UNIVERSIDAD POLITECNICA DE VALENCIA

DEPARTAMENTO DE INGENIERIA MECANICA Y DE MATERIALES

ELEMENTOS FINITOS

(E.T.S.I.I.V)

MODELADO POR ELEMENTOS FINITOS

LECCION 1.- INTRODUCCION

J. L. OLIVER
Dr. Ingeniero Industrial

Valencia, 2005

COURSE OVERVIEW

GOALS: To understand F. E. Theory
To model structural systems
To use commercial FEA codes

APPROACH: Lectures, laboratories, case studies

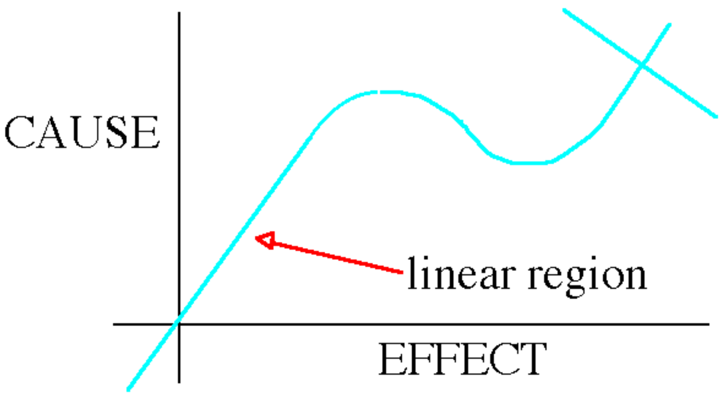
FOR: Engineers, physical scientists, mathematicians

LEVEL: Introductory

2

SUBJECT: Finite element analysis of linear, static systems

VARIABLES: Displacement, stress, strain



3

DEFINITION OF FINITE ELEMENT

FINITE ELEMENT:

A hypothetical subdivision of a structure or system, possessing a regular shape which can be analyzed.

The finite element method requires:

a) development of individual elements, often with concepts from classical mechanics

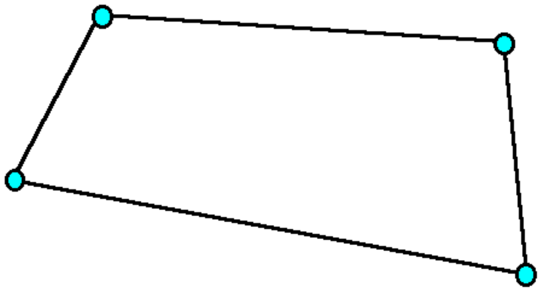
b) assembly of elements into structure or system

c) solution of the assembly using modern numerical analysis and computing

d) recovery of field variables (stress, strain) within the interior of the elements

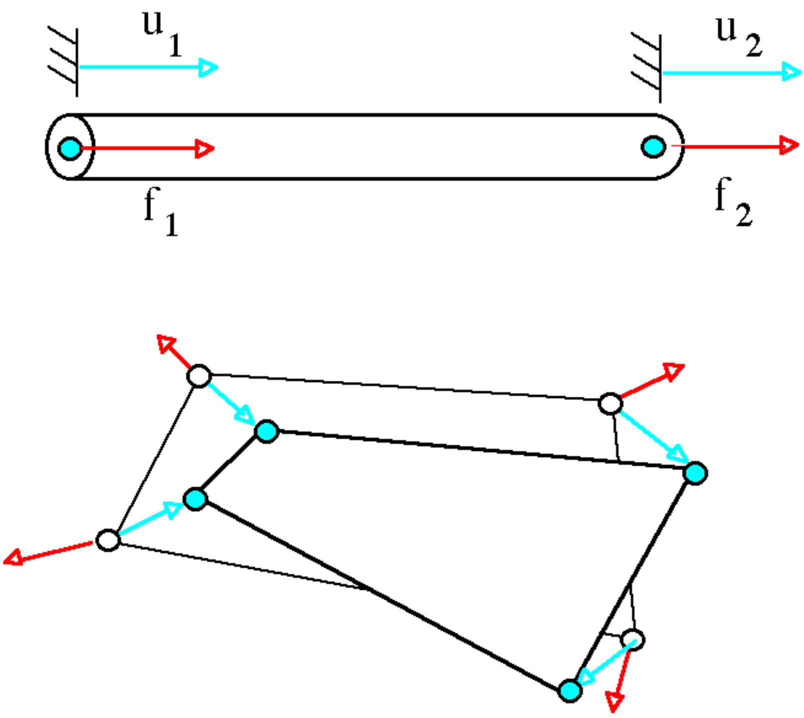
(6)

Line element (truss, beam, pipe, electrical resistor)



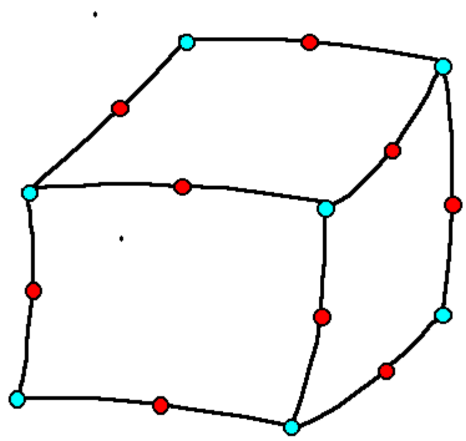
Two-dimensional element (membrane, plate, shell)

(7)



(deformation of element greatly enlarged)

8



Solid element

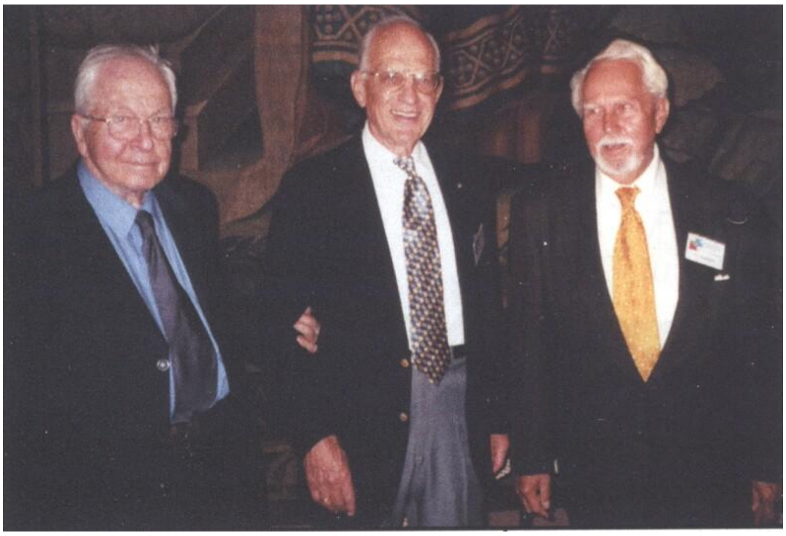
9

HISTORICAL REVIEW OF THE FINITE ELEMENT METHOD

1696	Gottfried Leibniz
1851	Karl Schellbach
1943	Richard Courant
1950's	John Argyris
1956	Turner, Martin, Clough and Topp

(10)

0011 JOHN ARGYRIS - RAY CLOUGH – OLEC ZIENKIEWICZ



0012 WALTER RITZ – BORIS GALERKIN

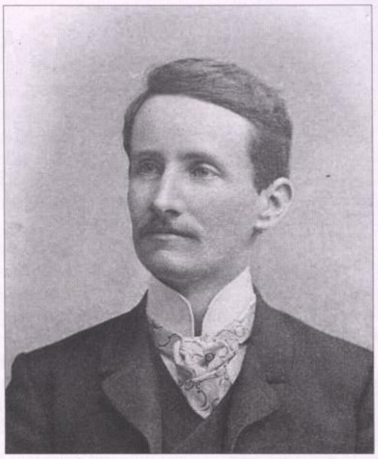


Figure 1:
Walter Ritz

Figure 2:
Boris Galerkin

0013 I. G. BUBNOV – R. COURANT

Figure 3:
I.G. Bubnov

Figure 4:
R. Courant

0014 FRAEIJS DE VEUBEKE

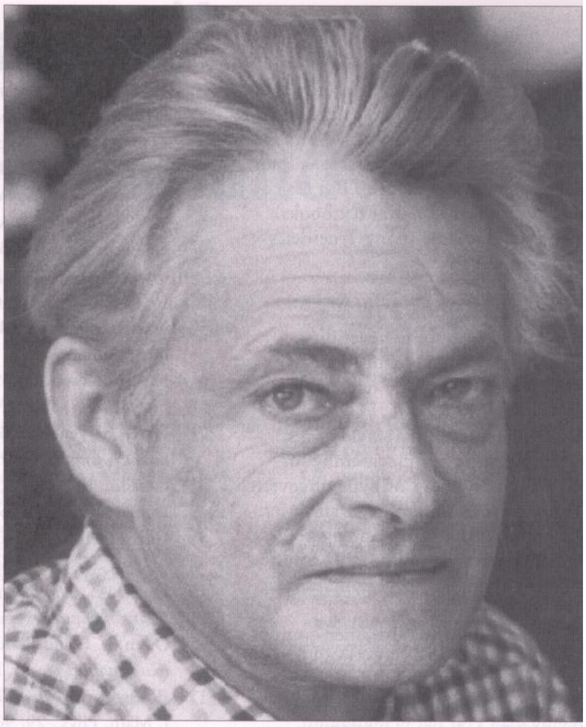
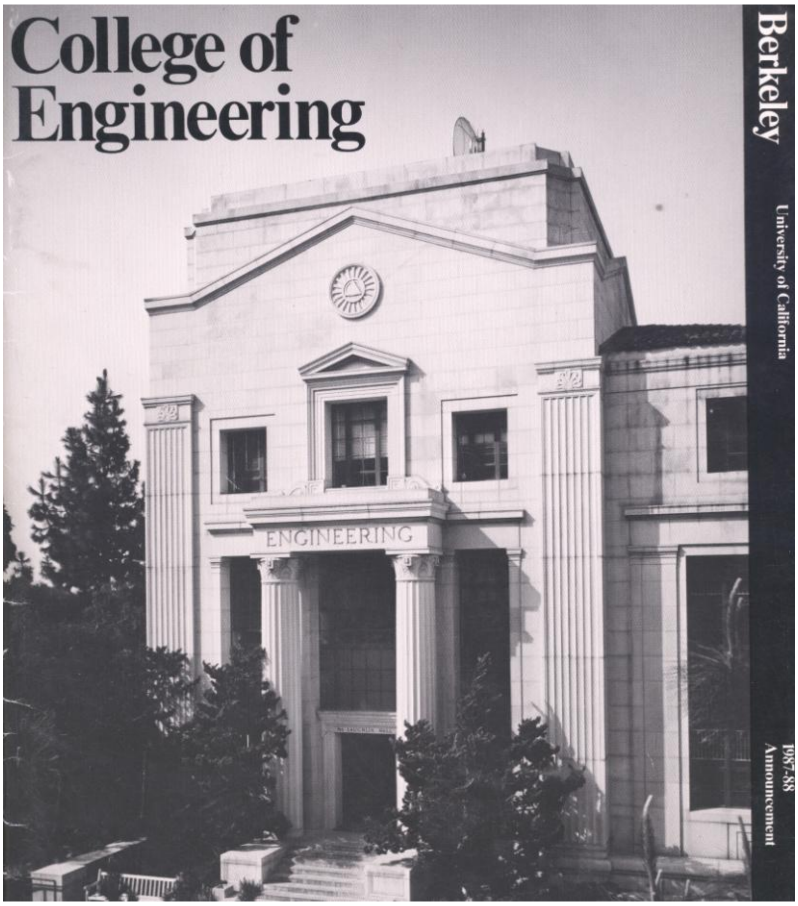
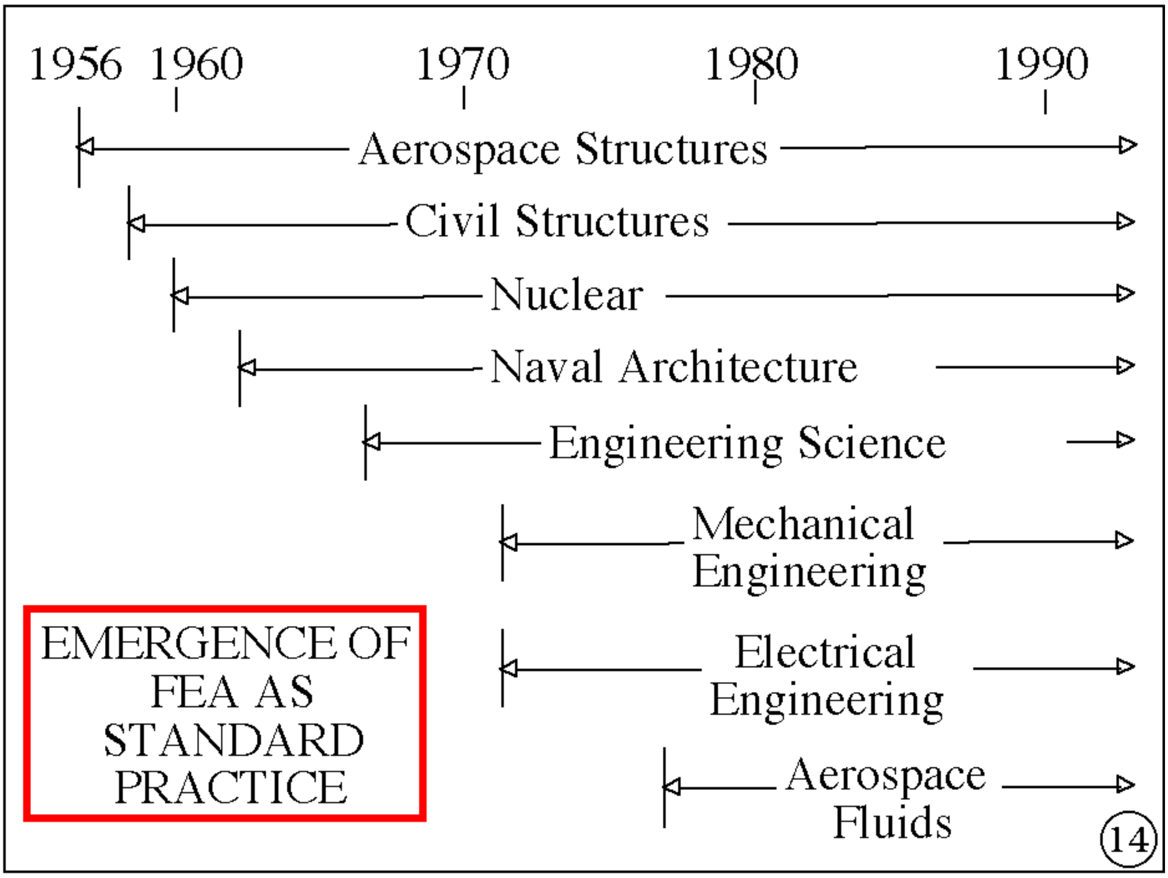


Figure 1:
Fraeijs de Veubeke

002 **DONDE** **CURSO 2005-6**

0022 UNIVERSIDADES




1956
1960
1970
1980
1990
Aerospace Structures
Civil Structures
Nuclear
Naval Architecture
Engineering Science
Mechanical Engineering
Electrical Engineering
Aerospace Fluids
EMERGENCE OF FEA AS STANDARD PRACTICE
14

