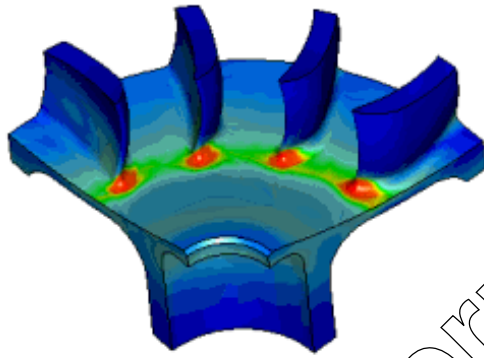
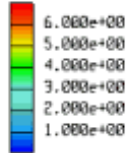
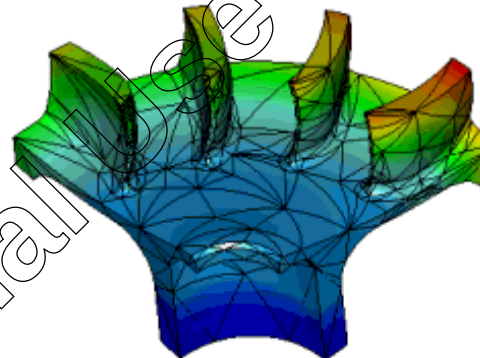
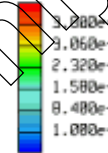


PTC Global Services

Stress von Mises (WCS)
(N / mm²)
Loadset: LoadSet1



Displacement Mag (WCS)
(mm)
Max Disp +4.1190E-03
Loadset: LoadSet1



**Structural and Thermal Simulation
with Pro/ENGINEER Wildfire**

T981-330-02

Copyright

Structural and Thermal Simulation with Pro/ENGINEER Wildfire

Copyright © 2003 Parametric Technology Corporation. All Rights Reserved.

User and training documentation from Parametric Technology Corporation (PTC) is subject to the copyright laws of the United States and other countries and is provided under a license agreement that restricts copying, disclosure, and use of such documentation. PTC hereby grants to the licensed user the right to make copies in printed form of this documentation if provided on software media, but only for internal/personal use and in accordance with the license agreement under which the applicable software is licensed. Any copy made shall include the PTC copyright notice and any other proprietary notice provided by PTC. This documentation may not be disclosed, transferred, modified, or reduced to any form, including electronic media, or transmitted or made publicly available by any means without the prior written consent of PTC and no authorization is granted to make copies for such purposes.

Information described herein is furnished for general information only, is subject to change without notice, and should not be construed as a warranty or commitment by PTC. PTC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the United States and other countries. It may not be copied or distributed in any form or medium, disclosed to third parties, or used in any manner not provided for in the software licenses agreement except with written prior approval from PTC. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

Registered Trademarks Of Parametric Technology Corporation or a Subsidiary

Advanced Surface Design, CADDs, Computervision, Computervision Services, Electronic Product Definition, EPD, EPD.Connect, Expert Machinist, Flexible Engineering, HARNESSDESIGN, Info*Engine, InPart, Optegra, Parametric Technology, Parametric Technology Corporation, PHOTORENDER, Pro/DESKTOP, Pro/E, Pro/ENGINEER, Pro/HELP, Pro/INTRALINK, Pro/MECHANICA, Pro/TOOLKIT, PTC, PT/Products, Shaping Innovation, and Windchill.

Trademarks of Parametric Technology Corporation or a Subsidiary

3DPAINT, Associative Topology Bus, Behavioral Modeling, CDRS, CounterPart, Create Collaborate Control, CV, CVact, CVaec, CVdesign, CV-DORS, CVMAC, CVNC, CVToolmaker, DataDoctor, DesignSuite, DIMENSION III, DIVISION, DVS, DVSAFEWORK, EDE, e/ENGINEER, Electrical Design Entry, EMX, eNC Explorer, Expert MoldBase, Expert Toolmaker, GRANITE, ISSM, KDIP, Knowledge Discipline in Practice, Knowledge System Driver, ModelCHECK, MoldShop, NC Builder, PartSpeak, Pro/ANIMATE, Pro/ASSEMBLY, Pro/CABLING, Pro/CASTING, Pro/CDT, Pro/CMM, Pro/COLLABORATE, Pro/COMPOSITE, Pro/CONCEPT, Pro/CONVERT, Pro/DATA for PDGS, Pro/DESIGNER, Pro/DETAIL, Pro/DIAGRAM, Pro/DIEFACE, Pro/DRAW, Pro/ECAD, Pro/ENGINE, Pro/FEATURE, Pro/FEM-POST, Pro/FICIENCY, Pro/FLY-THROUGH, Pro/HARNESS, Pro/INTERFACE, Pro/LANGUAGE, Pro/LEGACY, Pro/LIBRARYACCESS, Pro/MESH, Pro/Model.View, Pro/MOLDESIGN, Pro/NC-ADVANCED, Pro/NC-CHECK, Pro/NC-MILL, Pro/NCPOST, Pro/NC-SHEETMETAL, Pro/NC-TURN, Pro/NC-WEDM, Pro/NC-Wire EDM, Pro/NETWORK ANIMATOR, Pro/NOTEBOOK, Pro/PDM, Pro/PHOTORENDER, Pro/PIPING, Pro/PLASTIC ADVISOR, Pro/PLOT, Pro/POWER DESIGN, Pro/PROCESS, Pro/REPORT, Pro/REVIEW, Pro/SCAN-TOOLS, Pro/SHEETMETAL, Pro/SURFACE, Pro/VERIFY, Pro/Web.Link, Pro/Web.Publish, Pro/WELDING, Product Development Means Business, Product First, Products First, ProductView, PTC Precision, Shrinkwrap, The Product Development Company, The Way to Product First, Wildfire, Windchill DynamicDesignLink, Windchill PartsLink, Windchill PDMLink, Windchill ProjectLink, and Windchill SupplyLink.

Third-Party Trademarks

Adobe is a registered trademark of Adobe Systems. Advanced ClusterProven, ClusterProven, and the ClusterProven design are trademarks or registered trademarks of International Business Machines Corporation in the United States and other countries and are used under license. IBM Corporation does not warrant and is not responsible for the operation of this software product. AIX is a registered trademark of IBM Corporation. Allegro, Cadence, and Concept are registered trademarks of Cadence Design Systems, Inc. AutoCAD is a registered trademark of Autodesk, Inc. Baan is a registered trademark of Baan Company. CADAM and CATIA are registered trademarks of Dassault Systems. COACH is a trademark of CADTRAIN, Inc. DOORS is a registered trademark of Telelogic AB. FLEX/m is a registered trademark of GLOBETrotter Software, Inc. Geomagic is a registered trademark of Raindrop Geomagic, Inc. EVERS, NC, GROOVE, GROOVEFEET, GROOVE.NET, GROOVE NETWORKS, iGROOVE, PEERWARE, and the interlocking circles logo are trademarks of Groove Networks, Inc. Helix is a trademark of Microcadam, Inc. HOOPS is a trademark of Tech Soft America, Inc. HP-UX is a registered trademark and Tru64 is a trademark of the Hewlett-Packard Company. I-DEAS, Metaphase, Parasolid, SHERPA, Solid Edge, and Unigraphics are trademarks or registered trademarks of Electronic Data Systems Corporation (EDS). InstallShield is a registered trademark and service mark of InstallShield Software Corporation in the United States and/or other countries. Intel is a registered trademark of Intel Corporation. IRIX is a registered trademark of Silicon Graphics, Inc. MatrixOne is a trademark of MatrixOne, Inc. Mentor Graphics and Board Station are registered trademarks and 3D Design, AMPLE, and Design Manager are trademarks of Mentor Graphics Corporation. Netscape and the Netscape N and Ship's Wheel logos are registered trademarks of Netscape Communications Corporation in the U.S. and other countries. Oracle is a registered trademark of Oracle Corporation. OrbixWeb is a registered trademark of IONA Technologies PLC. PDGS is a registered trademark of Ford Motor Company. RAND is a trademark of RAND Worldwide. Rational Rose is a registered trademark of Rational Software Corporation. RetrievalWare is a registered trademark of Convera Corporation. RosettaNet is a trademark and Partner Interface Process and PIP are registered trademarks of "RosettaNet," a non-profit organization. SAP and R/3 are registered trademarks of SAP AG Germany.

Copyright

SolidWorks is a registered trademark of SolidWorks Corporation. All SPARC trademarks are used under license and are trademarks or registered trademarks of SPARC International, Inc. in the United States and in other countries. Products bearing SPARC trademarks are based upon an architecture developed by Sun Microsystems, Inc. STHENO is a trademark of CAD Schroer GmbH. Sun, Sun Microsystems, the Sun logo, Solaris, UltraSPARC, Java and all Java based marks, and "The Network is the Computer" are trademarks or registered trademarks of Sun Microsystems, Inc. in the United States and in other countries. VisTools is a trademark of Visual Kinematics, Inc. (VKI). VisualCafé is a trademark of WebGain, Inc. WebEx is a trademark of WebEx Communications, Inc. Microsoft, Windows, Windows NT, Visual Basic, and the Visual Basic logo are registered trademarks of Microsoft Corporation in the United States and/or other countries.

Licensed Third-Party Technology Information

Certain PTC software products contain licensed third-party technology: Rational Rose 2000E is copyrighted software of Rational Software Corporation. RetrievalWare is copyrighted software of Convera Corporation. VisualCafé is copyrighted software of WebGain, Inc. VisTools library is copyrighted software of Visual Kinematics, Inc. (VKI) containing confidential trade secret information belonging to VKI. HOOPS graphics system is a proprietary software product of, and is copyrighted by, Tech Soft America, Inc. G-POST is copyrighted software and a registered trademark of Intercim. VERICUT is copyrighted software and a registered trademark of CGTech. Pro/PLASTIC ADVISOR is powered by Moldflow technology. Moldflow is a registered trademark of Moldflow Corporation. The JPEG image output in the Pro/Web Publish module is based in part on the work of the independent JPEG Group. DFORMD.DLL is copyrighted software from Compaq Computer Corporation and may not be distributed. METIS, developed by George Karypis and Vipin Kumar at the University of Minnesota, can be researched at <http://www.cs.umn.edu/~karypis/metis>. METIS is © 1997 Regents of the University of Minnesota. LightWork Libraries are copyrighted by LightWork Design 1990-2001. Visual Basic for Applications and Internet Explorer is copyrighted software of Microsoft Corporation. Adobe Acrobat Reader is copyrighted software of Adobe Systems. Parasolid © Electronic Data Systems (EDS). Windchill Info*Engine Server contains IBM XML Parser for Java Edition and the IBM Lotus XSL Edition. Pop-up calendar components Copyright © 1998 Netscape Communications Corporation. All Rights Reserved. TECHNOMATIX is copyrighted software and contains proprietary information of Technomatix Technologies Ltd. Apache Server, Tomcat, Xalan, and Xerces are technologies developed by, and are copyrighted software of, the Apache Software Foundation (<http://www.apache.org/>) – their use is subject to the terms and limitations at: <http://www.apache.org/LICENSE.txt>. UnZip (© 1990-2001 Info-ZIP, All Rights Reserved) is provided "AS IS" and WITHOUT WARRANTY OF ANY KIND. For the complete Info-ZIP license see <ftp://ftp.info-zip.org/pub/infozip/license.html>. Gecko and Mozilla components are subject to the Mozilla Public License Version 1.1 at <http://www.mozilla.org/MPL/>. Software distributed under the MPL is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the MPL for the specific language governing rights and limitations. Technology "Powered by Groove" is provided by Groove Networks, Inc. Technology "Powered by WebEx" is provided by WebEx Communications, Inc. Acrobat Reader is Copyright © 1998 Adobe Systems Inc. Oracle 8i run-time, Copyright © 2000 Oracle Corporation. The Java™ Telnet Applet (StatusPeer.java, TelnetIO.java, TelnetWrapper.java, TimedOutException.java), Copyright © 1996, 97 Matthias L. Jügel, Marcus Meißner, is redistributed under the GNU General Public License. This license is from the original copyright holder and the Applet is provided WITHOUT WARRANTY OF ANY KIND. You may obtain a copy of the source code for the Applet at <http://www.mud.de/se/jta> (for a charge of no more than the cost of physically performing the source distribution), by sending an e-mail to leo@mud.de or marcus@mud.de— you are allowed to choose either distribution method. The source code is likewise provided under the [GNU General Public License](#). GTK+The GIMP Toolkit are licensed under the [GNU LGPL](#). You may obtain a copy of the source code at <http://www.gtk.org/>, which is likewise provided under the [GNU LGPL](#).

UNITED STATES GOVERNMENT RESTRICTED RIGHTS LEGEND

This document and the software described herein are Commercial Computer Documentation and Software, pursuant to FAR 12.212(a)(b) (OCT'95) or DFARS 227.7202-1(a) and 227.7202-3(a) (JUN'95), is provided to the US Government under a limited commercial license only. For procurements predating the above clauses, use, duplication, or disclosure by the Government is subject to the restrictions set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227-7013 (OCT'88) or Commercial Computer Software-Restricted Rights at FAR 52.227-19(c)(1)-(2) (JUN'87), as applicable. 110102

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

PRINTING HISTORY

<u>Document No.</u>	<u>Date</u>	<u>Description</u>
T981-330-01	03/07/03	First Printing of Structural and Thermal Simulation with Pro/ENGINEER Wildfire.
T981-330-02	12/31/03	Second Printing of Structural and Thermal Simulation with Pro/ENGINEER Wildfire.

Order Number DT-981-330-EN
Printed in U.S.A



PTC Telephone and Fax Numbers

Education Services Registration in North America

Tel: (888) 782-3773

Fax: (781) 370-5553

Technical Support (Monday - Friday)

Tel: (800) 477-6435 (U.S.)

(781) 370-5332 or (781) 370-5523 (outside U.S.)

Fax: (781) 370-5650

License Management

Tel: (800) 216-8945 (U.S.)

(781) 370-5559 (outside U.S.)

Fax: (781) 370-5795

Contracts

Tel: (800) 791-9966 (U.S.)

(781) 370-5700 (outside U.S.)

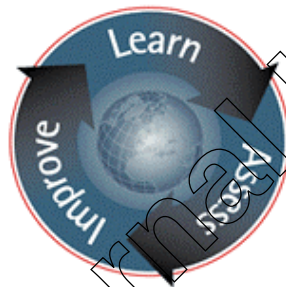
In addition, you can access the PTC Web site at **www.ptc.com**. Our Web site contains the latest training schedules, registration information, directions to training facilities, and course descriptions. You can also find general information about PTC, Pro/ENGINEER, Consulting Services, Customer Support, and Pro/PARTNERS.

Precision Learning

THE PRECISION LEARNING METHODOLOGY

PTC Global Services is dedicated to continually providing the student with an effective, comprehensive learning experience. Toward this goal, PTC developed Precision Learning, which matches the *right* training to the *right* people at the *right* time using the *right* method.

Precision Learning is based on a three-stage, **Learn—Assess—Improve** methodology.



Stage 1: LEARN

The student attends a PTC training course, including any:

- Instructor-led training course at a PTC training center.
- On-site training course.
- Customized training course.
- Web-based training (WBT) course.

Stage 2: ASSESS

The impact of a training course is assessed using Pro/FICIENCY. Pro/FICIENCY is a Web-based skills assessment and development-planning tool. It is designed to deliver information that will help improve the skills and productivity of the student.

Stage 3: IMPROVE

Pro/FICIENCY enables customers to identify areas for improvement. The training wizard will direct customers to the appropriate class based on their job responsibilities.

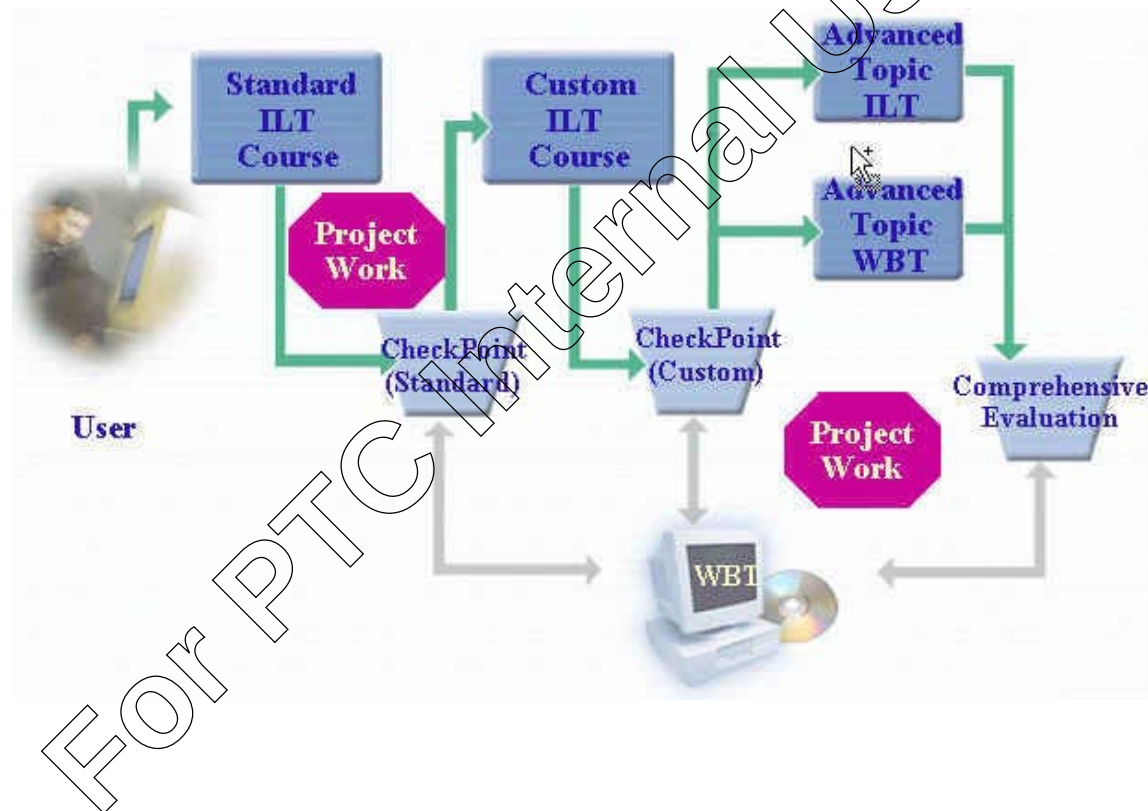
Precision Learning

Customers have access to a range of resources that include:

- Internal and external user groups.
- PTC technical support resources.
- Web-based courses and lessons.

CONTINUOUS IMPROVEMENT

The Precision Learning methodology provides a continuous cycle of knowledge expansion and improvement.



Precision Learning

PRECISION LEARNING IN THE CLASSROOM

The **Learn—Assess—Improve** Precision Learning methodology is also implemented in selected PTC instructor-led courses. Throughout the class, students will take Pro/FICIENCY assessments to evaluate their own comprehension. The group results are also used to identify areas for the instructor to review with the class as a whole. At the end of the class, each student will complete an Education Circuit form. This Education Circuit is the student's *action plan*, identifying topics for improvement, as well as the steps to take in order to enhance the skills in those areas.

The following pages provide a sample Education Circuit action plan, and a blank action plan. Instructions for using the Education Circuit action plan will be discussed in the course.

For PTC Internal Use Only

Precision Learning

EDUCATION CIRCUIT EXAMPLE

The following is an example of a student's Education Circuit at the end of the **Designing Products with Pro/ENGINEER Wildfire** training class.

Pro/FICIENCY Assessment Results

After reviewing assessment results for this course, the following lists the questions I answered incorrectly and need to research further:

Question	Improve Action
Weak and strong dimensions	Practice creating simple features with the desired dimensioning scheme. Web Lesson <i>Dimensioning Scheme</i>
Draft Features	See colleague at work for advice and product examples.
Configuration file options	Consult company user group for guidelines.

Class Evaluation Form Topics

After reviewing the questions on the class Evaluation form, the following lists the topics I need to research further:

Objective	Improve Action
Setting up the default view of a part	Practice on simple parts using different sketching planes and reference planes.
Creating sweeps	Web Lesson <i>Swept Forms</i>
Resolve Mode	Create some simple models and make them fail.
Resolve Mode	Web lesson <i>Resolve Mode</i>

Future Courses

After reviewing the Role Based Training guidelines, the following lists the courses recommended to improve my skills and enhance my job performance:

Next Courses	Next Courses
Advanced Assembly Management with Pro/ENGINEER Wildfire	
Advanced Surface Modeling with Pro/ENGINEER Wildfire	

Precision Learning

Pro/FICIENCY Assessment Results

After reviewing assessment results for this course, the following lists the questions I answered incorrectly and need to research further:

Question	Improve Action

Class Evaluation Form Topics

After reviewing the questions on the class Evaluation form, the following lists the topics I need to research further:

Objective	Improve Action

Future Courses

After reviewing the Role Based Training guidelines, the following lists the courses recommended to improve my skills and enhance my job performance:

Next Courses	Next Courses

Courseware Development Software

Structural and Thermal Simulation with Pro/ENGINEER Wildfire

The following software versions were used in developing this course:

Build Code(s)

Title/Version	Build
Pro/ENGINEER Wildfire for Windows NT 4.0 2000 and XP for Intel based systems	Production Build 2003250

Training Agenda

Structural and Thermal Simulation with Pro/ENGINEER Wildfire

Day 1

Introduction

Validating Designs Using Mechanica

Optimizing Designs Using Mechanica

Simplifying Designs Using Idealizations

Day 2

Optimizing Models for Analysis

Assigning Properties to Models

Applying Constraints

Simulating Applied Loads

Day 3

Creating and Running Structural Analyses

Comparing MPA and SPA Results

Evaluating Analysis Results

Day 4

Running Sensitivity Studies

Running Optimization Studies

Day 5

Applying Mechanica Analysis Concepts

Table of Contents

Structural and Thermal Simulation with Pro/ENGINEER Wildfire

THE PRECISION LEARNING METHODOLOGY.....	i
CONTINUOUS IMPROVEMENT	ii
PRECISION LEARNING IN THE CLASSROOM	iii
EDUCATION CIRCUIT EXAMPLE	iv
INTRODUCTION	1-1
Summary	1-3
VALIDATING DESIGNS USING MECHANICA	2-1
Module 2 Lab Exercises.....	2-3
Exercise 1: Creating the Bracket.....	2-3
Exercise 2: Adding Material Properties	2-9
Exercise 3: Defining Constraints	2-10
Exercise 4: Defining Loads	2-13
Exercise 5: Idealizing the Model	2-16
Exercise 6: Running an Analysis.....	2-19
Exercise 7: Displaying Results.....	2-21
Summary	2-25
OPTIMIZING DESIGNS USING MECHANICA	3-1
Module 3 Lab Exercises.....	3-3
Exercise 1: Creating Design Parameters	3-3
Exercise 2: Creating and Running Global Sensitivity Studies	3-8
Exercise 3: Optimizing the Bracket	3-11
Summary	3-16
SIMPLIFYING DESIGNS USING IDEALIZATIONS	4-1
Module 4 Lab Exercises.....	4-3
Exercise 1: Applying Mass, Spring, and Beam Idealizations	4-3
Exercise 2: Applying Shell Idealizations.....	4-9
Exercise 3: Applying Solid Idealizations.....	4-14
Exercise 4: Applying Rigid Connections	4-18
Exercise 5: Applying End and Perimeter Welds.....	4-20

Summary	4-26
OPTIMIZING MODELS FOR ANALYSIS	5-1
Module 5 Lab Exercises	5-3
Exercise 1: Suppressing Cosmetic Features	5-3
Exercise 2: Defining Mid-Surface Using Auto Detect	5-9
Exercise 3: Defining Mid-Surface for Shells	5-12
Exercise 4: Creating Regions	5-15
Exercise 5: Creating Volume Regions	5-20
Exercise 6: Modeling Structural Assemblies	5-22
Exercise 7: Modeling a 2-D Plane Stress Plate	5-25
Summary	5-28
ASSIGNING PROPERTIES TO MODELS	6-1
Module 6 Lab Exercises	6-3
Exercise 1: Assign Structural and Thermal Material Properties	6-3
Exercise 2: Adding New Materials to the Library	6-5
Exercise 3: Edit and Delete Materials	6-6
Summary	6-8
APPLYING CONSTRAINTS	7-1
Module 7 Lab Exercises	7-3
Exercise 1: Edge Constraint	7-3
Exercise 2: Point Constraint	7-6
Exercise 3: Surface Constraint	7-10
Exercise 4: Constraining Shell Models	7-12
Exercise 5: Using Coordinate System Constraints	7-14
Exercise 6: Using Cyclic Symmetry Constraints	7-17
Summary	7-19
SIMULATING APPLIED LOADS	8-1
Module 8 Lab Exercises	8-3
Exercise 1: Applying General Loads	8-3
Exercise 2: Applying Spatial Load Variations	8-7
Exercise 3: Varying Load Direction and Magnitude	8-11
Exercise 4: Pressure and Gravity Loads	8-13
Exercise 5: Creating Load Distributions	8-16
Summary	8-19

CREATING AND RUNNING STRUCTURAL ANALYSES	9-1
Module 9 Lab Exercises.....	9-3
Exercise 1: Setup and Run a Static Analysis	9-3
Exercise 2: Setup and Run a Thermal Analysis	9-10
Exercise 3: Calculating Thermal Stresses	9-18
Exercise 4: Combining Structural and Thermal Analyses	9-23
Summary	9-29
COMPARING MPA AND SPA RESULTS	10-1
Module 10 Lab Exercises.....	10-4
Exercise 1: Comparing MPA vs. SPA	10-4
Summary	10-12
EVALUATING ANALYSIS RESULTS	11-1
Module 11 Lab Exercises.....	11-4
Exercise 1: Roll Mill Bearing Stress Analysis.....	11-4
Exercise 2: Frying Pan Thermal and Structural Analysis.....	11-12
Exercise 3: Analyzing the Turning Fork	11-22
Summary	11-29
RUNNING SENSITIVITY STUDIES	12-1
Module 12 Lab Exercises.....	12-4
Exercise 1: Belt Clip Stress Analysis	12-4
Exercise 2: Belt Clip Sensitivity Studies.....	12-15
Summary	12-22
RUNNING OPTIMIZATION STUDIES	13-1
Module 13 Lab Exercises.....	13-3
Exercise 1: Optimizing the Belt Clip	13-3
Summary	13-8
APPLYING MECHANICA ANALYSIS CONCEPTS	14-1
Module 14 Lab Exercises.....	14-9
Exercise 1: Spot Weld Analysis	14-9
Exercise 2: Contact Analysis.....	14-15
Exercise 3: Transient Thermal Analysis.....	14-21
Exercise 4: Large Deformation Analysis	14-26
Exercise 5: Dynamic Time Analysis.....	14-32
Exercise 6: Dynamic Frequency Analysis	14-42

For PTC Internal Use Only

For PTC Internal Use Only

Introduction

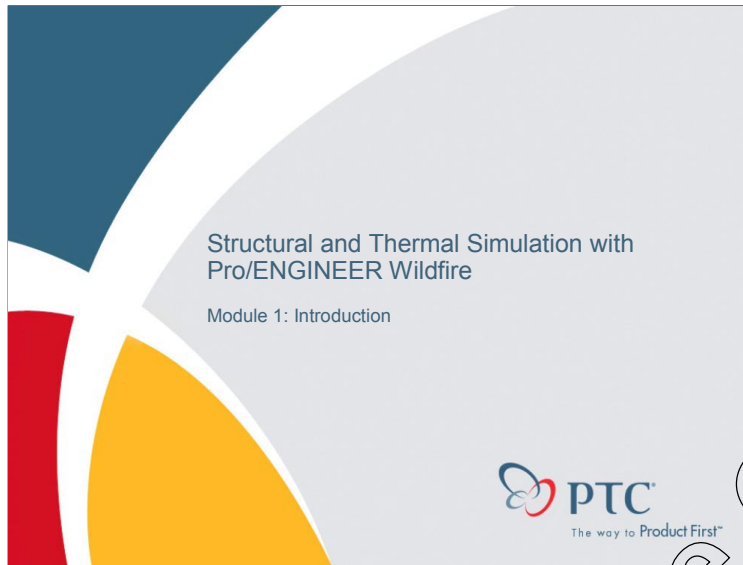
Introduction

Mechanica allows engineers to test and optimize designs for structural, dynamic, thermal and durability performance before physical prototyping. Mechanica Structure allows design engineers to evaluate and optimize the static and dynamic structural performance of their designs in a real-world environment. Precise representations of CAD geometry and unique adaptive solution technology provide fast, accurate solutions automatically. This results in improved product quality, while decreasing costs associated with extended development times and ineffectual prototypes.

Objectives

After completing this module, you will be able to:

- Describe Mechanica Structure.
- Describe Mechanica technology.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

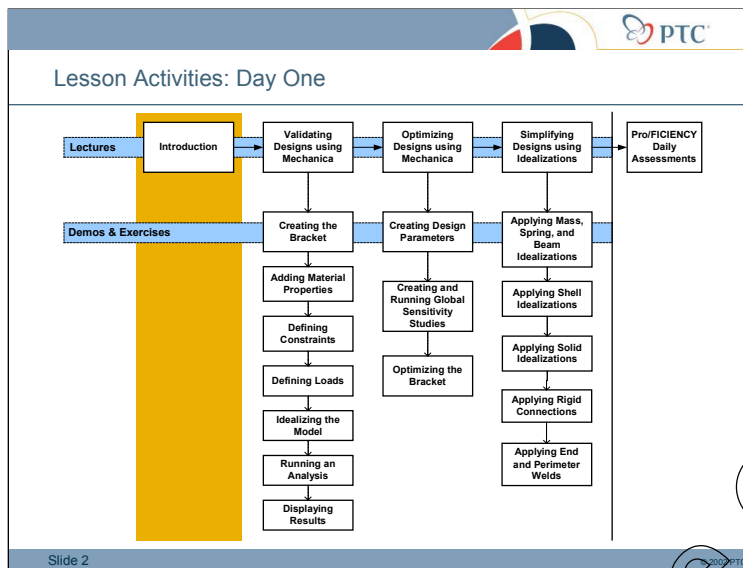
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos (0)
- Labs (0)
- Total: 20 min



Objectives

After completing this module, you should be able to:

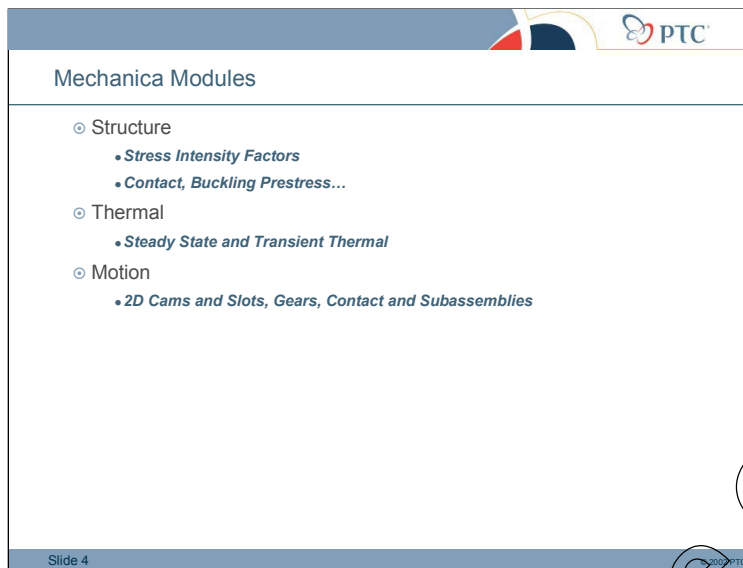
- Describe Mechanica Structure.
- Describe Mechanica technology.

Slide 3

Objectives

Mechanica allows engineers to test and optimize designs for structural, dynamic, thermal and durability performance before physical prototyping. Mechanica Structure allows design engineers to evaluate and optimize the static and dynamic structural performance of their designs in a real-world environment. Precise representations of CAD geometry and unique adaptive solution technology provide fast, accurate solutions automatically. This results in improved product quality, while decreasing costs associated with extended development times and ineffectual prototypes.

In this module, you will learn the basic process of using Mechanica.



Mechanica Products

These modules “talk” to each other and multi-disciplinary analyses can be performed. For example:

- Reaction forces and inertial loads can be transferred from motion to structure.
- Temperature distributions can be transferred from Thermal to Structure for thermal-stress analysis.
- Optimization studies can be done on structural as well as thermal criteria.

There are three main Mechanica modules: Structure, Thermal, and Motion.

- *Mechanica Structure* – allows the user to perform **Static, Modal, Buckling, Contact, Prestress Static, Prestress Modal, and Vibration** analyses. (Additionally, it includes Dynamic Time Response, Dynamic Frequency Response, Dynamic Random Response, and Dynamic Shock Response.)
- *Mechanica Thermal* – allows the user to perform **Steady-State Thermal and Transient Thermal** analyses. (Note: Conduction and Convection can be modeled but not Radiation.)
- *Mechanica Motion* – is a rigid body dynamics (Dynamics of Mechanisms) package which allows mechanism modeling, including 2D cams and slots, gears (simplified geometry is used), contact (collisions and impact between parts can be modeled in detail; certain restrictions apply in terms of the type of regions that can come into contact), and subassemblies.

Note: Only Structure and Thermal products will be used in this course.

PTC

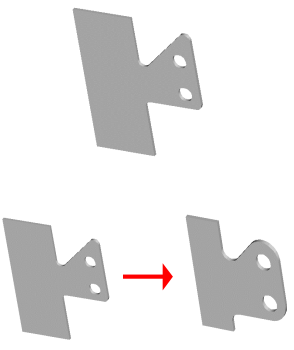
Mechanica: Structure

Analyze parts and assemblies to find:

- Deformed shapes.
- Maximum stresses.
- Natural frequencies.
- Temperatures.
- Buckling loads.
- Responses to forcing functions.

Design Optimization Tool

- Part shape and properties can be parameterized (Design Parameters).
- Vary the model parameters to find the best design.

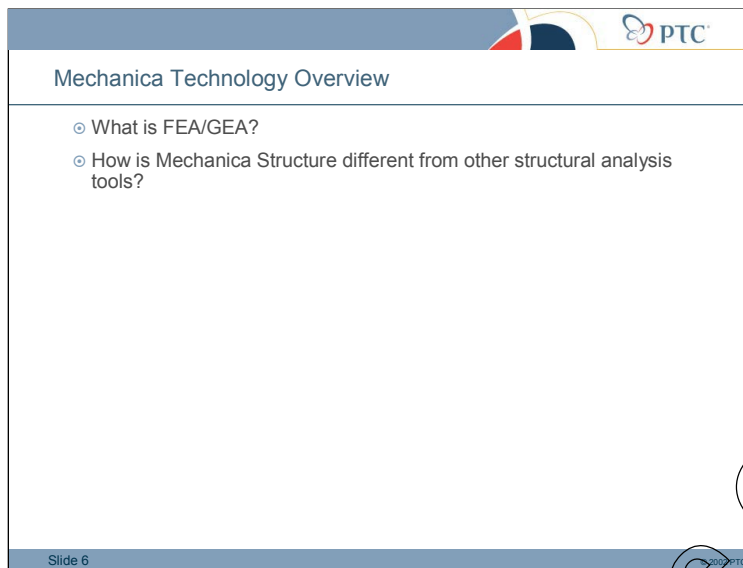


Slide 5

Mechanica: Structure

Mechanica Structure can be used to analyze parts and assemblies to determine deformed shapes, maximum stresses, natural frequencies, buckling loads, and responses to forcing functions.

- These are the “How Much” questions. Most analysis software will stop here (design validation).
- Being able to answer the “What if ” questions early and often is where Mechanica brings added value.
- Part shapes and properties can be parameterized to quickly generate and evaluate many software prototypes of your design and determine which is the best. This is illustrated here on a belt clip for an electronic pager.



What is Finite Element Analysis?

Finite Element Analysis (FEA) is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can be used to analyze either small or large-scale deflection under loading or applied displacement. It can analyze elastic deformation, or "permanently bent out of shape" plastic deformation. The computer is required because of the astronomical number of calculations needed to analyze a large structure. The power and low cost of modern computers has made Finite Element Analysis available to many disciplines and companies.

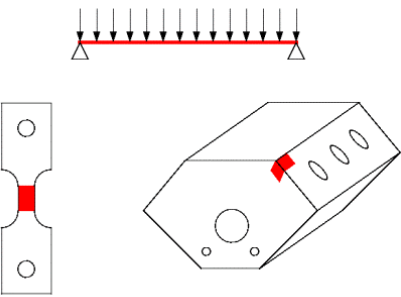
In the finite element method, a structure is broken down into many small simple blocks or elements. The behavior of an individual element can be described with a relatively simple set of equations. Just as the set of elements would be joined together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the behavior of the whole structure. The computer can solve this large set of simultaneous equations. From the solution, the computer extracts the behavior of the individual elements. From this, it can get the stress and deflection of all the parts of the structure. The stresses will be compared to allowed values of stress for the materials to be used, to see if the structure is strong enough.

Mechanica minimizes physical prototyping, increases creativity and helps deliver better products in less time. With Mechanica design engineers can evaluate and optimize static and dynamic structural performance of their designs as well as use expert tools to simulate the behavior of parts and assemblies subject to thermal loading. Unique adaptive technology provides fast, accurate solutions automatically. Of course, as an integral part of Pro/ENGINEER, Advanced Structural & Thermal Simulation simplifies the job since there's no awkward data transfer issues common with other systems.

Mechanics Technology Overview (cont.)

Closed Form Solution

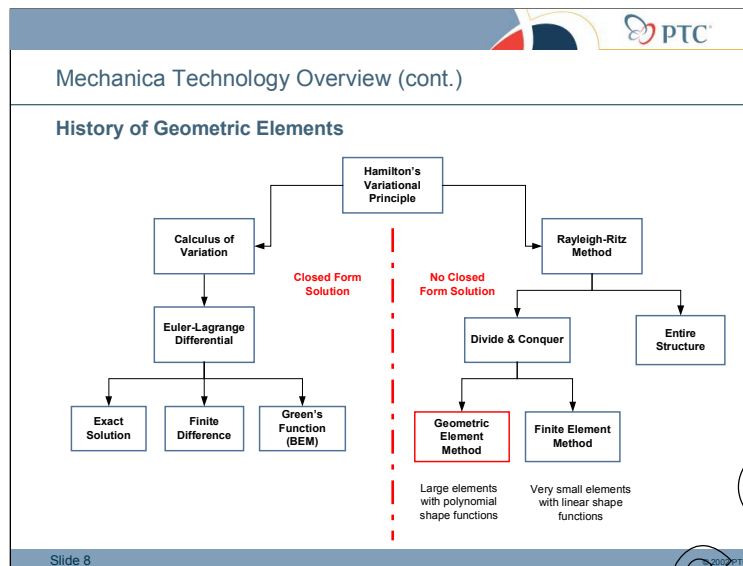
- ⊙ $\delta_{\max} = ?$
- ⊙ $\sigma_{vm, \max} = ?$
- ⊙ $F_{mode, 1} = ?$
- ⊙ $f(\delta, \sigma) = ?$



Slide 7

Closed Form Solution

- Exact solutions to differential equations are only known for very basic geometries and boundary conditions (for example: tensile test specimen, simply supported beam). To extract useful information from these solutions, we have to make some simplifying assumptions and then, using an engineering handbook, find the exact solution which best fits the application.
- For complex geometries, the above approach can only yield “ball park” figures. For example, no closed form solution exists for this engine block. If we want accurate answers, we have to resort to mathematical approximations and numerical methods.
- These methods proceed by using a “divide-and-conquer” approach: Break up the engine block into small, manageable elements (for example: elements which deform in known ways). Compute quantities of interest for each element and then put the solution together by gathering results from all elements.



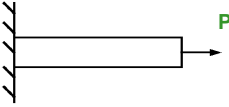
History of Geometric Elements

We therefore have two classes of problems: Those for which we have closed form solutions and those for which we don't. This chart is a brief history of mathematical formulations for engineering problems. Starting from Hamilton's Variational Principle:

- Calculus of variation leads to the formulation of Euler-Lagrange Differential Equations. Such equations can be formulated for a large class of problems; finding a solution to these differential equations, however, has proven to be extremely difficult for most situations except textbook problems. Other closed form solutions can be derived using Finite Difference Methods and Green's Functions (boundary element method), but again only for a limited class of problems.
- Faced with the difficulties of solving complex differential equations, engineers have developed alternative methods. One such alternative approach is the Rayleigh-Ritz method which assumes a deformed shape for the loaded body (shape-function) and works backward towards the solution. The success of this method depends largely on how well we have selected the shape function. For complex structures adequate shape-function cannot be found easily.
- This drawback is eliminated by using a "divide and conquer" approach: break a complex structure or body into small, manageable elements (for example: elements for which shape functions are well known), solve equations for each element and then assemble a solution for the complete structure.
- Two methods have been developed. The Finite Element Method (which uses many small elements with linear shape functions) and the Geometric Element Method (which uses larger elements with polynomial shape functions. Mechanics uses the GEM).

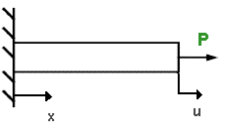
Mechanica Technology Overview (cont.)

Derivation



Derivation: Basic Assumptions

- One Degree of Freedom (DOF) in x-direction.
- Cross sectional area of bar is constant and bar is in static equilibrium.



Slide 9

Derivation

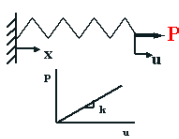
The material in this section is meant to be presented as a derivation. Figures provide a guideline for what should be written on the board.

- The following discussion is meant as an introductory example to demonstrate how Geometric Element Analysis (GEA) works. We will consider a basic engineering problem, a bar loaded in tension, and derive a solution using the FEA formulation
- FEA is a numerical procedure for analyzing systems for which no closed form solution is available. The basic idea is to subdivide the system under consideration into smaller pieces. Each piece is called an element.
- Different types of elements are used to mathematically represent the geometry and the elasticity of systems. The bar (or truss) element is the simplest type and will be used in our discussion.

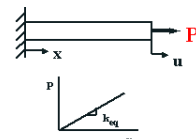
Mechanics Technology Overview (cont.)

Derivation: Linear Relationship Assumptions

- Deformations are elastic.
 - $\sigma = E \epsilon$
- Governing equation is linear.
 - $P = k u$



Spring Element



Bar Element

Linear Relationship

Slide 10

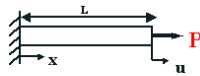
Derivation: Basic Assumptions (cont.)

- Deformations are elastic. If the load is removed, the bar comes back to its initial length.
- The cross-sectional area is constant throughout the bar element.
- Consider the bar element and a linear spring. Linear springs exhibit a linear relationship between force and displacement. This same relationship exists between force, stress, and displacement in a linear isotropic bar.

PTC

Mechanica Technology Overview (cont.)

Derivation: Equivalent Stiffness of a Bar



$$P = k_{eq} u$$

$$\sigma = E \epsilon$$

$$\epsilon = \frac{\partial u_i}{\partial i} = \frac{u}{L}$$

$$\sigma = \frac{P}{A}$$

$$\frac{P}{A} = E \frac{u}{L}$$

$$P = \frac{AE}{L} u$$

$$k_{eq} = \frac{AE}{L}$$

Slide 11

Derivations: Stiffness of Bar

- Mechanics is called a displacement-based code because it solves for displacements first and then derives all other results from there: Solving $\{u\} = [K]^{-1} \{F\}$, where:
 - $[K]$ is called the stiffness matrix, by analogy with the spring (this stiffness matrix, as we will see is derived from material as well as geometric properties).
 - $\{u\}$ is the displacement vector.
 - $\{F\}$ is the applied load.
- From displacements, Mechanics calculates the strain defined as: $\epsilon_i = du_i/di$ where “ u_i ” is the displacement in a particular direction “ i ”. For the situation at hand, and assuming a linear variation of displacement along the length of the element, strain reduces to the ratio of displacement “ u ” over initial length “ l ”. $\epsilon = u/l$.
- Assuming linear material behavior, Hooke’s law is used to calculate stress:

$$\sigma = E * \epsilon$$
 where σ is stress, E is Young’s Modulus and ϵ is strain.
- Now, coming back to the spring analogy, we can derive the equivalent spring constant of the bar in terms of stress and strain.

PTC

Mechanics Technology Overview (cont.)

Derivations: Computing Global Stiffness Matrix

$\rightarrow$ Force generated due to the unit displacement
 $\leftarrow$ Force generated due to the reaction at the nodes

$P = k u$ $\Sigma F_x = 0$

$$\left. \begin{aligned} F_{00} - F_{01} &= 0 \\ u_0 k_1 - u_1 k_1 &= 0 \\ -F_{10} + F_{11} &= 0 \\ -u_0 k_1 + u_1(k_1 + k_2) - u_2 k_2 &= 0 \\ -F_{21} + F_{22} &= 0 \\ -u_1 k_2 + u_2 k_2 &= 0 \end{aligned} \right\} \begin{bmatrix} P_0 \\ P_1 \\ P_2 \end{bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{bmatrix} u_0 \\ u_1 \\ u_2 \end{bmatrix}$$

Slide 12

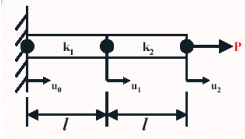
Derivations: Computing Global Stiffness Matrix

- As mentioned before, the main idea behind FEA/GEA is to subdivide the system in smaller manageable elements and then solve for displacements and stresses.
- An important step in this process is to assemble the global stiffness matrix of the system, which includes the stiffness contribution of each element.
- Rewriting the linear equation in matrix form, we obtain the global stiffness of the system.
- For the case of a system with bar elements the same procedure can be used. In the case of the bar, "k" is defined as EA/l (where "l" is the length of one element).

PTC

Mechanics Technology Overview (cont.)

Derivations: Computing Displacements



$$\{P\} = [K]\{u\} \rightarrow \begin{Bmatrix} 0 \\ P \end{Bmatrix} = \frac{EA}{l} \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

$$\{u\} = [K]^{-1}\{P\} \rightarrow \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \frac{l}{EA} \begin{bmatrix} 1 & 1 \\ 1 & 2 \end{bmatrix} \begin{Bmatrix} 0 \\ P \end{Bmatrix}$$

$u_1 = \frac{Pl}{EA}$

$u_2 = \frac{2Pl}{EA} = \frac{PL}{EA}$

PTC

Slide 13

Derivations: Computing Displacements

- In this case, node 0 has a fixed boundary condition and a load is applied at node 2.
- We will be solving the equation $Ku=P$, where K is the stiffness matrix, u is the unknown nodal displacements and P is nodal load vector.
- Because of the fixed boundary condition at node 0, the first row and column of the stiffness matrix are deleted. This leave us with 2x2-stiffness matrix that we need to invert and multiply by the load vector in order to solve for the nodal displacements.
- The final solution yields $u_1=Pl/EA$ and $u_2=2Pl/EA$. Now, considering that “ l ” is the length of the element and L the total length of the beam, we have $L=2l$, and therefore $u_2=PL/AE$, which is the closed form solution for uni-axial strain in a constant cross-section rod under tension.

What 's a shape function?

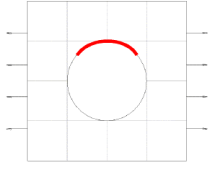
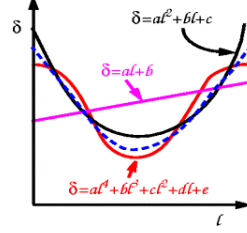
A shape function is a mathematical function used to describe the variation of displacement along the edge of an element.

PTC

Mechanica Technology Overview (cont.)

Geometric Element Analysis (GEA)

- Solution Quality
 - Built-in Convergence.
 - Comparison of solution from one polynomial level to next.
 - Quality of your solution is known.
 - Adaptive - adds computer resources only where needed.

Slide 14

Mechanica Technology Overview

Note: For the more experienced analyst, some of this discussion will appear very basic.

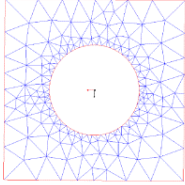
- GEA uses polynomial-order shape functions to calculate displacements. Each edge in a geometric element model is associated with a polynomial order.
- Convergence is defined as the situation where the answer quits changing as a function of mesh characteristics (be it polynomial order or number of elements)
- Illustrated here is a plate with a hole loaded in tension. We will consider the deformation of the highlighted edge as the GEA algorithm proceeds through the analysis.
- On this graph displacement (δ) is plotted as a function of edge length (l). The dashed line represents the exact solution (which, in practice, is of course unknown). The algorithm is said to have “built-in convergence” because it proceeds as follows:
 1. Perform the analysis with a linear shape function ($\delta = al + b$) and record results.
 2. Perform analysis with a quadratic shape function ($\delta = al^2 + bl + c$) and compare answer with results obtained in step (1). If the answer has changed by more than a given percentage, increase the p-level of the edge to the next order.
 3. Iterate until the previous and the current answers are close enough to be considered identical (within a user-defined tolerance). This is what we call a converged solution.
- As a result of the above process:
 1. The quality of your solution is known. Mechanica will allow you to verify how well or how poorly your solution has converged. (*How is convergence obtained with FEA? By performing mesh refinement studies which are extremely time consuming. As a result, very few analysts actually perform these studies and design decisions are made on un-converged FEA results.*)
 2. The solution is adaptive: higher order polynomial are only assigned to edges experiencing large deformations. Changing the nature of the problem (by displacing the load for instance) will affect polynomial orders but not necessarily the mesh.

PTC

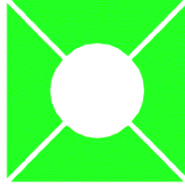
Mechanica Technology Overview (cont.)

Finite Element Analysis (FEA) and Geometric Element Analysis (GEA)

- Independent of load cases.
 - No previous knowledge of the answer is required.
 - Larger, more "flexible" elements vs. smaller, more "rigid" ones.



Traditional FEA Shell Mesh



Mechanica Structure GEA Shell Mesh

Slide 15

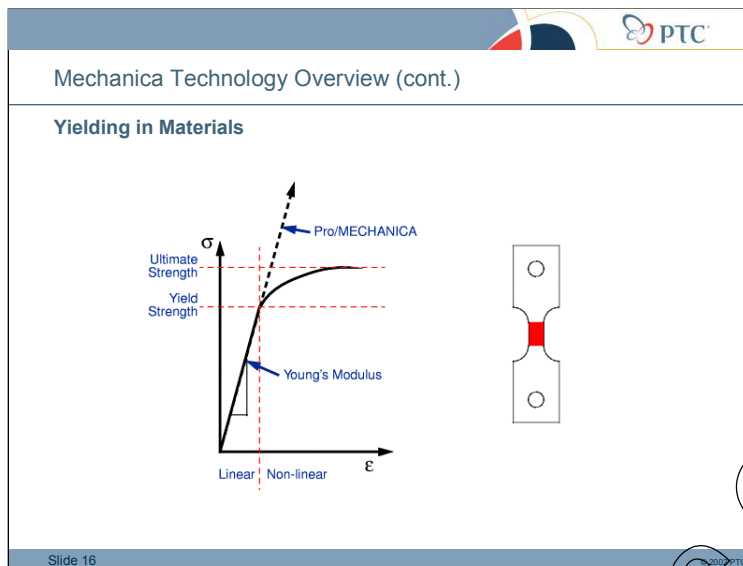
Mechanica Technology Overview

FEA and GEA

Mechanica uses Geometric Element Analysis (GEA) as opposed to Finite Element Analysis (FEA). This discussion covers the main differences between FEA/GEA as well as how Mechanica is different from other structural analysis tools.

FEA Meshes versus GEA Meshes

- An FEA mesh is made of elements with straight edges. As a result many elements are needed to produce an exact representation of the geometry. GEA elements map the geometry exactly. Only four elements needed in this example.
- FEA uses linear and at best quadratic shape functions to map deformations. Convergence is achieved through successive mesh refinement studies.
- As explained earlier, Mechanica GEA uses polynomial shape functions (up to order 9) and convergence is achieved by increasing the polynomial order of individual edges during a single run.
- Unlike FEA models (where many elements are needed at expected stress concentrations), GEA models are independent of load cases and no previous knowledge of the answer is required (p-levels needed to accommodate large stress gradients are determined during runtime).
- Finally, (and again due to polynomial shape functions), GEA elements are larger and more flexible (they can deform into irregular shapes as each edge can go to order 9). FEA elements are smaller (to map geometry) and more rigid (they can only deform slightly due to the linear shape functions).
- Other advantages of GEA meshes include: No transition regions, fewer element types, results provided everywhere in the mesh (continuous stress field) as opposed to only at nodes for FEA.



Yielding in Materials

- The continuous line on the graph represents a typical stress-strain curve for a ductile metal (*use tensile specimen example*). In the linear range of the graph (elastic range), deformations are reversible (ie: remove the load and the body will revert to its initial configuration). The slope of that curve is a constant called Young's Modulus (E).
- If the body is loaded beyond its yield strength, then non-elastic deformations occur (ie: remove the load and the body will remain in a deformed state) until the ultimate strength is reached.
- Mechanica is a linear analysis code and will compute results according to the linear range of their material stress-strain curve. If the material is loaded beyond its yield point, Mechanica results will deviate from reality (material non-linearity). This means the user must make sure the stress remains below the yield strength.
- Another situation to watch for is the large displacement non-linearity: typical example is a flat plate problem. If deformation is greater than $\sim \frac{1}{2}$ the thickness of the plate, some nonlinear effects appear in reality (shear becomes an important factor and Mechanica will overestimate displacements).
- Note that Mechanica can handle certain types of non-linearity such as buckling (the load at which buckling will occur is computed, not the post-buckling behavior), contact, and prestress loading.

Note: Large Deformation Analysis is covered in another module later.

PTC

Mechanica Technology Overview (cont.)

Failure Criteria

- von Mises and Tresca:
 - *Specifies Tensile Yield Stress.*
- Modified Mohr
 - *Specifies Ultimate Tensile Strength.*
 - *Specifies Ultimate Compressive Strength.*
- Produce Fringe Plot of Failure Index Actual Stress/Yield Stress.
 - *< 1: material below yield point*
 - *= 1: material at yield point*
 - *> 1: material past yield point*

Slide 17

Yielding in Materials

- How do you know when stress has exceeded yield strength? Which component of stress should you consider? This is what failure criteria are used for. They allow you to consider only one value with which to compare with the yield of your material. Mechanics will provide you with values for several of these criteria. You will have to choose the one that best fits your needs
- For materials whose properties are identical in all directions (isotropy), a general state of stress can be described using 9 components arranged in a 3x3 matrix (stress tensor, see figure below). This stress matrix is symmetric and reduces to 6 independent components: 3 normal-stress components and 3 shear-stress components.
- By combining stresses in all directions into a single figure, failure criteria avoid having to compare each component with the yield strength of the material.
- Using a convenient geometric transform, it is possible to orient the coordinate system such that all shear stresses are reduced to zero (Mohr's circle for stress). Once this is done, we are left with three normal stress components called principle stresses.
- Mechanics calculates stress criteria based on these principle stresses: The von Mises stress criterion and the principle stress criterion. These values can then be compared with the material yield strength.
- von Mises Stress criteria should be used for ductile metals (ie: most engineering metals). The principle stress criteria should be used for brittle materials (for example: glass, ceramics).



Summary

After completing this module, you should be able to:

- ◉ Describe Mechanica Structure.
- ◉ Describe Mechanica technology.

Slide 18

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Summary

After completing this module, you will be able to:

- Describe Mechanica Structure.
- Describe Mechanica technology.

For PTC Internal Use Only

For PTC Internal Use Only

Validating Designs Using Mechanical

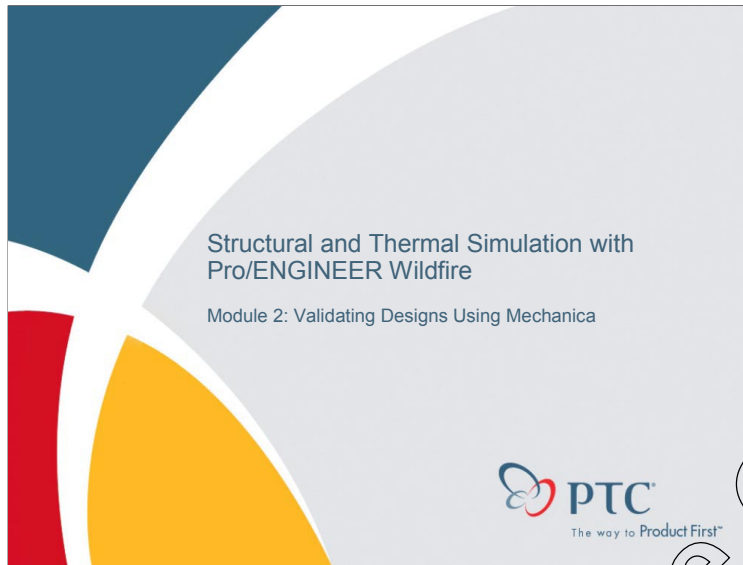
Introduction

Design processes require engineers to calculate the stress distributions and displacement of new designs. These processes allow engineers to validate design intent.

Objectives

After completing this module, you will be able to:

- Identify design requirements.
- Apply the Mechanical process to basic structure simulations for design validation.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanics*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

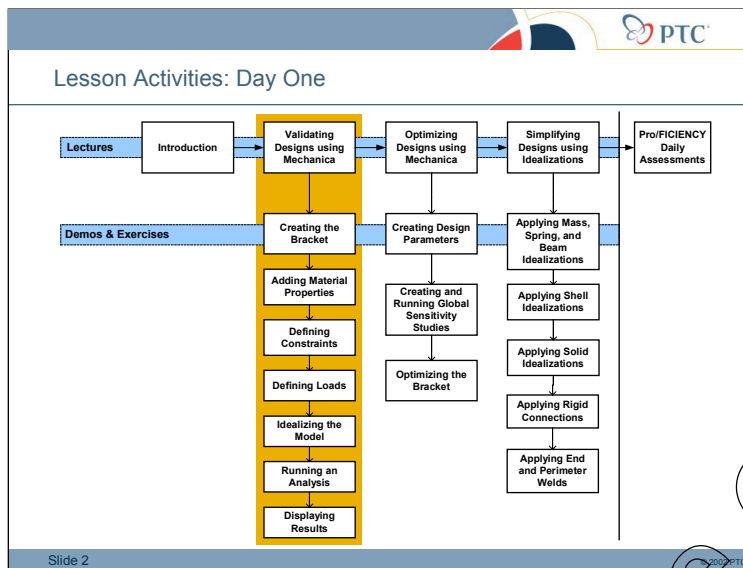
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 45 min
- Labs: 90 min
- Total: 2 hours 35 min



Objectives

After completing this module, you should be able to:

- Identify design requirements.
- Apply the Mechanica process to basic structure simulations for design validation.

