THE FINITE ELEMENT METHOD

A short tutorial giving an overview of the history, theory and application of the finite element method.

Disclaimer!

This tutorial is not a substitute for detailed study of the body of knowledge of the finite element methodology.

FEM – The Finite Element Method

The FEM is a computer aided mathematical technique for obtaining numerical solutions to the abstract equiations of calculus that predict the response of physical systems subjected to external influences.

The FEM solution may be exact for the approximated model of the real system.

Finite Element vs Finite Difference

- Finite Difference
    - Used for problems dealing with time only as independent variable
    - Mathematical technique based on power (Taylor) series expansion

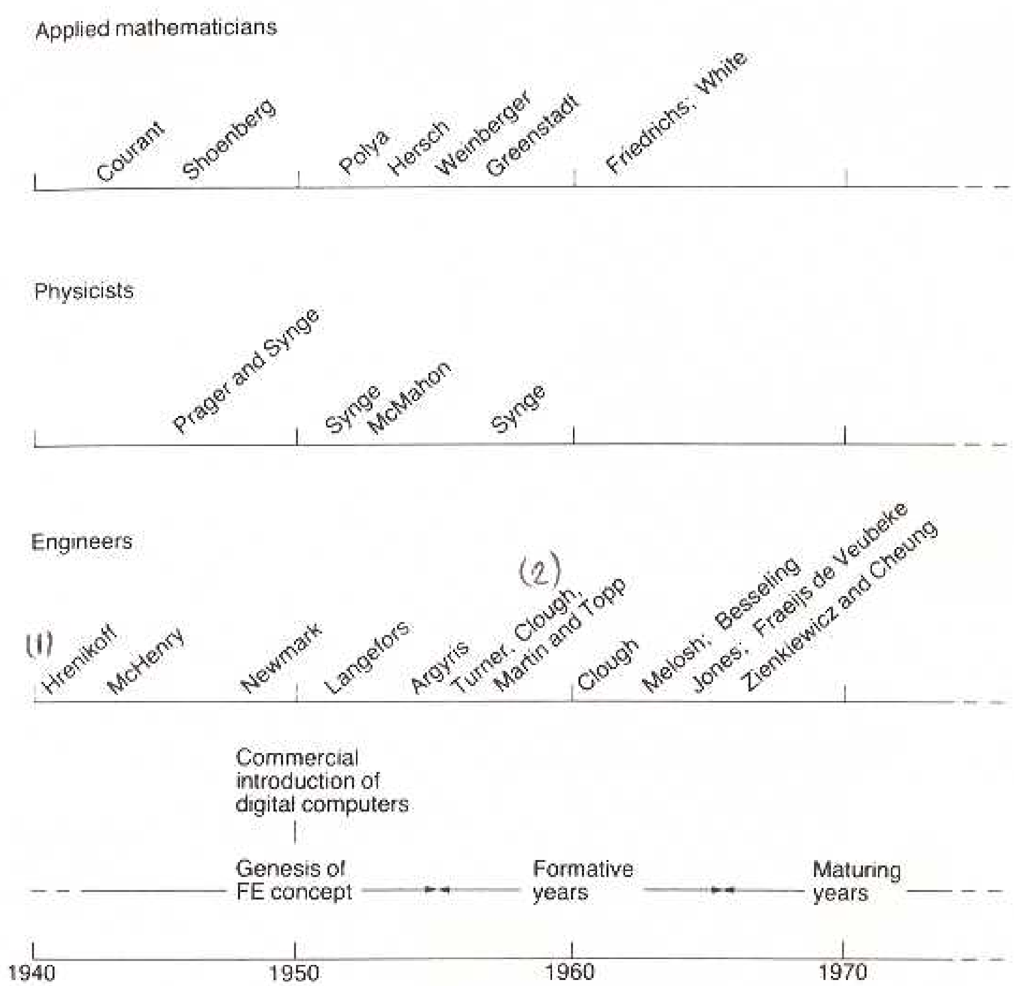
$$f(x) = f(a) + h \cdot f'(a) + \frac{h^2}{2!} \cdot f''(a) + ...$$

    - The increments or elements are of equal size, where “h” = (x-a)

      The “h” are of fixed size

- Finite Element
    - Deals with both space (x, y, z) and time (t) as independent variables
    - Mathematical technique based on intricate use of algebraic expressions and optimizing techniques
    - Elements can vary in size within the domain of the system

Historical Background



Notes:

(1) Used framework of physically separate 1D rods and beams to model elastic behaviour of a continuous plate

(2) Used assembly of triangular panels to model complete aircraft wing panel; used all the methodological elements of modern day FEM

