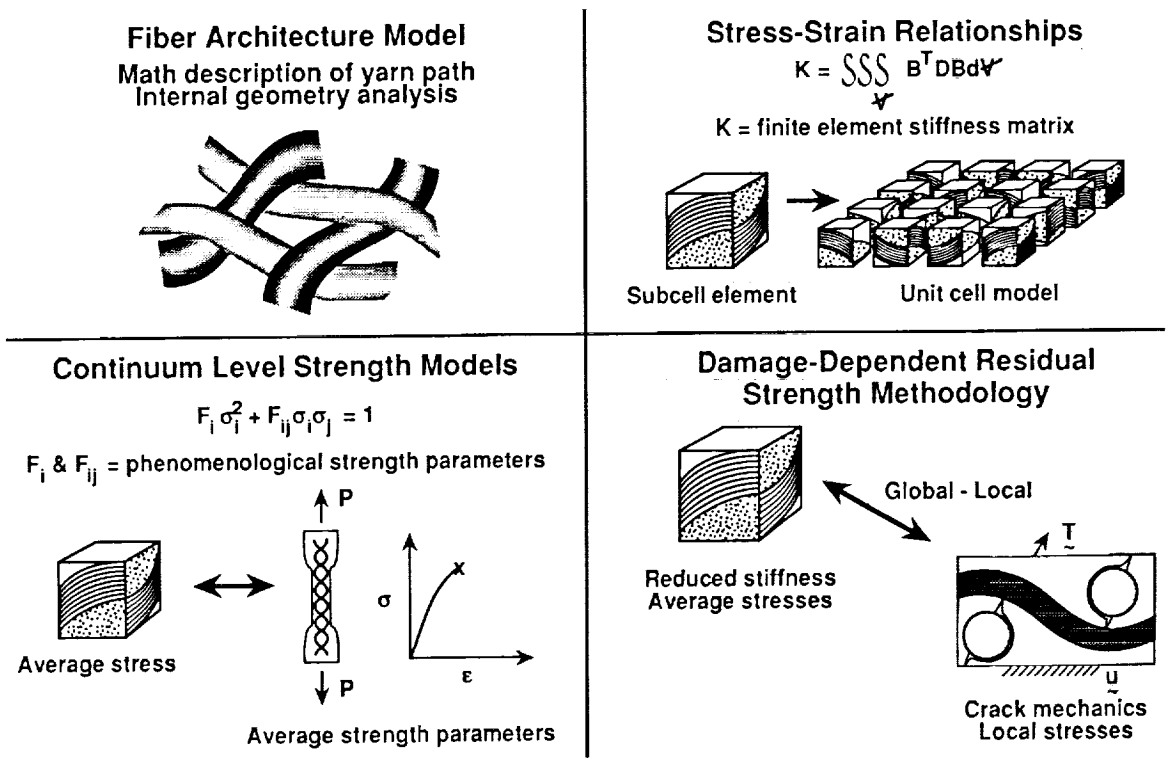
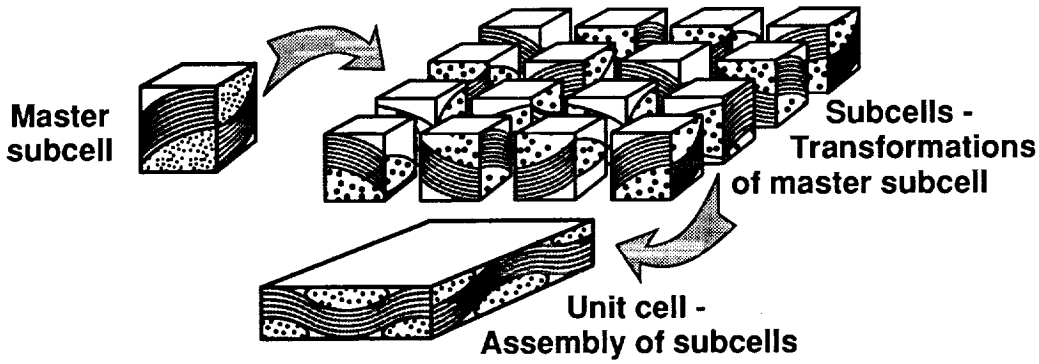


Figure 7

Elastic constants measured in experiments (Exp) and predicted with the unit cell model in Figure 7 (Pred) are shown for textile composites made from two different woven fabrics. One fabric is a plain weave (one over and one under), and the other is a 5-harness weave (one over and four under). The elastic constants, which are in the plane of the fabric, include Young's modulus, shear modulus, Poisson's ratio, and thermal coefficient of expansion. The measured and predicted values are in reasonably good agreement.



Weave		E warp (MSI)	G wf (MSI)	ν wf	α w (in./in./°F)
Plain	(Pred)	9.29	0.82	0.05	1.11
	(Exp)	9.13	--	0.10	--
5 harness	(Pred)	9.62	0.84	0.05	1.11
	(Exp)	10.59	0.76	0.05	1.12

Figure 8

Often, experimental results from coupon tests are used as allowables for structures on the basis of kinetic energy alone. The impact force calculated using plate theory with inertia terms is plotted against the reciprocal of impacter mass for a given value of kinetic energy. The horizontal dashed lines represent calculations with simple energy balance equations assuming a quasi-static response. For large masses, the plate response is quasi-static, and impact force depends on plate configuration but not impacter mass. However, for small masses, impact force depends on impacter mass but not on plate configuration. For intermediate masses, impact force depends on impacter mass and plate configuration. Thus, kinetic energy is not sufficient to define impact response.

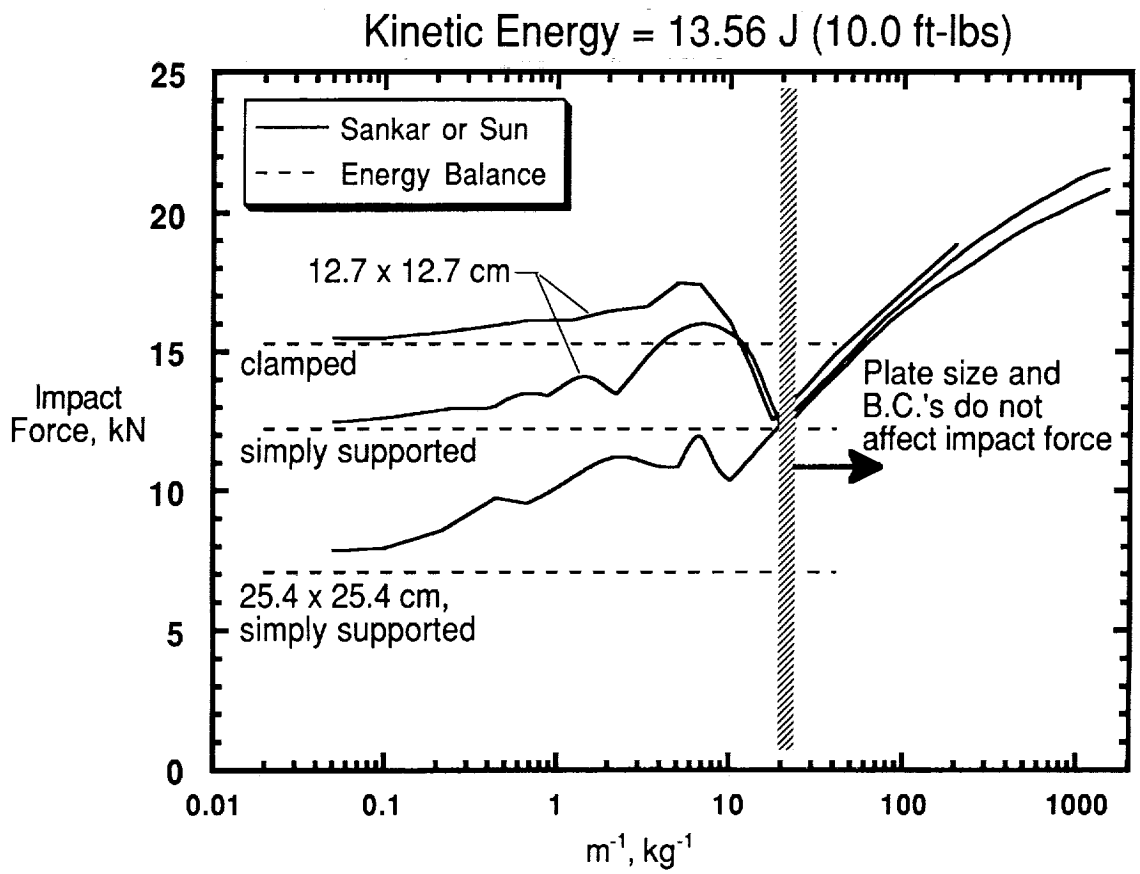
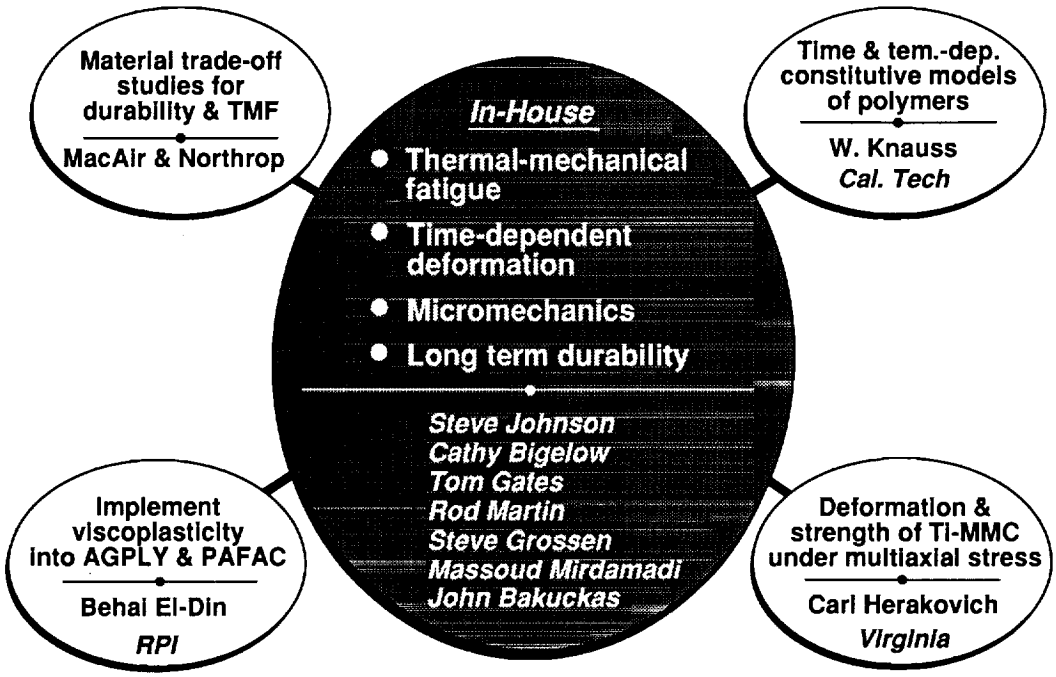


Figure 9

Elevated Temperature Mechanics of Composites is the third thrust in the base program. The goal is to develop thermomechanical fatigue and long-term durability characterization of metal and organic matrix composites for supersonic and hypersonic aircraft. In-house activities and supporting researchers are shown in the center oval. The activities involve thermal-mechanical fatigue, time-dependent deformation, micromechanics, and long term durability. These activities also support the *National Aerospace Plane* system program. The co-located researchers include NASA civil servants and nonpersonal services contractors. The in-house work is supported by contracts and grants with the companies or universities and principal investigators shown in the outer ovals.

Goal: Develop the thermomechanical fatigue and long-term durability characterization of metal and organic matrix composites for supersonic and hypersonic aircraft



NASP: Thermomechanical fatigue of TMC's

Figure 10

ANALYSIS OF UNIDIRECTIONAL TITANIUM METAL MATRIX COMPOSITES FOR THERMOMECHANICAL FATIGUE

Thermo-mechanical fatigue tests (simultaneous cycling of temperature and loads) were conducted on unidirectional silicon-carbide/titanium (SCS/6Ti-15-3) composites to determine their durability in a NASP type vehicle. The tests were conducted with temperature and loads increased in-phase (IP) and out-of-phase (OOP) at a cyclic period of 180 s. (See the graph in the upper-left corner.) The fatigue lives are plotted in the graph in the upper right for the various nominal (average) stresses. The fatigue lives were much shorter for the IP tests than the OOP tests. Fiber and matrix stresses were calculated using a thermo-viscoplastic analysis code for laminated plates called VISCOPLY. The stresses in the individual fibers and matrix are plotted in the graph in the lower left. The fiber stress is larger for the IP tests than the OOP tests, and the matrix stress is the opposite. Thus, in a relative sense, the matrix is more highly stressed for the OOP tests than the IP tests. The fatigue lives are plotted in the lower right graph for the various values of calculated fiber stress. The fatigue lives for the IP and OOP tests are essentially equal for a given fiber stress, indicating that fiber failure controls composite failure. Also, extensive cracking of the titanium was observed for the OOP tests but not for the IP tests. This observation is consistent with the relatively larger matrix stress that was calculated with VISCOPLY.

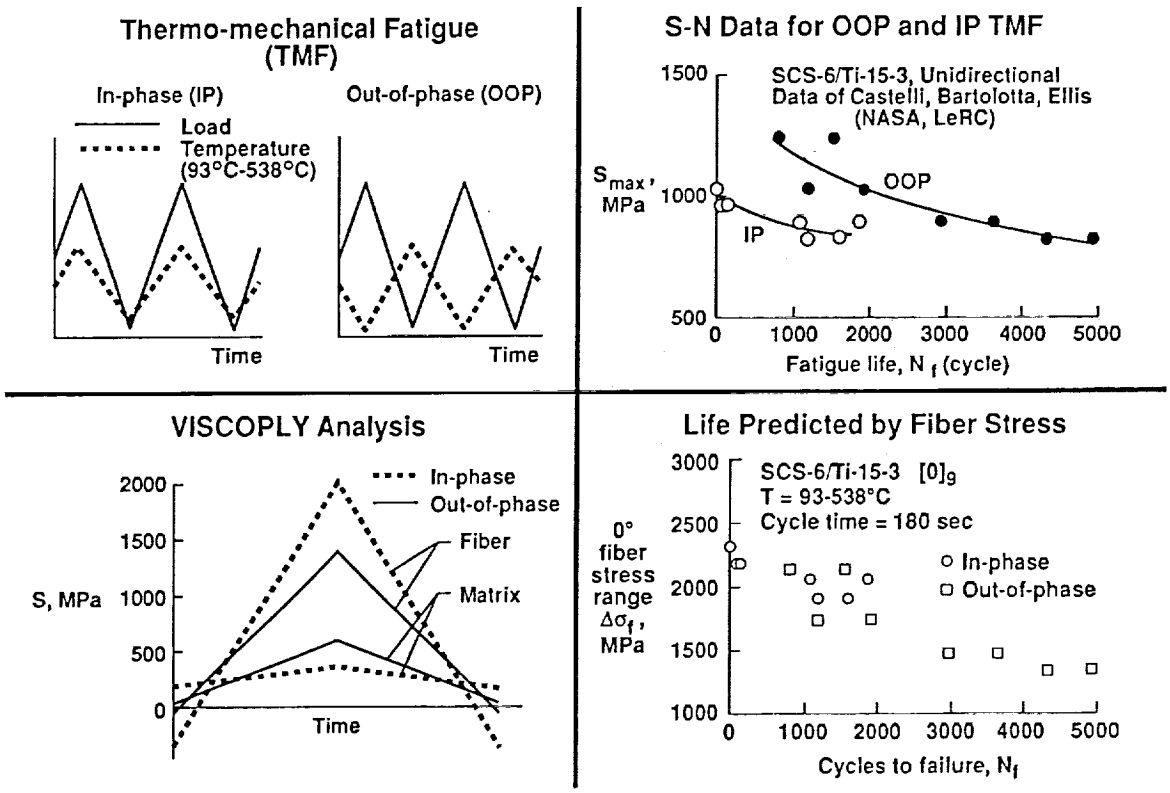


Figure 11

Advanced polymer matrix composites are being considered for the next generation commercial supersonic transports. Elastic/viscoplastic constitutive models are useful in evaluating these materials. The results from such a model are shown in the two bottom graphs for IM7/RadelX (graphite/thermoplastic). Predicted and measured room temperature results are plotted in the bottom-left graph for uniaxial specimens loaded 15° to the fiber direction and in the bottom-right graph for uniaxial specimens loaded 30° to the fiber direction. The effect of creep is indicated in the lower-left graph and the effect of stress relaxation in the lower-right graph. Results for off-axis unidirectional specimens offer the best test for an elastic/viscoplastic constitutive model. Agreement between the model and test results are very good. The predictions were made using the unidirectional results in the upper graphs and lamination theory. The upper-left graph reflects quasi-static response and the upper-right graph reflects strain-rate response to an overstress. Results for a graphite/bismaleimide (IM7/5260) are also plotted in the upper graphs. The time dependent behavior for the two materials is quite different.

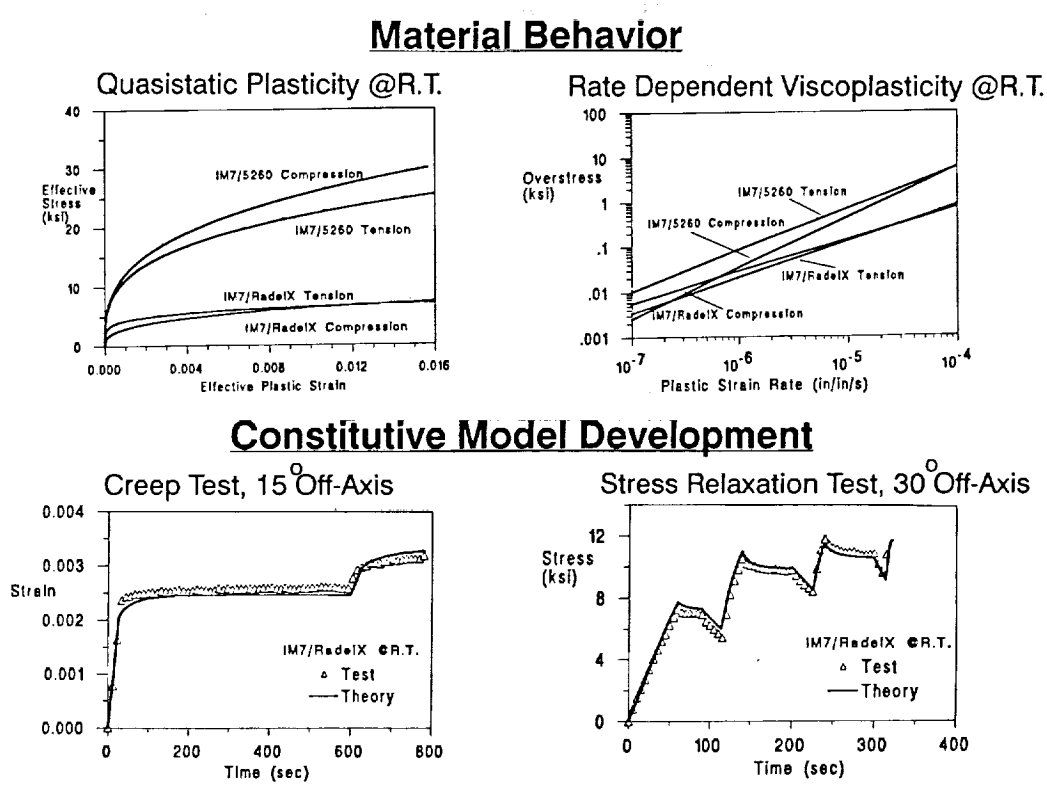
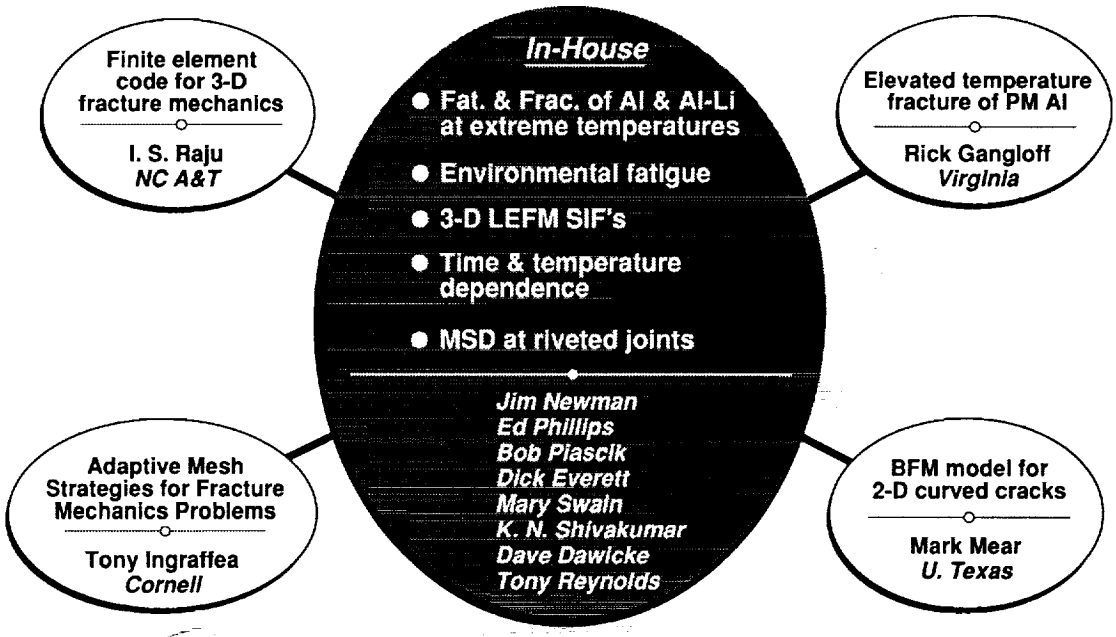


Figure 12

Fracture Mechanics of Metallic Materials is the fourth thrust in the base program. The goal is to develop fracture mechanics and fatigue crack growth models for metals for subsonic and supersonic aircraft. In-house activities and supporting researchers are shown in the center oval. The activities involve fatigue and fracture of aluminum and aluminum-lithium alloys at extreme temperatures, environmental fatigue, 3-D linear elastic fracture mechanics stress intensity factors, time and temperature dependence, and multi-site damage at riveted joints. These activities also support the *Airframe Structural Integrity* system program. The co-located researchers include NASA and Army civil servants and nonpersonal services contractors. The in-house work is supported by grants with universities and principal investigators shown in the outer ovals.

Goal: Develop fracture mechanics and fatigue crack growth models for metals for subsonic and supersonic aircraft



NASIP: Fatigue crack growth prediction methodology

Figure 13

PREDICTION METHODOLOGY FOR TOTAL FATIGUE LIFE

Recent research on aluminum alloys indicates that micro-cracks initiate very early from inclusion particles or voids 3 to 6 μm in size as illustrated in the upper-left photomicrograph. Crack-tip stresses are proportional to the stress intensity factor K. Traditionally, fatigue crack growth rates are correlated with the stress intensity factor range, ΔK, which is the alternating component of K. Fatigue crack growth rates are plotted against ΔK in the upper-right graph. The symbols represent test data for small cracks like those in the upper-left graph, and the pair of lines represent the upper and lower range of test data for large (long) cracks. The value of ΔK is proportional to load range and, in this case, increases with increasing crack length. Thus, for a given rate, a higher load is required for a smaller crack. The vertical lines indicate a threshold below which large cracks do not grow. However, small cracks do grow below the large crack threshold and even grow faster just above the threshold. The difference between the small and large crack data is primarily due to "crack closure" where plastic behavior at the crack tip causes large cracks to close prematurely and effectively reduces the stress intensity range. By accounting for this closure, the fatigue lives in the lower-left graph for spectrum loading were predicted assuming a small crack grows from an initial size equal to that of an inclusion particle to a size sufficient to reduce the strength below that of the subsequent load. The agreement with test data is good. In the lower-right graph, initial crack size is plotted against fatigue life using small-crack and large-crack data. The use of large-crack data is unconservative for very small initial cracks. An extrapolation of the large crack data as though there were no threshold is also unconservative.

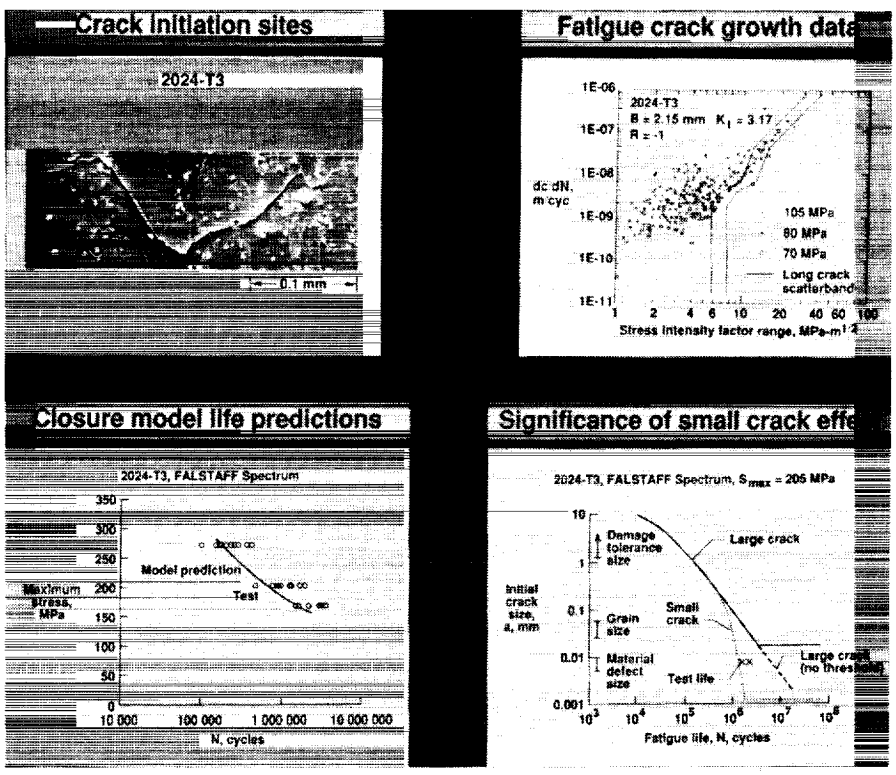


Figure 14

An alternating indirect boundary element (AIBE) method for evaluating the stress-intensity factors for multiple interacting cracks was developed and incorporated into a computer code for predicting fatigue crack growth lives. Fatigue tests were conducted on 300-mm wide, 2.3-mm thick sheets of aluminum alloy 2024-T3. The sheets contained 10 co-linear open holes with cracks propagating out of both sides of each hole. Fatigue crack growth lives were predicted on the basis of equal crack lengths and actual or unequal crack lengths. The AIBE method was used to account for crack interaction effects resulting from the unequal crack lengths. Results are plotted as average crack length versus load cycles. The AIBE predictions of cycles for unequal crack lengths were within 20% of the measured cycles; whereas, the predictions of cycles based on equal crack lengths exceeded the measured cycles as much as 100%. The closer than average proximity of the actual crack tips caused higher fatigue crack growth rates and shorter lives.

PREDICTION OF MSD CRACK GROWTH BEHAVIOR

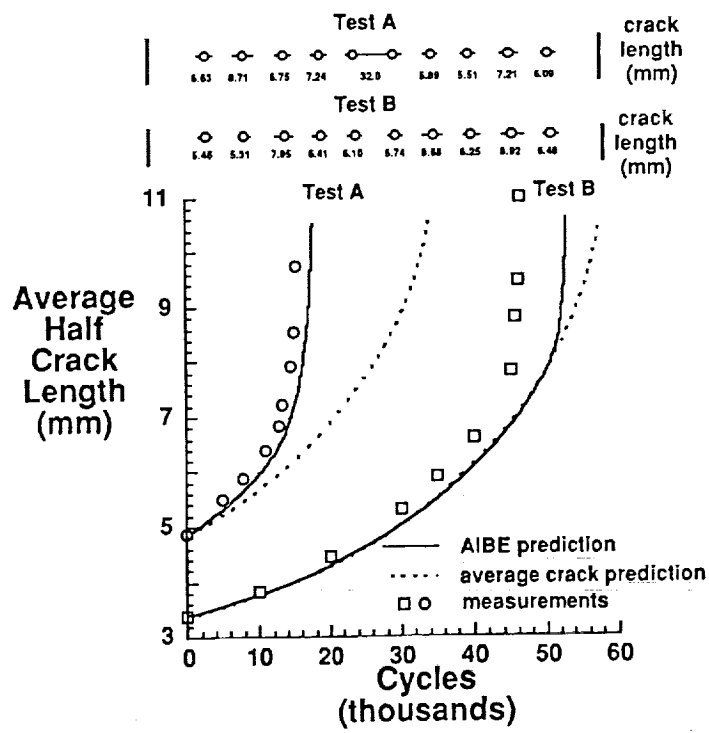


Figure 15

N92-25918

Analysis and Design Technology for High-Speed Aircraft Structures

James H. Starnes, Jr. and Charles J. Camarda
NASA Langley Research Center
Hampton, Virginia

INTRODUCTION

Viable supersonic and hypersonic aircraft structures must be structurally efficient and designed to operate reliably with combined mechanical and thermal loads. To provide such structures requires the development of verified structural analysis and design technology that is necessary to predict the response and failure characteristics of wing and fuselage structures made from advanced metallic and composite materials. Research is being conducted at NASA Langley Research Center to understand the response and failure characteristics of high-speed aircraft structures and to develop the necessary structural analysis and design technology for future high-speed aircraft.

The present paper describes selected recent high-speed aircraft structures research activities at NASA Langley Research Center. Selected topics include: the development of analytical and numerical solutions to global and local thermal and structural problems, experimental verification of analysis methods and identification of failure mechanisms, and the incorporation of analysis methods into design and optimization strategies. The paper describes recent NASA Langley advances in analysis and design methods, structural and thermal concepts, and test methods.

AIRCRAFT STRUCTURES RESEARCH

High-speed aircraft structures research at NASA Langley Research Center is focused on the development of structural mechanics technology for supersonic and hypersonic aircraft primary structures. A goal of the research is to understand the thermal and structural behavior of complex structures made from advanced metallic and non-metallic materials using advanced fabrication techniques. Another goal is to develop structurally efficient and cost-effective structural concepts which exploit the beneficial characteristics of advanced metallic and non-metallic composite materials.

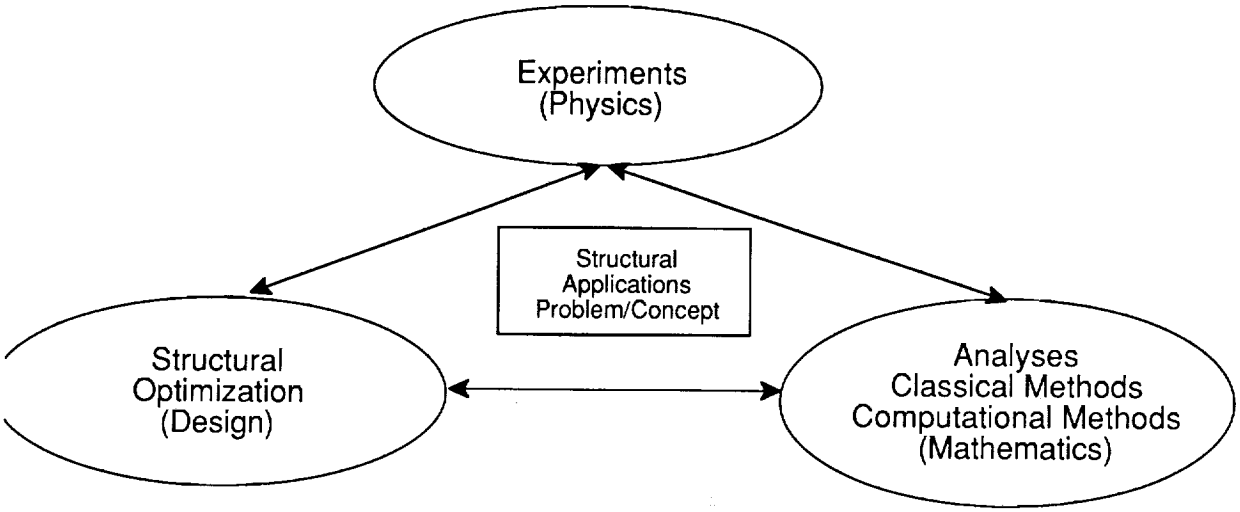
- Provide the scientific basis and structural mechanics technology for aircraft primary structures

- Develop structurally efficient, cost-effective structural concepts that exploit the benefits of advanced composite and advanced metallic materials

STRUCTURAL MECHANICS RESEARCH APPROACH

The key to understanding the physics of a structural mechanics problem is the ability to conduct precise structural and thermal experiments. Optimally designed experiments, which insure satisfaction of prescribed boundary and loading conditions and the accurate measurement of response quantities, are essential. Experiments are designed so parameters can be varied systematically and representative failure mechanisms can be identified and understood. The development of verified analytical methods, whether classical or numerical or combinations of both, is closely coupled to the experiments. The ability of the analysis methods to predict accurately the actual response and failure mechanisms verifies the methods. Anomalies in the correlation between analysis and experiment are resolved by careful studies of the observed behavior and, if necessary, additional experiments are conducted and improved analytical methods and models are developed. Once the analytical methods are verified and an understanding of a given problem is assured, the next logical step is the appropriate simplification of the analysis to increase computational efficiency and enable its incorporation into a formal optimization or structural sizing procedure. During the optimization process, many analysis and response sensitivity calculations must be performed and, hence, it is often necessary to simplify the analyses as much as possible to make the optimization process tractable. Iteration between the development of analysis and optimization procedures is necessary to insure accuracy of the analysis and proper representation of constraint boundaries. Iteration between experiments and optimization methods development is needed to assure that optimally designed structures represent actual physical behavior and to insure an accurate physical description of the response.

The overall goal of structural mechanics research is to provide a better understanding of the physics of the problems of interest, including the true limits of performance, which leads to less conservatism in the design and, hence, a more structurally efficient design for wing and fuselage primary structures.



- Systematically vary parameters
- Understand true limits of performance
- Identify failure mechanisms

STRUCTURAL MECHANICS RESEARCH APPROACH

STRUCTURAL ANALYSIS AND DESIGN TECHNOLOGY

One contribution that will decrease the time needed to design a new aircraft structure is a structural modeling tool that will decrease the time needed to develop a discrete structural model of a complex aerospace vehicle. Work is underway at NASA Langley to enhance the Solid Modeling Aerospace Research Tool (SMART) to enable rapid structural modeling of external and internal structure from a given aerodynamic shape. In addition, it is intended that the enhancements to SMART will allow the rearrangement and resizing of internal and external structural elements in a rapid manner.

Both global and local analysis methods are being developed to predict the structural and thermal response for static and transient, linear and nonlinear problems. Some of the analytical methods currently being investigated are: advanced reduced basis methods, operator splitting techniques, flux- or stress-based finite element algorithms, Ritz-based methods, and classical solution methods.

Several methods for optimizing large structural systems, such as a future supersonic transport wing which has thousands of degrees of freedom and is subjected to hundreds of mechanical and thermal constraints, are currently being explored. Equilibrium programming and other structural sizing methods are being developed to address the structural optimization of large nonlinear systems. Efficient methods to calculate structural and thermal sensitivity derivatives are being implemented into existing general-purpose finite element codes to facilitate formal optimization and to explore other uses for sensitivity derivative information such as parameter estimation. These structural sizing tools are being used to tailor structural designs to exploit the beneficial properties of advanced materials.

- Structural modeling tools for internal structural configurations from aerodynamic external shape
- Global analysis methods for combined mechanical, internal pressure and thermal loads
- Detailed and local analysis methods for combined mechanical and thermal loads
- Design, optimization and tailoring methods for mechanical and thermal constraints
- Sensitivity analyses to identify important geometric and material parameters that affect structural performance

THERMAL-STRUCTURAL CONCEPTS

The degree of coupling between the structural, thermal, and fluids disciplines increases proportionally as the speed of the vehicle increases. For a hypersonic vehicle such as the National Aero-Space Plane, which is designed to cruise at Mach numbers exceeding 16 and to fly single stage to orbit, the selection of a hot structures concept, an insulated structural concept, and a cooled structural concept is not straight forward and often requires re-evaluation of the concepts at the vehicle design level. A good understanding of fluid flow, heat transfer, and structural mechanics is often necessary early in the design cycle to insure proper synergism in the design. Examples of solutions to coupled problems include the design of a refractory-metal/refractory-composite heat-pipe-cooled wing leading edge and a liquid-metal-cooled engine cowl leading edge. Both of these concepts will be discussed in detail later in the paper.

Supersonic vehicles such as a future supersonic transport may not experience the same degree of coupling between the thermal and structural disciplines as a hypersonic vehicle; however, important structural design options will be governed by heat transfer. For example, the selection of wing structural concepts which contain fuel may be governed by the inherent insulative properties of a sandwich structure that may be lighter in weight than a stiffened structure with insulation. Organic composite and metallic structures will be compared in the preliminary design of a supersonic transport. Cost is an important consideration in the competitiveness of a commercial supersonic transport in addition to weight. Also, damage tolerance, durability and thermal stability may become critical design constraints in addition to traditional strength and buckling constraints. Details of preliminary studies for a wing structure will be presented later in the paper.

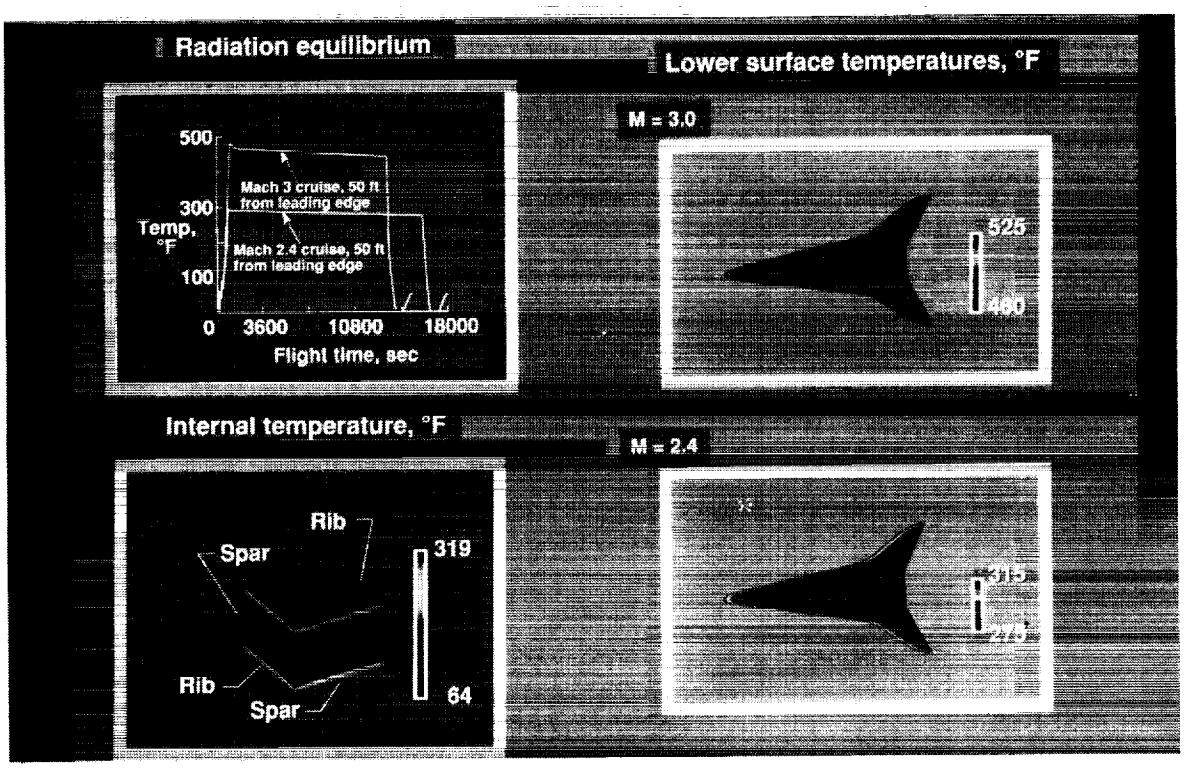
Some hypersonic structural concepts currently being investigated include: a carbon-carbon elevon (hot structure), a refractory-composite/heat-pipe-cooled wing leading edge (passively cooled structure), and a liquid-metal-cooled engine cowl leading edge (actively cooled structure). Structural materials for these concepts include advanced refractory-composite materials and advanced metal-matrix composites.

- Speed, weight, cost, and supportability influence concept selection
  - Uncooled or hot structures
  - Cooled structures
  - Thermal protection systems and insulated structures
- Current HSCT wing and fuselage concept candidates
  - Sandwich structure
  - Stiffened-skin structure with or without fuel tanks and fuselage insulation
  - Organic composites and metals
- Some current HSCT structural issues
  - Weight and cost
  - Damage tolerance and damage containment
  - Durability for 60,000 flight hours at Mach 2.4 cruise
  - Thermal stresses and thermal stability at global and local levels
- Some current hypersonic structural concepts
  - Actively cooled structure
  - C/C, C/SiC, advanced metal matrix
  - Heat pipes

THERMAL-STRUCTURAL CONCEPTS

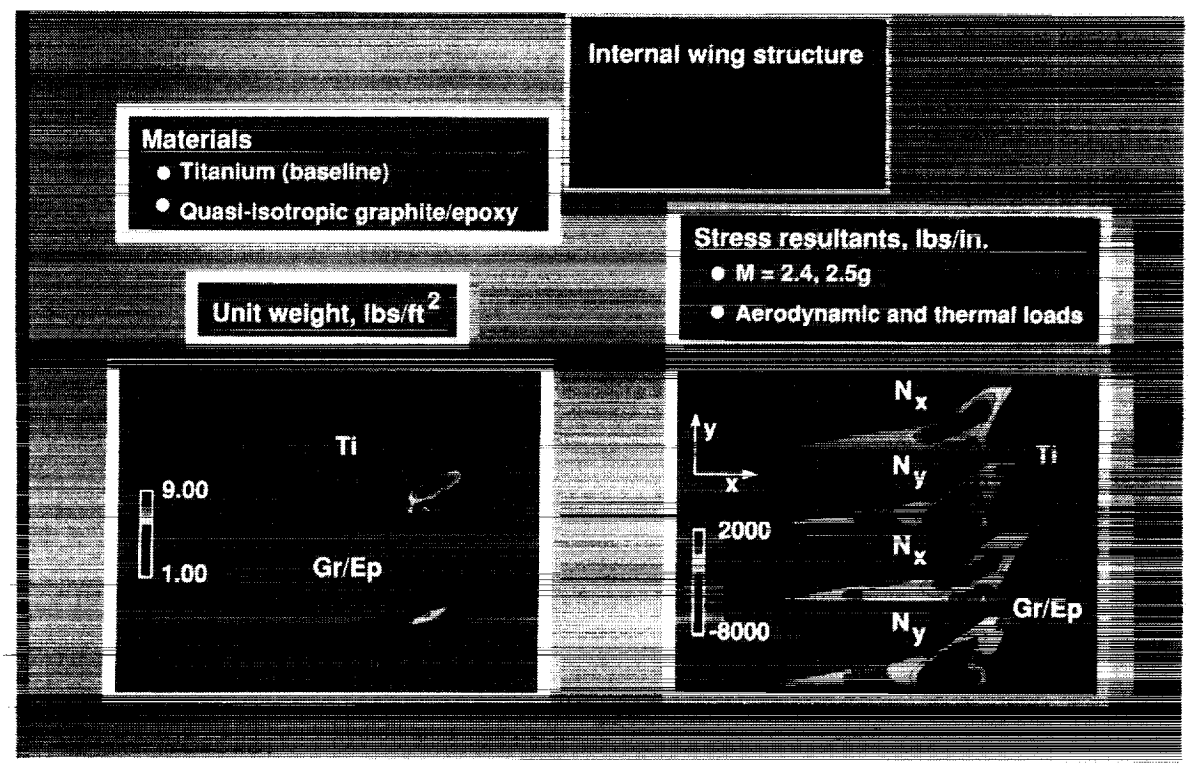
TEMPERATURE DISTRIBUTIONS FOR HIGH-SPEED SUPERSONIC CIVIL TRANSPORT WINGS

Temperatures of a future high-speed supersonic civil transport (HSCT) wing were calculated for two different flight cruise Mach numbers. Temperatures were calculated assuming radiation equilibrium on the heated surfaces and neglecting conduction. As shown in the upper left figure, a Mach 3 cruise flight condition results in temperatures close to 500 °F, 50 ft. aft of the leading edge. Corresponding maximum temperatures for a Mach 2.4 cruise flight condition are approximately 300 °F. The large difference in wing temperatures between a Mach 2.4 and a Mach 3 cruise condition has a significant impact on the selection of materials to satisfy the life and durability requirements of the vehicle. Lower-surface skin temperature distributions for both the Mach 3 and Mach 2.4 cruise flight conditions are shown in the figures at the right. For a Mach 3 flight trajectory, temperatures over the acreage areas on the lower surface of the vehicle are about 460 °F and temperatures along the stagnation lines of the wing leading edges are approximately 525 °F. Temperatures along lower surface acreage areas for the Mach 2.4 flight case are 275 °F and are 315 °F near the leading edges. Temperature gradients through the depth of a three-dimensional model of a section of the wing bounded by ribs and spars is shown in the figure at the lower right for a Mach 2.4 flight condition. The model accounts for flow of heat by conduction near massive sections of the wing such as ribs and spars. The model accounts for unsteady flow of heat through the honeycomb wing skins to the fuel contained within the wing structure. Detailed three-dimensional thermal models are necessary to predict accurately the temperature gradients which are necessary for accurate determination of thermal stresses.



WEIGHTS AND STRESS RESULTANTS FOR HIGH-SPEED SUPERSONIC CIVIL TRANSPORT WING COVER

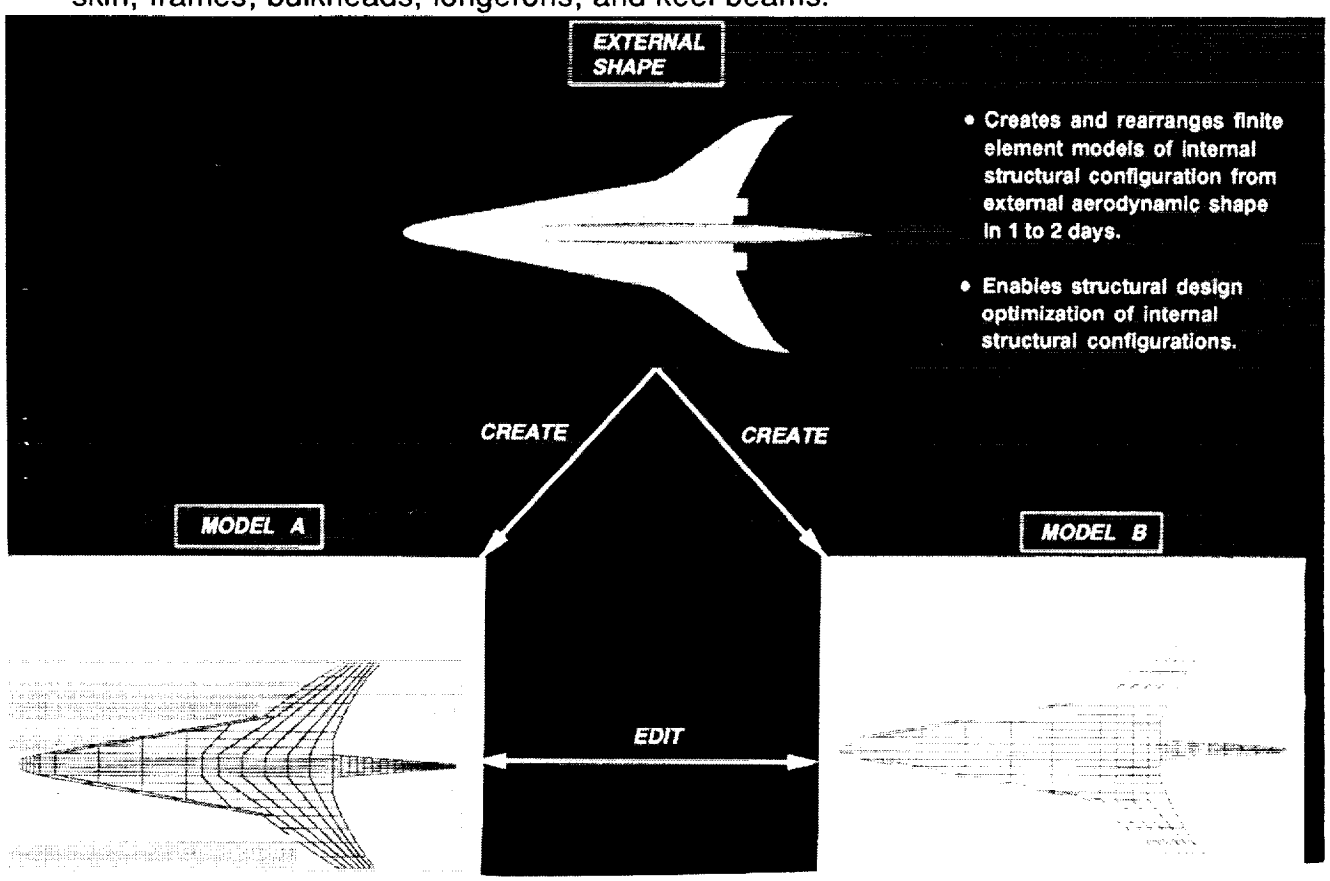
Stress resultants in a future high-speed supersonic civil transport wing cover are shown in the figure. Thermal and mechanical loads for a 2.5g pullup maneuver were used to size the vehicle and to determine stresses and unit weights. Two different materials were used for the wing structure, a conventional titanium alloy, Ti-6Al-4V, and a quasi-isotropic graphite reinforced organic matrix composite material. The normal stresses in the wing upper cover panels are shown in the figures at the right. As shown in the figure, maximum normal stresses occur in the section of the wing where the leading edge crank occurs. In addition, thermal stresses were considerably lower for the composite structural design than for the titanium structural design due to the lower coefficient of thermal expansion for the composite material. Unit weight distribution in the upper wing cover is shown in the figure at the left. As expected, the weight of the composite wing cover is much less than that of the titanium wing cover.



INTERACTIVE MODEL GENERATION CAPABILITY FOR AIRFRAME INTERNAL STRUCTURAL CONFIGURATION RESEARCH

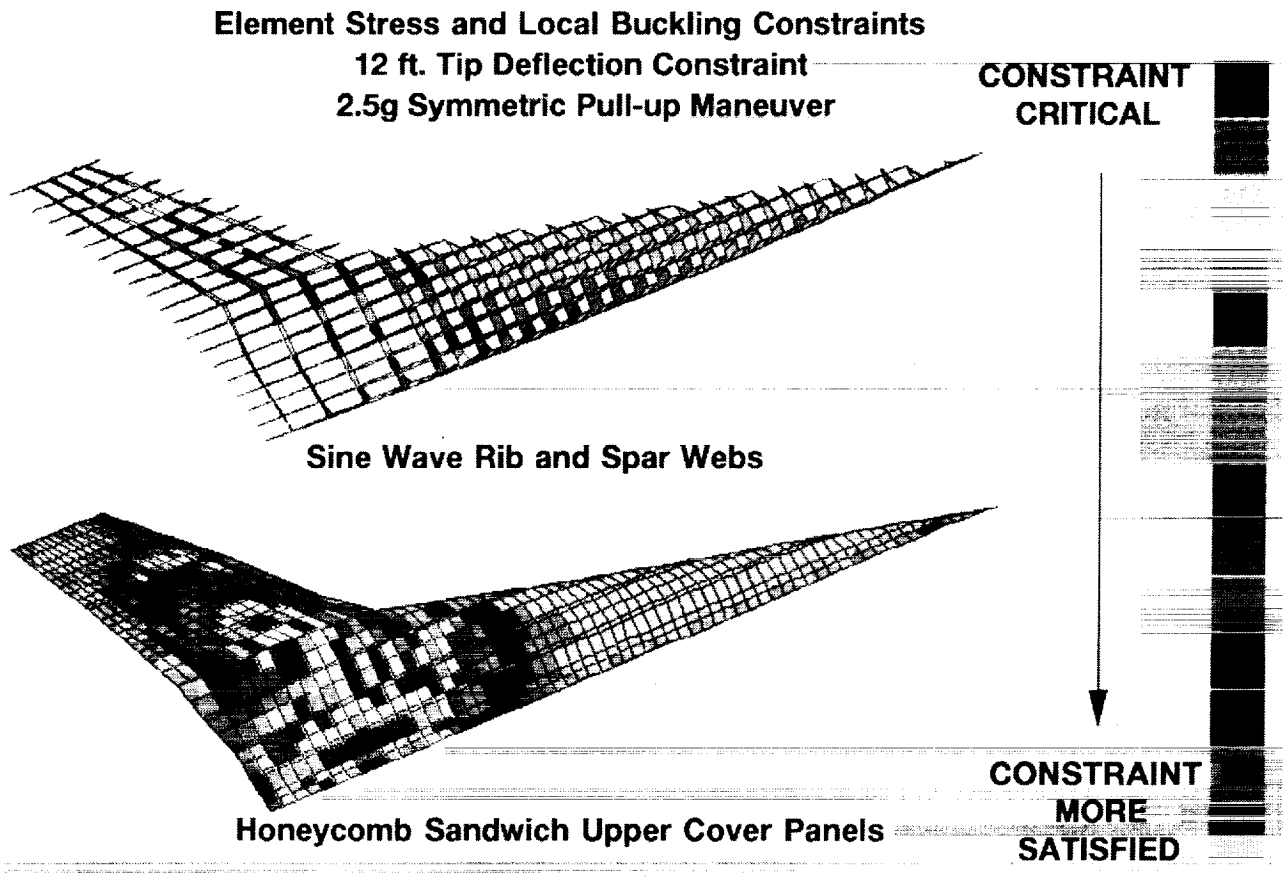
The time required to generate a three-dimensional structural finite element model from an external aerodynamic shape can be very long, on the order of months. Work is underway at NASA Langley to enhance the Solid Modeling Aerospace Research Tool (SMART) to enable rapid structural modeling of external and internal structure from a given aerodynamic shape. Currently, the computer time required for linear structural analysis is much less than the actual time to create and modify a structural finite element model. As shown in the figure, the purpose of the present research is to reduce the time to model internal and external structure from months to days. This reduction in modeling time will enable a more rapid assessment of internal structural dimensions and arrangements and speed up the optimum placement and sizing of internal structure. Planned enhancements will enable the generation of internal structural configurations such as those shown and enable the rapid rearrangement of internal structures as illustrated by the different structural arrangements in the models shown.

Planned enhancements to SMART include a means for: 1) creating and editing structural elements for the wing and fuselage; 2) integrating wing and fuselage structural components; 3) integrating tail and fuselage components; 4) remapping aerodynamic loads data to the structural model; 5) applying point and distributed loads to the structural model; and 6) preparing loads data for visual presentation. Examples of structural elements and components to be modeled include wing spars, ribs, shear webs and cover panels and fuselage skin, frames, bulkheads, longerons, and keel beams.



DESIGN CONSTRAINT CRITICALITY FOR MINIMUM WEIGHT HIGH-SPEED SUPERSONIC CIVIL TRANSPORT WING DESIGN

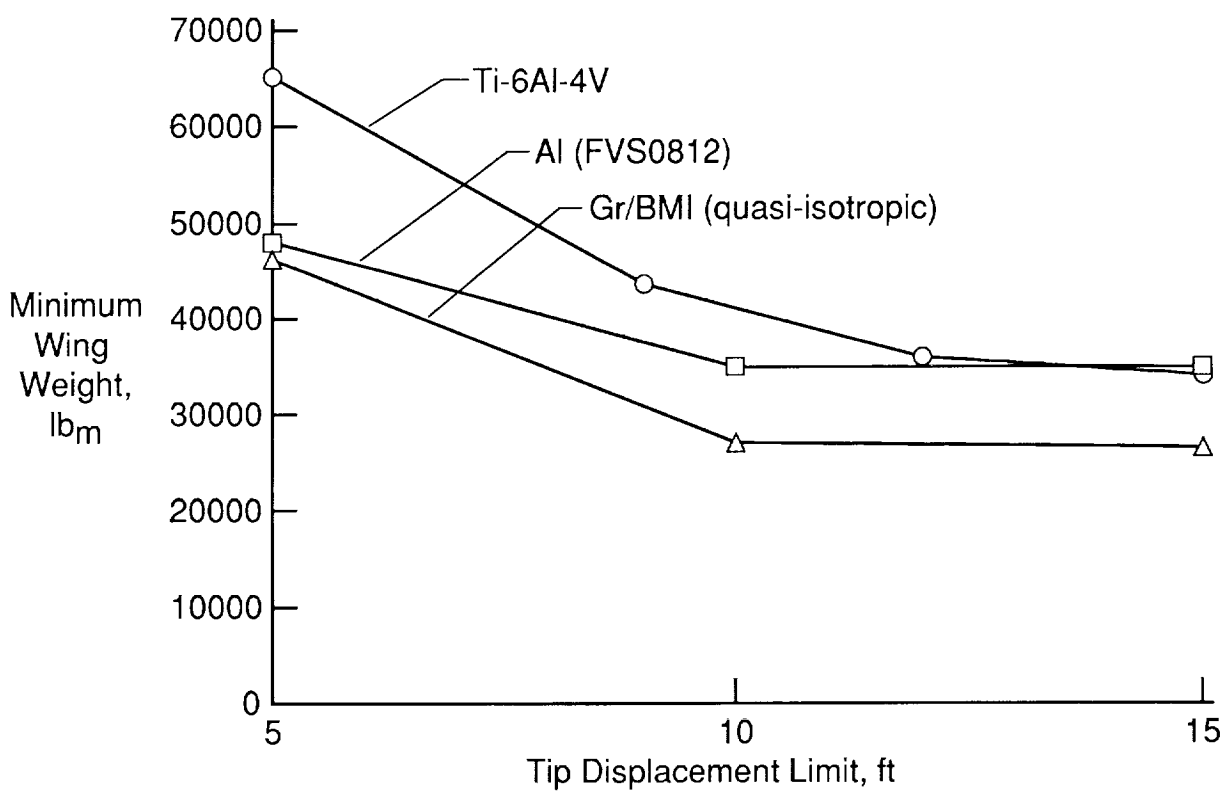
A future high-speed supersonic civil transport (HSCT) model was optimized using formal optimization procedures to satisfy element stress, local buckling, and displacement constraints. The vehicle structure was sized for a 2.5g pullup maneuver and a 12-foot tip deflection constraint. Sine-wave rib and spar elements were used to minimize thermal stresses in the wing and honeycomb sandwich panels were used for the upper and lower cover panels. Various regions of the structure were governed by different design constraint conditions, either element stress, local buckling or minimum gage constraints. The degree of criticality of each element can be easily monitored by using a simple color coding scheme to identify regions where constraints are critical or satisfied. Upon investigation of a critical region, it can be determined which constraint is approaching criticality.



SENSITIVITY OF MINIMUM-WEIGHT HIGH-SPEED SUPERSONIC CIVIL TRANSPORT WING DESIGNS TO MATERIAL PROPERTIES

The sensitivity of a future high-speed supersonic civil transport (HSCT) to the wing-tip deflection constraint is illustrated in the figure. The minimum-weight wing design of a candidate supersonic transport wing is plotted as a function of wing-tip deflection limit for three different structural material choices: a titanium alloy(Ti-6Al-4V), an advanced aluminum alloy (FVS0812), and a quasi-isotropic graphite-bismalimide (Gr/BMI) material (IM7/5260). When the tip displacement is relaxed (e.g., for a tip displacement limit of 15 feet), the titanium and advanced aluminum designs are similar in weight and considerably heavier than the Gr/BMI composite design. As the dip deflection limit is reduced and approaches 5 feet, the advanced aluminum and Gr/BMI designs become similar in weight and considerably less than the titanium design. The advanced aluminum chosen is similar in stiffness to a quasi-isotropic Gr/BMI composite structure. The benefits of the composite material in producing a lighter weight design can be realized if tailoring the structural design to exploit the benefits of the directional properties of the material is permitted. Future work will address the potential benefits of structural tailoring on minimum weight design.

2.5 G Supersonic Pull-up
Strength, Buckling, and Tip Displacement Constraints

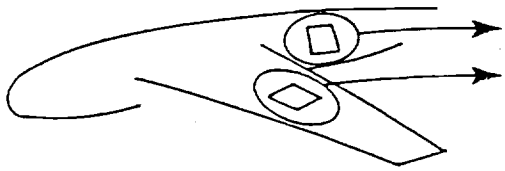


STIFFENED PANEL DESIGN CODE — PASCO

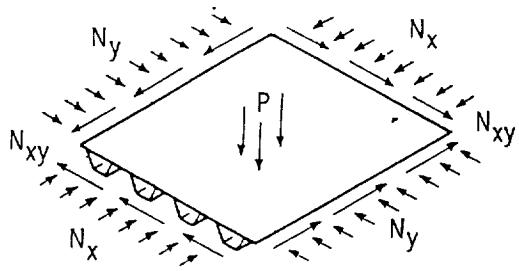
The Panel Analysis and Sizing Code (PASCO) (refs. 1 and 2) is a computer design code which combines a rigorous buckling analysis with a nonlinear mathematical optimization algorithm to perform structural analysis and minimum-weight optimization of longitudinally-stiffened composite panels. PASCO is restricted to prismatic structures having an arbitrary cross section. The PASCO program can accommodate the design of fuselage and wing structural panels which can be loaded by any combination of in-plane loads, lateral pressure and thermal loads. Initial "bow-type" imperfections in the panel geometry can also be analyzed using PASCO.

PASCO uses a linked-plate representation of geometry in which individual plates are assembled to construct a structural panel cross section such as those shown in the figure. Two or more individual plates are assembled to form a substructure, which is repeated to create the entire panel cross section. Plates are constructed as a balanced symmetric laminate of a prescribed number of plies with orthotropic material properties. PASCO can perform a local or global buckling analysis of a stiffened panel for various combinations of free or supported boundary conditions along the panel edges. Local buckling loads and mode shapes, such as those shown in the figure, are routinely calculated by PASCO. In addition, PASCO can perform a structural analysis of the loaded panel and minimize panel weight subjected to various stress and buckling constraints.

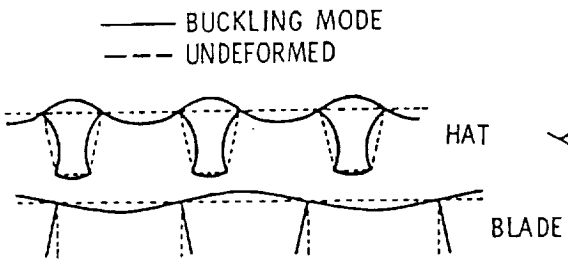
A MacIntosh version of PASCO has been developed (ref. 3) and an interactive graphical interface to the MacIntosh version of PASCO, called MacPASCO, has also been developed. The graphical interface was created to simplify user input and model checkout (ref. 4). PASCO, the MacIntosh version of PASCO, and the graphical interface, MacPASCO, are available through COSMIC.



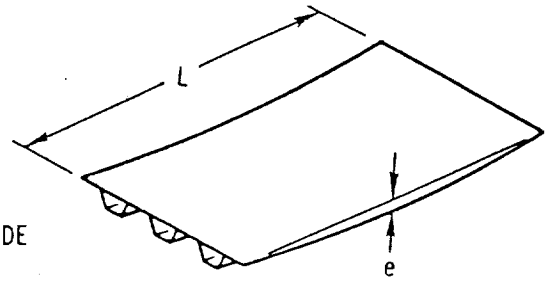
STRUCTURAL PANEL



LOADING



COMPLEX BUCKLING MODES OF ARBITRARY PANEL CONFIGURATIONS

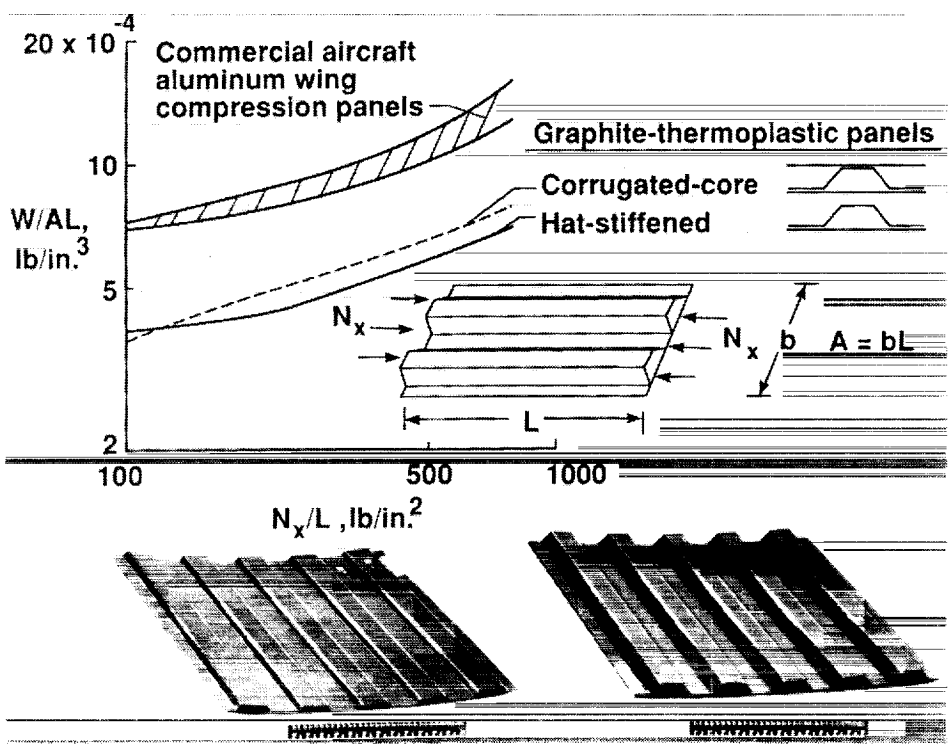


BOW-TYPE IMPERFECTION

STIFFENED PANEL DESIGN CODE — PASCO

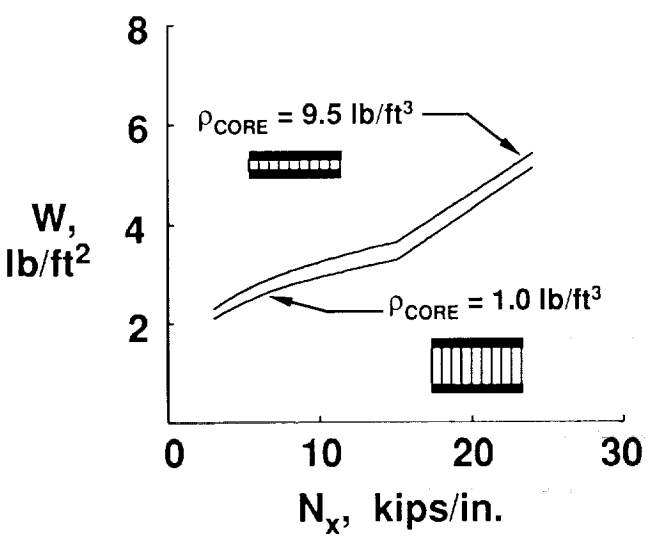
STRUCTURAL EFFICIENCY DETERMINED FOR OPTIMIZED GRAPHITE/THERMOPLASTIC PANELS

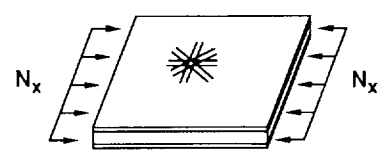
One application of PASCO to size structurally efficient stiffened composite panels is shown in the figure. Minimum weight designs for compression-loaded graphite-thermoplastic panel concepts were developed for a range of loading intensities and the results are compared with current aluminum designs in the figure. The weight, normalized by the planform area, A, and the panel length, L, is shown for different values of applied load, N_X, normalized by the panel length. Two graphite-thermoplastic concepts are compared, one with a corrugated core and two face sheets and one with a hat-stiffened configuration. The concepts are based on a cost-effective fabrication process that allows the corrugated core and hat-stiffened section to be thermoformed and subsequently attached to the face sheets.



STRUCTURAL EFFICIENCY OF OPTIMIZED SANDWICH COVER PANELS IS INSENSITIVE TO CORE DENSITY, ρ_{core}

A structural optimization study of a sandwich panel concept with composite face sheets has been conducted which includes damage tolerance constraints as well as strength and bucking constraints (ref. 5). The results of the study indicate that imposing a maximum strain constraint of 0.0045 in./in. will provide designs with thick enough face sheets to tolerate reasonable low-speed impact damage. The results of the study indicate that core density ρ_{core} does not significantly affect the weight, W, of the designs over a range of applied compressive loads, N_X. Since core density did not strongly affect these design weights, a heavier more damage-tolerant core can be used for the design with minimum weight increase.



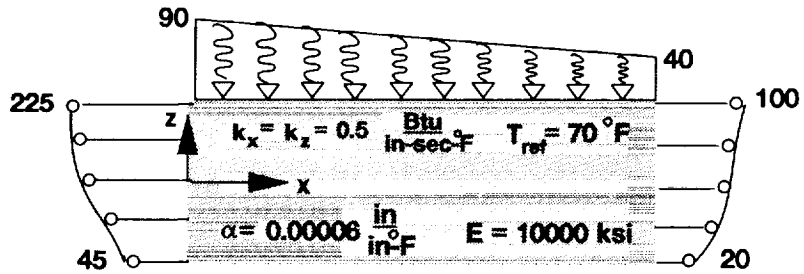


- Orthotropic facesheets
- Response mechanisms: global buckling, facesheet wrinkling, material failure
- Damage tolerance constraint: $\varepsilon_x < 0.0045$ in./in.

EFFICIENT RITZ-BASED ELEMENTS DEVELOPED FOR THERMAL-STRUCTURAL ANALYSIS OF BEAM AND PLATE STRUCTURES

The structural design of supersonic and hypersonic aircraft requires the efficient and accurate calculation of structural temperatures, the transfer of these temperatures to a discrete structural model, and the efficient and accurate calculation of the structural response. Discrete thermal and structural models are often dissimilar and require some form of mapping to transfer temperatures from the thermal model to the structural model. In addition, discrete structural elements like beam and plate elements have no analogous thermal elements which can be used to calculate the temperature distribution through the element thickness. Thermal models require significantly more detail to predict the through-the-thickness variation of temperature. The objective of the present research is to develop thermal elements which are compatible with structural beam and plate elements. Compatible thermal and structural elements can reduce modeling time, problem size, and the computation time required to obtain accurate thermal stresses. Ritz-based thermal and structural elements were developed as a means to alleviate some of the problems associated with thermal-structural analysis.

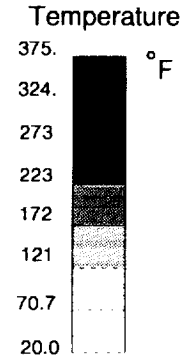
Structural and thermal energy functionals are developed which include parallel formulations for internal energy and the energy associated with boundary conditions. The Ritz method is used to develop the governing equations for the thermal and structural elements. Temperature and thermal stress results for a Ritz-based analysis of a heated beam are compared to linear finite element results as shown in the figure. The structure has mixed thermal boundary conditions and is fixed against translations and rotations at each end as shown in the upper sketch. Results from finite element thermal and structural analyses are shown for comparative purposes. Both the Ritz and finite element results were chosen from convergence studies which examined changes in the temperature and stress as a function of degrees of freedom. Convergence studies for the Ritz-based element required only an increase in the interpolation function order while finite element studies necessitated mesh refinement and associated changes in loads and boundary conditions defined at nodes. The Ritz-based analysis requires only 12 degrees-of-freedom for accurate prediction of temperatures and 34 degrees-of-freedom for accurate prediction of thermal stresses. The conventional finite element analysis requires 99 degrees-of-freedom for accurate temperature calculation and 198 degrees-of-freedom for accurate thermal stress calculation. The Ritz-based elements are capable of representing mixed boundary conditions including convection for a steady-state thermal analysis and prescribed displacements for structural analysis. Orthotropic and layered media can also be modeled with the Ritz-based elements. More detailed information on Ritz-based elements can be found in reference 6.



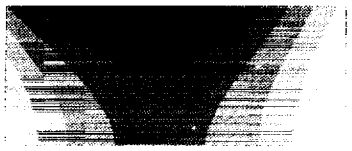
ends restrained against translation and rotation

prescribed temperature °F

heat load Btu / in.

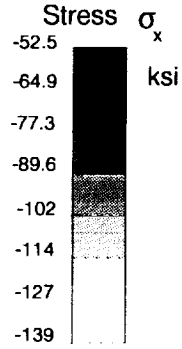


Thermal Analysis - Temperature



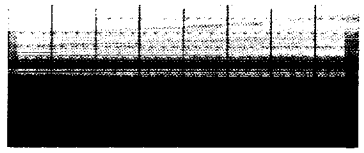
Ritz-based element
12 degrees of freedom

Finite element - 99 degrees of freedom



Structural Analysis - Thermal Stress

Ritz-based element
34 degrees of freedom

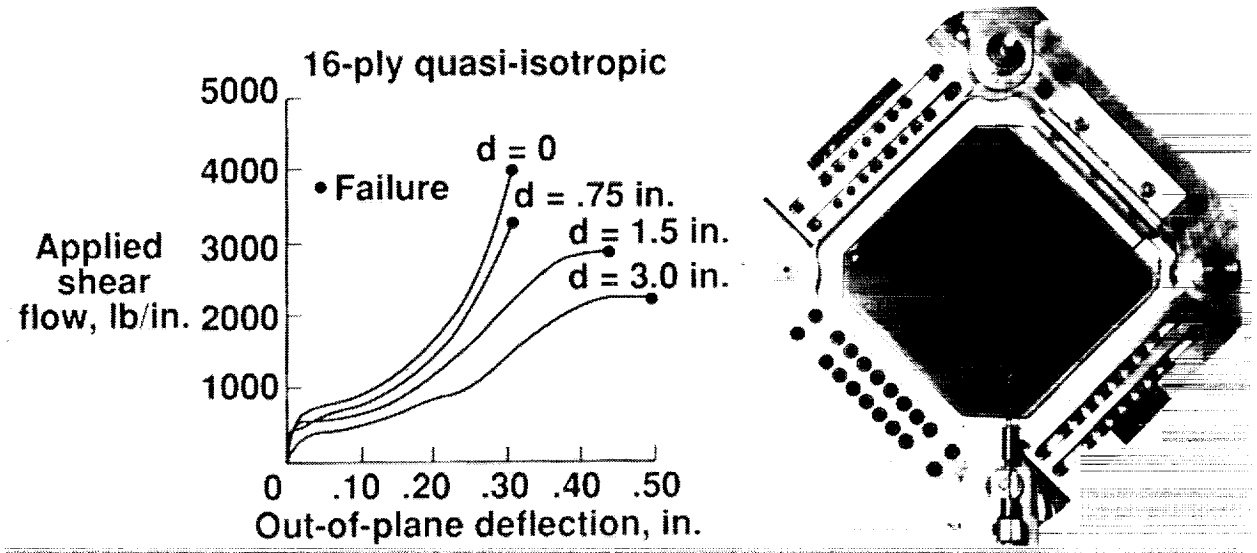


Finite element - 198 degrees of freedom

EFFICIENT RITZ-BASED ELEMENTS DEVELOPED FOR THERMAL-STRUCTURAL ANALYSIS OF BEAM AND PLATE STRUCTURES

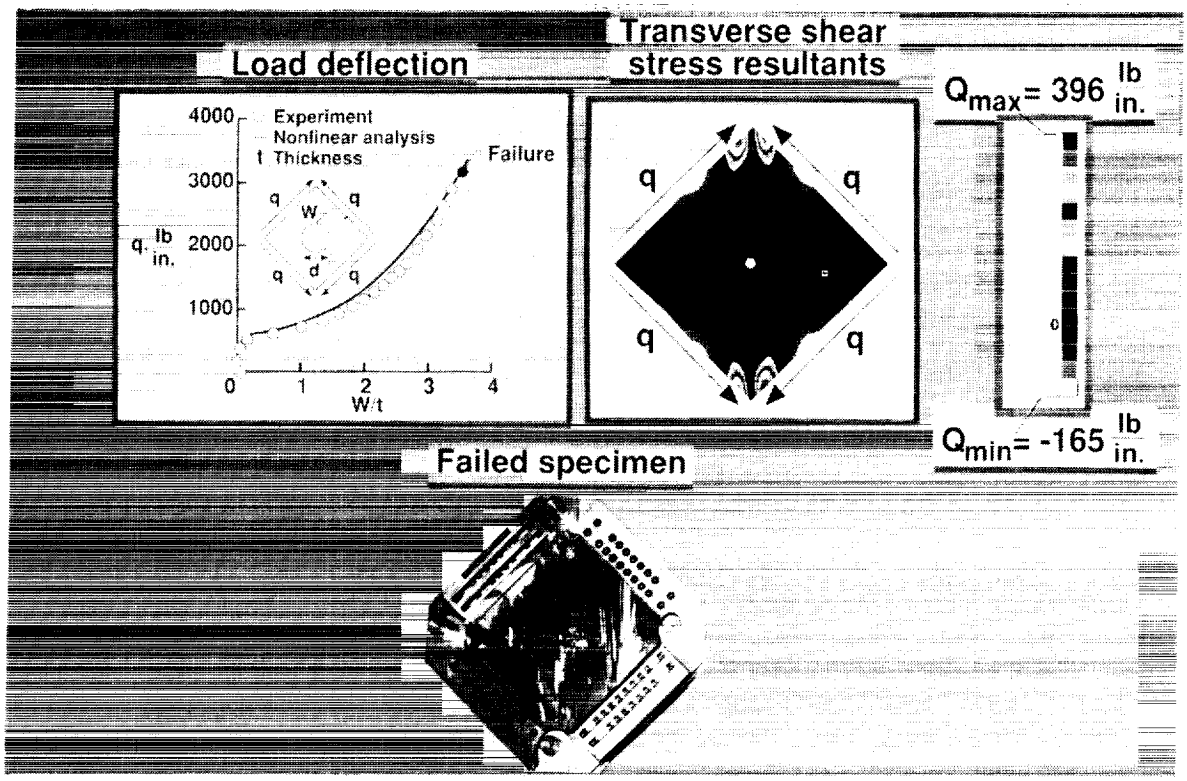
NASA/BOEING STUDY OF POSTBUCKLING BEHAVIOR FOR THERMOPLASTIC SHEAR WEBS WITH HOLES

Graphite-thermoplastic panels are being considered for supersonic aircraft applications. The results of an experimental study of graphite-thermoplastic shear webs with circular cutouts is shown in the figure (ref. 7). The cutout size was varied in the study from a diameter of 0 to 3 inches and the specimens were loaded to failure in a picture-frame shear test fixture. All panels buckled before failure. The results in the figure show the out-of-plane deflection as a function of applied shear load for difference cutout sizes. All panels had out-of-plane deflections ranging from approximately 4 to 6 times the 0.080 inch-thick shear-web thickness.



POSTBUCKLING RESPONSE AND FAILURE OF THERMOPLASTIC SHEAR WEBS

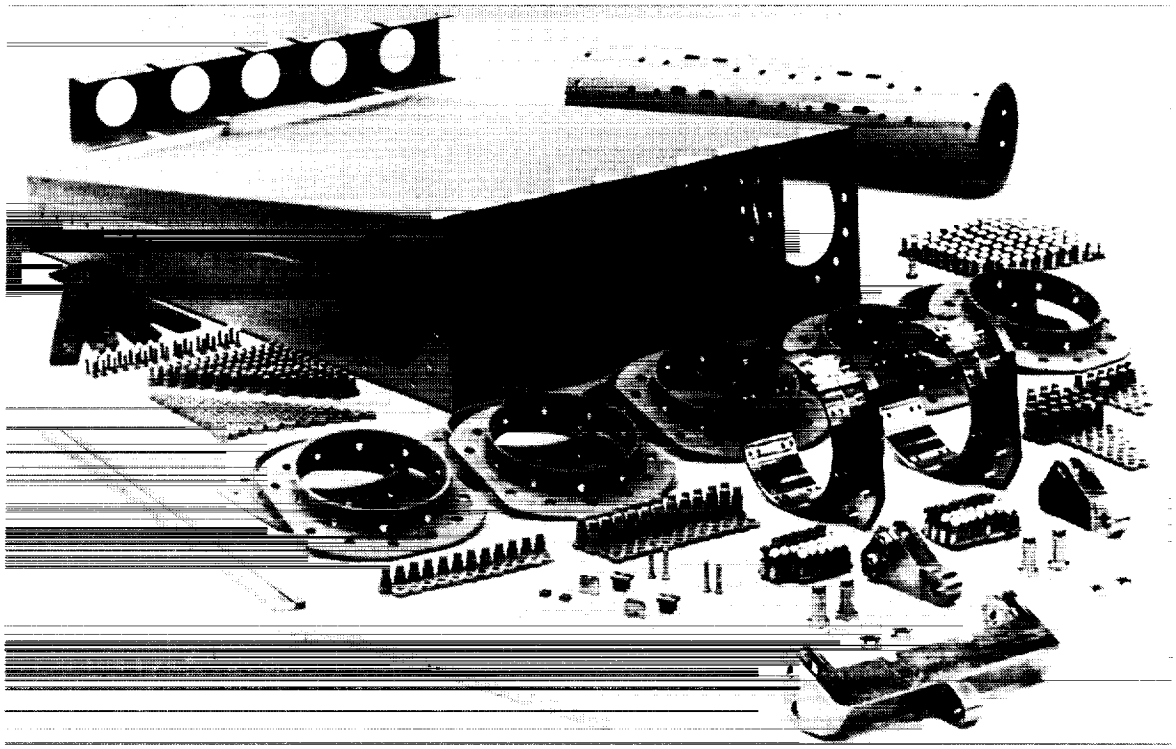
The results of a typical postbuckling analysis of a graphite-thermoplastic shear web with a 0.75-inch-diameter cutout is shown in the figure. Large shear stress gradients are shown in the right figure that correspond to the locations of failure in the panel shown in the lower figure. The analytical response prediction compares well with the test data as shown in the left figure.



CARBON-CARBON CONTROL SURFACE TEST COMPONENT (UNASSEMBLED)

Based on results of previous conceptual studies, it is believed that a refractory-composite material such as Advanced Carbon-Carbon (ACC) offers significant advantages which warrant its selection for control surfaces on a hypersonic vehicle. A generic elevon configuration was selected which would carry significant mechanical loads and reach maximum temperatures as high as 3000 °F. At the present time the elevon system has been designed, fabricated and assembled. The elevon was fabricated by LTV Corporation under the direction of NASA Langley. The major carbon-carbon (C/C) and refractory-metal (Rene' 41) parts which compose the elevon assembly are shown in the figure. The C/C parts consist of a 3-foot by 5-foot built-up structure with rib and skin panels, a torque tube, ten rib-to-tube attachment fittings, a closure piece, and many C/C fasteners and nuts. Rene' 41 pieces include: attachment rings, lugs, fasteners and cleats used to join the C/C torque tube to the wing support structure. Sub-component testing has begun on the C/C torque tube at NASA Langley as will be shown later in the paper. Testing of the full-scale elevon component will be performed in the structures laboratory at NASA Dryden.

Significant advancements in C/C design, analysis, and fabrication technology were necessary for fabrication of the test component. The large size of the test article and the need for close tolerances were a significant challenge that required advancement in the state of the art. Fabrication of the rib and skin panel built-up structure, the torque tube, and the rib-to-tube attachment fittings required significant expertise to develop. The high design torque loads and the large difference in coefficients of thermal expansion between the C/C torque tube and the Rene' 41 actuator lug posed a major challenge. The high design temperatures (up to 3000 °F) required the use of C/C fasteners for the assembly and is the first application of C/C fasteners to a structure that will be subjected to cyclic thermal and mechanical loads. Further details of the C/C elevon can be found in reference 8.

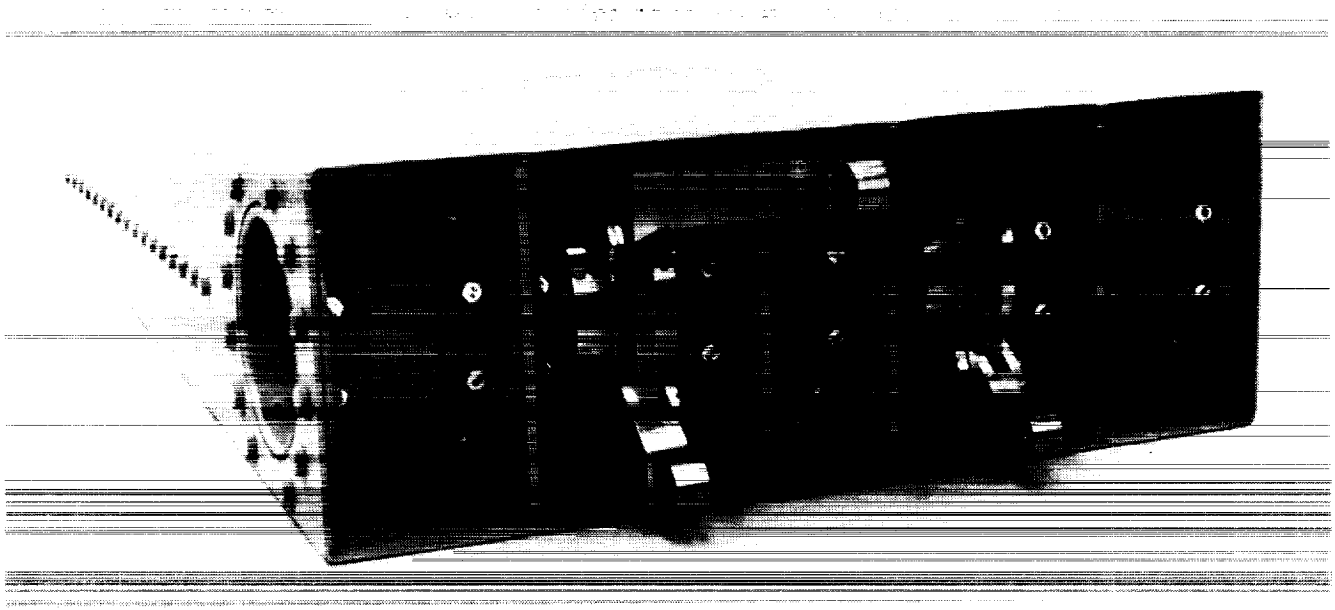


CARBON-CARBON CONTROL SURFACE TEST COMPONENT
(UNASSEMBLED)

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

CARBON-CARBON CONTROL SURFACE TEST COMPONENT (ASSEMBLED)

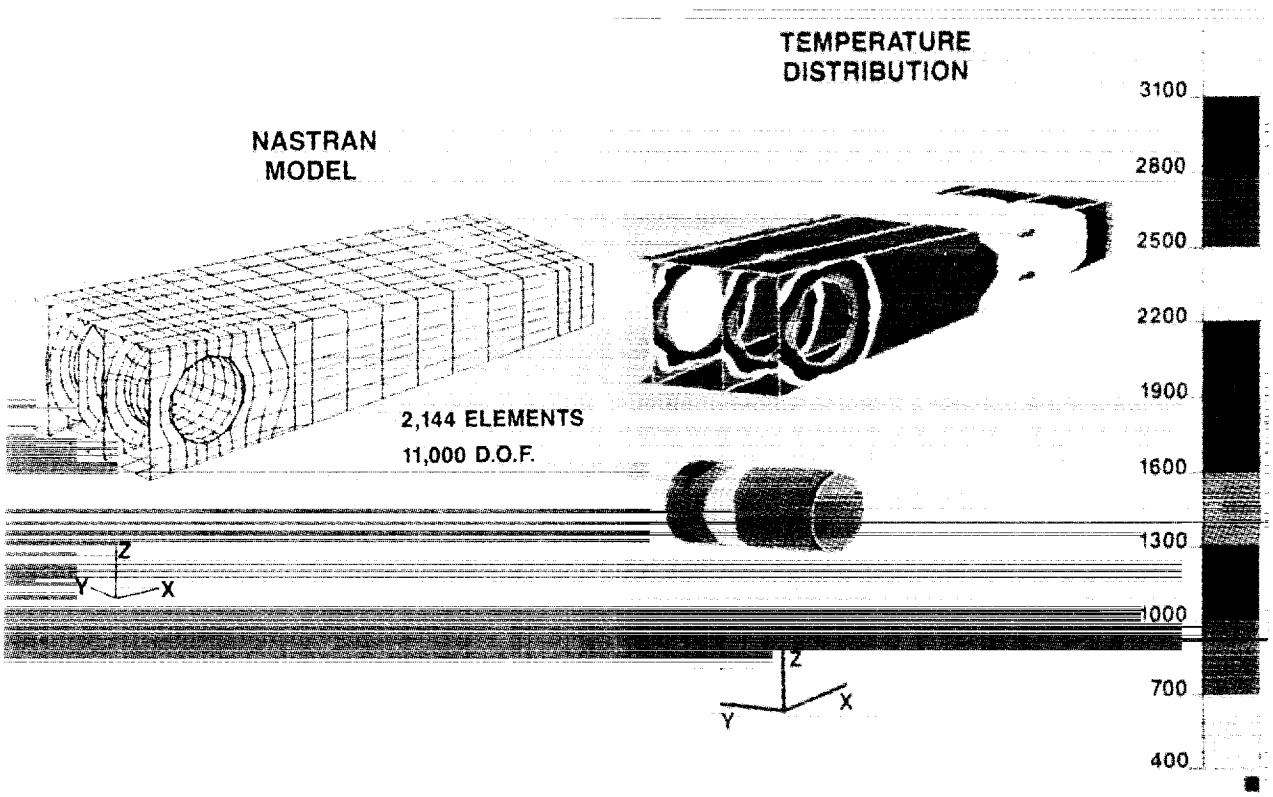
The figure illustrates a completely assembled Carbon-Carbon (C/C) elevon for a hypersonic vehicle complete with C/C fasteners and attachment rings and Rene' 41 fasteners and attachment pieces. The accurate fit-up and adherence to close dimensional tolerances is a testament to the significant advances in C/C manufacturing and rigorous fabrication procedures made during this program.



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

ELEVON NASTRAN MODEL AND ITS TEMPERATURE DISTRIBUTION

A three-dimensional NASTRAN thermal finite element model of the carbon-carbon elevon was developed to calculate detailed temperatures and temperature distributions. The detailed model was necessary to accurately predict temperatures and thermal loads necessary for accurate thermal stress calculation. Temperatures at critical regions where the refractory-metal Rene' 41 fasteners and fittings are located also necessitated accurate temperature prediction. The finite element model uses over 2000 elements and has over 11,000 degrees of freedom. Temperature contours in the elevon and torque tube for a generic flight profile which simulated ascent, cruise, and descent are shown at the right of the figure. Maximum temperatures in the ACC carbon-carbon elevon are 3000 °F and maximum temperatures of the Rene' 41 fasteners are 1600 °F.



DETAILED STRESS ANALYSIS OF CARBON-CARBON TORQUE TUBE

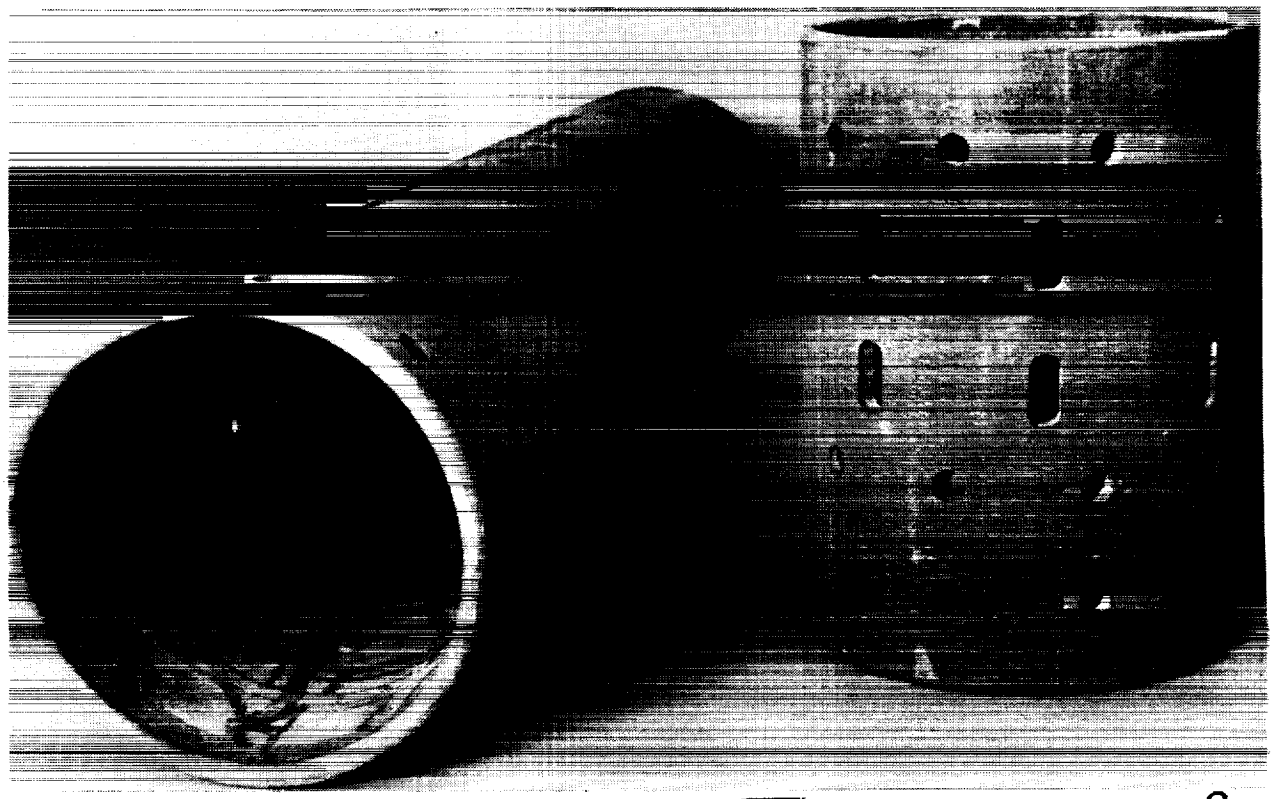
The carbon-carbon short torque tube is being tested to obtain preliminary performance data for a 9.5-in.-diameter torque tube constructed of 42 plies of woven carbon-carbon material. All loads and attachments are similar to those of the carbon-carbon elevon torque tube which will be tested later.

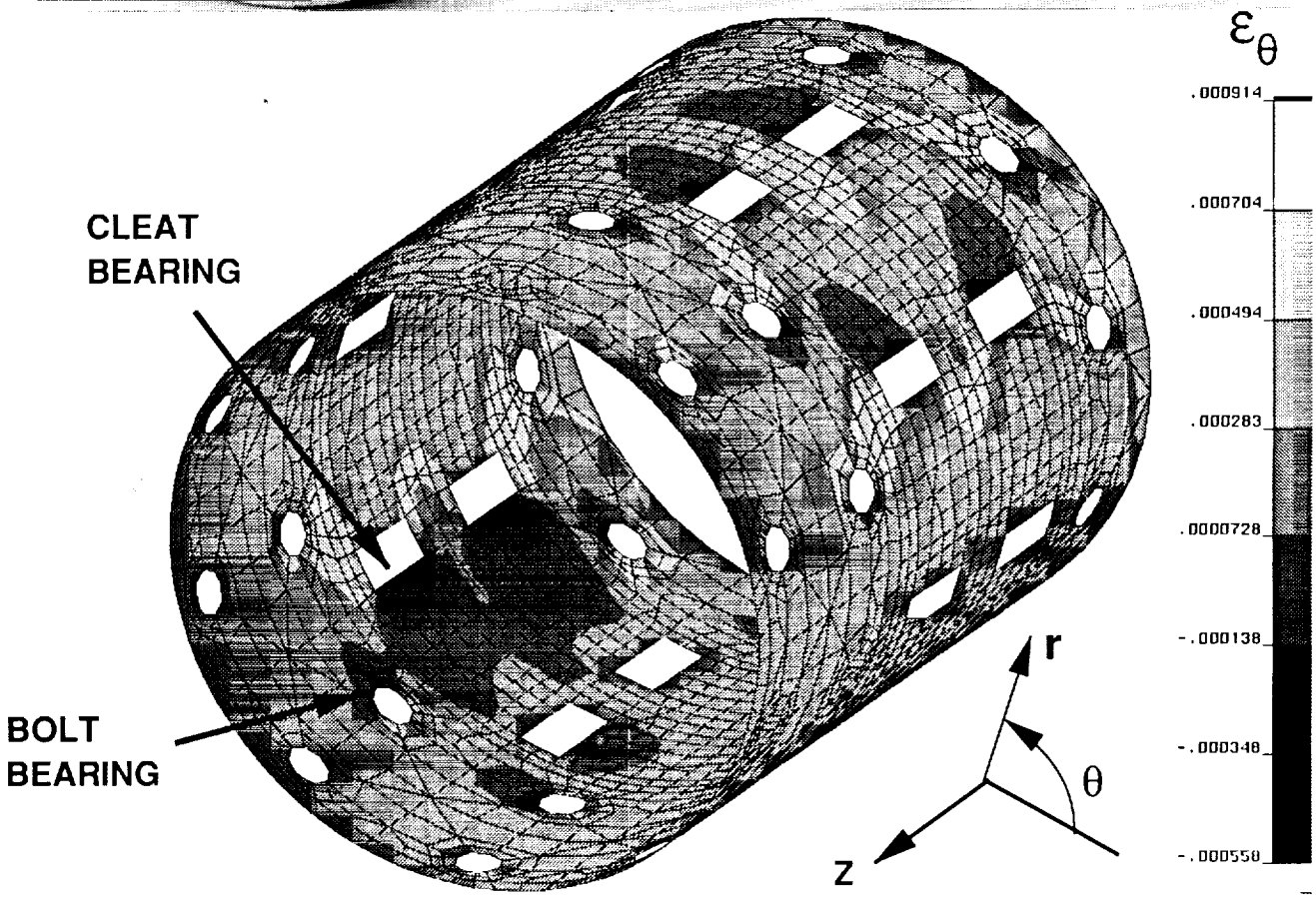
The short torque tube model, shown in the figure, was taken from a larger model which consisted of the torque tube, attachment rings, load arms, and support collar. The 0.5-inch-walls of the torque tube were modeled with plate bending elements having quasi-isotropic material properties. Cleat holes and bolt holes were included in the model since the proximity of these openings has an influence on the stress field. Mechanical fasteners were modeled with rigid beam elements which transmit only compressive loads to bearing surfaces. Perfect fit-up was assumed at all bolt holes and cleat holes.

The torque loads are transferred to the tube at the bolt holes resulting in a clockwise rotation. These loads are reacted by the cleat holes of the support collar. The compressive bearing strains at the bolt holes occur on the opposite face from the compressive strains at the cleat holes. The largest compressive strains in the torque tube occur where the couple forces from the load arms are applied to the tube. The compressive strain limit of .0013 in./in. for a carbon-carbon lamina is about one-third of the tensile strain limit.

There is some local bending at the cleat holes and bolt holes since single-lap joints are used to transfer loads. A local three-dimensional finite element model was used to investigate bending effects. Displacements from the planar model analysis were specified as boundary conditions for the local model analysis. The outer fiber compressive strains from the solid model analysis were about 2.5 times greater than the membrane strains obtained from the planar model analysis. This maximum value is approximately equal to the allowable compressive strain specified by the LTV Corporation which fabricated the torque tube.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH





DETAILED STRESS ANALYSIS OF CARBON-CARBON TORQUE TUBE

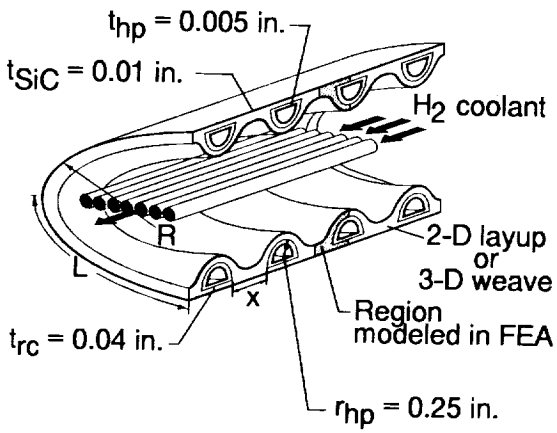
REFRACTORY-METAL/REFRACTORY-COMPOSITE HEAT-PIPE-COOLED WING LEADING EDGE

A refractory-metal/refractory-composite heat-pipe-cooled wing leading edge concept, shown schematically in the left of the figure, is currently being considered for use on a hypersonic aircraft. The heat-pipe concept has the potential to reduce leading edge weight by 50 percent over an all actively cooled leading edge and to result in a more reliable and redundant design. In addition, the heat-pipe-cooled leading edge concept eliminates the need for active cooling during descent and even the more severe ascent portions of the trajectory. Elimination of active cooling from the leading edge design greatly reduces systems complexities and weight.