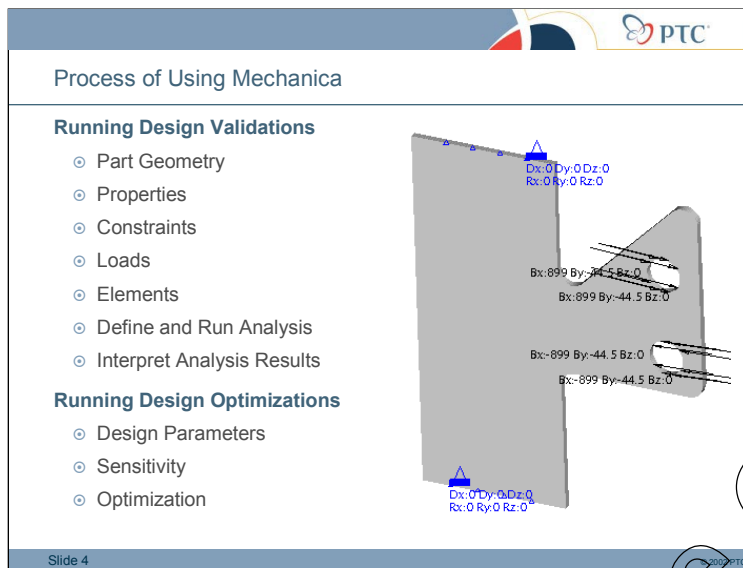
Slide 3

Objectives

Design processes require engineers to calculate the stress distributions and displacement of new designs. These processes allow engineers to validate design intent.

In this module, you will:

- Identify the design requirements using the exercise provided in this module.
- Use the design validation phase to prepare a model for static (structural) analysis.
- Create and run an analysis.
- View and interpret the results.



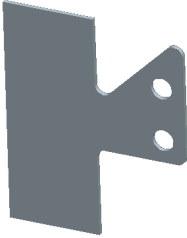
Process Overview

- A typical structural simulation consists of the design validation and optimization phases. These two phases consist of ten steps.
- The 10 steps listed here define the Mechanica Structure process. This process is reflected in the menu structure; it is also an outline for the course. This discussion is meant as an overview and each of these topics will be covered in detail in following modules.
- The first seven steps comprise the **Design Validation** phase. These steps will allow us to answer the “How much” questions (how much stress, displacement...) and validate our design for its intended use:
 - Create the Model* — Create the part or assembly that satisfies the design intent.
 - Idealize the Model* — Prepare the model for automatic mesh generation. This includes specifying appropriate idealization types and any required idealization properties.
 - Set Units and Material Properties* — Specify the appropriate system of units and assign material properties.
 - Identify Constraints* — Define realistic constraints that simulate how the model will function in the real world.
 - Set Loads* — Define loads to simulate how the model is loaded in the real world.
 - Define and Run the Analysis* — Define the appropriate analysis type (static, modal, etc.) based on the required results. You can also define the convergence settings. Run the defined analysis.
 - View and Interpret the Results* — The converged results can be visualized by generating graphs, fringe plots, etc.
- This completes the design validation phase of the process. From this point on we want to answer the “what if” questions. What if we wanted to make this design lighter? What would have to change? Can this design be improved? How does the stress vary as a function of this dimension?

PTC

Example: Identifying the Design Requirements

- ⦿ The design objective is to minimize the bracket mass.
- ⦿ The design must support a 20 pound load at a specific point.
 - *Maximum stress must be less than 93 N/mm².*
 - *The maximum allowable displacement is 0.04 mm.*
 - *The distance between the two bolt holes and sheet metal thickness must remain fixed.*



Slide 5

Identifying the Design Requirements

This example illustrates a typical structural simulation. The objective is to find a design that is stronger and lighter than the existing system, yet still easy to assemble resulting in these design requirements.

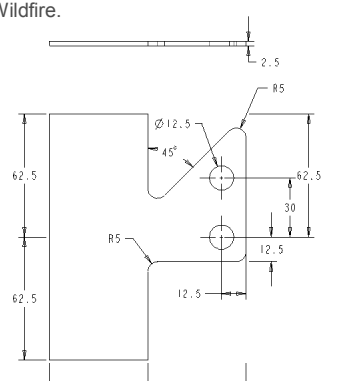
- The maximum stress must be less than 93 N/mm² (MPa). This maximum allowable stress is calculated using the tensile yield strength of the available steel (231 N/mm²) with a safety factor of 2.5.
- The maximum allowable displacement is 0.04 mm. This is necessary because a small deformation in the bracket can cause large displacement at the end of the rack it carries.
- The distance between the two bolt holes (30 mm) and sheet metal thickness (2.5 mm) must remain fixed. This is necessary due to assembly constraints.

PTC

Example: Validating Designs Using Mechanics

Bracket Design

- Create in Pro/ENGINEER Wildfire.
- Reflect the design intent.
- Dimensioning scheme.



The technical drawing shows a bracket with the following dimensions: a total width of 100 (50+50), a total height of 125 (62.5+62.5), and a hole diameter of $\varnothing 12.5$. It includes fillets with radii of R5 and a 45-degree chamfer. A dimensioning scheme on the right shows a vertical line with segments corresponding to the 62.5, 30, and 12.5 dimensions.

Slide 6

Bracket Dimensioning Scheme

Note: Relations can be used to capture the inter-relationship among the dimensions.

- The four rounds are created as two separate features: the inner and outer rounds. This dimensioning scheme provides the flexibility of altering each feature independently to improve the design.
- According to the design constraints, the dimensions of the hole will remain fixed during the optimization.

Validating Designs Using Mechanics (cont.)

Material Properties

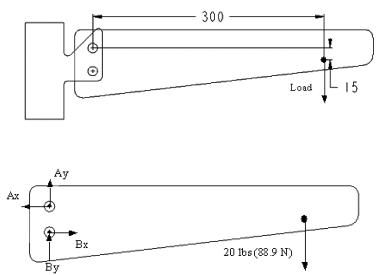
Defining Constraints

Loads

- Bracket bearing loads.
 - Rack Assembly
 - Free Body Diagram of Rack

Defining Elements

- Shell model.



Slide 7

Defining Loads

The bracket load must be calculated using realistic loading conditions. In reality, a rack is attached to the bracket using two bolts. A 20 lb. load is applied to a point as shown in the top figure.

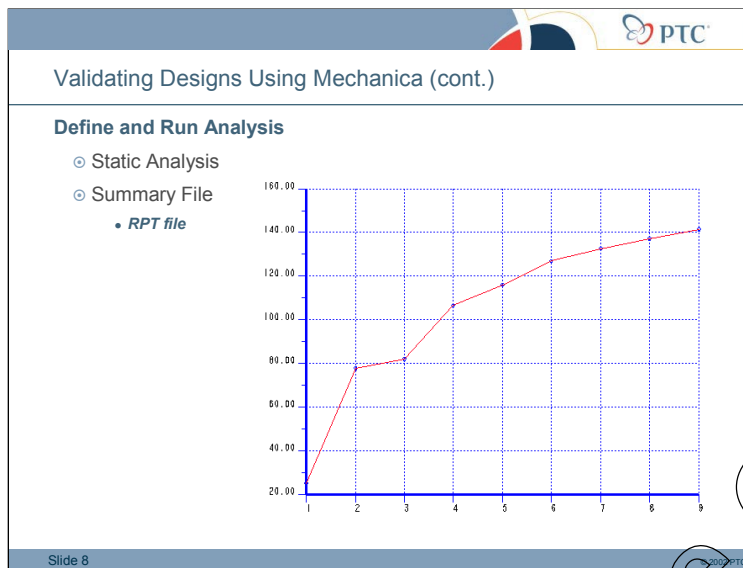
Calculating the Bracket Bearing Load

A Free Body Diagram (FBD) can be created as shown in the bottom figure.

Since the system of equations is indeterminate, the assumption $A_y = B_y$ is made. The static calculation yields the following result:

$$A_x = -889 \text{ N}; A_y = 44.5 \text{ N}; B_x = 889 \text{ N}; B_y = 44.5 \text{ N}$$

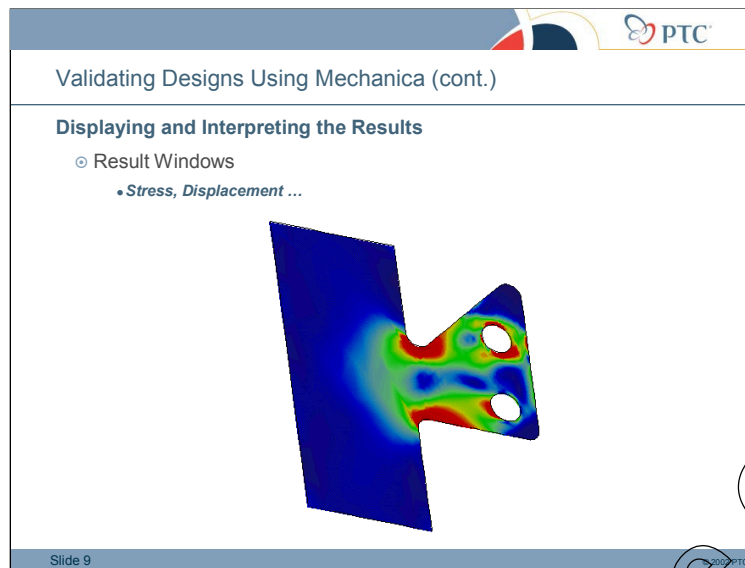
The force components of the bracket bearing load are equal and opposite.



Define and Run Analysis

Running the Analysis

- An analysis is the calculation of a model's response to its boundary conditions. Mechanica Structure provides 12 types of analyses that span a wide range of actual boundary conditions. The analysis type depends on the type of problem. For this problem, a static analysis will be defined with single pass adaptive convergence criteria. A static analysis type is used to find the stress and displacement of a structure.
- A summary file can be used to monitor the progress of the analysis.

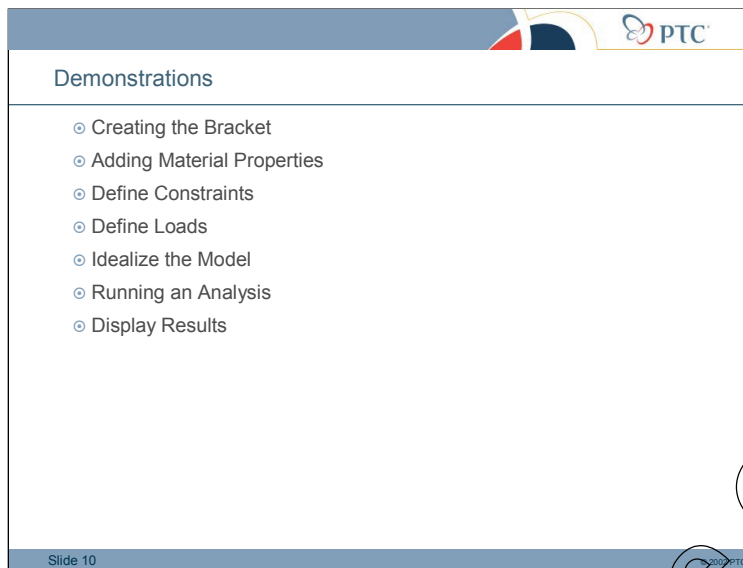


Displaying and Interpreting the Results

Result windows can be created to show the stress and displacement of the model. In this example, several result windows can be created:

- An animated fringe plot that shows the von Mises stress of the model.
- An animated fringe plot that shows the displacement of the model.
- An animated fringe plot that shows the principal stress of the model.
- A fringe plot used to dynamically query the stress of the model.

The results will show that the design needs to be improved in order to satisfy the design requirements. Since most of the part has very low stress, this part may be over-designed. This indicates that further improvement can be made to reduce weight.



Demonstrations

In this demonstration, I will:

- Create a part called BRACKET in Pro/ENGINEER Wildfire.
- Specify the material properties of the bracket as steel.
- Define how the bracket is constrained.
- Apply a bearing load on the top and bottom hole.
- Idealize the model by compressing it to a mid-surface.
- Define a static analysis.
- Create a result window.

You will repeat the same steps in the lab exercises.

Summary

After completing this module, you should be able to:

- Identify design requirements.
- Apply the Mechanica process to basic structure simulations for design validation.

Module 2 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Creating the Bracket

Introduction

In this demonstration, we will create a simple part named bracket using Pro/ENGINEER Wildfire. This part will include a protrusion, rounds, and holes. The design objective is to minimize the bracket mass resulting in the design requirements stated in the exercise scenario.

Objectives

After successfully completing this exercise, you will know how to:

- Create simple Pro/ENGINEER Wildfire parts.

Scenario

You are an engineer who works for a firm that designs and manufactures shelving systems for warehouses, storage units, and large hardware stores. Your manager assigns you to work on a new shelf design project. The design specification requires that the shelf system have a maximum stress less than 93 N/mm^2 and a maximum displacement less than 0.04 mm .

The objective is to find a design that is stronger and lighter than the existing system, yet still easy to assemble. This results in the following design requirements.

The design must support a 20 lb. (pound) load applied at a specific point.

The following design criteria must be satisfied:

- The maximum stress must be less than 93 N/mm^2 (MPa). This maximum allowable stress is calculated using the tensile yield strength of the available steel (231 N/mm^2) with a safety factor of 2.5.

- The maximum allowable displacement is 0.04 mm. This is necessary because a small deformation in the bracket can cause large displacement at the end of the rack it carries.
- The distance between the two bolt holes (30 mm) and sheet metal thickness (2.5 mm) must remain fixed. This is necessary due to assembly constraints.

The bracket design is created using Pro/ENGINEER Wildfire. When constructing the model, the dimensioning scheme should reflect the design intent and also provide enough design parameters for the design optimization.

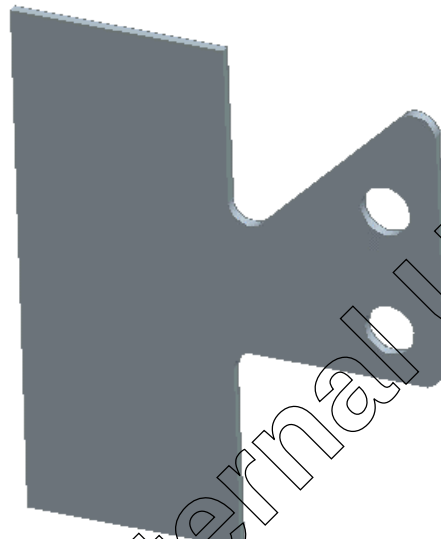


Figure 1: The Bracket Part

Step 1 Create the bracket part in Pro/ENGINEER Wildfire.

The first step is to create the part for your model in Pro/ENGINEER Wildfire. In this exercise, you create the shelving bracket; the relations and dimensioning scheme are a critical part of your model. The dimensions and relations you create when building your model in Pro/ENGINEER Wildfire become the design parameters that dictate how the size and shape of the model can be altered during optimization. Let us begin by starting the Pro/ENGINEER Wildfire application and begin creating our new part.

1. Start Pro/ENGINEER Wildfire.
2. Set the working directory to **C:\users\student\mechanica_330\module_02_03**.
3. Click **Tools > Environment**, and clear the Spin Center option.
4. Click **File > New > Part**, and name it **BRACKET**.
5. Edit units by selecting **Edit > Setup > Units**. Choose **mmNs > Set > OK > Close**.

1. Click **Insert > Round**. Select the edges as shown in the following figure. Remember to press and hold CTRL + left-click to add the second edge. Edit the radius to **5** using the dashboard and complete the feature.

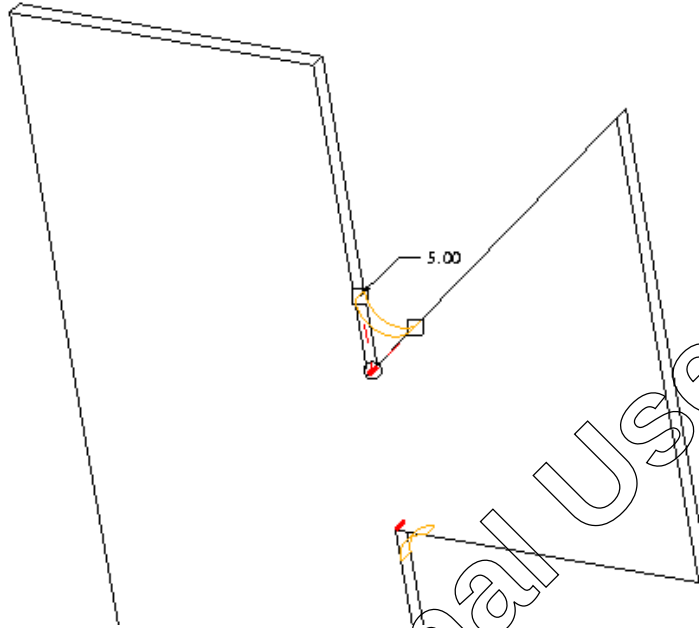


Figure 3: Inner rounds

2. Repeat the previous procedure to create another round on the two outer edges as shown in the following figure with a radius of **5**.

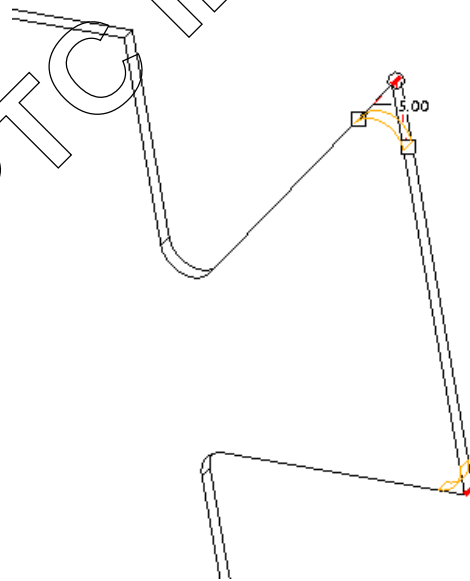


Figure 4: Outer Rounds

Step 4 Create and dimension the two holes.

According to the design constraints, the dimensions of the hole will remain fixed during the optimization.

1. Insert a straight hole on the front surface of the bracket. Click **Insert > Hole**.
2. Select the large surface as the primary reference.
3. Right-click and select **Secondary References Collector**.
4. Select the RIGHT datum plane and edit the distance to **-12.5**.
5. Press and hold CTRL + left-click to select to add the TOP datum plane. Edit the distance to **0.0**.
6. Edit the hole's diameter to **12.5**.
7. Using the dashboard, set the depth of the hole so that it intersects all surfaces as shown in the following figure.

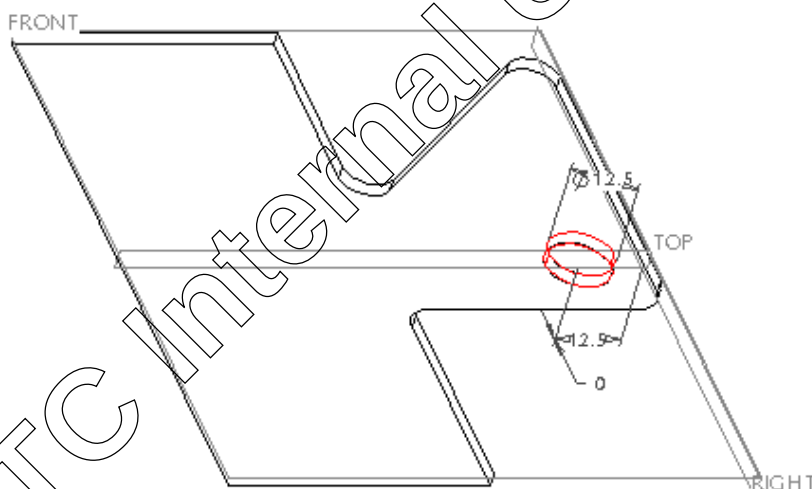


Figure 4b: Create Hole

Step 5 Create the second hole as an identical pattern.

1. With the hole selected, right-click, and select **Pattern**.
2. Select the 0.0 dimension as the pattern dimension for the first direction. Edit the dimension increment to **30**.
3. Enter **2** for the number of instances.
4. Complete the feature as shown in the following figure.

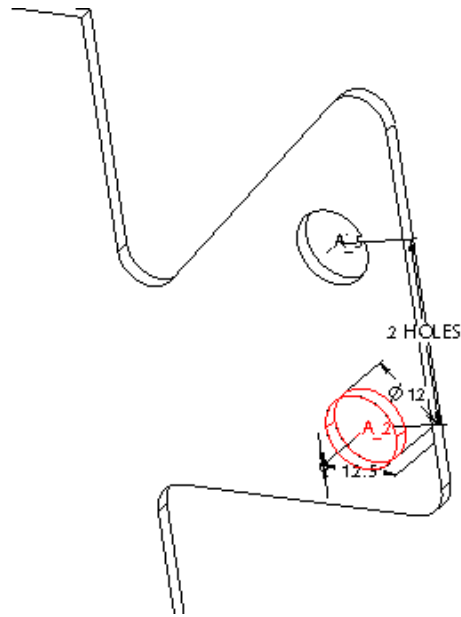


Figure 5: The Hole Pattern

5. Save the model.

Exercise 2: Adding Material Properties

Introduction

The next step in the Mechanics process is to assign material to a part. In this demonstration, we learn how to use Mechanics in an integrated mode to assign material to a bracket. The bracket is made of steel. The material properties can be assigned to the bracket using the available properties in the Mechanics library.

Objectives

After successfully completing this exercise, you will know how to:

- Use Mechanics to assign material to a part.

Scenario

The next step in the Mechanics process is to assign material to a part. In this exercise you will define the bracket as steel.

Step 1 Use Mechanics to specify the material properties of the bracket.

Before we can work with the bracket in Mechanics, we need to open the Mechanics application. From within the Wildfire application, we can start Mechanics. With the part still open in the Pro/ENGINEER window, open Mechanics by selecting Mechanics from the Applications menu. Continue and confirm your system of units. Since this is a structure simulation, we need to select Structure from the MECHANICA menu.

1. Enable Mechanics Structure by clicking **Applications > Mechanics**.
2. Check the system units in the dialog box and click **Continue**.
3. Select **Structure** from the MECHANICA menu.

Step 2 Specify the material properties of the bracket.

1. Click **Materials**.
2. Add STEEL to the **Materials in Model** list by selecting **STEEL** in the Materials in Library and left-clicking >>>.
3. Assign STEEL as BRACKET.PRT material by clicking **Assign > Part**. Select the bracket part and click **OK**.
4. Close the Materials dialog box.

Exercise 3: Defining Constraints

Introduction

In reality, the top and bottom surfaces of the bracket are welded to other components. As a result, these surfaces need to be rigidly fixed in Mechanica so that no movement is allowed there. In this demonstration, we are going to constrain the top edge of the part to simulate real operating conditions.

Objectives

After successfully completing this exercise, you will know how to:

- Use Mechanica to add constraints to a part.

Scenario

The next step in the Mechanica process is to add constraints to a part. In this exercise, you are going to constrain the top edge of the part to simulate real operating conditions. In reality, the bracket is attached by welding the top and bottom edges to the adjoining hardware.

Step 1 Define how the part is constrained.

1. Create an edge constraint by clicking **Constraints > New > Edge/Curve**.
2. Create a new constraint set named **weld**. Select the front top edge by left-clicking and front bottom edge of the bracket as curve references by holding CTRL + left-click. Fix all the DOFs (Degrees of Freedom) as shown in the following figure.

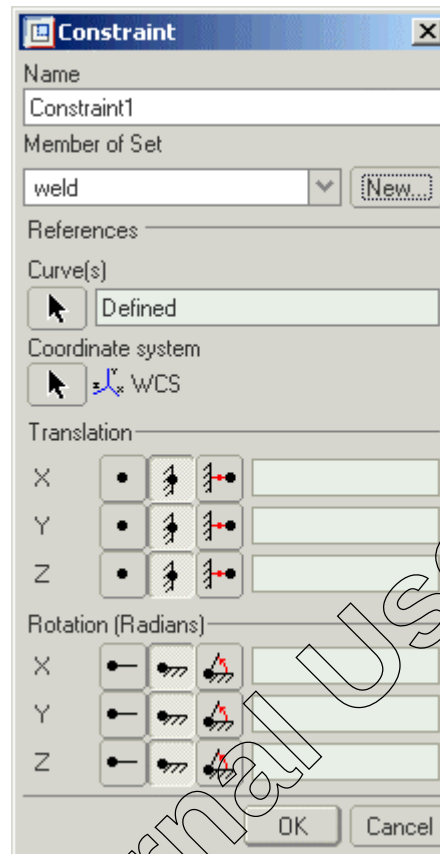


Figure 6: Constraints Dialog Box

3. After assigning the constraints, the model should appear as shown in the following figure.

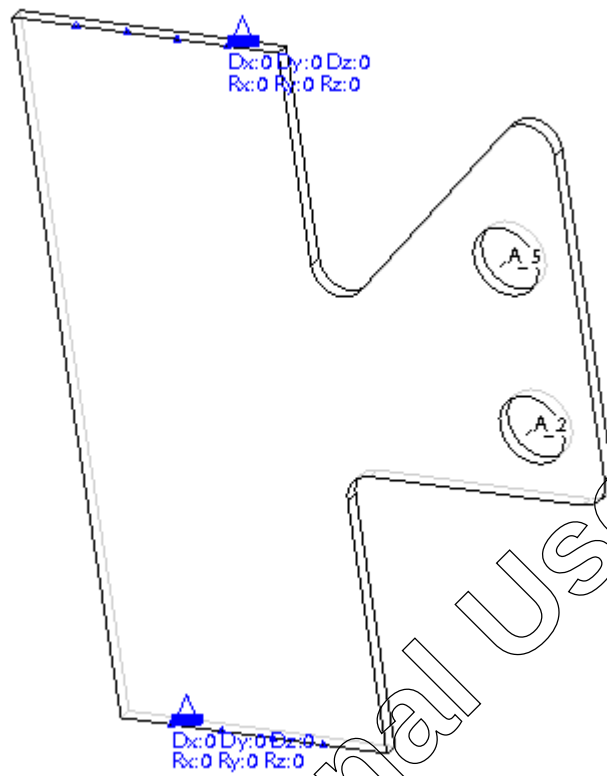


Figure 7: Bracket with Edge Constraints

Exercise 4: Defining Loads

Introduction

The bracket load must be calculated using realistic loading conditions. In reality, a rack is attached to the bracket using two bolts. A 20 lb. load is applied to a point on the rack. In this demonstration, we will add bearing loads on each hole based on the calculations presented in the exercise scenario.

Objectives

After successfully completing this exercise, you will know how to:

- Apply loads.

Scenario

After creating the part, assigning a material and adding constraints, you are going to add bearing loads on each hole.

Defining Loads

In this exercise, the bracket load must be calculated using realistic loading condition. In reality, a rack is attached to the bracket using two bolts. A 20-pound load is applied to a point as shown in the following figure.

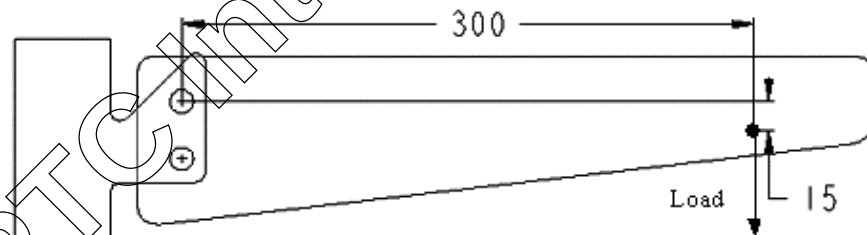


Figure 8: Rack Assembly

Calculating the Bracket Bearing Load

A Free Body Diagram (FBD) can be created as shown in the following figure.

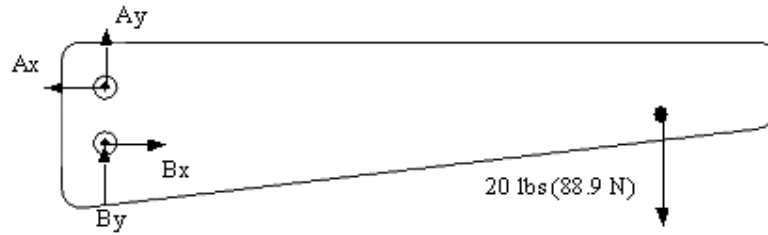


Figure 9: The Free Body Diagram of the rack.

Since the system of equations is indeterminate, the assumption $A_y = B_y$ is made. The static calculation yields the following result:

$$A_x = -889 \text{ N}; A_y = 44.5 \text{ N}; B_x = 889 \text{ N}; B_y = 44.5 \text{ N}$$

The force components of the bracket-bearing load are equal and opposite.

Step 1 Apply the bearing load on the top hole.

The bearing load type automatically creates a load with a cosine distribution on circular or cylindrical geometry. The peak direction of the load is defined by the components you specify.

1. Create a bearing load by clicking **Loads > New > Bearing**.
2. Create a new load set named **bearing_load**.
3. Select the front bottom edge of the top hole as the **Bearing hole**.
4. Do not change the Force default option **Components**. Edit the X component to **889** and edit the Y component to **-44.5**, and keep the Z component as **0**.
5. Preview the results. The system displays the load's distribution, as shown in the following figure.

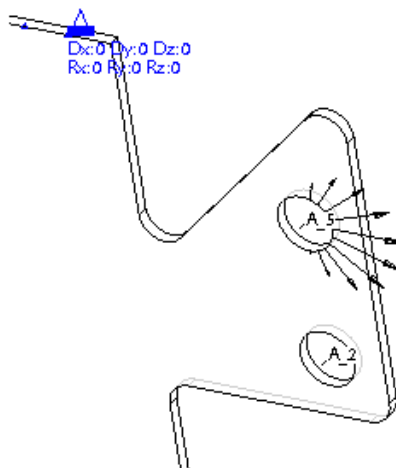


Figure 10: Bearing Load on the Upper Hole

Step 2 Apply the bearing load on the bottom hole.

1. Create another bearing load on the front bottom edge of the bottom hole using the same procedure.
2. Ensure that **bearing_load** is the load set name.
3. Do not change the Force default option **Components**. Edit the X component to **-889**, edit the Y component to **44.5**, and keep the Z component as **0**.

For PTC Internal Use Only

Exercise 5: Idealizing the Model

Introduction

Because this bracket is sufficiently thin, you will analyze it using shell elements. Therefore you must now develop a shell model by defining mid-surfaces.

Objectives

After successfully completing this exercise, you will know how to:

- Create a mid-surface for shell idealization.

Scenario

Since the bracket is thin, you can use shell elements instead of solid elements. In order for Mechanica to create shell elements a mid-surface has to be defined.

To define shell elements, surface pairs must first be defined. In this example, the front and back surface of the bracket model are defined as a surface pair. The surface pair will be compressed to form the mid-surface, as shown in the following figure. By compressing the three-dimensional geometry of the bracket into a surface, shell elements may be used to mesh the part.

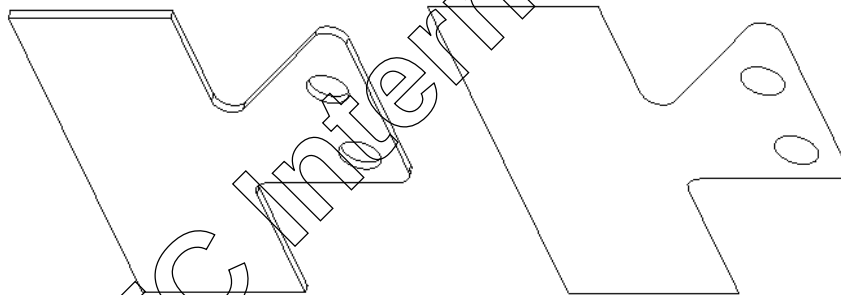


Figure 11: Mid-surface Compression

Step 1 Idealize the bracket part using a shell surface.

Since the bracket is thin, you can use shell elements instead of solid elements. In order for Mechanica to create shell elements, a mid-surface has to be defined.

1. Create a shell surface by clicking **Idealizations > Shells > Midsurfaces**.
2. Click **New**. Select the front surface by left-clicking and the back surface of the model by holding CTRL + left-click. Mechanica highlights the pairs in red and yellow.

Step 2 Verify the idealization and visualize the compressed model.

It is important to review the compressed model to be sure that the mid-surfaces created by Mechanica are correct.

1. Test the mid-surface compression by clicking **Compress > Shells Only > ShowCompress**. The system displays only the mid-plane geometry in yellow, as shown in the following figure.

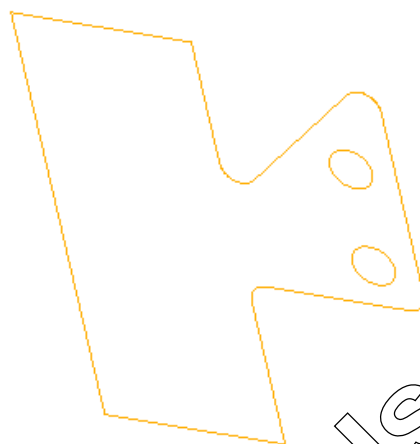


Figure 12: Compressed Bracket Mid-surface

Step 3 Create the elements to test the model.

Elements can be created on the model prior to running an analysis, or they can be created during the analysis process.

1. In the STRC MODEL menu, click **AutoGEM** (Automatic Geometric Element Mesh) to begin shell element generation.
2. Verify that Use Pairs is enabled from the Settings drop-down menu.
3. Click **Create** to automatically generate the shell elements. After the element generation is complete, the model should appear as shown in the following figure.
4. Click **File > Save Mesh** to save the created mesh. Close the dialog box.

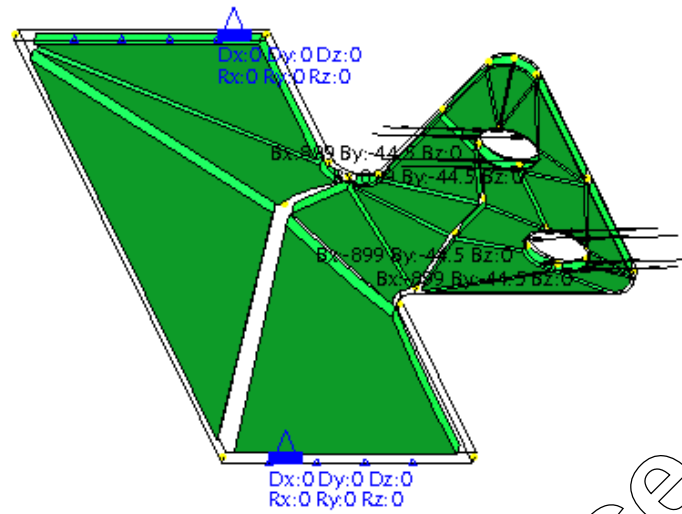


Figure 13: The shell elements in the mid-surface

Exercise 6: Running an Analysis

Introduction

In this demonstration, we will define and run an analysis. The type of analysis you choose depends on the type of problem you are trying to simulate. For this exercise, a static analysis will be defined with single pass adaptive convergence (SPA) criteria. The summary file can be used to monitor the progress of the analysis.

Objectives

After successfully completing this exercise, you will know how to:

- Create a static analysis and add constraints and loads.
- Modify the convergence and plotting grids.
- Run the analysis.
- Examine the summary file.

Scenario

After setting up the bracket with geometry, material, constraints, loads, and elements you want to find out what are the stress distribution and the maximum stress. To do this you will create a static analysis and run it.

Step 1 Define a structural static analysis to determine the stress and strain caused by the applied loads.

1. In the MEC STRUCT menu, click **Analyses/Studies**.
2. Create a new static analysis by clicking **File > New Static**.
3. Name the analysis **bracket_static**.
4. Click **weld** as the constraint.
5. Click **bearing_load** as the load set. Click **LoadSet1** to deselect it as a load set.
6. On the Convergence tab, ensure that convergence method is set to **Single-Pass Adaptive**.
7. On the Output tab, set the plotting grid output to **7**.
8. Click **OK** to close the Static Analysis Definition dialog-box.

Step 2 Run the defined analysis.

1. Click **Run > Start** to begin the bracket_static analysis.

2. Click **Yes** for Error Detection.

Step 3 Monitor the progress of the analysis.

*Check to see how the analysis is progressing by monitoring the summary file. This file shows you details about each solver pass as the analysis progresses. You do this by selecting **Status**. A summary window appears, displaying a report of the analysis's progress. The report should show information about Principal System of Unit, model type and geometry information, and element information.*

1. Click **Info > Status** to display the summary window.
2. Scroll down the report to view the progress.

Scroll down a little further and the report shows that the solver makes 2 passes. In the first pass, all of the edges are set to $p=3$, which is usually referred to as third order polynomial. Based on the results of this pass, the final edge order for all the edges is determined between three and nine. This polynomial order distribution is applied to the model and a second pass is performed.

3. Review and record the polynomial order (Maximum Edge Order) of the second pass.

Scroll down a little further and review the error estimate.

4. Review and record the error estimate.

Scroll down a little further. The report shows the constraint set and load set information.

5. Check the values for the total load (Resultant Load on Modal) in X, Y and Z directions.

Scroll down a little further and review the values of the measures. Some measures of interest may be the maximum displacements and stress components.

6. Review the Measures values.

Review the Memory and Disk Usage information. You can also find run time in this section.

7. Review the Memory and Disk Usage information.

At the end of the report, it indicates that the run is completed.

8. Click **Close** to exit the Summary window.
9. Click **Close** to exit the Analyses and Design Studies dialog box.

Exercise 7: Displaying Results

Introduction

In this demonstration, we will look at two results of your analysis: a color fringe plot of von Mises stress and an animation of the resulting deformations.

To look at any results from a Pro/MECHANICA run, you must create a result window. Create fringe plot of von Mises stress and an animation of the resulting deformations.

Objectives

After successfully completing this exercise, you will know how to:

- Create a fringe plot of von Mises stress.
- Create an animated fringe plot of displacement.

Scenario

After running the static analysis, to look at any results from a Mechanica run, you must create a result window. You will create fringe plot of von Mises stress and an animation of the resulting deformations and examine the stresses on the bracket.

Step 1 Create a Results window to look at the distribution of von Mises stress in the model.

1. Click **Results** to begin results creation. Do not save the model if prompted.
2. Click **Insert > Result Window** to create a new results window to display the stress.
3. Name the window **bracket_max_vm**.
4. Change the title to **Maximum von Mises**.
5. Select the Design Study directory by browsing to **bracket_static**, and click **Open**.
6. Ensure that the Display type is set to **Fringe**.
7. Ensure that the Quantity is set to display **Stress**.
8. Ensure that the Component is set to **von Mises** and that it includes all distributions from the shells.
9. On the Display Options tab, clear the Continuous Tone display option if necessary.
10. Check the Deformed option with a scale of **10%**.
11. Check the Animate option and specify **16** frames.

Note:

More frames will result in a smoother animation and will take longer to generate.

12. Ensure that the **Auto Start** and **Continuous Loop** options are enabled.
13. Click **OK and Show** to finish the definition and display the result window.
14. Use the animation control buttons to start, stop, and step the animation.
15. Proceed to Step 2 without closing the result window.

Step 2 Create a result window to display the displacement by copying the existing window.

Copying an existing result window simply eliminates the need to re-select the results directory when defining the new window.

1. Copy the existing results window.
2. Edit the name to **Bracket_Displacement**.
3. Change the title to **Bracket Displacement**.
4. Ensure that the Display type is set to **Fringe**.
5. Ensure that the Quantity is set to display **Displacement**.
6. Ensure that the Component is set to **Magnitude**.
7. On the Display Options tab, clear the Continuous Tone display option if necessary.
8. Check the Deformed option with a scale of **10%**.
9. Check the Animate option and specify **16** frames.
10. Click **Auto Start** and **Continuous Loop**.
11. Click **OK and Show** to finish the definition and display the result window.
12. By default, both results windows will now be displayed. Hide the Maximum von Mises result window using the Hide definition from selection tool.
13. Use the control buttons to start, stop, and step the displacement animation.
14. Proceed to Step 3 without closing the Displacement Result window.

Step 3 Create a result window to display the Maximum Principal Stresses by copying the existing window.

Maximum Principal Stress distinguishes tension from compression.

1. Copy the existing results window.
2. Change the name to **Bracket_Principal**.
3. Edit the title to **Maximum Principal Stress**.
4. Ensure that the Display type is set to **Fringe**.

5. Ensure that the Quantity is set to display Stress.
6. Ensure that the Component is set to **Max Principal Stress**.
7. Keep the other default settings.
8. Finish the definition and display the result window.

Again, both windows will be displayed.

9. Remove the Bracket Displacement window from display.
10. Proceed to Step 4 without closing the Maximum Principal Result window.

Step 4 Create a Dynamic Query result window to further investigate the results.

The Dynamic Query can be used to investigate the results at any location on the model. A Query window will display the results as you move the mouse over the model.

1. Copy the existing results window.
2. Change the name to **Bracket_vm_query**.
3. Edit the title to **von Mises Query**.
4. Ensure that the Display type is set to **Fringe**.
5. Ensure that the Quantity is set to display **Stress**.
6. Ensure that the Component is set to **von Mises**.
7. Clear the **Animate** checkbox.
8. Keep the other values as default.
9. Finish the definition and display the result window.
10. Remove the Maximum Principal window from display.
11. Click **Select Info > Dynamic Query** to investigate the stresses at any location on the model.

Step 5 Display the result windows and examine the results.

To control an individual window separately, select the window to activate it. The borders of the selected window should be highlighted in yellow.

To control multiple windows simultaneously, press SHIFT and select all the animated windows.

1. Show all the results windows.

Step 6 Interpret and record the results.

1. Write down the following observations:

- Maximum von Mises Stress _____;
- Maximum displacement _____;
- Does it satisfy the design requirement? _____.

2. Exit the result window and save the window. When prompted edit the name to **Original**.

3. Accept the changes to finish.

4. Save the model.

5. Close the window.

6. Erase components not displayed by selecting **File > Erase > Not Displayed**.

Summary

After completing this module, you will be able to:

- Identify design requirements.
- Apply the Mechanica process to basic structure simulations for design validation.

For PTC Internal Use Only

For PTC Internal Use Only

Optimizing Designs Using Mechanical

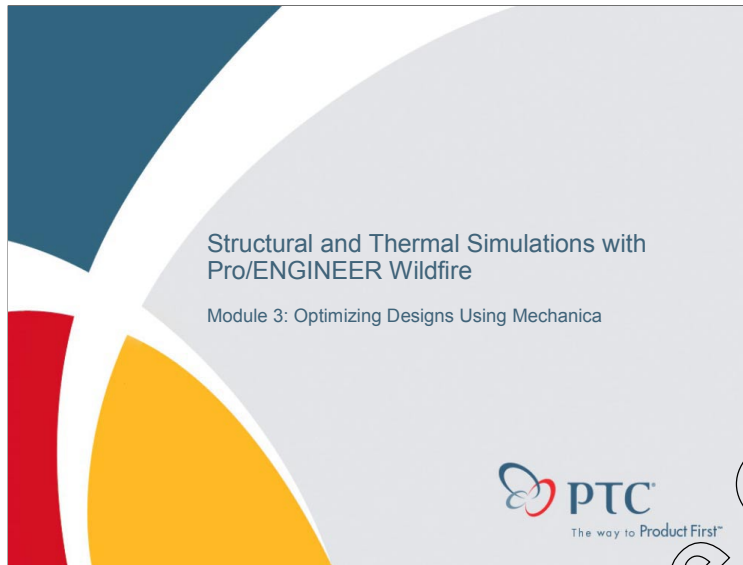
Introduction

To further improve a design, engineers may need to determine the key variables that control and influence the design. This information can be used to optimize designs.

Objectives

After completing this module, you will be able to:

- Create design parameters.
- Create and run a global sensitivity study.
- Create and run an optimization study.
- Display results and optimized models.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanics*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

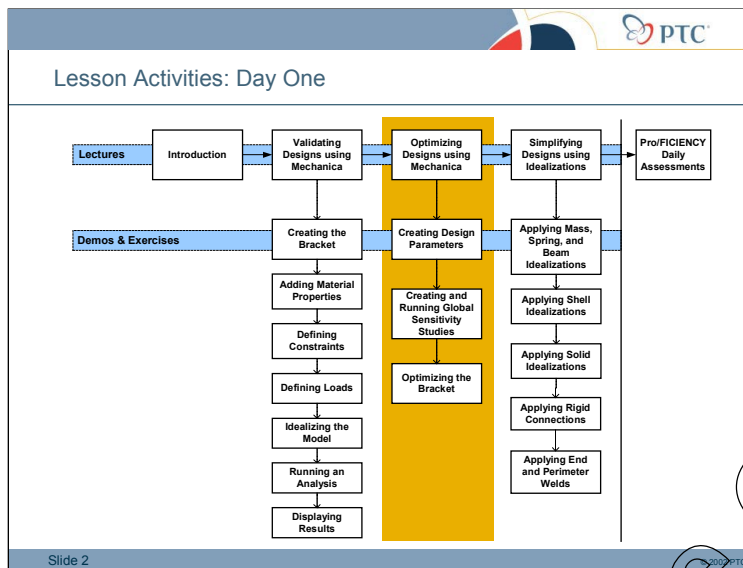
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 15 min
- Demos: 30 min
- Labs: 60 min
- Total: 1 hr 45 min



Objectives

After completing this module, you should be able to:

- Create design parameters.
- Create and run a global sensitivity study.
- Create and run an optimization study.
- Display results and optimized models.

Slide 3

Objectives

To further improve a design, engineers may need to determine the key variables that control and influence the design. This information can be used to optimize designs.

In this module, you will use the design optimization phase to:

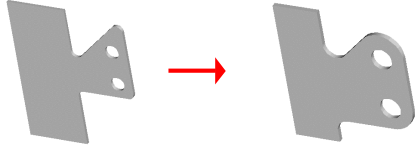
- Optimize the design.
- Find design parameters that best meet the design goal.
- Run sensitivity studies to determine the sensitivity to a change in the design parameters.
- Run optimization studies to find the best design parameters to satisfy the imposed constraints.

PTC

Optimizing Designs Using Mechanical

Design Optimization Tool

- Part shape and properties can be parameterized.
- Design Parameters can be created.
- Vary the design parameters parameters to find the best design.

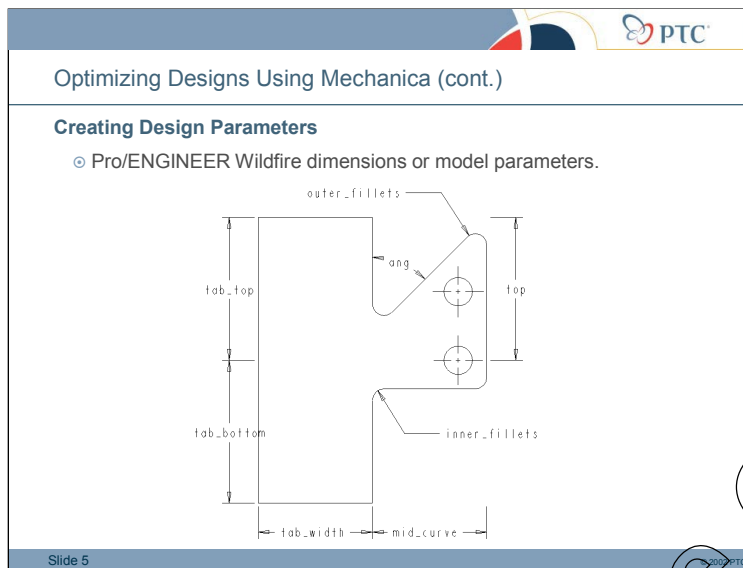


Optimization Process

- Design Parameters
- Sensitivity
- Optimization

Slide 4

There are no instructor notes on this slide.



Creating Design Parameters

Design parameters are variables of the model that can potentially affect the design objectives. Design parameters can be Pro/ENGINEER Wildfire dimensions or model parameters.

In a design optimization, the system changes the variables within a certain range to find the best values that satisfy the constraints and optimize the design goal. In the example of the bracket, there are eight design parameters.

Maintaining Design Intent Using Relations

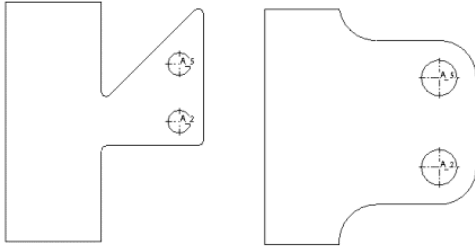
In Pro/ENGINEER Wildfire, relations can be created to capture the inter-relationship among parameters and dimensions.

In Mechanica, these relations remain valid and active in the simulation process. Whenever a design parameter is changed, Mechanica will re-evaluate the relations and regenerate the model. This ensures that the model satisfies the design intent.

PTC

Example: Optimizing Designs Using Mechanica (cont.)

- Maintaining design intent using relations.
 - Shape of the Bracket Using Relations
 - $\text{top} = 62.5 + (45 - \text{ang}) * 0.5$



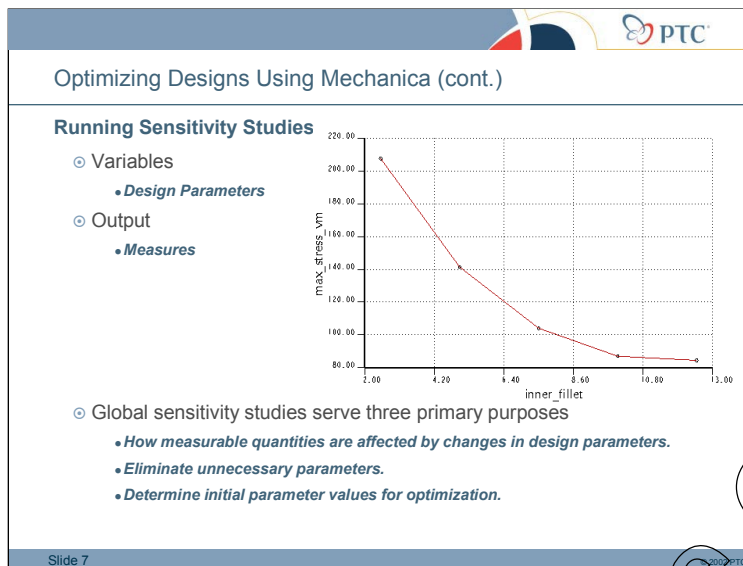
Slide 6

Shape of the Bracket Using Relations

In this example, the basic shape of the model should be maintained. When varying the dimension ANG within the allowable range (from 45 degrees to 90 degrees), the dimension top should be adjusted accordingly. To capture this design intent a relation will be created:

$$\text{top} = 62.5 + (45 - \text{ang}) * 0.5$$

Because of this relation, the dimension top will change as ANG changes from 45 to 90 degrees. Thus, the bracket model maintains its basic shape, as shown in this figure. The relation is used to maintain design intent during a sensitivity study. As a result, eight independent design parameters can be reduced to seven.



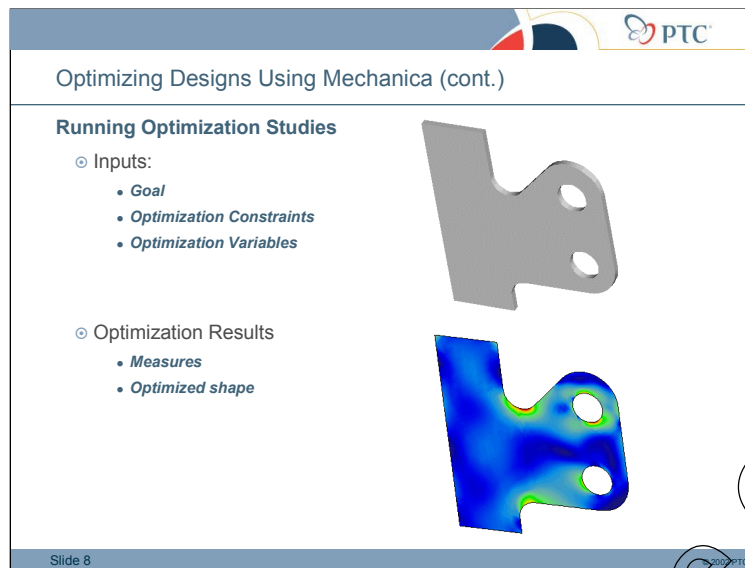
Running Sensitivity Studies

Sensitivity studies are used to determine whether a certain characteristic or property of the model is sensitive to a change in a design parameter. Specifically, the system calculates the changes in a model's measures (such as stress and displacement) when a parameter varied over a specified range.

Sensitivity studies can be performed by varying the parameters within a range (global sensitivity), or at a specific value (local sensitivity).

Global sensitivity studies serve three primary purposes:

- To help understand how a measurable quantity is affected as a parameter varies through its range.
- To help eliminate unimportant parameters from an upcoming optimization.
- To determine a good initial value for a parameter in an optimization.



Running Optimization Studies

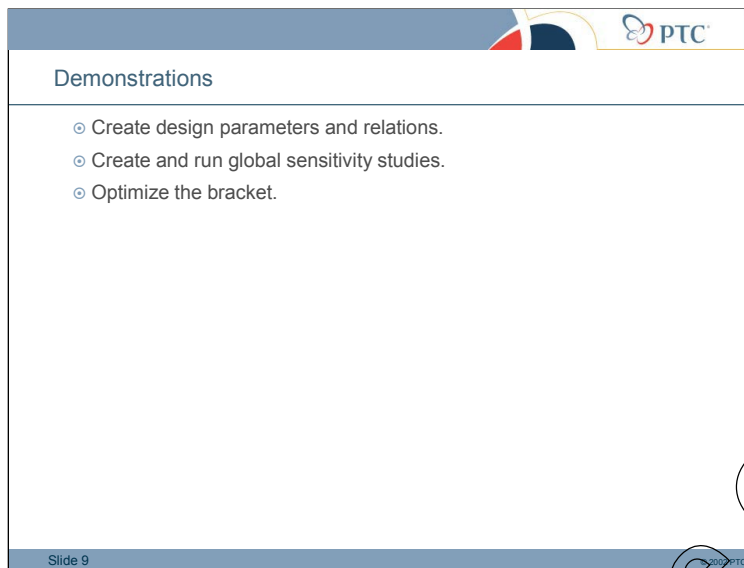
In an optimization study, the solver tries to find a set of design parameter values within their specified ranges that maximizes or minimizes a measurable quantity while satisfying all of the imposed constraints.

There are several key elements that need to be defined in an optimization study.

- **Goal:** A goal is a certain measurable quantity of the model that will be maximized or minimized in an optimization design study. In the case of the bracket, the goal is to minimize the mass of the bracket.
- **Optimization Constraints:** These are constraints that a model must satisfy. In the case of the bracket, the constraints are:
 - The maximum von Mises stress must be less than 93 N/mm^2 (MPa).
 - The maximum allowable displacement is 0.04 mm.
- **Optimization Variables:** In an optimization study, the solver changes the variables within their range to find the best values that satisfy the constraints and optimize the goal. In the case of the bracket, there are 7 variables shown in the figure (the ranges for these variables can be found in the exercise).

Reviewing Optimization Results

The types of results that should be reviewed are the plots showing von Mises stress compared to optimization pass. You may also want to review total mass compared to optimization pass.



Demonstrations

In this demonstration, I will:

- Create design parameters by renaming the dimensions in Pro/ENGINEER Wildfire.
- Investigate the parameters with a global sensitivity study.
- Create the optimization design study.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- Create design parameters.
- Create and run a global sensitivity study.
- Create and run an optimization study.
- Display results and optimized models.

Slide 10

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 3 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Creating Design Parameters

Introduction

In a design optimization, the system changes the variables within a certain range to find the best values that satisfy the constraints and optimize the design goal. In the bracket example, there are eight design parameters. The answers from the static analysis suggest that there is room for improvement, including reducing weight. You can use relations to further maintain design intent during a sensitivity study. In this demonstration, we will show that the relation is used to maintain design intent during a sensitivity study based on the exercise scenario. As a result, eight independent design parameters can be reduced to seven.

Objectives

After successfully completing this exercise, you will know how to:

- Create design parameters.
- Review the model shapes.

Scenario

In this example, the basic shape of the model should be maintained. When varying the dimension angle within the allowable range (from 45 degrees to 90 degrees), the dimension TOP should be adjusted accordingly. To capture this design intent a relation will be created:

$$\text{top} = 62.5 + (45 - \text{ang}) * 0.5$$

Because of this relation, the dimension TOP will change as the angle changes from 45 to 90 degrees. Thus, the bracket model maintains its basic shape, as shown in the following figure.

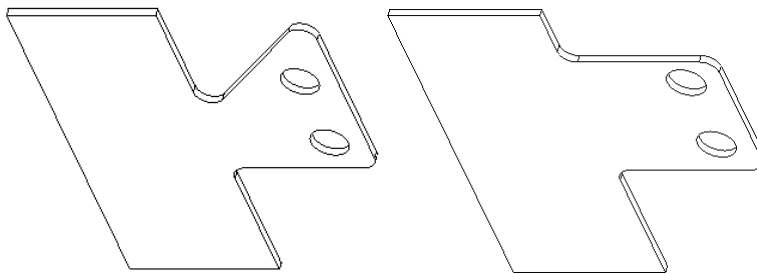


Figure 1: Shape of the bracket using relations.

In this exercise, the relation is used to maintain design intent during a sensitivity study. As a result, eight independent design parameters can be reduced to seven.

Step 1. Rename the dimensions in Pro/ENGINEER Wildfire.

Note:

If you were unable to complete the bracket in the previous exercise, you can retrieve a completed bracket from C:\users\student\mechanica_330\module_02_03. Open BRACKET_COMPLETED.PRT.

1. Open BRACKET.PRT.
2. **Edit** the main protrusion to display the dimensions. Select the **45** degrees angular dimension. Right-click and select **Properties**.
3. Select the Dimension Text tab from the Dimension Properties dialog box. Enter **ang** in the name field.
4. Edit the properties of the **62.5** dimension. Change the dimension name to **top** as shown in the following figure.
5. Toggle the dimensions between the numerical value and the symbolic name by clicking **Info > Switch Dimensions**. The model should look as shown in the following figure.

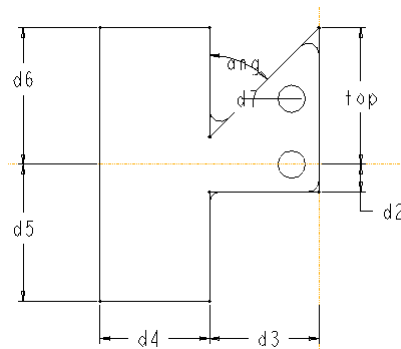


Figure 2: ang and top Dimensions

Step 2. Create a relation. As the angle changes, the bracket tip must rotate to maintain the desired part shape. Create a relation that captures this design intent.

1. Select **Tools > Relations** then add the following relations:

```
/* Maintain bracket height.

if ang >= 45 & ang <= 90

top = 62.5 + (45-ang) * 0.5

endif
```

2. Test the relations by changing the dimension from 45 to **89**.
3. Edit the dimension back to **45**.

Step 3. Define the design parameters in Mechanica.