Value of FEM for Plant Facilities

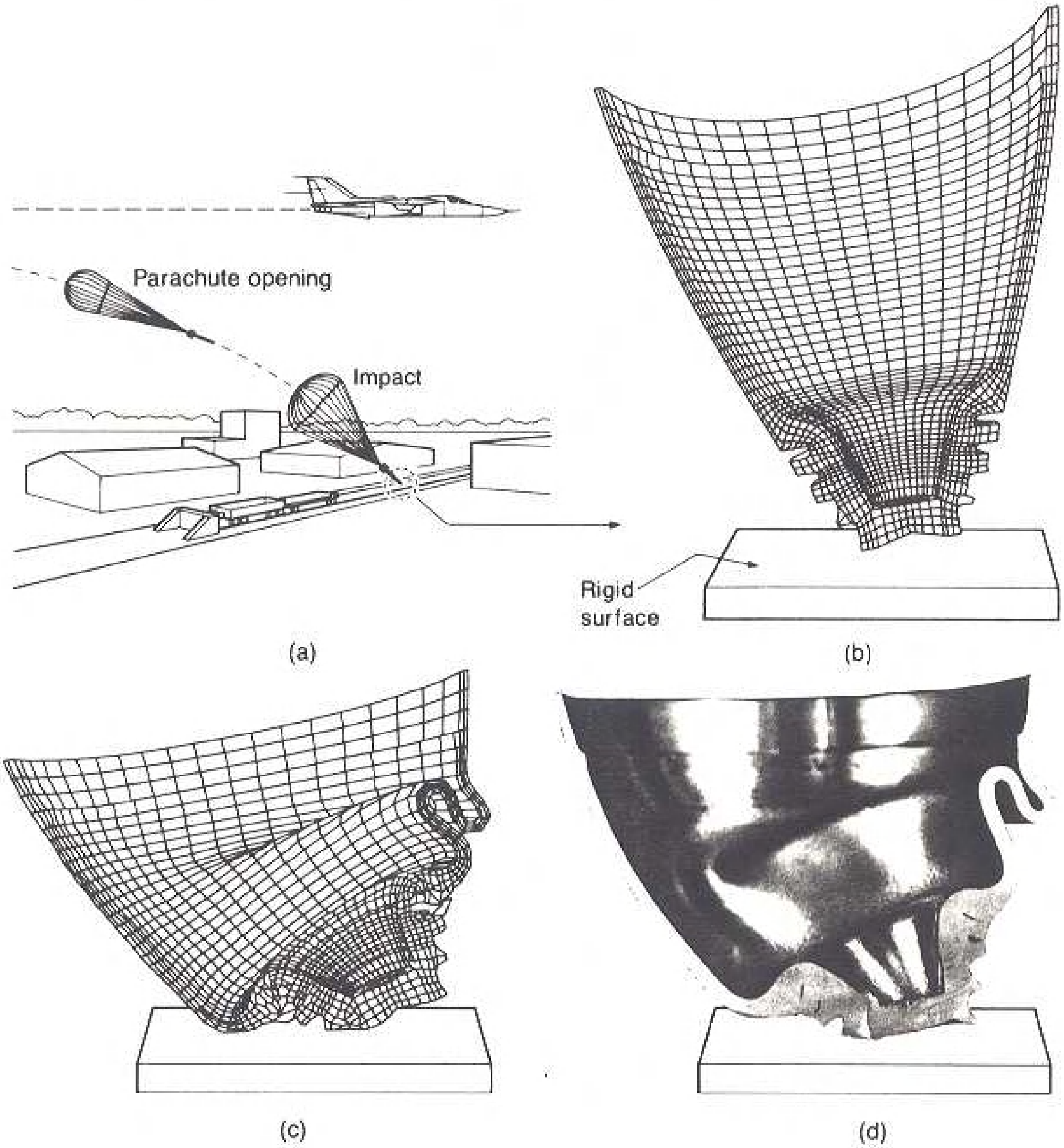
- Predictive — Design, reliability of proposed facilities reduce prototyping
- Confirmation — Quantify, assess evaluate existing condition
- Plant Support Areas
    - Engineering - damage and remaining life assessments
    - Construction - evaluate construction methods
    - Maintenance - evaluate maintenance methods
    - Inspection - codes & standards compliance
- Equipment Applications
    - Pressure vessels
    - Piping (special purpose FEM programs)
    - Steam generating equipment, fired heaters
    - Rotating equipment (pressure containing components)
    - Tankage
    - Valving, specialty & commodity
    - Structures
    - Heavy lift cranes, draglines
    - Refractory systems
    - Equipment auxiliaries
        - Bellows
        - Dampers
        - Fans
        - Internal structures
        - Stacks
- Cost / Benefit
    - FEM provides a means to quantify the performance of mechanical / structural equipment against a set of decision criteria such as
        - design & construction codes
        - monetary cost
        - safety
        - reliability
    - The results of properly conducted FEA are accepted by
        - facility insurer's
        - jurisdiction authorities, and
        - industry owners

Discipline Applications

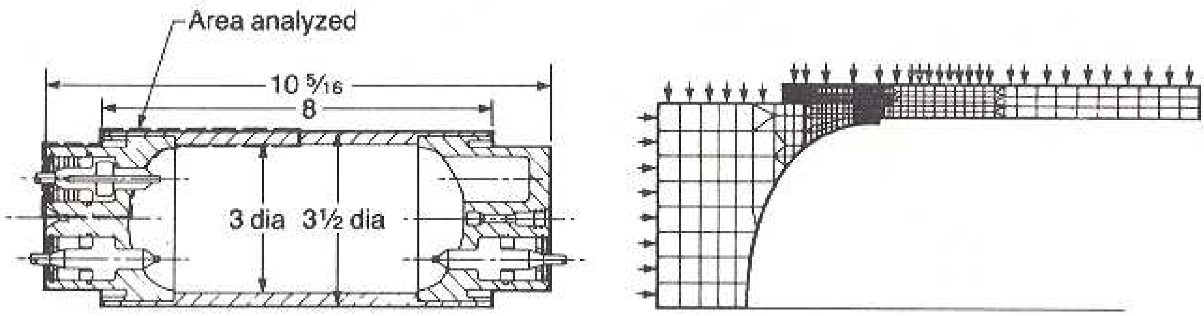
- Solid Mechanics
  - Elasticity
  - Plasticity
  - Statics
  - Dynamics
- Heat Transfer
  - Conduction
  - Convection
  - Radiation
- Fluid Mechanics
  - Laminar
  - Turbulent
- Acoustics
- Electromagnetism
- Solid State Physics
- Quantum Mechanics

FEM Example: Solid mechanics, plasticity (transient dynamic, nonlinear)

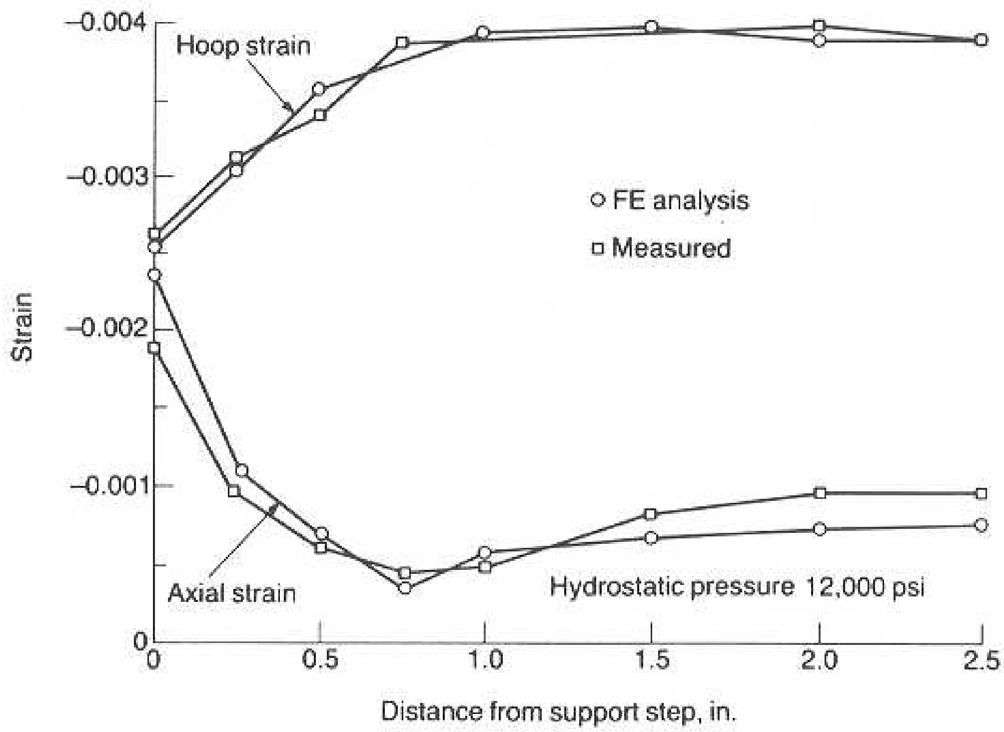
•Various parts of a modern strategic bomb are designed to mitigate considerable kinetic energy when impacting a hard surface in order to ensure the survivability of the internal components. Figure 1.14 shows the mesh used for a three-dimensional analysis of a steel nose cone during the few milliseconds immediately following impact. The crushing of the cone predicted by the analysis is revealed by the plot of the deformed mesh, which agrees extremely well with the photograph of an actual test specimen. ■



