

Basics of Finite Element Analysis

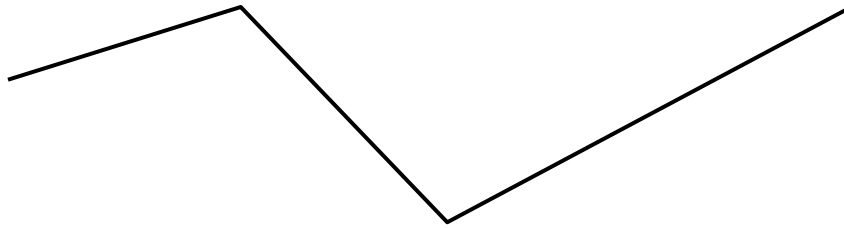
Strength of Materials, Solid
Mechanics

Outline of Presentation

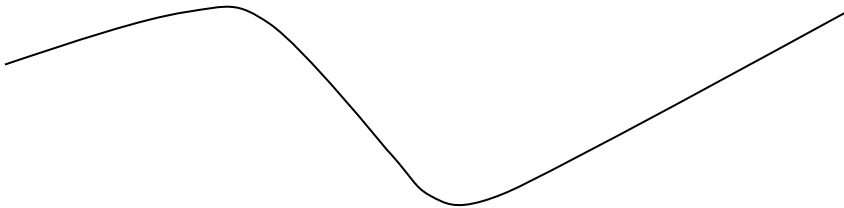
- Basic concepts in mathematics
- Analogies and applications
- Approximations to Actual Applications
- Improvisation of Traditional vs. Virtual Analysis

Basic Mathematical Concepts

- **Real Number** – Continuous variable – we can go to as much precision as required
- **Integers** – Discrete variables, only limited values available
- **Continuity:** There is no break when approached from the two dimensions, high – low, past – future (tangency not guaranteed)



- **Differentiability:** Continuity and tangency guaranteed.



- **Dimensions:** X, Y, Z and time

Analytical Calculation vs. Numerical Calculation

- **Analytical Results**

- Available as explicit or implicit form of an equation such as a quadratic equation.
- It is a continuous equation, available at each point in space (**Infinite Unknowns**)

- **Numerical Calculation**

- Available at discrete location in space / time dimensions (**Finite Unknowns**)
- It is an inherently discontinuous approach with some averaging / blending to ensure physical correctness

- **Examples**

- Simply supported beam - Analytical
- Plate temperature distribution - Numerical

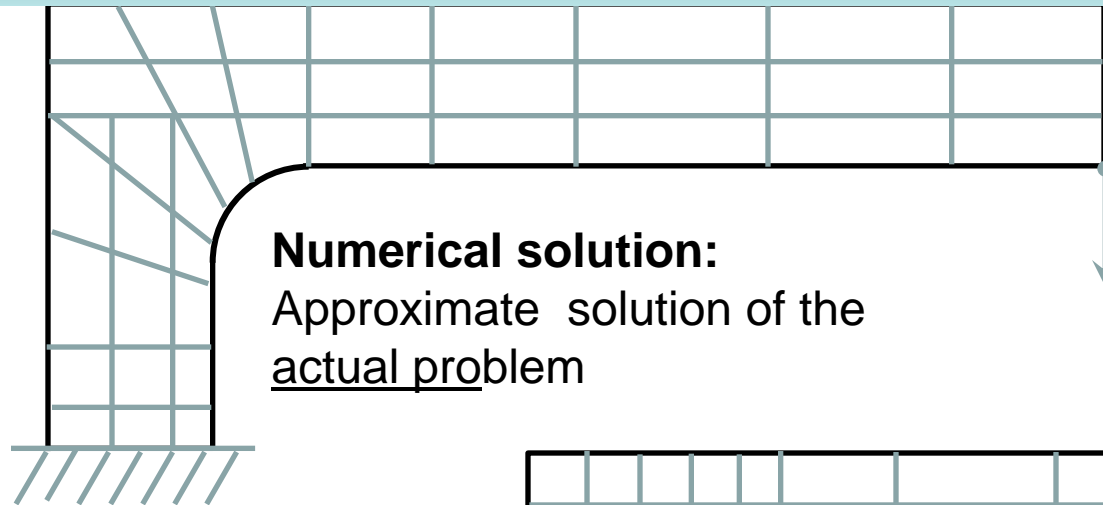


Physical System

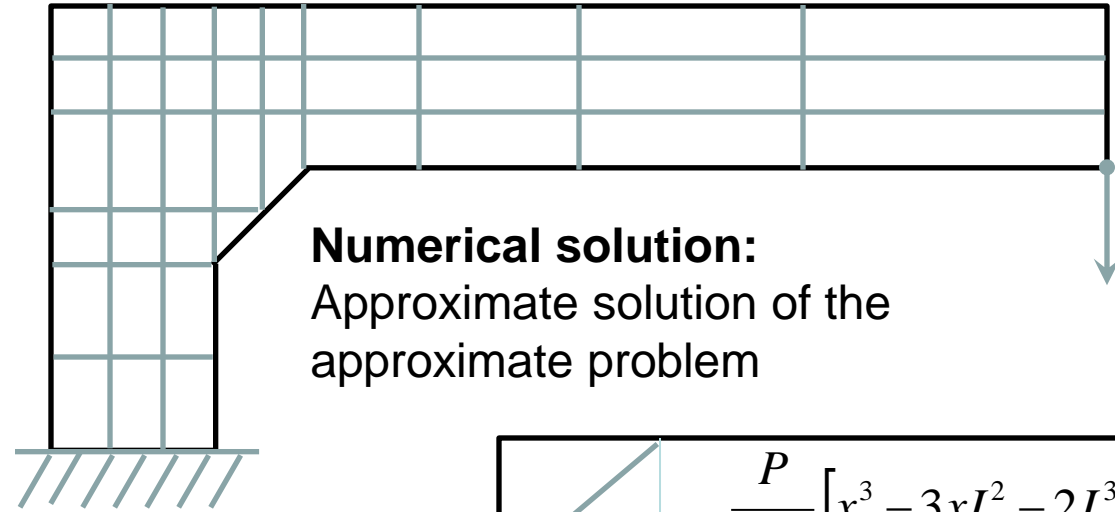


F.E. Model

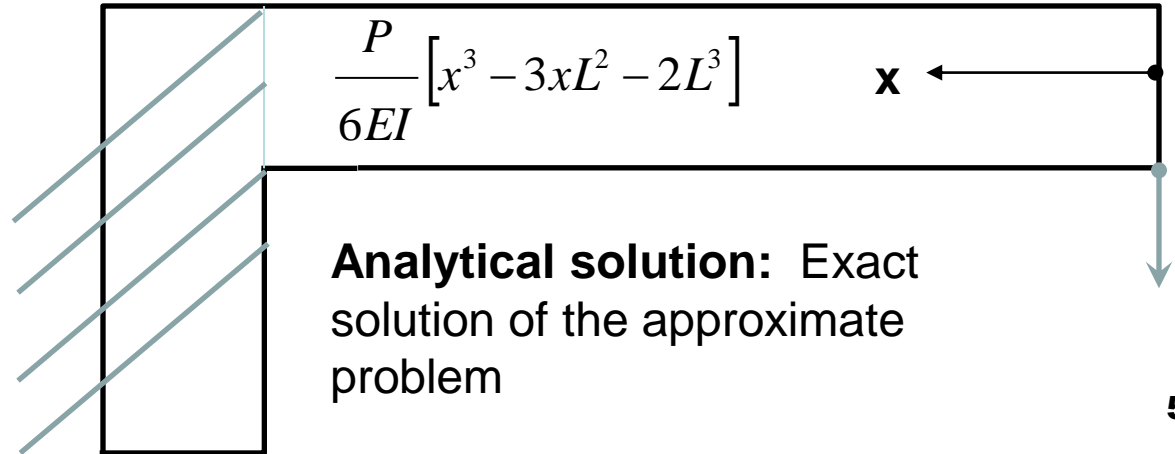
Analytical Calculation vs. Numerical Calculation



Numerical solution:
Approximate solution of the actual problem



Numerical solution:
Approximate solution of the approximate problem

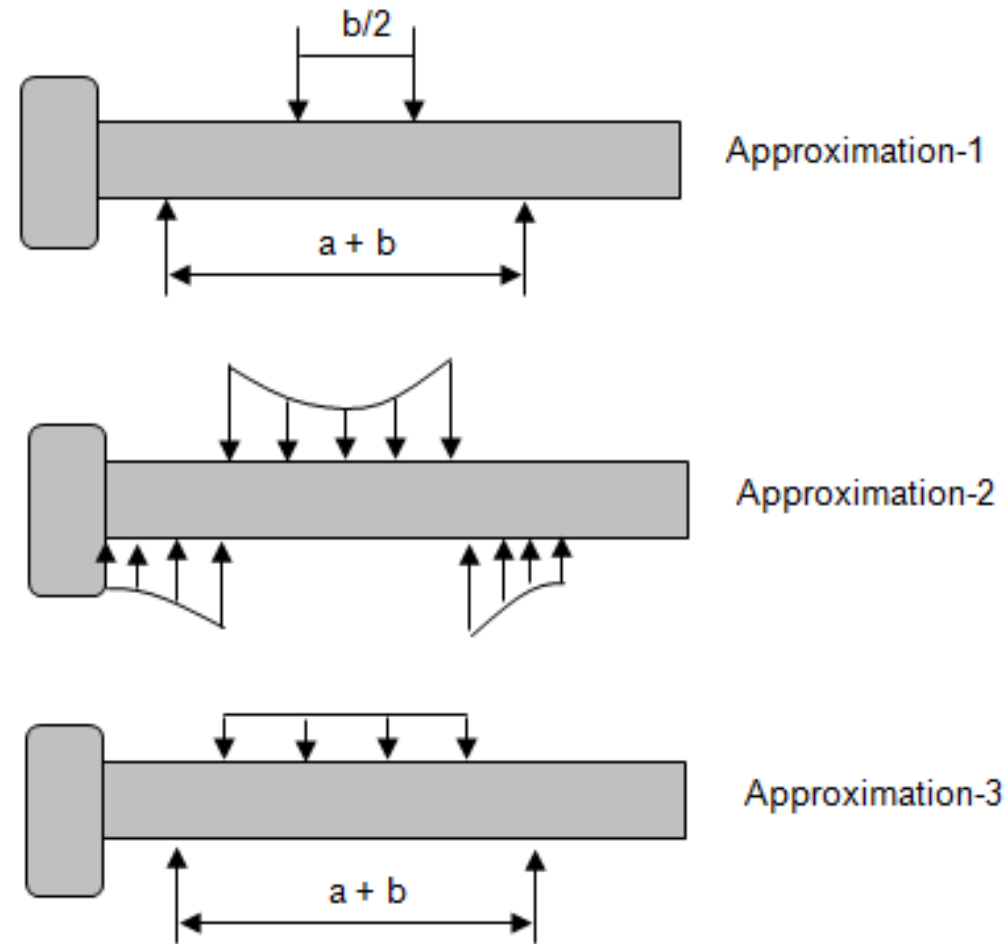
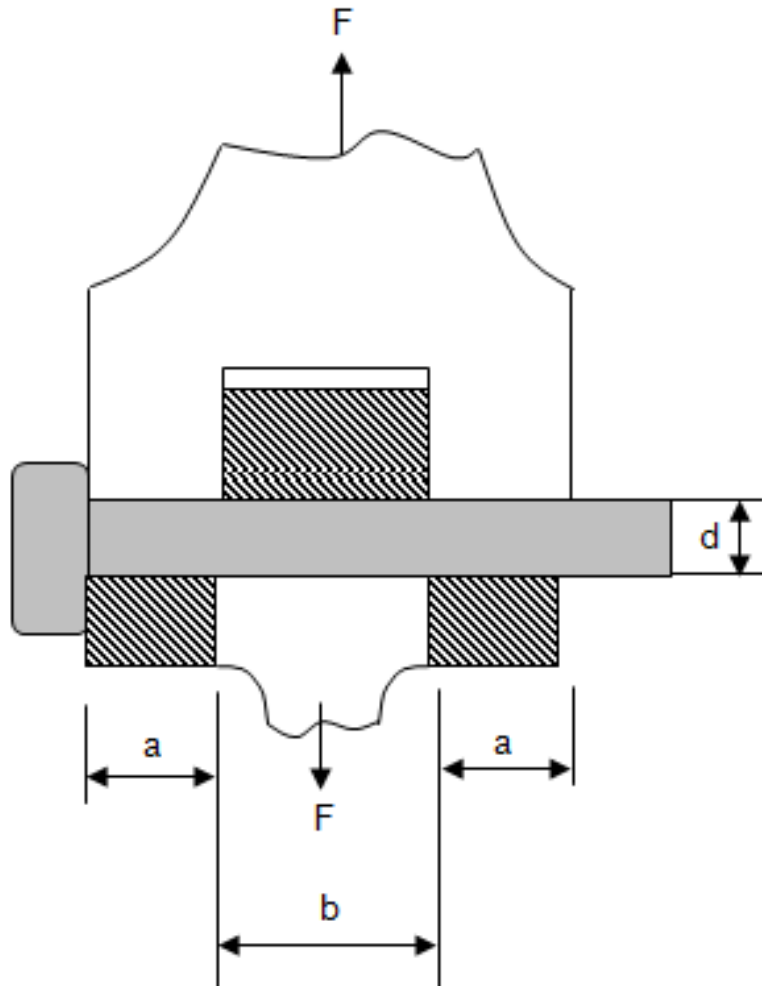


