

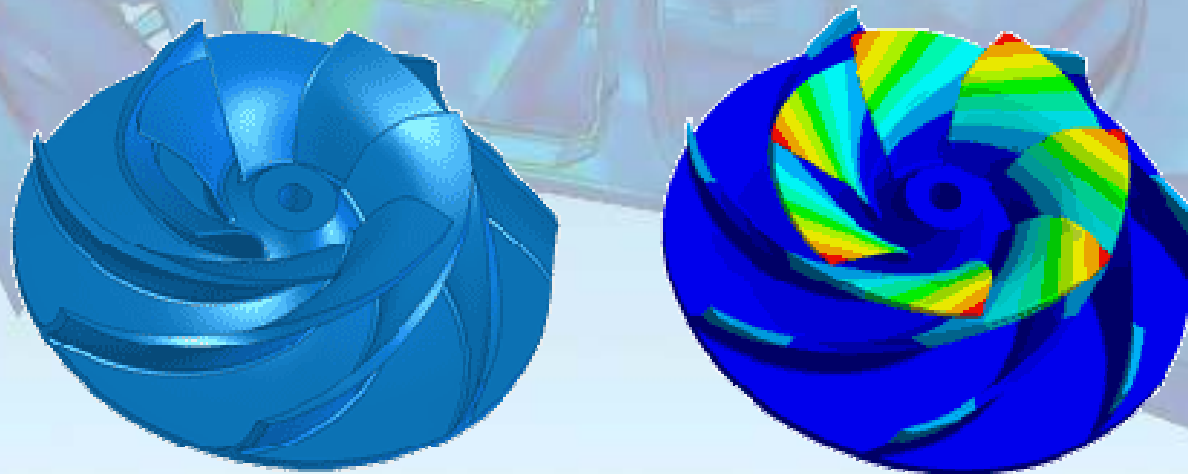


Finite Element Analysis (FEA)

- FEA is the computer application of the Finite Element Method (FEM) where models are created in a computer, stressed and analyzed for specific results.
- FEM is the mesh discretization of a continuous domain into a set of discrete sub-domains and solved using PDEs. It is the method by which FEA relies upon.
- The development of FEA, historically, can be attributed to Courant in 1943.
- However, Aeronautical engineers made amazing progress in the 1950s developing new formulations for elements which were largely unpublished due to company restrictions.
- In 1963, FEA was recognized as an application of the Rayleigh-Ritz method and gained respectability amongst the academic community as having a sound mathematical foundation.
- In the late 1960s and 1970s general-purpose FEA software was being produced but due to the high computing power requirements only large companies with large mainframes could operate the software.



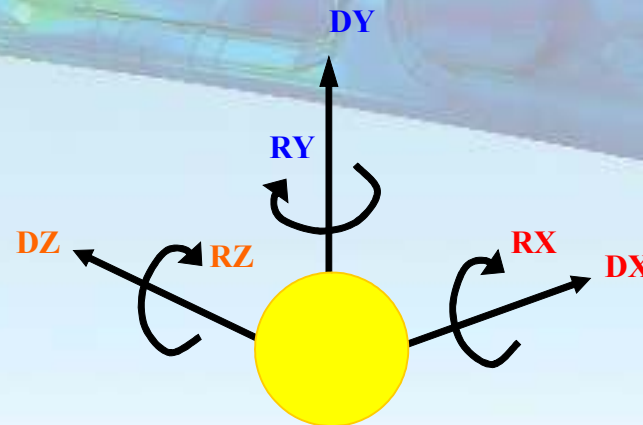
- The tedious nature of FEA meant it was only used for verification and validation in the developing stages of FEA.
- Since then developments in computer technology have gone hand in hand with early developments in FEA.
- With the advent of faster CPU's and graphics capabilities, FEA software is now used extensively as the number one design and analysis tool for design engineers.
- Designs can be created, analyzed and modified with relative ease using modern FEA packages.





