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

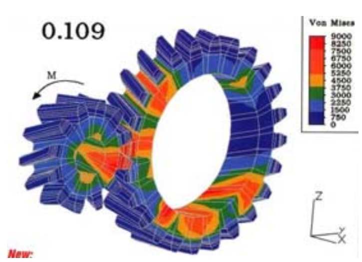
(Topics of Discussion)

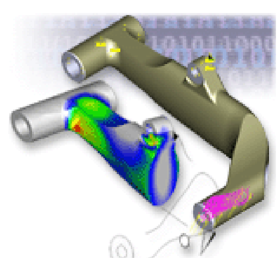
1. Use of FEA in CAD Environment, or Computer Aided Engineering – Pro/MECHANICA
2. Background of FEA Model Generation and Solution Procedure
3. Capabilities and Limitations of FEA Tools
4. The Use of CAD Model and FEA Tools for Optimizing a Design

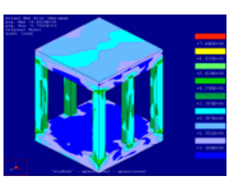
Finite Element Analysis (FEA) or Finite Element Method (FEM)

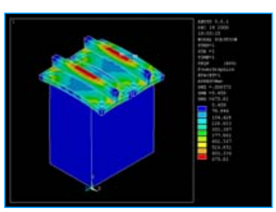
- The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics.
- Useful for problems with complicated geometries, loadings, and material properties where analytical solutions can not be obtained.

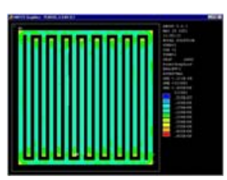
Examples of Finite Element Analysis (FEA) or Finite Element Method (FEM)











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

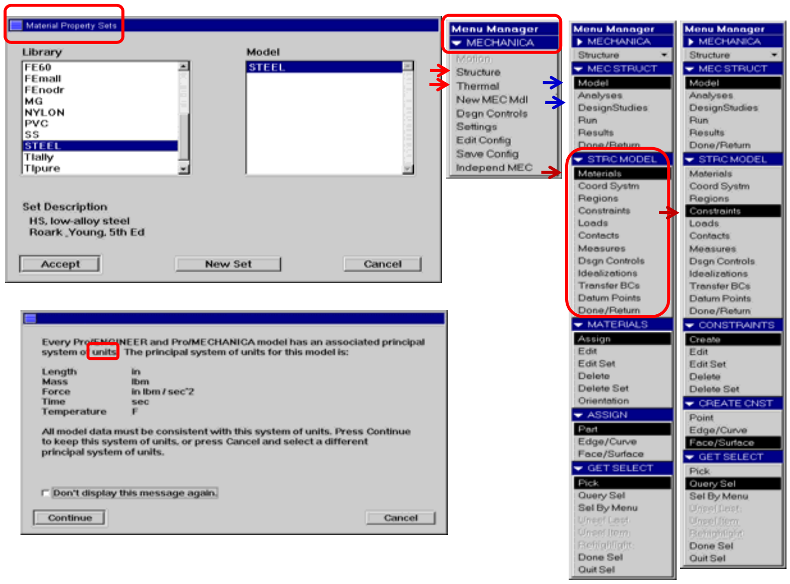


Table 1-1 Common Pro/M Mouse Functions

Function		Operation	Action
Selection (click left button)		LMB	entity or command under cursor selected
Direct View Control (drag holding middle button down)		MMB	Spin
		Shift + MMB	Pan
		Ctrl + MMB (drag vertical)	Zoom
		Ctrl + MMB (drag horizontal)	Rotate around axis perpendicular to screen
		Roll MMB scroll wheel (if available)	Zoom
Pop-up Menus (click right button)		RMB with cursor over blank graphics window	launch context-sensitive pop-up menus