- **FEM Example: Solid mechanics, elastic (static, linear)**



(a) (b)



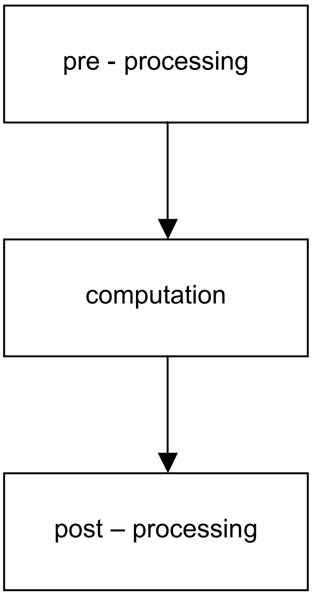
Note – this example shows validation of the theoretical results using experimentally determined results.

Why is FEM generally applicable to physical phenomena?

- Consider one-dimensional boundary value problems

Application	Unknown	Physical/material properties		Interior load	Flux	Balance equation	Constitutive equation	Governing equation (balance + constitutive eqs.)
General mathematical formulation	U	α	β	f	τ	$\frac{d\tau}{dx}+\beta U=f$	$\tau=-\alpha\frac{dU}{dx}$	$-\frac{d}{dx}\left(\alpha\frac{dU}{dx}\right)+\beta U=f$
Heat conduction	T (temperature)	k (thermal conductivity)	$\frac{hl}{A}$ (convection loss coefficient)	$Q+\frac{hl}{A}T_\infty$ Q = heat source $\frac{hl}{A}T_\infty$ = part of ambient convection	q (heat flux)	$\frac{dq}{dx}+\frac{hl}{A}T$ $=Q+\frac{hl}{A}T_\infty$	$q=-k\frac{dT}{dx}$	$-\frac{d}{dx}\left(k\frac{dT}{dx}\right)+\frac{hl}{A}T$ $=Q+\frac{hl}{A}T_\infty$
Elasticity	u (longitudinal displacement)	E (Young's modulus)		f (body force per unit volume)	σ (stress)	$-\frac{d\sigma}{dx}=f$	$\sigma=E\frac{du}{dx}$	$-\frac{d}{dx}\left(E\frac{du}{dx}\right)=f$
Cable deflection	W (transverse displacement)	T (tension in cable)	k (elastic modulus of foundation)	f (distributed transverse force)	F (transverse component of tension)	$\frac{dF}{dx}+kW=f$	$F=-T\frac{dW}{dx}$	$-\frac{d}{dx}\left(T\frac{dW}{dx}\right)+kW=f$
Electrostatics	Φ (electrostatic potential)	ϵ (permittivity)		ρ (charge density)	D (electric displacement)	$\frac{dD}{dx}=\rho$	$D=-\epsilon\frac{d\Phi}{dx}$	$-\frac{d}{dx}\left(\epsilon\frac{d\Phi}{dx}\right)=\rho$

The Elements of FEM



Mesh generation
Boundary Conditions
Loading

Element Equations
Assembly
Matrix Solution Methods

Determine stresses
Display
Print
Combine load cases

Mesh Generation

Superficially, mesh generation divides the domain of our system into elements. Associated with each element is a trial function(s) of algebraic expression. The element could be as large as abuilding or as small as small as the chip in a computer CPU. Hence, the term "finite" is used to describe the element.

If u(x) is the exact soluion to our problem, then $\tilde{u}(x{:}a)$ represents an approximate solution in algebraic form where:

$\tilde{u}^e(x{:}a) = a_1^e \cdot Ø_1^e(x) + a_2^e \cdot Ø_2^e(x) + \ldots\ldots\ a_n^e \cdot Ø_n^e(x)$

The $Ø_n(x)$ are set to form a Lagrange interpolation polynomial, i.e. when

$Ø_1(x) = 1$	$Ø_2(x) = 0$, and
$Ø_1(x) = 0$	$Ø_2(x) = 1$

This allows the a_n^e to constrain the equation of $\tilde{u}(x)$ to u(x) for all elements

If we let $K_{ij} = K_{ij}[Ø_n(x)]$, $F_i = F_i(Ø_n)$

Then

$$[K_{ij}]\ [a_n] = [F_i]$$

[K] = Matrix of coefficients that multiply the vector of unknown parameters

[a] = Unknown parameters dependent on the boundary conditions –i.e. sets the DOF, degree of freedom for the solution matrix

[F] = Load vector representing the exterior boundary and interior conditions

Element Equations

For each element, there is a K, a, F. These represent the element equations. For a single element with a beginning and ending node.

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

and for a succeeding element

$$\begin{bmatrix} K_{33} & K_{34} \\ K_{43} & K_{44} \end{bmatrix} \begin{bmatrix} a_3 \\ a_4 \end{bmatrix} = \begin{bmatrix} F_3 \\ F_4 \end{bmatrix}$$

If the matrix is written out in full:

$$\begin{bmatrix} K_{11} & \mathrm{K}_{12} & 0 & 0 \\ K_{21} & K_{22} & 0 & 0 \\ 0 & 0 & \mathrm{K}_{33} & K_{34} \\ 0 & 0 & \mathrm{K}_{43} & K_{44} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{bmatrix}$$

Assembly

For continuity, $u_1 = u_2$ at their common node point, i.e. where the elements connect. For our two element matrix, the means $a_2 = a_3$. The previous matrix can now be written as:

$$\begin{bmatrix} K_{11} & K_{12} & 0 \\ K_{21} & K_{22}+K_{33} & K_{34} \\ 0 & K_{43} & K_{44} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_4 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 + F_3 \\ F_4 \end{bmatrix}$$

This is known as assembly.

Matrix Solution Methods

As individual element equations are gathered into the system matrix, it is apparent that many zero terms arise; the non-zero terms being located on the main matrix diagonal. Since the non-zero terms predominate, the matrix is said to be sparse and allows for easy solution for the a_n. The width of the non-zero terms on the diagonal is called the bandwidth.