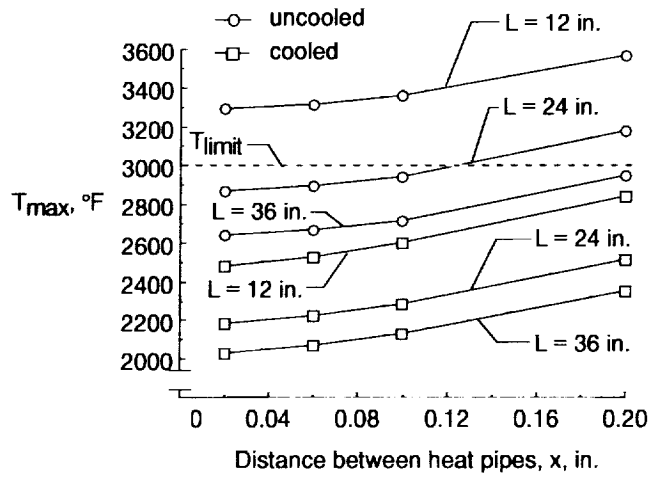
The concept uses thin refractory-metal "D-shaped" heat pipes embedded within a refractory-composite structure. The heat pipes are spaced close to one another and arranged normal to the leading edge. The heat pipes cool the stagnation region by efficiently transporting heat aft to the upper and lower surfaces of the wing where the heat is rejected by external radiation to space. The heat pipes effectively isothermalize the leading edge because the mechanism for transporting the heat is very efficient and relies on the evaporation and condensation of a high temperature working fluid; lithium in this particular design. The heat pipes are self contained and require no external pumping for the lithium working fluid. The lithium evaporates in the stagnation region and condenses in the aft sections of the heat pipe. The liquid condensate is pumped back to the stagnation region by the capillary pumping action of an internal wick structure. The heat pipes are sized to be redundant in the event of an individual heat pipe failure and the refractory-composite structure surrounding the heat pipes is also designed to offer ablative protection in the event of a massive heat pipe malfunction. If active cooling is necessary during ascent, it can be accommodated by internal radiation to an actively cooled heat exchanger as shown in the figure.

A thermal parametric trade study was performed (ref. 9) using a three-dimensional finite element model of a portion of a single heat pipe, shown by the shaded region in the figure on the left. Maximum temperatures for a 0.5-inch-radius design are shown in the figure on the right for the case of an uncooled leading edge and an internally cooled leading edge design. Parameters varied in the study were the wetted length of the heat pipes and the heat-pipe spacing. As shown in the figure, if the spacing between heat pipes is less than 0.1 in., a 24-in.-long heat pipe cools the stagnation region sufficiently to reduce maximum temperature below a 3000 °F temperature limit for the refractory-composite structure.

REFRACTORY METAL/REFRACTORY COMPOSITE HEAT-PIPE-COOLED WING LEADING EDGE



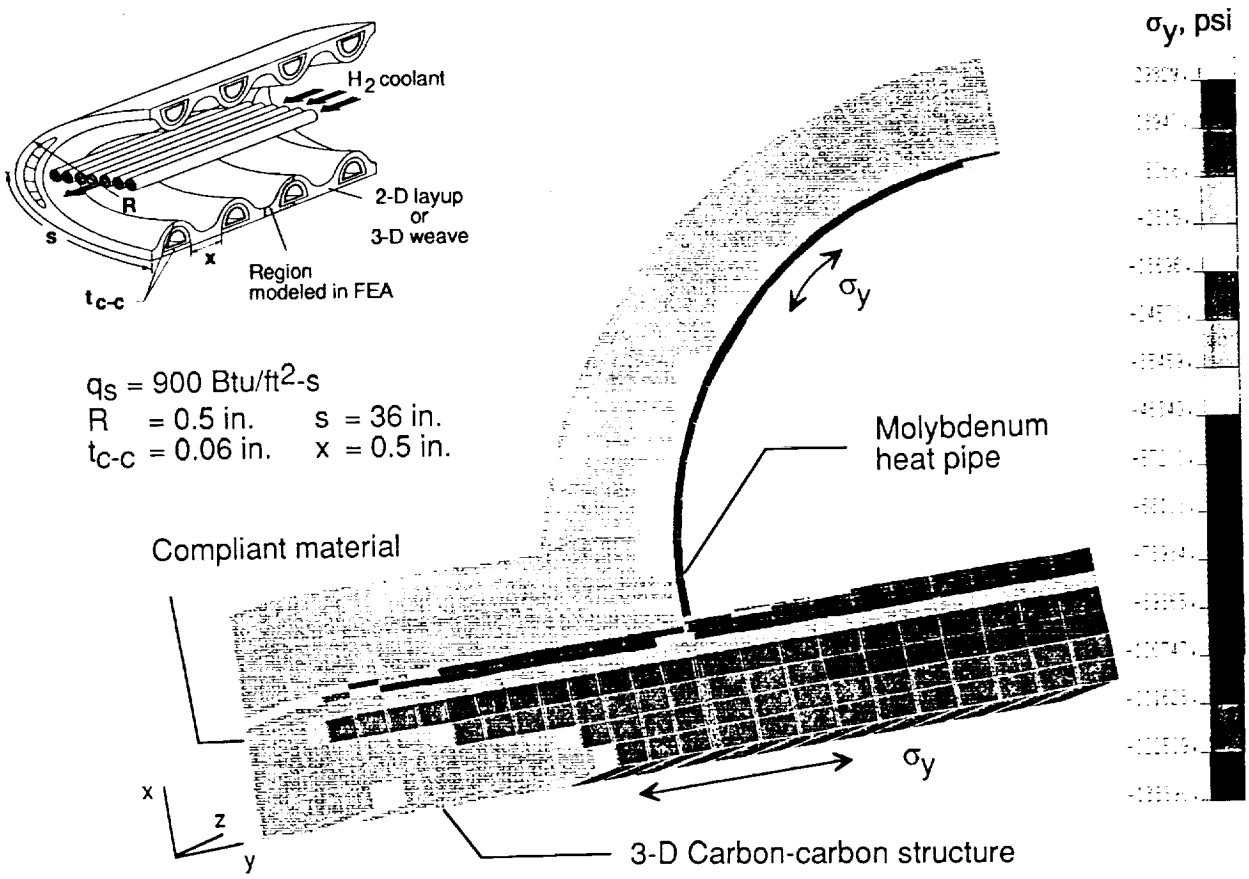
HEAT PIPES EMBEDDED IN REFRACTORY COMPOSITE WITH OPTIONAL INTERNAL RADIATIVE COOLING



THERMAL PARAMETRIC STUDY FROM 3-D FINITE ELEMENT ANALYSIS FOR R = 0.5 in.

CARBON-CARBON/HEAT-PIPE-COOLED LEADING EDGE THERMAL STRESSES AT THE STAGNATION LINE

A detailed thermal stress analysis of a molybdenum heat pipe embedded within ACC4 carbon-carbon structure, results in very high compressive stresses in the molybdenum tube as shown in the figure (ref. 9). At elevated temperatures, the refractory-metal molybdenum "D" tube expands into the lower coefficient of thermal expansion refractory-composite structure. The linear structural analysis does not account for the relief in thermal stress by yielding of the metallic tube. In addition, the structural analysis assumes a perfect bond between the refractory-metal and refractory-composite materials. Methods for alleviating the high thermal stresses are currently being investigated. The use of a soft carbon strain isolator material placed between the metal "D" tube and the refractory-composite structure is currently being investigated as a means of allowing the differential thermal expansion without the creation of excessive thermal stresses.

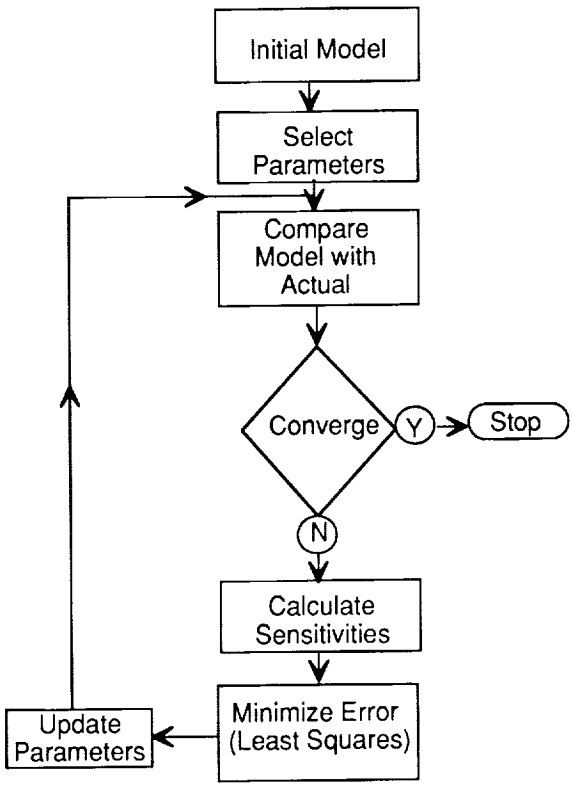


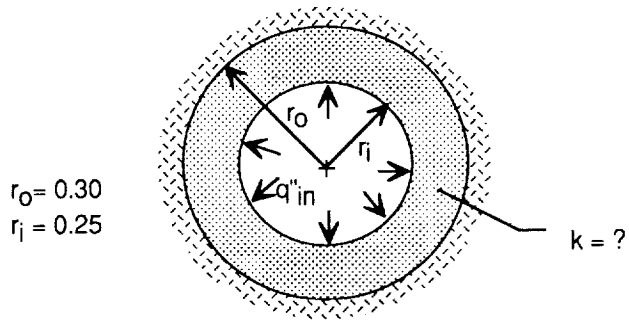
THERMAL-STRUCTURAL PARAMETER ESTIMATION

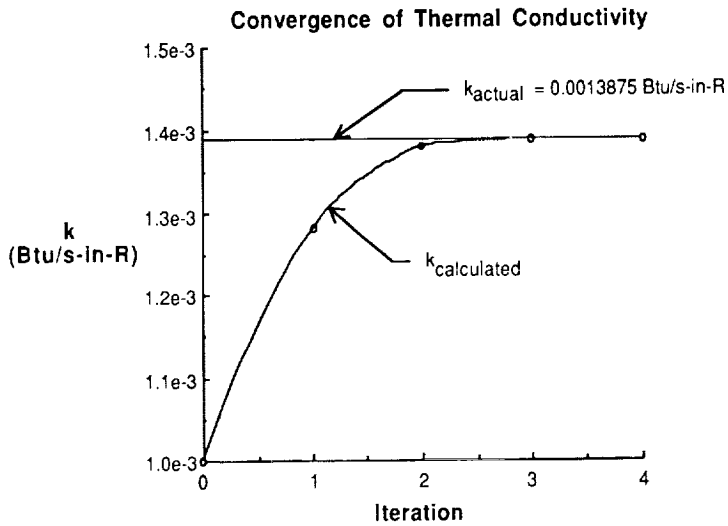
Parameter estimation methods are currently being developed to determine thermal as well as structural parameters and to help facilitate correlation between experimental and analytical results. Parameter estimation techniques are currently being used to determine the thermal contact resistance of refractory-metal heat pipes embedded within a refractory-composite structure. The estimation procedure is implemented within a commercially available thermal-structural finite element program called the Engineering Analysis Language (EAL) System (ref. 10). The flow chart illustrated in the figure depicts the procedural flow which begins with the development of a finite element model which represents a physical experiment. Certain parameters are considered known constants in the finite element model and certain parameters are considered to be variable. Analytical results are compared with actual or test results and a measure of the error or lack of correlation between results is calculated. If the correlation is poor, the sensitivity of the response to parameter variations is calculated and a least-squares procedure is used to minimize the error and predict a new value for the parameters. The procedure is repeated until the error or lack of correlation between actual and calculated responses is reduced below some prescribed tolerance.

A numerical experiment is illustrated in the figure to highlight the convergence of the parameter estimation procedure implemented into EAL. The transient thermal history of a point located on the outer surface of a tube heated internally is predicted numerically assuming a known value for thermal conductivity (k_{actual}=0.0013875 Btu/in.-s-R). The estimation procedure begins with an initial estimate for k_{actual} which is $k_{calculated}$=0.001 Btu/in.-s-R. The procedure automatically converges to the actual value for thermal conductivity in only three iterations. General methods for estimating multiple parameters of complex problems have been incorporated into the program. Future plans include extension of the methods to structural problems and the use of sensitivity information to optimally design experiments to determine accurately the variable parameter values.

THERMAL-STRUCTURAL PARAMETER ESTIMATION







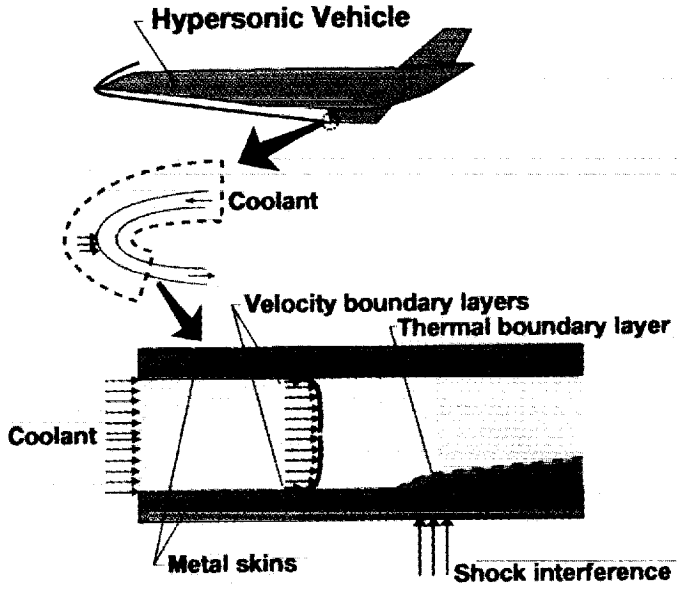
ANALYSIS OF LIQUID-METAL-COOLED NASP ENGINE COWL LEADING EDGE

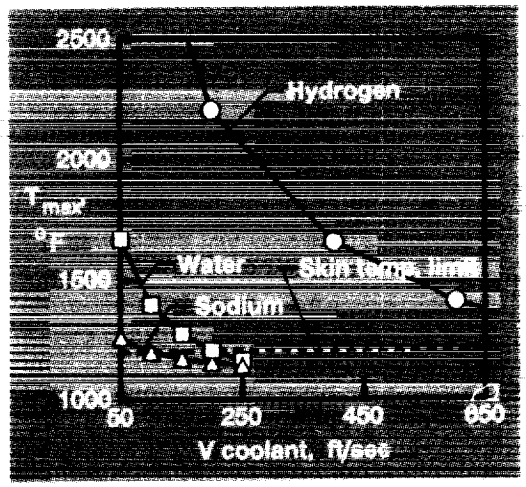
Hypersonic vehicles experience extremely high aerodynamic heating because of the high Mach numbers that the vehicle attains while traveling within the earth's atmosphere. A particularly severe condition exists when the vehicle accelerates and the bow shock from the nose of the vehicle intersects the bow shock of the engine cowl lip. The shock-shock interference produces extremely high and local heating, 50,000 to 100,000 Btu/ft^2-s acting over a region 0.01 to 0.02 inches wide. To predict accurately the thermal performance of a convectively cooled cowl leading edge subject to high local heating requires an accurate description of the internal fluid flow and heat transfer. Computational fluid dynamics (CFD) analysis of the coupled convective flow field and solid conduction to the cowl skin was used to predict accurately the maximum temperatures and temperature gradients within the thin copper skin of an engine cowl leading edge. Previous results, using engineering approximations to determine heat transfer characteristics, did not adequately represent the growth of a local thermal boundary layer and were overly conservative.

A section of the curved cowl lip (see figure) was analyzed as a planar section. The schematic diagram in the figure is not to scale; the true horizontal scale is on the order of inches, the vertical scale is on the order of hundredths of inches, and the heat pulse width is on the order of thousandths of inches. The coolant enters the cowl lip with a uniform temperature and velocity profile. Velocity boundary layers develop and cause the velocity profile to change to one having a nearly uniform velocity core with regions of high shear near the walls. Near the region of the high heat pulse, a thermal boundary layer begins to form in the coolant. The coolant outside this thermal boundary layer is at, or very close to, the inlet temperature while the temperature within the thermal boundary layer is hotter than the inlet temperature.

Results for a local 0.015-in.-wide heat pulse which has a magnitude of 50,000 Btu/ft^2-s is shown at the right of the figure. The cowl lip walls are 0.02-in.-thick copper with a temperature limit of 1200 °F. Results of maximum surface temperature as a function of coolant velocity are shown in the figure for three different coolants: hydrogen, water, and liquid sodium. The high conductivity and thermophysical properties of sodium result in high heat transfer coefficients and lower maximum temperatures. Results of the study indicate that, depending on coolant velocity constraints, several liquids could potentially accommodate high local heating representative of the shock-shock heating condition for a hypersonic vehicle.

ANALYSIS OF CONVECTIVELY COOLED NASP ENGINE COWL LEADING EDGE





AIRCRAFT STRUCTURES RESEARCH TOPICS

The development of verified structural analysis and design technology for future supersonic and hypersonic vehicles requires research on a number of topics as indicated on the figure. Advanced structural concepts are needed for cost-effective structurally efficient designs. Structural tailoring can be used to exploit the beneficial properties of advanced materials. Failure often initiates at stress gradients in a structure, so gradient producing discontinuities and eccentricities must be understood. Local two-dimensional and three-dimensional analyses of these local gradients are needed that are consistent with global two-dimensional plate and shell models. Nonlinear effects associated with postbuckling design philosophies and pressure and thermal loads must be accurately predicted analytically and minimum-weight designs developed for nonlinear structural response. Failure mechanisms must be understood and failure analyses developed to predict accurately the onset of these failure mechanisms. Damage tolerance requirements must be understood for future high speed vehicles and designs must be developed that safely tolerate local damage. The interaction between subcomponents and elements in built-up structures needs to be understood and minimum-weight joint technology needed to connect the elements and subcomponents is needed. Scaling laws for composite and metallic structures are needed to minimize vehicle development costs. Thermal effects and heat transfer into the structure must be predicted to determine thermal stresses and thermal buckling of a structure. In addition, light-weight thermal protection systems and actively cooled structural concepts are needed. High-speed aircraft structures must be designed to withstand combined mechanical, pressure and thermal loading during their flight profiles so the interaction of these loads on structural performance must be understood. All of the analysis and design methodology developed for high-speed vehicles must be verified in the laboratory with the appropriate experiments with panels and wing-box and fuselage-shell models.

- Structural efficiency studies and advanced concepts
- Structural tailoring and anisotropic effects
- Gradients, discontinuities, cutouts, and eccentricities
- 2-D global analysis with 2-D and 3-D local analysis
- Postbuckling and geometric nonlinear effects
- Nonlinear analysis and sizing procedures
- Failure mechanisms and failure analysis
- Damage tolerance and low-speed impact damage effects
- Subcomponent interaction and optimum joints
- Scaling laws for composite structures
- Thermal effects and heat transfer into interior structure
- Thermal stresses and thermal buckling
- Combined mechanical, pressure, and thermal loads
- Panels and subscale fuselage-shell and wing-box models

REFERENCES

1. Anderson, M. S.; Stroud, W. J.; Durling, B. J.; and Hennessy, K. W.: PASCO: Structural Panel Analysis and Sizing Code, User's Manual. NASA TM 80182, November 1981.

2. Stroud, W. J.; and Anderson, M. S.: PASCO: Structural Panel Analysis and Sizing Code, Capability and Analytical Foundations. NASA TM 80801, November 1981.

3. Lucas, S. H.; and Davis, Randall C.: User's Manual for the MacIntosh Version of PASCO. NASA TM 104115, September 1991.

4. Lucas, S. H.; and Davis, Randall C.: User's Manual for MacPASCO. NASA TM 104122, 1991.

5. Cruz, Juan R.: Optimization of Composite Sandwich Cover Panels Subjected to Compressive Loading. NASA CP 3104, Part 2, 1990.

6. Vause, R. F.: Compatible Thermal and Structural Elements for the Thermal Analysis of Beam and Plate Structures. Presented at the AIAA/ASME/ASCE/AHS/ASC 32nd Structures, Structural Dynamics, and Materials Conference, Baltimore, MD, April 8-10, 1991. AIAA Paper No. 91-1154.

7. Rouse, M.: Effect of Cutouts and Low-Speed Impact Damage on the Postbuckling Behavior of Composite Plates Loaded in Shear. Part 2, A Collection of Technical Papers presented at the AIAA/ASME/ASCE/AHS/ASC 31st Structures, Structural Dynamics, and Materials Conference, Long Beach, CA, April 2-4, 1990, pp. 877-892. AIAA Paper No. 90-0966.

8. Ho, T.; Matza, E. C.; Medford, J.; and Watabe, S.: Design Concept Study for NASP Control Surface. NASA CR 18173, October, 1988.

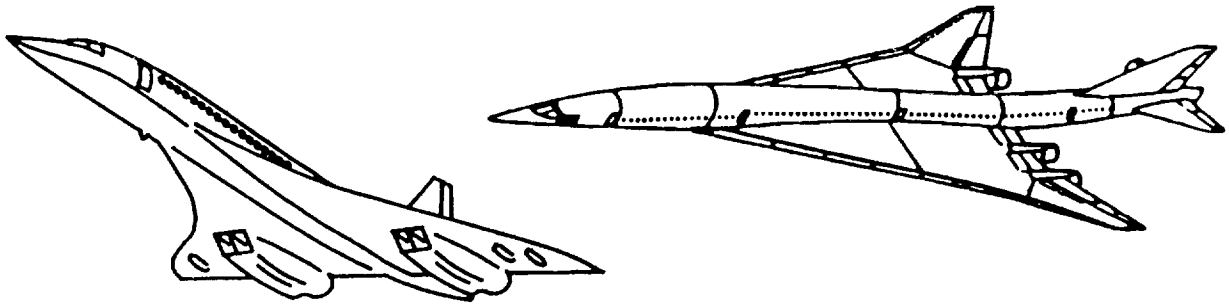
9. Glass, David D.; and Camarda, Charles J.: Preliminary Thermal/Structural Analysis of a Carbon-Carbon/Refractory-Metal Heat-Pipe-Cooled Wing leading Edge. Presented at the First Thermal Structures Conference, University of Virginia, Charlottesville, Virginia, November 13-15, 1990.

10. Whetsone, W. D.: EISI-EAL Engineering Analysis Language Reference Manual - EISI-EAL System Level 2091. Engineering Information Systems, Inc., July 1983.

N92-25919

High Speed Civil Transport

R. L. McKnight
General Electric Aircraft Engines
Cincinnati, Ohio



CONCORDE		HSCT
3,000	RANGE (NML)	5,000 TO 6,000
100	PAYLOAD (PASSENGERS)	250 TO 300
400,000	WEIGHT (LB)	750,000
EXEMPT	COMMUNITY NOISE STANDARD	FAR 36 STAGE III
PREMIUM	FARE LEVELS	STANDARD
APPROX 20	NO_X EMISSIONS (GM/KG)	3 TO 8

Comparison of the HSCT design requirements with the present day CONCORDE.

HIGH SPEED CIVIL TRANSPORT

- ENVIRONMENTAL IMPACT MAJOR DRIVER
  - EMISSIONS
  - NOISE
- CRITICAL COMPONENTS
  - HIGH TEMPERATURE COMBUSTOR
  - LIGHT WEIGHT EXHAUST NOZZLE
- NEW AND ADVANCED MATERIALS
  - HIGH TEMPERATURE CMC'S
  - HIGH TEMPERATURE IMC'S/MMC'S

KEY COMBUSTOR MATERIAL REQUIREMENTS

- HIGH OPERATING TEMPERATURE
- HIGH THERMAL STRESS RESISTANCE
- ACOUSTIC/VIBRATORY DURABILITY
- ENVIRONMENTAL DURABILITY
- DAMAGE TOLERANCE
- SHAPE-FORMING CAPABILITY
- REASONABLE COST

Design projections indicate that a successful HSCT combustor material will need to possess the noted characteristics.

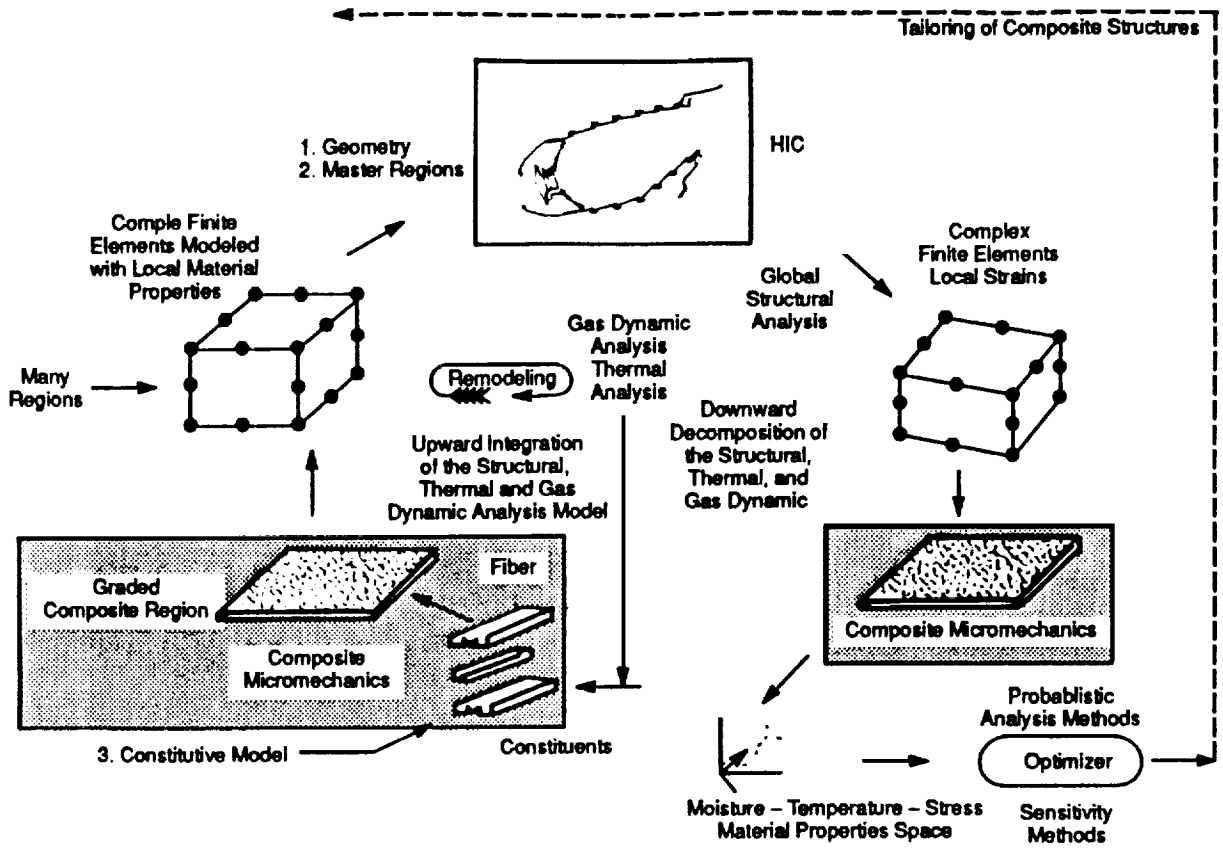
KEY NOZZLE MATERIAL REQUIREMENTS

- HIGH SPECIFIC STRENGTH
- THERMAL STABILITY
- ENVIRONMENTAL RESISTANCE
- THERMAL MECHANICAL/ACOUSTIC FATIGUE RESISTANCE
- THERMAL SHOCK/STRESS CAPABILITY
- DAMAGE TOLERANCE
- GOOD FABRICABILITY
- AFFORDABLE COST

Design projections indicate that a successful HSCT nozzle material will need to possess the noted characteristics.

HSCT PROGRAM WILL PROVIDE

- HIGH-TEMPERATURE ADVANCED COMPOSITE MATERIALS, INCLUDING FIBERS AND MATRICES
- IMPROVED PROCESSES FOR FABRICATING ADVANCED COMPOSITE COMPONENTS
- ANALYTICAL TOOLS FOR COMPOSITE AND COMPONENT DESIGN, FABRICATION, AND LIFE PREDICTION
- IMPROVED PROCEDURES FOR TESTING COMPOSITE SUBCOMPONENTS
- COMPOSITE SUBCOMPONENTS FOR ENGINE TEST
- ANALYTICAL TOOLS DEPEND ON MATERIAL CHARACTERISTICS
  - MANUFACTURING BEHAVIOR
  - FUNCTIONAL BEHAVIOR
  - FAILURE BEHAVIOR
- FAILURE MODES
- DAMAGE ACCUMULATION
- CONSTITUTIVE MODELS
  - THERMAL
  - STRUCTURAL
  - MACRO, MESO, MICRO
- LIFE MODELS
  - MACRO, MESO, MICRO
- CLOSE COUPLING BETWEEN LIFE MODELS AND CONSTITUTIVE MODELS



CSTEM OVERVIEW

"CSTEM" is the acronym for the computer program developed under the NASA contract, "Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures." The technical objectives for this program were to produce anisotropic, heterogeneous structures having high structural performance and low cost. The multiple disciplines involved are all highly nonlinear. They include anisotropic, large deformation structural analysis, anisotropic thermal analysis, acoustics, and coupled discipline tailoring. The CSTEM system is a computerized multidiscipline simulation specialized for the design problems of aeropropulsion structures. The enabling technical capabilities are implemented in a special 3D finite element formulated to tailor simultaneously the geometrical, material, loading, and environment complexities of composite structures for cost effective, optimum performance.

The CSTEM analysis begins with the constituent properties and uses a variety of composite models to obtain material characteristics for the laminate. The finite element model is then constructed and the global structural analysis carried out. The analysis is then carried down through the elements and micromechanics models to obtain local stresses and strains. Also shown are the mesomechanics models and how they are related to the CSTEM analysis system.

In each enabling technical discipline, a decoupled, stand-alone, 3D finite-element code has been developed. An executive program with controlling iterative solution techniques performs the nonlinear coupling among the participating analysis modules. Each analysis module is self contained, passing only the required input geometry and control information between the modules as well as returning any results that may be needed as input for an analysis by a following module.

The structural module uses 8-, 16-, or 20-noded isoparametric bricks and is similar to many other isoparametric finite element codes in many ways. It accommodates centrifugal, acceleration, nodal displacement, nodal force, temperature, and pressure loadings. The solution technique used is a multi-block column solver, which allows the solution of very large problems since it can work on portions of the set of equations separately.

The more advanced features of the structural module include its anisotropic, heterogeneous material capability. Material properties can be input relative to the material axes and then skewed integration point by integration point to obtain the desired orientation of the material with the global coordinate system. Material properties may also be referenced to the elemental coordinate system, with the orientation between elemental and global calculated internally. The structural module can also generate the anisotropic material properties it needs for composite materials, using the constituent properties that make up the composite. This is done using an internal adapted version of the computer program INHYD, which accesses a data bank containing the material properties of the constituents. The properties are calculated based on the volume ratios of the constituents.

- COUPLED STRUCTURAL/THERMAL/?
- COMPUTER HARDWARE/ALGORITHMS
- DESIGN OF EXPERIMENTS (TAGUCHI)
- TOO MUCH DATA-NEED EVALUATION STRATEGY/DATA BASE STRATEGY
- ANALYSIS-EXPERIMENT-ANALYSIS FEEDBACK LOOP
- HARDWARE/CODE/USER GOODNESS-OF-FIT
- PROBABILISTIC DESIGN
    - MATERIAL SYSTEM
    - STRUCTURES
- RISK ANALYSIS
    - DETERMINISTIC
    - PROBABILISTIC
- STRUCTURAL TAILORING AND OPTIMIZATION
    - MULTIDISCIPLINE
    - MULTIOBJECTIVE

HIGH SPEED CIVIL TRANSPORT

A

NEW

LEVEL

OF

COOPERATION

AMONG

GOVERNMENT/INDUSTRY/UNIVERSITIES

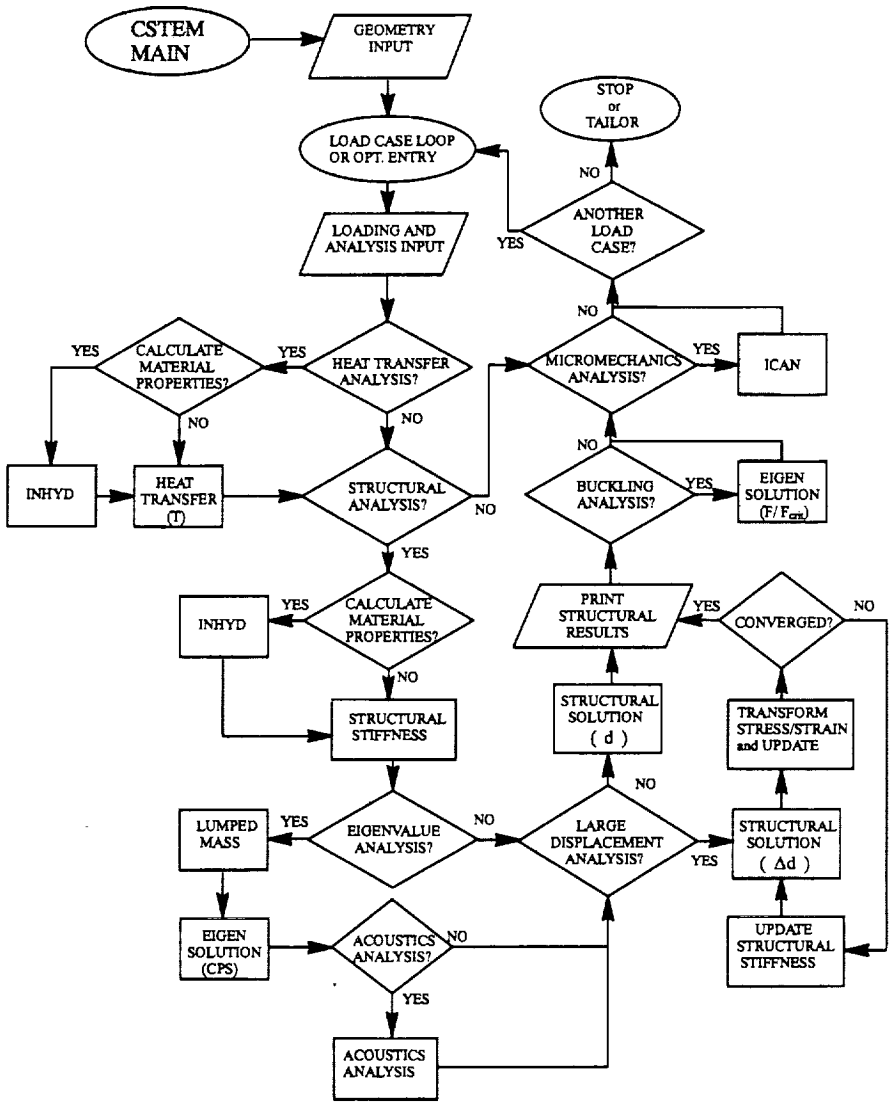


FIGURE 1. CSTEM FLOWCHART

Flowchart of the major analysis modules of CSTEM. These modules are used as a stand-alone analysis package with entry through the main executive routine, or as the analysis portion of the tailoring process - in which case the entry to these modules is at the load case level.

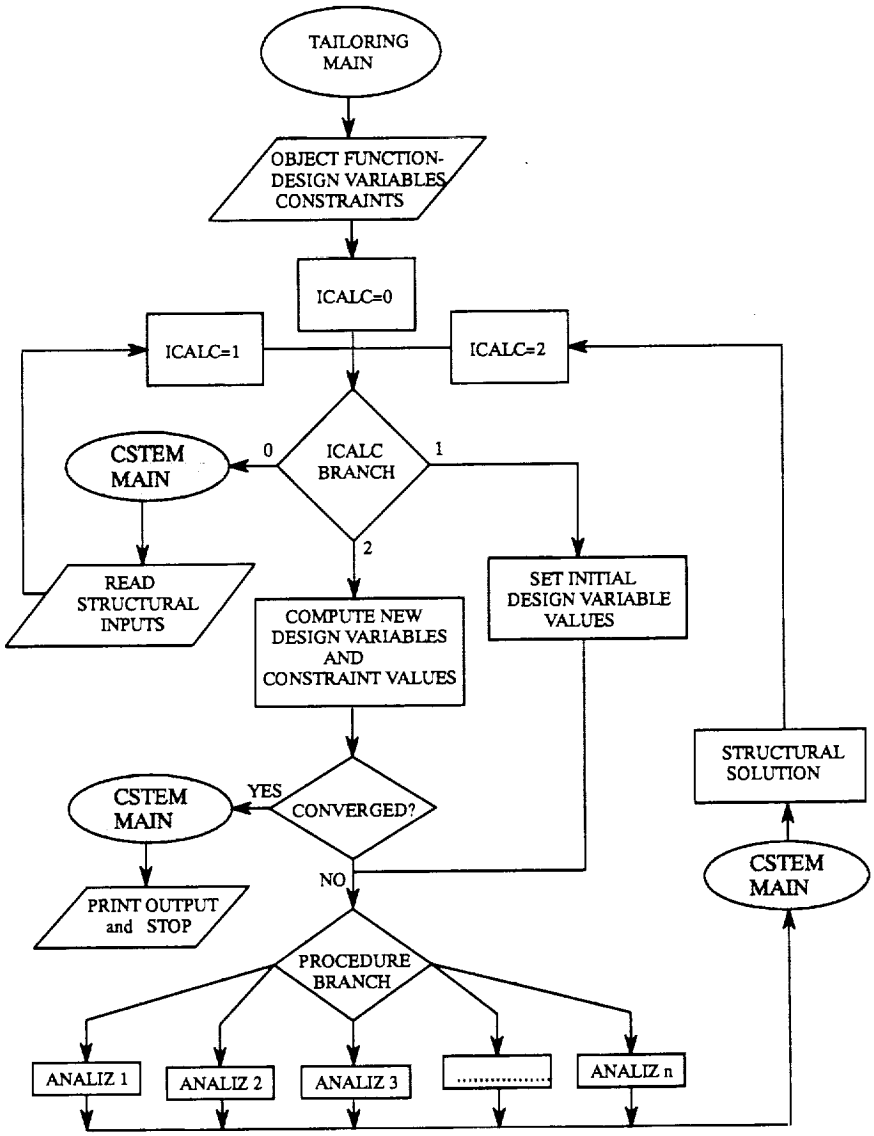
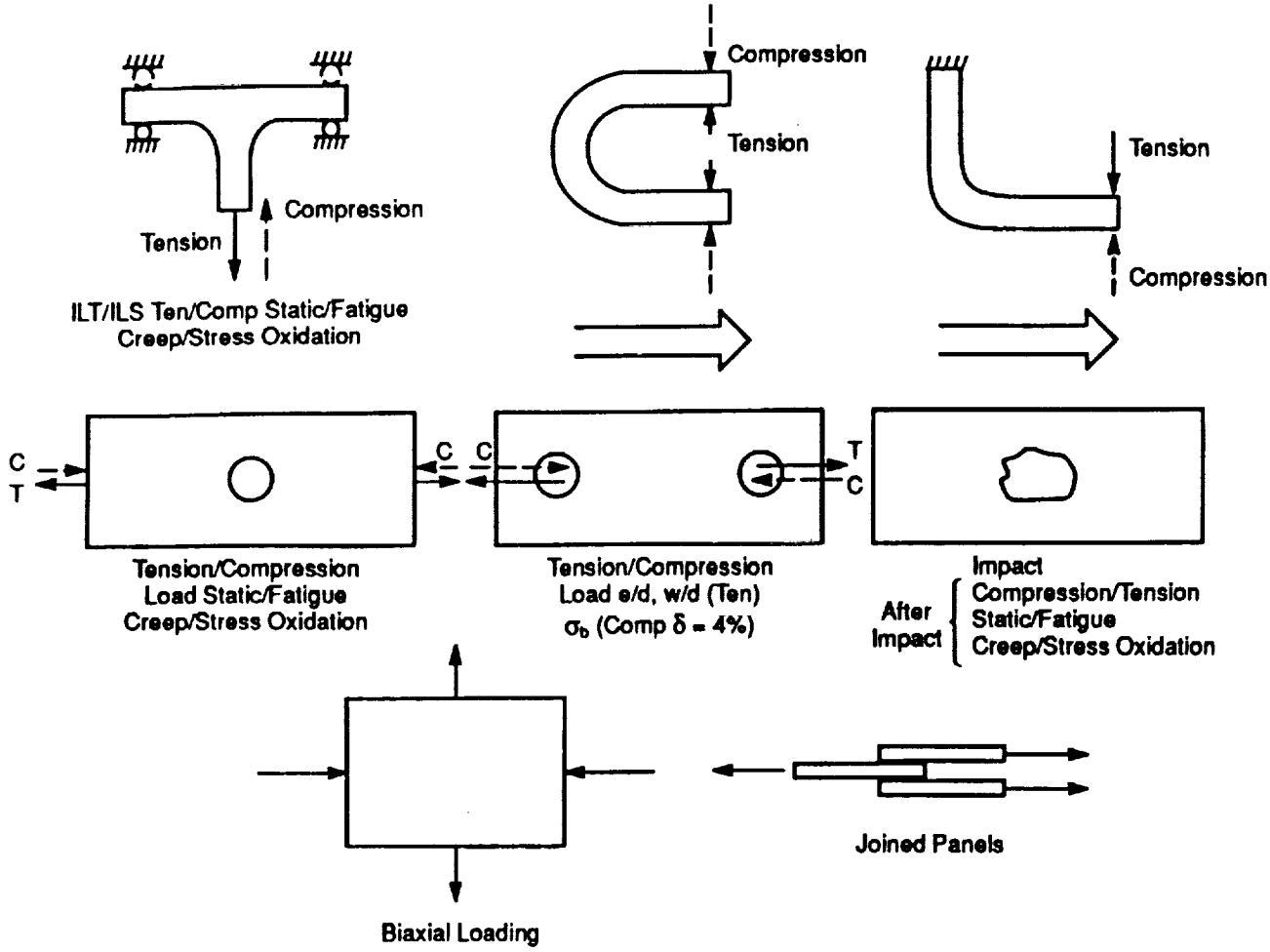


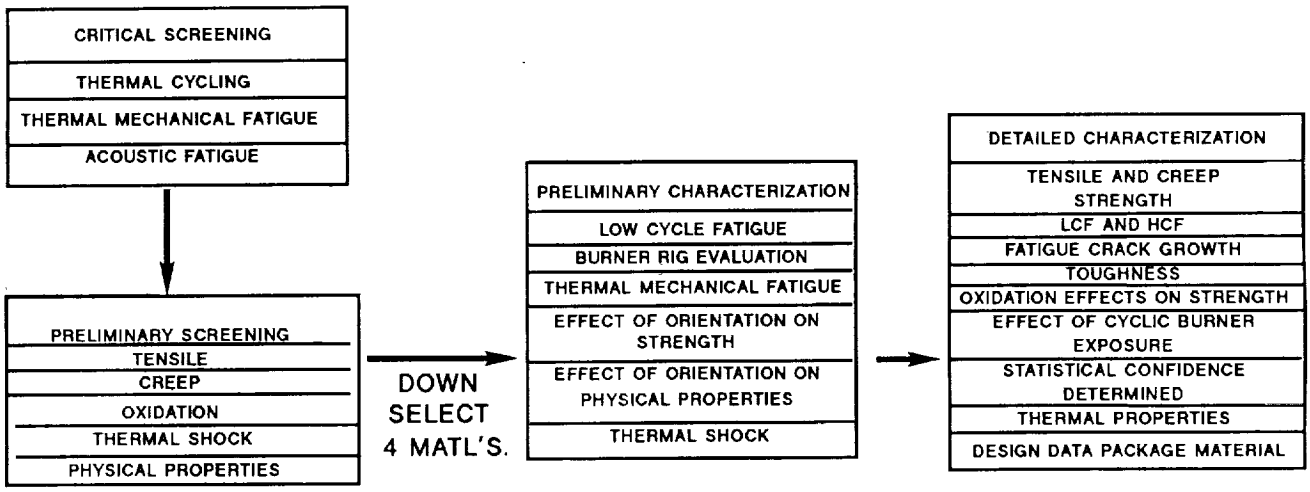
FIGURE 2. TAILORING PROCESS FLOWCHART

CSTEM tailoring capability has been built on the STAEBL computer program obtained from NASA Lewis. This program consists of two major modules: CONMIN, which performs the actual tailoring, and ANALIZ, which supplies the parameters to be tailored. The CONMIN module was abstracted from STAEBL and coupled with the CSTEM structural, thermal, and acoustic analysis modules.

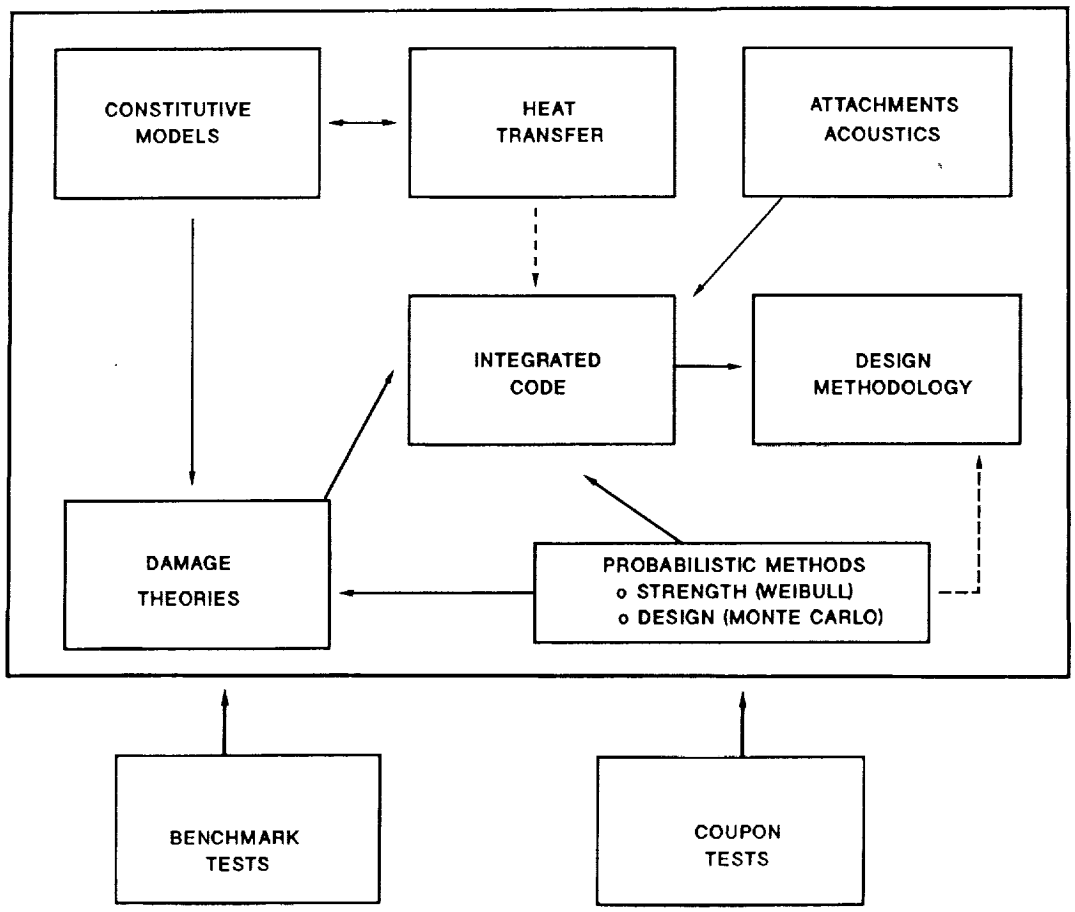


Validation of the analytical models and design methodology is an important part of the HSCT process. Validation will be required for the static, creep, stress oxidation and fatigue arenas. Configuration shaped laboratory specimens will be designed to be representative of generic component shapes. When tested these will account for fabrication difficulties and stress or failure mode interactions not represented in simpler laboratory specimens.

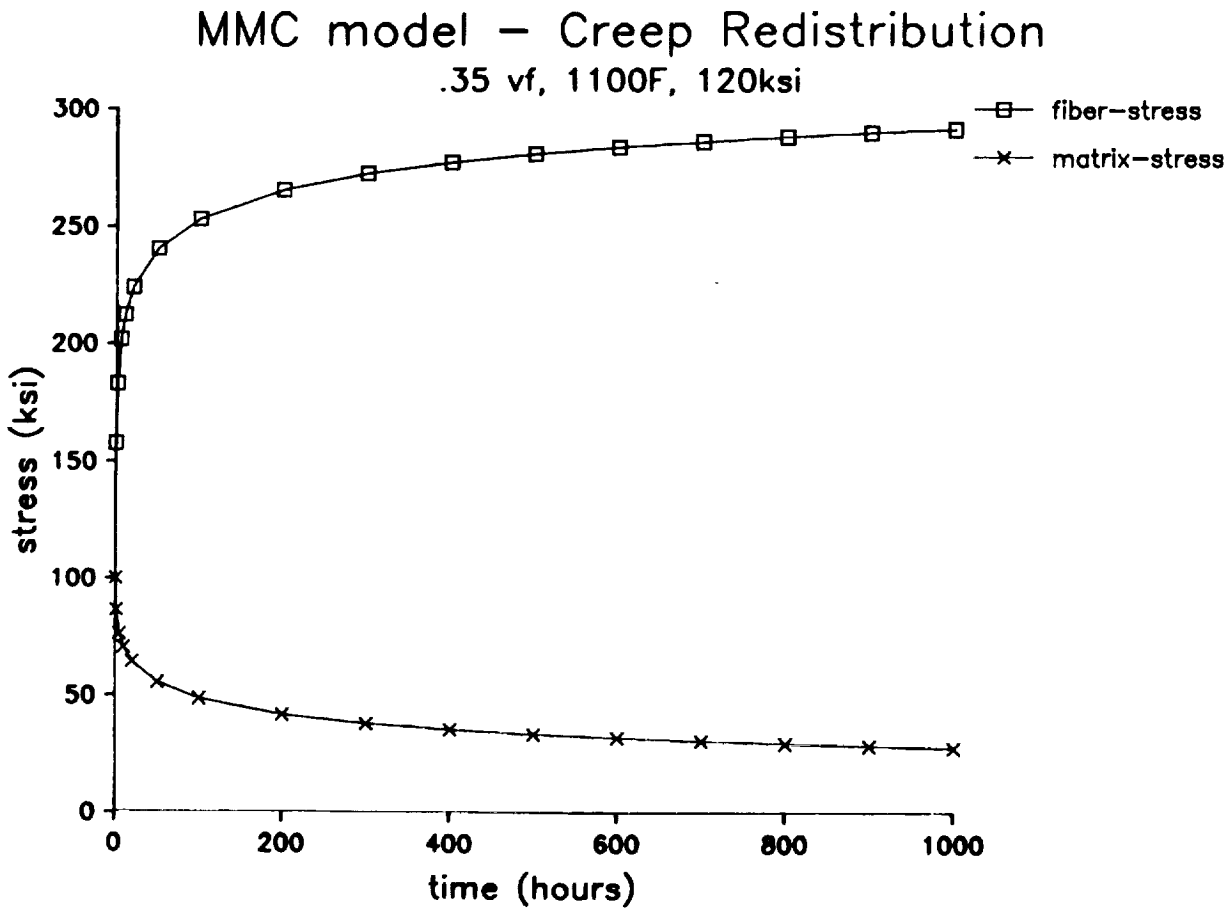
C-3



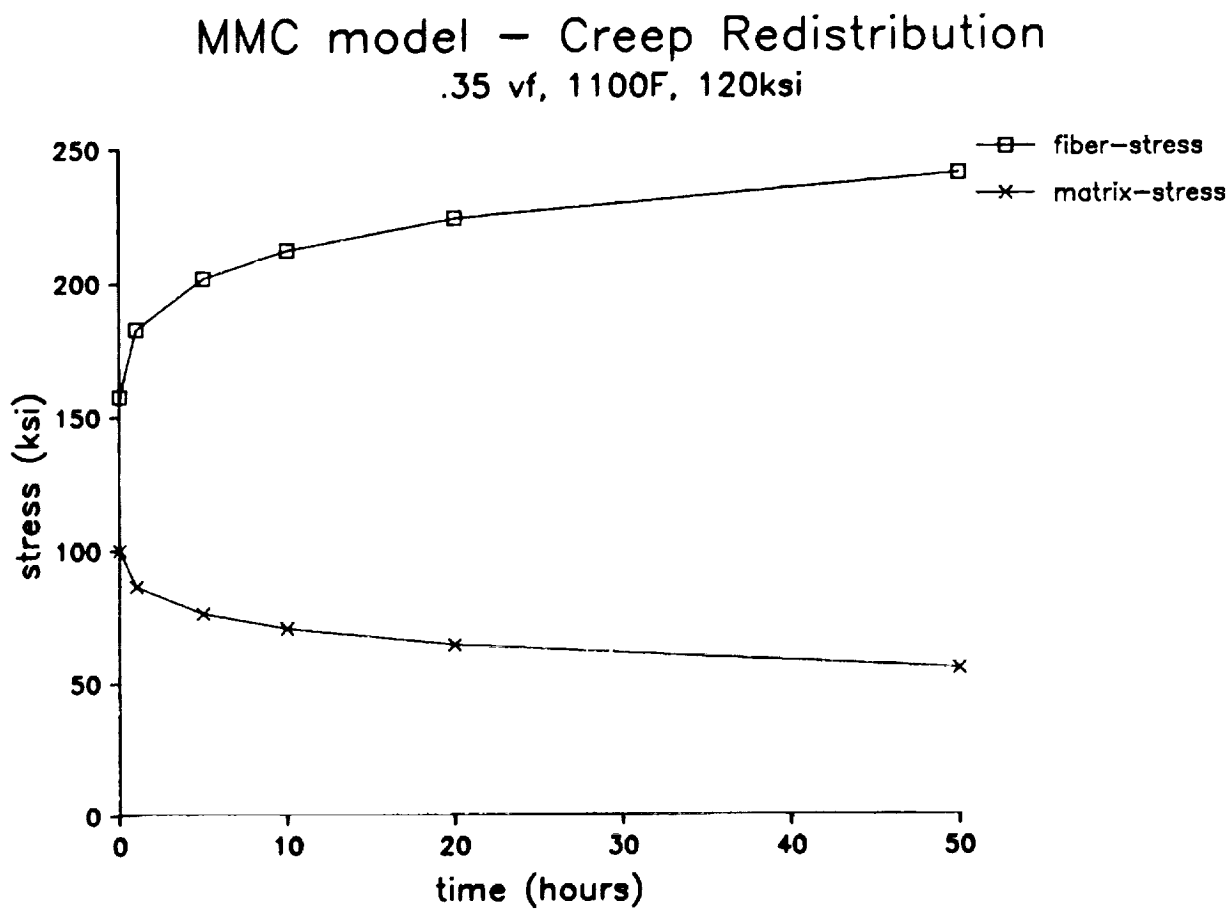
A detailed evaluation test plan for the candidate HSCT materials will be developed. It will include mechanical and physical testing, characterization techniques for environmental and thermal behavior, and candidate NDE techniques.



The theoretical models and analytical tools developed in the HSCT program will be integrated to provide an overall design methodology and life prediction system with a firm base established by the coupon and benchmark tests.



A creep simulation of an MMC unit cell utilizing an overlay model in a nonlinear finite element code. Stabilization has not occurred after 50 hours.



A creep simulation of an MMC unit cell utilizing an overlay model in a nonlinear finite element code. Stabilization is not totally complete after 1000 hours.

N92-25920

Structures Technology Applications for the National Aerospace Plane

T. E. Little
Pratt & Whitney
West Palm Beach, Florida

NASP Design Requirements Challenge Current Structure Technologies

Achieving the National Aerospace Plane (NASP) operational objectives of Mach 25 and single stage to orbit (SSTO) will subject the vehicle to extreme loading conditions and will require large, actively cooled structures while meeting very stringent weight goals.

- Mach 25
- Single stage to orbit (SSTO)

Loading

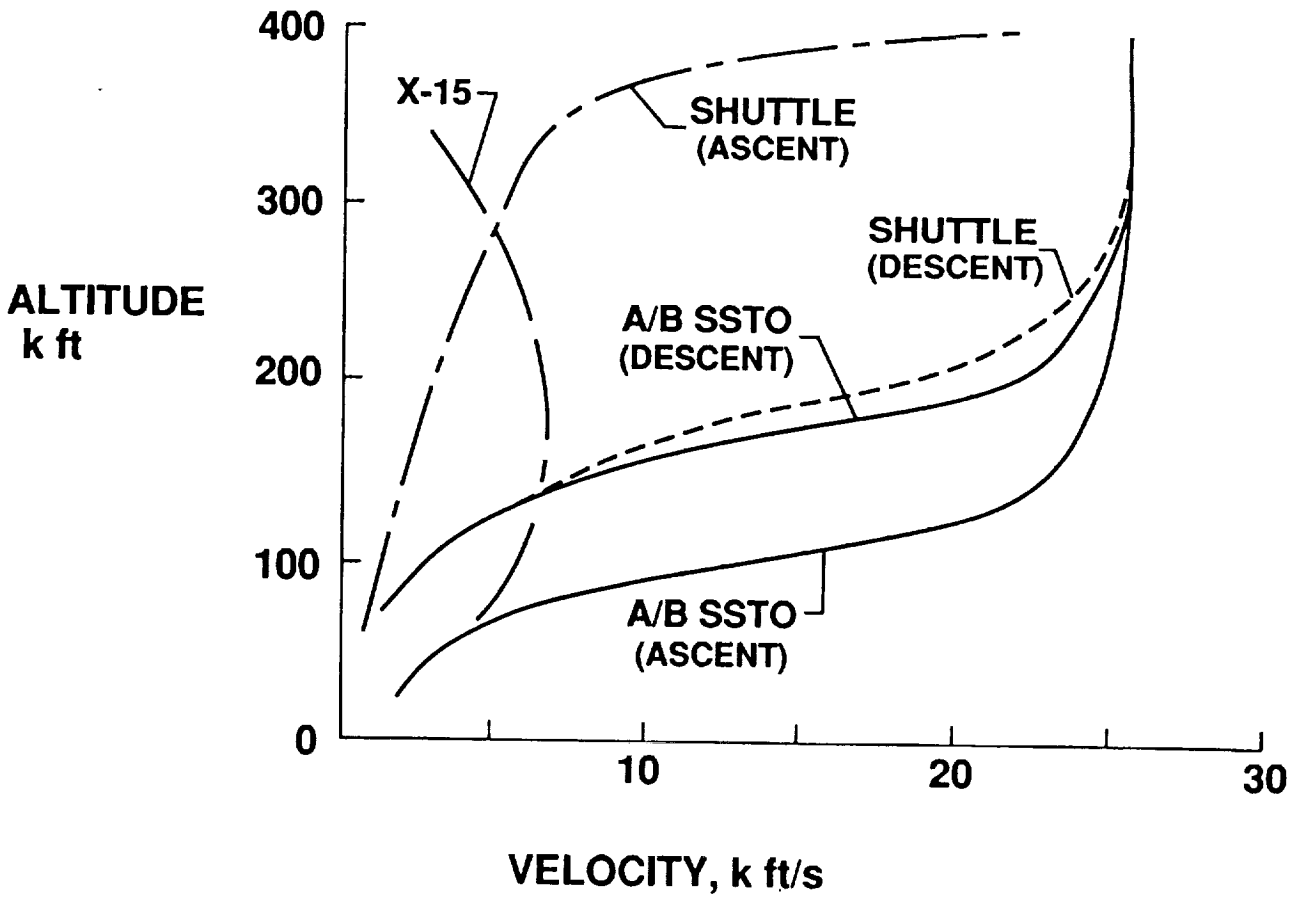
- High temperatures
- High acoustic loads
- High pressures
- Shock interactions
- Aerodynamic loads

Configuration

- Actively cooled structure
- Large panels
- Minimum weight

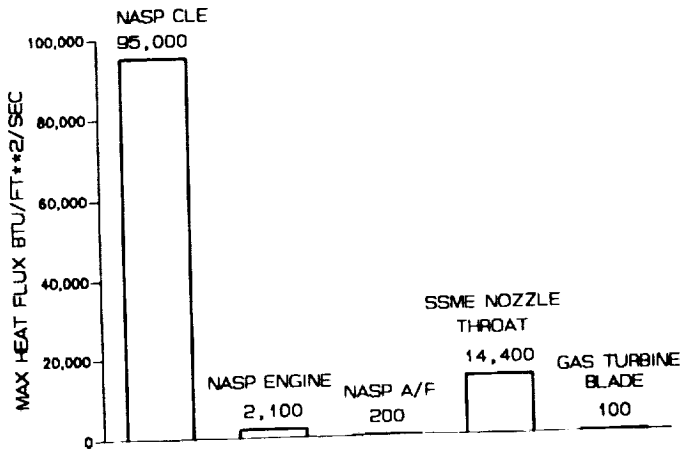
Trajectories For Hypersonic Vehicles

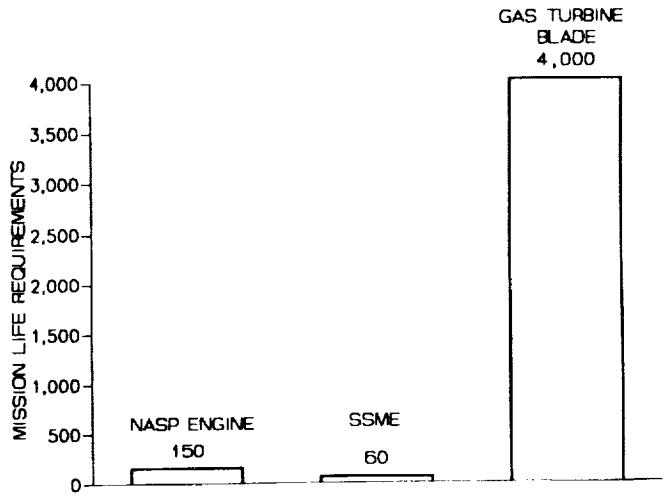
The NASP is an air breathing, single stage to orbit vehicle (A/B SSTO) which is to take off, achieve orbit and land under its own power. Typical trajectories for the X15 experimental aircraft, the Space Shuttle and NASP show that NASP will achieve much higher velocities at lower altitudes.



Heat Flux/Life Comparison

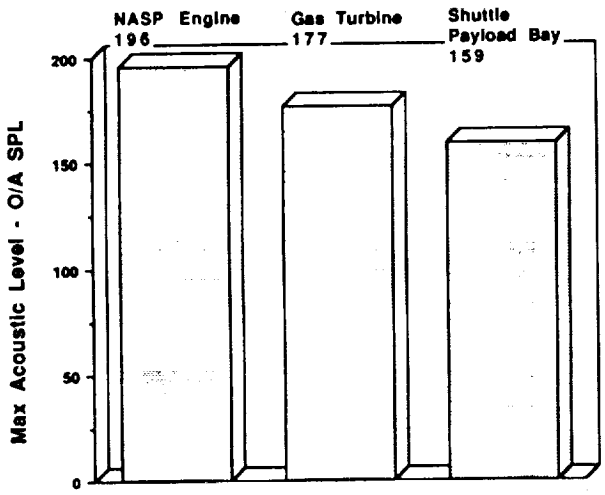
The combination of very high speed at relatively low altitudes causes the NASP airframe to experience heat fluxes greater than current gas turbine blades. The Cowl Leading Edge (CLE) is subjected to extremely high heat fluxes from the convergence of different shock waves on a very small position of the leading edge. Fortunately, the NASP design life requirement is "only" 150 cycles compared to 4000 cycles for a gas turbine engine. However, the cyclic application of such extreme thermal loads presents a very challenging structural problem.

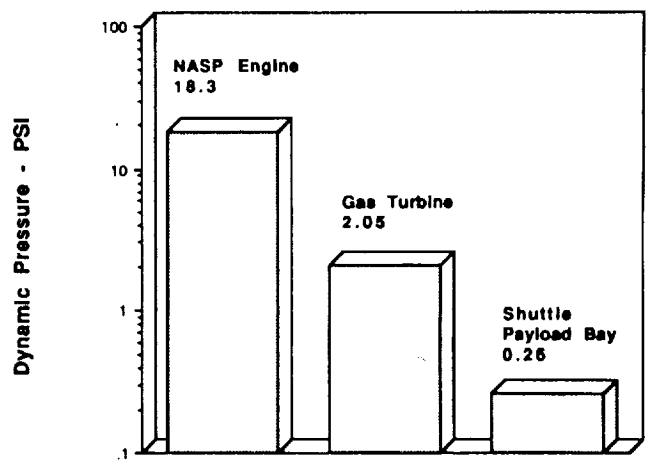




Engine Acoustic Load Comparison

Predicted acoustic levels are much greater than for a gas turbine engine. These acoustic loads are a very significant consideration in the NASP design. Dynamic pressure levels have been estimated based on the percentage of static pressure fluctuation in the combustor permissible for continued operation.



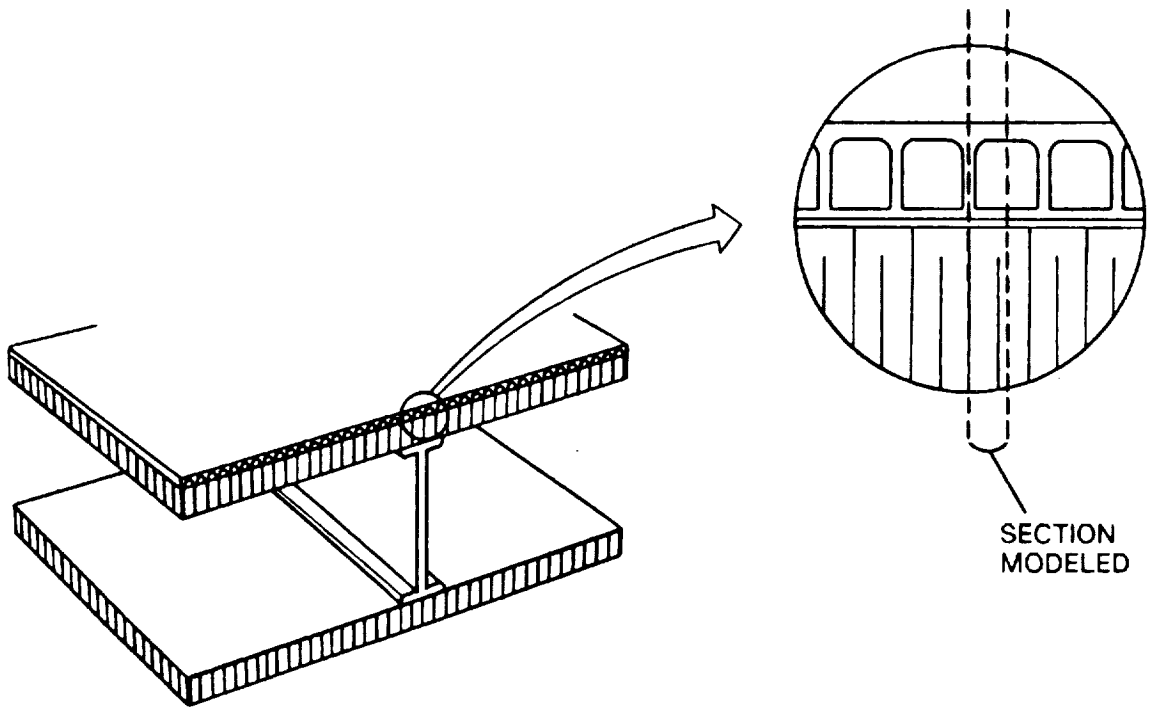


Aluminum 6061-T6 HEX Analysis

Three structural problems that have been addressed at Pratt & Whitney give insight into the magnitude of the problems that NASP presents and an idea of the approaches required for their solutions.

The first of the problems is the Heat Exchanger (HEX) analysis. The extremely high temperatures require large actively cooled structures. The integral heat exchanger (HEX) depicted is one approach to providing lightweight structural members that can withstand high surface temperatures. The key to the operation is the flowing of the coolant through channels at the hot surface and integrally supporting the cool side of the structure, in this case with honeycomb.

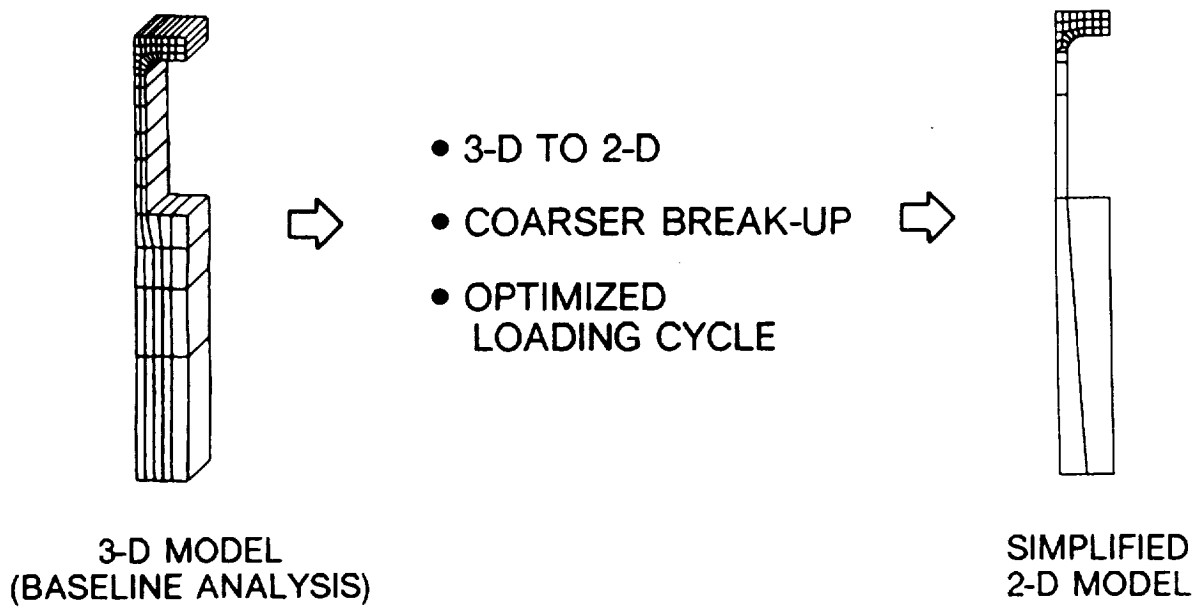
- **Analysis is for bodyside inlet integral HEX**



Simplified 2-D MARC Model

The heat exchanger is subjected to severe thermal gradients and high pressure loads which result in a many-facetted structural problem. Because of the relatively large deformation of the hot surface, a large-displacement, inelastic analysis was required. To achieve the necessary accuracy for the cyclic creep-ratcheting, the geometry had to be updated for each load increment in the solution. The original finite element model generated for the heat exchanger was a 3-D NASTRAN model which proved to be too costly for design iteration purposes. Subsequently, a 2-D analysis was generated which could be used for design. The model was verified by comparing it to the earlier 3-D results.

• Simplified model facilitates multi-cycle analysis



2-D Model Verification

Comparisons with the 3-D model verified that the 2-D model had sufficient accuracy for design purposes at greatly reduced costs.

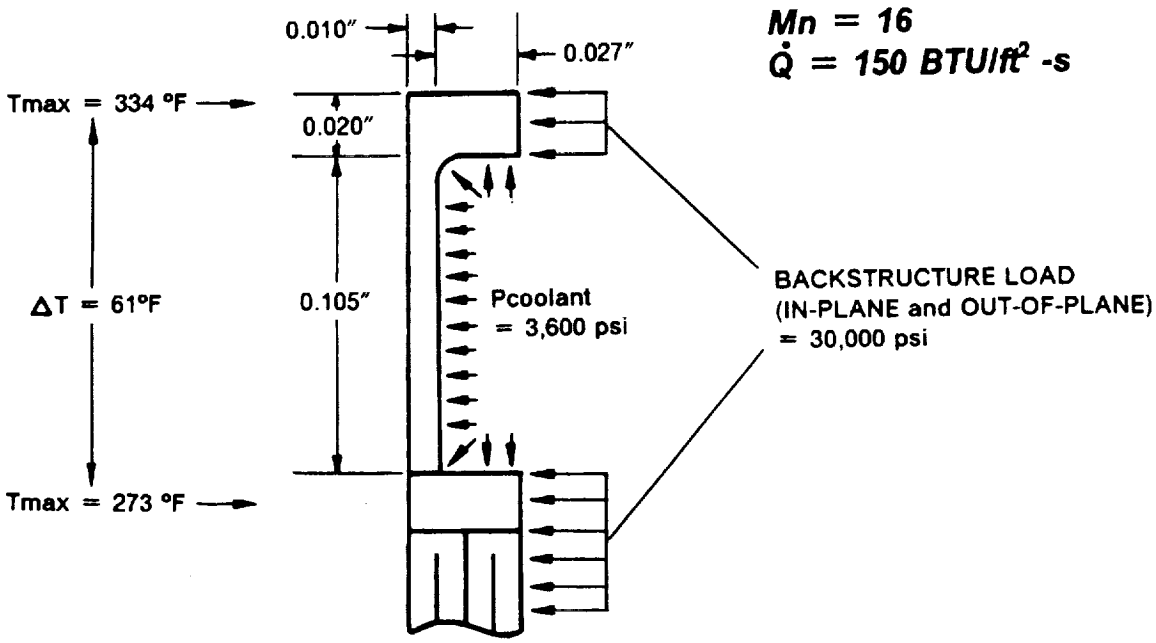
- **Simplified model is accurate and less costly**

1st LOAD CYCLE	BASELINE 3-D MODEL	SIMPLIFIED 2-D MODEL	DIFFERENCE
Max. Equiv. Von Mises Stress	53.990 ksi	54.040 ksi	0.1 %
Max. Equiv. Plastic Strain	0.8752 %	0.8889 %	1.6 %
Max. Equiv. Creep Strain	0.0730 %	0.0745 %	2.1 %
Degrees of Freedom	1695	253	Factor of 6.7
CPU Time (Cray)	75 min.	5 min.	Factor of 15

Aluminum 6061-T6 HEX Analysis

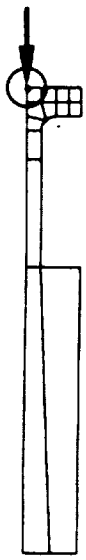
The loading conditions are shown for the MACH 16 flight condition that was determined to be the worst case for the HEX. The "average" mission used for the load cycle included an upload from start-up to 100% load, time for creep/dwell, and a download to shutdown.

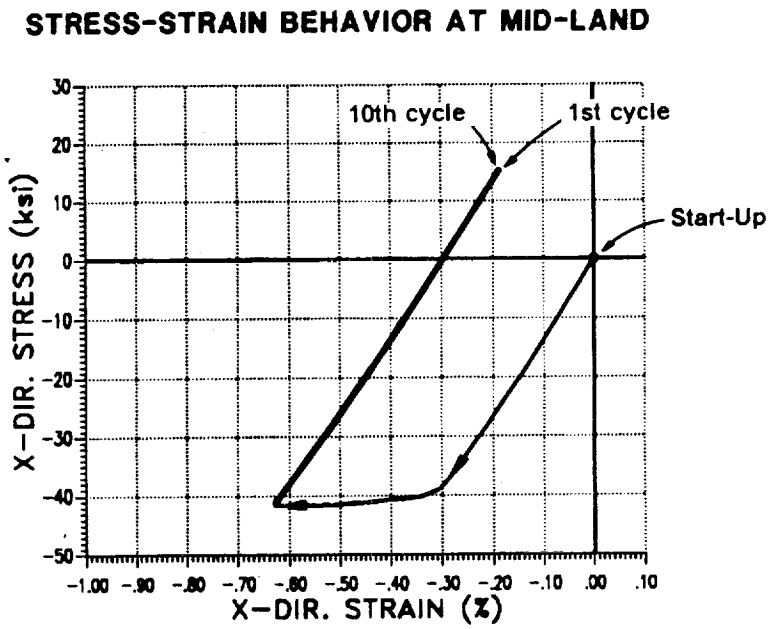
• Model simulates real conditions



The Aluminum HEX exhibited very little creep-ratcheting. This was not true for some more exotic materials that were analyzed. The stress-strain behavior becomes elastic in nature after the first complete cycle. This fact will make life predictions far more reliable.

- **Hot wall exhibits little creep-ratcheting**
- STRESS-STRAIN BEHAVIOR IS ELASTIC AFTER 1st CYCLE...



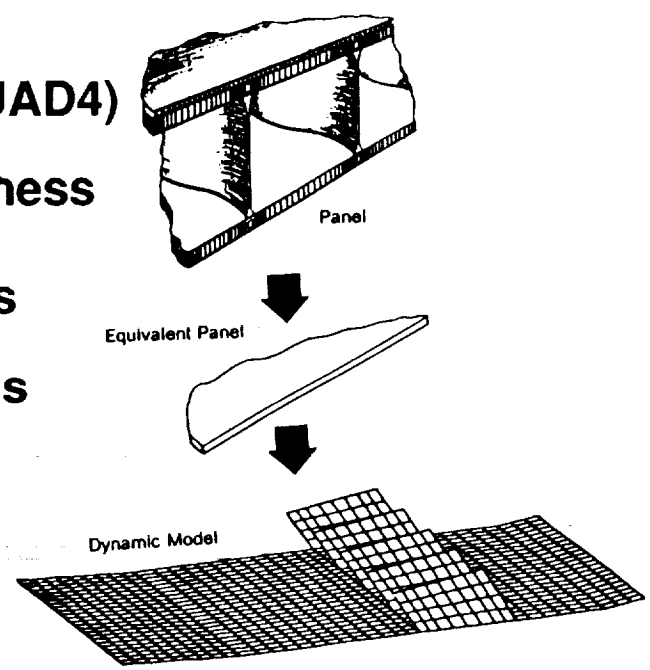


Dynamic Model Generator

The second problem to be discussed is the one of large panel flutter. The panel flutter analysis included using piston theory for the hypersonic flow regions and low supersonic wave theory in the difficult transonic regions. NASTRAN analyses were used to obtain natural frequencies and modeshapes. The problem of flutter for the NASP is complicated by the fact that, unlike a wing, there are great differences in airflow on either side of the panel. Across the cowl, the internal flow may be Mach 1 or lower while Mach 3 or higher on the outer surface.

The NASTRAN model simulated the actual component's honeycomb construction with "equivalent" panels that were modeled as laminated composite plates. The "equivalent" panels were non-symmetric with equivalent mass and stiffness and thus adequately modeled the structure at a much reduced cost in manpower and computing than would have been required for exact models.

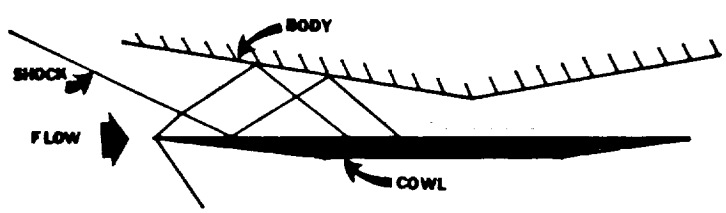
- **Built-up structure is modeled as a laminated composite**
  - **Quadrilateral plates (QUAD4)**
  - **Equivalent mass & stiffness**
  - **Non-symmetric sections**
  - **Equivalent thermal loads**
  - **User friendly**



Flutter Codes

The hypersonic/supersonic flutter code used for the analysis was developed at United Technologies Research Center (UTRC). The code uses the low supersonic wave theory in the Mach 1 to Mach 3 range because it is more accurate than the piston theory in that range. The code addresses unsteady wave actions and the variation in flow between the two boundaries. The piston theory is used for higher Mach numbers. The code accounts for the differences in flows between the top and bottom surfaces and addresses shock motion effects with a quasi-steady state analysis.

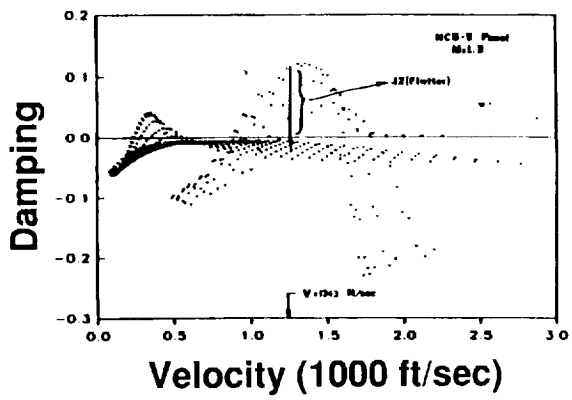
- **Low supersonic code is more accurate for Mach 1 to 3 range**
  - **Full unsteady wave model**
  - **Different top & bottom surface flows**
  - **Nastran modeshapes and frequencies of equivalent dynamic model**
  - **Automated stability (V-g) plots**

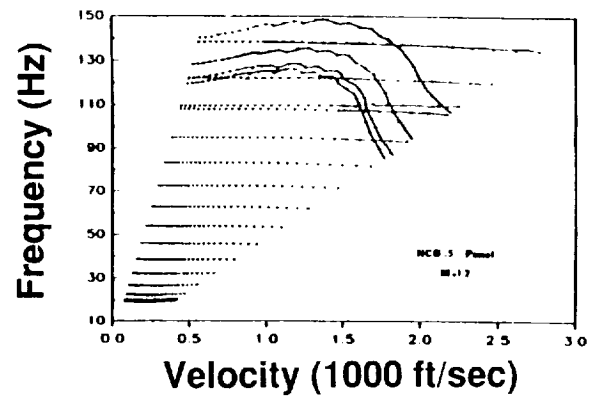


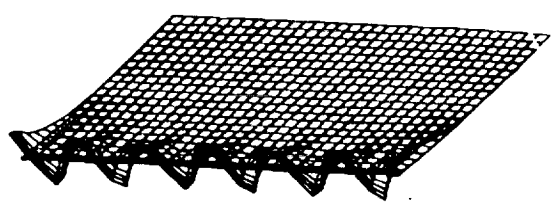
Typical Results

The hypersonic/supersonic flutter code provides automated output of the NASTRAN mode shapes, a damping vs. velocity plot and a frequency vs. velocity plot with which to evaluate the results.

- **Damping and velocity margins are required to satisfy stability**







NASTRAN Flutter Mode

Conceptual Design Analysis

The third analysis example is for preliminary design of complex structures to achieve minimum weight. Because weight is extremely critical to the NASP, complex analyses were performed in the conceptual and preliminary design phases to achieve the lightest possible designs. There are two basic traditional approaches to lightweight structures in the early phase of design. If similar vehicles have been designed previously, extrapolations can be made or, if enough experience is available, weight estimating algorithms can be developed. It is difficult to apply these two methods to the NASP since the design data base for a hypersonic vehicle is almost non-existent and the loads and failure modes expected for the NASP are unique.

- **Accurate weights are required for vehicle performance calculations**
- **Traditional methods:**
  - **Extrapolation from existing data**
  - **Weight estimation algorithms**

ST - Size Structural Sizing Code

There are many requirements to adequately address the problem of minimizing weight in complex structures. The primary requirements are:

- o Computation of accurate structural forces
- o Multiple and mechanical thermal loads
- o Temperature dependent material properties
- o Complex Structures
- o Multiple failure modes
- o Weight minimization methodology
- o Rapid computational methods

The ST-SIZE Structural Sizing Code is being developed at NASA Langley to address these requirements. ST-SIZE is based on a NASTRAN finite element analysis and iterates element force and stiffness to obtain a fully stressed solution.

- **Finite element (NASTRAN) based**
  - **Element force and stiffness iteration**
  - **Mechanical and thermal loads**
  - **Temperature dependent material properties**
  - **Composite/isotropic material capability**

ST-SIZE Structural Sizing Code

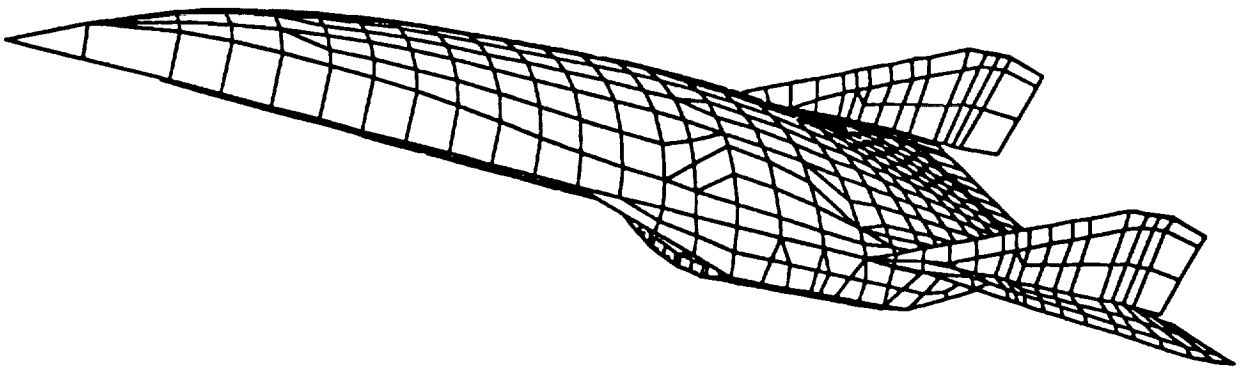
ST-SIZE iterates to a fully structural design while considering multiple failure modes. A "fully stressed" design has each element of the structure loaded to its maximum when compared to the possible failure modes. The code does not account for the effects of one element stiffness change on the loads in other elements. Several standard structural element models are built into the code thus greatly reducing the user's effort.

- **Separate stiffness decomposition for each thermal load case**
- **Iterates to a fully stressed design**
- **Multiple failure modes including strength, stability, and buckling checks**
- **Structural section library with several panel and beam types**

Typical Hypersonic Vehicle FEM

A typical ST-SIZE model is shown for a NASP vehicle. Output can be plotted to graphically show the controlling load case, controlling failure mode and panel unit weights as well as standard stress, load and temperature distributions for each element in the model.

1037 Elements
4644 DOF



Future Requirements For NASP Structures

As NASP transitions from the demonstrator preliminary design to final design, increasingly complex structural problems must be addressed and solved. Two requirements that typify the future of NASP structures are lifing methodology and the incorporation of statistical methods. The life system of the future must address thermally driven, inelastic stresses with high levels of vibration superimposed, consider multiple failure modes for complex, redundant structures and be accurate in the relatively low life ranges.

- **Life methodology**
  - **Inelastic steady stresses**
  - **Composite materials**
  - **Multiple failure modes**
  - **Superimposed vibrations**
  - **Redundant structures**
  - **Accurate at low lives**

The incorporation of statistical methods into the design process is required to achieve NASP goals. Addressing multiple failure with standard deterministic (single valued) analysis is inherently conservative and will lead to excessive weight. The use of probabilistic design methods in which the tolerances for the design drivers are addressed will yield a design that meets acceptable risk requirements.

- **Probabilistic design methodologies**
  - **Multiple failure modes**
  - **Composite and low ductility materials**
  - **Redundant structures**
  - **NDE capabilities for large structures**
  - **Large computing capacity required**

Summary

The NASP presents a unique set of very complex structural problems that challenge our computational capabilities. Complex analyses are required in the conceptual design phase to achieve sufficient accuracy to address the extreme load conditions and to adequately evaluate vehicle weight. The computational capability must be available to perform these analyses in a rapid manner to accommodate the design process.

- **NASP is a unique set of complex structural problems**
- **High fidelity analyses required in conceptual design phase due to weight criticality**
- **Rapid, user-friendly computational methods are necessary to accommodate evolutionary design**

N92-25921

Large Scale Optimization Using Astros An Overview

Vipperla B. Venkayya
Wright Laboratory
Wright Patterson Air Force Base, Ohio

In aircraft design, as in any other design, there are at least three or four stages. The conceptual design of an aircraft basically involves the mission, performance and aerodynamic shape. The payload, range and the expected maneuvers are the primary considerations in the mission definition. It is possible (and desirable) to use optimization quite effectively in conceptual design. The number of variables are generally few, and the constraints are usually defined by simple empirical relations. Also there are many optimization packages available for trade studies. However, since the analysis is grossly approximate, the performance predictions can at best be gross estimates. In conceptual design there is little or no definition of the structure to determine the flexibility effects. Only shape parameters such as the wing surface area, aspect ratio, sweep, chord to thickness ratio and propulsion requirements are identified in conceptual design. The aerodynamic conceptual design is followed by a definition of structural concepts in which the type of construction such as stiffened panel or sandwich, or the layout of spars and ribs are decided. At this stage there is enough definition of the aerodynamic planform as well as the internal structure to make a finite model for dynamic analysis. Design with finite element models is considered as preliminary design. Optimization in preliminary design is more productive, because performance predictions are more reliable. The detail design comes after preliminary design, and it involves stress models which include details of local effects such as cutouts and other discontinuities. The final design is the design that is actually scheduled to be built.

Elements of Design

Conceptual Design

⇓

Preliminary Design

⇓

Detail Design

⇓

Final Design

As stated earlier in preliminary design geometry details and candidate materials are assumed to be known (from the conceptual design).

Preliminary Design

Know

- Geometry (S_w, **AR**, Λ, t, *etc.*)
- Structural Concept (# spars, ribs, stiffeners, *etc.*)
- Materials
- Flight Conditions

The most difficult part of aircraft design is the definition of its operating environment. An aircraft is a highly dynamic system, and it is subjected to many types of accelerations and aerodynamic forces. A typical fighter aircraft mission includes takeoff, pullout, roll, landing and any combination of these, and these maneuvers induce high inertia and aerodynamic forces. Accurate determination of these forces is particularly important when design tolerances are reduced by optimization.

Flight Conditions

Meet certain maneuver requirements

- take-off
- pullout
- roll
- landing

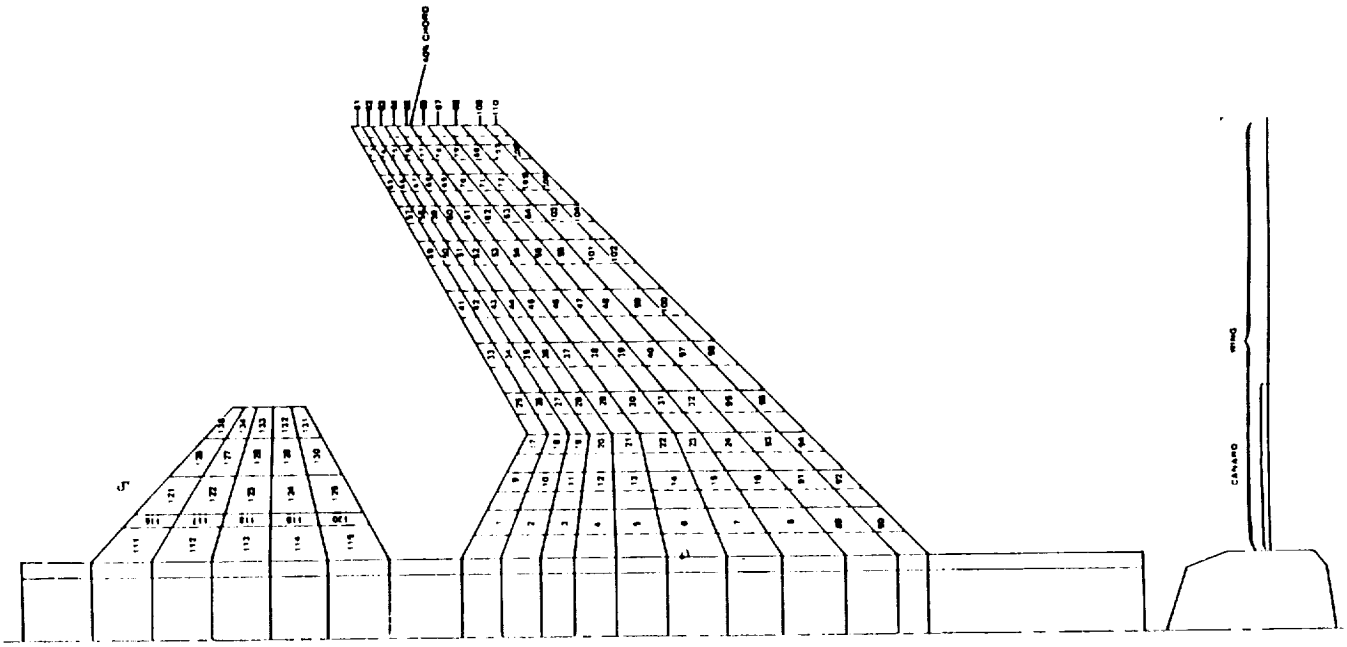
$\Downarrow$

Loads

- Inertia (ω, $\dot{\omega}$, a, n)
- Aerodynamic (M, α, q)

Aerodynamic panel methods are the mainstay of the current design practice for determining static airloads. These panel methods are based on steady aerodynamic theories. Doublet lattice, piston theory or strip theory are considered adequate for subsonic flight while some variations of these can be used for supersonic conditions.

Aerodynamic Panel Model



In general the aerodynamic forces are computed first excluding the flexibility effects, and then aeroelastic corrections are made using finite element models. In modern fighter aircraft the flexibility effects can be quite significant (30 to 40% change from the rigid airloads).

Aerodynamic Loads

Rigid Air Loads

$\Downarrow$

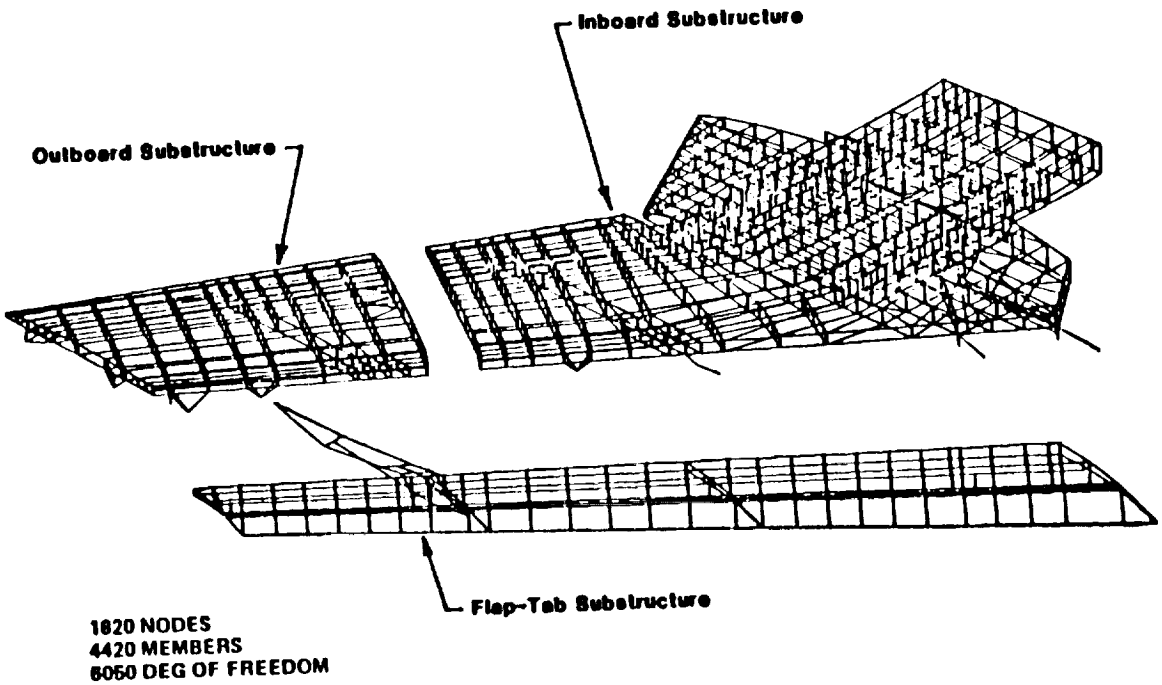
Make aeroelastic corrections

$\Downarrow$

Aerodynamic Lift Forces
(Static Air Loads)

A typical finite element model of a forward swept wing airplane is illustrated.

WING SUBSTRUCTURE



Having discussed the analysis models, it is appropriate to address the optimization model. An optimization problem involves three basic elements:

1. Definition of a performance function to be minimized or maximized.
2. Identification of a set of design variables that affect the performance.
3. Definition of the constraints which also depend on the same design variables.

Optimization Problem

Determine the *design variables* so as to minimize the weight of the structure, such that the behavorial/performance requirements (*constraints*) are met.

Design Variables

- skin, spar, & rib thicknesses
- spar cap, rib cap, & post areas
- composite fiber orientation

Constraints

- Strength
- Stiffness (Deflections)
- Natural Frequency
- Flutter speed
- Divergence speed
- Lift-curve slope
- Control-surface effectiveness

Optimization Problem

Minimize the objective function (weight), $\phi(X)$

Subject to the constraints, $g = \psi_i - \bar{\psi}_i \leq 0$

$\sigma_i \leq \bar{\sigma}_i$	allowable stresses
$u_j \leq \bar{u}_j$	allowable deflections
$\omega \geq \bar{\omega}$	minimum fundamental frequency
$V_f \geq \bar{V}_f$	flutter speed
$V_{div} \geq \bar{V}_{div}$	divergence speed
$\frac{dC_L}{d\alpha} \geq \bar{C}$	lift-curve slope

Having identified the performance measure (such as weight), the design variables and the constraints, a formal statement of the optimization problem is given in this slide.

$F(\underset{\sim}{x}) \rightarrow$ Performance Function

$z_j(\underset{\sim}{x}) \rightarrow$ Constraint Functions

$x_1, x_2, \ldots, x_n \rightarrow$ Design Variables

Minimize or maximize a function $F(\underset{\sim}{x})$

$$F(\underset{\sim}{x}) = F(x_1, x_2, \ldots x_n)$$

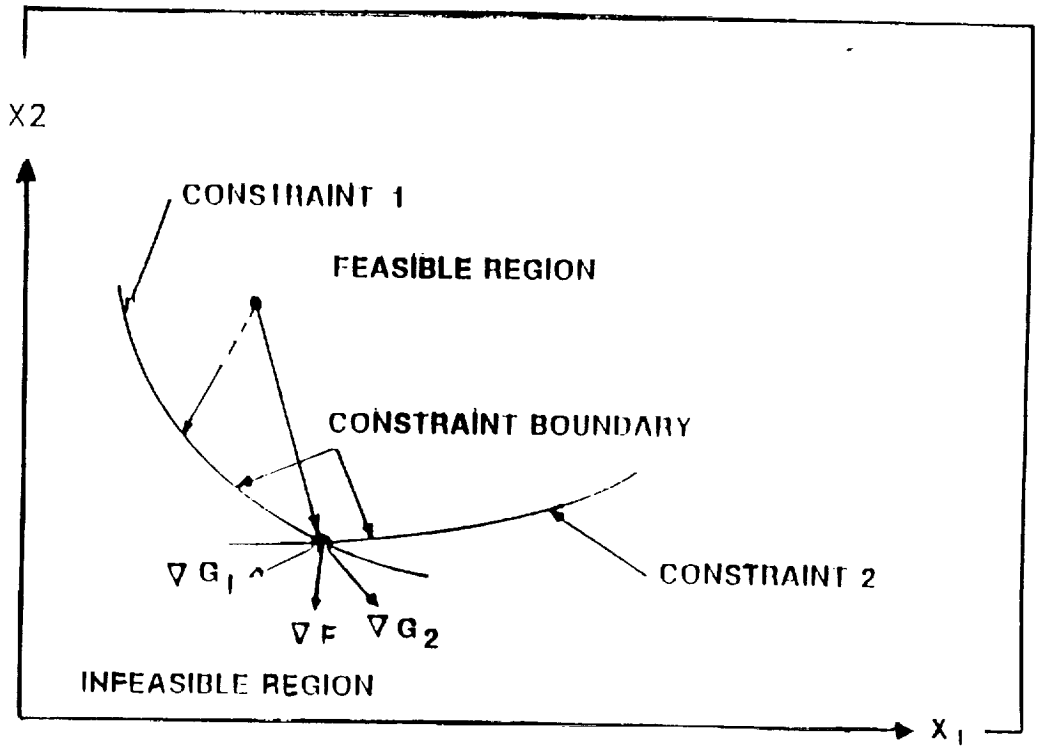
subject to the inequality constraints

$$z_j(x_1, x_2 \ldots x_n) \leq \bar{z}_j \qquad j = 1, 2 \ldots k$$

and equality constraints

$$z_j(x_1, x_2 \ldots x_n) = \bar{z}_j \qquad j = k+1, \ldots s$$

A two variable design space is illustrated with acceptable (feasible) and unacceptable (infeasible) regions separated by the constraint lines. The gradients of the objective function (performance) and the constraint functions are designated by ∇F and ∇G respectively. These gradients provide search information for locating the optimum (best) design.



The key elements of the preliminary design of an aircraft are illustrated in this schematic (block) diagram. The three main areas identified by dotted line boxes are:

1. Loads determination
   - Inertia loads (mass properties)
   - Steady Airloads

2. Structural model-optimization
   - Analysis
   - Sensitivity
   - Optimization

3. Aeroservoelasticity
   - Aeroelastic instabilities
   - Controls

Two key transformations (load and displacement transformations) often contribute to singularities due to improperly defined interpolations (spline or other interpolations).