1. First, you add a design parameter to vary the size of the inner fillets. Enable Mechanica Structure by choosing **Applications > Mechanica > Structure**.
2. Select **Model > Dsgn Controls > Design Params > Create**. Click **Select** and choose the inner fillet value as shown in the following figure in the following step. Edit the name to **inner_fillets**.
3. Define the range by editing the minimum to **2.5** and edit the maximum to **12.5**.
4. Animate the model using **Shape Animate**, and edit the **inner_fillets** from **2.5** to **12.5** and enter **2** for Number of Intervals to generate two models. Click **Animate**. Enter **Yes** to proceed to the next step.
5. Click **Yes** to return the model to its original shape.

Step 4. Define additional design parameters and animate the shape.

1. Using the procedures in the previous step, create more design parameters as shown in the figure below. The name and range of the design parameters can be found in the following table.

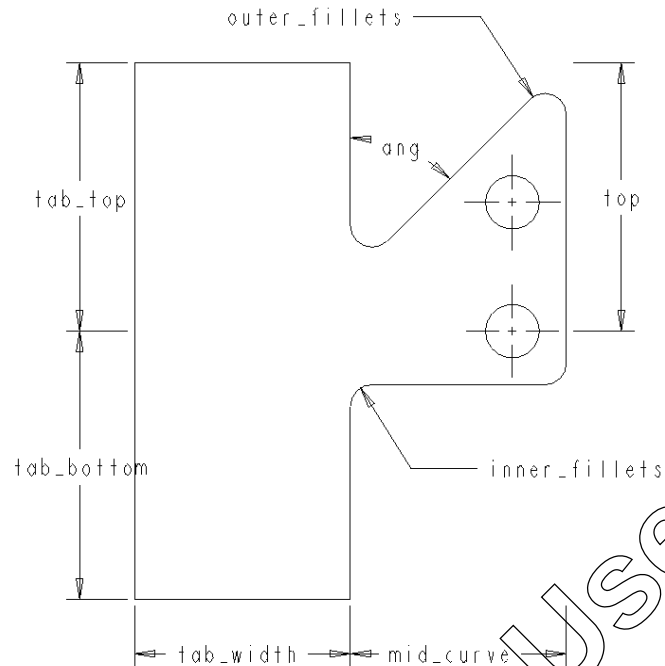


Figure 3: Design Parameters

Table 1: The design parameters.

Name	Min	Current	Max
tab_width	25.0	50.0	50.0
tab_top	53.0	62.5	62.5
tab_bottom	26.0	62.5	62.5
mid_curve	46.0	50.0	53.0
ang	45	45	90
outer_fillet	2.5	5	12.5

2. Animate the model with all parameters. Change their settings as shown in the following table.

Table 2: Shape Animate Settings

Parameter Name	Settings1:	Settings2:
tab_width	50.0	25.0
tab_top	62.5	53.0
tab_bottom	62.5	26.0
mid_curve	53.0	46.0
ang	45	90
inner_fillet	2.5	12.5
outer_fillet	2.5	12.5

3. Edit the NUMBER OF INTERVALS to 2 and animate.
4. The model should change its shape as shown in the following figure.

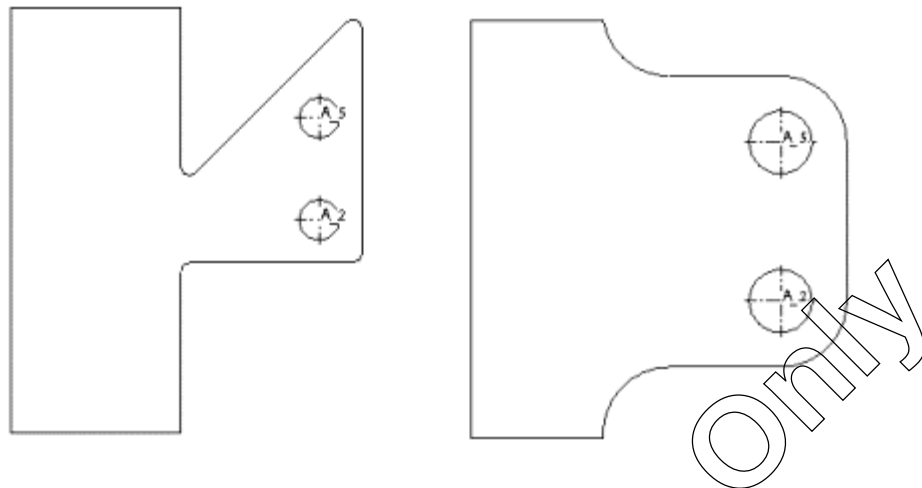


Figure 4: Shape animation on all design parameters

5. Restore the model to its original shape.

Exercise 2: Creating and Running Global Sensitivity Studies

Introduction

In this demonstration, we run a global sensitivity study on the parameter inner fillets. The results of this global sensitivity study will quantify the impact that inner fillets have on stress in your model.

Global sensitivity studies serve two primary purposes:

- *To determine how the variation of a parameter affects the design.*
- *To determine a good starting value for that parameter to use as an initial value in optimization.*

Objectives

After successfully completing this exercise, you will know how to:

- Create a global sensitivity study.
- Generate plot of measures vs. design parameters.

Scenario

In this exercise, you run a global sensitivity study on the parameter inner fillets. Change the inner fillet radius within its range (2.5 to 12.5 mm). All other parameters will stay at their current position. The results of this global sensitivity study will quantify the impact that inner fillets have on stress in your model.

Step 1. Create a global sensitivity study.

1. Create a new design study. From MEC STRUCT menu select **Analyses/Studies** then **File > New Design Study**.
2. In the Design Study Definition dialog box, select **Global Sensitivity** as type and edit the study name to **bracket_gs**.
3. Edit the description to **Sensitivity of the bracket to inner fillet radius size**.
4. Verify that BRACKET_STATIC (static) analysis is selected.
5. Select **inner_fillets** parameter and verify that the Start is **Minimum** and the End is **Maximum**.
6. Edit the number of intervals to **4**.
7. Select the Repeat P-Loop Convergence check box.
8. Click **Accept**.

Step 2. Run a global sensitivity study.

1. From the Analyses and Design Studies dialog box, select **Run > Settings**. Ensure that **Use Elements from an Existing Study** check box has been selected and the path is set to **C:\users\student\mechanica_330\module_02_03\results\bracket_opt**.
2. Click **OK** to close the RUN SETTINGS dialog box.
3. **Run** the global sensitivity study with error detection.
4. Check the summary file to monitor the study's progress.

Step 3. Create a results window to graph the von Mises stress vs. inner_fillet radius.

1. Create a result window. Select **Results** from MEC STRUCT menu.
2. Click **Insert > Result Window**, and do not save the model if prompted.
3. Edit the name to **vm_sens**.
4. Edit the directory for the study to **bracket_gs**.
5. Define the result window contents.
 - Edit the title to **Max von Mises Stress vs. inner fillets radius**.
 - In the QUANTITY tab, ensure that MEASURE is selected. Select **max_stress_vm** from the list of predefined measures.
 - In the GRAPH LOCATION section, click **inner_fillets** as the Design Var.
 - Click **OK and Show**. The results should look as shown in the following figure.

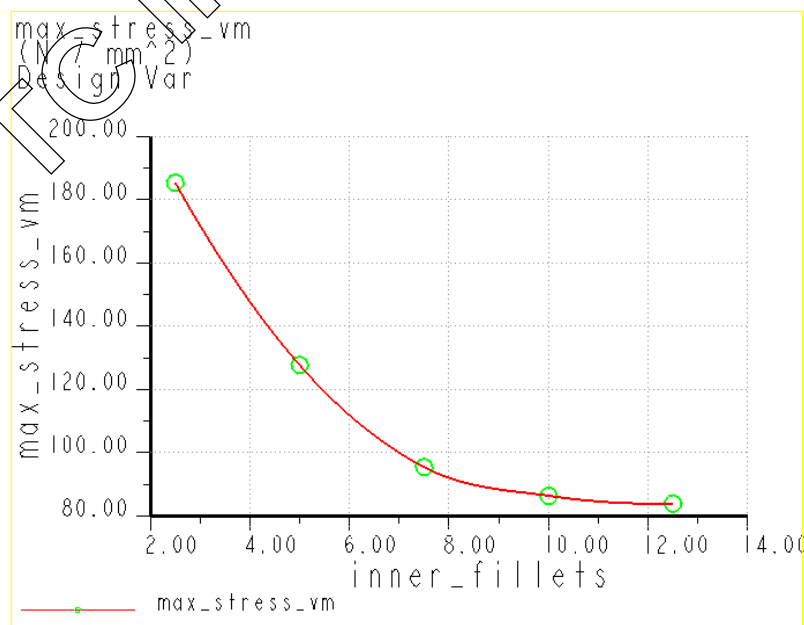


Figure 5: Max von Mises Stress vs. inner_fillets radius Graph

Step 4. Review and interpret the results of a global sensitivity study.

The curve quantifies the impact that inner fillets have on stress in your model.

1. Write down the inner fillet value that corresponds to the stress design constraint _____. This value can be used as the initial value of inner_fillets in the optimization.
2. Exit the result window. Do not save the result windows, when prompted.

For PTC Internal Use Only

Exercise 3: Optimizing the Bracket

Introduction

In this demonstration, we define an optimization design study. An optimization study adjusts one or more parameters to best achieve a specified goal or to test feasibility of a design, while maintaining specified limits. In the case of the bracket, the goal is to minimize the mass of the bracket while satisfying all of the imposed constraints.

Objectives

After successfully completing this exercise, you will know how to:

- Create and run an optimization study.
- Identify critical parameters.
- Set the convergence criteria.
- Update the model.

Scenario

In an optimization study, the solver tries to find a set of design parameter values within their specified ranges that maximizes or minimizes a measurable quantity while satisfying all of the imposed constraints.

There are several key elements that need to be defined in an optimization study.

- **Goal:** A goal is a certain measurable quantity of the model that will be maximized or minimized in an optimization design study.

In the case of the bracket, the goal is to minimize the mass of the bracket.

- **Optimization Constraints:** These are constraints that the model must satisfy.

In this case, the constraints are:

- The maximum von Mises stress must be less than 93 N/mm² (MPa).
- The maximum allowable displacement is 0.04 mm.

Step 1. Create the optimization design study.

1. Create another design study. From MEC STRUCT menu select **Analyses/Studies** then **File > New Design Study**.
2. In the Design Study Definition dialog box, select **Optimization** as type and edit the name to **bracket_opt**.
3. Edit the description to **Optimization study for the bracket**.

4. Edit the Goal to **Minimize** and the Measure to **total_mass**.
5. Create a limit on max_disp_mag and edit it to less than **0.04**.
6. Create another limit on max_stress_vm. Accept the default less than (<) sign and enter **93** as the limit value.

You can use the scroll bar in the PARAMETERS area to display all the parameters.

7. Define the variables, in the PARAMETERS section; select all seven buttons under Parameters.
 - Verify that the Min is set to **Minimum** and the Max is set to **Maximum** for all parameters.
 - Assign the initial parameter values. Enter the values listed in the following table.

Table 3: Parameter Range and Initial Values

Parameter	Min	Init	Max
inner_fillet	Minimum	3.75	Maximum
outer_fillet	Minimum	3.75	Maximum
tab_width	Minimum	50.00	Maximum
tab_top	Minimum	62.50	Maximum
tab_bottom	Minimum	62.50	Maximum
mid_curve	Minimum	50.00	Maximum
ang	Minimum	45	Maximum

8. Edit the OPTIM CONVERGENCE (%) to **2**. Click **Accept**.

Step 2. Create a result window to display the graph of von Mises Stress vs. Optimization Pass using the previous results.

Do not run the optimization study-it takes roughly 2 hours. Instead create a Result Window from already existing results. Create a Result Window, name it vm_history and navigate to C:\users\student\mechanica_330\module_02_03\results sub_directory for the bracket_opt result directory. Plot maximum von Mises stress as a function of the optimization pass.

Note:

Do not run the optimization as it may take several hours. In case you want to run the optimization, start the run at the end of the day.

1. Click Results in the MEC STRUCT menu. Do not save the model if prompted.
2. Create a new Result Window, and edit the name to **vm_history**.
3. Edit the title to **Max von Mises Stress vs. Optimization Pass**.
4. Navigate to the folder **C:\users\student\mechanica_330\module_02_03\results** and set **bracket_opt** as the Design Study.

5. Edit the Display type to **Graph**.
6. Edit the Quantity tab to **Measure**.
7. Measure **max_stress_vm**.
8. Click **OK and Show**.

Step 3. Create a Result Window to display the graph of mass vs. optimization pass using an existing window definition.

1. Create a new Result Window to plot the total mass. Edit the name to **mass_history**.
2. Edit the title to **Total Mass vs. Optimization Pass**.
3. Set **bracket_opt** as the Design Study.
4. Edit the Display type to **Graph**.
5. Edit the Quantity tab to **Measure**.
6. Measure **total_mass**.
7. Click **OK and Show**.

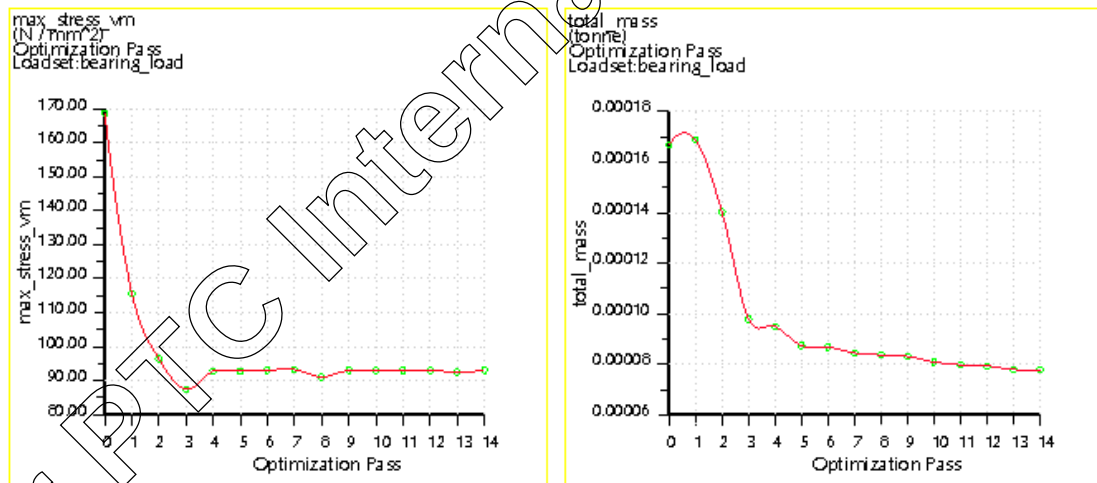


Figure 6: Optimization Graphs

8. Record the Initial mass _____.
9. Record Final mass _____.
10. Exit the result interface when you finish reviewing the result.
11. Save the result window and edit the name to **bracket_optimization**.

Step 4. Replay the optimization process and update the design.

1. Explore the optimization history. Set the working directory to **C:\users\student\mechanica_330\module_02_03\results**, where the optimization design study is located. Select **Model > Dsgn Controls > Optimize Hist > Search Study** and click on **bracket_opt**.
2. Press **Enter** when prompted to review the next step. Repeat to advance to the next step.
3. Update the design parameters so that the model remains at the optimized state. The model should look as shown in the following figure.

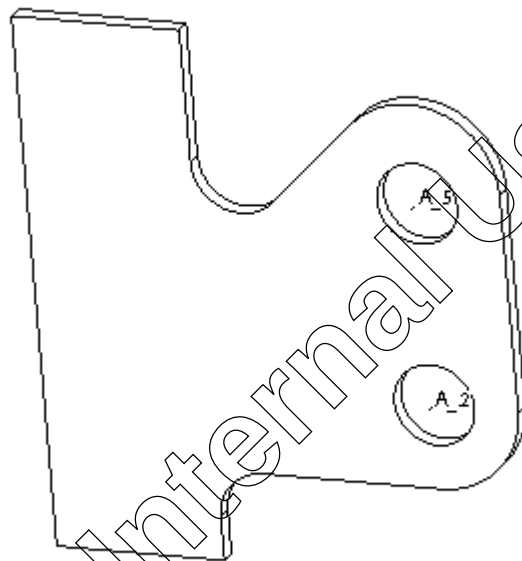


Figure 7: The optimized bracket model.

Step 5. Create a results window to show the von Mises Stress fringe plot of the optimized model.

1. Go to results. Do not save the model when prompted.
2. Create a window to display the stress and name it **vm_final**.
3. Edit the title to **von Mises Stress - Optimized Shape**.
4. Accept **bracket_opt** as the Design Study directory.
5. Ensure that the Display type is set to **Fringe**.
6. Ensure that **Stress** and **von Mises** are selected in the Quantity tab.
7. On the Display Options tab, ensure that the options **Continuous Tone** and **Average** are not selected.

8. Click Deformed and enter **10%**.
9. Edit the Animate number of frames to **16**.
10. Select the **Auto Start** and **Reverse** options.
11. Click **OK and Show**.
12. Record the Maximum von Mises stress _____.
13. Record the Maximum Displacement _____.
14. Calculate how much the mass has been reduced _____.
15. Exit the result interface when you finish reviewing the result. Save the result window when prompted and name it **final**.
16. Click **Applications > Standard**.
17. Save the model.
18. Close the window and erase all models from session.

For PTC Internal Use Only

Summary

After completing this module, you will be able to:

- Create design parameters.
- Create and run a global sensitivity study.
- Create and run an optimization study.
- Display results and optimized models.

For PTC Internal Use Only

Simplifying Designs Using Idealizations

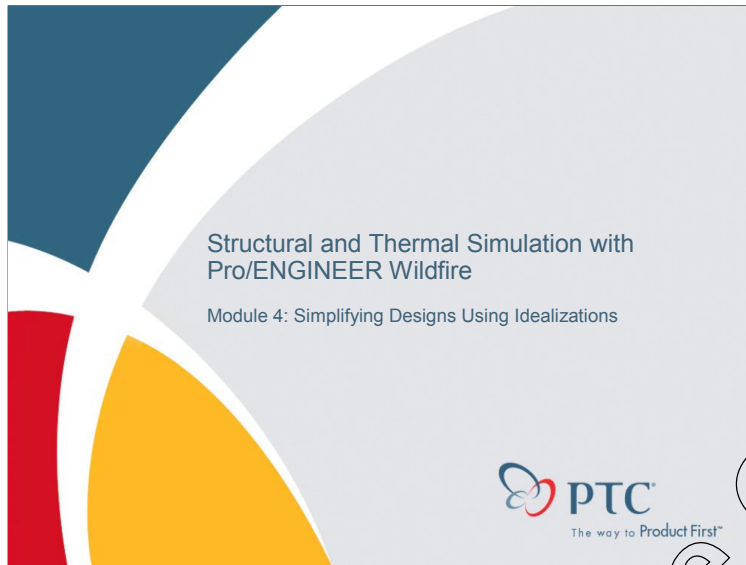
Introduction

Idealizations are mathematical approximations of the model geometry. Mechanica uses idealizations to simulate the behavior of the design. Mechanica calculates stress, strain and other values for each idealization or element. You can choose from several different types of idealizations to simplify designs when running analyses. In this module, you will examine several ways to idealize Mechanica elements.

Objectives

After completing this module, you will be able to:

- Describe the types and applications of idealizations.
- Simplify designs using idealization.
- Describe the purpose of rigid connections.
- Describe the purpose of end and perimeter welds.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

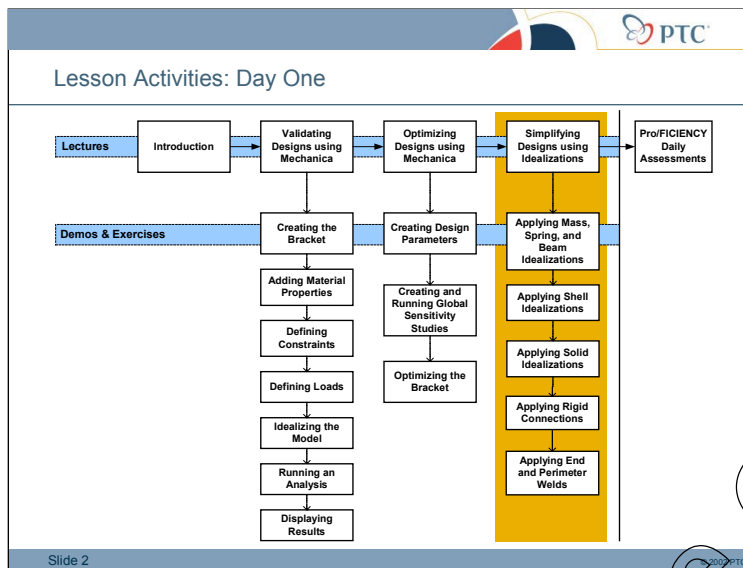
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 30 min
- Labs: 60 min
- Total: 1 hr 50 min



Objectives

After completing this module, you should be able to:

- Describe the types and applications of idealizations.
- Simplify designs using idealization.
- Describe the purpose of rigid connections.
- Describe the purpose of end and perimeter welds.

Slide 3

Objectives

Idealizations are mathematical approximations of the model geometry. Mechanica uses idealizations to simulate the behavior of the design. Mechanica calculates stress, strain and other values for each idealization or element. You can choose from several different types of idealizations to simplify designs when running analyses. In this module, you will examine several ways to idealize Mechanica elements.

There are several simplifications of idealizations of Mechanica elements. It is important to understand the various idealization types and how they affect the results you will get for any given analysis.

PTC

Geometry: Generating the Part

Geometry is the foundation of Mechanics models.

- Created in Pro/ENGINEER Wildfire.
- Imported from other CAD products.

Modeling geometry.

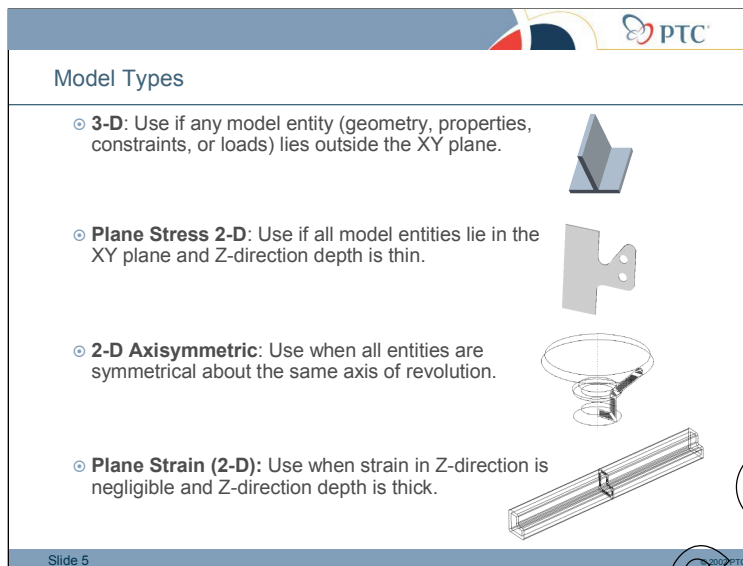
- Ease of modeling with Pro/ENGINEER Wildfire.
- Use recommended part generation techniques.

Slide 4

Geometry: Generating the Part

Decisions made early in generating parts with Pro/ENGINEER Wildfire will affect ease of modeling with Mechanics. The key is to plan ahead and to think in terms of analysis from the beginning. You want to generate “analysis friendly” parts. Here are some examples:

1. Selecting an appropriate **Base Feature** for your model can save you considerable amounts of time. For instance, thin protrusions allow automatic **Mid-surface Compression** for shell creation (an operation which must be done manually for other features).
2. Including certain features in your sketches instead of later in part mode will prevent you from easily performing **Feature Suppression**. External rounds (and any nonstructural feature) are generally suppressed before performing an analysis as they increase the number of elements in the mesh without affecting the results.
3. The **Dimensioning Scheme** will affect your ability to perform **Future Shape Changes** as well as **Optimization** and **Sensitivity Studies** on certain parameters of your part.
4. Managing **Relations** will also be important as they might affect parameters to be considered in the analysis.
5. **Datum Points** and **Curves** are useful to specify areas of local mesh refinements. They will also allow you to create **Regions** for precise application of **Loads** and **Constraints**.

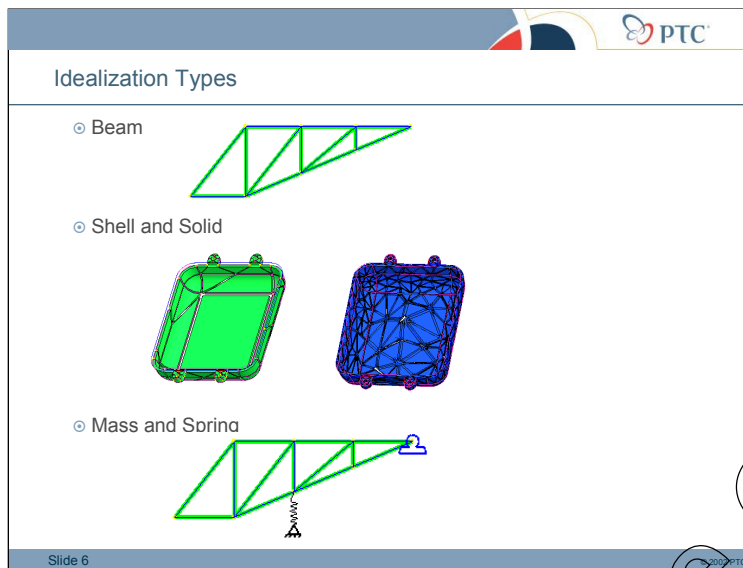


Model Types

- There are four model types available in Mechanical **Independent Mode** (3D, Plane Stress, Plane Strain and 2D Axisymmetric). The choice of model type significantly affects run times. Whenever possible, we want to take advantage of model types other than 3D.

Note: Independent Mode will not be used in this course.

- The **3D** model type is by far the one that requires the most computer resources. It is used when no simplification can be applied to the model. Note that this is the only model type available in Integrated Mode.
- The **2D-Plane Stress** model type is used for thin parts when all model entities (geometry, properties, constraints and loads) lie in one plane. (namely: no gradients exist in the transverse direction). In these cases, doing away with the third dimension will significantly decrease analysis times while accurately representing the physical situation
- The **2D Axisymmetric** model type, as its name implies, is used when all model entities (geometry, properties, constraints and loads) are symmetrical about an axis of revolution.
- The **2D-Plane Strain** model type is used for thick, extruded parts when all model entities (geometry, properties, constraints and loads) are identical for each cross-section of the structure (for example: dams, underwater pipes). Again, the third dimension in the model is ignored (as any cross-sectional result is valid for the entire structure) and run times are significantly reduced from the 3D representation.
- Elements, properties, loads, and constraints all depend on the model type selected. As a result, if the model type is changed, all that information is lost.



Idealization Types

“Idealizations” is the term used to designate Mechanical elements. This term puts emphasis on the fact that elements constitute mathematical representations of reality; in particular, we will review the assumptions tied to each type of element.

The following are the types of idealizations available along with most common uses:

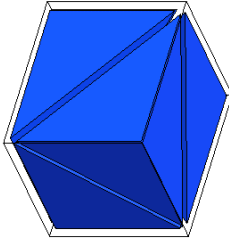
1. **Beam Elements** are represented as single lines in space. They are extremely economical from a computational point of view and therefore useful to model overall behavior of large structures (typically trusses, bridges and frames made of standard sections).
2. **Shell Elements** are represented as surfaces with no thickness. They are generally used to represent parts that are thin relative to their other dimensions (plates, shelled solids, pipes, pressure vessels, panels...). Also computationally economical when compared with 3D Elements.
3. **Solid Elements** can be used to model any type of structure or part, however they are the most computationally expensive elements
4. **Mass Elements** are used to represent any component which only contributes mass and no stiffness to a system, structure or part (for example: motor on a robotic arm). This is particularly useful for modal analysis.
5. **Spring Elements** are used to represent any component which contributes stiffness (or flexibility) but no mass to a system, structure or part (for example: driving belt). Spring elements are linear-elastic elements (double the applied force, you get double displacement), and can connect two points or a point to ground. Stiffness can be defined in translation (force per unit length) as well as torsion (torque).

When deciding on which elements to use, consider the structure that is being modeled and how that structure behaves, rather than the geometry. This will help in selecting the type of elements for your model.

PTC

Terminology Overview

- Geometric Entities**
 - Points
 - Curves
 - Surfaces
 - Volumes
- Element Entities**
 - Shells
 - Edges
 - Faces
 - Solids
- Integrated Entities**
 - Shell = Feature Type
 - Edges/Curves = Geometry
 - Faces/Surfaces = Geometry
 - Solids = Volumes



Slide 7

Terminology Overview

- Being familiar with terminology is critical when dealing with the Mechanica user interface.
- Terms used interchangeably in common language will mean different things in Mechanica. In particular, we need to distinguish between Geometric entities and Element entities.

As far as Integrated Mode is concerned:

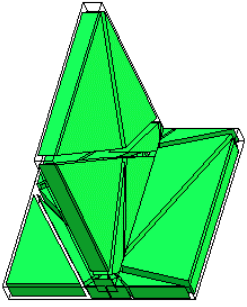
- A Shell is a feature type (not to be confused with shell elements of Mechanica).
- Edges/Curves are used to designate geometry.
- Faces/Surfaces are used to designate geometry.
- Solids and volumes become equivalent.

PTC

Simplifying Designs Using Idealizations

Using Shell Elements

- Constant shell thickness.
- Linear stress distribution through the thickness (t).
- Length to thickness ratio:
 - $10 < l/t < 1000$
- Shell radius of curvature (r):
 - $r > (2/3) * t$

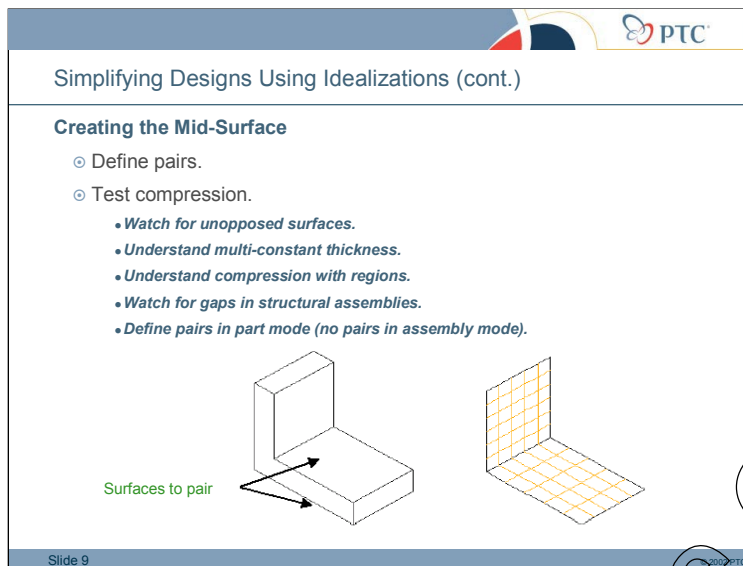


Slide 8

Using Shell Elements

Assumptions:

- Shell Elements are represented as surfaces with no thickness. They are used to represent parts that are thin relative to their other dimensions (plates, shelled solids, pipes, pressure vessels, panels)
- Shell element results are derived from classical plate theory. Again, you have to ensure underlying assumptions apply:
 - Shell thickness is constant throughout the element.
 - Bending, rather than shear is the dominant deformation mode. As a result, a shell element has a linear stress distribution through the thickness from tension at the top to compression at the bottom.
 - The length to thickness ratio must be greater than 10 and less than 1000. The lower bound (10) because thick parts deviate from the through-thickness linear stress distribution assumption (shear effects become significant), the upper bound (1000) because very thin parts must be handled with particular numerical formulations (to avoid ill-conditioned stiffness matrices).
 - The shell radius of curvature must satisfy $r > (2/3) * t$ (for example, sharp bends can not be handled with shells). Shell elements represent a 3D physical reality; from a geometrical point of view the smallest possible radius is $r = t/2$. However, for numerical results to be reliable a value of at least $2/3 * t$ must be used.
- You need curved underlying geometry to create curved shells.
- Mechanica allows you to mix elements created manually and elements created by AutoGEM. Node to node connectivity must be observed to avoid cracks in model (more on this in next lecture). In Integrated Mode, the user can specify where shell elements will be created on a part (at a mid-plane, on a face, on a region or on a datum plane).
- Pro/ENGINEER Wildfire users can hand mesh parts and surfaces by importing geometry.



Creating the Mid-Surface

To meet Mechanica's criteria for shell models, a part must have a constant or multi-constant thickness. In other words, a shell model can not be a tapered part, but a part can have multiple constant thickness areas. The thickness of the shell element is always distributed uniformly about the surface on which it is placed. For this reason, it is important that this surface actually represent the "mid-plane" of the model.

To go from 3D Pro/ENGINEER Wildfire geometry to 2D surfaces representing shell elements in Mechanica, the user must define pairs (or groups) of surfaces which will be compressed to a mid-plane surface.

This is a three step process:

- 1. Defining shell pairs** - define pairs of surfaces which will be compressed. A pair can consist of two or more surfaces on opposite sides of a volume.
- 2. Verify that the "Use Pairs" option is set** – (from the Pro/ENGINEER Wildfire menu, use Applications > Mechanica > Structure > Model > AutoGEM > Settings and check the "Use Pairs" option) Mechanica uses this setting to determine whether to perform solid or shell meshing.
- 3. Test Compression** – (from Structure Menu use Model>Idealizations>Shells>Compress). Use this option to compress the pairs of surfaces onto their mid-plane and preview the shell "interpretation" of your part. This is important as improperly paired surfaces will cause fatal errors in your analysis run.

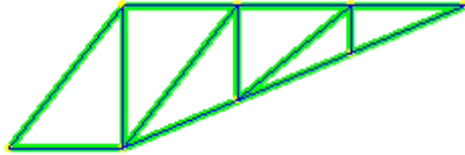
For complex parts, the pair definition process can become extremely tedious. This process can be automated using base features that Auto Pair (Model: Idealizations: Shells: Pairs: Auto Pairs). The following features will Auto Pair : Thin Protrusions (Pro/Surface Quilts), Shelled features, Sheetmetal features, Ribs and certain types of drafts (those having parallelograms as cross sections). Tapered solids can not be paired as shells have a constant thickness. In this sense, a general draft will not Auto Pair.

PTC

Simplifying Designs Using Idealizations (cont.)

Beam Elements

- Constant beam cross-section.
- Symmetrical cross-section, but there are Channel and "L" beams.
- Length to X-section ratio should be $\geq 10:1$.
- Results available are axial, bending, torsion, and total beam stresses.
- Must be planar (no corkscrews).



Slide 10

Using Beam Elements

Assumptions:

- A beam element is represented as a single line in space. Typically used for modeling large structures such as trusses, bridges and frames.
- Beam element results are derived from classical beam theory. It is the user's responsibility to ensure that the following assumptions apply when using beam elements:
 - The beam has a constant cross section.
 - The beam has a symmetric cross section and loads are applied along the plane of symmetry. Torsional effects can become important for highly un-symmetrical sections and are not well handled by classical beam theory. However, note that channels and "L" sections can be handled.
 - The length-to-width and length-to-depth ratios are greater or equal to 10:1. When this assumption is satisfied, the deformation developed in the beam is mainly due to bending stresses, as opposed to shear stresses.
 - Results include axial, bending, torsion, and total beam stresses. Note that shear stresses are not included. Shear effects can not be captured with beam elements.
 - The beam cross-section does not rotate about the neutral axis (no corkscrew).
- In Integrated Mode, the mesh is generated at run time by AutoGEM.

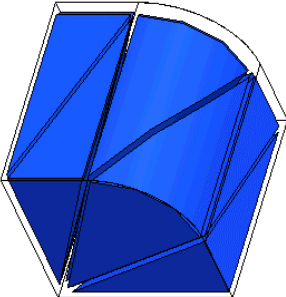
Note: You need curved underlying geometry to create curved beams.

PTC

Simplifying Designs Using Idealizations (cont.)

Solid Elements

- 30:1 Aspect ratio.
- Types:
 - **Brick**: six quad faces (8 points).
 - **Wedges**: two tri and three quad faces (6 points).
 - **Tetrahedral**: four tri faces (4 points).
- AutoGEM ensures:
 - Edge and face connectivity.
 - Solids do not overlap.
 - Adjacent elements have edge-to-edge connectivity.



Slide 11

Solid Elements

A solid model is a part that is modeled using solid elements like tetrahedral, bricks, or wedges. In Integrated Mode, Mechanica uses only tetrahedrals for solid modeling by default. Solid elements should be used when a part is “chunky,” or when a part’s width, length or thickness are not sufficiently small enough to assume that it can be modeled with a shell element.

- Being the most expensive in terms of run time, solid elements are used to model parts for which none of the beam, shell or 2D assumptions apply.
- *Assumptions*: For reliable results, solid elements with aspect ratios greater than 30:1 should not be used.
- The following types of solids can be created:
 - **Brick** (or hexahedron) with six quadrilateral faces and eight points.
 - **Wedge** (or pentahedron) with two triangular faces, three quadrilateral faces, and six points.
 - **Tet** (or tetrahedron) with four triangular faces and four points.
- In Integrated Mode, mesh generation occurs “behind the scenes”, and meshes are created using Tetrahedral elements only.
- Remember to turn off the setting of “Use Pairs”.

Guidelines for Creating Solids

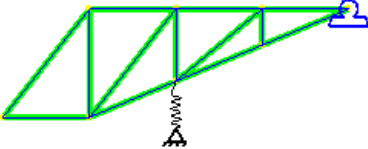
- To ensure good point, edge and face connectivity (namely, to avoid cracks in models), hand-meshed elements should not be mixed with AutoGEM elements.
- Solids must not overlap.
- Adjacent elements must have edge-to-edge connectivity. Any unmatched edge creates a crack in your model (discussed in the next lecture). Divide large arc spans into separate segments by placing points along them or by splitting them before selecting them as edges.

PTC

Simplifying Designs Using Idealizations (cont.)

Mass and Spring Elements

- Mass Elements
 - **Point Mass**
 - Simple
 - Advanced
- Spring Elements
 - **References**
 - Point to Point
 - Point to Point Pairs
 - Point to Surface
 - Point to Edge
 - **Types**
 - Simple
 - Advanced
 - Point to Ground



The diagram shows a truss structure with green members. A spring element, represented by a coiled line, connects a point on the truss to a fixed support (ground). A point mass, represented by a blue circle, is attached to the right end of the truss.

Slide 12

Mass and Spring Elements

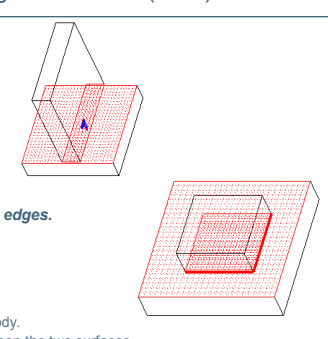
- Mass - Used to represent a concentrated or point mass without a specified shape (Structure only).
- Spring - Used to represent a linear elastic (six degrees-of-freedom) spring connection (Structure only).

PTC

Simplifying Designs Using Idealizations (cont.)

Connections

- End Welds
 - Requires 2 surfaces.
- Perimeter Welds
 - Requires 2 surfaces and edges.
- Spot Welds
 - Requirements:
 - One surface on each body.
 - A gap has to exist between the two surfaces.
 - A datum point for each spot weld location.
 - Diameter of the weld.
 - Material of the weld.



Slide 13

Creating Connections

A connection is the point of contact between two or more parts or subassemblies. In Mechanics, two kinds of connections can be used—end welds and perimeter welds.

End Welds

End welds can be used in assembly models to connect plates. The plates may be curved and placed at oblique or right angles, such as T or L configurations. By using the end weld, the shell mesh from one plate is extended to meet the mesh from the base plate.

End welds can be used to join:

- Two thin wall components at a right angle.
- Two thin wall components at an oblique angle.
- Two offset thin wall components mated at a right angle with a gap between the compressed surfaces of the components.

Two offset thin wall components mated at an oblique angle with no contact between the components.

Perimeter Welds

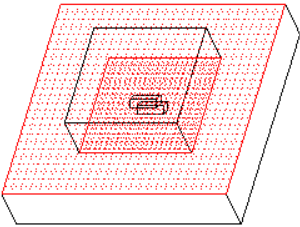
Perimeter welds can be used in assembly models to connect parallel plates, which may be curved, along the perimeter of one of the plates. During mesh generation, a series of surfaces is automatically created to connect the selected edges of the top plate to the base plate. Mechanical creates shell elements on the selected surfaces. A series of welds on one or more of the perimeter edges of the top plate establishes a connection to the base plate. The resulting compressed surfaces, however, will still be parallel to one another and will not touch.

PTC

Simplifying Designs Using Idealizations (cont.)

Creating Rigid Connections

- Requirements
 - Points, curves and surfaces.
 - Impose equal displacement conditions on the reference.
 - Identical to placing an infinitely stiff spring between the references.



Slide 14

© 2002 PTC

Creating Rigid Connections

A rigid connection connects geometric entities, such as surfaces, curves and points, so that they remain rigidly connected during an analysis. When entities are connected in this way:

- They move together as if part of a single rigid body.
- They do not deform, but the rigid body can move as a whole.

Because Mechanica uses linear constraint equations to enforce the rigid rotations, rather than equations with trigonometric functions (such as sine and cosine), rigid connections should only be used for small rotation angles of rigidly connected entities. Rigid connections should always be used in this way, even when modeling for a large deformation analysis. Mechanica supports rigid connections for 3-D models only.

PTC

Simplifying Designs Using Idealizations (cont.)

AutoGEM Limits Settings

- ⊙ Allowable Edge Angle – Angle created between any two element edges.
 - Shells: 25 min – 150 max
 - Solids: 5 min – 175 max
- ⊙ Allowable Edge Turn – Maximum arc which an edge can span.
 - Shells: 80
 - Solids: 95
- ⊙ Allowable Aspect Ratio – The longest edge divided by the shortest edge.
 - Shells: 10
 - Solids: 30
- ⊙ Allowable Face Angle – Minimum and maximum angle between any two adjacent element faces.
 - Solids 5 min – 175 max

Slide 15

AutoGEM

This is the automatic element generation. Underlying geometry is required. Note that in Integrated Mode, the mesh is generated by AutoGEM for the user at runtime.

Limits

- The global density of the mesh can be controlled using the AutoGEM Element Limit Settings:
 - The Allowable Edge Angle defines the minimum and maximum values for the angle between two edges. Bringing these values closer together creates smaller elements (mesh will be denser).
 - The Allowable Face Angle defines the minimum and maximum value for the angle between two faces in a solid. Again, bringing these values closer together will create more and smaller elements.
 - The Max Allowable Edge Turn: Geometric Elements map geometry exactly. Therefore elements will span arcs; this setting defines the maximum arc which can be spanned by an element edge. A small value will create more elements.
 - The Max Allowable Aspect Ratio: The aspect ratio of an element is defined as the ratio of its longest edge to its shortest. A square has an aspect ratio of 1. A smaller aspect ratio will create more elements for the same geometry.
 - Note that changing these settings will greatly influence your run times as well as convergence characteristics (more on this when we look at convergence).
 - These element settings are accessible in Integrated Mode under *Mechanica > Structure > Mode > AutoGEM > Settings > AutoGEM > Limits*.
 - Note that the default values are optimized for solid elements and will therefore necessitate changes for most shell models.
- Defaults based on much testing and validation.
- Changing the settings may result in longer run times.
- Poorly shaped elements may go to higher P-levels.
- Element Limits form can be changed - AGEM Settings: Define/Review Element Limits.



Demonstrations

- ◉ Using Mass, Spring, and Beam Idealizations.
- ◉ Using Shell Idealizations.
- ◉ Using Solid Idealizations.
- ◉ Using Rigid Connections.
- ◉ Using End and Perimeter Welds.

Slide 16

Demonstrations

In this demonstration, I will

- Create beam, mass, spring, shell, and solid elements.
- Simplify a design using shell idealizations.
- Define rigid connections.
- Define end and perimeter welds.

You will repeat the same steps in the lab exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 17

Precision Learning

- Learn: by listening to lectures, watching demos, and completing lab exercises.
- Assess: your progress with Pro/FICIENCY assessment questions.
- Improve: The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Note to Instructors:

Getting Started

- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Exams

For each course, Pro/FICIENCY exams are based upon the topics covered each day.

How to use it

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After completing this module, you should be able to:

- Describe the types and applications of idealizations.
- Simplify designs using idealization.
- Describe the purpose of rigid connections.
- Describe the purpose of end and perimeter welds.

Slide 18

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 4 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Applying Mass, Spring, and Beam Idealizations

Introduction

In this demonstration, we examine mass, spring, and beam idealizations. The model we will be working on is a truss. Since we will be defining the beam section properties (I-beam, solid circle, etc.) in Mechanica, all we need to create for geometry is the datum curve framework to set up beam idealizations.

Objectives

After successfully completing this exercise, you will know how to:

- Create beam, mass and spring elements.
- Assign beam properties to a curve.
- Assign mass and spring properties.

Scenario

Due the size of the models, the solution of a structural or a thermal analysis may be time consuming. Simplification of elements is helpful, at times necessary. If one tried to model large structures, beam elements could be considered. If only the mass or the stiffness of a system are important, mass and spring elements can be used. In this exercise, you examine mass, spring, and beam idealizations. The model you will be working on is a truss. Since you will be defining the beam section properties (I-beam, solid circle, etc.) in Mechanica, all you need to create for geometry is the datum curve framework to set up beam idealizations. For the mass and the spring, datum points are required.

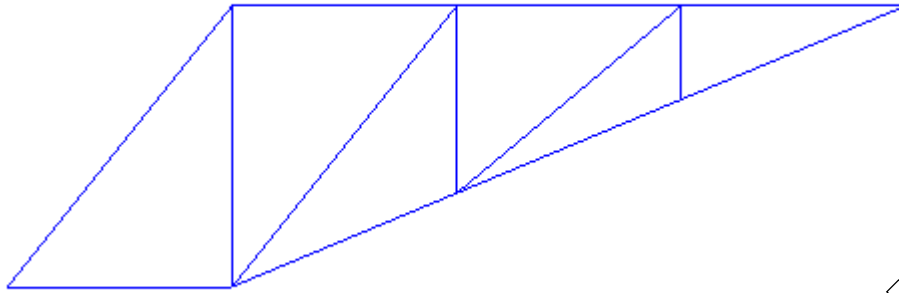


Figure 1: A truss structure made up of datum curves.

Step 1. Create the truss structure.

Create a new part called **truss** with the default template. Change the units from the Pro/ENGINEER Wildfire default to **mmNs**.

Create a sketched datum curve as shown in the figure below. Select the FRONT as the sketching plane and TOP datum plane for orientation.

Note:

If you have problems creating the truss, you can retrieve a completed truss model from C:\users\student\mechanica_330\module_04. Open TRUSS_COMPLETED.PRT.

1. Set your working directory to **C:\users\student\mechanica_330\module_04**.
2. Create a new part called **truss**. Leave the Use Default Template check box selected. The new part will have three default datum planes and a csys.
3. Turn off the spin center and bell if necessary.
4. Change units from the Pro/ENGINEER Wildfire default to **mmNs**. Select **Edit > Setup > Units**.
5. Create a sketched datum curve, as shown in the following figure. Select the FRONT as the sketching plane and TOP as a Top reference.

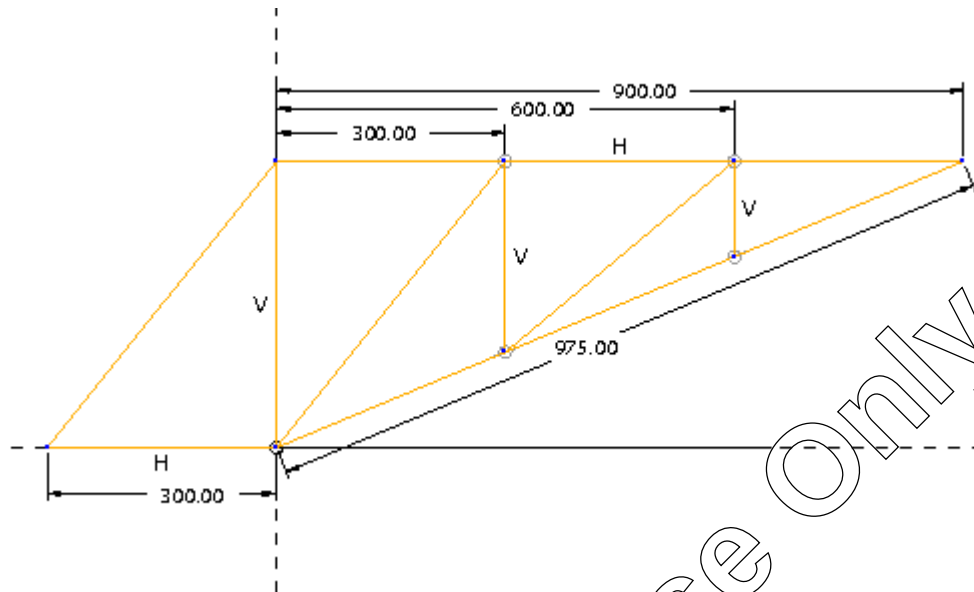


Figure 2: Sketch the datum curves.

Note:

If you have problems creating the truss, you can retrieve a completed truss from C:\users\student\mechanica_330\module_04. Open TRUSS_COMPLETE.PRT.

Step 2. Create the beams.

Enter **Mechanica Structure** and add a beam. For References select **Edge/Curve** and pick all curves in the model for beam references. Assign **Steel** as to the material. For the **Y** direction, select **Vector in WCS** and enter **0, 0, 1** respectively for X, Y and Z directions and for the **Section**, add a **Hollow Circle** with an outer radius of **12.5** and inner radius of **8.75**.

1. Enable **Mechanica Structure** mode. Notice the unit info window.
2. Add a new beam named **hollow_tube**. From **MEC STRUCT** menu, select **Idealizations > Beams > New**.
3. For References, select **Edge/Curve** and pick all curves in the model for beam references.
4. For material, select **More** and assign **STEEL** to the model.
5. For the **Y** direction, select **Vector in WCS** and enter **0, 0, 1** respectively for X, Y and Z directions.
6. For the **Section**, click **More** to add a section.
7. Click **New** to add a Beam Section named **Hollow_Circle**. Set the section type to **Hollow Circle** with an outer radius of **12.5** and inner radius of **8.75**.

8. When finished, icons appear at intervals along each beam, as shown in the following figure.
This gives you a visual idea of the size and shape of the hollow tubing.

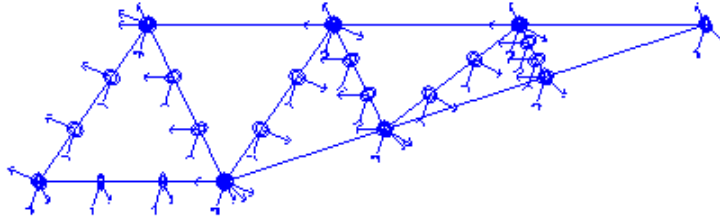


Figure 3: Truss model with beams.

Step 3. Attach a weight (mass) to the tip of the truss and model a flexible support (spring) attached at the center of the truss and the ground.

Create two datum points, PNT0 and PNT1, as shown below.

Create a mass on PNT0. Enter **100** as the mass.

Create a spring between PNT1 and the ground. For the spring properties, enter **10**, **1000**, **10** for Kxx, Kyy and Kzz respectively and accept other defaults.

Note:

The Mass and Spring idealizations are placed at datum points on your model. You will create a datum point in Mechanica.

1. Switch to the front view and create a datum point PNT0 at the right most vertex of your model by selecting **Features > Datum Point > Create**.
2. Create another datum point PNT1 on a vertex as shown in the following figure.

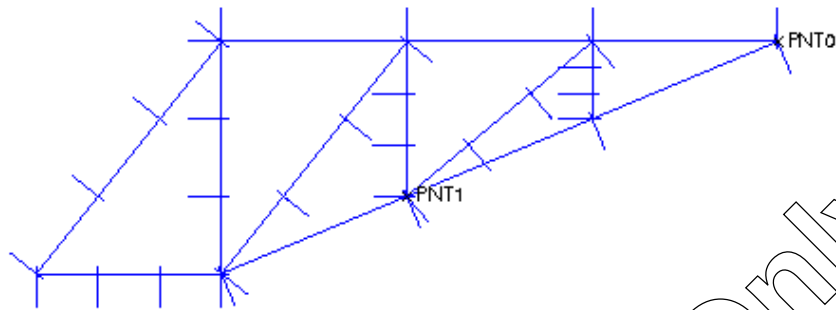


Figure 4: Truss Model

3. Create a mass on PNT0 you just created. Select **Idealizations > Masses > New**.
4. Pick PNT0 for reference and **Simple** for type. Enter **100** as the mass property. The Mass icon should appear.
5. Create a spring between PNT1 and the ground. Select **Idealizations > Springs > New**.
6. Select **To Ground** as type and PNT1 as reference. For the spring Properties, click on **More** and **New** and enter **10, 1000, 10** for Kxx, Kyy and Kzz respectively and accept other defaults.

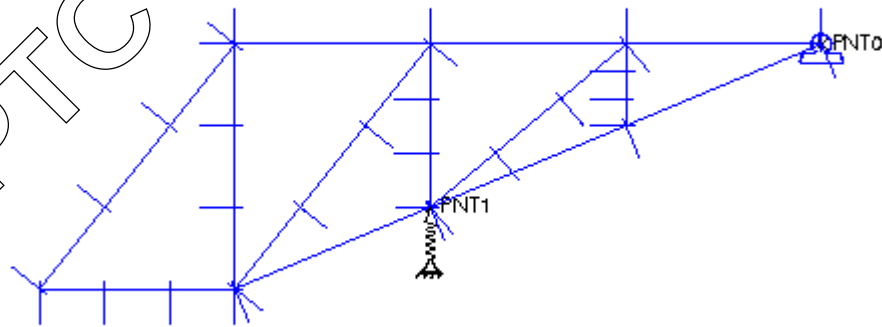


Figure 5: The truss model with beams, mass and spring idealizations.

Step 4. Hide the idealization icon display.

1. You can hide the idealization icons from the simulation display. Select **View > Simulation Display** and pick **Modeling Entities** tab.

2. Hide beam sections, masses and springs icons. The model is now ready for applying load and defining analyses.
3. Save the model.
4. Close the window and erase all models from session.

Tips & Techniques

Items, such as Loads, Constraints, and Simulation features are displayed together with Pro/ENGINEER Wildfire features in the Model Tree. You can use the Model Tree as a “shortcut” to manipulate these items. Right-click an object in the Model Tree. A pop-up menu will appear. You can use the pop-up menu to manipulate these items.

Exercise 2: Applying Shell Idealizations

Objectives

After successfully completing this exercise, you will know how to:

- Create shell features.
- Create shell pairs and perform a mid-surface compression.
- Create shell elements using Auto GEM.

Scenario

In this exercise you will explore the use of shell elements as an idealization. Shell elements are created on a mid-surface that is defined by the shell feature. You will be creating the pipes part shown below.

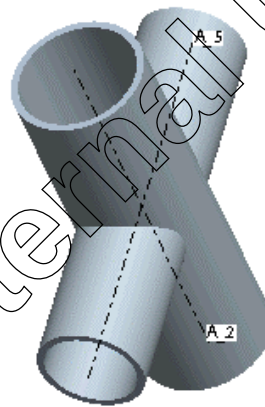


Figure 6: The finished pipes part.

Step 1: Create two intersecting pipes.

Note:

If you have problems creating the pipes, you can retrieve a completed pipes model from C:\users\student\mechanica_330\module_04. Open PIPES_COMPLETED.PRT.

1. Create a new part called **PIPES** using the default template.
2. Change units from the Pro/Engineer Wildfire default to **mmNs**.
3. Create a solid protrusion to represent the first pipe. Select TOP datum plane for sketching plane and RIGHT datum plane as the right orientation reference.

4. Sketch a circle that is **250 mm** in diameter. Extrude with a blind depth, for a depth enter **750** mm.
5. Change the default attribute to extrude the protrusion symmetrically on both sides.
6. Orient to Standard Orientation.

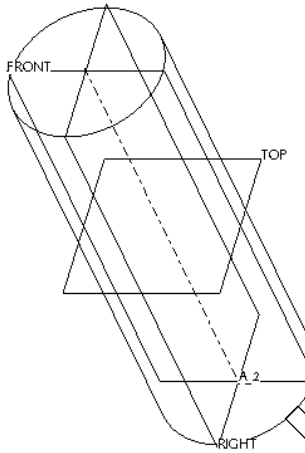


Figure 7: Finished Base Protrusion

7. Repeat the preceding procedures to create a second solid protrusion to represent the second pipe. Use FRONT as the sketching plane and use RIGHT as a Right references.
8. Sketch a circle that is **200 mm** diameter. Extrude both sides symmetrically to a blind depth of **750** mm. When finished, the model should appear as shown in the following figure.
9. Orient to Standard Orientation.

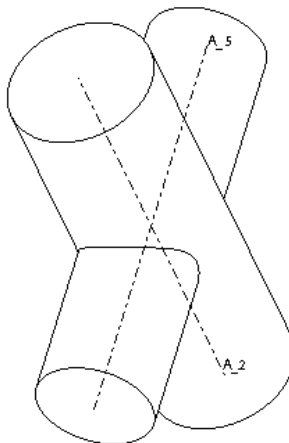


Figure 8: PIPES.PRT

10. Using the Shell Tool, shell the pipes with a thickness of **6 . 25**. Select the four end surfaces to remove, as shown in the following figure.

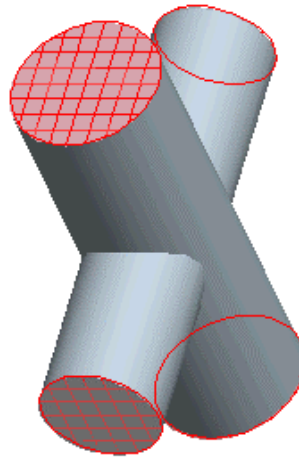


Figure 9: Unshelled PIPES.PRT.

11. The model should look as shown in the following figure. Save the model.

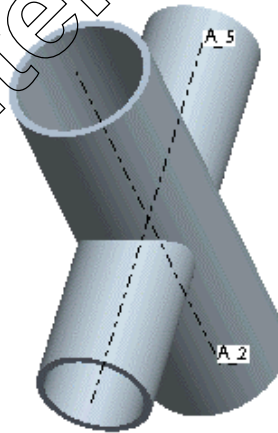


Figure 10: Completed PIPES.PRT

Step 2. Create the mid-surface of the shell model.

1. Enable Mechanica Structure mode.
2. Turn off the datum planes, axes, and coordinate systems and shade the model.

3. Determine the shells automatically by selecting **Idealizations > Shells> Midsurfaces > Auto Detect**.

Note:

Mechanica highlights one set of surfaces in red and the opposing surfaces in yellow. The red surface acts as a point of reference for the pair. Mechanica uses this surface as a viewpoint when determining which opposing surfaces in the model is part of the pair.

4. Test the shells by compressing the model and view the shell model. The pairs get compressed and displayed as shown in the following figure.