$$\frac{P}{6EI} [x^3 - 3xL^2 - 2L^3]$$

Analytical solution: Exact solution of the approximate problem

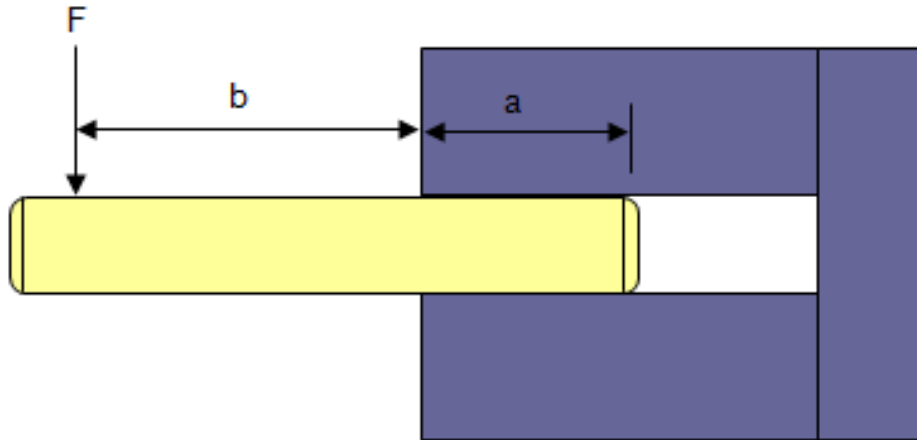
Analytical Calculation vs. Numerical Calculation

Engineering Assumptions: Example - 1:



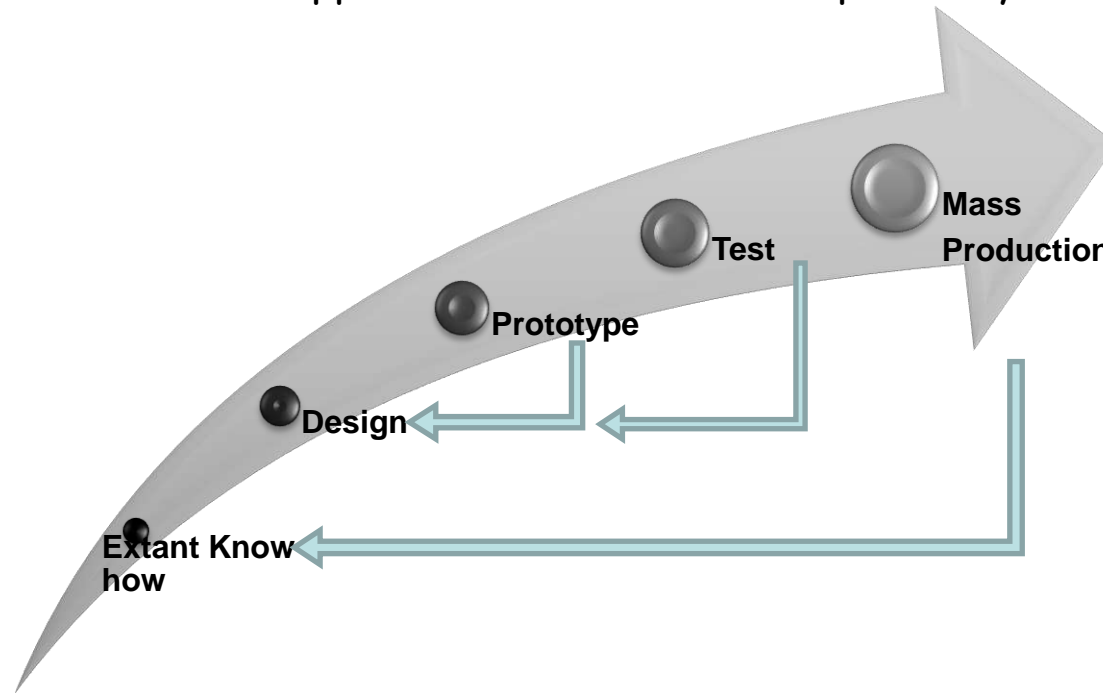
Analytical Calculation vs. Numerical Calculation

Engineering Assumptions: Example - 2:

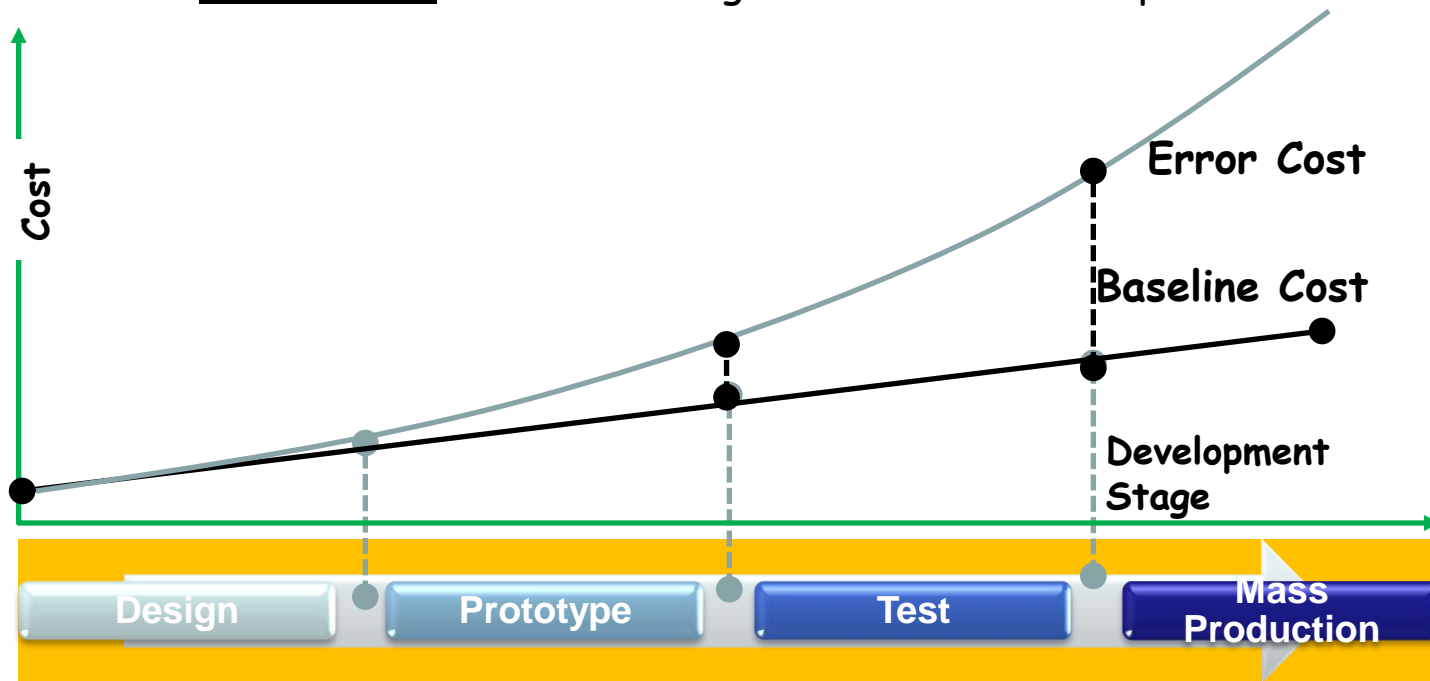


In the arrangement above a pin is press fitted into a hole of a relatively rigid material. A general analysis assumption is that of a concentrated reaction R and moment M at the edge of the rigid structure (distance b). Suppose the reaction is distributed along distance a . Use FEA analysis method to ascertain whether resulting moment reaction is larger or smaller than the concentrated reaction. Also calculate the loading intensity 'q' for the length 'a'.

Traditional Approach of Product Development Cycle

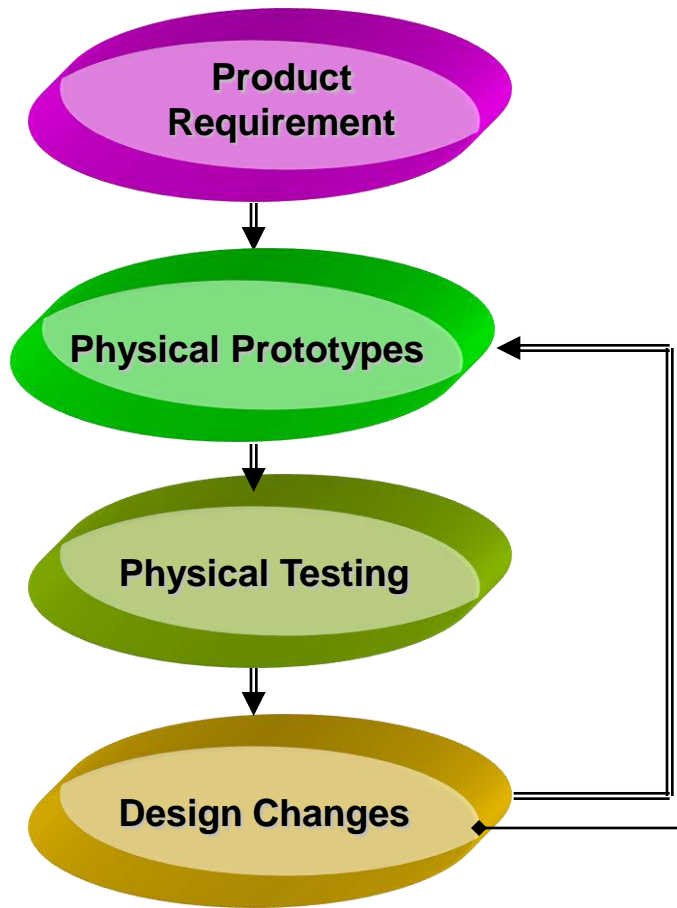


Error Cost at Various Stage of Product Development



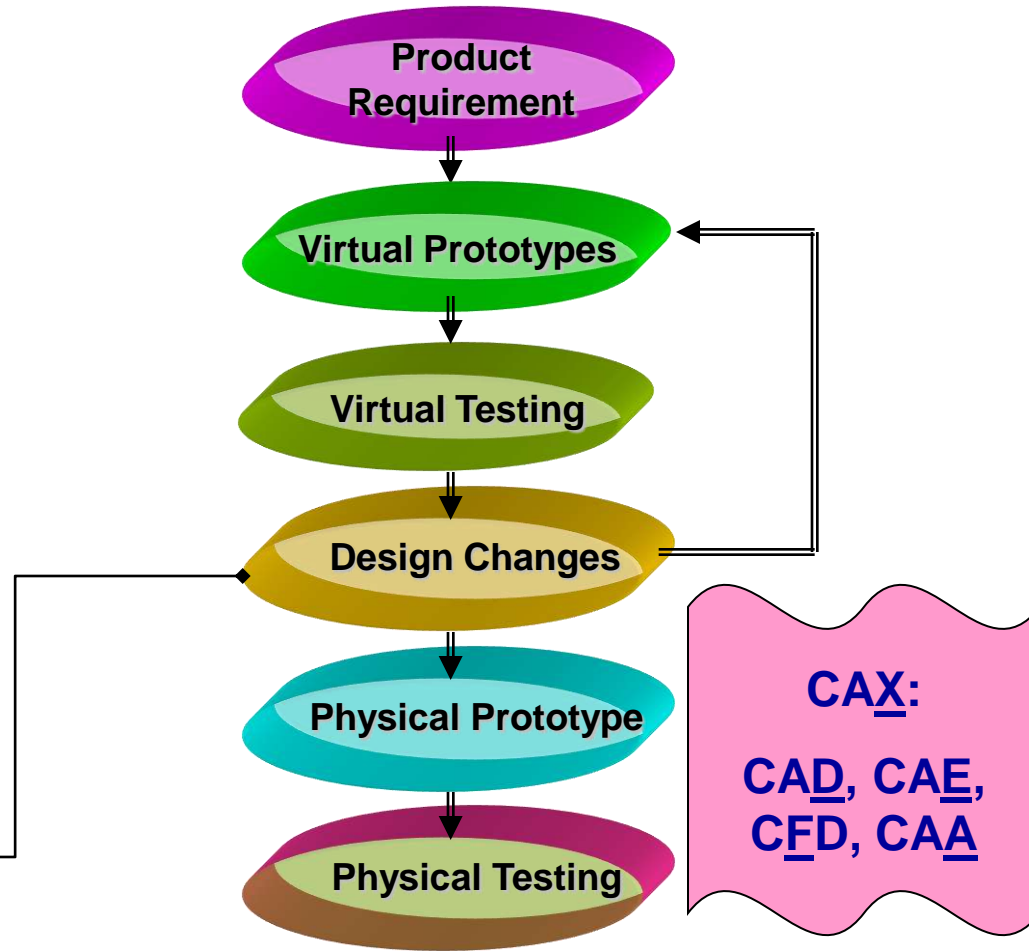
Traditional vs Virtual Design Evaluations