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

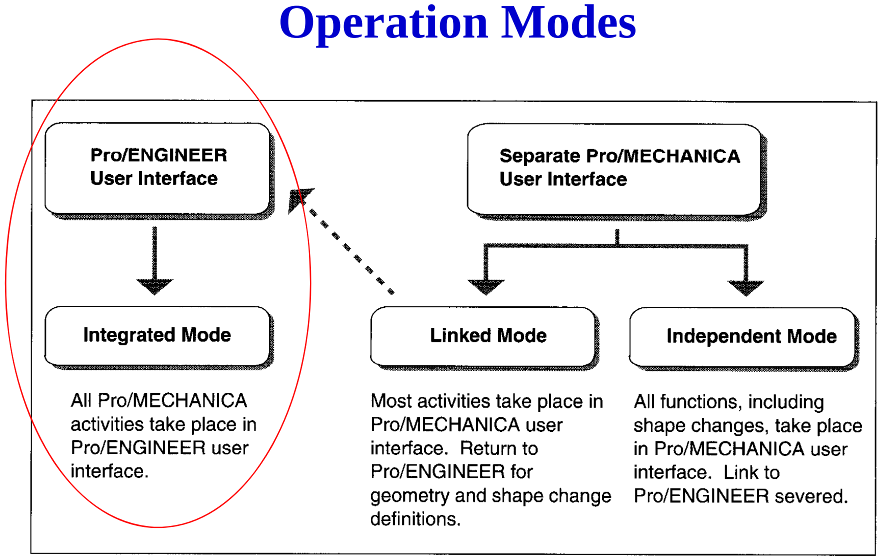
Pro/Mechanica Motion

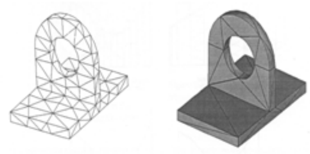
3D static, kinetic, dynamic, and inverse dynamic analyses as well as interference checking

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

Operation Modes

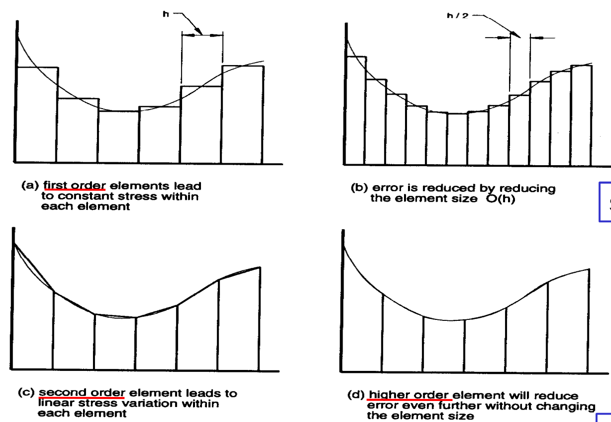




Two Approximation Methods

h element

p element



size

order

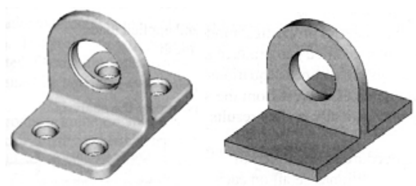
Approximation of stress function in a model

General Process

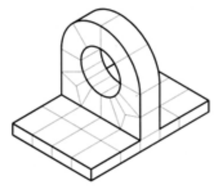
Develop the Model

- build your part
- define coordinate systems, if desired
- add materials, constraints, loads, contact regions, and measures
- create structural idealizations for your model

FEA in Pro/MECHANICA
Discretization



The "Real World" Object   The idealized physical model



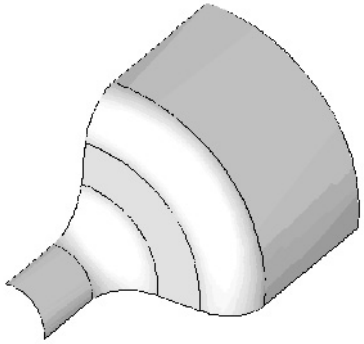
A mesh of solid brick elements

Real Word → Simplified (Idealized) Physical Model → Math. Model → Discretized (mesh) Model → Math. Model