Figure 11: Compressed PIPES.PRT

Step 3. Assign the material steel to the PIPES.PRT.

Note:

Before running Auto GEM, material properties are required.

1. Assign the material **STEEL** to PIPES.PRT.

Step 4. Create and view the elements using Auto GEM.

Note:

In Mechanica, you create and view the element mesh in integrated mode. This is NOT a required step. Normally, you will set up an analysis and then run it without creating the elements. When you run an analysis in Integrated Mode, Auto GEM will create the elements.

1. Change the model view to the Standard Orientation.

2. Adjust the element display settings from the display menu. Select **View > Simulation Display** and pick **Mesh** tab. Set Shrink Elements to **20%**.
3. From the MECHANICA menu, create the elements using Auto GEM. Select **Model > AutoGEM > Settings**.
4. Ensure that the **Use Pairs** setting is turned on. It should be on as a result of shell idealization. This setting tells Mechanica to place shell elements on the mid-surfaces you just defined. If you want to model this part as a solid instead, you would turn off this setting by clearing the checkbox.
5. Select **Create** and read the SUMMARY window.
6. Visually examine the model as shown in the following figure.
7. Save the mesh.

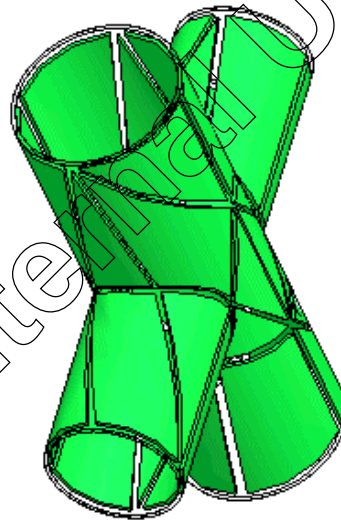


Figure 12: The meshed PIPES.PRT.

8. Save the part.
9. Close the window and erase all models from session.

Exercise 3: Applying Solid Idealizations

Objectives

After successfully completing this exercise, you will know how to:

- Create solid elements in integrated mode using Auto GEM.
- Create elements in independent mode using Auto GEM.
- Hand mesh in independent mode.

Scenario

In this exercise, you will create solid elements in integrated and independent mode. You will also perform hand meshing in independent mode. Solid elements are created on the volume of the part. First, you will create a T_BRACKET part shown in the following figure.

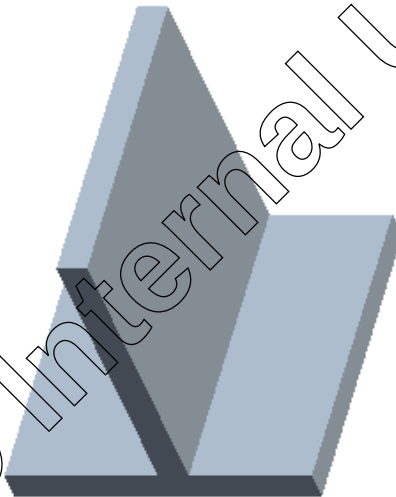


Figure 13: Finished Bracket Geometry

Step 1. Create a new part T_BRACKET.PRT and create the base feature.

Note:

If you have problems creating the t-bracket, you can retrieve a completed t-bracket from C:\users\student\mechanica_330\module_04. Open T_BRACKET_COMPLETED.PRT.

1. Create a new part called T_BRACKET. Use the default template.
2. Change the units to **mmNs**.
3. Create the base feature of T_BRACKET.PRT using a Thin protrusion. Select the FRONT datum plane as the sketching plane and RIGHT datum plane as a Right reference.

4. Sketch the T_BRACKET.PRT base feature as shown in the following figure. Since the base is symmetrical about the RIGHT datum plane, use a vertical centerline.

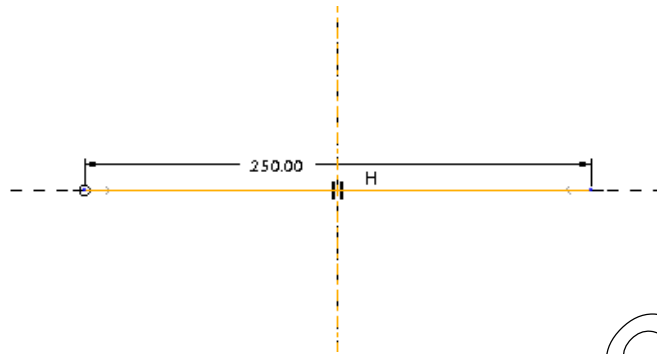


Figure 14: Sketch a horizontal line symmetric on FRONT.

5. Extrude to a symmetric depth of 250.
6. Enter **25** for the thickness.
7. Complete the feature.

Step 2. Create the second protrusion for T_BRACKET.PRT.

1. Create a Thin protrusion, select the FRONT datum plane as the sketching plane and RIGHT datum plane as a Right reference.
2. Sketch the section as shown in the following figure.

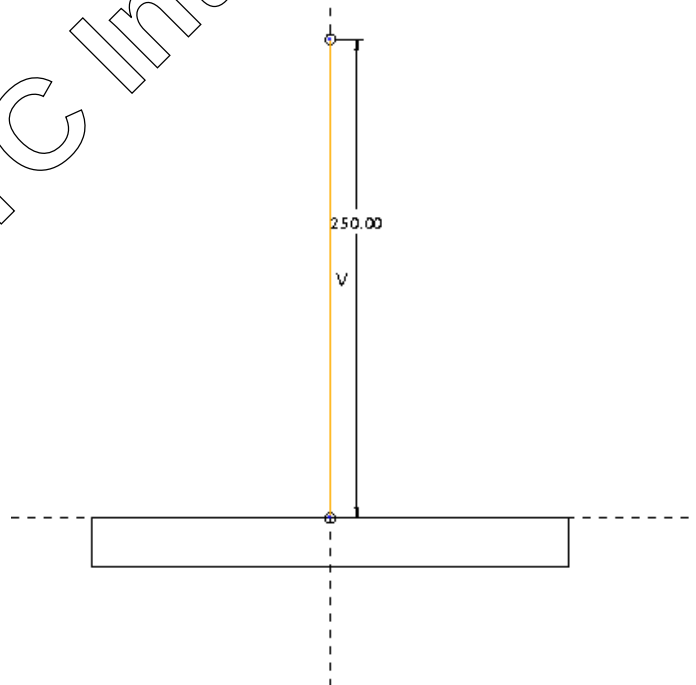


Figure 15: Sketch a vertical line on FRONT.

3. Extrude to a symmetric depth of 250.
4. Select the Thicken Sketch option and enter **25** for the thickness.
5. Complete the feature and save the file.

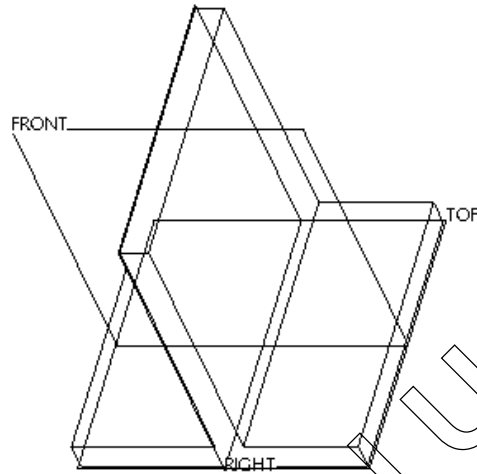


Figure 16: Finished Bracket Geometry.

Step 3. Specify the material properties of the bracket part.

1. Enable Mechanical Structure mode.
2. Assign the material **STEEL** to the bracket part.

Step 4. Create and view the elements using AutoGEM, in Mechanical.

1. Orient to the Standard Orientation.
2. Adjust the element display settings from the Display menu. Smooth shade the elements and Shrink Elements to **20%**.
3. From the Mechanical menu, create the elements using **AutoGEM**.
4. Read the SUMMARY window. Note the types of entities created.
5. Visually examine the model. The model should look like the following figure.
6. Save the mesh.

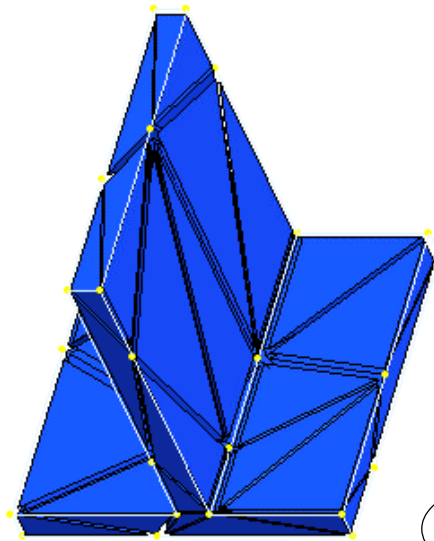


Figure 17: Bracket with solid elements.

7. Save the model.
8. Close the window and erase all models from session.

Exercise 4: Applying Rigid Connections

Objectives

After successfully completing this exercise, you will know how to:

- Create rigid connections.

Scenario

In this exercise, you will retrieve a container assembly shown below and define rigid connections. Rigid connections are similar to boundary conditions; they prescribe same displacement on the geometry selected.

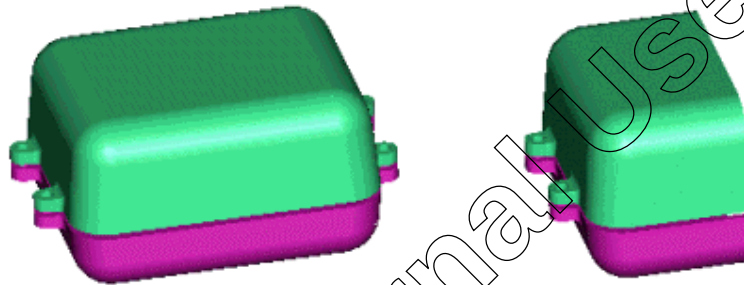


Figure 18: A symmetric plastic container is cut in half.

Step 1. Create a rigid connection on assembly RIGID_CONNECTION.ASM.

1. Open the RIGID_CONNECTION.ASM. The model is symmetric. All external load and constraints are also symmetric about the same center plane.
2. Enable Mechanical Structure mode. Notice the units.
3. Create a rigid connection. Select **Model > Connections > Rigid Connections > Create**.
4. Click the Select icon under Points, Curves, Surfaces. Select both halves of the two hole surfaces as shown in the following figure.

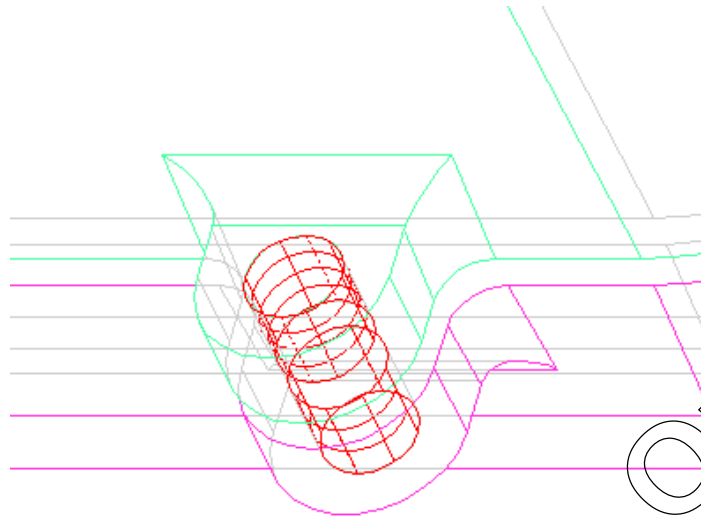


Figure 19: Select the indicated surfaces to create the rigid connection.

5. Repeat the procedure on the other pair of tabs. The system displays the Rigid Connection icons as shown in the following figure.

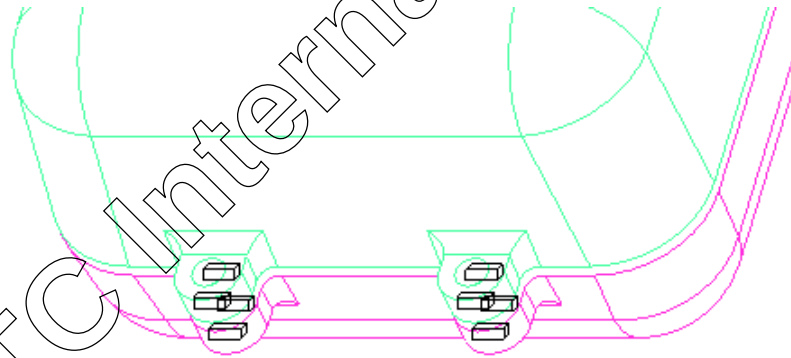


Figure 20: The rigid connection icon.

6. Save the model.
7. Close the window and erase all models from session.

Exercise 5: Applying End and Perimeter Welds

Objectives

After successfully completing this exercise, you will know how to:

- Create end weld.
- Create perimeter welds.

Scenario

In additions to rigid connections, weld connections can be created. In Mechanica, you can use two kinds of weld connections—end welds and perimeter welds.

- End Welds: Use end welds in assembly models to connect plates. Plates can be curved and placed at oblique or right angles, such as T or L configurations. Using the end weld, the shell mesh from one plate is extended to meet the mesh from the base plate.
- Perimeter Welds: Use perimeter welds in assembly models to connect parallel plates, which may be curved, along the perimeter of another plate. During mesh generation, a sequence of surfaces is automatically created to connect the selected edges of the top plate to the base plate. Mechanica creates shell elements on the selected surfaces. A series of welds on one or more of the perimeter edges of the top plate connects it to the base plate. In this case, the components are touching. The resulting compressed surfaces, however, are parallel to one another and do not touch. For this type of geometry, you should use a perimeter weld to connect the two plates.

In this exercise, you will retrieve a frame assembly and create and end weld and perimeter weld.

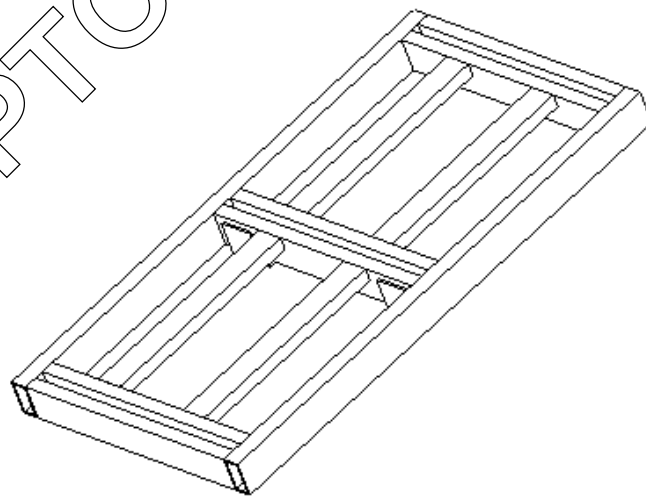


Figure 21: FRAME.ASM.

Step 1. Retrieve FRAME.ASM and create an assembly cut.

1. Open FRAME.ASM. Due to model symmetry, the analysis can be performed on one quarter of the model.
2. Create a cut. Specify ASM_TOP datum plane as the sketching plane, and ASM_RIGHT datum plane as the RIGHT reference plane. Use the Search tool to select the planes if necessary.
3. Specify the appropriate references and sketch the section, as shown in the following figure.

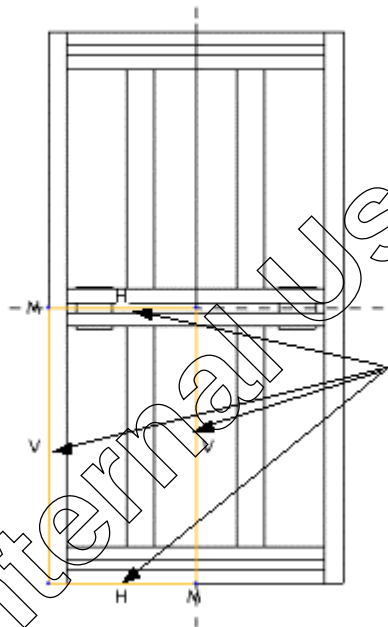


Figure 22: Sketch the cut section.

4. Set the cut to remove all material on both sides.
5. Remove the geometry outside of the rectangular section.

Step 2. Compress the mid-surface of the assembly in the assembly mode.

1. Enable Mechanica Structure mode.
2. Go to **Idealizations** and **Compress** the mid-surface.
3. The system highlights the MID_BAR.PRT and informs you that the shell model is not defined in this component.
4. Click **yes** to show the compressed surface. The system displays the shell models except the MID_BAR.PRT.

Note:

The shell models have been created in all components except MID_BAR.PRT. In assembly mode, you can compress shell models that have been previously created in part mode, however, you cannot create shells in the assembly mode.

Step 3. Create the mid-surface of the shell model in part mode.

1. Enable Standard mode.
2. Open MID_BAR.PRT.
3. Enable Mechanica Structure mode.
4. Use Idealizations to create a shell pair using auto detect and mid-surface.
5. Ensure the mid-surface is created by showing the compressed part.
6. Close the window.

Step 4. Compress the mid-surface of the assembly in the ASSEMBLY mode again.

1. Enable Mechanica Structure.
2. Test the shell compression again. Because all the shell models have been created in all the components in the part mode, the system compresses and shows all components.
3. Reorient to the TOP view and zoom in on the area indicated in the following figure. Notice that there is a gap between the MID_BAR shell and the CROSS_BAR shell.

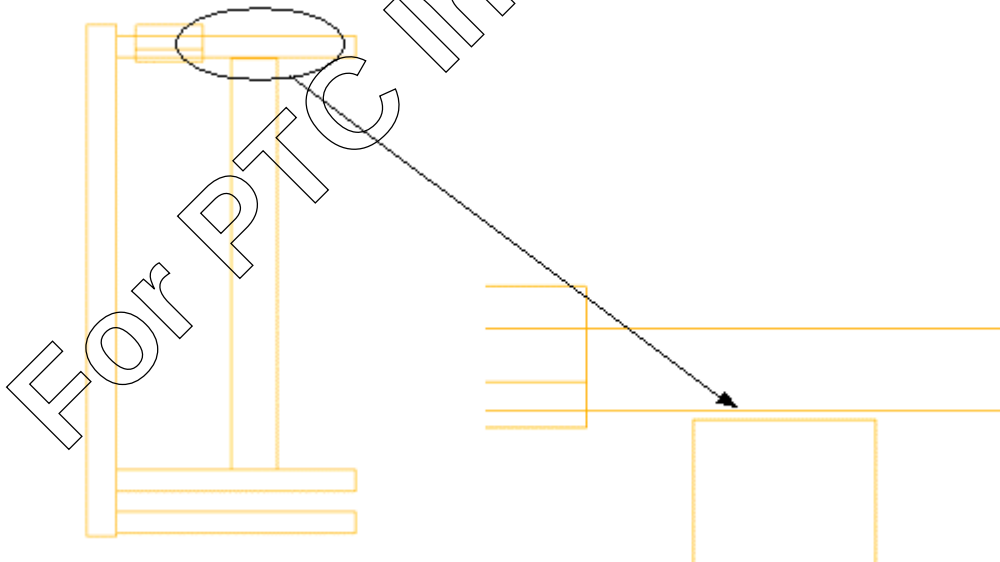


Figure 23: The gap between two compressed shell models.

Step 5. Create end welds.

1. Reorient to the Standard Orientation, and zoom in on the area indicated by the circle in the following figure.
2. Create end welds at one end of a sheet metal part, as shown in the following figure. Most end welds have been created for your convenience.

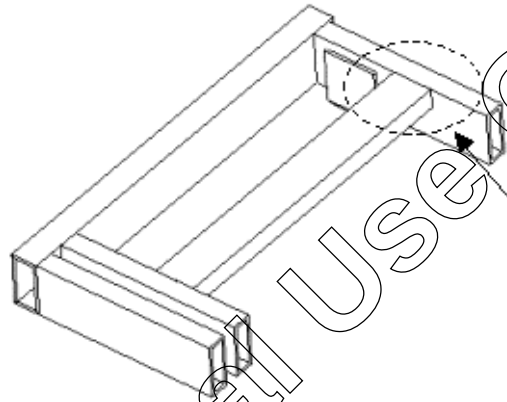


Figure 24: Area for end welds.

3. Create an End Weld connection. Select **Connections > Welds > End Welds > New**.
4. Specify the end weld surfaces. Select the hidden end surface of the MID_BAR.PRT. Then select the other surface, as shown in the following figure.

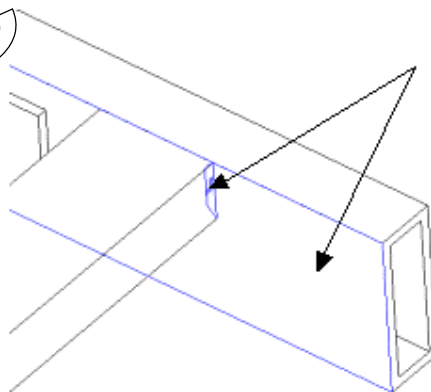


Figure 25: End weld surfaces.

5. Create 3 more end welds on the end of the MID_BAR.PRT, using the same procedures. The system displays the weld icons as shown in the following figure.

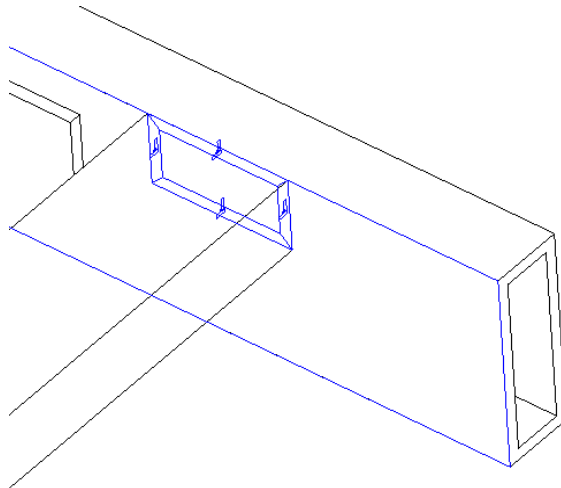


Figure 26: End welds at end of sheet metal part.

Step 6. Compress the mid-surface of the assembly in the assembly mode, to verify the end weld connections.

1. From the Idealizations menu, compress the assembly.
2. Orient to the RIGHT view.
3. Notice that the MID_BAR shell and the CROSS_BAR shell are now connected. However, if you change your view to RIGHT there is a gap between BRACKET.PRT and CROSS_BAR.PRT, as shown in the following figure.

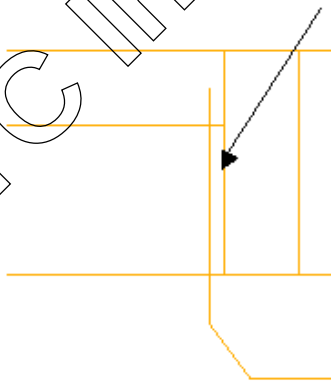


Figure 27: Gap between the BRACKET.PRT and the CROSS_BAR.PRT.

Step 7. Create a perimeter weld.

1. Create a perimeter weld connection by clicking **Connections > Welds > Perim Welds** then **New**.
2. Select the hidden surface of BRACKET.PRT as the doubler surface.

3. Select the three edges of the doubler surface to define weld location.
4. Select the shown surface of CROSS_BAR.PRT as the base surface, as shown in the figure below.

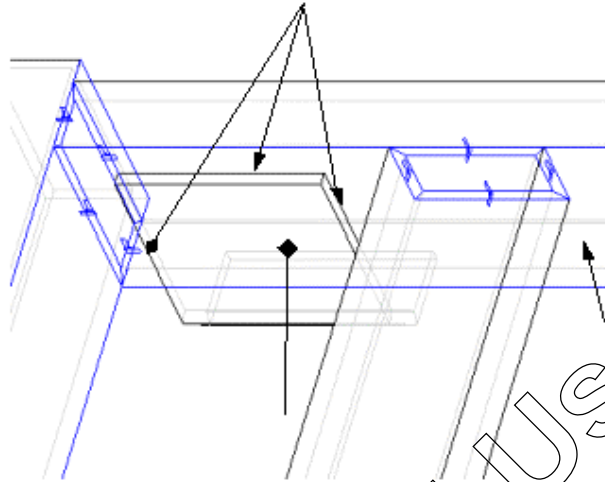


Figure 28: Create the perimeter welds.

Note:

For a perimeter weld, geometry will not be created until analysis is run. Hence, there will still be a gap between the parts.

5. Compress the model again to verify the connections.
6. Save the model.
7. Close the window and erase all models from session.

Summary

After successfully completing this module, you should know how to:

- Describe the types and applications of idealizations.
- Simplify designs using idealization.
- Describe the purpose of rigid connections.
- Describe the purpose of end and perimeter welds.

For PTC Internal Use Only

Optimizing Models for Analysis

Introduction

The first step in the Mechanics analysis process is part creation. You need to build a part before Mechanics can analyze and optimize it. The feature creation techniques used when you build your part will have a significant positive or negative influence on your modeling experience. The integrated mode of Mechanics unites geometry creation with analysis. Consequently, you need to take into consideration downstream activities such as analysis, sensitivity, or optimization as you build your part. You need to think modeling, not just geometry, and you need to plan ahead to increase efficiency.

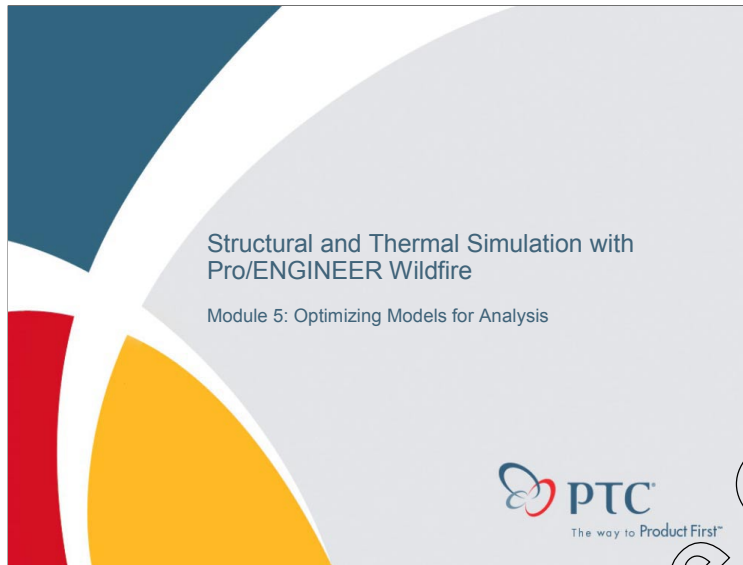
To promote a flexible approach to part modeling, Mechanics enables you to define your model as either a solid or shell. The way you define your model determines the type of elements Mechanics uses to model your part.

In this module, you learn how different geometry creation techniques used in Pro/ENGINEER Wildfire integrate with Mechanics. You also learn Pro/ENGINEER Wildfire techniques that reduce calculation time in Mechanics while maintaining accurate results.

Objectives

After completing this module, you will be able to:

- Create Pro/ENGINEER Wildfire models for analysis.
- Use shell elements, solid elements, and regions.
- Model structural assemblies.
- Create 2-Dimensional models.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Objectives

After completing this module, you should be able to:

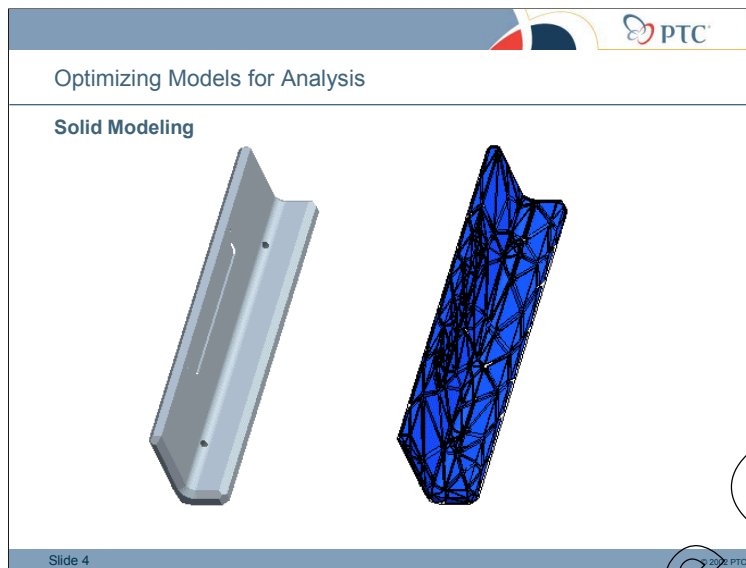
- Create Pro/ENGINEER Wildfire models for analysis.
- Use shell elements, solid elements, and regions.
- Model structural assemblies.
- Create 2-Dimensional models.

Slide 3

Objectives

Geometry must be created in Pro/ENGINEER Wildfire before it can be analyzed and optimized. You must take into consideration the future activities, such as analysis, sensitivity, or optimization as the geometry is created. Therefore, proper modeling techniques must be used.

To promote a flexible approach to analysis, a model may be defined with shell or solid elements. The way that a model is defined determines the type of elements Mechanica will use to model the part.



Solid Modeling

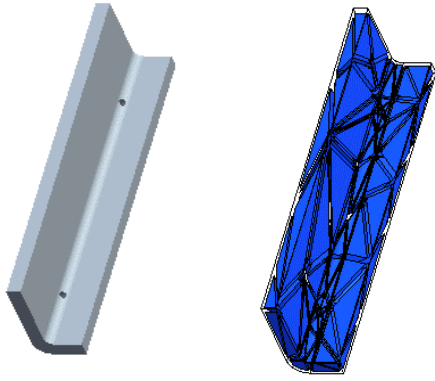
A solid model consists of geometry that can be represented by solid elements like tetrahedrals, bricks, or wedges. In Integrated Mode, Mechanical uses only tetrahedrals for solid modeling, by default. As a rule, solid elements should be used when geometry is as thick and wide as it is long. The geometry's thickness, however, can vary non-uniformly.

In Integrated Mode, element creation is automatic and transparent, with the analysis beginning immediately after the mesh is created. The elements cannot be seen in Integrated Mode until after an analysis is complete (when viewing results). Thus, the user must enter the Independent Mode in order to view the elements prior to running an analysis.

PTC

Optimizing Models for Analysis (cont.)

Suppressing Cosmetic Features



Slide 5

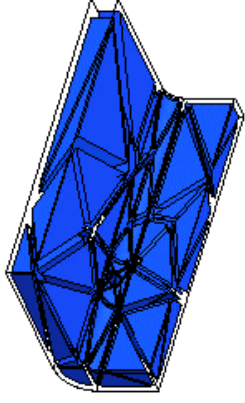
© 2006 PTC

There are no instructor notes on this slide.

PTC

Optimizing Models for Analysis (cont.)

Geometric Symmetry



Slide 6

© 2006 PTC

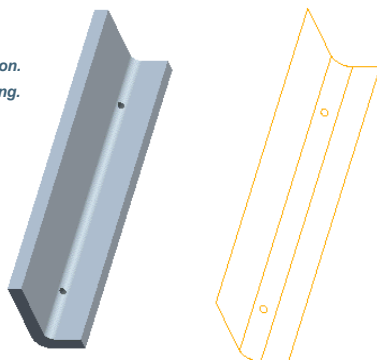
There are no instructor notes on this slide.

PTC

Optimizing Models for Analysis (cont.)

Modeling Shells

- Defining Mid-Surfaces
 - Defining shell pairs.
 - Test the pairs compression.
 - Verify the Use Pairs setting.



Slide 7

© 2006 PTC

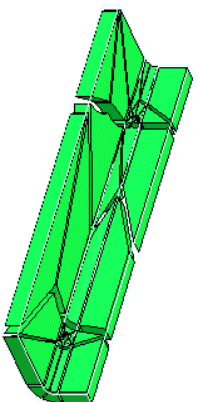
There are no instructor notes on this slide.

PTC

Optimizing Models for Analysis (cont.)

Modeling Shells (cont.)

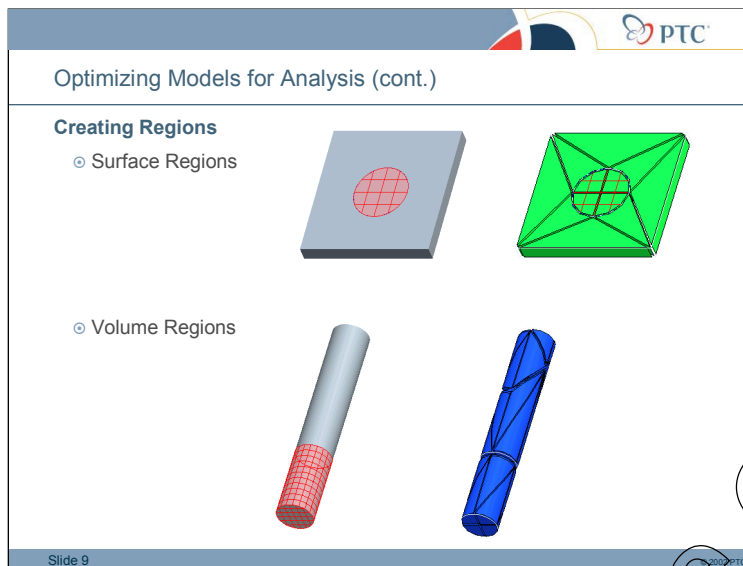
- Automatically create the mid-surface.
 - *Auto Detect*
 - *Shell features*
 - *Thin Features*



Slide 8

© 2006 PTC

There are no instructor notes on this slide.



Creating Regions

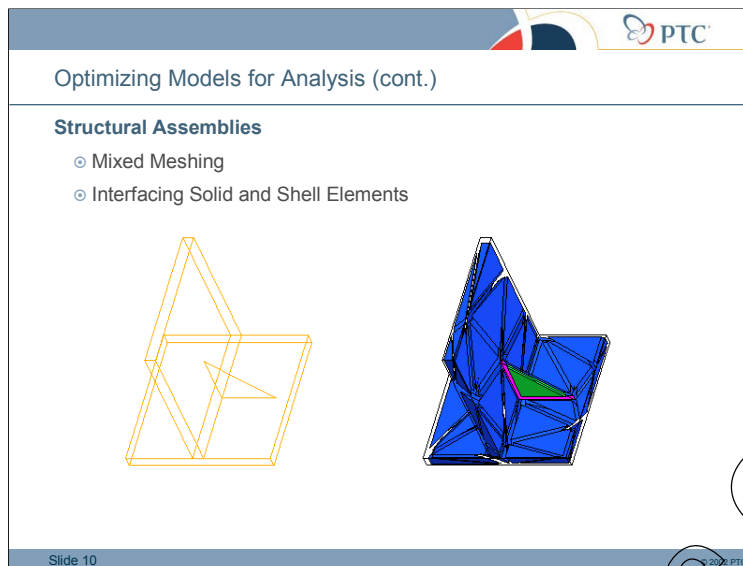
Regions are used to apply loads and constraints to particular “footprints” on a model. Surface Regions may be created in Mechanica to apply loads and constraints to specific localized surface areas. Surface region creation is a two-step process.

- **Defining the region boundary** is accomplished either by creating a datum curve feature to represent the boundary or by sketching the datum curve on the fly. A separate datum curve feature can be created for each region that must be defined. Multiple region boundaries cannot be defined with only one datum curve feature.
- **Creating the region** consists of splitting a model surface into smaller surfaces, or regions, using the datum curve as a boundary.

As of Release 2000i, features such as Datum Points, Curves and Coordinate Systems can be created either in Mechanica or in Pro/ENGINEER Wildfire.

Regions must be defined before mid-surfaces for shell modeling, because creating regions can invalidate existing shell pairs.

While Surface Regions may be created in Mechanica to apply loads and constraints to specific localized surface areas, similar functionality allows solids to be split into three-dimensional regions. This is accomplished by creating Volume Regions. Creation of the Volume Region simulation feature is similar to that of the Cut feature in Pro/ENGINEER Wildfire. It can be created in parts or assembly models and inherits the material properties from the solid geometry within which they are created. Since results may be viewed by volume, these regions are beneficial in preparing a model for post processing, making it easier to view internal stresses, strains, etc. Since elements must be created within a volume region, they can also be used as an effective means of increasing mesh density when required.



Structural Assemblies

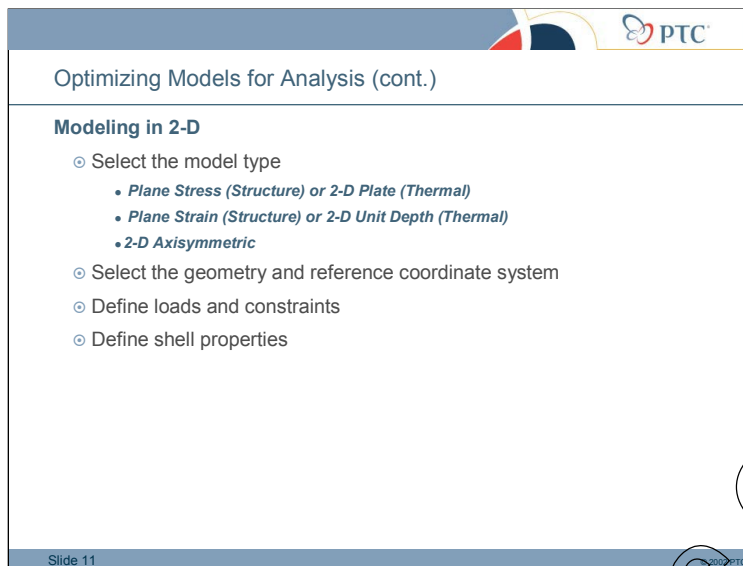
While understanding how parts behave individually is important, many times these parts are bolted or welded to other parts in an assembly. Their interaction may require analyzing the parts joined together in a structural assembly. Thus, the user must understand how to prepare an assembly for a shell model and for a solid model, as well understand the differences between shell and solid models.

Layers and Groups

Layer functionality is fundamental to organization and working efficiently in Pro/ENGINEER Wildfire. When a model is transferred to Independent Mode in Mechanica, there is an option for transferring Pro/ENGINEER Wildfire layers into Mechanica groups. The advantages of groups in Independent Mode are numerous. Groups facilitate ease of use when clicking or manipulating entities—especially elements. They are very useful in both model preparation, as well as in post-processing. For example, if a user wanted to view the stress fringe results for only one part in an assembly (as opposed to viewing the results on the entire assembly), the results can be viewed by group.

Mixed Meshing

In some cases, different element types can be combined. This is done simply by providing Mechanica with the geometry associated with the desired element type and by verifying settings. In Mechanica, if **Settings> Use Pairs** is checked on, any existing shell pairs will be meshed with shell elements, while any remaining unpaired part geometry will be meshed with solid elements. This functionality also works on single parts that are partially paired. Mixing element types is sometimes critical in balancing time savings with required accuracy.

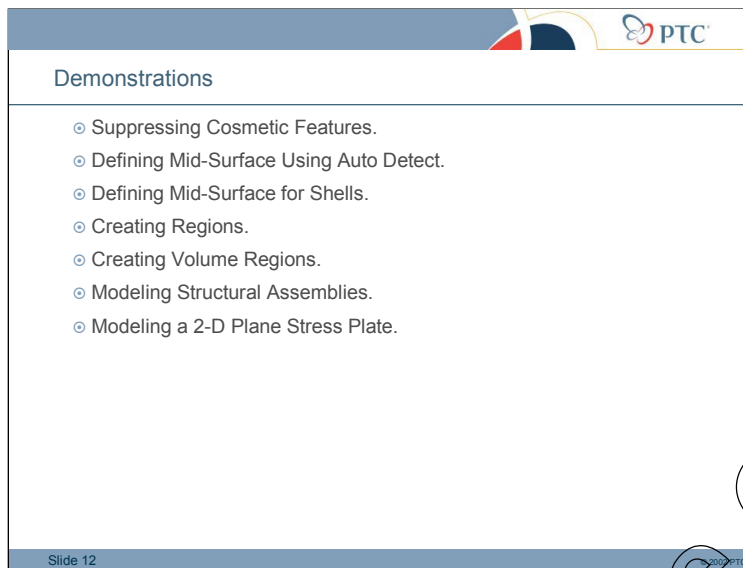


Modeling in 2-D

It is possible to define 2-D model types in Mechanica Integrated Mode. In Structure, the available 2-D model types are Plane Strain, Plane Stress, and 2-D Axis-symmetric. In the Thermal module the available model types are 2-D Plate, 2-D Unit Depth, and 2-D Axis-symmetric. The default model type is 3-D.

The suggested procedure to define a model for a 2-D analysis in Integrated Mode is as follows:

- **Define a reference coordinate system** - To prepare the model for a 2-D analysis, a reference coordinate system must be created before the model type is changed from 3-D. The coordinate system can be created in Pro/ENGINEER Wildfire or in Mechanica.
- **Select the model type** - The model type should be selected before any loads, constraints, or material properties are defined on the model. Changing the model type from 3-D to 2-D causes all modeling entities to be deleted. The available model types are:
 - **3-D** - Use this option if any aspect of the model lies out of the WCS XY plane. Most models will be 3-D. This is the default model type.
 - **Plane Stress (Structure) or 2-D Plate (Thermal)** - Models should be thin and all modeling entities (properties, constraints, loads, and geometry) must lie in the XY plane of the reference coordinate system.
 - **Plane Strain (Structure) or 2-D Unit Depth (Thermal)** - Model should be sufficiently long such that strain in the transverse z-direction is negligible. All modeling entities must lie in the XY plane of the reference coordinate system.
 - **2-D Axisymmetric** - Geometry and all modeling entities should be symmetric about an axis. All modeling entities must lie in the positive x portion of the XY plane of the reference coordinate system.
- **Select the geometry and reference coordinate system** - Once a model type is selected, the user is required to select the geometry and reference coordinate system for the model. If either selection is invalid, the model type will not be changed.



Demonstrations

In this demonstration, I will:

- Suppress structurally insignificant features
- Use shell elements, solid elements, and regions in Mechanica.
- Define mid-surfaces for shell modeling.
- Create a shell model using auto detect.
- Create regions.
- Create volume regions.
- Model simple structural assemblies.
- Model a 2-D plane stress plate.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- ◉ Create Pro/ENGINEER Wildfire models for analysis.
- ◉ Use shell elements, solid elements, and regions.
- ◉ Model structural assemblies.
- ◉ Create 2-Dimensional models.

Slide 13

Summary

There are no instructor notes on this slide.

Module 5 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Suppressing Cosmetic Features

Introduction

In this demonstration, we will de-feature a Pro/ENGINEER Wildfire model to simplify and save time running Mechanica analyses. We will learn the importance of suppressing purely cosmetic features on your part in preparation for analysis. This exercise is designed to give you practice in identifying features worthy of suppression. We also investigate how to further reduce model size by taking advantage of geometric symmetry.

After successfully completing this exercise, you will know how to:

- Identify and suppress structurally insignificant features.

Scenario

To help reduce the number of elements that AutoGEM makes, you can *de-feature* a model.

In this exercise, you will learn the importance of suppressing purely cosmetic features on your part in preparation for analysis. This exercise is designed to give you practice in identifying features worthy of suppression. You also investigate how to further reduce model size by taking advantage of geometric symmetry.

Step 1. Open HANDLE.PRT.

Open the handle part.

1. Set your working directory to **C:\users\student\mechanica_330\module_05**.
2. Open HANDLE.PRT.

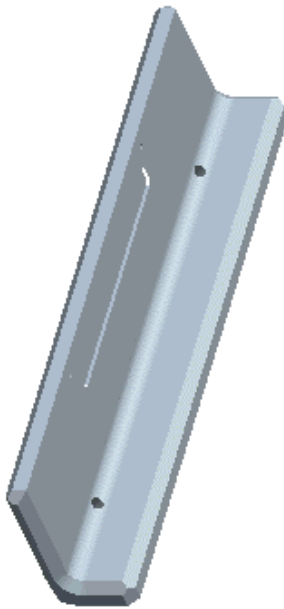


Figure 1: Handle Model

Step 2. Create and review the elements in Integrated Mode with the full model.

Note:

Find how many elements are created using AutoGEM. This will give you an indication of how model complexity can affect the number of elements, and consequently solution time, and disk space requirements for a fully detailed part.

Enable Mechanics and assign any material to the part.

Create solid elements using AutoGEM. Review the number of elements created and the CPU time. Shrink the elements display by 20% and hide the mesh points, the model should like the figure below. Close the file and erase from memory without saving.

1. Enable Mechanics Structure and check the units.
2. Assign material to the handle part.
 - Click **Materials** in the STRC MODEL menu.
 - Select **AL6061** and move it to the Material in Model box.
 - Click **Assign > Part**. Select the handle part and click **OK**.
 - Click **Close** to close the dialog box.
3. Create solid elements on handle part using AutoGEM.
 - Click **AutoGEM** in the STRC MODEL menu.

- Ensure that All with Properties is selected in the drop-down list. Click **Create**.
4. Review the model summary and confirm that there is a single volume and the solid elements were created.
 5. Note the number of elements and how long it took from the AutoGEM Summary dialog box.
 - The number of elements _____
 - Time _____
 - Element type _____
 - Reason for generating this type of element _____
 6. When the elements are created change their display. Shrink the elements display by 20% and hide the mesh points.
 - From the top menu, click **View > Simulation Display**.
 - Select the **Mesh** tab.
 - Unselect the checkbox **Mesh Points**.
 - Select the checkbox **Shrink Elements** and enter percentage value of **20**. Click **OK**.

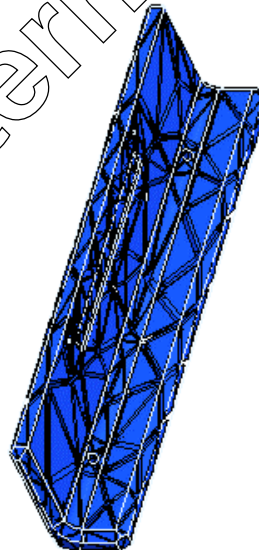


Figure 2: Meshed handle model

7. Click **Close** on the AutoGEM Summary dialog window.
8. Click **Close** on AutoGEM dialog window.
9. Do not save the mesh. Click **No**.

Step 3. Suppress the structurally insignificant features and AutoGEM the remaining geometry.

Re-open the handle part.

Suppress the chamfer, the pinholes, and the recess cut by clip-suppressing the chamfer.

Enable Mechanica and assign any material to the part.

Create solid elements using AutoGEM, review the number of elements created and the CPU time. Notice the difference from the previous step. Close the file and erase from memory without saving.

1. Enable Standard mode by clicking **Applications > Standard**.
2. Suppress the locator pinholes, the chamfers, and the recess cut by clicking **Edit > Feature Operations > Suppress > Clip**. Select the Chamfer feature in the Model Tree (Chamfer id 258). The system suppresses the chamfer and all the subsequent features.
3. Enable Mechanica Structure.
4. Create solid elements on handle part using **AutoGEM** and record the following:
 - Number of Elements _____.
 - Time to mesh _____.
5. Compare this mesh summary to the previous result.
6. Do not save the Mesh.

Step 4. Reduce the element count by removing symmetric geometry.

This technique is only valid under symmetric situations, which will be discussed in detail in later chapters. Open the handle part again. Resume all the features. Create a cut and cut the part in half as shown below.

1. Enable Standard mode by clicking **Applications > Standard**.
2. If necessary, resume all the suppressed features.

Note:

Examine the part and notice that part is symmetric about RIGHT. When the load and constraints are also symmetric about RIGHT, the model can be cut in half.

3. Cut the part in half at RIGHT. Create a cut, sketch on TOP datum, with RIGHT as the top reference plane.
4. Sketch a rectangle to encompass the redundant geometry, as shown in the following figure.

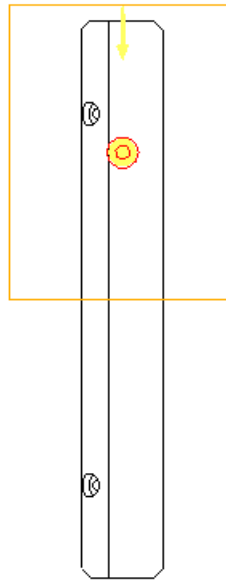


Figure 3: Sketch for the symmetry cut.

5. Make the cut thru all. Flip the arrow if necessary, so that the model appears as shown in the following figure.

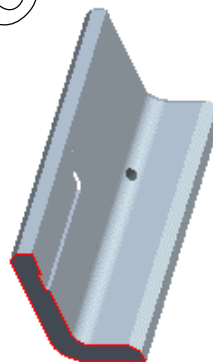


Figure 4: Half of the Handle Model.

Note:

When you AutoGEM this fully featured half model, you would get about half the number of elements. Using these simple, but powerful, techniques will drastically reduce analysis time.

Step 5. Mesh the model using AutoGEM and compare the element count.

Enable Mechanica and create elements using AutoGEM. Show that the number of elements is half.

1. Repeat earlier steps and create solid elements on the model using AutoGEM. Record the number of elements and the time below.
 - Number of Elements _____
 - Time to mesh _____.
2. Save the model.
3. Close the window and erase all models from session.

For PTC Internal Use Only

Exercise 2: Defining Mid-Surface Using Auto Detect

Introduction

In this demonstration, we will show how to use Auto Detect to shell the handle part.

Objectives

After successfully completing this exercise, you will know how to:

- Create shell mid-surfaces using Auto Detect.
- Compress the shells to mid-surface.

Scenario

In a certain cases, Mechanica can automatically find the mid-surfaces for the shell.

In this exercise, you will use Auto Detect to shell the handle part model.

Step 1. Retrieve the handle part and attempt to Auto Detect and compress the surface pairs.

Open the handle part again and resume all the suppressed features.

Auto Detect and compress the shell. Notice the warning.

1. Open HANDLE.PRT.
2. If necessary, resume all the suppressed features by clicking **Edit > Resume > All**.
3. Enable Mechanica Structure.
4. Create mid-surfaces by clicking **Idealizations > Shells > Midsurfaces > Auto Detect > Compress > Shells Only**. Note that the system will display a warning that some of the surfaces are unpaired. Click **Done**.

Step 2. Suppress the counter-bores in Pro/ENGINEER Wildfire and retry creating the mid-surfaces using Auto Detect.

Go back to standard Pro/ENGINEER Wildfire and clip-suppress the HOLE ID 221.

Auto Detect the mid-surfaces and compress the shells. Notice that the rounds could not be compressed.

Manually add the two round surfaces as a new pair and compress again.

1. Enable Standard mode by clicking **Applications > Standard**.
2. Suppress the unpaired geometry.
 - Click **Edit > Feature Operations > Suppress > Clip**.

- Select the feature HOLE ID 211. The system suppresses all the subsequent features. The handle part should appear as shown in the following figure.

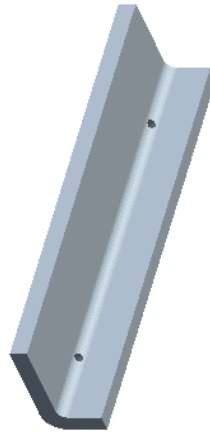


Figure 5: Handle part with suppressed features

3. Enable Mechanical Structure.
4. Auto Detect the mid-surfaces, by clicking **Idealizations > Shells > Midsurfaces > Auto Detect**. Note that not all the surfaces are paired. The unpaired geometry is the two separately created rounds.
5. Manually pair the inner and outer rounds by creating a **New** pair and select the inner and outer round surfaces. Hold CTRL + left-click to select multiple surfaces.
6. Click **Compress > Shells Only**. The model should appear as shown in the following figure.

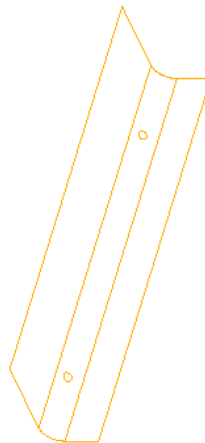


Figure 6: Compressed Shell of Handle Part

7. Assign the material AL6061 to the handle part.
8. Create solid elements using AutoGEM on the handle part.
9. Save the model.
10. Close the window and erase all models from session.

For PTC Internal Use Only

Exercise 3: Defining Mid-Surface for Shells

Introduction

In this demonstration, we will create a shell mid-surface by defining the shell pairs. We will then test the compression and show the compressed model.

Objectives

After successfully completing this exercise, you will know how to:

- Create shell mid-surfaces and define shell pairs.
- Compress the shells to mid-surface.

Scenario

Using shell elements can result in models with fewer elements that will run faster and require less disk space than a solid model.

In this exercise, you create a shell model from a T-bracket and create the mid-surfaces both manually and automatically.

Step 1. Prepare the model for shell elements.

Open T_BRACKET.PRT and go to Mechanics.

Use Idealization and create a new mid-surface for shells. Create a shell mid-surface by selecting the surface pairs. To select more than one surface, use the [Ctrl] key

1. Open T_BRACKET.PRT.
2. Enable Mechanics Structure.
3. Use **Idealizations** to create a new mid-surface for shells.
4. For the surfaces to compress, select the two surfaces to define the first pair as shown in the following figure. To select more than one surface CTRL + left-click.

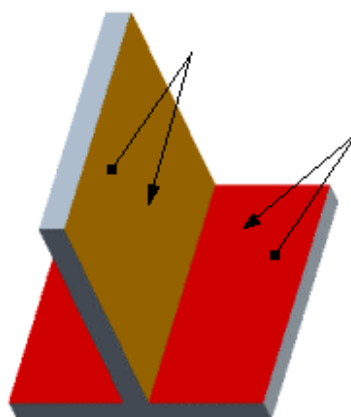


Figure 7: Create two surface pairs.

5. For the second pair select the other two surfaces as shown in the previous figure.

Step 2. Test the mid-surface compression to verify that you have defined the pairs correctly.

Test the mid-surface compression and show shell only.

1. Compress the model to display the mid-surface and shell only by selecting **Compress > Shells Only**. The pairs get compressed to their mid-surfaces and displayed in yellow.
2. You can also display the original geometry or display both the original geometry and the compressed mid-surface simultaneously.

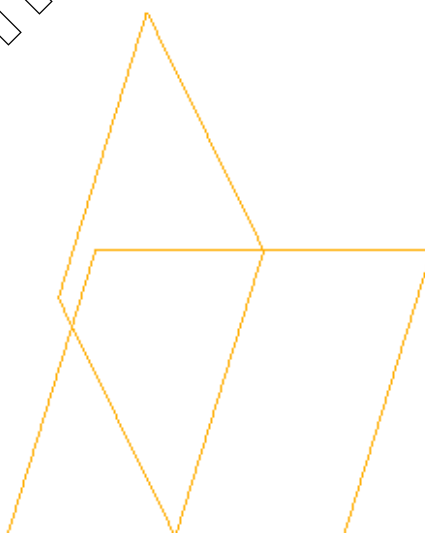


Figure 8: Compressed mid-surfaces.

Step 3. Use the Auto Detect option to generate mid-surfaces in the T_BRACKET.PRT.

Delete all previous pairs.

Create new mid-surfaces using Auto Detect and test the compression. The model should look like the previous figure.

1. Delete all previous pairs by selecting **Delete > Select All > Yes**.
2. To automatically create the pairs, select **Auto Detect**. Mechanica highlights the pairs that it was able to automatically create. One surface in each pair is red and the other is yellow.
3. Test the compression and show the shells.
4. Assign the material AL6061 to the bracket part.
5. Create solid elements using AutoGEM on the bracket part.
6. Save the mesh.
7. Save the model.
8. Close the window and erase all models from session.

Exercise 4: Creating Regions

Introduction

In this demonstration, we will show how to create surfaces regions by sketching a curve and splitting a surface using the curve.

Objectives

After successfully completing this exercise, you will know how to:

- Create surface regions.
- Create curves for surface regions.
- Split surfaces into surface region using curves.

Scenario

In order to constrain or load a specific portion of a part's surface, you need to create a region on the part. A region is created by splitting a specific surface on the part into multiple sections using a datum curve.

You will use the simple 254 x 254 x 25.4 steel plate and create circular regions on each side of the plate where you want to apply loads.

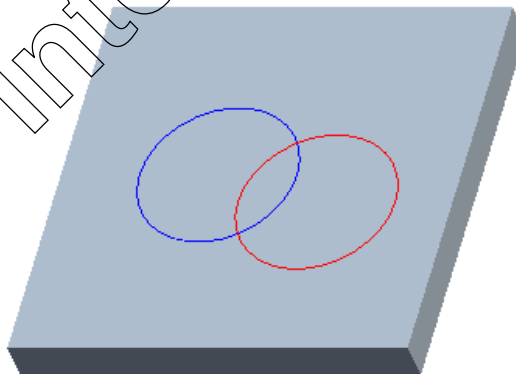


Figure 9: Regions on the plate

In this exercise, you will learn how to create regions and understand their effects on pairing by creating a simple sketched region.

Step 1. Create the first datum curve.

Retrieve the regions part and open Mechanica.

Create a datum curve feature. Use the top surface of the plate as a sketching plane. Use the bottom surface as bottom reference.

Sketch the curve shown below.

1. Open REGIONS.PRT.
2. Enable Mechanics Structure.
3. Create a sketched datum curve feature on the top of the plate.
4. Select sketching references and sketch the section as shown in the following figure.

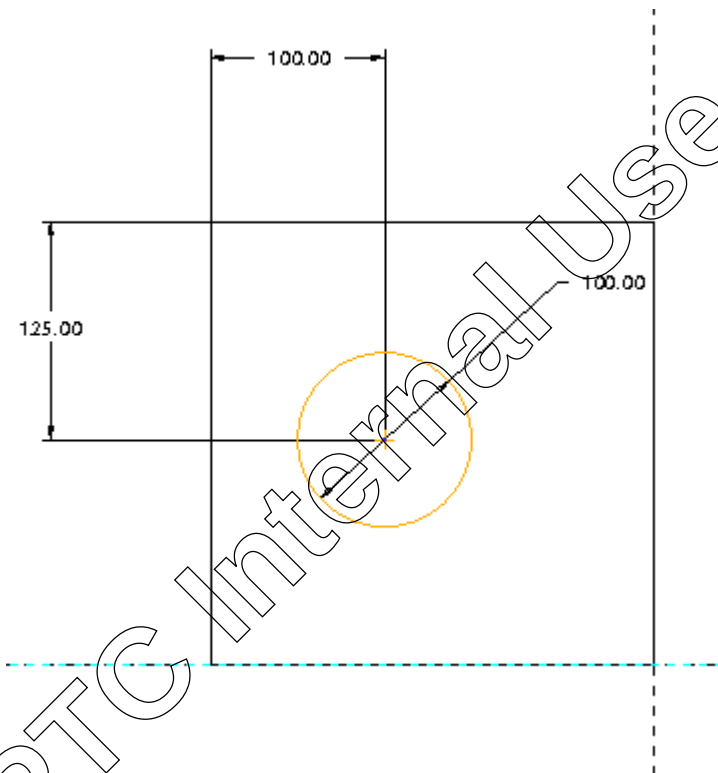


Figure 10: The sketched section for first region

Step 2. Create the second datum curve.