Traditional Design Methodology



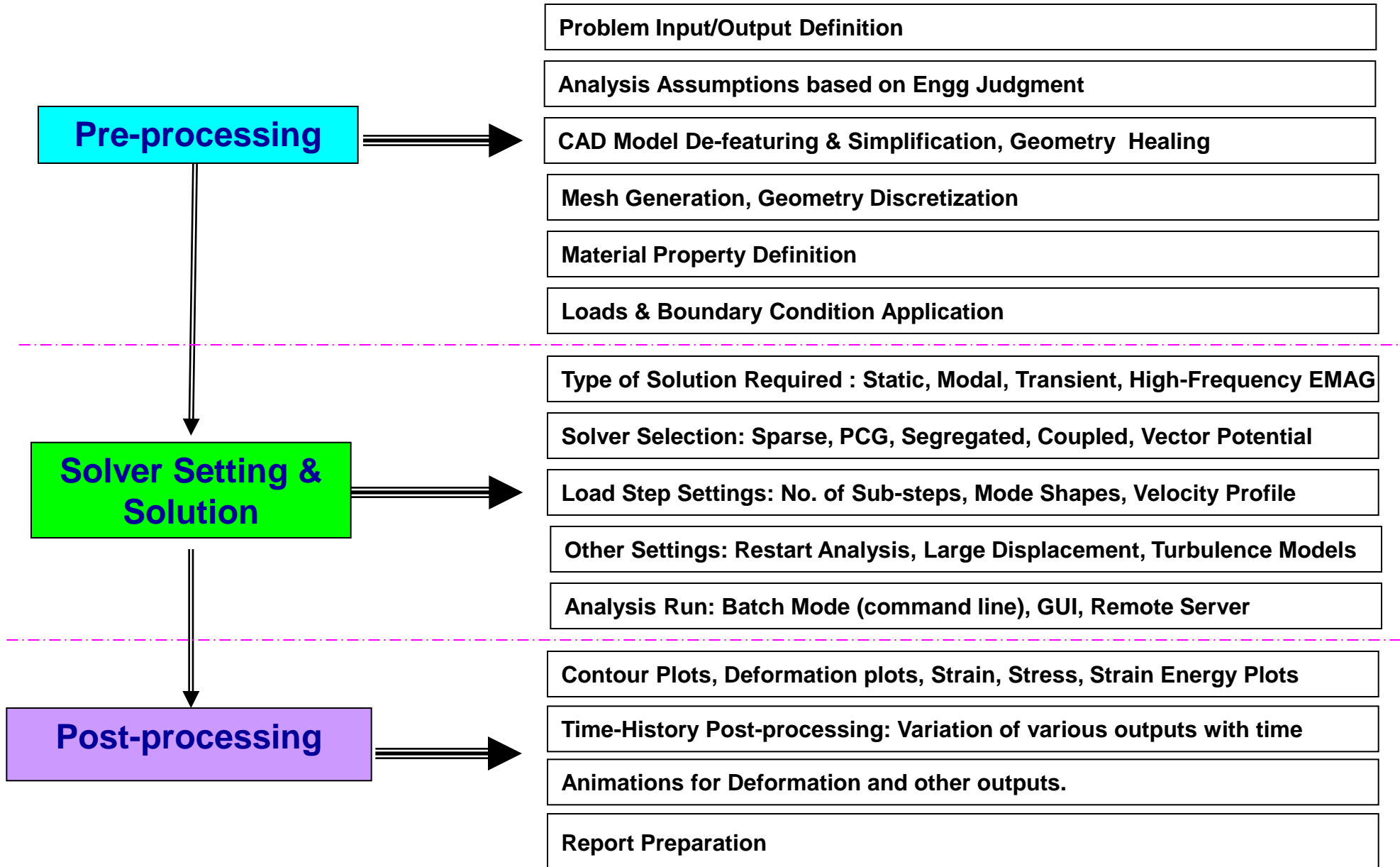
- Long Product Development Cycle
- Many prototypes
- Very Costly

Modern Design Methodology



- Minimal Prototypes and Cost Effective
- Product Development Cycle Drastically Reduced
- Better Insight into various Design Aspects

Typical Flow Chart for Numerical Analysis



Structural Calculations : Variables, Definition and Convention

- **Stress:** Force acting per unit area
 - Normal Stress, σ : Force acting normal to the area / Area
 - Shear Stress, τ : Force acting in the plane of the area / Area
 - Total Stress: Vector Sum of Normal and Shear Stress

$$\text{Resultant stress} = \sqrt{\sigma^2 + \tau^2}$$

- Strictly speaking, stress is not a vector quantity, in addition to magnitude and direction, we must specify the plane on which the stress acts. Stress is therefore a **tensor**, *its complete description depending on the **two vectors of force and surface of action***. Hence, stress is not invariant to the “Co-ordinate System” though true vectors are.
- **Tensor Plots:** Show the direction and magnitude of the stress (tension or compression) at each node

Structural Calculations : Variables, Definition and Convention

Normal stresses directed away from their related surfaces are tensile and positive, opposite compressive stresses are negative. Shear stresses are positive when they act in the positive direction of the relevant axis in a plane on which the direct tensile stress is in the positive direction of the axis. If the tensile stress is in the opposite direction then positive shear stresses are in directions opposite to the positive directions of the appropriate axes.

Non-linear Elasticity: Conservative process with complete strain reversal, no permanent inelastic strain.

Plane Stress: A state of stress in which the normal stress and the shear stress directed perpendicular to the plane are assumed to be zero.

Constraints are necessary to eliminate the rigid body modes which are associated with the element stiffness matrix as well as the stiffness matrix of the whole structure. From statics, it is well-known that for a plane problems at least 3 independent displacement must be constrained.

Structural Calculations : Variables, Definition and Convention

Co-ordinate System: All forces, displacements, and other direction-dependent nodal quantities are interpreted in the nodal coordinate system. A nodal coordinate system is attached to every node in the model. By default, the nodal CS is parallel to Global Cartesian, i.e. all applied forces and displacement constraints are interpreted in Global Cartesian by default. Unlike the structural analysis displacement and force boundary conditions, the analogous thermal analysis temperature and heat flow boundary conditions are not dependent on the nodal coordinate system.

Equations in Structural Mechanics:

1. Equilibrium: Force and Mass Balance
2. Kinematics Relation (strain-displacement): $e_{xx} = du/dx$
3. Constitutive Relation: $s_{xx} = E * e_{xx}$

1. Static Strength: Most common & the most fundamental form of strength calculation , includes calculations in Magneto-static and conductors carrying direct current.

Added complexity:

Geometry Non-linearity: Frictional Contact, Interference Fits

Material Non-linearity: Deformation beyond Yield Limit, Strain Hardening

2. Modal Analysis: To calculate natural frequencies and deformation shape of the design. This calculation is also called “Mode-Frequency Analysis”.

3. Thermal Loading:

1. Stress due to temperature gradient which in turn is due to non-uniform heating or cooling of the component.
2. Stress due to constraining the thermal expansion (heating) or thermal contraction (cooling) of the component even though there is no temperature gradient.

4. Transient Analysis: All calculations are time-dependent. Calculations related to vibrations, impact, varying magnetic field, short-circuit and electrical faults

- Steady transient (system behavior repeated in some periodic fashion)
- Unsteady transient (system response reached to a steady dynamic state or reduces to a standstill).

5. Harmonic Response Analysis: Combination of Modal, Transient and Static Analysis. Sometimes also known as Spectrum Analysis, though strictly not identical.

6. Buckling Analysis: Large deformation analysis where the assumptions of linear elasticity fails.

7. Sub-structuring Analysis: Sub-structuring is a procedure that condenses a group of finite elements into one element represented as a matrix. The single-matrix element is called a super-element.

Type of Calculations

Type	Input	Output	$\overbrace{[M]\{\ddot{u}\}}^{F_{\text{inertial}}} + \overbrace{[C]\{\dot{u}\}}^{F_{\text{damping}}} + \overbrace{[K]\{u\}}^{F_{\text{stiffness}}} = \overbrace{\{F\}}^{F_{\text{applied}}}$
Modal	None	1. natural frequencies & corresponding mode shapes 2. stress/strain profile	F(t) set to zero; [C] usually ignored
Harmonic Response	Sinusoidal excitations across a range of frequencies		F(t) and u(t) assumed to be sinusoidal
Spectrum	Spectrum representing the response to a specific time history	Maximum response if the model were subjected to the time history	Input is a known spectrum of response magnitudes at varying frequencies in known directions
Random	Spectrum representing probability distribution of excitation	Response within specified range of probabilities	Input is a probabilistic spectrum of input magnitudes at varying frequencies in known directions
Transient	Time-varying loads	Time-varying response	The complete, general form of the equation is solved

- 1. Stress Stiffening:** It is the stiffening or weakening of the structure due to state of stress. Also known as “Geometric Stiffening”, “Incremental Stiffening”, “Initial Stress Stiffening” or “Differential Stiffening”.
 - This needs to be normally considered for thin structures with bending stiffness very small as compared to axial stiffness.
 - Examples are cables, thin beams and shells
- 2. Spin Softening:** What is it and how to account for it in FE calculations?

Analysis of Mechanical Systems involves three Basic Steps:

- Study of Forces
 - ❖ Discretization of Mechanical System
 - ❖ Free Body Diagram
- Study of Motion (Kinematics) and Deformations
- Application of laws relating forces to motion and deformations

1. $F = m \cdot a$

2. $F = K \cdot x$

3. $\sigma = E \cdot \varepsilon \rightarrow [\sigma] = [D] \{\varepsilon\}$

4. $\varepsilon = \delta / L \rightarrow [\varepsilon] = [B] \{\delta\}$

$$\begin{bmatrix} \sigma_X \\ \sigma_Y \\ \sigma_Z \\ \tau_{XY} \\ \tau_{YZ} \\ \tau_{ZX} \end{bmatrix} = [D] \left\{ \begin{array}{l} \varepsilon_X \\ \varepsilon_Y \\ \varepsilon_Z \\ \varepsilon_{XY} \\ \varepsilon_{YZ} \\ \varepsilon_{ZX} \end{array} \right\}$$

Any structural calculation involves Stress, Strain, Deformation and relation between Stress and Strain. Relation between Forces and Deformations is more critical in such cases.

Structural CAE : Concepts

In matrix form:

$\{\sigma\} = [E] \{\varepsilon^{EL}\}$, where EL stands for elastic strain vector, the strain primarily responsible for stress

Let's take an example of a rod fixed at both ends and heated uniformly:

$\{\varepsilon^{TOT}\} = \{\varepsilon^{EL}\} + \{\varepsilon^{TH}\} = 0 \Rightarrow \{\varepsilon^{EL}\} = -\{\varepsilon^{TH}\}$ and hence stress will be introduced.

