CAD - How Computer Can Aid Design?

- Automating **Drawing** Generation
- Creating an Accurate **3D Model** to Better Represent the Design and Allowing Easy Design Improvements
- Evaluating **How Good** is the Design and Finding Design **Flaws** – Analysis (**FEA**)  
- How to Improve the Design (where to start and what to change) – **Sensitivity Analysis**
- Optimizing the Design - **Optimization**

**Finite Element Analysis (FEA) – A Useful Tool for Evaluating Design Performance**  
(Topics of Discussion)

1. Use of FEA in CAD Environment, or Computer Aided Engineering – **Pro/MECHANICA**
2. Background of FEA **Model** Generation and **Solution** Procedure
3. Capabilities and Limitations of FEA Tools
4. The Use of **CAD Model** and **FEA Tools** for Optimizing a Design
Finite Element Analysis (FEA) or Finite Element Method (FEM)

- The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics.

- Useful for problems with complicated geometries, loadings, and material properties where analytical solutions can not be obtained.

Examples of Finite Element Analysis (FEA) or Finite Element Method (FEM)
Introduction to Pro/MECHANICA

What is Pro/Mechanica?

• Pro/MECHANICA is an integrated and also independent Finite Element Analysis (FEA) module of Pro/E CAD/CAE/CAM system.
  – Pro/MECHANICA STRUCTURE
  – Pro/MECHANICA THERMAL
  – Pro/MECHANICA MOTION
Start Pro/Mechanica from Pro/E

Table 1-1 Common Pro/Mouse Functions

<table>
<thead>
<tr>
<th>Function</th>
<th>Operation</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selection (click left button)</td>
<td>LMB</td>
<td>entity or command under cursor selected</td>
</tr>
<tr>
<td>Direct View Control (drag holding middle button down)</td>
<td>MMB, Shift + MMB, Ctrl + MMB (drag vertical), Ctrl + MMB (drag horizontal)</td>
<td>Spin, Pan, Zoom, Rotate around axis perpendicular to screen</td>
</tr>
<tr>
<td>Pop-up Menus (click right button)</td>
<td>RMB with cursor over blank graphics window</td>
<td>launch context-sensitive pop-up menus</td>
</tr>
</tbody>
</table>
**Pro/Mechanica Structure**

- Static, Buckling, Contact, and Pre-stress Analyses
  - Linear static stress analysis -- most structures, except non-linearly elastic materials (such as rubber) and structures with large deformation (such as shells) (WF4 with nonlinear analysis capability)
  - Bucking analysis -- stability of slim posts.
- Vibration
  - Modal analysis (mode shapes and natural frequencies) - dynamic and vibration problems.
- Sensitivity Study (identify design parameters)
- Optimization (identify the best values of design parameters)

**Pro/Mechanica Thermal**

- Steady state and transient thermal modeling
- Sensitivity study
- Optimization

**Pro/Mechanica Motion**

3D static, kinetic, dynamic, and inverse dynamic analyses as well as interference checking
Operation Modes

- **Integrated (within Pro/E)**
  - Easy design change
  - Cannot see mesh, less FEA
- **Linked (Pro/E & Pro/M)**
  - Both interfaces; combination of the other two modes
  - Comparably more difficult to use
- **Independent (Pro/M)**
  - Strong FEA
  - Independent to Pro/E; hard to modify

Pro/ENGINEER User Interface

Integrated Mode

All Pro/MECHANICA activities take place in Pro/ENGINEER user interface.

Separate Pro/MECHANICA User Interface

Linked Mode

Most activities take place in Pro/MECHANICA user interface. Return to Pro/ENGINEER for geometry and shape change definitions.

Immediate Mode

All functions, including shape changes, take place in Pro/MECHANICA user interface. Link to Pro/ENGINEER severed.
Two Approximation Methods

h element

- (a) first order elements lead to constant stress within each element
- (b) error is reduced by reducing the element size.

p element

- (c) second order element leads to linear stress variation within each element
- (d) higher order element will reduce error even further without changing the element size

Approximation of stress function in a model

General Process

- Develop the Model
  - build your part
  - define coordinate systems, if desired
  - add materials, constraints, loads, contact regions, and measures
  - create structural idealizations for your model
**Difference between CAD Model and FEA Model**

- A CAD model is to provide a detailed document for manufacturing
- A FEA model simply captures the *rough geometry* of the design and its *loading conditions*.
  - Elimination all unimportant design details that have minor effect on the results of FEA.
  - Use of part symmetry to dramatically reduce the size of the model.
  - Elimination of uninterested portion of the design.
  - Due to the limited computation power of today
Use of Model Symmetry to Reduce the Computation Complexity – $\frac{1}{2}$ and $\frac{1}{4}$ Model