Create another datum curve feature on the other side of the plate. Use the bottom surface as bottom reference.

Sketch the curve shown in the figure below.

1. Create another sketched datum curve feature on the opposite side of the plate.
2. Select the opposite surface to the one selected for the initial datum curve as the sketching plane.

3. Specify the sketching references and sketch the section shown in the following figure.

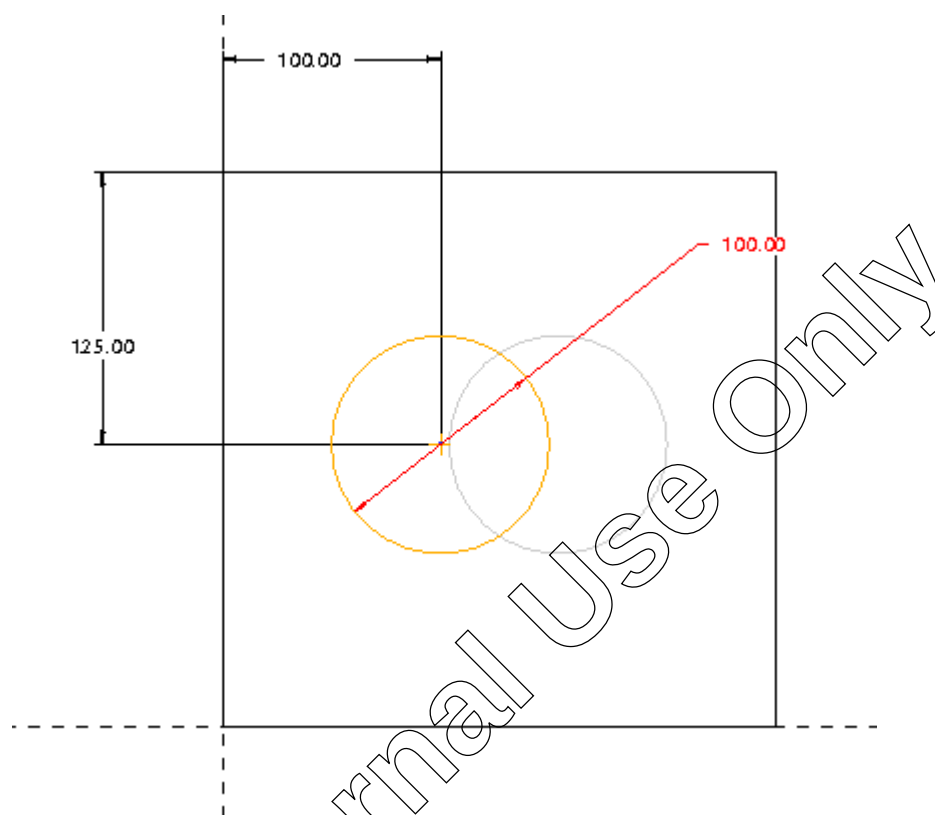


Figure 11. Sketch of the other curve

Step 3. Use the datum curves you created to define the regions.

Create a surface region. Select the top datum curve and select the top surface as the surface to split.

Follow the same procedure to create the bottom region.

Verify the defined regions by meshing the surfaces.

1. Create a surface region by selecting **Model > Features > Surf Region > Create > Select > Done**.
2. Select the top datum curve.
3. Select the top surface as the surface to split.
4. Follow the same procedure to create the bottom region.
5. Verify the defined regions by meshing the surfaces. Select **View > Model Setup > Mesh Surface**. Click on the surfaces within the datum curves. The model should appear as shown in the following figure.

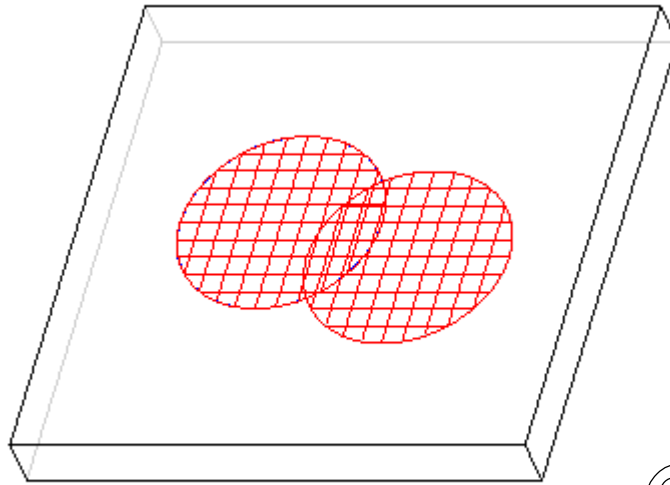


Figure 12: Meshed surface regions

Note:

Mesh surface in the previous step is a Pro/ENGINEER Wildfire functionality for visualizing surfaces. The mesh surface does not contribute to element generation in any way.

Step 4. Manually define the mid-surface.

Create shells then define new mid-surfaces. Select the top and bottom surfaces of the model (outside of the circular region).

Test the compression, shell only. Notice the error message in the message box regarding unpaired surfaces.

Edit the pairs and add the top circular region and the bottom circular region. Notice the color change.

Test the compression, shell only.

1. Create shell mid-surfaces. Define new mid-surfaces by selecting **Idealizations > Shells > Midsurfaces > New**.
2. Ensure that the model is shaded. Select the top and bottom surfaces of the model outside of the circular region.
3. Test the compression shell only. Notice the error message in the message box regarding unpaired surfaces.

Note:

When you created the regions, you actually split the surfaces. The top and bottom faces now each consist of two surfaces. You need to include all four surfaces in the pair definition.

4. Add the top circular region and the bottom circular region by clicking **Edit > Edit Pair > Add Surface** and selecting the surfaces.
5. Test the compression, shell only. The entire model should appear as shown in the following figure.



Figure 13: Compressed mid-surface

Step 5. Automatically define the mid-surface.

Delete the existing shell and create a shell using Auto Detect to automatically pair the part.

Test the compression. Notice how Auto Pair automatically included the regions you defined.

Save and erase the model.

1. Delete all the existing shell idealizations.
2. Create shell mid-surfaces using **Auto Detect** to automatically pair the part.
3. Test the compression. Notice how Auto Pair automatically included the regions you defined.

Note:

Since the plate model is created as a thin feature, you can automatically create the pairs.

4. Save the model.
5. Close the window and erase all models from session.

Exercise 5: Creating Volume Regions

Introduction

In this demonstration, we will show how to create volume region.

Objectives

After successfully completing this exercise, you will know how to:

- Create a volume region.

Scenario

While surface regions may be created in Mechanica to apply loads and constraints to specific localized surface areas, similar functionality allows solids to be split into three dimensional regions. This functionality is known as creating volume regions.

Volume regions can be created in part or assembly models. These regions inherit properties from the solid.

In this exercise, you will create a volume region.

Step 1. Open VOLUME_REGION.PRT.

Retrieve the volume region part and open Mechanica Structure.

1. Open VOLUME_REGION.PRT.
2. Enable Mechanica Structure.

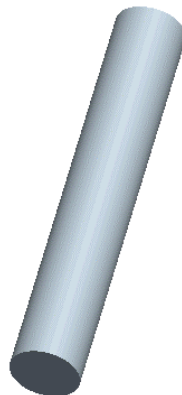


Figure 14: VOLUME_REGION.PRT

Step 2. Create a volume region.

Create an extruded volume region feature. Use either end face of the shaft as a sketching plane.

*In Sketcher, Use Edge and select both semi-circular edges to form a circular sketch. Enter a depth of **10**.*

Verify the defined region by meshing the surface of the created region.

1. Create a volume region feature using extrude. Select **Model > Features > Volume Region > Create > Extrude > Done**.
2. Select front flat end face of the shaft as a sketching plane.
3. Accept the direction of the feature creation into the cylinder.
4. In sketcher, **Use Edge** and select both semi-circular edges to form a circular sketch.
5. Enter a blind depth of **10**.
6. Verify the defined region by clicking **View > Model Setup > Mesh Surface**. The system meshes the selected surfaces as shown in the following figure.

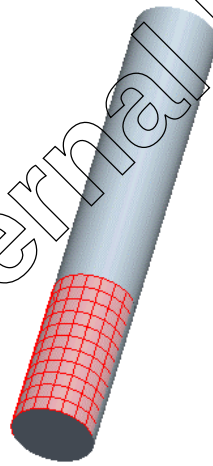


Figure 15: Meshed volume region surface

7. Assign the material STEEL to VOLUME_REGION.PRT.
8. Create solid elements using AutoGEM on VOLUME_REGION.PRT.
9. Save the Mesh.
10. Save the model.
11. Close the window and erase all models from session.

Exercise 6: Modeling Structural Assemblies

Introduction

In this demonstration, we will show how to use Auto Detect to shell the handle part.

Objectives

After successfully completing this exercise, you will know how to:

- Create shells mid-surfaces using Auto Detect.
- Compress the shells to mid-surface.

Scenario

While understanding how parts behave individually is important, many times these parts are bolted or welded to other parts in an assembly. Their interaction may require analyzing the parts joined together in a structural assembly.

In order to compress an assembly in Mechanica, one must create the shells in the part level.

In this exercise, you will create shell mid-surfaces in parts and verify that the shells transfer into the assembly. You will learn how to prepare an assembly for a shell model and for a solid model. Additionally, you will see how to take advantage of the differences between shell and solid models.

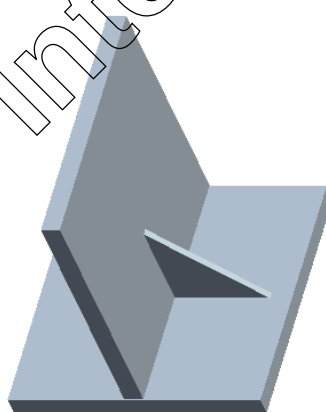


Figure 16: The Gusset Assembly

Step 1. Open the GUSSET.PRT and prepare it for shell modeling.

Open the gusset part and open Mechanica Structure.

Create shells. For new mid-surfaces, select both triangular surfaces. Use CTRL key to select more than one surface. Save the model.

1. Open GUSSET.PRT.
2. Enable Mechanica Structure.
3. Define a shell pair on mid-surfaces by selecting both triangular surfaces. Hold CTRL + left-click to select more than one surface.
4. Save the model.
5. Close the window.

Step 2. Retrieve the GUSSET.ASM and verify the shell and solid idealization in the model.

Retrieve the gusset assembly and verify that the assembly units have been set to mm N s.

Open Mechanica Structure. Test the shell compression. Use shells and solids. Retrieve the gusset assembly. This model is created using assembly constraints, such as MATE and ALIGN. GUSSET.PRT has been idealized as shell. The plate is not idealized as shell, hence it is a solid.

1. Open GUSSET.ASM.
2. Verify that the assembly units have been set to **mmNs**.
3. Enable Mechanica Structure.
4. Test the shell compression by using **Shells and Solid**. The model should appear as shown in the following figure.

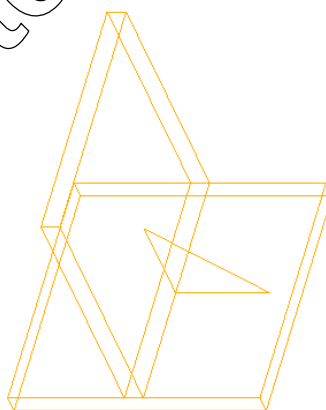


Figure 17: Gusset Assembly with Shell and Solid Idealizations.

5. Assign the material AL6061 to all parts of the Gusset assembly.
6. Create solid elements using AutoGEM on the Gusset assembly.
7. Save the mesh.

Note:

The interface between the shell GUSSET.PRT and the plate parts are handled by the system. If you create and run an analysis in Independent mode, the system will create links between the solid and shell idealization.

8. Save the model.
9. Close the window and erase all models from session.

For PTC Internal Use Only

Exercise 7: Modeling a 2-D Plane Stress Plate

Introduction

In this demonstration, we retrieve a plate and define it as a 2-D model.

Objectives

After successfully completing this exercise, you will know how to:

- Define 2-D models.
- Define shell properties.

Scenario

It is possible to define 2-D model types in Mechanica Integrated mode. For Structure, the available model types are Plane Strain, Plane Stress, and 2-D Axisymmetric. For the Thermal module, the available model types are 2-D Plate, 2-D Unit Depth, and 2-D Axisymmetric. The default model type is 3-D.

When the changes in the direction of stress are small, a 2-D Plane Stress model can be used.

Step 1. Define the PLATE_2D.PRT as a plane stress model.

Open the 2-D plate part and open Mechanica structure.

Define a 2-D plane stress using the front surface of the plate and the coordinate system. This is not possible since the front surface does not lie on the coordinate system.

Change to the back surface.

1. Open PLATE_2D.PRT.

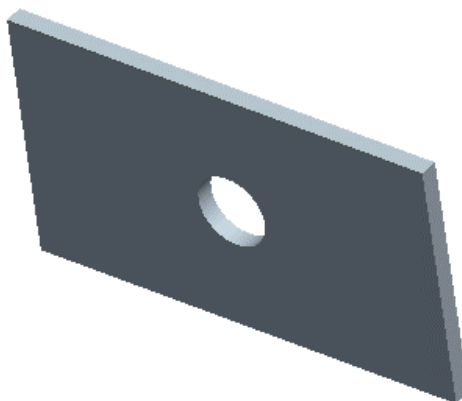


Figure 18: PLATE_2D.PRT

2. Orient to the Standard Orientation. Notice that the back surface of the model lies in the X-Y plane.
3. Enable Mechanica Structure.
4. Specify the model type.
 - Select **Model > Model Type**.
 - In the Model Type dialog box, select **Plane Stress**. Note that the Select Geometry and Select Coordinate System options become available.
 - Click **Select Geometry** and select the front surface of the model.
 - Click **Select Coordinate System** and select the coordinate system from the Model Tree. When finished, the system informs that the surface does not lie in the X-Y plane. Close the information window.
5. Define the Plane Stress model using the back surface. Remove the front surface by selecting it. Then select the back surface of the model.
6. Confirm that all entities will be deleted.

Step 2. Define shell properties to assign material and thickness to the model.

Define a new shell on the back surface. Keep the default name, simple and a thickness of 3. Assign AL6061 as material.

1. Click **Idealizations > Shell > New** to create a new shell.
2. Accept the default name for the shell.
3. Select the same surface that is used to define the plane stress model as the shell surface.
4. Accept the default type **Simple** and enter **3** as the thickness.
5. Assign the material **AL6061** to the model.

Step 3. AutoGEM the plate.

AutoGEM the 2-D plate and review the summary. Notice that 2-D plate elements were created.

1. Create solid elements on the part using **AutoGEM**.
2. Review the summary and notice that Mechanica creates 2-D Plates as elements.

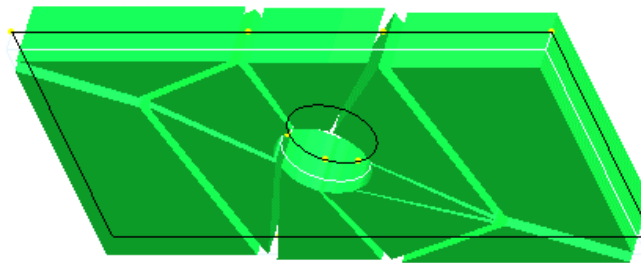


Figure 19: Meshed 2-D plain stress plate.

3. Save the Mesh.
4. Save the model.
5. Close the window and erase all models from session.

Summary

After completing this module, you will be able to:

- Create Pro/ENGINEER Wildfire models for analysis.
- Use shell elements, solid elements, and regions.
- Model structural assemblies.
- Create 2-Dimensional models.

For PTC Internal Use Only

Assigning Properties to Models

Introduction

Before running any Mechanics analysis, you must have a thorough understanding of some basic mechanics of materials, including structural material properties and engineering units. You should be able to answer some of these questions.

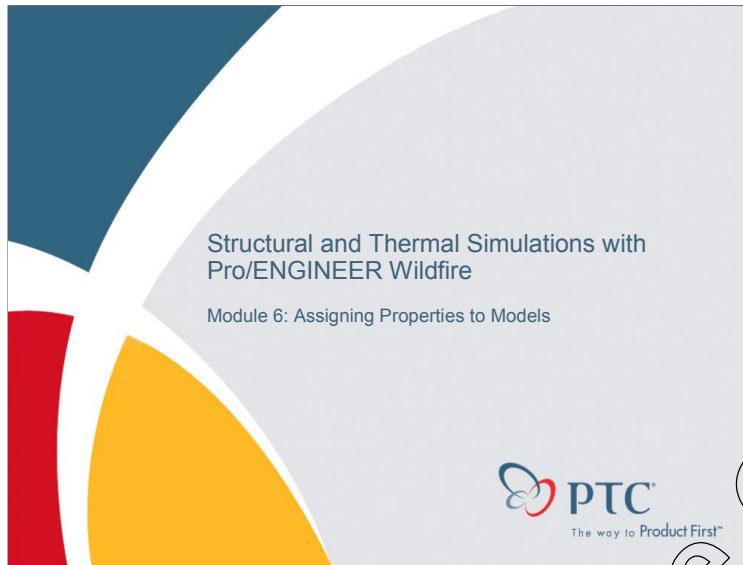
- What is the value of density and what are its units?
- What is the value of Young's Modulus and what are its units?
- What is Poisson's Ratio and what are its units?
- What is the coefficient of Thermal Expansion and what are its units.

Engineering units and materials must be set before running analyses in Mechanics. If a material does not exist in the Mechanics material library, then you can add additional materials to a custom material library.

Objectives

After completing this module, you will be able to:

- Assign structural and thermal properties to parts.
- Create materials libraries.
- Define temperature-dependent material properties.
- Edit and delete material properties.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

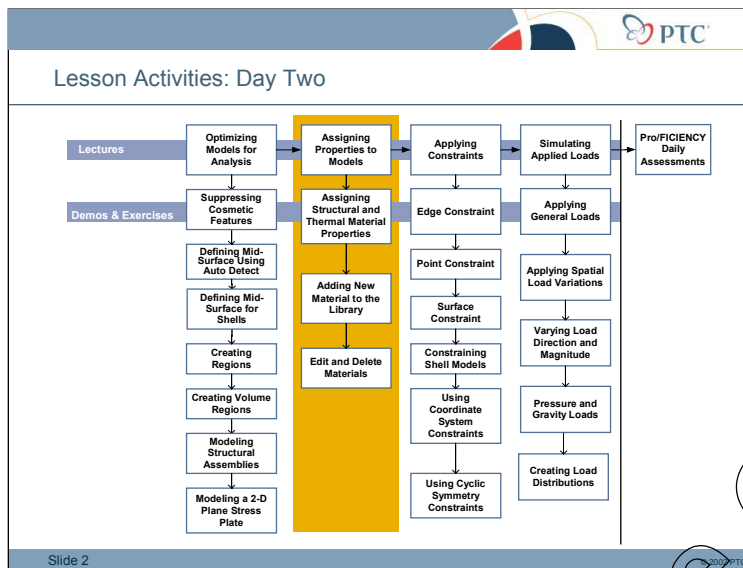
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 15 min
- Demos: 15 min
- Labs: 30 min
- Total: 1 hour



Objectives

After completing this module, you should be able to:

- ◉ Assign structural and thermal properties to parts.
- ◉ Define linear and nonlinear properties.
- ◉ Create materials libraries.
- ◉ Define temperature-dependent material properties.
- ◉ Edit and delete material properties.

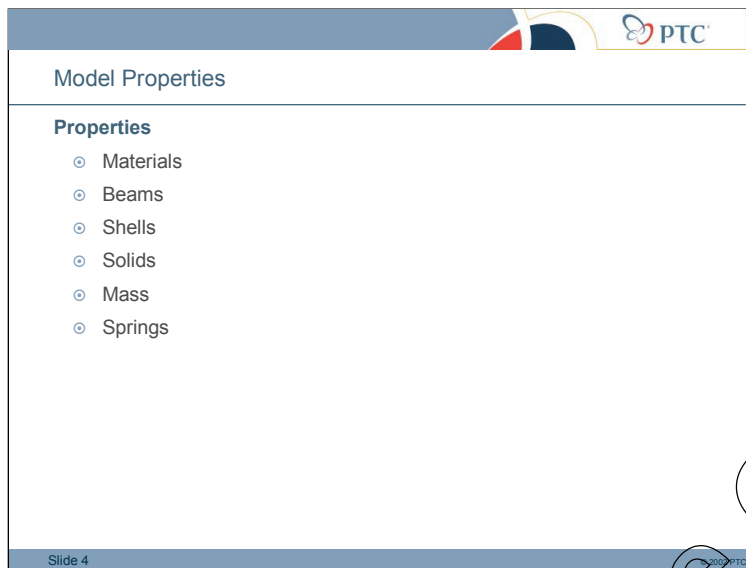
Slide 3

Objectives

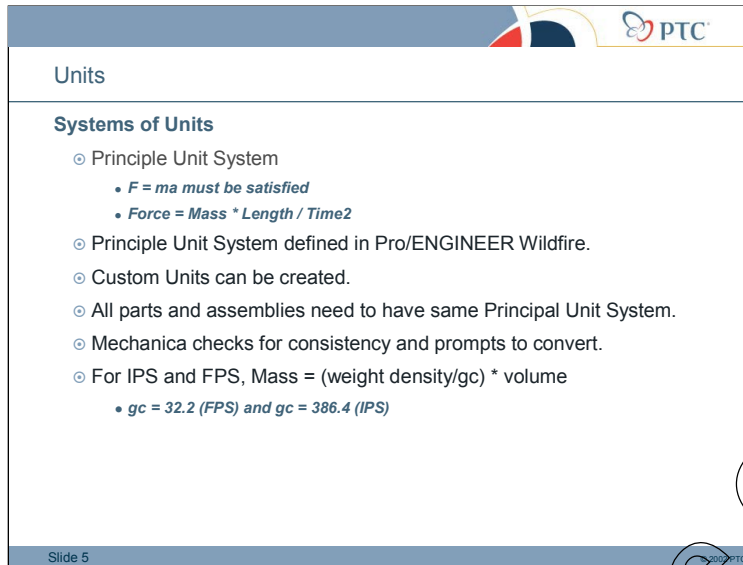
Before running any Mechanics analysis you must have a thorough understanding of some basic of mechanics of materials. This includes structural material properties and engineering units. You should be able to answer some of these questions.

- What is the value of density and what are its units?
- What is the value of Young's Modulus and what are its units?
- What is Poisson's Ratio and what are its units?
- What is the coefficient of Thermal Expansion and what are its units.

Engineering units and materials must be set before running analyses in Mechanics. If a material does not exist in the Mechanics material library, then you can add additional materials to a custom material library.



There are no instructor notes on this slide.

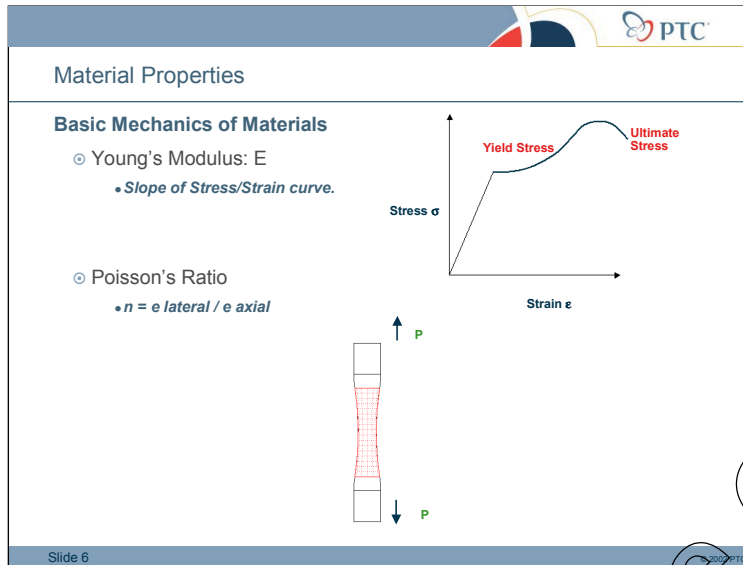


Systems of Units

- Four basic quantities are required to describe a structural system: Force, Length, Time and Mass.
- You must specify units for three out of the four quantities. The units for the fourth quantity is determined in terms of the three others according to (a dimensional analysis on) Newton's Law: $F = ma$. Mechanica has four basic unit systems (MKS, mmNs, FPS, IPS).
- Example: Consider a stress analysis for a computer chip. Assume we select millimeters (mm) for units of Length, Newtons (N) for force and seconds (s) for units of Time.
Substituting units in the equation $F = m \cdot a$ and recalling that $N = \text{kg m/sec}^2$, we have:
 $\text{Kg m / sec}^2 = \text{mass\_unit} * \text{mm / sec}^2$ and $\text{mass\_unit} = 1000 \text{ kg !!}$ or metric Tonnes
Having chosen mmNs as a unit system implies that mass for this computer chip will be expressed in metric tonnes.

Note to the instructors: The following table is given for your reference only.

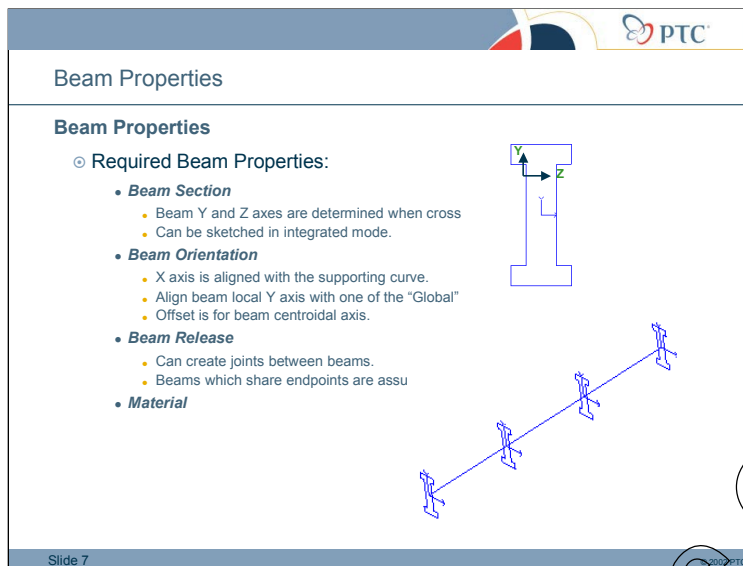
Principle System	Length	Time	Temp	Mass/Force	Derived Mass/Force
inch lbm second (Pro/E default)	in	sec	°F	lbm (mass)	lbm - in/sec ² (force)
MKS	m	sec	K	kg (mass)	kg - m/sec ² = N (force)
CGS	cm	sec	°C	g (mass)	g - cm/sec ² = μN (force)
mmNs	mm	sec	°C	N (force)	1000 kg = tonne (mass)
FPS	ft	sec	°F	lbf (force)	lbf - sec ² /ft = slug (mass)
IPS	in	sec	°F	lbf (force)	lbf - sec ² /in (mass)



Basic Mechanics of Material

Young's Modulus

This slide shows a sketch of a general stress-strain curve. This curve is created by applying an increasing axial load to a test specimen and by measuring the load and deformation simultaneously. From this data, the stress (the vertical axis) can be plotted against the strain, or percent elongation (the horizontal axis). As the axial load is increased, the strain increases in a linear fashion. The slope of this line is called Young's Modulus. The units for Young's Modulus are stress (load/area) over strain (change in length/original length), or in other words, the same units as stress.



Beam Properties

Beam Elements are represented as one-dimensional lines in space. The following properties must be provided:

1. The **Beam section** defines the cross section geometry and/or properties for beam elements.
2. The **Beam orientation** specifies the orientation of the section relative to the model.
3. **Beam release** – Allows the definition of joints between beam elements.
4. A **Material** must be specified for the element.
 - In Integrated Mode, Mechanica displays an icon representing the beam section and orientation as they have been specified.

Beam Sections

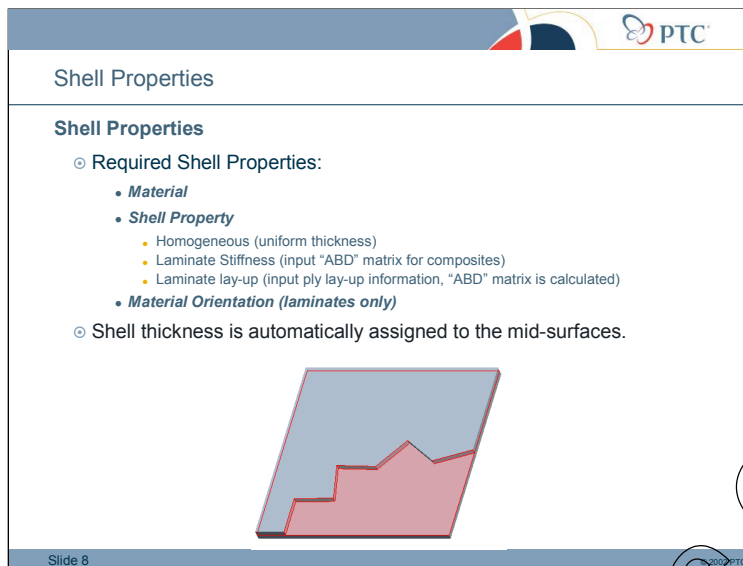
- Beam Section Properties are specified using the following fields:
 1. **Section Name:** This allows the user to define a library of sections for further reference.
 2. **Description:** Use this field to document the section type for easy reference.
 3. **Type:** Specify the type of section to be defined (for example: Channel, I-Beam). This will determine which geometric inputs are needed to fully define the section.
- When the “Review Properties” option is selected, Mechanica will calculate and display area and moments of inertia for the specified section.
- The specification of section properties implies the definition of a local coordinate system for the beam element. By default, the X-axis is oriented along the longitudinal direction. The Y and Z axis directions are defined using the “Beam Orientation” element property.

Beam Release

Use the Releases command to specify the degrees of freedom you want to release for a beam end or ends. If you do not define a beam release, Pro/MECHANICA assumes that all degrees of freedom participate in the connection at the ends of the beam.

Beam releases determine the degrees of freedom that do not participate in a connection at the end of a beam. You can specify beam releases for both straight and curved beams.

In independent mode, a beam release is considered an element property. You associate a beam release with one or more beams and one or more endpoints of these beams. If you select a single beam, you select one of its endpoints. If you select multiple beams, Pro/MECHANICA associates the beam release with all the endpoints of the selected beams.



Shell Properties

Shell elements require the following properties:

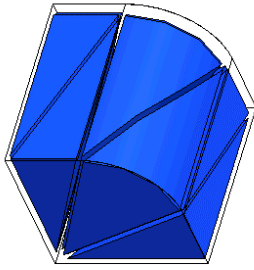
- 1. Thickness** – A shell element must be associated with a thickness in order to be fully defined geometrically. In Integrated mode this is done at the pair definition stage (the thickness of the compressed feature is known). In Independent Mode, the thickness is entered in a shell definition form.
- 2. Material** – In Independent mode, the material property should be placed on surfaces when possible.
- 3. Shell Property** – Shell Elements can be used to model laminates. Additional properties are necessary to define a laminate, namely the equivalent stiffness matrices (A,B,D). These can be entered directly or calculated by Mechanica given a laminate specification (ply thickness, ply material and orientation).

PTC

Solid Properties

Solid Properties

- ⊙ Required Solid Properties
 - **Isotropic**
 - $(E1 = E2 = E3)$
 - **Orthotropic**
 - $(E1 \neq E2 \neq E3)$
 - **Transverse Isotropic**
 - $(E1 \neq E2 = E3)$
- ⊙ Material Orientation
 - **Orthotropic and Transverse Isotropic only.**



Slide 9

Solid Properties

- Being the most expensive in terms of run time, solid elements are used to model parts for which none of the beam, shell or 2D assumptions apply.
- For reliable results, solid elements with aspect ratios greater than 30:1 should not be used.
- The following types of solids can be created:
 - **Brick** (or hexahedron) with six quadrilateral faces and eight points.
 - **Wedge** (or pentahedron) with two triangular faces, three quadrilateral faces, and six points.
 - **Tet** (or tetrahedron) with four triangular faces and four points.
- In Integrated Mode, mesh generation occurs "behind the scenes", and meshes are created using Tetrahedral elements only.
- Remember to turn off the setting of "Use Pairs".

Guidelines for Creating Solids

- To ensure good point, edge, and face connectivity (for example: to avoid cracks in models), hand meshed elements should not be mixed with Auto GEM elements.
- Solids must not overlap.
- Adjacent elements must have edge-to-edge connectivity. Any unmatched edge creates a crack in your model (discussed in the next lecture). Divide large arc spans into separate segments by placing points along them or by splitting them before selecting them as edges.

PTC

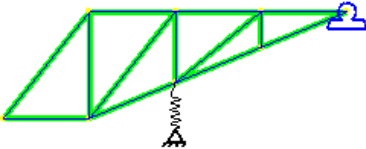
Mass and Spring Properties

Mass Element Properties

- Properties for masses.
 - *Mass and mass matrix.*

Spring Element Properties

- Spring Stiffness
- Spring Orientation
 - *Align Spring local Y axis with one of the global axes; WCS Y axis is the default.*

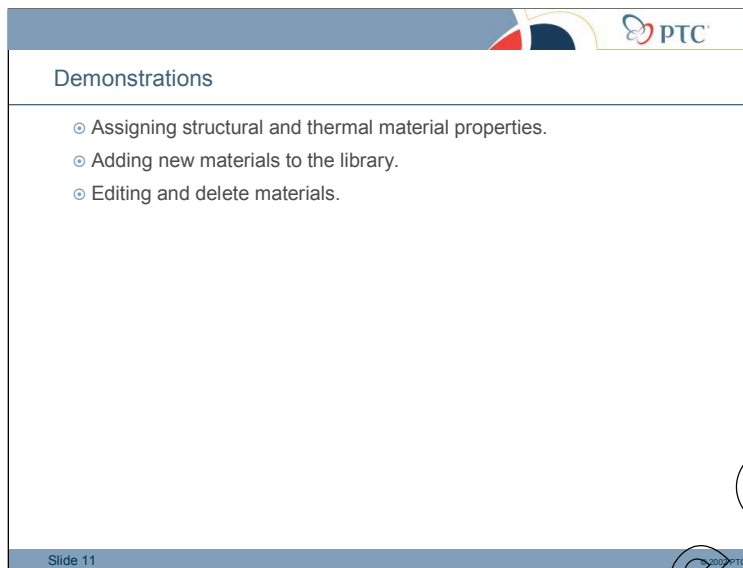


The diagram shows a truss structure with green members. A spring is attached to the bottom of the structure, and a mass element (represented by a blue circle with a '2') is attached to the top right node.

Slide 10

Mass Element Properties

- Mass elements are used to simulate components in a system which contribute mass but no stiffness. A typical example is a motor on a robotic arm.
- Two properties can be used to define a mass element:
 1. The **Mass Value** must be entered in appropriate units (remember this is mass, not weight. You must divide weight by gravity to get the appropriate mass value).
 2. Values for the **Moments of Inertia** are used for analyses where inertial loads are considered. Using moments of inertia is an accurate way to simulate the presence of complex parts without modeling their geometry.
- Note that this representation of parts (Mass and Inertia) is similar to the rigid body dynamics representation in Mechanica Motion.



Demonstrations

In this demonstration, I will

- Assign Structural and Thermal Material Properties
- Add new Materials to the Mechanics library.
- Edit and delete materials.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- ◉ Assign structural and thermal properties to parts.
- ◉ Define linear and nonlinear properties.
- ◉ Create materials libraries.
- ◉ Define temperature-dependent material properties.
- ◉ Edit and delete material properties.

Slide 12

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 6 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Assign Structural and Thermal Material Properties

Introduction

In order to run any Mechanica analysis, you need to assign the structural and/or thermal material properties to a part. For Mechanica to calculate stresses and displacement, the material stiffness (Young's Modulus) is required.

Also, you must understand that failure begins whenever the stress in the part exceeds its Yield Strength. Mechanica always assumes that the Young's modulus has a constant slope. Analyses that only consider the straight-line portion of the stress-strain curve (constant Young's Modulus) are considered linear. If the maximum stress in your model exceeds the material's Yield Stress, then the reported values are not accurate. In this demonstration, we will open a part, check the units, and add structural and thermal material properties.

Objectives

After successfully completing this exercise, you will know how to:

- Assign structural and thermal material properties to a part.

Scenario

In order to run any Mechanica analysis, you must first assign the structural and/or thermal material properties. For Mechanica to calculate stresses and displacement, the material stiffness (Young's Modulus) is required.

In this exercise, you will open a part, check the units, and add structural and thermal material properties.

Step 1. Define a system of units for your model in Pro/ENGINEER Wildfire.

Open the t-bracket and check the units. Make sure the units are mm N s.

Enter Pro/MECHINCA Wildfire Structure and assign STEEL to the part.

Note:

The default units in Pro/ENGINEER Wildfire are inches pounds-mass seconds Fahrenheit.

1. Set your working directory to **C:\users\student\mechanica_330\module_06**.
2. Open T_BRACKET.PRT.
3. Check the units. Ensure the units are set to **mm N s**.
4. Enter Mechanica Structure.
5. Assign the material properties of **STEEL** to T_BRACKET.PRT.

Step 2. Review the properties.

Edit the STEEL Properties and review them. Add failure criteria and enter 231 N/mm² for tensile yield stress.

1. Edit the STEEL properties. The Material Definition dialog box for STEEL appears.
2. Review the values for:
 - Density
 - Poisson's Ratio
 - Young's Modulus
 - Coefficient of Thermal Expansion
3. Select **Failure Criterion** tab.
4. Select the Distortion Energy (von Mises) option from the drop-down list. Change the tensile yield stress to **231 N/mm²**.
5. Close all dialog boxes.

Exercise 2: Adding New Materials to the Library

Introduction

In this demonstration, you will add a new material to the library and assign it to the part.

Objectives

After successfully completing this exercise, you will know how to:

- Add new material to materials libraries.

Scenario

If the material you want does not exist in the Mechanics library, you can simply add it. You will create a new material that is not in the library. Assign this new material to the T-Bracket and save this new material to your own custom material library.

Step 1. Add a new material to the Mechanics library.

Create a new material shown in the table below and assign it to the part.

1. Open the Material dialog box and click **New** to create a new material.
2. Select the appropriate units and enter the properties listed below.

Table 1: New material properties.

Material Name	Nickel
Description	Nickel Material Properties
Structural	Isotropic
Mass Density	8802 kg/m ³ or 17.08 slug/ft ³
Poisson's Ratio	0.31
Young's Modulus	206843 N/ mm ² or 30E06 psi
Coefficient of Thermal Expansion	1.296E-05 /C or 7.2E-06 /F

Note

You could flip from unit to unit by selecting it, Mechanics will do the conversion accordingly.

3. Add the new material to the library. In the Materials dialog box, click on the **Arrow** pointing toward Materials in Library. Click **Yes** to add the property to the library
4. Assign the new material to the part. Select the new material from the Materials in Model column. Click **Assign > Part**. Select the part. Click **OK > Yes**.

Exercise 3: Edit and Delete Materials

Introduction

In this demonstration, you will edit a material from the materials library and define temperature dependent properties. You will edit all the AL6061 in the Material Library and make Poisson's Ratio temperature dependent.

Objectives

After successfully completing this exercise, you will know how to:

- Edit existing materials from the library.

Scenario

Sometimes existing material requires editing for changes. Mechanica allows you to edit and change materials.

In this exercise, you edit a material properties file and define temperature dependent material properties.

Step 1. Edit AL6061 and make Poisson Ratio temperature dependent.

Assign AL6061 to the part. Edit AL6061 and make Poisson Ratio temperature dependent. Use Table 2 for data.

1. Assign the material AL6061 to the bracket part.
2. Edit the material AL6061 from the Materials in Model list.
3. Change Poisson's Ratio and make function of temperature. From the Material Definition dialog box, pick the function icon **f(x)** next to Poisson's Ratio.
4. In the Function Definition dialog box, name the function **Poisson_Function**, and the description **Temperature Dependent Poisson's Ratio**.
5. Edit the Type to **Table**.
6. Add 5 rows and enter table values shown below:

Table 2: Values for Poisson's Ratio.

Temperature	Poisson Ratio
60	0.31
70	0.32
80	0.33
90	0.34
100	0.35

7. Note the value of Poisson's Ratio is now Poisson_Function as you defined.
8. Place the material back in the library. Click **Yes** to overwrite.
9. Save the model.
10. Close the window and erase all models from session.

For PTC Internal Use Only

Summary

After successfully completing this module, you should know how to:

- Assign structural and thermal properties to parts.
- Define linear and nonlinear properties.
- Create materials libraries.
- Define temperature-dependent material properties.
- Edit and delete material properties.

For PTC Internal Use Only

Applying Constraints

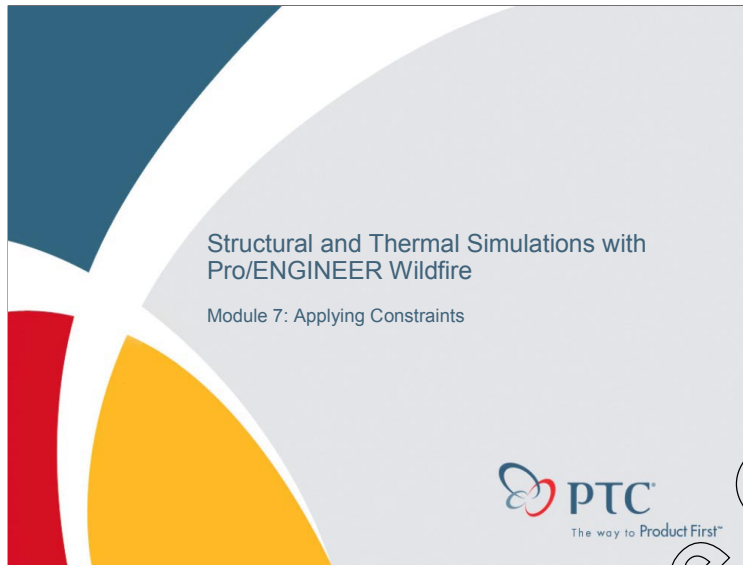
Introduction

Constraints and loads are configurable features within Mechanica that simulate the real-world environment that you expect your model to encounter. Mechanica uses this information to calculate the behavior of your model during analyses and sensitivity studies.

Objectives

After completing this module, you will be able to:

- Create different types of constraints.
- Constrain design models in Mechanica.
- Set the active coordinate system.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

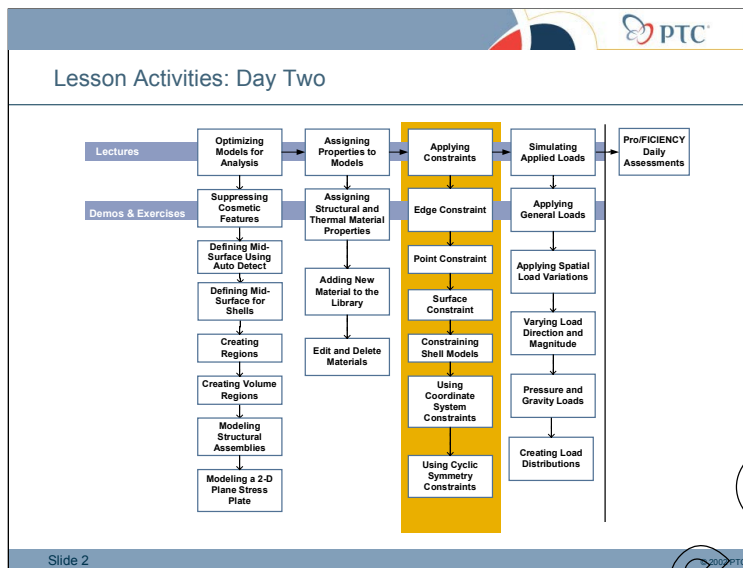
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 30 min
- Labs: 60 min
- Total: 1 hour 50 min



Objectives

After completing this module, you should be able to:

- Create different types of constraints.
- Constrain models in Mechanica.
- Set the active coordinate system.

Slide 3

Objectives

Constraints and loads are Mechanica entities that simulate the real-world environment that the model is expected to encounter. A model's stress and strain distributions depend on the constraints and loads applied to a model.



Applying Constraints

Constraints

- Constraining Degrees of Freedom
- Utilizing Symmetry
- Example application

Slide 4

There are no instructor notes on this slide.

PTC

Applying Constraints (cont.)

What is the goal?

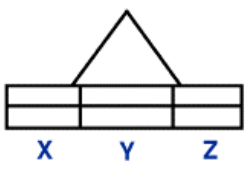
- Simulate the real world conditions.
- Eliminate rigid-body motion.

Apply Constraints to:

- Points
- Curves
- Surfaces

Degrees of Freedom (DOF)

- Translation & Rotation.
- Beams, Shells, Solids.
- Constrain DOFs or Enforce Displacements.



The diagram shows a truss structure consisting of a triangular top member supported by three vertical members. These vertical members are supported by three distinct points on a horizontal base, labeled X, Y, and Z from left to right. The entire structure is rendered in black lines on a white background.

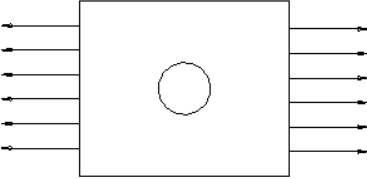
Slide 5

There are no instructor notes on this slide.

PTC

Example: Applying Constraints (cont.)

How do we constrain this model?



Slide 6

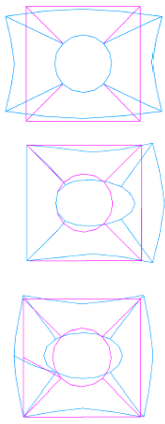
There are no instructor notes on this slide.

PTC

Example: Applying Constraints (cont.)

- ⦿ Constrain the hole.
- ⦿ Constrain the edge.
- ⦿ Constrain using three point technique or use symmetry.

Slide 7

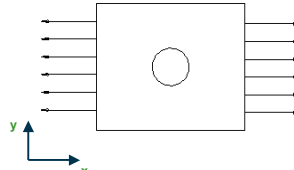


There are no instructor notes on this slide.

Applying Constraints (cont.)

Symmetry

- Use If:
 - Geometry
 - Properties
 - Loads
 - Constraints
- Are all symmetrical about the same cutting plane?
- Symmetry decreases:
 - Model size.
 - Your time.
 - Time to solve.
 - Disk space requirements.



Is there a plane of symmetry?

Slide 8

Constraints (cont.)

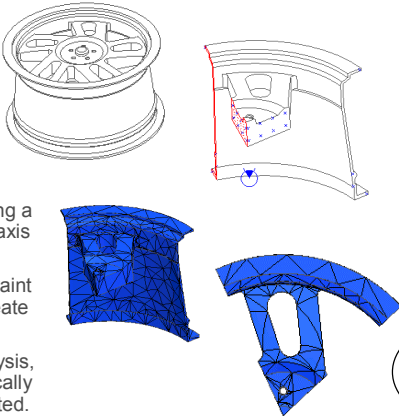
- Mechanics allows the definition of three types of constraints:
 - Point
 - Edge/Curve
 - Face/Surface
- The constraint definition dialog box contains the following fields:
 - **Constraint Set** : Constraints are grouped into constraint sets. This allows the user to define multiple constraints for the same model. Specify the *Set Name* to which this constraint belongs (a constraint can only belong to one Set). This is used later on at the analysis definition stage. Only one Constraint Set can be applied to an analysis.
 - **Fixed**: Specify which degrees of freedom are fixed for the selected entity.
 - **Free**: Specify which degrees of freedom are free for the selected entity
 - **Displacement**: Specify a prescribed displacement for the selected entity. This is useful when the input to the model is a known displacement.
- Based on the above fields, Mechanics will display an icon with flags (free or fixed) representing each degree of freedom for the selected entity.
- Constraints are applied in the current coordinate system. User defined coordinate systems can be used for complex orientations.
- Use regions (Integrated Mode) or surfaces (Independent Mode) to constrain only a portion of an entity.
- In Independent Mode, constraints should be applied to the geometry (the mesh can be regenerated during sensitivity and optimization runs). Note that in Integrated mode, the mesh is only created at run time, and therefore all modeling is done on the geometry.
- In Integrated Mode,
 1. When using shells, some surfaces will be compressed as pairs are defined. These surfaces can not be constrained.
 2. Constraints applied in part mode are not carried over in assembly mode.

PTC

Applying Constraints (cont.)

Cyclic Symmetry

- Use if the entire model:
 - Geometry
 - Properties
 - Loads
 - Constraints
- Can be reproduced by copying a pie shaped wedge about an axis an integer number of times.
- Apply cyclic symmetry constraint before using AutoGEM to create mesh.
- For modal and buckling analysis, any modes that are not cyclically symmetric will not be calculated.



Slide 9

Cyclic Symmetry

A cyclic symmetry constraint allows a section of a cyclically symmetric model to be analyzed, while simulating the behavior of the whole part or assembly. This relational constraint greatly reduces meshing and analysis time.

The original model (part or assembly) must exhibit cyclic symmetry, that is, the cut section can be copied about a common axis a specified number of times to reproduce the whole model. (The number of times must be an integer.) The model must exhibit cyclic symmetry in all of the following:

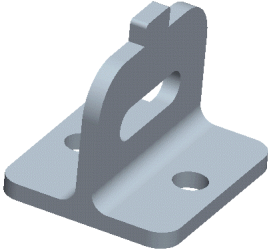
- Geometry
- Loads
- Constraints
- Material type and orientation

In Mechanics, a cyclic symmetry constraint prescribes rotation and displacement on two boundaries to be the same. In the Thermal module, a cyclic symmetry constraint prescribes the temperature distribution on two boundaries to be the same.

PTC

Applying Constraints (cont.)

- Constraints applied in current coordinate system.
- Use regions to constrain partial surfaces.
- Choose the appropriate level of detail.
- Apply in assembly mode.
- Don't apply to entities that get compressed away in shell models.



Slide 10

Assume - in the assembly, this bracket is bolted to a large block and a horizontal load is applied to the small tab at the top of the bracket. (a picture of such an assembly would be great)

We need to verify that the stress in the fillet will not exceed yield under this loading.

How should we constrain this model?

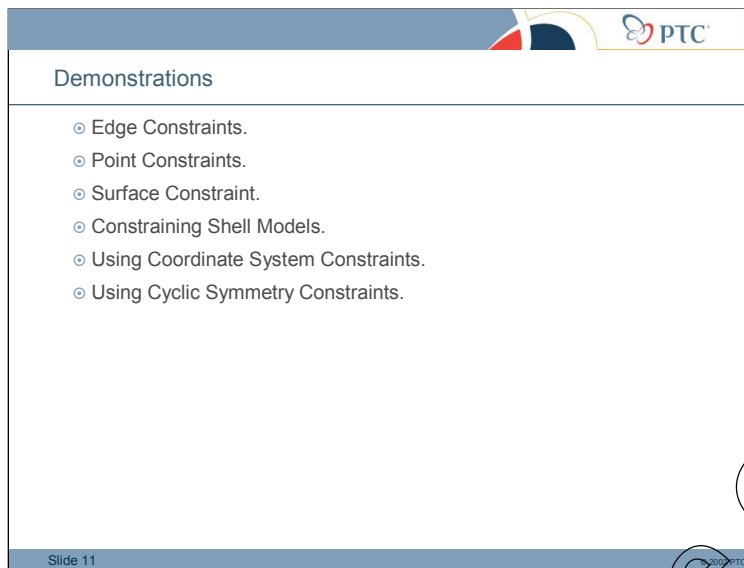
One choice – fully constrain inner hole surfaces – what will happen? Is this realistic for the behavior of the hole? No, we will see artificial stress concentrations. But, are we concerned with the stress around the hole? No. So this will probably be fine.

Choice two – if we are not concerned with the stress around the holes, we could even suppress the holes and use surface regions to replace them. Constrain the surface regions (perhaps top and bottom). Again, this will create artificial stress concentrations. Are we concerned about these?

Choice three – what if we are concerned with the stresses around the holes? How can we model this 'bolt-type' constraint?

The slide says that constraints are applied in a current coordinate system. Describe or demo what they will do in the exercises. This will allow the hole to flex much like it would in reality. This will reduce artificial stress concentrations around the hole.

The choice of which constraint scheme to use is up to you – what are you concerned about?



Demonstrations

In this demonstration, I will:

- Use edge and point constraints.
- Constrain a shell model in Mechanica.
- Set an active coordinate system.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- Create different types of constraints.
- Constrain models in Mechanica.
- Set the active coordinate system.

Slide 12

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 7 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Edge Constraint

Introduction

Before you can run an analysis, you need to constrain your model. The first step in applying constraints to your model is to select the type of entity you want to constrain. The next step is to fill out the dialog box that will determine how these entities will, or will not be allowed to move during the analysis. After the dialog box is filled out, a constraint icon that graphically represents the allowable movements will be attached to the entity and, at a glance, you can visually determine how your part is constrained.

In this demonstration, we will open a part, open Mechanica and add an edge constraint.

Objectives

After successfully completing this exercise, you will know how to:

- Add edge constraints.

Scenario

The first step in applying constraints to your model is to select the type of entity you want to constrain. The next step is to fill out the dialog box that will determine how these entities will, or will not be allowed to move during the analysis.

In this exercise, you will apply an edge constraint to remove some degrees of freedom from the model.

Step 1. Fix the edge of a plate from translating in the Z-direction and rotating with respect to the X and Y axes. All other Degrees Of Freedom (DOF) will be free.

Open the plate part and enable Mechanics Structure. Create an edge constraint.

1. Set your working directory to **C:\users\student\mechanica_330\module_07**.
2. Open PLATE.PRT.
3. Enable Mechanics Structure.
4. Create an edge constraint. Name the constraint **edge_constraint** and leave ConstraintSet1 as Set name.
5. Click to add the Curve reference as shown in the following figure.

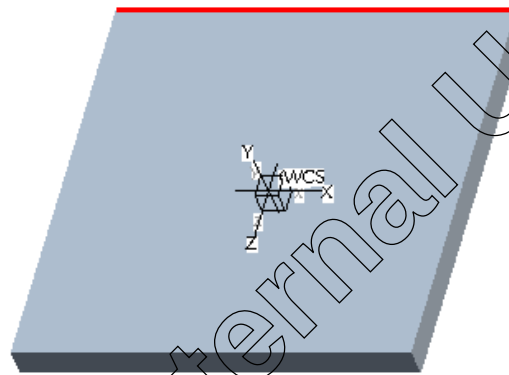


Figure 1: Edge constraint.

6. Ensure that the WCS is set as the coordinate system.
7. Select the appropriate icons to Free Translation in X, Translation in Y, and Rotation in Z. Fix all remaining Translation and Rotation directions. The dialog box should look as shown in the following figure.

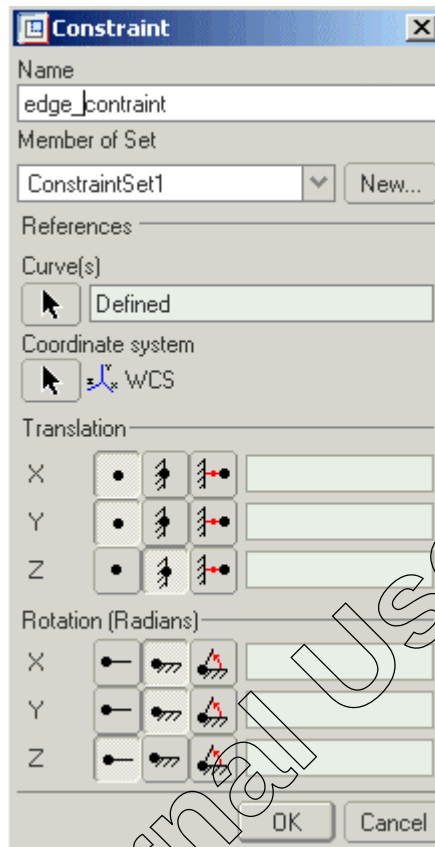


Figure 2: The finished constraint dialog-box.

Exercise 2: Point Constraint

Introduction

In this demonstration, we will open a part, open Mechanica, and add point constraints.

Objectives

After successfully completing this exercise, you will know how to:

- Add point constraints.

Scenario

In this exercise, you will apply point constraints to remove all six degrees of freedom from the model.

Step 1. Delete the edge constraint.

Delete the constraint you just created.

1. Delete the constraint you just created.

Step 2. Create datum points at the locations to constrain.

Create three datum points on vertices.

1. Create three datum points on vertices as shown in the following figure. Select **Model > Features > Datum Point > Create**.
2. Pick the three vertices as shown in the following figure.

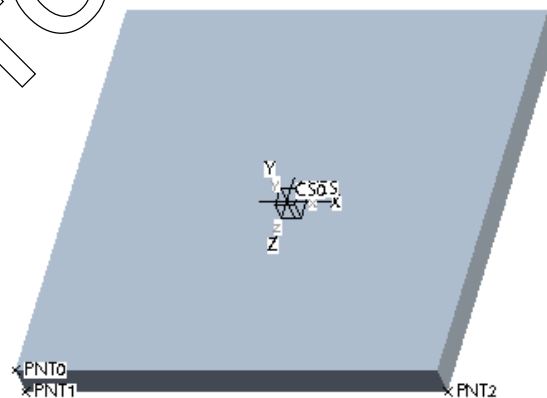


Figure 3: Datum points on vertices of the model.

Step 3. Constrain the first point.

Constrain PNT0 in all translational DOFs and leave rotational DOFs free.

1. Create a point constraint on PNT0, by clicking **Constraints > New > Point**.
2. Keep the default Constraint1 and ConstraintSet1 names.
3. Keep WCS as the coordinate system.
4. Fix all the Translational DOF and free all the Rotational DOF, as shown in the following figure. This constraint removes all Translational DOF, but the part is still free to rotate around the X, Y, and Z-axes.

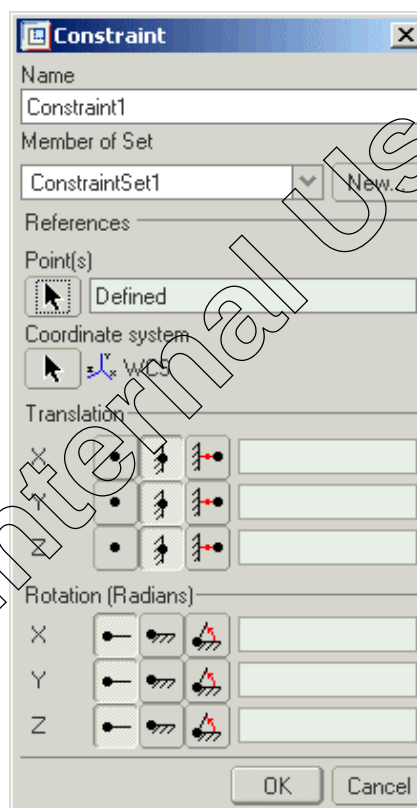


Figure 4: Constrain the first point.

Step 4. Constrain the second point.

Constrain PNT1 in X and Z translations and leave Y translation and all rotational DOF's free.