And when the rod is allowed to expand freely,

$\{\varepsilon^{TOT}\} = \{\varepsilon^{EL}\} + \{\varepsilon^{TH}\} = 0 + \{\varepsilon^{TH}\} = \{\varepsilon^{TH}\}$

Since, $\{\varepsilon^{EL}\} = 0$, $\{\sigma\} = 0$.

Conclusion:

$\{\varepsilon^{TOT}\} = [E]^{-1} \{\sigma\} + \{\varepsilon^{TH}\}$ where $[E]^{-1}$ is called Flexibility or compliance matrix.

Symmetry: In a structural analysis, for example, a symmetry boundary condition means that out-of-plane translations and in-plane rotations are set to zero, and an anti-symmetry condition means that in-plane translations and out-of-plane rotations are set to zero.

Acceleration:

Translational acceleration accounts for the structural effects of a constant linear acceleration. If desired, acceleration can be used to simulate gravity (by using inertial effects) by accelerating a structure in the direction opposite of gravity (the natural phenomenon of). That is, accelerating a structure vertically upwards (say +Y) at 9.80665 m/s^2 , applies a force on the structure in the opposite direction (-Y) inducing gravity (pushing the structure back towards earth).

Acceleration due to Gravity:

Alternative to what is explained above, the **Standard Earth Gravity** can be used to produce the load due to the effect of gravity.

- Gravity and Acceleration are essentially the same type of load except they have opposite sign conventions and gravity has a fixed magnitude.
- For **applied gravity**, a **body tends to move in the direction of gravity** and for **applied acceleration**, a **body tends to move in the direction opposite of the acceleration**.

Any structural numerical calculation or as a matter of fact any numerical calculation involves calculation about a set of discrete points. Hence, they necessarily form a matrix. Followings are the typical matrices. It must be noted that a vector is also a type of matrix where there are only one row or column for more than 1 column or rows respectively.

1. Elastic stiffness, elasticity or stress-strain matrix, $[D]$, (analogous to stiffness in 1D)
2. Strain-displacement matrix, $[B]$
3. Flexibility or compliance matrix = Inverse of elasticity matrix
4. Stiffness matrix, $[K]$
5. Displacement Vector
6. Damping matrix, Mass Matrix
7. Stress Stiffness Matrix (also called geometric, incremental, initial stress, or differential stiffening)
8. Pressure Vector
9. Force Vector
10. Temperature Vector

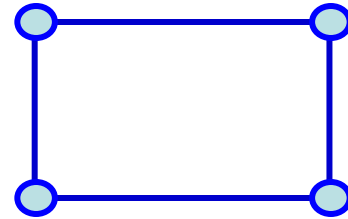
- 1. Nodal values** across a free surface can be integrated using INTSRF command. The free surface is determined by a selected set of nodes which must lie on an external surface of the selected set of elements.
 - Main Menu > General Postproc > Element Table > Sum of Each Item
 - Utility Menu > Parameters > Get Scalar Data
 - In the "Type of data to be retrieved" filed, highlight "Result data" and "Elem table sums".
- 2. Error approximation technique for Displacement-based Problems [post1->prerr].** The usual continuity assumption used in many displacement based finite element formulations results in a continuous displacement field from element to element, but a discontinuous stress field. To obtain more acceptable stresses, averaging of the element nodal stress is done. Then returning to the element level, the stress at each node of the element are processed to calculate energy error for the element.

FE Mesh Quality Targets					
S. No	Criteria/Parameter	For 90 % elements		For Remaining 10%	
		2D	3D	2D	3D
1	Aspect Ratio, MAX	5	8	10	10
2	Warpage	5	5	7	7
3	Minimum angle				
	Quad	60	50	30	30
	Tri	30	30	15	15
4	Maximum angle				
	Quad	120	140	170	170
	Tri	110	120	160	160
5	Tetra Collapse	*	0.4	*	0.1
6	Tetra Aspect Ratio	*	5.0	*	25
7	Jacobian	0.6	1.0	0.4	1.0

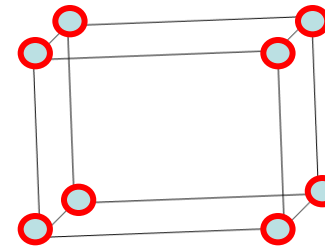
Structural CAE : Common Elements (In Brief)



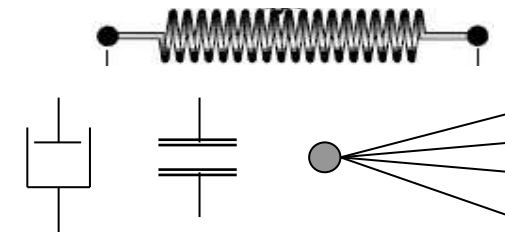
1-D



2-D



3-D



Others

- **Element Shape**

- Line

- **Element Type**

- Rod/Bar
- Beam
- Pipe
- Axi-symmetric shell
- Others

- **Practical Application**

- Long Shafts
- Beams
- Pin Joints
- Connection Elements

- **Element Shape**

- Tria, Quad

- **Element Type**

- Thin Shell
- Plate
- Membrane
- Axi-symmetric Solid
- Plain Stress/Strain

- **Practical Application**

- Sheet Metal parts
- Plastic Components
- Thin pressure vessels

- **Element Shape**

- Tetra, Penta, Hex, Pyramid

- **Element Type**

- Solid

- **Practical Application**

- Transmission Casing
- Engine Block
- Crankshaft
- Turbines

- **Element Type**

- Mass, Spring, Damper, Contact, Rigid

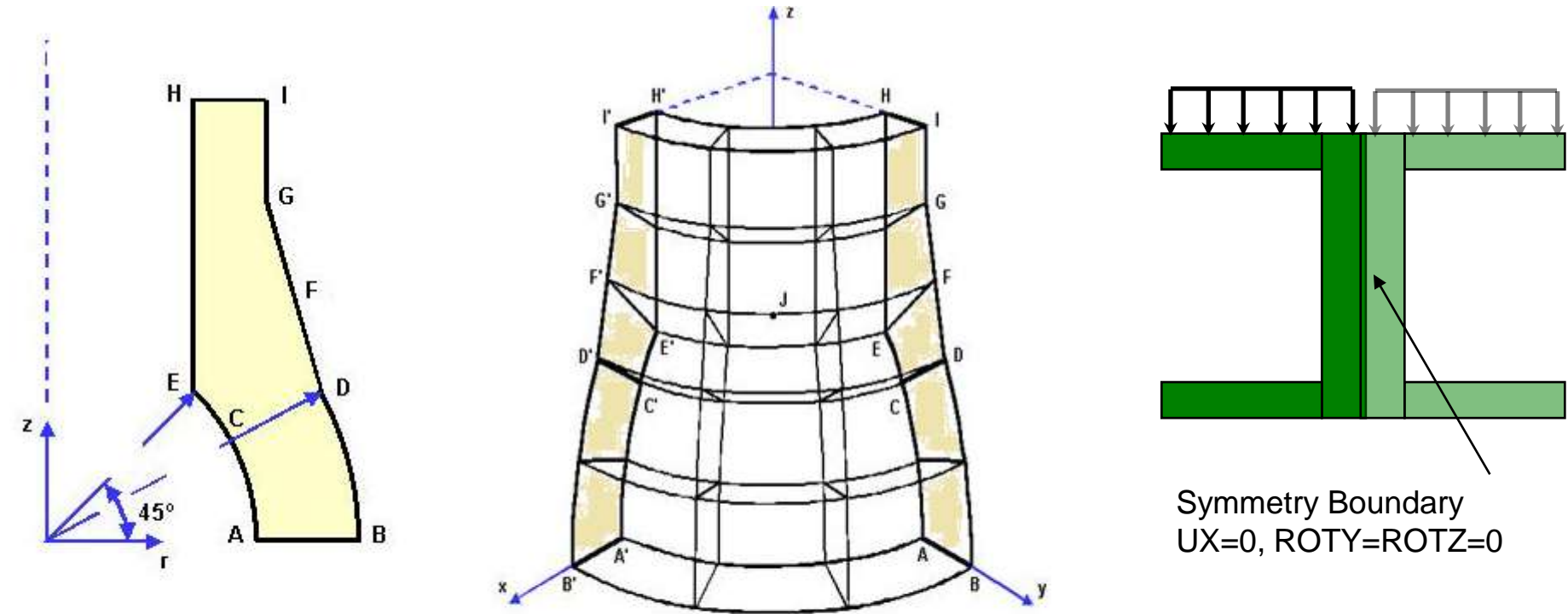
- **Practical Application**

- Concentrated Mass formulation
- Friction Formulation
- Damping effects
- Rigid connections
- Lumped Formulation

Structural CAE : Meshing Tips & Tricks



Symmetry Boundary Conditions

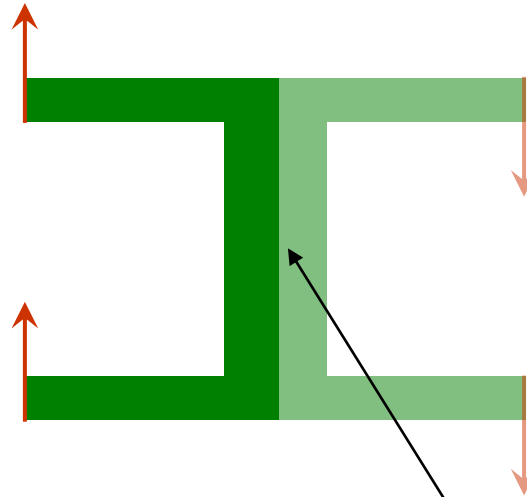


Symmetry Boundary
 $UX=0, ROTY=ROTZ=0$

Thick Shell with Pressure Loading: What is the B.C. for axisymmetric FEA Model?

Out-of-plane displacement and in-plane rotations are fixed
Input value of the force (load vector) are based on full 360 deg. Example, if force on a periphery of a cylindrical cell is P N/m, force to be applied on half-symmetrical model would be $2\pi r.P$ Output value of the force are also based on full 360 deg.

Anti-Symmetry Boundary Conditions



Antisymmetry Boundary
 $U_Y=U_Z=0, ROT_X=0$

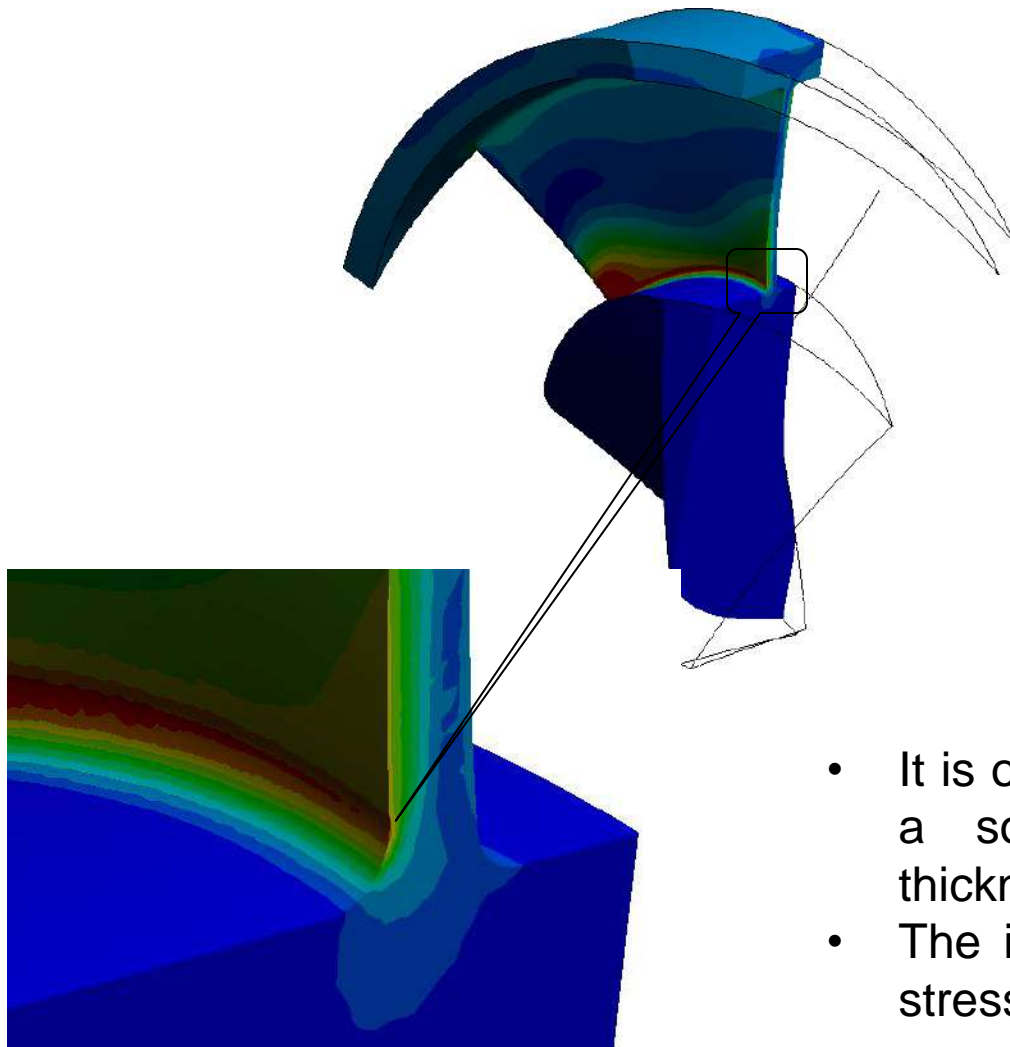
In-plane displacement & out-of-plane rotations
are fixed

Symmetric Boundary Conditions

1. In general, a symmetry condition will result in degree of freedom constraints being applied to the nodes on the symmetry plane.
2. For volume elements, the translational degree of freedom normal to the symmetry plane will be constrained.
3. For shell and beam elements, the rotational degrees of freedom in the plane of symmetry will be additionally constrained.
4. For nodes which have multiple symmetry regions assigned to them (for example, along the edge between two adjacent faces), the combined constraints associated with the two symmetry planes will be enforced.