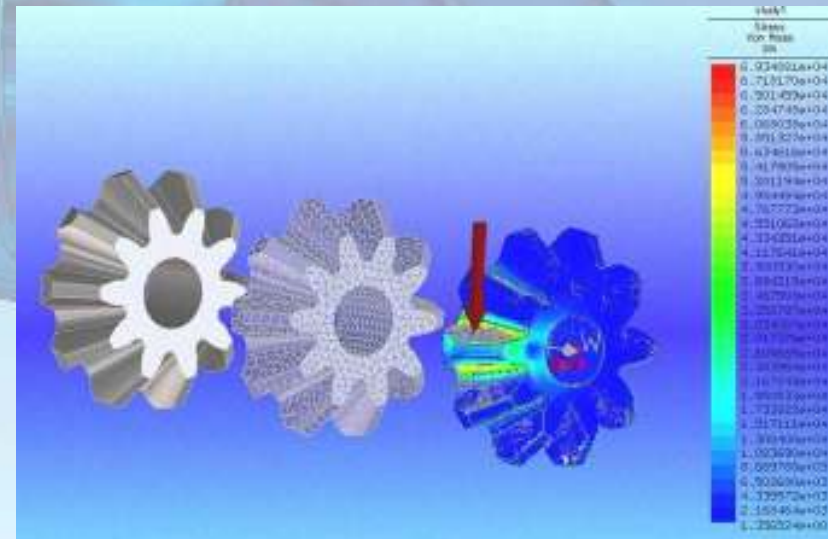
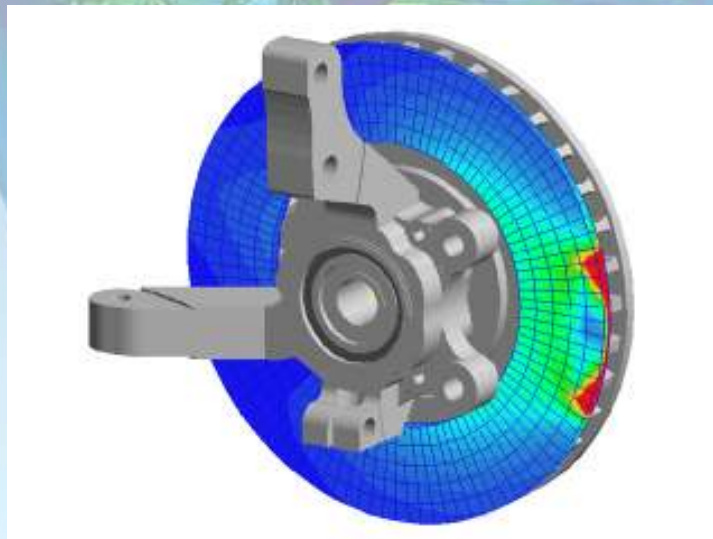
FEA – An Introduction

- The fundamental building block of all finite element models is the **Node**. A **Node** is used to create elements and provides continuity between other elements in the model. A single node has 6 degrees-of-freedom; 3 translational (DX, DY, DZ) & 3 rotational (RX, RY, RZ). These can be manipulated to simulate actual loading and displacement scenarios.
- For a group of elements to sufficiently transfer loads from one end to the other, within a structure, the **Nodes** of individual elements must be physically coincident.





- Connecting nodes are elements. Elements are the basis to which a constitutive law/mathematical model is applied defining the behavior of a material.
- Elements may be assigned material properties, geometry and loads. Elements are combined to form models.
- Elements may take the form of rods/bars/trusses, beams, plates, bricks, links and other higher order elements.





Solving Problems Using FEA

- There are essentially 6 steps involved in formulating a FEA model

1. Problem Classification

2. Mathematical Modeling

3. Preliminary Analysis

4. Finite Element Analysis

5. Check Results

6. Model Revision



Problem Classification

- The analyst must determine what is important for the model in question.
 - Is the problem time-dependent (dynamic), time-independent (static)?
 - Are there any perceived areas of stress concentration? And how will that affect the mesh density?
 - Should buckling be considered?
 - Can the model be simplified using lower order elements?
 - Are the materials linear-elastic or nonlinear?
 - Can we use symmetry to reduce CPU memory requirements?
- Remember – the engineer and not the software is responsible for results obtained. We have an ethical obligation to ensure that what we are doing is Right! If someone dies because of something we designed in FEA that wasn't correct; it is not the software companies problem... it is Yours!



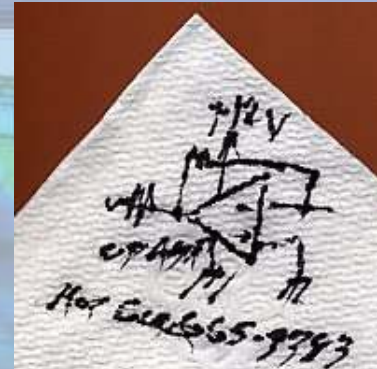
Mathematical Model

- The mathematical model consists of a simplified problem with the analysis theory to be applied in solving it.
- This involves deciding what parts are important and what unnecessary detail can be omitted.
 - i.e. disregard any small geometric irregularities, regard some loads as being concentrated, homogenise composite material properties, idealise materials to be linear-elastic.
- Then we choose the theory which best describes the behaviour of the model.
 - i.e. is the behaviour best described by beam theory, plate-bending theory, plane elasticity, plane strain or plane stress formulations?
- When modelling something in FEA attention must be paid to what you are actually trying to achieve. What results are you looking for? This means weighing up factors like: the required accuracy and the CPU limitations and capabilities.



Preliminary Analysis

- Before solving your Model you should conduct a preliminary analysis; a “back of the napkin calculation” that indicates a ball park figure in which your results should lie.
- This can be done using what ever means are available to you.
 - Analytical calculations
 - Handbook formulas
 - Trusted previous solutions
 - Experimental results
 - Research papers
- If you have some reference material for which you can compare your FEA results, the natural tendency of searching for answers to support the FEA is reduced.
- In some cases the problem may be too complicated to do such a short calculation





Finite Element Analysis

- There are three commonly used steps employed in general purpose FEA:
 1. *Preprocessing*: This is basically creating the model, inputting geometry, material properties, boundary conditions and loads. Some software provides automatic mesh generation algorithms but some will require extensive user input to build the desired mesh. Thought must be given to symmetry possibilities, mesh element size, element aspect ratios, geometric limitations etc.
 2. *Numerical Analysis*: This is where the software automatically generates element stiffness matrices that represent the behaviour of each element, combines them into the global element stiffness matrix equations that represents the complete structure, and solves these equations to determine values at the nodes.
 3. *Post Processing*: The results of the FEA are displayed graphically with the user determining which quantities they wish to look at. Most software will output tables, deformation, stresses and much more



Results Check

- Prior to accepting the results ask yourself: “do the results look right?”
 - Are there any obvious errors?
 - Isn't there supposed to be an expected symmetry?
 - Does the structure deform in an expected manor?
 - Are the boundary conditions correctly simulating the actual deformation of the structure?
 - Are there any unexpected points of high stress concentrations?
 - Has the mesh deformed too much?
 - Has the mesh deformed too little?
 - Are there any irregularities in the stress contours?
 - Am I displaying the results correctly?
- The FEA results should be compared with the preliminary analysis to see if there are any major discrepancies.



Model Revision

- Model Revision allows you to make changes to the model in order to improve the outputted results.
 - Based on the results of the FE Analysis is the current model adequate?
 - Do I need to make changes in order to extract the relevant results?
 - How can I improve the boundary conditions to better represent the actual behaviour?
 - Can I change the element types to achieve a better solution?
 - Can I optimise the model for faster solution times?
 - Am I able to use symmetry to increase the mesh resolution?
 - Is the mesh too coarse?



Basic Elements

M. Daniel K. Wood

- Rod/Bar/Truss – 1 dof (1 axial translation), Capable of taking only axial tension and compression loads (automatically assigned pin-jointed ends). Used to simulate pin jointed frames and trusses.
- 2D Beams – 3 dof (2 translations, 1 rotation) Capable of taking axial tension, compression and bending loads. Used simulate welded frames, structural load bearing beams in 2D.
- 3D Beams – 6 dof (3 translations, 3 rotations), full motion simulation in all in and out-of-plane situations. Used simulate welded frames, structural load bearing beams in 3D.
- Linear Triangle (Constant Strain Triangle, CST, Tri3) – linear displacement field thereby producing a constant strain field. CST's are unable to predict linearly varying stresses and strains across their boundaries. Can not capture pure bending.
- Bilinear Rectangle (Quad4) – Has a linear displacement field, can not capture inplane bending.