Once the system equations are assembled, each a_n is determined by Gaussian elimination.

Once the a_n have been determined, back substitute into the appropriate solution:

$\tilde{u}^e(x{:}a) = a_1^e \varnothing_1^e(x) + a_2^e \varnothing_2^e(x) + \ldots\ldots\ a_n^e \varnothing_n^e(x)$

For each element, what does the $\tilde{u}^e(x{:}a)$'s represent?

Temperature	→	heat flow
Displacement	→	stress
Voltage	→	current
Hydraulic head	→	fluid flow

Techniques for minimizing error

Convergence

- Convergence checks that the answer is unique

- Element labelling should ensure that most of the matrix coefficients end up on the main diagonal. Most programs ignore terms outside a bandwidth of 10.

- Refine mesh between successive runs and determine convergence

Mesh refinement	-	"h" method
Increase equations	-	polynomial trial solutions
	-	complex elements
	-	"p" method

- Check continuity across elements for flux terms

- Use alternate elements

Validation

- Validation checks that the answer is close to the exact solution

- Compare answers to known set of answers [benchmark problems]

- Determine solution accuracy for simple models with closed form solution

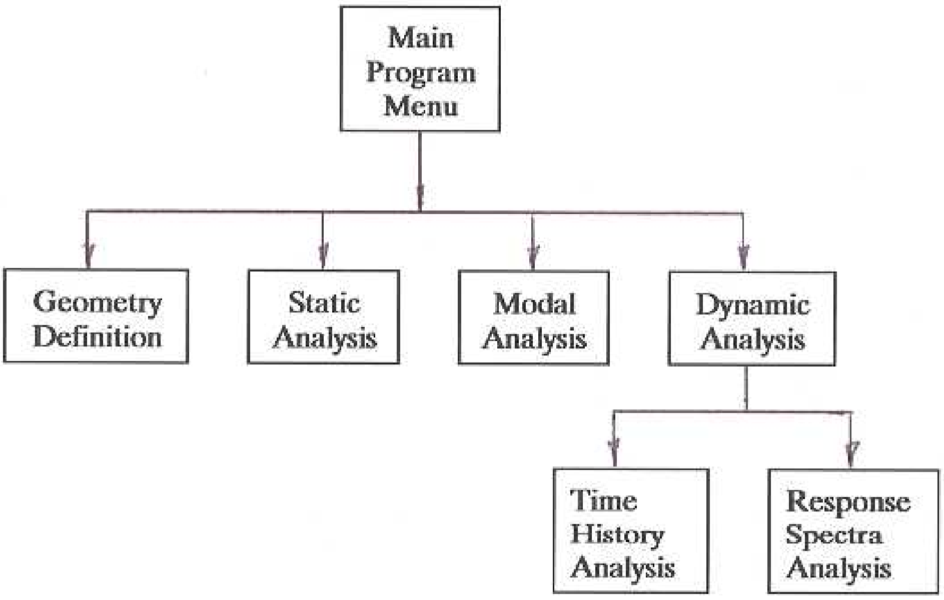
Caution: Error improvement cannot take place if only a few of the mesh elements become arbitrarily small → the large elements will have introduced some errors already.

Using an FEA program

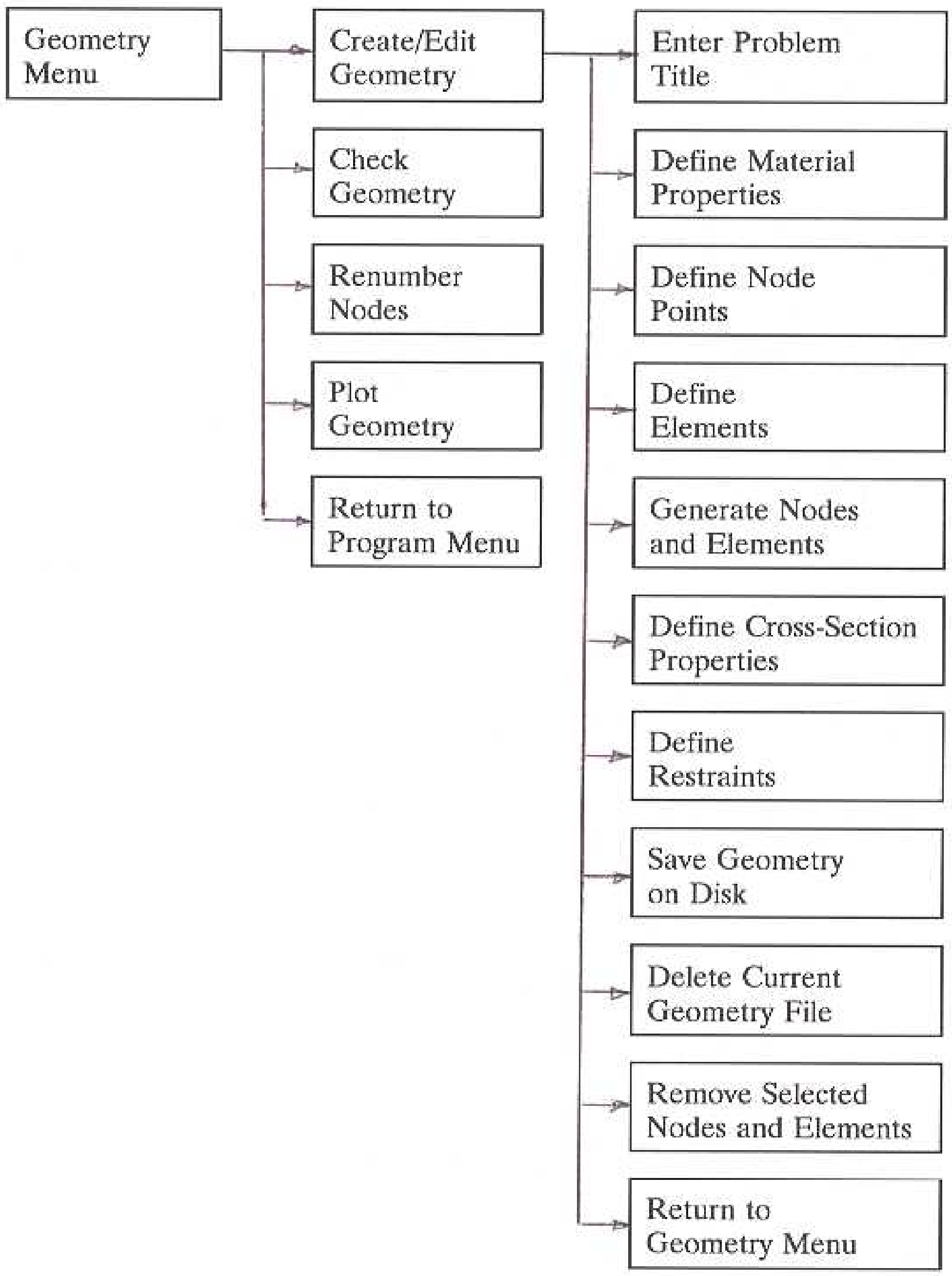
Many engineering problems can be analyzed on microcomputers and allow for many design iterations → sometimes, too many since the computational time is shortened drastically for many practical problems encountered in an industrial setting.