Symmetric Boundary Conditions

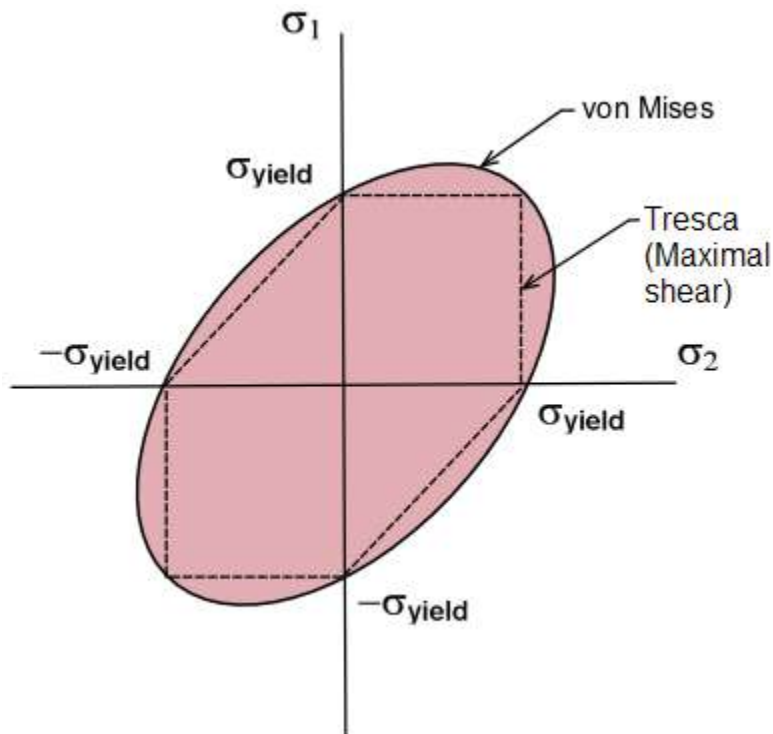
1. Axisymmetric structures – Any structure that displays geometric symmetry about a central axis (such as a shell or solid of revolution) is an axisymmetric structure. Examples would include straight pipes, cones, circular plates, domes, and so forth.
2. Models of axisymmetric 3-D structures may be represented in equivalent 2-D form. You may expect that results from a 2-D axisymmetric analysis will be more accurate than those from an equivalent 3-D analysis. By definition, a fully axisymmetric model can only be subjected to axisymmetric loads.
3. In many situations, however, axisymmetric structures experience non-axisymmetric loading. You must use a special type of element, known as a general axisymmetric element, to create a 2-D model of an axisymmetric structure with non-axisymmetric loads.



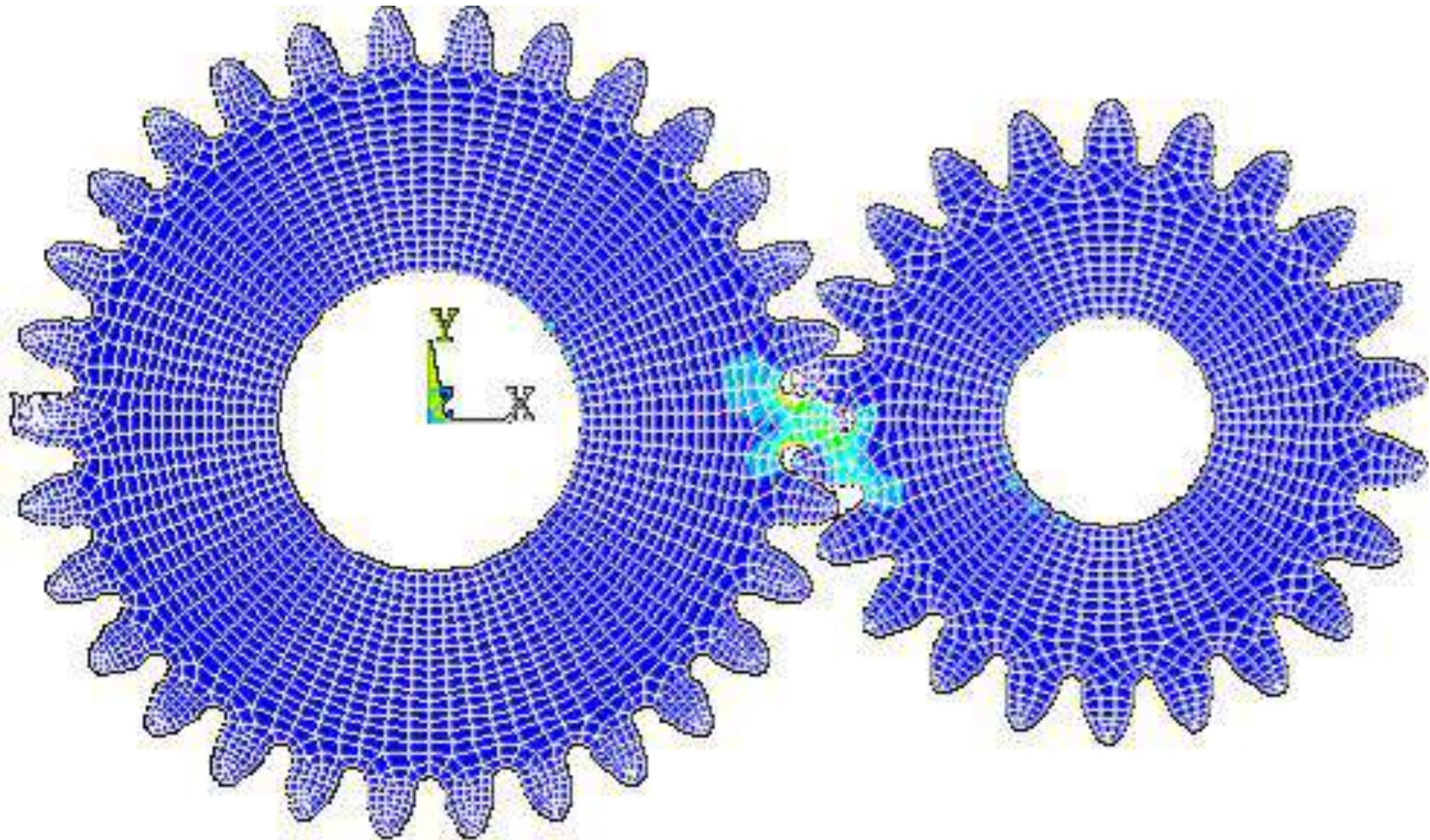
- It is often useful to slice through the thickness of a solid model to investigate through-the-thickness stresses
- The image at the left shows that the high fillet stress barely penetrates into the blade

Structural Calculations : Result Analysis

- von Mises stress is used to predict yielding of materials under multi-axial loading conditions by combining individual principal stresses into a single “equivalent” stress
- This calculated equivalent stress can then be compared to material yield data in order to determine if failure will occur.
- High von Mises stresses can sometimes be mitigated by determining which principal stress is responsible and changing geometry appropriately

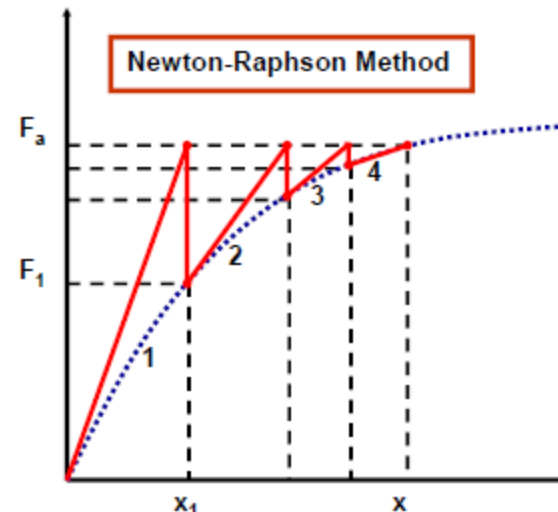
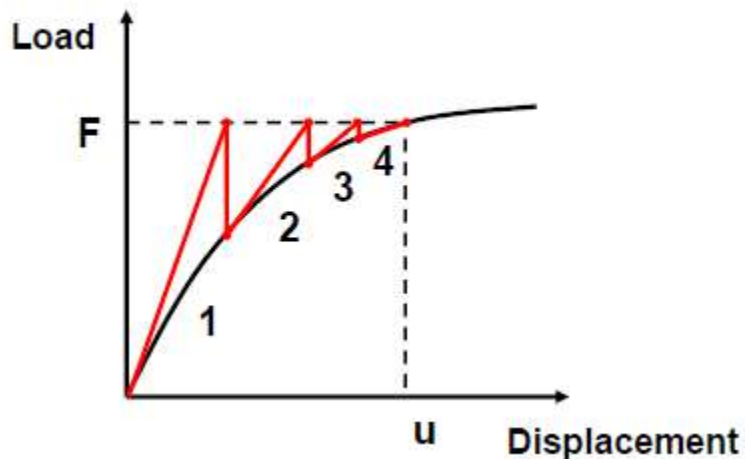


Non-linear Analysis



Non-linear Analysis

1. In a nonlinear analysis, the response cannot be predicted directly with a set of linear equations. The actual relationship between load and displacement (shown with a blue dotted line) is not known beforehand.
2. However, a nonlinear structure can be analyzed using an iterative series of linear approximations, with corrections.
3. ANSYS uses an iterative process called the *Newton-Raphson Method*. Each iteration is known as an *equilibrium iteration*.
4. A full Newton-Raphson iterative analysis for one increment of load. (Four iterations are shown.)



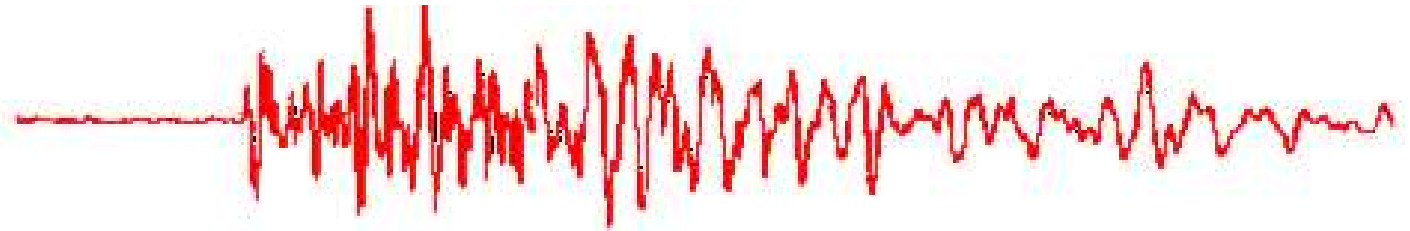
Non-linear Analysis

1. the total load F_0 is applied in iteration 1. The result is x_1 . From the displacements, the internal forces F_1 can be calculated.
2. If $F_0 \neq F_1$, then the system is not in equilibrium. Hence, a new stiffness matrix (slope of red line) is calculated based on the current conditions. 1
3. The difference of $F_0 - F_1$ is the out-of-balance or residual forces. The residual forces must be 'small' enough for the solution to converge.
4. This process is repeated until $F_0 = F_1$. In this example, after iteration 4, the system achieves equilibrium and the solution is said to be converged.
5. The difference between external and internal loads, $\{F_0\} - \{F_N\}$, is called the residual. It is a measure of the force imbalance in the structure.
6. The goal is to iterate until the residual becomes acceptably small; that is, until the solution is converged.
7. When convergence is achieved, the solution is in equilibrium, within an acceptable tolerance.