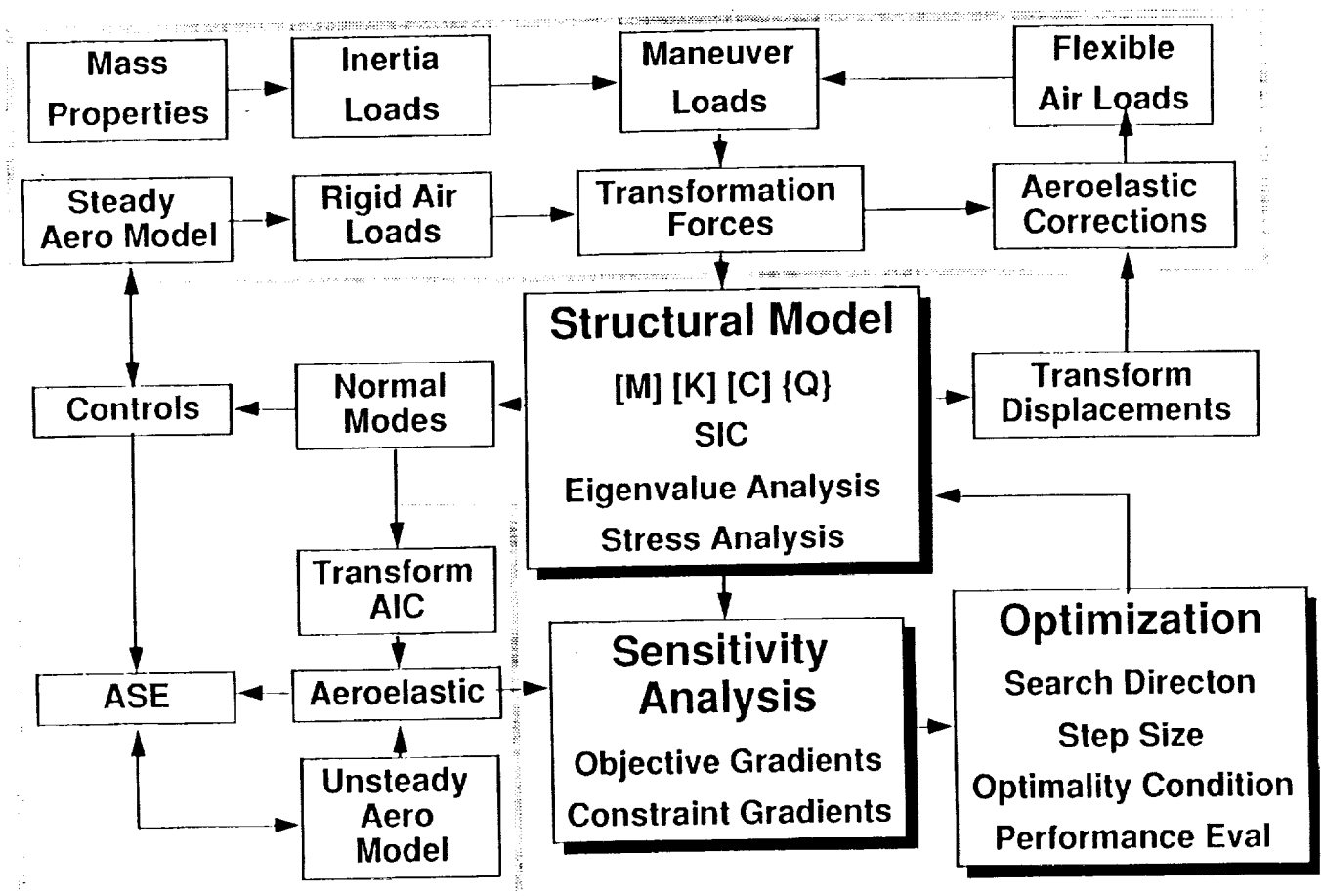


Fig 1 Interdisciplinary Design

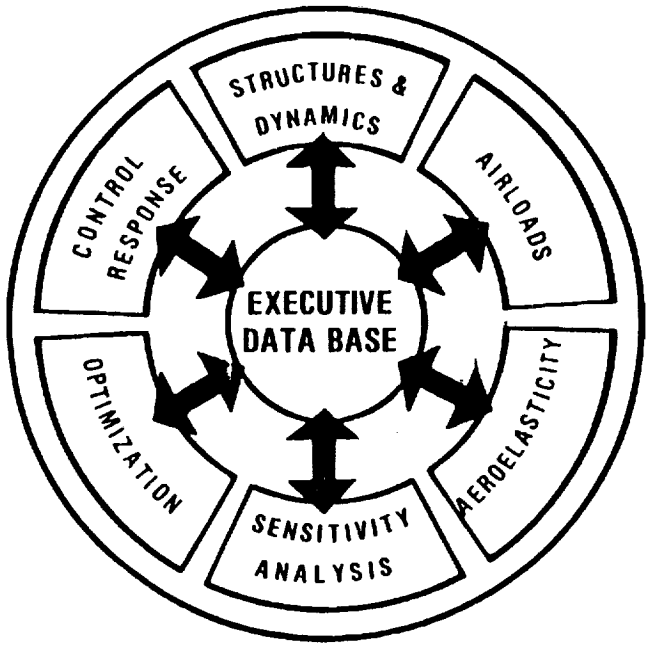
Interest in interdisciplinary systems is widespread around the world. A number of structural optimization systems are being developed and tested. Some examples are listed here. ASOP - FASTOP - TSO are programs developed for the Air Force Flight Dynamics Laboratory during the seventies. They have established the feasibility of integrating various disciplines. The program ''STARS'' is basically a structural optimization system developed by the Royal Aircraft Establishment (RAE) in England. At present, it is being enhanced to include aeroelasticity. ''LAGRANGE'' is an optimization system developed by MBB in Germany. It includes loads - structures - aeroelasticity and optimization. ELFINI is a similar system developed (DASSAULT) in France. It is being marketed in the US as well as other countries. ''ASTROS'' - Automated STRuctural Optimization System was developed for the Flight Dynamics Directorate of Wright Laboratory. The remaining viewgraphs describe the ASTROS system.

Review of Structural Optimization Systems

- ASOP—FASTOP
- TSO
- STARS
- LAGRANGE
- ELFINI
- ASTROS

ASTROS is a design optimization system with a highly sophisticated computer architecture. Control of ASTROS is through an executive system which is written in a high-level computer language called MAPOL (Matrix Abstraction Program Oriented Language). This executive system is supported by a well designed scientific database called CADDB. The interactive version of CADDB is called ICE. Both the executive system and the database are highly flexible and provide an extremely user friendly environment. The executive system and database are supported by six engineering modules. The structures and dynamics model is central to ''ASTROS'' and is based on the finite element method with a library of elements applicable in optimization. Eigenvalue analysis and frequency and transient response analysis constitute dynamic analysis. The static analysis includes a variety of mechanical and thermal loadings. The airloads module is based on steady aerodynamic theory and is used for the purpose of calculating maneuver loads. The aeroelasticity module is based on unsteady aerodynamic theory for flutter calculation. The sensitivity analysis is based on analytical gradients of the constraints and the objective function. Stress, displacement, frequency, flutter and a variety of static aeroelastic constraints are included in the sensitivity analysis. The optimization module is at present the ADS package, and planned enhancements include a method based on optimality criteria. The control response module is the weakest, and it does not participate in optimization.

AUTOMATED STRUCTURAL OPTIMIZATION SYSTEM



OBJECTIVES AND PAYOFFS

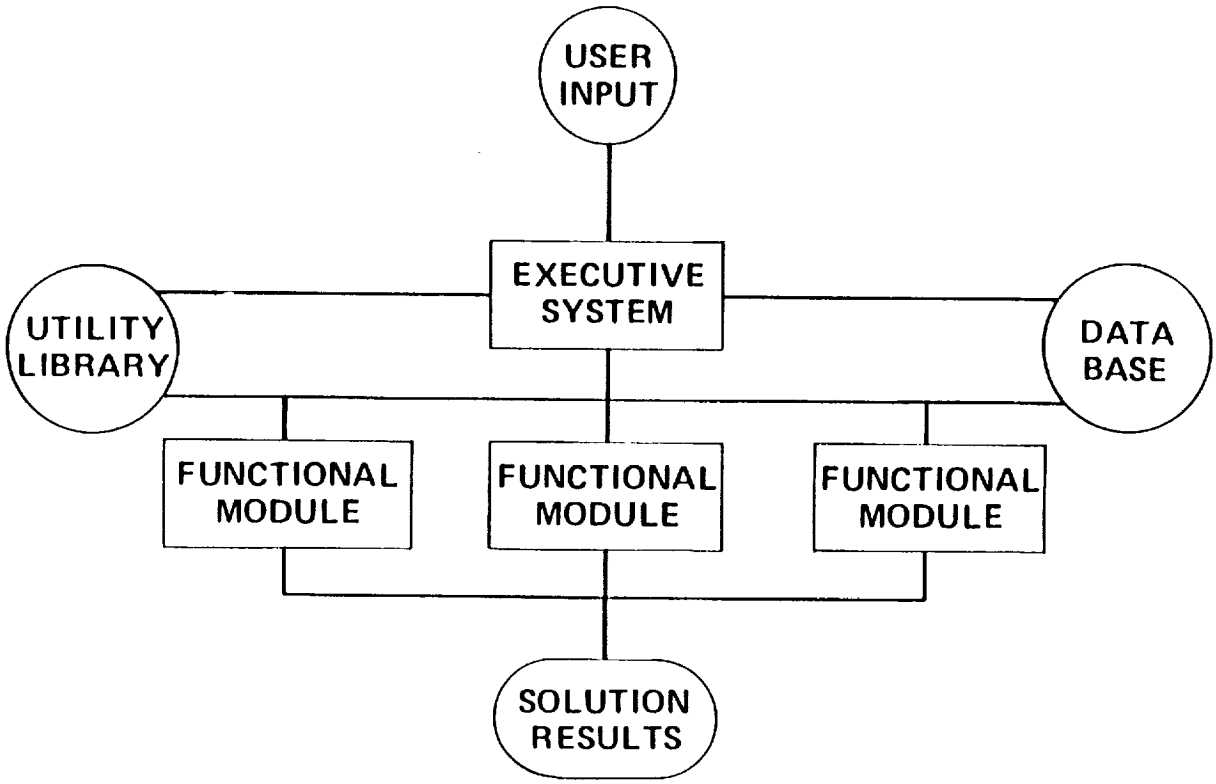
OBJECTIVES

- AN AUTOMATED TOOL FOR PRELIMINARY STRUCTURAL DESIGN
- EMPHASIZE INTERDISCIPLINARY FEATURES OF THE DESIGN TASK
- PROVIDE A NATIONAL RESOURCE

PAYOFFS

- IMPROVED COMMUNICATION AMONG DESIGN TEAM MEMBERS
- IMPROVED DESIGN
- REDUCED DESIGN TIME

ASTROS ARCHITECTURE

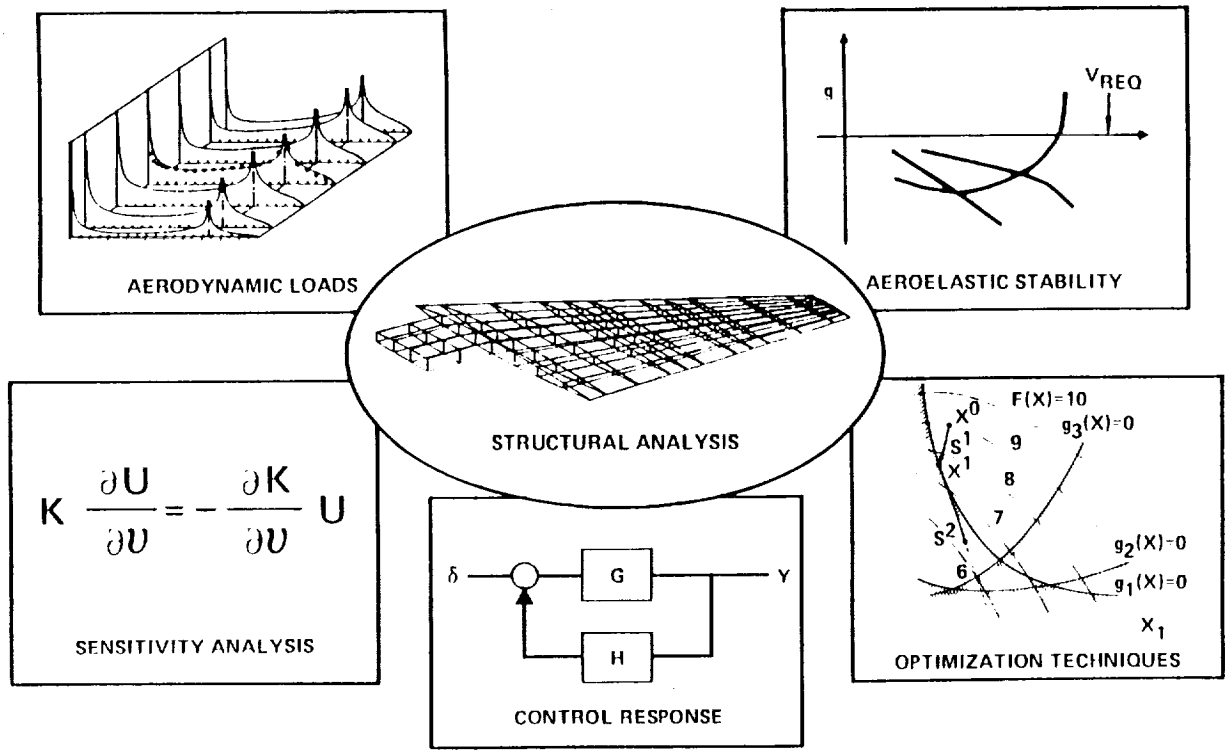


Architectural Highlights

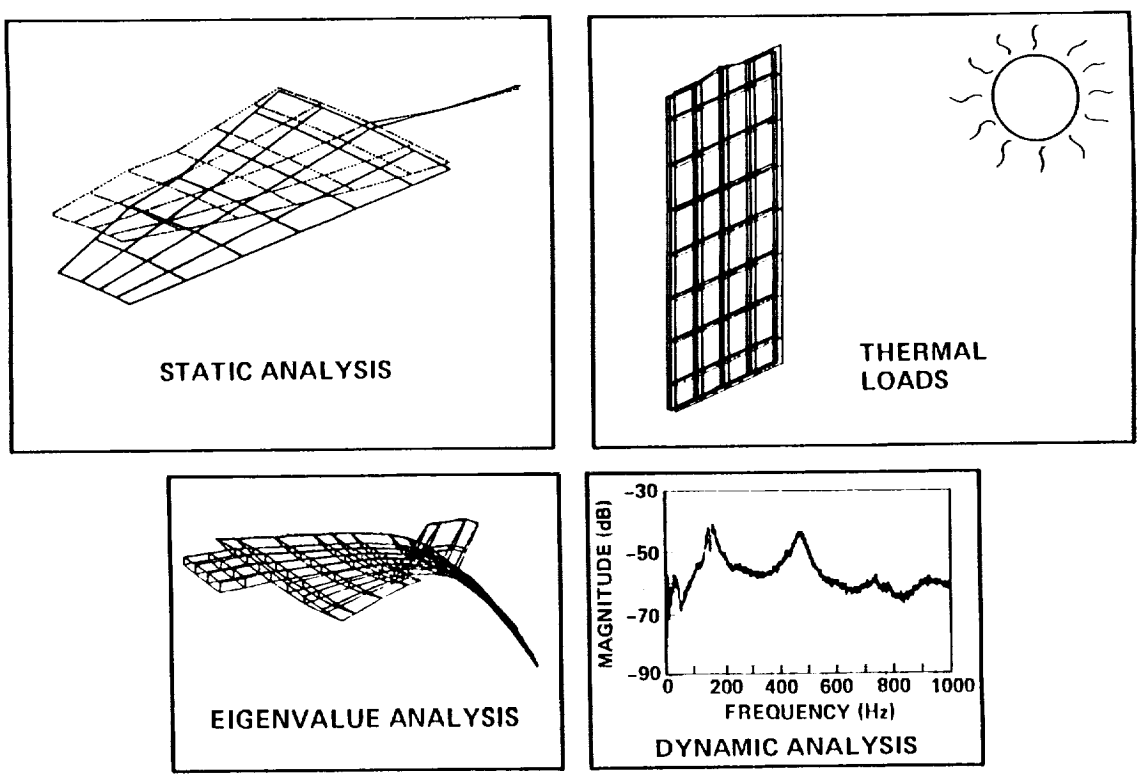
- **Executive System**
  - Provides High Level Control
  - Enables Multidisciplinary Design
- **Database**
  - Customized for Engineering Analysis and Design
  - Necessitated Major Recoding of Software Resources
- **Dynamic Memory**
  - Enables Unrestricted Problem Size
  - Provides Programmer with Precise, Explicit Control
- **Utility Library**
  - Special Purpose Routines Required By Modules (Sort, Search, etc.)
  - Emphasis Placed on High Quality, Robust, Self Documented Algorithms
  - Machine Dependent Functions Isolated
- **Modules**
  - Distinction Between Functional and Utility Modules Blurred
  - Each Module :

    Establishes Base Address in Memory
    Opens Required Data Base Entities
    Closes All Data Base Entities Prior to Exit
    Frees All Memory Blocks Prior to Exit
  - Intermodular Communication is Through the Data Base

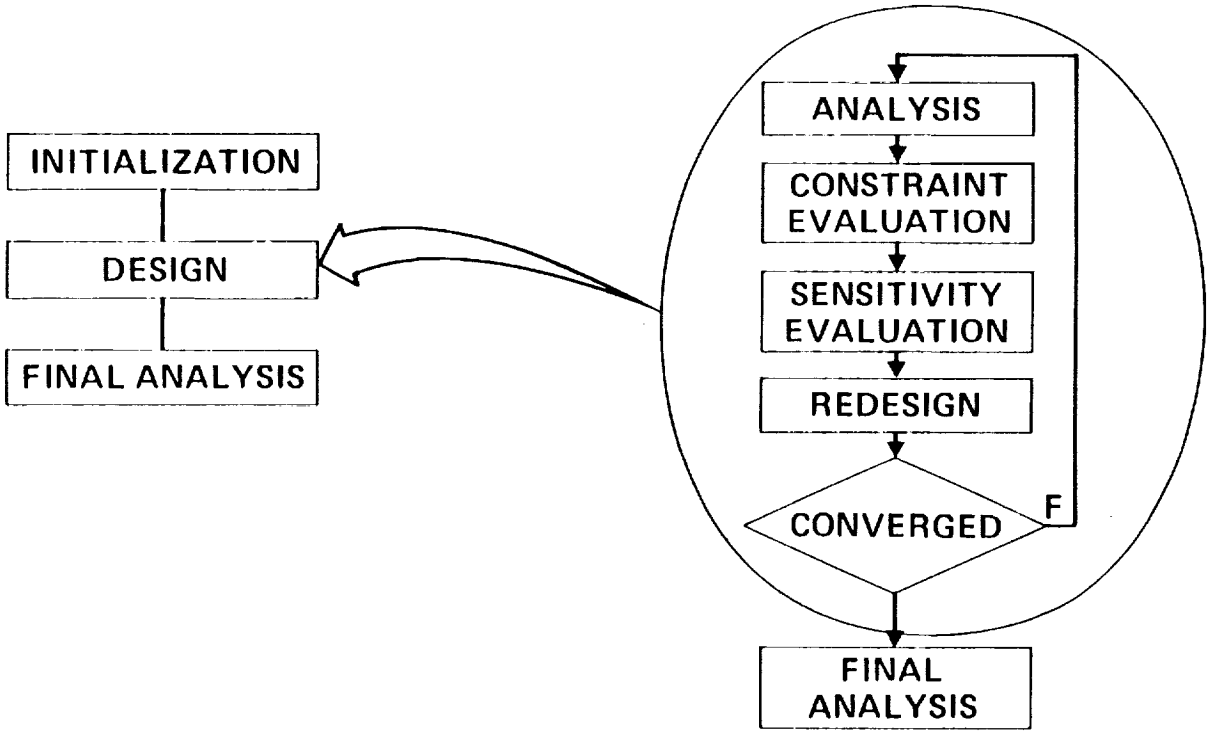
ENGINEERING DISCIPLINES



STRUCTURAL ANALYSES



BASIC ASTROS SEGMENTS



Software Resources for ASTROS

Structural Analysis	—	NASTRAN
Static Aerodynamic Loads	—	USSAERO
Unsteady Aerodynamic Loads	—	Doublet Lattice CPM
Optimization Algorithms	—	MDOT

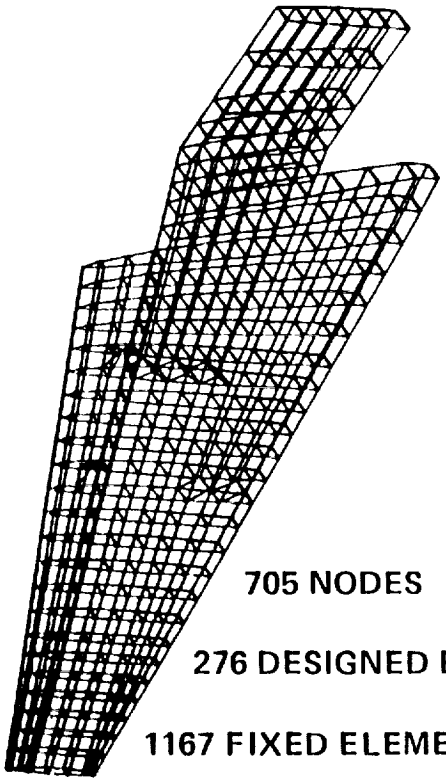
Ten Software Contributions of ASTROS

- Framework For Multidisciplinary Analysis and Design
- Engineering Data Base
- High Level Executive System
- Obsolescence of Rigid Formats
- Unlimited Problem Size
- Exploitation of Microcomputers
- Built In Maintenance Features
- Improved Special Purpose Utilities
- Balanced Approach to Software Design
- Integration of Dispersed Development Team

Ten Engineering Contributions of ASTROS

- Multidisciplinary Analysis and Design
- Analytical Sensitivity Analysis
- Approximation Concepts in a Production Code
- QUAD4 Element in the Public Domain
- Improved Supersonic Unsteady Aerodynamics
- Innovative Flutter Design Technique
- Nuclear Blast Analysis with Finite Elements and Advanced Aerodynamics
- Advanced Methods of Dynamic Reduction
- Design Variable Linking
- Aerodynamic Influence Coefficients For Static Aeroelasticity

AN ARCHETYPICAL ASTROS APPLICATION



GIVEN:
- STRUCTURAL CONFIGURATION
- MATERIAL PROPERTIES
- DESIGN FLIGHT CONDITIONS
- DESIGN ALLOWABLES

DETERMINE
- THICKNESSES OF DESIGNED ELEMENTS
- OPTIONALLY – MASS BALANCE VALUES

POSSIBLE DESIGN CONSIDERATIONS
- MULTIPLE BOUNDARY CONDITIONS
- MULTIPLE FLIGHT CONDITIONS
- MULTIPLE STORE LOADINGS

705 NODES

276 DESIGNED ELEMENTS

1167 FIXED ELEMENTS

DESIGN PARAMETERS

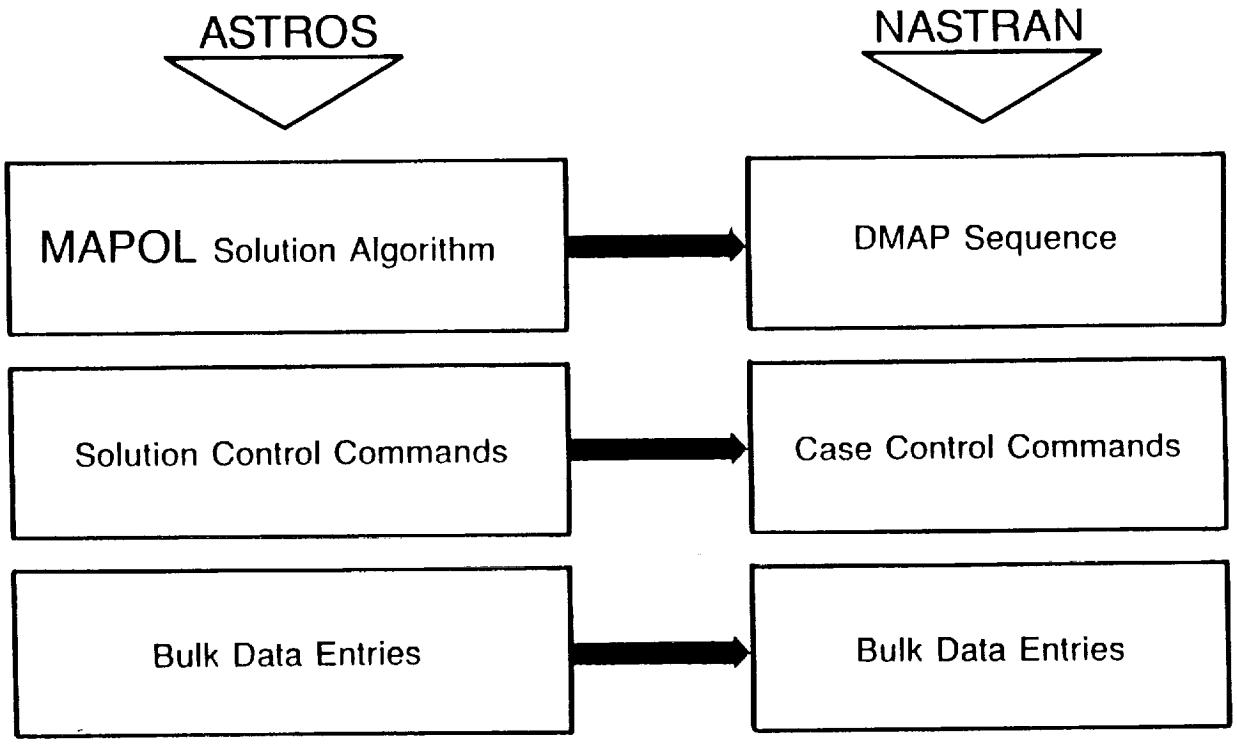
DESIGN VARIABLES

- ROD AREAS
- SHEAR ELEMENT THICKNESSES
- MEMBRANE ELEMENT THICKNESSES
- BARS
- CONCENTRATED MASSES

CONSTRAINTS

- STRESS-STRAIN
- DISPLACEMENT
- MODAL FREQUENCY
- AEROELASTIC EFFECTS
  - LIFT EFFECTIVENESS
  - AILERON EFFECTIVENESS
  - DIVERGENCE SPEED
- FLUTTER RESPONSE

User Input Data Stream



ASTROS Documentation

- **Theoretical Manual**

  Describes ASTROS Methods
  Emphasizes Innovative Features

- **User's Manual**

  Input and Output Description
  Techniques to Obtain Additional Output
  Creation and Modification of MAPOL Sequences

- **Application Manual**

  Documentation Resources
  Modeling Guidelines
  Sample Cases

- **Programmer's Manual**

  Code Installation
  Module Description
  Data Base Calls
  Utility Calls

Conclusions

- Potential for Realization of CAD in Multidisciplinary Environment
- Widespread Interest Around the World
- Need Extensive Applications to Assure Reliability
- Validation

- SHORTER SCHEDULES
- EXTENSIVE PARAMETRIC STUDIES
- OPTIMAL DESIGNS – BEST PERFORMANCE
- LIGHT WEIGHT STRUCTURES
- TECHNOLOGY TRANSFER
  - RESEARCH TO APPLICATION
  - SYSTEM TO SYSTEM (LESSONS LEARNED)

N92-25922

Light Thermal Structures and Materials for High Speed Flight

Earl A. Thornton
Light Thermal Structures Center
University of Virginia
Charlottesville, Virginia

INTRODUCTION

As a hypersonic vehicle accelerates at high speeds in the atmosphere, shocks sweep across it interacting with local shocks and boundary layers. These interactions expose structural surfaces to severe local pressures and heat fluxes. One example is leading edges of integrated engine structures which experience intense, highly localized aerothermal loads.

Until recent years the study of structural response at elevated temperatures due to dynamic loads was not possible because of an inability to model inelastic material behavior. Over the last twenty years, unified viscoplastic constitutive models have evolved to meet this need. These constitutive models provide a means for representing a material's response from the elastic through the plastic range including strain-rate dependent plastic flow, creep and stress relaxation. Rate-dependent plasticity effects are known to be important at elevated temperatures.

The purpose of this paper is to describe computational and experimental research programs underway at the Light Thermal Structures Center focused on the investigation of the response of structures and materials to local heating. In the first part of the paper, finite element thermoviscoplastic analysis is highlighted. In the second part of the paper, the thermal-structures experimental program is outlined.

FLOW INTERACTION FOR AEROSPACE PLANE COWL LEADING EDGE

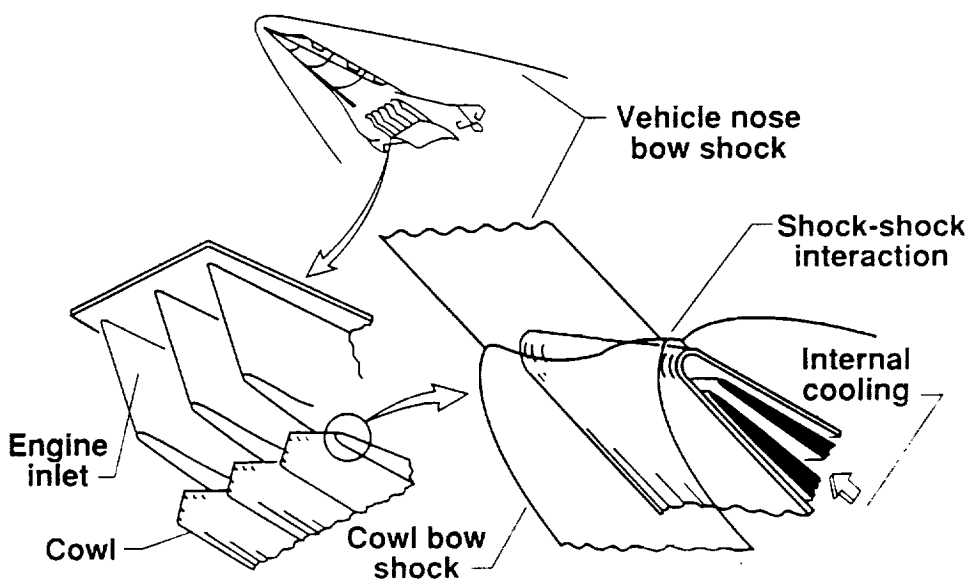


Figure 1

RESEARCH OBJECTIVES

Finite element analysis with unified constitutive models has been under development for about 15 years and provides an important simulation capability. Reference 1 describes past efforts and presents a thermoviscoplastic finite element computational method for hypersonic structures. Applications to convectively cooled hypersonic structures illustrate the effectiveness of the approach and provide insight into the transient viscoplastic behavior at elevated temperatures. References 2-3 use the computational method presented in ref.1 to perform quasistatic finite element thermoviscoplastic analysis of aerospace structures subjected to localized heating.

There is a need for further studies to: (1) understand the phenomena that occur in the viscoplastic response of structures for intense local heating, (2) investigate finite element modeling techniques required to represent thermoviscoplastic behavior, (3) perform thermal-structural experiments to validate thermoviscoplastic computational methods, and (4) understand the high-temperature behavior of difficult design problems such as leading edges of hypersonic vehicles.

- Investigate Thermoviscoplastic (TVP) response of thin panels subject to intense local heating.
- Evaluate finite element Thermal-Structural analyses with unified TVP constitutive models by comparison with experimental data.

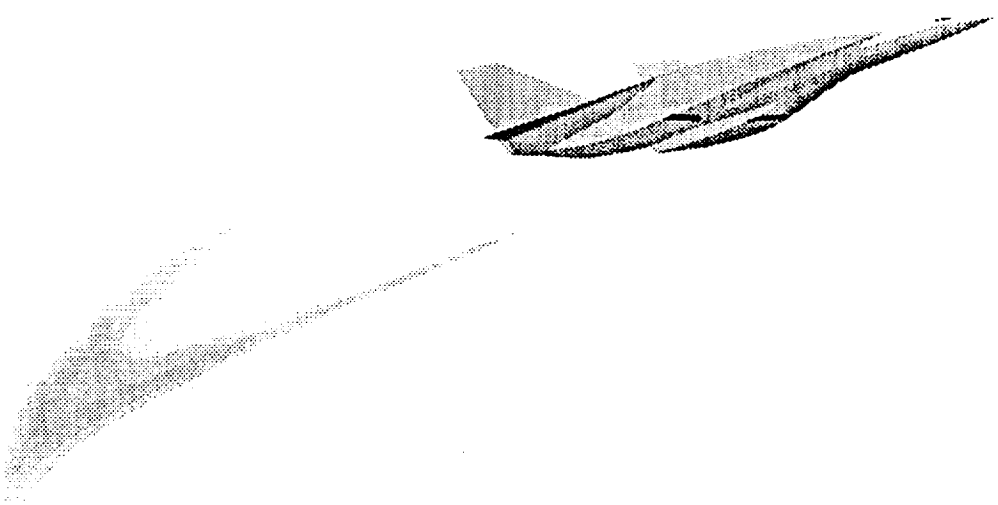


Figure 2

The behavior of a thermoviscoplastic structure subjected to intense heating is analyzed assuming that: (1) thermo-mechanical coupling in the conservation of energy equation can be neglected, (2) the structural response is quasistatic, and (3) deformations are infinitesimal. With these assumptions, an unsteady thermal analysis may be performed first to determine the temperatures. Then, using these temperatures, the structure's viscoplastic response is determined. The structural response is obtained by solving the equilibrium equations written in rate form. The solution is obtained by solving the initial value-boundary value problem by time marching (ref. 1).

$$[K]\{\dot{\delta}\} = \{\dot{F}_p\} + \{\dot{F}_T\} + \{\dot{F}_\sigma\} + \{\dot{F}_B\}$$

where:

$$[K] = \int_{\Omega_e} [\,B\,]^T\,[\,E(T)\,]\,[B]\,d\Omega \qquad \textbf{Stiffness Matrix}$$

$$\{\dot{F}_p\} = \int_{\Omega_e} [\,B\,]^T\,[\,E(T)\,]\,\{\dot{\varepsilon}^P\}\,d\Omega \qquad \textbf{Plastic Strain}$$

$$\{\dot{F}_T\} = \int_{\Omega_e} [\,B\,]^T\,[\,E(T)\,]\,\{\alpha(T)\}\,\Delta\dot{T}\,d\Omega \qquad \textbf{Temperature}$$

$$\{\dot{F}_\sigma\} = \int_{\partial\Omega_e} [N]^T\,\{\dot{\bar{\sigma}}\}\,ds \qquad \textbf{Surface Tractions}$$

$$\{\dot{F}_B\} = \int_{\Omega_e} [N]^T\,\{\dot{B}\}\,d\Omega \qquad \textbf{Body Forces}$$

Figure 3

The Bodner-Partom constitutive model is of the internal state variable type that is based on phenomenological observations and supported by physical concepts related to dislocation dynamics. The model has gone through several modifications and was extended for anisotropic work hardening materials. The current model (ref. 4) also includes thermal effects.

The strategy employed in the viscoplastic algorithm is as follows: with the initial distribution of stress, temperature and internal variables specified, the equilibrium equations are solved to obtain the nodal displacement rates. Then the constitutive equations are integrated forward in time at the element Gauss integration points. With updated values of the stresses, temperatures and internal variables at the new time, the equilibrium equations are solved again. This sequence of determining the nodal displacement rates, then advancing the constitutive equations in time is continued until the desired history of the initial-value boundary-value problem has been obtained.

1. At time = t, initialize σ_{ij} and Z
2. Calculate $\dot{\varepsilon}^P_{ij} = (S_{ij} / \sqrt{J_2})\, D_0 \exp[- \frac{1}{2} (Z^2 / 3J_2)^n]$
3. Assemble and solve $[K]\{\dot{\delta}\} = \{\dot{F}\}$
4. Calculate $\{\dot{\varepsilon}\} = [B]\{\dot{u}\}$
5. Calculate $\{\dot{\sigma}\} = [E]\{ \dot{\varepsilon} - \dot{\varepsilon}^P \} - [E]\{\alpha\}\Delta\dot{T}$
6. Calculate $\dot{Z} = m_1(Z_1 - Z)\dot{W}_P - A_1 Z_1[(Z - Z_2)/ Z_1]^r$
7. Integrate $\dot{\sigma}_{ij}$ and $\dot{Z}$ forward in time to get σ_{ij} and Z
8. If $(t + \Delta t) < t_{final}$, return to step 2

Figure 4

In reference 1 a two-dimensional, plane strain model of a convectively cooled hypersonic structure with a region of intense heating was analyzed. The model represented a segment of the wall of a scramjet engine. The analysis was performed with the finite-element model shown in Fig. 5a. Figure 5b presents the temperature history on the aerodynamic surface indicating the very high temperature the structure may experience. Although there is a temperature gradient through the skin thickness there is little axial conduction because of the low thermal conductivity of the nickel superalloy material. The resulting thermal "hot spot" induces large in-plane inelastic compressive stresses. Figure 5c presents the history of the normal stress component at the aerodynamic surface. Note the material yields early in the response. After the temperature drops there is a rapid decay of the stress, and there are residual tensile stresses. The residual stresses suggest the possibility of cumulative damage under repeated load cycles.

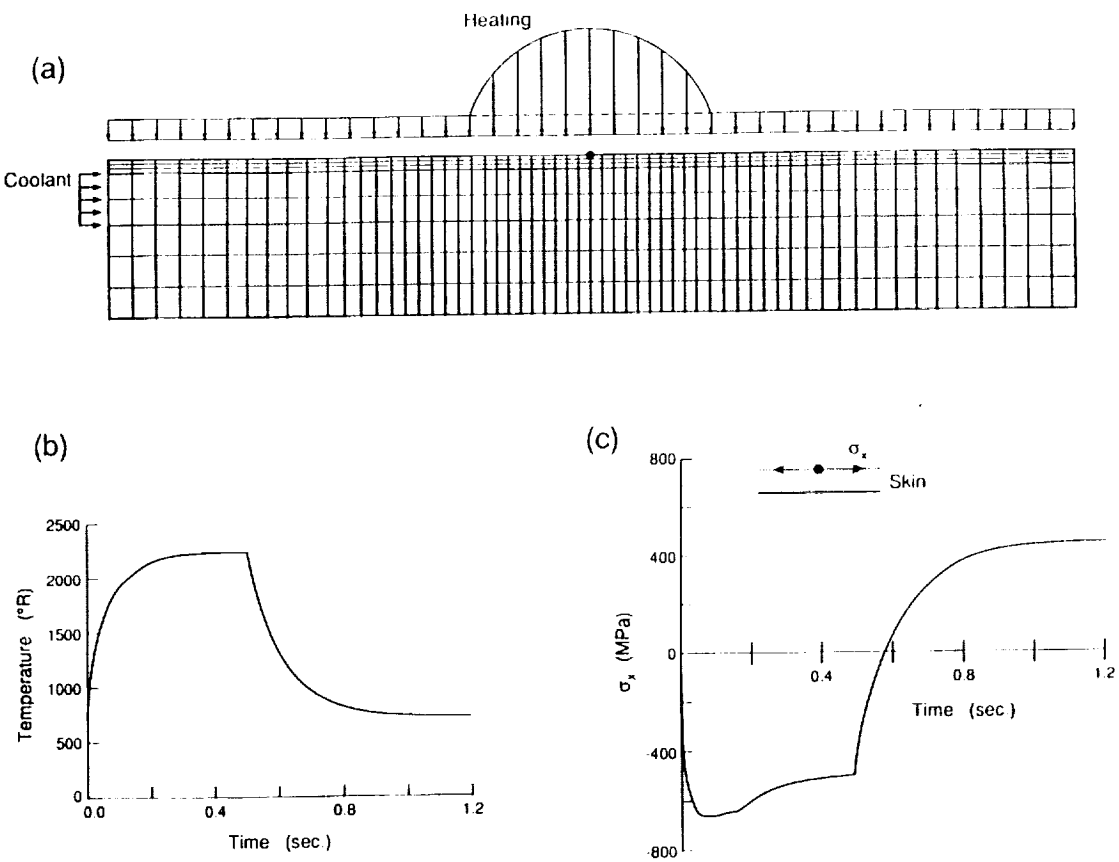


Figure 5

In the early 1950s, Richard R. Heldenfels and co-workers at NASA Langley (refs. 5-6) investigated the elastic thermal-structural behavior of flat plates. The investigation consisted of theoretical studies based on approximate analytical solutions and experimental studies of rectangular aluminum plates. In the experimental studies, simple "tent-like" steady-state temperature distributions were introduced by heating a rectangular plate along a centerline with a heating wire and maintaining constant temperatures along parallel edges by water flow through coolant tubes. Top and bottom surfaces of the plate were insulated to produce uniform one-dimensional, linear temperature variations between the heated centerline and cooled parallel edges. In the first experimental study (ref. 5), in-plane plate displacements were permitted to occur freely, but out-of-plane displacements were prevented by restraints that forced the plate to remain flat. In the second experimental study (ref. 6), the out-of-plane constraints were removed except along the plate edges, and plate bending displacements were permitted. An important result of these studies was to demonstrate that out-of-plane plate bending displacements occur due to in-plane temperature variations. Out-of-plane displacements are initiated by small initial plate warpage.

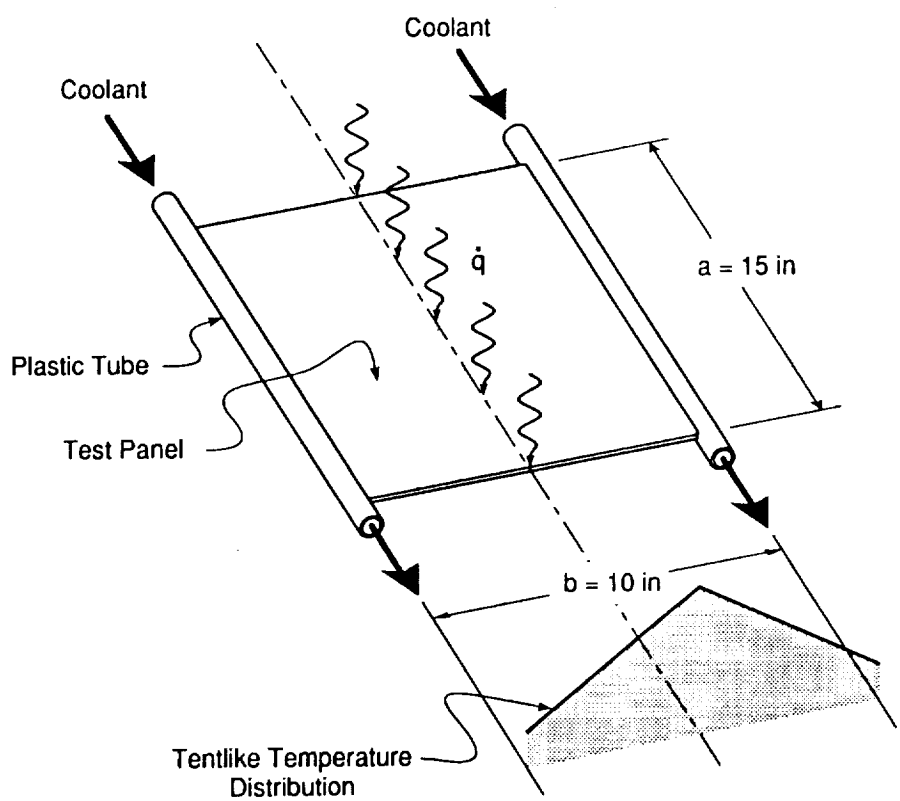
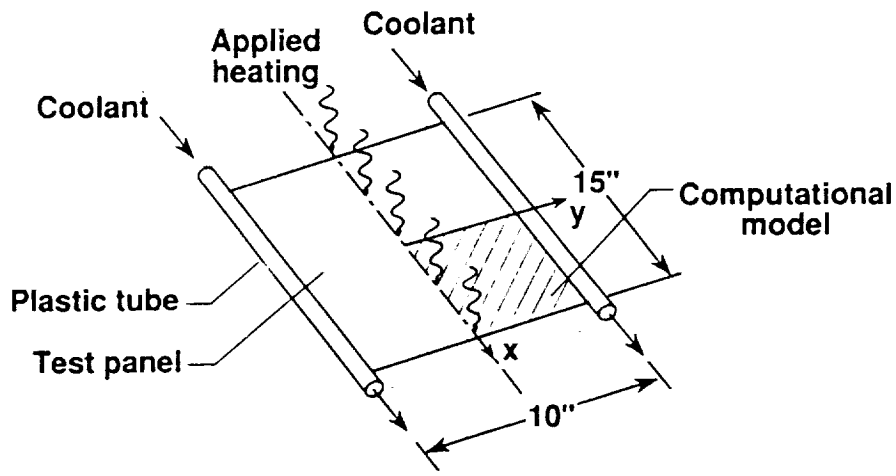


Figure 6

In ref. 3 the thermoviscoplastic behavior of a Heldenfels plate is studied assuming the plate is perfectly flat and a state of plane stress occurs. Preliminary elastic analysis of an initially warped plate shows that membrane stresses are larger than bending stresses so that a plane stress thermoviscoplastic analysis will provide valuable insight into the transient, inelastic plate response. Assuming symmetry, one-quarter of the plate is modeled as shown. The plate material is Hastelloy-X.

The plate was analyzed for two cases of heating. In the first case, a low heating rate is used, and plate temperatures are assumed to be in thermal equilibrium at each time. In the second case, a high heating rate is used, and a thermal transient ensues. The strain rate in the slowly heated plate is about 1 microstrain/s, and the maximum initial strain rate in the rapidly heated plate is approximately 1000 microstrain/s.



Finite Element Meshes:

	Slowly Heated	Rapidly Heated
Nodes:	176 uniform	187 stretched
Elements:	150 quads	320 triangles

Figure 7

Slowly heated plate: In the slowly heated plate, a tentlike temperature distribution is assumed at each instant so that temperatures decrease linearly from the heated plate center to the cooled edges. Temperatures were increased over a 90-minute period to levels that induce significant plastic deformation. Temperature variations over the one-quarter symmetry finite element model are shown. A time step of 10s was used to compute the plate response for the 90-minute duration.

Rapidly heated plate: In the rapidly heated plate, the temperature increases quickly with time along the plate centerline at y=0 and demonstrates a steep spatial gradient. Note that due to the high local heating rate, temperatures at the plate centerline reach 2300 R in 180s. The temperatures shown were computed from an analytical solution that assumes constant thermal properties. The viscoplastic response was computed for 180s using a time step of 0.125s.

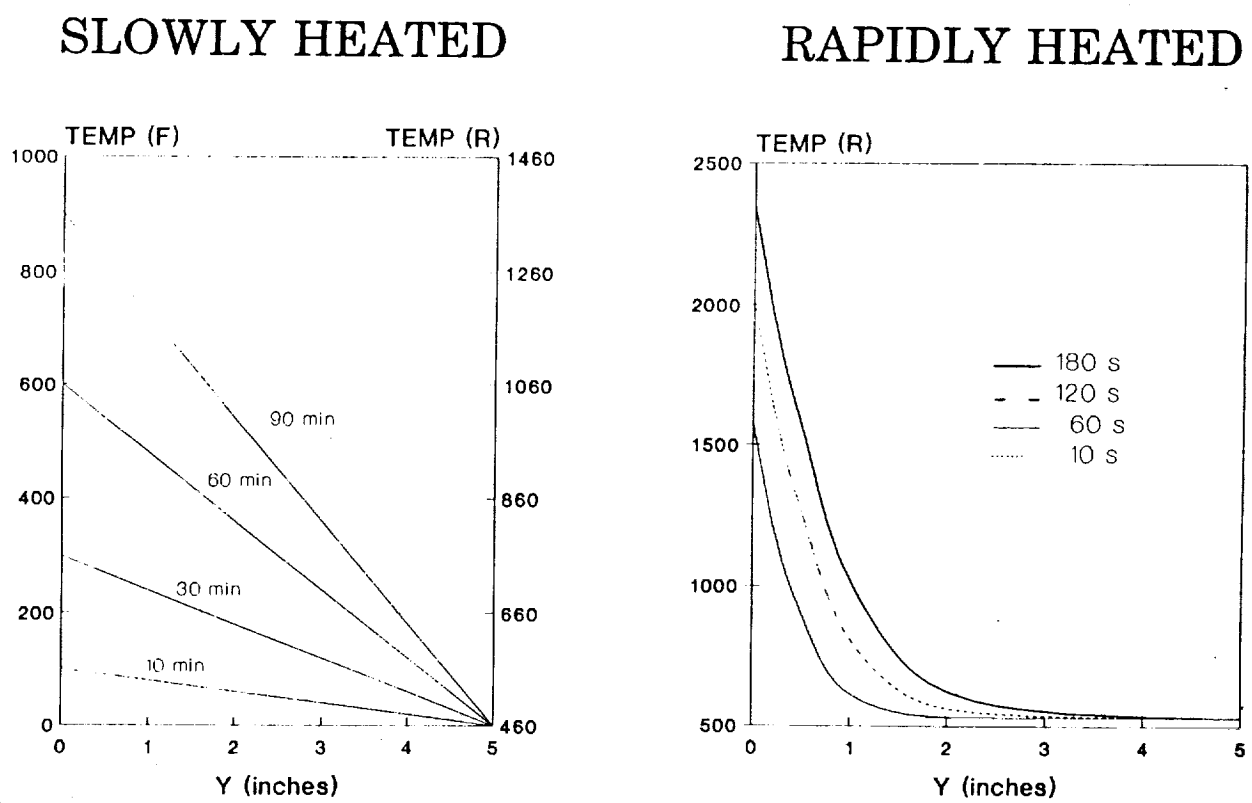


Figure 8

Slowly heated plate: In the first 30 minutes of heating the plate remains elastic. For this period, the finite element computed stresses compare well with results computed from an approximate complementary energy solution given in ref. 5. The inelastic behavior of the plate is shown for three later times in the response. The figure clearly shows that the plate yields in both compressive and tensile stress regions. The yielded regions grow with time along the y axis to cover almost the entire cross-section at 90 minutes.

Rapidly heated plate: In comparison to the slowly heated plate, yielding occurs only near the plate centerline. On the outer plate edge, the tensile stress is well below the yield stress. In ref. 3, the response histories near the plate center for slow and rapid heating are studied in more detail. The rapidly heated plate shows an initial increase in yield stress that is likely due to strain rate effects. Additionally, beginning at about 80 s, a decrease in yield stress begins as plate temperatures exceed 1800 R. This effect is shown in the stress distribution below at 180 s.

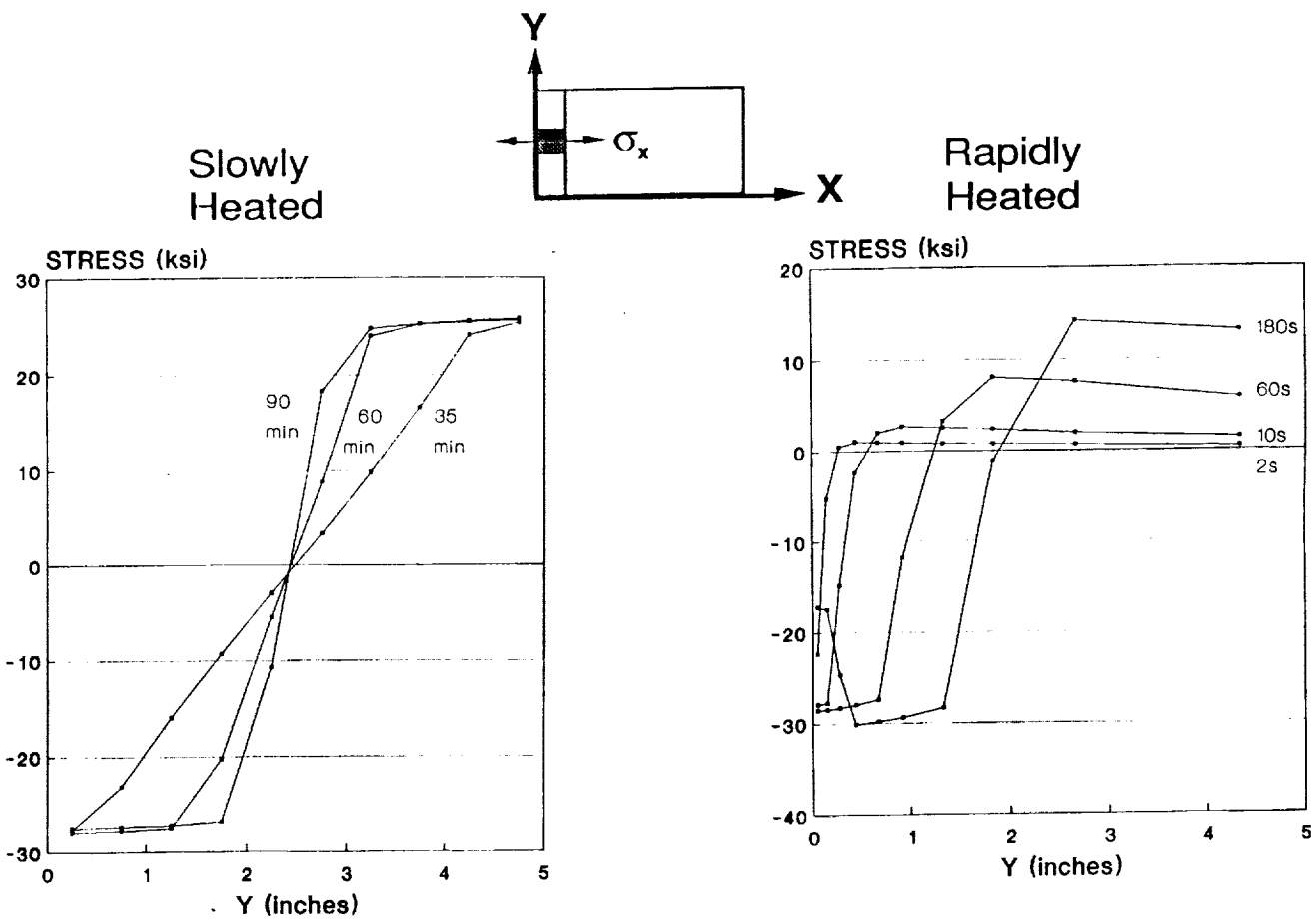


Figure 9

CONCLUSIONS FROM THERMOVISCOPLASTIC ANALYSES

Unified constitutive models implemented in finite element programs provide an important simulation capability. Applications to locally heated structures illustrate the approach and provide insight into transient viscoplastic behavior at elevated temperatures. There has, however, been virtually no validation of the finite element simulation capability by correlations with experimental data for airframe structures.

- TEMPERATURE RISE TIMES AND LEVELS SIGNIFICANT
- FOR RAPID TEMPERATURE RISES:
  - -HIGHER YIELD STRESSES FROM STRAIN-RATE EFFECTS
- AT ELEVATED TEMPERATURES:
  - -MATERIAL YIELD STRENGTH AND STIFFNESS DEGRADE RAPIDLY
  - -PRONOUNCED PLASTIC DEFORMATION
- EXPERIMENTAL DATA NEEDED FOR VALIDATION

Figure 10

EXPERIMENTAL PROGRAM

Research is underway to investigate the nonlinear response of panels subjected to localized heating. The panels' response includes nonlinear geometric effects and inelastic material behavior. An objective of the experimental program is to provide data that can be used for validation of finite element thermoviscoplastic analysis. To achieve this objective, attention is focused on experimentally providing: (1) clearly defined thermal-structural boundary conditions, (2) a known external heat flux distribution, and (3) reliable material characteristics. The balance of this paper describes recent experimental progress and presents preliminary results for elastic buckling from a slowly heated panel test.

- PANELS SUBJECTED TO LINE HEATING
  - -TRANSIENT SPATIAL TEMPERATURE GRADIENTS
  - -GEOMETRIC AND MATERIAL NONLINEARITIES
  - -WELL DEFINED BOUNDARY CONDITIONS

- CONSTITUTIVE MODEL TESTS
  - -BODNER-PARTOM DATA
  - -MATERIALS: HASTELLOY
    8009 ALUMINUM ALLOY

Figure 11

In the experimental study, panels are tested in a fixture based upon the original Heldenfels' concept but with important differences. To obtain sufficiently high heat flux rates to induce inelastic behavior, the panels are heated by a quartz lamp with an elliptical reflector. The reflector focuses the incident heat flux along a narrow band along the panel centerline. The edges of the panels are cooled with a chill-water system to maintain constant edge temperatures. A test panel is supported at only four points to permit well-defined structural boundary conditions and minimize support's heat losses. One support prohibits displacements in both the transverse and in-plane directions, and the other three supports prohibit transverse displacements while permitting in-plane motions. Test panels are instrumented to measure temperatures, displacements and strains.

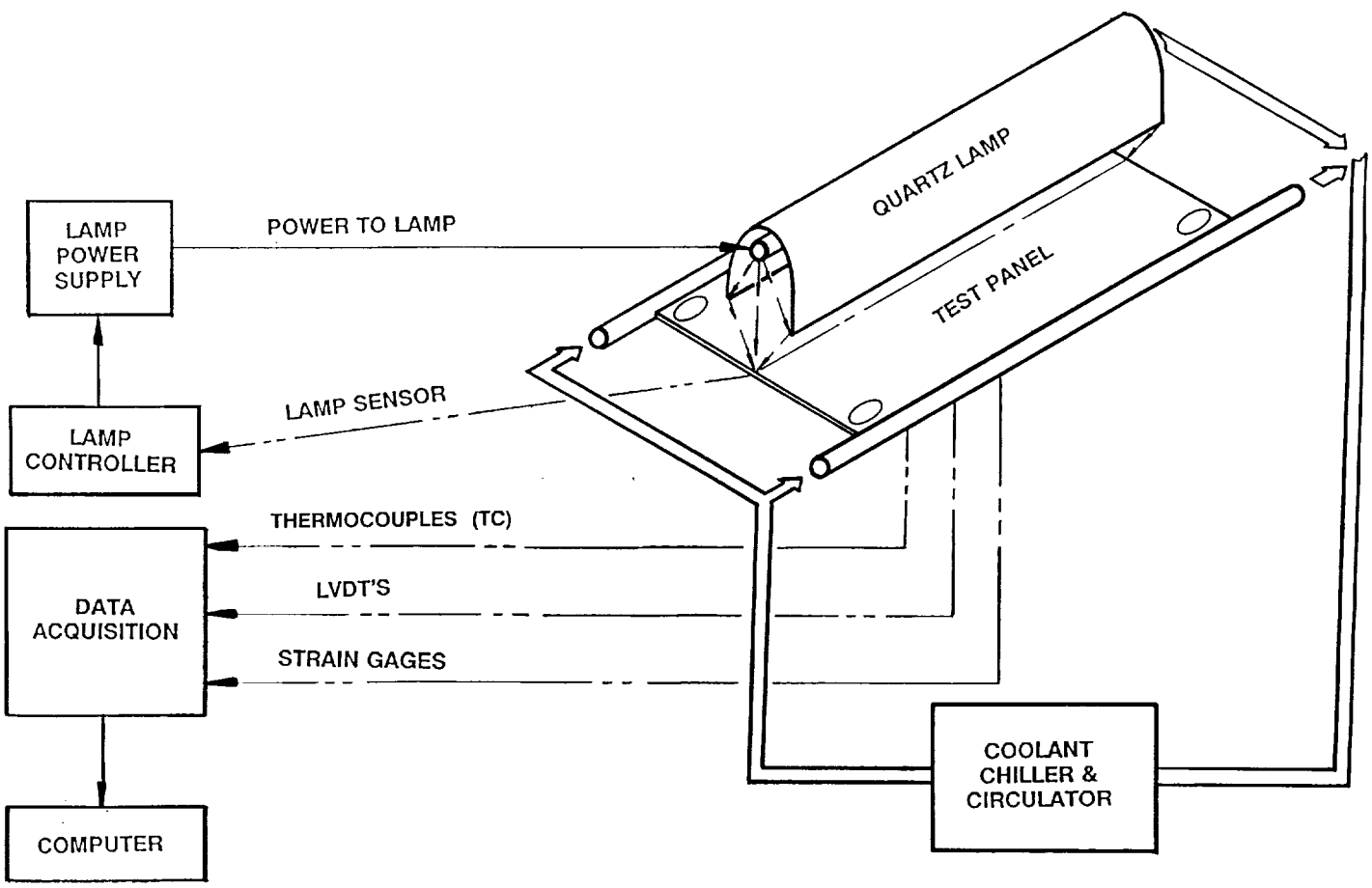


Figure 12

PANEL INITIAL DEFORMATIONS

Test panel initial deformations (warpage) determine the direction of panel buckling due to the in-plane spatial temperature gradients. The initial warpage of each test panel was carefully determined with a coordinate measuring machine at NASA Langley Research Center. A typical surface plot of measured initial displacements of a 1/8 in. thick Hastelloy-X test panel is shown.

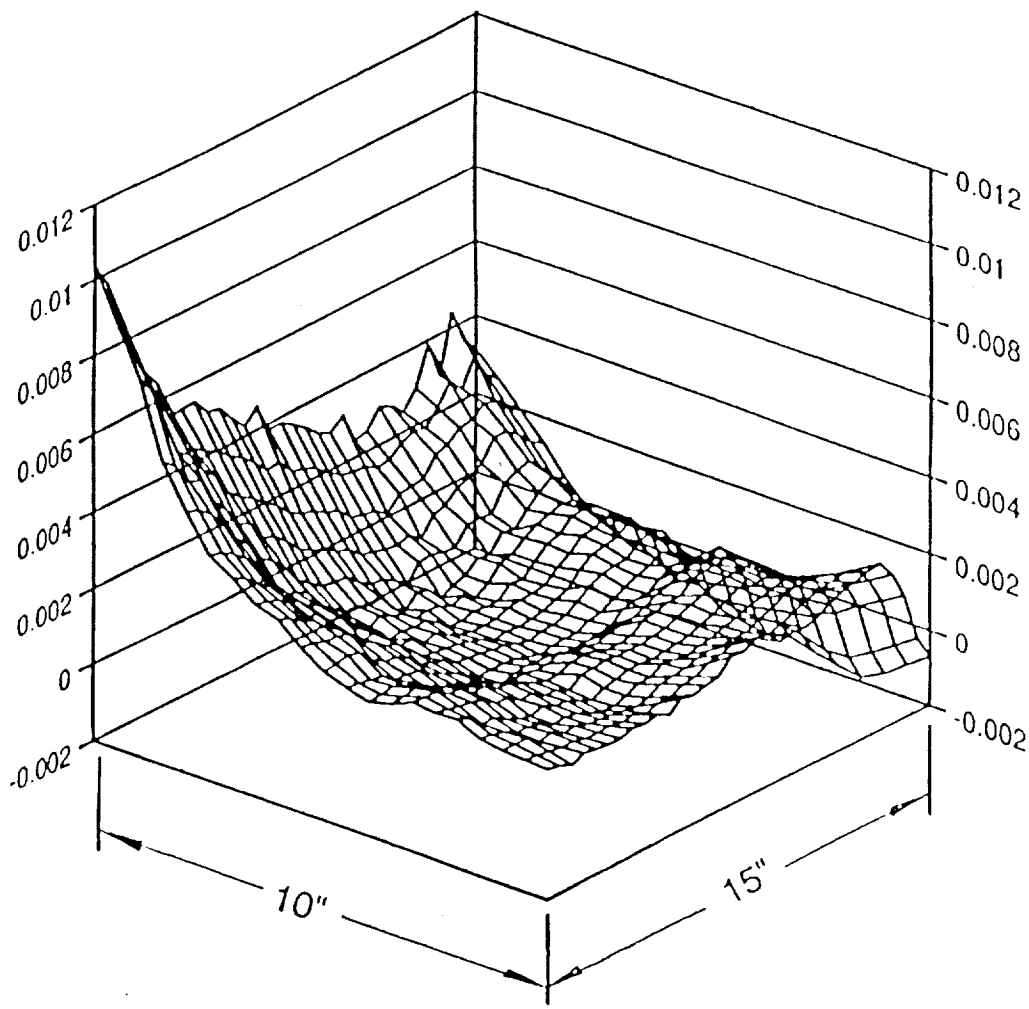


Figure 13

HEAT LAMP TEST FIXTURE

Knowledge of the distribution of the incident heat flux on a test panel surface is important for predicting the thermal response. Although the lamp reflector is designed to focus the heat flux along a narrow band, preliminary measurements indicate a variation of the flux over incident and adjacent areas. In addition, there is a concern that the deflecting panel, either toward or away from the lamp, may alter the incident flux. To address these issues, a heat lamp test fixture has been designed and constructed to permit measurement of the flux distribution over a surface. The test fixture permits x-y-z variations of the lamp position.

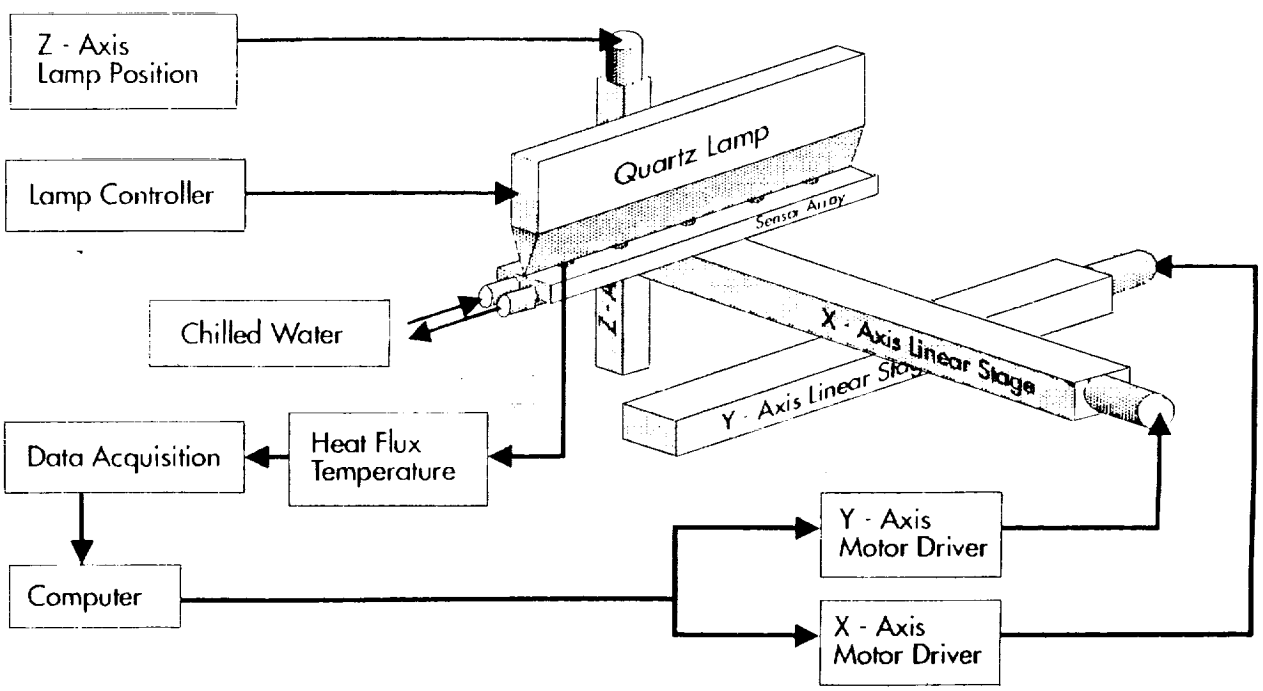
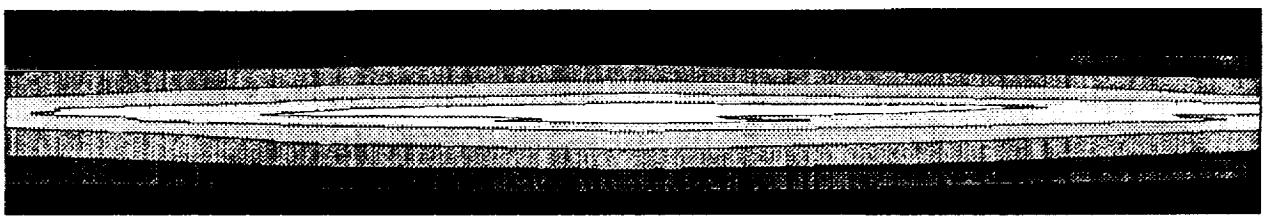


Figure 14

Preliminary results from an incident heat flux survey are shown. The results demonstrate that the heat flux varies strongly with the y coordinate across the lamp width indicating the focusing effect of the lamp reflector. The flux is nearly uniform in the x-direction over the central portion of the lamp but decreases near each end. Lamp heat flux measurements are continuing, and data will be gathered for various heating rates. In addition, heat flux variations with the z coordinate will be investigated.



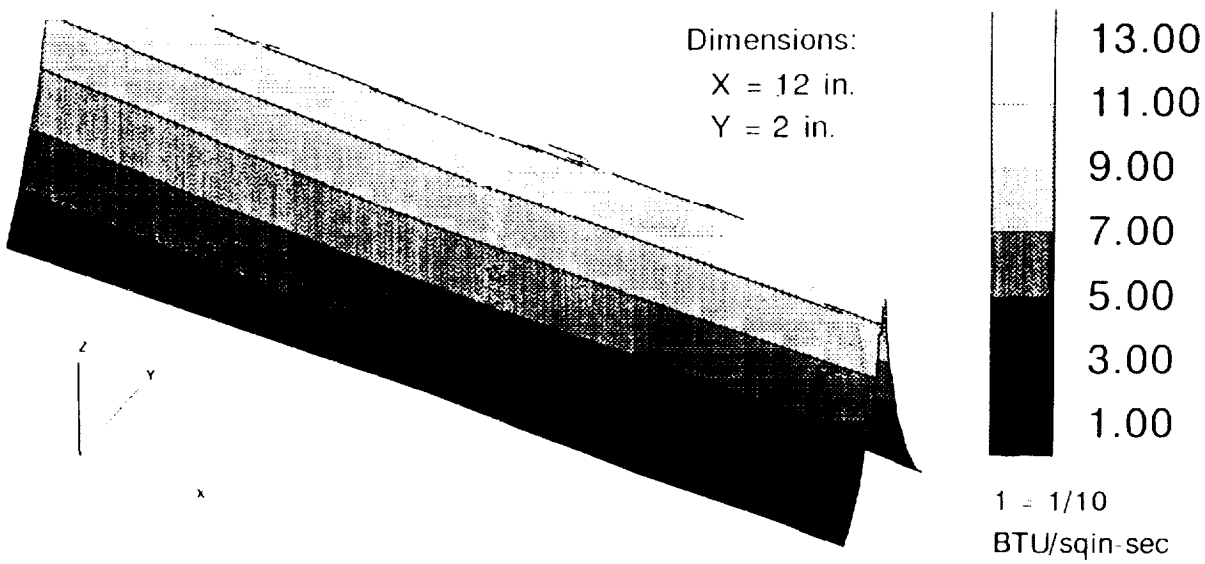


Figure 15

EXPERIMENTAL PANEL TEMPERATURE DISTRIBUTIONS

Preliminary results for a slowly heated Hastelloy X panel are shown in the next three figures. For the low heating rate used, the panel approached thermal equilibrium in about 90 minutes. Figure 16 presents measured temperature profiles along the Y axis at three times in the response. The profiles are symmetric about the X-axis and reach a linear variation at steady state. Temperature variations along the X axis (not shown) are nearly uniform. These results indicate that the desired thermal boundary conditions have been achieved. The results provide validation for the effectiveness of the cooled edges, point supports and insulation system.

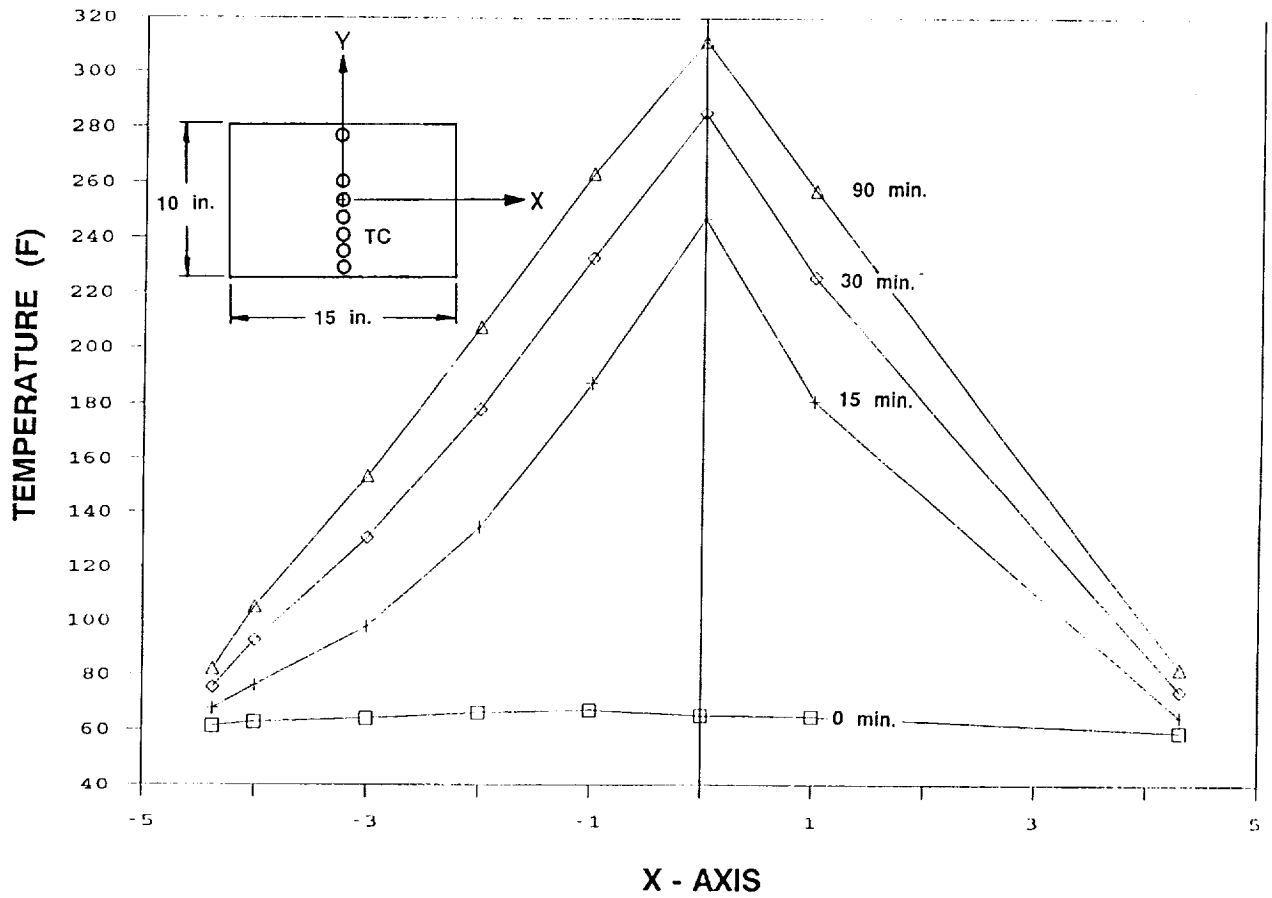


Figure 16

EXPERIMENTAL PANEL TEMPERATURE RESPONSE

The transient temperature response measured by two "back-to-back" thermocouples on top and bottom surfaces of the panel are shown. These responses show temperatures approaching steady-state in 90 minutes as previously discussed. The results also demonstrate that temperatures on the top and bottom surfaces of the panel follow each other closely. Thus there is only a small temperature gradient through the panel thickness at this point of about 5°F. Other pairs of thermocouples show similar results.

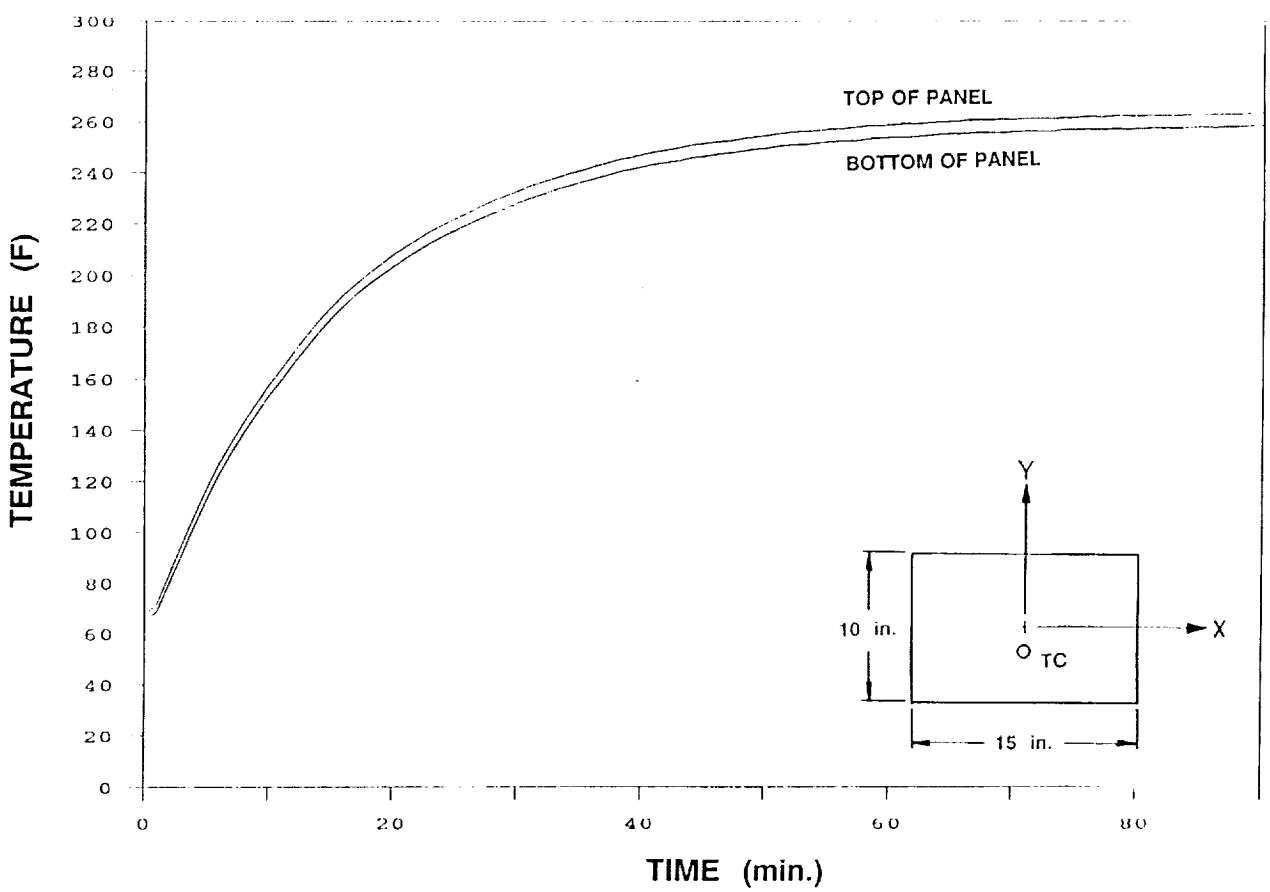


Figure 17

EXPERIMENTAL PANEL DISPLACEMENT RESPONSE

The temperature-transverse displacement response measured at the center of the panel is shown. The result clearly demonstrates the nonlinear bending behavior of the panel. Observations of panel response in several tests indicate that overall panel bending takes place in both the x and y directions. The direction of the panel bending is determined by the initial panel warpage. This result has been demonstrated by turning the panel "upside down" in the test fixture. Panel deformations then occur in the opposite direction. The measured temperature displacement response is consistent with results presented in Ref. 6.

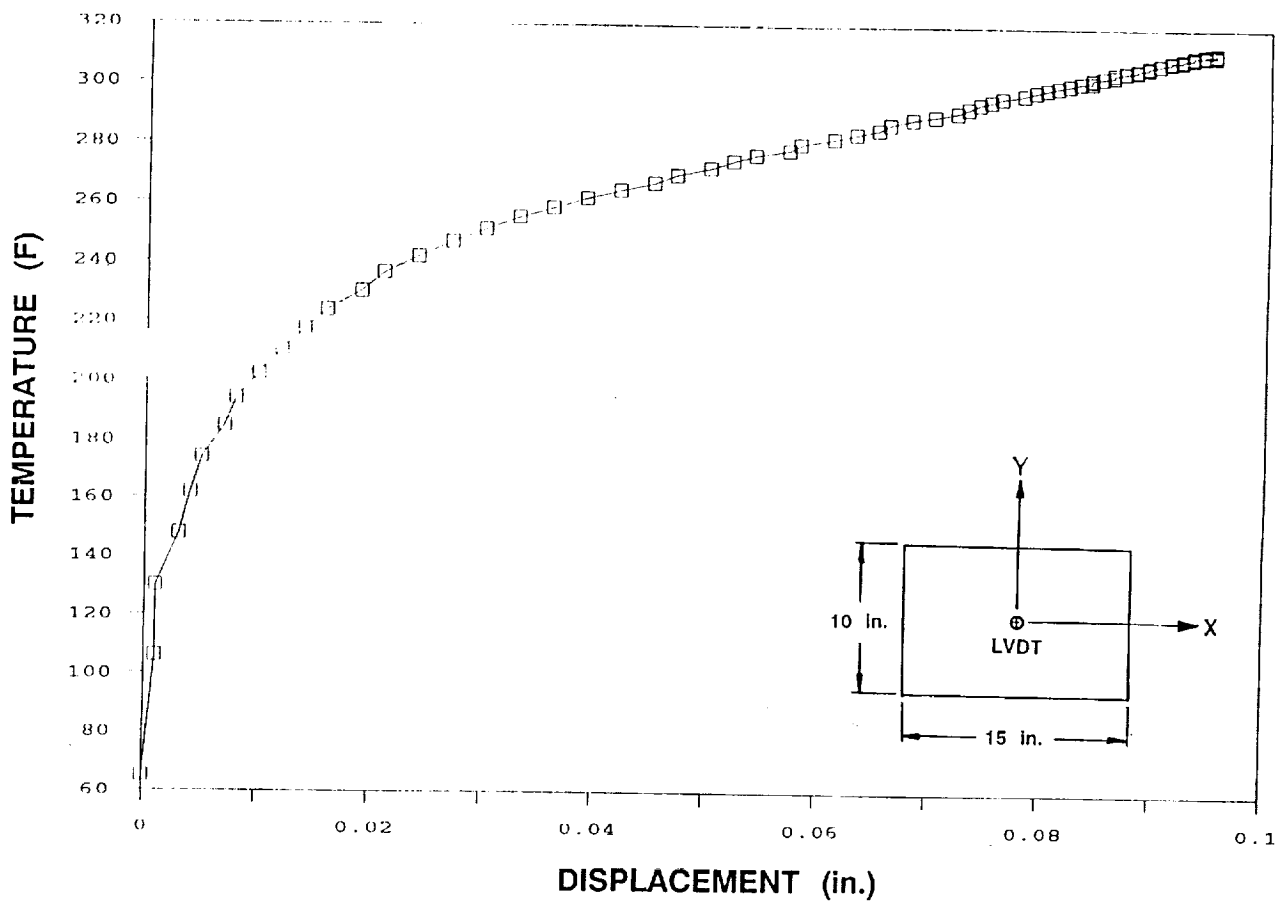


Figure 18

CONCLUDING REMARKS

Recent progress of a research program focused on understanding the thermoviscoplastic behavior of structural panels is described. The program has three elements: (1) finite element simulations of nonlinear material and geometric behavior, (2) experimental determination of parameters for Bodner-Partom constitutive models of panel materials, and (3) thermal-structural tests of panels subjected to localized heating. This paper highlights finite element plane stress computations with inelastic material behavior for different experimental conditions. The panel test program is described and preliminary test results are presented.

This research program represents only a small portion of the current activities of the Light Thermal Structures Center. There are currently 16 active structures and materials research projects supported by the UVA Academic Enhancement Program. Thirteen faculty members from four departments are involved. Approximately 25 graduate students are supported by the university funded research. There are also a substantial number of closely related funded research projects. Current and future research is focused on the development of new materials and their application in structural components with thermal gradients.

- CURRENT RESEARCH ACTIVITIES IN TVP BEHAVIOR OF STRUCTURAL PANELS DESCRIBED:
  - -FINITE ELEMENT TVP ANALYSES HIGHLIGHTED
  - -EXPERIMENTAL PROGRAM OUTLINED

- UVA LIGHT THERMAL STRUCTURES CENTER
  - -ACADEMIC ENHANCEMENT PROGRAM SUPPORTS FACULTY/STUDENT RESEARCH
  - -FOCUSED ON INTEGRATION OF NEW MATERIALS WITH THERMAL STRUCTURES APPLICATIONS

Figure 19

REFERENCES

1. Thornton, E. A.; Oden, J. T.; Tworzydlo, W. W.; and Youn, S. K.: Thermo-Viscoplastic Analysis of Hypersonic Structures Subjected to Severe Aerodynamic Heating. Journal of Aircraft, Vol. 27, No. 9, Sept. 1990, pp. 826-835.

2. Pandey, A. K.; Dechaumphai, P.; and Thornton, E. A.: Finite-Element Thermo-Viscoplastic Analysis of Aerospace Structures. Proceedings of the First Thermal Structures Conference, University of Virginia, pp. 169-189. To appear as a volume in the AIAA Progress in Astronautics and Aeronautics series.

3. Thornton, Earl A.; and Kolenski, J. D.: Viscoplastic Response of Structures with Intense Local Heating. AIAA Paper No. 91-1149, AIAA/ASME/ASCE/AHS/ASC 32nd Structures, Structural Dynamics and Materials Conference, April 8-9, 1991, Baltimore, MD.

4. Bodner, S. R.: Review of a Unified Elastic-Viscoplastic Theory. *Unified Constitutive Equations for Creep and Plasticity*, A. K. Miller, Editor, Elsevier Applied Science Publishers, 1987.

5. Heldenfels, R. R.; and Roberts, W. M.: Experimental and Theoretical Determination of Thermal Stresses in a Flat Plate. NACA TN 2769, 1952.

6. Gossard, M. L.; Seide, P.; and Roberts, W. M.: Thermal Buckling of Plates. NACA TN 2771, 1952.

ACKNOWLEDGEMENT

The author is pleased to acknowledge research support from NASA Langley under grant numbers NAG-1-1013 (Allan R. Wieting, Technical Monitor) and NAG-1-745 (James H. Starnes, Jr., Technical Monitor). Additional support is acknowledged from the University of Virginia Academic Enhancement Program through the Center for Light Thermal Structures.

N92-25923

Modeling "Brittle" High-Temperature Composite Structures

K. T. Kedward, R. M. McMeeking and S. Janson
University of California
Santa Barbara, California

PRECEDING PAGE BLANK NOT FILMED

MODELING "BRITTLE" HIGH-TEMPERATURE COMPOSITE STRUCTURES

A major inhibitor to the projected application of brittle matrix composite structures, such as ceramic matrix composites (CMC), to advanced, high temperature airframes is premature matrix microcracking. However, the phenomenon has been experienced with polymer matrix composite (PMC) systems that are now in an advanced stage of development. Consequently, the subsequent mechanical behavior of these composite systems must be predictable which demands a more complete understanding of the mechanisms such as nonlinear features, localized breakdown of the matrix, interface and fiber phases.

To establish the need for such developments we will commence with a review of the key differences between Polymer, Ceramic as well as Metal Matrix Composites (MMC). Next some lessons learned from the development of the most mature PMC structures will be discussed to indicate how this experience may be carried forward to assist the development on high temperature composites subjected to monotonic and cyclic loads as discussed and selected areas for future technical development are indicated.

FIGURE 1

ENGINEERING OBJECTIVE OF ADVANCED COMPOSITES R & D

- High Specific Stiffness
- High Specific Strength
- Adequate Fracture Toughness & Damage Tolerance
- Adequate Fatigue Resistance & Durability

⇒

- Superior Product/ System Performance

FIGURE 2

THE MATERIAL PROPERTIES PRIMROSE PATH

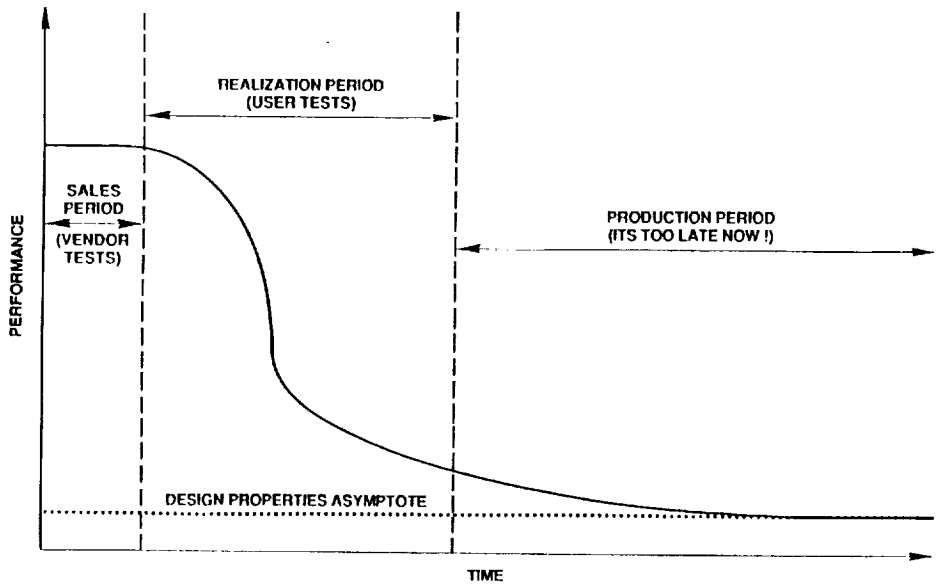


FIGURE 3

SUCCESSFUL APPLICATIONS OF MMC - SPACE SHUTTLE TUBULAR STRUTS

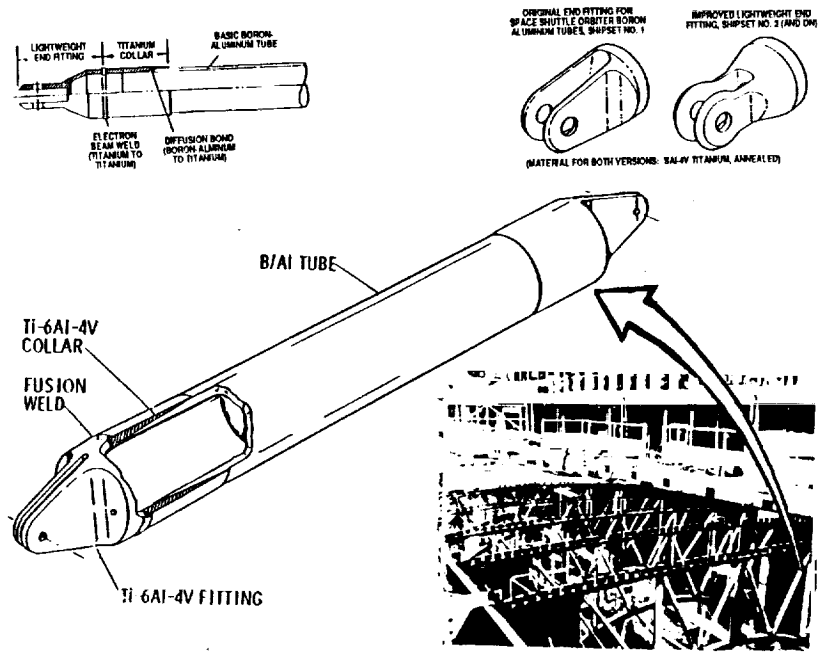


FIGURE 4

PMC vs CMC vs MMC COMPOSITE DESIGN

	PMC	CMC	MMC
UD Material Usage	Not recommended (handling & gen'l damage tolerance concerns)	Not recommended (handling & gen'l damage tolerance concerns)	Can prove feasible for dominant axial loads
Nonlinear Effects	Usually not important	Nonlinearity due to matrix breakdown and interface failure	Nonlinearities due to plasticity and failures in matrix/interface
Temperature Capability	Poor	Good	Moderate
Fatigue	Not major issue (if design avoids out-of-plane loads)	Potential concern	Potential concern
Stiffness, Strength CTE Anisotropy	Can lead to significant problems due to coupling, Joining, etc.	Moderate concern due to shear strength interface variability, and residual stress	Not major issue

FIGURE 5

POTENTIAL OUT-OF-PLANE FAILURES IN INTEGRALLY-STIFFENED TORQUE BOX DESIGN

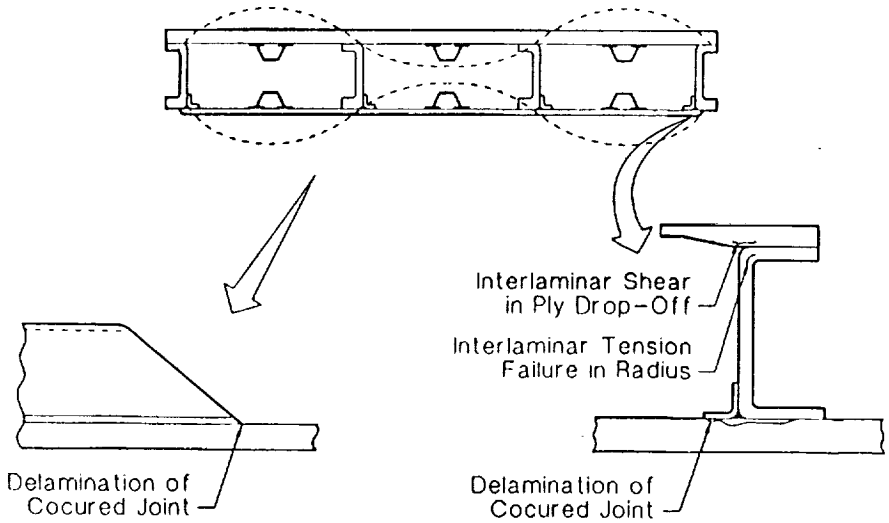
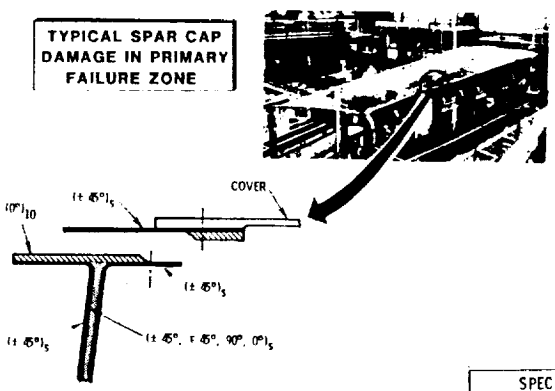


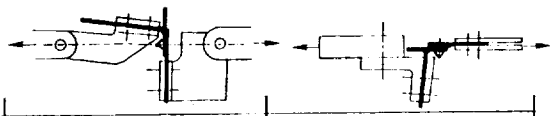
FIGURE 6

DETERIORATION HAS BEEN MEASURED ON FULL SCALE COMPOSITE HARDWARE

(Reference 1)



L-1011 composite vertical fin static test failure at 98% design ultimate load

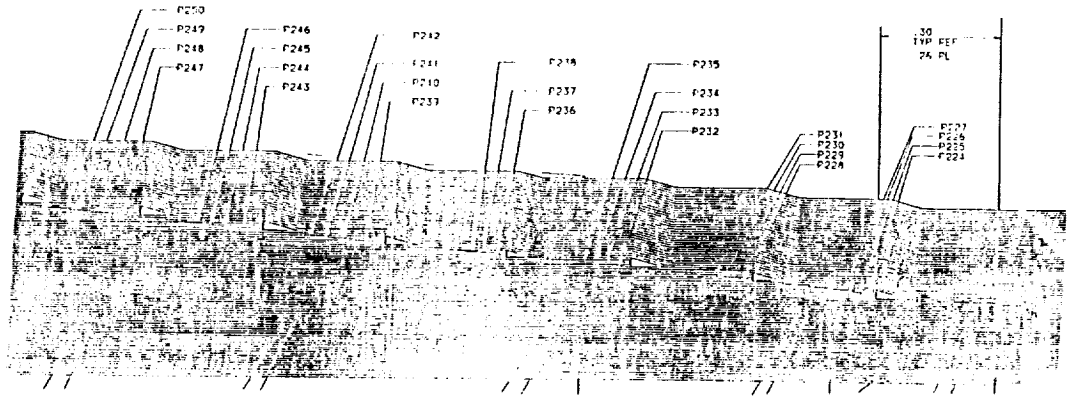


SPECIMEN CONDITIONING	INTERLAMINAR TENSION (LB./FASTENER)	TRANSVERSE TENSION (LB./IN.)
NO PRIOR LOADING	88	445
SEGMENT OF SPAR OF FAILED GROUND TEST UNIT	34	256
PRIOR LOADING EQUAL TO GROUND TEST UNIT	56	225

L-1011 composite vertical fin- influence of load cycling on interlaminar strength

FIGURE 7

LAMINATE TRANSITION



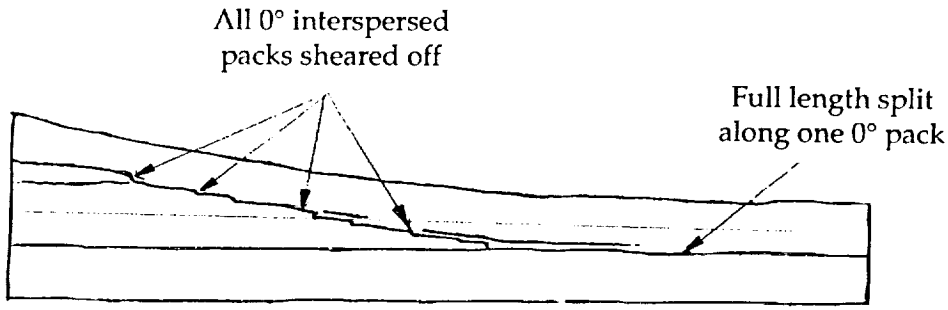


FIGURE 8

LOADS ON TAPERED FLANGE

- Axial tension load in the continuous outer plies at the flange taper require radial (σ_r) stresses for equilibrium.

$$\sigma_r = \frac{\sigma_t \, t}{R}$$

- A series of internal ply drops (typically 4 - 12 mil plies) in the tapered region provide interlaminar stress concentration sites from which delaminations can propagate.

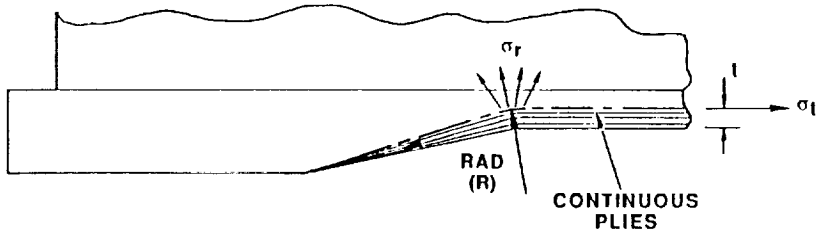


FIGURE 9

IMPEDIMENTS AND DANGERS

- Attitude
- Lack of Broad Systems Viewpoint
- Rigid, Formalized Design Rules
  e.g. Thermoset - to - Thermoplastic Matrix
  Polymer - to - Ceramic Matrix
- Appreciations of Sources of Delamination and Effects of Anisotropy (Stiffness, Strength, Thermal, etc.)