1. Create a point constraint on PNT1.
2. Keep the default Constraint2 and ConstraintSet1 names.
3. Keep WCS as the coordinate system.

4. Fix only the X and Z Translational DOF. Ensure that the Y Translation and all three rotational DOF remain free as shown in the following figure.

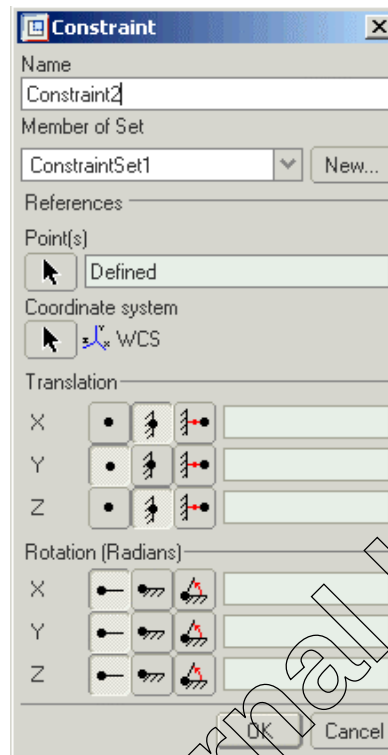


Figure 5: Constrain the second point.

Step 5. Create a point constraint for the third point.

Add a third constraint on PNT2. Fix point PNT2 only in the Z translation. (Even though this constraint scheme does not explicitly remove a rotational DOF, the two bottom points cannot translate in the Z direction, which removes rotation around the Y-Axis.)

1. Create a point constraint on PNT2.
2. Fix PNT2 only in the Z translation as shown in the following figure. (Even though this constraint scheme does not explicitly remove a rotational DOF, the two bottom points cannot translate in the Z direction, which removes rotation around the Y-Axis.)

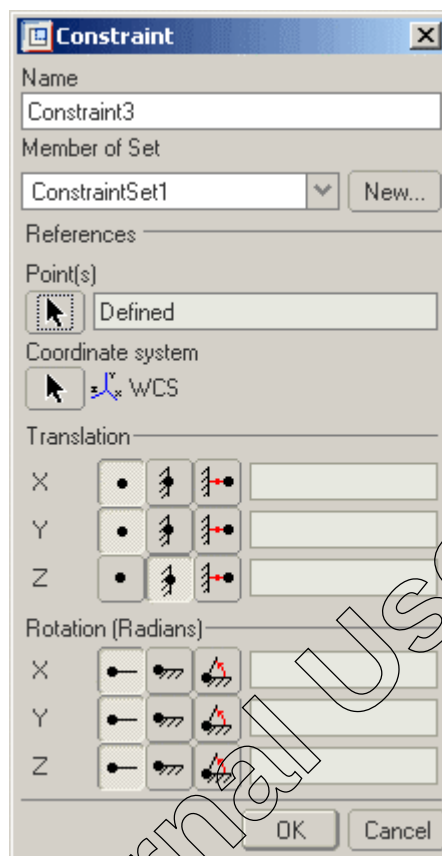


Figure 6: Constrain the third point.

Note:

The three translational constraints have removed all six DOF, including rotations, even though a rotational box was never explicitly fixed in any of the constraint dialog boxes. It is not necessary to explicitly fix each DOF in a constraint dialog box in order to remove all six DOF from a model.

Exercise 3: Surface Constraint

Introduction

In this demonstration, we will open a part, open Mechanica, and add a surface constraint.

Objectives

After successfully completing this exercise, you will know how to:

- Add surface constraints.

Scenario

In this exercise, you will apply a surface constraint to remove all translation degrees of freedom from the model.

Step 1. Delete the three point constraints.

1. Delete the constraints you just added.

Step 2. Create a surface constraint.

Create a surface constraint on the vertical sides of the plate. Fix all the translational DOFs. Leave the rotational DOF free.

1. Create a surface constraint on the vertical sides of the plate as shown in the following figure. Keep the default constraint and constrain set names.
2. Keep WCS as the coordinate system and fix all Translational DOF. Leave all Rotational DOF free.
3. Save the model.
4. Close the window.
5. Erase not displayed.

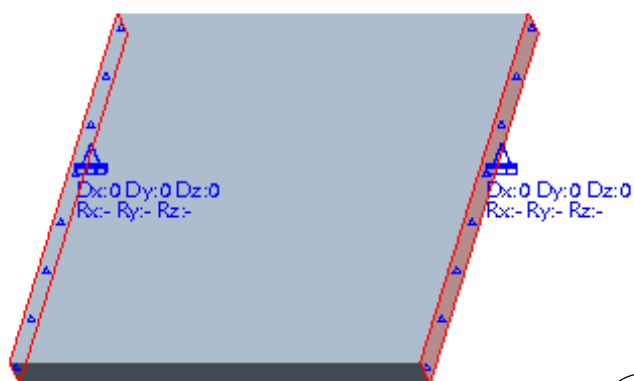


Figure 7: Surface Constraints

Exercise 4: Constraining Shell Models

Introduction

In this demonstration, we will open a part, open Mechanica and test the shell compression. Since a shell model is used, surfaces that are going to be compressed cannot be constrained, therefore you will add edge constraints.

Objectives

After successfully completing this exercise, you will know how to:

- Add edge constraints on shells.

Scenario

In this exercise, you will apply edge constraints to a model that is going to be compressed. Since the side surfaces will be compressed you can constrain them.

Step 1. Apply constraints to shell models using the T_BRACKET.PRT in order to hold the two ends of the T_BRACKET.PRT fixed in all DOF for the model.

Open T_BRACKET.PRT and verify that the units are mmNs.

Enter Mechanica Structure. And define the surface pairs for shell using auto detect and test the mid-plane compression.

1. Open T_BRACKET.PRT and verify that the units are **mmNs**.
2. Enable Mechanica Structure.
3. Define the surface pairs for shell mid-surface using auto detect and test the mid-plane compression. The model should appear as in the figure below.

Note:

The first step to constraining a part is to think about how this part will actually be held in place. Then, apply the appropriate constraints to the model. In this example, the part is welded on the right and left edges of the base into a larger assembly.

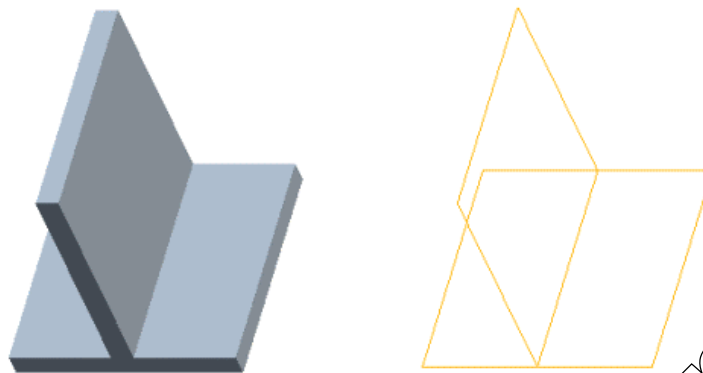


Figure 8: T_BRACKET.PRT.

Step 2. Fully constrain the two vertical surfaces of the T_BRACKET.PRT

Create an edge constraint on two edges one from each side surfaces. Keep the default constraint and constrain set names. Keep *WCS* as the coordinate system, fix all DOFs in the dialog box, and accept the dialog box.

1. Create an edge constraint on two edges one from each side surfaces by holding CTRL + left-click to select multiple edges.
2. Keep the default constraint and constrain set names. Keep *WCS* as the coordinate system and fix all DOFs in the dialog box and accept the dialog box.
3. Save the model.
4. Close the window.
5. Erase not displayed.

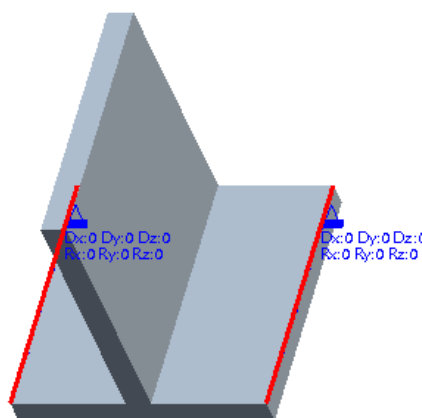


Figure 9: Edge Constraints

Exercise 5: Using Coordinate System Constraints

Introduction

In this demonstration, we will open a part and open Mechanica. Create a cylindrical coordinate system and add a constraint referencing the new coordinate system.

Objectives

After successfully completing this exercise, you will know how to:

- Create a coordinate system.
- Add a constraint and reference it to a coordinate system.

Scenario

Sometimes constraints are better represented as different types of coordinate systems such as Cartesian, cylindrical or spherical. In this exercise, you will create User Defined Coordinate Systems and apply constraints with respect to the new coordinate system.

Step 1. Create a cylindrical coordinate system aligned with the hole to define a constraint with respect to the new coordinate system.

Retrieve the T_BRACKET_HOLE.PRT and enter Mechanica Structure.

Create a cylindrical coordinate system. Use the axis of the first hole and the top edge of the plate as references.

Use the axis A1 to determine Z and the edge to project T=0. Keep the default coordinate system name.

1. Open T_BRACKET_HOLE.PRT.
2. Enable Mechanica Structure.
3. Create a cylindrical coordinate system. Select **Model > Features > Coord System > Create**.
4. From the **Origin** tab, left-click to select the axis of the first hole and hold CTRL + left-click to select the top edge of the plate as references as shown in the following figure. With the second reference, the location of the coordinate system is fully defined.

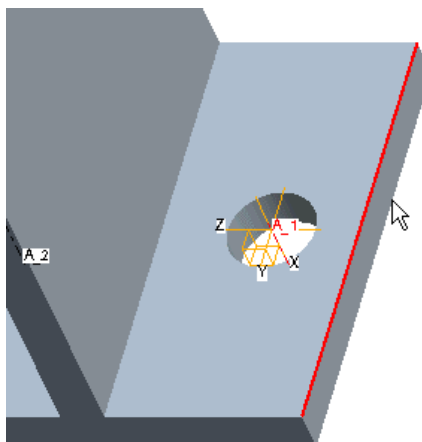


Figure 10: Edge and Axis to define the coordinate system.

5. Set the Type to **Cylindrical**. From the Orientation tab, select axis A.1 to determine Z and the edge to project T=0.
6. In the **Properties** tab, keep the default coordinate system name.

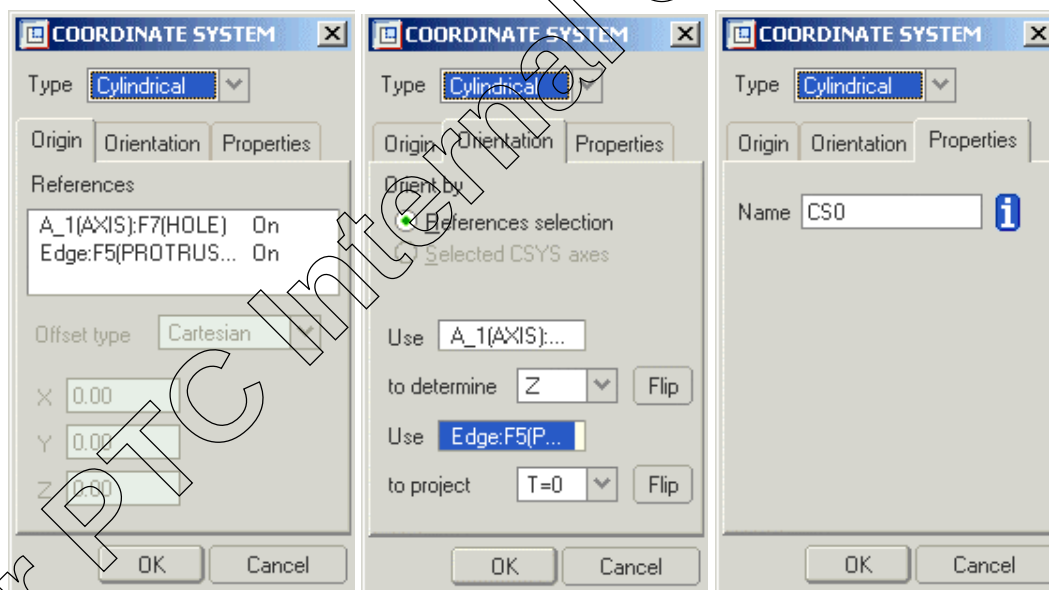


Figure 11: Datum point dialog box.

Step 2. Create another coordinate system on the other hole.

Repeat the above steps for the other hole to create CS1.

1. Repeat the above steps for the other hole to create CS1.
2. Flip the direction of the Z-axis as necessary, so that it has the same direction as CS0.

Step 3. Simulate bolted bracket by constraining the edges of the holes.

Create an edge constraint to constrain the two top edges that define the top of the right hole. Name it **bolt1**. and keep the default constraint set name. For coordinate system use CS0.

Free R and fix other Degrees of Freedom.

Constrain the second hole on the left hole using the same procedure and name it **bolt2**.

1. Create an edge constraint to constrain the two top edges that define the top of the right hole.
2. Name it **bolt1** and keep the default constraint set name. Assign CS0 as the coordinate system.

Note:

Constraints and Loads are always applied with respect to a current coordinate system by default. Any local coordinate system can be made current or referenced during the constraint creation.

3. Free Translation in R and fix all other Degrees of Freedom.
4. Create an edge constraint to constrain the two top edges that define the top of the left hole.
5. Name it **bolt2** and keep the default constraint set name. Assign CS1 as the coordinate system.
6. Free Translation in R and fix all other Degrees of Freedom.
7. Save the model.
8. Close the window and erase all models from session.

Exercise 6: Using Cyclic Symmetry Constraints

Introduction

In this demonstration, we will open a part and open Mechanica. Review the cut in the part to show the cyclic symmetry. Create a cyclic symmetry constraint.

Objectives

After successfully completing this exercise, you will know how to:

- Create a cyclic symmetry.
- Identify parts that exhibit cyclic symmetry.

Scenario

If the original model (part or assembly) exhibits cyclic symmetry, you can analyze a section of it instead of the whole model. The section used must reproduce the whole model if copied a certain times about a common axis.

In this exercise, you will create a cyclic symmetry constraint on a section of wheel.

Step 1. Create the cyclic symmetry constraint

Retrieve the part and review the cut to explain the cyclic symmetry.

1. Open WHEEL.PRT.
2. Edit the definition of the last cut feature. Quit the edit definition without changing the cut.

Note:

The wheel part is cyclically symmetric. The last cut is created to remove geometry, as shown in the following figure. The geometry you are going to perform analysis on is the one slice of the entire model.

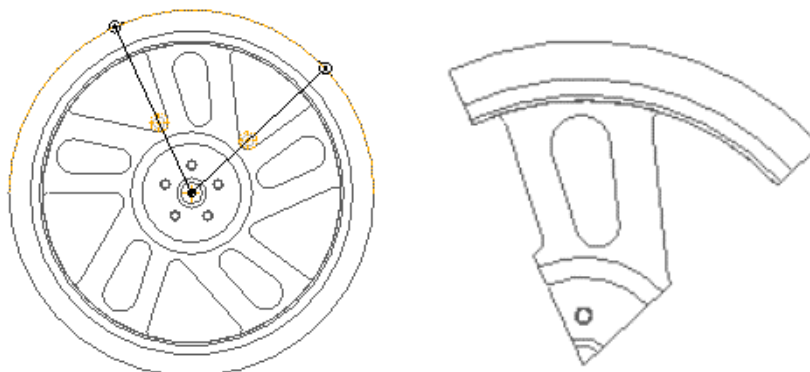


Figure 12: The cyclic symmetric wheel

Step 2. Create the cyclic symmetry constraint.

Enter Mechanics and check the units.

Create a new cyclic symmetry. For the first side, select the one pie cut surface as the first side and the second pie cut surface as the second side.

1. Enable Mechanics Structure.
2. Create a new cyclic symmetry. Select **Model > Constraints > New > Cyclic Symm**.
3. Select one pie cut surface as the first side and select the second pie cut surface as the second side, as shown in the following figure below. The axis is determined automatically; it is the intersection of these two surfaces.

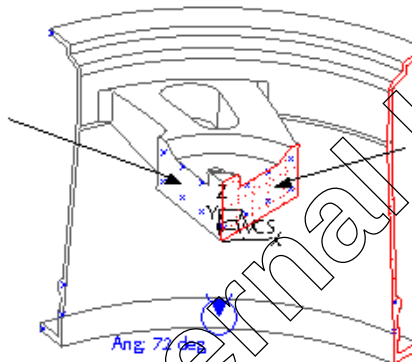


Figure 13: Surfaces for the cyclic symmetry constraint.

4. Save the model.
5. Close the window and erase all models from session.

Summary

After successfully completing this module, you should know how to:

- Create different types of constraints.
- Constrain models in Mechanica.
- Set the active coordinate system.

For PTC Internal Use Only

For PTC Internal Use Only

Simulating Applied Loads

Introduction

For Mechanica to perform most types of analyses, you must load at least one area of your model. Mechanica provides a wide variety of load types. The loads that Mechanica Structure supports the following loads:

- Point
- Edge/Curve
- Face/Surface
- Bearing
- Centrifugal
- Gravity
- Pressure
- Temperature

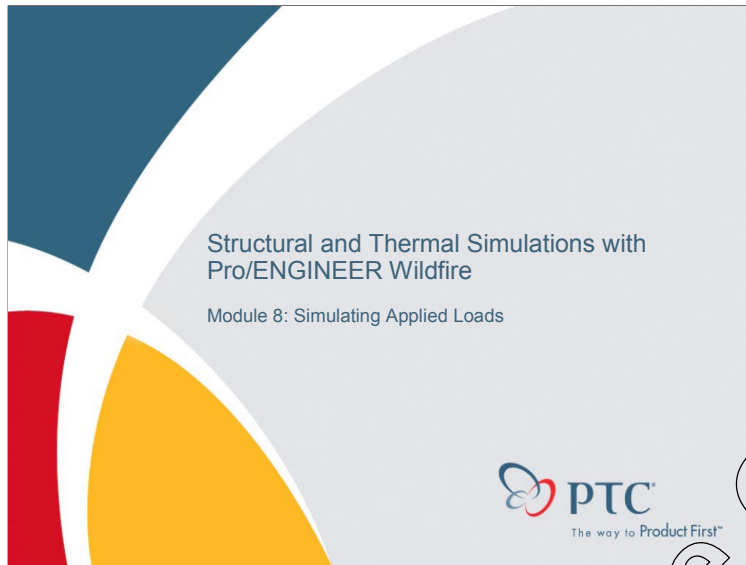
Mechanica Thermal supports:

- Heat Loads

Objectives

After completing this module, you will be able to:

- Create different loads.
- Describe the difference between point, edge, and surface loads.
- Describe when to use each load type.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

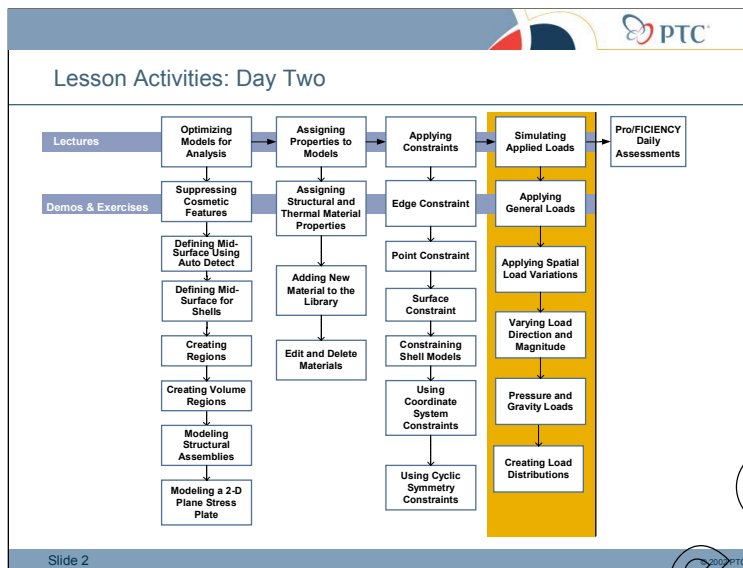
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 30 min
- Labs: 60 min
- Total: 1 hour 50 min



Objectives

After completing this module, you should be able to:

- Create different load types.
- Describe the difference between point, edge, and surface loads.
- Describe when to use each load type.

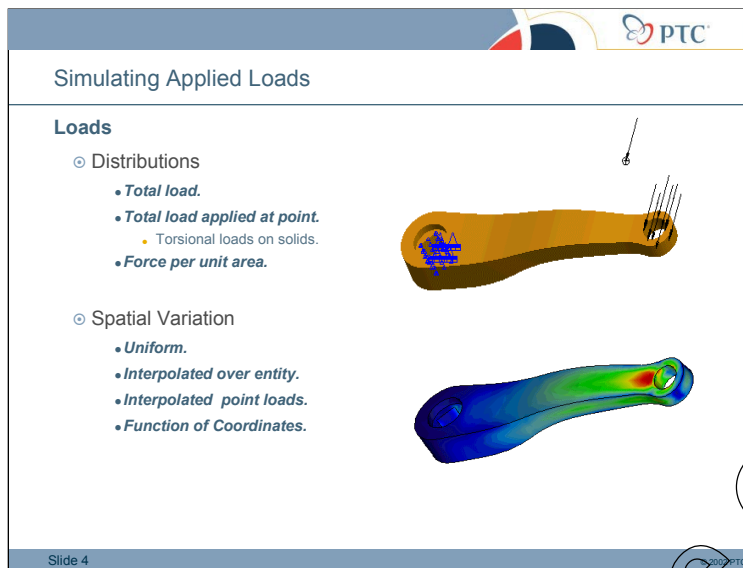
Slide 3

Objectives

Mechanica provides a wide variety of load types. Loads must be applied to at least one area of a model in order for Mechanica to perform most types of analyses.

In this module you will:

- Create Point, edge, and surface loads.
- Apply spatial variations to loads.
- Apply varying load directions and magnitude.
- Apply pressure and gravity loads.
- Differentiate between point, edge, and surface loads.



Loads

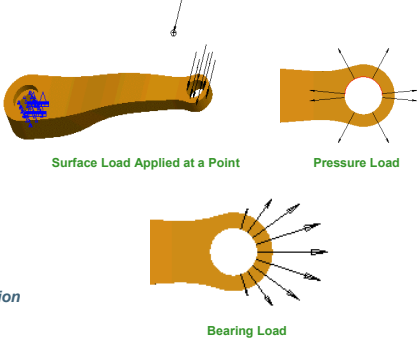
- Mechanics allows the definition of a variety of load types:
- Point, Edge/Curve, Face/Surface.
 - Bearing loads to simulate contact between cylindrical surfaces (Hertzian contact load distribution).
 - Centrifugal and gravity loads to simulate body forces.
 - Pressure loads (always perpendicular to the applied surface).
 - Temperature loads to compute thermal stresses.
- The load definition dialog box contains the following fields:
 - **Load Set** : Loads are grouped into load sets. This allows the user to isolate the effect on various loads on the model. Specify the Set name to which this load belongs (a load can only belong to one Set). This is used later on at the analysis definition stage. Multiple load sets can be applied to one analysis.
 - The **Distribution** field defines how the load magnitude is interpreted (mathematically):
 1. **Total load** - distributes the load along the length or area of the selected entity such that the integral of the load over the selected entity equals the total prescribed value.
 2. **Total load at point (TLAP)** – computes and applies the equivalent shear, moment and torsion of a point force acting at a specified distance from a surface.
 3. **Force per unit area/length** – allows the user to specify a load per unit length, or load per unit area.
 - The **Spatial Variation** field specifies how the load is applied over the entity:
 1. **Uniform** - The load has no spatial variation.
 2. **Interpolated Over Entity** – Interpolation points with an associated scaling factor are specified
 3. **Function of Coordinates** – The load is distributed as a function of a specified coordinate value in the current coordinate system.

PTC

Simulating Applied Loads (cont.)

Loads Types

- Point
- Edge/Curve
- Surface
- Pressure
- Bearing
- Gravity
- Centrifugal
 - Angular velocity
 - Angular acceleration
- Temperature
 - Global Temp
 - MEC/T Temp
 - External Temp



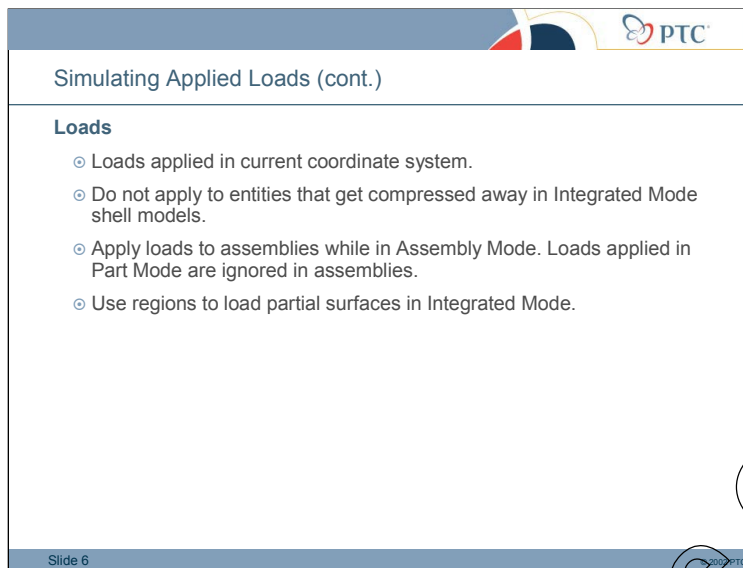
Surface Load Applied at a Point

Pressure Load

Bearing Load

Slide 5

There are no instructor notes on this slide.



Loads (cont.)

- Loads are applied in the current coordinate system. User defined coordinate systems can be used for complex load orientations.
- Use regions (Integrated Mode) or surfaces (Independent Mode) to load only a portion of an entity
- Unlike the “Force per Unit Area” option, the “Pressure” load type can only act normally to a surface.
- Note that in Integrated mode, the mesh is only created at run time, and therefore all modeling is done on the geometry.
- In **Integrated Mode**
 1. When using shells, some surfaces will be compressed as pairs are defined. These surfaces can not be loaded.
 2. Loads applied in part mode are not carried over in assembly mode (since parts are usually loaded to simulate the presence of other components).

Simulating Applied Loads (cont.)

Singularities

- Stress = Force/Area = Infinite (Singular)
 - If Area = 0 and F ≠ 0

Load and Constraint Types	Beams	Shells	Solids
Point	OK	No	No
Curve or Edge	OK	OK	No
Surface or Edge	N/A	OK	OK

- A load or constraint must act on an **AREA** to avoid singularity.

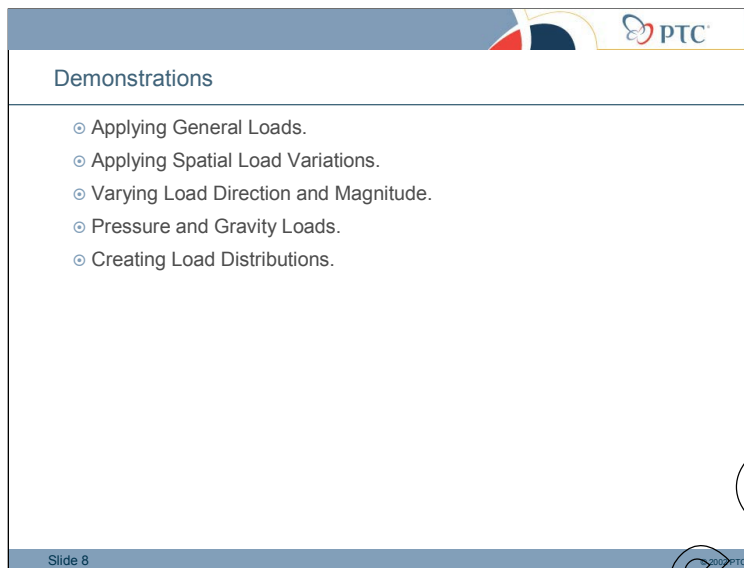
Slide 7

Singularities

- Stress can be defined as load over area: $s = F/A$. As a result, if any load is applied on an entity that has no area, the stress becomes infinite. This is called a singularity.
- Notice that singularities can be created by applying loads or constraints (applying a constraint to an entity creates a reaction force on that entity). We therefore have to ensure that loads and constraints are applied to entities which have area associated to them.
- The table specifies which load/constraint types are acceptable for each element type.

Load/Constraint Type	Beams	Shells	Solids
Point	Finite	∞	∞
Curve or Edge	Finite	Finite	∞
Surface or Face	N/A	Finite	Finite

- Point loads/constraints can only be applied on beam elements, where they are considered to act on the associated cross-section (defined as an element property). A point load on a shell or on a solid will create a singularity.
- Edge loads/constraints can be applied to beam elements (again, the cross-section property can accommodate loads) as well as to shell elements (where they are considered to act on the area defined by the edge and the shell thickness)
- An edge load/constraint on a solid element will create a singularity.
 - Surface loads can be applied to shell and solid element types (which both have area) but not to beam elements (no surface to pick).
- 3-Point constraints (refer to PCL 12) violate the above rules. Valid 3-Point constraints can get away with constraining points on solids and shells because they constrain locations where the net force is zero. Therefore, in the equation $s = F/A$, both F and A are zero and stress goes to zero (no singularity).



Demonstrations

In this demonstration, I will:

- Create point, edge, and surface loads.
- Apply spatial variations to loads.
- Apply varying load directions and magnitude.
- Apply pressure and gravity loads.
- Apply spatial variations to loads.

You will repeat the same steps in the lab exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 9

Precision Learning

- Learn: by listening to lectures, watching demos, and completing lab exercises.
- Assess: your progress with Pro/FICIENCY assessment questions.
- Improve: The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Note to Instructors:

Getting Started

- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Exams

For each course, Pro/FICIENCY exams are based upon the topics covered each day.

How to use it

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After completing this module, you should be able to:

- Create different load types.
- Describe the difference between point, edge, and surface loads.
- Describe when to use each load type.

Slide 10

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 8 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Applying General Loads

Introduction

To run any analysis, Mechanica requires load sets. In exercise 1, you apply Point, Edge, and Surface Loads to the T bracket part and show how to review and verify the created loads. In exercise 2, you investigate some of the spatial variations that you can assign to loads using linear, quadratic, and cubic interpolation options. In exercise 3, you explore how to apply a load using Direction and Magnitude. In exercise 4, you apply a pressure load that varies as a function of a coordinate system. You also apply a gravity load and finally in exercise 5, you will create a load distribution using the Total Load at Point Option.

Objectives

After successfully completing this exercise, you will know how to:

- To create and review Point, Edge, and Surface Loads. You also review and verify the created loads.

Scenario

In this exercise, you practice applying various loads as Point, Edge, and Surface Loads to T-Bracket. You also review the created loads.

Step 1. Open T_BRACKET_HOLE.PRT and create datum points for loads creation.

First, set the correct working directory.

Open the T bracket and enter Mechanica Structure.

Create the four points on vertices as shown in the following figure.

1. Set your working directory to **C:\users\student\mechanica_330\module_08**.
2. Open **T_BRACKET_HOLE.PRT**.
3. Enable **Mechanica Structure**.
4. Turn off the display of the constraint symbols.
 - Click **View > Simulation Display** in the main menu.
 - Select the **Loads/Constraints** tab and unselect the check box **Constraints**.
 - Click **OK**.
5. Create four datum points on the vertices as shown in the following figure.

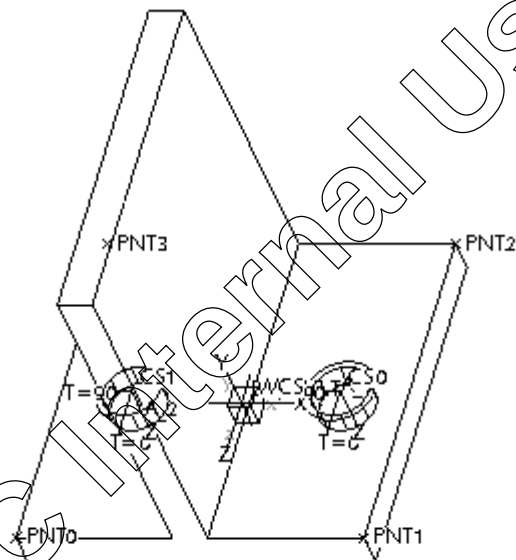


Figure 1: Points Created For Applying the Load

Step 2. Create a point load.

Create a new point load on **PNT0** named **Point0**. Place it in a new load set named **point_load**. Keep the default coordinate system **WCS**. The **Y** component of the load is **-10**.

1. Create a new point load named **Point0**, by clicking **Loads > New > Point**.
2. Edit the name of the load set to **point_load**.
3. For the point reference, select **PNT0**.
4. Keep **WCS** as the default coordinate system.

5. Set the Y component of the load to **-10**. Click **OK**. Note the Load icon at point PNT0.

Note:

Point and curve loads can introduce theoretically infinite stresses and distort your results for both solid and shell models.

To avoid stress concentrations, you can define regions and apply the load to the region instead of the point or curve.

Step 3. Review the load you applied to the T_BRACKET_HOLE.PRT.

Review the load just created on PNT0. Notice that FY is -10 and the moments are all zero due to the zero moment arm.

1. Review the load just created. Click **Rev Tot Load** from LOADS menu.
2. When prompted for the point, select **PNT0**. When prompted for a load to review, select the **point_load** from the model. Click **OK**.
3. The Load Resultant dialog box is displayed. Notice that the moments are zero, because the load passes directly through the point. Close the dialog box.

Repeat the same step for the other points PNT1, PNT2, and PNT3.

4. Repeat reviewing the resultant load at PNT1, PNT2, and PNT3 due to the **point_load**. For example, select **PNT1** and then **point_load**. Then select **OK**.
5. Notice that FY is -10 for all the points. The values of the resultant moments are different at each point of the moment arm.

Step 4. Apply an edge load.

Create a new edge load named edge_load1 on the edge as shown in the following figure. The X component of the load is -1 the rest are zero. Place it in a new load set named edge_load. Keep the default coordinate system WCS.

1. Create a new edge load named **edge_load1**, by clicking **Loads > New > Edge/Curve**.
2. Edit the name of the load set to **edge_load**.
3. Select the edge as shown in the following figure.

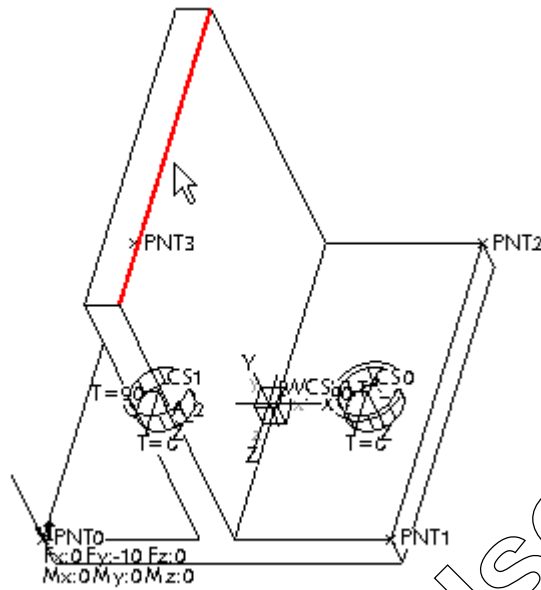


Figure 2: Apply load at the indicated edge.

4. Keep the default coordinate system WCS.
5. Set the X component of the load to **-1**. The total load of 1 N is uniformly distributed over the entire edge, producing a unit load of 0.004 N/mm. (The edge length is 250 mm.)
6. Preview the load.
7. Complete the load application and close the dialog box.

Exercise 2: Applying Spatial Load Variations

Objectives

After successfully completing this exercise, you will know how to:

- Create spatial loads variations using linear, quadratic, and cubic interpolation options.

Scenario

In this exercise, you investigate some of the spatial variations that you can assign to loads using linear, quadratic, and cubic interpolation options.

Step 1. Edit the edge load to linearly interpolate over the edge length.

Redefine the edge load created previously and change the distribution from Uniform to Interpolated Over Entity. Define the Interpolation by entering [0] for point 1 and [1] for point 2.

Preview the load, it should load appears as shown in the following figure.

1. Edit the edge load.
2. Change the Uniform load distribution to **Interpolated Over Entity**, from the drop-down list.
3. Select **define** to enter the Interpolation values. Enter **0** for point 1 and **1** for point 2.
4. Preview the load as shown in the following figure.

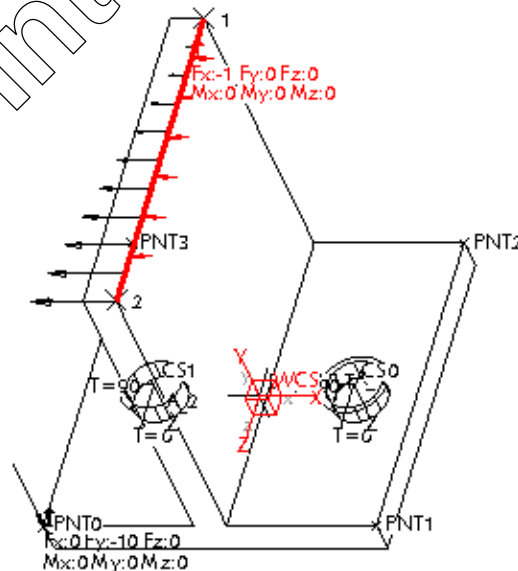


Figure 3: Interpolate over two points.

Step 2. Review the resultant load at the 4 datum points.

Review the edge load on any of the four datum points. The Load Resultant dialog-box should report $FX = -1$. Try reviewing the load at the other datum points. FX should remain a constant -1 , but the moments will vary.

1. Review the edge load on any of the four datum points.
2. The Load Resultant dialog-box should report $FX = -1$. Try reviewing the load at the other datum points. FX should remain a constant -1 , but the moments will vary.

Step 3. Create three additional datum points on the edge to use as interpolation points.

*Create three points on the edge you just applied the load to. Use offset ratio of **0.25**, **0.5** and **0.75**.*

1. Create a datum point on the edge you just applied the load to.
2. Enter **0.25** for the offset ratio.

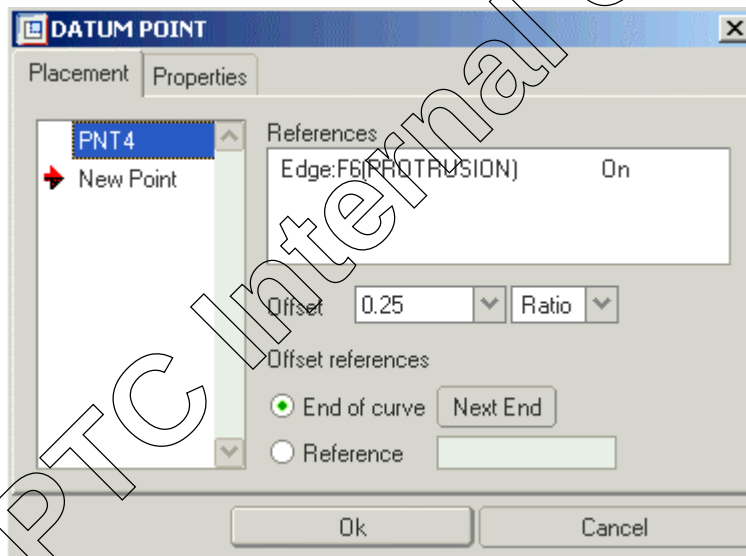


Figure 4: Datum point dialog box

3. Repeat the previous steps to create points at **0.5** ratio and **0.75** ratio along the curve as shown in the following figure.

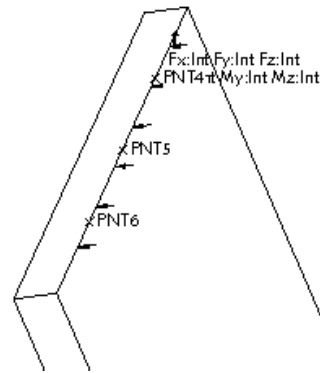


Figure 5: Create three interpolation points.

Step 4. Edit the edge_load to change its interpolation scheme to quadratic.

Redefine the edge load again and define the Interpolation. Add the center point PNT5 and enter [0], [0], [1] for the interpolation point values. Preview the load and note the parabolic distribution of the load along the edge.

1. Edit the edge load.
2. Click **Define** to change the Interpolation. Click **Add** and select the center point PNT5.
3. Enter **0, 0, 1** for the interpolation point values.
4. Preview the load and note the parabolic distribution of the load along the edge as shown in the following figure.

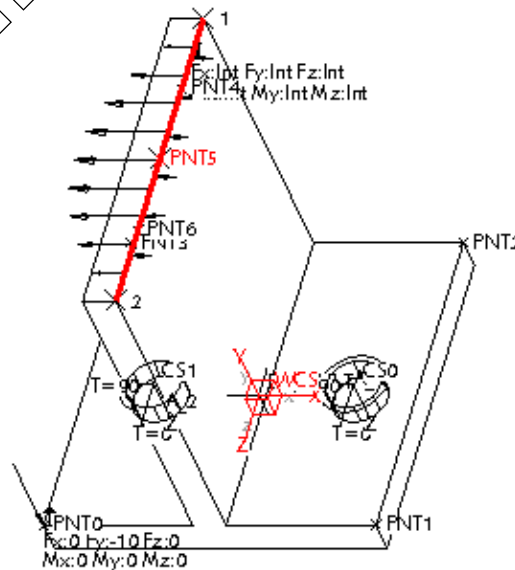


Figure 6: Parabolic distribution.

Step 5. Add additional datum points to the edge.

To develop a cubic interpolation, add additional datum points to the edge. Add PNT4 to the interpolation. Set the values for the four remaining points as shown in the figure below. Preview the distribution.

1. Add PNT4 to the interpolation using the procedure described above.

Try and add PNT6. Only 4 points are allowed for interpolated loads. Read the error message.

2. Click **Add** to add PNT6 to the interpolation. Read the message in the message window. Click **OK** to close the message window.

Note:

When you try to add the 5th interpolation point, you get an error message. Only 4 points are allowed for interpolated loads.

3. Set the values for the four remaining points as **0, -1, 0, 1**.
4. Preview the cubic distribution as shown in the following figure. Click **OK**. Close all dialog boxes.

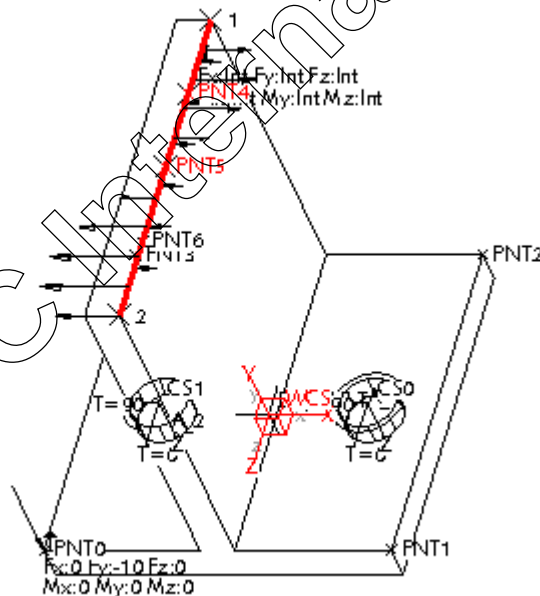


Figure 7: Define a cubic interpolation.

Note:

For the Interpolated option of Spatial Variation, specifying two points results in a linear interpolation, three points in a quadratic interpolation, and four points in a cubic interpolation.

Exercise 3: Varying Load Direction and Magnitude

Objectives

After successfully completing this exercise, you will know how to:

- Apply a load using Direction and Magnitude.

Scenario

In this exercise, you explore how to apply a load using Direction and Magnitude. You will edit the edge load to be at 45 degrees to the XZ plane, and to have a magnitude of 10.

Step 1. Enter the edge_load directions and then input a magnitude in the Load dialog-box.

Change the force from Components to Direction Vector and & Magnitude and enter -1 for FX, 1 for FY and 10 for magnitude.

Review the load created on any of the datum point. The Load Resultant dialog-box should report $F_x = -7.071$ and $F_y = 7.071$ N.

1. Click **Loads > Edit**. Select the load **edge_load1** from the model.
2. Change the Force from Components to **Dir Vector & Mag**.
3. Enter -1 for X, 1 for Y and 10 for the Magnitude.
4. Preview the resulting load distribution, and note how it is now at a 45° angle to the XZ plane as shown in the following figure. Click **OK**.

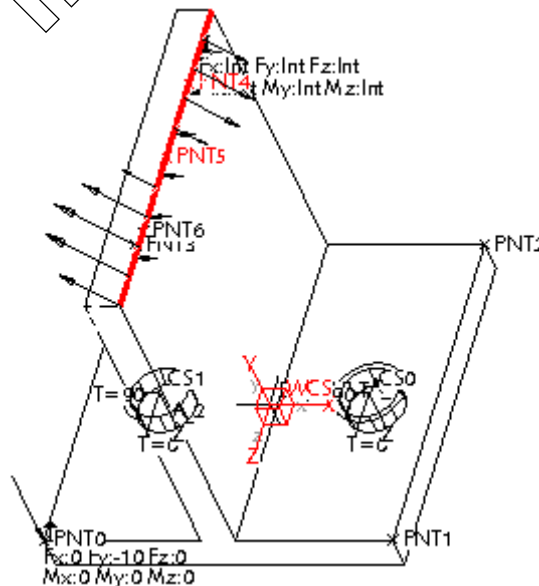


Figure 8: Preview the Load Distribution

Note:

If you defined this same load using Componets, the components would be:

$$F_x = -F_y = -\sqrt{\frac{(10)^2}{2}} = -7.071$$

This can be verified by measuring the resultant of this load at any location on the model.

5. Review the load on any datum point with **Rev Tot Load**. The Load Resultant dialog-box should report $F_x = -7.071$ and $F_y = 7.071$ N.
6. Close all dialog boxes.

Exercise 4: Pressure and Gravity Loads

Objectives

After successfully completing this exercise, you will know how to:

- Apply a pressure load that varies as a function of a coordinate system.
- Apply a gravity load.

Scenario

In this exercise, you apply a pressure load on the vertical plate and a gravity load on the entire model.

Step 1. Apply a pressure load on the right side of the vertical plate.

Create a new pressure load on the surface shown below: The pressure magnitude is 1. Name the load `surface_load1` and place it in a new load set named `surface_load`.

Define a new distribution function. Name it `y_squared` and enter the function y^2 as symbolic.

1. Click **Loads > New > Pressure** to create a new pressure load named **surface_load1** on the surface shown in the following figure.

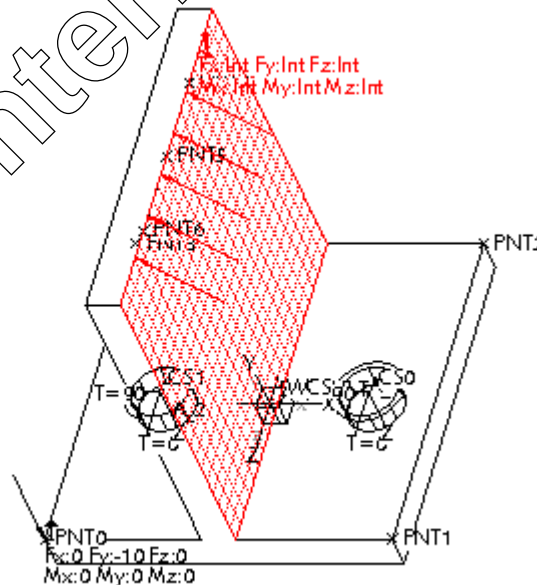


Figure 9: Surface for pressure load.

2. Edit the name of the load set to **surface_load**.
3. Select the surface as shown in the previous figure.

4. From the Distribution drop-down list, click **Function of Coordinates**.
5. Enter a magnitude of **1** for the P.
6. Define a distribution function by selecting **$f(x)$** . Create a new function named **y_squared**.
7. Select **Symbolic** as the type and enter **y^2** as the function as shown in the following figure.

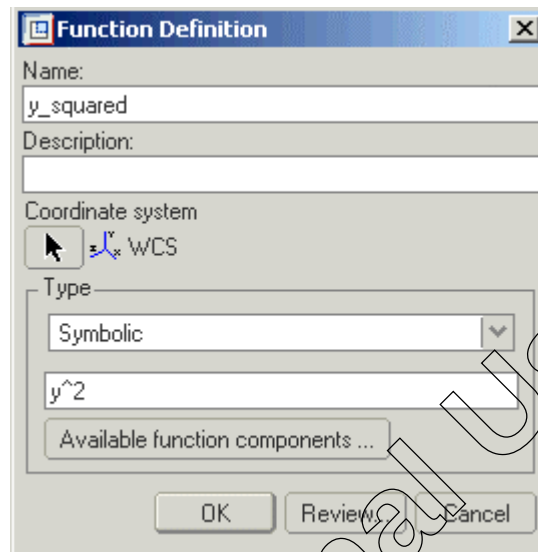


Figure 10: Define a load function.

Step 2. Review the function.

Review the function from 12.5 to 250. Show how:

Zoom in on a section of the graph, display the full graph and display the exact values of a point.

1. **Review** the function. Enter **12.5** and **250** respectively for the lower and for the upper limit.
2. Display the distribution by clicking **Graph**.
3. Zoom in and expand a portion of the graph.
4. Display the full graph using **refit**.
5. Display the exact value at a specific point by simply selecting the point from the graph.
6. Exit the graph. Click **Done** on the Graph Function dialog box.
7. Click **OK > OK** to close the Function dialog boxes.
8. Preview the load distribution should look as shown in the following figure. Click **OK**.

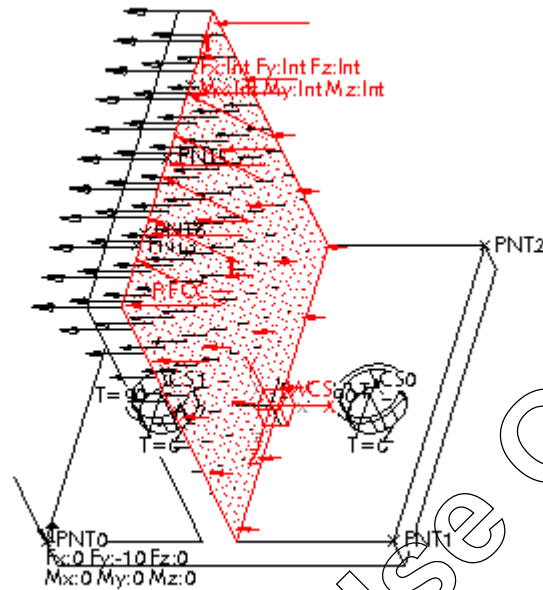


Figure 11: Preview the pressure load.

Step 3. Apply a gravity load to the T_BRACKET_HOLE.PRT.

Create a gravity load and enter -9810 in the y direction.

Note:

In a lot of cases, the gravity load is negligible in magnitude relative to the applied loads; however, vibration analyses, in particular, often rely on heightened gravity loads to simulate drop or shock conditions.

1. Click **Loads > New > Gravity** to create a new gravity load named **gravity1**.
2. Edit the name of the load set to **gravity**.
3. Enter **-9810** for Y direction. In SI units the gravity is 9.81 m/s^2 . Note the units in the part are mmNs, the gravity should be entered as 9810 mm/s^2 . Notice the addition of the G icon at the default coordinate system origin.
4. Click **OK > Done/Return**.
5. Save the model.
6. Close the window and erase all models from session.

Exercise 5: Creating Load Distributions

Objectives

After successfully completing this exercise, you will know how to:

- Create a load distribution using the Total Load at Point option.

Scenario

In this exercise, you will create a load distribution using the Total Load at Point option.

Step 1. Retrieve CRANK.PRT and create datum points on the crank to prepare for load creation.

Retrieve the crank part and enter Mechanica Structure.

Create a datum point at the center of the curve as shown in the following figure. Use the datum point icon.

Create another point offset normal from the surface as shown in the following figure and the first point. The offset value is 3.

1. Open CRANK.PRT.
2. Enable Mechanica Structure.
3. Create a datum point at the center of the curve using the Datum Point Tool as shown in the following figure.

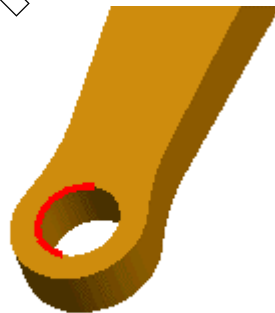


Figure 12: Edge for datum point creation.

4. Create a second datum point by selecting PNT0 and holding CTRL + left-click on the surface.
5. Enter **3** as the offset.

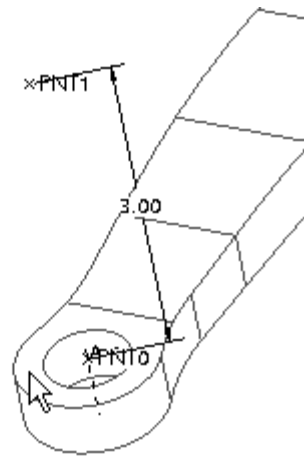


Figure 13: Select the indicated surface as the offset reference.

Step 2. Create a surface load with the Total Load at Point DISTRIBUTION option.

Create a new surface load on the surface shown in Figure . The load is total load at a point PNT1 with a Z magnitude of 350.

1. Create a new surface load on the surface shown in the following figure.



Figure 14: Select the load surface.

2. Accept the default world coordinate system WCS.
3. Click **Total Load at Point** from the Distribution drop down list and select PNT1 as shown in the following figure.

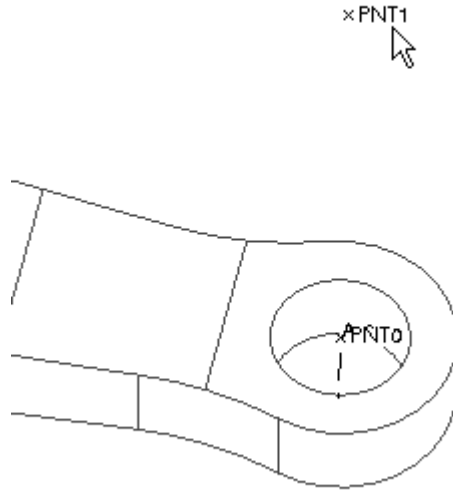


Figure 15: Select the point for load distribution.

4. Enter **350** for the FZ.
5. Click **OK > Done/Return**.
6. Save the model.
7. Close the window and erase all models from session.

Summary

After completing this module, you will be able to:

- Create different loads.
- Describe the difference between point, edge, and surface loads.
- Describe when to use each load type.

For PTC Internal Use Only

For PTC Internal Use Only

Creating and Running Structural Analyses

Introduction

An analysis is the calculation of a model's response to a loading condition and its boundary conditions. Mechanical provides several types of analyses that span a wide range of actual loading and boundary conditions.

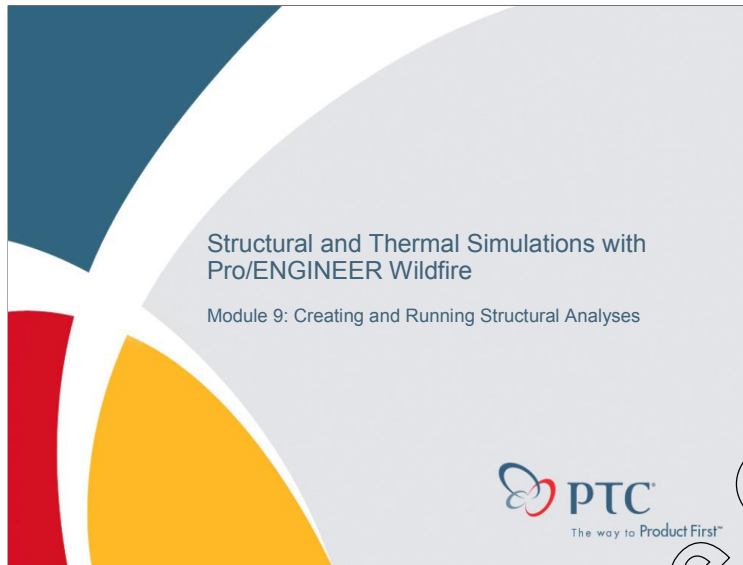
Structural analyses are performed on a structural product to find the stress and displacement distribution over a structure. Thermal analyses perform heat transfer calculations using heat loads and boundary conditions to determine all heat fluxes and temperature distribution throughout a structure.

In most instances, we are not only concerned with thermal stresses, but Mechanical stresses as well. You will combine both thermal and Mechanical loads into a structural analysis so that you may examine the combined effects of both loading conditions.

Objectives

After completing this module, you will be able to:

- Setup models for structural and thermal analyses.
- Run structural and thermal analyses and view the results.
- Transfer thermal loads into structure to determine thermal stresses.
- Create thermal/mechanical and gravity loads.
- Create a static analysis with multiple load sets.
- Create results windows with different combinations of load sets.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

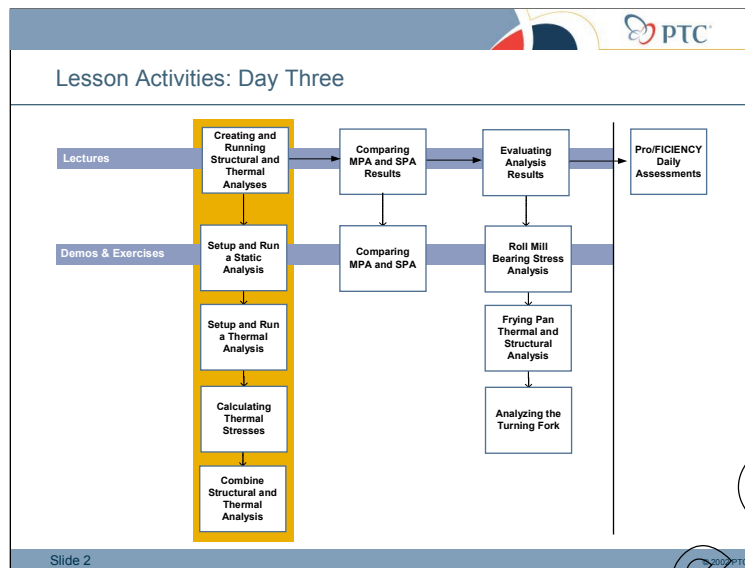
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 30 min
- Labs: 90 min
- Total: 2 hours 20 min



Objectives

After completing this module, you should be able to:

- Setup models for structural and thermal analyses.
- Run structural and thermal analyses and view the results.
- Transfer thermal loads into structure to determine thermal stresses.
- Create thermal/mechanical and gravity loads.
- Create a static analysis with multiple load sets.
- Create results windows with different combinations of load sets.