Fatigue

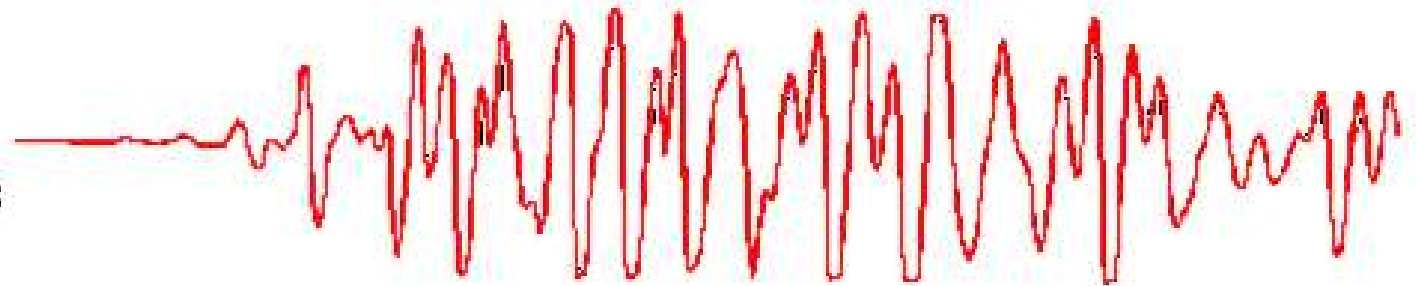
Theory & FEA

**Tectonic like
Earthquakes**



Courtesy: U.S. Geological Survey

**Shallow
Volcanic
Earthquakes**

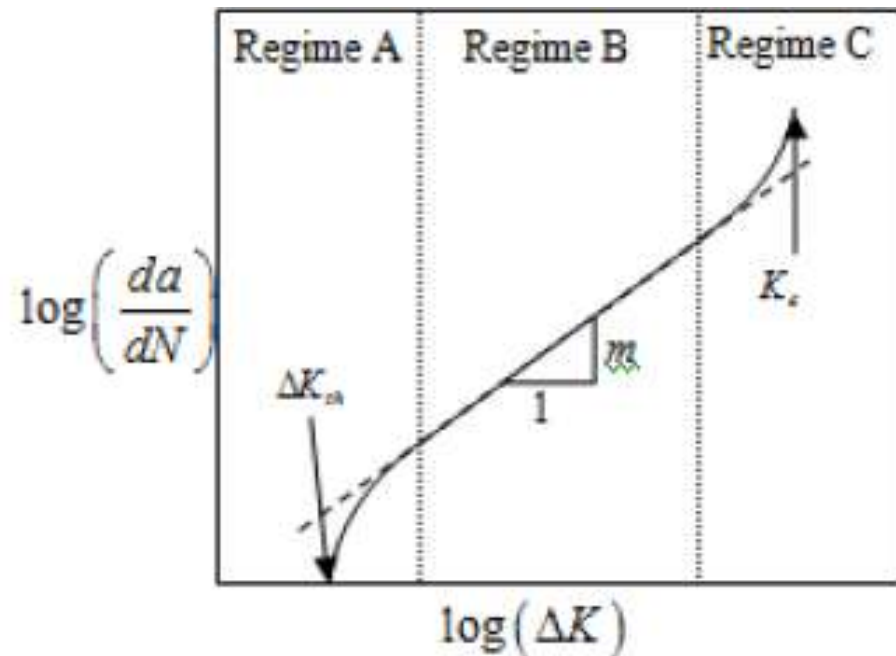
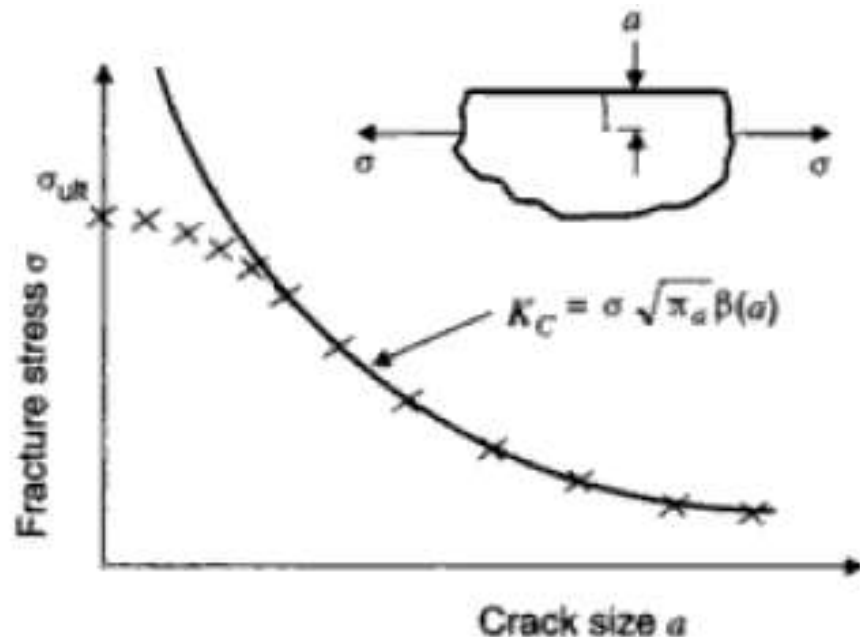


Fracture Mechanics

Fracture mechanics is a field concerned with predicting failure of a structure containing a crack –uses methods of analytical and experimental solid mechanics to characterize a material's resistance to fracture.

Fracture Mechanics

- Paris (1962) derived relationships for the stage II crack growth with cycles N , in terms of the cyclical component ΔK of the Stress Intensity Factor K .
- Two fracture modes possible –*ductile or brittle* –*relative terms* –*whether a fracture is ductile or brittle depends on situation.*
- All fractures involve crack formation followed by crack propagation. Mode of fracture highly dependent on crack propagation mechanism.
- Ductile crack propagation – stable, Brittle crack propagation –unstable.



Fracture Mechanics

- K = Stress Intensity Factor
$$K = \sigma \cdot \sqrt{\pi a} \cdot \beta$$
- σ = Applied Stress (nominal, away from the crack zone)
- a = Crack Length
- b = Dimensionless Number = function of crack length and component geometry
- K_C = Fracture Toughness = Critical Stress Intensity Factor
- N = Number of Cycles
- C, m = Material Constants

$$\frac{da}{dN} = C \cdot \Delta K^m \quad \Rightarrow \quad \log\left(\frac{da}{dN}\right) = m \log(\Delta K) + \log(C)$$

- Stress Intensity Factor range at the Crack Tip

$$\Delta K = (K_{MAX} - K_{MIN}) = (\sigma_{MAX} - \sigma_{MIN}) \cdot \sqrt{\pi a} \cdot \beta = \Delta \sigma \cdot \sqrt{\pi a} \cdot \beta$$

- **Only tensile stress needs to be use as Crack Growth stops in Compression**

Fatigue Life Prediction

$$\frac{da}{dN} = C \cdot \Delta K^m \quad \Rightarrow \quad dN = \frac{da}{C \cdot \Delta K^m} = \frac{da}{C \cdot \Delta \sigma \cdot \sqrt{\pi a} \cdot \beta(a)}$$

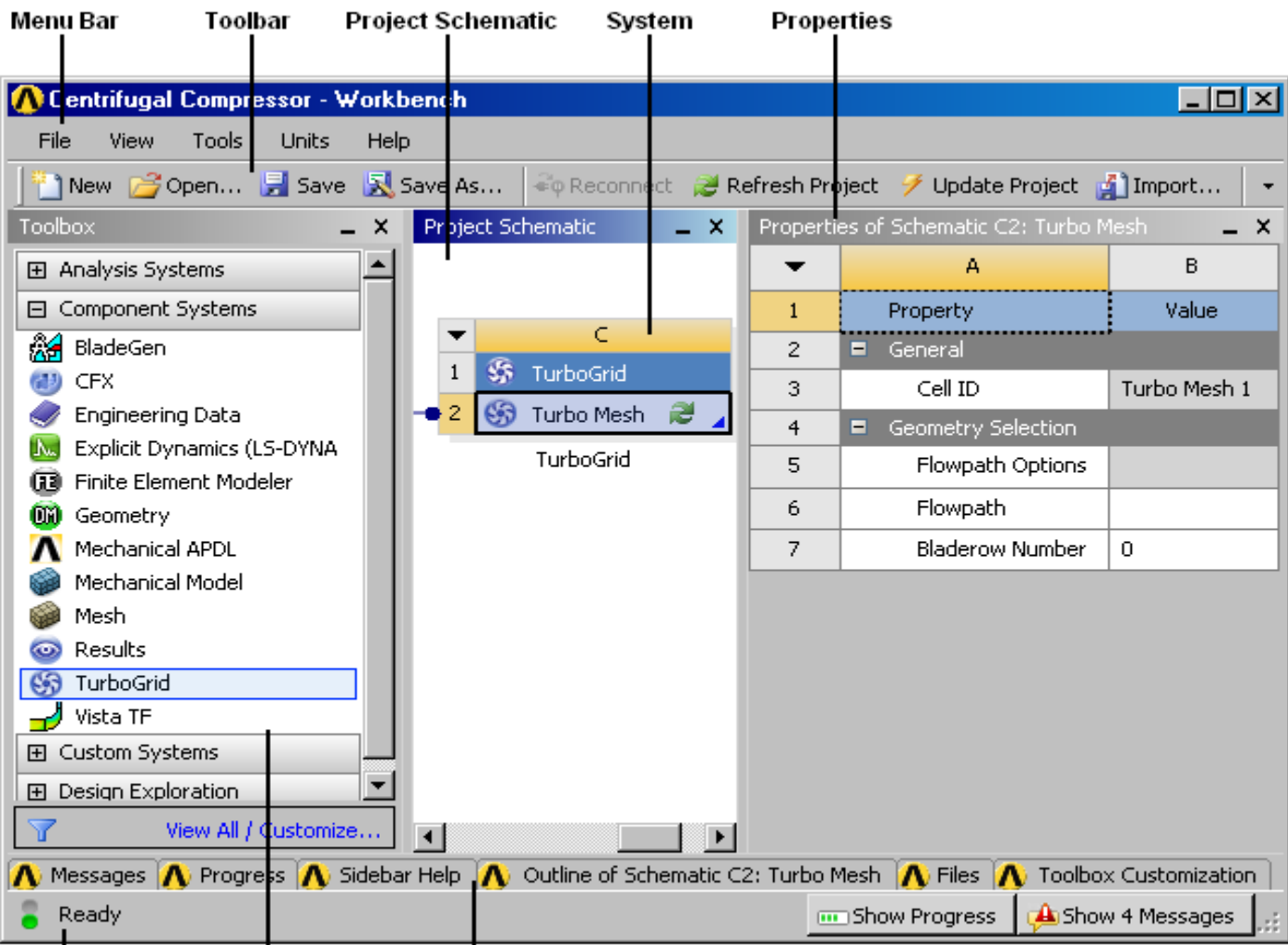
- Typically β is function of crack length a . If we assume that $\beta \neq f(a)$

$$N_F = \int_0^{N_F} dN = \int_{a_0}^{a_F} \frac{da}{C \cdot \Delta K^m} = \int_{a_0}^{a_F} \frac{da}{C \cdot [\Delta \sigma \cdot \sqrt{\pi a} \cdot \beta]^m}$$

$$N_F = \frac{1}{C \cdot (\Delta \sigma \cdot \beta)^m \cdot \pi^{\frac{m}{2}}} \int_{a_0}^{a_F} \frac{da}{a^{\frac{m}{2}}} = \frac{2}{(2-m) \cdot C \cdot (\Delta \sigma \cdot \beta)^m \cdot \pi^{\frac{m}{2}}} \left[\frac{1}{a_F^{1-\frac{m}{2}}} - \frac{1}{a_0^{1-\frac{m}{2}}} \right]$$

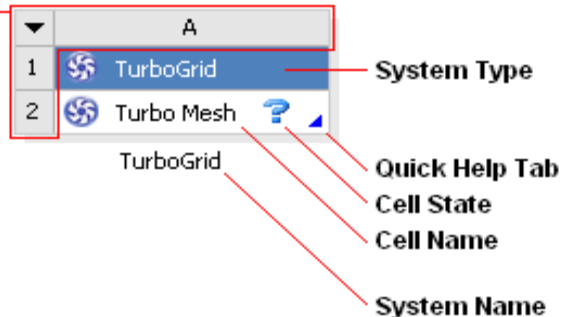
FEA using Ansys Workbench

Basics



Status Bar **Toolbox** **View Bar**

System Coordinates



Keyboard Support

The same functionality is available via your keyboard provided the **NumLock** key is enabled. The numbers correlate to the following functionality

0 = View Isometric

1 = +Z Front

2 = -Y Bottom

3 = +X Right

4 = Previous View

5 = Default Isometric

6 = Next View

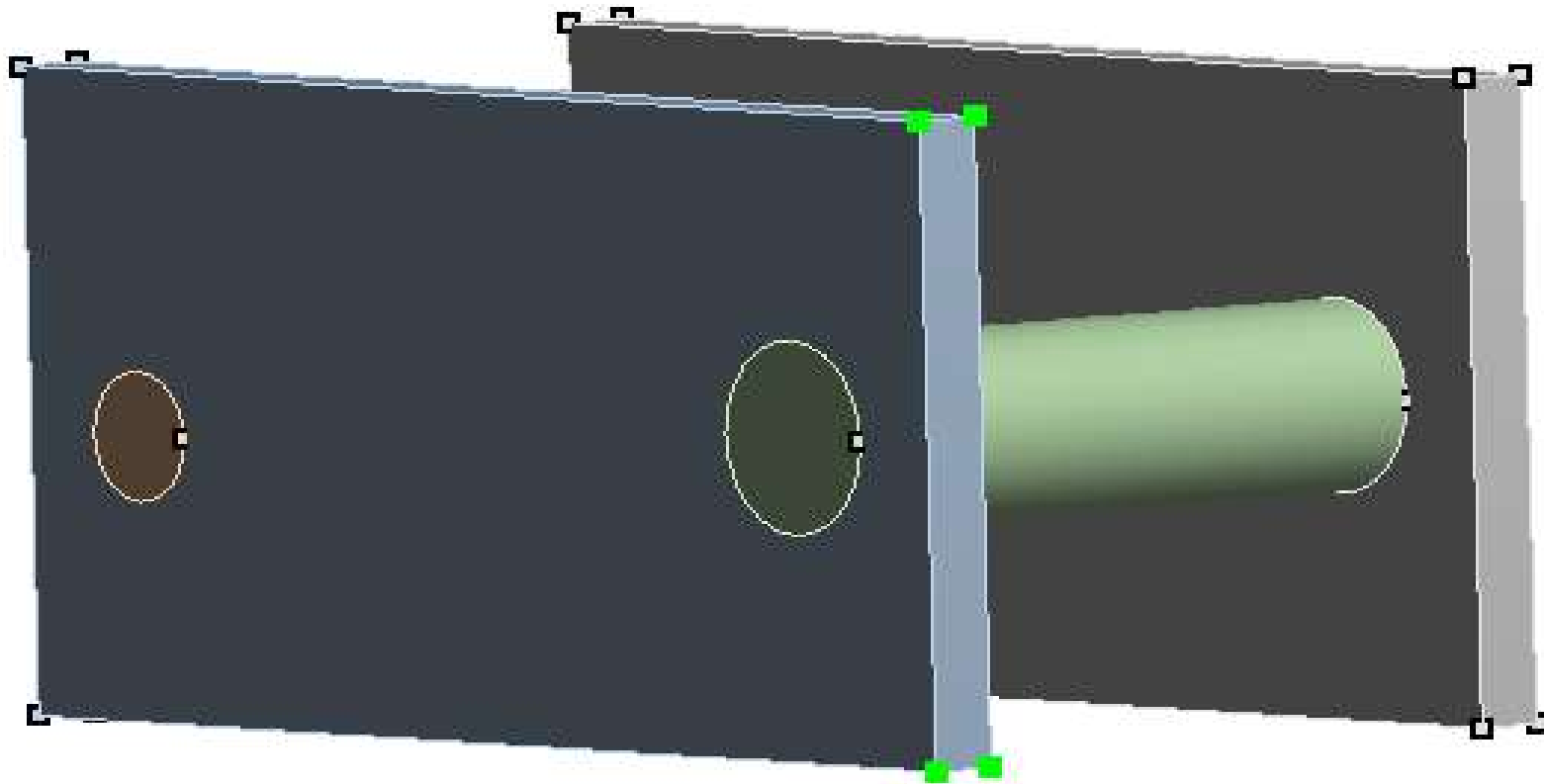
7 = -X Left

8 = +Y Top

9 = -Z Back

. (**dot**) = Set Isometric

7 -X Left	8 +Y Top	9 -Z Back
4 ← Previous	5 Default Isometric	6 Next →
1 +Z Front	2 -Y Bottom	3 +X Right
0 View Isometric		. Set Isometric



Selection Information				
Coordinate System:		Global Coordinate System	<input checked="" type="checkbox"/>	Show Individual and Summary
Entity	X(m)	Y(m)	Z(m)	Body
4 Vertices, Summary	0.1	2.5e-002	5.25e-002	
Vertex 3	0.1	0.	5.5e-002	SidePlate2
Vertex 2	0.1	5.e-002	5.5e-002	SidePlate2
Vertex 1	0.1	5.e-002	5.e-002	SidePlate2
Vertex 4	0.1	0.	5.e-002	SidePlate2

Getting Information of Geometrical Entities

The screenshot displays the ANSYS software interface. The main window shows a 3D model of a mechanical part, a green plate with a hole and a cylindrical protrusion. A context menu is open over the model, with the 'Selection Information' option highlighted. The status bar at the bottom indicates '1 Body Selected: Volume = 2.3724e-005 m³'.

ANSYS

File Edit View Units Tools Help | Solve | Worksheet | i

Shaded Exterior and Edges
Shaded Exterior
Wireframe
Graphics Options

Model

Outline

Project

- ✓ Thick Shells and Beams
- ✓ Visual Expansion
- ✓ Annotations
- Custom Annotations
- ✓ Ruler
- ✓ Legend
- ✓ Triad
- ✓ Eroded Nodes
- ✓ Annotation Labels
- ✓ Large Vertex Contours

Outline
Toolbars
Windows

Messages
Mechanical Wizard
Graphics Annotations
Section Planes
Selection Information
Reset Layout

Details of "Model"

Filter Options

Control	Enabled
---------	---------

Lighting

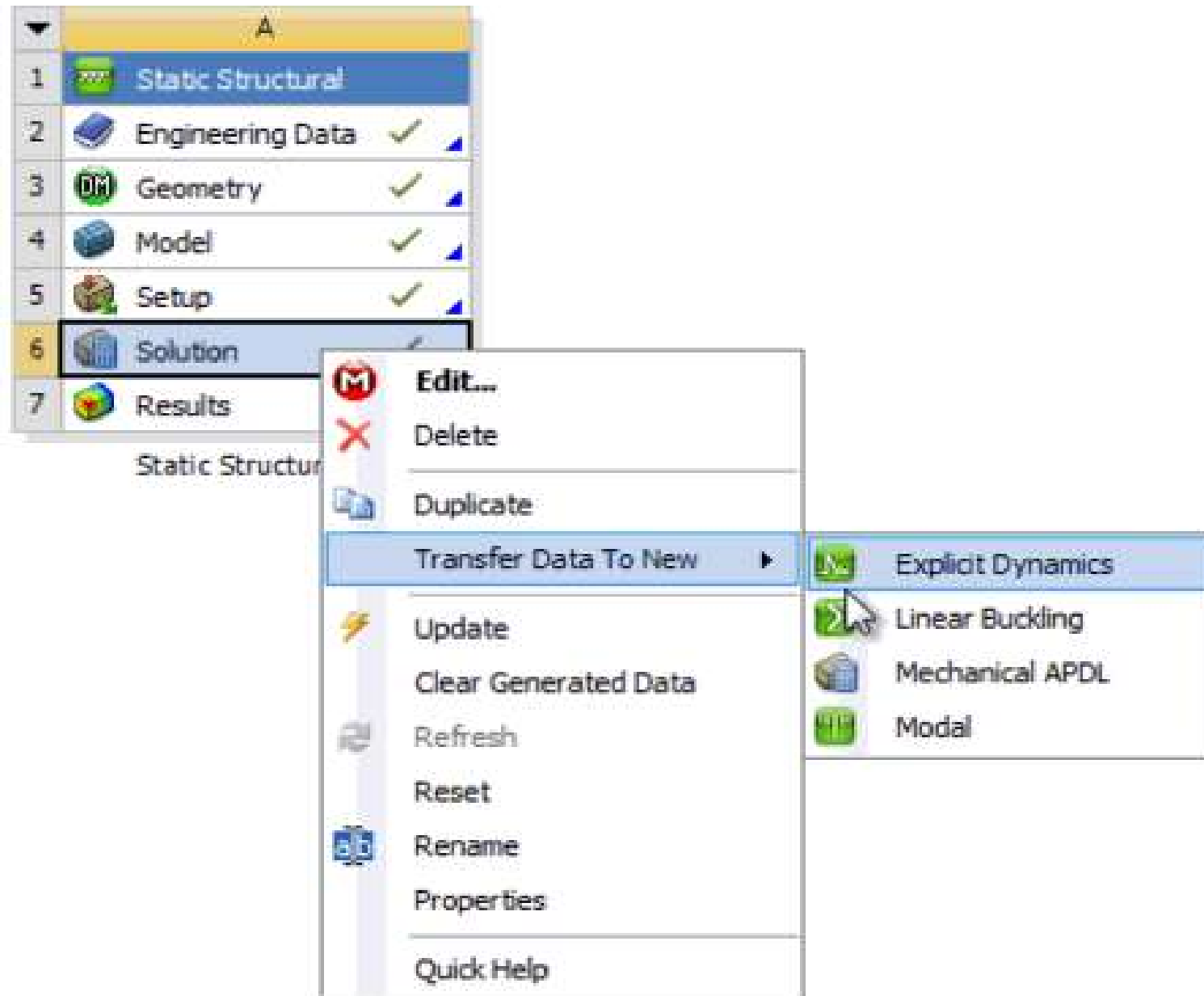
Ambient	0.1
Diffuse	0.6
Specular	1
Color	

0.000 0.025 0.050 (m)

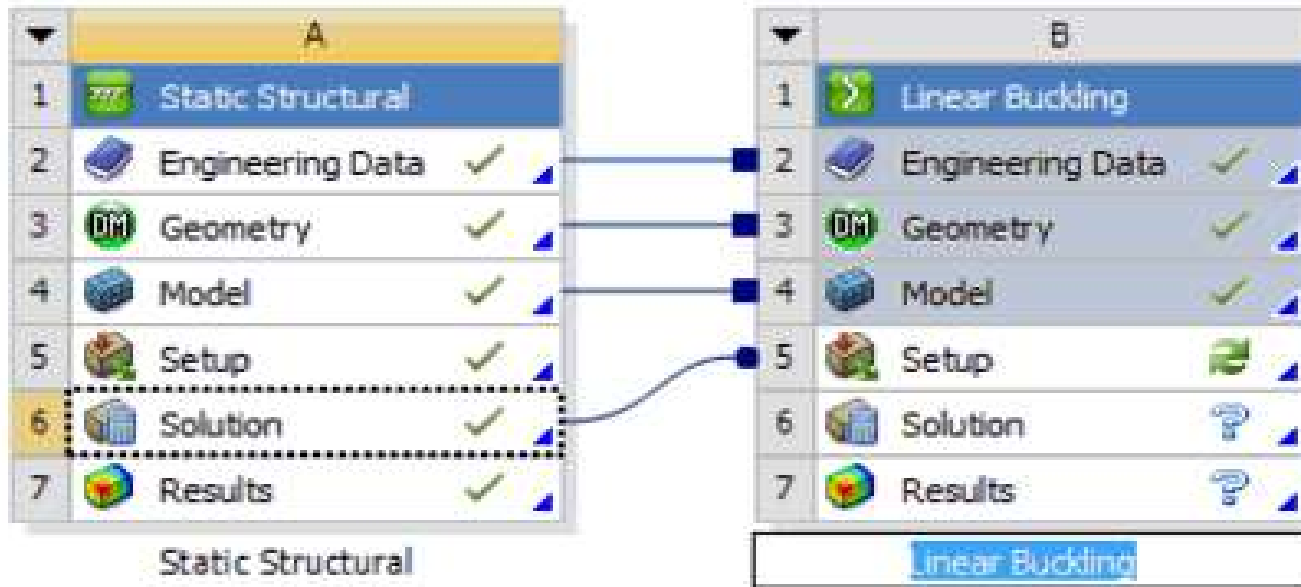
Geometry | Print Preview | Report Preview