FIGURE 10

"BRITTLE MATERIALS, WHICH FAIL SUDDENLY WITH NO PLASTIC ELONGATION, SHOULD NEVER BE USED FOR AIRCRAFT STRUCTURAL MEMBERS"

David J. Peery "Aircraft Structures"

McGraw - Hill, 1950, Page 302

FIGURE 11

TOUGHNESS IN "BRITTLE" COMPOSITE SYSTEMS

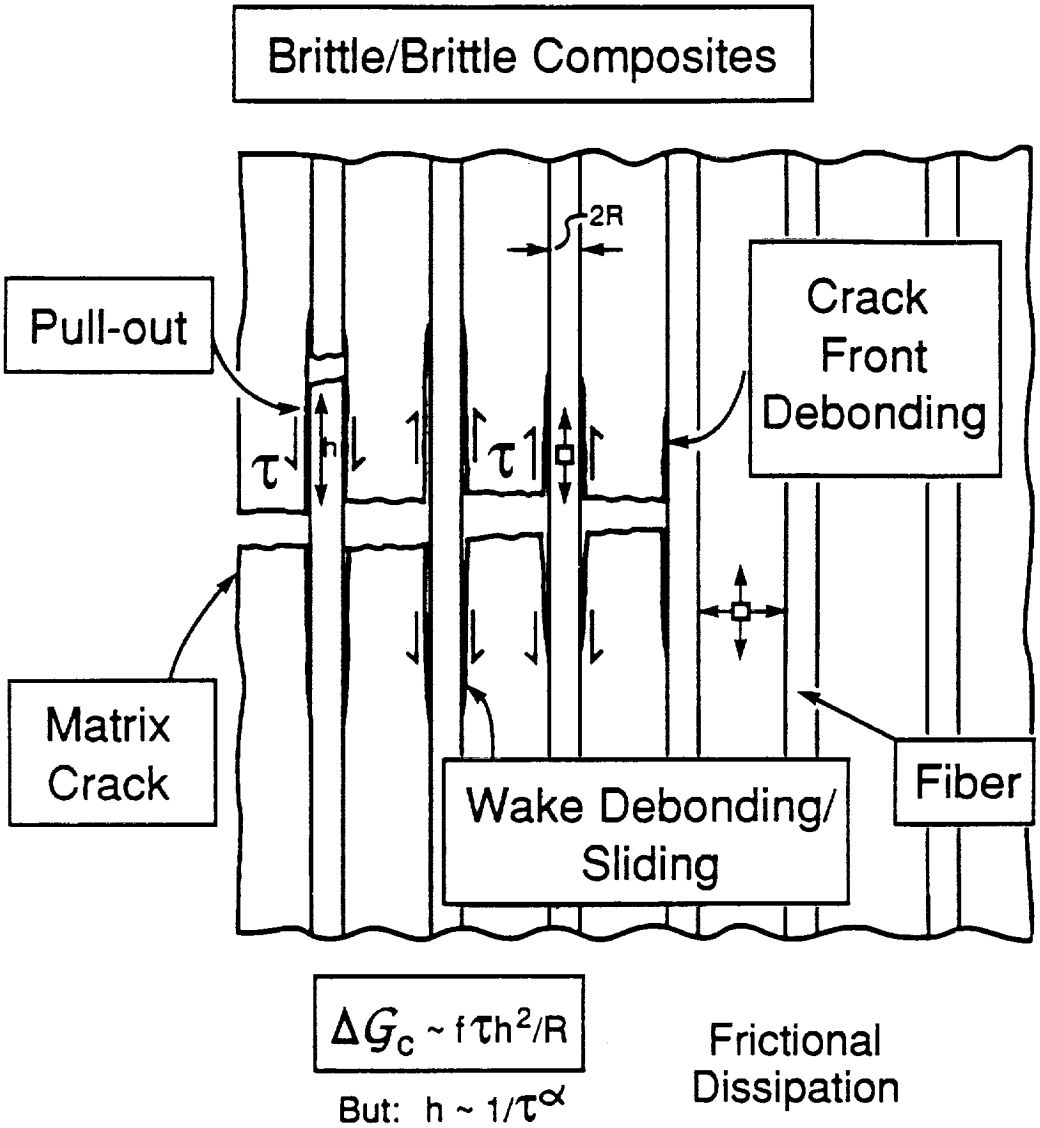


FIGURE 12

THE EFFECT OF A "WEAK" INTERFACE ~ LONGITUDINAL TENSION

'Weak' Interface

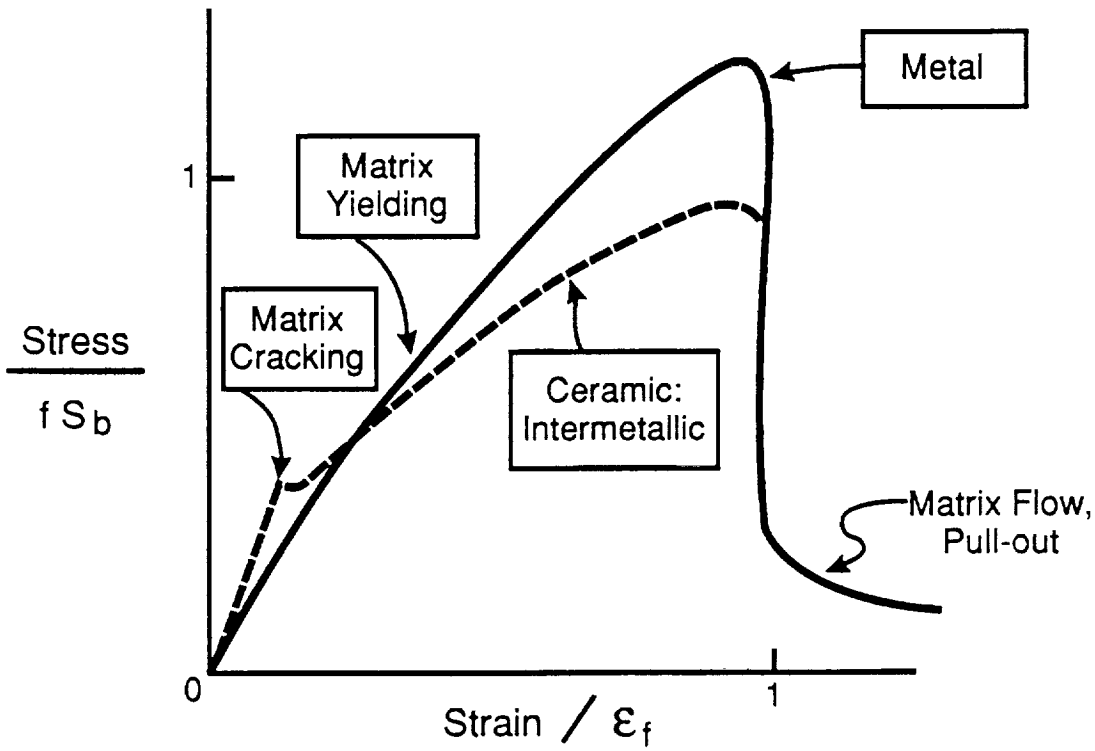
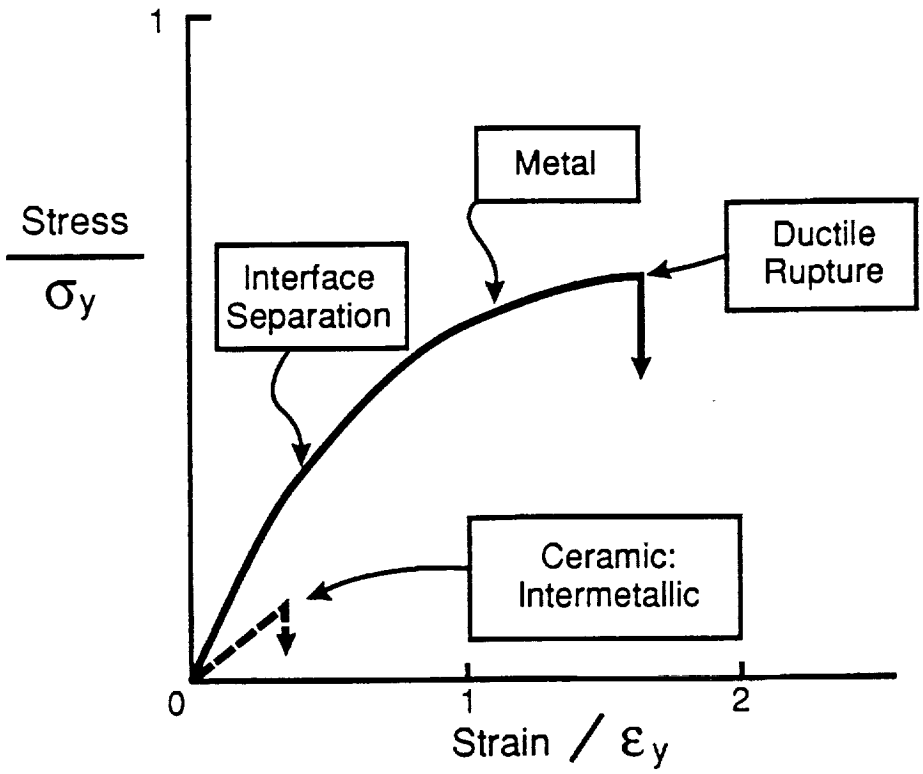


FIGURE 13

THE EFFECT OF A "WEAK" INTERFACE ~ TRANSVERSE TENSION

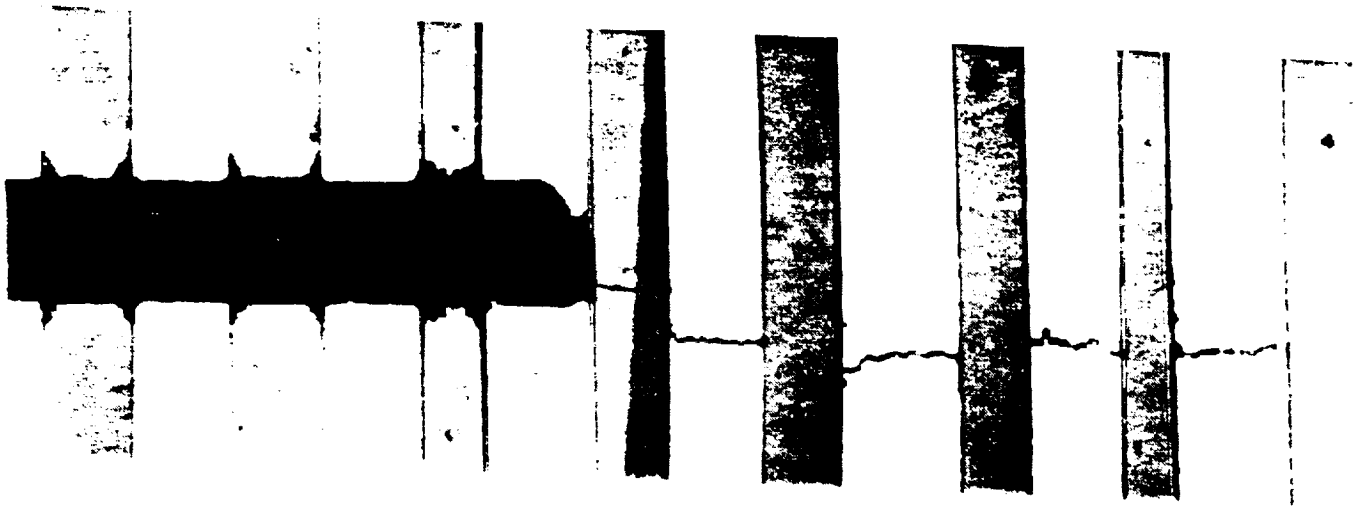
'Weak' Interface



Ceramic/Intermetallic Composites Must Be Laminated/Woven

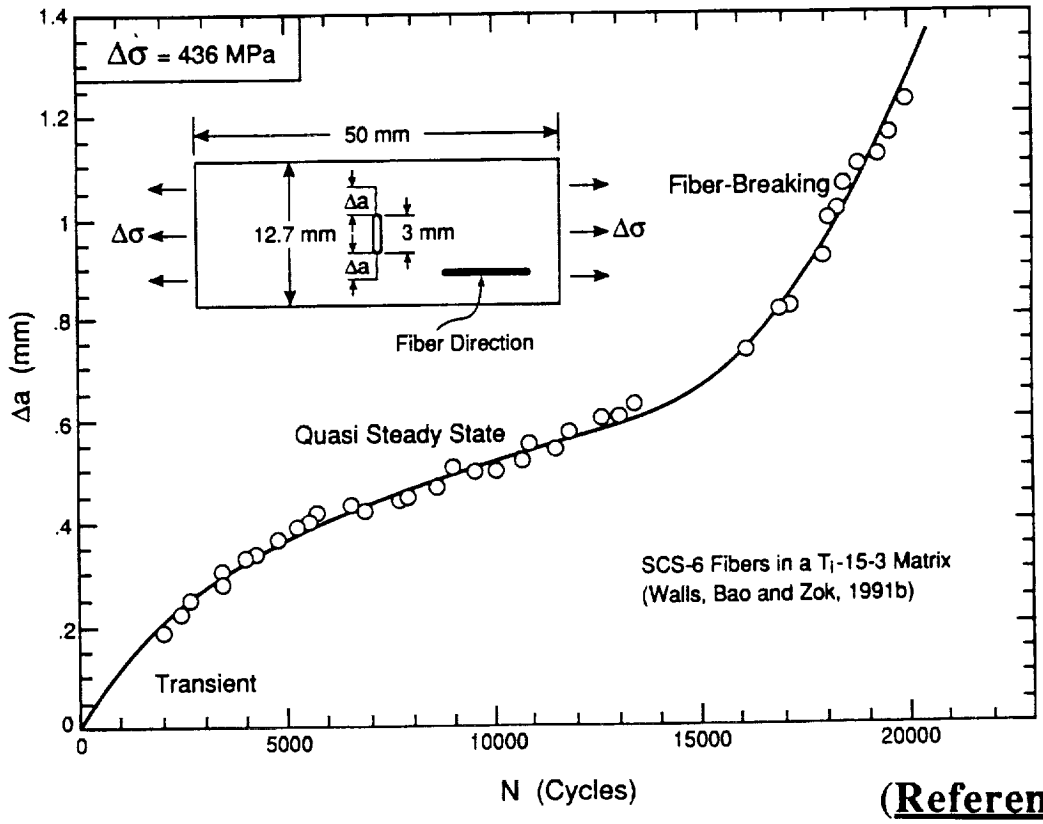
FIGURE 14

CRACK GROWTH FROM A NOTCH



Wright (GE)

(Reference 2)



(Reference 3)

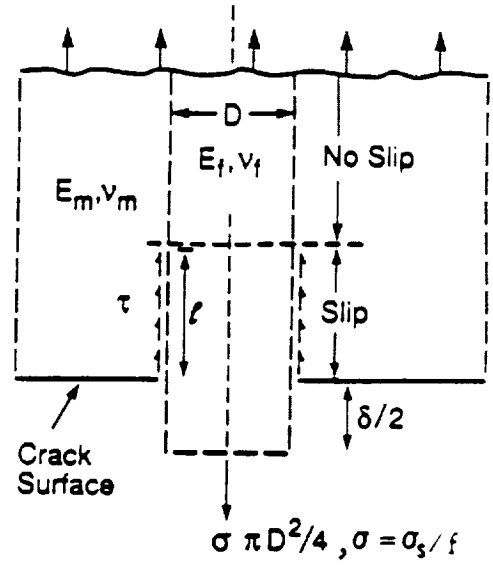
FIGURE 15

THE MECHANISM OF FIBER BRIDGING

BRIDGING LAW

$$\delta = \lambda \sigma_s^2, \ \lambda = \frac{D(1-f)E_m}{4E_f E f^2 \tau}$$

$$\Delta\delta = \pm\frac{1}{2}\lambda(\Delta\sigma_s)^2$$



PARIS LAW

$$da/dN = \beta(\Delta K_{tip}/E_m)^n$$

SHIELDING

$$\Delta K_{tip} = \Delta K_A - \Delta K_F$$

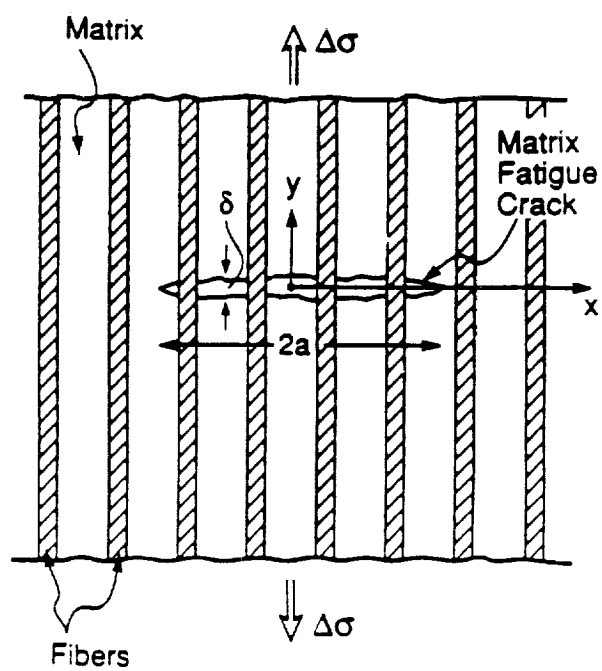
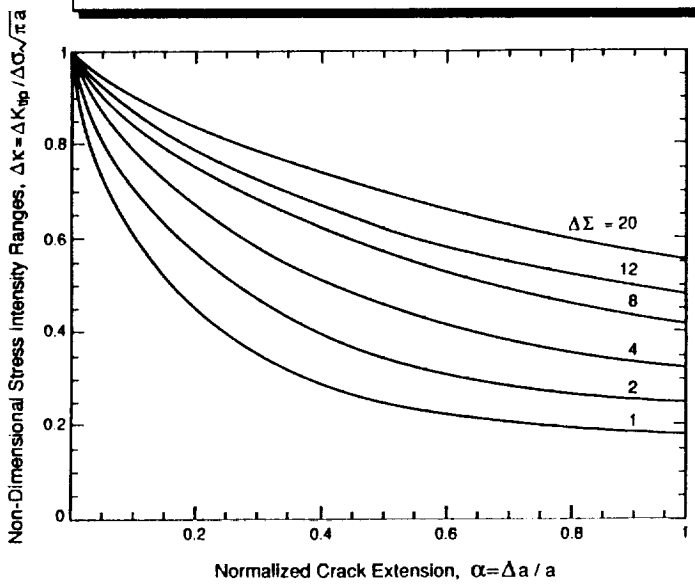


FIGURE 16

A MODEL FOR STRESS INTENSITY AT THE CRACK TIP



$$\Delta K_{tip}/\Delta K_A = \exp(-\sin\alpha\,[A(\Delta\Sigma)+B(\Delta\Sigma)\alpha+C(\Delta\Sigma)\alpha^2]/\alpha^{1/4})$$

FIGURE 17

CRACK EXTENSION RATES AT INCREASING STRESS AMPLITUDES WITH τ = 45 MPA

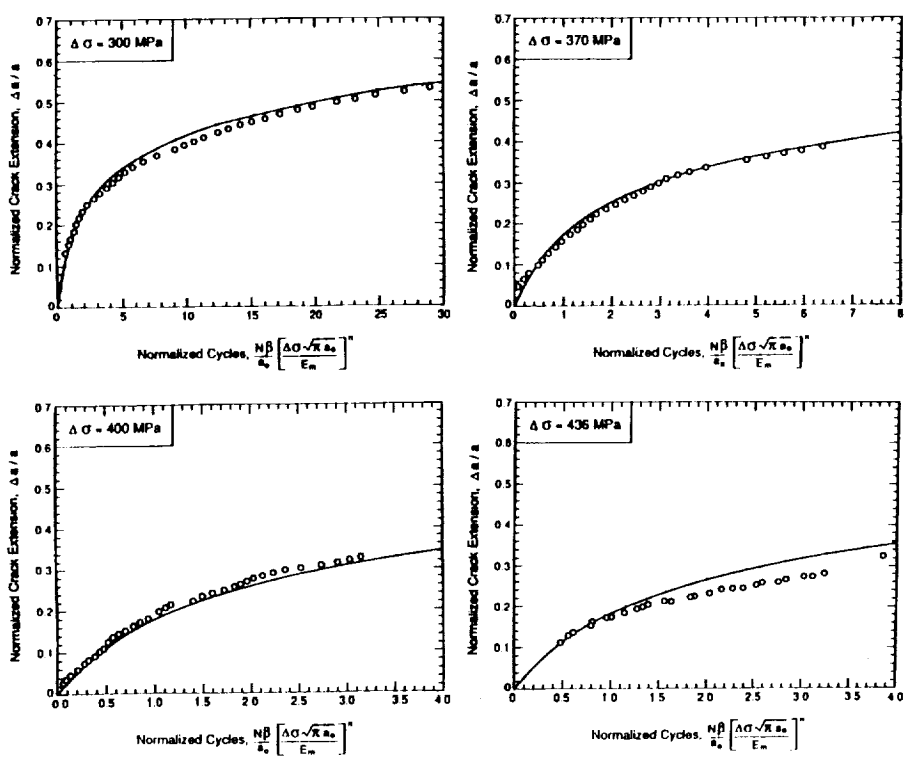
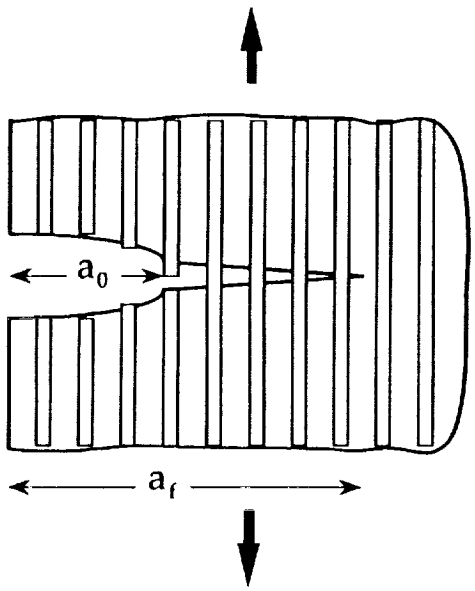


FIGURE 18

EFFECT OF THE ONSET OF FIBER FAILURE

$\sigma_{max} = \Delta\sigma / (1 - R)$

$(\sigma_s)_{max} / f = S$



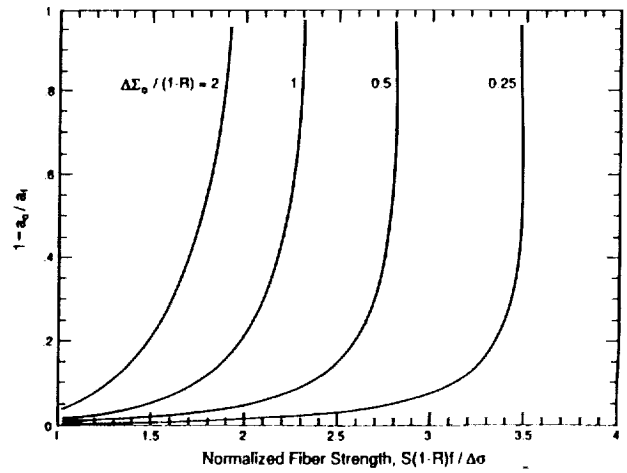


FIGURE 19

FATIGUE CRACKING WITH FIBER FAILURE (1)

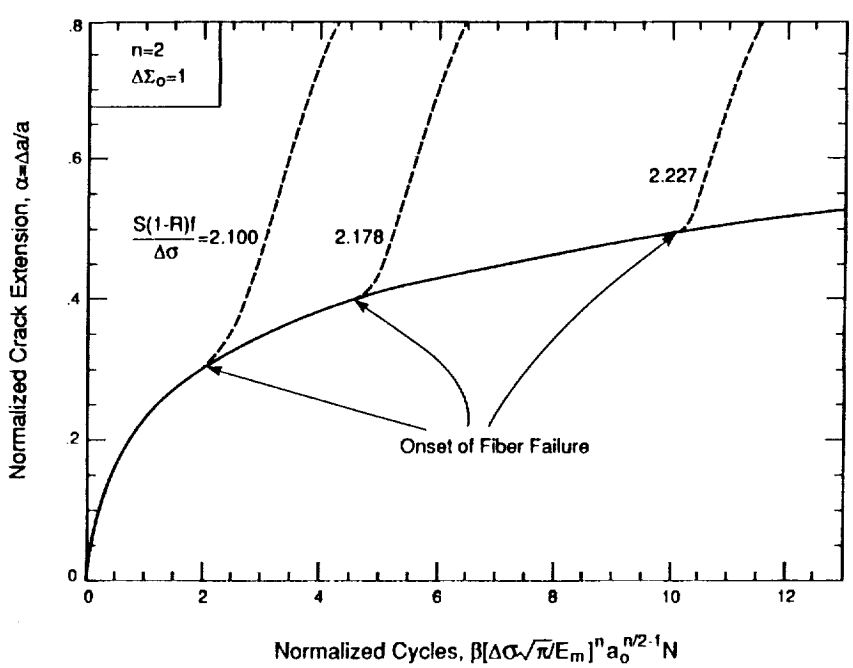


FIGURE 20

FATIGUE CRACKING WITH FIBER FAILURE (2)

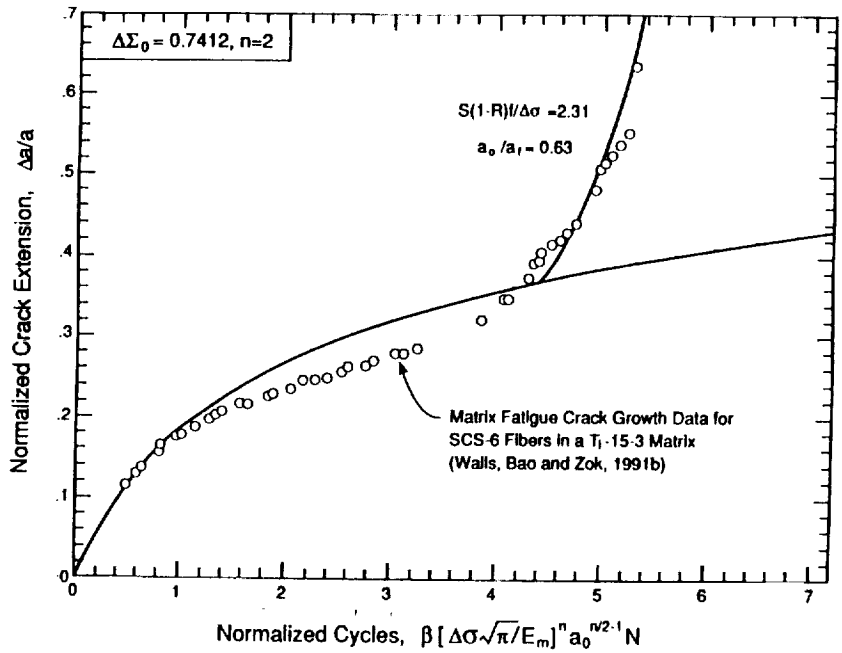


FIGURE 21

SUMMARY

- Cautioned the tendency for extrapolating PMC expertise to high temperature systems.
- Identified need for enhancement in our approach to design detail evaluation for most mature PMC's.
- Discussed fatigue modeling/evaluation technique for "brittle" high temperature composite that can be integrated into general purpose finite element codes.
- Analysis and software development will be required to refine the treatment of high temperature composites including:
  - more complex fiber architectures
  - refined matrix fatigue characterization
  - address potential changes of failure mode
  - consider implications of fiber strength and interface strength variations
  - effect of variable load amplitude

REFERENCES

1. Bohon, H.L.; Chapman III, A.J.; and Leybold, H.A.: Ground Test Experience with Large Composite Structures for Commercial Transport. NASA TM-84627, March 1983.

2. Wright, P.K.; Nimmer, R.; Smith, G.; Sensmeier, M.; and Brun, M.: 'Interfaces in Metal-Ceramic Composites.' The Minerals, Metals and Materials Society, 1989, pp. 559-581.

3. McMeeking, R.M.; and Evans, A.G.: Matrix Fatigue Cracking in Fiber Composites. Mechanics of Materials, Vol. 9, 1990, pp. 217-227.

N92-25924

Algorithmic Development in Structures Technology

Frank Sagendorph
General Electric Aircraft Engines
Cincinnati, Ohio

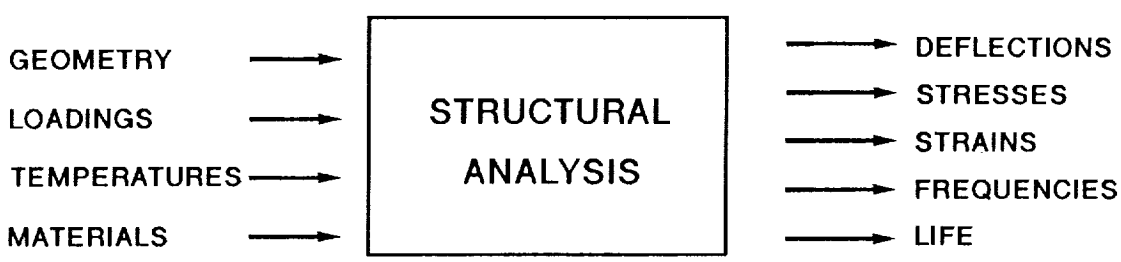
A wide variety of topics about the types of algorithms in use or in development for the design of gas turbine engines will be covered in this report, ranging from why we need better algorithms, to how we would use them. Selected examples will be referred to; by intention, there will be no detailed derivations or equations in this paper. The selection of areas covered is not intended to correspond with an established level of importance, nor is it intended to reflect an official GE Aircraft Engines viewpoint.

Areas to be addressed

- Why development is needed
- How to prioritize it
- What is the computing environment
- Who needs new methods
- A bunch of examples
- The importance of validation
- A personal closing message

The focus of this report will be on the actual structural and life analysis task, ignoring the equally important (and, in some cases, more-so) task of determining the inputs to the analysis.

THE PROCESS



Many people believe that the real developments in structural analysis have been completed and we should move on to other fields. That belief is not correct.

Why we need algorithm development

- We have solved many easy problems
- We have solved many difficult problems
- Many hard problems not yet solved effectively
- Substantial improvements required to handle
  - New materials
  - Increased technical requirements

Business and Technical requirements are drivers

No one can afford to work on everything at the same time. One way to prioritize development activities is to use the matrix shown here. This can be used to rank effects in productivity, capability and accuracy versus the time required to implement them. As always, other factors such as cost and business needs must be considered.

DEVELOPMENT CRITERIA

PURPOSE	← TIMING →		
	IMMEDIATE	12-18 MONTHS	LONG RANGE
IMPROVED PRODUCTIVITY			
INCREASED ACCURACY			
ADDITIONAL CAPABILITY			

Engineers have more computing power available than ever before. UNIX workstations and 386 PC's offer high performance at their desks. Fiber optic networks allow high speed data transfer to and from main frame computers. In many cases, navigating through this environment can be as difficult as actually solving the structural problem.

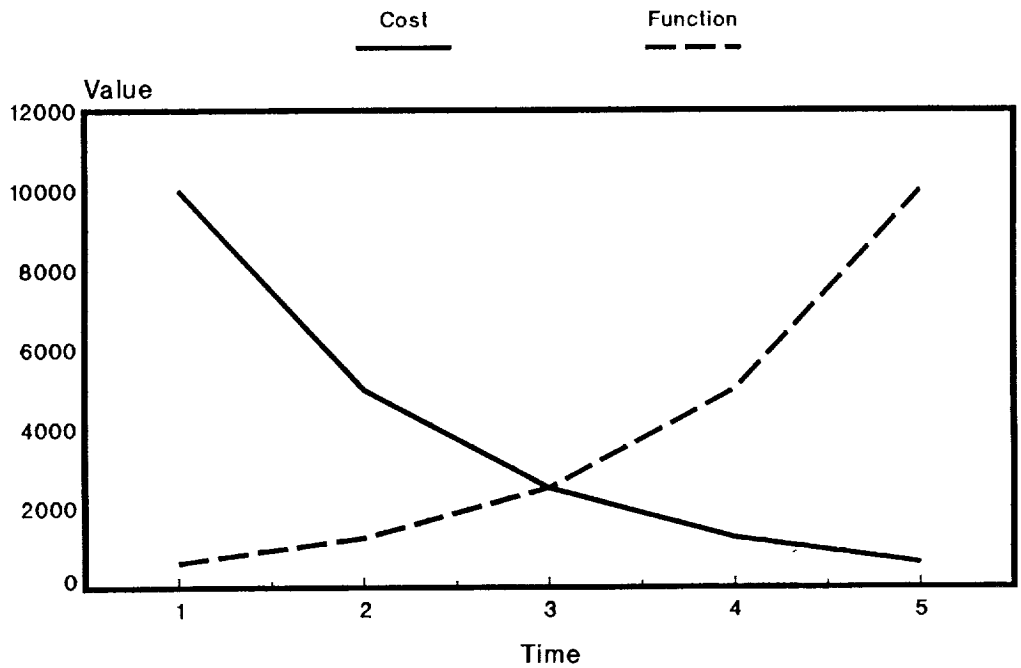
The Computing Environment

- A wide range of processors and operating systems
- Power moving to the desktop
- High speed networks for data access and transfer
- Graphical presentation of input and results demanded
- Gradual transition from FORTRAN toward C
- Cost is always an issue

Algorithms must address Machine Portability

This is really a universal curve. In this instance, it shows the recognized fact that as time goes on, computer hardware costs are going down, while performance for that hardware is going up. Upper management knows this also and frequently asks why the overall cost of computing doesn't go down. It's a good question.

Trends in Cost and Function



Like every other population, there is considerable variation among computer users. They are all, however, trained, knowledgeable people. Some do analysis infrequently, while others devote their entire career to it.

The Users

- Trained, knowledgeable people
- Designers may spend 1 week a month on design/analysis
  - Continual interaction with other disciplines
- Expert users spend all day every day on detailed analysis
  - Focus on solutions to very complex problems
- They both want algorithms that work reliably

Algorithms must address User Needs

Lots of things must fit into place before an algorithm really becomes accepted. We will talk about many of them during this paper. But one fact continually stands out. Successful algorithms have an internal champion.

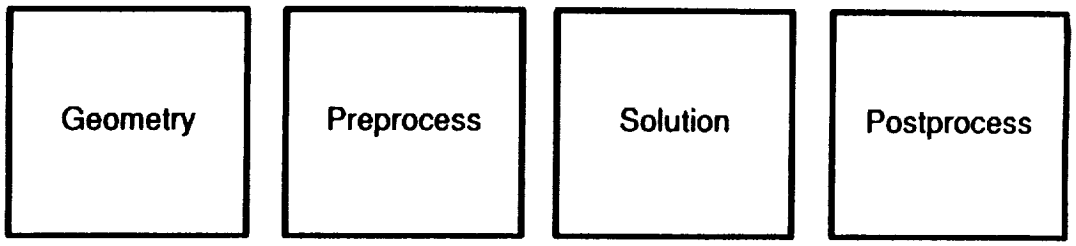
Summary for successful algorithms

- Code which meets user's needs
- Runs on user's equipment
- Robust and reliable
- Integrated with user's work flow
- Well documented
- Training and support structure in place

Requires "Champion" for wide acceptance

These next four charts depict our SIESTA system (System for Integrated Engineering STructural Analysis) which forms a framework for productivity improvements. The first chart shows a "big picture" of the four major phases of structural analysis.

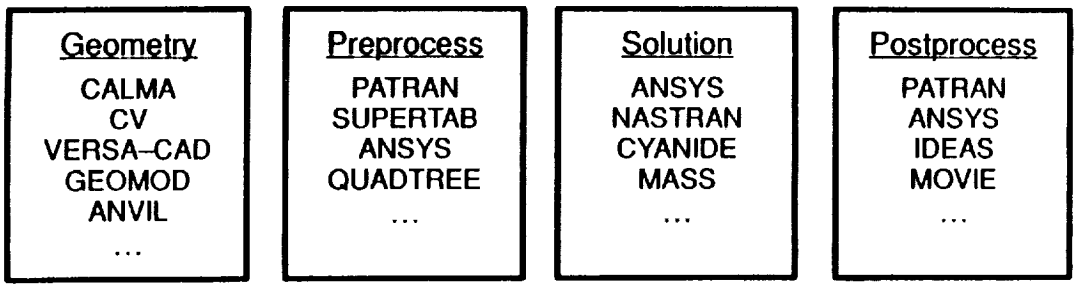
Design Analysis



Major phases of analysis

Actual stress analysis is not so easy because each phase has many programs.

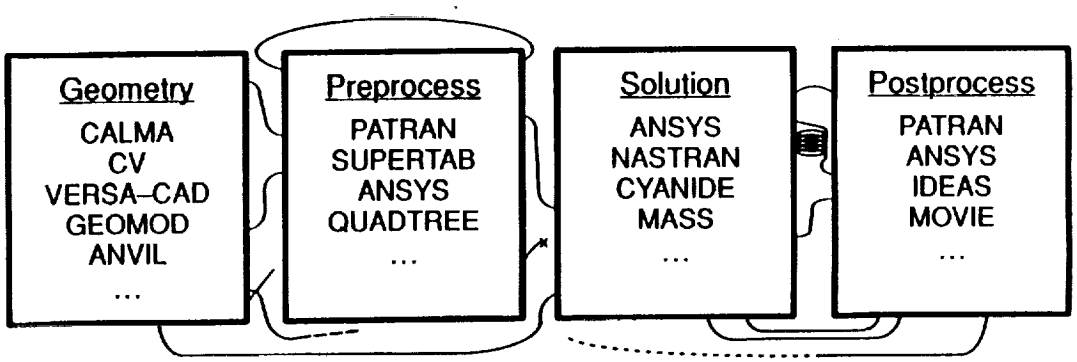
Real Design Analysis



Many different tools are used, but much of the input and output is common.

In real life, going from one phase to the other can be very difficult and sometimes impossible.

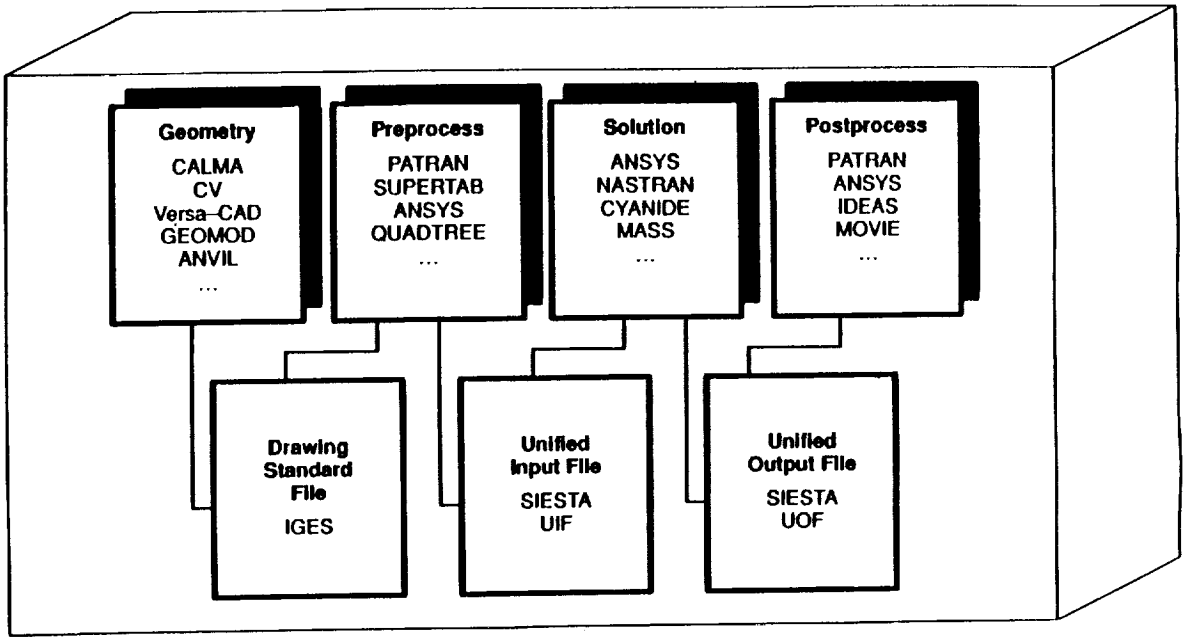
The LA Freeway Design Analysis System



Dead-ends, one-way streets, no feed-back loops

The development of standard interfaces allows the user to easily move from phase to phase and use the appropriate algorithm.

The SIESTA System



SIESTA integrates the tasks and the software.

C-4

Geometry lies at the heart of all structural analysis problems, and improved algorithms for constructing and sharing 3-D geometry are needed.

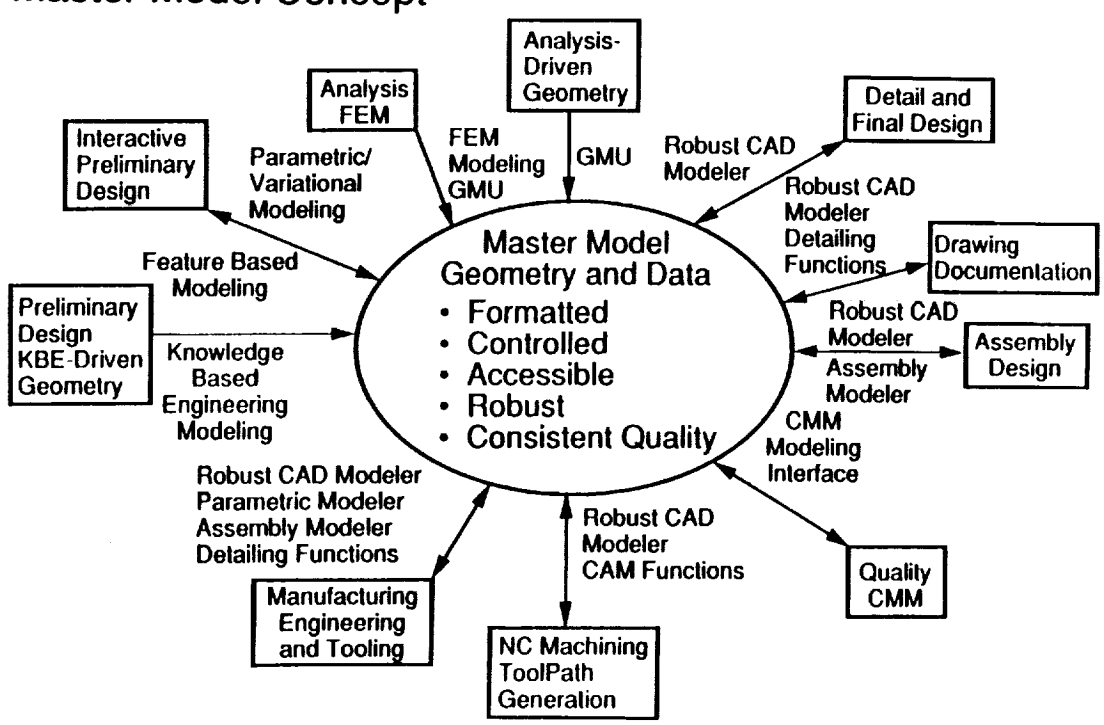
Geometry and the Master Model

- Geometry evolves through the design process
- Information needed by many disciplines
- Drawings not the most effective way to communicate
- Algorithms for 3-D solid and surface definitions
- Parameter driven, feature based systems - Not just new CAD
- Consistent geometry essential for concurrency

A Productivity and Product Issue

A "Master Model" concept, as shown here, means that one geometric definition is the source of information for all functions. Achieving this goal is more difficult than drawing the pictures.

Master Model Concept



Automatic meshing algorithms can have immense effects on productivity. Their acceptance is growing, but because the meshes look so different from our past experience, there is a cultural as well as a technical barrier to overcome.

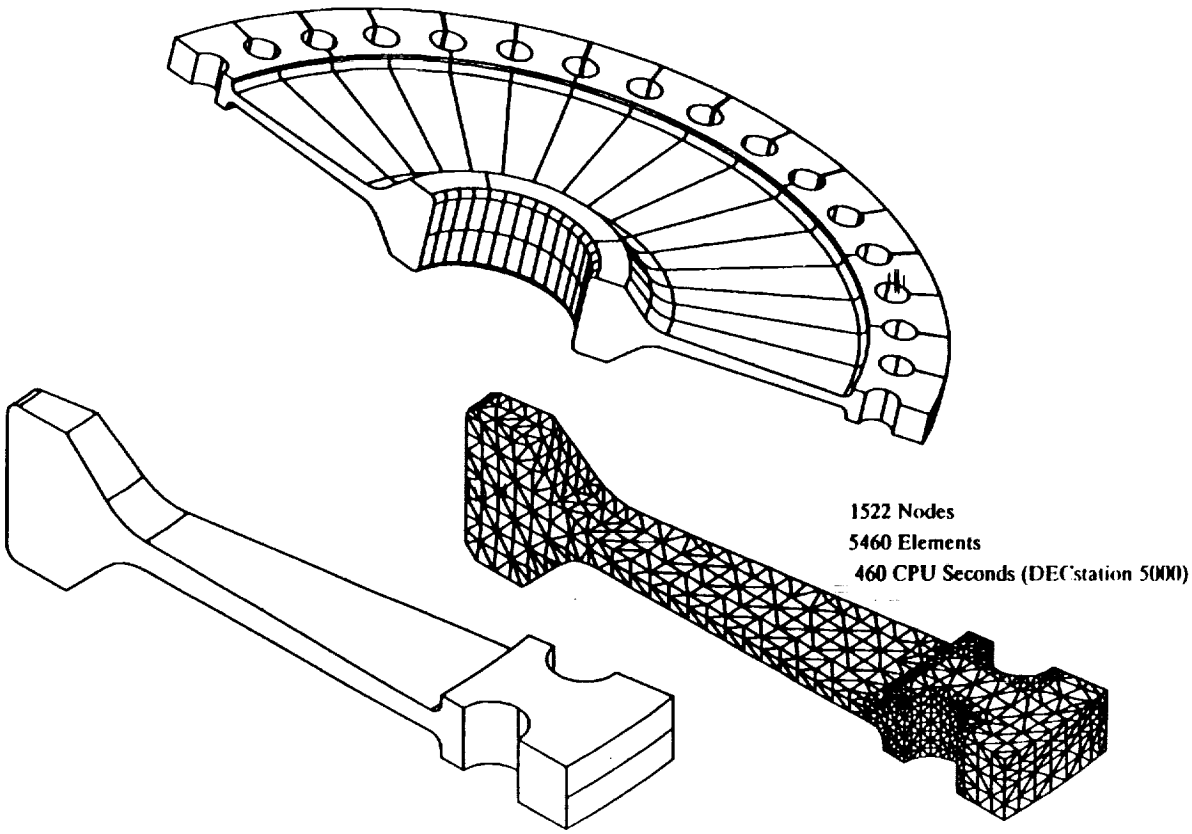
Analytical Modeling Algorithms

- Can be the major roadblock in the design process
  - Too much time means not enough iterations
- Boundary element techniques help, but not all cases
- 3-D mapped mesh techniques still predominate
- Automatic meshing algorithms gaining popularity
  - Produces many tetrahedron elements

A combined Technical and Cultural change

Here is a 3-D example of a model turbine disk, meshed with our internal OCTREE code. It has about three times the number of elements as a conventional mapped mesh would have, but it was easier to produce.

3-D Automatic Mesh Generation



Often, in our eagerness for technical excellence, we forget about the iterative nature of the designer's job. Algorithms to help in this task can be different from those based on tensor algebra, but they are still important.

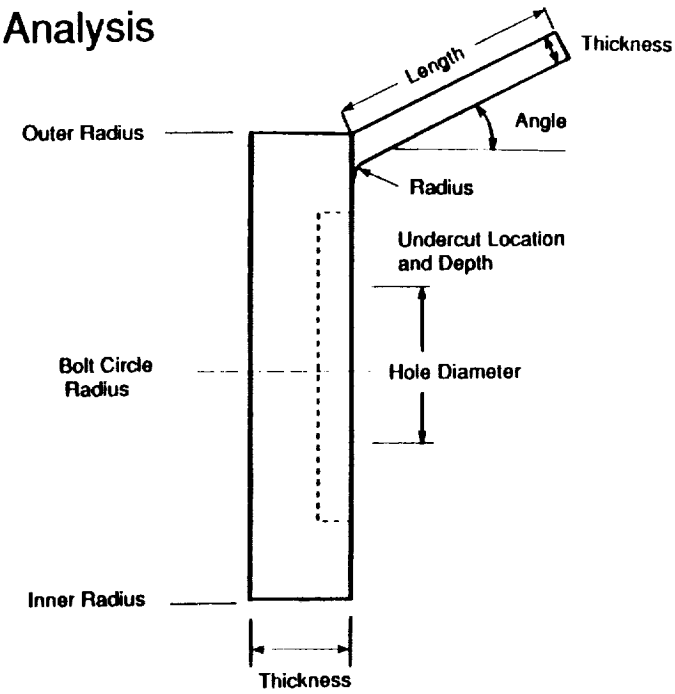
Algorithms for design

- May be based on experimental correlations
- May use actual finite element or boundary models
- Initial models have reasonable element density
- Used for sizing and initial life calculations
- Results shared with other disciplines
- Runs on mainframes or workstations

Used to iterate on a design

As an example, designers need to size bolted flanges for blade-out loads. This chart shows a generic parametric flange definition.

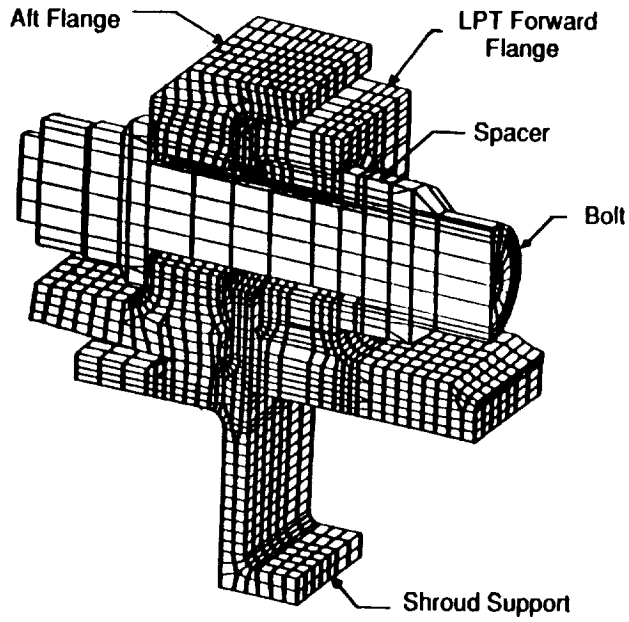
Bolted Flange Analysis



Generic flange description.

Here is a full 3-D finite element model of a bolted flange joint. We used this to confirm the adequacy of our simplified tool. A second phase of this project is to construct most of the 3-D model automatically from the generic input.

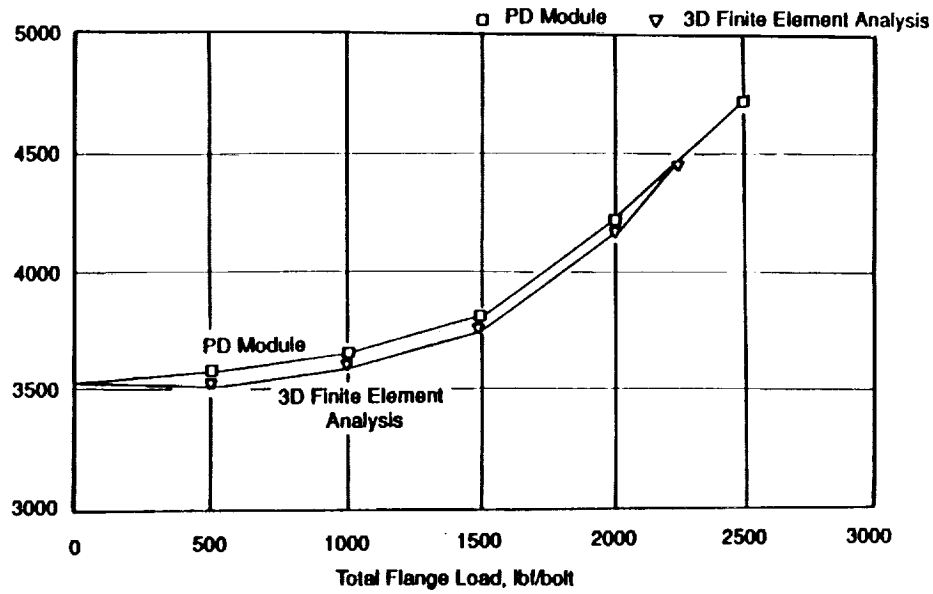
LPT Forward Flange



ANSYS finite-element model.

The generic definition was used as input for a relatively simple finite element analysis to predict bolt load versus applied flange load. We got excellent agreement.

Complex Flange Load



Comparison of PD module and 3D finite-element analyses.

At the other end of the spectrum, some analysts spend all their time making very complex models. These models, which typically require a CRAY to run, include fine detail in high stress regions, as well as contact and friction elements.

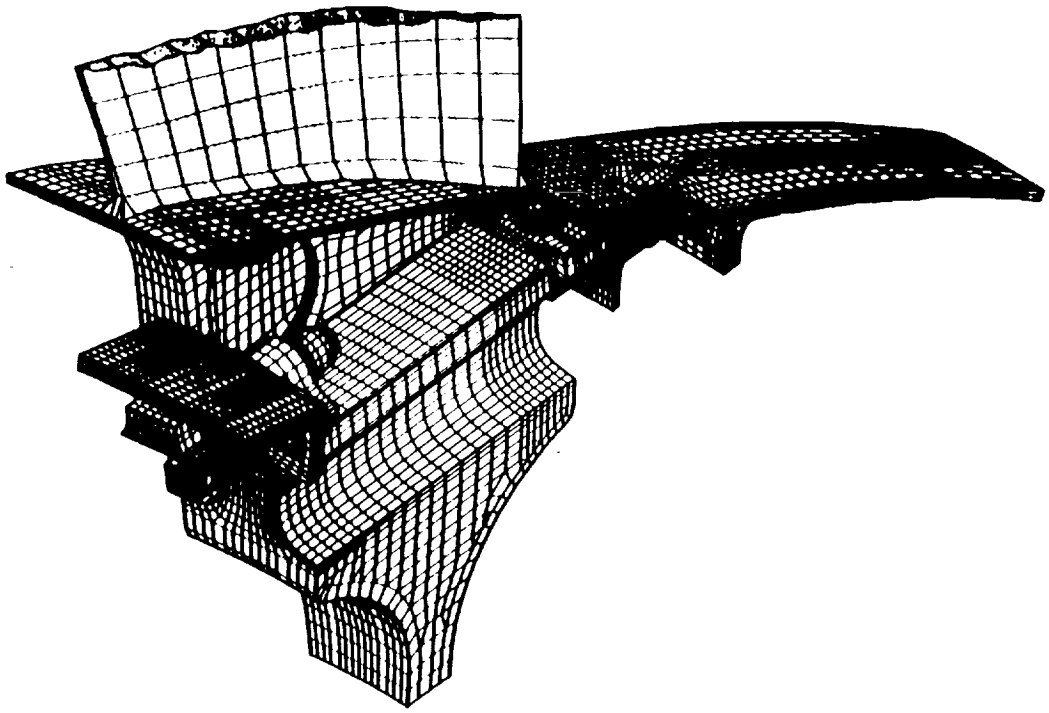
A detailed 3-D finite element model

- Generally produced by "experts"
- Initial model has fine element density
- Successive mesh refinement in key areas
- Routine use of friction and contact elements
- Uses high speed computer for solution

Produced to validate a design

A Typical Detailed 3-D Model

Detailed 3-D Finite Element Model



Instead of asking, "How good is my solution?", users should be encouraged to ask, "How good a solution do I need?". In any event, current work in adaptive solutions and error estimations will provide the answers.

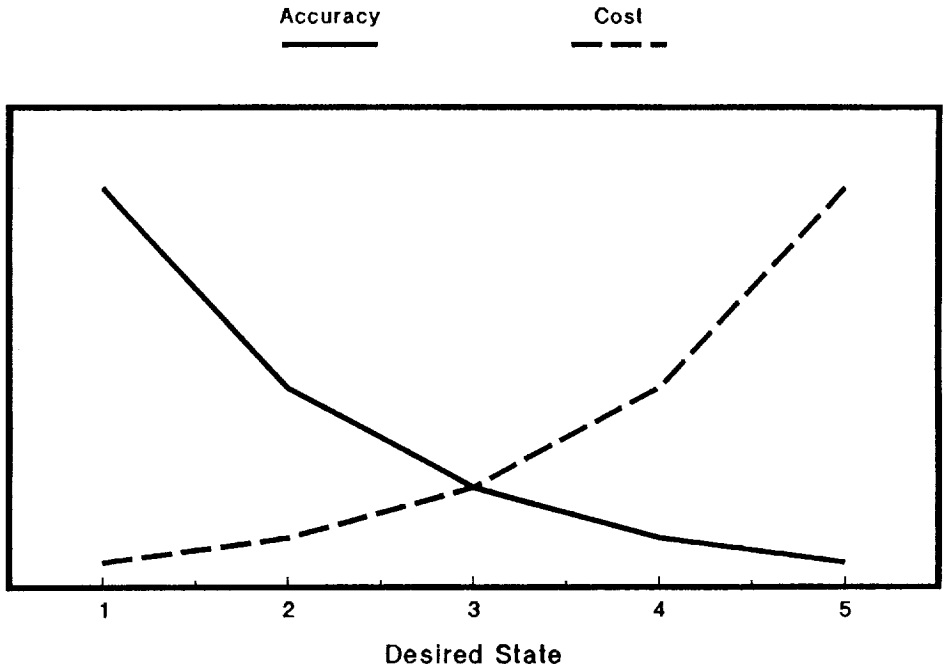
How good is my solution ?

- How good do you need it to be ?
- Mesh density often a matter of experience
- Global convergence studies can be expensive
- Local h-, r-, and/or p- convergence preferred
- Improvement in error estimates needed

Adaptive Analysis Algorithms Are An Answer

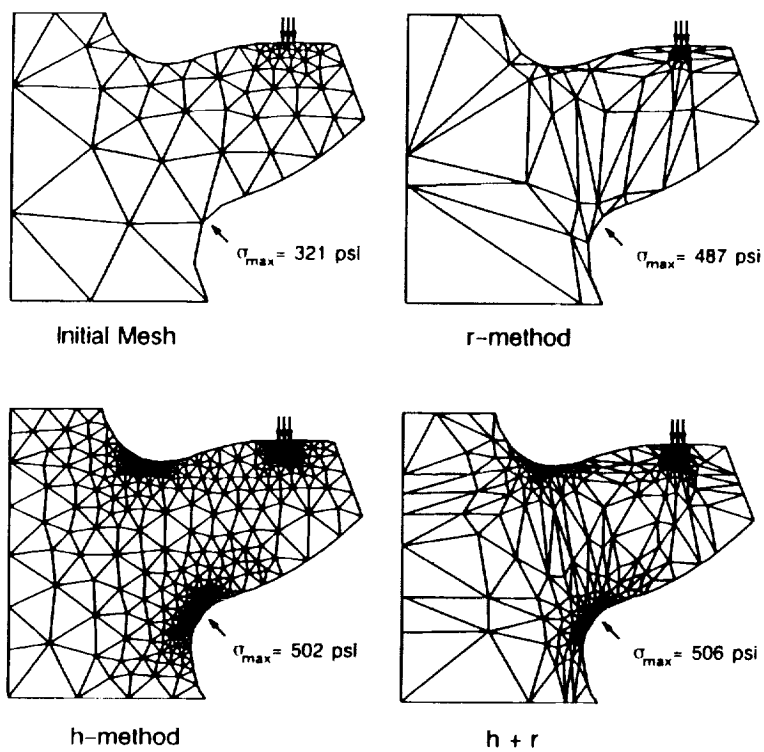
The universal curve again, this time showing that as the accuracy level decreases (with stresses within 1%), the cost increases.

Trends in Cost and Accuracy



Here's an actual example of combined H and R local adaptive analysis. The results agree closely with photoelastic data.

Adaptive Mesh Analysis



Cyclic symmetry algorithms allow users to analyze a complete stage of blades by modeling only a single one. Complex constraint equations handle the mathematics. Sounds easy, but it really isn't.

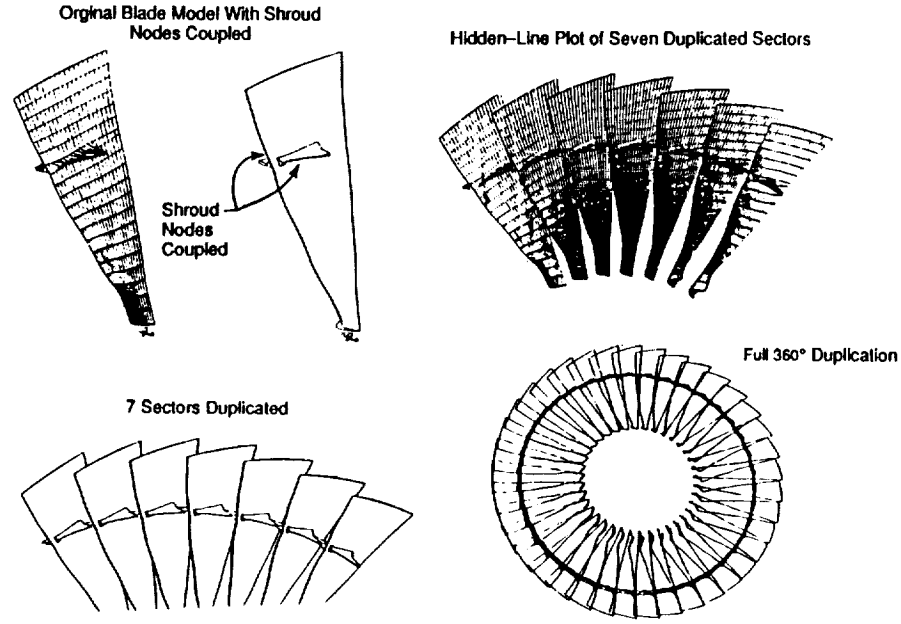
Cyclic symmetry: Expanding the domain

- Use single blade model to predict stage frequencies
- Basic theory goes back to 1918
- Difficult to make model and to interpret results
- New pre & post processing methods simplify user's tasks
- Powerful computers and large scale solution algorithms

Success requires efficient algorithms

Specialized algorithms for pre and post processing have allowed more users to employ this powerful technique.

SIESTA Graphics Duplication



Begins with a single blade model and internally duplicates geometry.

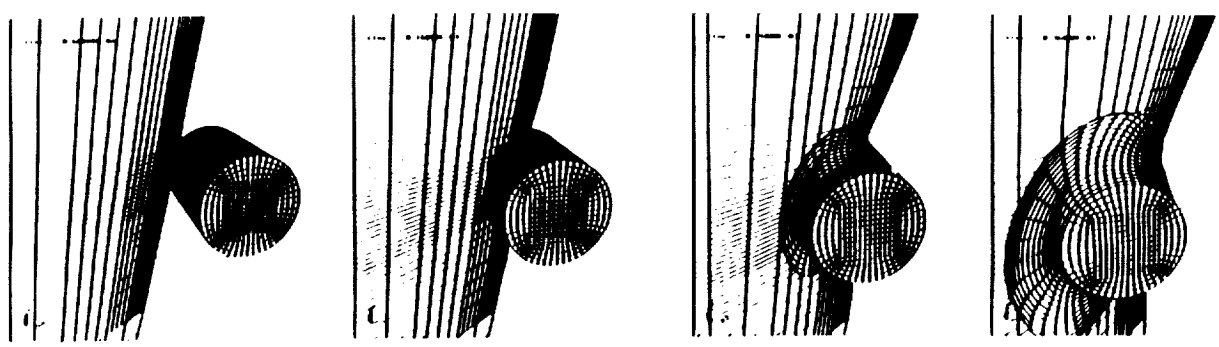
Predicting the effect of bird impact on fan blades through analysis provides cost reductions from the conventional design and test cycle. Unfortunately, commercially available software does not always do the entire job, and additional algorithms must be utilized.

Dynamic impact analysis

- Needed to predict effect of bird impact on fan blades
- Our work based on DYNA3D code from Lawrence Livermore
- Modified for composite materials
- Modified for soft body impact
- Successfully used for impact test predictions
- Not simple task, nor is it completed

Powerful algorithms need powerful understanding

This figure shows the progression of the bird as it impacts and is deformed by the blade. Experimental results show good correlation with strain gage data for this type of analysis.



DYNA3D BIRD-BLADE IMPACT SEQUENCE

As we strive for increased performance and improved thrust to weight, temperatures go up, and the importance of correctly calculating the effects of plasticity and creep increases. Much of our work is based on an internal code, CYANIDE, and the following examples show some of the uses.

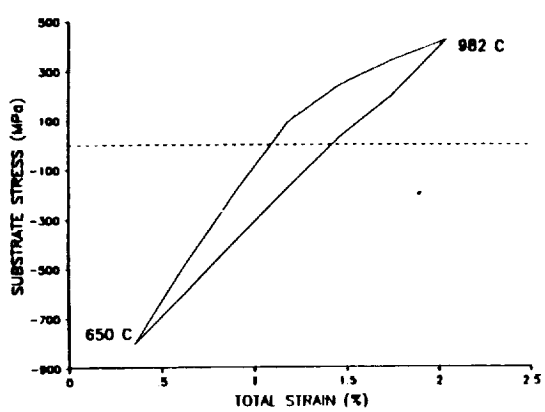
Combined Creep and Plasticity

- Must be accounted for in hot parts design
- Constitutive equations are material dependent
- Limited test data to use for validation
- Issue of 3, 5 or 10 term creep law
- Algorithms need accurate response and fast execution
- Successful analyses based on in-house CYANIDE code

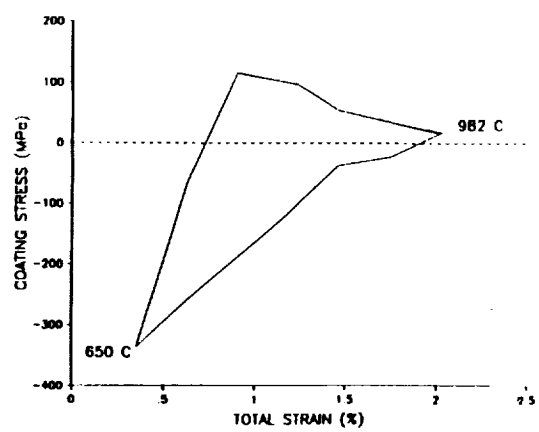
Developments tied to materials and applications

Thermal barrier coatings are frequently used to help protect the basic metal structures. These coatings are themselves a particularly interesting structural analysis problem, primarily because of the mis-match in thermal coefficients between the coating and the base metal. This plot, conducted under a contract for Naval Air Development Center, shows the calculated strains for a complete thermo-mechanical fatigue cycle for both coating and substrate.

Stress–Strain Response for In–Phase TMF Cycle



Rene'80 Substrate



Aluminide Coating

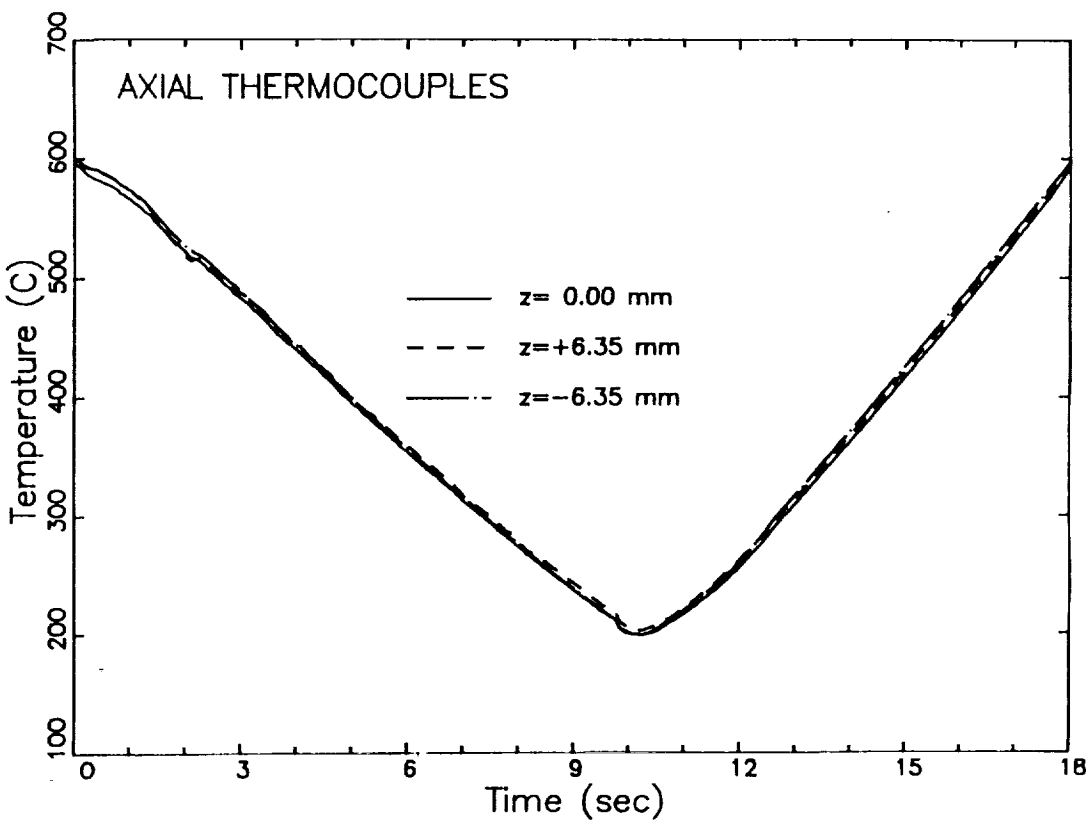
Moving into the life calculation area, the theories become more complex (generally), and validation is more difficult. Additional work in all areas, including global as well as local failure mechanisms, is required. In addition, a representative, coordinated and controlled validation test plan is essential in order to use the new algorithms for production applications.

Algorithms to predict failure mechanisms

- Global mechanisms
  - Limit loads, Buckling, Burst, Progressive Failure, ...
- Local mechanisms
  - Low Cycle Fatigue, Fatigue Crack Growth
- Majority of testing is for simple laboratory conditions
- Components have complex stress/temperature cycling

Validation testing is critical

A particularly interesting validation test concerns thermo-mechanical fatigue cycling. By designing a test facility which can capture the magnitude of the thermal and stress cycles, we believe we will be able to improve our understanding of the degree of conservatism in our designs.



Another needed area of development concerns the applicability of linear elastic fracture mechanics for high temperature and stress environments. A current contract with NASA Lewis on Elevated Temperature Crack Growth is providing data and correlation for certain aspects of this phenomenon. Additional work will be required to provide a full capability for engine structures.

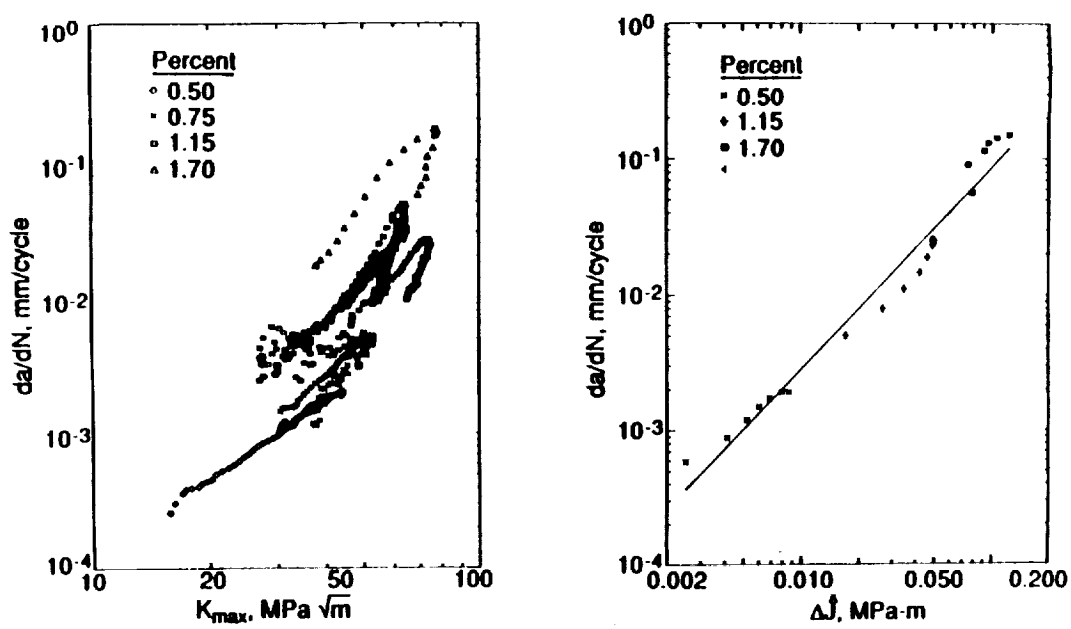
Non-Linear Elastic Fracture Mechanics (NLEFM)

- Materials are being used in high temperature/stress regimes
- In these regimes, LEFM predictions may not be accurate
- NASA contract identifying parameters for through cracks
- Engine components require extension for surface cracks
- Virtual crack extension or Path independent integrals ?

Localized stress relaxation can be important

This plot shows the correlation mentioned for through cracks in Alloy 718. The strain levels in the tests were high enough to induce bulk inelastic deformation of the specimens. The analytical results are based on the path independent integral proposed by Kishimoto. The data could not be correlated using conventional LEFM.

Alloy 718 da/dN at 538°C



The area of composite materials offers considerable promise for designing lightweight/high strength engine components. It also offers considerable challenges in order to establish a set of validated algorithms which account for the detailed stress and strain calculations as well as the "randomness" of the manufactured material. When viewed in conjunction with the wide variety of failure mechanisms exhibited by composites, the analytical challenges are especially high.

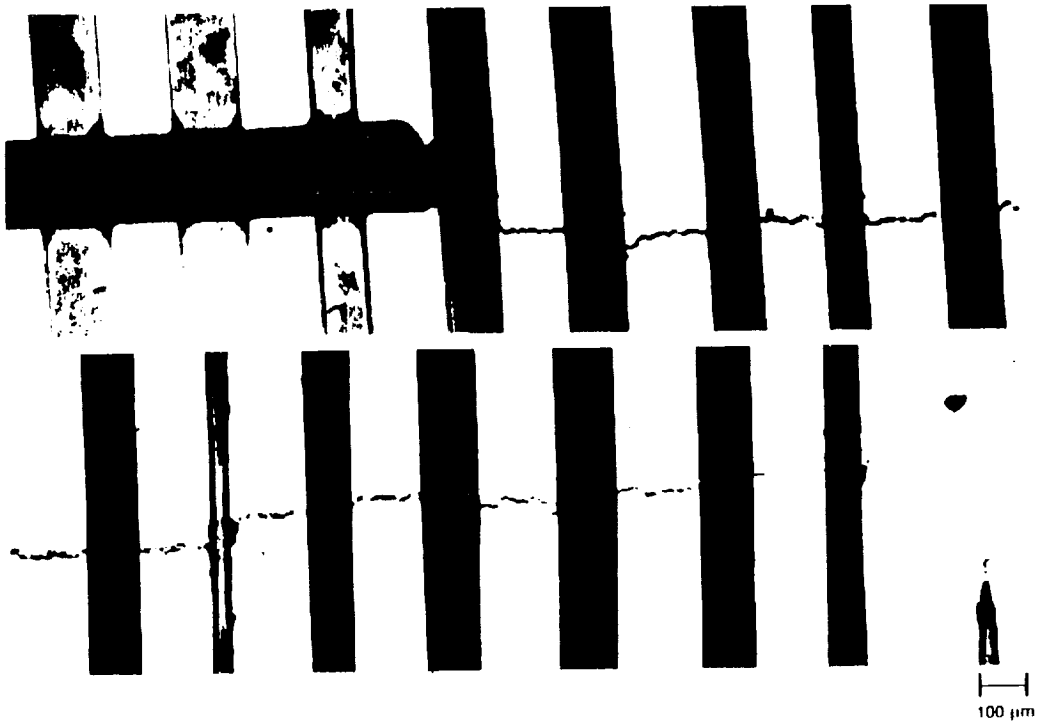
Composite materials: An opportunity

- Increased use of composites in designs
- Gross structural response well understood
- Macro- , Meso- , and Micro-Mechanics being worked
- Extension to failure criteria still major challenge
- Innovative testing and inspection programs needed
- Close ties with manufacturing and design

Need validated algorithms for effective designs

This chart simply shows an example of the complex failure mechanisms in composites, in this case, fiber bridging.

Fiber Bridging in Ti–Aluminide Composite



The CSTEM code was developed under a NASA Lewis contract and is used extensively for algorithm development. It was designed to handle composite materials as well as integrate with other disciplines. It has been applied to a variety of unusual design problems and has produced valid, useful results. As such it demonstrates an excellent transition path from research to production use.

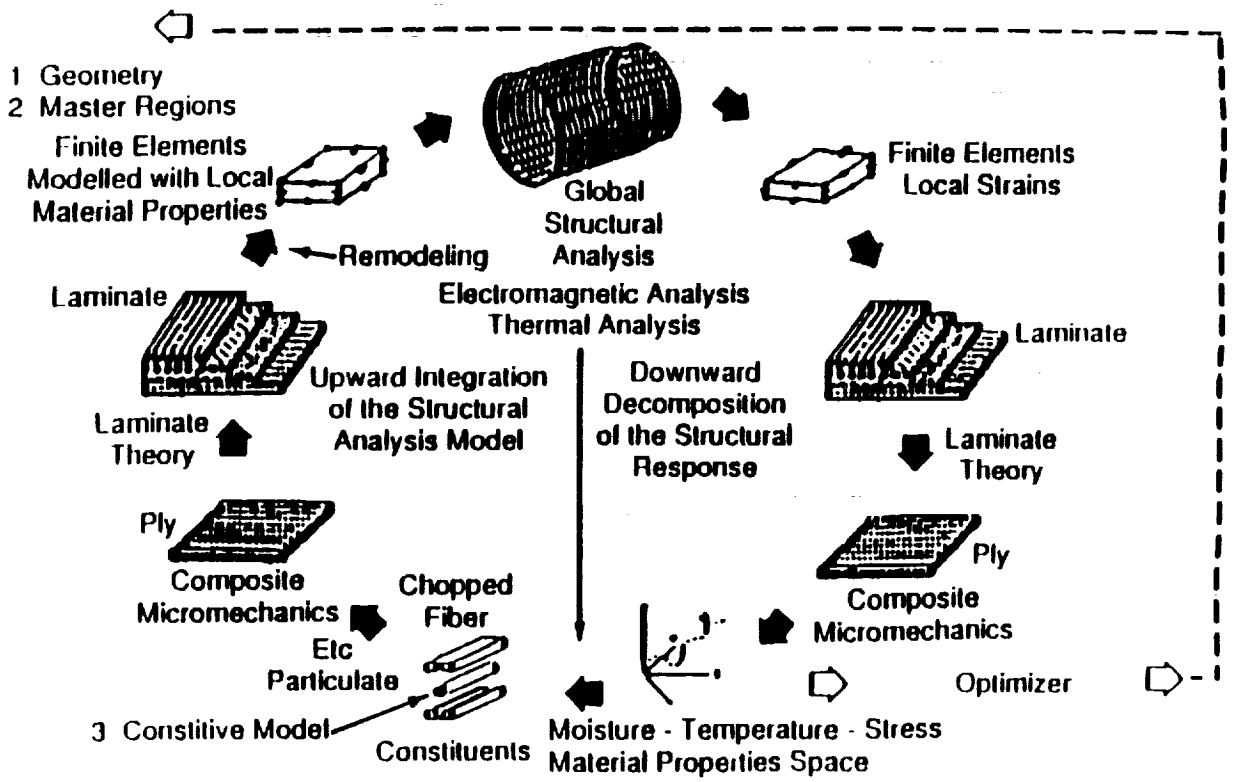
Multi-disciplinary analysis - CSTEM

- CSTEM code developed under NASA contract
  - Coupled Structural/Thermal/Electro-Magnetic analysis/tailoring of graded composite structures
- Special finite element for composite materials
- Includes large deformation analysis capabilities
- Code expandable to couple additional disciplines

Test bed for algorithm development

This may be a familiar chart, but I have included it here to show the upward integration and subsequent downward decomposition which is at the heart of the CSTEM philosophy.

CSTEM Integrated Analysis of Composite Structures



As we develop tools to get our analyses done earlier in the design cycle, the opportunity for use of optimization increases. The mathematics are well understood for simple problems, and computing technology is fast enough to allow us to use these techniques. Additional work is clearly warranted in true shape optimization as well as in multi-discipline optimization. Additional changes will have to occur in our ability to pose the problem for an optimizer.

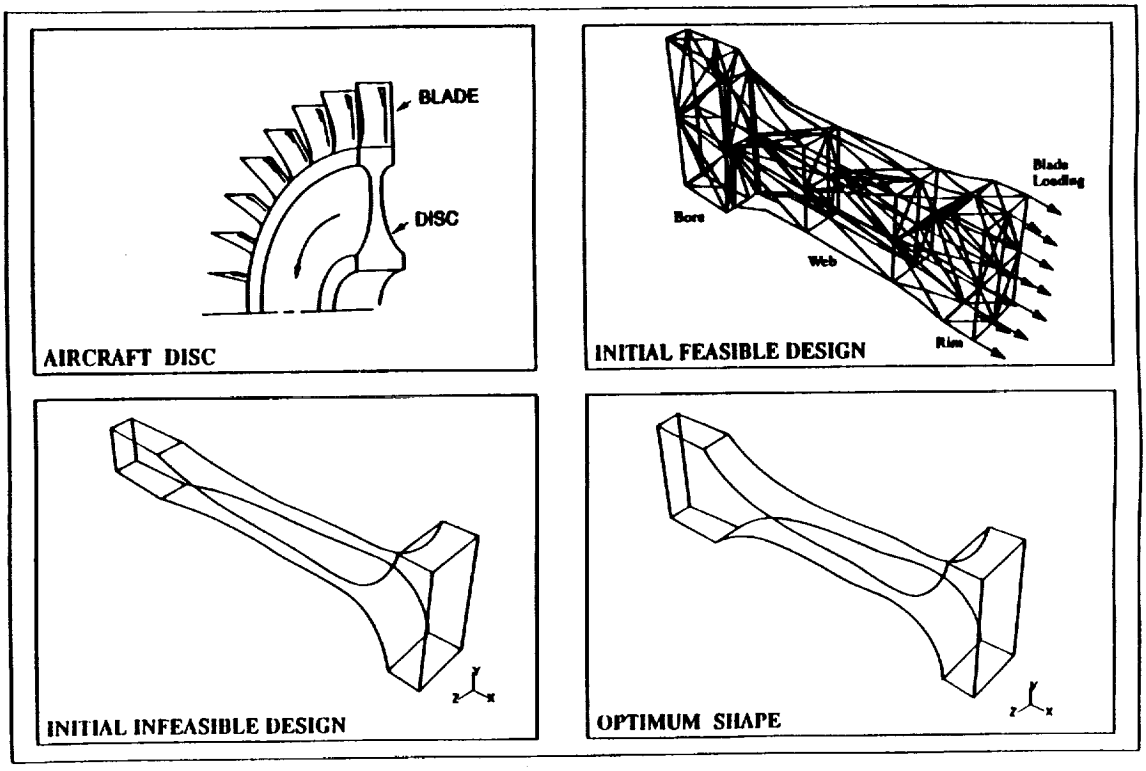
Optimization and structural tailoring

- CSTEM designed with optimizing functionality
- Simpler tools used early in design cycle
- Increased emphasis on producibility and cost
- Concurrent engineering needs multi-discipline optimization
- Effective algorithms will require problem reformulations
- Mixture of heuristic and mathematical rules

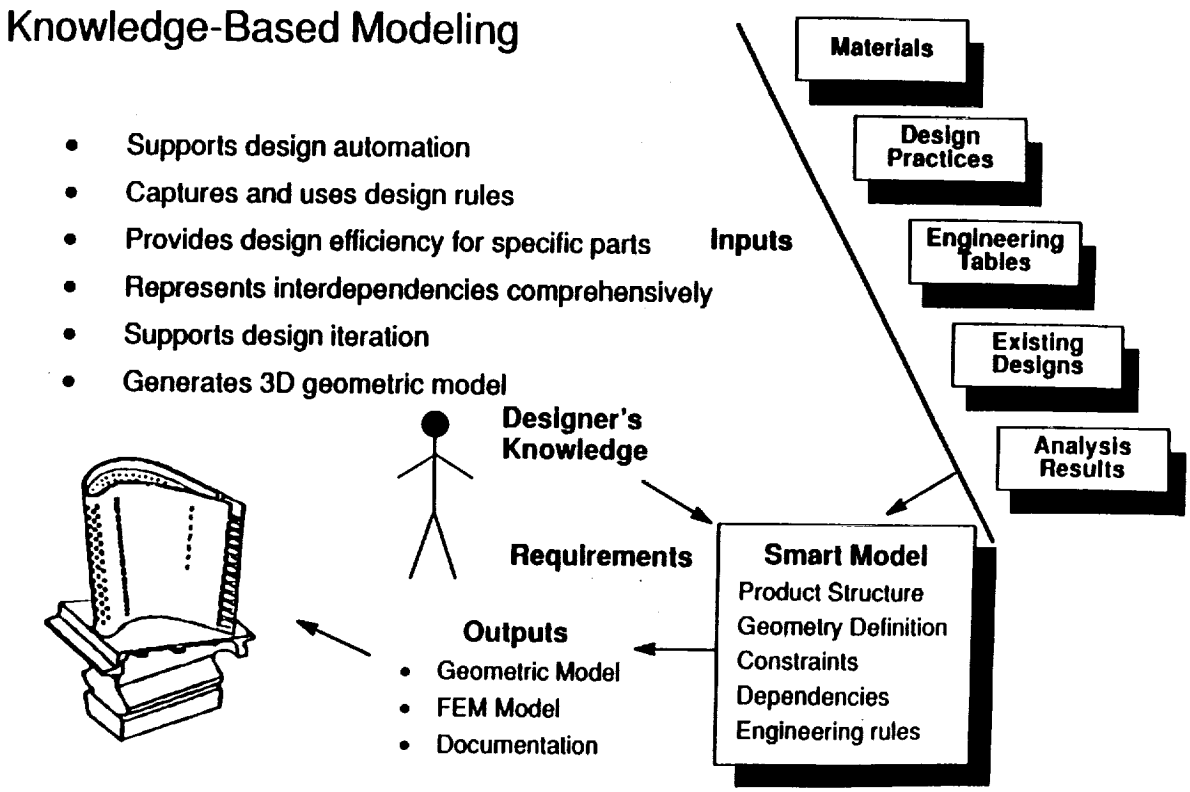
Optimization offers improved designs faster

This example shows a straightforward disk optimization result. This shows that not only does it produce a good design, but it also produces the same design from two radically different starting points.

AIRCRAFT TURBINE DISC - SHAPE OPTIMIZATION



Related to optimization is knowledge based modeling. A designer's knowledge must be captured, as well as the rules of related disciplines, and then brought together in a geometric model as well as a finite element model, along with any related documentation to produce the part.



Another technology which deserves future work is the area of probabilistic analysis. Once again, the power of modern computing systems enables us to perform the immense numbers of calculations required for this type of analysis. It seems almost intuitive that everything has some degree of variation inherent in its construction, and this technology offers a way to quantify the effects of such variations.

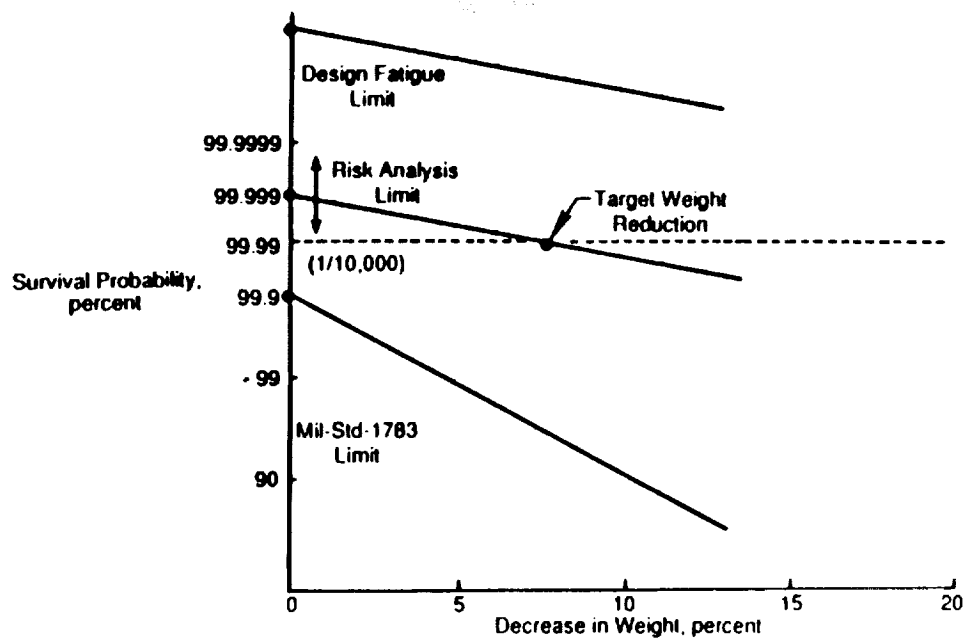
Probabilistic Algorithms

- Early work done to predict life in presence of defects
- Coupling of probability and deterministic crack growth
- MISSYDD code validated through controlled experiments
- NESSUS code developed under NASA sponsorship
- Air Force contract for Probabilistic Rotor Design System
  - Variations in loadings, material response, geometry, ...

Applications of probabilistic algorithms growing

One possible application of probability methods would be to enable a lighter weight design with high survival probability. This chart schematically portrays such an approach. We are under an Air Force contract to develop a Probabilistic Rotor Design System, so that we can account for variations in our designs.

Schematic of Objective - Potential Weight Reduction with High Survival Probability



No matter how much computer power you have, it's never enough to solve the REALLY HARD PROBLEMS. Coming to the rescue here is a new generation of parallel processing machines. They are already available and are being used in compressible fluid dynamics problems with great success. The NPSS project will showcase these kinds of computers and let us actually simulate an entire engine in sufficient detail.

Parallel processing algorithms

- Parallel computing architectures no longer "Futures"
- Algorithms will be specialized for the particular architecture
- Massively parallel computations capability probably required for large scale system simulations
  - e.g. Numerical Propulsion System Simulation

Parallel computing is the future for large systems

A personal closing message

- Pressure on businesses to reduce the development cycle
- Need new algorithms for productivity, accuracy, capability
- New algorithms need champions to become established
- Close integration of functions (concurrency) will help
- Analysis function is a service function in design process

Algorithms provide the right answer, right away

N92-25925

Structural Mechanics Simulations

Johnny H. Biffle
Sandia National Laboratories
Albuquerque, New Mexico

Structural Mechanics Simulations

by

Johnny H. Biffle

Sandia National Laboratories

Albuquerque, New Mexico

Sandia National Laboratory has a very broad structural analysis capability. This capability has been designed to apply to a wide variety of simulation. Work has been performed in support of reentry vehicles, nuclear reactor safety, weapon systems and components, nuclear waste transport, strategic petroleum reserve, nuclear waste storage, wind and solar energy, drilling technology and submarine programs. The analysis environment contains both commercial and Sandia developed software. Included are mesh generation capability, structural simulation codes and visualization codes for examining simulation results. The analysis software consists of the PRONTO series of codes for transient dynamic simulations and the JAC and SANTOS codes for static simulations. To effectively simulate a wide variety of physical phenomena, a large number of constitutive models have been developed. These include models for elastic plastic, viscoplastic, creep, crushable foam, soil, glass solidification, polymer curing, wood, explosive detonation, fracture, fragmentation and continuum damage. Examples of dynamics structural mechanics simulations at Sandia include impact of nuclear waste shipping casks, dynamics response of an electronic assemble, projectile and blast penetration studies, forming sheet metal, and the impact of a nose cone. Static simulations include crush of a half-cylinder, welding of electronic components, and modeling of the gas metal arc welding process.

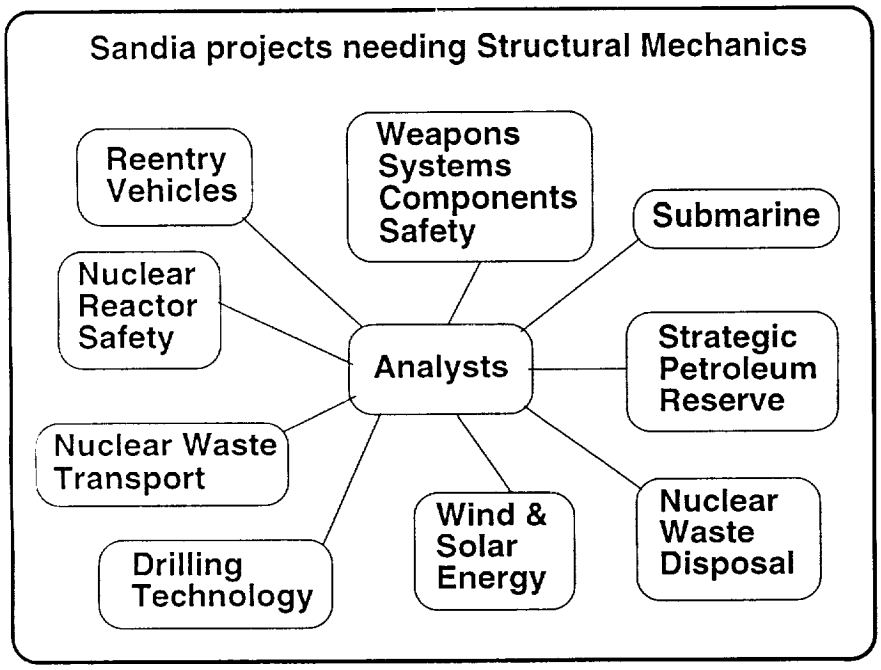


Figure 1. Because Sandia National Laboratory is involved in many different projects, structural mechanics analysis support is required. The range of capability is very broad and supports many projects.

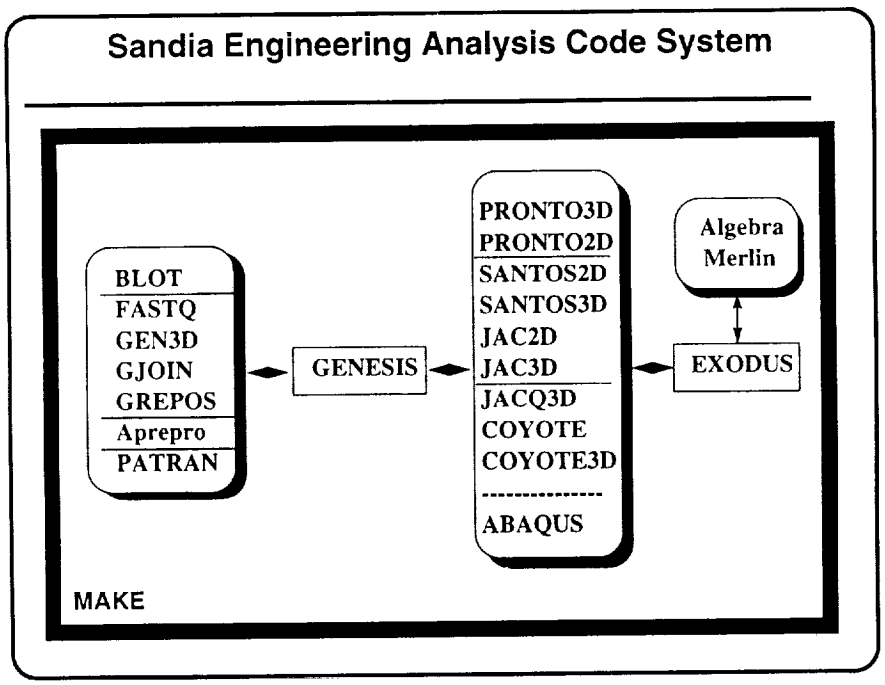


Figure 2. Most of the structural mechanics capability is in the Sandia Engineering Analysis Code System. It includes pre- and post-processing codes such as FASTQ and BLOT. The analysis codes are used to predict nonlinear transient dynamics, quasi- static and thermal response. The codes use the GENESIS and EXODUS data formats for communication between modules.

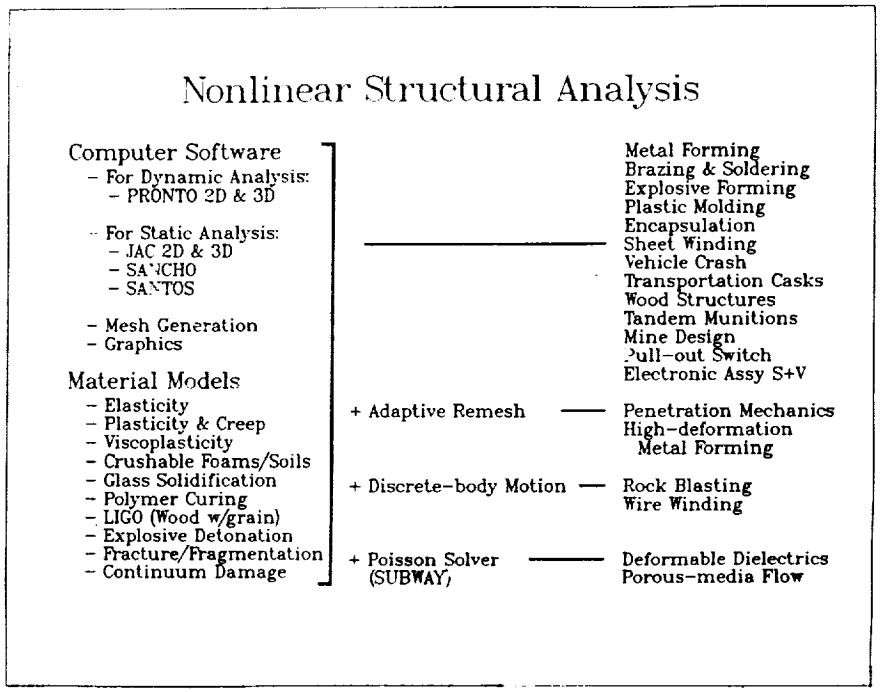


Figure 3. A wide range of structural mechanics simulations can be performed with the capability at Sandia National Laboratory. This capability has been applied to a broad range of problems.

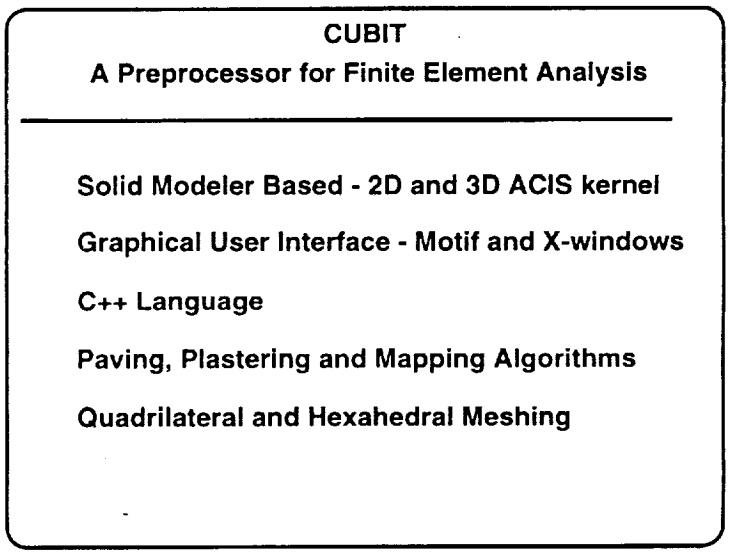


Figure 4. The CUBIT mesh generation product will integrate solid modelling with a set of mesh generation tools. The product will use the ACIS kernel for solid modeling tasks, an X-window user interface and the C++ programming language. It will be capable of quadrilateral and hexahedral meshing with the very powerful PAVING (ref.1) and PLASTERING algorithms. Mapping algorithms are also included. PAVING will provide an all quadrilateral mesh for 2D and 3D surfaces and PLASTERING will fill an arbitrary 3D solid with an all hexahedral mesh.

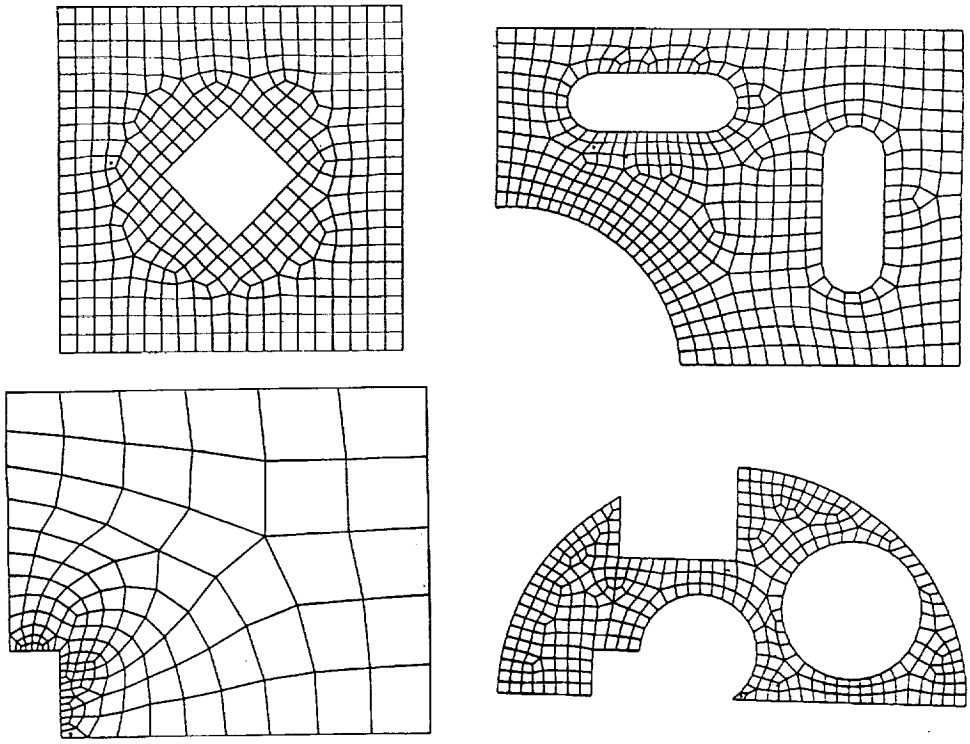


Figure 5. Four examples are shown of PAVING 2D surfaces. The PAVING algorithm is very good at producing well-shaped elements, especially on boundaries. It also has very good transitional capability.

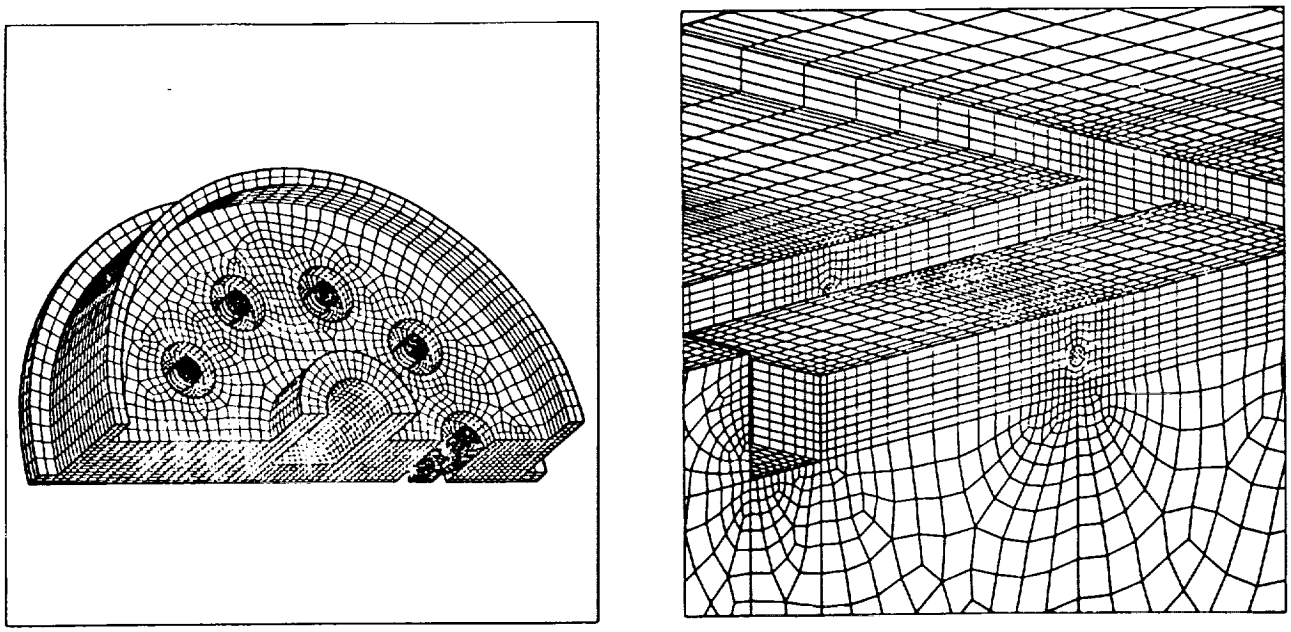


Figure 6. Two examples are shown of using PAVING for 2D surfaces and then mapping the mesh in the third direction to create a 3D mesh. The GEN3D program is used for the mapping, and the GJOIN program is used to connect several 3D meshes together.

