UNIVERSITY OF MANITOBA

Introduction to Basics of FEA and Pro/MECHANICA

25.353 Lecture Series

G. Gary Wang
Department of Mechanical and Manufacturing Engineering
The University of Manitoba

What is Pro/MECHANICA

Pro/MECHANICA is an integrated and also independent Finite Element Analysis (FEA) module of Pro/E CAD/CAM system.

- Pro/MECHANICA Structure
- Pro/MECHANICA Thermal
- Pro/MECHANICA Motion

Pro/MECHANICA Structure

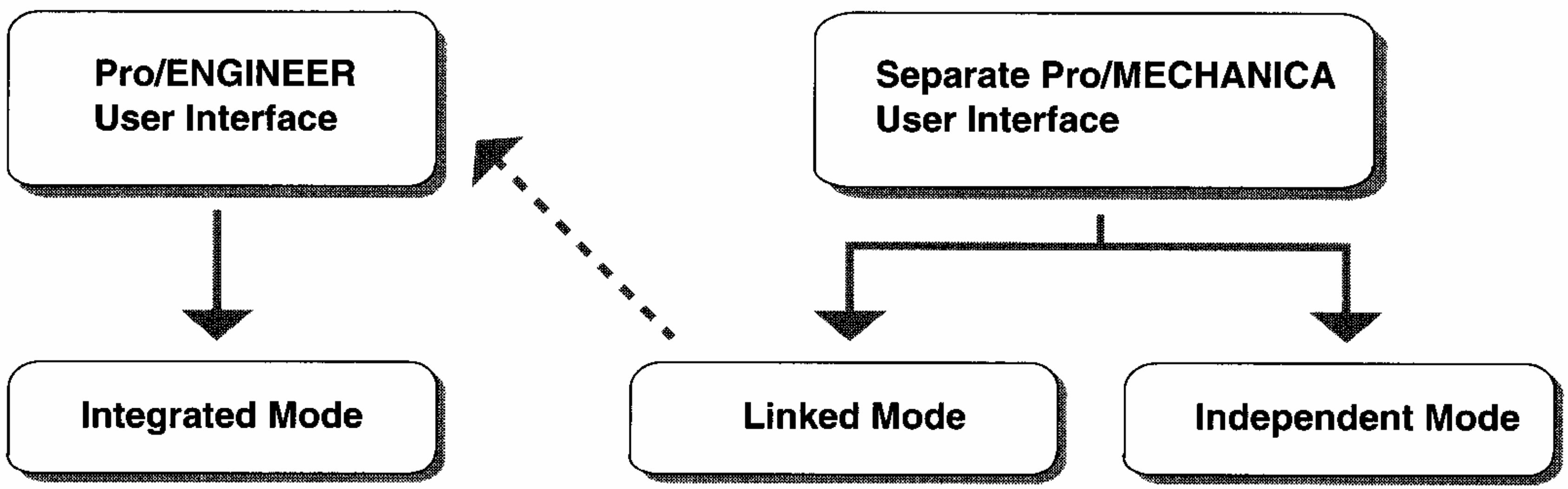
- Linear static stress analysis
- Modal analysis (mode shapes and natural frequencies)
- Buckling analysis
- Large deformation analysis (non-linear)

- **Pro/MECHANICA THERMAL** — a thermal analysis package that features many of the capabilities of Structure along with heat transfer analysis and thermal design optimization.
- **Pro/MECHANICA MOTION** — a motion analysis package that provides mechanism modeling and mechanism design optimization capabilities. This product enables you to analyze your mechanism's motion and forces.

Operation Modes

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

Operation Modes



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

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

Independent Mode: All functions, including shape changes, take place in Pro/MECHANICA user interface. Link to Pro/ENGINEER severed.

Modes of Operation

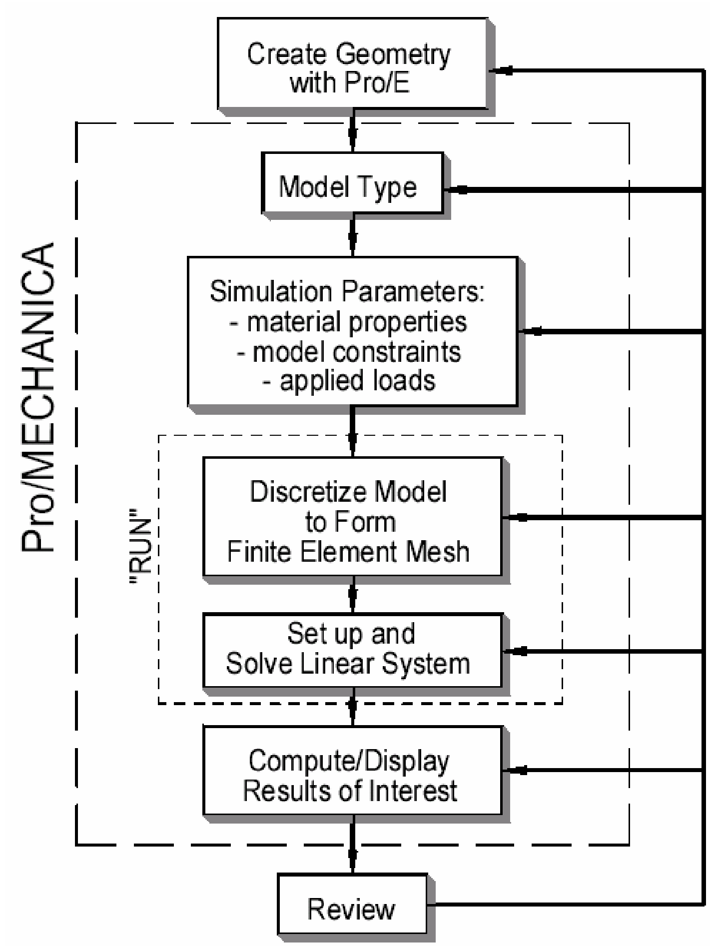
Integrated Mode	Independent Mode
Pro/E interface	Pro/M interface
all analyses available	all analyses available
2D and 3D models	2D and 3D models
some measures of results not available	all measures available
some analysis options not available (eg excluding elements)	all options available
all elements generated automatically	element creation manual or automatic
sensitivity and optimization using Pro/E parameters only	sensitivity and optimization uses Pro/M variables

Common Unit System

Quantity	System and Units			
	SI MNS	Metric mm-N-s	English FPS ft-lb-sec	English IPS in-lb-sec
length	m	mm	ft	in
time	s	s	sec	sec
mass	kg	tonne (1000 kg)	slug	lbf-sec^2 / in
density	kg/m^3	tonne/mm^3	slug/ft^3	lbf-sec^2 / in^4
gravity, g	9.81 m/s^2	9810 mm/s^2	32.2 ft/sec^2	386.4 in/sec^2
force	N	N	lbf	lbf
stress, pressure, Young's modulus	N/m^2 = Pa	N/mm^2 = MPa	lbf/ft^2	lbf/in^2 = psi

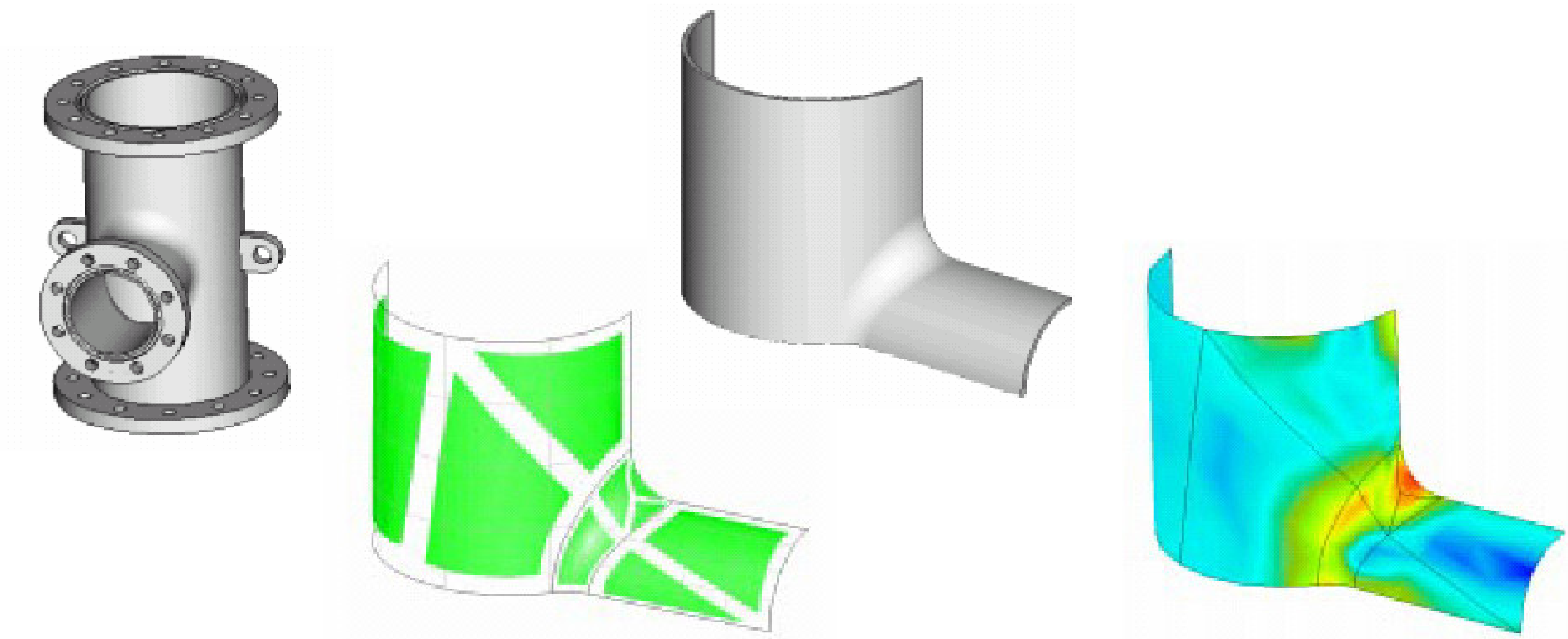
Steps in Preparing an FEA Model for Solution