Slide 3

Objectives

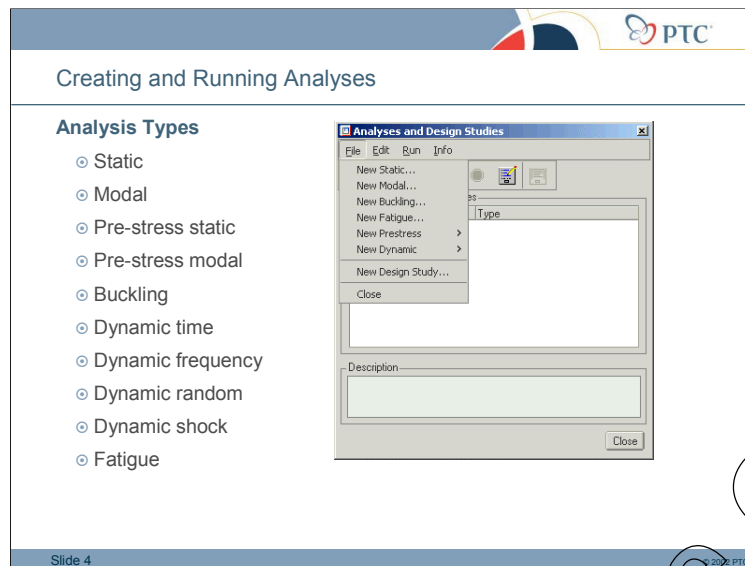
A Structural analysis is the calculations of a model's response to a loading condition and its boundary conditions. Mechanics provides several types of analyses that span a wide range of actual loading and boundary conditions.

Static analysis is performed on a structural product to find the stress and displacement distribution over a structure.

Mechanics will be used to investigate the performance of an industrial controller module, which has had reliability problems due to the CPU chip overheating.

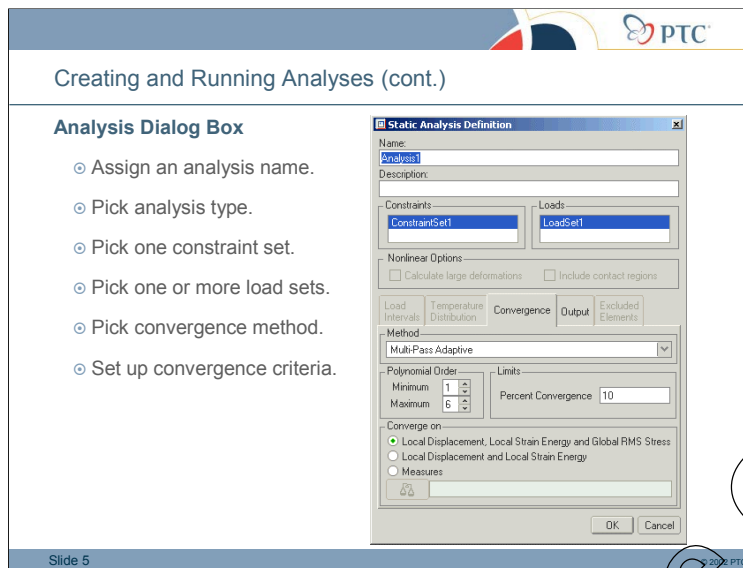
The controller module consists of a printed circuit board (PCB) with a CPU, several ICs, and some on-board power-supply components - an AC power connector, transformer, and a small vertical heat sink for a voltage regulator. Many electrical components are not included in the model - only those that generate, conduct, absorb or dissipate significant amounts of heat are important for the analysis.

First, a structural analysis will be performed to evaluate the stress due to the weight.



Analysis Types

- **Static:** Calculates deformations, stresses and reactions in response to specified loads and subject to specified constraints.
- **Modal:** Calculates the natural frequencies and mode shapes of your model.
- **Pre-Stress Static:** Enables the user to account for the presence of pre-stress in a structure under static load (pre-stress can strengthen or weaken a structure). A Static analysis must be performed prior to the Pre-Stress analysis.
- **Pre-Stress Modal:** Enables the user to account for the presence of pre-stress in the determination of natural frequencies and mode shapes of a structure.
- **Buckling:** Uses results from a previously defined static analysis to determine the critical magnitudes of loads at which a structure will buckle, as well as the model's stresses, strains, and deformations at the onset of buckling. Note that post-buckling behavior can not be modeled using this analysis type.
- **Contact:** Calculates deformation, stresses, strain, contact pressures and total contact area in response to specified loads and subject to specified constraints. Contact regions must be defined prior to running the analysis. Note that this is a non- linear analysis as the contact area (and therefore stress) varies with the applied load.



Analysis Form

The analysis definition form requires the following input:

- **Analysis Name:** This name will be used to create the results directory.
- **Type:** There are ten different analysis types available in Mechanica Structure. These analysis types are described in the next slide.
- **Constraint Set:** Select a single constraint set for this analysis. Only one constraint set can be used.
- **Load Set:** Select one or more load sets for this analysis.
- **Convergence Method:** Pick a convergence method, SPA or MPA. These will be explained shortly.
- **Convergence Criteria:** Specify a percentage for convergence and a set of quantities on which to converge.
- **Excluded Elements:** This option is only available in Independent mode. It allows the user to exclude troublesome elements from the convergence process, thereby ignoring known singularities (high-stress areas).
- **Polynomial Order:** Specify the minimum and maximum polynomial order for this analysis. This can be used to stop an analysis at a particular p-level, inspect the results and then restart it to complete the convergence process.
- **Plotting Grid:** Specify the number of intervals along edges at which results will be computed. This topic will be covered in detail in slide A&R-7.
- **Calculate:** This option allows the user to toggle "off" unwanted results.

PTC

Creating and Running Analyses (cont.)

Analyses Types:

- Static
 - Calculates deformations, stresses, and reactions in response to specified loads and constraints.
- Modal
 - Calculates natural frequencies and mode shapes of model.
- Pre-stress Static
- Pre-stress Modal



Slide 6

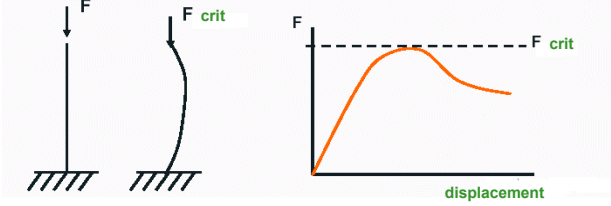
Pre-stress Static Type Analysis

- Accounts for the presence of pre-stress in a structure under static load. Must perform a static analysis first.
- Accounts for the presence of pre-stress in the determination of natural frequency and mode shapes of a structure. Must perform a modal analysis first.

PTC

Creating and Running Analyses (cont.)

- ⊙ Buckling
 - Uses results from a static analysis to determine the critical magnitudes of loads at which a structure will buckle, as well as the model's stresses, strains, and deformations at onset of buckling.
 - No post-buckling behavior.



Slide 7

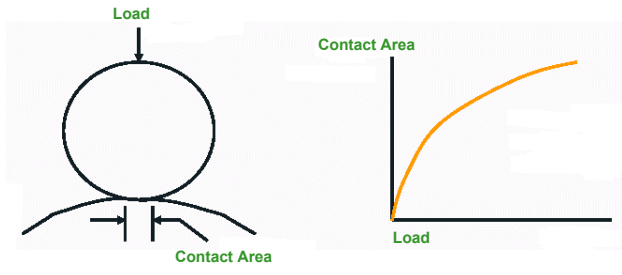
Buckling Type Analysis

- Non-linear.
- Uses results from a static analysis to determine the critical magnitudes of loads at which a structure will buckle, as well as the model's stresses, strains, and deformations at onset of buckling.
- No post-buckling behavior.

PTC

Creating and Running Analyses (cont.)

- Contact:
 - Calculates deformation, stresses, strain, contact pressures, and total contact area in response to specified loads and subject to specified constraints.
 - Contact regions must be defined prior to running the analysis.



The diagram on the left shows a sphere being compressed by a downward force labeled 'Load'. The resulting flattened region at the bottom is labeled 'Contact Area'. The graph on the right plots 'Contact Area' on the vertical axis against 'Load' on the horizontal axis. The curve starts at the origin and rises with a decreasing slope, representing a non-linear relationship.

Slide 8

Contact Type Analysis

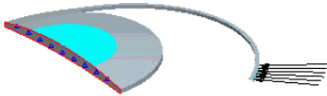
- Non-linear.
- Calculates deformation, stresses, strain, contact pressures, and total contact area in response to specified loads and subject to specified constraints.
- Contact regions must be defined prior to running the analysis.
- As load increases, contact area changes non-linearly.

PTC

Creating and Running Analyses (cont.)

Large Deformation Analysis

- **Use if:**
 - Deformation due to load is large
 - Small Strain
- **Only available for Static analysis.**
- **Must use 3D or 2D solid elements.**
- **Only models with isotropic material properties allowed.**
- **No temperature dependent properties are allowed.**
- **Loads must be with respect to WCS.**
- **No bearing loads or loads as a function of coordinates are allowed.**
- **Loads should not be near critical buckling loads.**



Slide 9

There are no instructor notes on this slide.

PTC

Creating and Running Analyses (cont.)

Dynamic Type Analyses

- Dynamic Time
 - Calculates displacements, velocities, accelerations, and stresses in the model in response to time-varying loads (transient response).
- Dynamic Frequency
 - Calculates amplitude and phase displacements, velocities, accelerations, and stresses in response to cyclic loading.
- Dynamic Random
 - Calculates RMS (root-mean square) responses based on loading conditions which use a statistical random function. Input is Power Spectral Density plot.
- Dynamic Shock
 - Calculates maximum values of displacements and stresses in response to base excitation (seismic).

Slide 10

Dynamic Analysis Type

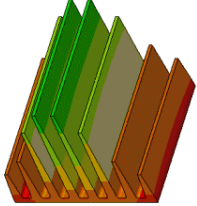
- Dynamic Time - Calculates displacements, velocities, accelerations, and stresses in the model in response to time-varying loads (transient response).
- Frequency - Calculates amplitude and phase displacements, velocities, accelerations, and stresses in response to cyclic loading.
- Random – Calculates RMS (root-mean square) responses based on loading conditions which use a statistical random function. Input is power spectral density plot.
- Shock - Calculates maximum values of displacements and stresses in response to base excitation (seismic).

PTC

Creating and Running Analyses (cont.)

Thermal Analyses

- Steady State Thermal
 - Calculates temperatures through a structure given heat loads, convection conditions.



Slide 11

Analysis Type: Thermal

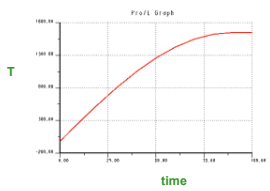
- Steady State Thermal** - Calculates temperatures through a structure given heat loads, convection conditions, and known temperatures.

PTC

Creating and Running Analyses (cont.)

Thermal Analyses (cont.)

- Transient Thermal
 - *Calculates time varying temperatures through a structure given time-varying heat loads and/or convection conditions.*



Slide 12

Analysis Type: Thermal

- **Transient Thermal** - Calculates temperatures through a structure given time-varying heat loads and/or convection conditions.



Demonstrations

- ◉ Setup and Run a Static Analysis.
- ◉ Setup and Run a Thermal Analysis.
- ◉ Calculating Thermal Stresses.
- ◉ Combine Static and Thermal Analysis.

Slide 13

Demonstrations

In this demonstration, I will:

- Setup and run a static analysis.
- Setup and run a thermal analysis.
- Calculating Thermal Stresses
- Combine static and thermal analysis.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- Setup models for structural and thermal analyses.
- Run structural and thermal analyses and view the results.
- Transfer thermal loads into structure to determine thermal stresses.
- Create thermal/mechanical and gravity loads.
- Create a static analysis with multiple load sets.
- Create results windows with different combinations of load sets.

Slide 14

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 9 Lab Exercises

Demonstration Instructions

Preparation

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Setup and Run a Static Analysis

Introduction

In the following three exercises you study a behavior of PCB board under thermal and structural loads. First, you will create a structural analysis to look at the stress distribution in a PCB board under its own weight.

Objectives

After successfully completing this exercise, you will know how to:

- Setup materials, loads and constraints.
- Setup and run structural analyses and review the results.

Scenario

Mechanica will be used to investigate the performance of an industrial controller module, which has had reliability problems due to the CPU chip overheating.

The controller module consists of a printed circuit board (PCB) with a CPU, several ICs, and some on-board power-supply components - an AC power connector, transformer, and a small vertical heat sink for a voltage regulator. Many electrical components are not included in the model - only those that generate, conduct, absorb or dissipate significant amounts of heat are important for the analysis.

First, a structural analysis will be performed to evaluate the stress due to the weight.

Step 1. Open the PCB_INIT.ASM and enter Mechanica Structure.

Open PCB_INIT.ASM and review it.

Enable Mechanica Structure.

1. Start Pro\ENGINEER Wildfire and modify your working directory to **c:\users\student\mechanica_330\module_09**.
2. Open the PCB_INIT.ASM.
3. Enable Mechanical Structure.
4. Clear the **Spin Center** option in the environment settings.

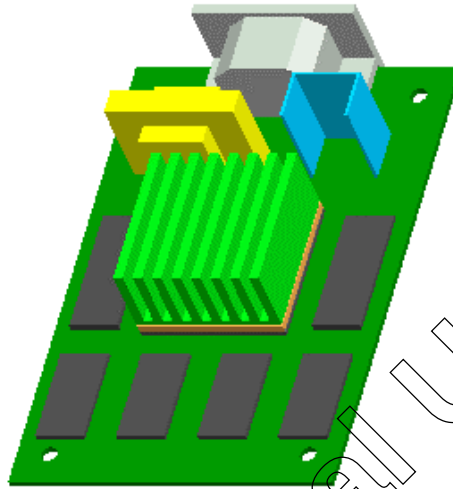


Figure 1: PCB_INIT.ASM

Step 2. Material properties of components are required to run any analysis. Some of the components have already been assigned materials to reduce repetition. You will assign material properties for the remaining components.

Assign steel material to the transformer, XFRMR.PRT.

*Create a new part named **Sink** and assign it to both the HEAT_SINK_INIT.PRT and SINK.PRT components.*

Repeat the same for IC1 components.

Repeat the same for DIE_BOND1.PRT components. Note that the thermal properties are transversely isotropic.

Repeat the same steps for the PCB.PRT component.

For all the properties use the table below.

1. Assign **STEEL** material to XFRMR.PRT.
2. Create a **new** material named **Sink**. Use the properties shown in the following table:

Table 1: Sink Properties

Name:	Sink
Specific Heat Capacity c (mm ² /sec ² C):	8.43 E+08
Thermal Conductivity k (N/sec C):	100
Density ρ (tonne/mm ³):	2.71 E-09
Poisson's ration ν :	0.3
Young's modulus E (N/mm ²):	68940
Coefficient of thermal expansion α (1/C):	2.34 E-05

3. Assign this new material to both the HEAT_SINK_INIT.PRT and SINK.PRT components.

4. Repeat the same steps for the IC1 components with the following material properties:

Table 2: IC1 Properties

Name:	IC1
Specific Heat Capacity c (mm ² /sec ² C):	1.05 E+09
Thermal Conductivity k (N/sec C):	2
Density ρ (tonne/mm ³):	1.30 E-09
Poisson's ration ν :	0.37
Young's modulus E (N/mm ²):	3000
Coefficient of thermal expansion α (1/C):	6.00 E-05

5. Repeat the same steps for the chip bond component DIE_BOND1.PRT with the material parameters in the following table. Change the Thermal properties from Isotropic to Transversely Isotropic from the pull-down arrow:

Table 3: Chip Bond Parameters

Name:	Chip_Bond
Specific Heat Capacity c (mm ² /sec ² C):	1.00 E+08
Thermal Conductivity k (N/sec C):	$k_1 = 1$ $k_2 = k_3 = 0.25$
Density ρ (tonne/mm ³):	1.30 E-09
Poisson's ration ν :	0.37
Young's modulus E (N/mm ²):	3000
Coefficient of thermal expansion α (1/C):	6.00 E-05

6. Repeat the same steps for the PCB.PRT component with the following material parameters in the following table. Leave the Thermal properties as Isotropic in this case.

Table 4: PCB Component Properties

Name:	PCB
Specific Heat Capacity c (mm ² /sec ² C):	1.05 E+09
Thermal Conductivity k (N/sec C):	2
Density ρ (tonne/mm ³):	1.30 E-09
Poisson's ration ν :	0.37
Young's modulus E (N/mm ²):	3000
Coefficient of thermal expansion α (1/C):	6.00 E-05

7. Close the Materials dialog box.

Step 3. Now you will create two spot welds to simulate AC-SKT.PRT screwed onto the PCB.

Create a spot weld connection on PNT0 and PNT1 between PCB.PRT and AC-SKT.PRT. Use Screw as material and a diameter of 5 mm.

1. Create a Spot Weld by selecting **Connections > Spot Welds > Create**.
2. Select the top surface of the PCB.PRT and the bottom surface of the AC-SKT.PRT for reference surfaces.
3. Select both the point of set PNT0 as the spot welding location. Click **OK**.
4. As the value of the weld diameter enter **5 mm** and for material select the **SCREW** material.
5. Similarly create another spot weld using point set PNT1. Click **Done/Return**.

Step 4. Constrain the four holes.

Fully constrain all surfaces of the four holes on the PCB part.

1. In the STRCT MODEL menu click **Constraints > New > Surface**.
2. Name the constraint **insert_tabs**. Leave it in the default constraint set. Leave WCS as the coordinate system.
3. Using the selection tool, select the four inside surfaces of the mounting holes on PCB board as shown in the following figure.

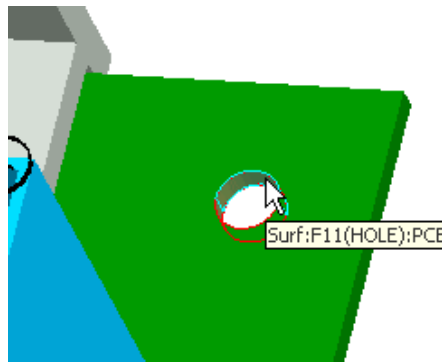


Figure 2: Hole surfaces for constraints

4. Click **OK** to apply the constraints.

Step 5. Create a load to simulate the effect of gravity.

Create a gravity load named **gravity** that has a value of 9810 mm/s^2 in the negative y direction.

1. Click **Loads > New > Gravity** to create a new gravity load.
2. Name the load **gravity** and leave it in the default load set.
3. The gravity is $g = 9.81 \text{ m/s}^2$ which is **9810** mm/s^2 acting in the negative y direction as shown in the following figure. Enter the value in the dialog box. Click **OK**.

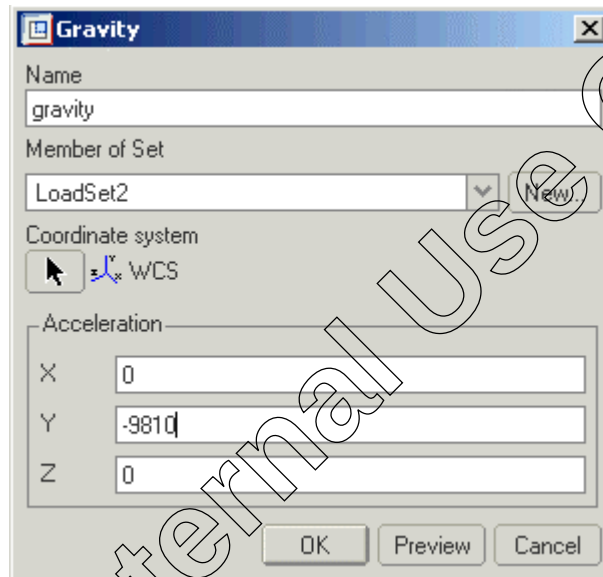


Figure 3: Gravity load dialog box

4. Click **Done/Return**.
5. PCB_INIT.ASM should appear as shown in the following figure.

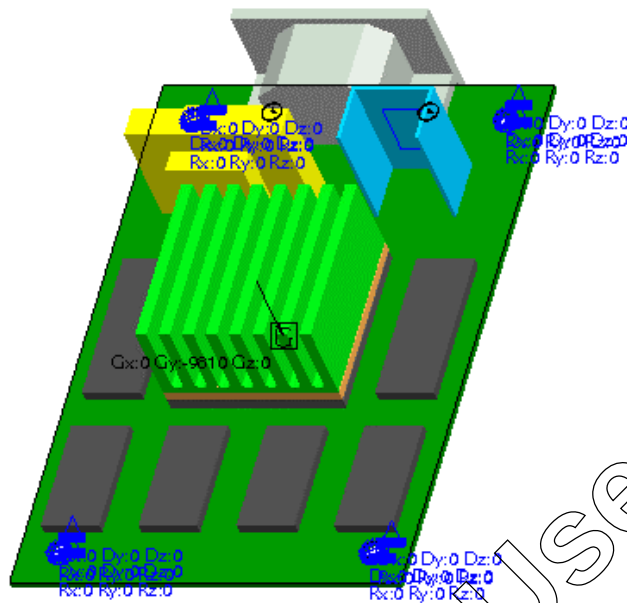


Figure 4: PCB_INIT.ASM with loads and constraints

Step 6. Create a new static analysis to study PCB_INIT.ASM as constrained under its own weight.

Create a static analysis named **pcb_static_spa**. Use SPA as the integration method.

1. In the MEC STRUCT menu click **Analyses/Studies**.
2. Click **File > New Static** to create a new static analysis named **pcb_static_spa**.
3. Note the constraints and the loads applied for the analysis.
4. Select the **Output** tab and set the plotting grid to **7**.
5. Click **OK**.

Step 7. Run the analysis and monitor the run.

Note to the instructors:

Instead of running the analysis you can show the report file from the already run analysis. The analysis results are saved in C:\users\student\mechanica_330\module_09\pcb_static_spa. Discuss the errors and convergence.

Note:

The analysis will take a considerable amount of time to complete. The time taken to run the analysis is dependent on the complexity of the model and the hardware resources available.

Do not run the analysis you created. The results from the pcb_static_spa analysis are available for review.

1. Open the report files located in C:\users\student\mechanica_330\module_09\pcb_static_spa with notepad and examine the run.
 - What was the maximum p-order set to in Pass 2?
 - What was the maximum von Mises stress in the model?
 - What is the RMS Stress Error reported for the static analysis?
 - How long did the analysis take to complete?

Note:

You may run the analysis in your spare time. Ensure that the RAM allocation is set to half of the system RAM, in the analysis settings. You can monitor the run by checking the status, and view the element creation progress. Toggle back and forth between the Status window and the Log window by checking the Detailed Summary option on the bottom-left corner of the Status window.

Step 8. You will create a fringe plot to examine von Mises stress. In addition, you will create displacement plots.

Create a result window displaying fringe plot of von Mises stress. Name it vm_gravity_load.

1. Click **Results** in the MEC STRUCT menu and create a result window named **vm_gravity_load**.
2. Select **pcb_static_spa** for design study. Do not double click on the analysis directory just highlight it.
3. Select von Mises stress to plot.
4. On the Display Options tab, enable the Continuous Tone option.
5. Click **OK and Show** to display the result.
6. Repeat the above steps to create another result window to show fringe plot of displacements.

Step 9. Display the results windows.

Display the previously created result window.

1. Show both result windows just created. Notice that the stresses are not significant. In the following modules, we will explore the effects of thermal stresses on the model, which are much greater.
2. Click **File > Exit Results**. Do not save the result.
3. Save the model.

Exercise 2: Setup and Run a Thermal Analysis

Objectives

After successfully completing this exercise, you will know how to:

- Create thermal loads and boundaries conditions.
- Setup and run a thermal analysis and review the results.
- Setup models for thermal analyses.

Scenario

In this exercise, we will explore the effects of heat loads from IC components on the entire PCB assembly.

Step 1. Before we transfer the model into Mechanica Thermal, we need to do some preliminary work. In order to create heat loads, you will need to create a number of surface regions for the IC components. These regions should be created in part level, so open IC1.PRT.

You need to create surface regions on IC1 and IC2 parts to place heat load.

Open IC1.PRT and enable Mechanica Thermal. Create 10 mm square surface regions to split the top surface of IC1 part.

1. Open IC1.PRT.
2. Enable Mechanica Thermal by clicking **Applications > Mechanica > Thermal**.
3. Create a surface region.
 - Click **Features > Surf Region > Create > Sketch > Done**.
 - Select the top planar surface of the IC as the sketching plane and select **DTM3** as bottom reference for sketch view.
 - Sketch two centerlines for symmetry. Create a square cross-section symmetric about the two DTM1 and DTM 3 with a dimension of 10 mm as shown in the following figure.

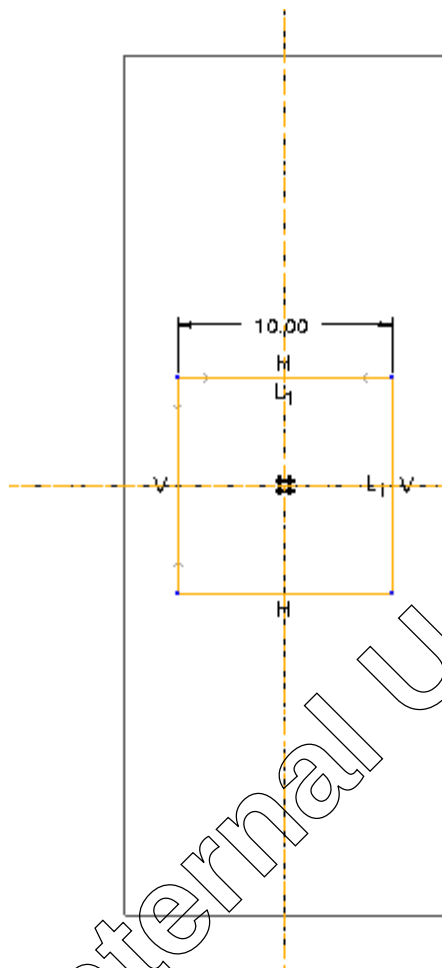


Figure 5: Surface region sketch

4. When finished, select the top planar surface again to split. The surface should be split into two regions.
5. Save the model.
6. Close the window.

Step 2. Create the same surface region in IC2 part.

Create another 10 mm square surface region to split the top surface of IC2 part.

1. Open IC2.PRT.
2. Enable Mechanics Thermal.
3. Repeat same procedure for the IC2 component.
4. Save the model.
5. Close the window.

Step 3. You will now create heat loads in the PCB assembly on all IC parts.

From View and Simulation display, turn off the display of all thermal conditions except heat loads. You will notice some heat loads already exist. This is to reduce setup time and repetition.

*Revert back the PCB assembly and create a heat load called **IC1** on the square surface regions created earlier of both IC1 components. The load value is 1000mW.*

1. Activate PCB_INIT.ASM.
2. Enable Mechanical Thermal.
3. You will notice some heat loads already exist, in order to reduce the setup time and repetition. From the main menu, click **View > Simulation Display**. Select the **Loads / Constraints** tab and unselect all check boxes for Thermal conditions except Heat Loads.
4. Create a surface heat load, by clicking **Model > Heat loads > New > Surface**.
5. Name the heat load **IC1**.
6. For surfaces, select the small square section surface regions on both IC1.PRT.
7. All IC1 components dissipate 1 W of heat, or 1000mW. Enter **1000** for Q as shown in the following figure.

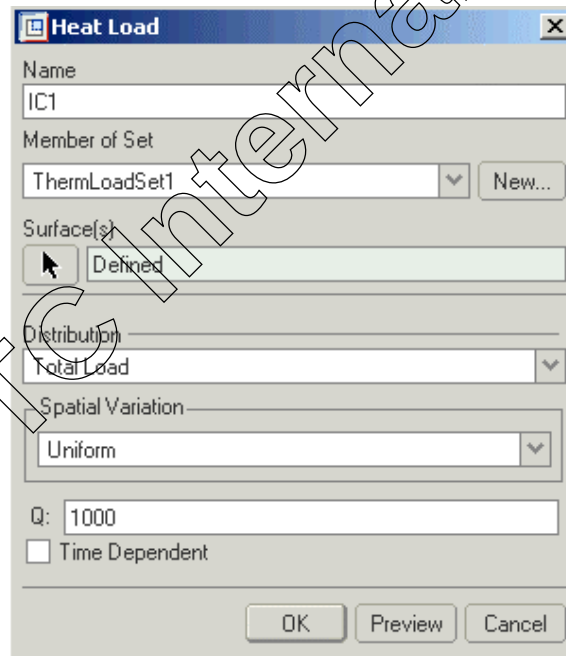


Figure 6: Heat load dialog box

Repeat the previous steps to create a heat load IC2 on all the IC2 parts. The load value is 2000mW

8. Repeat the above steps to create a second heat load named **IC2** and enter **2000** for Q. This time, select the four small, square cross-sectional surface regions belonging to the IC2 components.

Step 4. Create another heat load on CHIP_DIE1.PRT.

Create a third heat load on the bottom surface, CHIP_DIE1.PRT. To access the necessary surface, hide the first three components from CPU_INIT.ASM sub-assembly.

Name the load **CHIP** and 10,000 mW for the heat value.

1. Create a third heat load named **CHIP** on the bottom surface of CHIP_DIE1.PRT. To access the necessary surface. Hide the HEAT_SINK_INIT.PRT, CHIP_CASE1.PRT, and DIE_BOND1.PRT in CPU_INIT.ASM sub-assembly.

Note:

Select components to hide in the Model Tree and click **View > Visibility > Hide**.

2. The heat load coming from the silicon die dissipates 10 Watts, or 10,000 mW. Enter **10,000** for Q.

Step 5. Create prescribed temperature to simulate area of the PCB that remains at fixed temperature during thermal analysis.

Now you will create a prescribed temperature to simulate an area of the PCB that must remain at a fixed temperature during the thermal analysis. Turn on the temperature icon. Create a prescribed temperature boundary condition on all surfaces of the four holes on the PCB part. Name it **Fixed_Temp** and enter 25 for value.

1. Click **View > Simulation Display**. Select the **Loads / Constraints** tab and select the check box **Prescribed Temperature**. Click **OK**.
2. Create a prescribed temperature boundary condition, by clicking **Model > Bndry Conds > New > Prescr Temp > Surface**.
3. Name the boundary condition **Fixed_Temp**.
4. Select all surfaces that make up the four holes on the PCB by holding CTRL + left-click during the selection process.

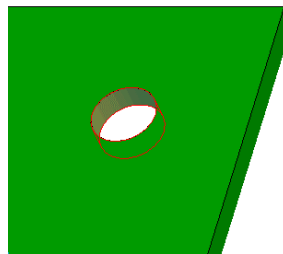


Figure 7: Hole surfaces

5. Enter **25** for the temperature value. Click **OK**.
6. Turn off the display of Prescribed Temperature from the Simulation Display dialog box.

Step 6. Create convective boundary conditions to simulate natural convection over the entire model.

Now you will create several convective boundary conditions to simulate natural convection over the entire model. This will be accomplished through several convection coefficients over certain surfaces of the model. Turn on the Convective icons and create a convective boundary condition. Name it IC.

*Select the top surfaces of all IC part (including the surface regions created earlier) and enter **0.01** as the value for the film coefficient and **25** for the reference temperature.*

Note:

The convection coefficient can be estimated by hand calculation, from thermal design handbooks, by experiment, or from a detailed computational fluid dynamics (CFD) analysis. The Bulk Temperature of the cooling air – here the room temperature, will be around 25 °C. You will also notice that some boundary conditions already exist. This, again, is to reduce repetition.

1. Turn on the display of Convection Conditions in the Simulation Display dialog box.
2. Create a Convective Boundary Condition named **IC** by clicking **New > Conv Cond > Surface**.
3. Select the top surfaces of each IC part (including the surface regions created earlier)
4. Enter **0.01** for the Convection Coefficient and **25** for the Bulk Temperature.
5. Click **OK**.

Step 7. Create a boundary condition on the SINK.PRT surfaces.

*Create another boundary condition on the vertical, outer surfaces of SINK.PRT. Name it **SINK** and enter **0.02** for the film coefficient, and **25** for the reference temperature.*

1. Create another convection boundary condition on the vertical, outer surfaces of SINK.PRT as shown in the following figure.

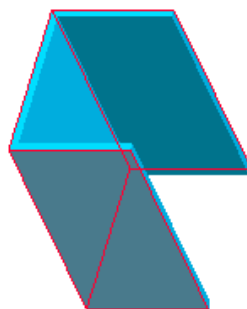


Figure 8: Sink surfaces

2. Name the boundary condition **SINK**.

3. Enter **0.02** for the Convection Coefficient, and **25** for the Bulk Temperature. Click **OK**.

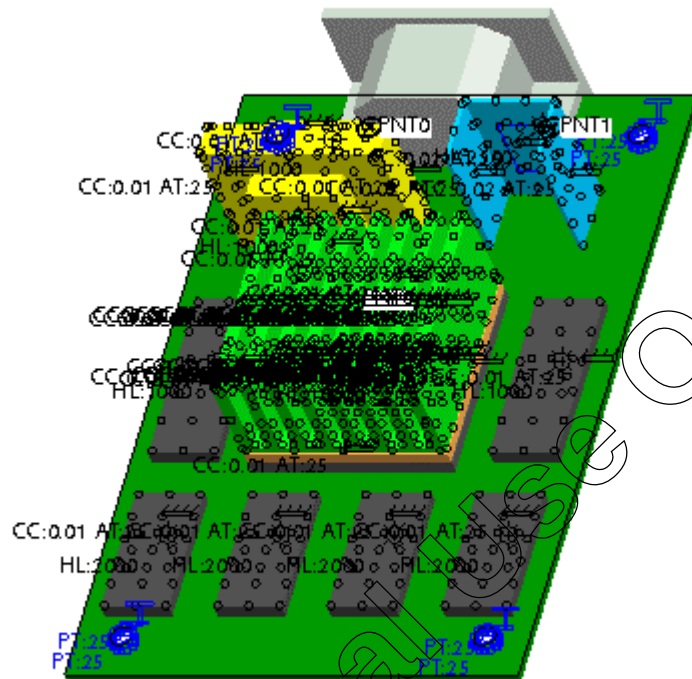


Figure 9: PCB board with all loads and boundary conditions

Step 8. Review the material properties assigned to the heat sink.

All the materials properties are already assigned. Once again, we have already completed the process for all components to reduce repetition.

1. Review the material properties for HEAT_SINK_INIT.PRT.

Step 9. Create a thermal analysis to evaluate the temperature distribution over the PCB model.

*Create a steady state thermal analysis. Name the thermal analysis **pcb_init_spa**.*

*Set the convergence method to **Single Pass** and the plotting grid to **7**.*

1. Create a steady state thermal analysis named **pcb_init_spa** as shown in the following figure.

- Click **Analyses/Studies** in the MEC THERMAL menu.
- Click **File > New Steady State Thermal**.

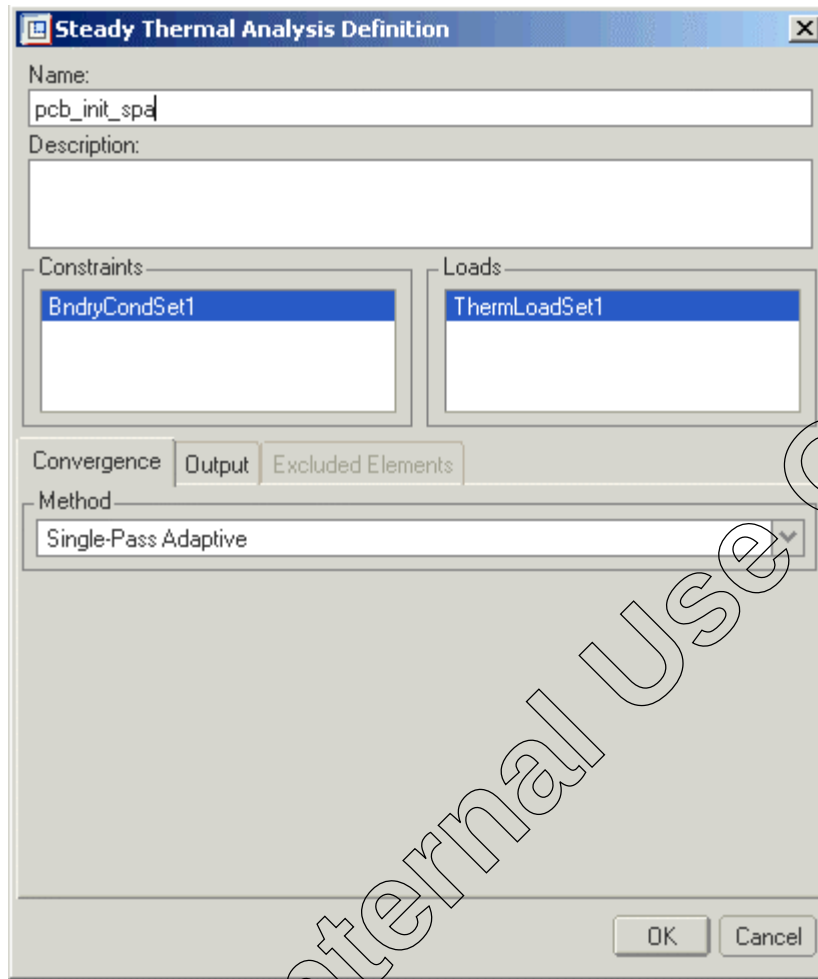


Figure 10: Thermal analysis dialog box

2. Set the convergence method to **Single-Pass Adaptive**. Select the **Output** tab and set the plotting grid to **7**. Click **OK**.

Step 10. Run the thermal analysis

Discuss the errors and convergence.

1. Before running the analysis, set the memory allocation from settings to half of the system RAM, by clicking **Run > Setting**. Then set RAM to $\frac{1}{2}$ of available RAM.
2. Run **pcb_init_spa** thermal analysis.
3. Check the progress of the thermal analysis from the status.

Step 11. Review the results of the thermal analysis.

*The last step of the exercise is to review the results based on the thermal analysis just ran. We will determine the maximum temperature in the assembly as well as identify the location of the maximum temperature. Create a result window displaying fringe plot of temperature. Name it **pcb_temperature**.*

1. Create a new result window named **pcb_temperature** to display the temperature throughout the model.
2. For the design study, select **pcb_init_spa** as shown in the following figure.

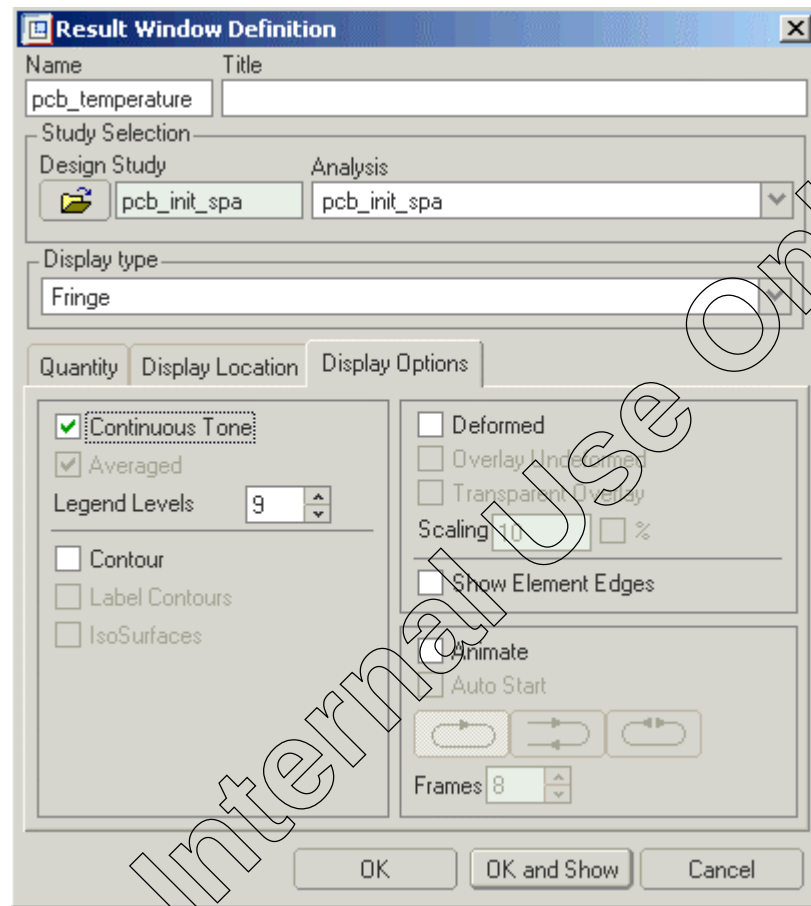


Figure 11: Result dialog box

Display the previously created result window.

3. Show the result window.
4. Show the maximum temperature of the model.
5. Create a cutting plane and change it dynamically to see the temperature distribution in the model. You should now have a better view of the location of the maximum temperature. If not, you can also zoom in and rotate your model if necessary.
 - What is the maximum temperature?
 - Where is the maximum temperature located?
 - Is this where you would expect the maximum temperature to take place?
6. Exit the result window. Do not save the result.
7. Click **Done/Return**.

Exercise 3: Calculating Thermal Stresses

Introduction

Following the previous exercise, you enter calculate the thermal stresses resulting from the temperature distribution.

Objectives

After successfully completing this exercise, you will know how to:

- Transfer thermal loads into structure to determine thermal stresses.

Scenario

Now that you have determined the temperature distribution in the model, you will proceed to examine the resulting thermal stresses. You will create a static analysis using Mechanical/Thermal load.

Step 1. You will transfer the PCB_INIT.ASM into Mechanica Structure and create a temperature load to simulate the effect of temperature on stresses.

With the PCB_INIT.ASM opened enter Mechanica Structure. Create a new temperature load as Mechanical/Thermal. Name the load temperature_load. Use pcb_init_spa for the thermal analysis. Make sure to deselect the Use previous design study option.

1. With the PCB_INIT.ASM opened enable Mechanica Structure.
2. Create a new load for Temperature by clicking **Model > Loads > New > Temperature > MEC/T Temp.**
3. Create a new Load Set named **LoadSet3**. Name the load **temperature_load**.
4. Select **pcb_init_spa** for the thermal analysis as shown in the following figure. Ensure that Use previous design study option is not enabled.
5. Enter **25** as the Reference Temperature.

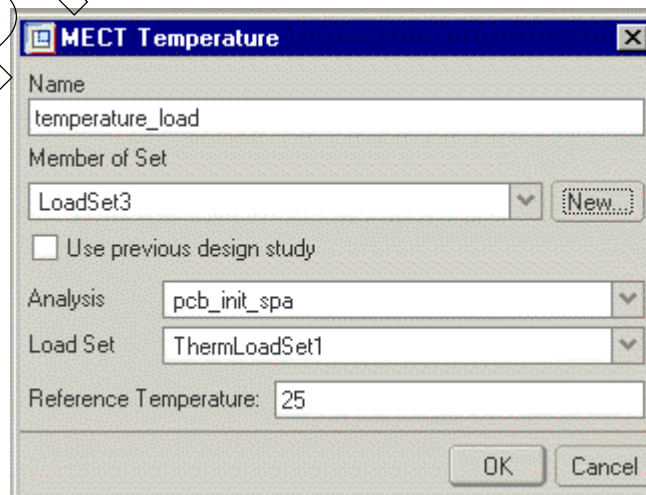


Figure 12: Temperature load

Step 2. Create a static analysis

Create a static analysis. Name the thermal analysis **thermal_static_spa**. Select the Mechanical/Thermal load. Set the convergence method to **Single Pass** and the plotting grid to 7.

1. Create a static analysis named **thermal_static_spa** as shown in the following figure.
 - Click **Analyses/Studies** in the MEC STRUCT menu.
 - Click **File > New Static**.

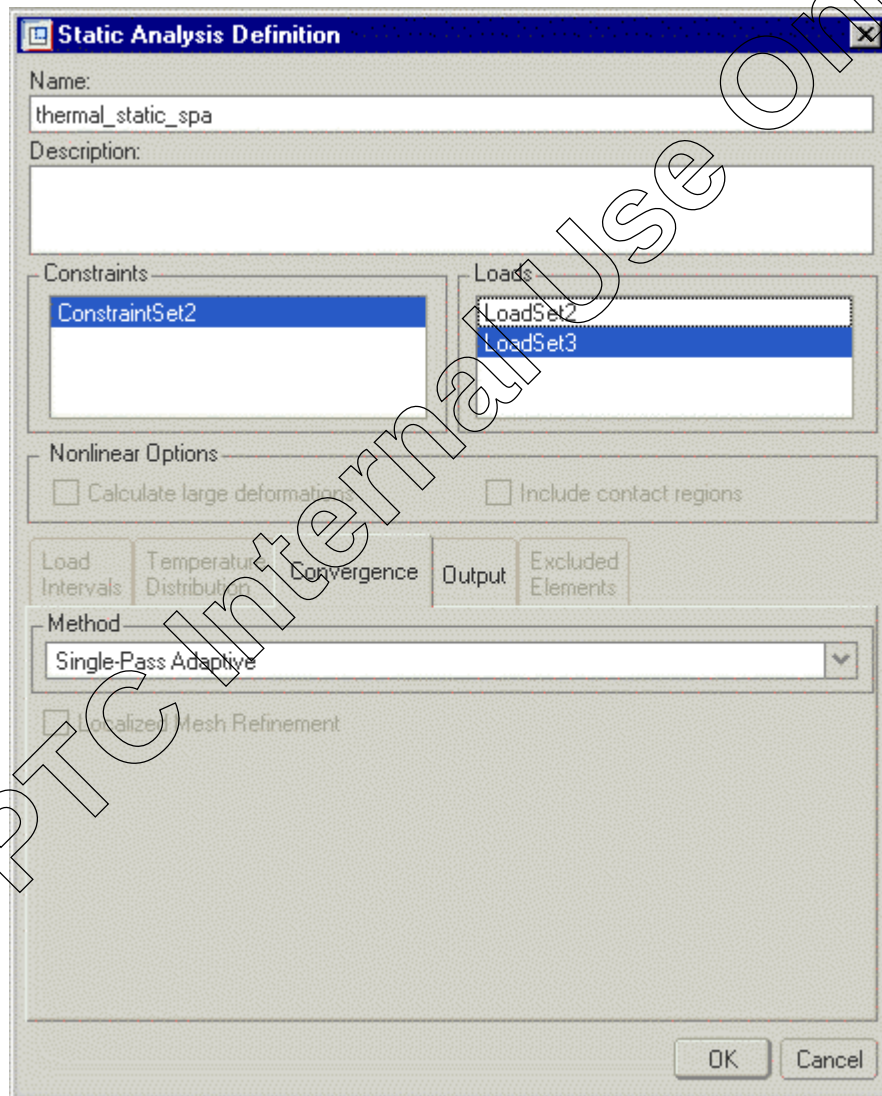


Figure 13: Static analysis dialog box

2. For the integration method, select **Single-Pass Adaptive** and set the plotting grid to **7** from the output tab.
3. Click **OK**.

Step 3. Running the static analysis.

Note to the instructors:

Instead of running the analysis you can show the report file from the already run analysis. The analysis results are saved in C:\users\student\mechanica_330\module_09\thermal_static_spa. Discuss the errors and convergence.

Note:

The analysis will take a considerable amount of time to complete. The time taken to run the analysis is dependent on the complexity of the model and the hardware resources available.

Do not run the analysis you created. The results from the thermal_static_spa analysis are available for review.

1. Open the report file and review it.

- What was the maximum p-order set to in Pass 2?
- What was the maximum von Mises stress in the model?
- Notice the stress this time is much higher than the one from the gravity load.
- What is the RMS Stress Error reported for the static analysis?
- How long did the analysis take to complete?

Note:

You may run the analysis in your spare time. Ensure that the RAM allocation is set to half of the system RAM, in the analysis settings. You can monitor the run by checking the status, and view the element creation progress. Toggle back and forth between the Status window and the Log window by checking the Detailed Summary option on the bottom-left corner of the Status window.

Step 4. You will create a fringe plot to examine von Mises stress.

Create a result window displaying fringe plot of von Mises stress. Name it **thermal_stress**.

1. Create a result window named **thermal_stress**.
2. Select **thermal_static_spa** for design study.
3. Select **von Mises stress** to plot and set the display option as shown in the following figures.

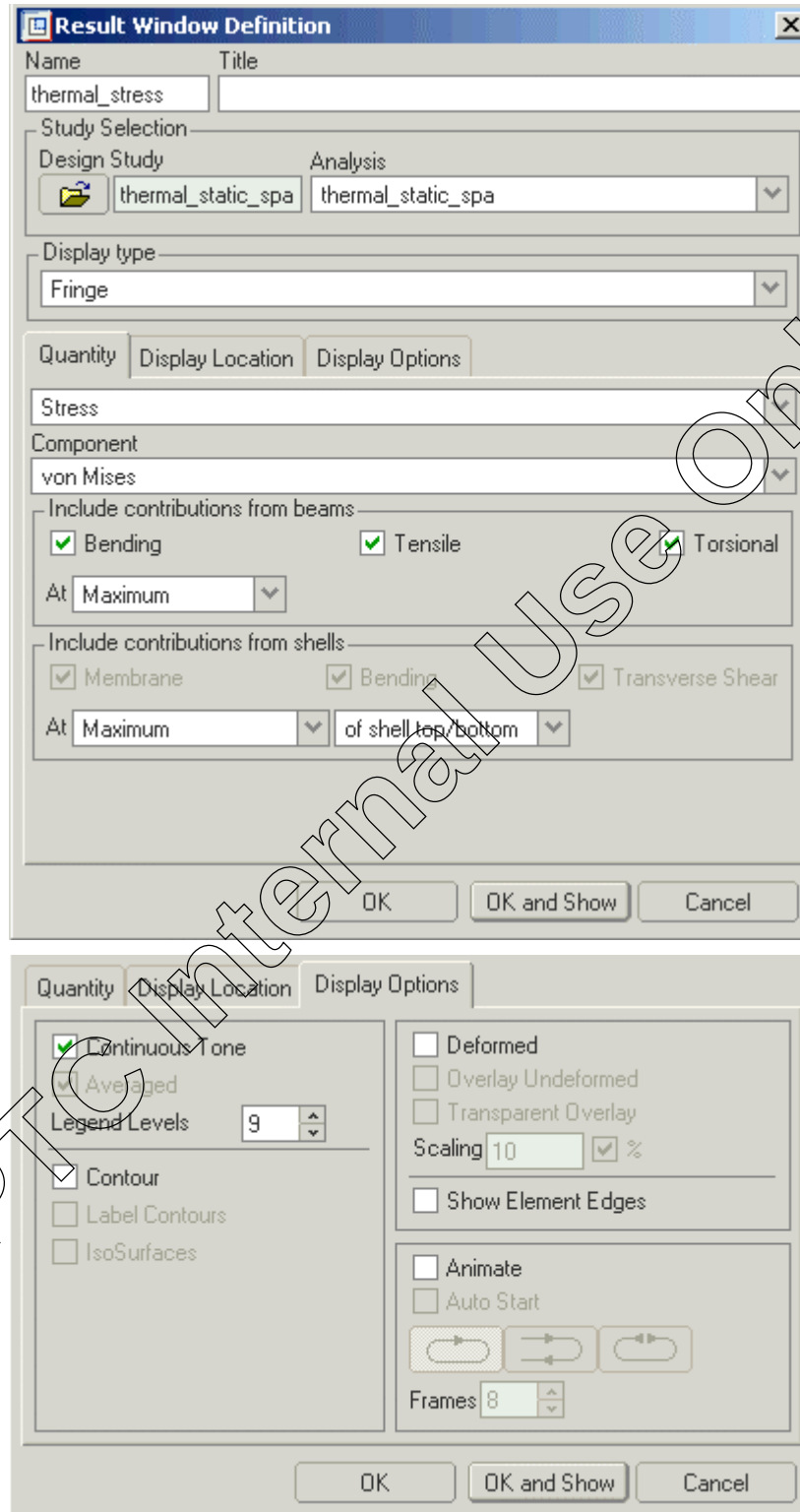


Figure 14: Result dialog box

Step 5. Display the results windows.

View the result window just created.

1. Show the result window. Notice that the stresses are significant compared to the previous exercise.
2. Exit the result window. Do not save the result.
3. Save the model.

For PTC Internal Use Only

Exercise 4: Combining Structural and Thermal Analyses

Introduction

Following the previous exercise, you create a static analysis with two load sets. One load is for gravity, and the other for the Thermal/mechanical load.

Objectives

After successfully completing this exercise, you will know how to:

- Create thermal Mechanical and gravity loads.
- Create a static analysis with multiples load sets.

Scenario

In this module, combined thermal and structural static analysis will be defined. A thermal load will be defined in the structural analysis that will apply the calculated thermal distribution on the PCB as a structural load. A gravity load will also be defined.

Both loads will be created in two different load sets and a static analysis with two load sets will be created and run.

Step 1. Create a static analysis

Create a static analysis. Name the thermal analysis **thermal_mech_static_spa**. Make sure to select both load sets.

Set the convergence method to **Single Pass** and the plotting grid to **7**.

1. Create a static analysis named **therm_mech_static_spa**.
 - Click **Analyses/Studies** in the MEC STRUCT menu.
 - Click **File > New Static**.
 - Name the analysis **therm_mech_static_spa**.
2. Make sure to select both load sets containing Mec/T and gravity loads.
3. For the integration method, select **Single-Pass Adaptive** as shown in the following figure.

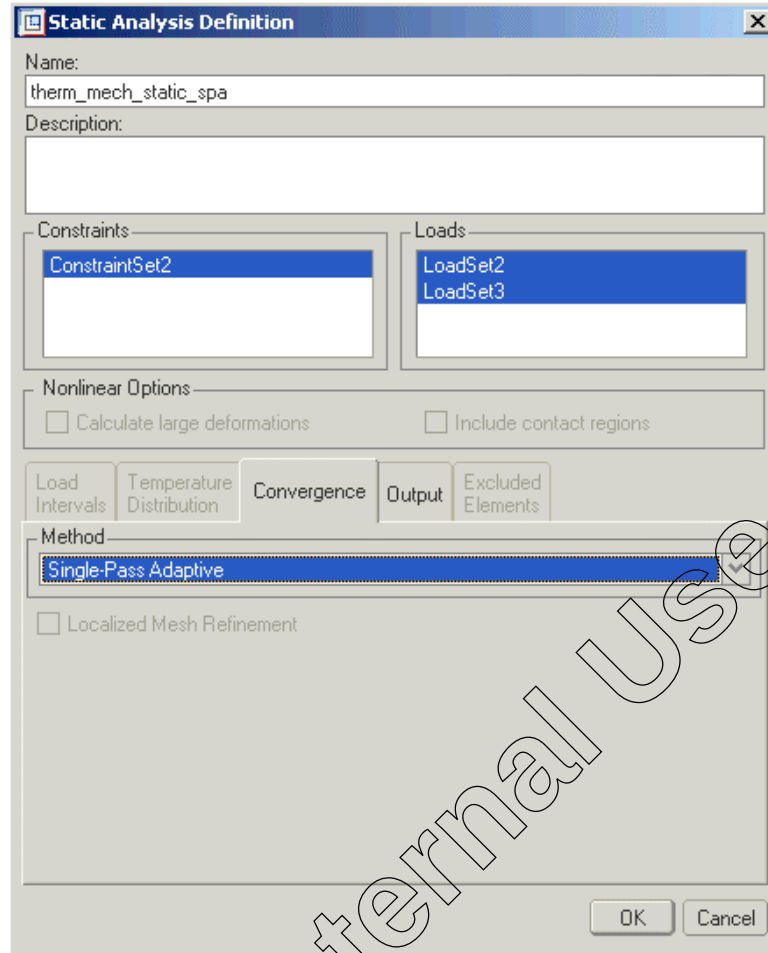


Figure 15. Static analysis dialog box

4. From the Output tab, set the plotting grid to 7.
5. Click **OK**.

Step 2. Running the static analysis.

Note to the instructors:

Instead of running the analysis you can show the report file from the already run analysis. The analysis results are saved in

C:\users\student\mechanica_330\module_09\therm_mech_static_spa.

Discuss the errors and convergence.

Note:

The analysis will take a considerable amount of time to complete. The time taken to run the analysis is dependent on the complexity of the model and the hardware resources available.

Do not run the analysis you created. The results from the therm_mech_static_spa analysis are available for review.

1. Open the report files located in
C:\users\student\mechanica_330\module_09\therm_mech_static_spa with notepad and examine the run.
 - What was the maximum p-order set to in Pass 2?
 - What was the maximum von Mises stress in the model?
 - What is the RMS Stress Error reported for the static analysis?
 - How long did the analysis take to complete?

Note:

You may run the analysis in your spare time. Ensure that the RAM allocation is set to half of the system RAM, in the analysis settings. You can monitor the run by checking the status, and view the element creation progress. Toggle back and forth between the Status window and the Log window by checking the Detailed Summary option on the bottom-left corner of the Status window.

Step 3. You will create a fringe plot to examine the contribution of the thermal load only von Mises stress.

Create a result window displaying fringe plot of von Mises stress due to thermal load only. Name it thermal_vm_stress. Check load set 3 only.

1. Create a result window named **thermal_vm_stress**.
2. Select **therm_mech_static_spa** for design study.
3. Select the load set **LoadSet3** only.
4. Select **von Mises** stress to plot.
5. On the Display Options tab, enable the Continuous Tone option.
6. Click **OK and Show** to display the result.

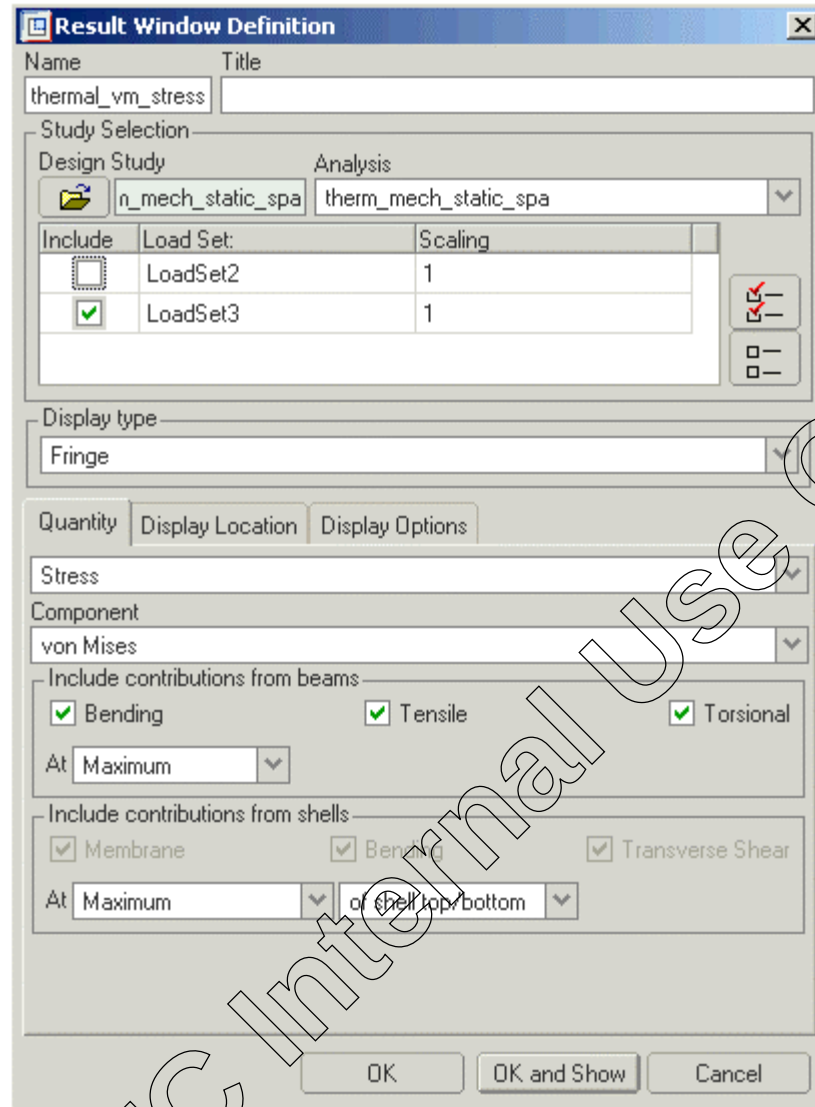


Figure 16: Result dialog box for thermal load.