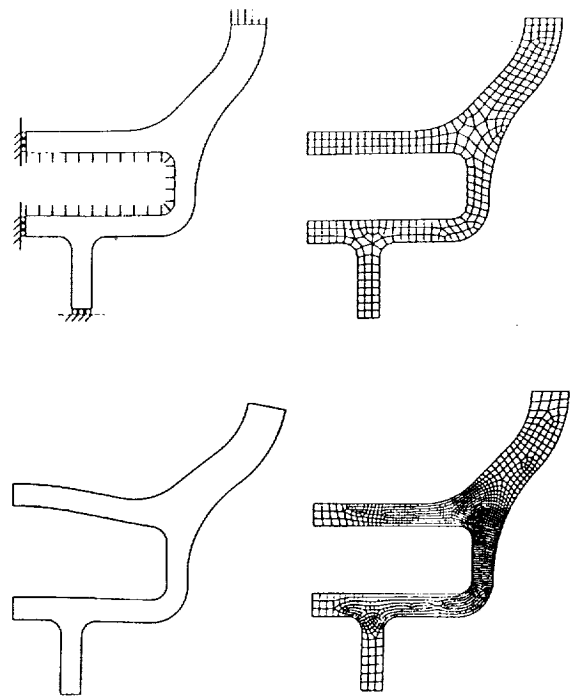


Figure 7. The PAVING algorithm is very powerful when used as the meshing algorithm for adaptive analysis. This is an example of a static simulation. A value of equilibrium error is produced for each element in the original mesh and a new mesh is created based on the error. At present the method converges to a reasonable equilibrium tolerance in four iterations. (ref.2)

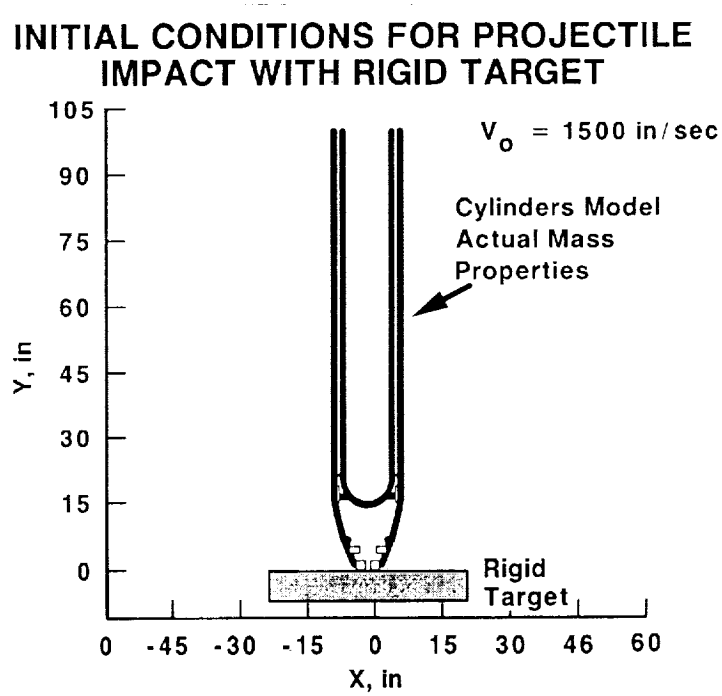


Figure 8. The impact of a nose cone on a rigid target is an example of the ability to model the large deformation response of metals.

COMPUTED NOSE CRUSHUP

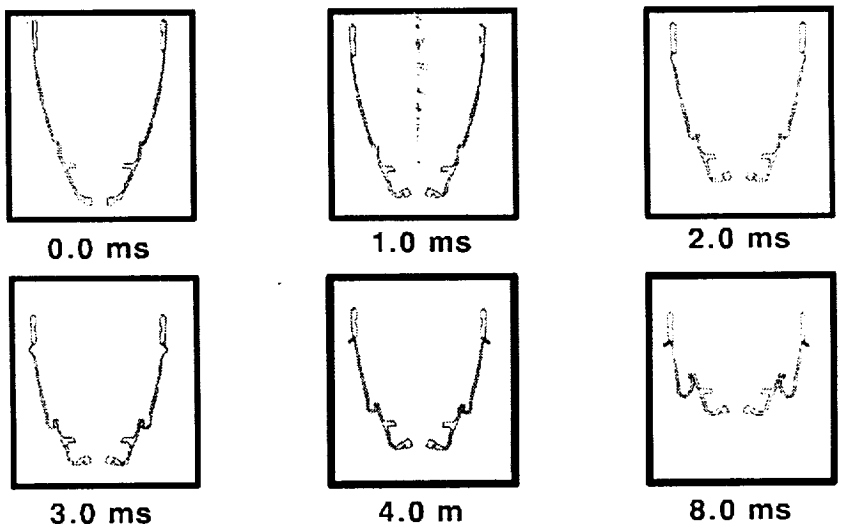


Figure 9. The response is predicted for 8.0 ms. The nose is designed to form a plastic hinge and for the plastic hinge to roll up the sides of the nose to absorb energy during crushup.

COMPUTED (RIGHT) AND EXPERIMENTAL (LEFT) DEFORMED NOSES

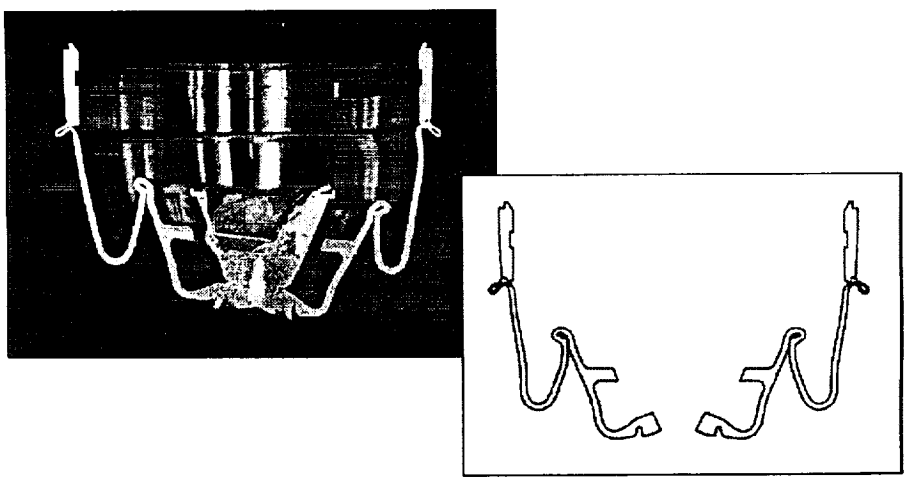


Figure 10. (Same as Viewgraph)

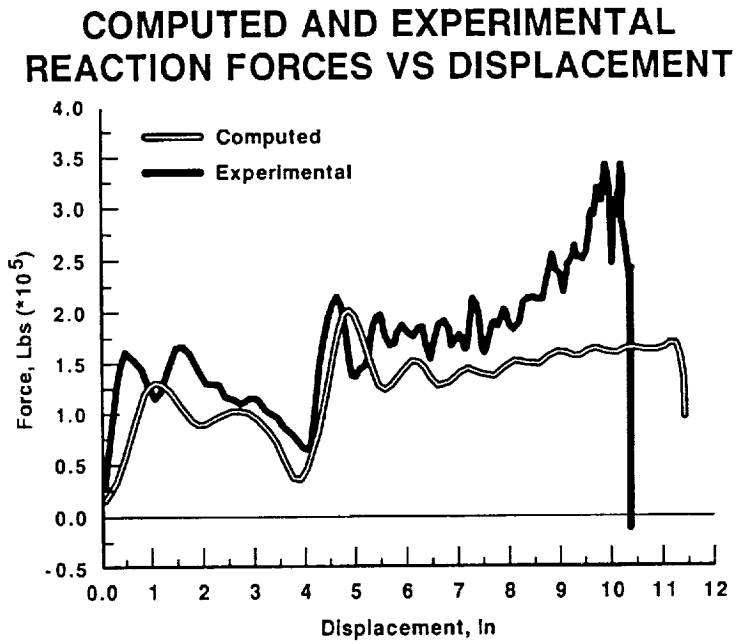


Figure 11. The comparison between experiment and computed force displacement response is very reasonable.

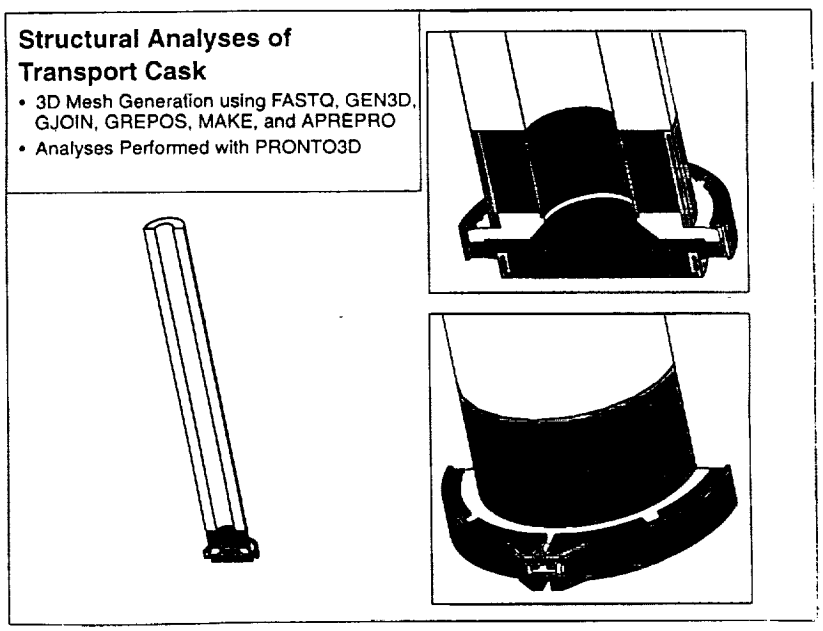


Figure 12. Another example of the robustness of the codes is the structural analysis of a transport cask to accident scenarios. The response is predicted for a cask which has a two-piece clamp for a closure mechanism.

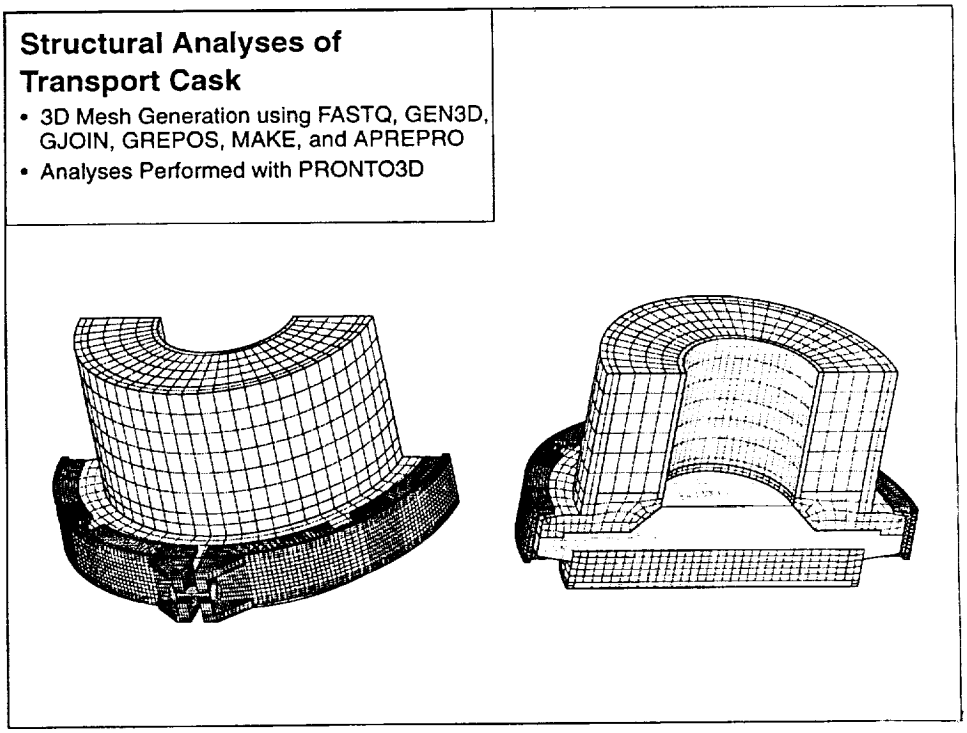


Figure 13. Depicted is the finite element mesh for the end of the transport cask. The mesh contains 33712 nodes and 24848 elements. A contact surface algorithm is used to model contact between parts.

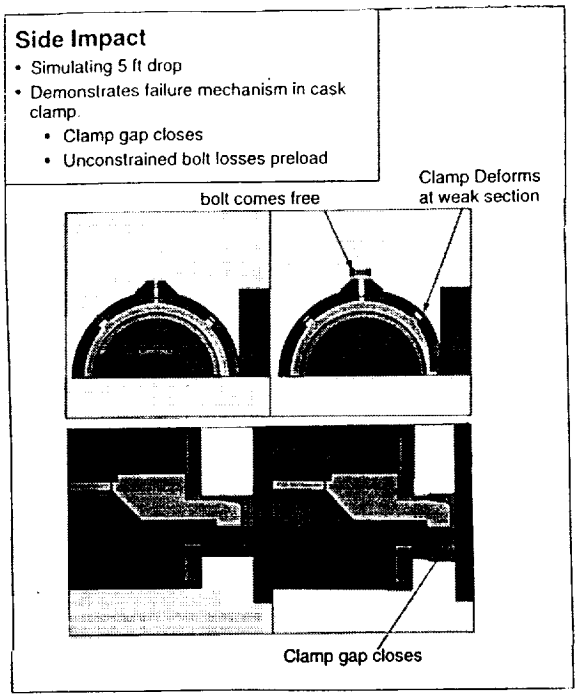


Figure 14. Upon impact from a side drop the clamp mechanism fails when the bolt comes free.

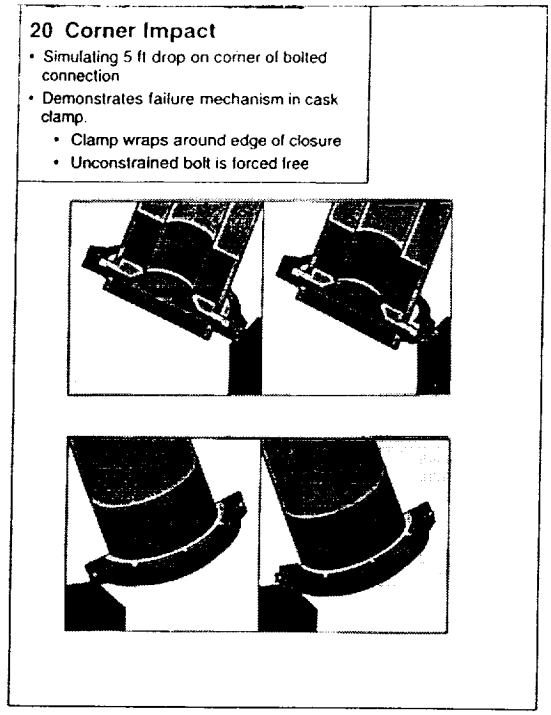


Figure 15. The clamp is removed in a 20 degree corner impact.

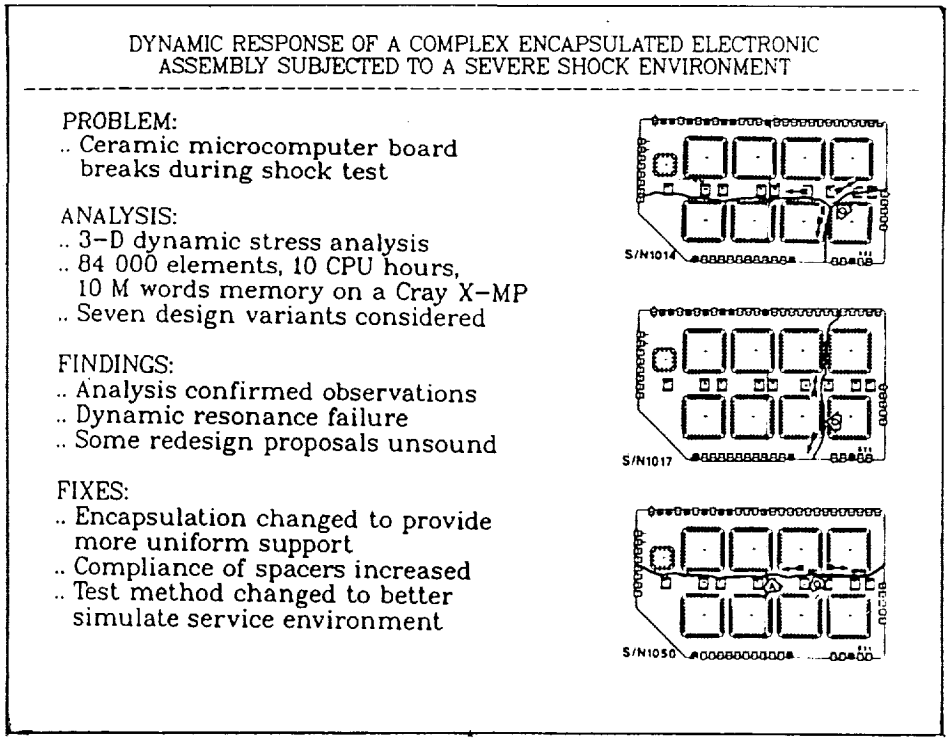


Figure 16. The dynamic response of a complex encapsulated electronic assemble subjected to a sever shock environment is an example showing the modeling capability of the analysis system.

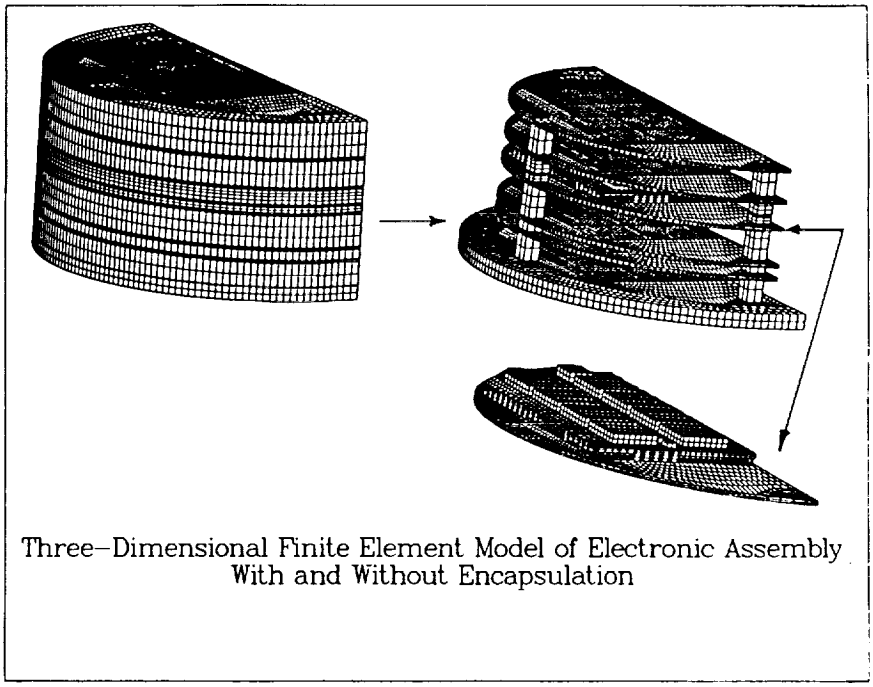


Figure 17. The finite element model.

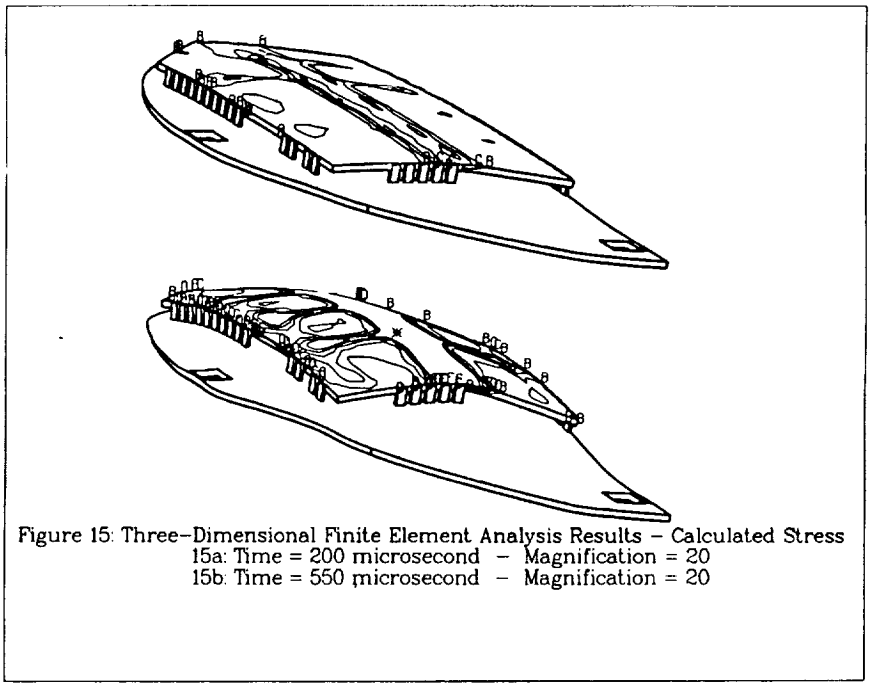
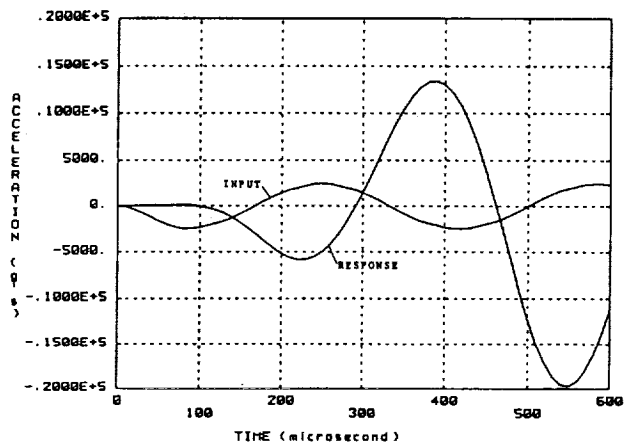
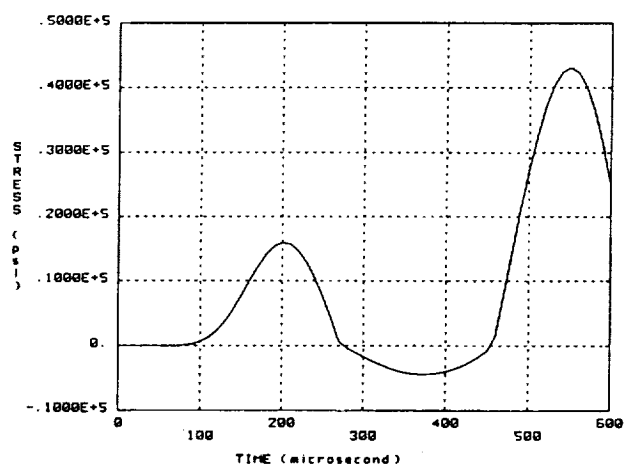


Figure 18. Dynamic response.



Acceleration Input and Calculated Acceleration Response at Center of Ceramic Microcomputer Substrate

Figure 19. Acceleration input versus response.



Calculated Maximum Principal Stress in Ceramic Microcomputer Substrate

Figure 20. Stress response versus time.

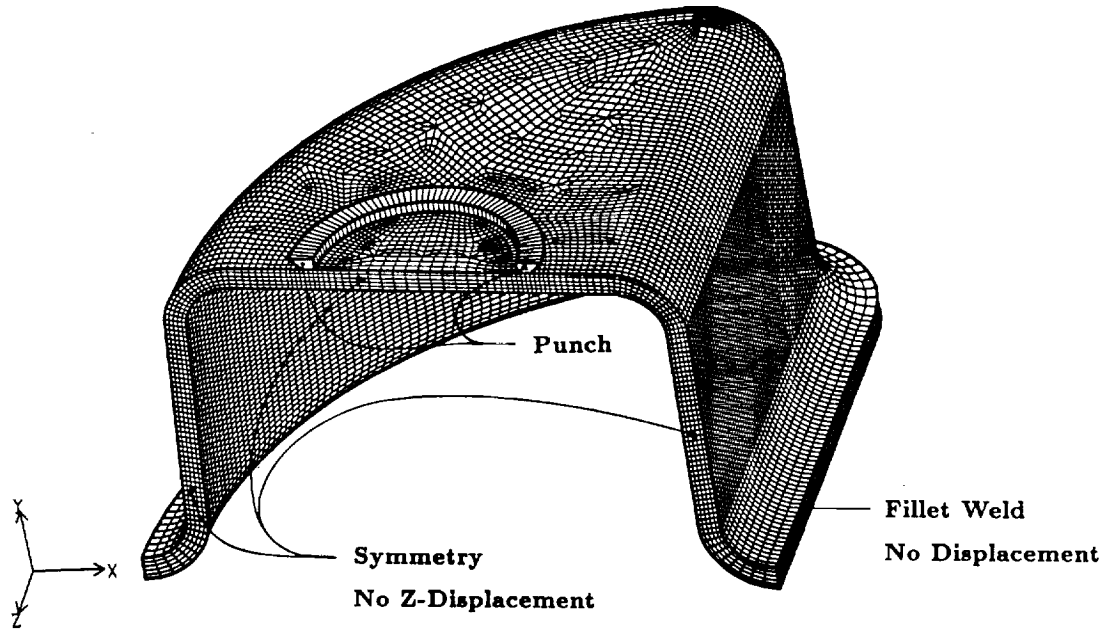


Figure 21. The response to the static deformation of a hat section is an example of the predictive capability for large deformation response of metals. The material is 304L and a power law hardening model was used for the simulation. A rigid punch was used to crush the section. The finite element model which was used for the simulation is depicted.

Deformed Shape for 304L Stainless Steel Half-Cylinder

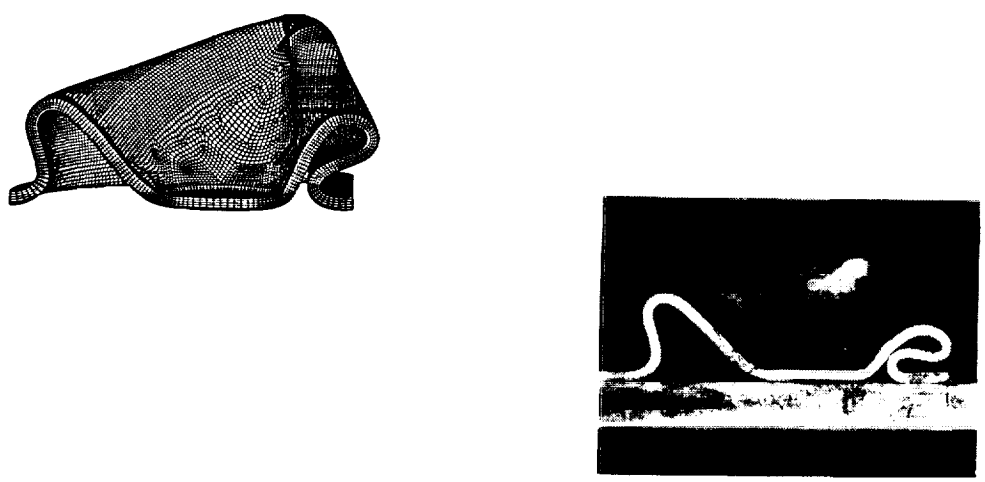


Figure 22. The experimental results and predicted deformation from the simulation is shown.

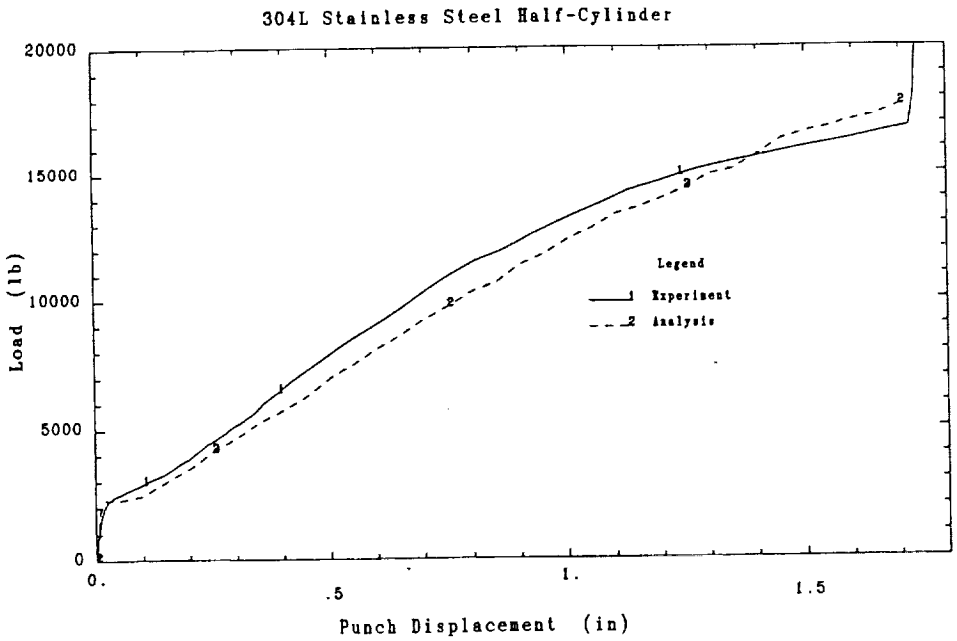


Figure 23. The comparison between the experiment and analysis load-displacement response is shown. Very good agreement is predicted using the power law hardening model.

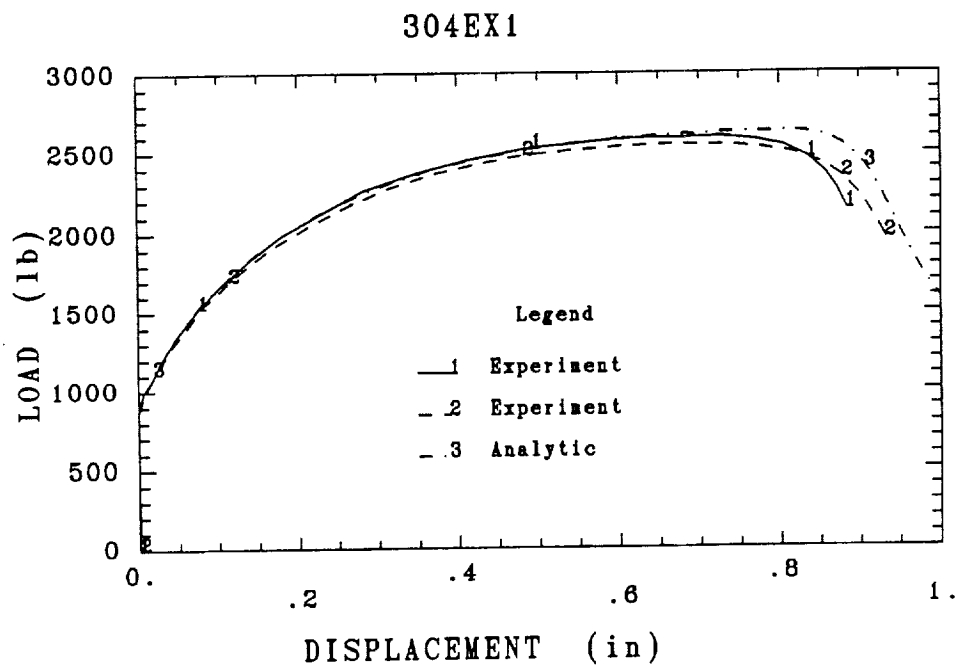


Figure 24. To check the power law model, the material test which was used to obtain material constants for the power law model was analyzed. The power law fit yielded good results when compared with experimental data.

Sandia has a program to model metal arc weld processes. It involves coupled thermomechanical analysis, superheated filler metal deposition, accurate high-temperature constitutive material modelling and advanced measurement techniques for input and validation of the process. The goal is to model the three stages of welding; early metal deposition, the complete filler metal deposition and the cool down period. A heat input model has been derived that partitions energy between arc heating and superheated filler metal deposition. The parameters in the model are determined through the measurement of several quantities; the temperature of superheated droplet, the filler metal deposition rate, the radius of metal deposition, the arc radius and the net power into the workpiece.

(Color viewgraphs are not available)

Materials Available

	ROOM TEMP	WITH TEMP	RATE EFFECT	DAMAGE
6061-T6	+	*	*	+
304L SS	+	*	*	+
HY-130	+	*	*	+
21-6-9	+	*	*	
COPPER	+	+	*	
TI 10-2-3 (140)	+		*	
TI 10-2-3 (170)	+		*	
MAR 250	+		*	*
MAR 300	+		*	*
4340 (120)	+		*	*
4340 (180)	+		*	*
4340 (220)	+		*	*
HP9-4-20	+		*	*
AF1410	+		*	*

+ based on reliable data
* based on questionable or limited data

Figure 25. The work in high temperature metal deformation problems used a material model which has been characterized for a number of materials. The model is capable of simulating temperature effects, rate effect, plastic flow and damage.

Codes with Model Implemented

Code Name	Plasticity	Damage
DYNA2D	D	D
DYNA3D	D	D
NIKE2D	D	
ABAQUS	D	
PRONTO2D	*	*
PRONTO3D	*	*
PASTA2D	D	

* = In Progress
D = Done

Unclassified

Figure 26. The high temperature material model has been implemented into the following computer programs.

Thin Sheet Metal Forming Example

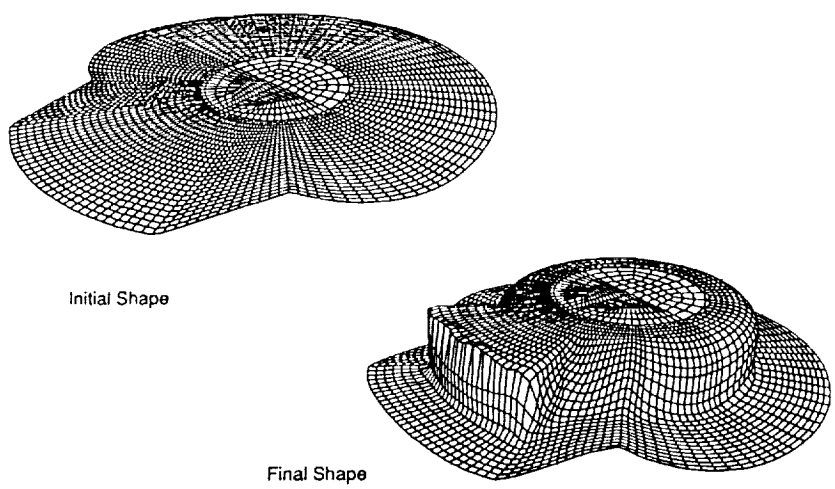


Figure 27. Research is also being performed in sheet metal forming simulation. The following is an example of the tearing which resulted when forming a part.

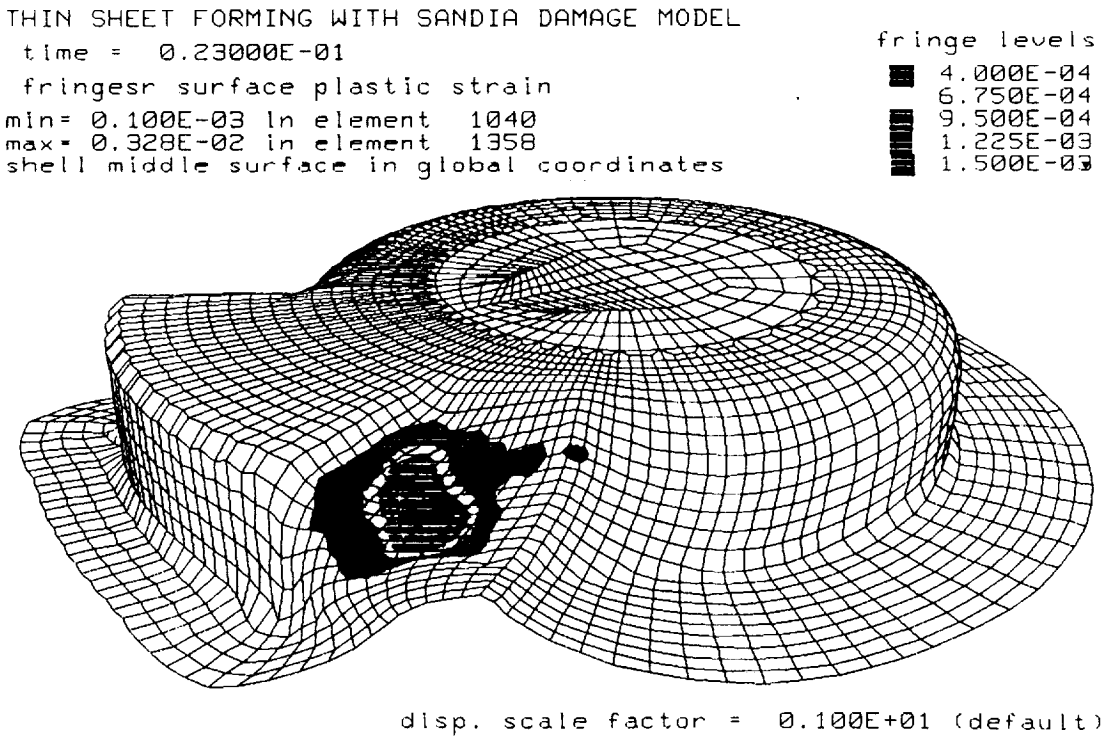


Figure 28. The tearing of the previous example can be eliminated by changing material. Material damage is predicted.

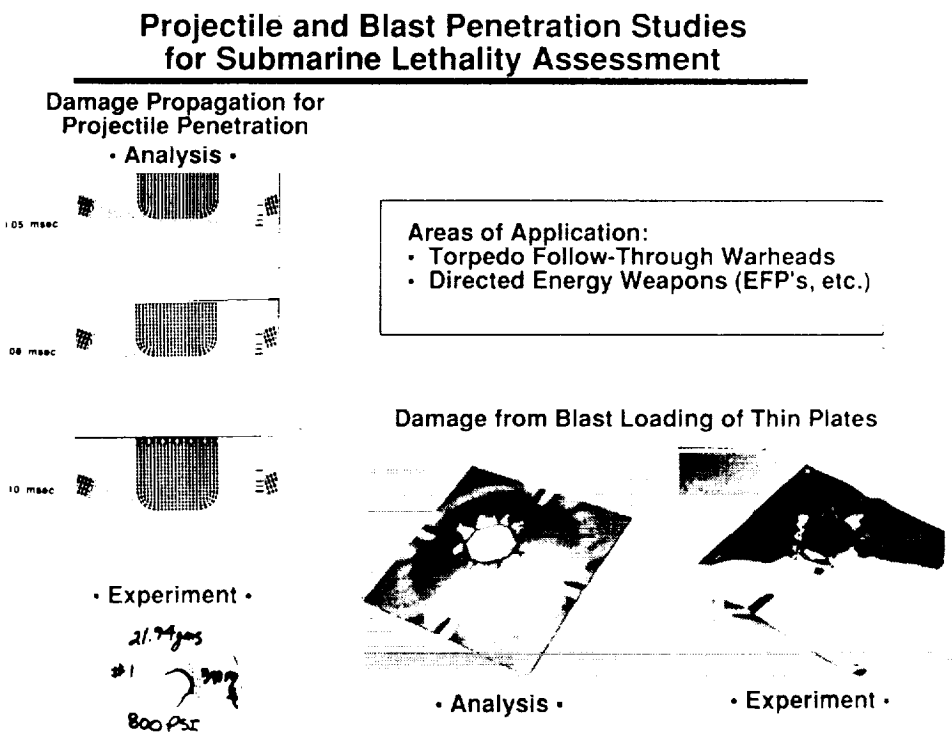


Figure 29. The damage capability of the high temperature material model, is used to predict tearing in a projectile penetration event. Very good correlation between experiment and analysis was obtained.

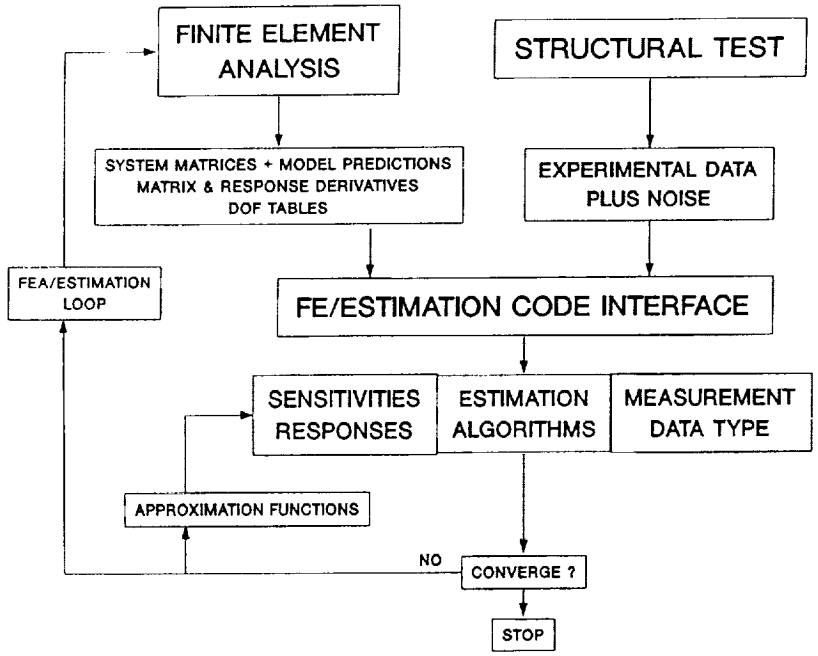


Figure 30. Sandia is involved in developing techniques for system identification. Finite element analyses and structural test results are used with an estimation code interface to adjust parameters in the finite element model.

ESTIMATION TECHNIQUES

FOUR ALGORITHMS – THREE MEASUREMENT TYPES

* Measurements: Modal Frequency, Mode Shape, Frequency Response Function

* Estimation Algorithms:
  - Bayes Estimation / Weighted Least Squares
  - Kalman Filter for Parameter Estimation (Recursive processing of the data)
  - Optimization / Math Programming Techniques
  - Non Iterative (One step) Methods

Figure 31. Several estimation techniques are under investigation. At present the Bayes estimation algorithm has been used for system identification.

REFERENCES

1. Blacker, T. D. and Stephenson, M.B.: Paving: A New Approach to Automated Quadrilateral Mesh Generation, SAND90-0249. Sandia National Laboratories, 1990.

2. Blacker, T. D.; Jung, J.; and Witkowski, W. R.: An Adaptive Finite Element Technique Using Element Equilibrium and Paving. ASME 90-WA/CIE-2, 1990.

N92-25926

Structural Analysis for Preliminary Design of High Speed Civil Transport (HSCT)

Kumar G. Bhatia
Technology and Product Development
Boeing Commercial Airplane Group
Seattle, Washington

OVERVIEW

In the preliminary design environment, there is a need for quick evaluation of configuration and material concepts. The simplified beam representations used in the subsonic, high aspect ratio wing planforms are not applicable for low aspect ration configurations typical of supersonic transports. There is a requirement to develop methods for efficient generation of structural arrangement and finite–element representation to support multidisciplinary analysis and optimization. In addition, empirical data bases required to validate prediction methods need to be improved for HSCT type configurations. NASA and industry cooperation is needed to improve state–of–the–art for HSCT prediction methods.

- PD CONFIGURATION DEVELOPMENT SUPPORT FOR QUICK EVALUATION OF CONFIGURATION AND MATERIAL CONCEPTS.
- MULTIDISCIPLINARY ANALYSIS & OPTIMIZATION.
- NASA–INDUSTRY COOPERATION IN DEVELOPING RELIABLE PREDICTION METHODS.

PD CONFIGURATION DEVELOPMENT

The next generation of HSCT has to be economically viable, and a reliable estimate of structural weight is required to evaluate airplane performance. The weights statistical data base for subsonic airplanes is generally reliable enough to perform performance evaluations for different concepts. The lack of an equivalent data base for HSCT makes it necessary to determine structural sizing early in the preliminary design phase to estimate structural weights.

Large, flexible HSCT wing with engines mounted towards the wing trailing edge is susceptible to flutter clearance problems, and may have to be stiffness designed. Aeroelastic effects on control surfaces are also increased for HSCT configurations, relative to subsonic configurations, due to a flexible wing and a long, flexible body.

It is anticipated that active flight controls would play a significant role in reducing weight of an HSCT configuration by such concepts as load enveloping, flutter suppression, etc. In order to predict aeroservoelastic effects in the preliminary design environment, it is necessary to improve steady and unsteady aerodynamic prediction methods.

Structural heating at supersonic speed affects stiffness and aeroelastic performance. Emergency descent conditions would have to be considered in determining design loads.

- STRUCTURAL SIZING REQUIRED FOR PERFORMANCE ESTIMATES.
- STIFFNESS REQUIREMENTS FOR FLUTTER CAN RESULT IN A LARGE INCREASE IN OEW.
- AEROELASTIC EFFECTS SIGNIFICANT FOR STABILITY & CONTROL.
- AEROSERVOELASTICITY CONSIDERATIONS IMPORTANT FOR FLIGHT CONTROL DESIGN.
- AEROTHERMOELASTICITY CONSIDERATIONS IMPORTANT FOR LOADS AND SIZING.

REQUIREMENTS FOR THE FINITE–ELEMENT MODEL

An integrated external load, internal loads, flutter, sizing and weights analysis with a common model is necessary for an HSCT configuration due to common requirements for a detailed structural representation. A half–model finite–element representation is adequate at the preliminary design stage. All aerodynamic surfaces including control surfaces are represented in our model for performing an integrated analysis. Major structural load paths are represented in the preliminary design model to avoid unnecessary simplifications requiring compromises that can be difficult to evaluate.
In order to support optimization for determining a minimum weight structure, structural mass properties are separate. In addition fuel and payloads mass properties are separated to determine critical combinations of payload and fuel for external loads and flutter.

- A COMMON ELFINI MODEL FOR EXTERNAL LOADS, INTERNAL LOADS AND FLUTTER WITH INTERFACE TO STRESS AND WEIGHTS ANALYSES.
- HALF–AIRPLANE PD MODEL REPRESENTING MAJOR AERODYNAMIC SURFACES, AND CONTROL SURFACE MECHANISMS.
- MAJOR STRUCTURAL LOAD PATHS INCLUDING ENGINE ATTACHMENTS, LEADING AND TRAILING EDGES, CUTOUTS FOR DOORS AND GEARS.
- MASS PROPERTIES SEPARATED FOR STRUCTURAL, SYSTEMS, PAYLOADS, AND FUEL WEIGHTS.

  MASS FACTOR DATA TO SUPPORT STRUCTURAL OPTIMIZATION.

REQUIREMENTS FOR AEROELASTIC ANALYSIS

It needs to be recognized that a satisfactory aerodynamic data base for HSCT does not exist, and needs to be developed. This is particularly true of control surface aerodynamics and unsteady aerodynamics. Traditional correction methods for high aspect ratio wings have relied upon "rigid" aerodynamic model data. The low aspect ratio configuration of HSCT makes "rigid" model data harder to interpret due to larger aeroelastic effects. In order to predict airplane loads and dynamic characteristics, camber bending effects inherent to low aspect ratio surfaces need to be reliably predicted. Effects of Reynolds number for control surface steady and oscillatory derivatives are well appreciated. Finally, design of an active control system for flutter suppression, ride comfort, load enveloping, etc. requires reliable aerodynamic methods that can predict accurate pressure distributions. Ability to accurately predict steady and unsteady aerodynamics has a critical effect in choosing a configuration, and determining structural sizing and control system design. Complexities of a low aspect ration configuration require application of formal optimization methods in an interdisciplinary environment. Aeroelastic optimization is a logical starting point for development and application of multidisciplinary optimization.
Nonlinear analysis methods are required in the context of detailed finite–element representation to account for attachment and actuator nonlinearities as well as aerodynamic nonlinearities.

- AERODYNAMIC PRESSURE DISTRIBUTIONS FOR :

  STATIC LOADS,

  GUST,

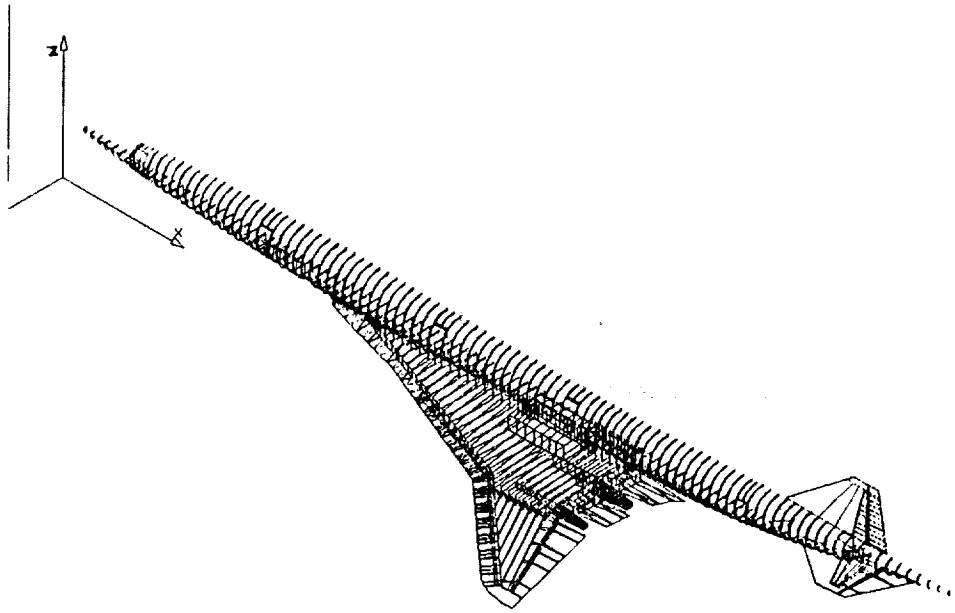
  FLUTTER, AND

  ACTIVE FLIGHT CONTROLS.

- AEROELASTIC OPTIMIZATION.

- NONLINEAR ANALYSIS.

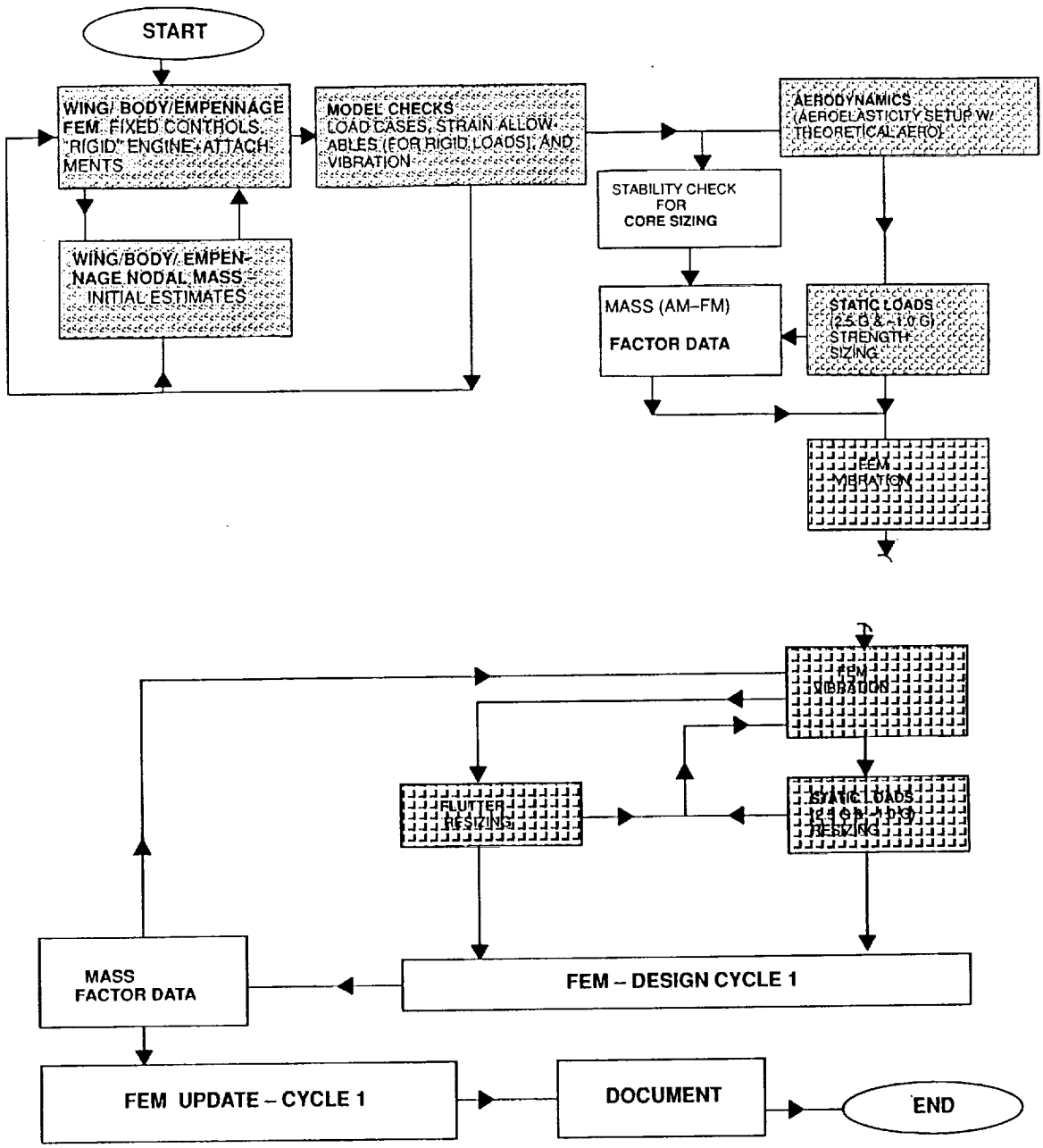
HSCT MODEL



A wireframe representation of structural configuration is shown. The wireframe was used in the Catia system to generate a finite–element mesh for Elfini analysis.

HSCT FINITE–ELEMENT (ELFINI) MODEL – WING SIZING – CYCLE 1

Preliminary design cycle of HSCT is illustrated. One of the important things to note is the interfaces required for capabilities not available in a standard finite–element code. These interfaces affect how optimization is performed. In particular, minimization of total weight requires reasonable estimates of actual to theoretical mass distribution. Each company has its own weights methodology, and the methodology is not available in commercial finite–element software.



PRELIMINARY DESIGN AND CURRENT ENVIRONMENT

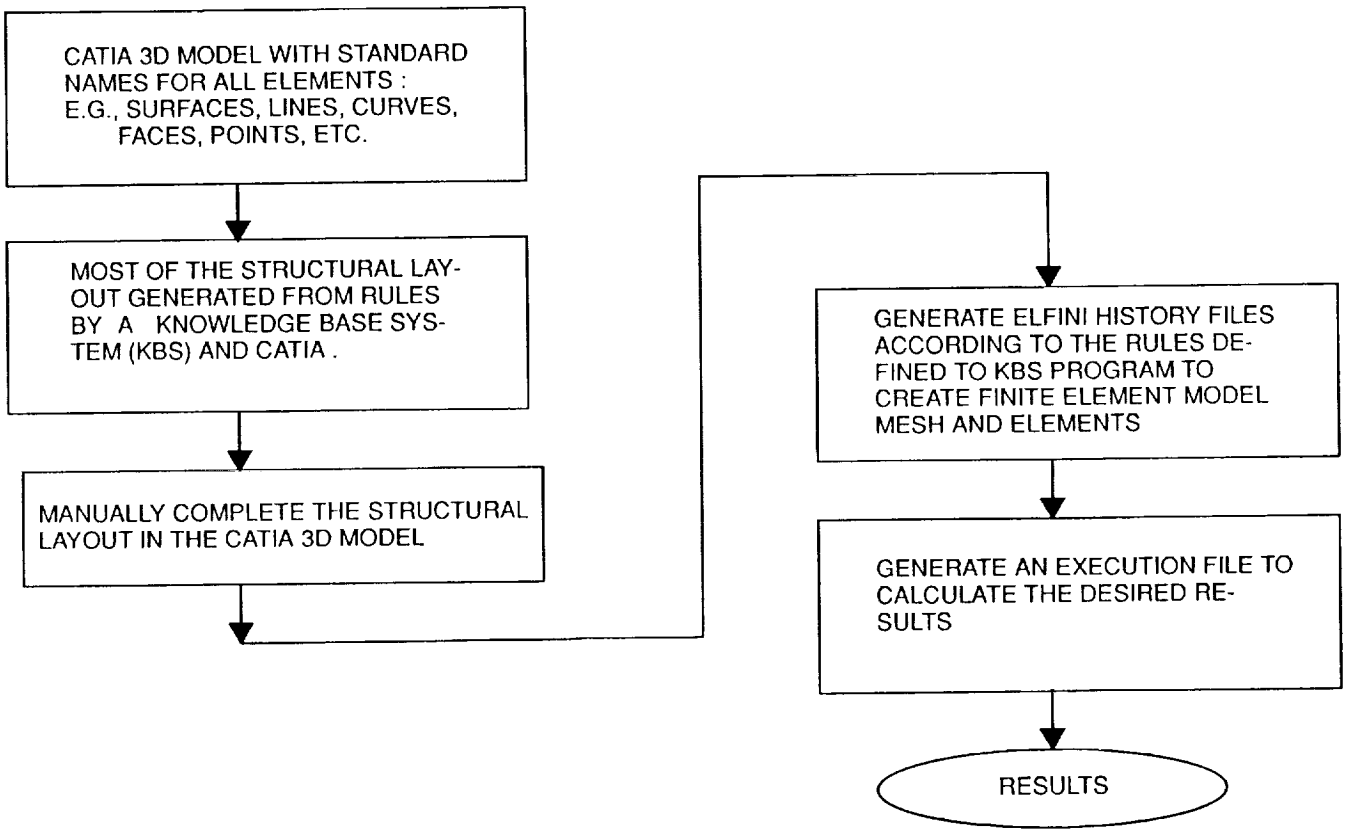
Detailed analysis required for an HSCT configuration results in an unacceptably long flow time. The primary contributor to the long flow time is the effort required to generate a finite–element model representation. One of the important aspects of preliminary design environment is sensitivity studies particularly those dealing with configuration modification. Such modifications require a time consuming procedure susceptible to errors.

It is possible to improve the current process for finite–element model generation and modification by using a knowledge base system approach.

- A SIGNIFICANT EFFORT AND A LONG FLOW TIME ARE REQUIRED TO DEFINE A STRUCTURAL ARRANGEMENT IN A CATIA MODEL.
- A SIGNIFICANT EFFORT AND A LONG FLOW TIME ARE REQUIRED TO CREATE A FINITE–ELEMENT MODEL FROM A CATIA MODEL.
- MODIFICATION OF THE CATIA GEOMETRY, THE STRUCTURAL ARRANGEMENT, OR THE FINITE–ELEMENT MODEL IS A TIME CONSUMING PROCESS.
- THE CURRENT ENVIRONMENT FOR GENERATING FINITE–ELEMENT MODELS CAN BE IMPROVED TO BETTER SUPPORT THE PRELIMINARY DESIGN PROCESS.

IDEALIZED AUTOMATED FINITE–ELEMENT MODEL GENERATION

An idealized environment for finite–element model generation and analysis is shown. It is recognized that even in an idealized environment, a certain percentage of work would be done manually rather than automatically through a computer.



EFFICIENT CATIA/ELFINI FINITE–ELEMENT ANALYSIS
BENEFITS

A reduction in flow time from over 6 months to under 6 weeks appears possible in creation of a finite–element model suitable for preliminary design usage.
Sensitivity studies currently not performed due to lack of time and resources in a preliminary design organization can be more easily accommodated to gain better insight into the configuration.
Rules for checking the finite–element representation, developed using a KBS, can be incorporated in the KBS itself. Additionally efficiency improvements can be achieved by using KBS to automate interfaces with other programs.

- FASTER CREATION OF THE FINITE–ELEMENT MODEL TO SUPPORT PD CONFIGURATION DEVELOPMENT – REDUCTION IN FLOW TIME FROM 6 MONTHS TO 6 WEEKS?

- FACILITY TO ANALYZE SENSITIVITIES TO IMPORTANT CONFIGURATION VARIATIONS SUCH AS ENGINE PLACEMENT, STRUCTURAL MEMBER ORIENTATION, ETC.

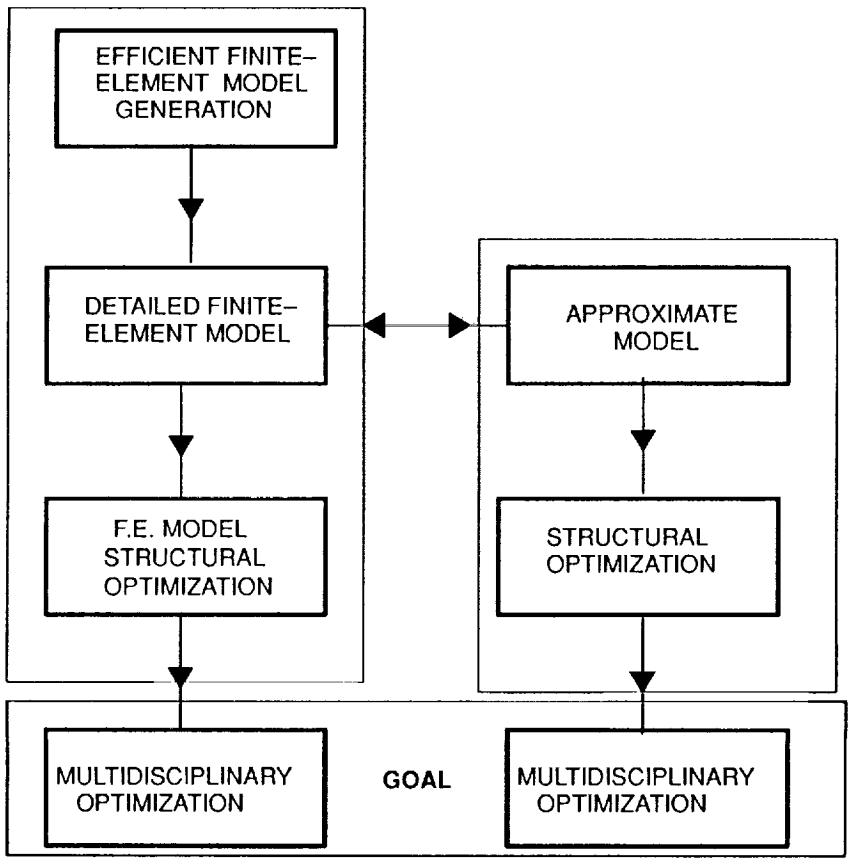
- POTENTIAL FOR

  1. FACILITATING THE FINITE–ELEMENT MODEL CHECKING.

  2. AUTOMATING INTERFACES WITH OTHER PROGRAMS.

STRUCTURAL ANALYSIS FOR PRELIMINARY DESIGN OF HSCT

An alternate approach to a detailed finite–element analysis is to analyze a simpler approximation to actual structure. A simplified representation usually results in quicker turnaround for analysis results, and facilitates optimization. In particular, it may be much easier to incorporate multidisciplinary optimization in a simplified analysis than in a detailed finite–element analysis. The simplified analysis must provide results of acceptable accuracy, and therefore must be calibrated against the more detailed analysis. A parallel approach is needed to develop both the capability for efficient detailed analysis, and a rapid approximate analysis.



STRUCTURAL ANALYSIS FOR PRELIMINARY DESIGN OF HSCT

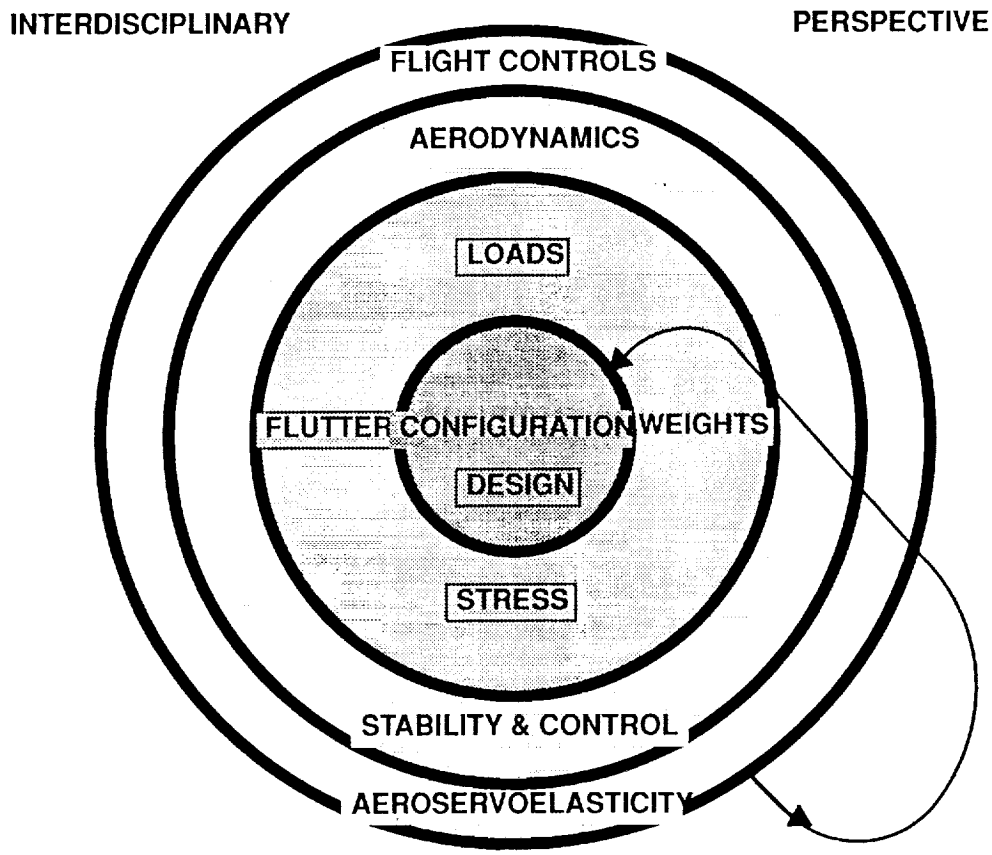
An interdisciplinary analysis has to be a team activity utilizing expertise and talents of engineers from different disciplines. Initially the team may consist of loads, flutter, stress and weights groups to generate the basic structural representation, perform aeroelastic analysis, calculate structural sizing and determine mass distribution.

Once a basic model is available, it can be refined using better aerodynamic data. Aerodynamics group can use the information to further examine off–design performance, and Stability and Control group can take into account aeroelastic effects in control surface sizing. This team would include some of the engineers from the original team, and representatives from Aerodynamics and Stability and Control.

The second team would have improved the aeroelastic model, and perhaps refined the configuration including resizing of control surfaces, if necessary. At this point, a third team can be formed including some of the members from the second team and engineers from Flight Controls or Avionics. The third team would review flight controls design using aeroservoelastic analysis, and make trade studies involving loads, flutter, airplane response, handling, etc.

Results of work by the three teams would be fed into configuration development as appropriate. The three teams may continue to work in parallel until all issues with respect to a given configuration are either resolved, or deferred for further study with a subsequent configuration.

An interdisciplinary activity currently takes place during configuration development phase for current transonic transports. A greater level of coordination is required for a supersonic transport.



NASA–INDUSTRY COOPERATION FOR HSCT

Current NASA plans for High Speed Research – Phase II (HSR II) emphasize materials aspect of structural development. It is our opinion that the resources allocated towards structural analysis methods and structural dynamics need to be increased from their current levels. In particular, there is a need to develop an empirical aerodynamic data base to support reliable prediction of external loads and flutter, and performing aeroservoelastic analysis. A coordinated wind tunnel test program is required to determine adequacy of the current methods and the extent of future developments required to predict steady and unsteady pressure distributions near flight Reynolds number. The wind tunnel test program also needs to address status of and requirements for *predicting* aeroservoelastic performance. The overall emphasis of the wind tunnel program should be to determine if we can *predict* rather than *correlate* pressure distributions.

It is important to develop efficient methods for preliminary design process in order to eliminate surprises after a configuration selection is made. Efficient generation of finite element models certainly should rank first in improving the response time in preliminary design process.

Optimization methods need to be applied to realistic configurations since in theory the methods have been available for some time. It is imperative for HSCT that one aspect of configuration not be optimized with detrimental results for some other aspect; thus a multidisciplinary approach is required.

- MORE EMPHASIS REQUIRED ON STRUCTURAL ANALYSIS METHODS AND STRUCTURAL DYNAMICS.
- CALIBRATION OF PRESSURES AND FLUTTER CHARACTERISTICS AT HIGH REYNOLDS NUMBERS.
- IMPROVED METHODS FOR AEROSERVOELASTIC PREDICTIONS.
- EFFICIENT FINITE–ELEMENT GENERATION AND APPROXIMATE METHODS FOR REDUCTION IN ANALYSIS CYCLE TIME.
- MULTIDISCIPLINARY OPTIMIZATION.

CONCLUDING REMARKS

In conclusion, economic viability of HSCT requires improved methods to improve performance. NASA and industry has a unique historical opportunity for a long term, system approach to improve methods and tools used to develop a configuration that will be certified around the year 2005.

- HSCT REQUIRES IMPROVED METHODS & TOOLS TO MEET ECONOMIC GOALS.
- HSCT CERTIFICATION PLANS OFFER UNIQUE OPPORTUNITY FOR A SYSTEMATIC, LONG TERM EFFORT.

N92-25927

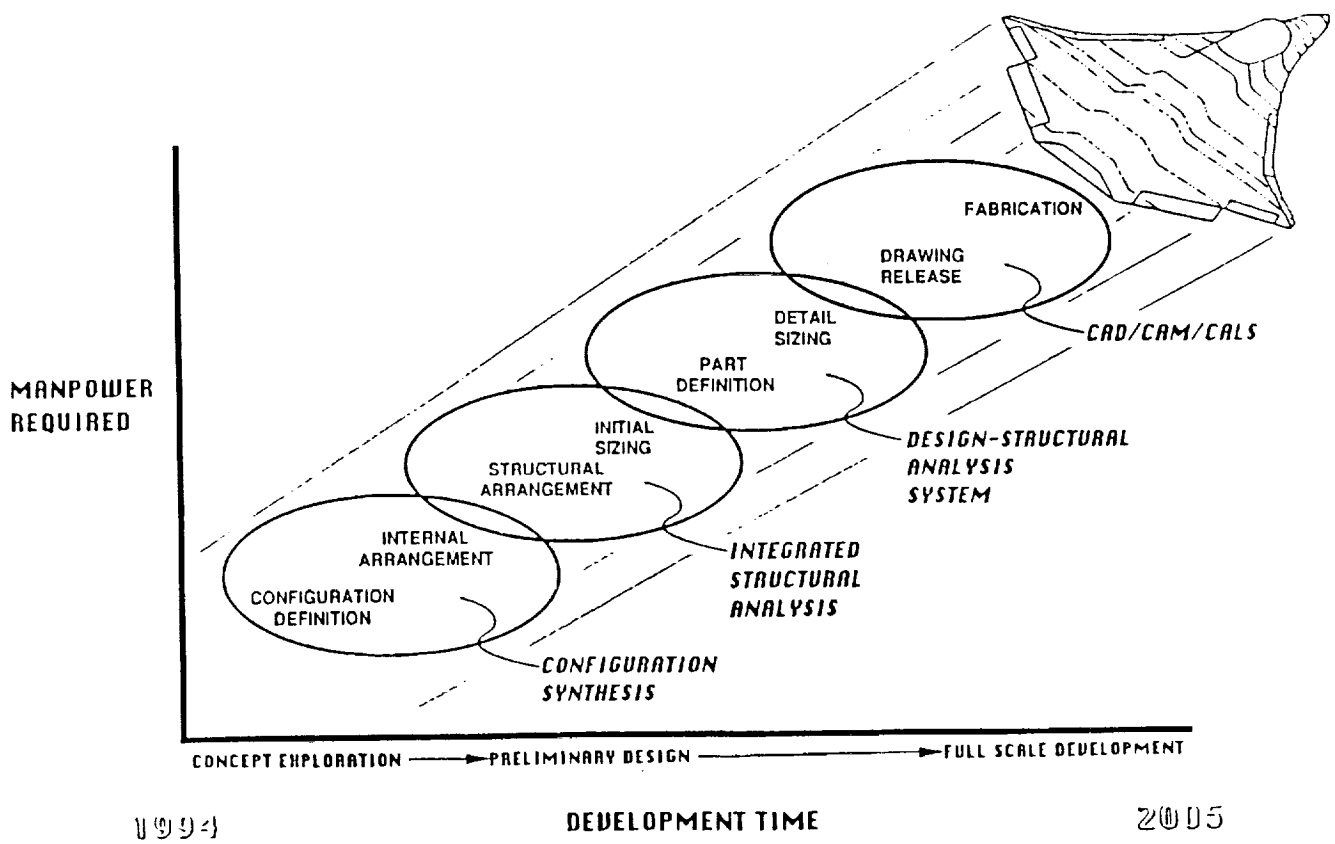
Application of Integrated Structural Analysis to the High Speed Civil Transport

C. R. Saff
McDonnell Douglas Company
McDonnell Douglas Corporation
St. Louis, Missouri

INTEGRATED STRUCTURAL ANALYSIS (ISA) - A MULTIDISCIPLINARY TOOL FOR PRELIMINARY DESIGN

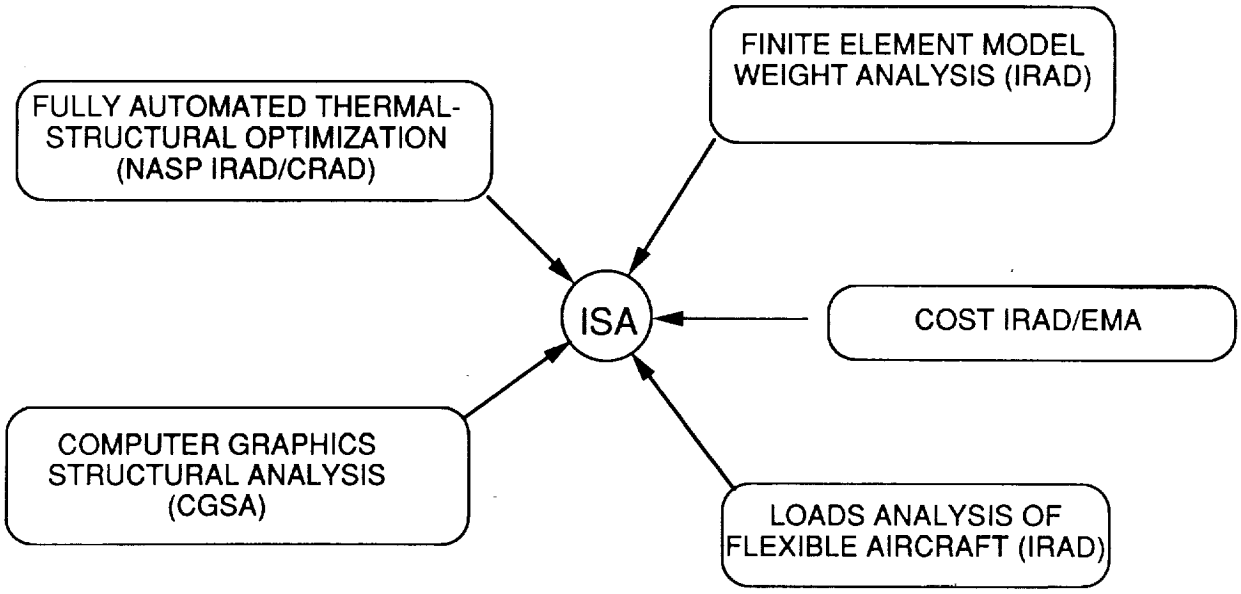
The Integrated Structural Analysis (ISA) system was developed in response to the need to define lightweight structures early in the design process for military aircraft. With the advent of stealth technologies, which change both outer moldline configurations and determine much about the skin materials, the need for lightweight, efficient internal structures becomes crucial for development of high performance aircraft. In studying the design processes used to produce previous aircraft, we found that many of the decisions that added weight to the airframe were made early in the design process when the number of people assigned to the project are small and experience plays a key role in the efficiency of the design. We also learned that much of the weight added in later portions of the design process were the result of not having all disciplines represented in the design development and review process.

The objective of the ISA system is to permit rapid development of an optimized skin and big-bones structure that insures that the requirements of strength, dynamics, loads, and weights are met. While the system was developed originally for fighter aircraft, it is applicable to commercial transports or any other structure. The design requirements dictate the structural arrangement and sizing limitations to be imposed.



THE ISA SYSTEM INTEGRATES SEVERAL SYSTEMS INTO SINGLE DESIGN SYSTEM

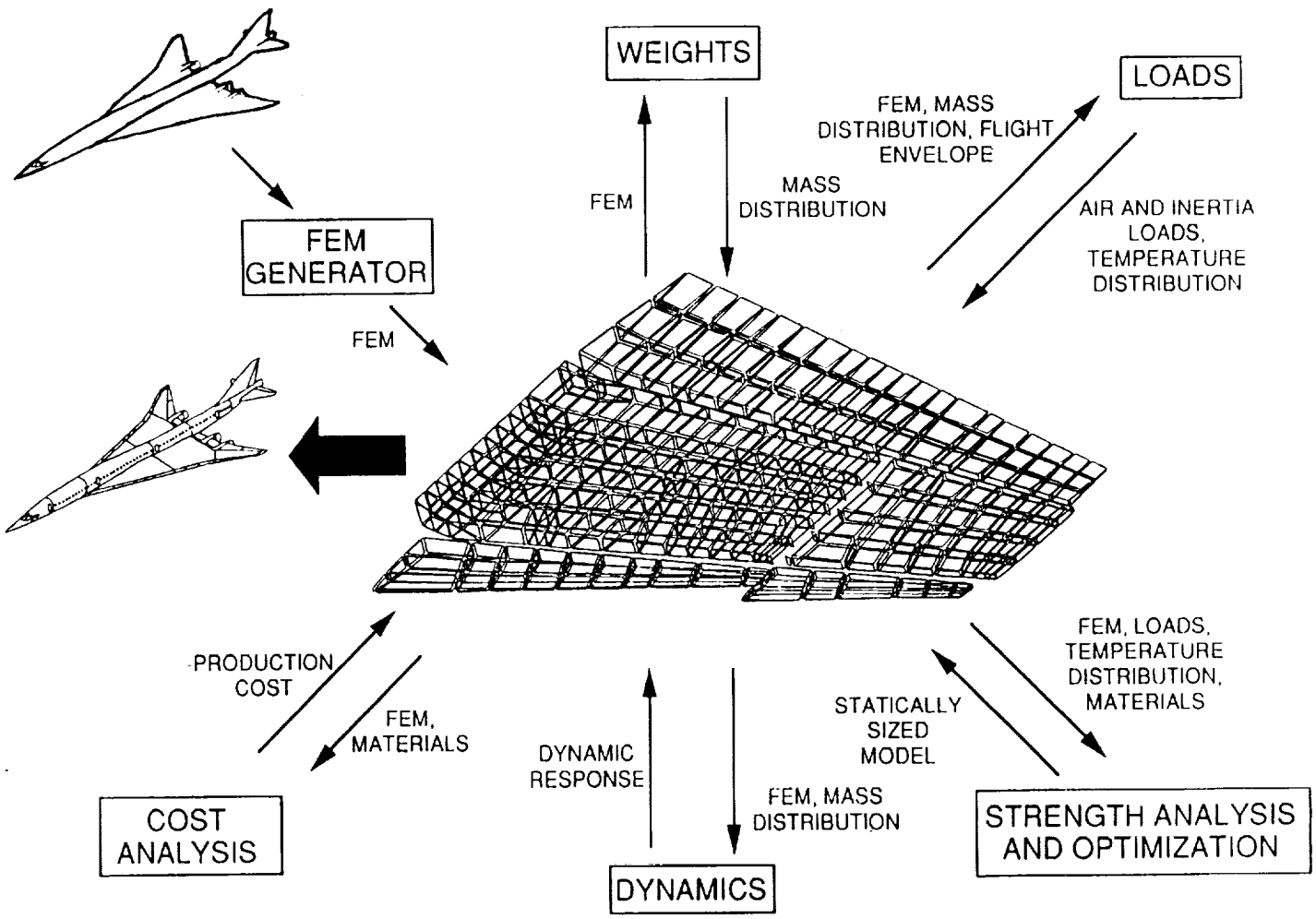
The Integrated Structural Analysis (ISA) system couples panel and bar optimization codes developed for NASP to a sophisticated weights estimation technique that works directly from the data incorporated into a finite element model, to a loads module that computes both aerodynamic loads and inertial loads for dynamics analysis. All of these analyses are linked through the development of a simplified finite element model developed using a graphical model development environment (labeled CGSA in the chart). The system is modular and components of the system are being improved and replaced as better products are verified. The system does not always use internally developed codes. For example NASTRAN is used as the finite element analysis module and the PATRAN neutral file is used to communicate certain information to various disciplines and codes.



NASTRAN IS USED AS THE PRIMARY FINITE ELEMENT PROCESSOR

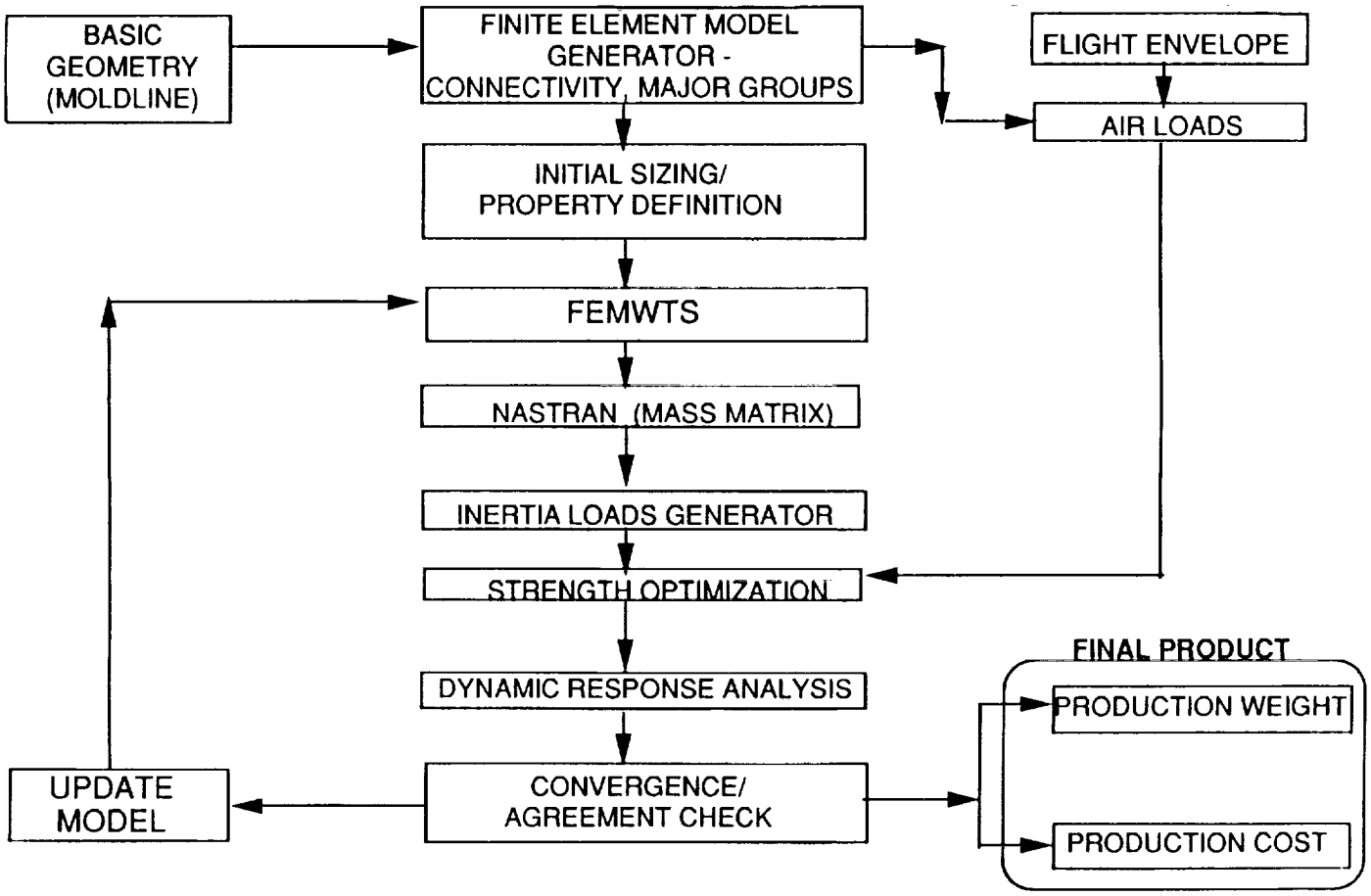
THE ISA SYSTEM USES THE FINITE ELEMENT MODEL TO LINK DISCIPLINES AND COMMUNICATE STRUCTURAL DEFINITION

The key to the ISA system is that each discipline communicates through the finite element model. Each software package that performs the required analysis gets its information concerning the geometry, materials, stiffness, densities, strengths, and loads from the finite element model and returns its results to the finite element model. In order to insure that each discipline receives the data required and communicates the results required, the development group contains members from each discipline. These representatives define how much of the structure the model must represent, how the structure will be modeled, and what each element represents. These decisions form the communication base that insures that when one discipline examines a portion of the model, the structure represented by the model is understood. When advanced structures are modeled, the group reconvenes to insure that the communication path is still intact. In spite of the wonders of automation, and the speed with which the optimization can now be performed, it is really these communication meetings that insure that each discipline understands the language of the finite element model.

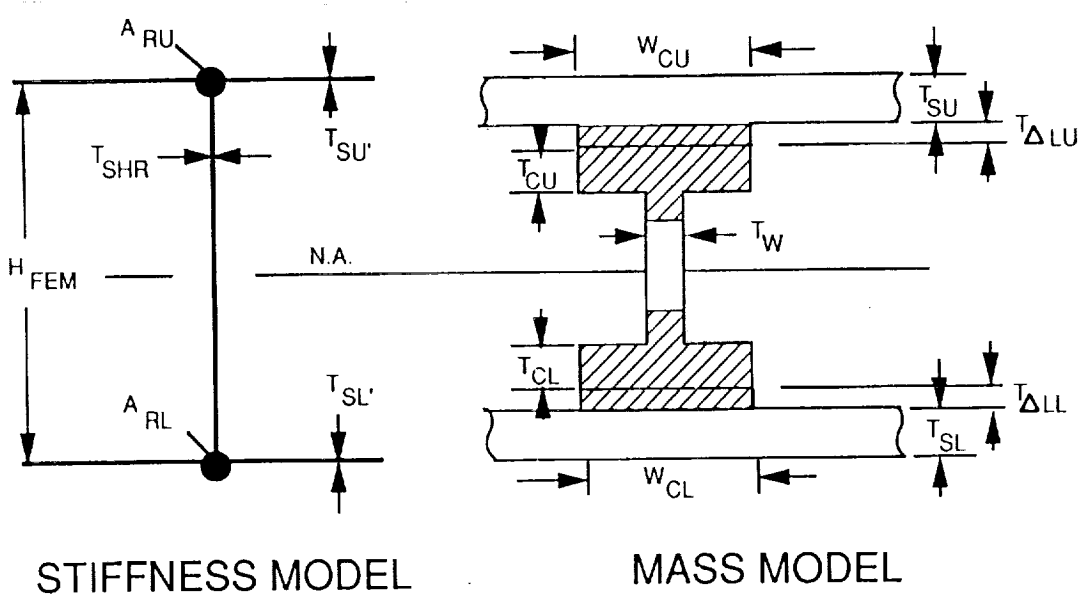


GLOBAL FLOW CHART

While the original goal of the ISA system was to develop a concurrent engineering approach to definition of optimized internal structural arrangement, it was found necessary to perform tasks sequentially to insure that each discipline had the data required for its analysis. For example, weights analysis is required to determine the mass distribution through the structure for inertia loads and dynamics analysis, and balanced aerodynamic loads are required before strength can do sizing. It was found that much in the model definition required dynamics considerations to define element types, connectivities, etc. Moreover, dynamics must check a strength optimized design for global buckling, overall aeroelastic deflections, local dynamic problems, as well as flutter speed. Considerable effort goes into file management to insure that each discipline knows what version (pass through the optimization process) is being worked. Convergence is obtained when component weights do not change by more than five (5) percent between iterations. This usually occurs within three iterations. Once convergence is obtained, then final costs and weights are computed to allow comparisons with other configurations.

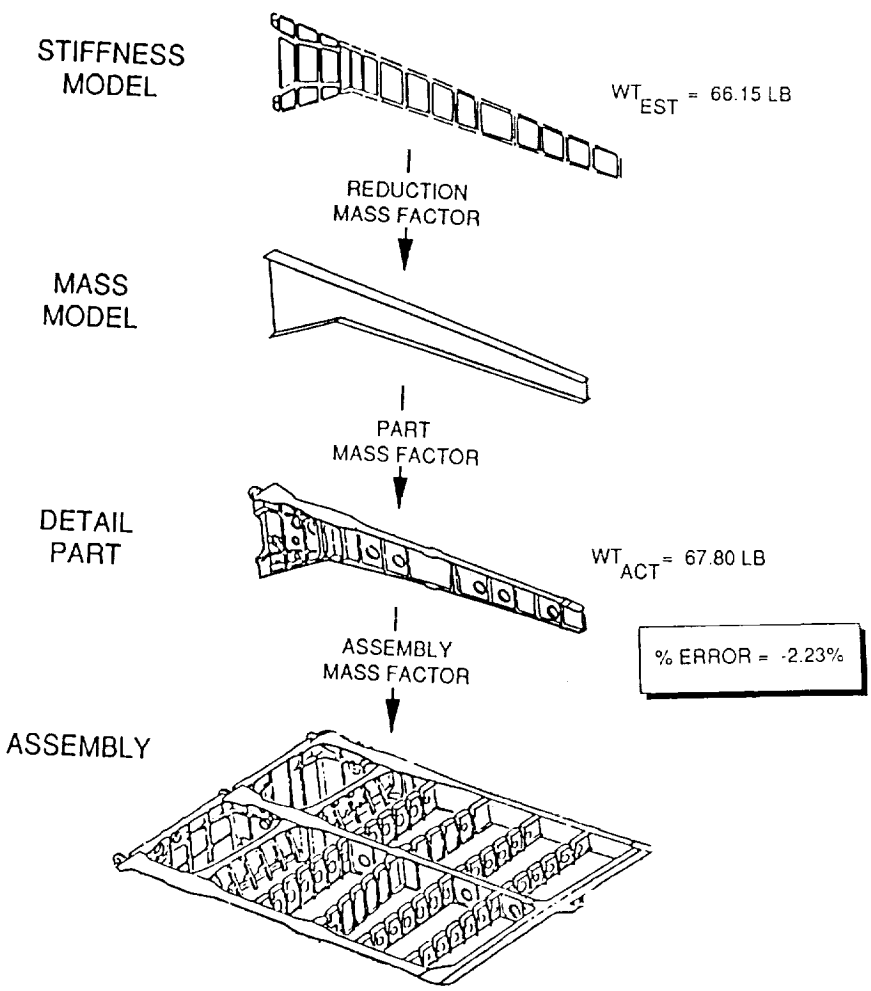


As an example of how the finite element model is translated in each discipline, consider the problem of distributing mass through the model. The finite element model (finite element model) can be formulated a number of ways. In the method shown the skins and nodes are located at the outer moldline of the part. To estimate the mass of the part as closely as possible, the geometry defined by the model must be understood and agreed upon by both the modeler and the weights analyst. In this case an I-beam spar, skin, and skin land are modeled. The weights analyst transforms this finite element model into a mass model that accounts for the skin thickness, the cap stiffness, and the web thickness. In this case skin and web thicknesses are left as defined in the finite element model, and most of the effort goes into determining the area and mass of the cap/land combination modeled by the bar in the finite element model. This analysis matches the neutral axis of the section and the stiffness of the cap/land combination. Once the cap thicknesses are established, then the web length is determined and the masses are used to adjust the densities in the finite element model. This adjustment is known as the reduction mass factor.



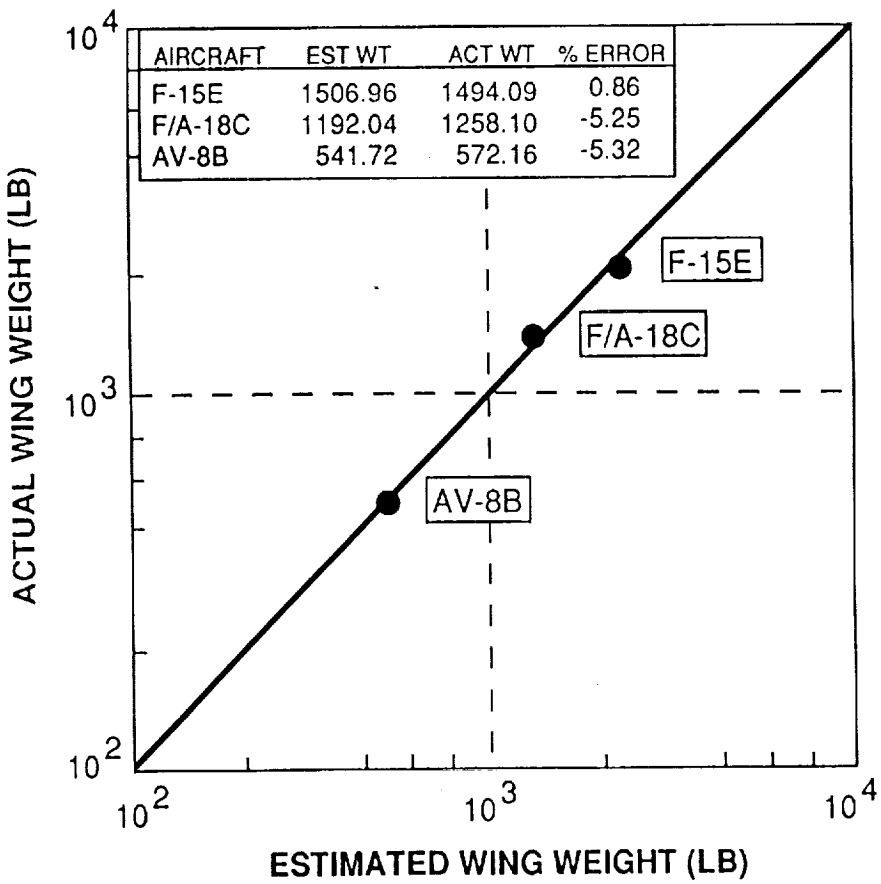
WEIGHT ANALYSIS PROCESS

The weight analysis process involves several mass factor adjustments to go from the stiffness, or finite element model to the part mass factor in its assembled condition including attachments. We described the reduction mass factor in the previous chart. There is a part mass factor which takes that mass model and relates it to a detailed part using a historical data base that accounts for the detail of the model and additional masses that are associated with parts of this type. The database includes all of the aircraft built at MCAIR, and, unfortunately, is specific to MCAIR manufacturing and design methods. Therefore, it probably is not useful outside of MCAIR. What would be useful is the database setup and the techniques used to include advanced technology components. These components have to be added to the database in order to effectively estimate weights for advanced design programs. Data for the HSCT will be added to the database using these routines. This chart shows a comparison between the weight estimated from the stiffness model of an F-15 wing spar and the weight of the part as weighed. Mass times density weights are often in error by two orders of magnitude.



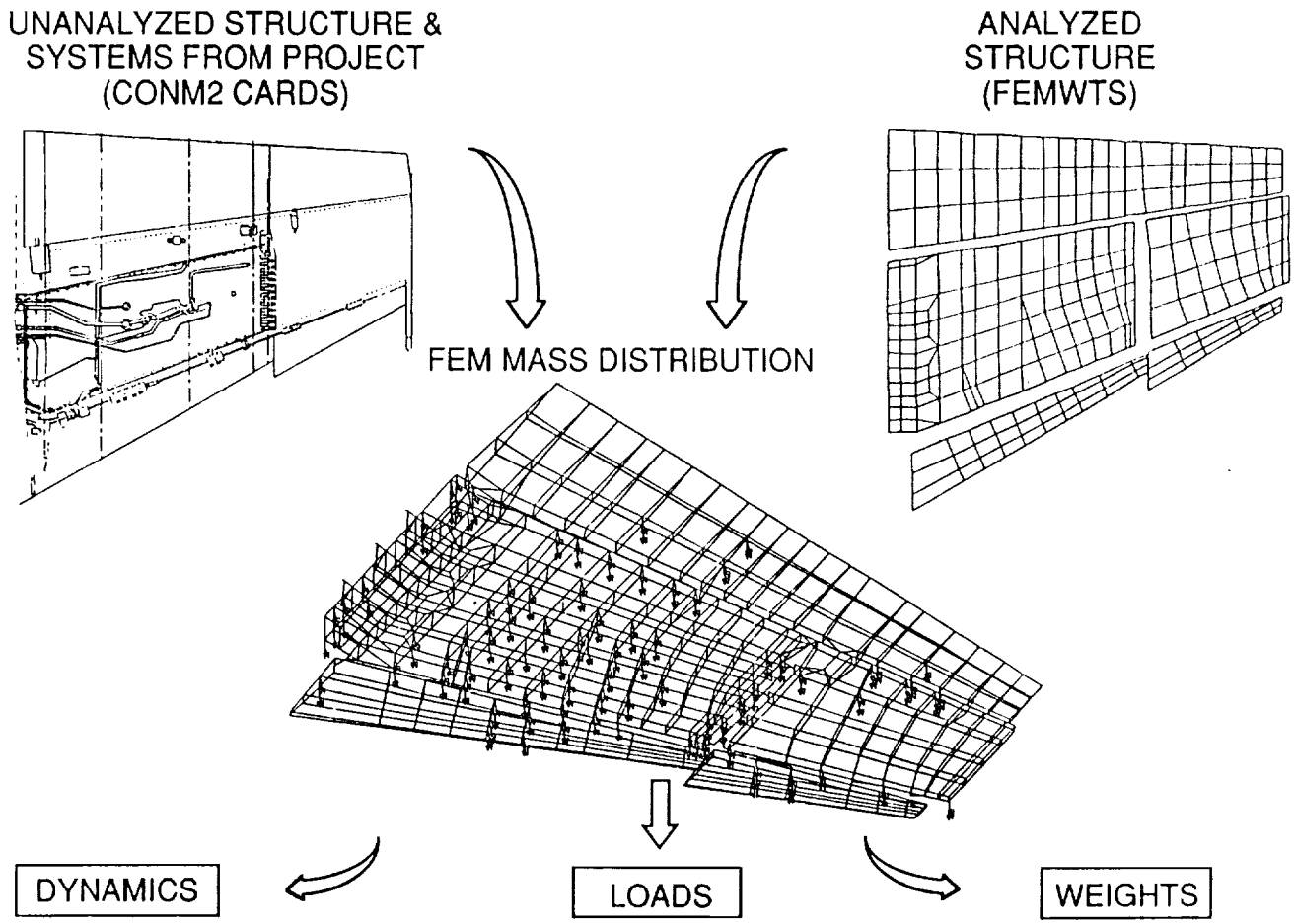
WING TORQUE BOX AND TRAILING EDGE WEIGHT CORRELATION

To verify our weight predictions, especially from very simple finite element models, like those used in preliminary design, we compared the weights estimated from simple models of existing aircraft with actual weights. Although the analysis method was developed using detailed, production finite element models for the aircraft chosen for verification, only data from skins, spars, and ribs were included for this verification exercise. As shown, for skins ranging from all metal (F-15E), composite skins over metal substructure (F-18C), to all composite (AV-8B), the predictions were within six percent of the actual weights.