1. Identify the model type
2. Specify the **material properties**, model **constraints**, and applied **loads**
3. Discretize the **geometry** to produce a finite element mesh
4. Solve the system of equations
5. Compute items of interest from the solution variables
6. Display and critically review results and, if necessary, repeat the analysis



A CAD Model is NOT a FEA Model!

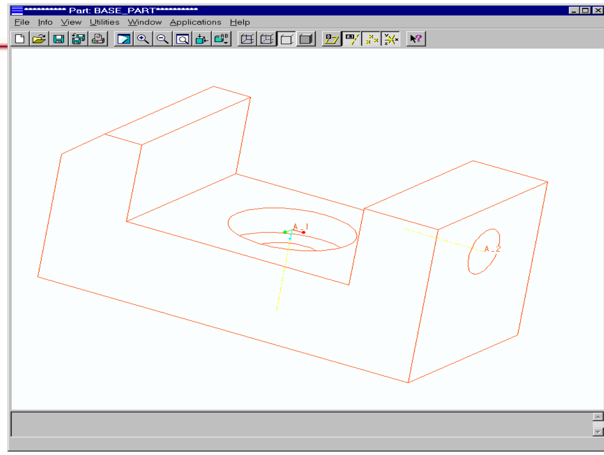
Use the simplest model possible that will yield <u>sufficiently reliable results</u> of interest at the <u>lowest computational cost</u>.

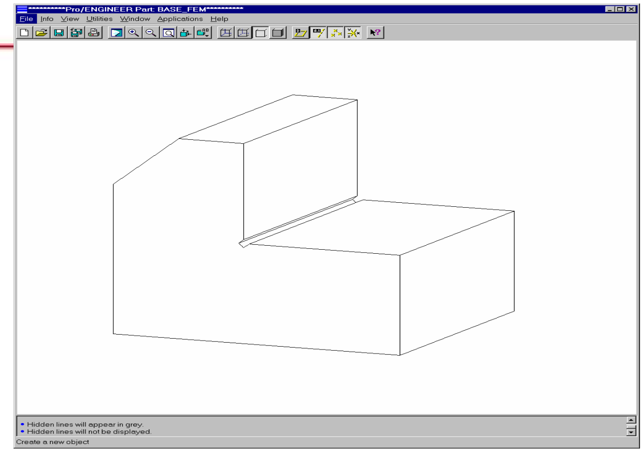


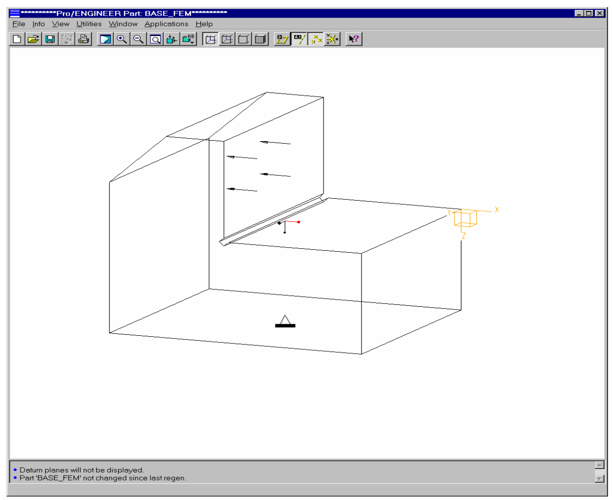
Difference between a CAD & a FEA Model

- 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.