0013

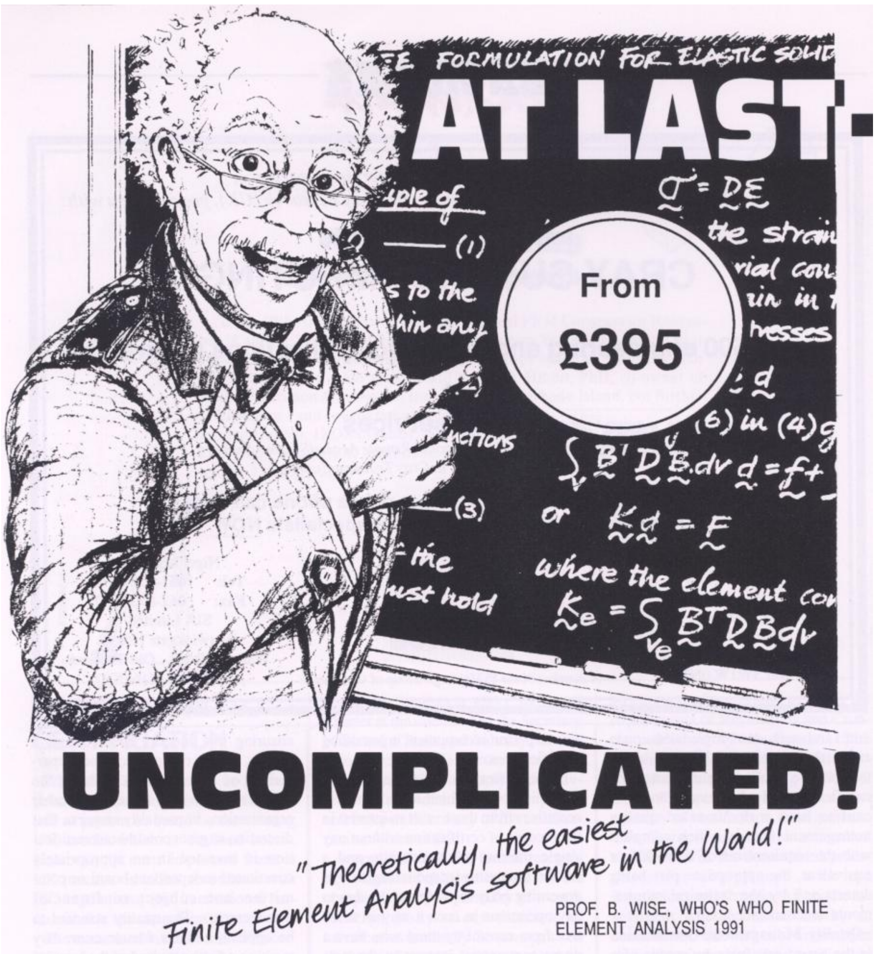
0014

Figure 4:
Aeroplane from the 50´s

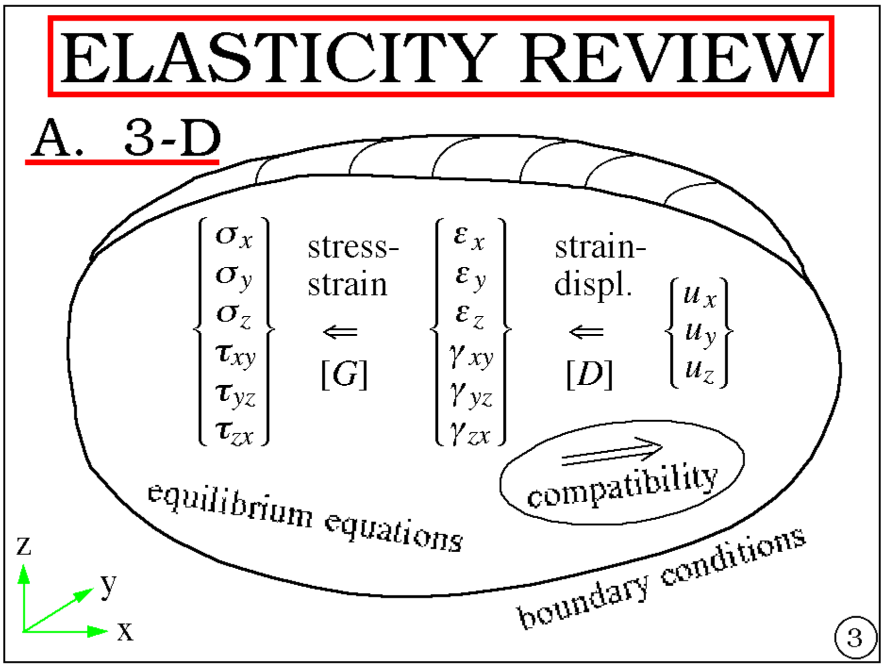
0014

Figure 4
Seat layout of Boeing 747.

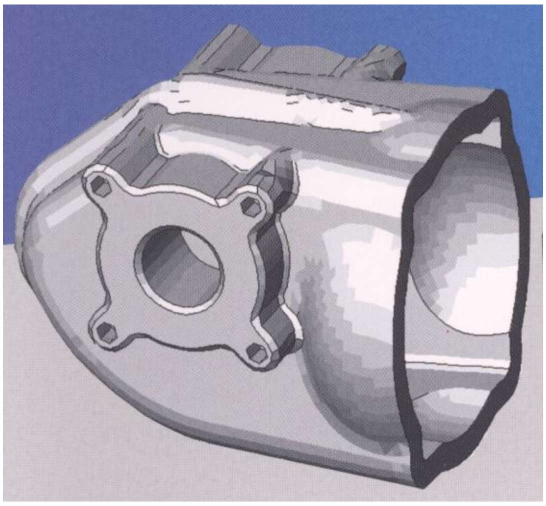
0021 PROBLEMAS FISICOS FORMULADOS MEDIANTE ECUACIONES DIFERENCIALES EN DERIVADAS PARCIALES



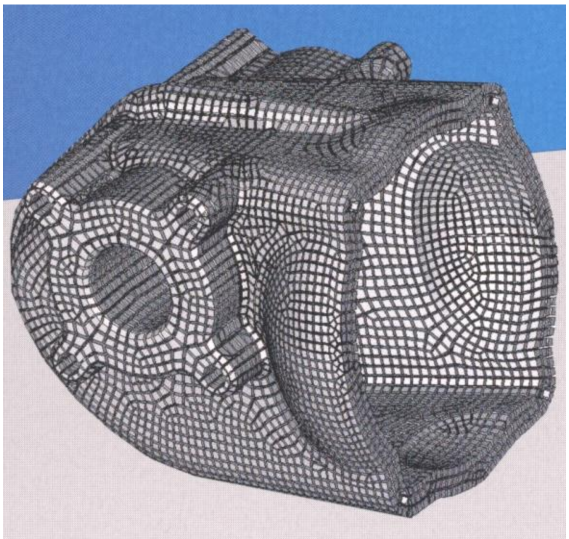
0022 TEORIA DE LA ELASTICIDAD



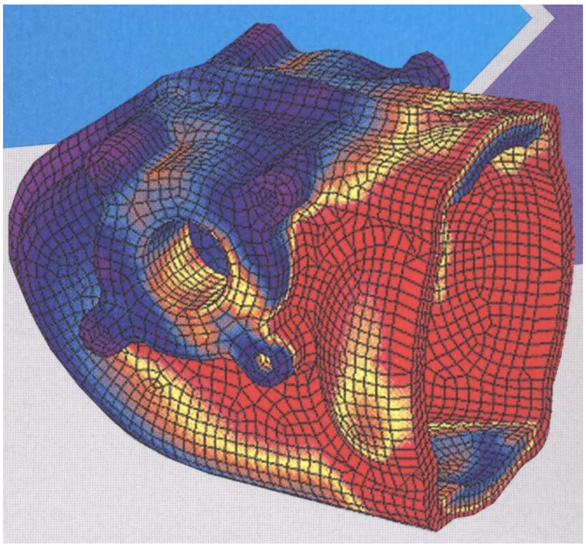
0031 IMPOSIBLE RESOLVER LAS ECUACIONES DIFERENCIALES EN UN PROBLEMA REAL



0032 POSIBLE RESOLVERLAS EN DOMINIOS CON FORMAS GEOMETRICAS SENCILLAS



0033 APROXIMADAMENTE LA SOLUCION ES SUMA DE LAS SOLUCIONES EN LOS ELEMENTOS FINITOS



0041

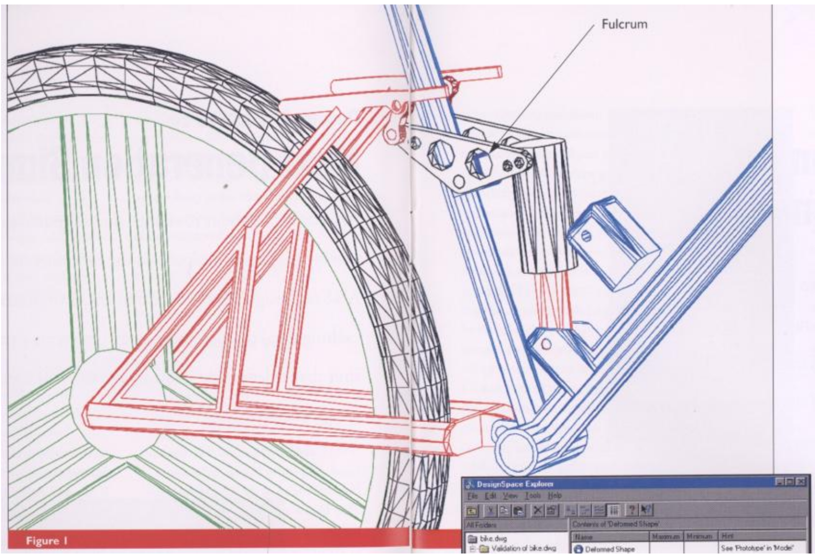


Figure 1

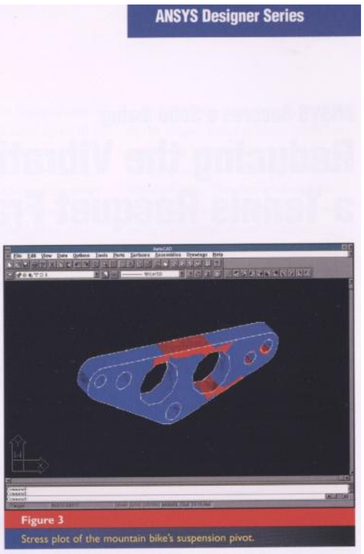
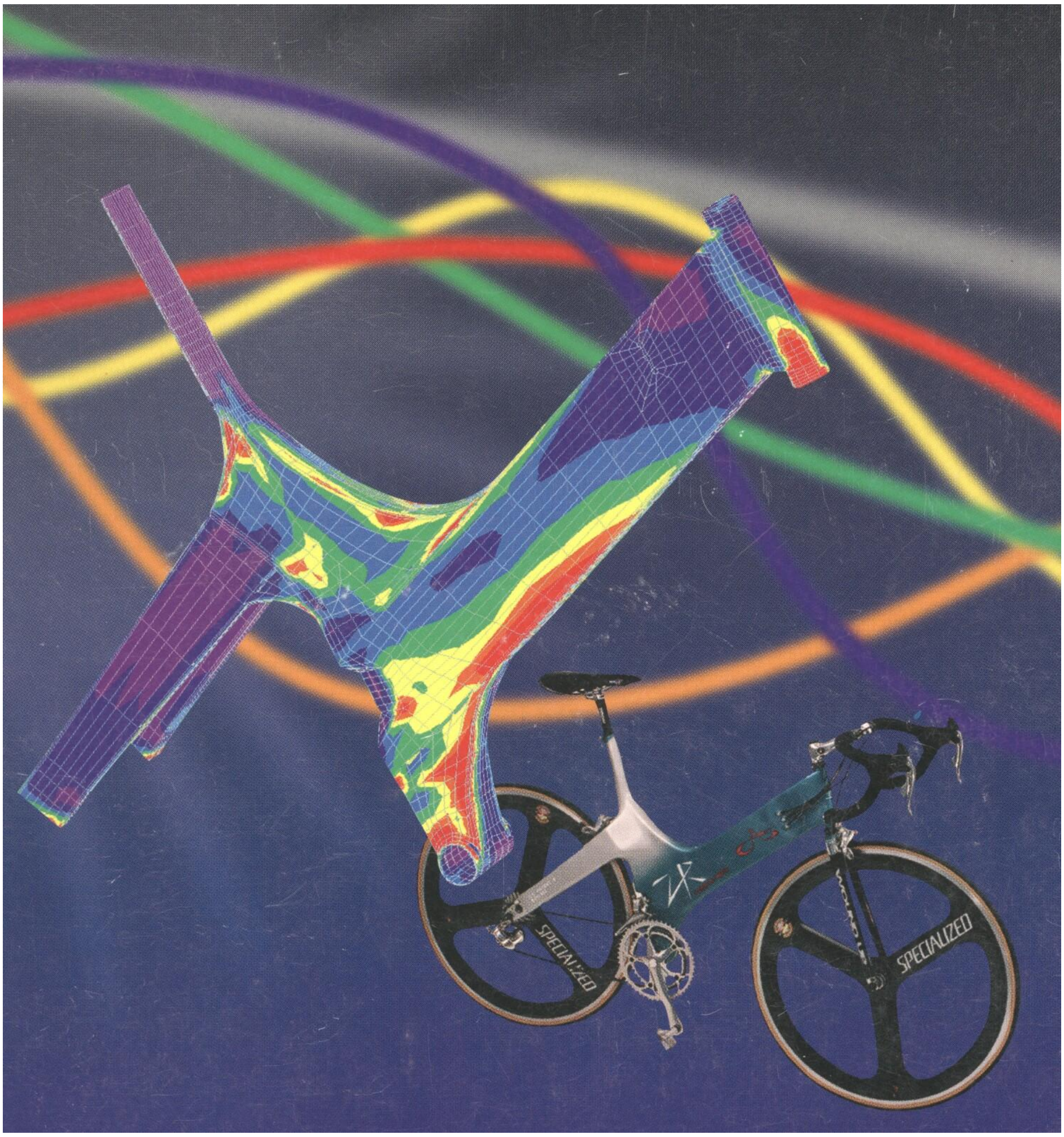
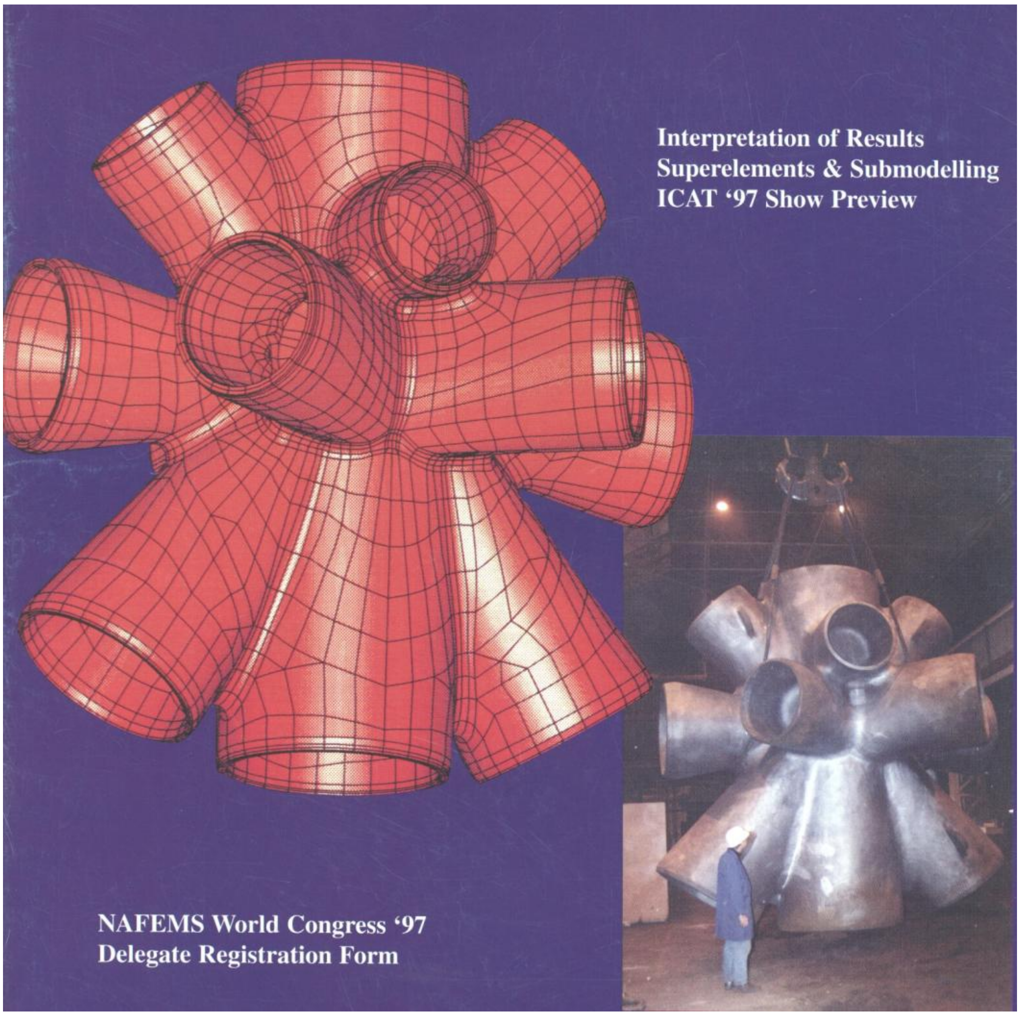
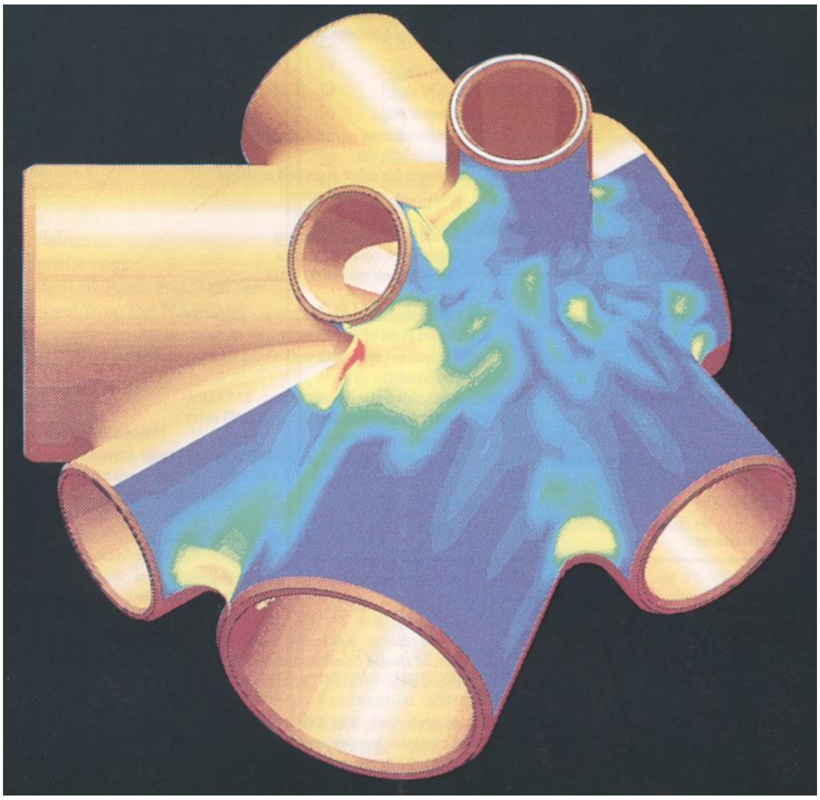