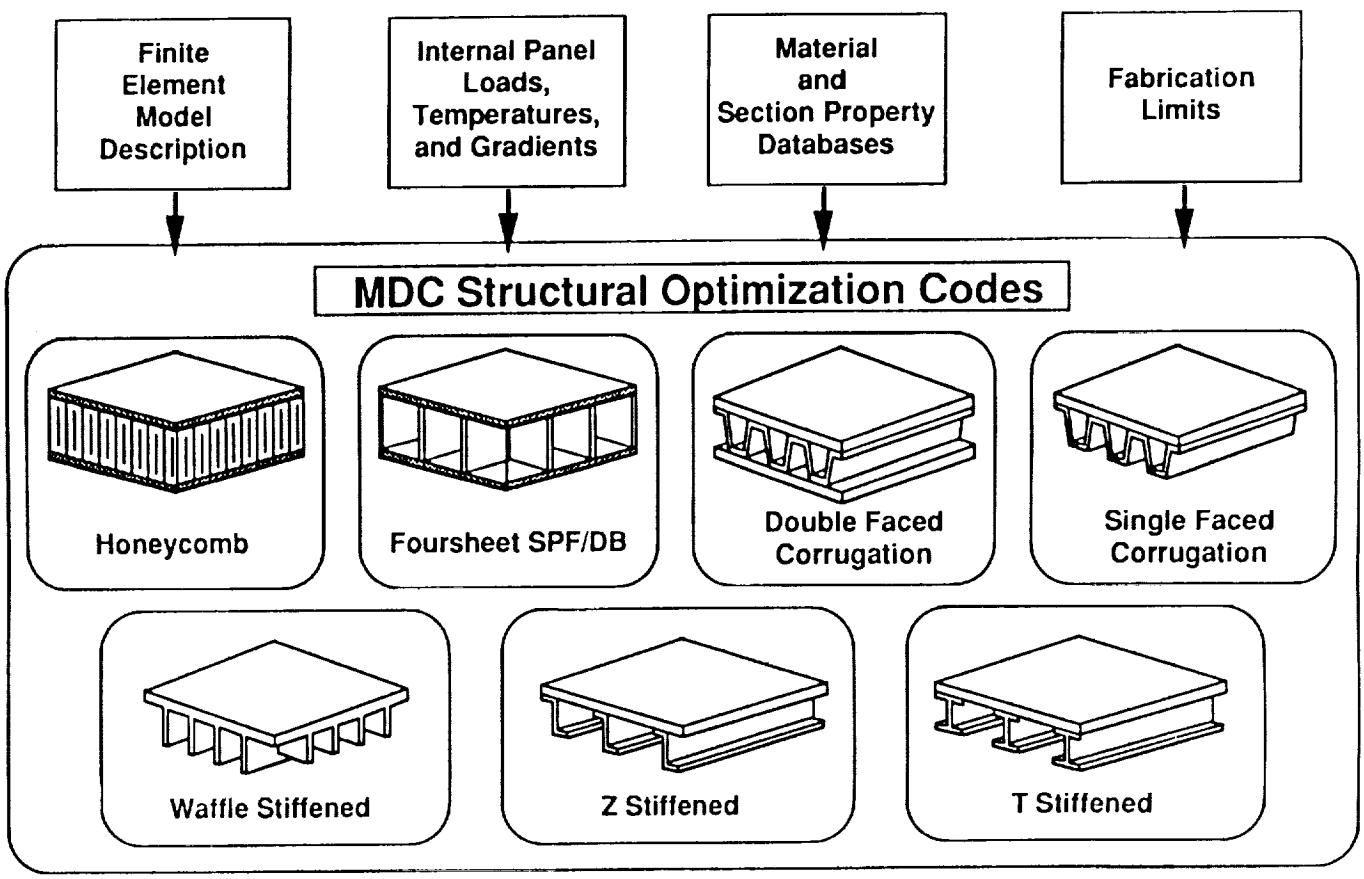
WING MASS DISTRIBUTION

Structural masses are only half of the weights problem. Non-structural masses make up at least half of the total weight, and these masses must be included to develop accurate predictions of inertial loads, natural frequencies, vibration modes, or flutter speeds. Methods and software have been developed to distribute these masses depending upon system location within the structure. Currently, these masses are computed from the database system but must be added to the finite element model by hand. Efforts toward automating this procedure are underway.



OPTIMIZATION PROGRAMS ARE USED TO SIZE SKIN PANELS

Years ago MCAIR developed some simple codes to optimize skin thickness and stiffener spacing and sizing for monolithic metal panels. Under the NASP program, these codes were updated to include composite skins, honeycomb, SPF/DB, waffle stiffened, and hat stiffened panels. These analyses pull geometry, material, stiffness, thermal, and mechanical load data from the finite element model. Fabrication limits are imposed through lower limits on thicknesses and thickness increments. These codes use a simple fault tree type of optimization routine to find the least weight design that satisfies the strength criteria imposed. Under the fault tree analysis scheme, weights of potential solutions are checked against weights of solutions that have passed the strength criteria. If the weight is greater, then the potential solution is discarded; if less, then the strength criteria are checked. Because the strength checks are not done unless the potential solution is lower weight, many potential solutions are discarded, and the minimum weight solution is found very rapidly even though all solutions are checked for weight.



STRUCTURAL CRITERIA CONTAINED IN THE OPTIMIZATION CODES

The structural failure criteria examined in the optimization codes include static failures due to exceeding ultimate strength, overall panel buckling, local buckling of stiffeners or unsupported face sheets, crippling, or beam column stability. The optimization codes examine each of the load conditions included in the finite element analysis, including thermal loads and thermal gradients. Material properties are extracted from a material property database that includes metal and composite properties at a number of temperatures. The temperatures for each element are used to interpolate the mechanical and physical properties in the database. The codes include internal checks for manufacturing limitations on minimum gages, minimum laminate thickness, ply thicknesses, and orientations. The user has control over these checks. They can be applied over the full range of variables over limited ranges. Certain variables can be eliminated from consideration by the user if they are not of interest for the particular component.

- **Section Failure Modes**
  - Ultimate Stress for All Components and All Directions
  - Tsai-Hill Interaction for Composite Materials
  - Overall Panel Buckling (Bi-axial and shear loadings)
  - Local Component Stability Failure
    (e.g., Cross Corrugation Buckling - TWOSHEET
    or Facesheet Wrinkling - HONEYCOMB)
  - Beam Column Stability
  - Local Crippling
- **Load Conditions**
  - Multiple Load Conditions Satisfied Simultaneously
  - Thermal Loads and Thermal Gradients
- **Material Properties**
  - Material Database
  - Isotropic and Orthotropic Material Properties
  - Composite Materials
  - Mechanical and Physical Properties
- **Manufacturing Capabilities**
  - Internal Checks for Allowable Geometries

PANEL WEIGHTS CALCULATED USING PANEL OPTIMIZATION CODES

Douglas Aircraft Company has used the panel optimization codes described above, to evaluate various panel configurations for isolated locations on their HSCT design. Fuselage load conditions included 150 percent of design limit pressure and mechanical loads including 125 percent of thermal loads when they are additive or reduced by 100 percent of thermal loads when they reduced the mechanical loads. Other conditions checked included: 150 percent of mechanical loads alone (with appropriate thermal loads) and 200 percent of limit fuselage pressure. The thermal loads include loads due to thermal expansion and those due to thermal gradients.

FUSELAGE DESIGN LOAD CONDITIONS CHECKED:

1) 1.5 x (LIMIT MECH + LIMIT PRESS) OR
   + 1.25 x (THERMAL ADDITIVE)
   - 1.00 x (THERMAL SUBTRACTIVE)

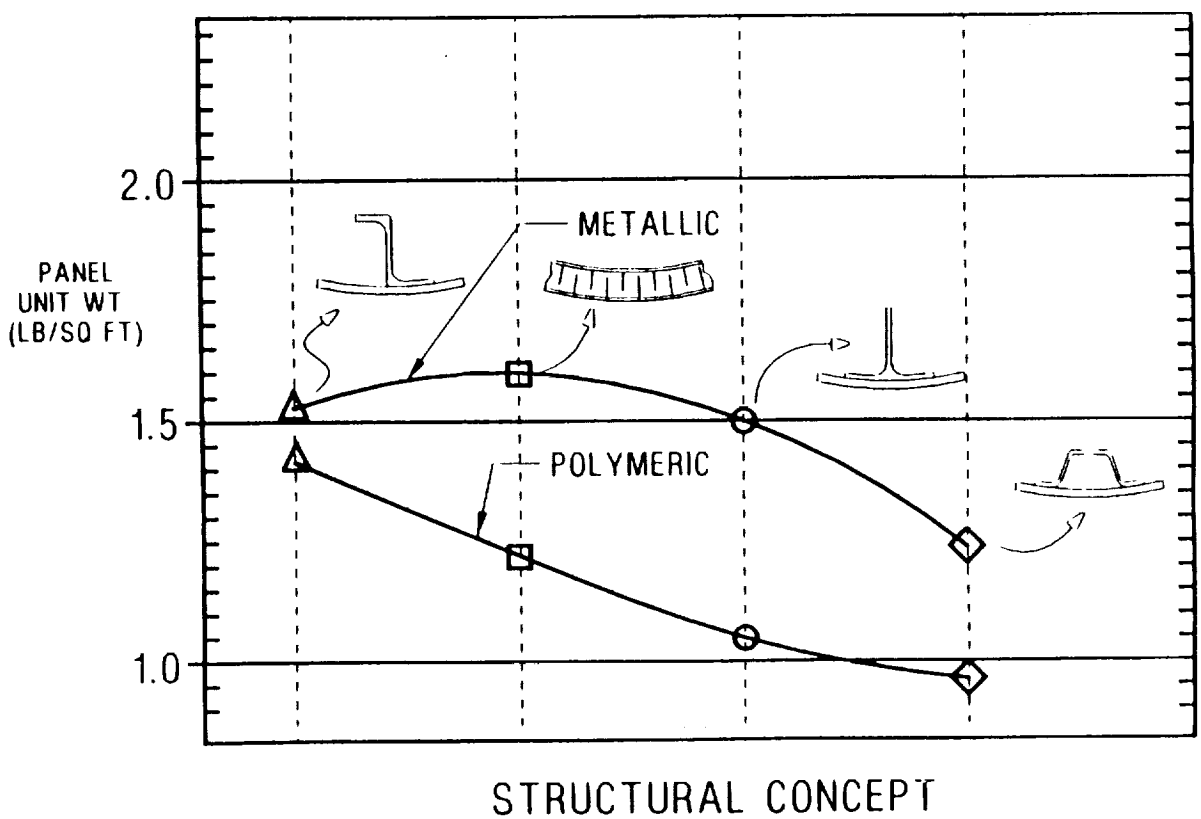
2) 1.5 x (LIMIT MECH) OR
   + 1.25 x (THERMAL ADDITIVE)
   - 1.00 x (THERMAL SUBTRACTIVE)

3) 2.0 x (LIMIT PRESS)

NOTE: THERMAL LOADS INCLUDE $\alpha \Delta T$ (Nx,Ny) AND THROUGH THICKNESS GRADIENT LOADS (Mx,My).

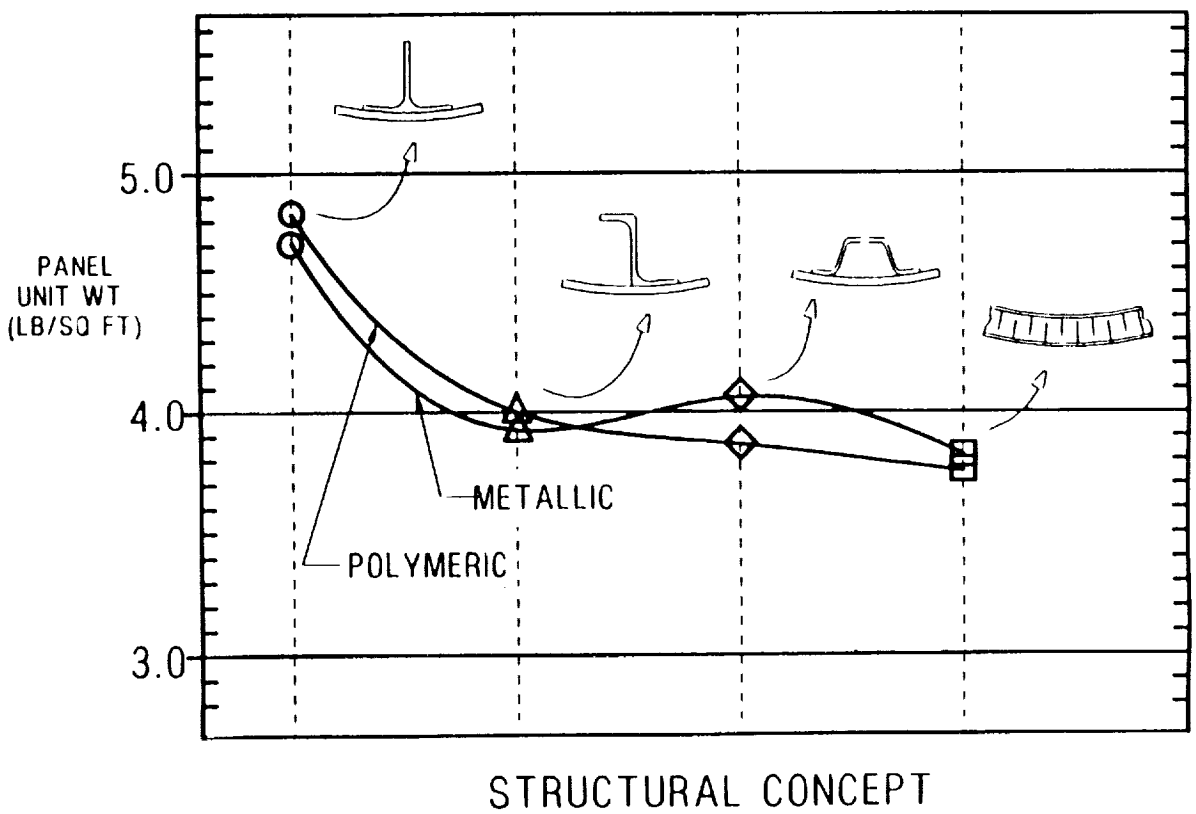
POINT DESIGN PANEL WEIGHTS (FORWARD FUSELAGE)

For the lightly loaded panels in the forward fuselage, optimization codes showed that resin matrix composites show substantial payoff over metallic designs. In both material systems, hat stiffened design showed the greatest weight savings, greater than honeycomb stiffened panels. This result was important because Douglas has been trying to eliminate honeycomb from its structures as much as possible because maintenance of such structures is very difficult and Douglas expects the airlines will consider honeycomb structures as a life cycle cost detriment to HSCT designs. Discretely stiffened designs can be repaired using conventional repair techniques (with some care to insure the integrity of the composite materials).

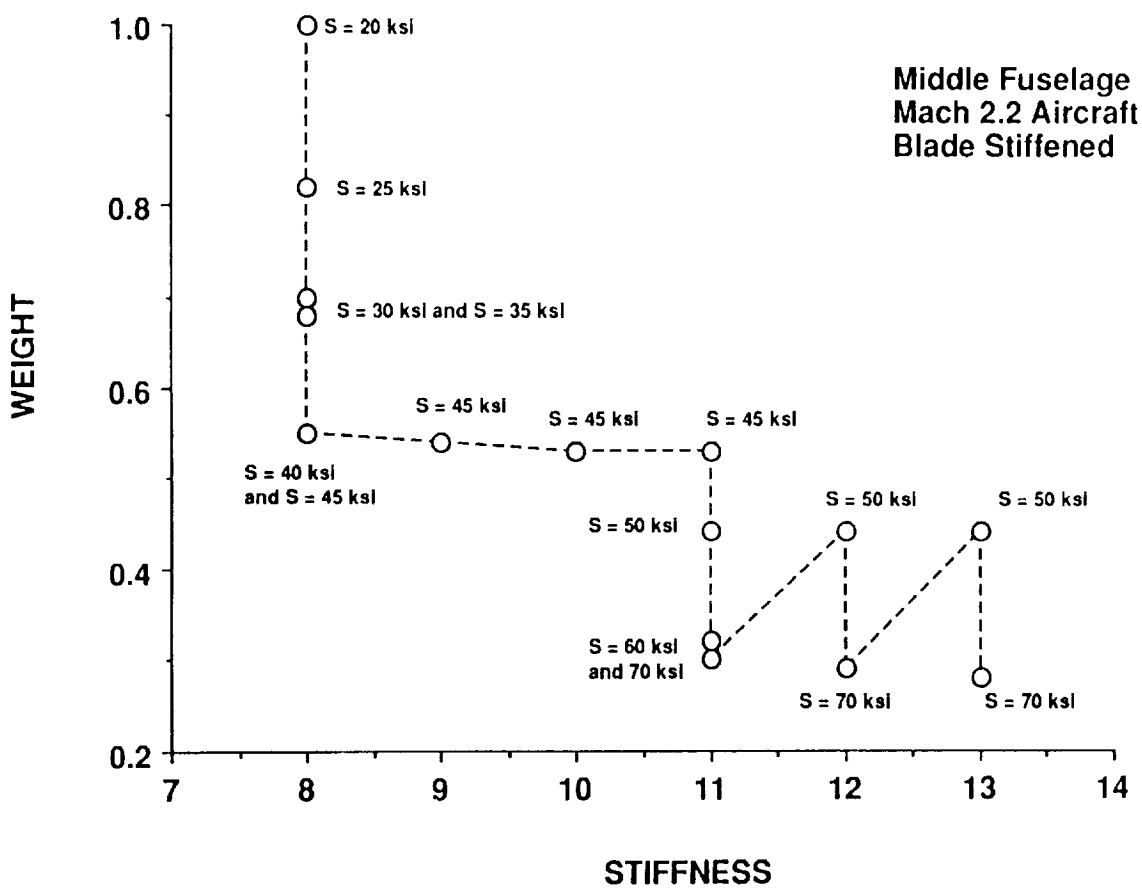


POINT DESIGN PANEL WEIGHTS (AFT FUSELAGE)

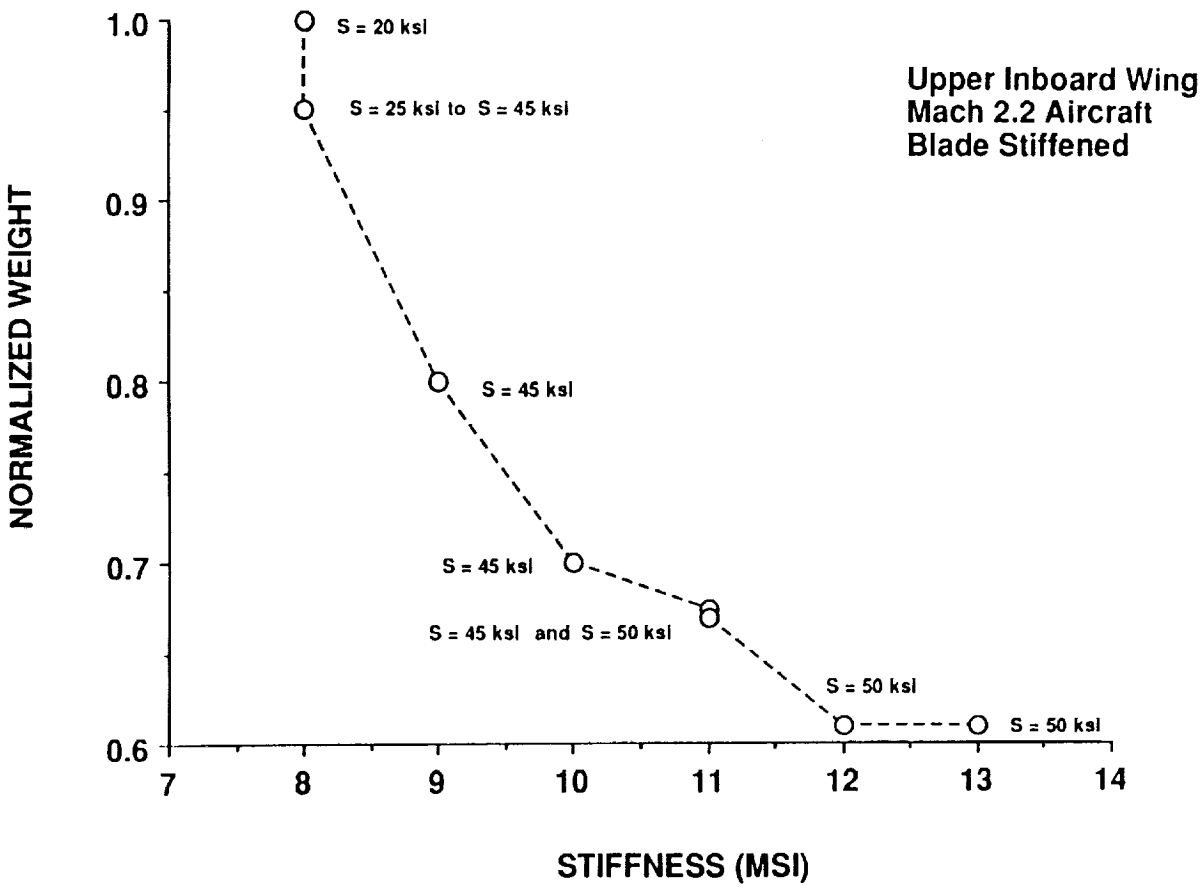
For the more highly loaded fuselage panels of the aft fuselage, there is little difference between metallic and composite minimum weight designs. Here the honeycomb panels show slightly lower weights than the other panel configurations, however; the difference is very small. In fact, there is very little difference in panel weights between honeycomb panels, hat-stiffened, and z-stiffened panels. Blade stiffened panels are significantly heavier. Because mechanical loads and load intensities are high on these panels, stiffener caps are necessary to provide panel stability. Therefore, the blade stiffened panel shows considerable weight increase over the other configurations.



The optimization codes are also being used to determine the best materials for the HSCT airframe. To determine the effects of material stiffness and strength changes on panel weight in the center fuselage area, the analysis was initially used to size an isotropic panel. The panel results were normalized to a 20 ksi operating stress level in an 8 msi stiffness isotropic panel. As strengths were increased to reflect improved material systems with the same stiffness and density, the panel weights decrease. However, once the strength levels increase to the point that operating stress levels can be 40-45 ksi, buckling criteria begin to define failure and the weight cannot be driven lower at that stiffness. To drive the weight down stiffnesses were increased to reflect materials having improved moduli. This drives the weight lower until the stiffness hits 11 msi. At that point, strength again limits the weight. In the remaining analysis, strengths and stiffnesses are alternately increased to determine minimum weight designs. This type of data can be used to formulate curves that represent minimum weight panels with given operating stress levels and stiffnesses. Then given a new material with projected strength and stiffness, the payoff in panel weight for that material system can be determined. In the development shown in this chart, the change from strength to stiffness driven structures results in rather abrupt changes in panel weights.

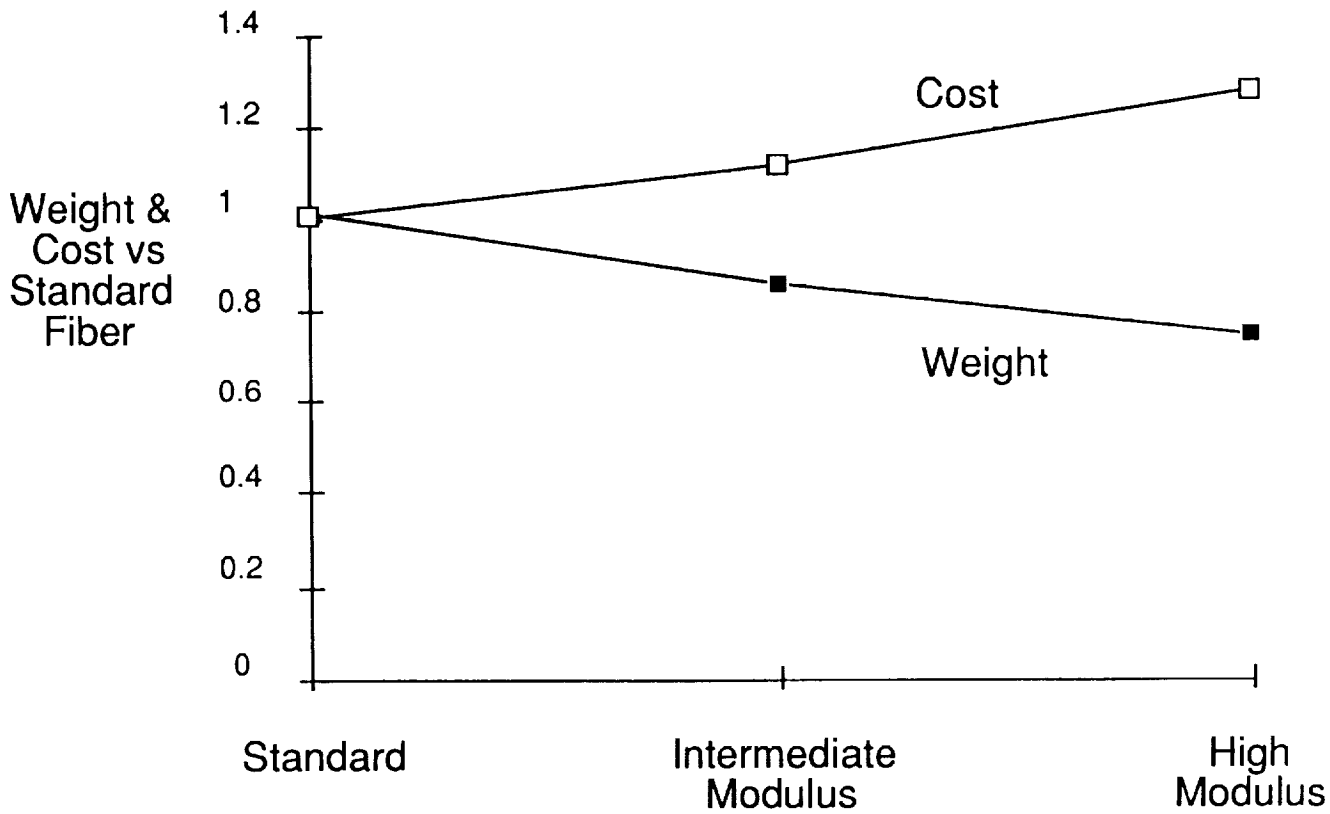


In this example of material evaluation using the optimization routines, strengths and stiffnesses are increased to show the minimum weight blade stiffened panels that can be fabricated for the range of material stiffnesses shown. At 10 msi stiffness the maximum useful operating stress level is 45 ksi. Any higher stress level changes the failure mode but does not decrease the panel weight. Similarly, at 45 ksi the maximum useful stiffness is 11 msi. Any higher stiffness makes the panel strength critical and does not drive the weight down. Again, comparison with material systems proposed for the HSCT wing will rapidly demonstrate the system having least weight for the blade stiffened concept.



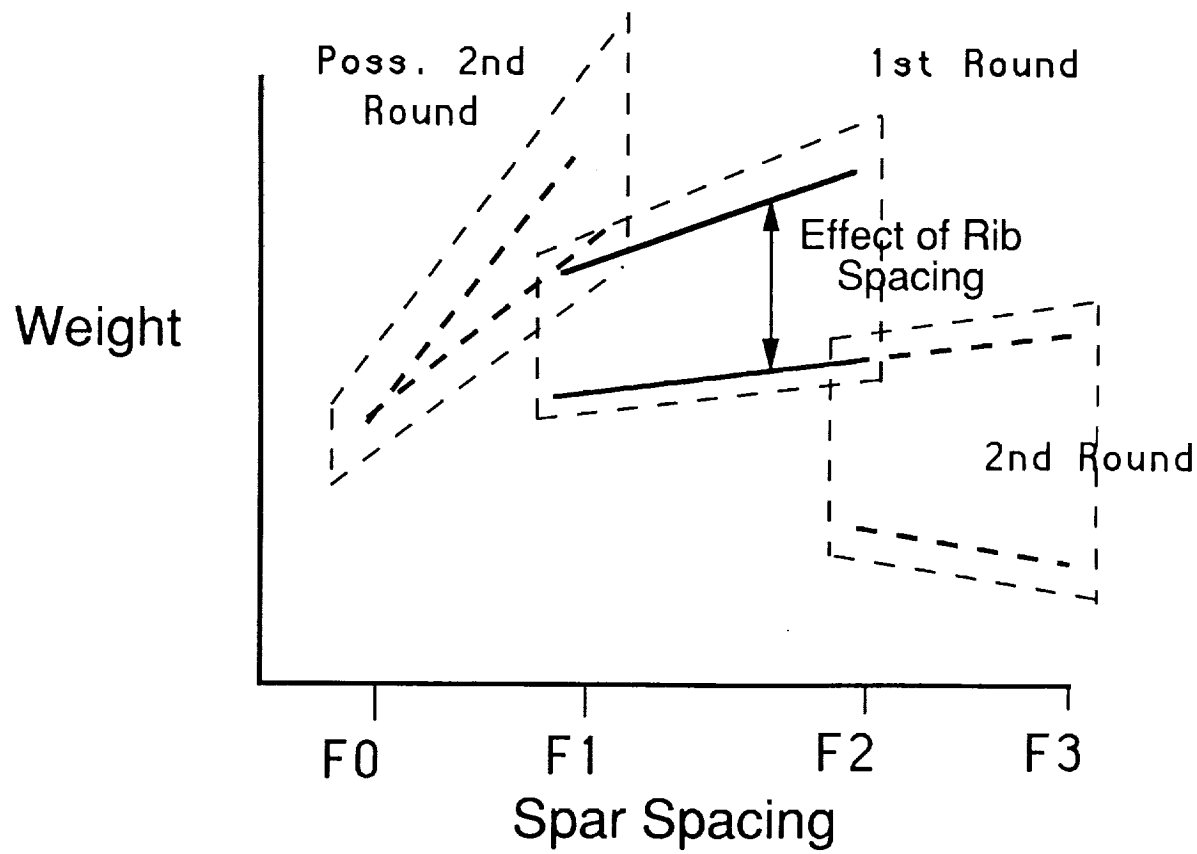
PAYOFF ASSESSMENT

Least weight is not the only feature that must be balanced in the HSCT design. Costs must be low enough that the passenger fares fall within the target range. Therefore, payoff assessments, like that shown in this chart, are performed to determine the cost of a pound of weight savings. These types of analysis are crucial to determining which material systems will find their way onto the HSCT airframe.



TAGUCHI ANALYSIS USED TO DETERMINE OPTIMUM CONFIGURATION

To determine the best internal arrangement of spars and ribs, the ISA analysis uses a Taguchi design of experiments approach. In this analysis the analyst chooses a few configurations to be evaluated for least weight and cost. The Taguchi analysis provides a systematic approach for determining which variables to include in each of the configurations to be analyzed. In this method a design space containing more than one hundred test conditions can be sampled with seven strategically selected tests. Based on the results of those few tests it is possible to extrapolate or interpolate the results to a new, and potentially lower weight, design. If desired this new design point can be verified by test or new conditions surrounding this design point can be identified for another round of focused testing (or analysis in this case). The advantage of the Taguchi method is that the analyst has control of the design conditions that are traded and the ranges over which they are traded, while obtaining the maximum information in the minimum number of samples.



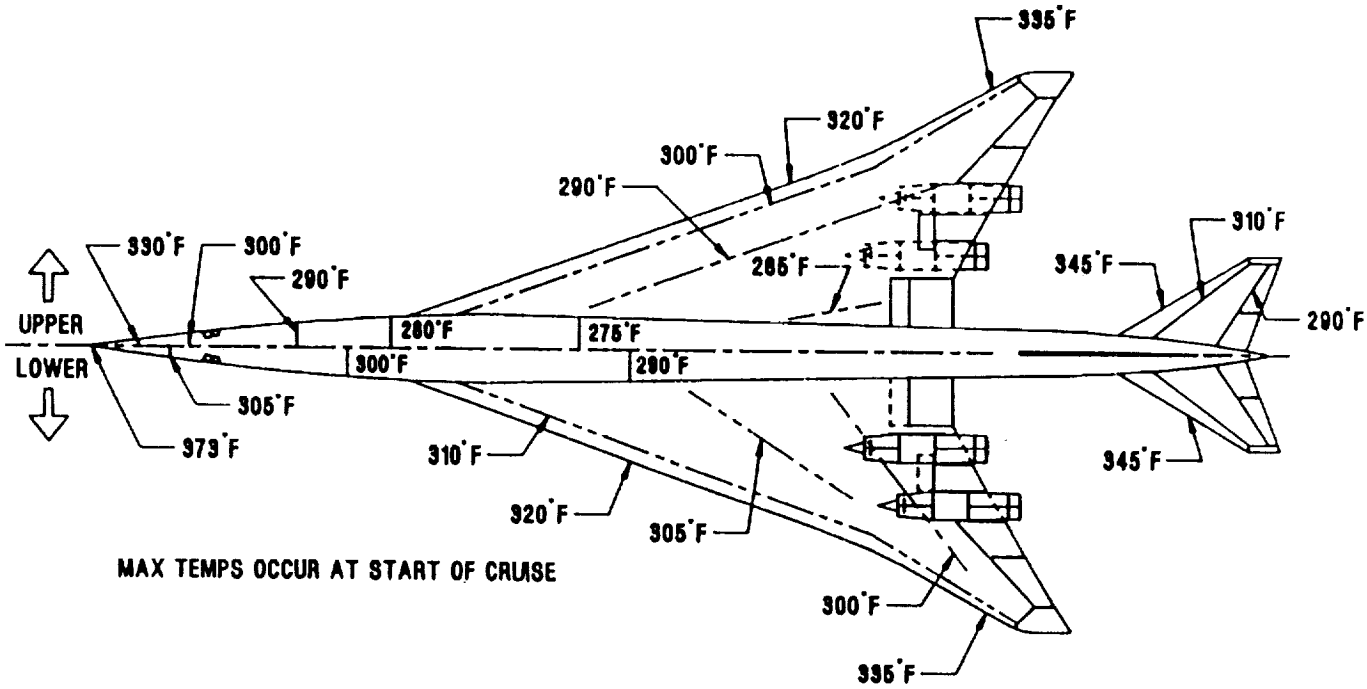
KEYS FOR HSCT ANALYSIS

The primary concerns for the HSCT airframe that have received little attention so far are thermal mismatch between subcomponents and the effects of this mismatch on joint loads, and the need to include durability and damage tolerance requirements that reflect the properties of materials aged at temperature for 60,000 hours of flying. In the large, cocured structures used to minimize the cost of composite components, thermal gradients from skin to substructures could be a source of durability problems. To avoid these problems, representative structures must be fabricated and subjected to the expected environment to assure that thermodynamic analyses can predict the thermal gradients, and that subsequent predictions of joint loads and strengths are accurate. Furthermore element tests of aged joints are required to insure that the adhesive or bolted attachment properties and strengths are properly included in the analyses.

- **THERMAL LOADS DUE TO CTE MISMATCH MUST BE CONSIDERED**
- **JOINT LOADS/STRENGTHS MUST BE DEFINED**
- **DAMAGE TOLERANCE/DURABILITY REQUIREMENTS MUST BE REFLECTED IN ALLOWABLES**

TEMPERATURE REQUIREMENTS FOR A MACH 2.4 AIRCRAFT

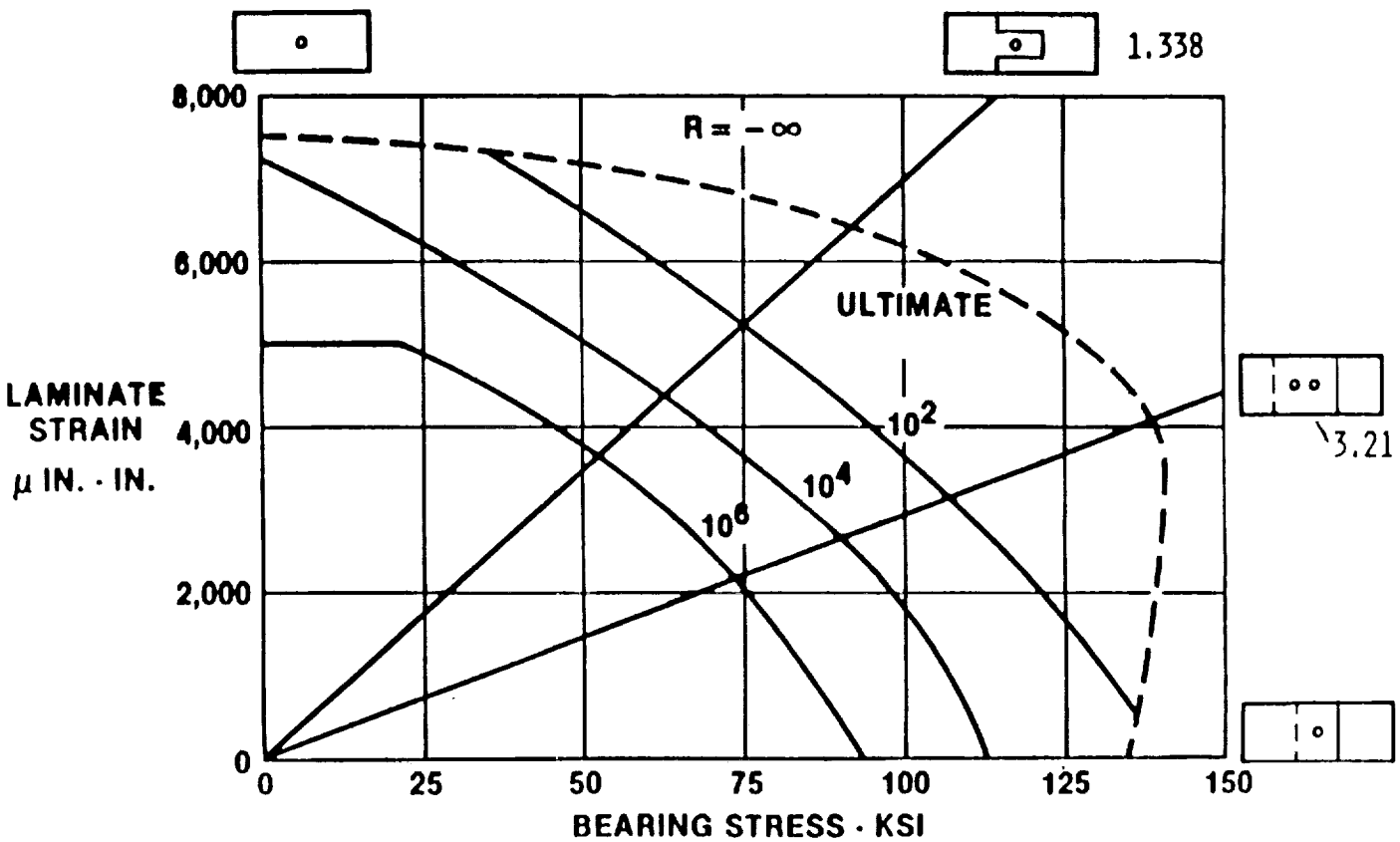
At Mach 2.4 and 375 degrees Farenheit, thermal mismatch is not likely to lead to large joint loads between components. Rather, the thermal mismatch problems are likely to be local problems between skin and substructure where ply orientation differences can create large strain mismatches. Such problems can also be caused by thermal gradients that produce large strain differences between hot skins and cooler substructures. This will be more of a concern in primary wing box and fuselage structures where the substructure is cooled by fuel or by air conditioning of the passenger compartment. To achieve minimum weight, one wants to minimize insulation that does not carry load. This can be done either by choosing insulation that can carry some load or by choosing structures that provide some insulation capability of its own. This drives the interest in honeycomb structures for the HSCT. But as we mentioned previously, honeycomb structures have historically resulted in maintenance problems, driving manufacturers away from its use.



MAX TEMPS OCCUR AT START OF CRUISE

JOINT DURABILITY

Data from element tests of joints subject to accelerated loads and thermal environments will have to be performed to determine the allowable loads for a range of joint types. Current programs are focused on determining accelerated aging tests for composite materials. Similar things must be done for joints, both bolted and bonded, that will simulate long term exposure to high loads and temperatures but not artificially change the failure modes induced in these attachments. Data like this will be a key to all HSCT designs.



Data from the joint and coupon durability tests must be incorporated into the databases used to design HSCT components and structures. In the database used within the ISA optimization codes, properties for a range of operating temperatures are input. Then the properties used for any element are interpolated from these tables to determine both the mechanical and strength properties to insure that load paths in the model represent those that would occur in the airframe at the end of its service life. For the HSCT, which sees long soak times at temperature during its overseas flights, the primary thermal loads probably occur at the beginning and end of flight as the aircraft accelerates to cruise speed or slows from cruise to landing approach speeds. These transient conditions may be important for the local failures due to thermal gradients.

Typical Properties

- **Available Materials**
  - Al 2024-T3 Plate
  - Al 7150- T61511 Plate
  - Al 7150 - T61511 Bar
  - Ti 6Al - 4V Plate
  - IM7 / Epoxy
- **Material Additions Made By User**

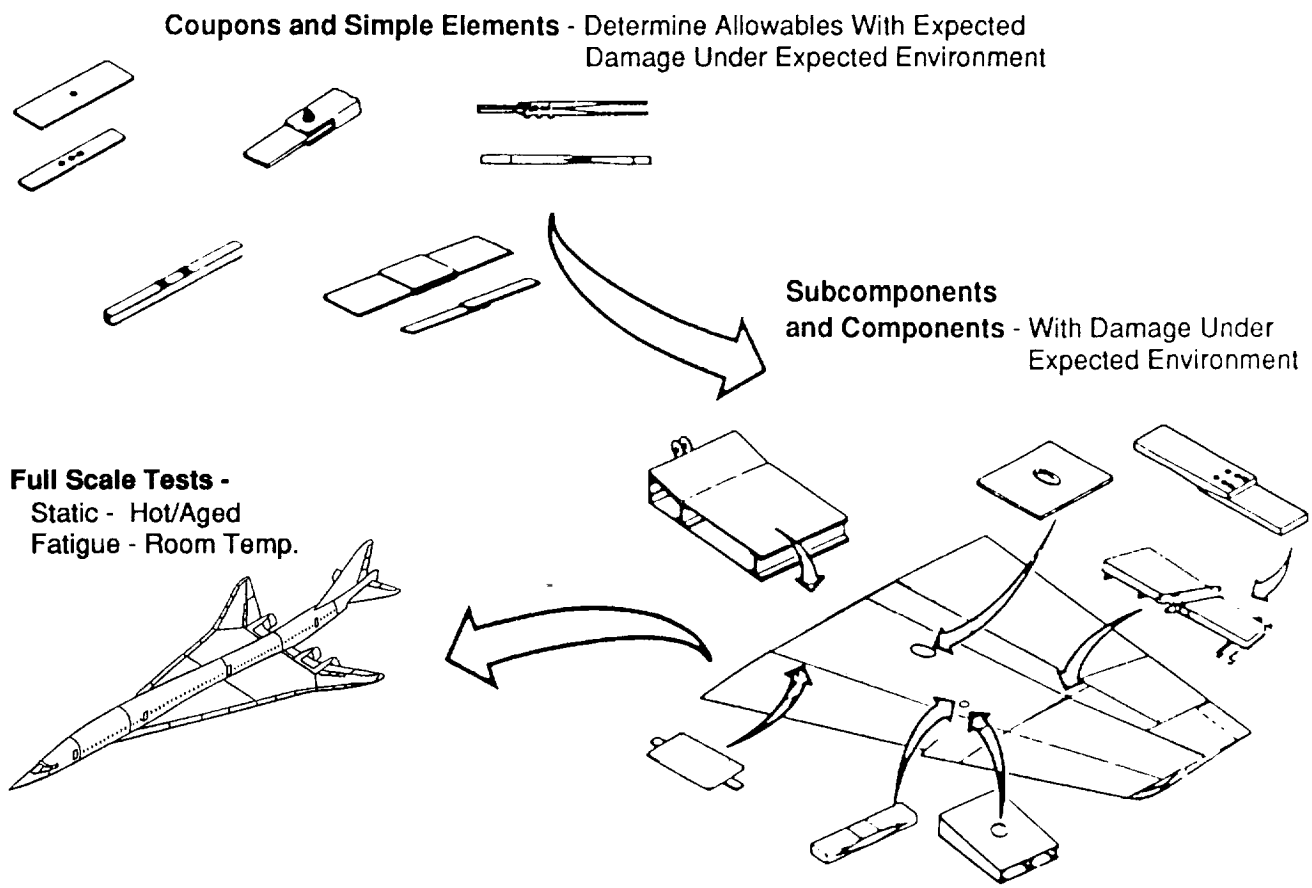
SOLID MATERIAL NUMBER 1 ENTERED BY T.M.Wilson ON 25-OCT-90
Aluminum 2024-T3 Sheet Mil-HndBk 5

DENSITY = 0.10100 LB/IN**3

MATERIAL PROPERTY		ANALYSIS PROPERTY	DATABASE PROPERTIES USED 1	2	3	4	5
1	TEMP	150.	70.	200.	300.	400.	500.
2	E1	0.1057E+08	0.1070E+08	0.1049E+08	0.1017E+08	0.9520E+07	0.8560E+07
3	E2	0.1057E+08	0.1070E+08	0.1049E+08	0.1017E+08	0.9520E+07	0.8560E+07
4	FTU1	60849.	64000.	58880.	52480.	48000.	32000.
5	FTU2	60849.	64000.	58880.	52480.	48000.	32000.
6	FALL1	60849.	64000.	58880.	52480.	48000.	32000.
7	FALL2	60849.	64000.	58880.	53480.	48000.	32000.
8	FCY1	38040.	39000.	37440.	35880.	32540.	25740.
9	FCY2	38040.	39000.	37440.	35880.	33540.	25740.
10	FSU	38040.	39000.	37440.	35490.	31980.	21840.
11	GXY	0.3951E+07	0.4000E+07	0.3920E+07	0.3800E+07	0.3560E+07	0.3200E+07
12	ALPHA1	0.127E-04	0.125E-04	0.129E-04	0.132E-04	0.133E-04	0.136E-04
13	ALPHA2	0.127E-04	0.125E-04	0.129E-04	0.132E-04	0.133E-04	0.136E-04
14	ALPH12	0.127E-04	0.125E-04	0.129E-04	0.132E-04	0.133E-04	0.136E-04
15	U12	0.33000	0.33000	0.33000	0.33000	0.33000	0.33000

STRUCTURAL ANALYSIS TIES TESTING TO CERTIFICATION

The computational structural analysis methods used to design the HSCT vehicle will be a key to certifying the airframe. Full-scale tests, particularly the durability tests, cannot be economically performed under the widely varying temperatures that the vehicle will see. Maximum temperature conditions can be induced in the static test simply to verify load path predictions under combined temperature and mechanical loads, but failure loads are likely to be pulled without temperature. Therefore, structural analyses will be used to link the results of the full-scale static and fatigue tests to the subcomponent, element, and coupon test results to verify the integrity and margins for the structure. Coupon and element tests will be performed under the environments of concern both with and without damage. Subcomponents and components will be tested under limited environmental conditions, perhaps with the maximum uninspectable damage. Analysis is the only way to link these tests with the data from the full scale test to insure that the structure meets the design requirements. Airframe certification will drive the need for the computational structural analyses required for the HSCT vehicle.



N92-25928

Overview of Computational Structural Methods for Modern Military Aircraft

J. N. Kudva
Manager, Structural Methods and Applications Department
Northrop Corporation, Aircraft Division, Hawthorne, California

PRECEDING PAGE BLANK NOT FILMED

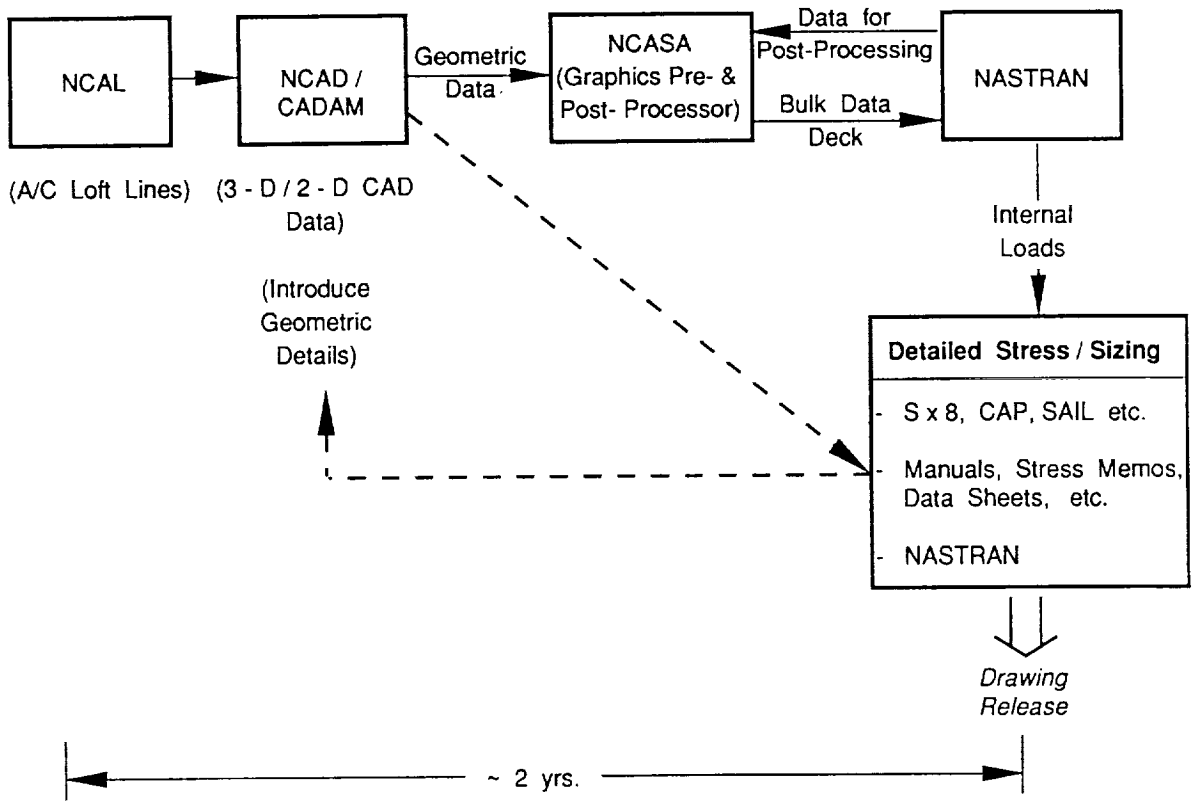
Introduction: Computational structural methods (taken here to broadly mean any method which requires a computer to obtain useful results) are essential for designing modern military aircraft. This briefing deals with computational structural methods (CSM) currently used. The methods' shortcomings, as well as planned improvements to support efficient structural design/analysis of modern military aircraft, are discussed. Most of the work presented here is based on studies and projects conducted by the Structures Research Department of the Aircraft Division of Northrop Corporation. While this is necessarily a partial and selective view of the process, based on exchanges with representatives from other aerospace companies, the results and conclusions presented here are typical of the aircraft industry.

The agenda for the briefing is given below. First a brief summary of modern day aircraft structural design procedure is presented. Following this, several ongoing CSM related projects at Northrop are discussed. Finally, shortcomings in this area, future requirements and summary remarks are given.

Agenda

- Introduction
- Overview of Aircraft Structural Design Cycle
- Current Research Activities
- Summary Remarks

Overview of Structural Analysis/Design Cycle

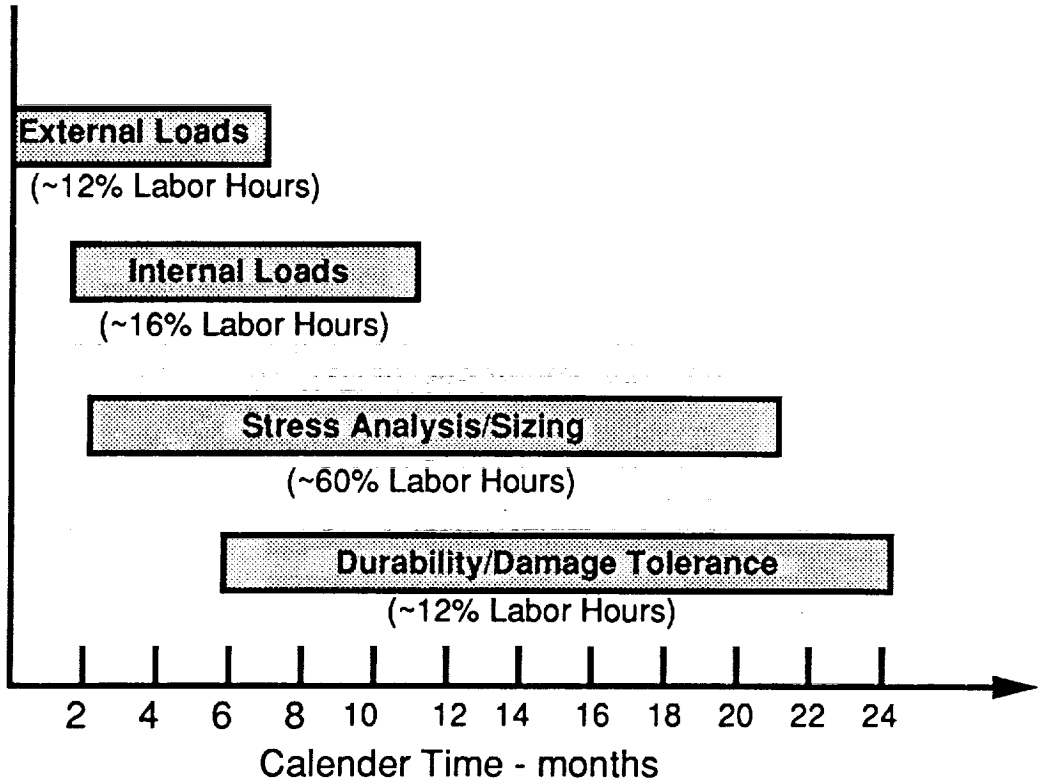


NOTE: NCASA (Northrop Computer Aided Structural Analysis), NCAD (Northrop Computer Aided Design), NCAL (Northrop Computer Aided Lofting), SAIL (Structural Analysis Integrated Library) etc., are Northrop proprietary codes developed and maintained at Northrop. Similar proprietary or commercially leased programs are used at other companies.

Structural design of a new airplane typically starts with 'loft' lines determined by aerodynamic performance considerations and mission requirements. This information is used in a 2-D and/or a 3-D CAD package to define outer and inner moldlines of the aircraft. The structural details at this stage are known only approximately and are based on simplified analyses, past experience and standard practices. From the CAD model a finite element model is generated using a finite element modeling package such as PATRAN or NCASA. The NASTRAN model is used to generate internal loads which are used for detailed design. Detailed design is performed using a combination of special purpose computer programs, manuals, stress memorandums and R&D reports. The detailed sizing information is sometimes (but not always) used to update and recheck the internal loads model.

The entire cycle takes approximately 2 years from project start to initial drawing release. However, during the production life of an aircraft, minor as well as significant reanalysis, design modifications, etc., are continuously performed.

Structural Analysis/Design Tasks



The above figure shows the approximate division of labor hours among the various tasks within the structural design cycle. Several points need to be noted here.

1. NASTRAN (or a similar finite element analysis program) is used for generating internal loads for various flight load conditions. 90 to 95% of this analysis consists of linear static or normal modes analysis. Also, the time taken to generate good finite element internal loads models (weeks or months) is significantly higher than the CPU time (hours).

2. The last two tasks should be considered together as the detailed design process; the relative division between them varies significantly from aircraft to aircraft.

3. By far the majority of the effort is taken up by the detailed design process (stress analysis, sizing and durability and damage tolerance analyses). This process is not fully automated or standardized; significant cost and time savings can be achieved by doing so.

Current Projects

- RAMCOMP (Rapid Multidisciplinary Computational Methods)
- FEVES (Finite Element Validation Expert System)
- Global (FE) / Local (BE) Analyses (COMET Related R&D)
- Manuals and Software Development

There are several CSM related activities currently underway at Northrop. Of these, three are of particular interest and they are discussed in greater detail. These are:

1. RAMCOMP: This is an umbrella project undertaken to speed up the entire aircraft design process by bringing together engineers from various disciplines to develop better tools within and interfaces among the disciplines.

2.FEVES: This is an expert system to help validate and classify NASTRAN models.

3. Global/Local Analyses : This task involves development of combined finite element (FE)/boundary element(BE) techniques for efficient analysis of large complex structures.

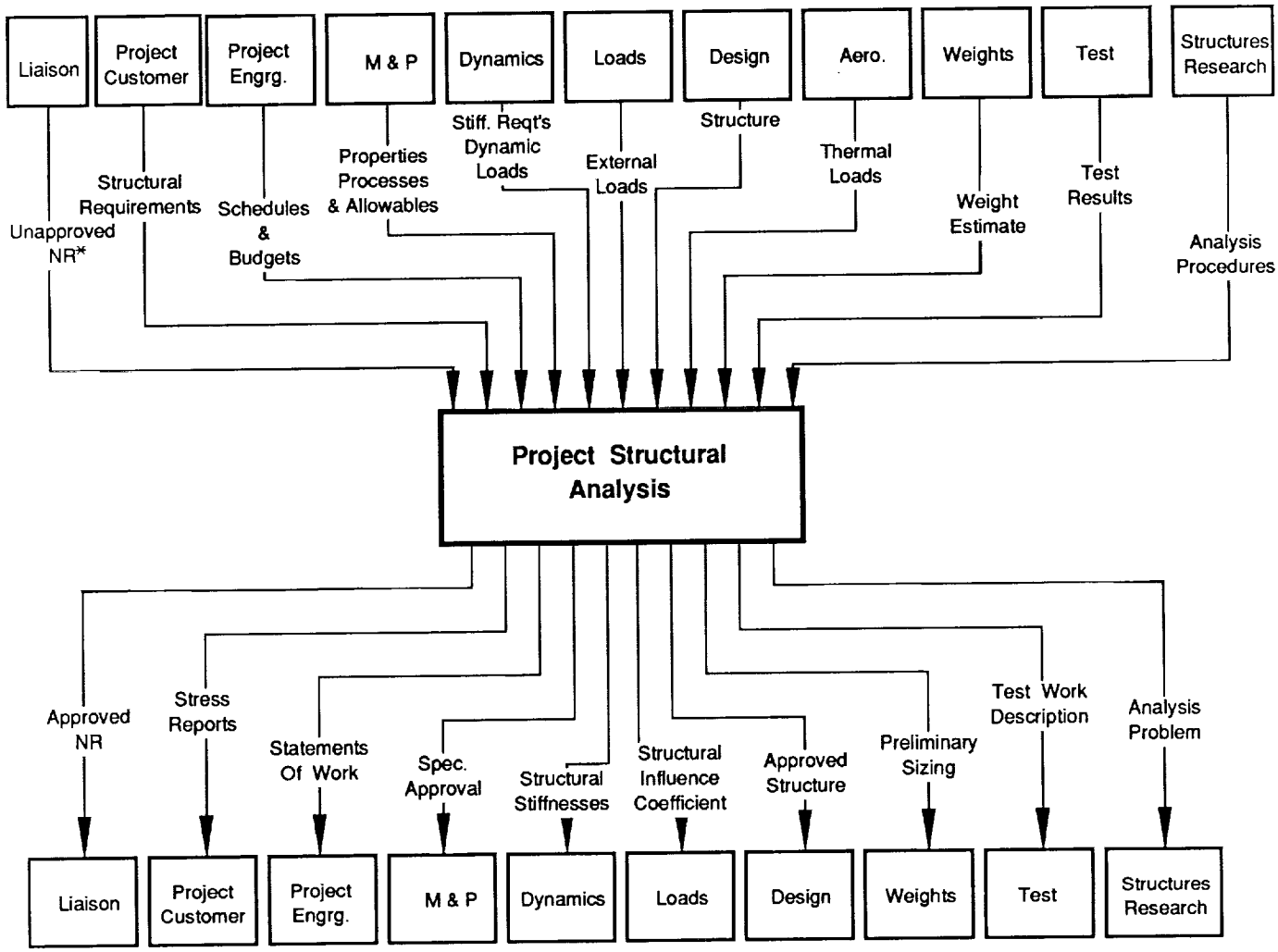
In addition to the above, structural analysis and design manuals and special purpose computer codes (for joints analysis, postbuckling design, etc.) are continuously being developed and updated.

RAMCOMP (Rapid Multidisciplinary Computational Methods)

- RAMCOMP Represents the Continual Evolution of a Standardized, Integrated, Computer Based Design Environment for Vehicle Synthesis Encompassing the Disciplines of Aerosciences, Dynamics, Loads and Criteria, Mass Properties, Structural Analysis and Vehicle Management System (VMS)

Under the RAMCOMP project, engineers from various disciplines, in concert, are trying to speed up the overall aircraft design process. This entails developing better tools (computer programs, manuals, etc.) and better interfaces among the tools, common formats and configuration control procedures and minimization of redundancies.

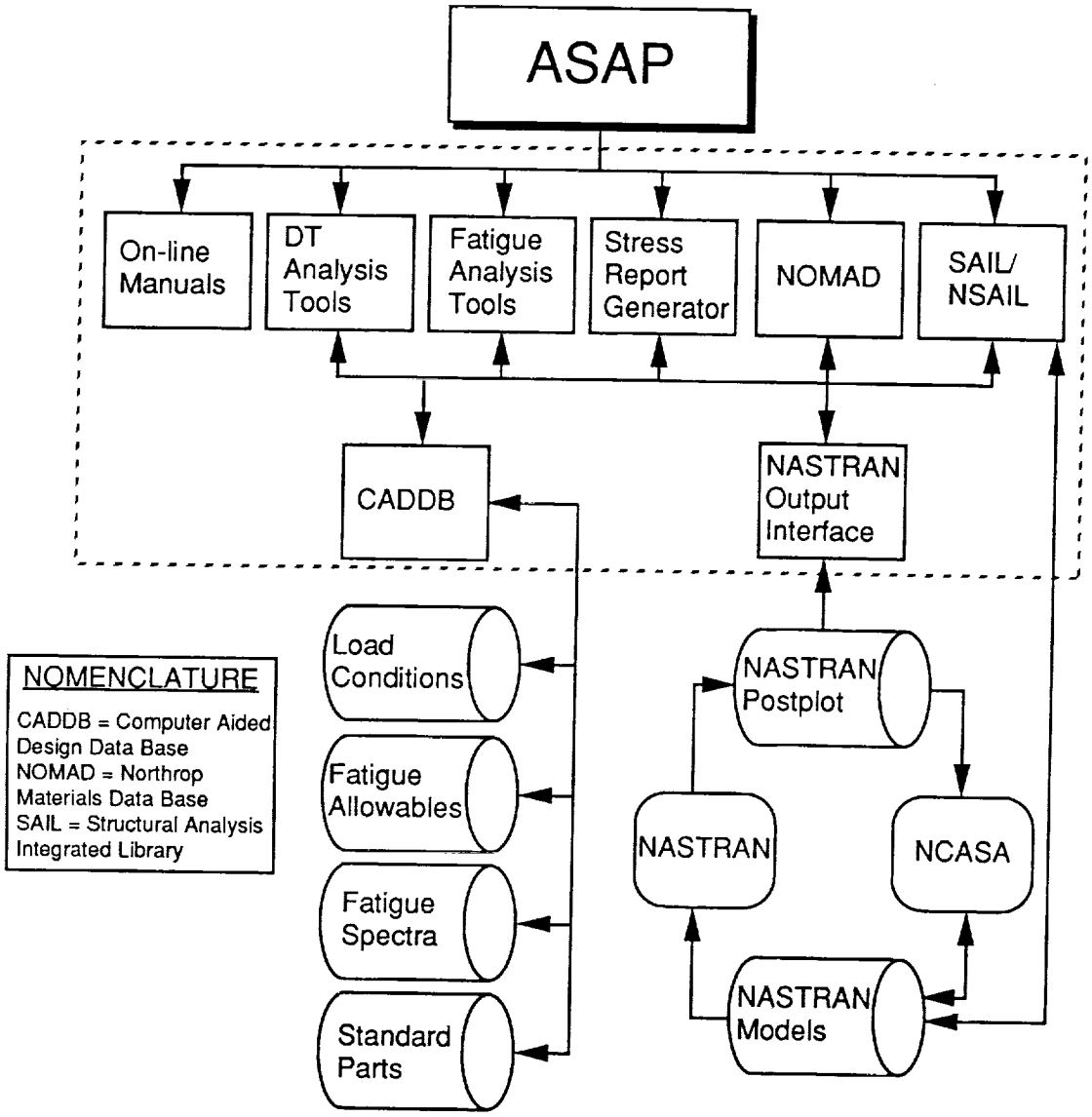
Project Structural Analysis Functional Diagram



(* NR = Nonconformance Reports)

The above figure schematically illustrates the various functional departments which exchange information with the structural analysis department in an aircraft design project. A significant portion of the time taken and labor hours expended in a project involve interaction and data transfer among various groups; RAMCOMP is an attempt to speed up the overall process by improving the format, method and content of the data transfers.

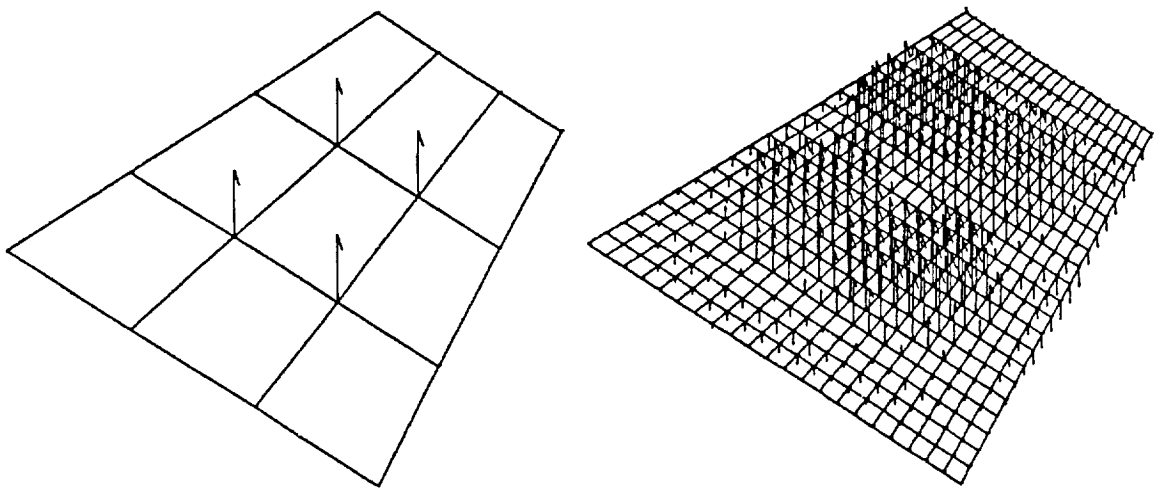
Automated Structural Analysis Program



ASAP, being developed under the RAMCOMP project, is an attempt to provide one convenient, integrated environment for stress analysts on a project to do their job. In addition to several tightly integrated tools for stress and damage tolerance analysis, it includes a materials database which is automatically accessed by the analysis programs. Links to automatically extract required local loads information from NASTRAN output are also included. After completing the analysis, the user can generate a stress report which can be directly incorporated in the final stress analysis documentation.

ASAP is expected to significantly decrease the time taken for performing and documenting stress analysis on future Northrop aircraft design projects.

Load Transformation Program-LOADTRAN



Coarse Mesh Fine Mesh

LOADTRAN is a good example of a task performed under the RAMCOMP program which has demonstrated considerable time savings. It transforms a set of applied point loads (at finite element grids) from a coarse mesh to a refined mesh.
External loads on an aircraft structure are provided to the design/analysis group by the loads group; these are generated using complex CFD or other aerodynamic codes. When the loads are initially generated, the structural design is usually at a preliminary stage and the finite element model on which they are used will be a coarse mesh model. As the design proceeds, a fine mesh model may be developed and loads at the new set of grid points would be needed. To obtain the external loads again from the loads department would require reruns of the aerodynamic codes; the calender time involved would typically be around three days. LOADTRAN side steps this process by transforming the loads using equilibrium and equivalent stiffness considerations; the new loads can thus be generated in a matter of minutes. LOADTRAN is also useful when a finite element mesh is modified for any reason.

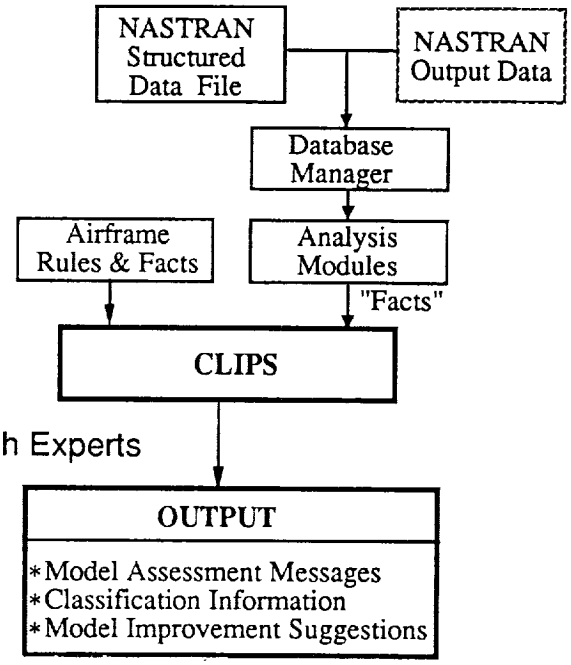
Finite Element Validation Expert System-FEVES

- Architecture

  - NASTRAN Bulk Data Input
  - 'CLIPS' Expert System Shell
  - C++ Graphical Interfaces

- Analysis

  - Rules Generated from Interviews with Experts
  - Geometry/Component Based Rules

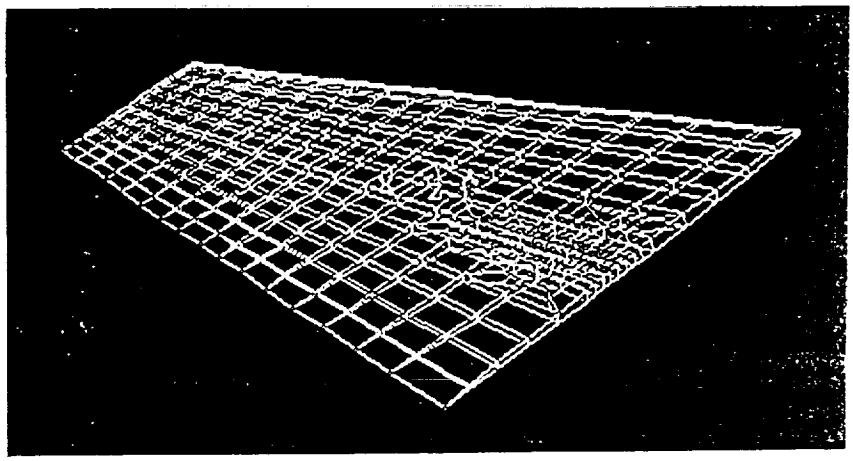


It takes years to develop expertise in using sophisticated finite element programs such as NASTRAN to analyze complex aircraft structures. FEVES is an expert system which helps evaluate a NASTRAN finite element model prior to performing a NASTRAN analysis run. Given a NASTRAN input deck, FEVES detects bad modelling practices and 'errors' not detected by NASTRAN. In other words, the particular model, if run, would in all probability yield results which look reasonable but would actually be inaccurate. Specifically, FEVES performs the following tasks : 1. determines if appropriate finite elements are used to model particular components; 2. evaluates compatibility of elements at nodes; 3. flags discontinuities of geometry and material properties; 4. checks for consistency between loads applied at a node and the types of elements connected to the node; and 5. divides the input model into structural subcomponents identifying the elements belonging to each subcomponent.

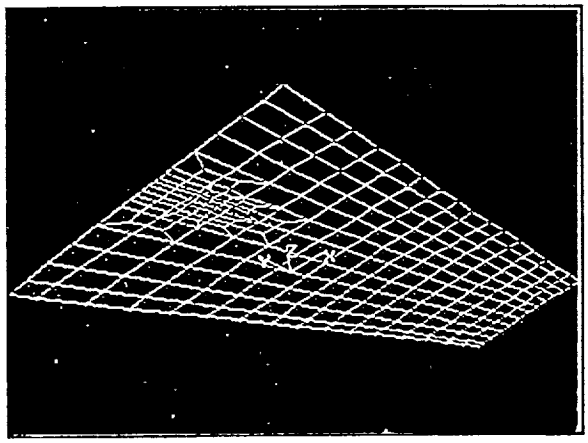
FEVES is useful in reducing the total modeling time, encouraging standard modeling practices and as a training aid for novice engineers. The classification capability can be of significant help to an analyst - on established production aircraft programs large internal loads models are constantly modified over the years. While consistent numbering schemes are often used to identify different subcomponents, these are not always followed. Often an analyst will have to laboriously try and figure out which elements belong to which subcomponent. FEVES automates this task, saving an analyst hours of tedious work.

The next page shows an example of the classification task in which an input aircraft wing model is decomposed into spars, ribs and top and bottom skins. (The figures shown are computer screen images).

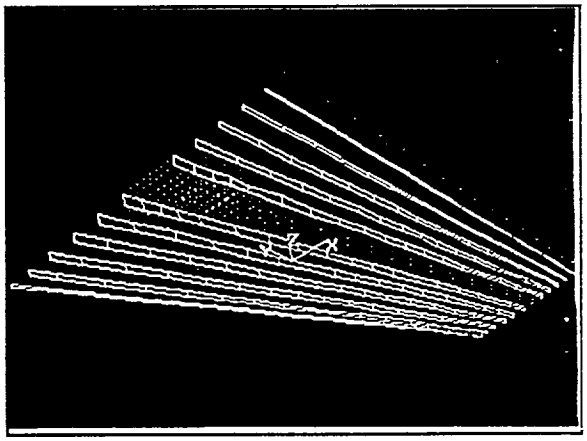
FEVES - Classification Example - Wing Model



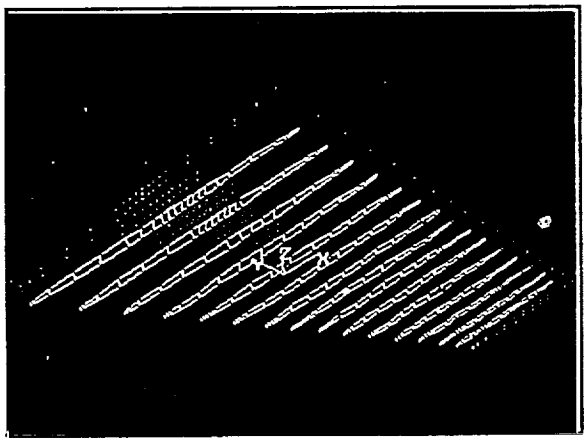
Complete Model



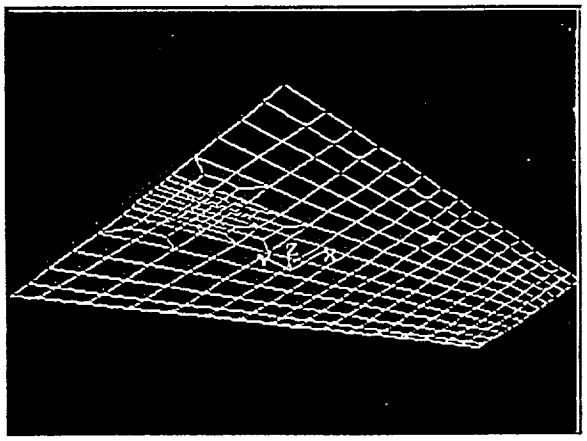
A. Top Skin



B. Spars

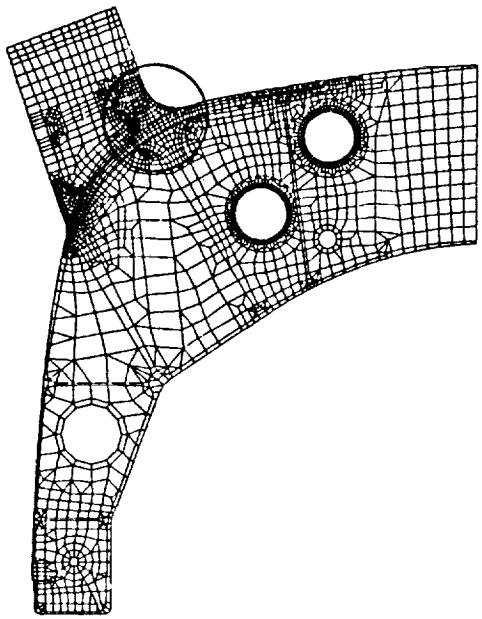


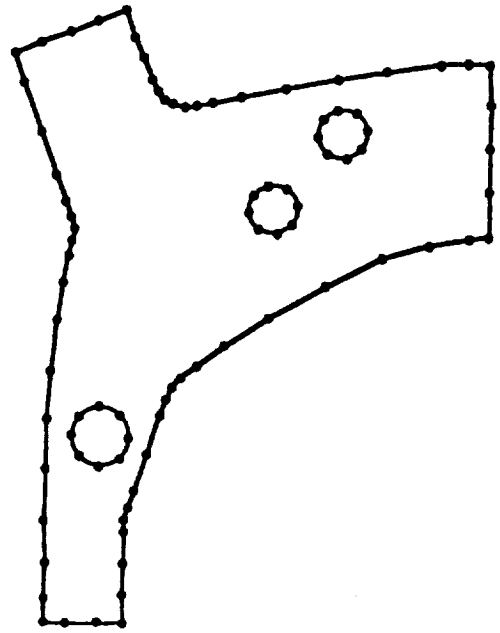
C. Ribs



D. Bottom Skin

Global(FE)/Local(BE) Analyses





- Versatile, Well Established
- Can be Used for Complex Built-up Structures
- Suitable for Global Analysis

- Reduced Dimensionality, Modeling Time, Cost
- Improved Accuracy
- Suitable for Local Analysis

The figures show FE and BE meshes for a section of an aircraft bulkhead. Cracks were detected in-service at the circled corner and accurate stresses were required to evaluate fatigue life of the structure. As can be seen from the figure, the FE model is quite complex; it took several weeks of an analysts' time to generate a good model to obtain accurate results. The equivalent BE model on the other hand is significantly simpler. However, to do the BE analysis, boundary stresses or displacements are needed; these are best obtained from the global FE model. Global/local analysis using combined FE/BE methods has significant payoffs in this and other similar situations.

The current research project is being conducted in cooperation with the Computational Structural Mechanics Branch at NASA Langley. The project involves developing generalized 2-D and 3-D boundary element codes and incorporating them in the NASA COMET code for global/local analyses.

Analysis Programs and Tools

- NASTRAN is the Mainstay of Structural Analysis at Northrop
- SX8, SS8, SO0 , CAP – Laminate Analysis
- BJSFM, SASCJ, SAMCJ, JOINT, etc. - Joints Analysis
- NORCRACK, FLAWGRO, etc. - Crack Analysis
- DoD and NASA Manuals, Reports etc.
- Northrop Manuals

Several special purpose computer programs are used for detailed stress analysis. These include programs developed at Northrop and other aerospace companies under Dod/NASA sponsorship, progams developed in-house and commercially available programs. Many of these programs resulted from R&D efforts and need to be validated and, if required, modified for real world problems. Also, there are no well established procedures or codes for design/analysis of several complex configurations. This is discussed next.

Needed Improvements

- Composite Design Procedures
  - Cut - Outs, Out-of-Plane Loading, Post - Buckling, Crippling, etc.
  - Bonded and Bolted Joints
- Finite Element Modeling Guidelines
- Buckling Of Complex Structures
  - Plates With Reinforced Holes
  - Non - Rectangular Webs
- Stress Concentration Factors For Complex Geometry & Loading Cases
- Fatigue Analysis Of Fastener Holes Under Complex Loading

Some of the areas in which improvements are needed are listed above. These include:

1. While numerous analytical and experimental results are available for joints, post-buckled panels and panels with cut-outs, they need to be condensed into standardized, easy to use codes and procedures for design.

2. Finite element analysis of large complex aircraft structures is more of an art than a science. It usually takes years for an engineer to be an expert and there are very few written guidelines available.

3. Buckling analysis of non-rectangular panels and panels with reinforced holes is often performed using finite element analysis. This takes considerable amount of time and number of iterations to obtain accurate results. Again, simple approximate methods will be helpful.

4. Fatigue life of an aircraft component is often dictated by the stress state around fastener holes. To predict fatigue life, validated and standardized methods to evaluate stress concentration and stress intensity factors of aircraft structures with holes under multiaxial loading conditions are needed.

Summary Remarks

- Finite Element Programs are Primarily Used for Determining Internal Loads
  - CPU Time << Time for Generating and Validating FE Models
  - Tools are Required to Automate Model Generation and Validation

- A Variety of Manuals, Programs, Reports etc., are Used for Detailed Stress Analysis
  - These need to be Integrated and Automated
  - Continuous Interaction with Users is Essential

If Tools are Not Easy to Learn/Use, they Will NOT be Used

Since the CPU times for large FE models are almost trivial compared to the time taken to generate good models, and since well established, validated and documented codes such as NASTRAN are readily available, the most significant contributions to reducing design cycle times will be tools which automate the generation and validation of FE models. Mesh generators and knowledge-based tools are steps in the right direction.

Detailed design takes up around 75% of the labor hours of an aircraft design effort. This is done using a variety of tools; these are usually not standardized or integrated. Again, doing so can significantly reduce design cycle time.

While there are still many shortcomings in currently available tools for structural analysis, there are very few, if any, unresolved theoretical issues. What is required are easier to use tools which include some assessment of the accuracy of the results derived.

Because of time and budget constraints, lack of theoretical knowledge and a variety of other factors, unless tools developed for stress analysts are validated on realisitic problems and are easy to learn and use, they will not be used.

Acknowledgements: The results and conclusions presented here are based on the works of several individuals within the structures departments at Northrop. While these are too numerous to mention here, the author would particularly like to acknowledge the contributions of Mr. Barry Megginson, Dr. Nasir Munir and Dr. Paul Tan.

N92-25929

A Perspective on Technical Needs in Computational Structures Technology

Frank F. Abdi, Gregory L. Savoni and Kenneth J. Newell
Rockwell International

PRECEDING PAGE BLANK NOT FILMED

A PERSPECTIVE ON TECHNICAL NEEDS IN COMPUTATIONAL STRUCTURES TECHNOLOGY

I) Micro-Macro Structural Integration

- Creep Analysis
- Consolidation Analysis Program for Parameter Selection

II) Correlation between Test & Mathematical Model

III) Transient Dynamic Analysis

- Impact Analysis
- Superplastic Forming

IV) Mathematical Optimization

- Multi-Disciplinary Optimization
- Transient Dynamic Optimization

V) Parallel Processing

VI) Summary and Conclusions

I-Micro-Macro Structural Integration

To accurately predict the behavior of the material, and hence that of the structure comprised of the material, the constitutive equations must take into account the processes occurring on the microstructural level.

Identify Microprocesses , Deformation Mechanisms

Develop Constitutive relationships and Equations

Interface Microanalysis with Macroanalysis

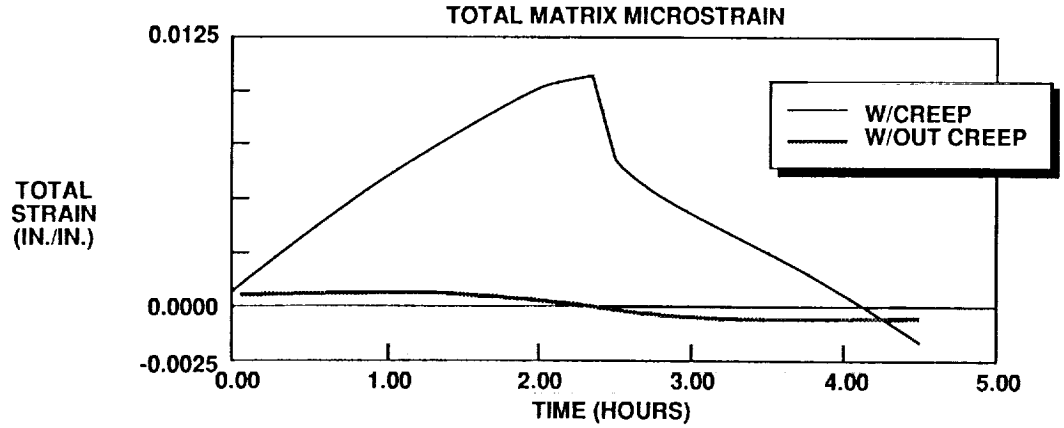
This approach has been used in integrating the Mechanical Equation of States (MEOS) with the METCAN program.

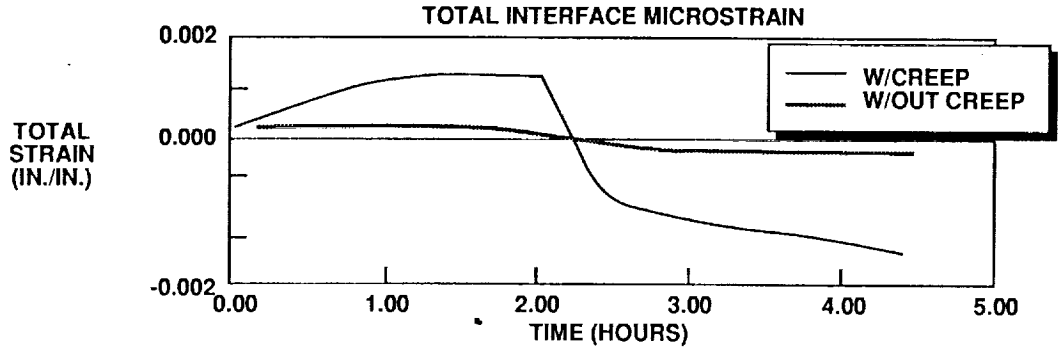
At Rockwell International , a prime contractor of the National AeroSpace Plane program, we have developed an analytical creep program that addresses specific High temperature cyclical Loading expected to occur in the Hypersonic flight regime. The creep methodology has been incorporated into the METCAN program of NASA/LERC. This methodology utilizes the Mechanical Equation of States as formulated by John Gittus which calculates the multiaxial inelastic(recoverable) plus plastic (nonrecoverable) strains. The emphasis in MEOS is on dislocation creep. The source of this dislocation creep is thermal climb and stress reversal will also remobilize dislocation links.

Another application of the MEOS algorithm was the analysis for the metal matrix filamentary composite fabrication process. This process constituted a Consolidation Analysis Program for Parameter Selection (CAPPS). The Consolidation parameters to be determined for a successful fabrication process are temperature, and pressure as functions of time. These parameters are applied to a composite lay-up of metal sheets, and filaments for the purposes of : 1)embedding the filaments in the metal sheet matrix by means of creep-forming the metal around the filaments, and (2) providing suitable conditions for diffusion bonding the metal-metal and metal -filament surfaces as they make contact.

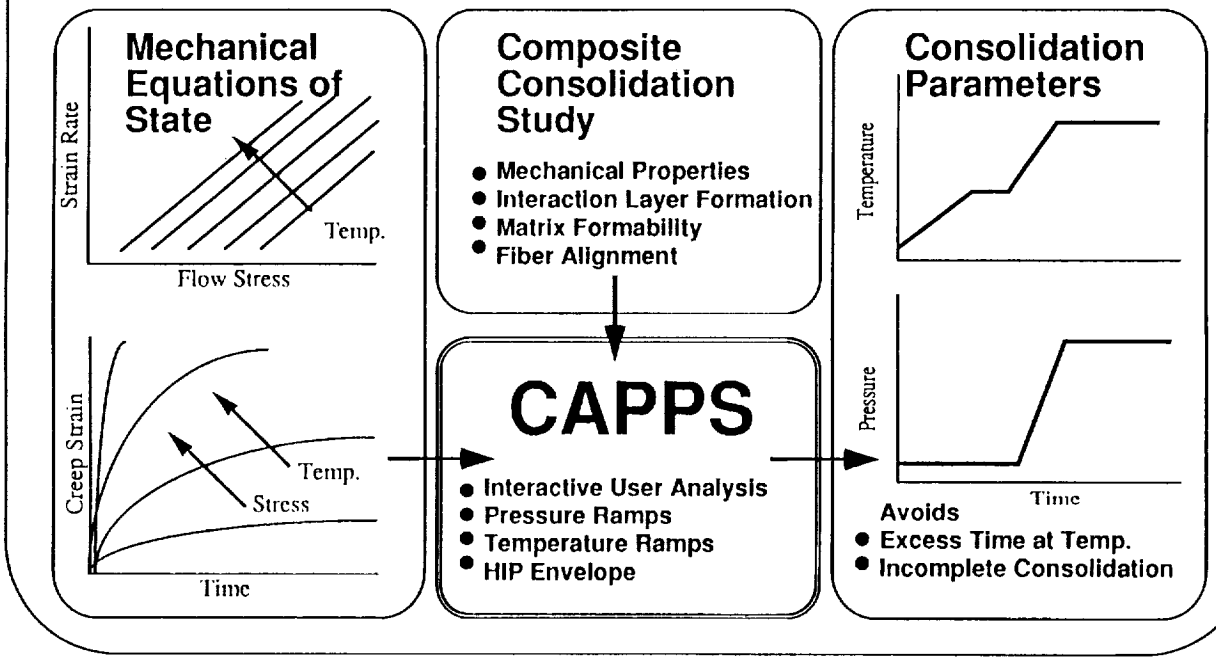
TOTAL MATRIX AND FIBER MICROSTRAINS

WBS: 2410.03 ANALYSIS METHOD DEVELOPMENT - CREEP





Consolidation Analysis Program for Parameter Selection (CAPPS)



Technical Needs in Micro Structural Deformation Mechanism Modeling

Other deformation mechanisms such as Nabarro-Herring, and Coble can be similarly formulated into a Micro-Macro Structural Analysis. This analysis will consider the dominant mechanism in the map of deformation variables.These additional formulations can be summarized as follows:

1- Analysis of creep behavior during tertiary stage

Monkman-Grant equation will be adopted to analyze tertiary creep,and time to rupture

2-Evaluation of creep behavior for the metal matrix interface material

3-Deformation Mechanism Map Formulation

Addition of Grain boundary sliding, Coble, Harper-Dorn and Nabarro-Herring to Mechanical Equation of State

4-Precipitate Hardening phenomenon for Metal Matrix Composites (beta 21s), and Titanium Aluminide

* TRANSMISSION OF ELECTRON MICROSCOPY
* BIMODAL DISTRIBUTION OF ALPHA PHASE PRECIPITATES IN BETA PHASE MATRIX
* INCIPIENT NUCLEATION OF ALPHA PHASE ALONG DISLOCATION LINES
* SIZE OF PARTICLES IS DIRECTLY MEASURED FROM THE TEM PHOTO -MICROGRAPH
* SEPARATION BETWEEN PARTICLES IS CALCULATED BY : $L = N/A$, $X = 1/L$

WHERE N IS THE AVERAGE NUMBER OF PARTICLES OF KNOWN SIZE, A IS THE AREA OF OBSERVATION,

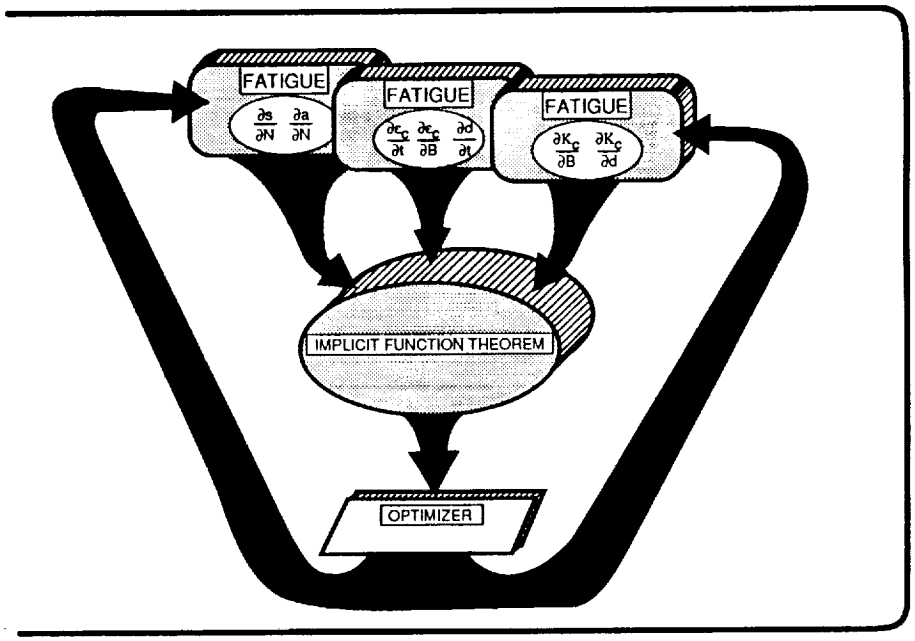
X IS THE DISTANCE BETWEEN PARTICLES, AND L IS THE DENSITY OF PARTICLES PRESENT IN THE OBSERVATION AREA

5-Crack growth void formation

6-Prediction of Composite material behavior for **Stitching, Stacking, Weaving**, and **Braiding**

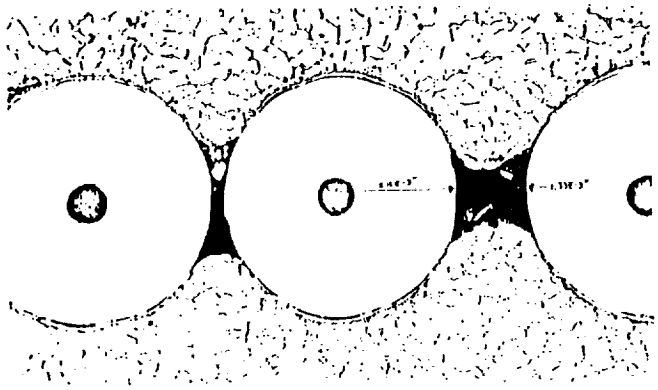
7-Interaction between Fatigue/creep/fracture

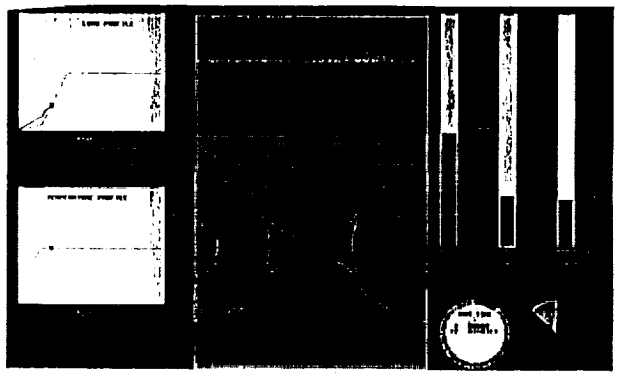
Combine microstructural failure mechanisms by use of Implicit Function theorem of Sobieski, and/or use of Multi Factor Equation of Chamis. These methods can also be applied to the prediction of material -structural life under anticipated service conditions.



II- Correlation between Test & Mathematical Model

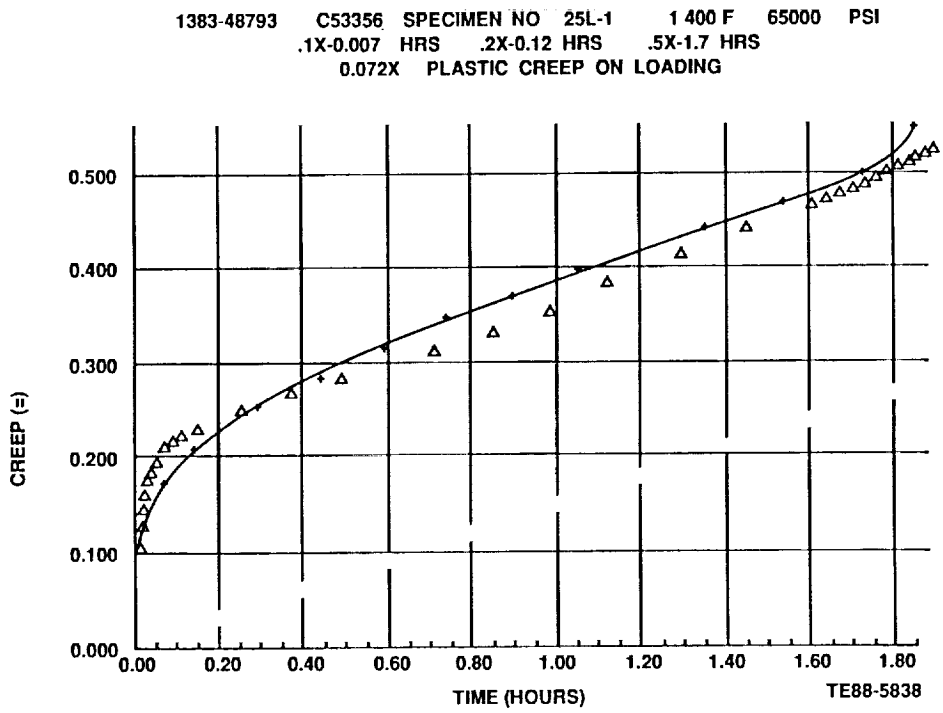
For advanced composite material characterization, correlation between test and theory is required. The modeling technique will require quantitative material parameters at the microstructural level, verification at the macrostructural level, and correlation at the component level.





Verification of MEOS Creep Analysis w/Test Data

- Nonlinear Creep analysis has been achieved and verified for lamina behavior



——— Allison Test Data

IR&D △ MEOS CREEP STRAIN PREDICTION

III) Transient Dynamic Analysis

The development of efficient tools in computational Science which address the **geometrically and materially** nonlinear transient thermodynamics solutions are necessary to assist in the design of a vehicle or component expected to successfully perform its mission.

The Finite Element/Difference Model must address the following physical/analytical characteristics:

1. Grid Distortion Effect(Lagrangian/Eulerian Coupling)

With Lagrangian meshing, the grid follows the deformation of the material. In hypersonic impact events, the distortion can become very severe, to the point of causing solution inaccuracies or inefficiencies.

In an Eulerian mesh, on the other hand, the material flows with respect to a fixed mesh, which is inherently more amenable to large deformations. The problem here is in tracking the materials to prevent false mixing of the materials or false tunneling of particles through barriers.

One way to solve this problem is to use a coupled Lagrangian/Eulerian solution method that uses the characteristics of the two approaches to best advantage.

2. Spall Effect(Spalling at Rear of Target)

Correct spall strength, adequate spatial resolution, and proper failure treatment are required in order to predict accurately the back surface spallation. This is especially important in impact events where spalled materials become projectiles.

Shock Wave Behavior and Spallation

In targets of intermediate thickness, during hypervelocity impact the primary shock wave detaches from the expanding crater surface, reaches the free surface at the rear of the target, and is reflected as a tensile wave back into the target, for projectiles having all dimensions of comparable length.

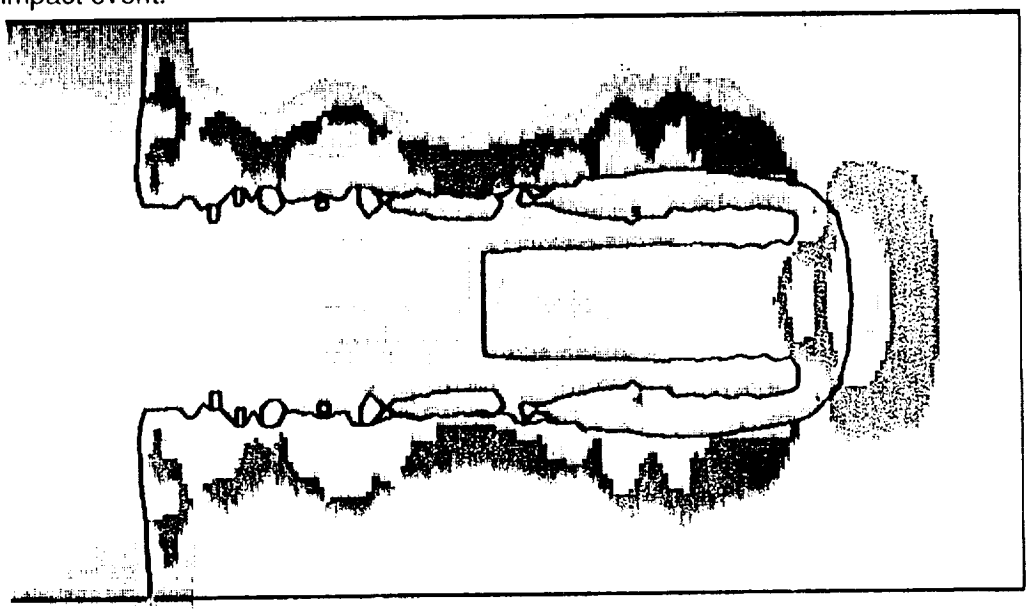
A spall plane fracture will occur where the ultimate dynamic tensile yield strength is exceeded. Part of the momentum of the impact event is trapped behind the spall plane, imparting a tendency for this material to break away from the main material body.

Once material has spalled away, successive spall planes and spallation fragments may result.

3. **Eroding Effect**(Erosion of Material during Penetration)

For thick targets impacted by malleable materials at velocities exceeding 1 km/s, there is a hydrodynamic erosion and inversion of the penetrator material against the receding face of the penetration channel in the target. The channel diameter becomes several times that of the penetrator.

Also, for contact algorithms, an "eroding" capability allows the automatic redefinition of the contact surface during the impact event.



4. **Master/Slave Contact Algorithm**(Interaction between Body Surfaces)

With a Lagrangian code, a general material interface treatment is necessary to allow different materials to slide relative to one another, separate from each other if the tensile strength of the interface is exceeded, or collide with each other if previously separated.

One surface may be defined as the master surface and the other surface may be defined as the slave surface.