Involved Models in the Tutorial







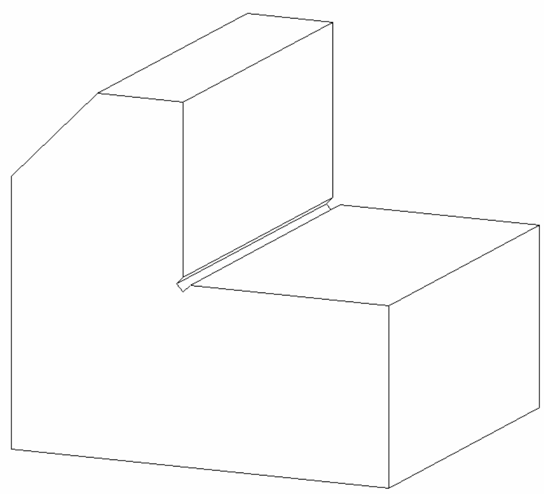
(a) A CAD Model

(b) A Simplified CAD Model

(C) A FEA Model

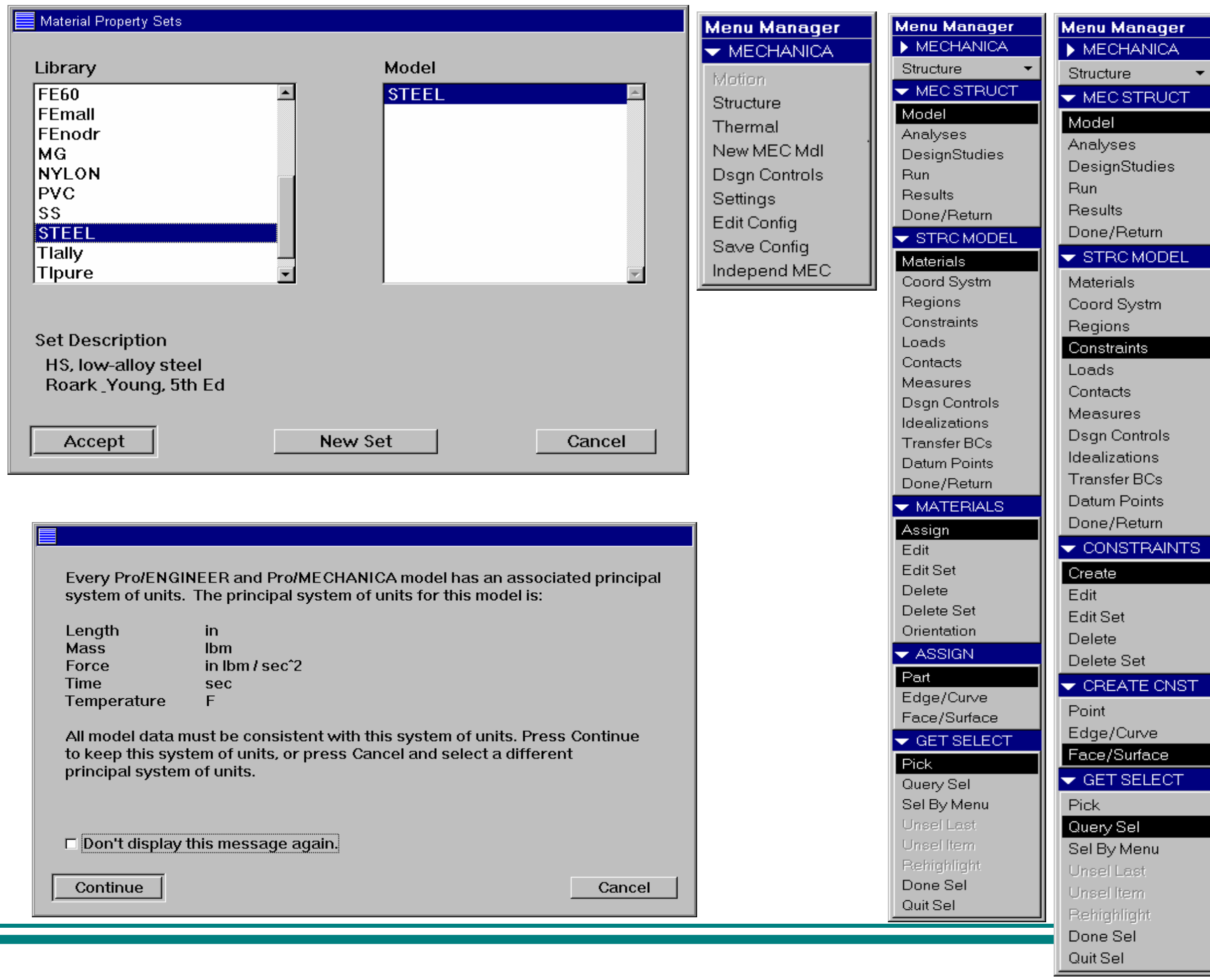
Procedure of the Lab

- Preparation of the Model / Create the geometry in Fig. 49



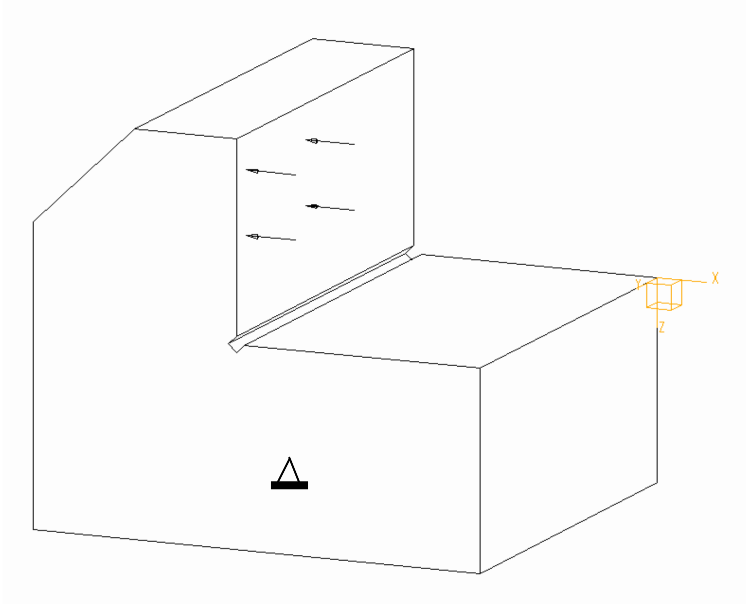
Procedure of the Lab

- Start Pro/Mechanica from Pro/E



Building a FEA Model

- Materials
- Loads
- Constraints



Pre-processing

- Invisible in the Integrated mode

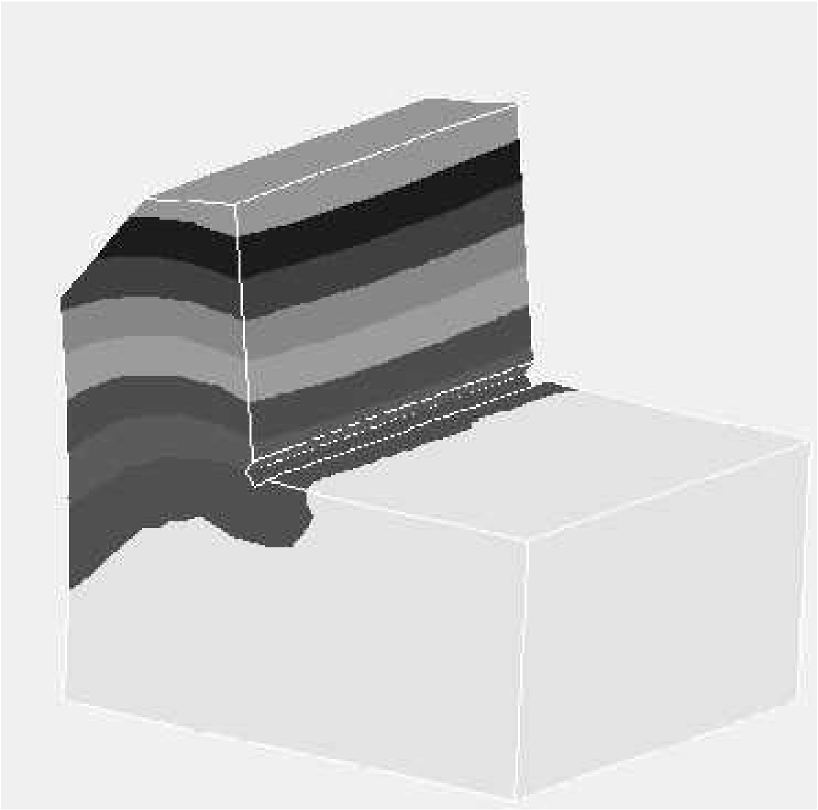
Analysis

- Quick Check
- Multi-pass Adaptive

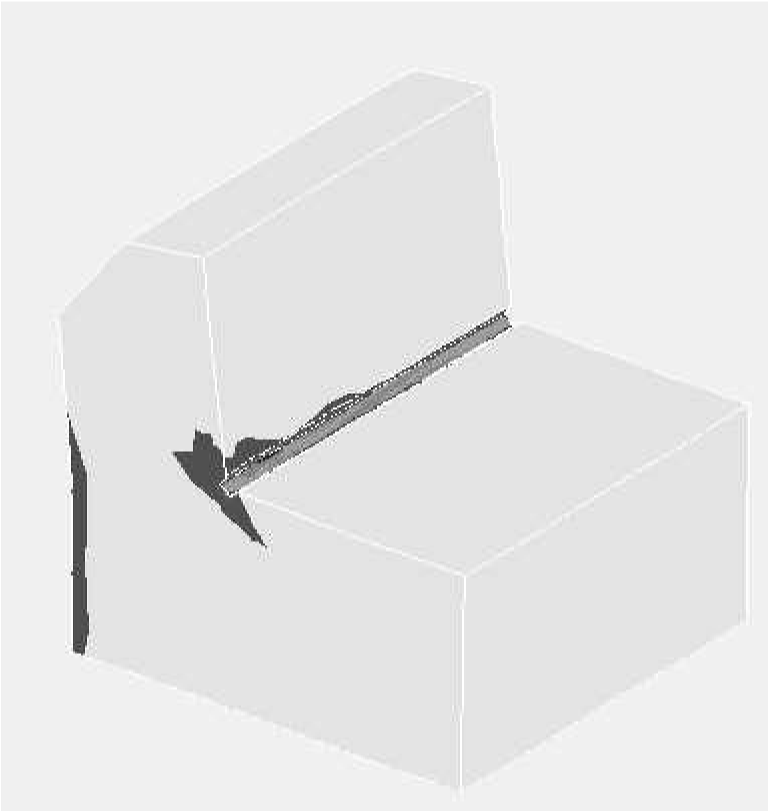
Post-processing

- Displacement
- Von-mises stress
- Strain energy

Result Models

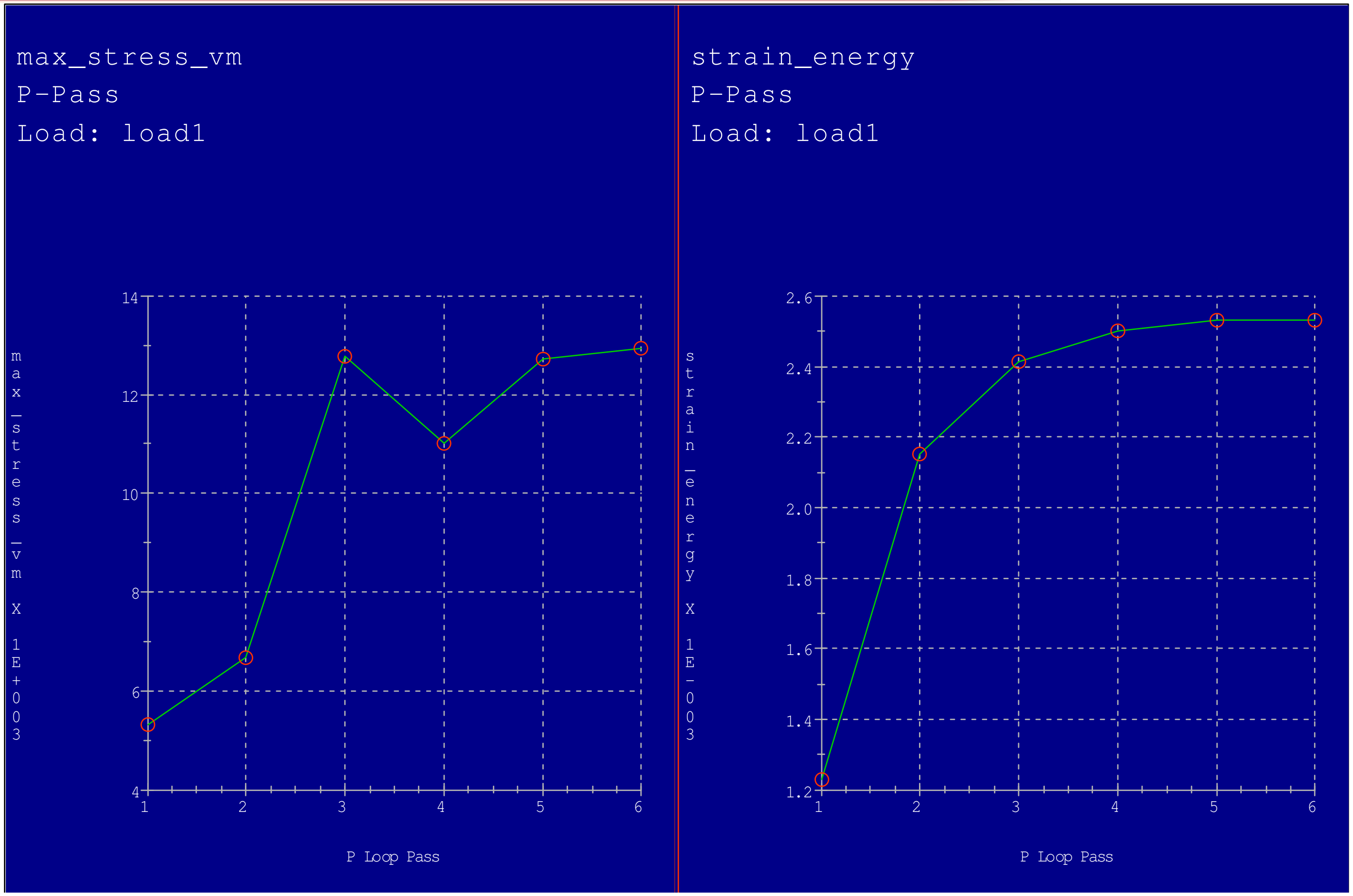


(a) Deformation



(b) Von Mises Stress

Convergence Check



(a) Von Mises (b) Strain Energy

General Design/FEA Cycle

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