All input data are entered through the use of menus which correspond to the program modules –

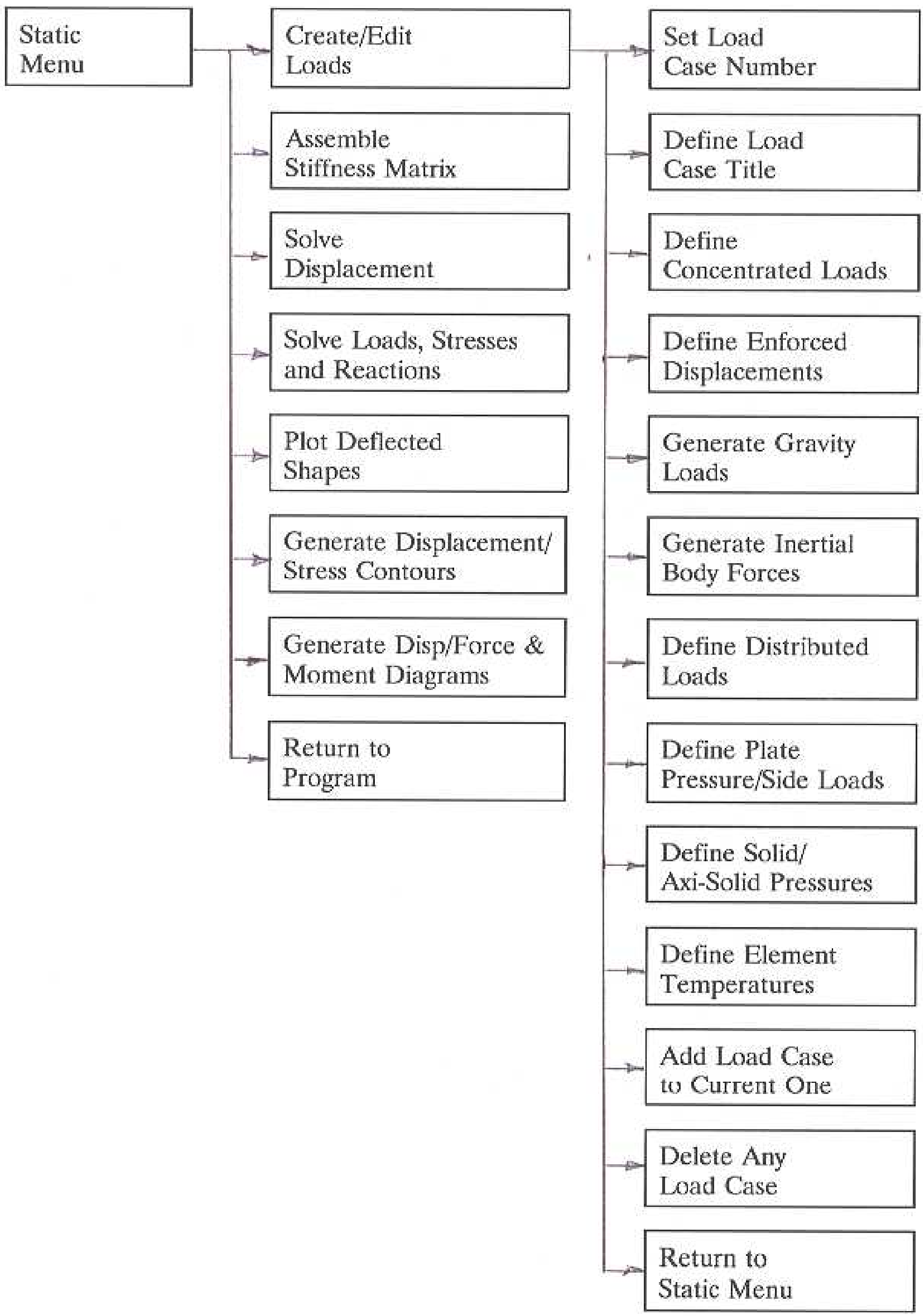
- Geometry definition
- Static analysis
- Modal analysis
- Dynamic analysis