Figure 3
Stress plot of the mountain bike's suspension pivot.

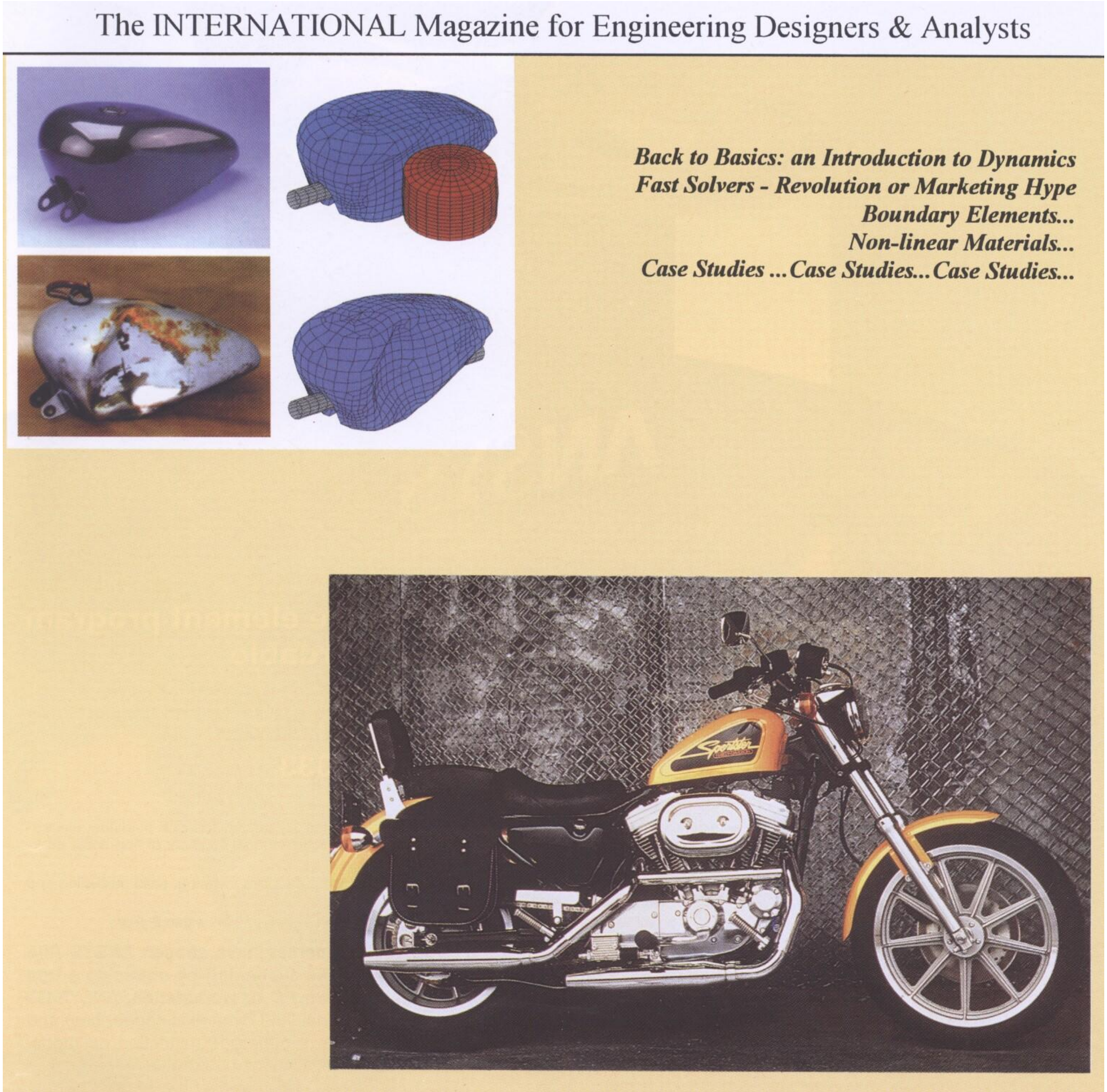


0043


Interpretation of Results
Superelements & Submodelling
ICAT '97 Show Preview
NAFEMS World Congress '97
Delegate Registration Form



0042



0044

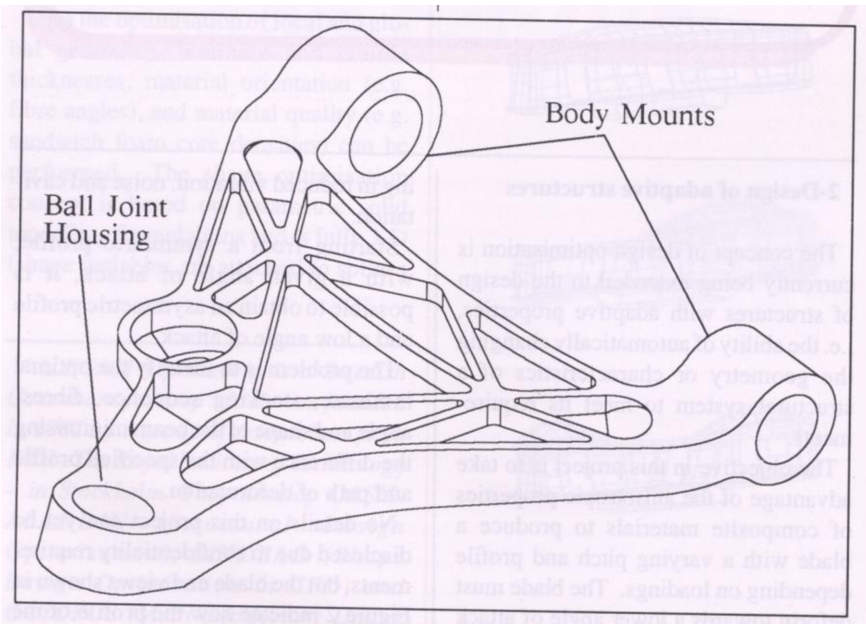


Figure 1. The Composite Suspension Arm

Figure 2. Photoelastic Plot of Overall Stress Distribution

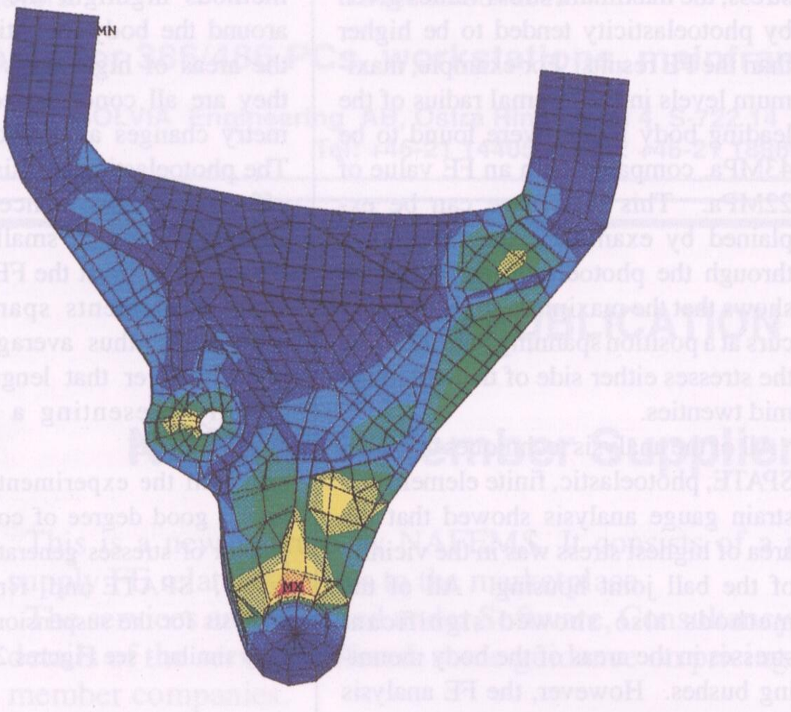


Figure 3. FE Plot of Overall Stress Distribution

Position	Strain Gauges	FE	Photoelastic
Ball joint housing	176 Mpa	165 Mpa	176 Mpa

0045


July 1998
BENCHmark
The INTERNATIONAL Magazine for Engineering Designers & Analysts
Back to Basics: Acoustics (Part II)
Thermal Transient Analysis
Validation of Composite Material Properties
Case Studies ...
Product Updates
Published by:
NAFEMS: The INTERNATIONAL Association for the Engineering Analysis Community
Creating Awareness Delivering Education & Training Stimulating Standards