3D Shell quarter-model
of transition between cylinders

An Example
Process of Using Pro/M Structure

Overall Steps in FEA Solution

Create Geometric Model → Licentiate Model to Form Finite Element Mesh → Supply Model Parameters - material properties, loads, constraints → Solve → Compute/Display Results of Interest → Review

Stress Strain/Deformation

FEA Results

Solid model of a part

Von Mises stress fringe plot

Deformation of the part
Convergence Methods

- **Quick Check**
  - Is *not* a convergence method since the model is run only for a single fixed (low) polynomial order.
  - For error check (in defining the analysis problem)
  - The result should never be trusted

- **Single Pass Adaptive**
  - More than a Quick Check, but less than a complete convergence
  - Unless the model is very computationally intensive and/or is very well behaved and known, avoid this method

- **Multi-Pass Adaptive**
  - The ultimate in convergence analysis.
  - Base your final conclusions on the results obtained using this convergence method.

Convergence Plots for the Maximum Von Mises Stress and Strain Energy

![Convergence Plots](image-url)
Tutorials for Pro/Engineer Wildfire

7 Pro/Mechanica for Structural Analysis, Sensitivity Analysis, and Design Optimization
   7.1 Prepare the Model
   7.2 Start Pro/MECHANICA
   7.3 Define the FEA model
   7.4 Run a static analysis
   7.5 Design parameter sensitivity study
   7.6 Design optimization

8 Pro/Mechanica – Standard Static Analysis
   8.1 Objectives
   8.2 Procedures

An Example

- Preparation of the Model
  - Base of a Vise

size of the groove; why?
FEA Model from CAD Model

(a) A CAD Model
(b) A Simplified CAD Model
(C) A FEA Model

Building a FEA Model

• Coordinate System
• Material
• Loads
• Constraints
**Pre-processing**
- Invisible in the Integrated Mode

**Analysis**
- Quick Check
- Multi-pass Adaptive

**Post-processing**
- Displacement
- Von-mises Stress
- Strain Energy

**Results**
(a) Deformation  (b) Von Mises Stress
Convergence Check

(a) Von Mises  (b) Strain Energy

Parameter Sensitivity Study

- Define a design parameter (groove size, \( d \))

- Define a design study
- Perform the study and plot displacement and stress
Sensitivity Study

Different groove size causes different results
Every point represents one FEA run.

Max Displacement

Max Stress

Variation of Design Objective with Change of Design Variable

Design Optimization

Objective: minimize the maximum stress in the structure
Constraints: maximum deformation of the L bracket

design variable
Result of the Optimization

Best groove size, $d$: 0.13 (with minimum Maximum Stress)

Every point represents one FEA run.

Different Optimization Result - I
(when large deformation is allowed)
**Different Optimization Result - II**
*(when large deformation is NOT allowed)*

\[ d \leq 0.11 \]

**Max Displacement**

\[ d^* = 0.11 \]

**Max Stress**

---

**An Different Design Optimization** with Two Design Variables

**Objective:** minimize the weight (mass) of the structure

**Constraints:** maximum load and deformation allowed

1. Define relations to control the model generation (two design parameters; one is the groove size and the other is the overall fixture size.)

2. Specify ranges of variables, objective, and constraints

3. Perform the optimization (about 15 min.)

4. Results plotting and convergence check
Pro/MECHANICA

- **Integrated Mode**: The other two programs in Pro/M (Thermal and Motion) are used for thermal analysis and motion analysis of mechanical systems, respectively.

  Both of these two programs can pass information (for example temperature distributions) back to Structure in order to compute the associated stresses.

- **Design Tool**: Pro/MECHANICA is a design tool since it will allow parametric studies as well as design optimization.

- **Limitations**: Pro/M Structure has limited ability to handle non-linear problems (e.g. stress analysis involving non-linearly elastic material or large deformation).

  Problems involving large geometric deflections can be treated, as long as the stresses remain within the linearly elastic range for the material.

Quick Questions

- Why a CAD model should be simplified and unimportant portion of the model should be removed for FEA?
- Does a FEA model only include information of product geometry, loads and constraints?
- What are Pro/MECHANICA’s three convergence methods?
- What is the ideal index for FEA convergence check?
- Can Pro/MECHANICA run independently to Pro/E?
- What are the three necessary components of an optimization problem?