Difference between CAD Model and FEA Model

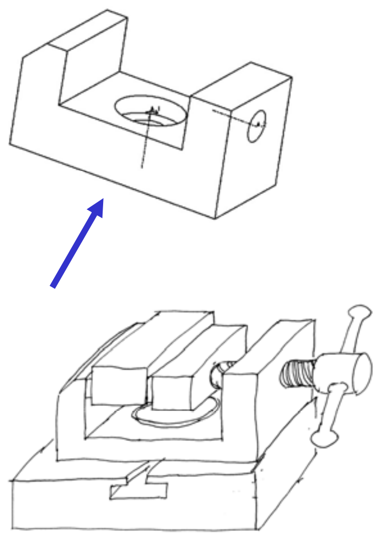
- A CAD model is to provide a detailed document for manufacturing
- A FEA model simply captures the *rough geometry* of the design and its *loading conditions.*
  - ◊ Elimination all unimportant design details that have minor effect on the results of FEA.
  - ◊ Use of part symmetry to dramatically reduce 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 – ½ and ¼ Model

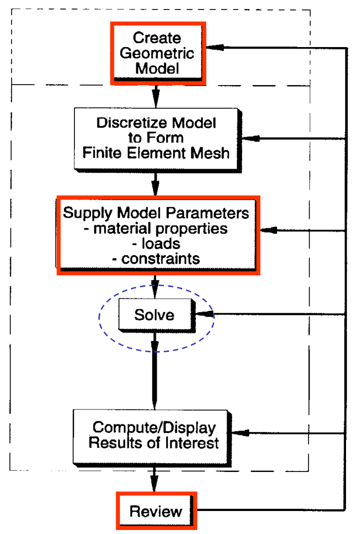


3D Shell quarter-model
of transition between cylinders

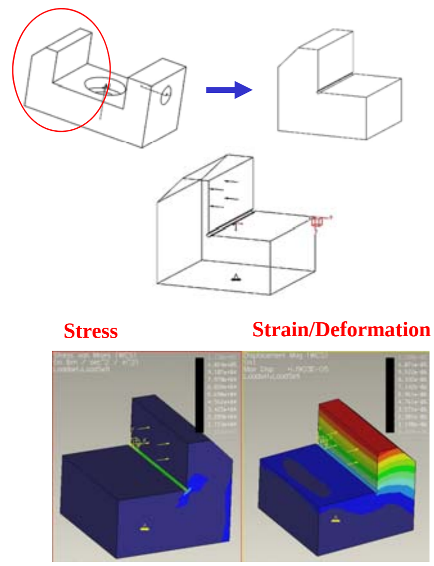
An Example



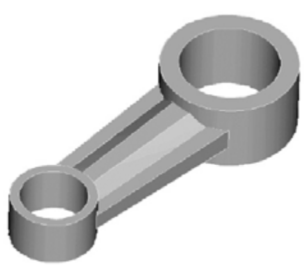
Process of Using Pro/M Structure



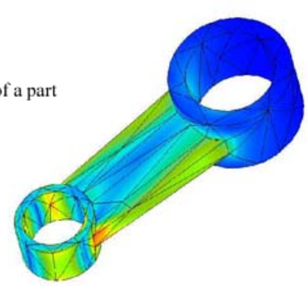
Overall Steps in FEA Solution



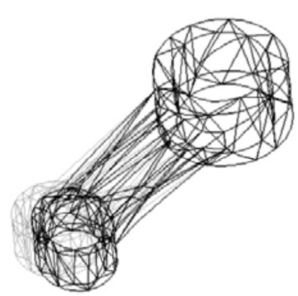
FEA Results



Solid model of a part



Von Mises stress fringe plot



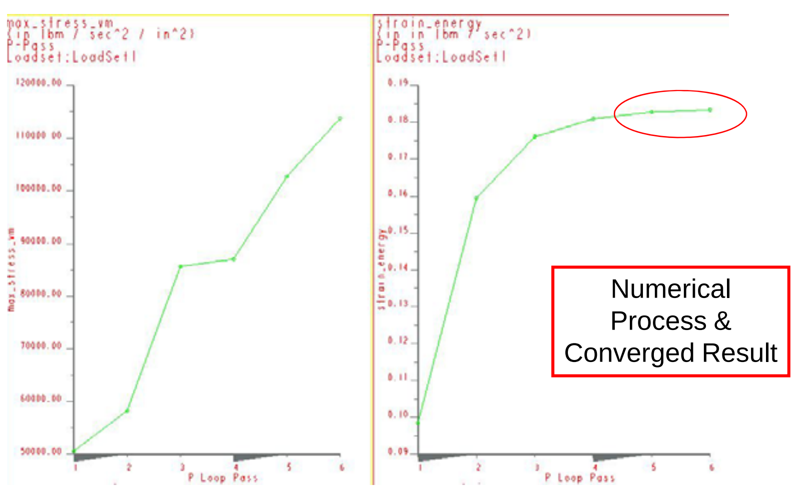
Deformation of the part

Convergence Methods

Numerical method – iterative process

- **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.