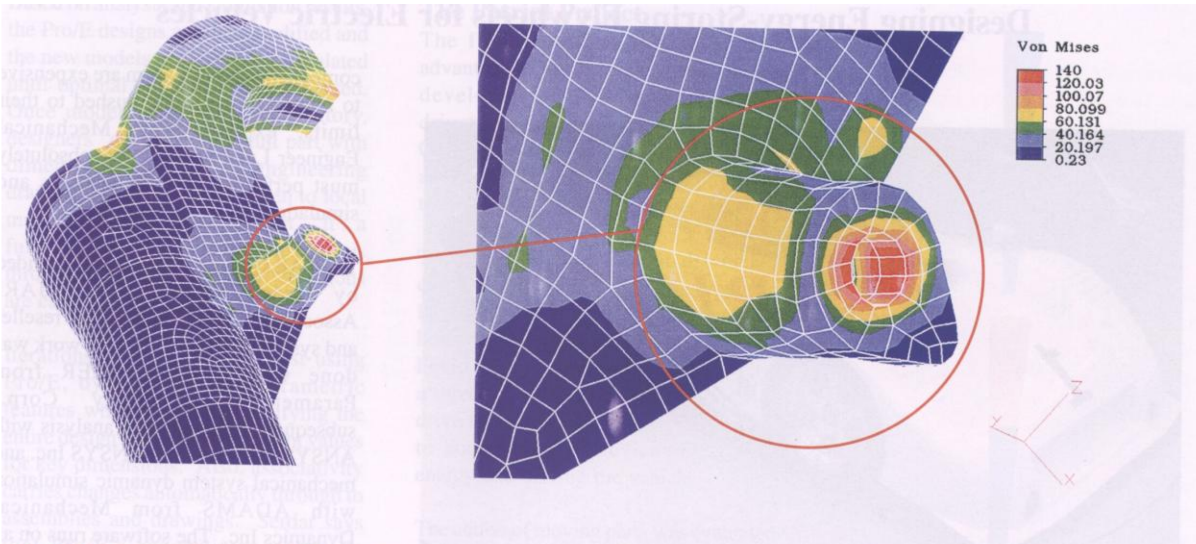
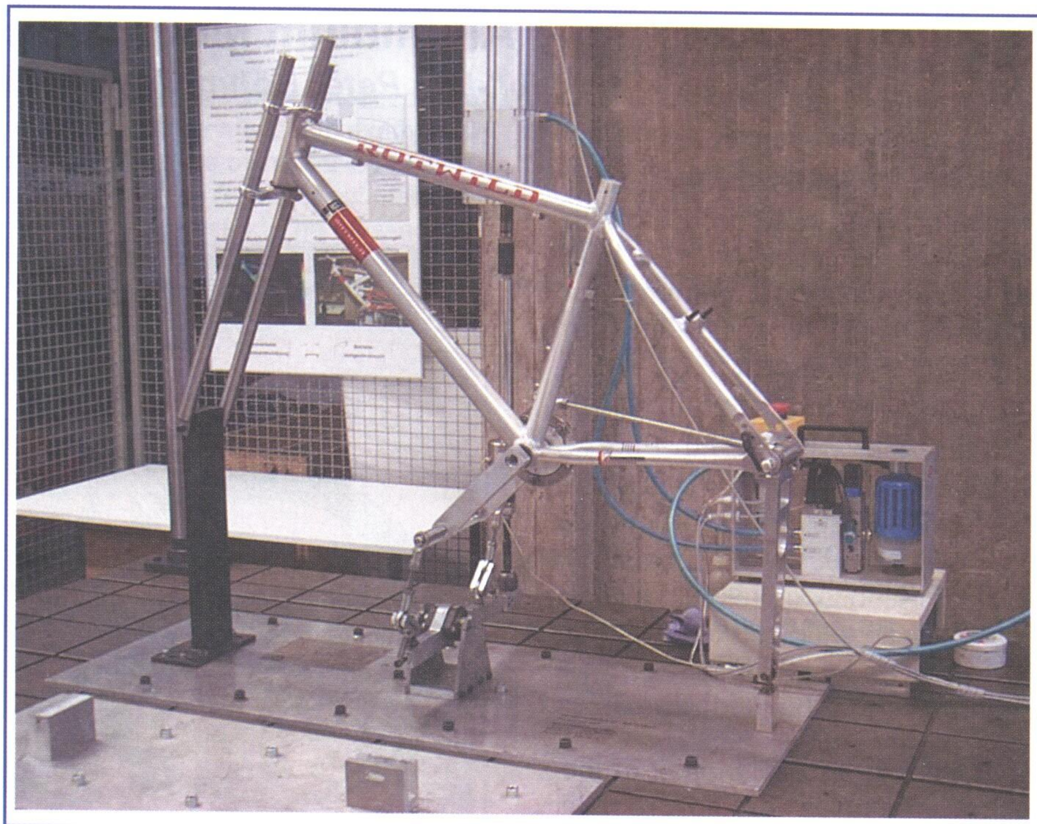
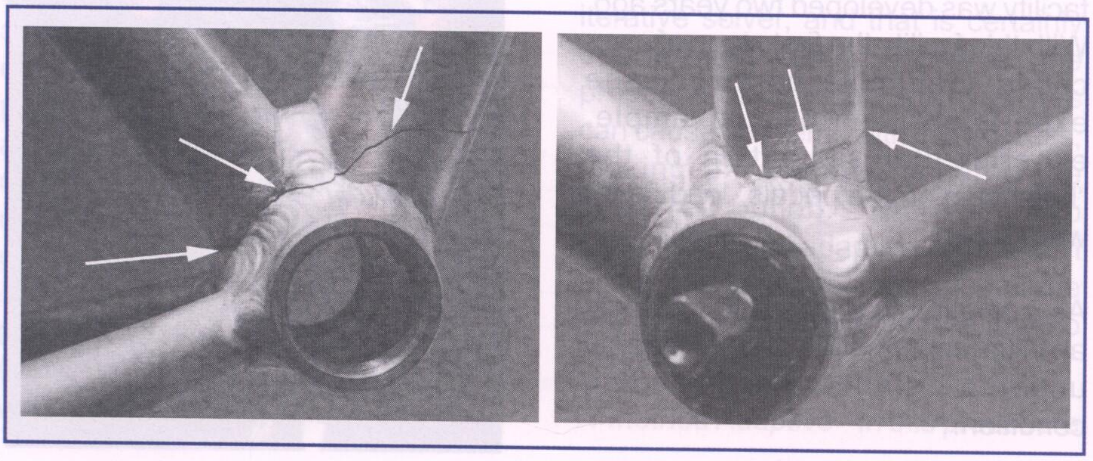
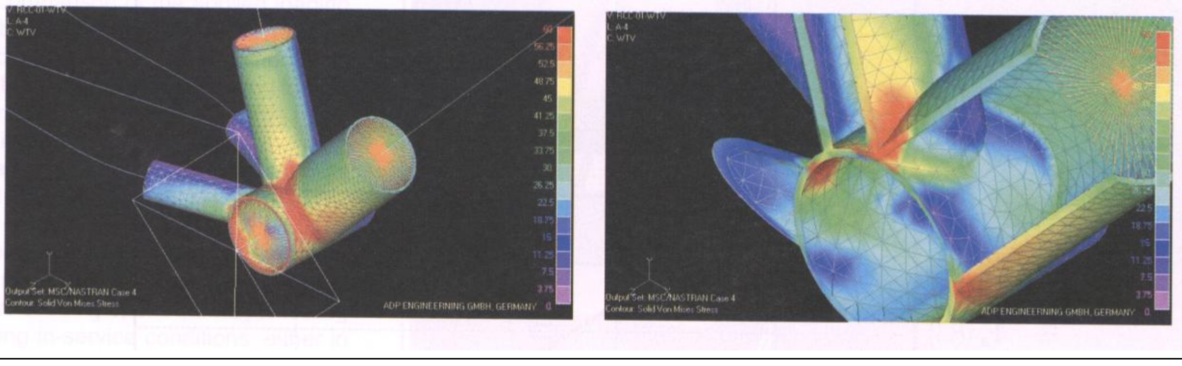


Figure 1: A finer mesh was applied around critical areas of the front fork

Figure 2: High localised stresses on this brake attachment result from the loading applied to the brake assembly






Output Set: MSC/NASTRAN Case 4
Contour: Solid Von Mises Stress
ADP ENGINEERING GMBH, GERMANY
Output Set: MSC/NASTRAN Case 4
Contour: Solid Von Mises Stress
ADP ENGINEERING GMBH, GERMANY

The Space Shuttle Discovery and its five man crew launched from Kennedy Space Center. See related story, page 18. Photo courtesy of NASA.

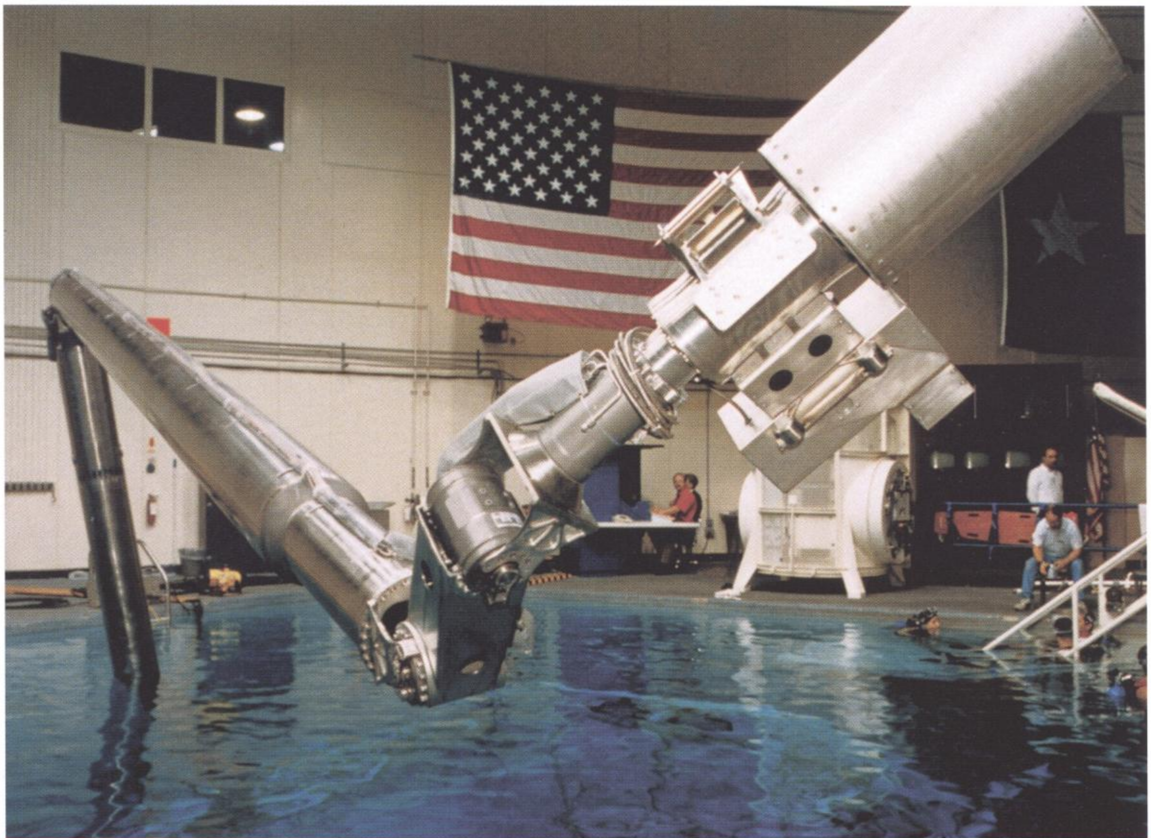
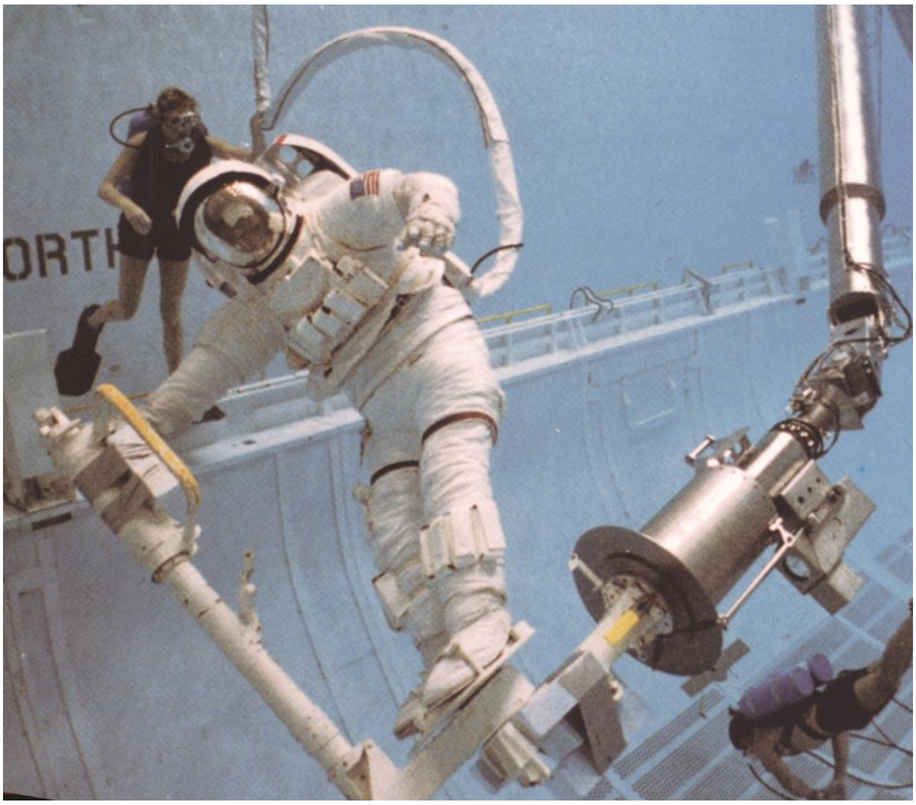


Figure 1

The WRMS demonstrates that it has the ability to operate out of the water, increasing the ease of maintenance. This arm can be serviced in approximately one-fourth the time of the old arm.



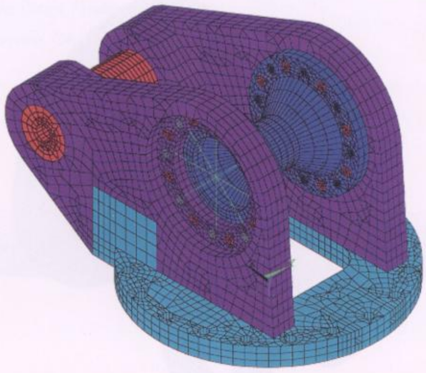


Figure 3

Finite element model of the shoulder pitch yoke including the hydraulic actuator mounting pin and stiffening spool.

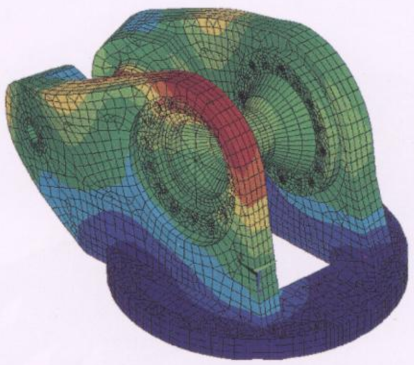


Figure 4

Deflected shape of the shoulder pitch yoke under worst-case loading as determined by the optimization routine.

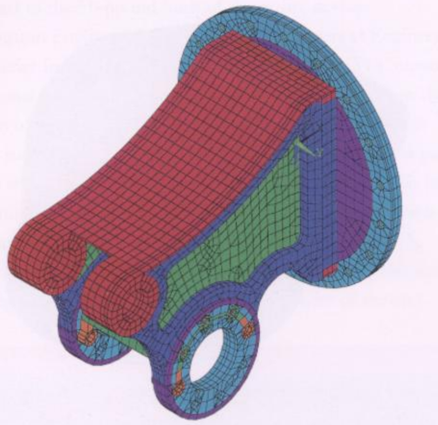


Figure 5

Finite element model of the elbow pitch yoke including the composite bearing.