Analyze the Model

- define an analysis
- run the analysis
- review the analysis results

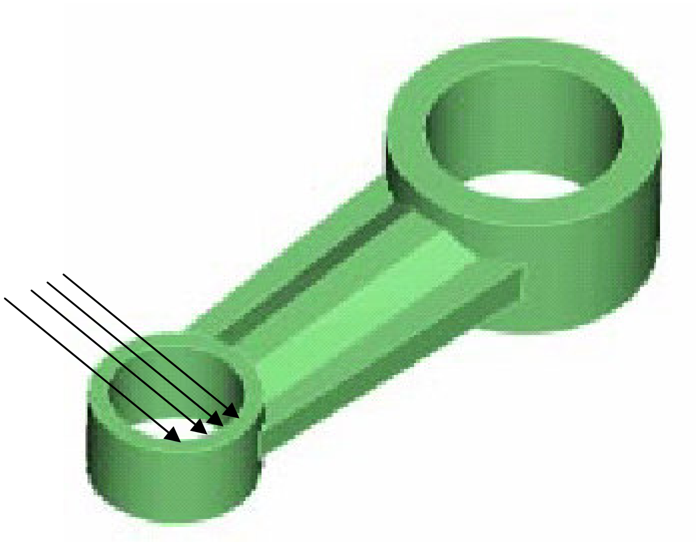
Define Model Changes

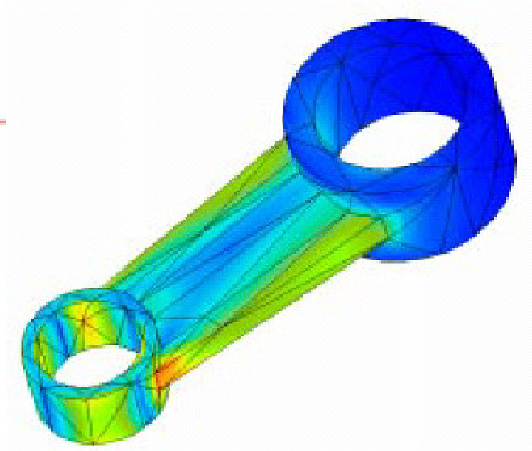
- define the design parameters and design variables
- review and modify the shape changes

Optimize the Model

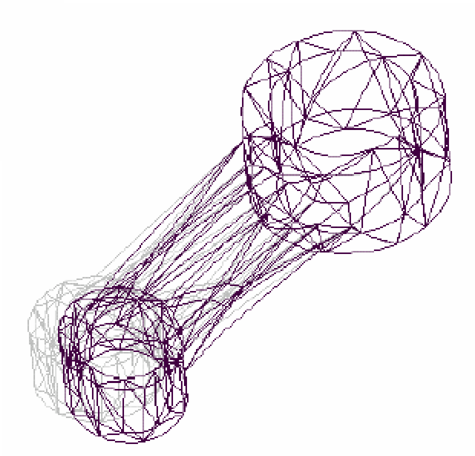
- define sensitivity and optimization studies
- run the studies
- review the study results
- accept the optimized design

Analysis



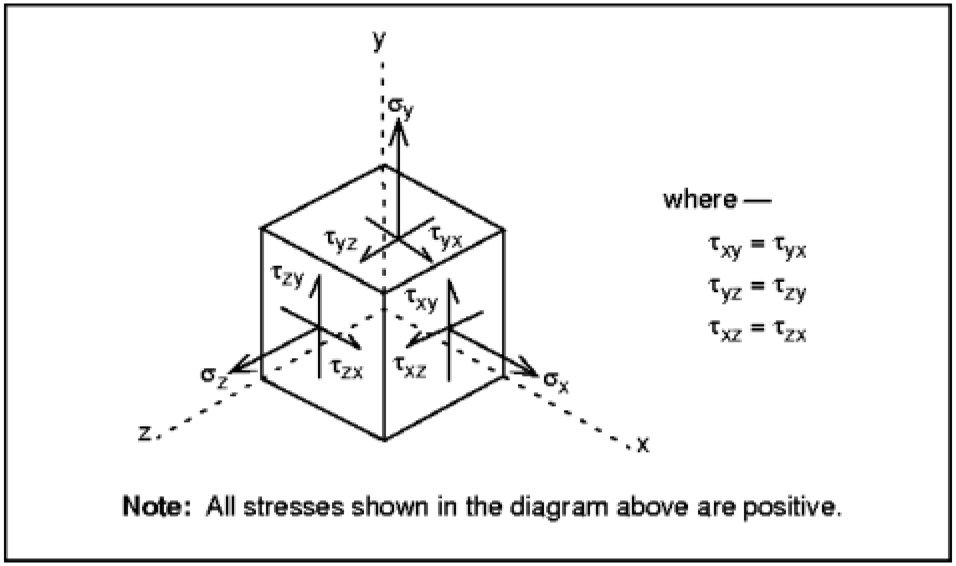


Contours of the Von Mises stress



Total deformation of the part

Von Mises Stress

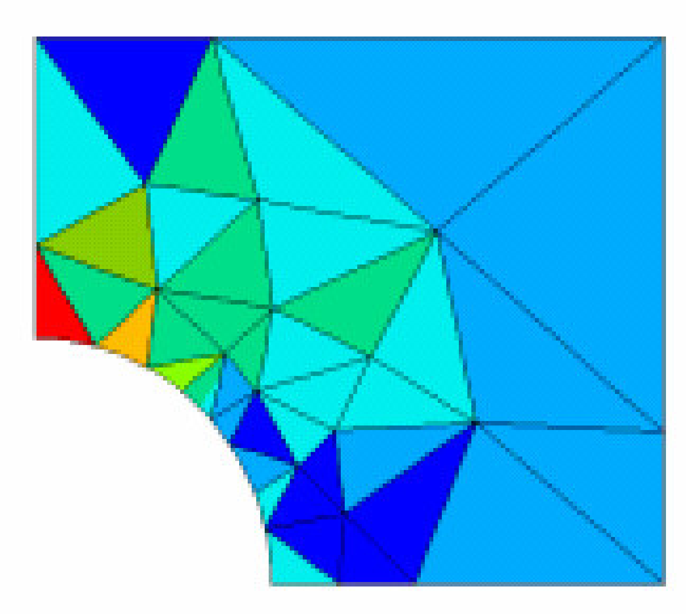


The Von Mises stress is obtained by combining all the stress components at a point

Image adapted from Ref. 2

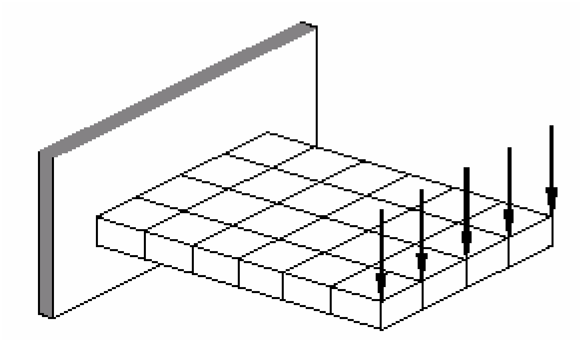
Convergence of FEA

- No FEA solution should be accepted unless the convergence properties have been examined