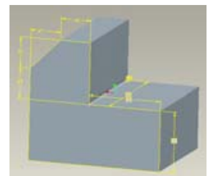
Convergence Plots for the Maximum Von Mises Stress and Strain Energy



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**

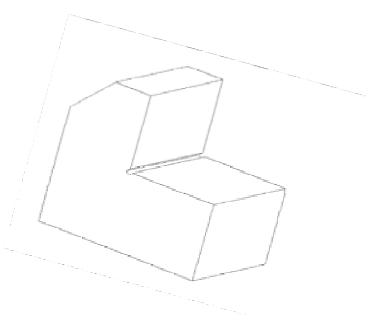


8 Pro/Mechanica – Standard Static Analysis

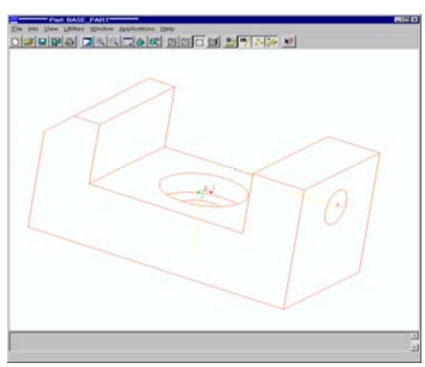
- **8.1 Objectives**
- **8.2 Procedures**

An Example

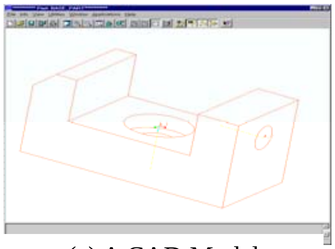
- Preparation of the Model
  - Base of a Vise



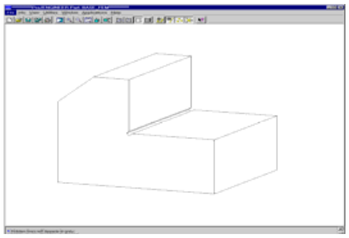
size of the groove; why?



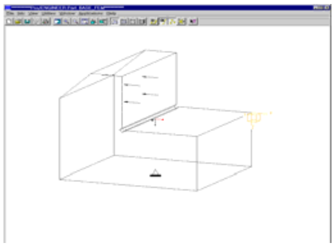
FEA Model from CAD Model



(a) A CAD Model



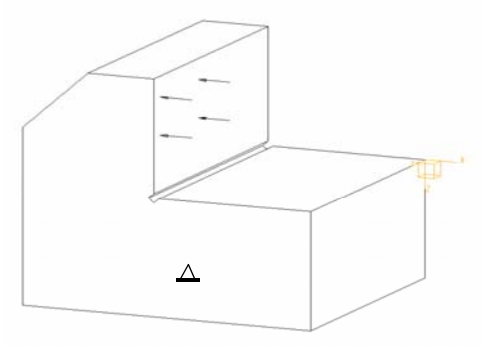
(b) A Simplified CAD Model



(C) A FEA Model

Building a FEA Model

- Coordinate System
- Material
- Loads
- Constraints



Pre-processing

- Invisible in the Integrated Mode

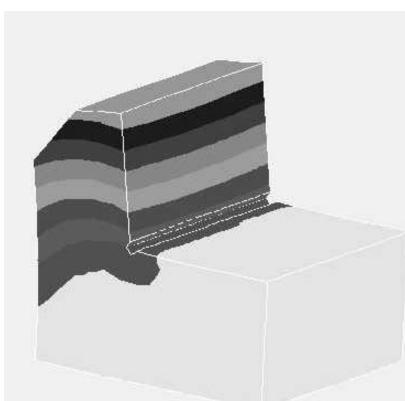
Analysis

- Quick Check
- Multi-pass Adaptive

Post-processing

- Displacement
- Von-mises Stress
- Strain Energy

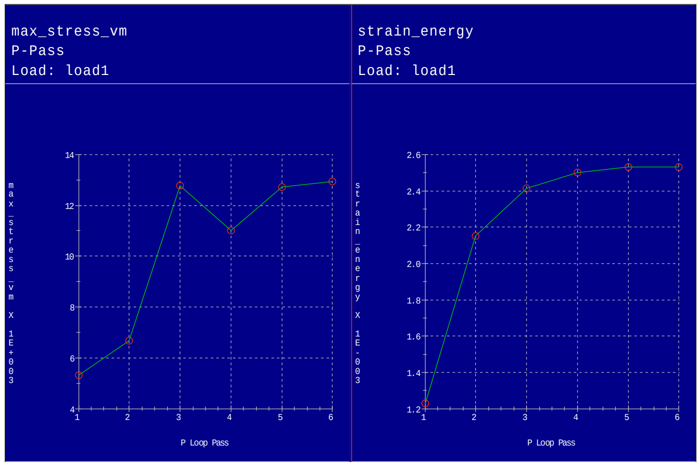
Results



(a) Deformation

(b) Von Mises Stress

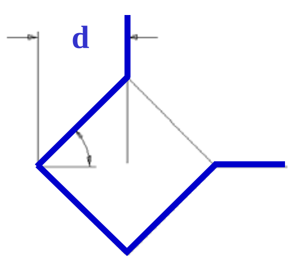
Convergence Check

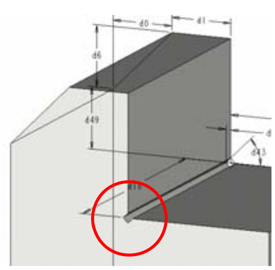


(a) Von Mises (b) Strain Energy

Parameter Sensitivity Study

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



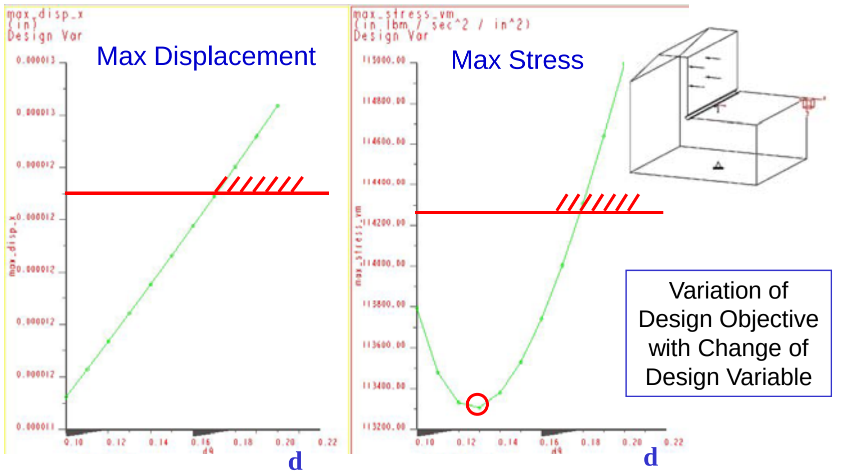


- Define a design study
- Perform the study and plot displacement and stress

Sensitivity Study

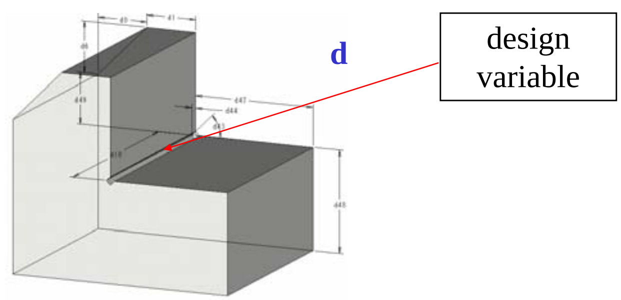
Different groove size causes different results

Every point represents one FEA run.