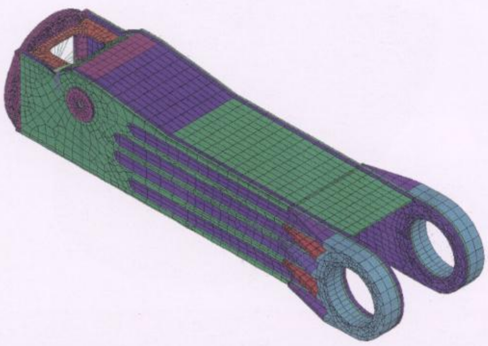
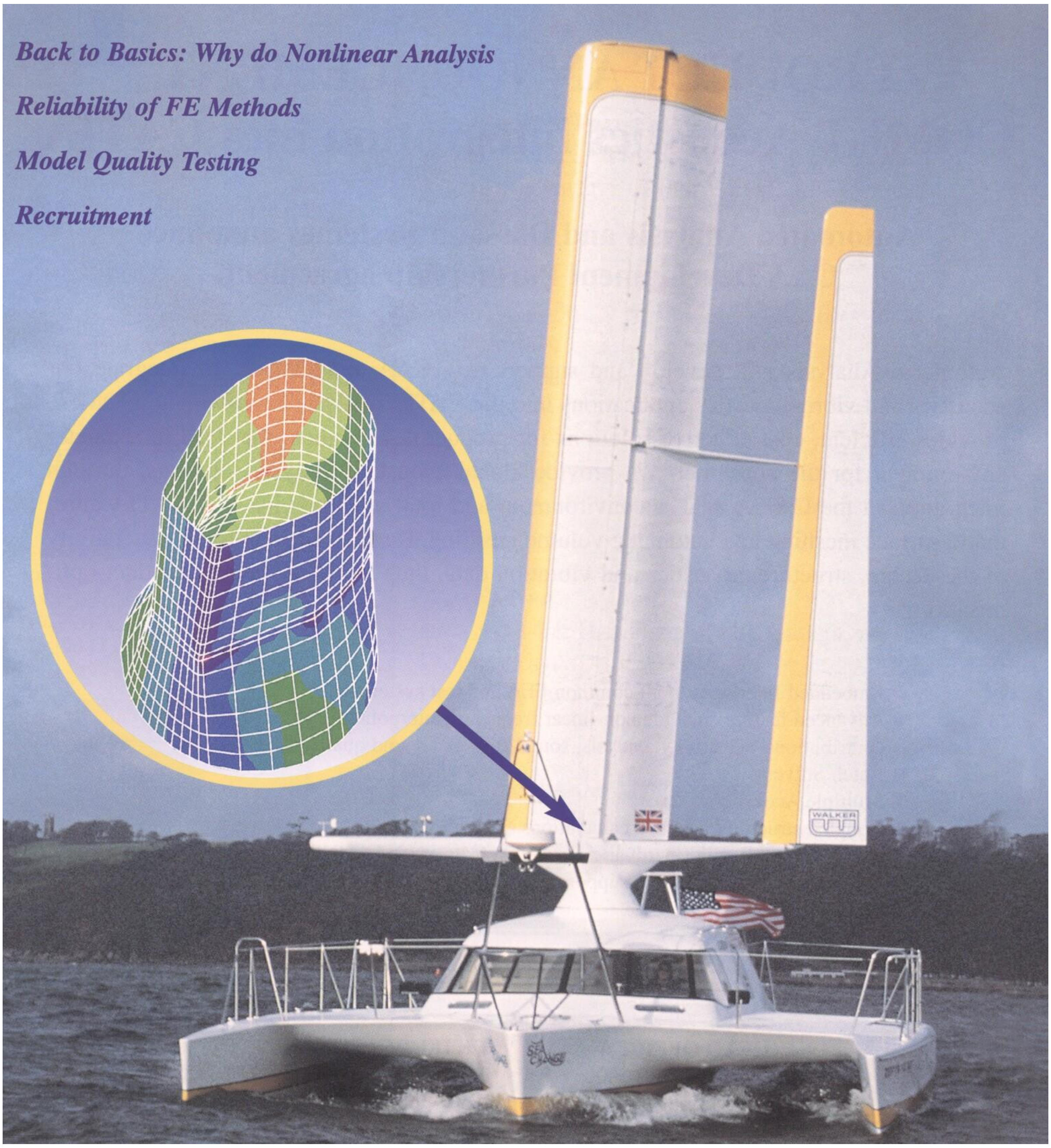
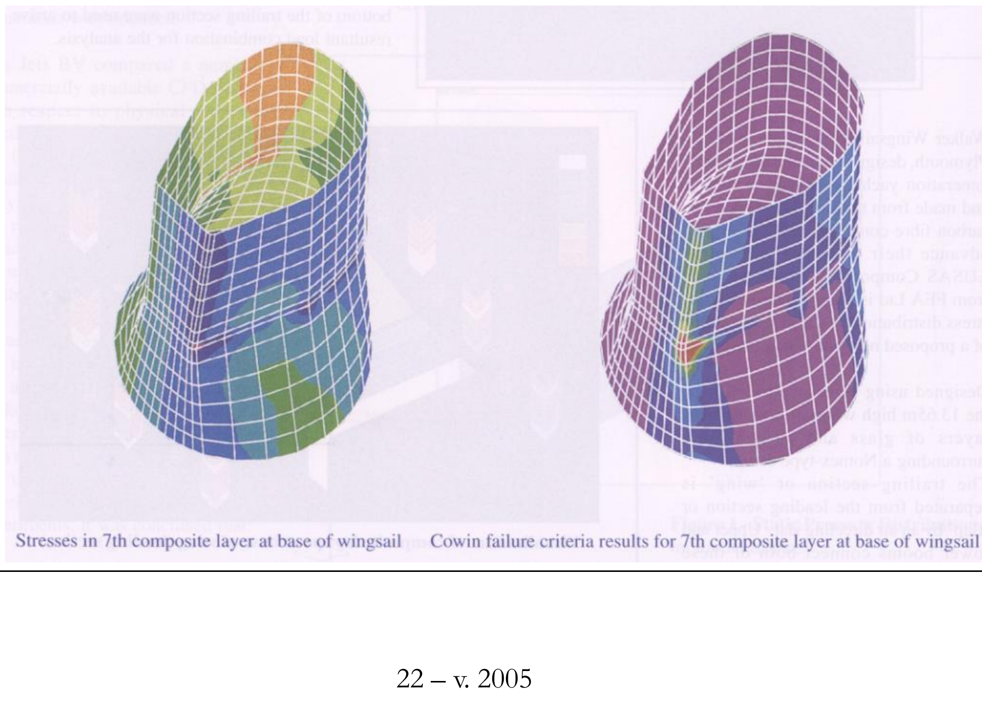


Figure 6

Finite element model of the main shoulder yoke.

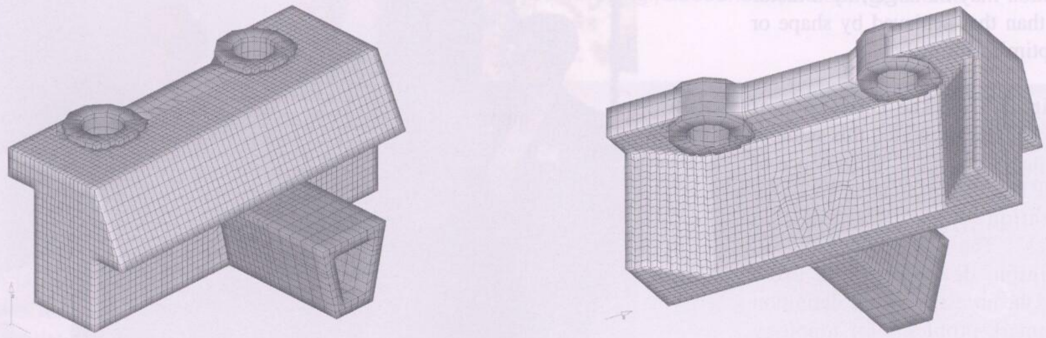
0047



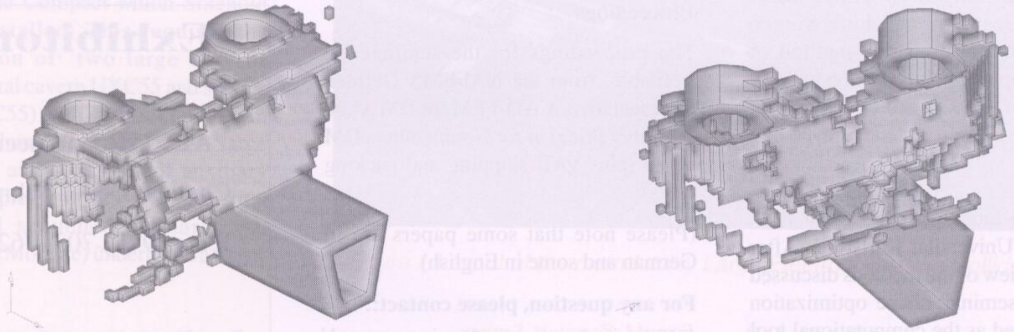


Stresses in 7th composite layer at base of wingsail   Cowin failure criteria results for 7th composite layer at base of wingsail

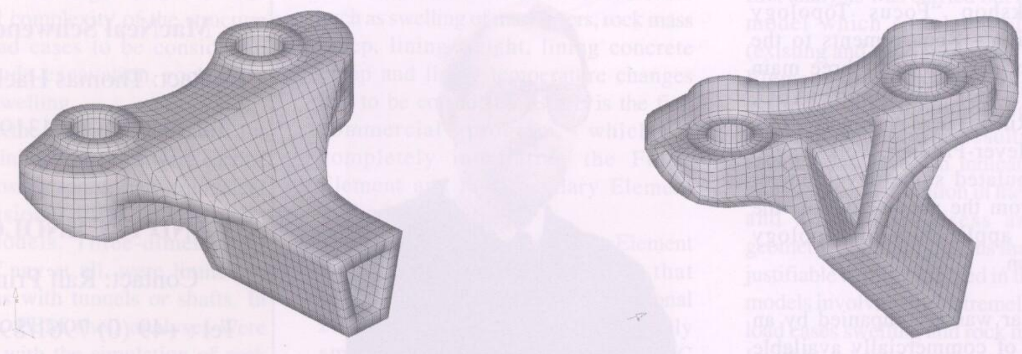
At Adam Opel the programs SKO(Soft Kill Option) and OptiStruct have been extensively used.



(Above)Two views of the design space for an engine mount.
(Below) Results of the topology optimisation for a volume fraction of λ = 0.35 Only elements with at least 99.8% of the maximum Young's Modulus are plotted.



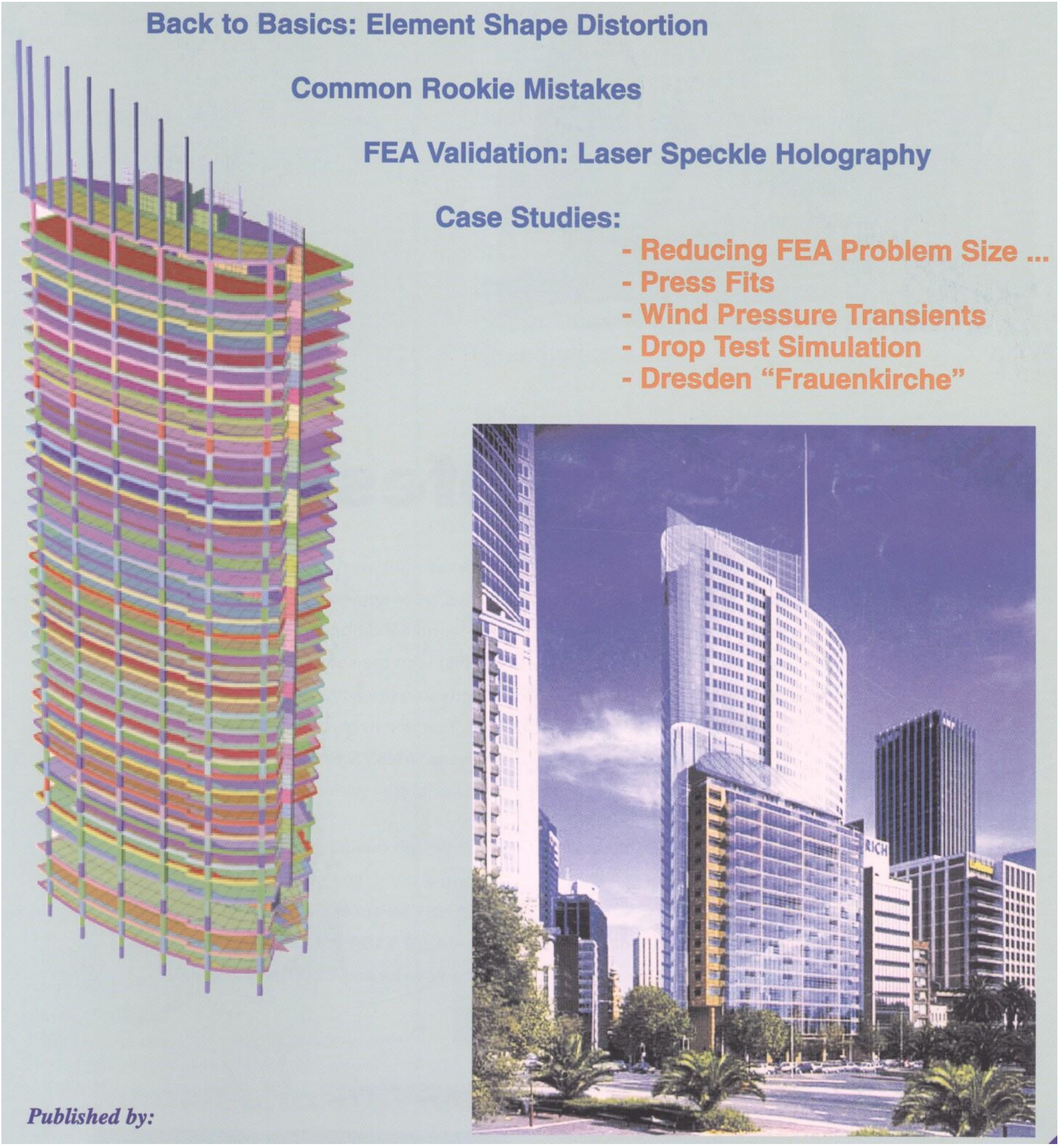
(Below) The final design shown at the bottom led to a stress reduction of 63%)