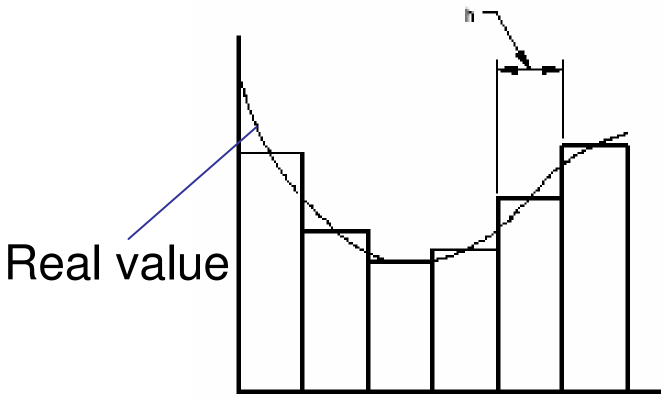


Cost vs. Accuracy

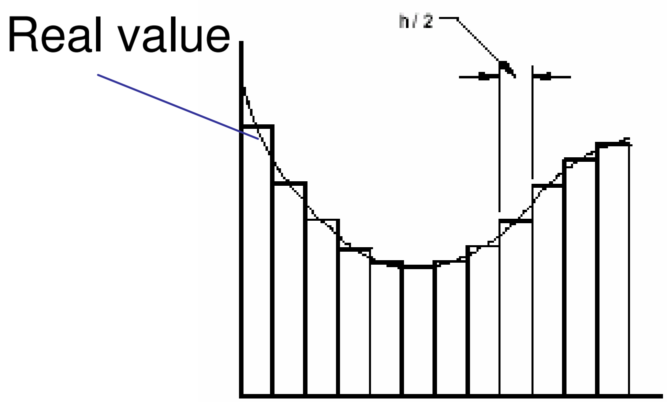
Simulation vs. Experiment



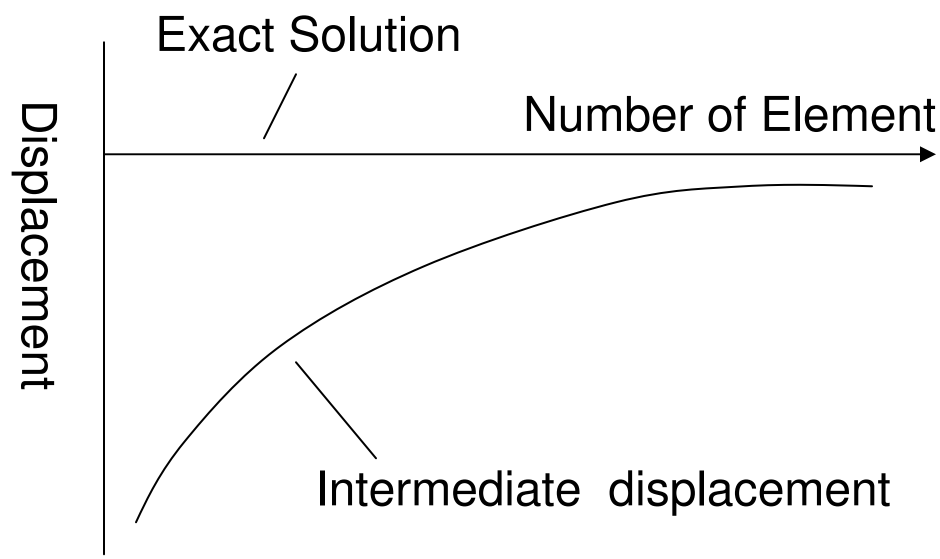
Convergence of H-elements



(a) first order elements lead to constant stress within each element



(b) error is reduced by reducing the element size O(h)



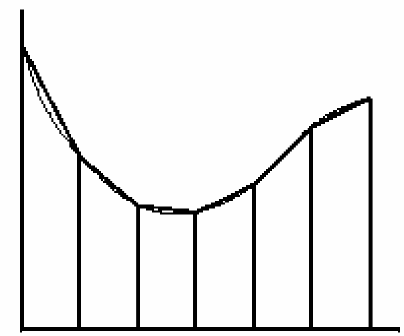
H – size of element

Convergence of P-elements

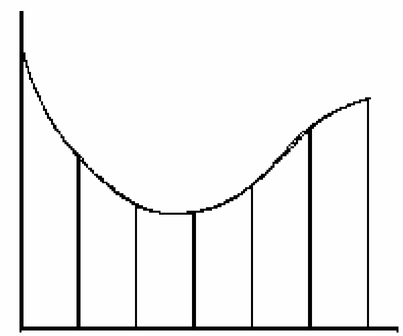
➢Up to 9th order polynomials in Pro/MECHANICA

➢The same mesh can be used throughout the convergence analysis, rather than recreating meshes or local mesh refinement required by h-codes.

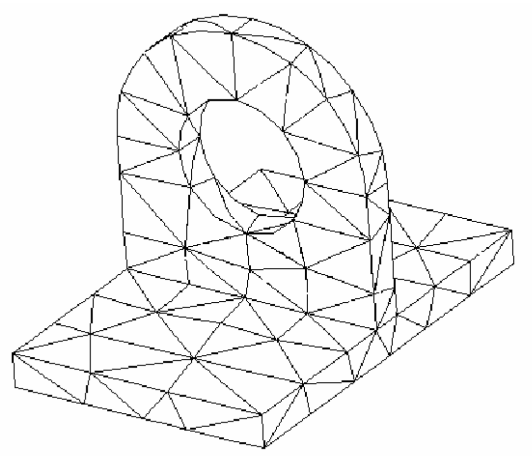
➢The mesh is virtually always more coarse and contains fewer elements than h-codes.



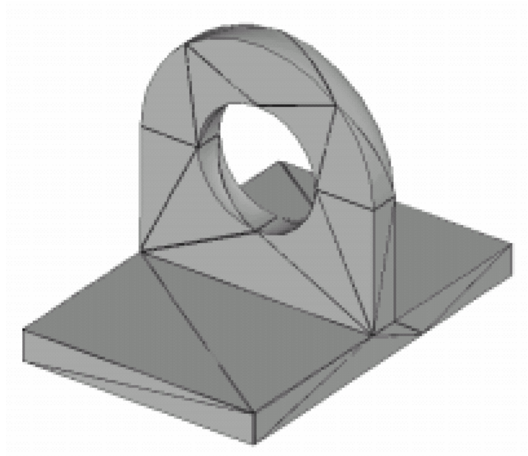
(c) second order element leads to linear stress variation within each element



(d) higher order element will reduce error even further without changing the element size



A mesh of h-elements

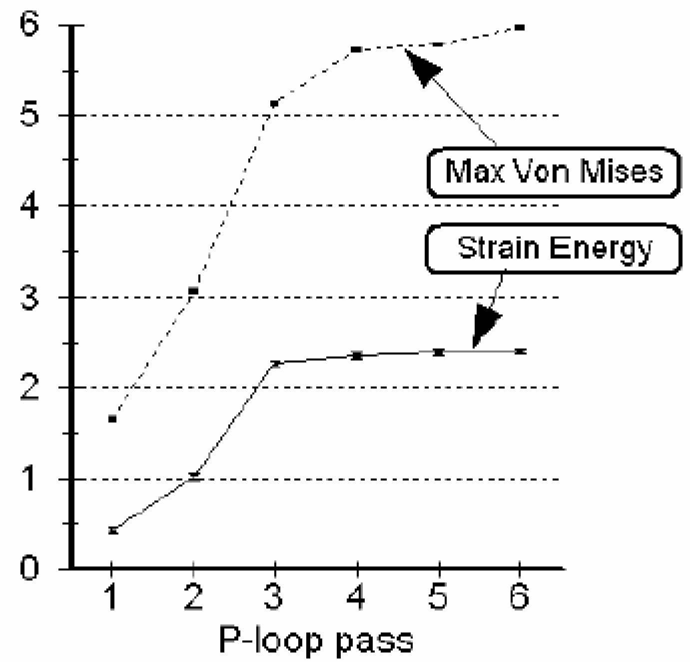


A mesh of p-elements

Convergence of P-elements (Cont'd)

- Automatic mesh generators is more effective for p-elements rather than h-element.

- Mesh is tied directly to the geometry.