Press F1 for Help | No Messages | 1 Body Selected: Volume = 2.3724e-005 m³ | Metric (m, kg, N, s, V, A) Degrees

Creating a connected system by Linking

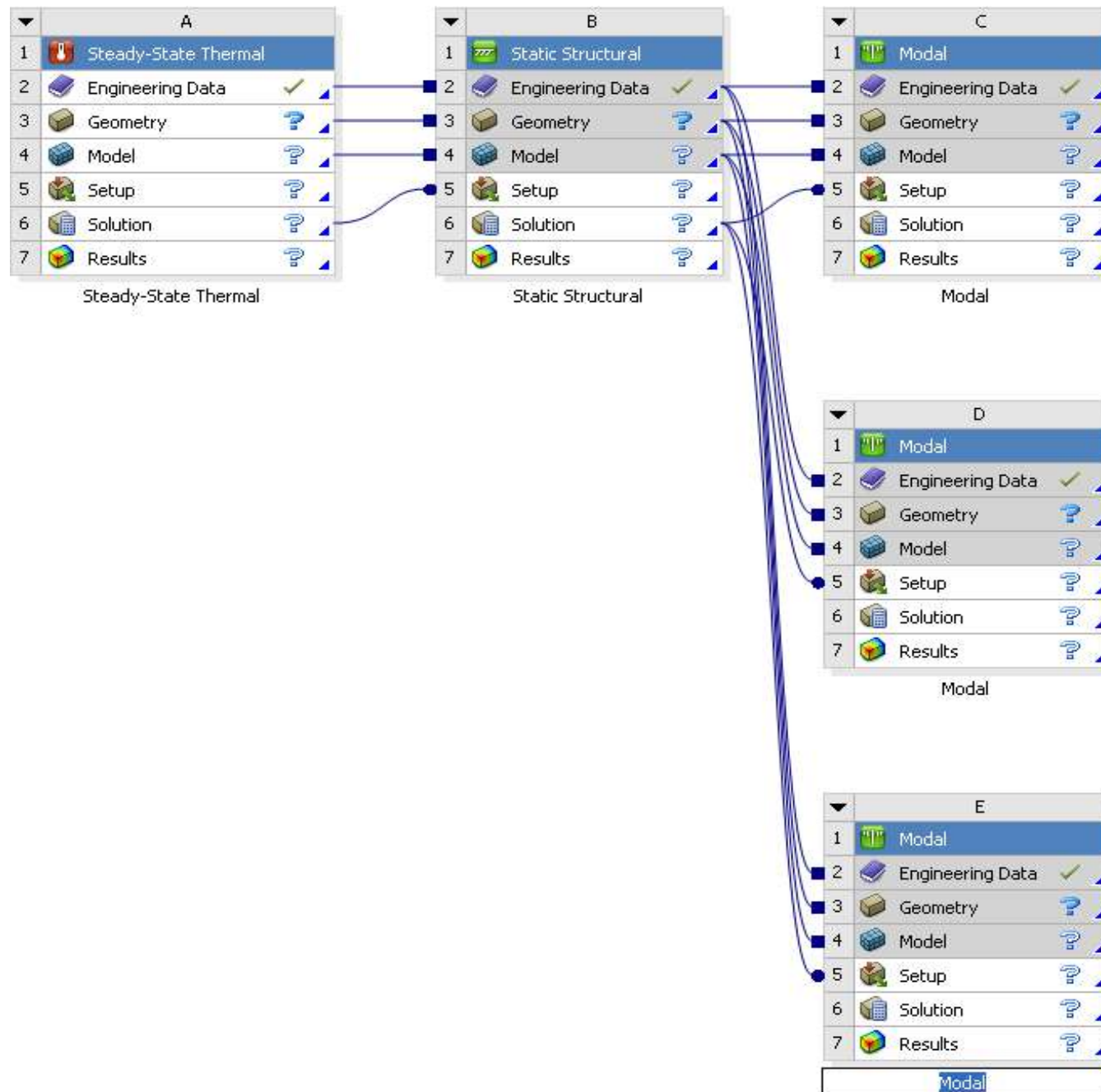


Creating a connected system by Linking

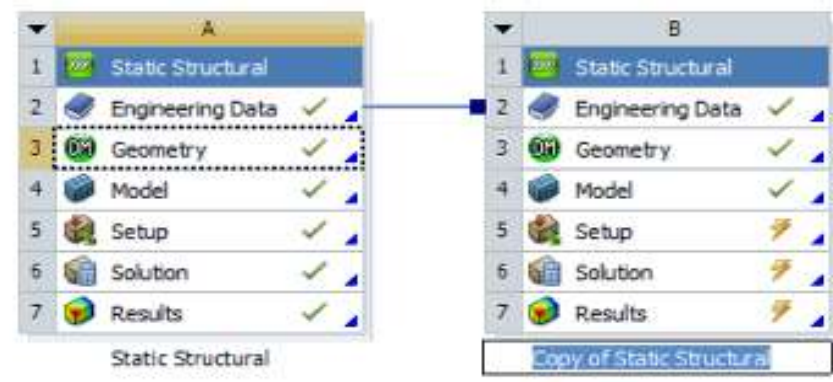
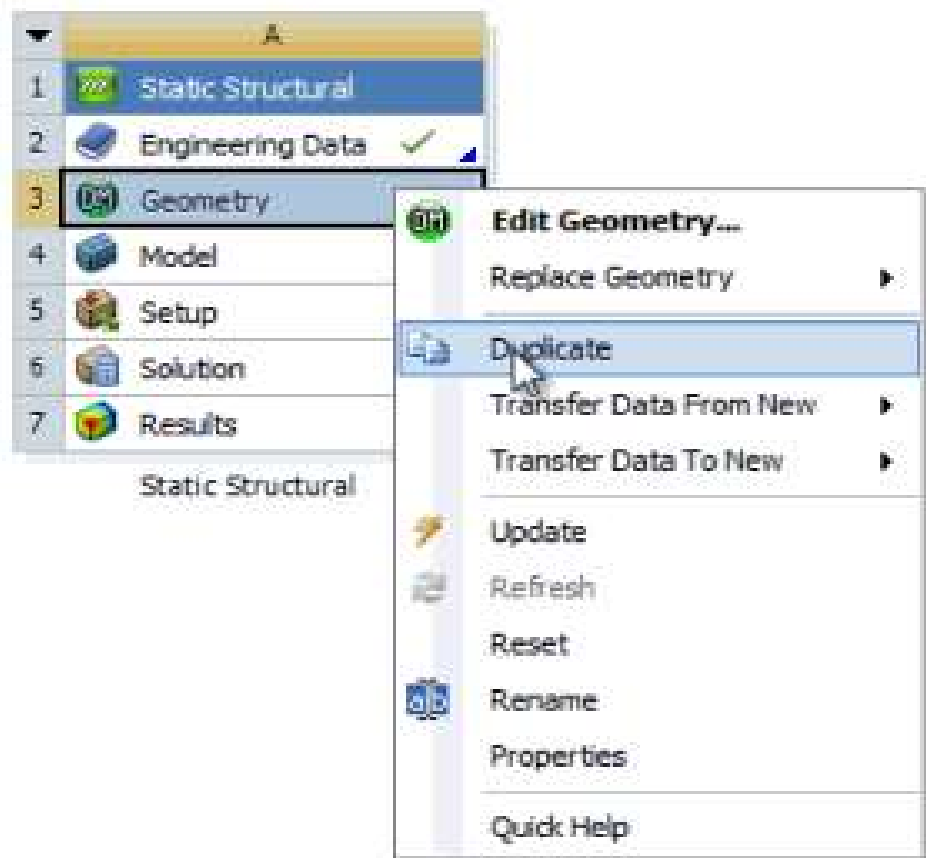


- Systems are added to the Project Schematic from left to right, and from top to bottom. Hence, all data transfer occurs from left (also called upstream) to right (also called downstream). You cannot transfer data from right to left.
- Therefore, when placing or moving systems, it is important that you place receiving systems to the right of sending systems.
- All processing of data (updates, etc.) also occurs left to right and top to bottom. Again, be aware of this order when placing or moving systems.

Creating a connected system

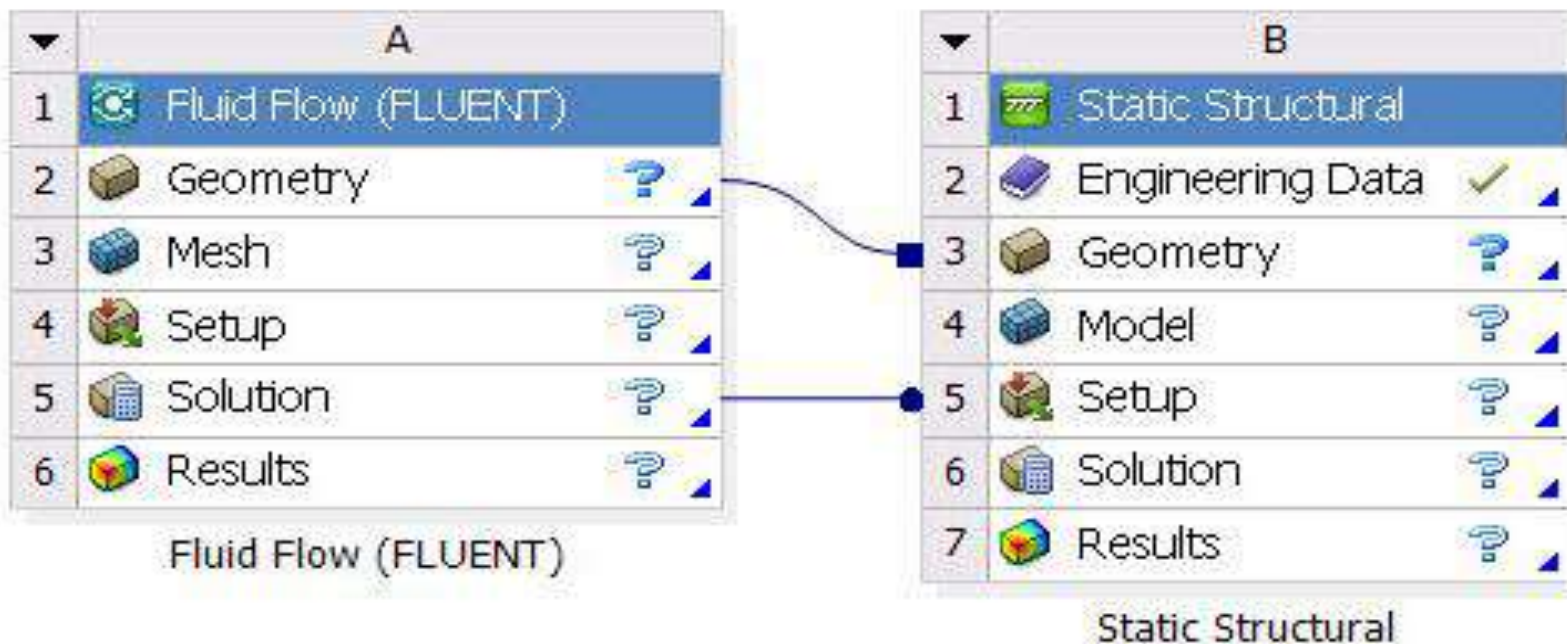


Creating a connected system by Duplication of a cell in the System



...the Engineering Data cell is shared.

Description of Links in Connected Systems



- The link with the **square terminator** indicates that the two systems share a common geometry.
- The link with the **round terminator** indicates that solution data from the Fluid Flow system is transferred as a load to the setup of the Static Structural system.
- Most systems are defined by three primary attributes: physics type, analysis type, and solver type. ANSYS Workbench uses these attributes to determine valid data transfer and system replacement possibilities.