5. **Phase Change**(Solid/Solid/Liquid/Gas/Mixed-Phase Phase Change governed by Thermodynamically Consistent Equation of State)

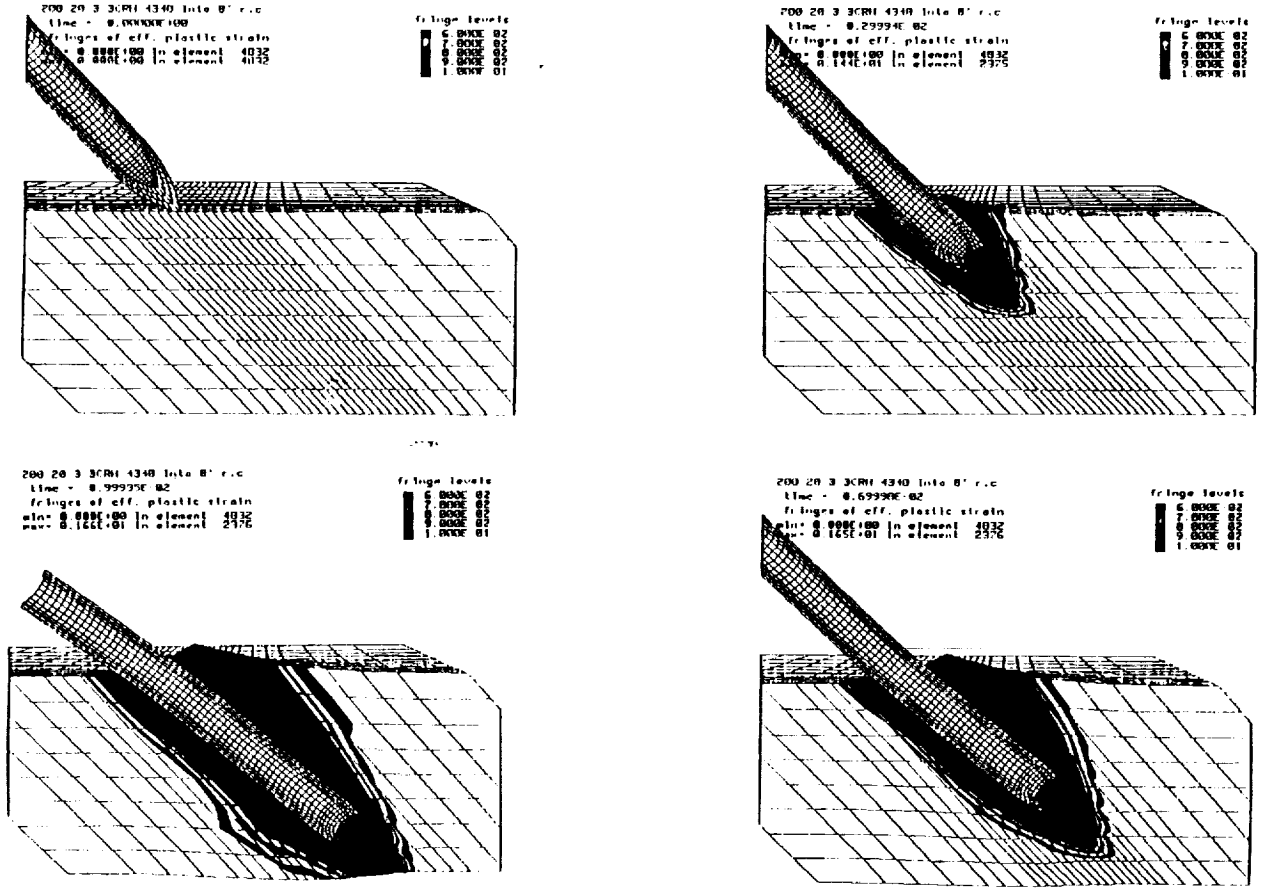
To completely model the behavior of material during impact, it is necessary to account for any phase change that may occur due to the energy-temperature-density equilibrium of the material.

The phase of the material will affect material behavior, the stress wave structure during impact, and the accuracy of the computational result.

6. **Wave Propagation Effect**(Controls Time Step and Determines Pressure)

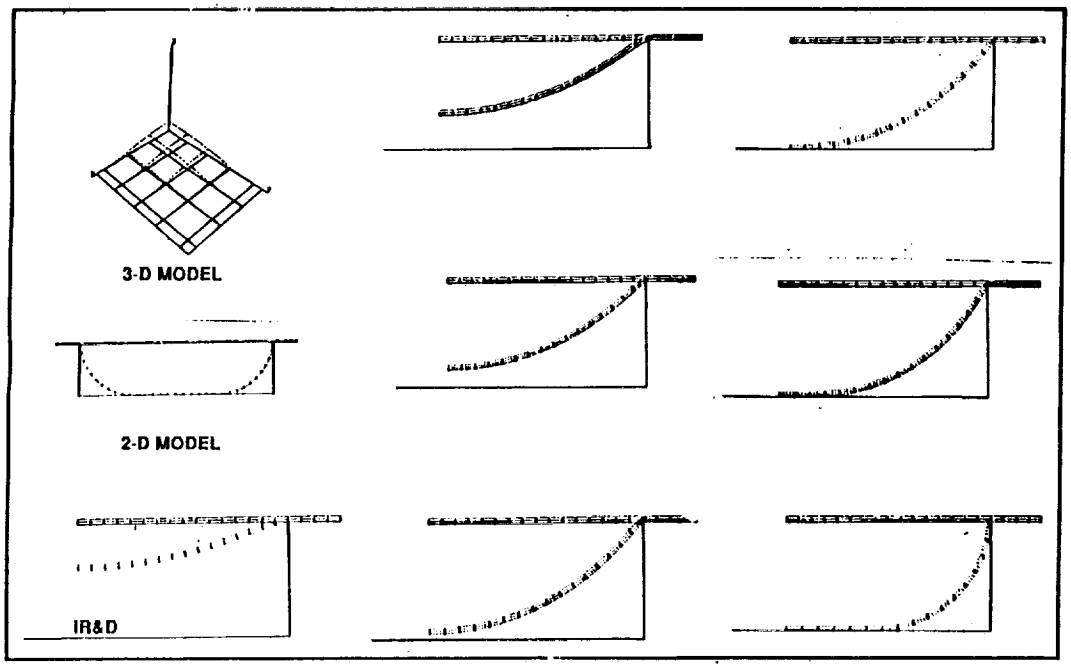
The time step in explicit hydrodynamic codes is controlled by the smallest dimension in the grid and the wave propagation speed.

If a shock wave exists, the energy equation relating the thermodynamic quantities of energy, pressure, and density must be satisfied. Energy conservation is used to obtain the internal energy, which in turn is used with the density to determine the pressure via the equation of state.


fringe levels
fringes of eff. plastic strain

Existing Computational methods for superplastic forming must be enhanced to include the **strain rate sensitivity , surface contact, friction, variable thermal/pressure profile and micro-Macro structural interaction.** The crucial relationship between flow stress, and strain rate as a function of temperature must act in conjunction with an instantaneous structural analysis in order to predict the correct pressurization-temperature schedule of the part.

PRELIMINARY DEMONSTRATION FOR FE ANALYSIS OF SUPERPLASTIC FORMING



IV- Multi-Disciplinary Optimization

Optimization methodology is most applicable in Design of aircraft, and Spacecraft. The nature of Aircraft design involves a wide spectrum of disciplines. The most efficient design will take into consideration all these various disciplines. The total fabrication and performance of aircraft can be broken into design process levels with local optimization taking place at each level. Primary emphasis is the structural behavior and material application, as well as including the Aerodynamic performance and mission capability as part of trade-off to optimize vehicle weight and overall performance.

OBJECTIVE

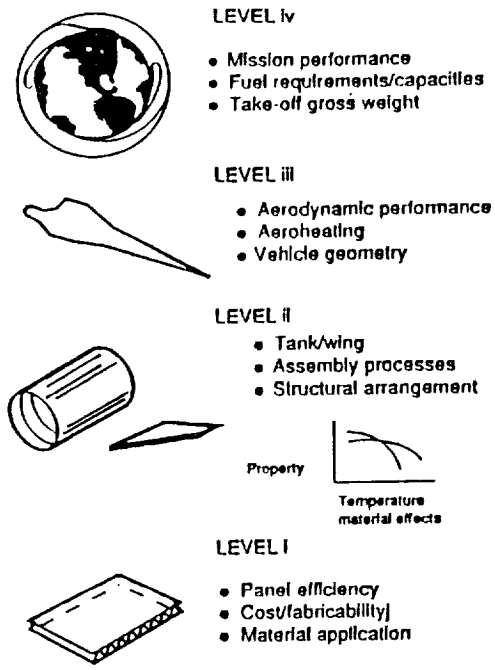
The objective is to maximize the **Effective Specific Impulse (ISP)**, with Minimum **Take-off Gross Weight (TOGW)** based on varying material application and flight conditions.

CONSTRAINT

The vehicle constraints were **volume, aerodynamic heating, and structural integrity**. The trajectory constraints included **maximum dynamic pressure, angle of attack, and maximum heating rate.**

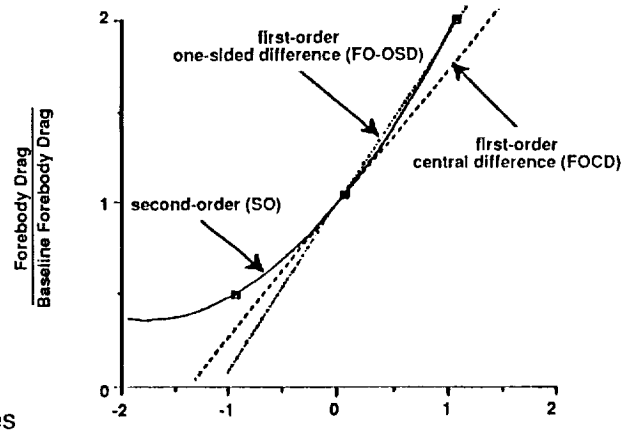
DESIGN VARIABLE IDENTIFICATION

Design parameters identified for the Generic Hypersonic Vehicle were determined by their importance in the optimization process . To reduce the complexity of the multi-disciplinary optimization, Design variable linking, reduction and grouping of constraints were necessary. Parameters under consideration were:**forebody length, forebody mean cone angle, fuselage upper surface height, and upper surface transition length.**



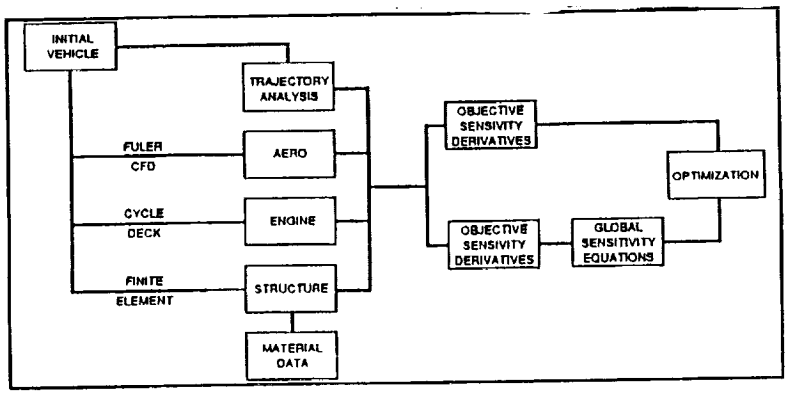
SENSITIVITY DERIVATIVES

Optimum Performance was achieved using sensitivity derivatives. We used **analytical derivatives** as was possible, and **finite differencing** was employed otherwise (**1st order:** one sided and central differencing; and **2nd order:** to capture nonlinear effects.) A **Global Sensitivity Matrix (GSM)** was constructed to consider cross coupling effects between the contributing analyses.



OPTIMIZATION STRATEGY

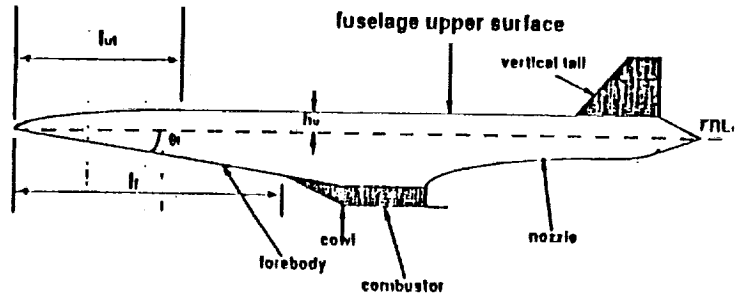
Multilevel optimization with linear decomposition technique resulted in the optimum shape. Trade studies for selected concepts(corrugated panel, honeycomb, SPF/DB beaded stiffened panel) were performed to achieve the minimum weight design using Multiconstraint optimization technique.

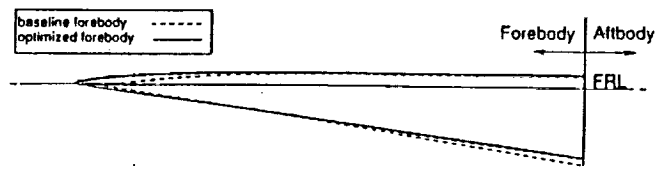


Design Study Strategy Was Enhanced Considering Structural Behavior and Material Responses

RESULTS OF SAMPLE CASE STUDY

Several cases of forebody shape optimization were performed. to demonstrate the validity of the methodology. Minimum takeoff gross weight and Maximum Effective Trimmed Isp were achieved varying the design variables.





Optimization Parameter	Baseline Value	Optimization Results
Design Variable		
1 - Forebody length	1.000	1.0209
2 - Cone angle	1.000	0.9693
3 - Upper surface height	1.000	1.0029
4 - Geometric transition length	1.000	1.0760
5 - Elevon deflection	1.000	0.8620
6 - Bodyflap deflection	1.000	1.0320
Objective		
Effective trimmed Isp	1.000	1.1259

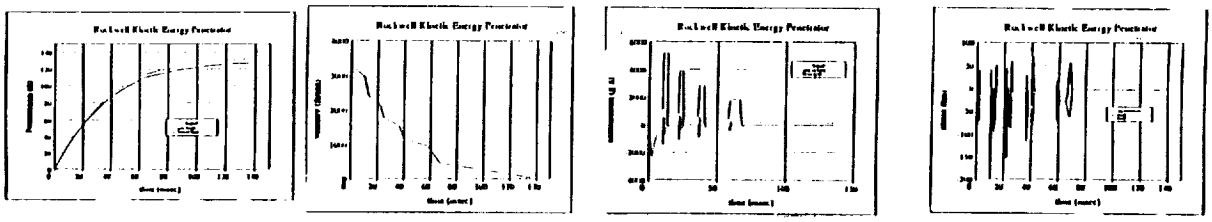
Transient Dynamic Optimization

We are developing aTransient Dynamic Optimization program as an integral part of our design system.

Our method uses the existing empirical , finite difference and finite element analysis programs as input to the Multiobjective, Multiconstraint Kreisselmeir-Stieinhauser formulation.

Conceptual	Preliminary	Detail	
SAMPLL (SNL)	CALE (LLNL)	CTH, CALE, and DYNA3D suite (SNL)	(LLNL)

take Time History baseline Results from Analysis tools as sensitivity input to:

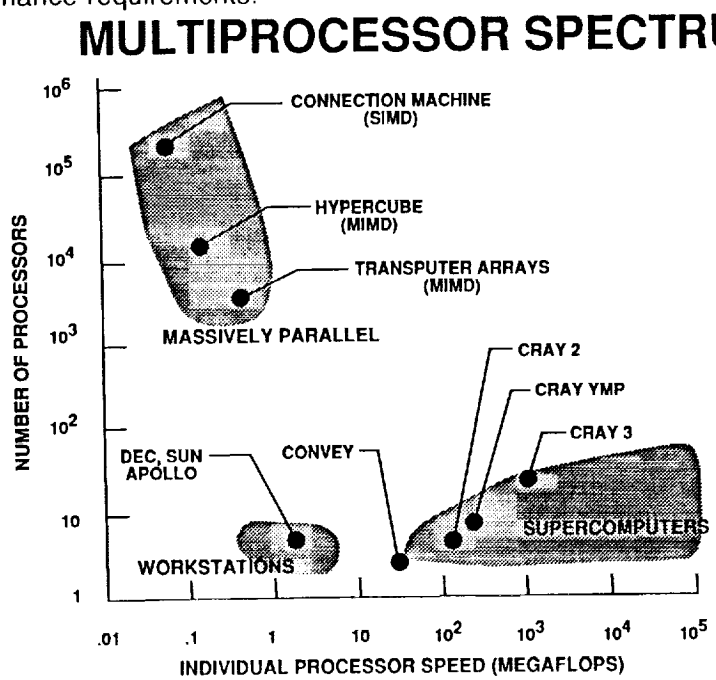


Rockwell Hypersonic Earth Penetrating Weapon Optimization Design System

(1) Minimum Weight Design (satisfying c.g. location, stability)
(2) Specific Penetration (target delivery scenario dependent: heaving, damage diameter)
(3) Design for Multiple projectiles
(4) Design for Multiple Components for each projectile
(5) Design for Survivability (structural integrity: fracture, melting, excessive plasticity)
(6) Design for the mission (available velocity, AOI/AOA)
(7) Design for Hypersonic Payload Constraints (type of H.E, weight, volume, glue/groove/baffle effect, sleeve)
(8) Achieve the Optimum Shape and Delivery Strategy for 1 thru 7

V- Parallel Processing

To date, one of the major factors preventing design integration of computational methods has been the lack of suitable computing hardware. The emergence of computer systems which utilize parallel architectures provides a possible solution to this current lack of powerful inexpensive computers. Parallel processing systems which incorporate multiple processors with relatively large distributed memories have recently become commercially available at prices significantly less than those of supercomputers. Preliminary analysis of commercially available designs such as the BBN Butterfly, The Floating Point Systems T series , The INTEL iPSC and the Thinking Machines CONNECTION MACHINE would suggest that with sufficient numbers of processors , such designs stand a chance of meeting computational performance requirements.



VI-Summary and Conclusions

WE have demonstrated the wide spectrum of needs and applications of Computational Structural Mechanics.The Spectrum ranges from Micro Structural Mechanics to The Optimization of the performance of aerovehicles.

Critical high leveraged technologies will require engineering process modernization and automation, affordable structures and materials, and advanced computational technology.

Recent advancements at Rockwell International in computational Structural Mechanics have addressed these issues and will further improve our ability to achieve Advanced Aircraft Objectives.

N92-25930

Computational Structures Technology at Grumman Current Practice/Future Needs

Allan B. Pifko
Applied Mechanics Laboratory
Corporate Research Center
Bethpage, New York

Harvey Eidinoff
Group Manager, Structural Mechanics
Aircraft Systems Division
Bethpage, New York

PRECEDING PAGE BLANK NOT FILMED

- Perspective and Activities - Structural Sciences
  Grumman Aircraft Systems Division
  - Harvey Eidinoff

  - Integrated Design/Analysis System
  - Structural Optimization
  - Structural Analysis -Case Study
  - Future Needs

- Perspective and Activities - Applied Mechanics Laboratory
  Grumman Corporate Research Center
  - Allan Pifko

  - Overview of Applied Mechanics Laboratory
  - Computational Modeling of Materials and Processes
  - Control of Intelligent Materials and Structures
  - Observations & Conclusions

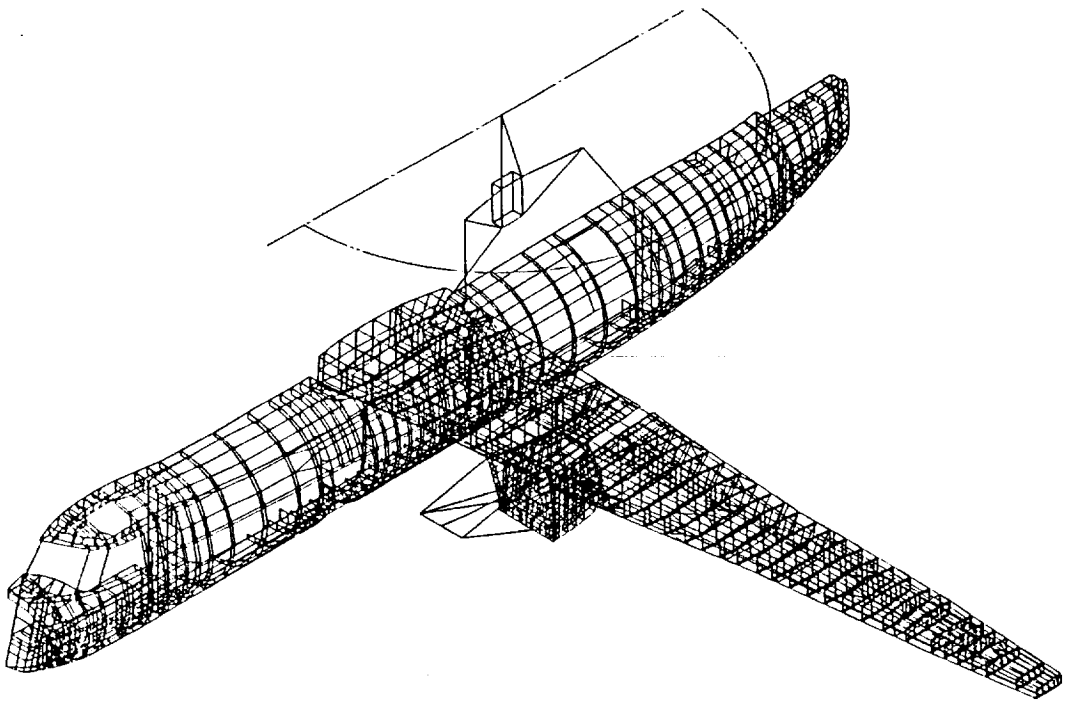
CURRENT PRACTICE

The current practice at Grumman for the design/analysis of new airframe structural systems is to construct a master finite element model of the vehicle in order to develop internal load distributions. The inputs to this model include the geometry which is taken directly from CADAM and CATIA structural layout and aerodynamic loads and mass distribution computer models. This master model is sufficiently detailed to define major load paths and for the computation of dynamic mode shapes and structural frequencies, but not detailed enough to define local stress gradients and notch stresses. This master model is then used to perform structural optimization studies that will provide minimum weights for major structural members. The post-processed output from the master model, load, stress, and strain analysis is then used by structural analysts to perform detailed stress analysis of local regions in order to design local structure with all its required details. This local analysis consists of hand stress analysis and life prediction analysis with the assistance of manuals, design charts, computer stress and structural life analysis and sometimes local finite element or boundary element analysis. The resulting final design is verified by full-scale fatigue (durability) test and full-scale static test to failure.

- Integrated Design/Analysis (COGS System)
  - Inputs from CADAM/CATIA
  - Inputs from Aerodynamics, Weights, Loads, Dynamics
  - Prepare Master Finite Element Model of Airframe
- Perform Structural Analysis (FEA)
  - Optimization
  - Internal load distributions
  - Mode shapes and frequencies
  - Preliminary fatigue screening
  - Buckling
- Global/Local Analysis
  - Regional Stress Analysis
  - Stress Concentrations
  - Elastic-Plastic Analysis
  - Fracture Mechanics
- Detail Structural Analysis & Verification by Test
  - Strength analysis
  - Life analysis

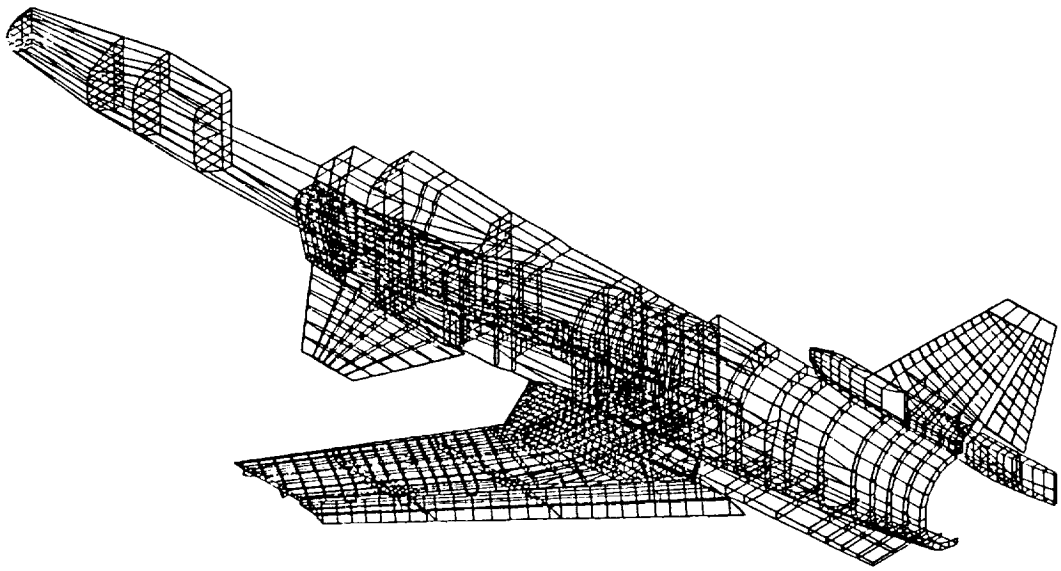
MASTER FINITE ELEMENT MODEL - E2C HAWKEYE

This is a drawing of a typical master (global) finite element model used to determine internal loads for the airframe. This particular example models only one-half of the airframe.



MASTER FINITE-ELEMENT MODEL -X-29

Another example of a master (global) finite element model shows a drawing of the X-29 forward swept wing demonstrator aircraft. This model has the entire airframe, including the wings, fuselage, and canards, but not the vertical fin.



HISTORICAL PERSPECTIVE AT GRUMMAN

This figure outlines the historical development of the finite element based design/analysis system called COGS which was developed at Grumman and is currently being used in-house for the design and analysis of airframe structural systems.

- 1963 - Development of Displacement Method - ASTRAL system
- 1964 - Development of Fully Stressed Design (FSD) capability - ASOP program
- 1967 - Development of Integrated Design & Analysis System (IDEAS)
  - Applied to Design of the F-14 (8 disciplines)
- 1972 - Development of RAVES Time Share System (15 disciplines)
- 1973 - 1981 - Development of FASTOP Optimization System
  - Flutter, Aeroelastic Effectiveness, Divergence speed constraints
- 1975 - Development of GEMS system - interactive graphics
  - CADAM, CATIA on IBM mainframe via IBM 5080
- 1978-1983 - Development of COGS system (subsystem of GEMS)
  - Applied to design of X-29, Lavi wing, et al.
- 1987 - Conversion of system to PHIGS standard - increase interactive graphics
- 1989 - Migration to high performance UNIX workstations

SCOPE OF ANALYSIS

The master finite element model of the entire vehicle is employed to perform the tasks of calculating internal loads for the major structural load paths, normal modes and frequencies, thermally induced loads, overall structural stability (buckling), flexibility coefficients for joining substructures, and in some cases, local detailed stress distributions. In addition, the master model is used to perform structural optimization analysis to determine the minimum weights for the major structural members. The master finite element model also provides boundary conditions for more detailed finite element analysis of local regions, i.e., for global/local analysis.

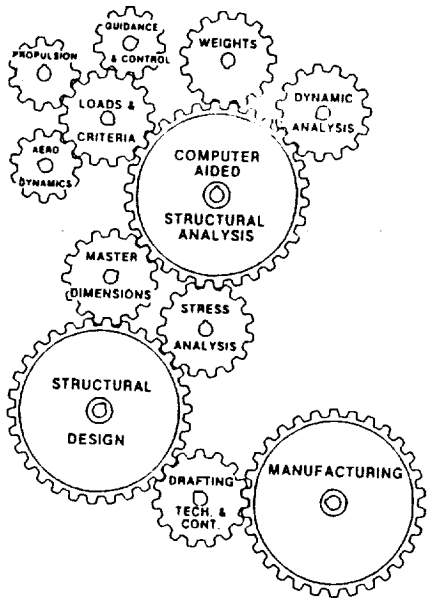
- Overall Vehicle for Internal Loads
- Local Detailed Stress Calculations
- Buckling Studies
- Normal Modes and Frequencies
- Thermal Stress Analysis
- Influence Coefficients
- Computation of Flexibility Coefficients
- Structural Optimization

COGS INTEGRATED DESIGN/ANALYSIS SCHEME

This figure illustrates the many disciplines with their analyses and databases that are linked to the COGS system. This enables rapid data transfer between all of the disciplines that must work together in order to rapidly design and produce an effective aerospace vehicle.

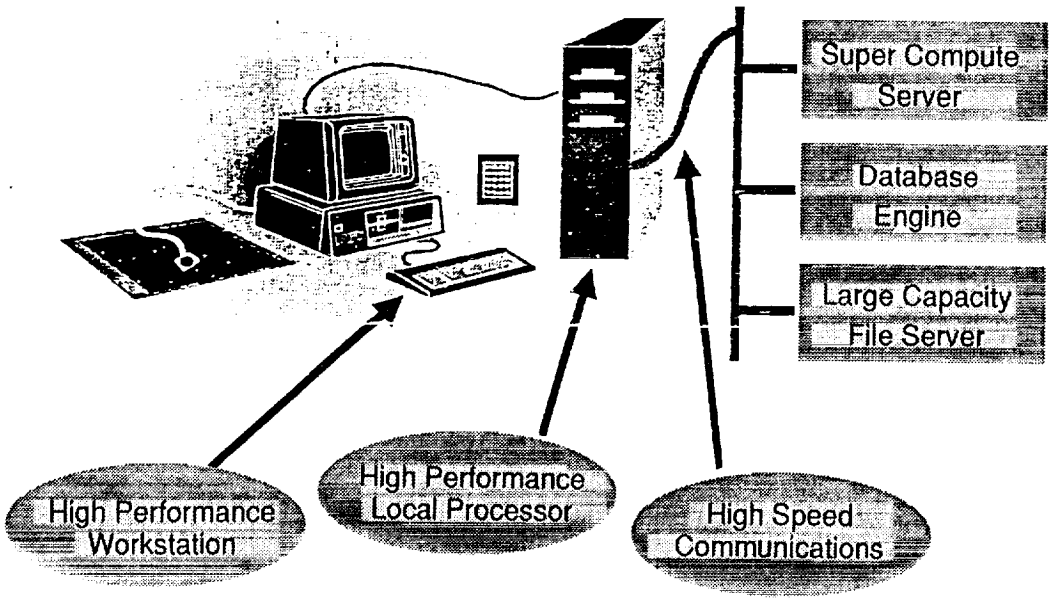
COGS

COMAP-ASTRAL INTERACTIVE GRAPHICS SYSTEM



COGS ENVIRONMENT

The COGS system is configured to operate on a mainframe computer or locally on a high performance work station. The current practice is to perform modeling, and pre-processing on a work station, major number crunching on a supercomputer, and post-processing of the results on the work station. The model and analysis results are stored in a database that resides in a mainframe computer and is accessible to all structural analysts working on the project.



LAVI COMPOSITE WING

The Lavi composite wing structure was designed and produced by Grumman in the late 1980's. The COGS design/analysis system was employed to design and analyze the wing structure. The structure consisted of graphite/epoxy wing covers with metallic ribs and spars, and an optimum (minimum weight) design to meet strength and control-surface effectiveness requirements was achieved.

WING STRUCTURE

- Multi-Spar Configuration
- Graphite - Epoxy Cover, Metallic Substructure
- Attached to Fuselage at 8 Points
- Movable Surfaces - Leading Edge Flap, Inboard and Outboard Elevons

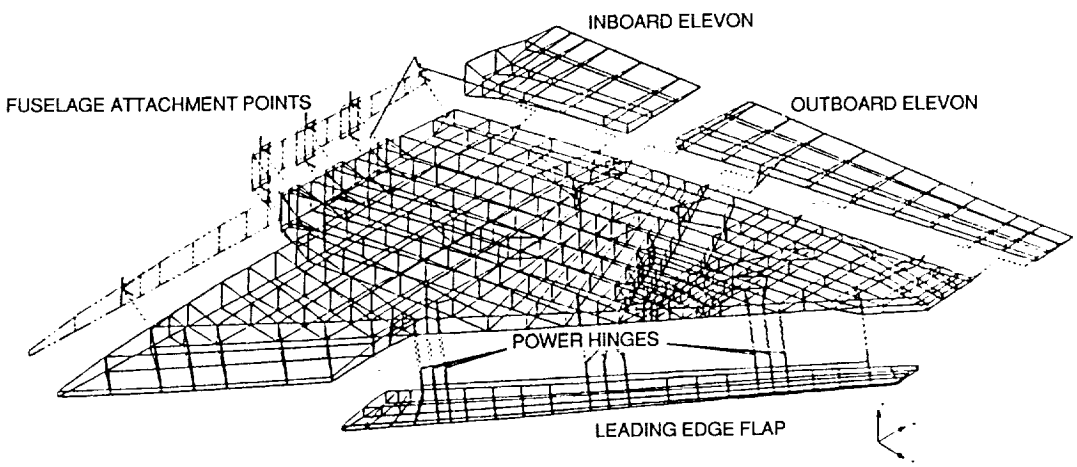
MODELING

- Covers - Modeled Using Anisotropic Membranes
- Ribs & Spars - Modeled Using Bars & Shear Panels
- Model Contained Approximately
  - 3100 Members
  - 3400 Degrees of Freedom
  - 6000 Design Variables (accounts for ply directions)

FINITE ELEMENT MODEL OF LAVI WING

The LAVI wing is a multispar configuration having graphite-epoxy covers and a metallic substructure. It is attached to the fuselage at 8 points. Movable surfaces consist of a leading edge flap and inboard and outboard elevons. The covers are modeled as anisotropic membrane panels; ribs and spars are represented by bars and shear panels. The total model contains about 3100 members, 3400 degrees of freedom and approximately 6000 design variables (which account for the individual ply directions in the covers).

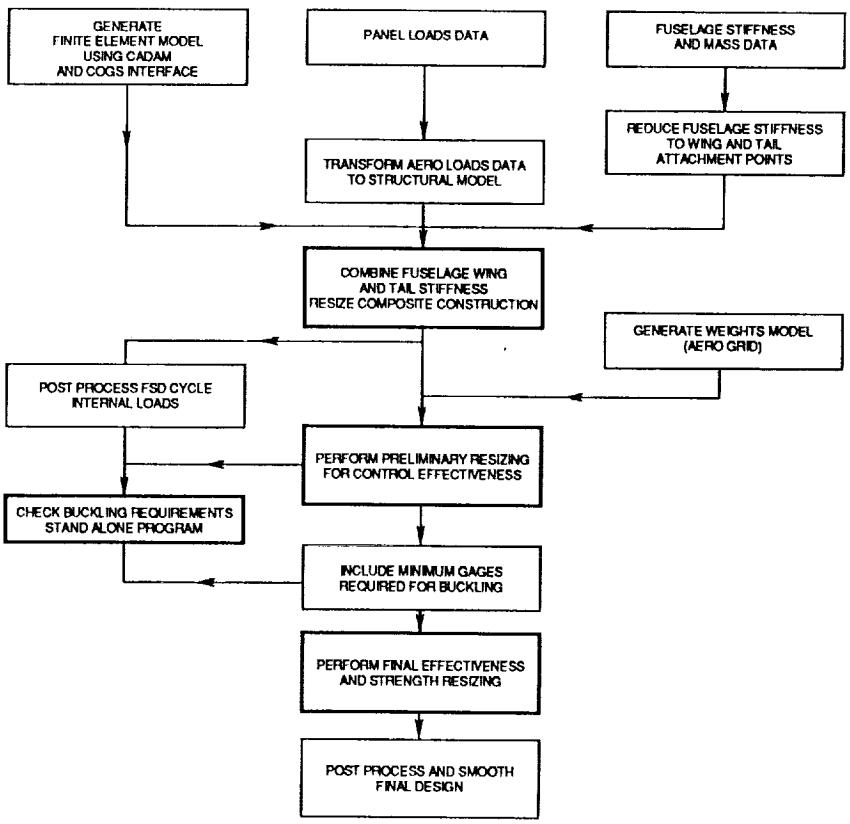
The structure was analyzed and sized to meet strength requirements for 102 flight design conditions. For the cover, strength requirements were based on maximum allowed fiber strains and panel buckling avoidance. Control-surface effectiveness requirements also played a major role in the design of this relatively thin wing. These requirements involved both pitch and roll, as well as ratios of pitch moment to hinge moment and roll moment to hinge moment, at Mach 0.9 and 1.2. The design was checked for flutter and leading-edge flap divergence, neither of which had any significant impact on the final design.



LAVI WING DESIGN/ANALYSIS CYCLE

The design/analysis cycle is shown below. Initial tasks consisted of generating the finite element model using CADAM and our COGS interface. The prime contractor supplied panel-point loads that were transformed to the structural model. They also provided stiffness and mass data for the fuselage. The fuselage stiffness matrix was reduced to the wing and tail attachment points and coupled with the wing and vertical tail stiffness matrices.

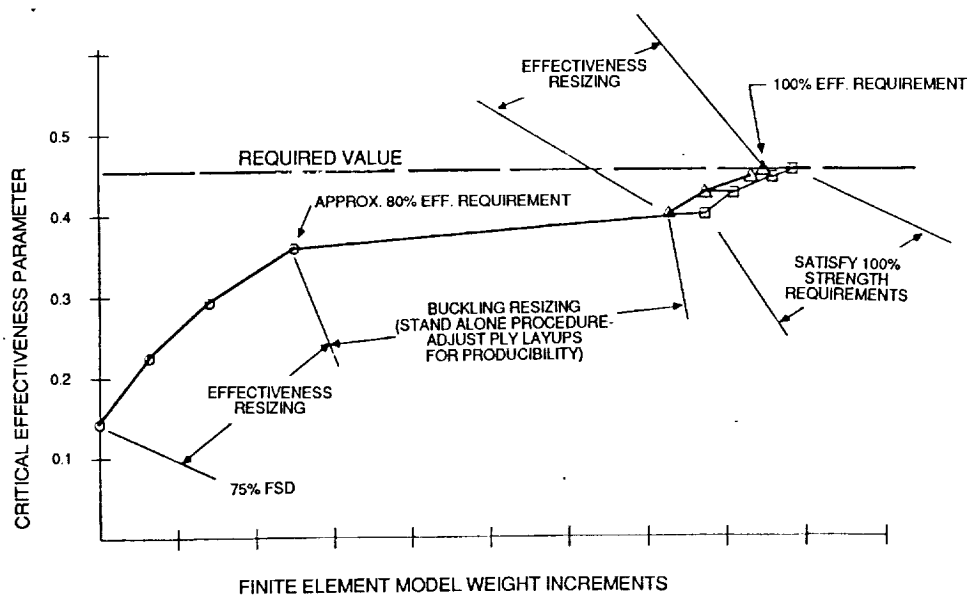
Several design/analysis cycles were performed by Grumman for the wing and vertical tail. Based upon experience gained in the early cycles, we established a rather pragmatic approach to obtain a near-minimum-weight design in the final design cycle, in which requirements for strength, panel buckling avoidance and control-surface effectiveness were treated in a somewhat interactive way.



LAVI WING SIZING PROCEDURE

The final sizing procedure and results are summarized in the figure below. We have plotted wing finite element model weight increments along the horizontal axis and the governing control-surface effectiveness parameter along the vertical axis. The required value of the parameter is shown as the horizontal line. Initially, we generated an FSD design for 75% of applied ultimate load. We then performed effectiveness resizing and brought the design to a point where the effectiveness parameter was approximately 80% of its required value. The buckling resizing and adjustments of the ply layups for producibility added additional weight increments and brought the effectiveness parameter to about 85% of the required value. Additional resizing to increase control effectiveness proceeded along the points marked by triangles in the upper portion of the curve. Along with each of these points are side-step increments required to satisfy 100% of ultimate load. All but the last of these latter points (marked by squares) represent designs which satisfy full strength and buckling avoidance requirements but which compromise the full effectiveness requirement, should such a compromise be desired in the face of the identified weight increments.

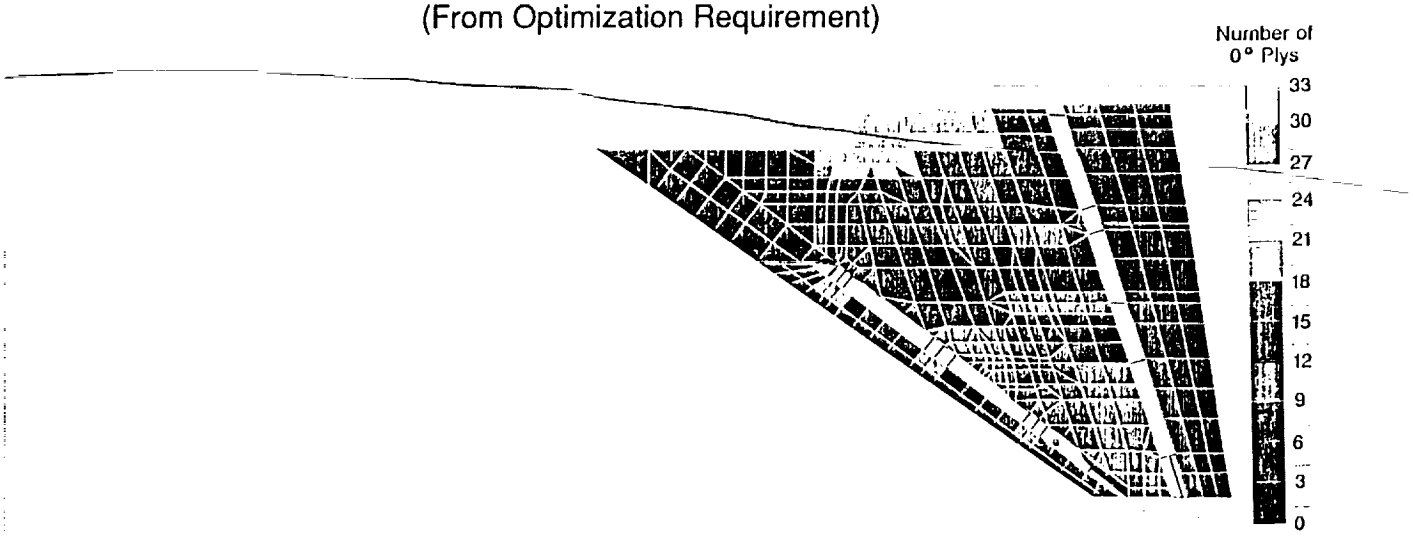
EFFECT OF RESIZING



LAVI WING 0° PLY DISTRIBUTION

Here we see the 0°-ply distribution for the lower cover of the LAVI wing. The number of plys are color coded. The COGS system allows us to display a wide variety of information in an interactive graphics environment. For example, since we store various derivatives within regions of the member data, we can display them as well. We have found displays of this type information to be particularly useful, not only in giving us important information about the design, but also as an aid in checking the realism of the model.

0° PLY DISTRIBUTIONS
(From Optimization Requirement)



LAVI WING CONCLUSIONS

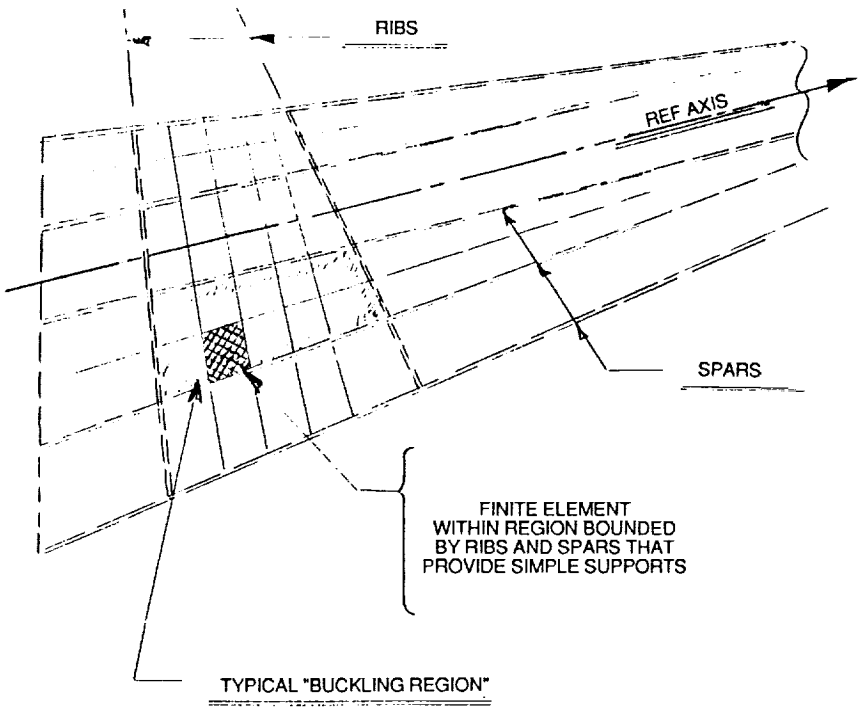
Integrated structural analysis and design systems and structural optimization procedures are being used in a production environment. Interactive computer graphics can and will play a significant role in the analysis/optimization/design/manufacturing area. Today, we talk about co-locating a team of people that include analysts, designers and manufacturing engineers on a given project so that they can interact via a common system. Practical structural optimization procedures are tools that must be made available to the team.

More work needs to be done to automate the detailed design and analyses process--more emphasis should be placed on the real design problems.

- Integrated design/analysis system produced successful production wing
- Optimized design achieved
  - control surface effectiveness
  - strength
- Needs more automation
  - Incorporate panel buckling and post-buckling analysis

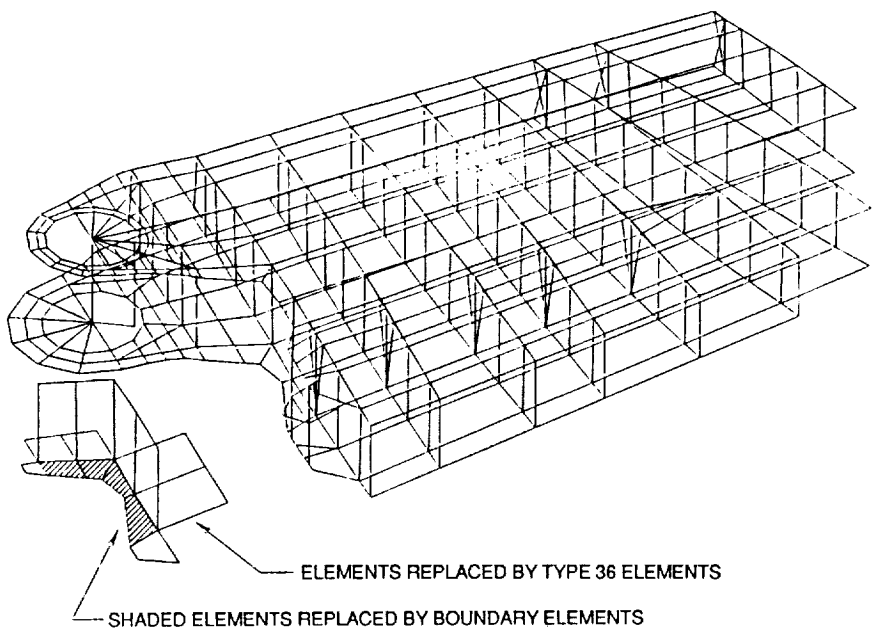
GLOBAL/LOCAL ANALYSIS-PANEL BUCKLING

A typical type of global/local analysis that needs to be performed is one that predicts local panel buckling. Here, one problem is to identify a group of finite elements in a wing or fuselage cover from the global model such that there are hard supports on all edges, and then to calculate the buckling modes of this local model. This could be done automatically or by a manual procedure that "lifts" the local model out of the global model and then performs the panel buckling analysis separately on a work station.



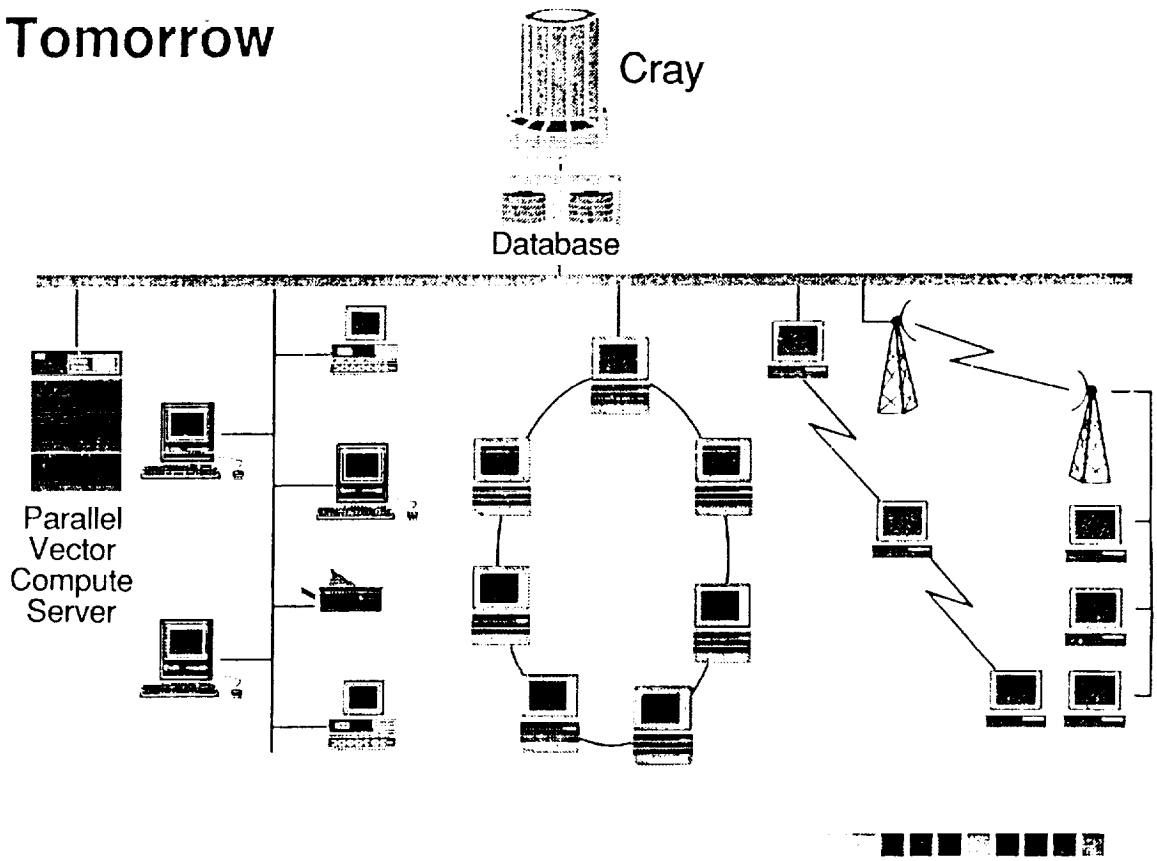
GLOBAL//LOCAL ANALYSIS-LOCAL STRESS CONCENTRATION

The determination of the stress concentration for a small, complex detail such as notch or fillet radius can be accomplished by embedding a local 2-dimensional boundary element model into the global 3-dimensional finite element model. This type of analysis was performed to investigate a fatigue critical location on the F-14 outer wing panel, where a complex fillet radius is located on the inboard end of the lower cover near the pivot lug. This analysis calculated the stress concentration factor for the initial design and was used to modify the design to achieve a lower, less critical value.



TOMORROW-COMPUTING NETWORKS

A COGS network linking local work stations where CATIA drawings are developed and structural analysis is performed with mainframe computers that serve as database storage and supercomputers was previously shown. Here, local networks at contractors' sites that are teamed together to develop new aerospace vehicles, and their subcontractors, are all linked together so that data can be exchanged among all the team members. This linkage will be important on future programs, as the trend on new programs is to have multiple contractor teams.



COMPUTATIONAL STRUCTURES TECHNOLOGY - FUTURE NEEDS

The topics that are listed on these three figures have been identified as areas where further research and development is needed in order to solve problems that are not currently being addressed or areas where more efficiency in terms of improved analytical results and/or more rapid solution time is required.

- Structural Optimization
  - Thermal Effects - Thermal Stress and Creep
  - Buckling and Post-Buckling
  - Survivability/Vulnerability - Optimize Damaged Structure
  - Improved "Model Tuning" Capability
  - Improved Smoothing & Slaving Procedures
  - Incorporate Manufacturing Constraints
  - Shape Optimization
  - Incorporate Passive Damping Elements
- Survivability/Vulnerability
  - Predict Damage
- Fatigue/Damage Tolerance
  - Improved Methods of Identifying Critical Regions
  - Global/Local Analysis for critical details
    - Local flexibility effects (fastener loads)
    - Stress concentrations
    - Elastic-plastic analysis for notches
    - Fracture mechanics
  - Improved fatigue analysis
  - Improved crack growth analysis

COMPUTATIONAL STRUCTURES TECHNOLOGY - FUTURE NEEDS

- Computing Systems
  - Supercomputer
  - Interactive Analysis - Workstation
  - Engineering Data base issues
- Explore the Use of Probabilistic Methods for Design and Analysis of Structures
  - NESSUS developed by Southwest Research Institute (NASA LeRC)
  - Composites version of NESSUS under development
  - Grumman application of composite version of NESSUS under NASA LRC, ACT program
- Multi-Disciplinary integration/interface
  - Geometry definition (PDES, CALS)
  - Aerodynamic model interface - feedback
  - Other emerging technology interfaces (Ex., NDI)

GRUMMAN CORPORATE RESEARCH CENTER

Six major disciplines are represented in the Grumman Corporate Research Center with computational structural mechanics (CSM) represented in Material and Structures Directorate, computational fluid mechanics (CFD), in the Aerosciences Directorate, and computational control technology in the Computing and Control Sciences Directorate.

- Approximately 100 Scientist, 60 Support
- Six Major Disciplines
  - Materials and Structures
  - Solid State Physics
  - Sensor Sciences
  - Aerosciences
  - Nuclear and Plasma Sciences
  - Computing and Control Sciences

OVERVIEW OF APPLIED MECHANICS LABORATORY

The activities of the Applied Mechanics Laboratory, part of the Materials and Structures Directorate, are outlined below. The major areas are computational structural mechanics which currently involves nonlinear static and dynamic analysis using our DYCAST code to predict impact behavior of composite airframe components. Crash safety, although not an "up front" enabling technology, is nevertheless an issue in occupant carrying vehicles including the HSCT. Our current work (with the NASA LaRC Impact and Dynamics Branch) involves a nonlinear composite beam element for A/C frames and progressive failure criteria for composite A/C structures. We are involved with the NASA/Industry Partners Development Program whose focal point is the CSM testbed code COMET. The COMET code serves as a computational workbench to assess, develop and transfer technology between NASA and the U.S. aircraft companies. Our initial efforts are to implement our progressive composite failure methodology into COMET as a means to become familiar with COMET.

The last area in CSM is simulation of sheet metal forming processes. While not specifically involving CST, it addresses the important issues of manufacturability and concurrent engineering.

Computational modeling of materials and processes is an area of cooperation between mechanics and materials scientists. Our efforts include micro-mechanics simulation of short fiber metal matrix composites (shown on subsequent visuals), ceramic-to-metal bonding and semiconductor crystal growth.

- Computational Structural Mechanics
  - Composite plate and curved beam elements
  - Anisotropic beam phenomenology
  - Improved lamina failure criteria
  - NASA CSM testbed
  - Nonlinear static and dynamic analysis
  - Simulation of sheet metal forming processes

- Computational Modeling of Materials and Processes
  - Micro-mechanics of short-fiber metal matrix composites
  - Ceramic-to-metal bonding
  - Semiconductor crystal growth
  - Sheet metal forming
  - Coldworking for fatigue life enhancement
  - Polymer composites processing tools design, thermal/curing distortion

- Smart Structures/Smart Materials
  - Piezoelectric fracture
  - Micromechanics of embedded devices

SIMULATION OF MANUFACTURING PROCESSES

Three areas have been identified as being important: sheet metal forming; tooling for polymer matrix composites; and simulation of resin transfer molding.

- An effort in Simulation of Manufacturing Processes was Begun at the Grumman Corporate Research Center in Concert with Manufacturing Engineering
- After an Extensive Study Three Areas were Identified:
  - Sheet metal forming
  - Tooling for polymer matrix composites
  - Simulations of resin transfer molding (RTM)

The next few figures outline areas involving Manufacturing Process simulation. It is shown here to emphasize our view that this is an important area to be considered. The area is not necessarily CST but qualifies as an important area in computational mechanics and is an important aspect of design/analysis of high performance aircraft.

PROBLEM

Manufacturing Processes are Becoming Increasingly Complex and Technically Demanding

- It is Necessary to Develop Physically Realistic Numerical Simulations of Manufacturing Processes in Order to Reduce the Time, Cost, and Risks Associated with Developing and Building New Airframe Structures
- While this Does not Strictly Qualify as Computational Structures Technology, It is an Important Aspect of Design/Analysis of High Performance Aircraft
- Research in this Area Involves Interdisciplinary Effort in Computational Mechanics, Materials Sciences, and Computer Science

SHEET METAL FORMING

The following outlines our initial efforts in simulation of sheet metal forming. Our efforts involve a partnership with material scientists and manufacturing engineers. The ultimate goal is to develop a comprehensive design/analysis system that goes from a geometric parts data base (CATIA) to an array of computational tools for simple 1 and 2 dimensional simulations on a work station to full three dimensional simulation a super computer. Initial candidates for use are the LLNL nonlinear codes, NIKE2D and DYNA3D.

- Finite Element Simulation of Various Sheet Metal Forming Operations
  - Simple 1 and 2 dimensional simulations (Implicit)
  - Comprehensive 3 dimensional simulation (Implicit or Explicit)
- Global Computational Strategies Seem to be in Place
  - Commercial and public domain codes
  - Computationally intensive
- Better Models Describing Friction and Material Behavior Required
- Essential Partnership with Material Scientists
  - Material testing
  - Friction testing
  - Qualify numerical simulations

The next few figures outline the Laboratories' work with Short-Fiber-Reinforced Composites (SFRC). This work was done in collaboration with the structural materials laboratory of CRC. The micromechanics and overall mechanical properties of short-fiber-reinforced metal matrix composites (MMCs) were investigated. See "Elastoplastic Finite Element Analysis of Short-Fiber-Reinforced SiC/A/Composites: Effects of Thermal Treatment," by Alvin Levy and John Papazian (Acta Met.&Mat., Vol. 39, No. 10, pp. 2255-2266). Composite materials provide improved stiffness/density, thermal conductivity/density and strength/density compared to conventional metal alloys. In addition, SRFCs offer excellent formability characteristics compared to continuous-fiber-reinforced composites. Because of these enhanced properties, and the potential to control the overall properties, SFRCs are receiving widespread attention as an aerospace structural material. In this investigation, elastoplastic FEA methods were developed and utilized to investigate the micromechanics and overall mechanical behavior of SFRCs so as to better understand and predict their behavior. This included thermomechanical behavior due to material processing as well as high service temperatures. In collaboration with the Materials Laboratory of CRC, fundamental tests were performed to obtain individual component and overall composite material properties. Obtaining accurate component properties enabled comparisons to be made between the finite element and experimental data.

SHORT FIBER METAL MATRIX COMPOSITES

- Mechanical Properties of Short Fiber Reinforced Metal Matrix Composites was Investigated Using Three-Dimensional Elasto-Plastic Finite Element Model

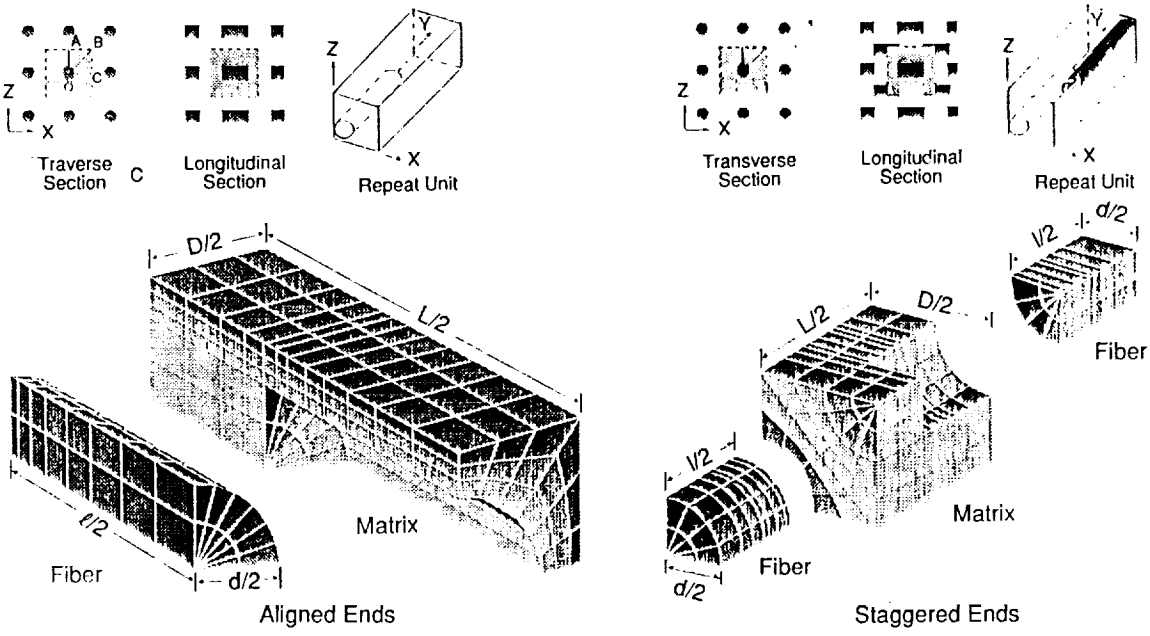
- Micro-Mechanics Approach - Modeled Fiber/Matrix Unit Cell

- Analysis Included
  - Elasto-Plasticity
  - Thermomechanical behavior
  - Residual stress

- Factors Critical to Overall Mechanical Behavior Investigated
  - Individual component properties
  - Fiber/matrix packing
  - Fiber volume content
  - Fiber orientation
  - Thermal residual stresses due to heat treatment

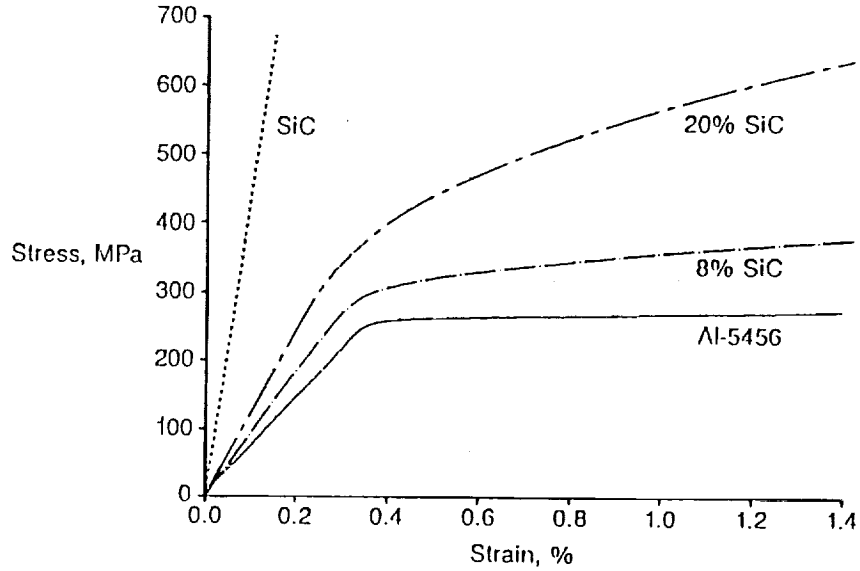
- Partnership with Material Scientists

FINITE ELEMENT MODELS OF SFRC



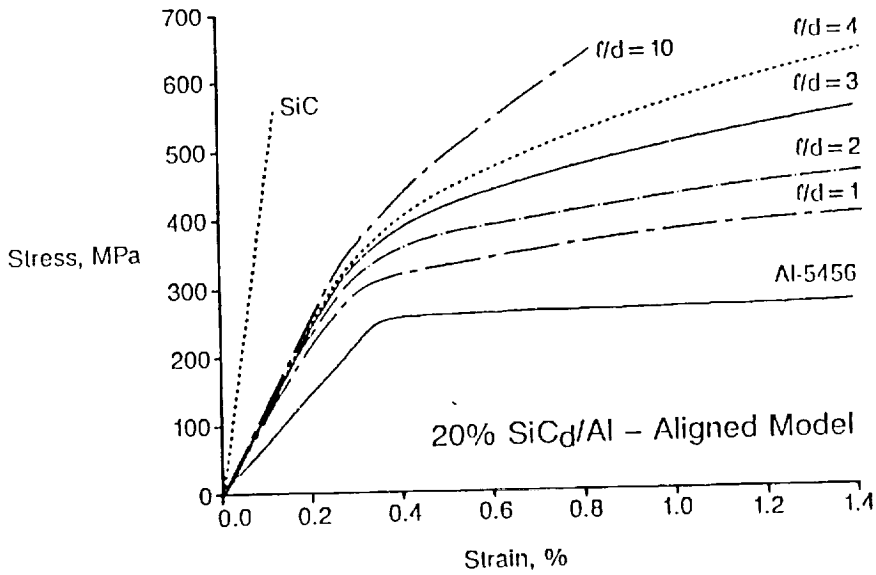
Stress-Strain Curves for SiC_w/Al

Various Fiber Volume Content



Stress-Strain Curves for SiC_w/Al

Various Fiber-Aspect Ratios



SHORT FIBER METAL MATRIX COMPOSITES

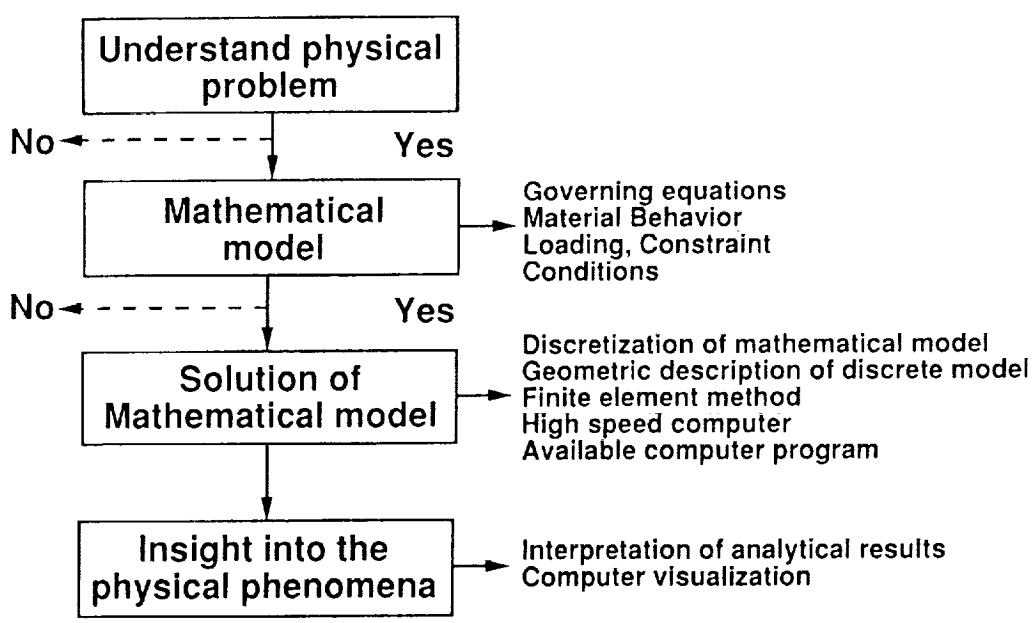
- Conclusions

  - Micromechanics leads to better understanding of mechanical behavior
  - Residual stresses affect mechanical behavior
  - Tensile and compressive properties different
  - Overall properties predicted and controlled
  - Thermal cooling affects mechanical properties

REQUIREMENTS FOR COMPUTATIONAL STRUCTURES TECHNOLOGY

The figure below is an attempt to characterize the essentials of an analysis process. Of vital importance and perhaps obvious is understanding the physical problem. This is particularly important as the phenomena that we wish to simulate become more and more complex, i.e., the HSCT. This understanding can come from experience or combined test and analytic studies, i.e., material modeling to component tests.

Once understanding the physics, the next and classical step is to create the mathematical model - the first level of modeling. One can create an accurate (in a numerical sense) discrete model of a structure based on an inadequate mathematical model - obvious issues here can be use of engineering technical theory versus full three-dimensional analysis, material constitutive behavior. Solution of the mathematical model is that area called computational mechanics that depends on discretizing the mathematical model, high-speed computing and ultimately an available computer program. Issues here are the validity and accuracy of the discrete method, and computer resources needed; also, the availability of a computer code, an ever increasingly expensive and time consuming item. And finally, the "bottom line" insight into the physical phenomena. A necessary ingredient is postprocessing capability and computer visualization, but an essential understanding of the three above items is essential. The analysis may not be "deterministic" but with the correct judgement it can give the appropriate insight.



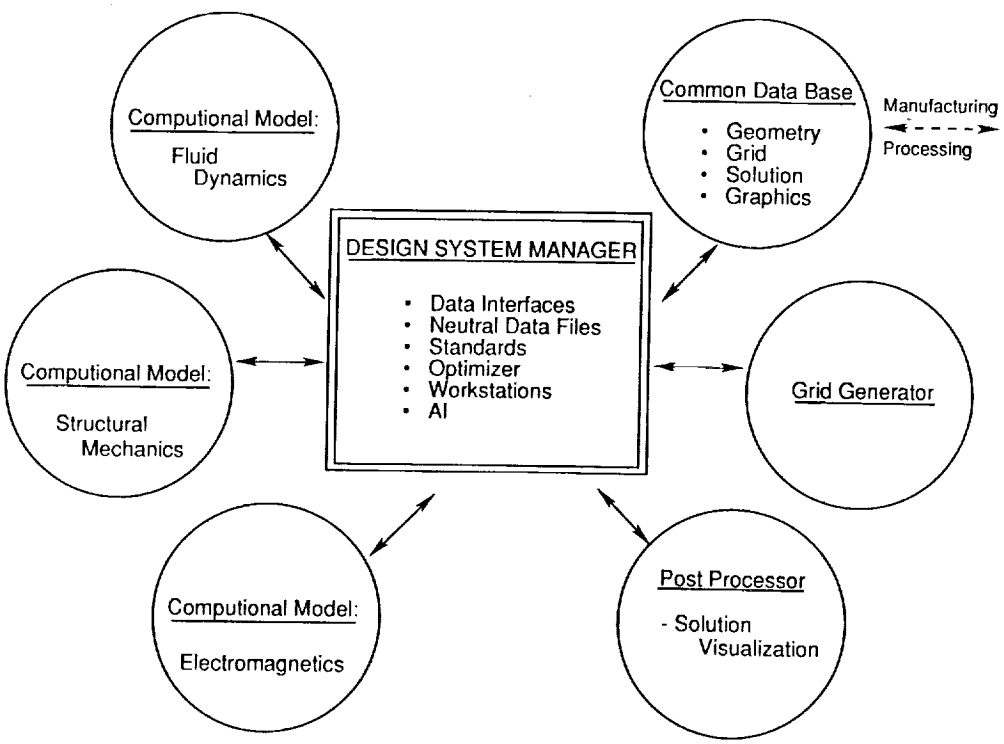
Which leads to the "punch line" on the following figure

THERE IS NO SUBSTITUTE FOR SOUND ENGINEERING JUDGMENT

COMPUTATIONAL STRUCTURES TECHNOLOGY

- Good News
  - A computer program will usually provide "Answers" to complex problems
- Bad News
  - A computer program will provide "Answers" to complex problems
- There is No Substitute for Sound Engineering Judgement

This figure combines the thoughts of CFD as well as CST. The structures area has addressed these issues for a long time primarily because computational mechanics efforts grew out of early use and development of FEM in the aircraft industry. Integrated systems exist in one form or other in each company. At Grumman we have the COGS system, discussed previously. However, these issues still exist; as does the question of how to integrate new state-of-the-art analysis/design methods into the system.



PROBLEM

THE USE OF STATE-OF-THE-ART COMPUTATIONAL METHODS FOR DESIGN AND ANALYSIS HAS BEEN LIMITED FOR A NUMBER OF REASONS

- The Weak Link Between Geometric Modeling and Computational Analysis Tools
  - It can take several months of labor-intensive effort to generate the geometric data necessary for input to computational tools
- Lack of Sufficient Computer Power has Impeded the use of Modern Computational Techniques in the Design Process
- Time Required for Analyses are Incompatible with Project Timetables so that State-of-the Art Computational Tools are not Used
- Weak links Between Computational Tools for the Individual Disciplines such as CFD, CMS, CEM, etc. also Severely Inhibits their use in Design Efforts
- Added Complexities Exist for Computational Structural Mechanics as Dictated by Global/Local
  - Loads/Boundary Conditions are Needed from Global Model

OBSERVATIONS AND CONCLUSIONS

The final three figures are self-explanatory and come from our experience and the three items cited. Classification and prioritizing are important and difficult. Perhaps an on-going airframe manufacturers' group can better address these issues. And finally, the questions raised on the last visual. Where will new technology come from, how is it transferred to industry, who should do it, what is NASA's role? If technology is ultimately translated into software, who writes the software? Can we envision industry-wide standards for a design/analysis system that can integrate commercial proprietary and government software? What role could a U.S. Aerospace Consortium play in such an endeavor?

- A Consensus Exists on the Important Research Areas in Computational Structural Mechanics
  - "Advances and Trends in Computational Structures Technology," by A. K. Noor, S. L. Venneri (eighty-six references) Computing Systems in Engineering, Vol. 1, No. 1, 1990
  - Draft Report, "Research Directions in Computational Mechanics," Editorial Committee, J. T. Oden, T. Belytschko, I. Babuska
- Objectives and Goals of NASA Langley Research Center Computational Structural Mechanics Branch Branch Head - Jerry Housner
- Research Issues:
  - Integrated detailed analysis of complex structures from global to local scales
  - Routine use of nonlinear methods in vehicle design/analysis
  - Adaptive methods and error estimation
  - Impact and implementation of new computing systems - parallel computation and the quest for the "Teraflop" computer
  - Prediction of failure of structural components composed of new materials
  - Discrete modeling via geometric database - CAD/CAE
  - Computer visualization
  - Qualify methods by comparison to test data

- Research Issues Should be Classified and Prioritized in each Class
- Class 1 - Do what we can do today better and faster
  - Integrated Detailed Analysis
  - Global/Local
  - Adaptive mesh and error indicators
  - More routine use of nonlinear analysis
- Class 2 - Things that we can not currently do
  - Failure prediction and optimization of new material systems (partition between material and structure is disappearing)
  - Impact of new computers (new parallel algorithms)
  - ?????
- Required Research and Development in Computational Structures Technology is Comprehensive and Costly
- Individual Aerospace Companies can no longer afford to "Go-It-Alone" and Develop all their Own Methods and Software
- New Technology Must Come From
  - Government - NASA, National Laboratories
  - Government/Industry Partnerships
  - Universities
- An Important Question is; How will this technology be integrated into design analysis systems for aerospace vehicles?
- How will Technology be Shared Among Competing Companies?
- Should the Commercial Market be Completely Relied Upon?
- What Role Should NASA Play?

N92-25931

Airframe Life Prediction

G. P. Sendeckyj
Wright Laboratory
Wright Patterson Air Force Base, Ohio

PRECEDING PAGE BLANK NOT FILMED

AIRCRAFT STRUCTURAL INTEGRITY PROGRAM REQUIREMENTS

Air Force aircraft are designed to meet the requirements of and are managed under the Aircraft Structural Integrity Program (ASIP). ASIP requires that the aircraft structure be designed to be damage tolerant and durable, and specifies the required verification testing to ensure that the requirements are met. Moreover, it requires a force management plan for managing the aircraft fleet in service.

- DURABILITY - economics issue
  - INITIAL DAMAGE ASSUMPTIONS
  - ANALYSIS REQUIREMENTS
- DAMAGE TOLERANCE - safety issue
  - INITIAL DAMAGE ASSUMPTIONS
  - ANALYSIS REQUIREMENTS
- VERIFICATION TESTING
- FORCE MANAGEMENT

Figure 1

DAMAGE TOLERANCE REQUIREMENTS

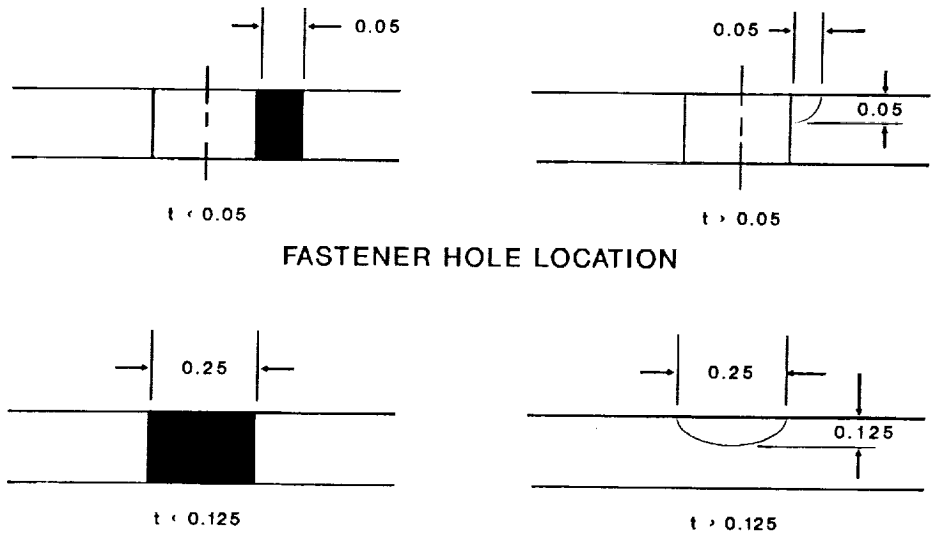
The Aircraft Structural Integrity Program requires that the airframe shall be designed to be damage tolerant, that is, safety-of-flight structural components of the airframe shall be capable of maintaining adequate residual strength in the presence of material, manufacturing and processing defects until damage is detected through periodic scheduled inspections. The damage tolerance requirement specifies what must be done to ensure that the airframe structure meets the required, but does not specify the exact procedures to be followed. For metallic structures, the requirements are further subdivided based on the number of load paths.

- Safety-of-flight and other selected structural components shall be capable of maintaining adequate residual strength in the presence of material, manufacturing and processing defects until damage is detected through periodic scheduled inspections.
- Single load path structure without crack arrest features shall be designed as slow crack growth structure.
- Structures utilizing multiple load paths and crack arrest features shall be designed as either slow crack growth or fail-safe under appropriate inspectability levels.

Figure 2

INITIAL PRIMARY DAMAGE ASSUMPTIONS

The damage tolerance requirements specify the type of damage that has to be assumed to be initially present in the most critical portions of the structure. The initial damage assumptions for metallic structures are based on a rogue flaw scenario, in which the flaw is missed during manufacturing and assembly inspections. At fastener holes, the initial primary damage assumption is either a 0.05 inch through-the-thickness crack or a 0.05 inch semi-circular corner depending on the material thickness. At locations other than a fastener hole, the initial primary damage assumption is either a 0.250 inch long through-the-thickness crack or a semi-elliptical surface crack that is 0.250 inches long by 0.125 inch deep.



NOTE: ALL DIMENSIONS IN INCHES

Figure 3

INITIAL CONTINUING DAMAGE ASSUMPTIONS

The damage tolerance requirement for metallic structures assumes that additional damage, corresponding to the durability requirements to be discussed later, is present near the initial primary damage. The initial damage assumptions are the presence of a 0.005 + Δ inch semi-circular crack at fastener holes, and a 0.02 + Δ by 0.01 + Δ inch semi-elliptic crack at locations other than fastener holes. Δ is the incremental growth of the indicated flaw prior to initial damage termination, such as the fracture of the material between fastener holes.

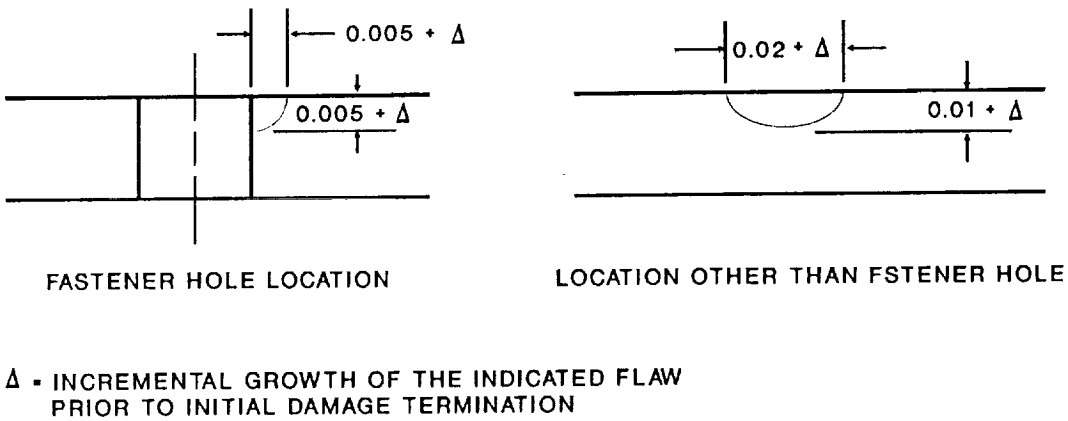


Figure 4

INITIAL DAMAGE ASSUMPTIONS - COMPOSITES

For polymer-matrix composite structures, the initial damage assumptions are the presence of a surface scratch, delamination or impact damage with the dimensions indicated in Figure 5. Analysis has shown that the surface damage is rather benign and, hence, it is normally not considered in the damage tolerance analysis. The delamination and impact damage sizes are based on worst case scenarios. They can be relaxed if a 100 ft-lb impact will not occur during assembly of the airframe and/or service.

DAMAGE/FLAW TYPE	DAMAGE/FLAW SIZE
SCRATCHES	SURFACE SCRATCH THAT IS 4.0-in LONG AND 0.02 in DEEP
DELAMINATION	INTERPLY DELAMINATION WITH AREA EQUIVALENT TO 2.0-in DIAMETER CIRCLE
IMPACT DAMAGE	DAMAGE CAUSED BY IMPACT OF 1.0-in DIAMETER HEMISPHERICAL IMPACTOR WITH 100 ft-lb OF KINETIC ENERGY OR WITH THAT KINETIC ENERGY REQUIRED TO CAUSE A DENT 0.10-in DEEP, WHICHEVER IS LEAST

Figure 5

RESIDUAL STRENGTH REQUIREMENTS

The damage tolerance requirement specifies that flaw/damage growth and residual strength analyses shall be performed, but does not indicate the type of the analysis. Normally, this is negotiated with the airframe manufacturer. The analysis should be within engineering accuracy and account for all important loading effects. If an adequate analysis procedure is not available, then the manufacturer has the option to do it through development testing. This is the current situation for composite structures. Moreover, development and full-scale damage tolerance tests are required to demonstrate that the airframe structure meets the requirements. These tests are also intended to uncover design errors.

ANALYSIS

- Flaw/damage growth and residual strength analyses shall be performed assuming the presence of flaw/damage placed at the most unfavorable location and orientation with respect to applied loads and material properties.
- The analysis shall predict the growth behavior of this flaw/damage in the chemical, thermal, sustained and cyclic stress environments.
- Spectrum interaction effects shall be accounted for.

TESTS

- Development and full-scale damage tolerance tests are required to demonstrate that the airframe structure meets the requirements.

Figure 6

GENERAL DURABILITY REQUIREMENTS

The durability requirement addresses the economics of ownership of the aircraft and, hence, it is an economic requirement. It applies to all primary and secondary structures and is necessary, but not sufficient to ensure safety-of-flight. Simplistically, it states that damage shall not develop in the airframe during one design lifetime to the point that it is not economical to repair and/or maintain the aircraft. This requirement is critical for commercial aircraft. The initial damage assumption for metallic structure is the initial continuing damage assumption of Fig. 4 with Δ set equal to zero. The requirement can be relaxed if the manufacturer can prove that the quality of his airframe is better than that implied in the initial damage assumption. The initial quality concept has been developed to support this. For composite structures, the initial damage assumptions are the same as the damage tolerance assumptions.

- DURABILITY OF THE AIRCRAFT SHALL BE ADEQUATE TO RESIST FATIGUE CRACKING, CORROSION, THERMAL DEGRADATIONS, DELAMINATIONS AND WEAR DURING OPERATIONAL AND MAINTENANCE USE.
- THE ECONOMIC LIFE SHALL BE SUFFICIENT TO WITHSTAND THE SERVICE LIFE AND USAGE.

- APPLIES TO ALL PRIMARY AND SECONDARY STRUCTURES
- NECESSARY, BUT NOT SUFFICIENT TO ENSURE SAFETY-OF-FLIGHT

Figure 7

SPECIFIC DURABILITY REQUIREMENTS

The specific durability requirements are outlined in Fig. 8. The analysis requirements are similar to those for ensuring damage tolerance, but with different initial damage assumptions. Full scale airframe testing is required to show that the economic life exceeds the required service life. Normally, the durability test is performed first. If the verification requirements are satisfied, then damage is introduced into the test airframe and damage tolerance testing performed.

- FATIGUE CRACKING/DAMAGE - Adverse cracking which would cause functional impairment or required costly maintenance action or both shall not occur within two lifetimes of expected usage.
- ANALYSIS shall be performed to show that costly to repair airframe structure cracks or other damage shall not reach sizes large enough to necessitate repair, modification, or replacement of components within the required time.
- DEVELOPMENT TESTING shall be performed to provide adequate data to meet the requirements.
- DURABILITY TESTS - A complete airframe or approved alternatives shall be durability tested to show that the economic life exceeds the required service life.

Figure 8

VERIFICATION ISSUES

The current practice (conventional wisdom) is to perform static tests to establish the durability and damage tolerance of composite structures and to perform spectrum fatigue tests for metallic structures. This practice is a consequence of differences in the fatigue behavior of composite materials and metals. The damage producing events in composites are high loads. Moreover, composites exhibit large scatter in fatigue lives and are insensitive to cyclic loading (that is, the slopes of the S-N curves are very shallow). As a result, it is not economical to fatigue test composite structures. In contrast, cracks grow in metallic structures at relatively low loads, while high loads tend to retard crack growth. As a consequence, the metallic components of the structure are the weak link.

- CONTRADICTORY REQUIREMENTS - Composites vs Metals
  - Composites have NO FATIGUE PROBLEMS
    - Degradation due to peak loads
    - Large fatigue data scatter
    - Design conservatism
  - Metals
    - LOW LOADS produce crack growth
    - HIGH LOADS retard crack growth
- CONSEQUENCES
  - Metal structure is weak link
  - Static test for composite structures
  - Fatigue test for metallic structure

Figure 9

LIFE PREDICTION METHODS

The life prediction methods for metallic structures, subjected to thermomechanical loading are reasonably good. Crack growth models that incorporate contributions from cyclic cracking and creep are adequate for engineering accuracy. Probabilistic life prediction methods are needed to provide confidence intervals on the predicted lives and improve the durability analysis methods. The current life prediction methods are not adequate for composite structures. One reason for this is that the user community has become convinced that composites do not have fatigue problems because the currently used design load levels are low. As a result, support in this area has been decreased. The residual strength degradation fatigue models that have been developed for resin-matrix composites seem to be adequate on the surface. In reality, they are too simplistic to provide reliable results. Damage state based models are being developed to improve the life prediction methods. For delamination growth, conventional crack growth type models are adequate.

METALS

- CYCLE-BY-CYCLE INTEGRATION OF CRACK GROWTH LAWS

$$\frac{da}{dt} = \underbrace{c\,(\Delta K)^n}_{\text{cyclic}} + \underbrace{B\,K^m}_{\text{creep}}$$

- PROBABILISTIC METHODS - relax initial crack assumptions

RESIN-MATRIX COMPOSITES

- RESIDUAL STRENGTH DEGRADATION MODELS
- DAMAGE STATE MODELS (e.g., Reifsnider)
- DELAMINATION GROWTH MODELS

$$\frac{da}{dn} = C\,(\Delta G)^n$$

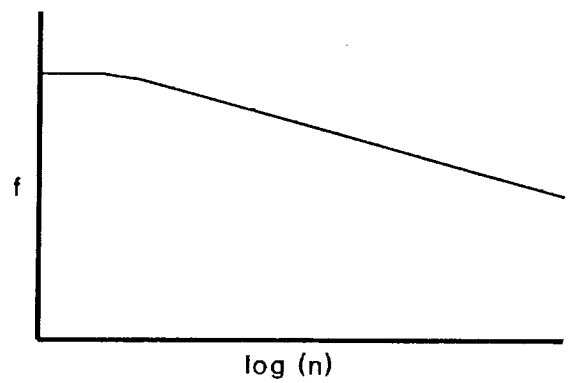
Figure 10

LIFE PREDICTION METHODS

The residual strength degradation models assume that the residual strength is an adequate metric and use a postulated residual strength degradation equation, which can be integrated to define the S-N curve. Since the parameters in the models are determined experimentally, the residual strength degradation models are just a fancy type of curve fitting. Considering the large number of possible stacking sequences in composite structures, they are rather uneconomical in practice. The damage state models have been developed to decrease the amount of required testing. They assume that the fatigue life of the composite is governed by a critical element normally assumed to be the 0 degree plies. The remaining plies play a secondary role. The damage state in the secondary plies effects the loads on the critical elements, but does not control the fatigue life.

RESIDUAL STRENGTH DEGRADATION (RSD) MODELS

$$\frac{df}{dn} = -(1/d)\, C\, f_a^{d} f^{1-d} \qquad f_e = f_a[(f_r/f_a)^{1/S} + C\,(n-1)]^{S}$$



DAMAGE STATE MODELS

- RSD MODELS CONTROL LIFE OF CRITICAL ELEMENT
- DAMAGE STATE EFFECTS LOADS ON CRITICAL ELEMENT

Figure 11

DAMAGE GROWTH IN MMC

Metal matrix composites exhibit complex behavior in thermomechanical fatigue. The damage accumulation process can consist of self-similar crack growth as in homogeneous metals or extensive matrix cracking as in resin matrix composites. Which type of damage accumulation process occurs depends on the test temperature. Figure 12 illustrates the cyclic damage growth in a typical unidirectional metal matrix composite. As can be seen from the figure, matrix cracks parallel to the loading direct develop first and continue to grow into the grips. Matrix cracks perpendicular to the loading develop later during the cyclic test and these grow and multiply until the specimen surface is covered with hundreds of surface cracks. The matrix cracks do not fracture the fibers and, hence, the residual strength is very high. Similar behavior occurs in unidirectional metal-matrix composites with crack-like slits.

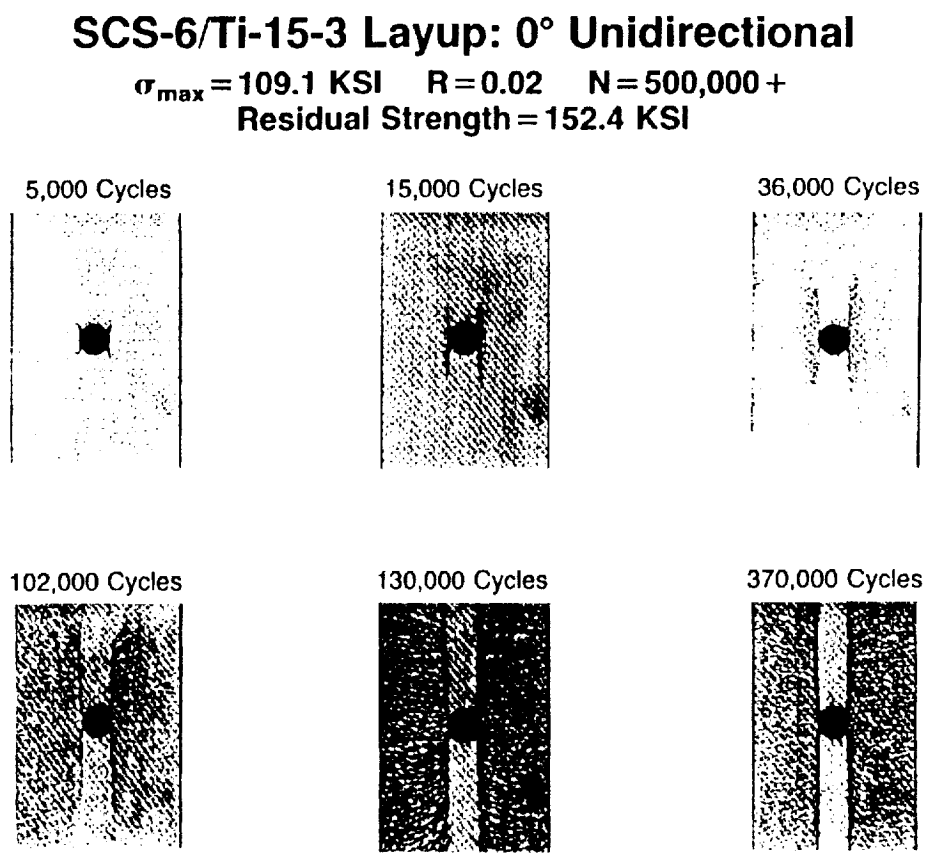


Figure 12

RESEARCH ISSUES

The required research to develop improved life prediction methods for metallic and composite structures under severe thermomechanical loading must include the development of a verified thermoinelastic fracture criterion. There has been much work in this area with many fracture criteria being proposed. Due to the lack of adequate experimental verification none of them are widely accepted. Research must also be performed to develop and implement improved thermoinelasticity theories that properly model large temperature excursions and high temperature gradient. This research is required to provide confidence in the simpler theories currently used for thermoinelastic analysis. Finally, experimental data is needed to define the behavior of and damage accumulation process in thermoinelastic materials. Special emphasis must be placed on understanding failure mode transitions under thermomechanical loading conditions.

THERMOINELASTIC FRACTURE CRITERIA

- MANY PROPOSED, e.g.
  - ΔT_C Integral (Atluri)
  - C^* PARAMETER (Landes and Begley)
  - STRAIN RANGE PARTITIONING
  - S Integral (Wagner)
- NONE FULLY ACCEPTED

THERMOINELASTIC THEORIES/ANALYSIS METHODS

- LARGE THERMAL GRADIENTS
- LARGE TEMPERATURE EXCURSIONS
- COUPLING EFFECTS

EXPERIMENTAL DATA

- FRACTURE with TEMPERATURE GRADIENTS
- TEMPERATURE INDUCED FAILURE MODE CHANGES

Figure 13

WL/FIBEC PROGRAMS

The Structural Integrity Branch of Wright Laboratory has two major efforts to develop life analysis methods for metal matrix composite materials. The first effort is a contractual program with McDonnell-Douglas Corporation to develop strength and life prediction methods, while the second effort is an in-house program to verify and extend the contractually developed methods to titanium matrix composites.

THERMOMECHANICAL LOAD HISTORY EFFECTS IN METAL MATRIX COMPOSITES - Contract F33615-87-C-3219

- OBJECTIVE: Develop Strength and Life Prediction Methodology for Metal Matrix Composites Subjected to Combined Mechanical and Thermal Loading

MATERIAL CHARACTERIZATION OF TITANIUM MATRIX COMPOSITES - NASP GWP No. 85G

- OBJECTIVE: Develop and Verify Life Analysis Methods for Titanium Matrix Composites Designed as NASP Structural Components
- APPROACH: Conduct Fatigue Crack Growth Tests on Center Cracked and Open Hole Specimens Made From SCS-6/Beta 21S Panels and Correlate Results With the Analytical Model

Figure 14

N92-25932

Probabilistic Design Applications for the Space Transportation Main Engines

Kevin O'Hara
Rockwell International
Huntsville, Alabama

PRECEDING PAGE BLANK NOT FILMED

Figure 1. Probabilistic Design Applications Agenda

Figure 2. Present Day Propulsion System Reliability Issues
Recently published reliability data indicates that the flight reliabilities of current propulsion vehicles do not exceed 96%. Atlas, Delta, Thor and Titan vehicles all achieve less than 96% flight reliability. This means that one failure occurs out of every 25 launches (minimum). Unfortunately, the Space Shuttle accident (mission 51L) failed on its 25th mission. Even the Arriane 4 has a flight reliability record of about 92% and the Soviet Proton has had 10 failures in 200 flights (95%). A review of the record indicates that over half of these failures can be attributed to the propulsion system.

All current flight vehicle propulsion systems were design based on deterministic techniques, qualitative reliability assessments and limited testing. Therefore, based on the record, an upper limit flight reliability of ~96% exists for this design methodology. Today, customers are demanding higher reliabilities for new propulsion systems (RM2000, DoD initiatives, Advanced Launch System statement of work). In order to meet contractual requirements and the new specifications for achieving a quantified propulsion system reliability, a new design approach is required.

Figure 3. Present Day Design Approach
All of the current rocket engine designs are based on deterministic design analysis methods. The most recently designed engine, Space Shuttle Main Engines (SSME), was designed and developed using these techniques in the mid-1970's through the early 1980's. Although limited probabilistic techniques were available at the time, none were used in the design process. During the design of the SSME, no formal quantitative reliability nor risk assessments were conducted. Both reliability and risk assessments were qualitative in nature with semi-quantitative support analysis/data.

Traditional reliability analyses usually predict a demonstrated propulsion system reliability at the end of the development or test program. For a new rocket engine design, it takes at least three to four years for component testing to be complete, and five to six years before sufficient engine test data is available. Knowing the reliability at this point in time does not permit many design changes to occur, if any. Traditional reliability approaches to design and development do not permit pro-active reliability assessments of the design during the design process.

Figure 4. STME Design Objective
The Space Transportation Main Engine (STME) design objective is to utilize a robust design methodology and to pro-actively assess and improve the design during the design process. The STME has two significant engine level reliability requirements:

- a demonstrated engine reliability of 0.99 at 90% confidence, and
- a design-to reliability of 0.999.

This design approach is a combination of the traditional deterministic methodologies and recently developed probabilistic methodologies. The designer/analyst must consider "lesson learned," utilize a more rigorous structural criteria, conduct pro-active reliability assessments (reliability allocations, FMEA's, reliability growth studies and design reliability

aggregation/assessment), employ a damage tolerant design philosophy and perform probabilistic analyses. Rigorous application of this structured methodology will develop a better engine product.

Figure 5. "Designing for Reliability"
A graph depicts a more rapid assessment of the engine reliability. The main point is with probabilistic techniques one can have an engine reliability assessment during the design process. Additionally, the engine reliability during the development and demonstration phases do not have to start at zero. The design analysis and material/component/engine level testing will define/augment/supplement the analysis reliability estimate.

Figure 6. "Designing for Reliability" - General Approach
"Designing for Reliability" is a six-step design process. The process begins with a reliability design goal (i.e., STME engine level design-to reliability is 0.999). Next, it considers historical data during design layout (incorporation of lessons learned), then it requires that an FMEA/CIL be conducted early (design out or desensitize problem areas).

Subsequently, the process identifies critical hardware and/or component failure modes via a component screening process and then requires that a probabilistic analysis be conducted for each component critical failure mode. During the design analysis process, a bottom-up reliability estimate (based on either probabilistic analysis or simplified reliability assessment) is calculated and compared to the allocation for each component. Only reliability estimates greater than the allocation would be considered as acceptable designs. Finally, an overall reliability assessment of the engine system (failure mode aggregation) is conducted, which assesses the overall calculated reliability for the engine to the design-to reliability goal.

Figure 7. Component Reliability Allocation
A component allocation is established based on historical data. The STME reliability allocation is based on SSME failure histories of each component with adjustments made for design differences/improvements (i.e., the STME does not have ball bearings and/or the associated bearing failure modes). The reliability allocation is a top-down assessment of component failure histories and these allocations can be taken down to the piece part (depending upon the resolution of the historical failure data). These reliability allocations represent the design-to values (analogous to factor of safety) and should be modified only if the design is changed.

Figure 8. Historical Data
Historical data must be utilized to the maximum extent possible. Designs should be the best they can be, and not be subject to failures that were experienced ten years earlier. A database of "lessons learned" should be established so that the designers and the analysts can check for past failures or problem areas. It is imperative to review all data and design out these potential problem areas. The second item is to minimize those failure modes or problem areas that cannot be completely designed out (i.e., process control, procedures, desensitize the design, etc.). For the remaining failure modes (critical to the engine operation), a control system must be in place to detect and take corrective action during engine operation.

Figure 9. Failure Modes and Effects Analysis (FMEA)
Most designers/analysts conduct FMEA's for new designs, but they usually are not conducted early enough. FMEA's must be conducted early to allow the designer/analyst to mitigate a particular failure mode via a design change. This permits pro-active reliability improvements

in the design process during the design. Additionally, the FMEA must be updated as the design matures. Many program managers have had unexpected failures, because the design is different than the analysis. Upon completion of the engine design/development, the FMEA should reflect all potential failure modes that can occur within the engine.

The most important analysis in the FMEA process is the estimated probability of failure for each failure mode. Many failure modes are catastrophic, but only a few of these failure modes will occur during the operational life of the part. Identification of these few critical failure modes will assist the designer and the analyst in eliminating or minimizing their failure mode potential.

Figure 10. Critical Component Screening
Once an FMEA and a simplified deterministic analysis has been conducted, usually in the preliminary design stage, a failure mode screening process can be conducted. This screening technique will identify which components/piece parts are failure mode critical. It is these components that require a probabilistic analysis. Components that would typically require this additional analysis are: main injector LOX posts, pump impellers, turbine blades, or flow splitters in a dense turbulent fluid. It should be noted that only one or two failure modes control a piece part (i.e., high and low cycle fatigue for turbine blades) and it is only these failure modes that require the probabilistic analysis.

Critical components are identified via a typical screening criteria, such as prior failure histories, criticality to the operation of the engine, potential for failure, complex of the design, sensitivity of the analysis or failure safe/level of redundancy. Components that have high potential for failure and are sensitive to their environment would be candidates for the additional analysis. It is estimated that approximately 20% of the hardware will require a detailed probabilistic analysis.

Figure 11. Simplified Probabilistic Analysis - Fatigue Strength
All hardware will be assessed using deterministic techniques (i.e., factor of safety yield or ultimate, high and/or low cycle fatigue life, fracture mechanics life). Approximately 20% of the hardware will receive a detailed probabilistic analysis, but the remaining hardware will be assessed via deterministic and simplified probabilistic techniques. The simplified technique will calculate a reliability estimate from the deterministic factor of safety. This figure shows the general approach for determining a reliability estimate.

The example of fatigue strength assessment shows the relationship between factor of safety of endurance and the failure region between operating stress and material capability. Reliability can be computed via a reliability/factor of safety interrelation nomograph. These failure mode specific reliability estimates can be aggregated with other reliability estimates for each component and compared to the reliability allocation.

The most important feature in conducting a simplified reliability estimate is the variance analysis that occurs when making the estimate. The simplified method requires that the analyst understand the mean, variance and distribution of both the operating stress and material capability. This variance analysis lets both the analyst and the designer have a better understanding of component versus engine criticality.

Figure 12. Damage Tolerant Design Approach
The STME requirements dictate a robust design (i.e., components that are damage tolerant). Damage tolerant hardware can be designed via fracture mechanics analysis. Damage

tolerance must be designed in. Hardware can be desensitized to material flaws, weld defects or fabrication/handling damage.

Figure 12 shows an example of damage tolerance to material porosity for castings. Each casting will be assessed for size and type of porosity inherently in the casting process. The component stress intensity will be computed for the casting based on porosity size and stress conditions. Then the design could be modified to reduce or eliminate flaw growth. Methods to accomplish this are to thicken material walls, minimize stress concentrations, or modify the casting process to eliminate flaws above a certain size.