Step 4. Create another fringe plot to examine the contribution of gravity load to von Mises stress.

Create another result window displaying fringe plot of von Mises stress due to gravity load only.

Name it `gravity_vm_stress`.

Check load set 2 only.

1. Create a result window named **gravity_vm_stress**.
2. Select **therm_mech_static_spa** for design study.
3. Select the load set **LoadSet2** only.
4. Select **von Mises** stress to plot.
5. On the Display Options tab, enable the Continuous Tone option.
6. Click **OK and Show** to display the result.

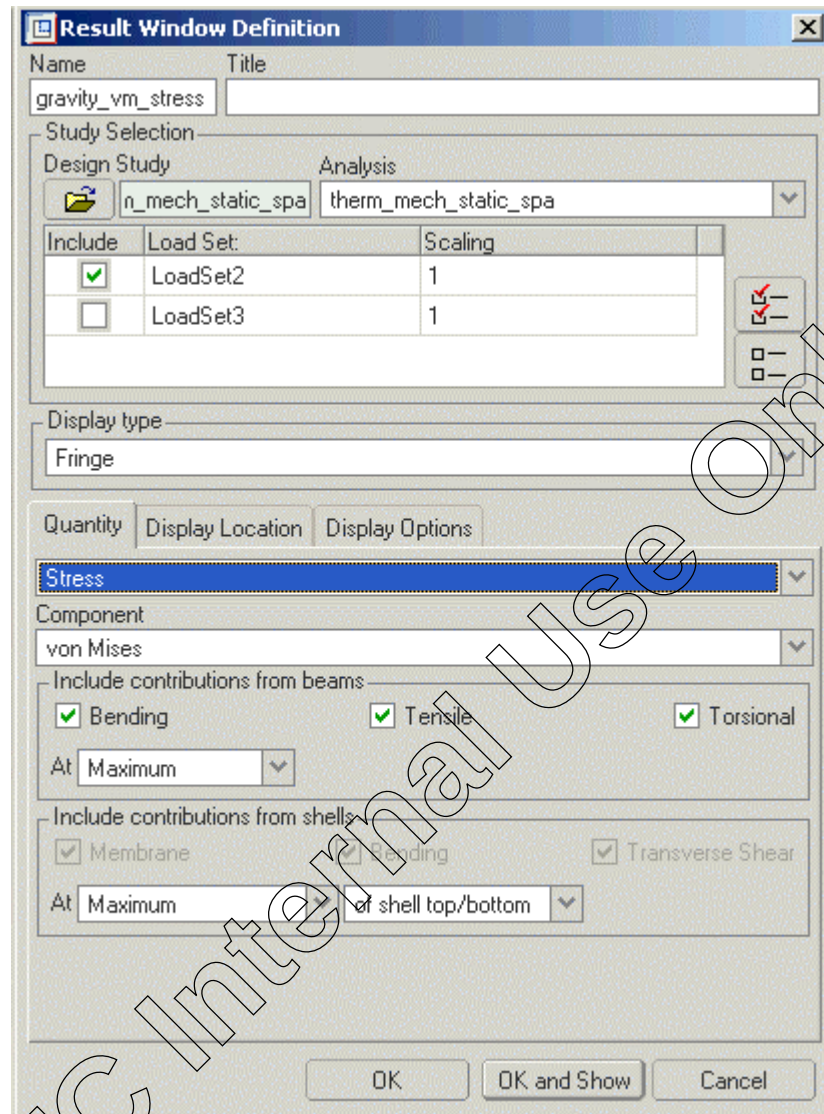


Figure 17: Result dialog box for gravity load.

Step 5. Create another fringe plot to show the contribution of both loads on von Mises stress.

Create another result window displaying fringe plot of von Mises stress due to the combination of both loads. Name it `combined_vm_stress`. Check both load sets.

1. Create a result window named **combined_vm_stress**.
2. Select **therm_mech_static_spa** for design study.
3. Select both the load sets, **LoadSet2** and **LoadSet3**.
4. Select **von Mises** stress to plot.
5. On the Display Options tab, enable the Continuous Tone option.
6. Click **OK and Show** to display the result.

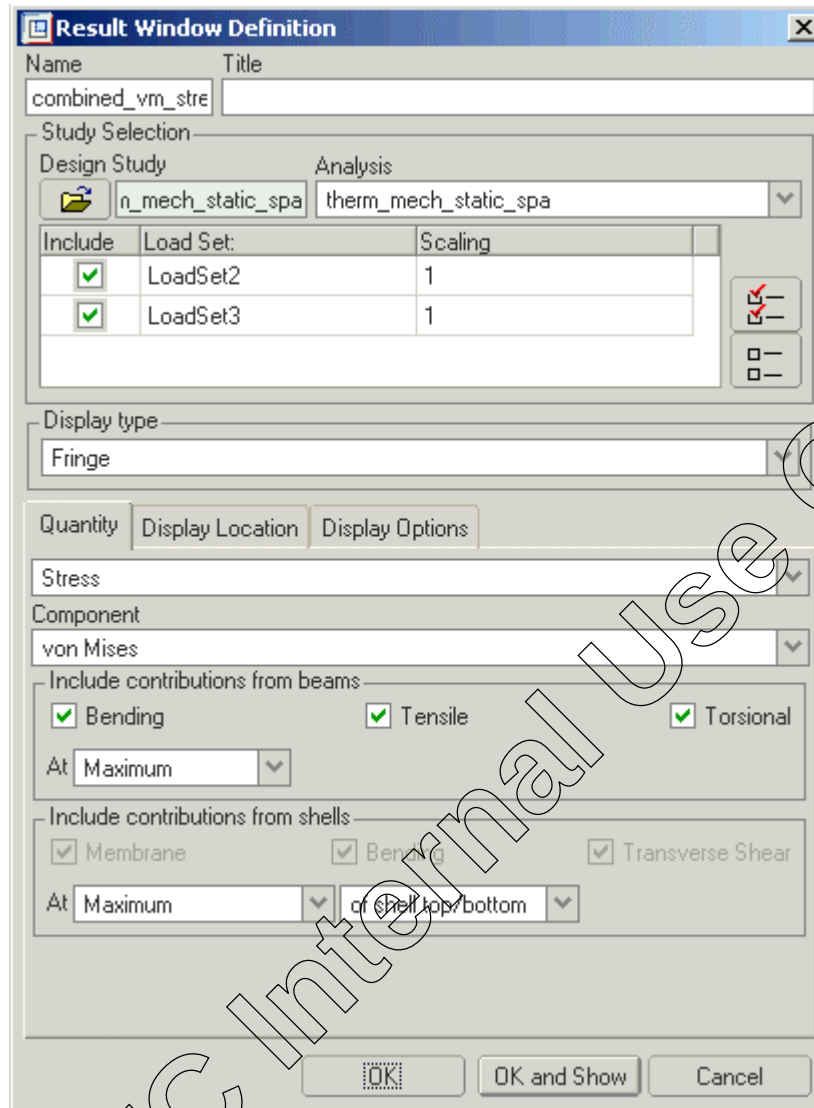


Figure 18: Result dialog box for thermal load.

Step 6. Review the results windows.

View all the result windows created.

1. Display all the result windows.
2. Notice that the stress due to the gravity is small compared to the thermal stress.
3. Exit the result window. Do not save.
4. Save the model
5. Close the window and erase all models from session.

Summary

After completing this module, you will be able to:

- Setup models for structural and thermal analyses.
- Run structural and thermal analyses and view the results.
- Transfer thermal loads into structure to determine thermal stresses.
- Create thermal/mechanical and gravity loads.
- Create a static analysis with multiple load sets.
- Create results windows with different combinations of load sets.

For PTC Internal Use Only

For PTC Internal Use Only

Comparing MPA and SPA Results

Introduction

There are two methods Mechanica can use to elevate these polynomials in order to converge the solution: the Multi-Pass Adaptive (MPA) approach, and the Single-Pass Adaptive (SPA) approach.

Single-pass adaptive convergence applies to static, modal, buckling, and contact analyses of all element types. Models may consist of isotropic and/or orthotropic materials. While the single-pass adaptive option is available for contact analysis, we do not normally recommend its use because it can increase run time.

Mechanica runs a first pass and determines a local estimate of stress error. Using this error estimate, Mechanica determines a new p-order distribution and performs a final pass.

In general, use single-pass adaptive convergence when it is available. Larger models run with single-pass adaptive convergence typically require less disk space. Also, single-pass adaptive convergence yields comparable results to multi-pass adaptive convergence of 10% with generally shorter run times.

With single-pass adaptive analysis, you do not control the convergence tolerance, but you do need to examine the stress error estimates reported in the run summary. If these error estimates are acceptable, then continue to use single-pass adaptive convergence for efficiency.

If the error estimates from the single-pass adaptive convergence tolerance are not acceptable—for example, the stress error estimates are too large—then switch to the multi-pass adaptive convergence strategy for subsequent design studies.

When you run a design study, using Multi Pass Adaptive, Mechanica performs calculations and increases the polynomial order for each element edge until the convergence criteria are satisfied. (The polynomial level for edges on which beams and 2D shells lie begins with either the third order or the order you enter on the analysis definition dialog box, whichever is higher). An analysis converges when the difference in the results of the current pass and the previous pass is within the percentage you specify under Convergence.

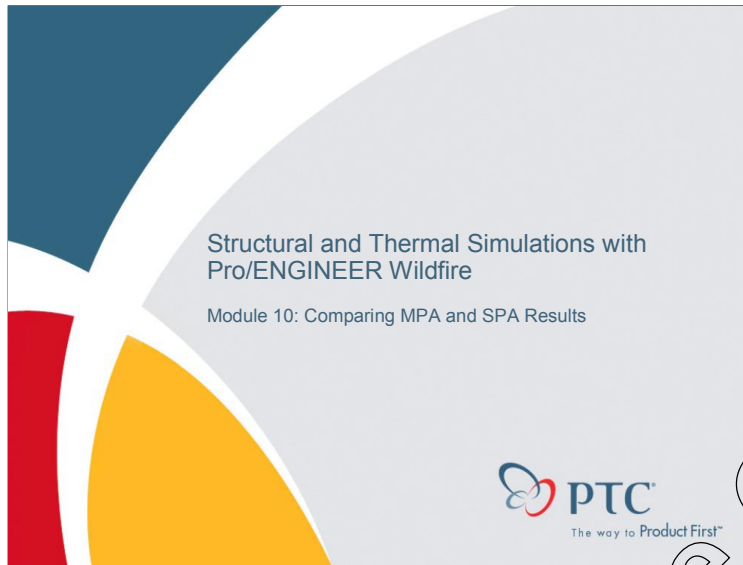
This module will discuss the difference between the two methods Mechanica can use to elevate the solution, the Multi-Pass Adaptive (MPA) approach, and the Single-Pass Adaptive (SPA) approach.

Objectives

After completing this module, you will be able to:

- Create a Single-Pass Adaptive analysis.
- Create a Multi-Pass Adaptive analysis.
- Describe the difference between Single-Pass Adaptive and Multi-Pass Adaptive analyses.

For PTC Internal Use Only



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

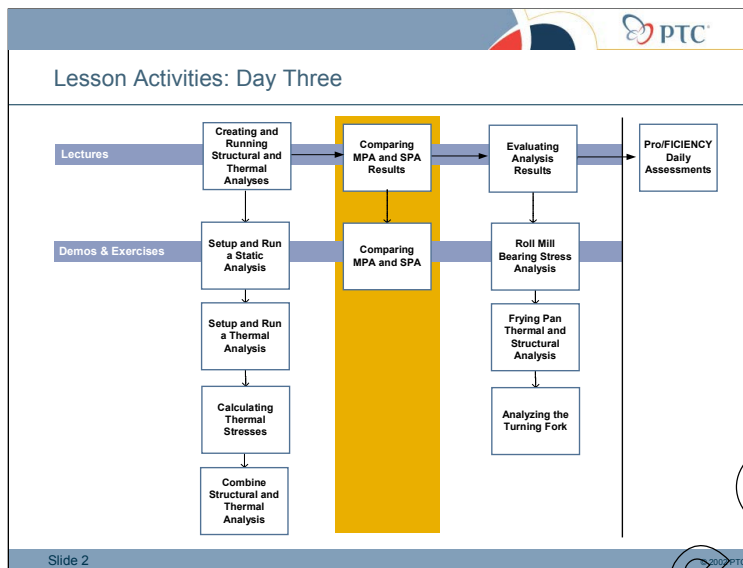
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 30 min
- Labs: 60 min
- Total: 1 hour 50 min



Objectives

After completing this module, you should be able to:

- Create a Single-Pass Adaptive analysis.
- Create a Multi-Pass Adaptive analysis.
- Describe the difference between Single-Pass Adaptive (SPA) and Multi-Pass Adaptive (MPA) analyses.

Slide 3

Objectives

There are two methods Mechanica can use to elevate these polynomials in order to converge the solution: the MPA approach, and the SPA approach.

SPA convergence applies to static, modal, buckling, and contact analyses of all element types. Models may consist of isotropic and/or orthotropic materials. While the single-pass adaptive option is available for contact analysis, we do not normally recommend its use because it can increase run time.

Mechanica runs a first pass at $p=3$ and determines a local estimate of stress error. Using this error estimate, Mechanica determines a new p -order distribution and performs a final pass.

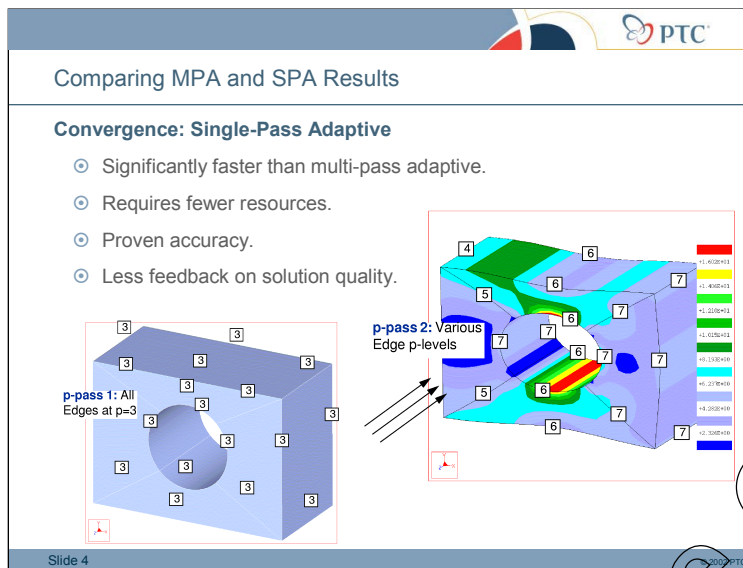
In general, use SPA convergence when it is available. Larger models run with SPA convergence typically require less disk space. Also, SPA convergence yields comparable results to MPA convergence of 10% with generally shorter run times.

With SPA analysis, you do not control the convergence tolerance, but you do need to examine the stress error estimates reported in the run summary. If these error estimates are acceptable, then continue to use single-pass adaptive convergence for efficiency.

If the error estimates from the SPA convergence tolerance are not acceptable—for example, the stress error estimates are too large—then switch to the MPA convergence strategy for subsequent design studies.

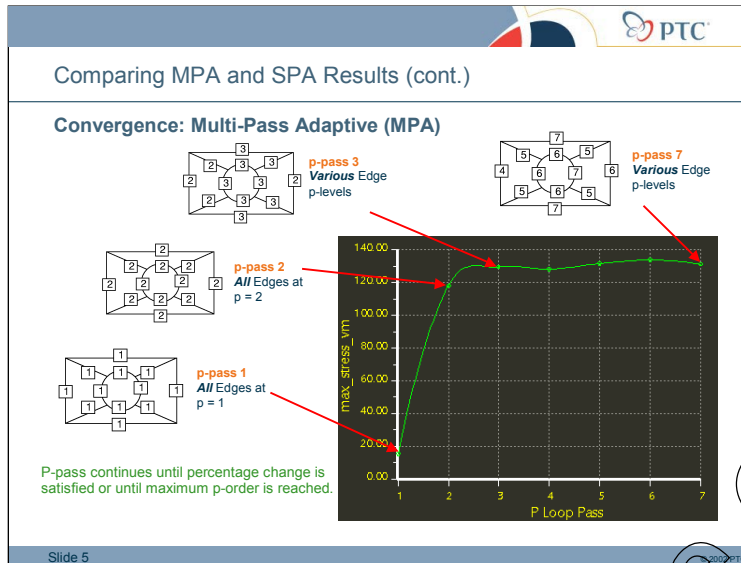
When you run a design study, using MPA, Mechanica performs calculations and increases the polynomial order for each element edge until the convergence criteria are satisfied. (The polynomial level for edges on which beams and 2-D shells lie begins with either the third order or the order you enter on the analysis definition dialog box, whichever is higher). An analysis converges when the difference in the results of the current pass and the previous pass is within the percentage you specify under Convergence.

This module will discuss the difference between the two methods Mechanica can use to elevate the solution, the MPA approach, and the SPA approach.



Convergence: Single-Pass Adaptive

- Solid and shell element types.*
 - Beams, if a shell or solid is present.*
 - Multiple may include both orthotropic and isotropic materials.*
 - All analysis types except pre-stress static.*
 - Steady State Thermal.*
- The SPA is significantly faster than MPA, requires fewer computer resources and provides results within 10% of MPA.
 - The drawback of SPA is that the user has less information on the quality of the solution and must therefore rely on experience to detect improperly converged results.
 - The convergence process for SPA is described below:
 - Mechanica solves the problem at p-level 3 and estimates local stress errors.
 - New polynomial orders are assigned to edges based on the magnitude of local stress errors.
 - The problem is solved with the updated polynomial orders and results are outputted.
 - The local stress errors are determined based on the difference between stress values computed using two different methods (super-converged stress and single element stress results).
 - A percentage is then defined for each element and compared to a target internal error estimator. The target error estimator % is set internally and has been determined experimentally.
 - Based on the ratio between actual and target stress error, the number of degrees of freedom (DOF's) needed to accurately solve for the stress can be determined. The p-level is then increased to the level required to solve the number of DOF's.
 - This method of improving stress accuracy and estimating stress error is based on the work by Zienkiewicz and Zhu ("The Superconvergent Patch Recovery and A Posteriori Error Estimates. Part 1: The Recovery Technique", International Journal for Numerical Methods in Engineering, Vol. 33, p. 1331 (1992)).
 - The final RMS stress error reported in the summary file is calculated by sampling the local error estimates (the same as those used to update polynomial orders).
 - If the iterative solver is used, Mechanica first solves the problem at polynomial order 2 with the block solver, then uses the iterative solver for the p-level 3 pass and for the final pass where edge p-levels have been updated to their final values.



Convergence: Multi-Pass Adaptive

- A solution is considered to have converged when the results are independent of mesh density and p-levels. More specifically, in the context of Mechanical, a solution is considered converged when results from the current and previous pass only differ by a user-defined percentage.
- Two convergence methods are available in Mechanical: Single-Pass Adaptive (SPA) and Multi-Pass Adaptive (MPA).
- The process of a Multi-Pass Adaptive run is described in the following example. Consider a static analysis on a plate with a hole and assume we have selected the MPA Convergence Method with 5% convergence on Displacement, Strain Energy and RMS Stress.
 1. In the first pass, the analysis is performed with all edges at p-level 1 (linear shape function). Results for displacement, Strain Energy and RMS Stress are stored.
 2. In the second pass, the analysis is performed with all edges at p-level 2 (quadratic shape function). Results are compared with the previous pass. A convergence check is now performed on edges *and* elements. Edges for which previous and current results differ by more than 5% are upgraded to p-level 3. Elements for which previous and current results differ by more than 5% have all their edges upgraded to p-level 3.
 3. The p-passes are repeated until the convergence percentage is satisfied for all specified quantities or until the maximum p-level is reached.



Comparing MPA and SPA Results (cont.)

Checking Convergence Information for SPA

- Check summary file (*.rpt).
 - *Look at the last pass % convergence values.*
 - *Maximum Edge Order should be less than 9.*
 - *Resultant loads should equal the applied load.*
- View Fringe Plots
 - *View plots with Average and Continuous Tone OFF.*
 - *Look for discontinuities between elements.*

Slide 6

There are no instructor notes on this slide.

PTC

Comparing MPA and SPA Results (cont.)

Checking Convergence Information: MPA

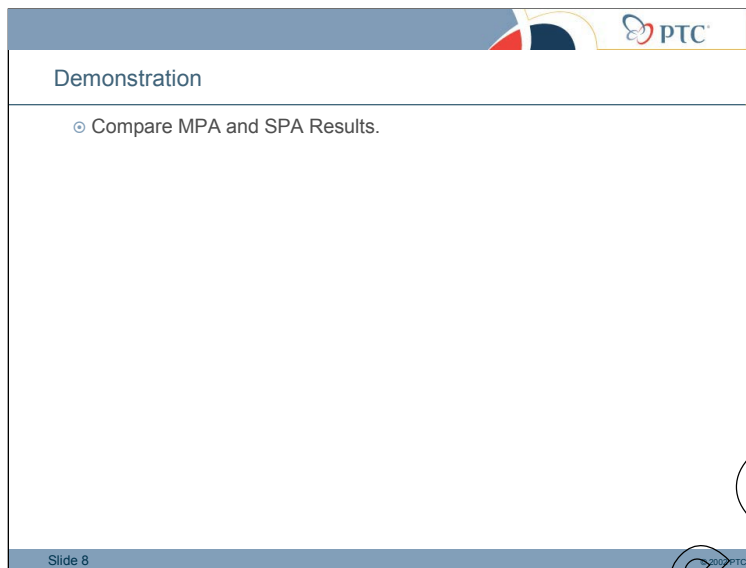
- Check summary file (*.rpt)
 - Look at the last pass % convergence values specially for measures of interest.
 - Maximum Edge Order should be less than 9.
 - Resultant loads should equal the applied load.
- Check Convergence Plots
 - Generate P-plots for all quantities of interest/concern.
 - Graph should asymptotically approach a single value. The last two values should be nearly identical.
- View Fringe Plots
 - View plots with Average and Continuous Tone OFF.
 - Look for discontinuities between elements.

Slide 7

Checking Convergence Information

How do we know if the solution is converged? Evaluating the quality of a solution is critical. The following procedure is recommended for MPA convergence:

1. From the Run Menu, select “**Summary**” (also available by editing the *.rpt file). Scrolling the file to the last p-pass, you want to verify the following:
 - a) The maximum edge order in the model. A value of 9 indicates that at least one edge went to the maximum edge order. This is a potentially un-converged result.
 - b) A message indicates whether the analysis has converged within the specified tolerance. This message *is not sufficient* to determine the quality of the solution. The table of convergence percentages should have all values close to ~10%.
 - c) The resultant force on the model should be equal to the applied load.
2. Generating **Convergence Graphs** is the best way to determine the quality of the solution. You need to create a graph for each quantity on which convergence-checks were performed. Convergence graphs can be generated by plotting quantities of interest (for example: max. displacement, max. stress) against p- ass. A good convergence graph has *both* of the following characteristics:
 - a) Clear trend approaching a value.
 - b) Ends with a zero slope (for example: last two passes results are identical).



Demonstrations

In this demonstration, I will:

- Create a static analysis.
- Choose SPA as integration method.
- Create another static analysis.
- Choose MPA as integration method.
- Run both analyses and compare the results.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- Create a Single-Pass Adaptive analysis.
- Create a Multi-Pass Adaptive analysis.
- Describe the difference between Single-Pass Adaptive (SPA) and Multi-Pass Adaptive (MPA) analyses.

Slide 9

Summary

There are no instructor notes on this slide.

Module 10 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Comparing MPA vs. SPA

Introduction

In this exercise, you will create two thermal analyses, open using SPA as the integration method, the other using MPA. After running the two analyses, you will compare the results.

Objectives

After successfully completing this exercise, you will know how to:

- Create a Single-Pass Adaptive analysis.
- Create a Multi-Pass Adaptive analysis.
- Describe the difference between Single-Pass Adaptive and Multi-Pass Adaptive analyses.

Scenario

Single Pass Adaptive (SPA) and Multi Pass Adaptive (MPA) convergence algorithm methods will be demonstrated through the PCB example. A Multi-Pass Adaptive static analysis will be run and the results will be compared with those from a Single-Pass Adaptive analysis.

Step 1. Open the PCB_INIT.ASM and enter Mechanica Thermal.

*Open the PCB assembly **pcb_init.asm** from the appropriate directory and enter Mechanica thermal. Examine the model. Notice that the materials, the loads and constraints are assigned.*

1. Change the your working directory to **c:\users\student\mechanica_330\module10**.
2. Open PCB_INIT.ASM.

3. Enable Mechanical Thermal.
4. Clear the **Spin Center** option in the environment settings and turn off the datum planes and coordinate systems. Review the components in the assembly.
5. Examine the model. Note that the materials, loads, and constraints are assigned.

Step 2. Create a steady state thermal analysis using SPA.

*Create a steady state thermal analysis. Name the thermal analysis **pcb_temperature_spa**. Set the convergence method to **Single Pass** and the plotting grid to 7.*

1. Create a steady state thermal analysis named **pcb_temperature_spa**.
 - Click **Analyses/Studies** in the MEC THERMAL menu.
 - Click **File > New Steady State Thermal**.
2. Set the convergence method to **Single-Pass Adaptive** as shown in the following figure.

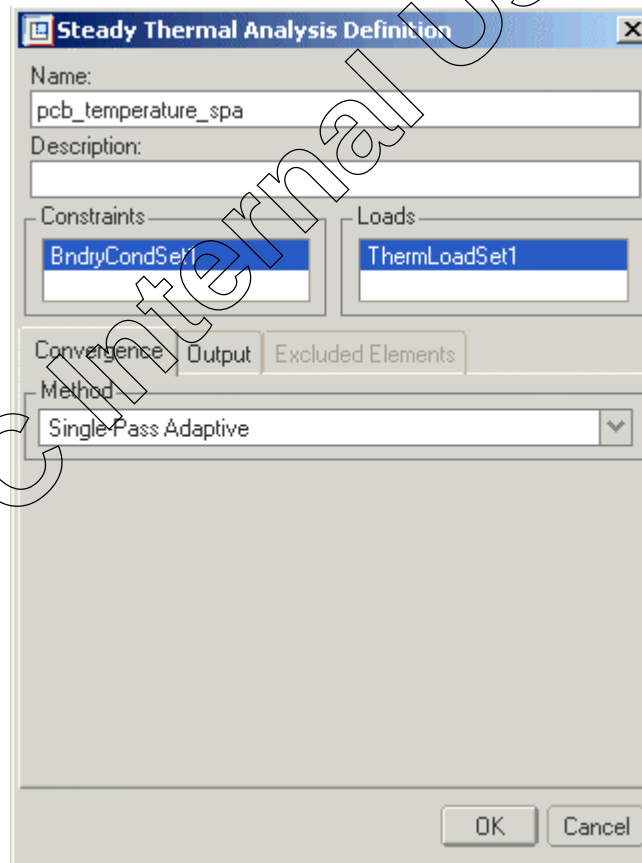


Figure 1: Thermal analysis using SPA dialog box

3. Select the Output tab and set the plotting grid to 7.
4. Click **OK**.

Step 3. Create a steady state thermal analysis, using MPA.

Create another steady state thermal analysis. Name the thermal analysis

pcb_temperature_mpa. Set the convergence method to Multi Pass and the plotting grid to 7.

1. Create a steady state thermal analysis named **pcb_temperature_mpa**.
 - Click **Analyses/Studies** in the MEC THERMAL menu.
 - Click **File > New Steady State Thermal**.
2. Set the convergence method to **Multi-Pass Adaptive** as shown in the following figure.

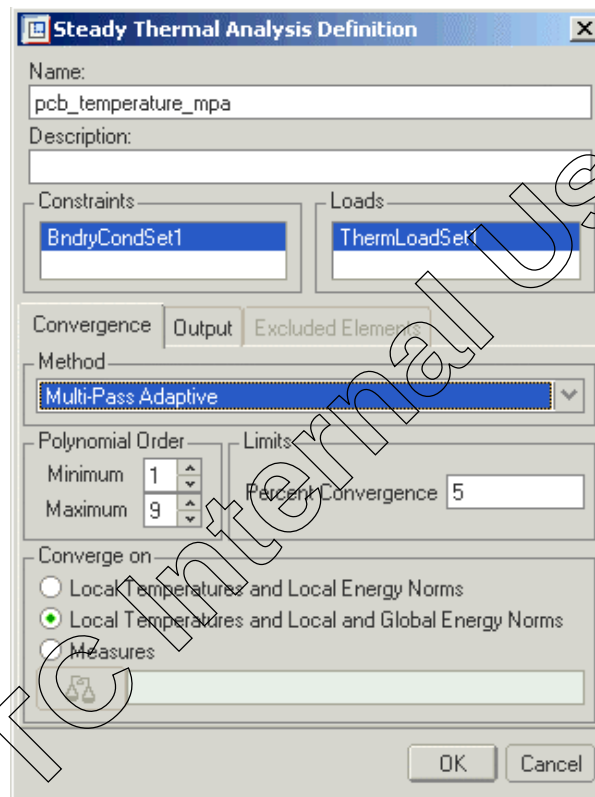


Figure 2: Thermal analysis using MPA dialog box

3. Select the Output tab and set the plotting grid to 7.
4. Click **OK**.

Step 4. Run pcb_temperature_spa analysis.

Run the **pcb_temperature_spa** analysis. Show the report file.

1. Before running the analysis, set the memory allocation from settings to half of the system RAM.
2. Run the **pcb_temperature_spa** thermal analysis.

3. Check the progress of the thermal analysis from the status.

Step 5. Run **pcb_temperature_mpa** analysis.

1. Run the **pcb_temperature_mpa** thermal analysis
2. Monitor the analysis. Notice the following changes as the analysis run progress.
 - Multiple passes are made.
 - Each time, the polynomial orders of the equations used in the solution are increased. Each pass reports fewer “elements not converged” and more equations than the pass before.
 - Eventually, all elements will converge to the specified percentage and the run will be complete.

Note:

If the analysis does not converge to the specified accuracy, Mechanica will report that, and allow you to look at the non-converged results. In either case, the overall solution quality of every run is known and can be controlled by changing the convergence percentage on the analysis dialog box. Review the Summary file carefully.

Step 6. Define a result window to display a fringe plot of temperature for the Single Pass run.

Create a result window displaying fringe plot of temperature. Name it **pcb_temp_spa**.

1. Enter the result mode from results.
2. Create a result window named **pcb_temp_spa**.
3. For the design study, select **pcb_temperature_spa**. Set the Display options as shown in the following figure.
4. Click **OK and Show**.

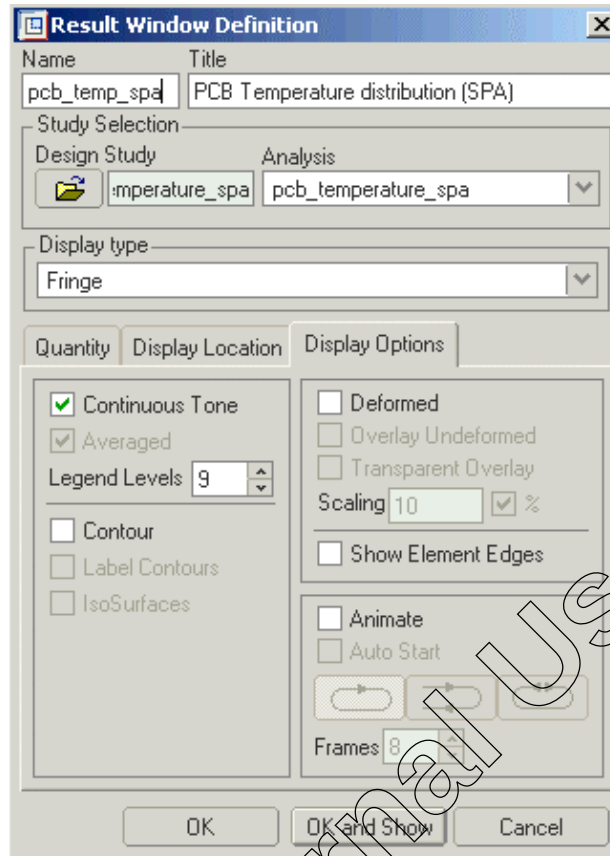


Figure 3: Result dialog box for SPA run

Step 7. Define a result window to display a fringe plot of temperature stress for the Multi Pass run.

Create another result window displaying fringe plot of temperature. Name it **pcb_temp_mpa**.

1. Create another result window named **pcb_temp_mpa**.
2. For the design study select **pcb_temperature_mpa**. Change the plotting grid to **7**. Set the Display options as shown in the following figure.
3. Click **OK and Show**.

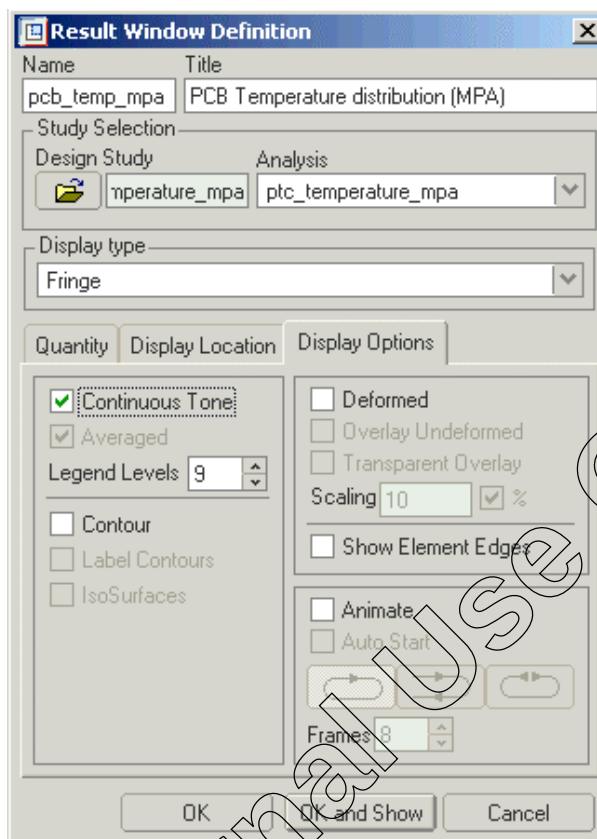


Figure 4. Result dialog box for MPA run

Step 8. Review the results.

Display both result windows `pcb_temp_spa` and `pcb_temp_mpa` created above.

1. Display both result windows for `pcb_temp_spa` and `pcb_temp_mpa` created previously.

Note:

The answers with the Multi-Pass Adaptive (MPA) and Single-Pass Adaptive algorithm (SPA) are within 10% of each other. For larger models SPA can be up to ten times faster and use up to two-thirds less disk space than MPA. As a default, you should use SPA as a solution method. MPA is good for cases where you would like to specify the convergence, and you want to manually control the convergence percentage.

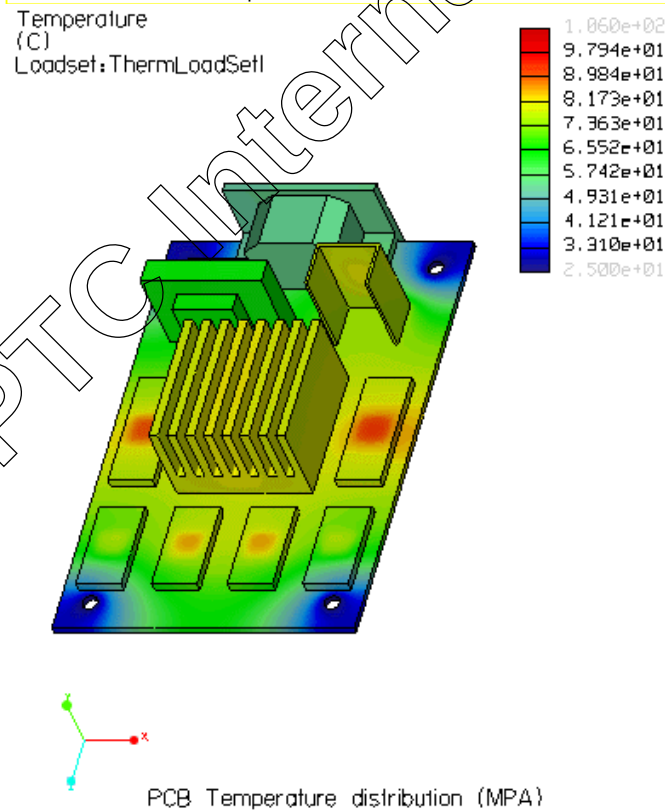
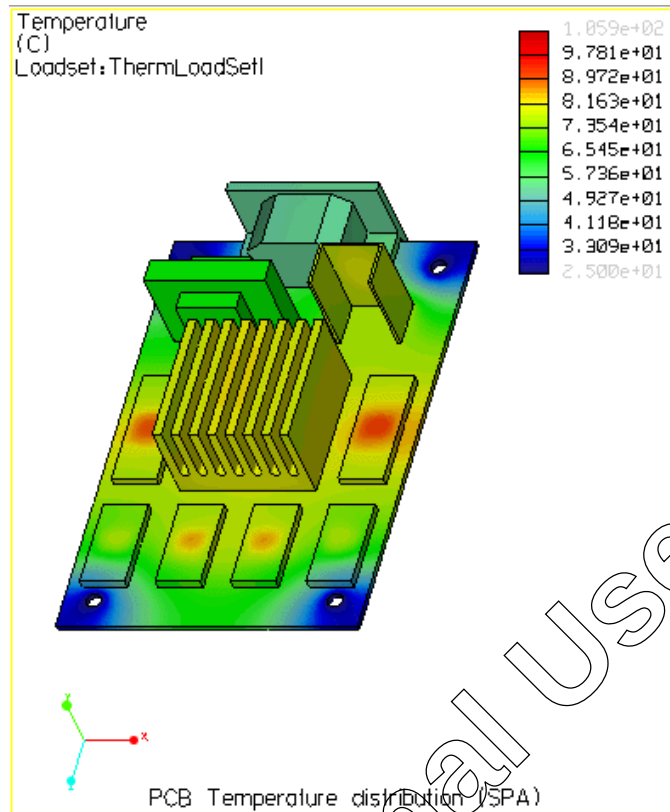


Figure 5: SPA and MPA results

2. Exit the result window. Do not save.
3. Save the model.
4. Close the window and erase all models from session.

For PTC Internal Use Only

Summary

After completing this module, you will be able to:

- Create a Single-Pass Adaptive analysis.
- Create a Multi-Pass Adaptive analysis.
- Describe the difference between Single-Pass Adaptive and Multi-Pass Adaptive analyses.

For PTC Internal Use Only

Evaluating Analysis Results

Introduction

When you begin applying Mechanica, you may select large, complex models to analyze. However, it is recommended that you initially attempt to apply Mechanica to simple, basic models in order to learn the process of building and analyzing Mechanica models.

Analyzing Models

You must apply the following steps when running Mechanica analyses:

1. Create or import geometry
2. Assign material properties
3. Define loads
4. Apply constraints
5. Create elements
6. Define the analysis
7. Run the analysis
8. Review the results
9. Assign design parameters
10. Run a sensitivity study
11. Run an optimization study
12. Update the part geometry

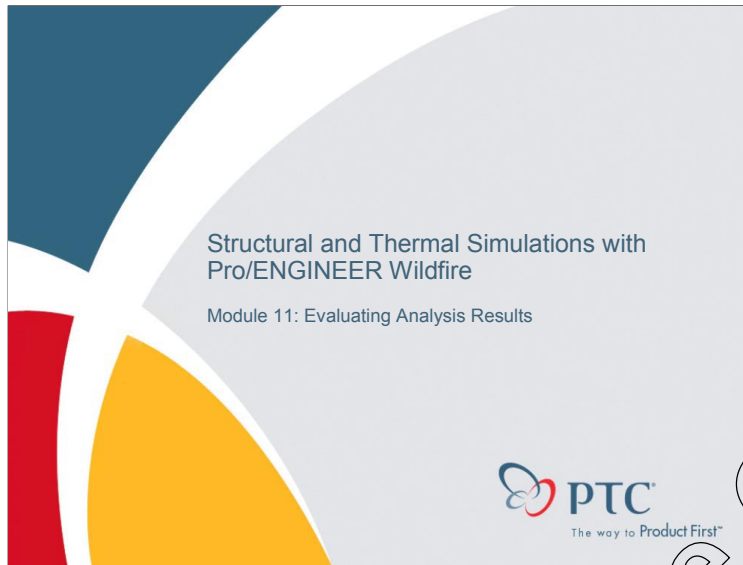
Steps 1 through 8 define an engineering analysis in order to understand how a model behaves under certain boundary conditions. Steps 9 through 12 enable you to improve a design.

Objectives

After completing this module, you will be able to:

- Describe the basic steps for building a Pro/ENGINEER Wildfire model for analysis.
- Set up and run a thermal analysis and a structural analysis, simultaneously.
- Set up and run a modal analysis and an optimization design study.

For PTC Internal Use Only



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

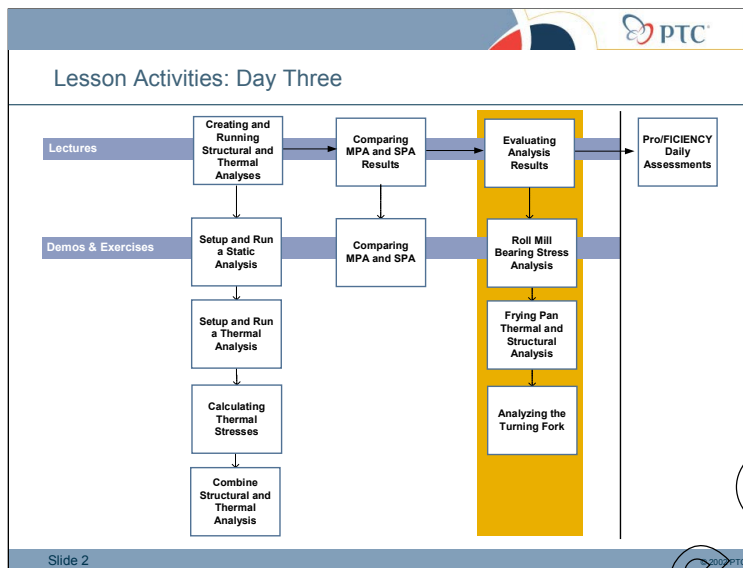
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 20 min
- Demos: 30 min
- Labs: 90 min
- Total: 2 hours 20 min



Objectives

After completing this module, you should know how to:

- Describe the basic steps for building a Pro/ENGINEER Wildfire model for analysis.
- Set up and run a thermal analysis and a structural analysis, simultaneously.
- Set up and run a modal analysis and an optimization design study.

Slide 3

Objectives

When you begin applying Mechanica, you may select large, complex models to analyze. However, it is recommended that you initially attempt to apply Mechanica to simple, basic models in order to learn the process of building and analyzing Mechanica models.

Analyzing Models

You must apply the following steps when running Mechanica analyses:

1. Create or import geometry
2. Assign material properties.
3. Define loads.
4. Apply constraints.
5. Create elements.
6. Define the analysis.
7. Run the analysis.
8. Review the results.
9. Assign design parameters.
10. Run a sensitivity study.
11. Run an optimization study.
12. Update the part geometry.

Steps 1 through 8 define an engineering analysis in order to understand how a model behaves under certain boundary conditions. Steps 9 through 12 enable you to improve a design.



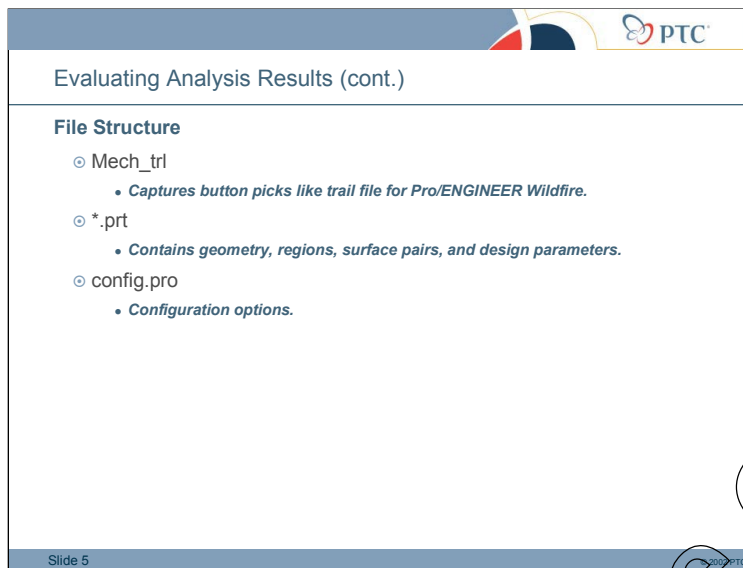
Evaluating Analysis Results

Running Analysis

- Start - Begins calculations.
- Batch - Creates a batch file called mechbatch.bat. Calculations start when the batch file is executed.
- Restart - Continues from last successfully completed p-pass.
- Summary - Monitor the run's progress.
- Directories for Output and Temporary files - Defaults to the directory where you started Mechanica.
- RAM Allocations - 50% of Physical RAM.

Slide 4

There are no instructor notes on this slide.



File Structure

Use the following for an example and assume that the following conditions apply:

- The working directory where Pro/ENGINEER Wildfire was started is mech_stuff.
- The model name is test_model.
- The analysis name is static_test.

Then, the following files are created:

- The mechevnt file (which provides a record of all commands in Mechanica) is appended to the current Pro/ENGINEER Wildfire trail file.
- The files test_model.prt.* and test_model.mdb (Mechanica database) are created upon the first *File:Save*. The .prt file contains all of the Pro/ENGINEER Wildfire geometry plus regions, surface pairs and design parameters. The .mdb file contains all Mechanica entities (loads, constraints, properties, and all analysis forms).
- The files test_model.mbk is created upon the second *File:Save* and can be used as a backup.
- The directory static_test is created when running the analysis. In this directory, the following files are saved: (1) a copy of the *.mdb and *.prt file is kept here for future reference, (2) the *.rpt is a text file containing the model and analysis summary, (3) the *.pas file contains a time log of various analysis and file operations, (4) the *.err file records any error messages occurring during the run 5. The directory static_test.tmp is created when running the analysis. This directory contains temporary files (typically the stiffness matrix) and is deleted upon analysis completion. Disk space requirements must be addressed.

Note: The config.mech file contains user defined configuration options for the Mechanica entities.

PTC

Evaluating Analysis Results (cont.)

Defining a Result Window

1. New result window
2. Window name
3. Highlight study directory
 - *Do NOT double click*
4. Desired results
 - *Quantity*
 - *Location*
 - *Display*
5. Accept and Show

Slide 6

Notes to Instructor:

- You will be displaying the next slide, which is a duplicate of this slide without the screen shots.
- Demonstrate the process using the screen shots.

Defining a Result Window

Defining a Result window is a four-step process:

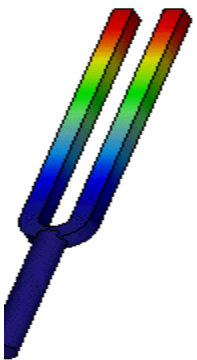
- Select "Create" from the Result Window Menu.
- Enter a name for the window to be created.
- Highlight the study directory where the desired results are stored and accept the window.
Note: When selecting the study directory, do not double-click on the directory name. This will access the next level in the directory structure.
- Define the result window (Quantity, Location, Display).
- Results from any directory are accessible independently of the model currently displayed.

PTC

Evaluating Analysis Results (cont.)

Defining a Result Window

1. New result window
2. Window name
3. Highlight study directory
 - *Do NOT double click*
4. Desired results
 - *Quantity*
 - *Location*
 - *Display*
5. Accept and Show

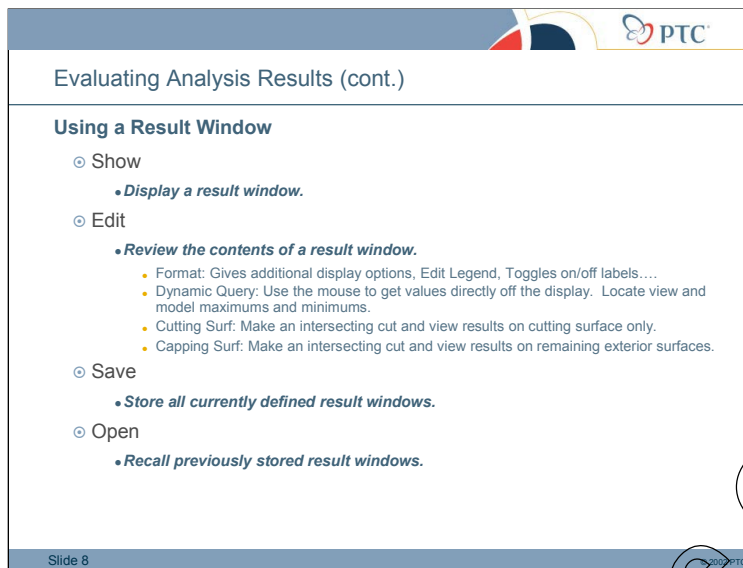


Slide 7

Defining a Result Window

Defining a Result window is a four-step process:

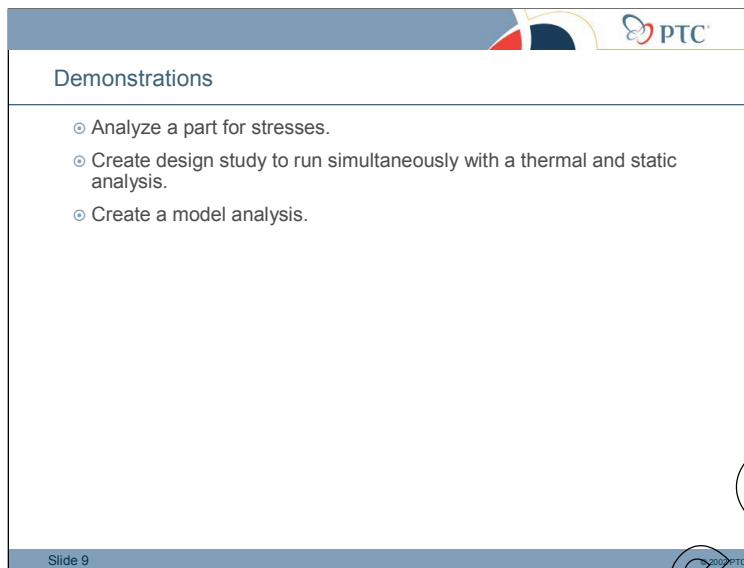
1. Select "Create" from the Result Window Menu.
 2. Enter a name for the window to be created.
 3. Highlight the study directory where the desired results are stored and accept the window.
Note: When selecting the study directory, do not double-click on the directory name. This will access the next level in the directory structure.
 4. Define the result window (Quantity, Location, Display).
- Results from any directory are accessible independently of the model currently displayed.



Using a Result Window

Taking a closer look at the Result Window:

- Note that the *Create, Review, Copy, Delete, Change, Save and Load* options only apply to the Edit column while the *Show* option only applies to the Show column. The **Review** option is used to redefine the Quantity, Location and Display Type for the window.
- The **Change** option is used to access results from different analyses in a multidisciplinary design study (for example: an optimization study with modal and static analyses).
- **Save & Open** – Allows you to store and recall the current result windows definitions.
- A summary of the result window parameters (Study, Analysis, Loadcase, Quantity, Location, Display) is displayed for the window currently selected in the Edit column.



Demonstrations

In this demonstration, I will:

- Review the basic steps for building a Pro/ENGINEER Wildfire model for analysis.
- Analyze a roller mill bearing part for stresses.
- De-feature a model.
- Analyze a pan assembly for heat distribution and strength
- Design a tuning fork for the musical note "G"

You will repeat the same steps in the lab exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 10

Precision Learning

- Learn: by listening to lectures, watching demos, and completing lab exercises.
- Assess: your progress with Pro/FICIENCY assessment questions.
- Improve: The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Note to Instructors:

Getting Started

- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Exams

For each course, Pro/FICIENCY exams are based upon the topics covered each day.

How to use it

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After completing this module, you should know how to:

- Describe the basic steps for building a Pro/ENGINEER Wildfire model for analysis.
- Set up and run a thermal analysis and a structural analysis, simultaneously.
- Set up and run a modal analysis and an optimization design study.

Slide 11

Summary

There are no instructor notes on this slide.

Module 11 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Roll Mill Bearing Stress Analysis

Introduction

In the first exercise, you will run a static analysis to compute stress distribution on the mill bearing part. First need to de-feature the part and cut it in half.

Objectives

After successfully completing this exercise, you will know how to:

- Analyze a roller mill bearing part for stresses.
- Simplify a model for analysis.

Scenario

You will de-feature a model and then set up and run an analysis. Understanding the Mechanica process will enable you to expand your skills into more advanced techniques such as optimization.

The model you will be analyzing in this exercise is a mill bearing part. Notice the model contains a large number of features, specifically, many holes that will increase the size of the Mechanica model. Remember that suppressing features that are inconsequential to the analysis is recommended practice. Also note the model is symmetric and that the boundary conditions are symmetric as well. Consequently, you will perform two tasks before entering Mechanica:

- Remove features from the part.
- Cut the model in half.

Step 1. In Pro/ENGINEER Wildfire, open the model. De-feature the production model to create a simulation model. The production model is carefully constructed. All the features after Cut id 2826 can be suppressed.

- Set the working directory to the folder that corresponds to the name of the current module. Open the mill_bearing.prt and check that units are mmNs.
- Suppress the hole ID 809 in the first pattern and all the features below. Use Clip Suppress.

1. Set the working directory to C:\users\student\mechanica_330\module_11.
2. Open MILL_BEARING.PRT.

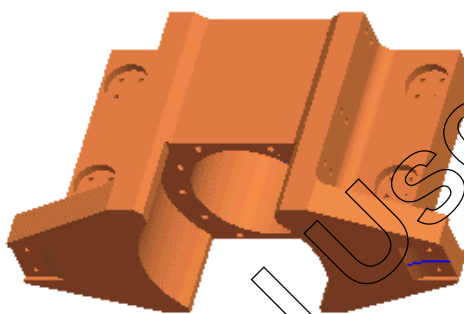


Figure 1: Mill bearing model

Note:

There are many pin-holes in your model that will not transmit any load but will increase the number of elements in the Mechanics model significantly.

The model is also symmetric. It can be cut in half to reduce the element count. The model is actually constructed in half, and merged to create the whole model.

3. Suppress the Hole id 809 in the first pattern and all the features below. Some features will be removed and the model will be cut in half for symmetry.
 - From the main menu click **Edit > Feature Operations > Suppress > Clip**.
 - Select the hole feature id 809 in the first pattern from the model tree.
 - Click **OK > Done**.

Step 2. Enter Mechanics Structure and apply material properties.

- Enter Mechanics Structure and assign Steel material properties to the part.

1. Enable Mechanics Structure.
2. Assign STEEL material properties to the model.

Step 3. Create two loads that belong to the same load set.

- Create a bearing load of 1,000,000 Newtons applied to the large inner cylindrical surface as shown below in the positive Y-direction.
- Create another surface load of 500,000 Newtons in the positive Z-direction applied to the lip of the inner protrusion as shown. Make to place it in the same load set as the previous load.

1. Create a bearing load of 1,000,000 Newtons in the positive Y-direction applied to the large inner cylindrical surface as shown in the following figure.

- Click **Model > Loads > New > Bearing**.
- Leave the name of the load as default.
- Using the selection tool, select the surface as shown in the following figure.

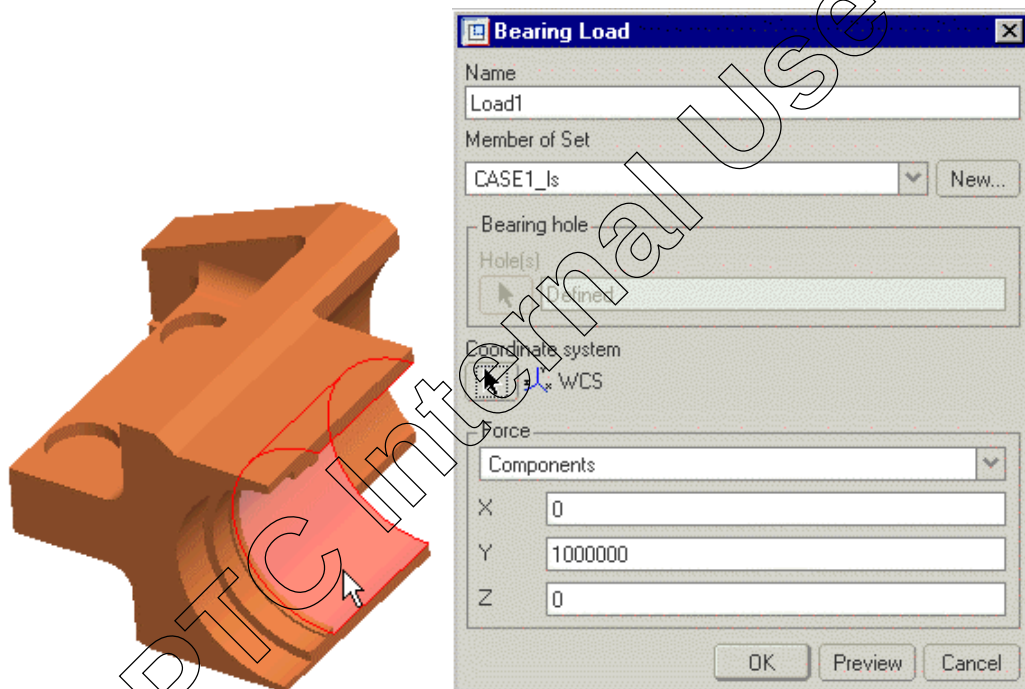


Figure 2: Bearing Load in the Y-direction

- Enter value **1000000** for Y.
 - Click **OK**.
2. Mechanica displays a warning that it is expecting an entire cylindrical surface for the bearing load. Accept it.
3. Create another surface load of 500,000 Newtons in the positive Z-direction applied to the lip of the inner protrusion as shown in the following figure.
- Click **Model > Loads > New > