COMPUTER AIDED ENGINEERING DESIGN (BFF2612)

FINITE ELEMENT MODEL

by
Dr. Mohd Nizar Mhd Razali
Faculty of Manufacturing Engineering
mnizar@ump.edu.my

COMPUTER-AIDED ENGINEERING (CAE)

Definition:

- Is used to analyse CAD geometry, allowing the operator to *simulate* and study how the product will behave so the design can be refined and optimized.
- To observe how the product will behave and catch any errors early in the design cycle thus optimizing the design and reducing overall product development time and cost.

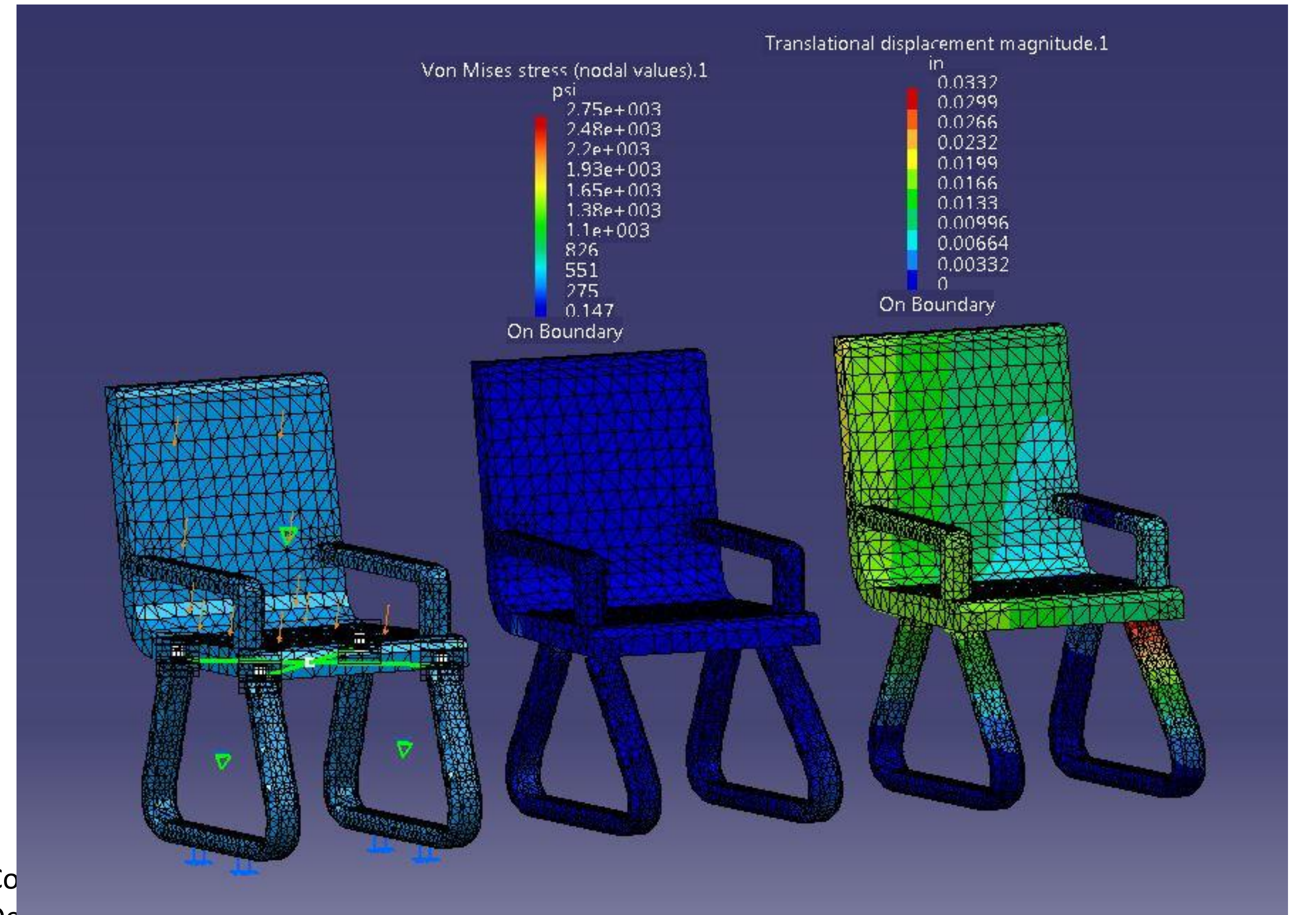
WHAT IS FEM OR FEA ?

- Finite element is the most popular numerical analysis technique for obtaining approximate solutions to a wide variety engineering problems.
- It is an efficient design tool by which designers can perform parametric design studies by considering various design cases (different shapes, materials, loads, boundary conditions, and so forth), analysing them, and choosing the optimum design.
- The method is based on (1) dividing a complex shape into small elements, (2) solving the equilibrium equations at hand for each element, and then (3) assembling the elements' results to obtain the solution to the original problem.

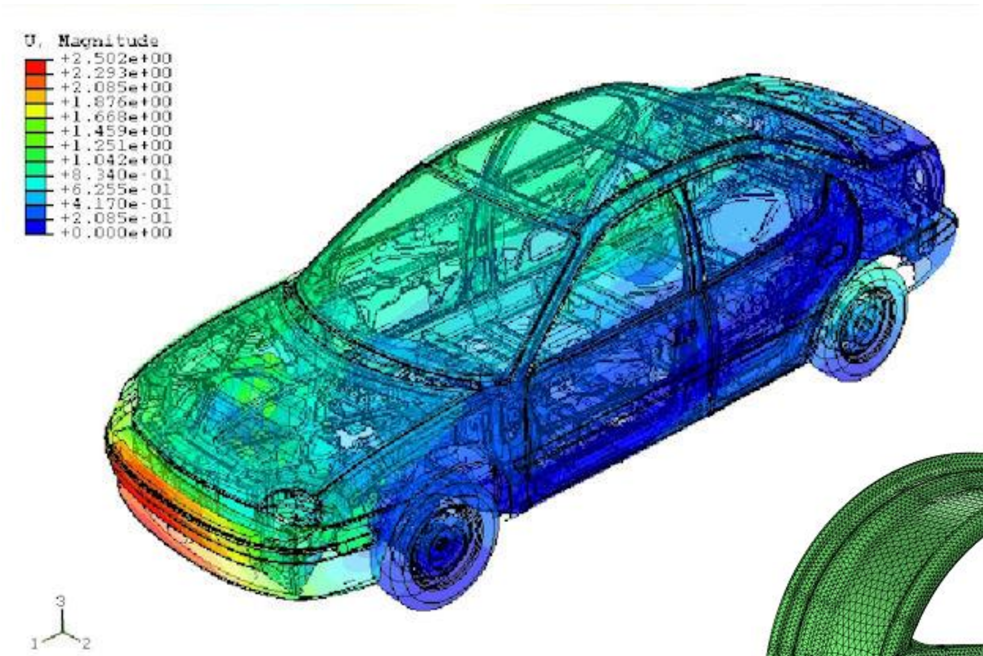
APPLICATIONS OF FEM

- Solid mechanic analysis (static, dynamic)
- Deformation studies
- Fluid-flow analysis
- Heat-flow analysis (conduction, convection, radiation)
- Magnetic-flux studies
- Acoustic analysis

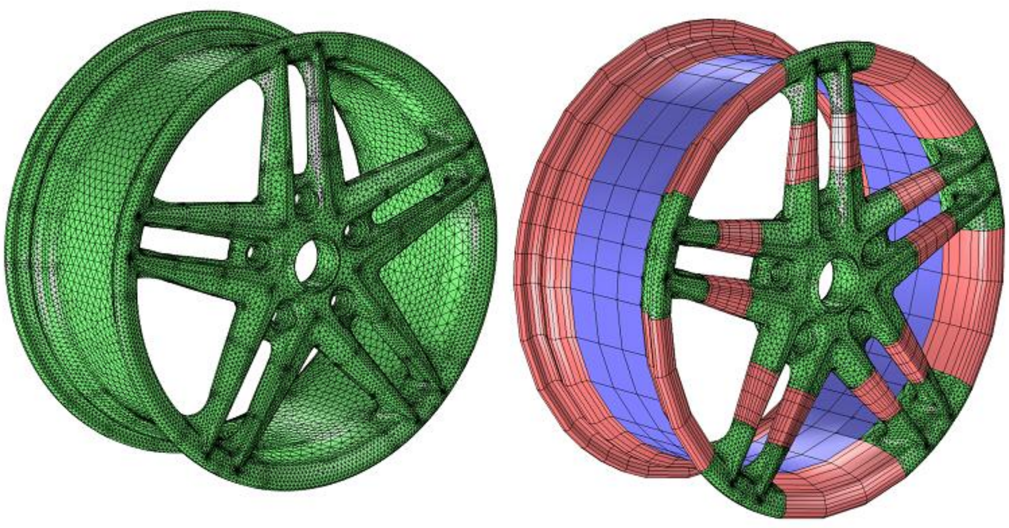
CHAIR


Von Mises stress (nodal values).1
psi
2.75e+003
2.48e+003
2.2e+003
1.93e+003
1.65e+003
1.38e+003
1.1e+003
826
551
275
0.147
On Boundary
Translational displacement magnitude.1
in
0.0332
0.0299
0.0266
0.0232
0.0199
0.0166
0.0133
0.00996
0.00664
0.00332
0
On Boundary

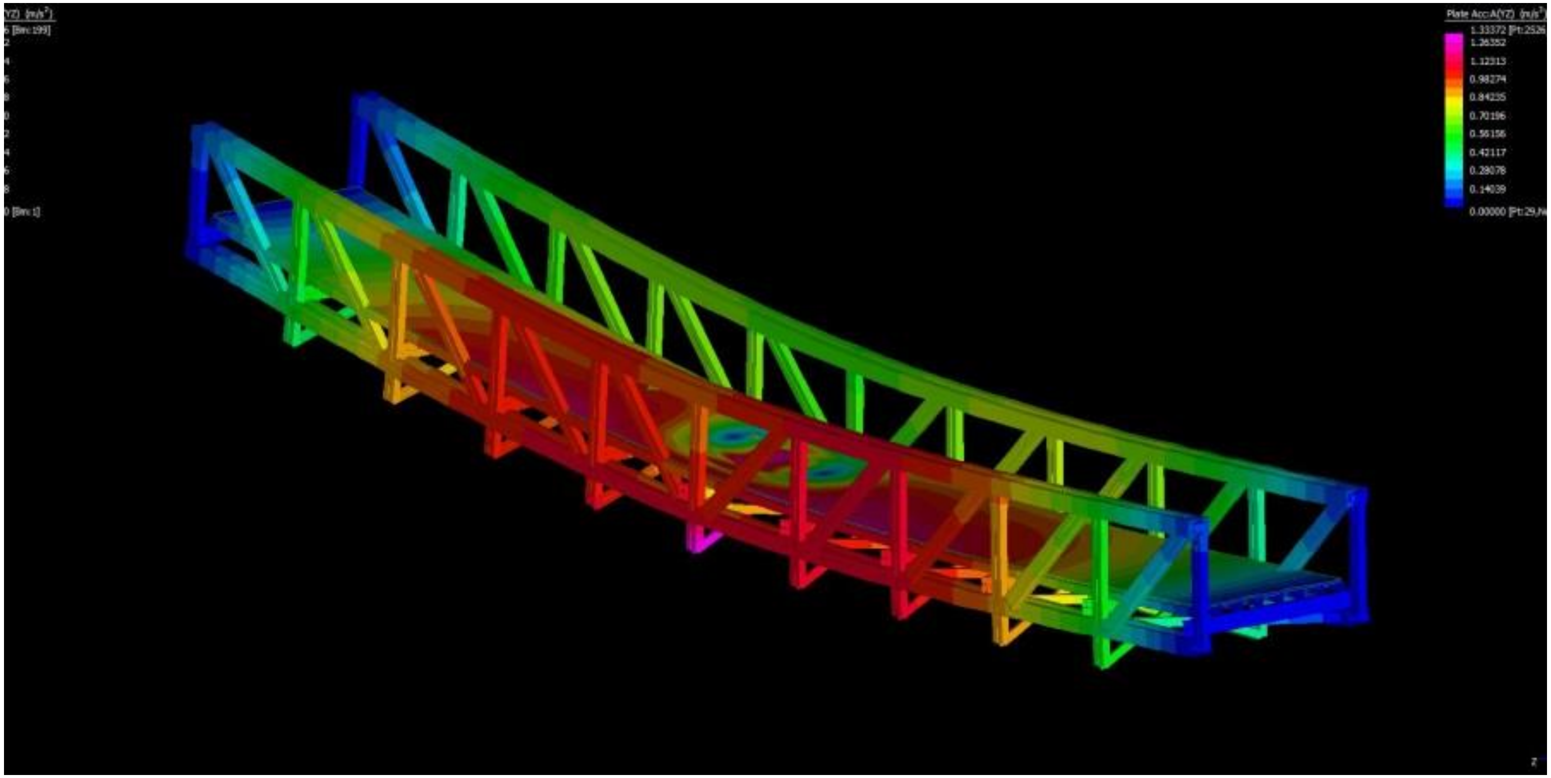
VEHICLE


U, Magnitude
+2.502e+00
+2.293e+00
+2.085e+00
+1.876e+00
+1.668e+00
+1.459e+00
+1.251e+00
+1.042e+00
+8.340e-01
+6.255e-01
+4.170e-01
+2.085e-01
+0.000e+00
3
1
2

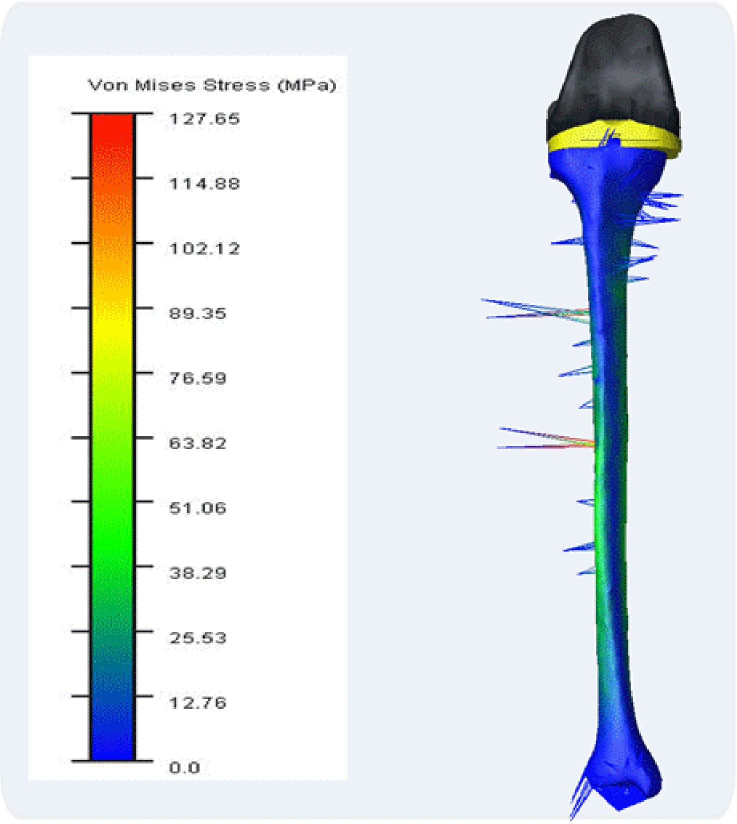
RIM

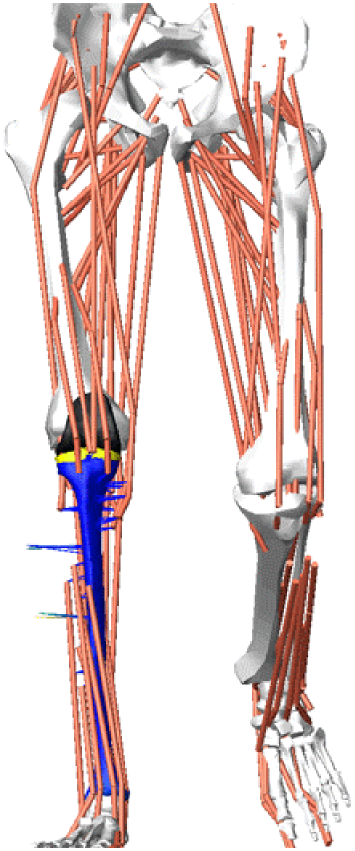


BRIDGE STRUCTURE



BIO-MEDICAL DEVICES



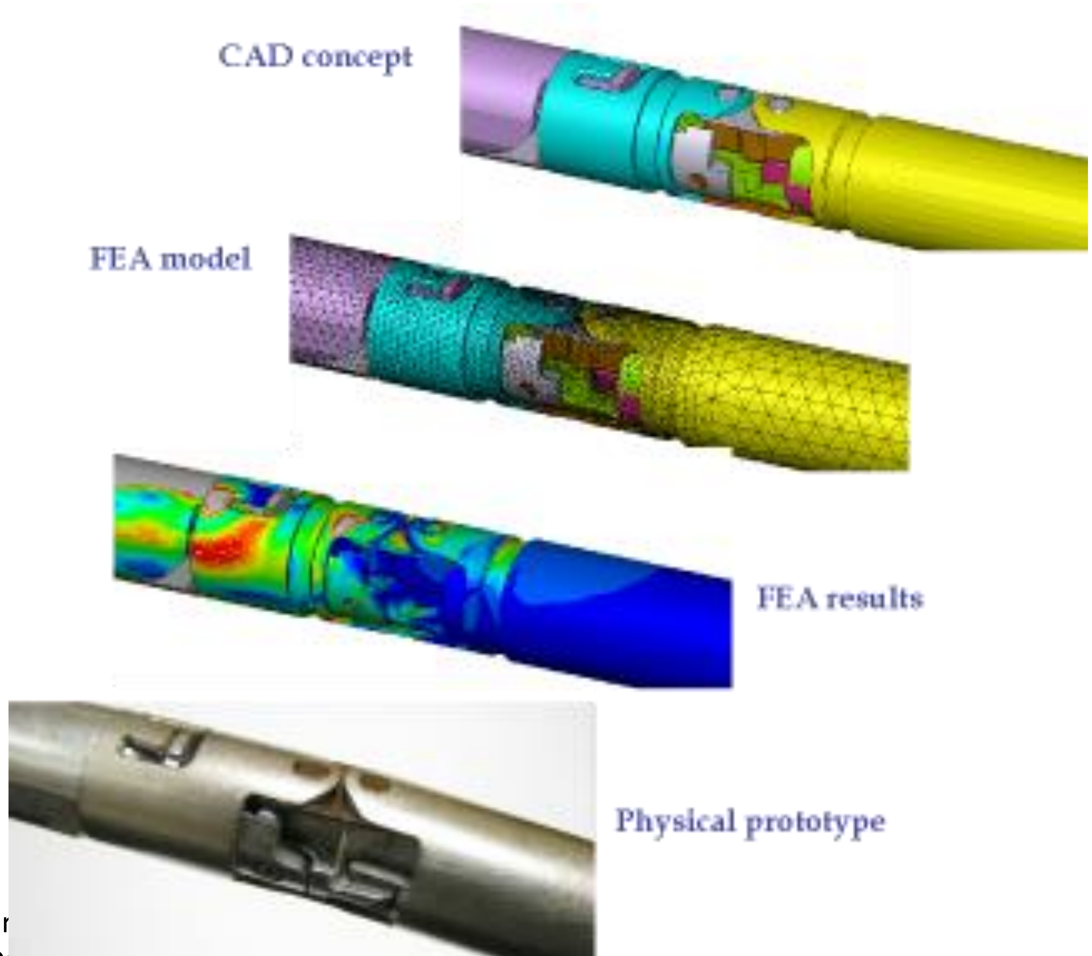


PROSTHESIS

ADVANTAGES OF FEM

- Parts with irregular geometries are difficult to be analysed by the use of conventional strength of material approaches. In FEM, any complex geometry can be analysed with ease.
- Parts made from different materials can be analysed using FEM.
- It is possible to analyse parts that have complex loading patterns with multiple types of forces acting on the geometry with large number of supports.
- The FEM procedure provides results throughout the part (all points).
- It is easy to change the model in FEM and generate a number of options with 'what if' scenarios. This helps in developing faster prototypes and speeds up time to market by shortening the design cycle.
- Testing of products can be done using FEM without expensive destructive testing.

FINITE ELEMENT PROCEDURE



FINITE ELEMENT PROCEDURE

Physical problem

FEM (generate nodes, elements, boundary conditions, material properties, loads, data file)

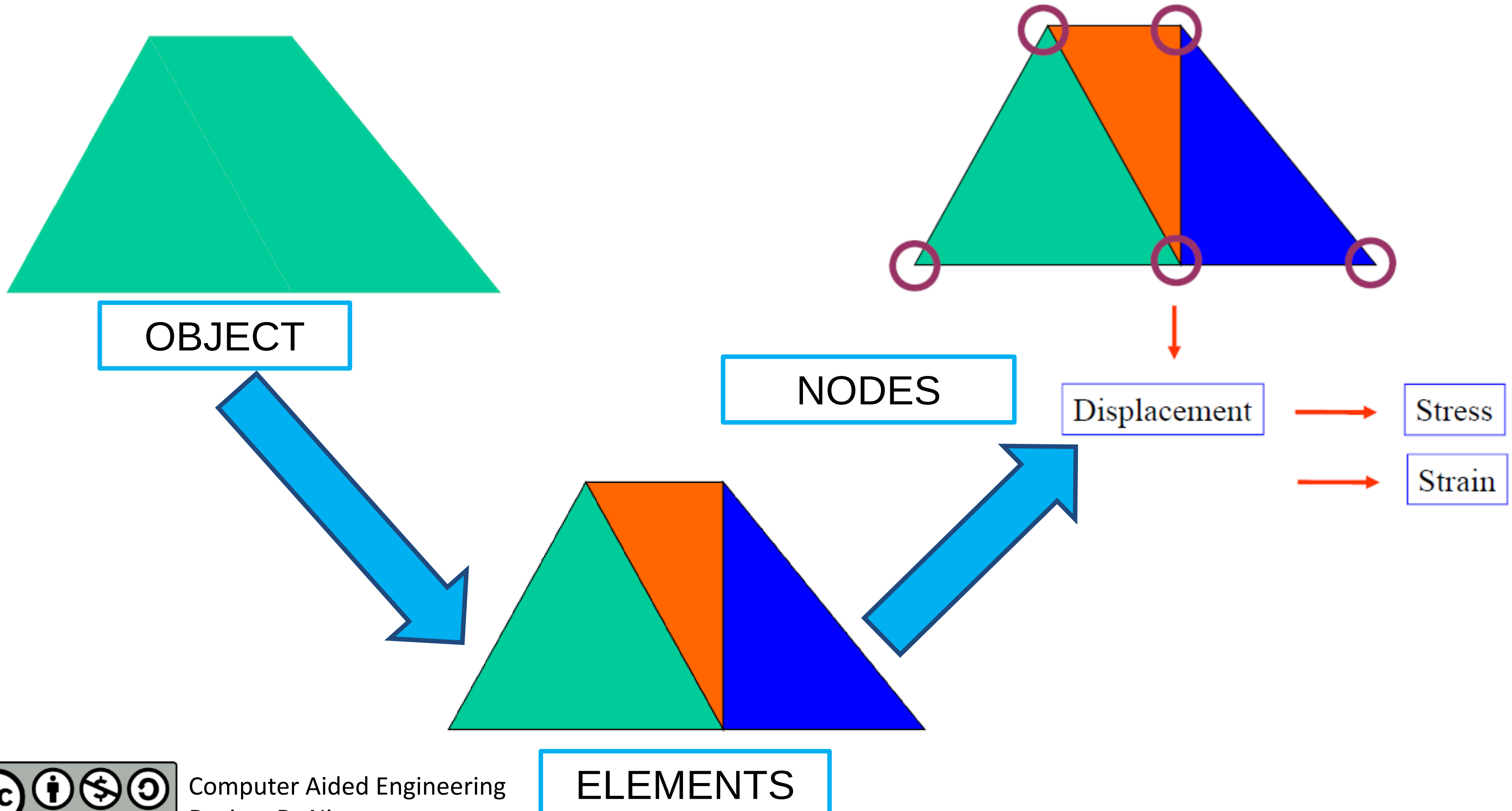
PREPROCESSOR

FEA (generate element matrices, compute nodal values, and derivatives, store results)

Analyse results (display curves, contours, deformed shapes)

POSTPROCESSOR

FINITE ELEMENT ANALYSIS



TERMS IN FEM

- **Mesh:** subdivided part geometry. The process of subdivision is called meshing.
- **Element:** small volume that divide the part geometry in finite element analysis to solve it easily.
- **Node (nodal point):** a set of points in each element. Nodes are usually located at the corners or endpoints of elements. In the higher-order elements, nodes are also placed on side or face as well as possible the interior of the element.
- **Degrees of freedom:** specify the state of the element. Normally, each node has six degree of freedom in static analysis. These are three linear displacements along the rectangular coordinate axes and the other three are the rotary motion about these axes.

TERMS IN FEM

- **<u>Nodal forces:</u>** forces that applied on the finite element model. The concentrated forces at the nodes.

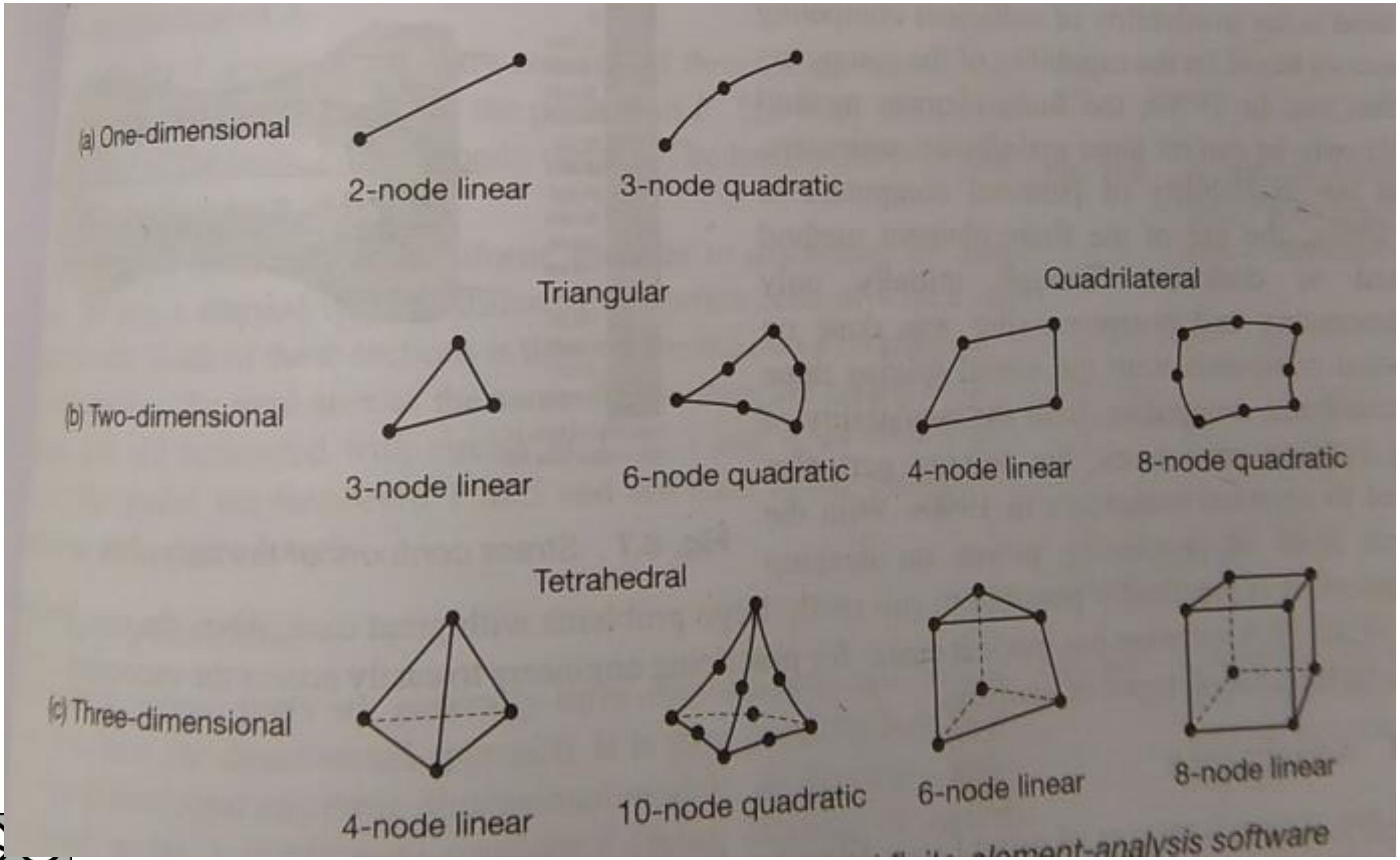
- **<u>Boundary conditions:</u>** specify the current state of some nodes in the finite element mesh. This is a way to specify how some of the nodes in the model are constrained.

- **<u>Dimensionality:</u>** element that are used in FEM can have intrinsic dimensionality of one, two or three space dimensions. This dimensionality can be expanded for higher dimensions by kinematic transformation. For example, a one-dimensional element such as a bar or a beam may be used to build a model in 2D or 3D space.

CRITICAL DECISIONS IN FEM

- Type of analysis
- Number of nodes
- Degree of freedom (components of the field variable) at each node
- Element shape and type
- Material type
- External loads
- Boundary conditions
- Interpretation of the results.

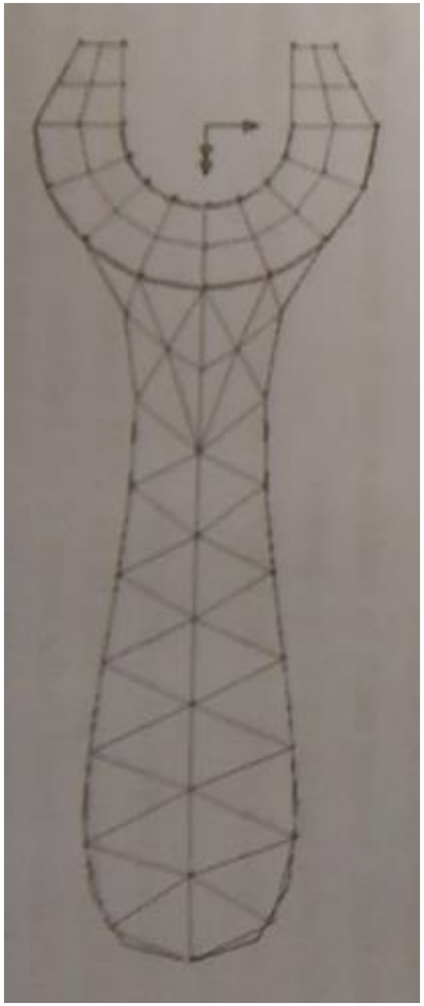
ELEMENT SHAPE



Design: Dr Nizar

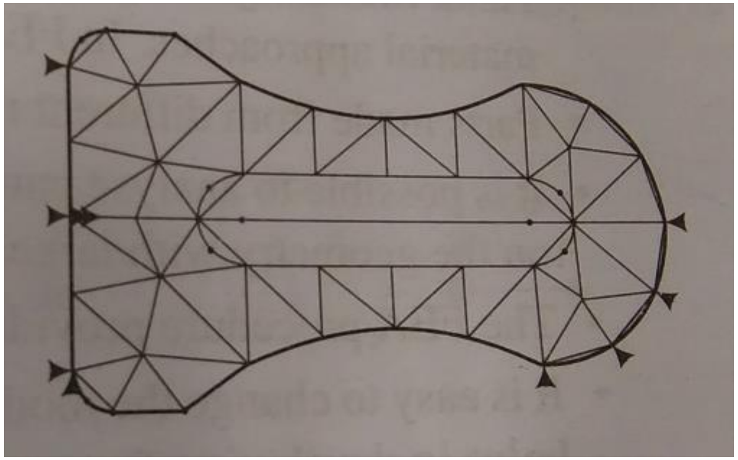
EXAMPLE (MESHING)

CAD

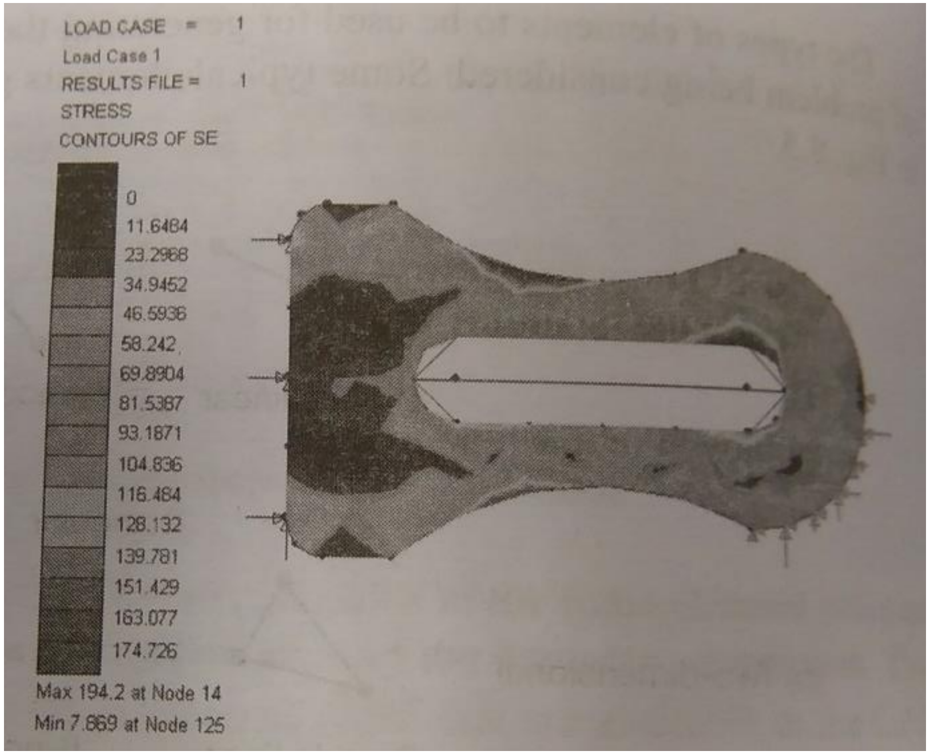


Generated mesh using FEA

EXAMPLE



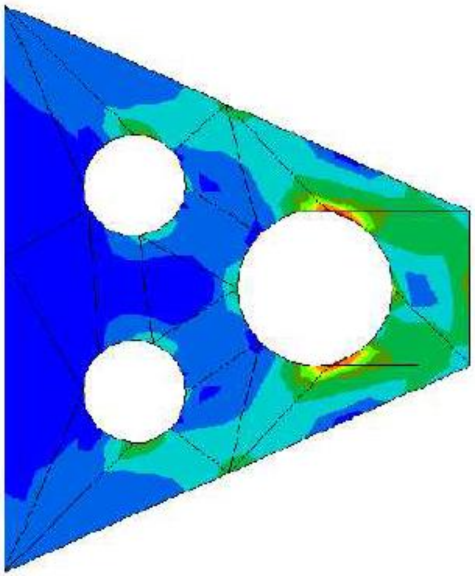
Cam plate showing the geometry, the finite element mesh and the boundary conditions used



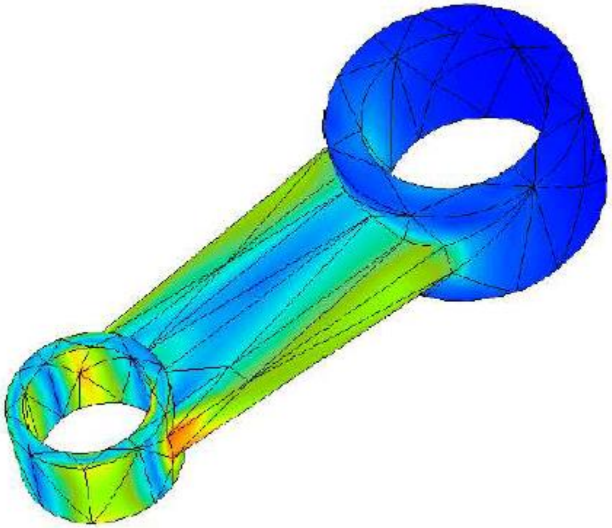
Stress contours of the cam plate

EXAMPLE

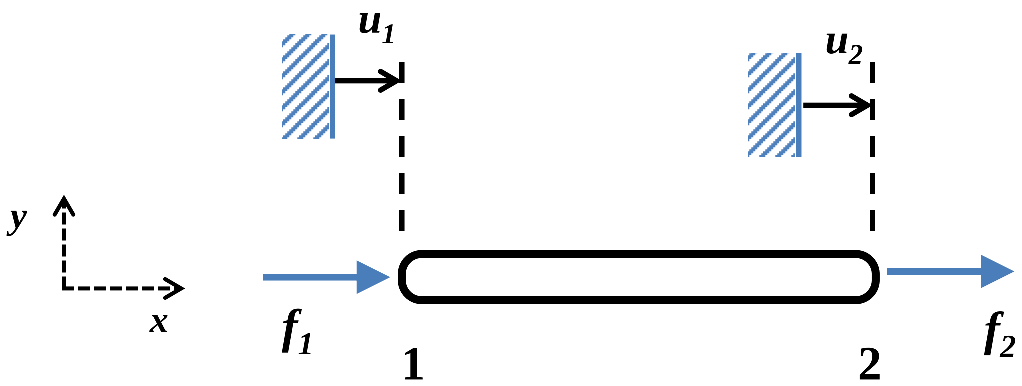
Examples of FEA - 2D



Examples of FEA – 3D



TRUSS (ONE BAR) ELEMENT (TWO-FORCE MEMBERS)



L = original length (m)
A = cross-sectional area (m²)

$$\varepsilon = \frac{u_2 - u_1}{L} \qquad \sigma = \frac{f}{A} \qquad E = \frac{\sigma}{\varepsilon}$$

$$\frac{EA}{L}(u_2 - u_1) = f \quad \text{or} \quad k(u_2 - u_1) = f$$

Where $\boxed{k = \frac{EA}{L}}$

STIFFNESS

E is Young's Modulus (Pa or N/m²)

u is element nodal displacement vector.

F is element nodal load vector.

TRUSS ELEMENT (TWO-FORCE MEMBERS)

Note that $f = f_2 = -f_1$, therefore

$$k(u_1 - u_2) = f_1$$

$$k(u_2 - u_1) = f_2$$

The pair of equations can be put into matrix equation form:

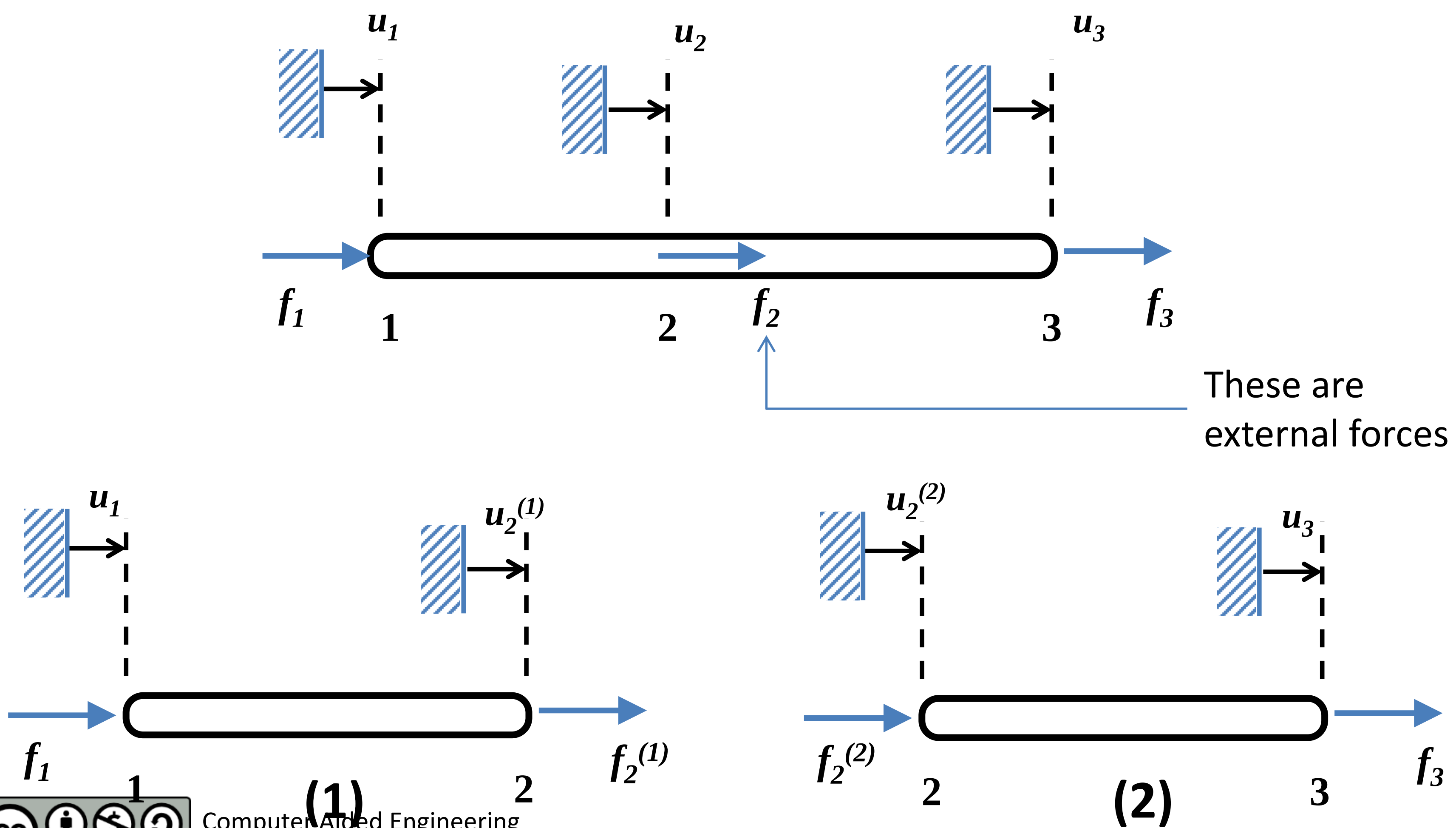
$$k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}$$ or $$[k]\{u\} = \{f\}$$

unknown

Where $[k] = k\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$

$[k]$ is stiffness matrix for one element

TWO BAR ELEMENTS



TWO BAR ELEMENTS

The general matrix equation for each element:

$$k_e \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}^{(e)} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}^{(e)}$$

Where e denotes the element number. Hence for elements 1 and 2:

$$k_1 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ {u_2}^{(1)} \end{Bmatrix} = \begin{Bmatrix} f_1 \\ {f_2}^{(1)} \end{Bmatrix} \Rightarrow k_1 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ {f_2}^{(1)} \end{Bmatrix}$$

$$k_2 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} {u_2}^{(2)} \\ u_3 \end{Bmatrix} = \begin{Bmatrix} {f_2}^{(2)} \\ f_3 \end{Bmatrix} \Rightarrow k_2 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} {f_2}^{(2)} \\ f_3 \end{Bmatrix}$$

Displacement compatibility conditions:

$$u_2^{(1)} = u_2^{(2)} = u_2$$

TWO BAR ELEMENTS

Expand each equation in matrix form:

$$k_1\begin{bmatrix}1 & -1 & 0\\ -1 & 1 & 0\\ 0 & 0 & 0\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}f_1\\ {f_2}^{(1)}\\ 0\end{Bmatrix}$$

$$\begin{bmatrix}k_1 & -k_1 & 0\\ -k_1 & k_1 & 0\\ 0 & 0 & 0\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}f_1\\ {f_2}^{(1)}\\ 0\end{Bmatrix}$$

$$k_2\begin{bmatrix}0 & 0 & 0\\ 0 & 1 & -1\\ 0 & -1 & 1\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}0\\ {f_2}^{(2)}\\ f_3\end{Bmatrix}$$

$$\begin{bmatrix}0 & 0 & 0\\ 0 & k_2 & -k_2\\ 0 & -k_2 & k_2\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}0\\ {f_2}^{(2)}\\ f_3\end{Bmatrix}$$

Summing member by member:

$$\begin{bmatrix} & & 0\\ & & \\ 0 & & \end{bmatrix}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}f_1\\ {f_2}^{(1)}+{f_2}^{(2)}\\ f_3\end{Bmatrix}$$

TWO BAR ELEMENTS

So that, the stiffness matrix:

$$[K] = \begin{bmatrix} & 0 \\ 0 & \end{bmatrix}$$

<u>*The stiffness matrix is:*</u>

1. *Symmetric*. This is a consequence of the symmetry of the forces (equal and opposite to ensure equilibrium).

2. *Singular* and therefore not invertible. That is because the problem as defined is incomplete and does not have a solution: boundary conditions are required.

FEA FOR MULTIPLE (MANY) ELEMENTS

$$\{F\} = [K].\{U\}$$

Array of applied forces
(one for each DOF)

Matrix of stiffnesses
(DOF x DOF)

Array of displacements
(one for each DOF)

$\{F\}$ is “known” (loads)

$[K]$ is “known” (geometry, material properties … elements)

$\{U\}$ is to be determined (displacements)

This can be solved mathematically using a matrix inversion method

$$\{F\} = [K].\{U\} \longrightarrow \{U\} = [K]^{-1}\{F\}$$

(First nodal quantity)

Universiti Malaysia PAHANG
Engineering • Technology • Creativity

FEA FOR MULTIPLE (MANY) ELEMENTS

$$\{U\} = [K]^{-1}.\{F\}$$

Once the displacements {U} are known, then strains and stresses can be determined:

$\varepsilon = \frac{\Delta u}{L}$ (1-D … more complicated for 2-D and 3-D strains)

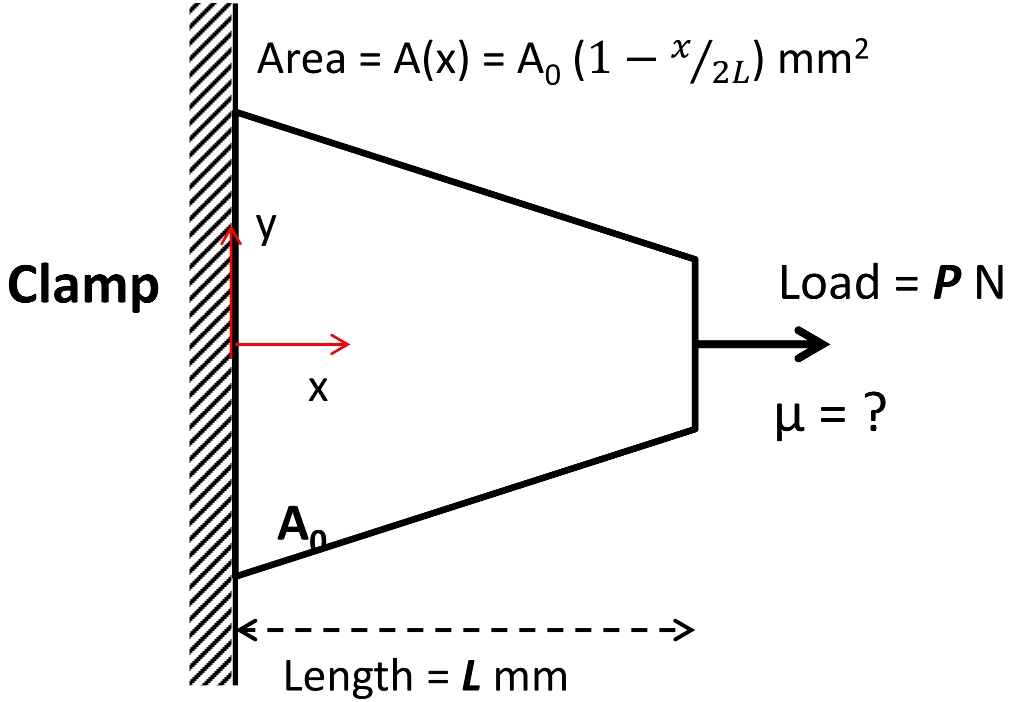
and $\sigma = E.\varepsilon$

And FOS (factor of safety) $= \frac{\sigma_y}{\sigma}$

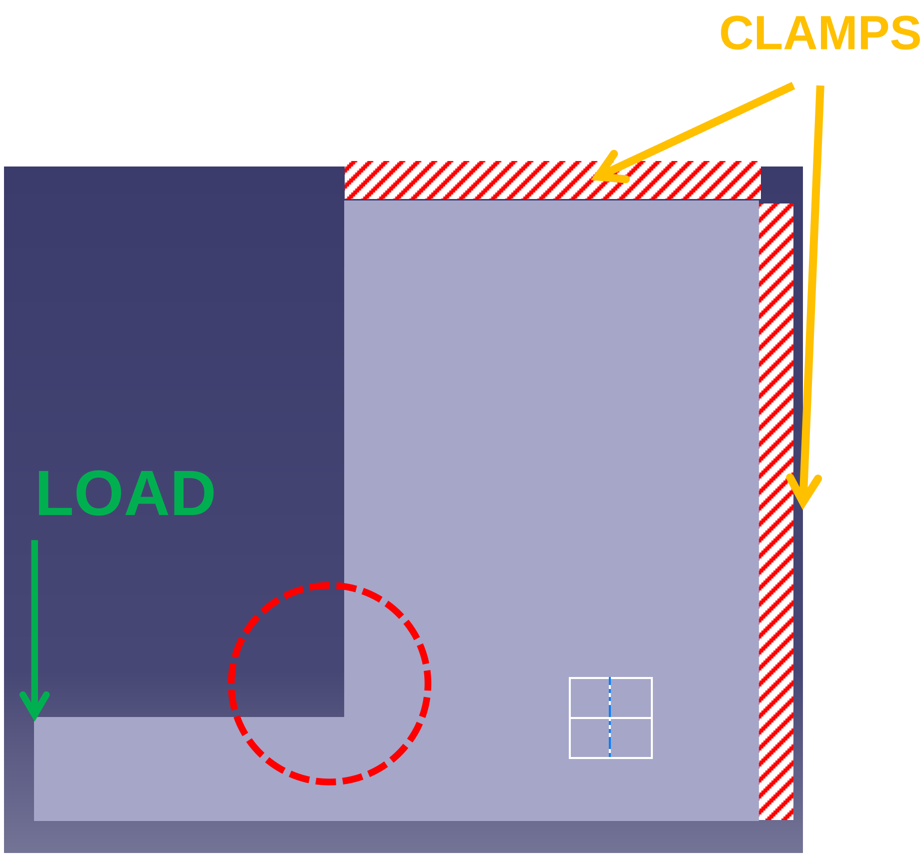
(second nodal quantities)

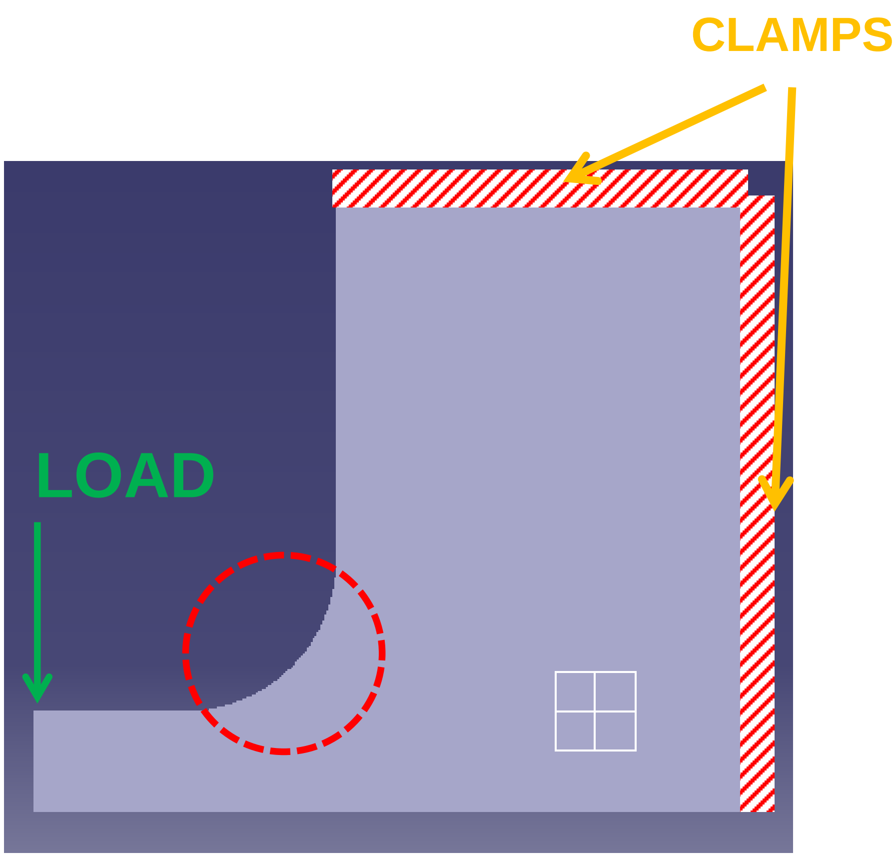
EXAMPLE OF CALCULATION

Finite Element Case for 2D element.



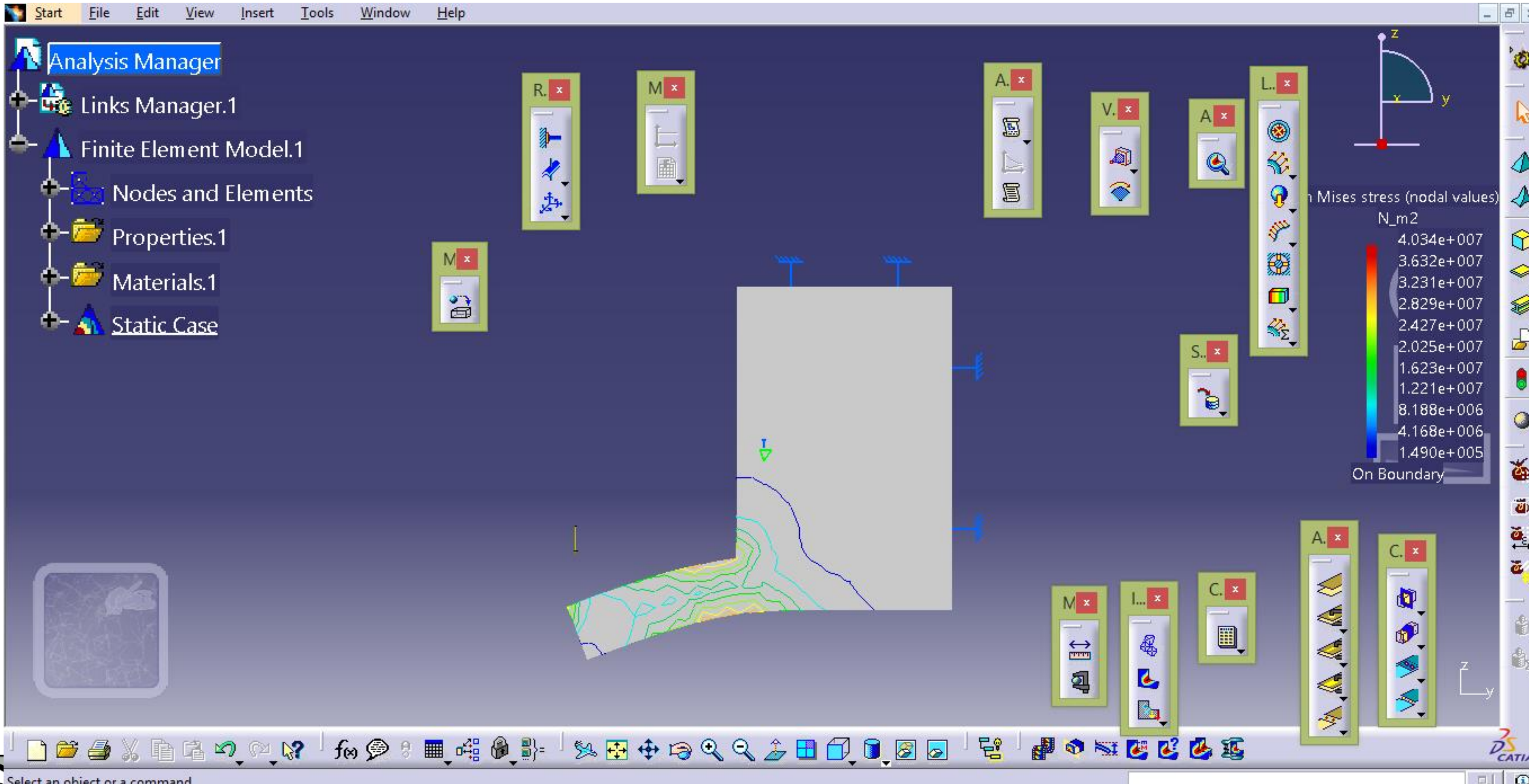
STRESS CONCENTRATION





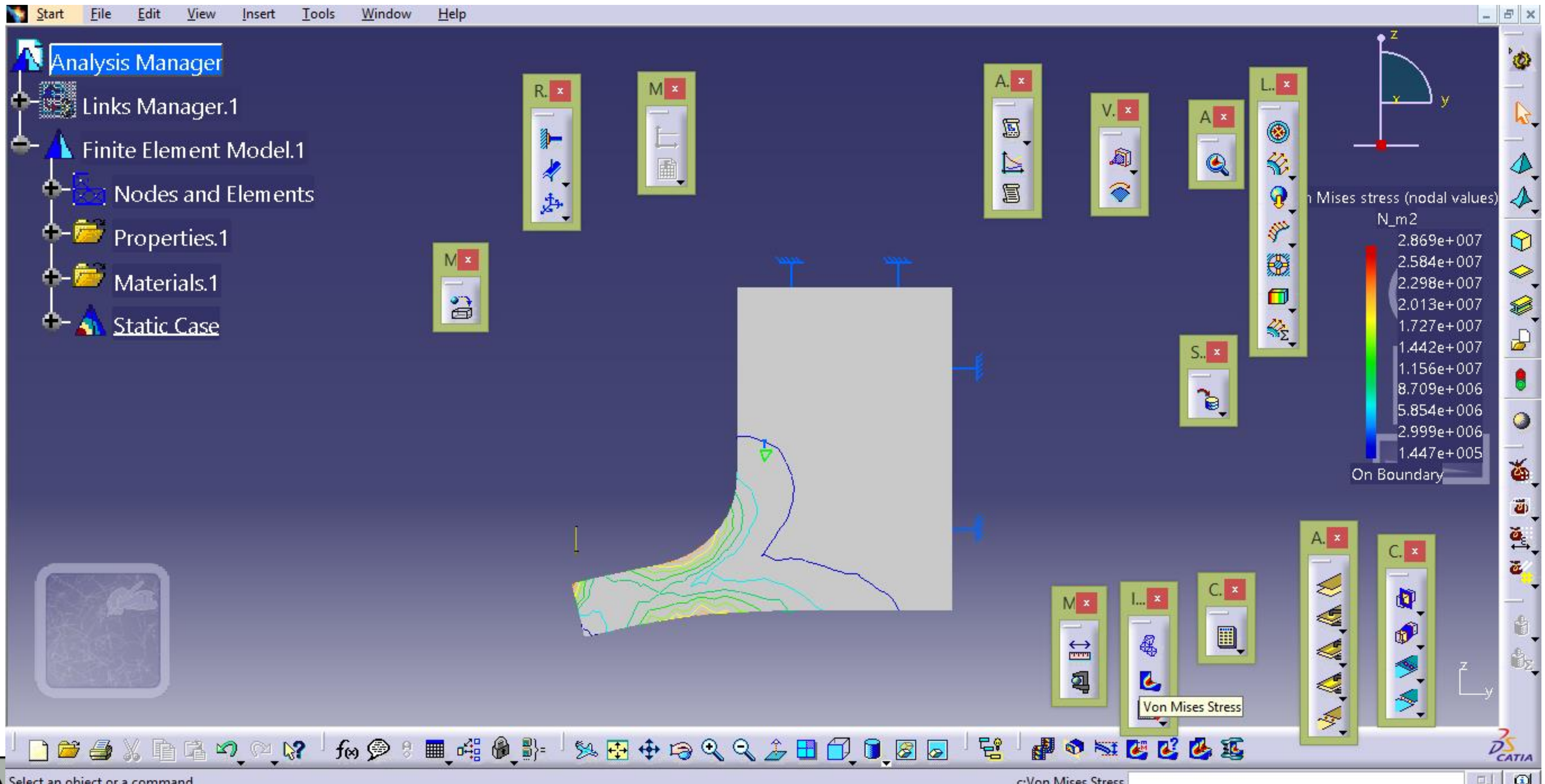
Which one has higher stress?

FEM Result



Design: Dr Nizar

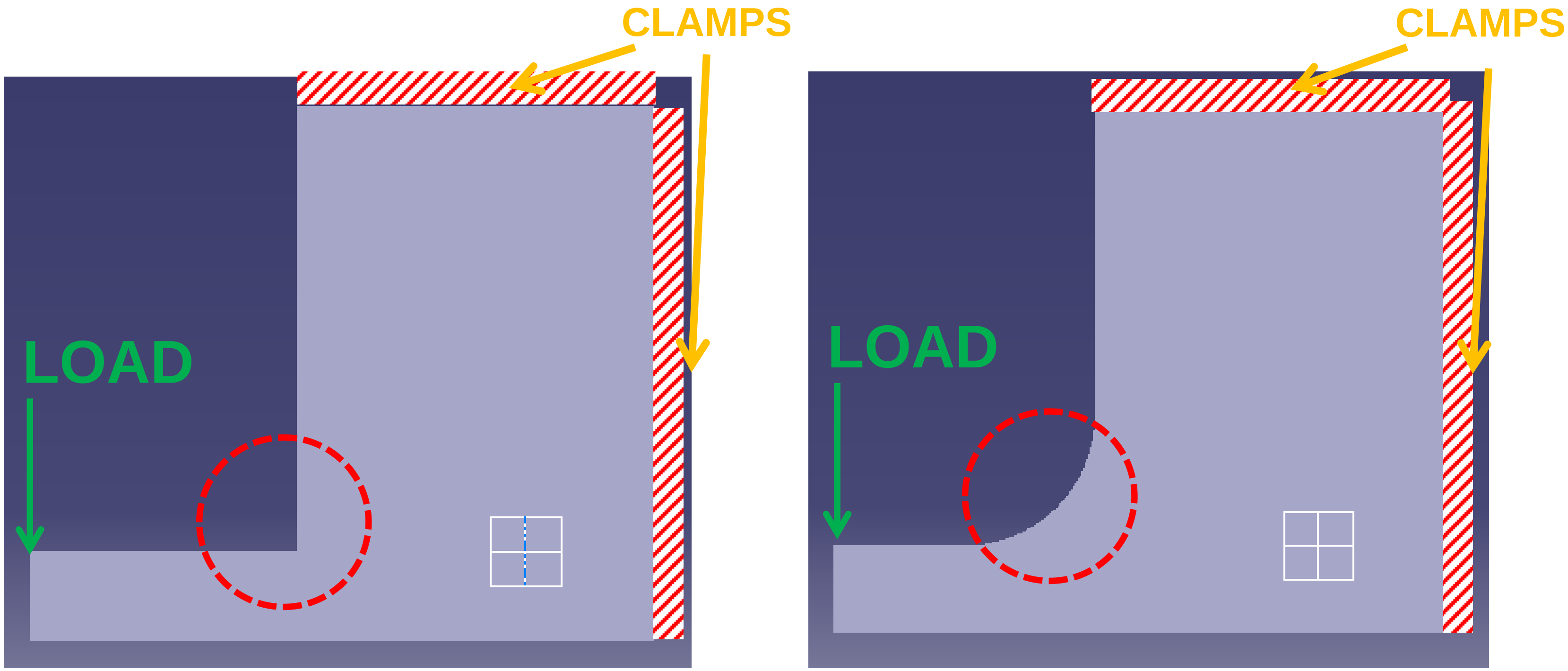
FEM Result



Design: Dr Nizar

MESH REFINEMENT (FINER ELEMENT SIZE)

To get more accurate results at specific location of interest.



Where should you apply mesh refinement?

Have any questions?

Thank you
and Have a nice day!

COMPUTER AIDED ENGINEERING DESIGN (BFF2612)

Dr. Nizar