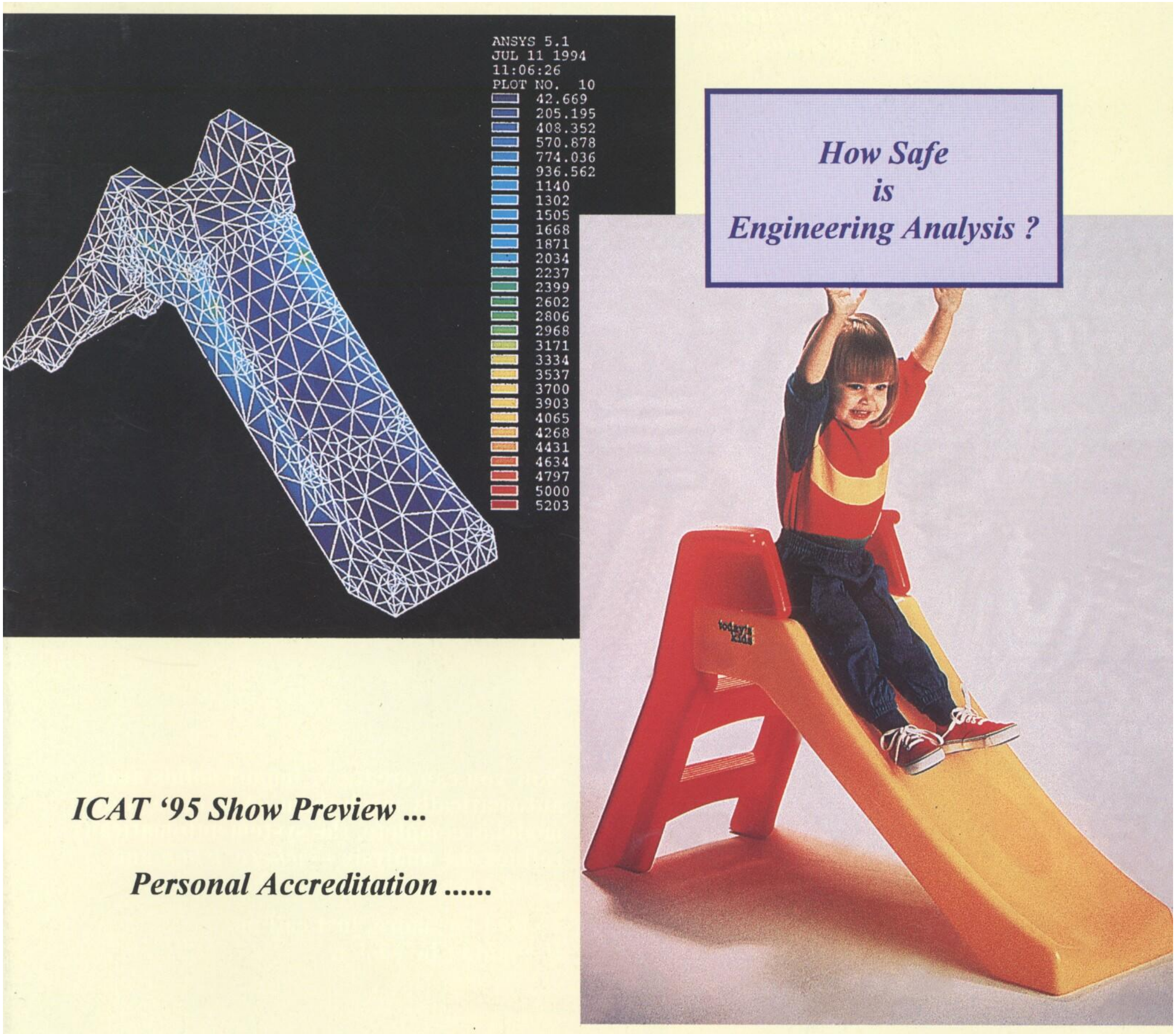

When lightning strikes, British Aerospace like to know where it is going
British Aerospace plc is solving the problem of protecting the Eurofighter from the effects of lightning strike.
Faced with expensive physical testing, BAe turned to software simulation and have assembled a system using electromagnetic analysis codes with the FEGS FAM modelling facility.
Chris Jones, Technical Specialist in the BAe EM group, says: "A model of this size and complexity could not have been considered before our FAM facility. FAM reduces the model development time by an order of magnitude, and allows us to do things that were not possible before." He continues: "Major gains will be accrued by re-using the model for several different types of analysis with different codes, and varying the aircraft configuration."
The model is tartan meshed to a 5 cm resolution. This FAM image shows the mesh of the pilot and his seat. The analysis of the entire aircraft uses 10 million cells.
FAM image of the Eurofighter geometry model, received from CATIA™. The aircraft is colour coded by structural part. A key requirement was to transfer complete aircraft geometries from the CATIA™ CAD system into FAM for analysis modelling.
BRITISH AEROSPACE
DEFENCE
MILITARY AIRCRAFT
+ FAM


NAFEMS World Congress '99
Delegate Registration
Design of Adhesive Joints
How to do Contact
Fatigue Analysis
Loading Verification

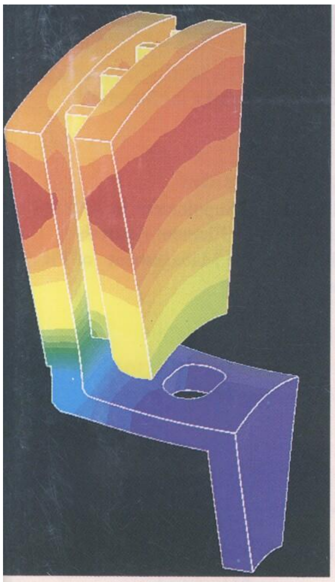
0411


Back to Basics: Element Shape Distortion
Common Rookie Mistakes
FEA Validation: Laser Speckle Holography
Case Studies:
- Reducing FEA Problem Size ...
- Press Fits
- Wind Pressure Transients
- Drop Test Simulation
- Dresden "Frauenkirche"
Published by:

0412


ANSYS 5.1
JUL 11 1994
11:06:26
PLOT NO. 10
42.669
205.195
408.352
570.878
774.036
936.562
1140
1302
1505
1668
1871
2034
2237
2399
2602
2806
2968
3171
3334
3537
3700
3903
4065
4268
4431
4634
4797
5000
5203
How Safe
is
Engineering Analysis ?
ICAT '95 Show Preview ...
Personal Accreditation ......

0413



NAFEMS WORLD CONGRESS '97

NAFEMS World Congress '97

Back to Basics: Understanding FE Jargon
The Analysis of Axisymmetric Rubber Mounts..
The World Wide Web as an Analysis Resource...

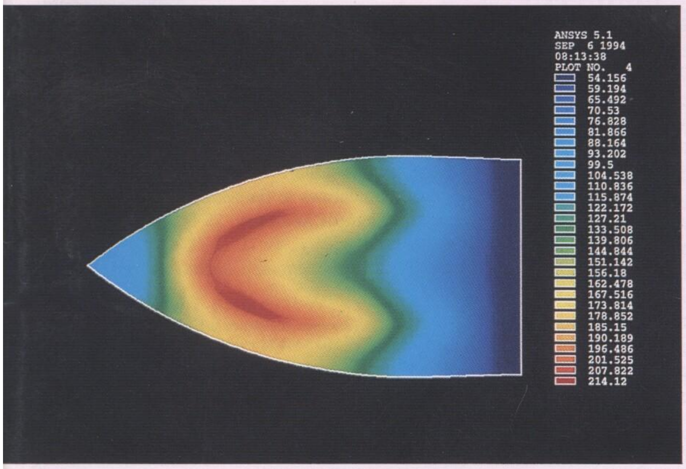
Official Launch

The NAFEMS Registered Analyst Scheme

REGISTERED ANALYST

BENCHmark

The INTERNATIONAL Magazine for Engineering Designers & Analysts



Iterative Linear Equation Solvers ...
Hex or Tet Meshing Controversy ! ...

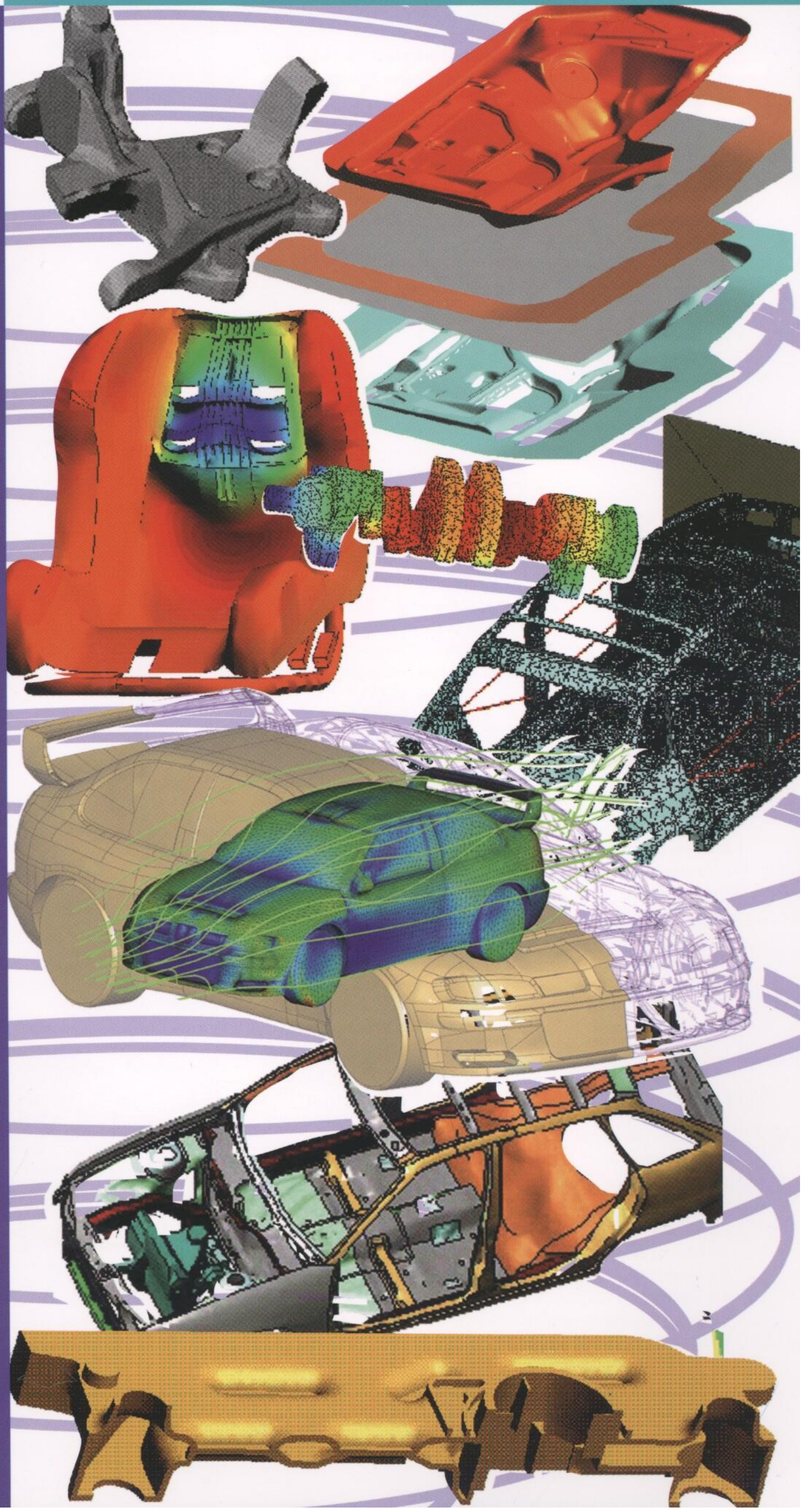
Lloyd's Challenge to FE Analysts ! ....
How to Interpret FE Results ...

Published By:
NAFEMS : **The INTERNATIONAL Organisation for the Engineering Analysis Community**

Creating Awareness • Delivering Education & Training • Stimulating Standards

July 1995

iacm expressions



Bulletin for The International Association for Computational Mechanics

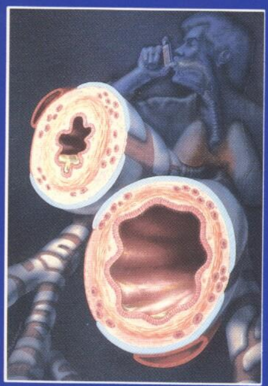
No. 12
Summer 2002

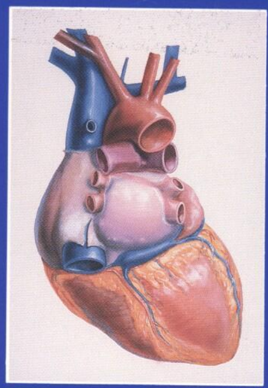
Third Issue, 1991

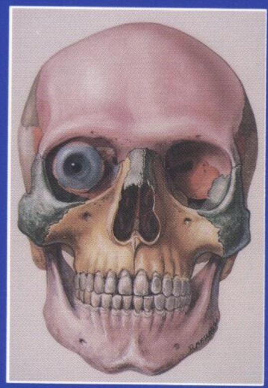
ANSYS news®

The ANSYS® Program in the Biomechanics Industry

See Related Story On Page 7.







Graphics courtesy of David Mascaro, Medical College of Georgia.

INSIDE THIS ISSUE...

LUSAS FEA NEWS
Finite Element Analysis
1995 Issue 2

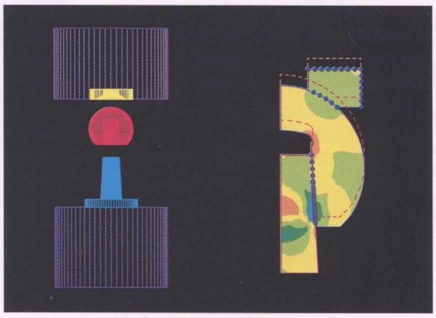
Morgan Matroc Analyses Artificial Hips

You would not normally associate "Hip Replacement" with Finite Element Analysis - unless you work for Morgan Matroc Ltd. They have recently bought the LUSAS Analysis System which they plan to use for stress analysis of various ceramic components including femoral heads used in total hip replacement. In orthopaedic applications, ceramic heads offer the advantages of low wear, high strength and excellent bio-compatibility.

Morgan Matroc are world leaders in ceramic technology and use the most up-to-date techniques to evaluate, design and test their ceramic implant devices.

The ceramic heads are made of either zirconia or alumina. The heads are highly polished to reduce wear on the hip socket which is made from Ultra High Molecular Weight Polyethylene (UHMWP). The head is then attached to a tapered titanium shaft which is cemented into the femur.

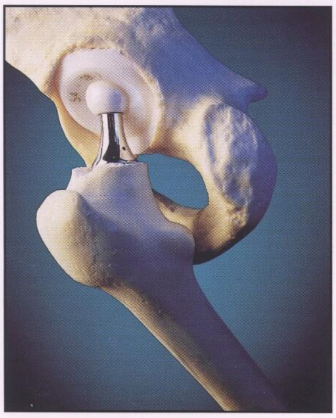
The heads are produced in a range of sizes with bore sizes and taper angles adjusted according to customer requirements. LUSAS will be used to investigate fatigue and impact for these various configurations.



FE model showing stresses induced when a ceramic head is forced onto a titanium shaft

Contact Analysis using Slidelines

Normally, the analysis of the contact between the shaft and head and between head and socket is a very difficult problem for traditional FE systems. However, LUSAS contains a "slideline" facility specifically designed for these types of problems. The slideline facility automatically calculates the contact conditions between the various parts in the structure and allows complex problems involving contact and friction to be modelled and analysed with relative ease.



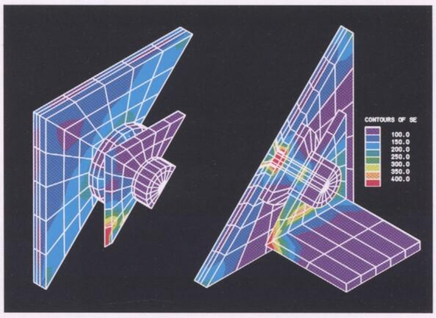
An artificial hip with ceramic head.

Contents

Pilkington plc buys LUSAS

Extensive Evaluation of LUSAS

After extensive evaluation of the LUSAS Analysis System, Pilkington plc is installing the software at two sites within the company - the Pilkington Architectural Division and Group Research.