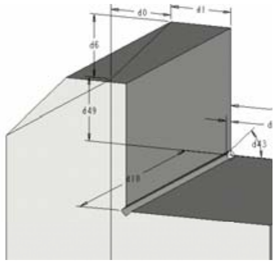


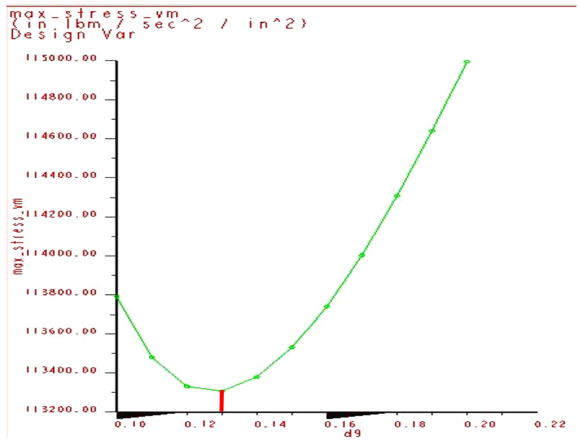
Design Optimization

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



Result of the Optimization

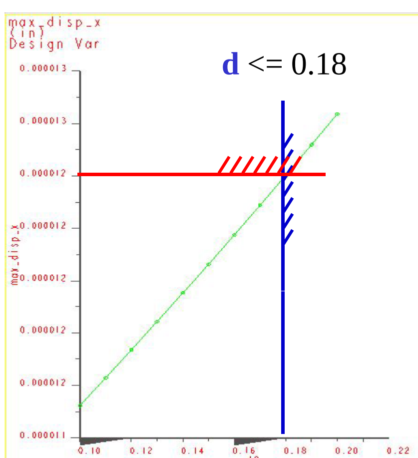




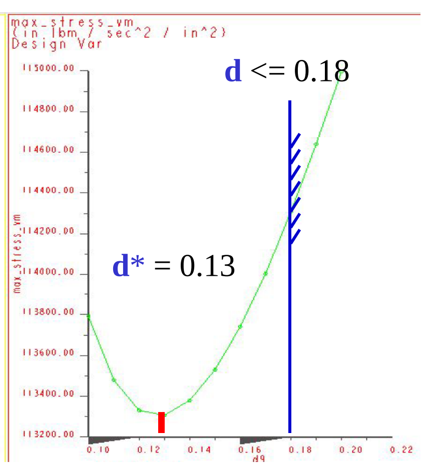
Best groove size, **d**: 0.13 (with minimum Maximum Stress)

Every point represents one FEA run.

Different Optimization Result - I (when large deformation is allowed)

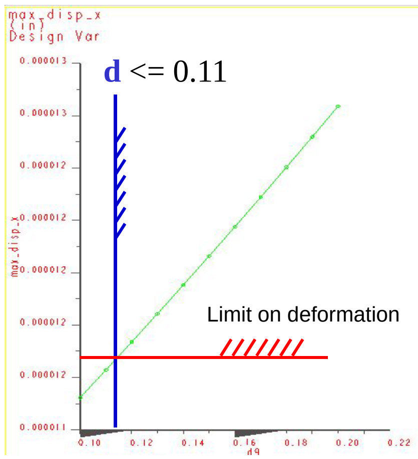


Max Displacement

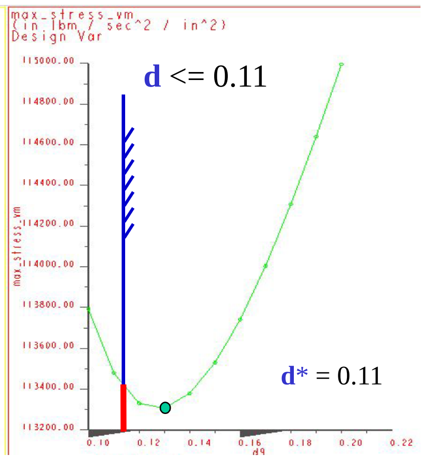


Max Stress

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



Max Displacement

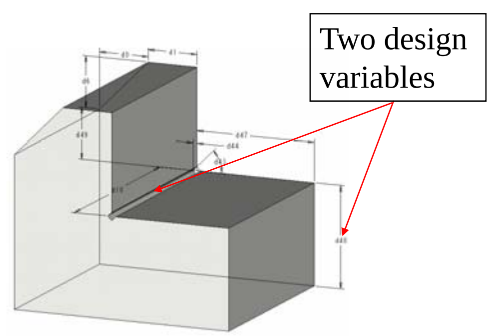


Max Stress

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?