Geometry Definition Menu Organization

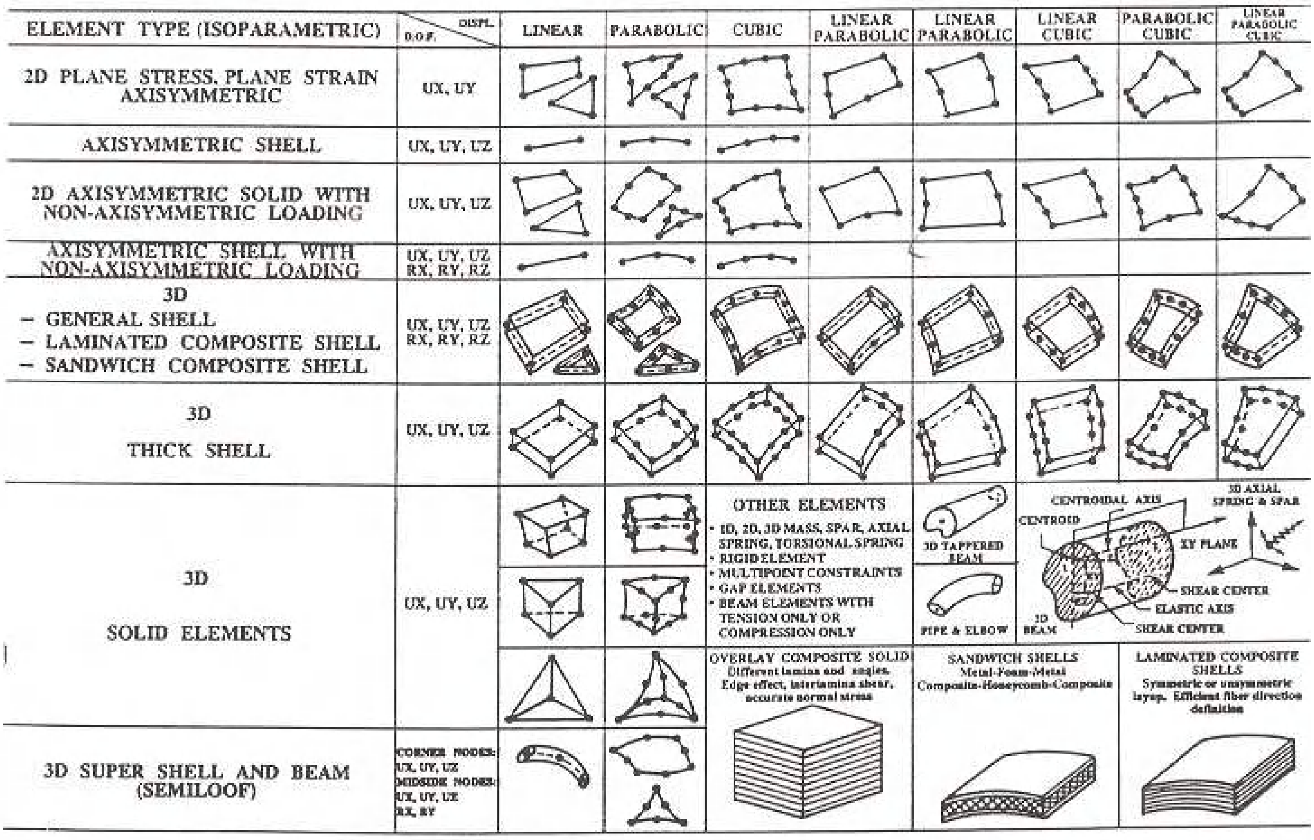


Static Analysis Menu Organization



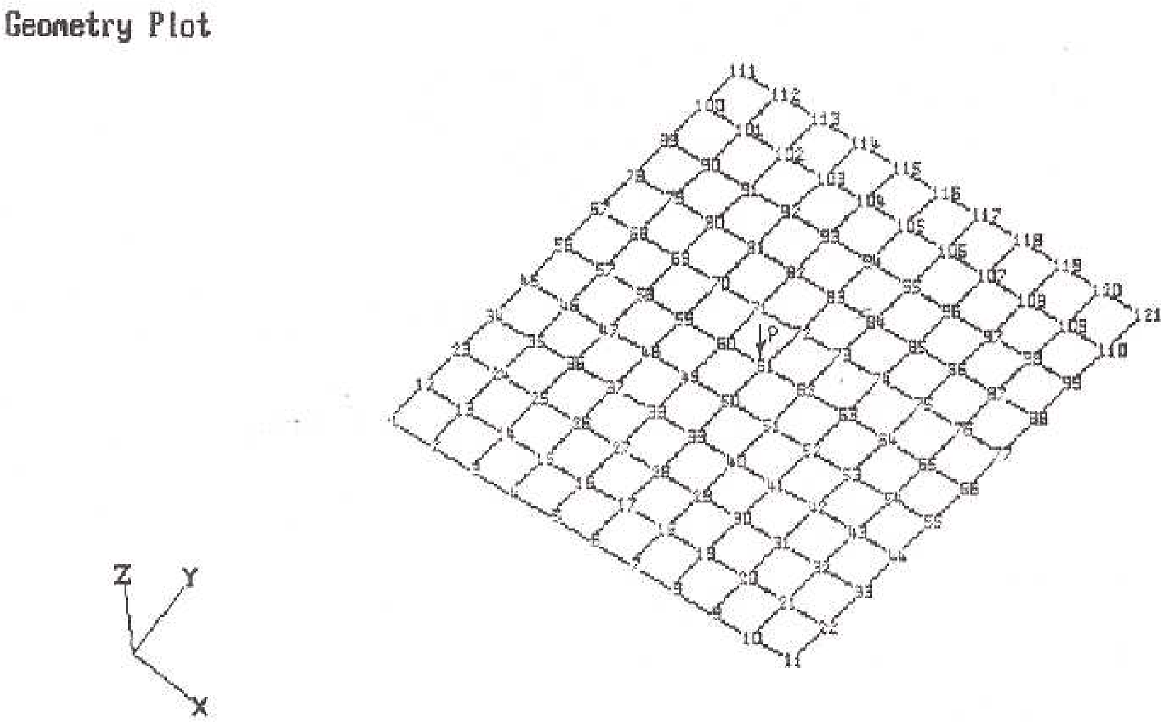
FEA Element Library

A typical program element library will provide sufficient elements types to conduct analyses to varying degrees of complexity in the discipline of interest, i.e. structures, fluids, electromagnetism, acoustics, etc.



Example of Static & Dynamic Analysis of a Plate

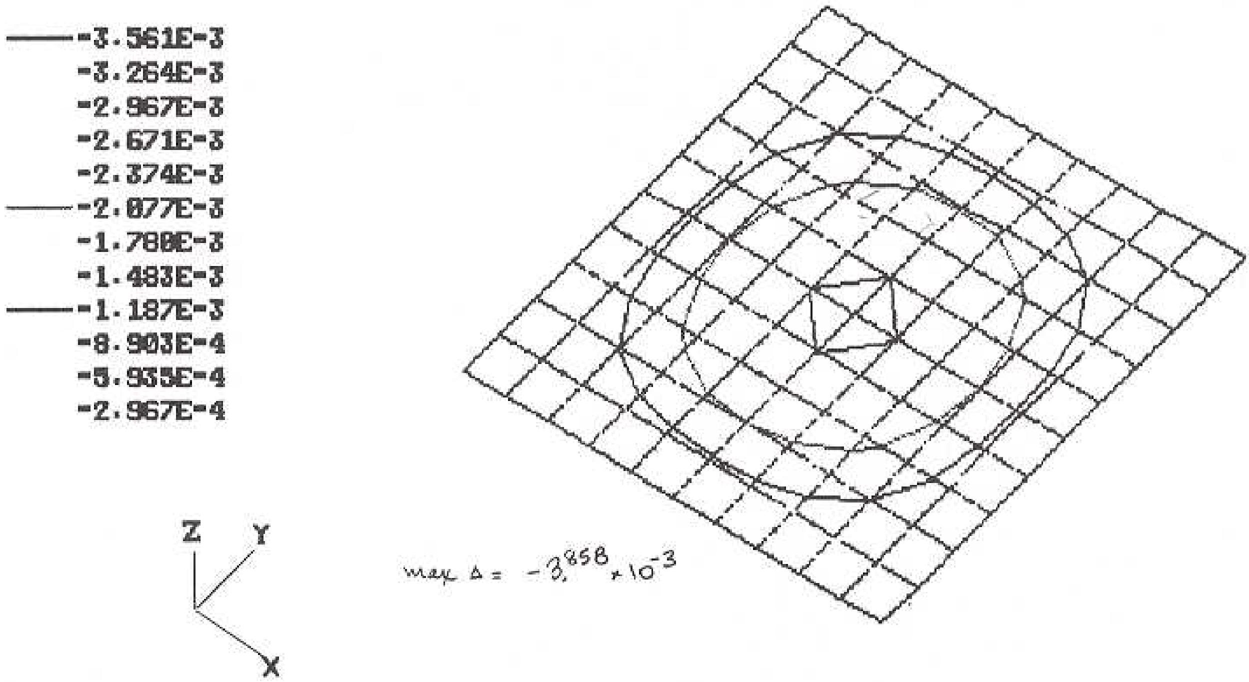
Static Example:



STATIC ANALYSIS OF PLATE FOR DEMONSTRATION
Wireframe Plot

t = 1 inch
l = w = 40 inches
P @ node 61 = 400 lb
weight density = 0.283 lb/m^3

Displacement Plot



Displacement Contour Plot
DZ

Solution

	Deflection	% Difference
Software	0.00386	0.8
Theory	0.00383	-

Reference: W.C. Young, "Roark's Formula's for Stress and Strain", 6^{th} Ed pg. 458

Detailed Calculation of Example Static Problem

REFERENCE

W.C. YOUNG, "ROARK'S FORMULAS FOR STRESS AND STRAIN", 6th Edition, pg 458

① CASE 1b

$$y = \frac{-\alpha W b^2}{E t^3} \qquad W = 400 + .283 \times 40 \times 40 \times 1 = 853^{\#}$$

$E = 29.6E6$

$\alpha = 0.1267$

$$= \frac{-0.1267 \times 853 \times 40^2}{29.6 \times 10^6 \times 1^3}$$

$$= 0.00058$$

② CASE 1a:

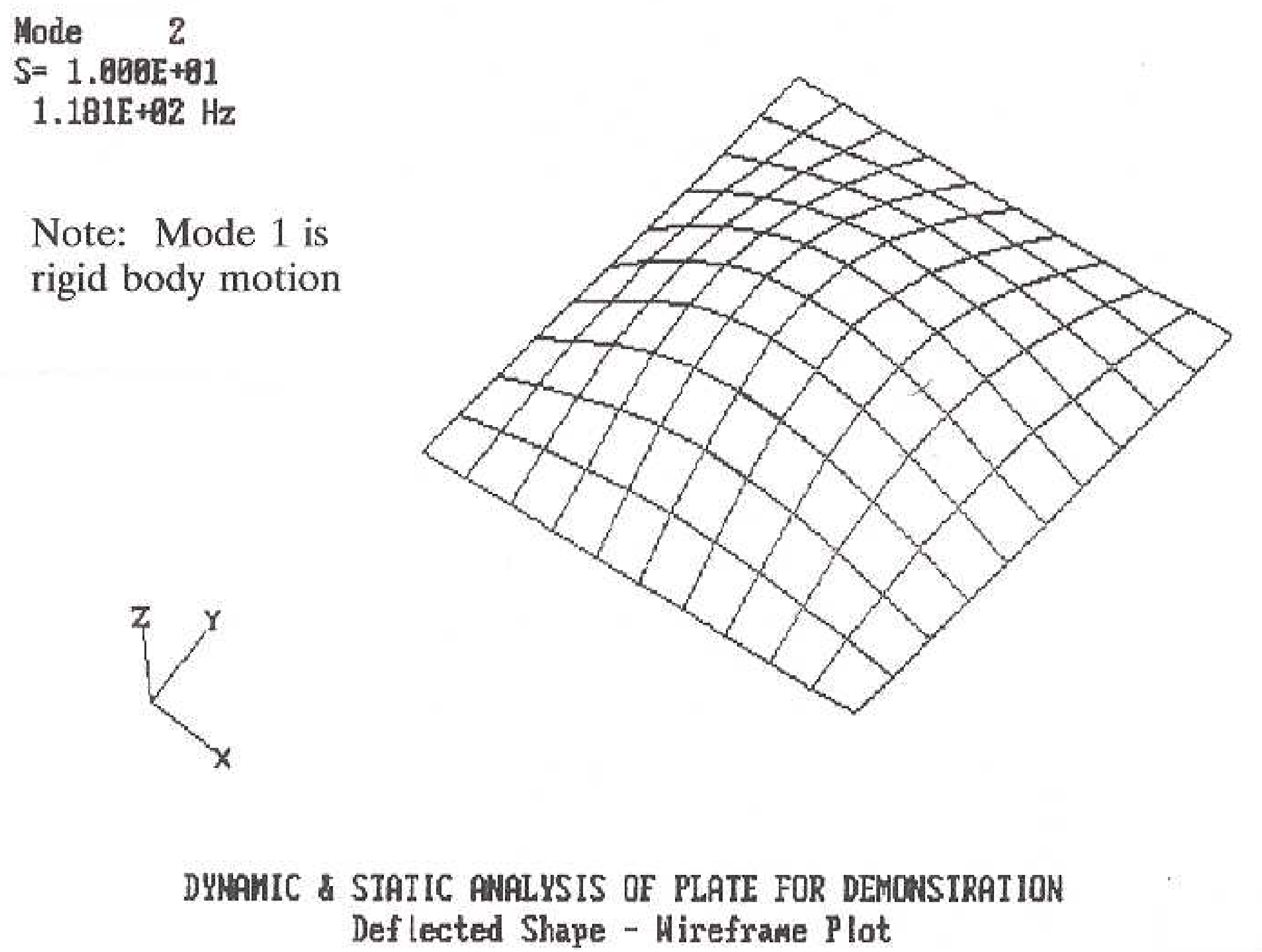
$$y_1 = \frac{-\alpha q b^4}{E t^3} = \frac{-0.0444 \times .283 \times 40^4}{29.6E6 \times 1^3} = .00109''$$

Case 1b:

$$y_2 = \frac{-\alpha W b^2}{E t^3} = \frac{-0.1267 \times 400 \times 40^2}{29.6E6 \times 1^3} = \underline{.00274}$$

$.00383''$

Dynamic Example



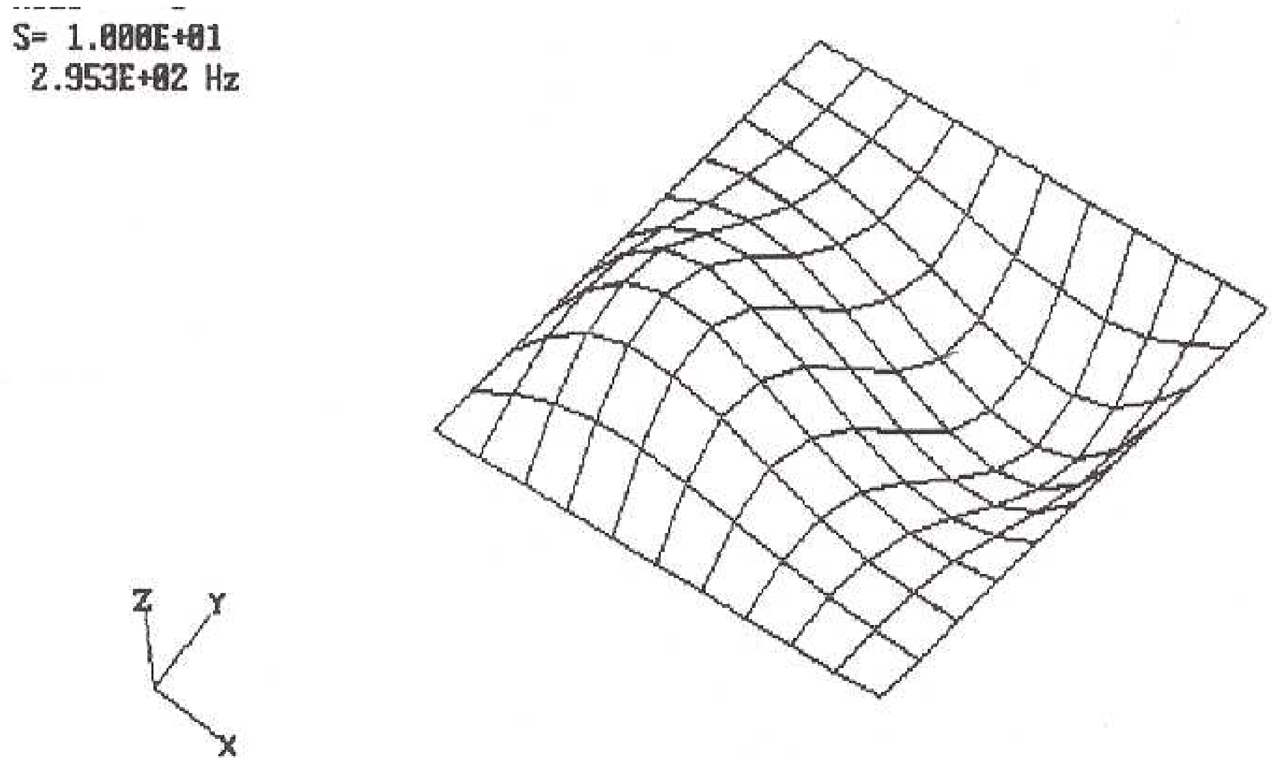
Solution

	f_1	f_2	f_3
Software	118.1	295.3	295.3
Theory	119.2	298.3	298.3
% Difference	0.8	1.0	1.0

Reference: W.C. Young, “Roark’s Formulas for Stress and Strain”, 6th Ed. Pg. 717

Dynamic Example [cont'd]

Mode 3



DYNAMIC & STATIC ANALYSIS OF PLATE FOR DEMONSTRATION
Deflected Shape - Wireframe Plot

Solution

	f_1	f_2	f_3
Software	118.1	295.3	295.3
Theory	119.2	298.3	298.3
% Difference	0.8	1.0	1.0

Reference: W.C. Young, "Roark's Formulas for Stress and Strain", 6th Ed. Pg. 717

Detailed Calculation of Example Dynamic Problem

REFERENCE

W.C. YOUNG; "ROARK'S FORMULAS FOR STRESS AND STRAIN", 6th EDITION, pg 717

CASE 16

$f = \frac{K_n}{2\pi}\sqrt{\frac{Dg}{\omega a^4}}$ $D = \frac{Et^3}{12(1-\nu^2)} = \frac{29.6\times10^6\times1}{12(1-.3^2)} = 2.711\times10^6 \; \frac{lb}{in^2}\times in^3$

$f_1 = \frac{19.7}{2\pi}\sqrt{\frac{2.711\times10^6\times386}{.283\times40^4}}$ $\omega = .283 \; lbs/_{in^2}$ $g = 386 \; in/_{sec^2}$

n	1	2	3
K_n	19.7	49.3	49.3

$\frac{lb\text{-}in \cdot \frac{in}{sec^2}}{\frac{lb}{in^2}\times in^4}$

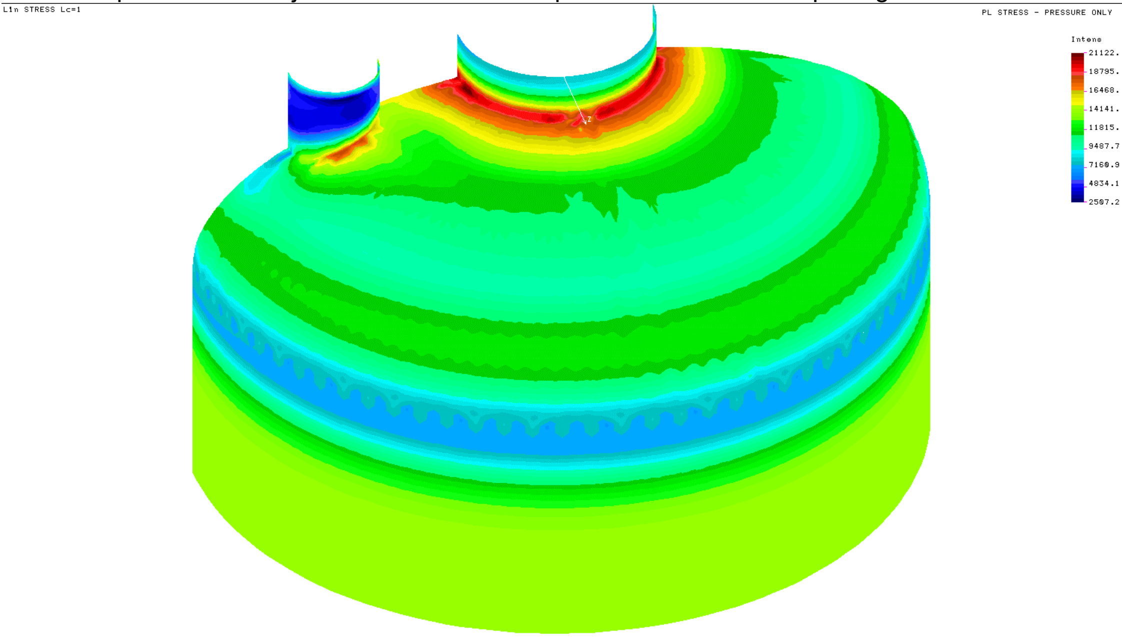
$= 19.7\times 6.05$

$= 119.2$ cps

$f_2 = f_3 = 298.3$ cps

Examples from actual problems

Vessel Top Head with Adjacent Nozzle – overlap of stresses around openings



Outlet Nozzle with Repad – note lack of penetration at repad ID weld