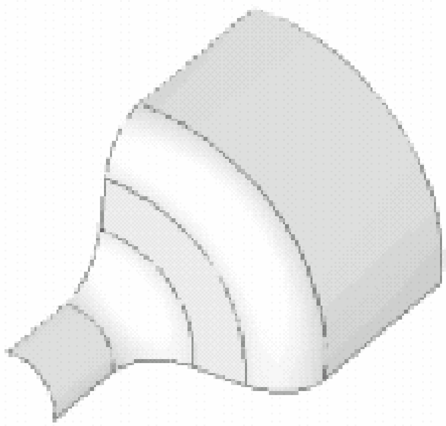
Convergence Methods

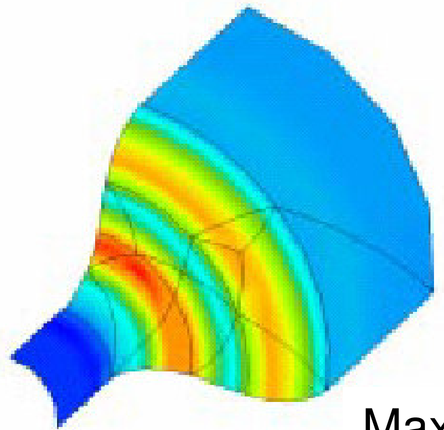
- **Quick Check** - the model is run only for a polynomial of order 3. The results of a Quick Check should never be trusted.

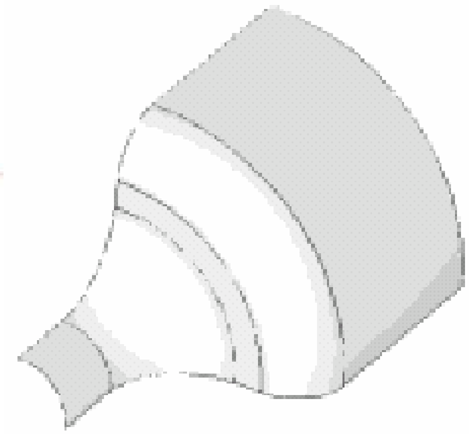
- **Single Pass Adaptive** - the single pass adaptive method performs one pass at a low polynomial order, assesses the accuracy of the solution, modifies the p-level of “problem elements”, and does a final pass with some elements raised to an order that should provide reasonable results.

- **Multi-Pass Adaptive** - The ultimate in convergence analysis. Multiple “p-loop” passes are made through the solver, with edge orders of “problem elements” being increased with each pass. This iterative approach continues until either the solution converges to a specified accuracy or the maximum specified edge order (default 6, maximum 9) is reached.

Sensitivity Study

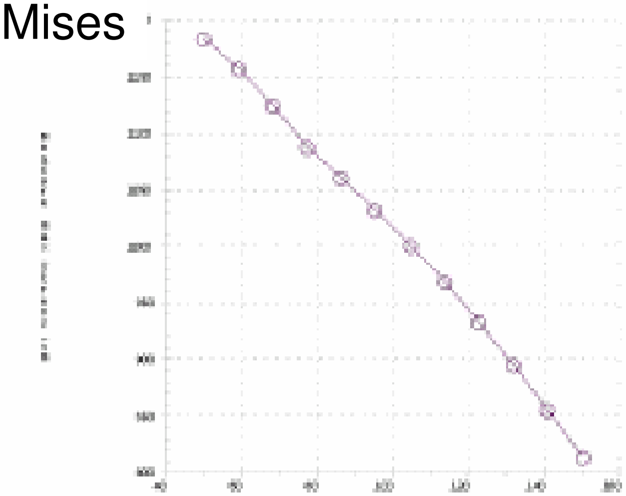




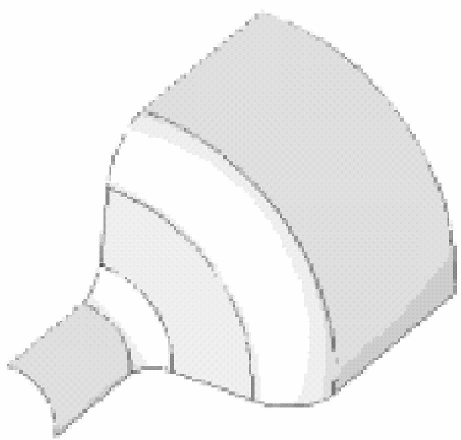


Max fillet radius

Max Von Mises



Fillet radius



Min fillet radius

Design Optimization

Optimization: A fascinating, useful, mathematical tool

An Optimization Model:

Min Mass of a Mug as a function of dimensions (D: Diameter, Height, Thickness) ***-- Objective***

Subject to

Mug Volume >= A Constant ***-- Inequality Constraint***

H/D = 1.65 ***-- Equality Constraint***

D, H, T > 0 ***-- Variables***

Find: D*, H*, and T* ***-- Optimum***

Design Optimization

Another Optimization Model:

<u>Min</u> Manufacturing Cost of the Mug -- ***Objective***

<u>Subject to</u>

Mug Volume > Constant 1 -- ***Inequality Constraint***

Mug Mass <= Constant 2

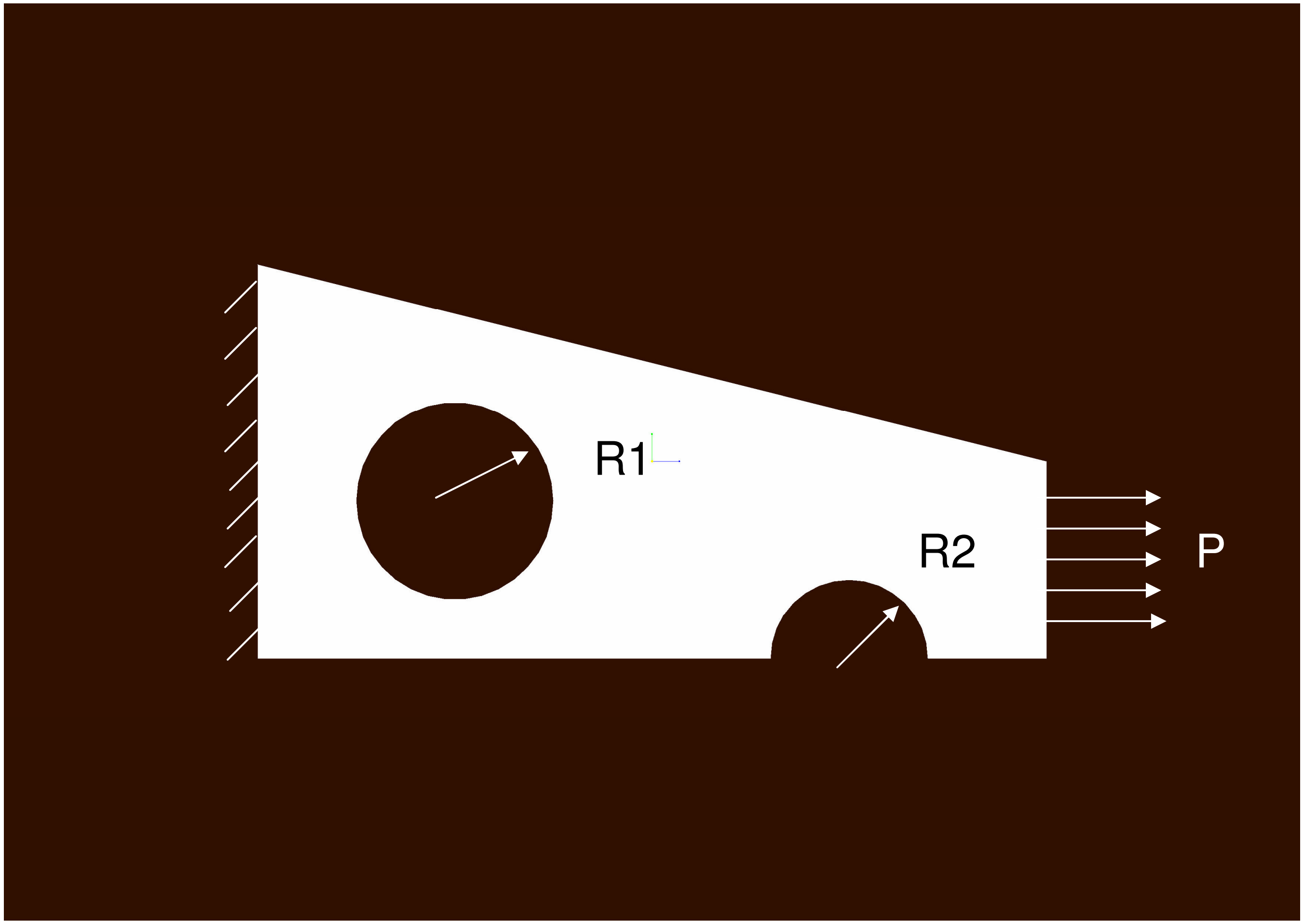
Strength > Constant 3

H/D = 1.65 -- ***Equality Constraint***

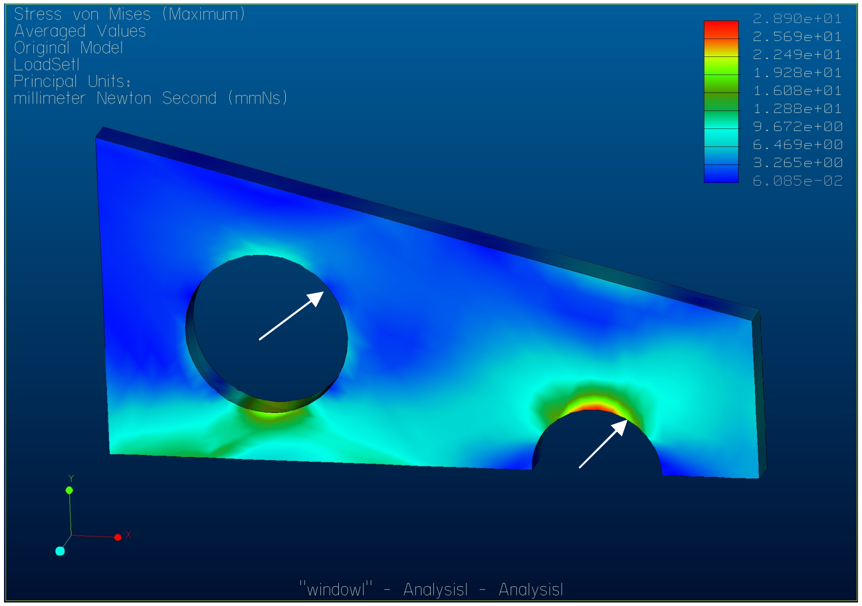
D, H, T, material, tolerances, etc. -- ***Variables***

Find: D*, H*, T*, etc.* -- ***Optimum***

Example



Design Optimization



Design Variables: R1 and R2

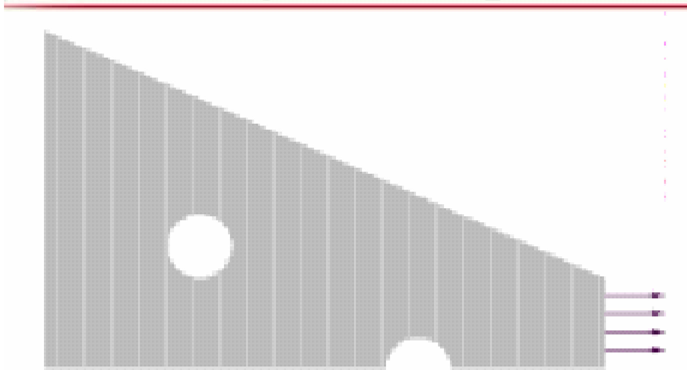
Constraints: Max stress

Min and max values for these variables

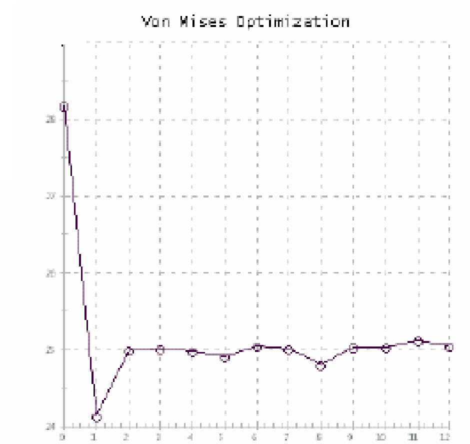
Objective Function: Reduce the total mass

Design Optimization

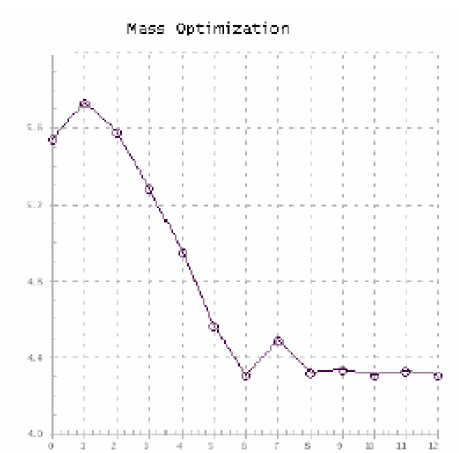
Maximum values of design variables



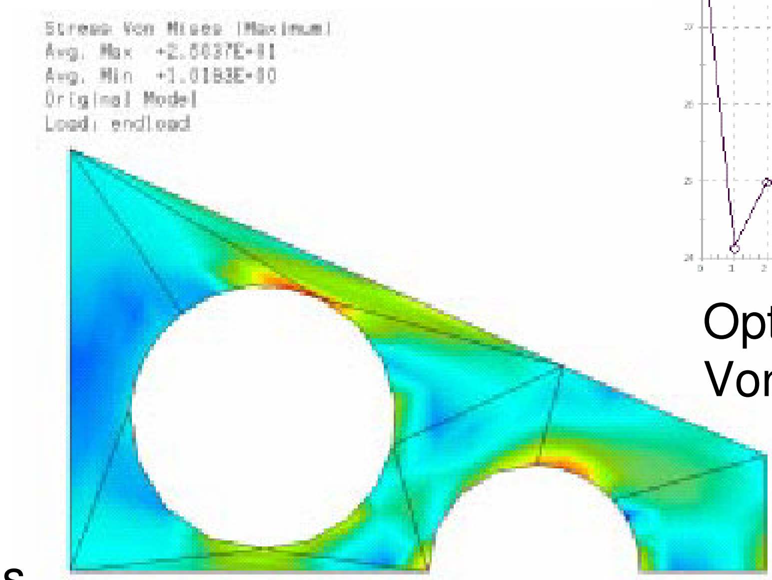
Minimum values of design variables



Optimization history: Von Mises stress



Optimization history: total mass



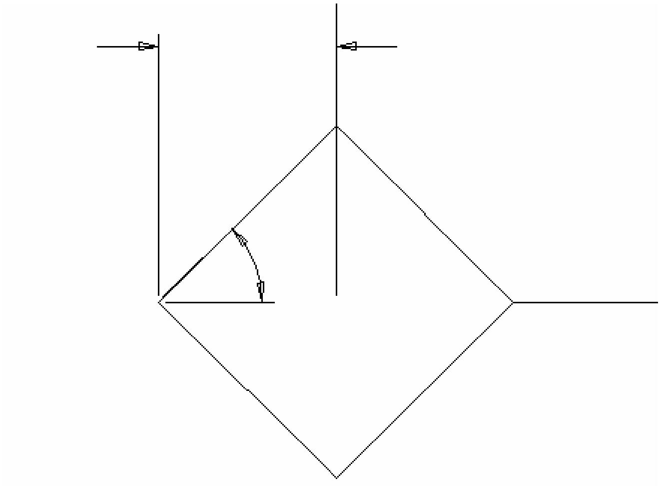
Von Mises stress distribution in optimized plate

For the lab: Design Studies

- A standard design study is the most basic and simple - Finite Element Analysis
- A sensitivity design study
  - design variables and their range
- Optimization
  - desired goal (such as minimum mass of the body)
  - geometric constraints (such as dimensions or locations of geometric entities), material constraints (such as maximum allowed stress)
  - and one or more design variables

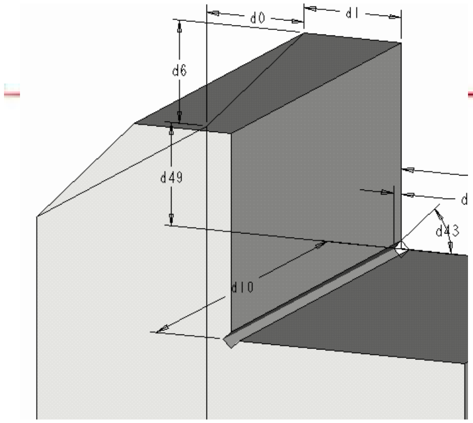
Parameter Sensitivity Study

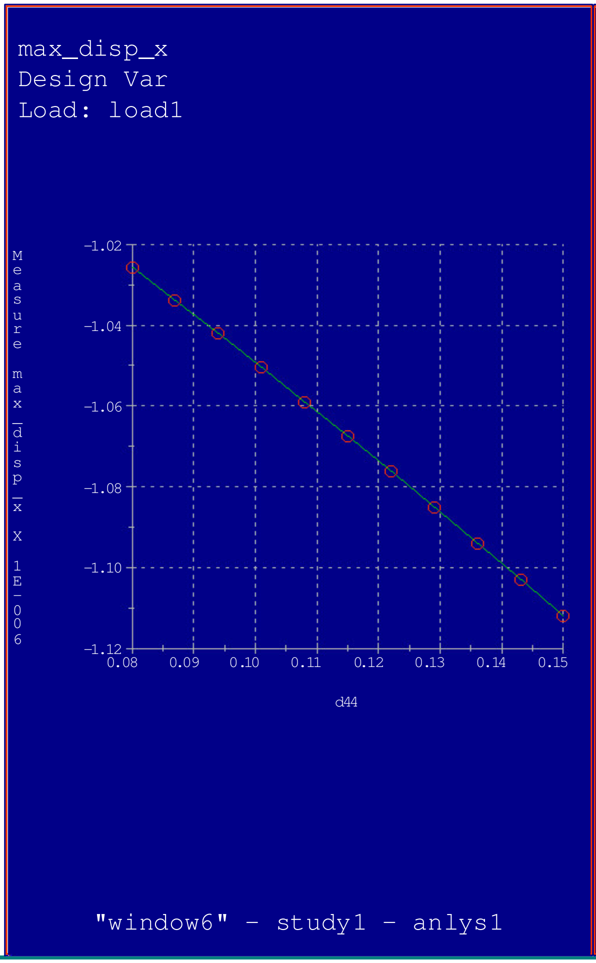
- Define a design parameter (groove size)

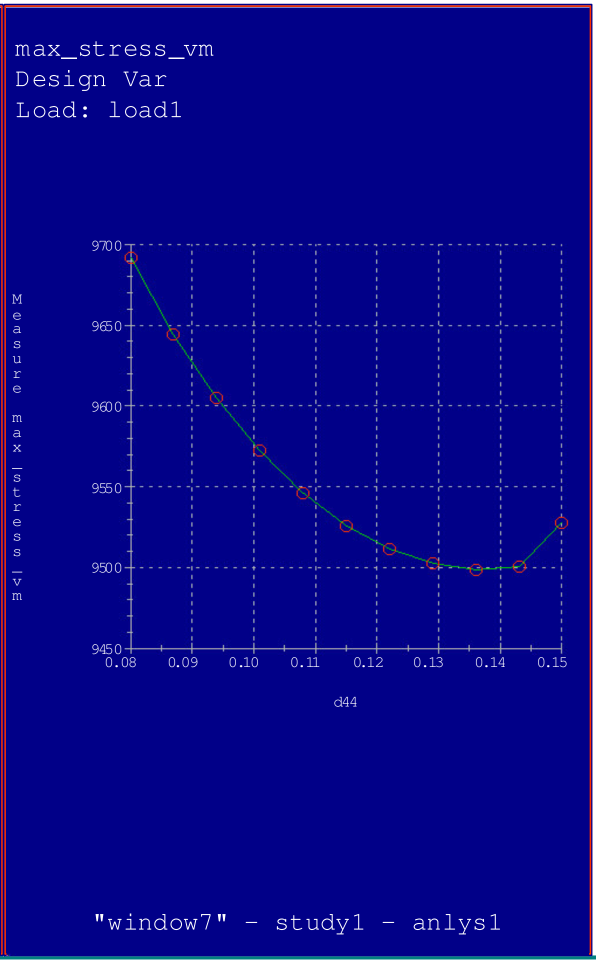


- Define a design study
- Perform the study and plot displacement and stress

Sensitivity Study



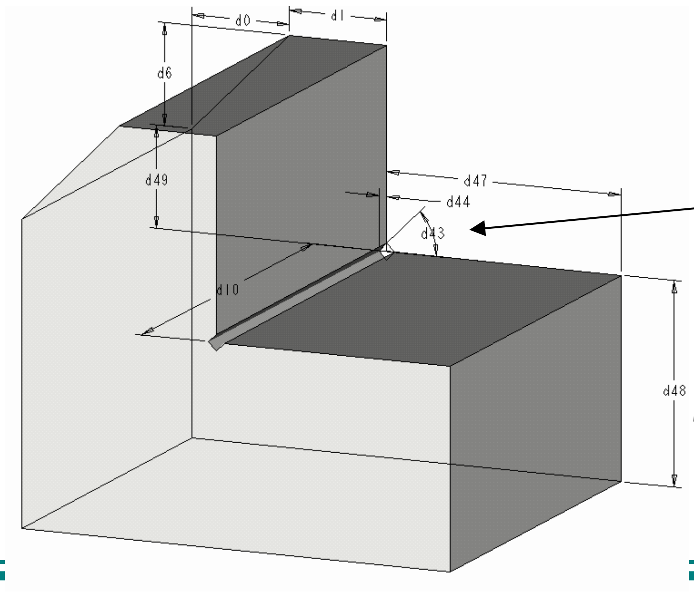




Design Optimization

- Objective: minimize the total mass
- Constraints: maximum load and deformation

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



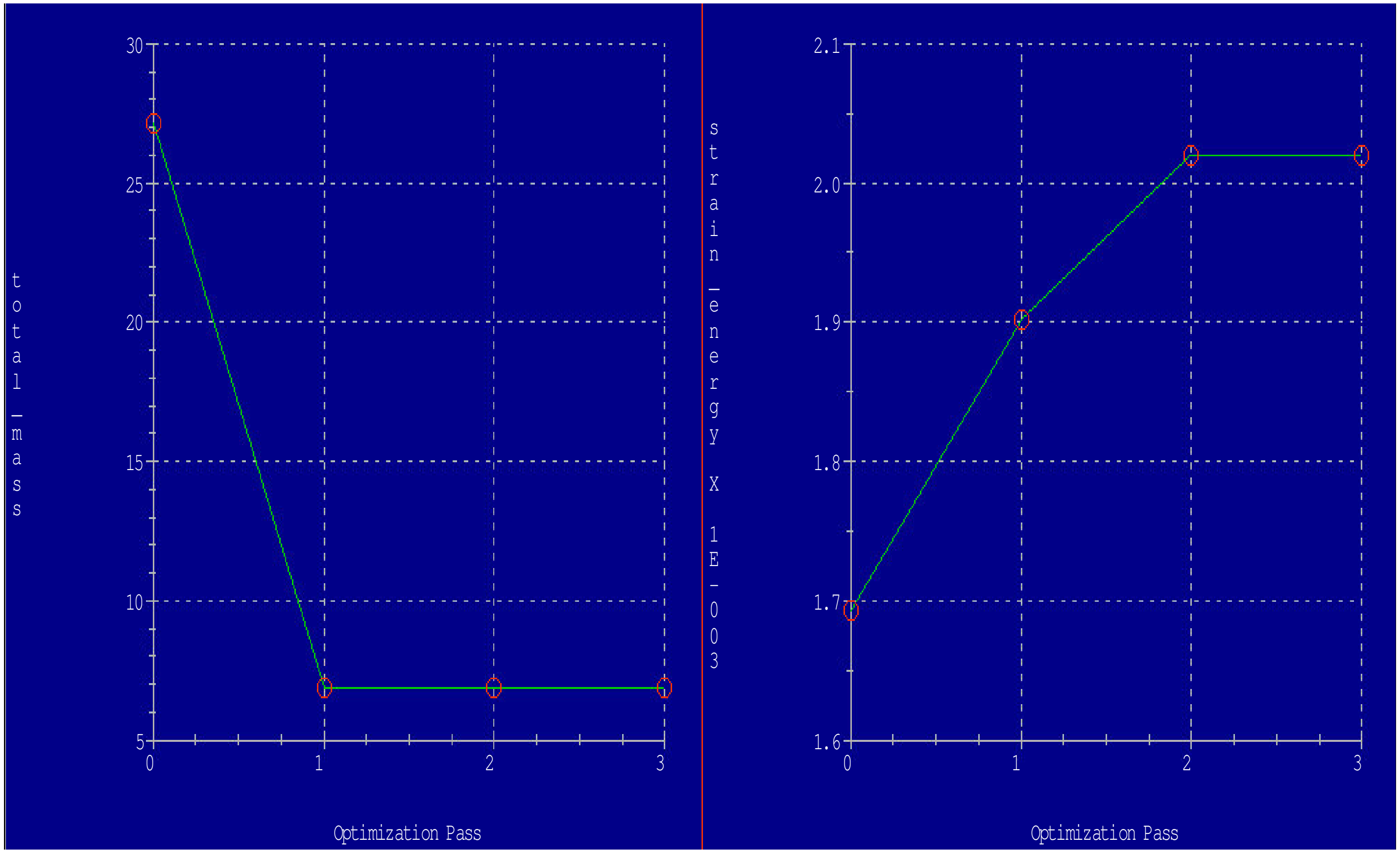
Two design variables

Design Optimization

- Objective: minimize the total mass
- Constraints: maximum Von Mises stress and deformation

2. Specify ranges of variables, objective, and constraints
3. Perform the optimization
4. Results plotting and convergence check

Optimal Design



The Total Mass and Strain Energy Convergence Plots in the Optimization

Quick Questions

- A CAD model should be simplified for FEA.
- Unimportant portion of a design can be eliminated if a FEA is carefully defined.
- A FEA model only includes information of product geometry, loads and constraints.
- Pro/Mechanica has three convergence methods, namely, quick check, single pass adaptive, and multi-pass adaptive.
- Von-mises stress is a better index than strain energy for analysis convergence check.
- Pro/Mechanica can run independently to Pro/E.
- What are the three necessary components of an optimization problem?

Summary

- General process of FEA
- Inputs to FEA (4 basic things)
- H and P-elements
- Convergence check
- Sensitivity Analysis
- Optimization