Model Creation in Strand7

- There are essentially 5 steps involved when creating a FE model in Strand7 are:

Select appropriate units (Newtons, millimetres)

1. Create Nodes (Input nodal coordinates)
2. Create elements
 - Define mesh and areas of importance
 - Refine mesh (subdivide mesh at critical areas)
 - Input material properties and sectional geometry (Young's Modulus, Poisson's Ratio, Shear Modulus, CTE, membrane thickness, sectional properties etc)
3. Enforce boundary conditions
 - Simulate local restraints (restraints that apply to individual nodes)
 - Simulate global restraints (restraints that apply to the entire model)
4. Apply loads
 - Solve first using check only mode ([Model checking and quality control.pdf](#))
5. Solve

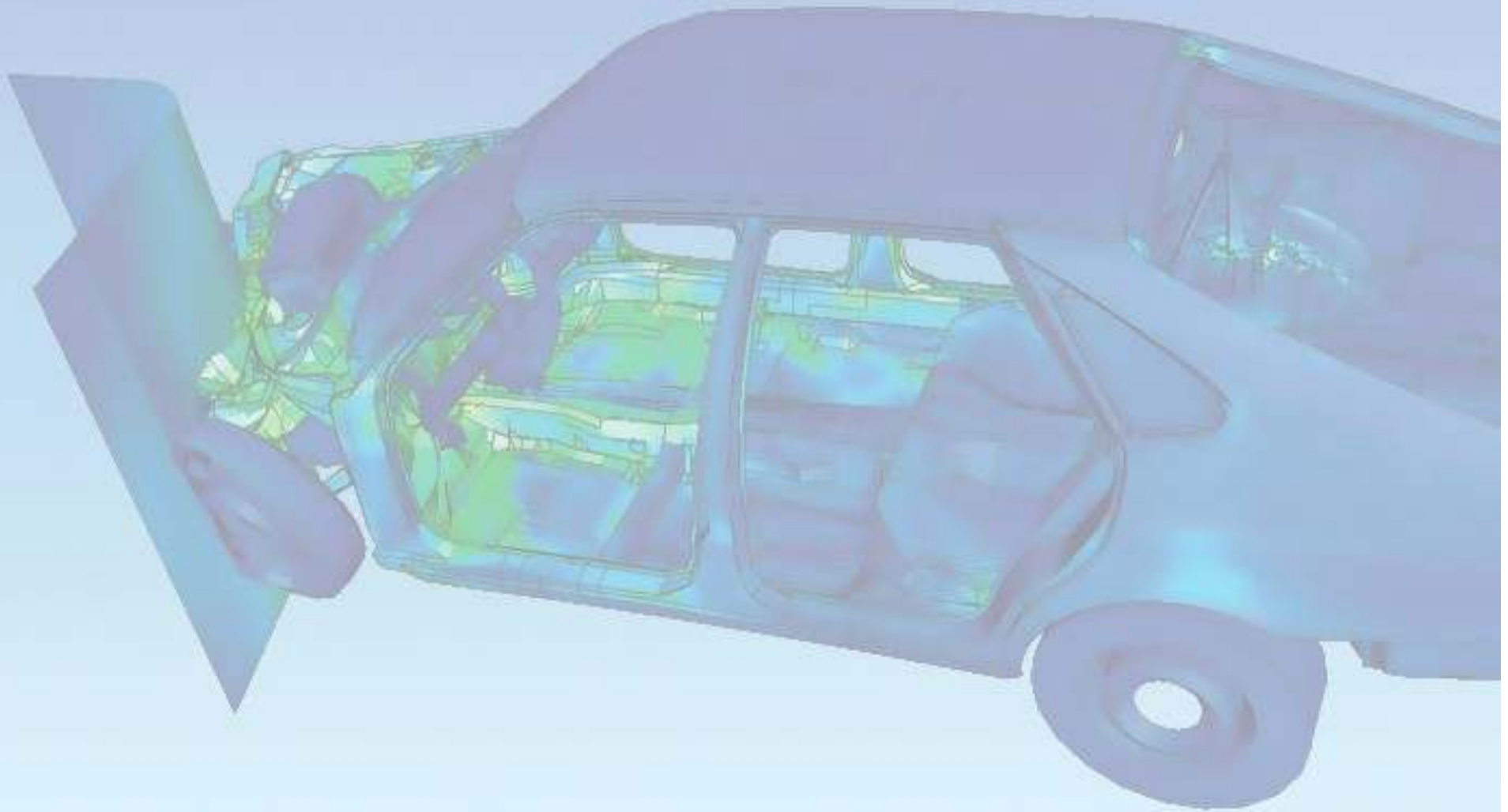
SAAB model



The University
of the South Pacific

School of Engineering – MM211

M. Daniel K. Wood



MM211-001
10/10/2010 10:10:10 AM
10/10/2010
10/10/2010 10:10:10 AM