Equivalent stresses in a Planar Support Fitting

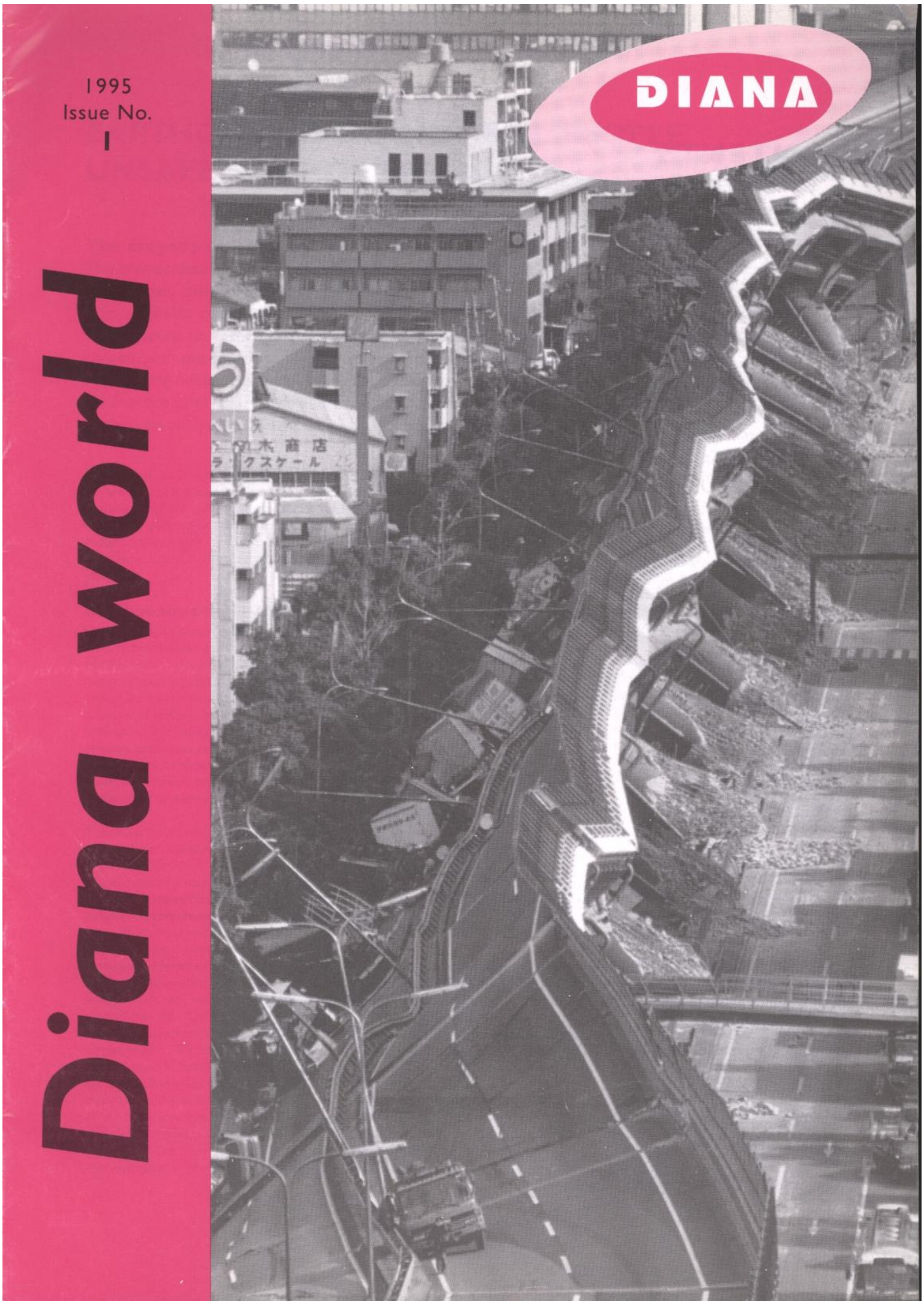
Typical Products made by Pilkington

Pilkington plc make glass for Building and Transport applications. Products include windscreens and windows for cars, trains and helicopters as well as architectural glazing for buildings.

The company plans to use LUSAS for a variety of applications including the stress analysis of glass and support mountings for architectural glazing systems and the impact analysis of windscreens.

...Continued on Page 4

0055



ALGOR DESIGN WORLD

When The Engineering Has to be Right

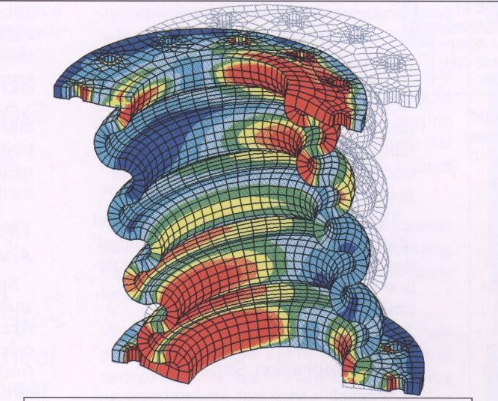
December 1994

Accupak Nonlinear Analysis Upgrade

More analysis choices, user controls and automatic features are highlights of new version.

The latest version of Algor's Accupak Nonlinear Analysis Package boasts a large number of added and enhanced design and analysis capabilities that help make it an even more powerful engineering tool. Accupak is primarily used by engineers whose designs require sophisticated nonlinear analysis due to nonlinear material properties or large deformation characteristics.

The new version of the software is designed to offer these engineers more analysis choices and help to simplify the process as much as possible.



The new Accupak supports the Mooney-Rivlin hyper-elastic material model for both 3-D solid "brick" (shown above) and plane stress elements. Above: A von Mises stress contour on a deflected view of an expansion joint subjected to stretching and lateral deflection (applied displacements).

Prescribed Displacement Load Curves

The new Accupak allows users to prescribe displacement versus time prior to analysis. These loads, called displacement load curves, act in much the same manner as a force load curve, allowing you to prescribe displacement levels and directions at various nodes in a structure. This capability is necessary for many analyses which include post-buckling and/or "snap-through" types of behavior.

Prescribed displacements are applied in Superdraw II, much like forces. The magnitude of the displacement, its direction, the load curve index and the active range index are all specified by selecting any node in the model.

(See "Accupak Nonlinear..." Page 2)

New Technology Improves Processing Speed

Algor's finite element analysis processors now contain a proprietary new technology that leads to much faster FEA results.

The new Algor technology accesses the matrices which numerically define the finite element model in a way that is more suitable to the newest generation of computers which have high-speed cache memory.

The advanced processing technology has been integrated into most Algor processors, including linear stress, nonlinear stress and vibration, steady-state and transient heat

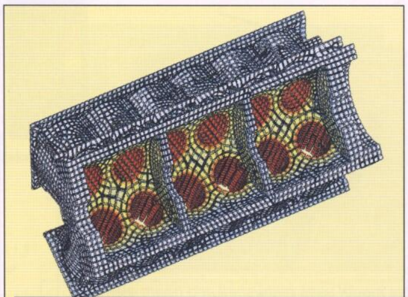
(See "New Software..." Page 4)

In This Issue:

Hexagen 4.0 adds More "Brick" Meshing Options

New version creates solid, 8-node "brick" models for analysis by any FEA processor.

Algor has introduced an enhanced version of Hexagen, the automatic solid mesh generator which creates, 8-node "brick" finite element models using solid models imported from CAD software such as Pro/ENGINEER, Unigraphics, SDRC, Aries, Catia, AutoCAD, Cadkey, IBM, Intergraph and more.



This FE model of a Formula One racing engine block was created from a Pro/ENGINEER CAD solid model part file. The surface mesh was enhanced in Houdini prior to automatic solid "brick" meshing by Hexagen.

High Accuracy Solid Models for any Processor

The resulting solid models are highly accurate because they use 8-node "brick" elements. Hexagen 4.0 has three solid meshing options, enabling the software to create models for analysis by virtually any FEA processor, including custom, "in-house" codes.

Three Solid Meshing Options

Hexagen uses a proprietary algorithm to generate models which are highly-accurate, but do not require excessive processing time. These models are pri-

(See "Hexagen 4.0..." Page 3)

ABAQUS / News

The Newsletter for ABAQUS Users Spring–Summer 1995

1995 ABAQUS Users' Conference

Contents

The 1995 ABAQUS Users' Conference will be held in Paris, France during May 31 to June 2, 1995. 64 papers will be presented, including the following invited lectures:

Overview of ABAQUS Applications at Pirelli, F. Mancosu, Pirelli Coordinamento Pneumatici S.P.A.

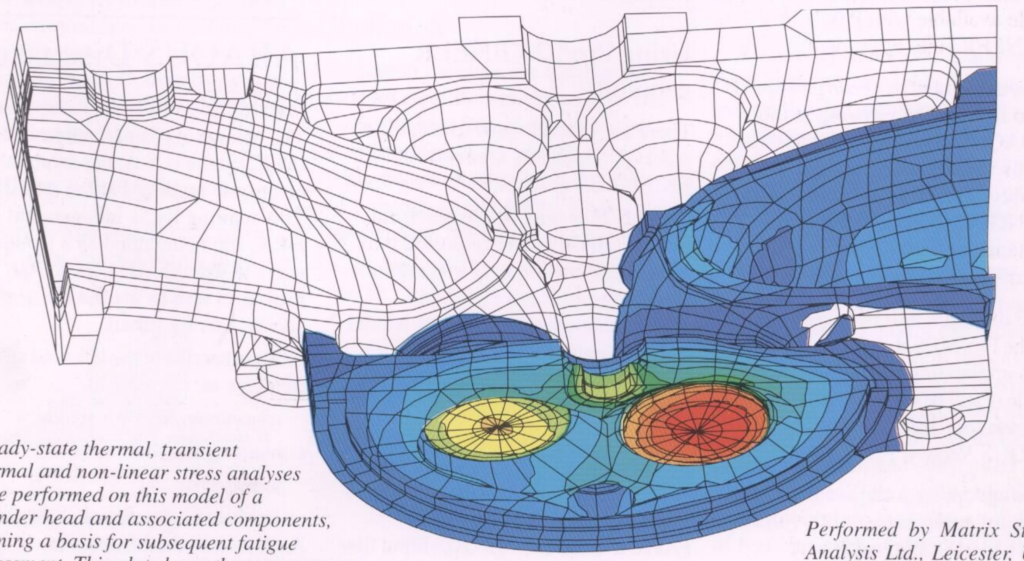
Overview of Nonlinear Finite Element Applications for Sheet Steel at Cockerill-Sambre Research and Development, M. Traversin, Cockerill-Sambre.

HKS will present 30 minute lectures on *Submodelling* and *Structural Elements,* as well as 1 hour seminars on *Solving Large Problems*, *Cavity Radiation*, *Viscoelasticity* and *Coupling ABAQUS/Standard and ABAQUS/Explicit.* We will also announce and describe the new features in ABAQUS Version 5.5.

On May 30, the day before the Conference, we will present one-day courses on the following topics:

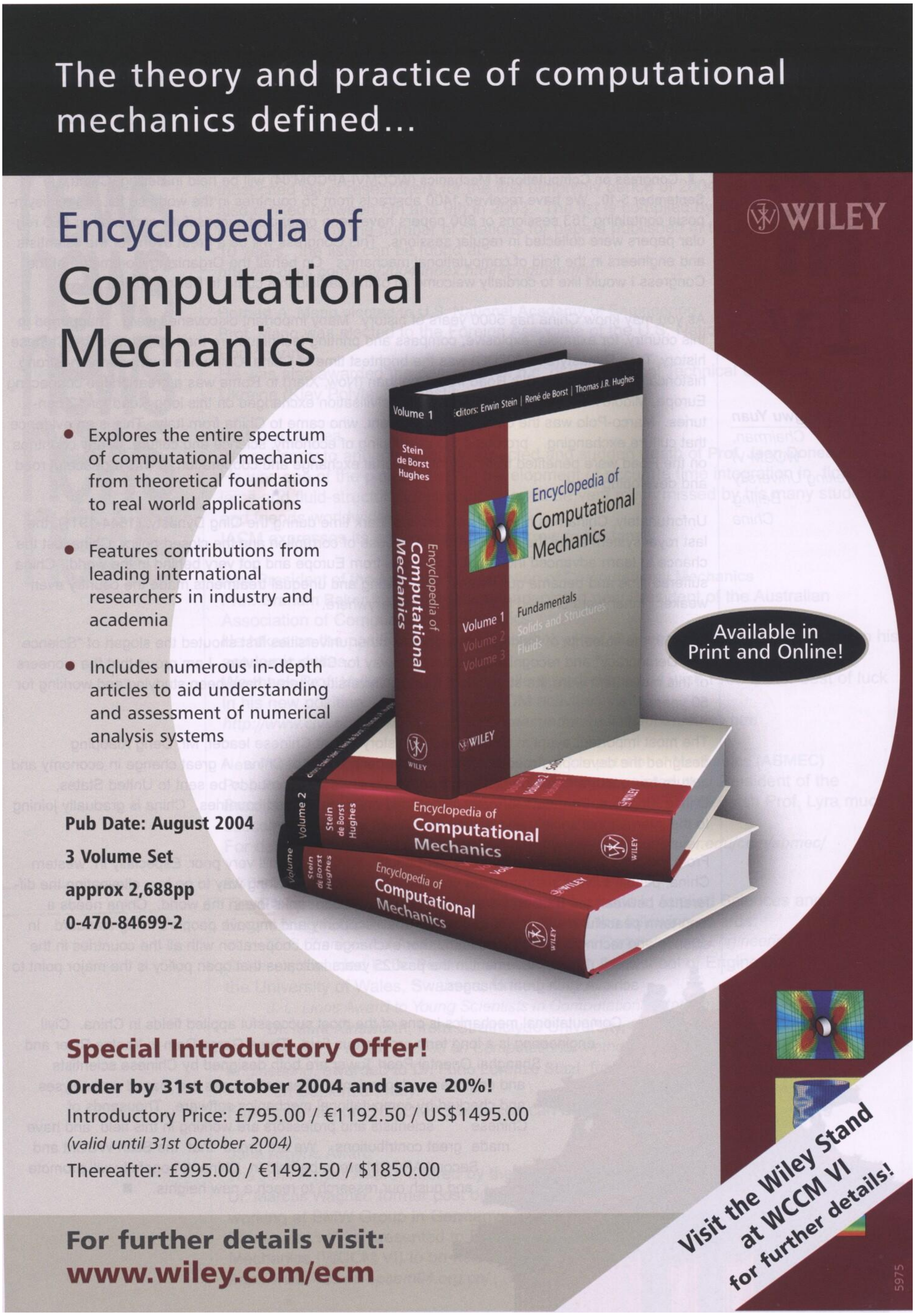
- Contact and Friction
- Fracture Mechanics
- Geomechanics and Soil Mechanics
- Heat Transfer

The Conference provides an ideal opportunity to meet other users, discuss the capabilities of ABAQUS and to meet and question HKS staff. Contact us for more information.



Steady-state thermal, transient thermal and non-linear stress analyses were performed on this model of a cylinder head and associated components, forming a basis for subsequent fatigue assessment. This plot shows the temperature distribution over the flame face for steady state conditions.

Performed by Matrix Stress Analysis Ltd., Leicester, U.K. on behalf of Ulstein Bergen AS, Bergen, Norway.


The theory and practice of computational mechanics defined...
WILEY
Encyclopedia of Computational Mechanics
Explores the entire spectrum of computational mechanics from theoretical foundations to real world applications
Features contributions from leading international researchers in industry and academia
Includes numerous in-depth articles to aid understanding and assessment of numerical analysis systems
Available in Print and Online!
Pub Date: August 2004
3 Volume Set
approx 2,688pp
0-470-84699-2
Special Introductory Offer!
Order by 31st October 2004 and save 20%!
Introductory Price: £795.00 / €1192.50 / US$1495.00
(valid until 31st October 2004)
Thereafter: £995.00 / €1492.50 / $1850.00
For further details visit:
www.wiley.com/ecm
Visit the Wiley Stand at WCCM VI for further details!

COMMERCIAL PROGRAMS (PARTIAL LIST)

USA STRUCTURAL FEA PROGRAMS

ASTROS®	US Air Force, WP-AFB
CSA/NASTRAN®	CSA Corp.
MSC/NASTRAN®	MacNeal-Schwendler
NASTRAN®	US Government, COSMIC
UAI/NASTRAN®	Universal Analytics
ANSYS®	Swanson Analysis Systems
ABAQUS®	Hibbitt, Karlsson, Sorensen
MARC	MARC Analysis Res. Corp.

(18)

ALGOR®	Algor, Inc.
COSMOS/M®	Structural Res. & Analysis
I-DEAS®	Structural Dyn. Res. Corp.
PATRAN®	PDA Engineering
GIFTS®	U. of ARIZONA, H. Kamel
ADINA®	ADINA R & D, Inc.
IMAGES®	Celestial Software
MECHANICA®	RASNA CORP.
mTAB/SAP386®	Structural Analysis, Inc.
NISA®	Engineering Mech. Res. Corp.
SAP IV, SAP6®	Univ. Calif., Ed Wilson
STARDYNE®	STARDYNE
STRUDL®	MIT

(19)

EUROPEAN FEA PROGRAMS

ASAS®	Atkins, England
ASKA®	IKOSS, Germany
BERSAFE®	Nuclear Electric, England
CASTOR®	CETIM, France
DIANA®	DIANA Analysis, Netherlands
FAM®	FEGS Ltd., England
LUSAS®	FEA, England
PAFEC®	PAFEC Ltd., England
PAM-CRASH®	PSI/ESI, France
PERMAS®	INTES, Germany

(20)

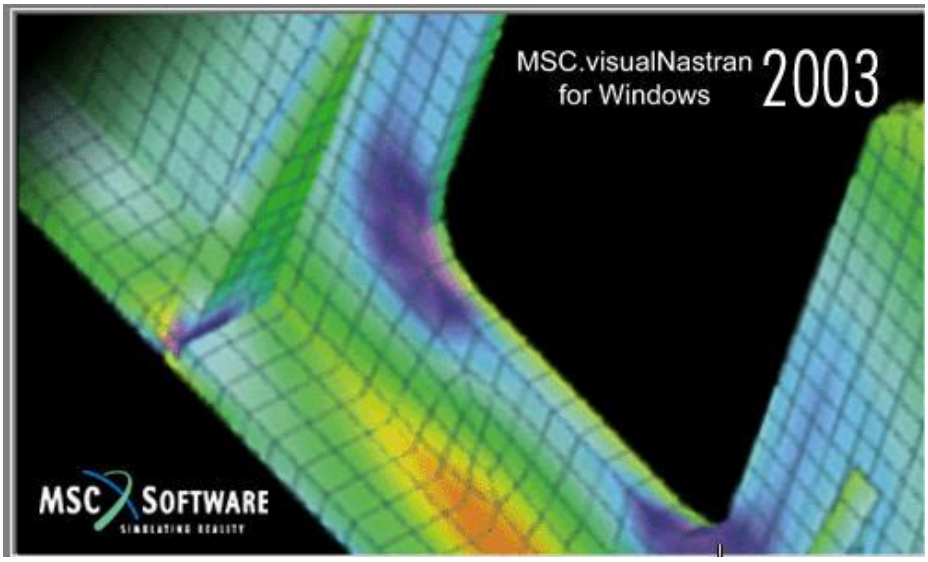
RADIOSS®	France
SAMCEF®	Belgium
SESAM®	VERITAS SESAM SYS., Norway
SOLVIA®	Solvia Engineering, Sweden
SYSTUS®	FRAMASOFT, France

•
•
•

(21)

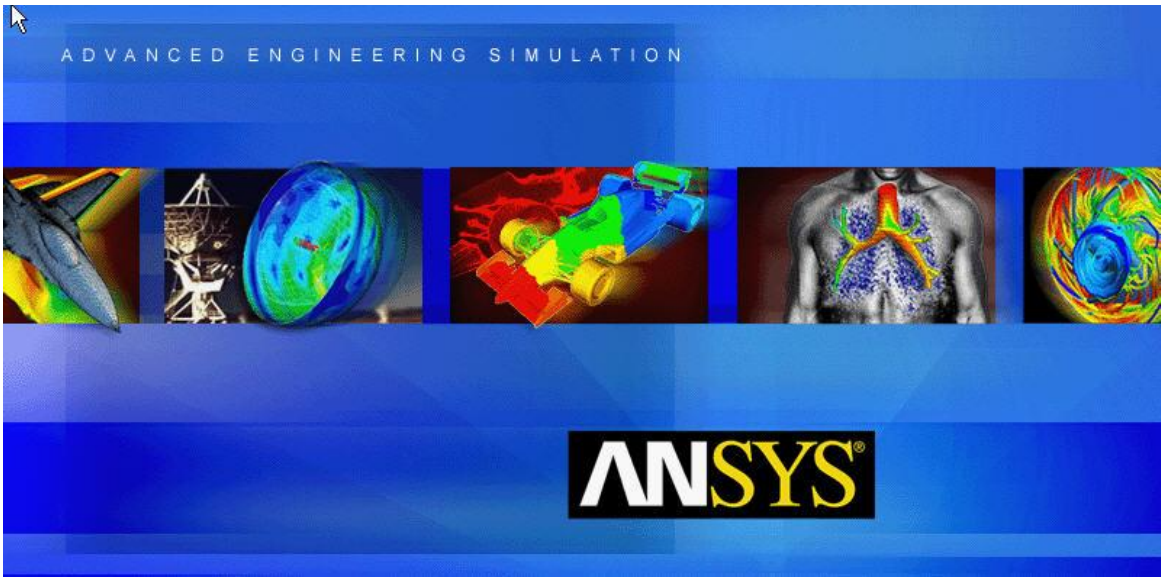
0071 **SOLIDWORKS - ANSYS - NASTRAN**

0072 **Visual NASTRAN for Windows**

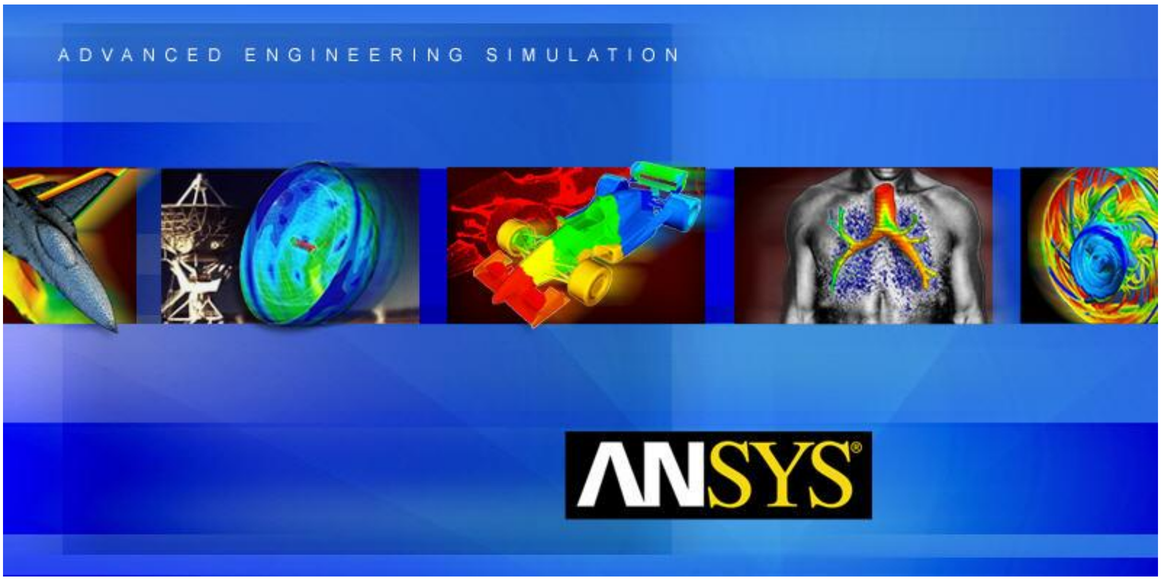


0073 **Visual NASTRAN Desktop**

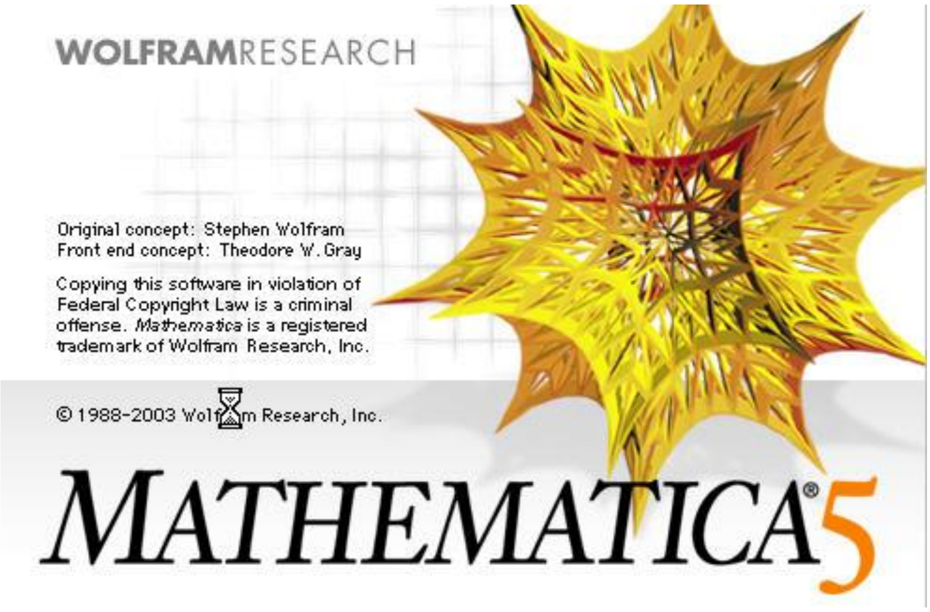
0074 ANSYS CLASSIC



0075 ANSYS WORKBENCH



0076 MATHEMATICA



FEM Terminology

degrees of freedom (abbrv: DOF)

state (primary) variables: displacements in mechanics

conjugate variables: forces in mechanics

stiffness matrix

master stiffness equations

$$\mathbf{K}\,\mathbf{u} = \mathbf{f}$$

$$\mathbf{K}\,\mathbf{u} = \mathbf{f}_M + \mathbf{f}_I$$

Physical Significance of Vectors u and f in Miscellaneous FEM Applications

Application Problem	*State (DOF) vector* **u** *represents*	*Forcing vector* **f** *represents*
Structures and solid mechanics	Displacement	Mechanical force
Heat conduction	Temperature	Heat flux
Acoustic fluid	Displacement potential	Particle velocity
Potential flows	Pressure	Particle velocity
General flows	Velocity	Fluxes
Electrostatics	Electric potential	Charge density
Magnetostatics	Magnetic potential	Magnetic intensity

Introduction to FEM

Attributes of Mechanical Finite Elements

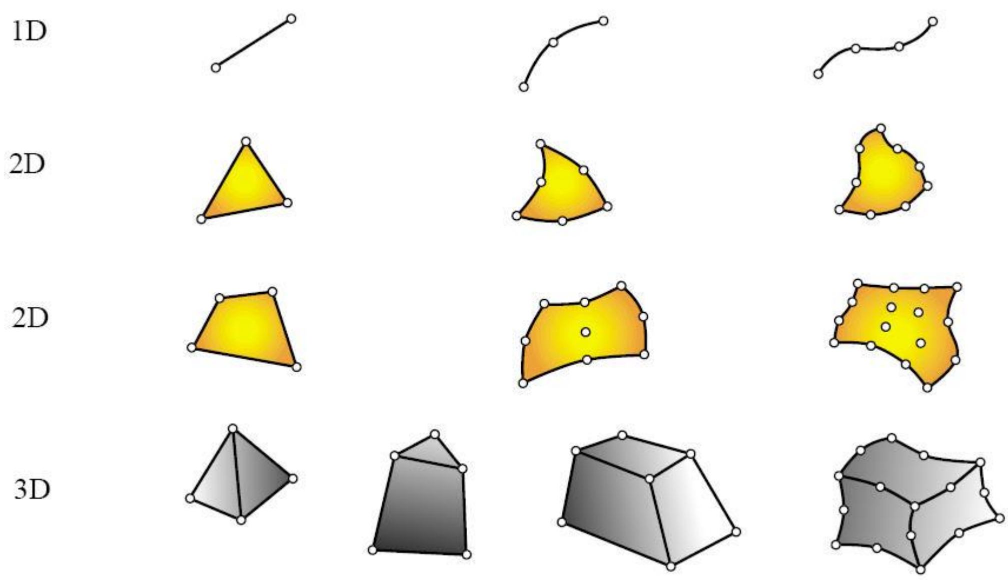
Dimensionality
Nodes serve two purposes
geometric definition
home for DOFs (connectors)

Degrees of freedom (DOFs) or "freedoms"
Conjugate node forces

Material properties
Fabrication properties

Introduction to FEM

Element Geometry Is Defined by Node Locations



Introduction to FEM

Classification of Mechanical Finite Elements

Primitive Structural

Continuum

Special

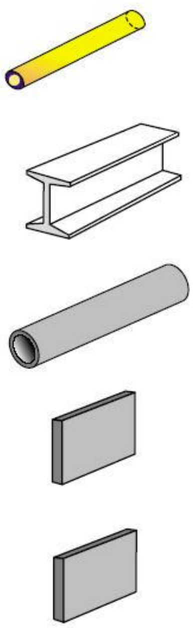
Macroelements } **Superelements**

Substructures } **Superelements**

Introduction to FEM

Primitive Structural Elements
(often built from MoM models)

Physical Structural Component	Mathematical Model Name	Finite Element Discretization



bar

beam

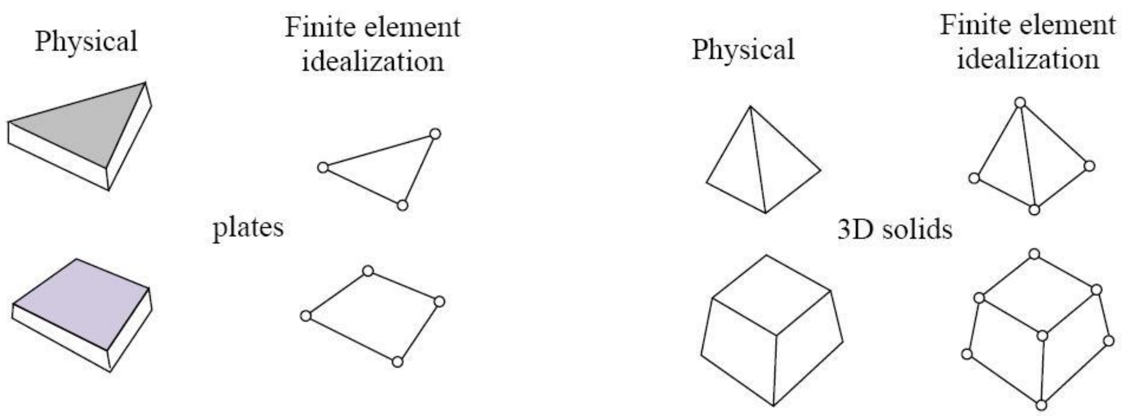
tube, pipe

spar (web)

shear panel (2D version of above)

Introduction to FEM

Continuum Elements



Introduction to FEM

Boundary Conditions (BCs)

The most difficult topic for FEM program users ("the devil hides on the boundary")

Two types
- **Essential**
- **Natural**

Boundary Conditions
Essential vs. Natural

Recipe:

1. **If a BC involves one or more DOF in a** *direct* **way, it is** *essential* **and goes to the Left Hand Side (LHS) of Ku = f**

2. **Otherwise it is** *natural* **and goes to the Right Hand Side (RHS) of Ku = f**