Figure 13. Detailed Probabilistic Analysis
A detailed probabilistic analysis is comprised of three main elements: component loads definition; component response calculation; and component failure rate or reliability estimate. These three steps may utilize expert opinion, specific load models, finite element/difference models, closed-form solutions, and/or computer simulations (Monte Carlo, Fast Probability Integration schemes) to assess component reliability. The next three figures will elaborate on this approach via an example.

Figure 14. Component Loads Analysis
Component loads can be obtained through three methods: scaled data; an empirical model; or expert opinion. Usually the component loads are determined via a combination of these methods. Figure 14 defines the process for establishing a turbine blade from an empirically derived engine model. This model was developed through NASA-LeRC and is called Composite Loads Spectra model. The approach relates engine primitive variables (thrust, inlet conditions, mixture ratio, pump speed, etc.) to component loading on a turbine blade. This relationship is established through sets of influence coefficients derived from older engines (where test data is compared to the analysis of terms of component loading). Design input variables are produced in a distributional format and are used as input to the component response analysis.

Figure 15. Component Response Analysis
Component response analysis can be accomplished using either a closed-form solution analysis or a finite element model which calculates component stresses. All inputs (component geometry and loads) are distributional in format and the component responses are output in a similar format. Figure 15 graphically depicts the analysis process.

The example shows a model of a turbine blade using the NESSUS finite element code. This code has been specifically developed for NASA-LeRC to conduct probabilistic response analysis. Both input and output variables are shown in a distributional format. NESSUS has the capability to compute the stresses or displacements for a particular model and conduct a reliability assessment as part of the analysis package.

Figure 16. Component Failure Rate
Component failure analysis can be accomplished as part of the component response analysis or computed separately. If computed separately, the input variables as shown in Figure 15 and the material capability are input to a failure model (tensile, fatigue, fracture mechanics, etc.). This failure model assesses the probability of failure for each failure mode.

The example shows a generic input model displaying the types of inputs required and the analysis flow logic. A failure simulation is computed using a Monte Carlo or FPI technique to

predict the expected failure rate. This failure rate can be converted to a reliability estimate for determining B1 of B.1 lives.

Figure 17. Benefits of Probabilistic Approach
The probabilistic approach has several benefits, five of which are listed in Figure 16. The three most important benefits that are realized are:

- A quantitative reliability estimate can be made during the design process.
- It forces the analyst to conduct a sensitivity analysis and ranks input variable effect to hardware reliability.
- Gaps in the design database are identified early and an optimum test program can be structured.

Figure 18. Space Transportation Main Engine (STME)
This figure provides background data on the STME for general audience understanding. This data supports the examples to come.

Figure 19. Combustion Chamber - Critical Hardware
This figure provides additional background data for forthcoming examples. It depicts the analysis locations of the combustion chamber liner and inlet manifolds.

Figure 20. Component Reliability Allocation
The STME initial reliability allocation is shown here. The engine level design-to goal of 0.999 is allocated down to the major component groups and then to the combustion device components.

Figure 21. STME Gas Generator Engine - Reliability Allocation
The STME entire initial reliability allocation has been tabulated to show how it is derived. This is a back-up chart.

Figure 22. Main Combustion Chamber
This figure depicts the physical features of the combustion chamber and shows both the chamber copper alloy liner and fuel inlet cross-sections. Section A-A defines the geometric configuration of the liner wall, including the coolant channel geometry.

Figure 23. Probabilistic Analysis of the MCC Liner
This figure show the steps for conducting the liner probabilistic analysis. The component loads model was used as defined in Figure 14. The liner example defines the primitive variables (engine cycle specific) and tabulates the output variables on the righthand side. The loads data was input into a one-dimensional thermal model to compute liner thermal gradients and calculate the correlated liner wall temperatures. Finally, all geometric data, thermal loading and material properties are input into a structural model for four failure modes. The results of that analysis are shown on the next four figures.

Figure 24. Probabilistic Analysis - Channel Bending Stress
A probability density plot indicates the relationship between liner bending stress and material yield strength. The analysis indicates a failure rate in excess of 0.000001. The analysis was stopped after a million calculations; therefore, the exact failure rate was not computed.

Figure 25. Probabilistic Analysis - Land Tensile Stress
A probability density plot indicates the relationship between land tensile stress and material yield strength. The analysis indicates a failure rate in excess of 0.000001. The analysis was stopped after a million calculations; therefore, the exact failure rate was not computed.

Figure 26. Probabilistic Analysis - Low Cycle Fatigue Life
A probability density plot shows the low cycle fatigue life of the liner. The analysis indicates a minimum fatigue life of 476 cycles and a maximum fatigue life of over 27,000 cycles. The combustion chamber liner requires only 15 engine starts (thermal cycles). The liner fatigue life requirement has been met and represents a good design.

Figure 27. Probabilistic Analysis - Ratcheting Failure Rate
This chart shows the life of the liner due to thermal ratcheting. The main issue defined by the chart is the low life for the thermal ratcheting failure mode. Review of field engines indicate that this liner is present and needs to be designed out. The thermal ratcheting failure data forces both the designer and the analyst to review the design. Two design changes are being evaluated as a result of this analysis:

1) lower the liner wall temperature via film coolant or
2) design in a Tungsten filament wire just below the surface of the liner hot wall. It eliminates the failure ratcheting mode.

Figure 28. MCC Aft Manifold - High Cycle Fatigue Analysis
A probability density plot shows the high cycle fatigue life of the aft manifold shell. The reliability estimate of the liner to meet the required fatigue life is $0.9^{9}85$. This design is acceptable as is. Another important set of data is the sensitivity factors. The input variable which most impacts the design reliability is the manifold stress concentration. Presently, it represents the highest effect of the stress, component fatigue life and reliability. If a design change was required, reduction of the stress concentration would have the most effect.

Figure 29. Reliability Assessment - Combustion Chamber Aft Manifold
This chart provides a summary description and the reliability estimate for the combustion chamber aft manifold. A probabilistic analysis was conducted for the aft manifold, similar to the liner analysis shown earlier. The input variables are shown and the resulting factors of safety and reliability estimates are shown for comparison.

Figure 30. Combustion Chamber Aft - Safety and Reliability Summary
This chart provides a summary of all the analysis for the combustion chamber. Both factor of safety and reliability estimates are tabulated for comparison. Presently, the liner meets its reliability allocation, as long as the liner ratcheting problem is resolved.

It must be noted that the liner ratcheting problem results in a small liner crack. This crack causes a small amount of hydrogen coolant to leak into the hot gas stream. It requires 15-20 cracks to cause a significant performance loss. Therefore, this failure is not a critical failure mode. The design change would eliminate the liner cracking, a benign failure.

Figures 31 and 32. Future Applications
These figures indicate the types of analysis that will be conducted on the STME engine hardware in the future. Analysis tools need to be developed for several failure modes. Additionally, a probabilistic methodology needs to be developed to assist in assessing process failures. The STME program is trying to develop fabrication process controls

(casting process, vacuum plasma spray process, etc.) to ensure consistently built hardware. Therefore, a probabilistic process methodology needs to be developed.

It should be noted that 50% of all propulsion system flight failures result from process failure modes. An example is the temperature sensor failure on Space Shuttle mission 51G. A sensor failure caused by a filament failure resulted from excessive winding tension. In fact, the winding tension could only be accomplished by one person correctly at the sensor fabrication facility. Probabilistic methods would have identified this process as a problem area, including how sensitive the process was.

Figure 33. Recommended Technology.
Five recommendations were made. The most important recommendation is the training and/or user friendliness of the new probabilistic methods. It is suggested that expert systems be provided to assist a junior level engineer in determining input variable distributions, defining the level of accuracy required for a particular analysis and applying the methodology. The second most important area is to establish a set of definitions and analysis standards for probabilistic analysis and then establish a meaningful output format for the reliability data, such as graphics. Finally, this data should be incorporated into a risk management assessment package to determine overall program technical risk.

PROBABILISTIC DESIGN APPLICATIONS

Agenda

- Present day design issues
- STME design objective
- General design approach
- STME applications
- Future applications
- Recommended technology

Figure 1

PRESENT DAY PROPULSION SYSTEM RELIABILITY ISSUES

- **Historical reliabilities of past flight vehicles indicate propulsion system reliabilities limited to 96% (1 failure every 25 launches)**
- **New propulsion systems are demanding higher reliabilities by design**
  - RM 2000 guidelines - "Design it right the 1st time"
    Design incentives/warranties
  - DOD Initiatives - "Assured access into space"
    "Launch on schedule"
  - NLS Program - specifies engine reliability level reliability
    - Demonstrated reliability of 0.99 @ 90% confidence
    - Design to reliability of 0.999

Therefore a new design approach required

Figure 2

PRESENT DAY DESIGN APPROACHES

- **Most baseline designs are selected based on qualitative assessments and deterministic analyses**
- **Traditional analysis techniques cannot quantitatively assess engine reliability during the design process**
- **Reliability assessments usually conducted by test**
  - Circumvents "Design for Reliability" approach
  - 2 to 3 years for components evaluation
  - 4 to 6 years for engine level evaluation

Therefore a new design approach required

Figure 3

STME DESIGN OBJECTIVE

Objective:

- Define a rigorous methodology to improve product reliability where it counts, in the design process

Rationale

- DoD - "Design it right the first time"
- NLS - Develop a robust design
  - Incorporate lessons learned
  - Apply a more rigorous structural criteria
  - Desensitize to external environment
  - Design for defect tolerance
  - Assess reliability using probabilistic analyses
- NLS - Design to reliability of 0.999

} Designing for Reliability

To design and develop a better product

Figure 4

"Designing for Reliability"

Proposed Approach

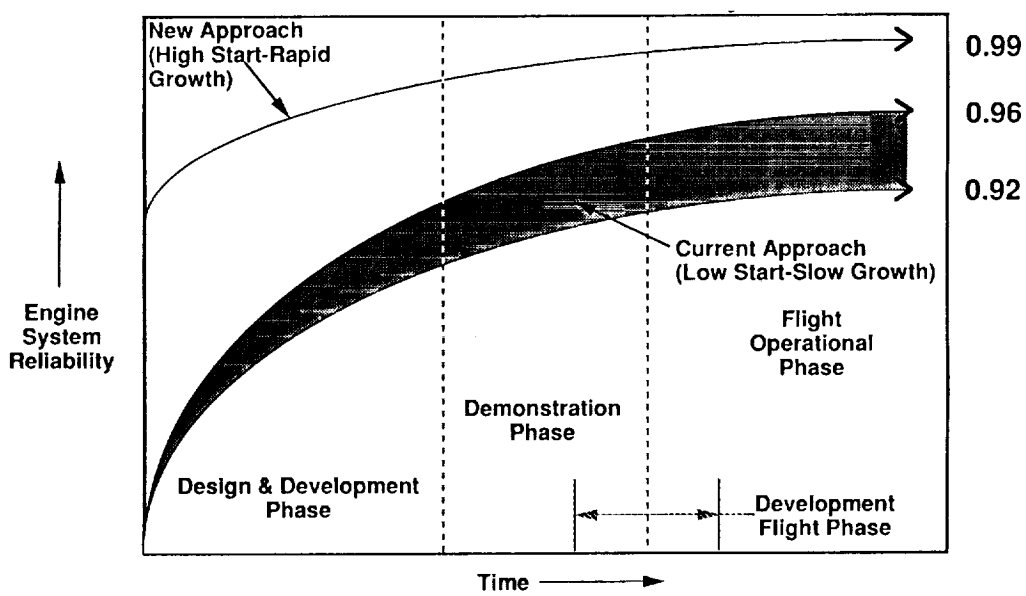


Figure 5

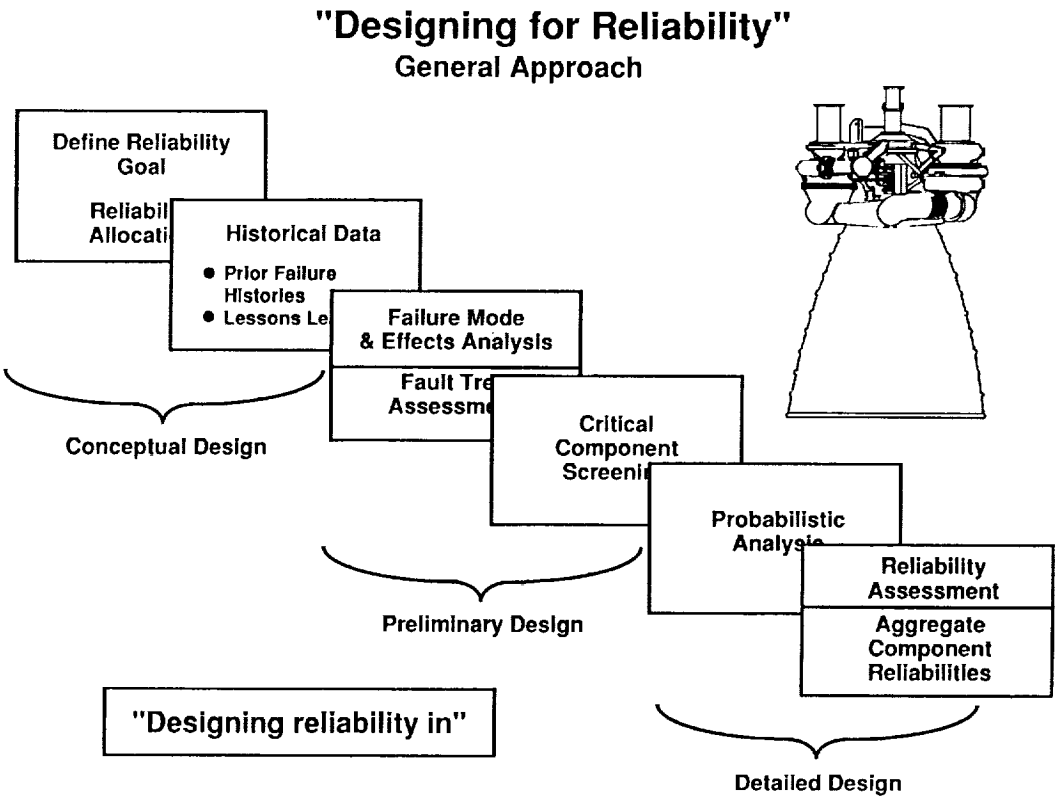


Figure 6

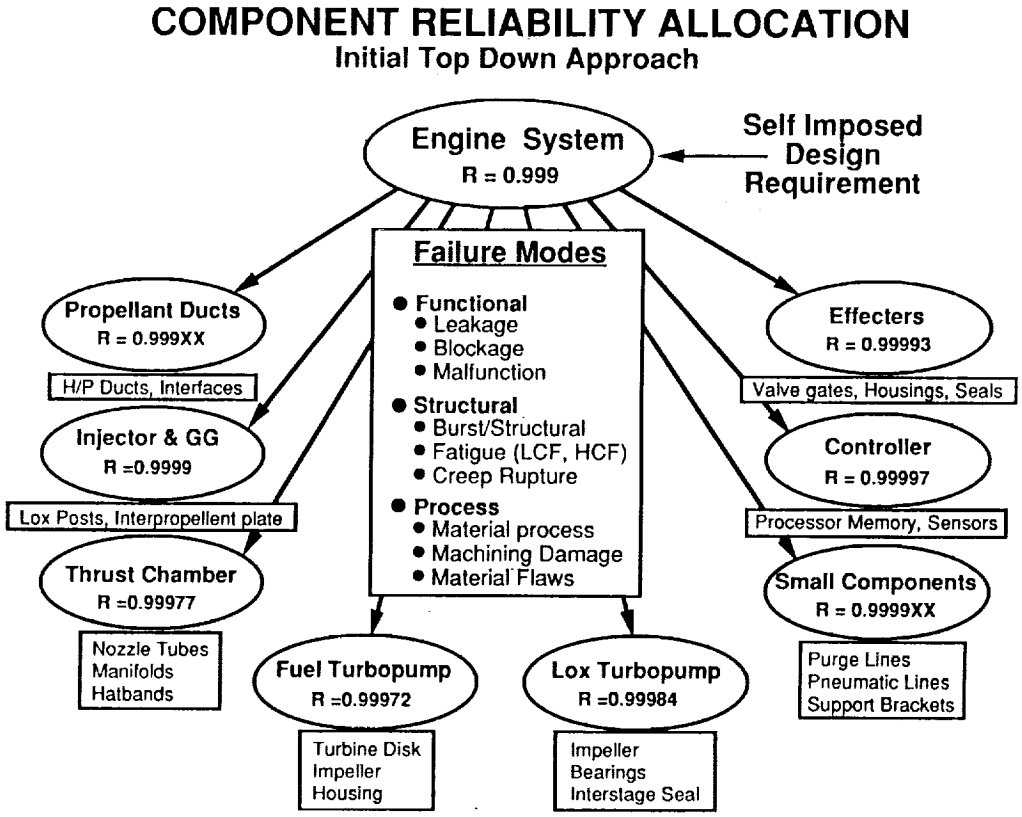


Figure 7

HISTORICAL DATA

- **Utilize established historical database**
  - Flight history failure modes
  - Development failure modes
  - Unsatisfactory condition report (UCR) hardware concerns/issues
  - Generic Material Reviews (MR) issues
  - New design practices

  } Lessons Learned Sources

- **Eliminate or minimize a priori hardware failure through pro-active design**

- **Control of all other failure modes**

Figure 8

FAILURE MODES & EFFECTS ANALYSIS

When, What & How

- **When to conduct FMEA?**
  - As soon as viable design concept is established!
  - Living document - must updated as the design matures
- **What kinds of FMEA's?**
  - Functional
  - Structural

  } Standard

  - Process ← Not usually done , but responsible for 50% of operating failures
- **How to conduct an FMEA?**
  - Establish a failure criticality - Crit 1,2 & 3 ... 1R ... 1S
  - Define worst case failure consequences for each component hardware
  - Categorize via failure criticality
  - Quantify probability of occurrence ← *Most Important Item*

Figure 9

CRITICAL COMPONENT SCREENING

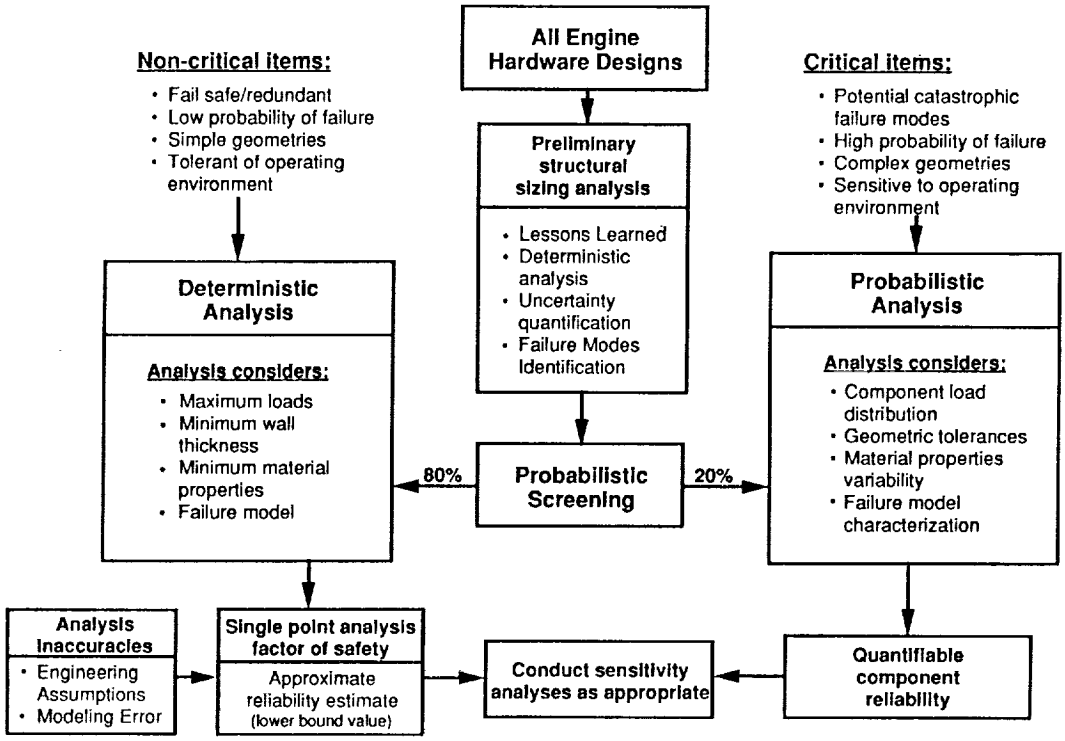
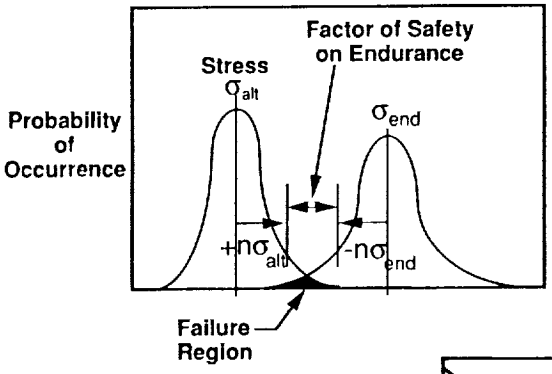


Figure 10

SIMPLIFIED PROBABILISTIC ANALYSIS

Fatigue Strength



Factor of Safety

$$F.S_{(end)} = \frac{Ftu_m - n\sigma_{end}}{S_m - n\sigma_{alt}}$$

n = 2 - 95% Reliability

n = 3 - 99.7% Reliability

Design Hardware to be Insensitive to Damage over Time

- Thicker Walls (lower σ)
- Low Stress Concentrations
- Smooth surface finishes
- Alternating Stresses below σ_{end}

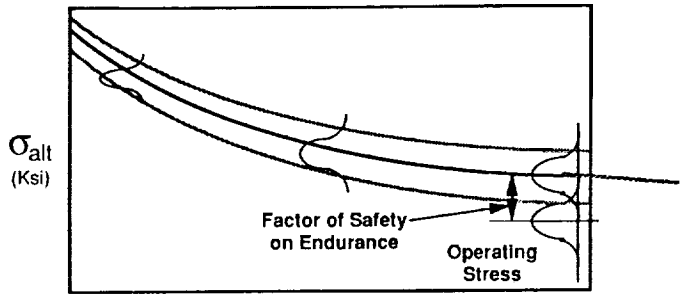
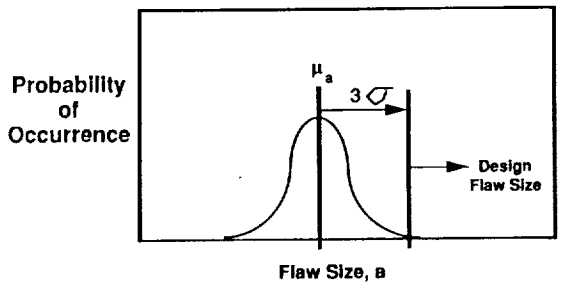


Figure 11

DAMAGE TOLERANT DESIGN APPROACH

Fracture Mechanics

Example: Castings



Characterize Material Flaw Sizes

- Establish a Flaw Size/Distribution Database
  - Collect historical databases - typical castings
  - Fully characterize material properties
  - Conduct 1st article - section castings
  - Section production hardware - random selection

Design Hardware to be Damage/ Flaw Tolerant

- Thicker walls (lower stress)
- Minimize stress concentrations
- Established maximum flaw size
- Stress intensity below ΔK_{th}

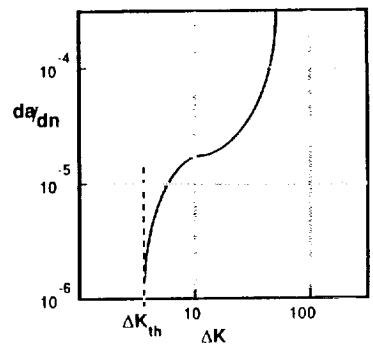


Figure 12

DETAILED PROBABILISTIC ANALYSIS

Most Sophisticated Approach

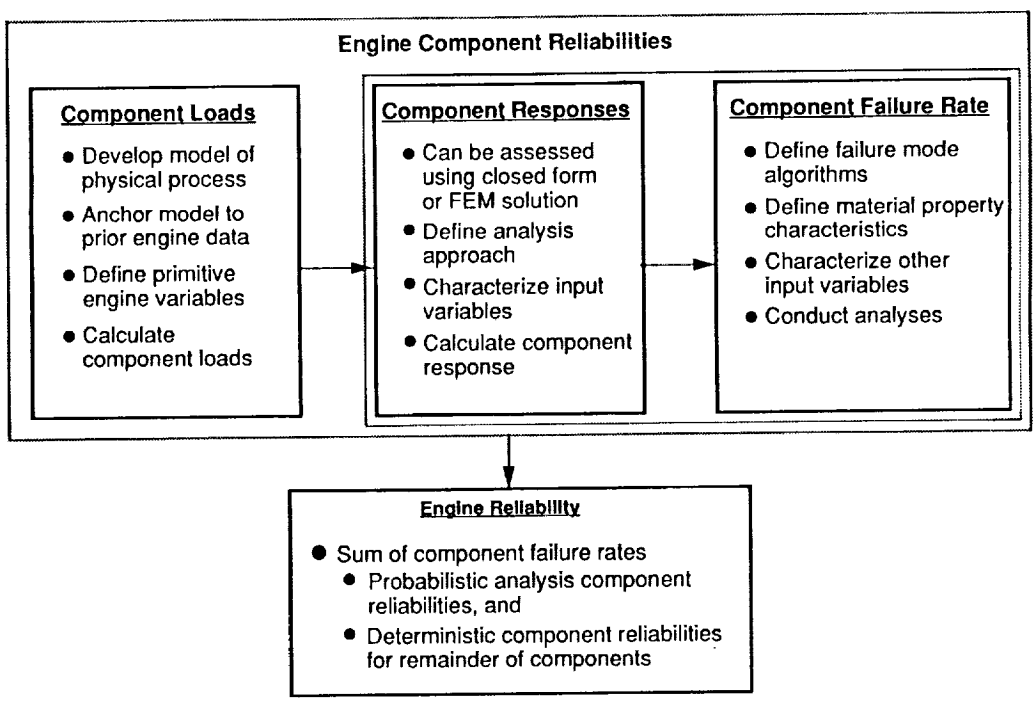


Figure 13

COMPONENT LOADS ANALYSIS

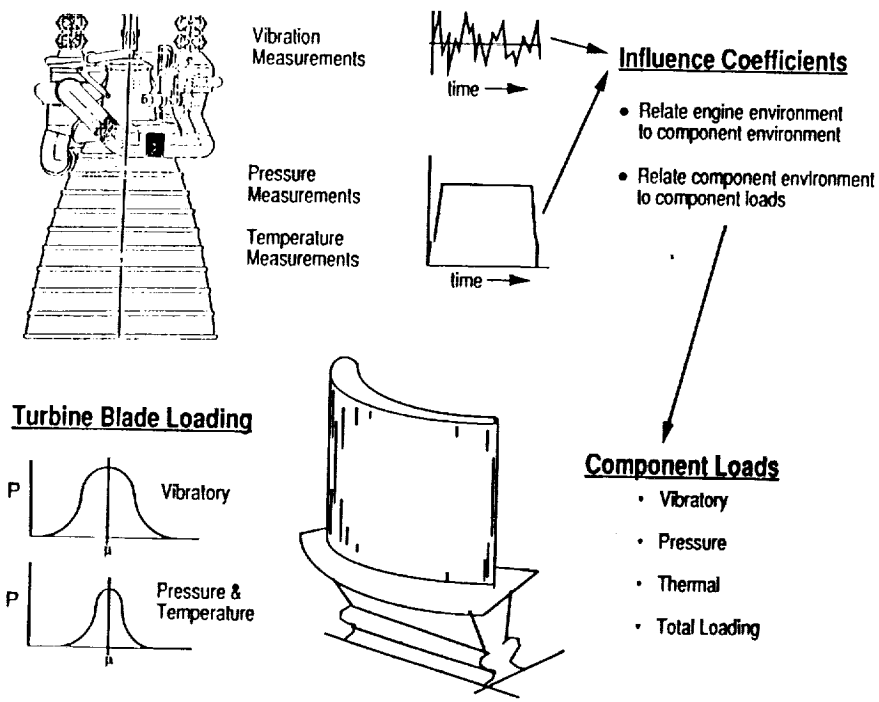


Figure 14

COMPONENT RESPONSE ANALYSIS

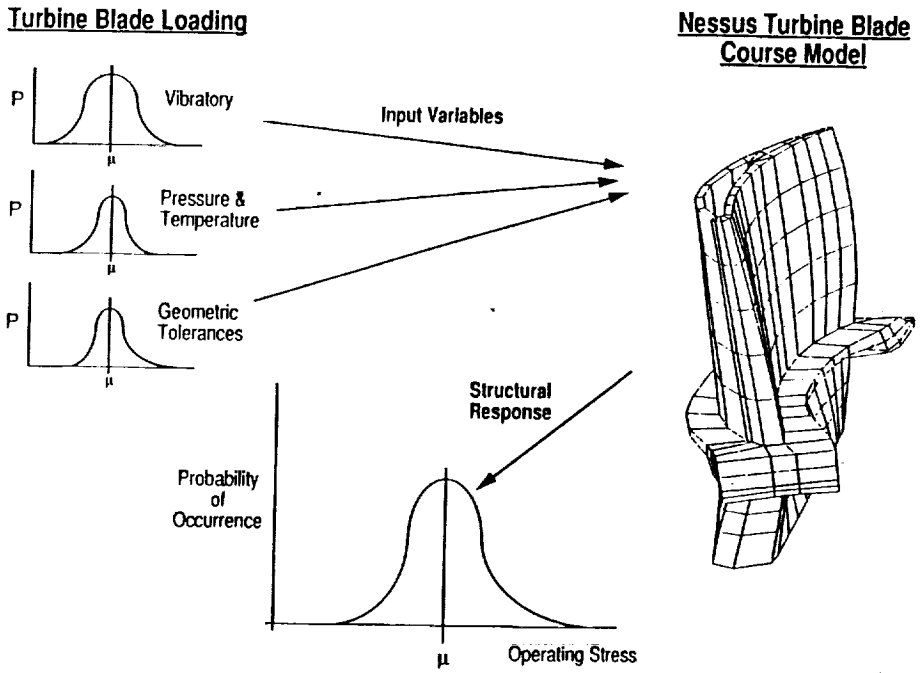


Figure 15

COMPONENT FAILURE RATE

Generic Model Process

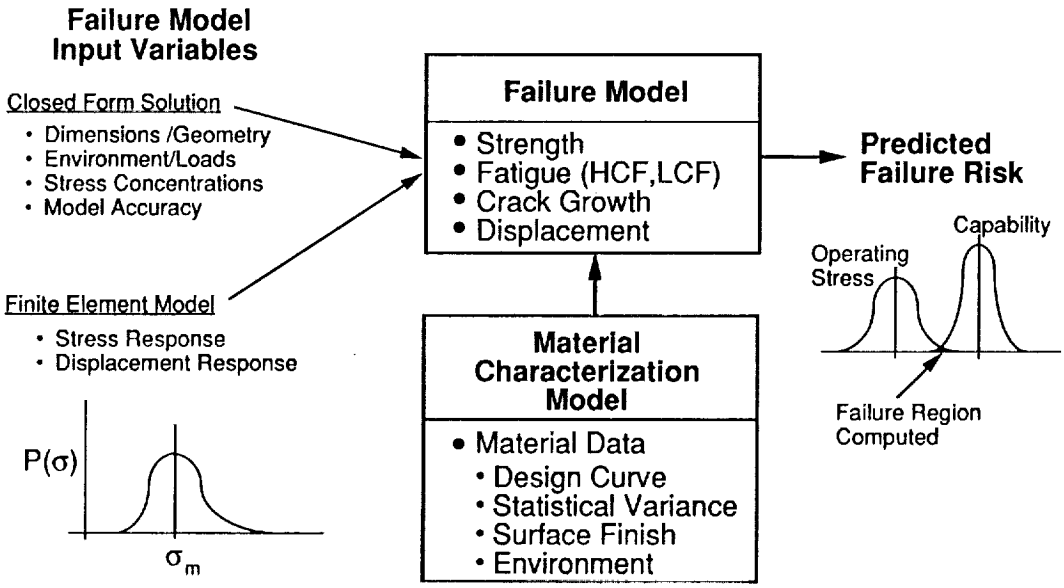


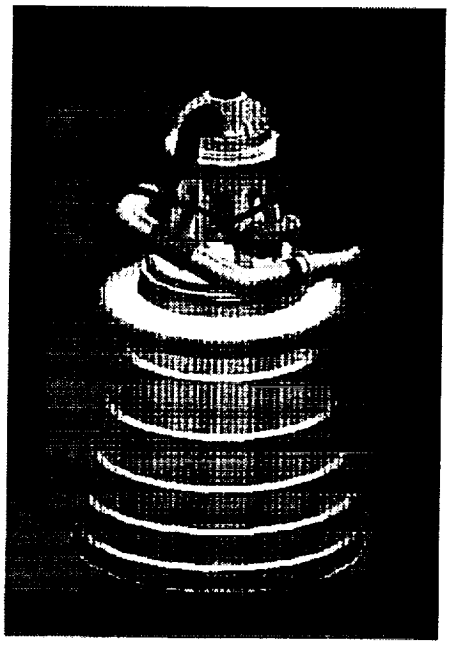
Figure 16

BENEFITS OF PROBABILISTIC APPROACH

- **Quantitative measure of integrity of design and reliability**
- **Uncertainty of each variable is explicitly considered in the analysis process**
- **Most significant design variables (drivers) are ranked in the order of their effect on reliability**
- **Design trade studies can be assessed via reliability**
- **Gaps in the design data are surfaced - program resources can be effectively used to obtain necessary data**

Figure 17

SPACE TRANSPORTATION MAIN ENGINE



Operational Parameters

• Thrust, klb (vac)	583
• Specific impulse, s (vac)	430
• Chamber pressure, psia	2,250
• Engine mixture ratio, o/f	6.0
• Area ratio, ϵ	45:1
• Mission life	10
• Weight, lb	7,900
• Gimbal capability, deg	±10
• Throttling	70%

Design Features

- Gas generator cycle
- Liquid hydrogen/liquid oxygen
- Milled channel chamber/tubular gas cooled nozzle
- 3 stage H_2 T/P-1 stage O_2 T/P
- Tank head start
- Open loop control
- No bleeds
- 152 in. long x 87 in. dia

Figure 18

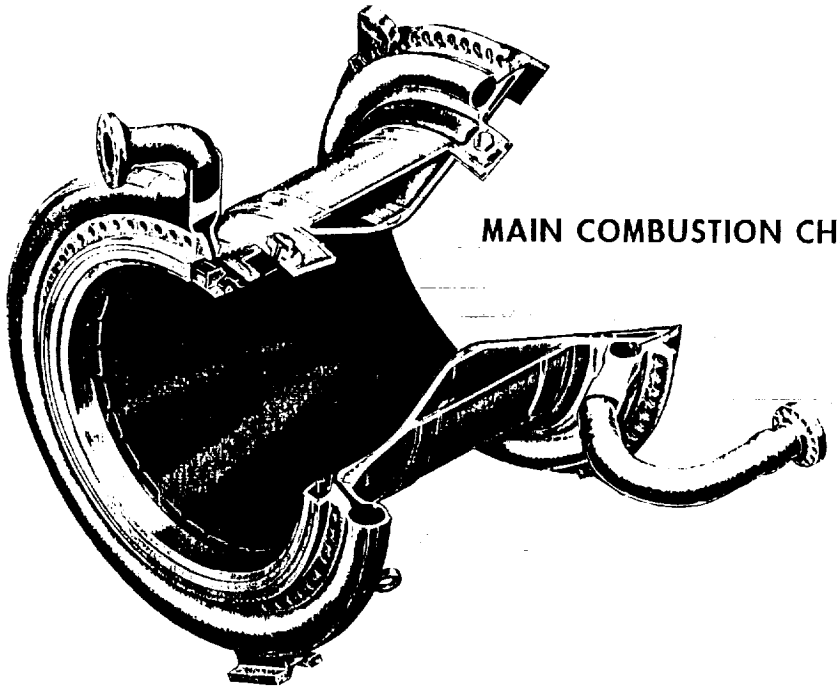
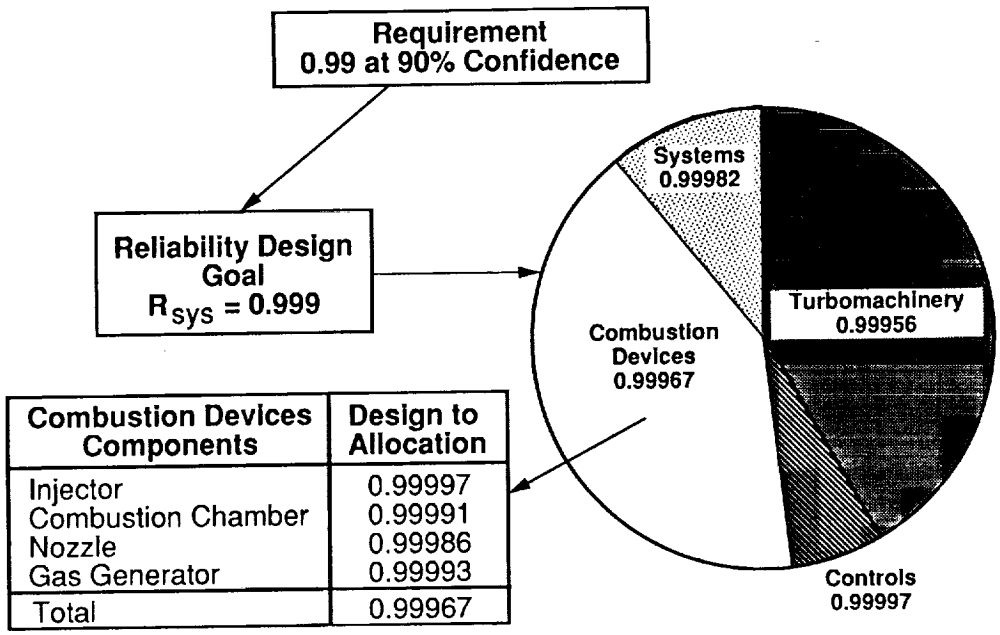


Figure 19

COMPONENT RELIABILITY ALLOCATION

STME Initial Reliability Allocation



Combustion Devices Components	Design to Allocation
Injector	0.99997
Combustion Chamber	0.99991
Nozzle	0.99986
Gas Generator	0.99993
Total	0.99967

Figure 20

STME GAS GENERATOR ENGINE

Initial Reliability Allocation

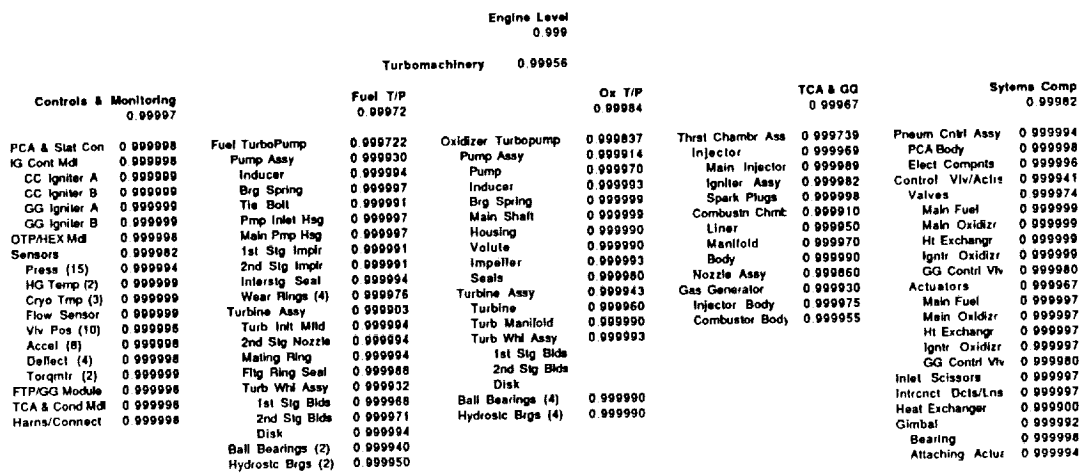


Figure 21

MAIN COMBUSTION CHAMBER

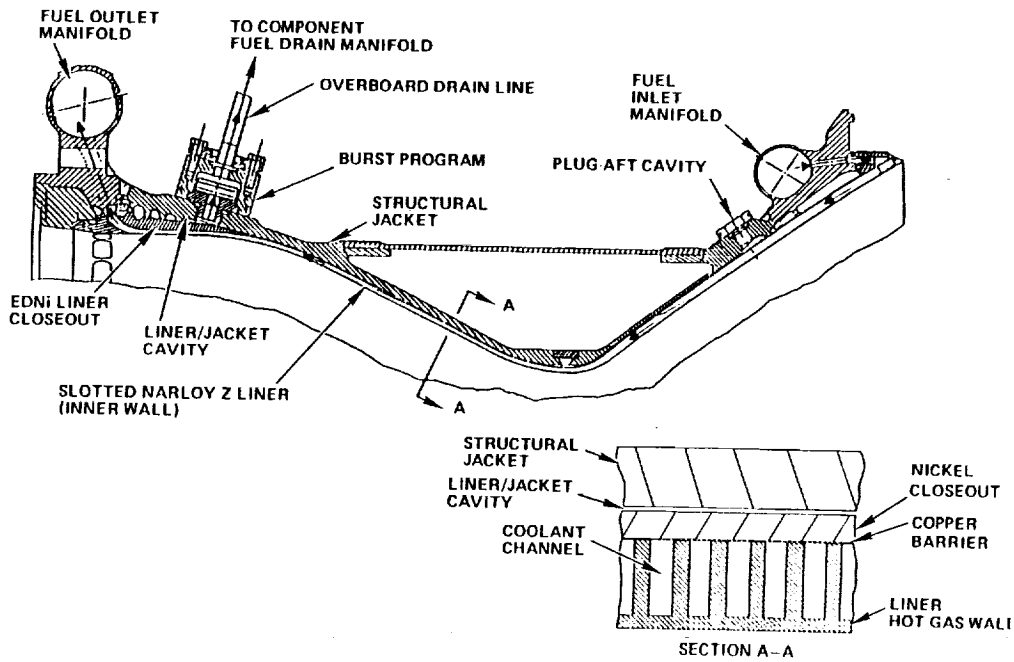


Figure 22

PROBABILISTIC ANALYSIS OF THE MCC LINER

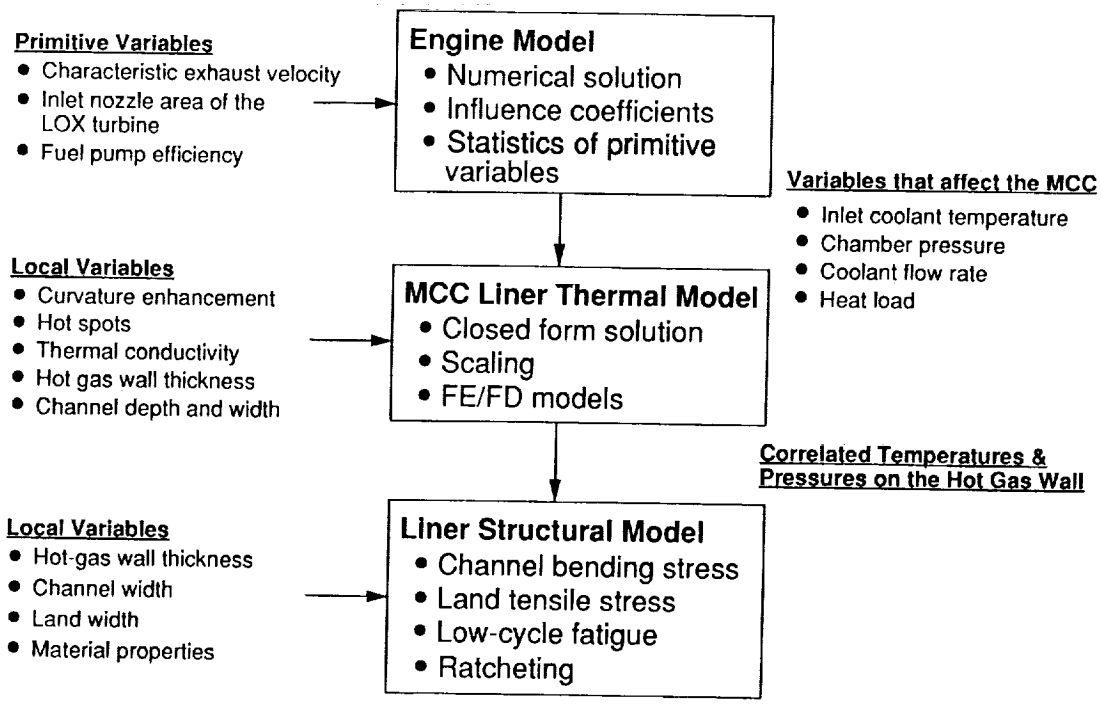


Figure 23

PROBABILISTIC ANALYSIS

Channel Bending Stress

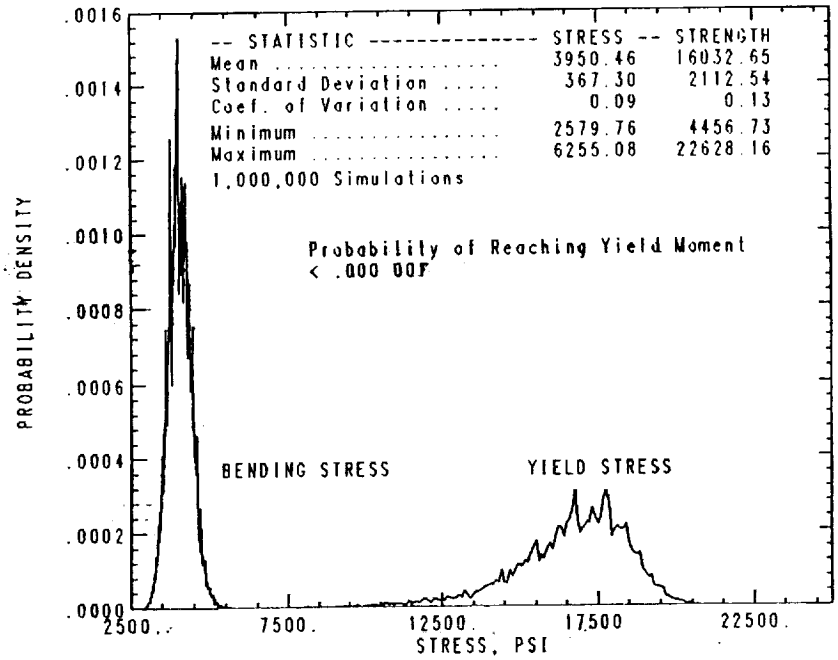


Figure 24

PROBABILISTIC ANALYSIS

Land Tensile Stress

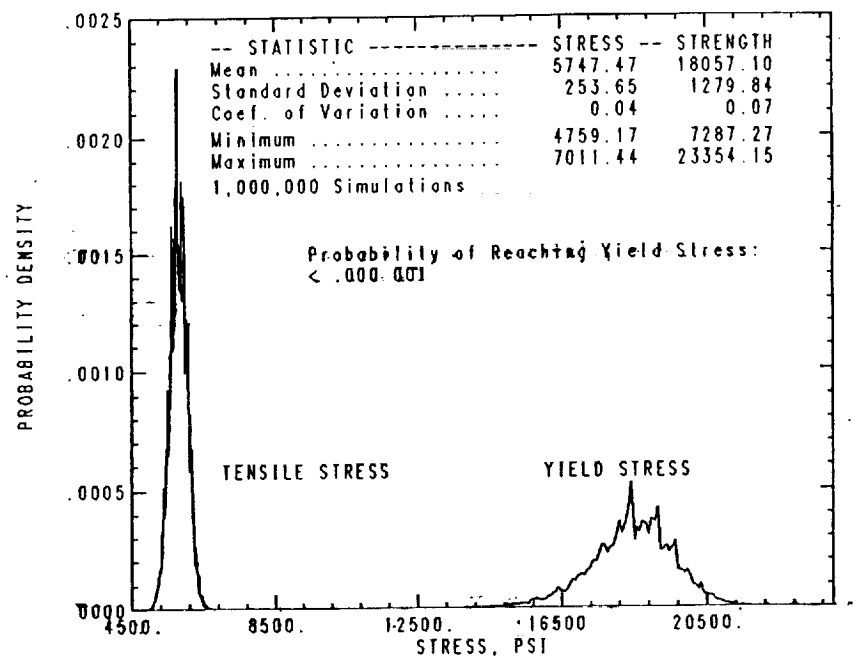


Figure 25

PROBABILISTIC ANALYSIS

Low Cycle Fatigue

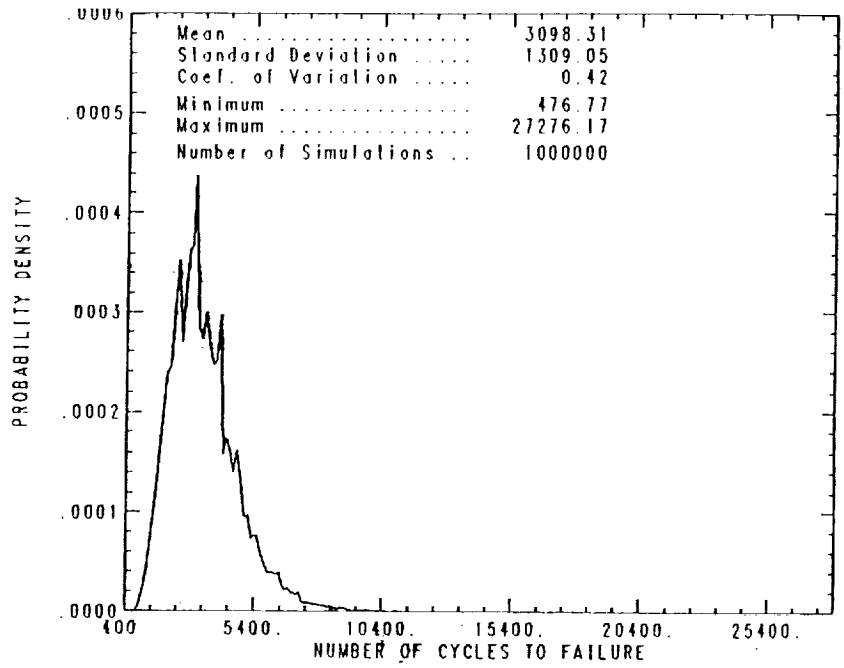


Figure 26

PROBABILISTIC ANALYSIS OF MCC LINER

Ratcheting Failure Rate at a Hot Spot

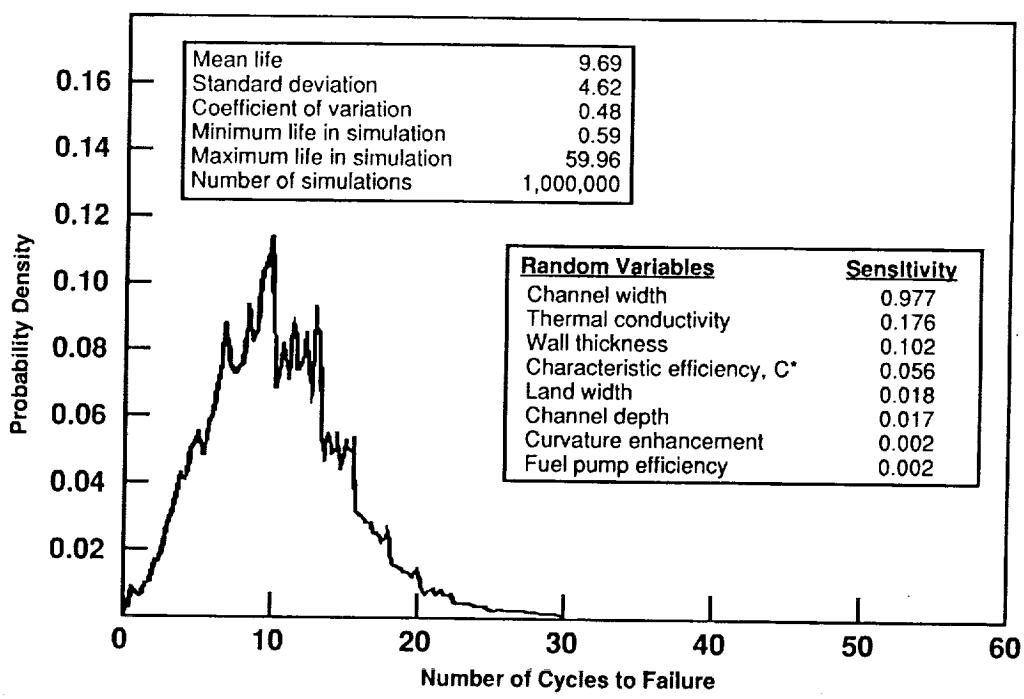


Figure 27

C-6

PROBABILITY DENSITY FUNCTIONS FOR HIGH CYCLE FATIGUE

Main Combustion Chamber AFT Manifold Casting

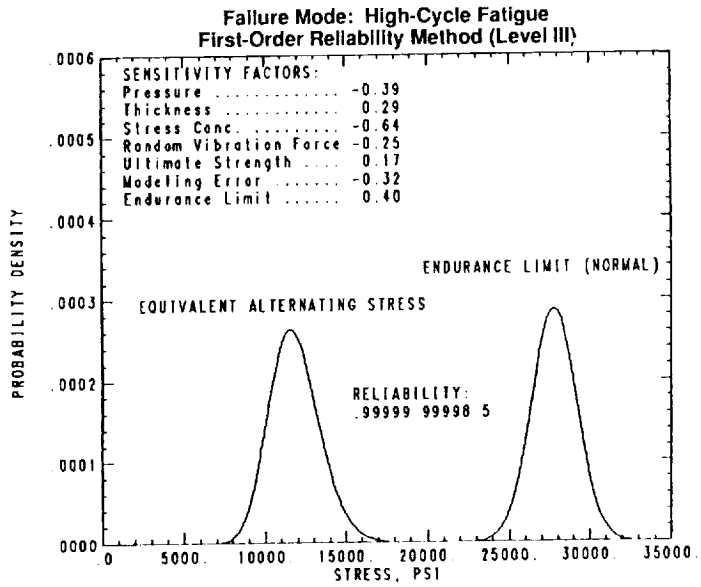
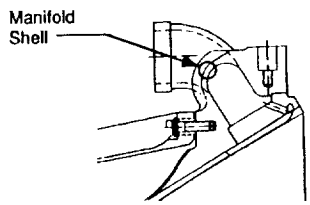


Figure 28

PRELIMINARY RELIABILITY ASSESSMENT USING PROBABILISTIC ANALYSIS

Combustion Chamber Aft Manifold



DETERMINISTIC SAFETY FACTORS

		(Design Reqm't)
Primary Stress:	3.9	(1.5)
High Cycle Fatigue:	2.2	(1.25)
Fracture Mechanics: (ΔK threshold/ΔK dynamic)	1.0	(1.0)

Probabilistic Analysis

Random Variables	Mean	STD Dev
Manifold Pressure	3,600 psi	360 psi
Shell Thickness	0.400 in.	.007 in.
Sideload	184,000 lbs	27,600 lbs
Vibration Load	87,000 lbs	4,350 lbs
Stress Concen. Factor	1.7	0.10
Ultimate Strength	96,700 psi	4,833 psi
Endurance Limit	27,800 psi	1,390 psi

Reliability

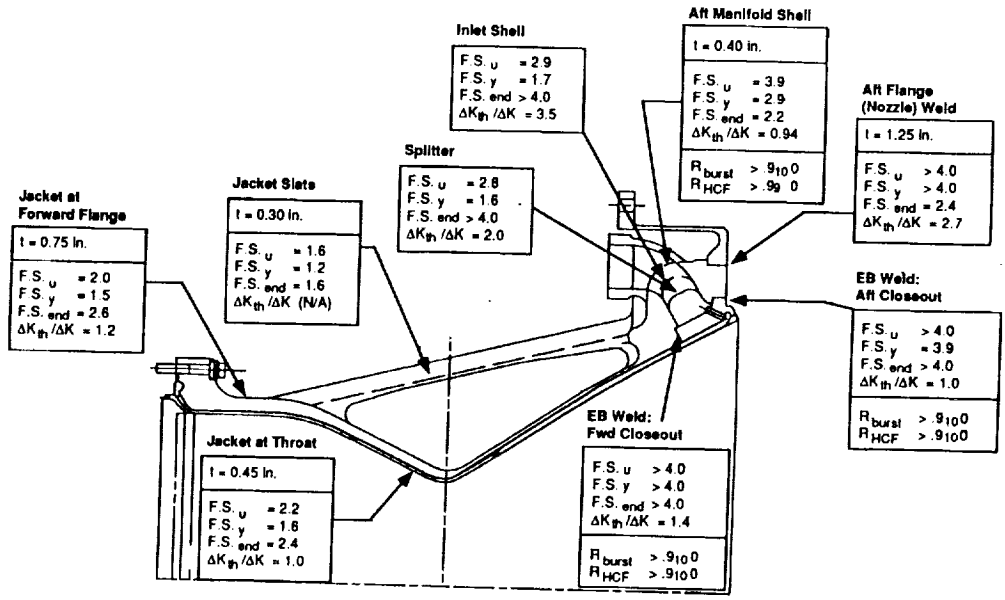
Burst	$R > 0.9_{10}0$
HCF Crack Initiation	$R = 0.9_{9}8$

Chamber Reliability Allocation:

$R = 0.99992$

Figure 29

COMBUSTION CHAMBER SAFETY AND RELIABILITY SUMMARY



Combustion Chamber Reliability Allocation = 0.99992

Figure 30

FUTURE APPLICATIONS*

Hardware/Process	HCF	LCF	Fract. Mech.	Wear	Displ.	Buckling/ Instability
Turbomachinery						
Turbine blades	√		√			
Turbine disk	√	√	√			
Hydrostatic bearings				√	√	
Inducer blades	√		√			
Impeller vanes	√		√			
Combustion Devices						
Injector LOX post	√	√				√
Injector close-out welds		√	√			
Interpropellant plate	√	√	√			
Fuel inlet manifold	√	√	√			
Fuel inlet splitter	√	√				
Manifold to Liner close-out		√	√			
MCC liner		√				√

*** Not complete list**

Figure 31

FUTURE APPLICATIONS*

Hardware/Process	HCF	LCF	Fract. Mech.	Wear	Displ.	Buckling/ Instability
Systems Hardware Interactions						
Fluid Coupling in LOX system	√	√				√
Stability issues	√				√	√
Process Control						
Casting flaw size			√			
Material surface finish	√	√	√			
Welds/braze joints	√	√	√			

*** Not complete list**

Figure 32

RECOMMENDED TECHNOLOGY

- **Simplified probabilistic analysis tools (rapid analysis)**
  - Required early in design process (sizing, trend analysis)
  - Closed-form solutions for estimating variable distributions
  - Simplified probabilistic analyses or estimates from FS solutions
- **Simplified reference guide for conducting PDA**
  - Many engineers are not familiar with process
  - Elimination result misinterpretation (due to lack of knowledge)
- **General purpose probabilistic tool for empirically derived equations**
  - Nonlinear solutions - define by empirical relationships
  - Process control problems - fabrication issues
- **Meaningful graphics display of results**
  - Graphic overlays - solution comparisons as a result of trade studies
  - Carpet plots showing reliability versus selected variables
- **Risk management tools for evaluation of technical risk**

Figure 33

N92-25933

Computational Structural Analysis and Advanced Commercial Engines

R. B. Wilson
Pratt & Whitney

PRECEDING PAGE BLANK NOT FILMED

ENGINE ENVIRONMENT - PRESENT

- Materials - Monolithic TI, NI alloys, limited composites (not primarily structure)
- Engine configurations primarily turbofans
- Limiting Issues
  - Compressor exit temperature (1150 to 1300° F)
  - Turbine inlet temperature (2000 to 3000° F)

The history of computational structural analysis at Pratt & Whitney will be reviewed and anticipated requirements for the design, development and support of advanced commercial engines will be discussed throughout the following paper.

The present commercial engine environment is comprised primarily of turbofan engines containing monolithic titanium and nickel alloys.

ENGINE ENVIRONMENT - FUTURE

- Materials
  - Monolithics with modestly higher capability
  - Composite primary structure
  - Woven and braided composites
  - Metal matrix and ceramic matrix composites in high temperature areas
- Engine Configurations
  - Derivatives of existing turbofans
  - Ducted or unducted propfans (efficiency and 1997 noise requirements)
  - HSCT - "Super" Turbojet
- Key Issues
  - Engine/airframe integration
  - Gear systems
  - Fan integrity
  - Reliance on load sharing

Future commercial engines will make extensive use of composite materials to meet demanding high temperature requirements and aggressive weight goals, since only modestly higher capabilities can be anticipated in monolithic materials. Engine configurations will include ducted and/or unducted propfans and "super" turbojets (HSCT) in addition to turbofans.

BUSINESS ENVIRONMENT

- Intense competition
  - Shorter design cycle
  - Reduced design/development costs
- Increased emphasis on:
  - Safety
  - Reliability
  - Durability
  - Cost
  - Efficiency
  - Environmental impact - emissions and noise

Competitive pressures will require a shorter design/development cycle and continuing reduction in costs. At the same time a variety of economic, durability and environmental issues will assume increasing importance.

DESIGN/ANALYSIS ENVIRONMENT

- Fewer $/Task
- Fewer, less experienced people
- Less time
- Vast increase in computing resources
- Increasingly complex analysis
  - Three-dimensional models required
  - Increasing nonlinear requirements
  - Detailed analysis of assemblies
  - Off-design analysis to assure robust designs
  - Closer fluids/structures coupling
- Sharing of analytical work among joint venture partners
- Increasing reliance on vendors for:
  - Software
  - Design
  - Analysis
  - Testing

Structural design and analysis will need to be accomplished more quickly, with fewer personnel resources. Increasingly complex analysis will be required. The process will benefit from vastly increased computing resources.

ROLE OF ANALYSIS IN ENGINE LIFE CYCLE

A brief historical overview of the role of structural analysis during the commercial engine life cycle is given in the next few figures.

PRE-HISTORY (1968?)

- Mechanical Design
  - Handbook calculations - sometimes computerized
  - One-dimensional analysis
  - Modest use of two-dimensional analysis

- Engine development/support addressed primarily through testing

"THE ONLY SIMULATION FOR AN ENGINE IS AN ENGINE!"

Twenty to twenty-five years ago design issues were addressed primarily through handbook calculations, with very modest use of two-dimensional analysis. Major structural issues arising during engine development and field support were addressed primarily through testing, frequently of full engines.

PRESENT

- Mechanical Design
  - Supported by extensive 2D and 2.5D analysis, and selective 3D analysis, during design cycle
  - Additional 3D analysis for design verification
  - Significant investment of highly trained personnel
- Engine Development/Field Support
  - Analysis drives design changes
  - Testing to provide analysis input and verify changes

"IT'S GOING TO TAKE <u>HOW</u> LONG?"

Presently, mechanical design is supported by extensive analysis, primarily two-dimensional. Development and field support are usually driven by analysis, with testing to provide analysis input and verify changes.

FUTURE (1993-2005)

- Mechanical Design
  - Use of "artificial intelligence" will require hands-off automation of much 2D and 3D analysis
  - Vastly increased 3D requirements
  - Routine use of nonlinear capabilities (2D and 3D)
  - Requirement for concurrent work by diverse organizations
  - Increased problem sizes - driven by automation and closer coupling in engines
  - Vast increase in analysis throughput
    - Design automation
    - Concurrent engineering
    - Evaluation of off-design conditions
- Engine Development/Field Support
  - Mostly 3D analysis needed
  - Frequently nonlinear
  - Rapid turnaround necessary

"YOU WANT ME TO BUY <u>HOW MANY</u> COMPUTERS?"

Future designs will require much more complex analysis types to be used on a routine basis. Shortened design cycles will require increasing automation of portions of the design and analysis process.

ADVANCES REQUIRED

1. Analysis - What problems can be solved?
2. Computing - What resources are available?
3. Analysis Process - What does the user have to do?
4. Data/Information Handling.

Advances in computational structural analysis will be required in several areas in order to support future requirements.

TRANSIENT/DYNAMIC ANALYSIS

- Fan and containment design more exotic - design concepts and materials
- Durability standards more demanding
- Pressure on weight and manufacturing
- Testing very expensive, with long lead times
- Analysis must be:

  - Timely
  - Reliable
  - Usable by non-specialist

- Improvements needed in:

  - Element technology
  - Time integration
  - Material modeling

- HSCT will introduce new problems - i.e., thermal shock

Transient and dynamic analysis will become more important due to economic pressures, the use of exotic materials and the requirements of new engine types.

COUPLED ANALYSES

- Typically CFD, thermal and stress analysis done separately
- Inefficient, error prone process
- When coupled analysis is done, level of individual analyses considerably simplified
- Advanced engines will require some coupled analyses in full detail - for example, thrust reverse transition for a propfan will require full Navier-Stokes sophisticated structural analysis.

Many problems in future engines will require coupling of structural analysis with thermal and/or computational fluid dynamics analysis.

MATERIAL MODELS

- Further refinement of state variable models for monolithic materials - cover entire range of thermal/mechanical loading
- Increased complexity of "traditional composites" - woven and braided fiber configurations
- Metal and ceramic matrix composites - material model must recognize environmental effects
- Stress analysis and damage accumulation become a coupled problem

Material modeling requirements, and the interfacing of these models with stress analysis programs will become vastly more complex. High temperature requirements will require further development of state variable models and computationally effective material models will be required for a variety of traditional and exotic composite materials.

OTHER AREAS

- Contact analysis
- Improved triangular and tetrahedral elements
- Quality assurance for analysis results
- Rezoning/remeshing/substructuring
- Hybrid (FEM/BEM) methods
- Sensitivity analysis
- Coupled stress/fracture mechanics

Advances in many other areas will also be required, especially to support timely, complex analysis by relatively inexperienced personnel.

COMPUTING CAPABILITY

Improvements in available computing capability will support increased analysis demands.

- All predictions are too conservative.
- All available capability will be utilized.
- Networked workstations of increasing power for pre-/post-processing and analysis
- Compute Servers
  - "Traditional" supercomputers
  - Massively parallel systems
  - Use of workstation networks as a parallel system
- Computer development is ahead of analytical software

Computational structural analysis will depend on networks including capability from workstations through supercomputers functioning as compute servers. In general, structural analysis software lags behind current hardware capabilities.

ANALYSIS PROCESS

The analysis process is driven by design/development requirements.

- Only three time scales matter
  - Impact the design process
  - Verify design while low cost changes possible
  - Respond quickly to development/field problems
- Processes must be usable by "ordinary" engineers and technicians

Analytical tools must be usable by line technicians and engineers. Analysis results must be available in a timely manner in order to have impact on design and development.

REQUIREMENTS

- Close coupling of stress pre-processors with CAD tools
- (Close to) automated meshing
- Order of magnitude improvement in load and boundary condition definition
- Simple geometry changes should be simple to evaluate!

Major improvements are needed to allow the more complex analyses required for future engines to impact the design process.

CONCLUSIONS

- Vastly more analysis will be required.
- More complex analysis will be required.
- Analytical development required
  - Material models
  - Dynamics
  - Nonlinear problems
- Analysis process improvements equally important.

Real challenges exist for all communities involved in the development and use of structural analysis tools - government, software developers and industrial users.

N92-25934

Military Engine Computational Structures Technology

Daniel E. Thomson
WL/POTC
Wright Patterson Air Force Base, Ohio

Agenda

IHPTET overview

Codes we use now

Codes we are developing

The Future

Summary

Integrated High Performance Turbine Engine Technology Initiative (IHPTET)

Goal: Double turbine engine propulsion capability by the year 2003

50% of goal will come from advanced materials and structures, the other 50% will come from increasing performance

▶ IHPTET's Effect on Computational Structures

- Advanced Materials
  - Ceramics
  - Intermetallics
  - Single crystal castings
- Composites
  - Organic Matrix Composites (OMC)
  - Metal Matrix Composites (MMC)
  - Ceramic Matrix Composites (CMC)
- Higher Temperature Conditions

▶ IHPTET's Effect on Computational Structures (continued)

- Advanced Tactical Fighter Service Life Requirements
  - 4000 hours/8000 TAC cycles - cold section
  - 2000 hours/4000 TAC cycles - hot section
- Advanced Structural Designs
  - Integrated structures/materials design
  - Integrated structures/aero design

▶ Codes We Use Now

MSC NASTRAN

NOSAP-M

X3D

UDHeat

CRACKS 90

PATRAN

▶ NOSAP-M

Soft body (birds, ice) impact analysis

Developed by General Electric under Air Force Contract

Scaled-down version of ADINA

- **20 node brick element**
- **Uses water jet to model impact**
- **Implicit time integration**

Orthotropic material capability

X3D

Nonlinear dynamics and impact analysis

Developed by University of Dayton Research Institute under Flight Dynamics Directorate contract

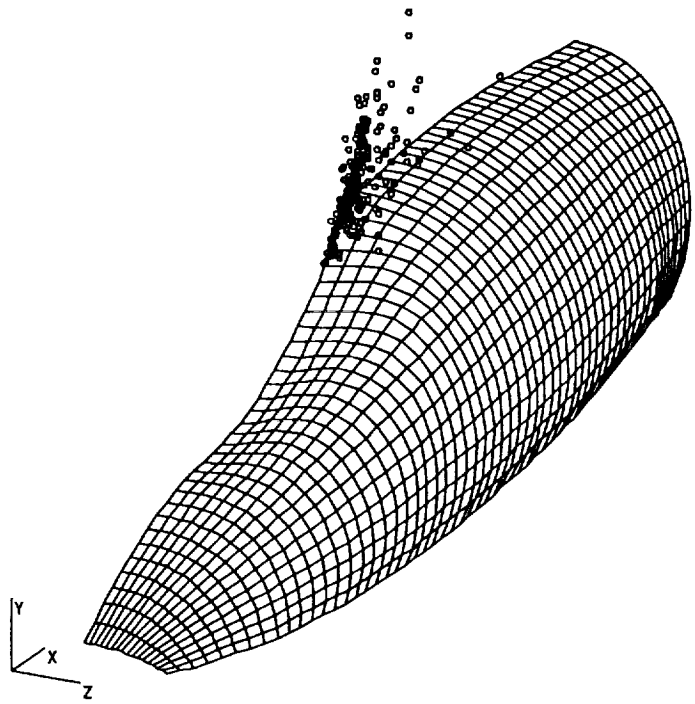
Explicit time integration

Model both the structure and the impactor

Includes solid and shell elements

Fiber failure and delamination analysis

Currently performing code verification



F-16 BIRD IMPACT AT F.S. 112, SYMMETRIC, 350 KNOTS
DISPLACEMENTS FOR TIME STEP = 1731 / TIME = 6.0000001E-03
05-JUN-91 04:27:38
F16BE - 0.5 INCH - POLYCARBONATE: SY=7140 / SU=16000

▶ UDHeat

Explicit code (finite element in space, finite difference in time) for steady state and transient conductive heat transfer

Boundary Conditions

- Convective
- Surface or volume heat flux
- Heat sources and sinks (applied to nodes)

2D elements

- Infinite in one direction
- Axisymmetric

3D elements

Orthotropic material capability

▶ CRACKS 90

Preprocessing

- Material database and plotting
- Spectrum profile graphs
- Stress intensity factor generation and plotting
- Generation of analysis input file
- Residual stress table generation

▶ CRACKS 90

Analysis

- Residual strength
- Crack growth life
- Residual stress
- Load interaction
- Crack growth rates
- Comprehensive tabular output
- Brief summary of results

Postprocessing

- Single or multi-curve growth rate plots
- Single or multi-curve life plots
- Residual strength plots

▶ Codes We Are Developing

Blade Life Analysis and Design Evaluation (BLADE)

Engine Structural Analysis Consultant (ENSAC)

CRACKS xx

Probabilistics

▶ Background (BLADE-ST)

Stress Technology Inc (STI) has worked under the sponsorship of the Electric Power Research Institute (EPRI) since 1980

Developed BLADE-ST

- **menu driven**
- **contains blade preprocessing library of over 40 generic blade and root types**
- **contains materials data base**
- **computes steady stresses, dynamic stresses, natural frequencies, mode shapes, and performs a life analysis**

▶ Recent Accomplishments

Integrally bladed rotor analysis

F110 high pressure turbine blade analysis

Blade Life Analysis and Design Evaluation for Gas Turbine Engines (BLADE-GT)

OBJECTIVE: To develop a user-friendly, "start-to-finish" finite element solver

FEATURES:

- User friendly geometry input
- Automated mesh generation
- Automated boundary condition generation
- Static and dynamic finite element solver
- Forced vibration solver
- Heat transfer and thermal stress solver
- Life analysis

Funding for BLADE-GT

Small Business Innovative Research (SBIR)

Jointly funded from four AF organizations

- Aero Propulsion and Power Directorate
- Engine System Program Office (SPO)
- B2 SPO
- Arnold Engineering Development Center

Engine Structural Analysis Consultant (ENSAC)

- Application Characterization

GOAL: Build an expert system that advises engineers in the use of general purpose structural analysis codes

USERS: Engineers with some background in structural analysis, but limited practical experience

▶ Application Overview

Preparation of input data files for general purpose finite element structural analysis programs

Find applicable and reliable material properties

Complex combination of software capabilities, hardware capabilities, and CPU costs

Analysis model must simulate correct physical behavior, preserve desired accuracy, and minimize CPU costs

A similar capability currently exists in an expert system called Structural Analysis Consultant (SACON)

▶ Status

Selected ART-IM as programming language

- pull down menus
- dialog boxes
- windowing

Preliminary demo due 1 July 1991

- prototype
- partial capability

▶ CRACKS xx

OBJECTIVE: Extend the capabilitiy of CRACKS90 to include crack growth in composite systems

Completed:

- An influence function-based method was developed to calculate the stress intensity factor as a function of crack length for either an edge crack or a surface crack in bimaterial construction

In Progress:

- Models of several additional damage accumulation modes are being developed
  - Interface crack
  - Delamination growth
  - Crack-to-delamination transition

Probabilistic Design System Development

Probabilistic Methodology

- Conventional, deterministic design uses single values for material properties, manufacturing tolerances, mission usage, and other parameters
  - Safety factors are applied to account for variability
  - Method is inherently conservative
- Probabilistic design makes full use of the known statistical distribution of all parameters, and combines them to produce a statistical prediction of the result
  - Conservatism is reduced to known, acceptable level
  - Design can be optimized for weight, performance, life, or any selected criterion

▶ Probabilistic Design System Development

Probabilistic Rotor Design System (PRODS) Program with General Electric (GE)

Started September 1990

Similar program was also started with P&W but was terminated due to lack of funding, in February 1991

Both GE and P&W have very large IR&D efforts in probabilistic method development

Anticipate one or more proposals on probabilistic design later this summer

Other engine companies also show interest in probabilistics

▶ GE PRODS Program

OBJECTIVE: Develop, validate and apply a probabilistic rotor design system methodology

APPROACH: Six phase program

- Data acquisition
- Method Development
- Validation
- Application
- Application Test
- Method Extension

PAYOFF: Improved design capability in advanced materials; predicted safety; and weight savings

FUNDING: $1.8 million, over 4 years

▶ Probabilistics Summary

Probabilistic design is real

Improved computational power makes it feasible

More knowledge of the variability of material properties, behavior, usage, and manufacturing effects is necessary

Efficient usage of composite materials makes it essential

But acceptance of the methodology will take time

- must be demonstrated, validated and applied
- must be sold as a design tool

▶ The Future

Improved composite analysis methods

AI Programming/Expert Systems

- Composite Life Prediction
- User Friendly Systems

Animation/Simulation of Dynamic Phenomena

- Crude animation of X3D results has revealed phenomena unseen in "snap-shots"

Structural optimization for composite components

Summary

- IHPTET goals require a strong analytical base
- Effective analysis of composite materials is critical
  - Life Analysis
  - Structural Optimization
- Accurate life prediction for all material systems is critical
- User friendly systems are desirable
- Post processing of results is very important

N92-25935

Computational Structures Technology Engine/Airframe Coupling

Bruce C. McClintick
General Electric Company
Cincinnati, Ohio

COMPUTATIONAL STRUCTURES TECHNOLOGY
ENGINE / AIRFRAME COUPLING
BC M^C^CLINTICK

There are many aspects of engine / airframe coupling related to structures. My personal experience, for the past 14 years, has been with the generation and analysis of full 3–dimensional engine structures with regard to the design and operation of the engine under static load conditions. However, several problems arose about 6 years ago which required the utilization of the 3–dimensional models under dynamic loads to more fully understand what was happening. Since then, full 3–dimensional engine structural models have been used on a regular basis to help understand dynamics related problems. One such problem, which this paper addresses, is engine related aircraft vibration.

Engine related aircraft vibration is noise within the fuselage of the aircraft which can be both felt and heard. This noise is generally caused by rotor imbalance and is transmitted from the engine through the structural components of the aircraft. Also, the primary cause of noise is from the lower speed rotor, or fan rotor, as this rotor more nearly matches frequencies of the aircraft system. Many of the problems encountered have also been non–linear and observed only during certain flight envelopes. Engine mounts becoming unloaded, bearings becoming unloaded, variations in both mount and bearing stiffness, and the types of bearings used play major roles in the transmission of vibration. To effectively predict engine related aircraft vibration the entire engine/airframe must be treated as a system.

ENGINE RELATED AIRCRAFT VIBRATION

- PERCEIVED NOISE IS BOTH FELT AND HEARD
- CAUSED BY ROTOR IMBALANCE
  - PRIMARILY FAN ROTOR
    - LARGEST ROTOR
    - LOWEST SPEED ROTOR
- CAN BE NON–LINEAR IN NATURE
  - BEARING TYPE
  - BEARING STIFFNESS
  - MOUNT STIFFNESS
- ROTORS ⇒ FRAMES ⇒ MOUNTS ⇒ STRUT ⇒ WING ⇒ FUSELAGE ⇒ CABIN/COCKPIT ⇒ PASSENGERS/CREW
  - MUST BE TREATED AS A SYSTEM

Historically the engine has always been treated as a necessary evil to 'make the airplane go' and little emphasis was given to the engine other than making sure it stayed on the wing. The engine manufacturers sized the engine for thrust, designed for ultimate loads, minimized the affects of rotor modes and provided a reasonable balance at factory acceptance. The main point is not to over simplify the aircraft and engine design, which is indeed quite complex, but to point out that little was done in the design of the system to reduce or understand noise within the fuselage.

HISTORICAL

- ENGINE / STRUT / WING SYSTEM
  - ENGINE A NECESSARY EVIL
  - FLUTTER STABILITY
  - ULTIMATE LOADS STRENGTH
  - LIVE WITH RESULTS
- ENGINE
  - SIZED FOR THRUST
    - LIKE ENTROPY – REQUIREMENT ALWAYS INCREASES
  - ULTIMATE LOADS STRENGTH
  - ENGINE MODES OUT OF ROTOR OPERATING RANGE
    - NOT AT NORMAL OPERATING SPEEDS
    - SIMPLE DYNAMIC ANALYSES UTILIZED
    - RELIANCE ON ENGINE TEST DATA
  - REASONABLE BALANCE AT FACTORY ACCEPTANCE

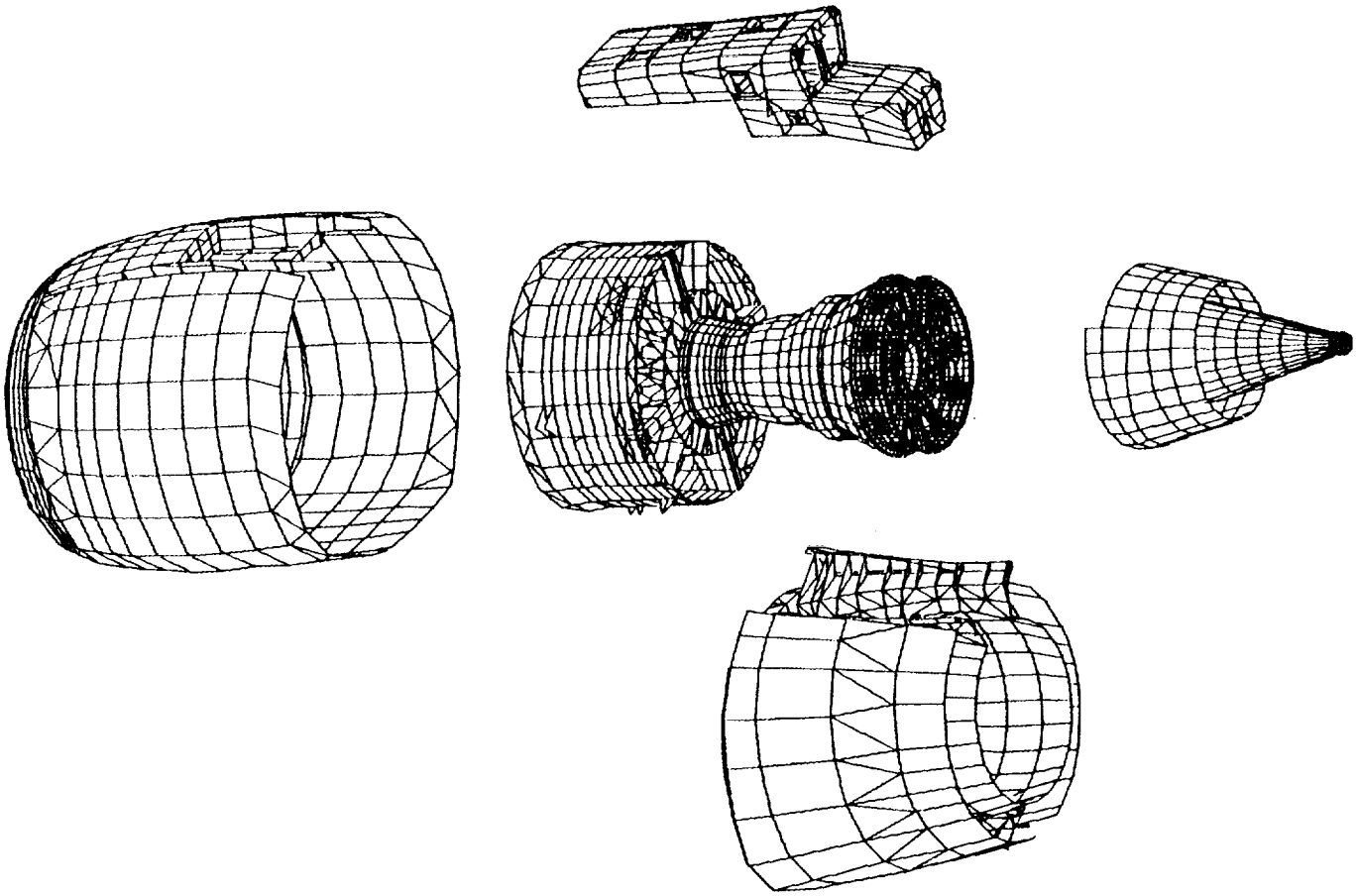
As new aircraft were developed it became more apparent that both the passengers and crew were less tolerant to noise and to maintain sales it was imperative to have a quiet, vibration free cockpit and cabin. Thus, tighter vibration limits were imposed on engine manufacturers and on–wing trim balance procedures were the norm. Also, as an aid to determining the cause of some of the more troublesome vibration problems, 3–dimensional finite element models, previously used only for static analyses, began to be utilized for dynamics. This marked the beginning of aircraft/engine design for structural borne noise.

EARLY 80'S

- PASSENGERS/CREW LESS TOLERANT TO PERCEIVED VIBRATION
  - SMOOTH AND QUIET REQUIRED TO SELL AIRPLANES
  - TIGHTER PRODUCTION VIBRATION LIMITS
  - ON–WING TRIM BALANCE PROCEDURES
  - AFTER MARKET STRUCTURAL CHANGES TO 'SYSTEM'
- 3–D FINITE ELEMENT MODELS
  - ENGINE / STRUT PRIMARILY FOR STATIC LOADS
  - LARGE, COMPLEX MODELS (100,000 – 200,000 DOF)
  - USED ONLY FOR SPECIAL PROBLEMS
  - SPECIAL ROUTINES WRITTEN FOR ROTOR DYNAMICS
- BEGINNING OF DESIGN FOR STRUCTURAL BORNE NOISE

Large, complex engine structural models used for static loads and deflections were the first models used for studying engine related aircraft vibration.

TYPICAL ENGINE STRUCTURAL MODEL FOR STATICS



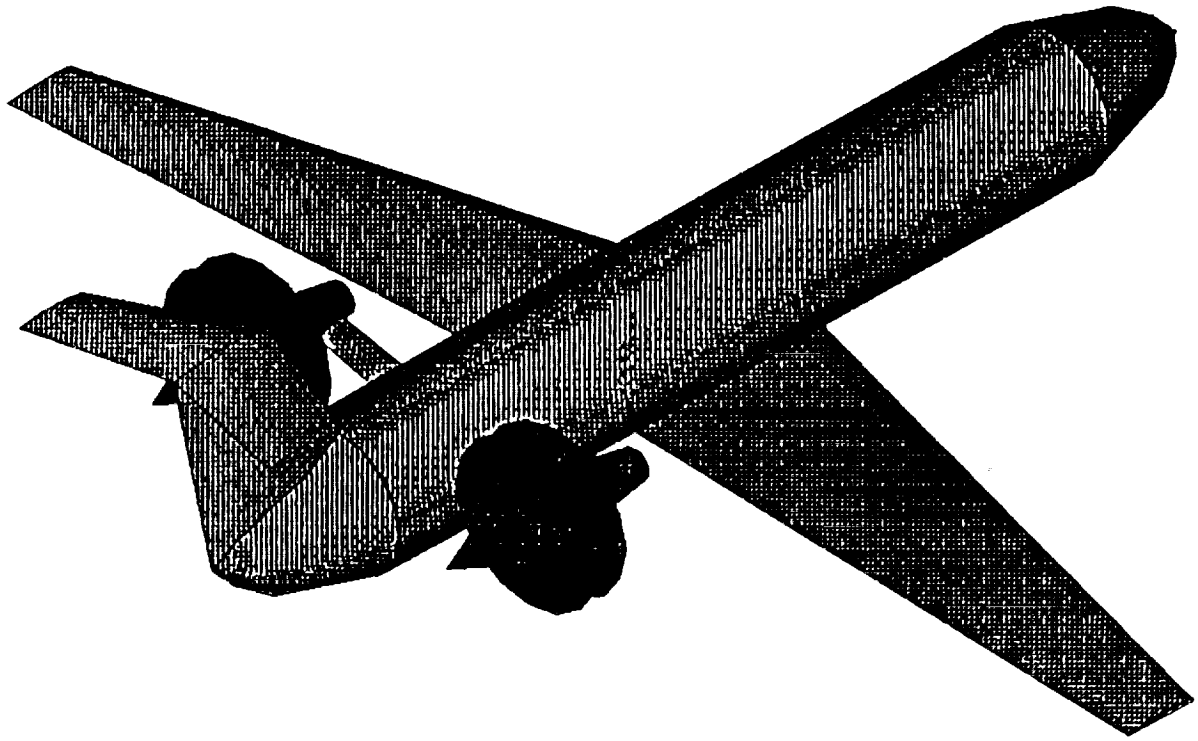
New aircraft development in the later 1980's saw the integration of the engine and airframe as a system in attempts to predict the response of the aircraft cabin and cockpit to rotor imbalance. Aircraft / engine teams were formed to develop the necessary methodology to analyze, test and modify the system design to decrease structurally transmitted noise from the engine.

LATE 80'S

- GE UNDUCTED FAN – UDF®
  - ENGINE RELATED AIRCRAFT VIBRATION
    - SIGNIFICANT DESIGN PARAMETER
    - ENGINE / AIRCRAFT TREATED AS A SYSTEM
  - ENGINE MOUNT DESIGN
    - BLADE OUT LOADS
    - AND – VIBRATION TRANSMISSION REDUCTION
  - 'ACTIVE' VIBRATION CONTROL

GE's Unducted Fan program utilized a full airplane/engine structural model to generate vibration related data throughout the airframe. Thus, cabin and cockpit vibration was analytically monitored for various design changes.

GE UNDUCTED FAN – UDF® DUAL ENGINE DYNAMIC ANALYSIS



Engine vibration related noise is now included in engine specifications. Thus, it is now necessary to include engine vibration transmission as a design parameter and develop the necessary methodology to predict noise within the aircraft before designs become 'fixed' and expensive to modify and change. Part of this methodology development is the correlation of the 3–dimensional finite element dynamic models with both engine testing and aircraft flight testing. Modelling techniques also need to be developed which address the specific needs for dynamic analyses and vibration transmission.

THE 90'S

- ENGINE SPECIFICATION – ENGINE VIBRATION RELATED NOISE
  - DESIGN PARAMETER – ENGINE VIBRATION TRANSMISSION
  - ENGINE MOUNTS DESIGNED TO REDUCE VIBRATION TRANSMISSION
  - ENGINE / AIRCRAFT TREATED AS A SYSTEM
  - HARDWARE DESIGNED AND BUILT TO REDUCE NOISE
- 3–D FINITE ELEMENT MODELS
  - ENGINE TESTS – PLANNED AND DESIGNED TO CORRELATE ENGINE MODELS WITH ACTUAL HARDWARE
  - FLIGHT TESTS – PLANNED AND DESIGNED TO CORRELATE SYSTEM MODEL PREDICTIONS TO TEST DATA
  - MODELLING TECHNIQUES USED SPECIFICALLY FOR DYNAMICS AND VIBRATION TRANSMISSION

The complex, many DOF static structural model is still utilized for dynamic analyses; however, each of the structural components are reduced to a more manageable number of DOF's using dynamic reduction. The dynamic reduction reduces the components to its boundary DOF plus a limited number of DOF used to describe its vibratory modes. Also, many routines have been written to include such items as rotor gyroscopic stiffening, structural and viscous damping, non–linear stiffness and damping versus load and/or frequency and non–linear rub springs.

CURRENT ANALYSIS TECHNIQUES

- 3–DIMENSIONAL STRUCTURAL SYSTEM MODEL
  - COMPLEX MODEL – ALSO USED FOR STATICS
  - REDUCTION OF COMPONENTS TO DYNAMIC EQUIVALENCY
    - RETAIN OVERALL COMPONENT FLEXIBILITIES
    - RETAIN LOCAL INTERNAL FLEXIBILITIES
    - RETAIN ALL INTERNAL IMPORTANT MODES
    - SIGNIFICANT REDUCTION IN DOF
  - ROTOR GYROSCOPIC STIFFENING TERMS
  - BEARINGS – NON–LINEAR
    - STIFFNESS VS ROTOR LOADING
    - STIFFNESS / DAMPING VS FREQUENCY
    - STIFFNESS / DAMPING VS BEARING LOAD
  - STRUCTURAL DAMPING
  - ENGINE MOUNT STIFFNESS VS LOAD VS FREQUENCY

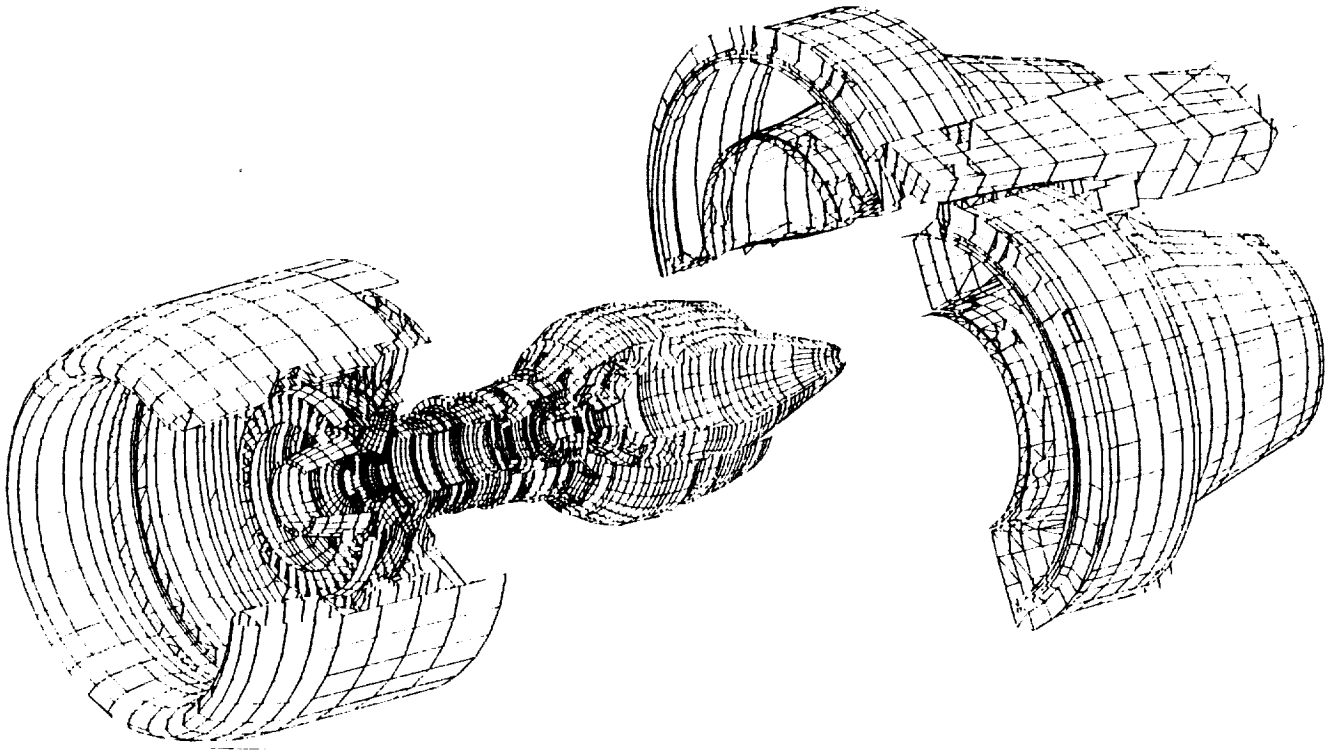
NASTRAN is currently used as a common analysis base between the aircraft and engine manufacturers and agreements are obtained between companies as to modelling number ranges. This permits ease of transmitting models to each other and similar DMAP can be utilized. Unique post–processing routines are utilized to review results from the steady state frequency response analyses.

CURRENT ANALYSIS TECHNIQUES (CONTINUED)

- COMMON ANALYSIS BASE – NASTRAN
  - DIRECT SOLUTION, STEADY STATE FREQUENCY RESPONSE
  - LARGE AMOUNT OF 'DMAP' WRITTEN
    - MATRIX INPUT
    - GYRO STIFFENING
    - NON–LINEAR BEARINGS
    - NON–LINEAR MOUNTS
    - SPECIAL OUTPUT REQUIREMENTS
- UNIQUE POST–PROCESSING ROUTINES
  - COMPONENT ENERGIES (POTENTIAL AND KINETIC)
  - RESPONSE (LOADS, DEFLECTIONS) VS FREQUENCY
  - ANIMATION PLOTS
  - BEARING / MOUNT RESPONSE

This current new generation strut/nacelle/engine structural model is being utilized for both static and dynamic analyses. For dynamic analysis the model will undergo dynamic reduction on approximately 15 major components to reduce the number of DOF from over 150,000 to less than 5000.

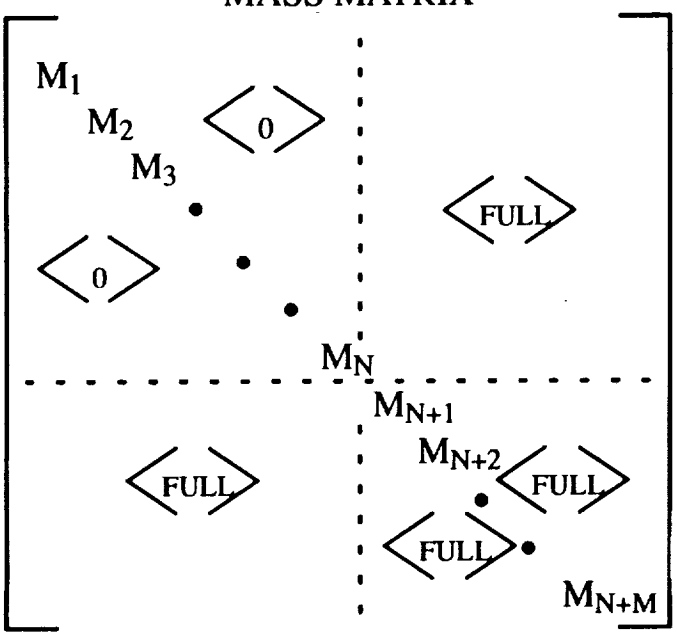
MODEL USED TO GENERATE DYNAMICALLY SIMILAR COMPONENTS



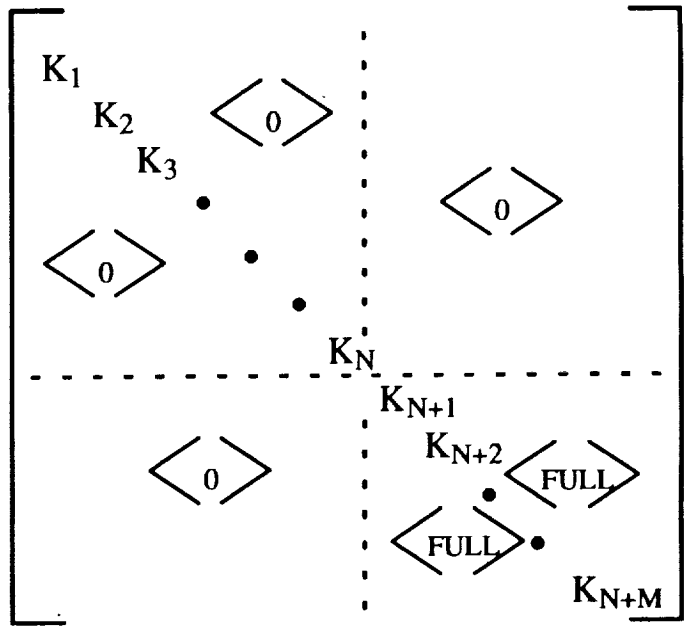
Component dynamic matrices are generated using fixed boundary component modes. The resulting mass and stiffness matrices are of the form shown. Each of the N component modes can be obtained from K_i/M_i and are coupled to the M boundary DOF by only the mass matrix. Also, full structures can be generated from a symmetric structure by combining the dynamic matrices generated from both a symmetric and anti–symmetric analysis. This is commonly done with the aircraft structure.

COMPONENT DYNAMIC MATRICES

N: COMPONENT DYNAMIC DOF
MASS MATRIX



M: COMPONENT BOUNDARY DOF
STIFFNESS MATRIX



Currently NASTRAN is utilized as the analysis tool in determination of engine related aircraft noise; however, large amounts of DMAP are required to obtain desired results. It is possible to use NASTRAN as the common model generation tool and output the required matrices for use in external solution routines. The solution matrices could then either be re–input into NASTRAN to obtain desired results or use external post–processing routines. Non–linear solution techniques also need additional development to describe items such as oil damped bearing properties, elastomeric and fluidic mounts and unloaded/loaded bearing and mounts. Test correlation of both the engine and airframe under dynamic loading is a necessity and there is some concern as to the procedures to be used if correlation is not obtained. The dynamic models used are quite complex, thus it will be difficult to locate and fix areas which do not correlate.

ISSUES

- ANALYSIS TOOL
  - NASTRAN IS ADEQUATE – IS IT CURRENTLY BEST??
  - LARGE, COMPLEX, 3–DIMENSIONAL, NON–LINEAR
  - DAMPED, FREQUENCY RESPONSE
- BEARINGS
  - METHODOLOGY TO DESCRIBE OIL DAMPED PROPERTIES
  - DEVELOPMENT OF STIFFNESS VS ROTOR LOAD
- ENGINE MOUNTS
  - STIFFNESS VS LOAD
  - STIFFNESS VS FREQUENCY – FLUIDIC
- TEST CORRELATION
  - TEST PLAN DEVELOPMENT
  - BAD CORRELATION – WHAT TO DO??

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	May 1992	Conference Publication

4. TITLE AND SUBTITLE
Computational Structures Technology for Airframes and Propulsion Systems

5. FUNDING NUMBERS
505-63-53-01

6. AUTHOR(S)
Ahmed K. Noor, Jerrold M. Housner, James H. Starnes, Jr., Dale A. Hopkins, and Christos C. Chamis, Compilers

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
NASA Langley Research Center
Hampton, VA 23665-5225

8. PERFORMING ORGANIZATION REPORT NUMBER
L-17049

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA CP-3142

11. SUPPLEMENTARY NOTES
Ahmed K. Noor, Center for Computational Structures Technology, Univ. of Virginia, Hampton, VA; Jerrold M. Housner and James H. Starnes, Jr., NASA Langley Research Center, Hampton, VA; Dale A. Hopkins and Christos C. Chamis, NASA Lewis Research Center, Hampton, VA.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified–Unlimited

Subject Category 39

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*
This conference publication contains the presentations and discussions from the joint UVA/NASA Workshops on Computational Structures Technology for Airframes and Propulsion Systems held at NASA Lewis Research Center, June 26–27, 1991 and at NASA Langley Research Center, September 4–5, 1991. The presentations made included NASA Headquarters perspectives on HSCT, goals and objectives of the UVA Center for Computational Structures Technology, NASA and Air Force CST activities, CST activities for airframes and propulsion systems in industry, and CST activities at Sandia National Laboratory.

14. SUBJECT TERMS
Computational structures technology; Structural mechanics; Failure and life prediction; Airframes; Propulsion systems

15. NUMBER OF PAGES
527

16. PRICE CODE
A23

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	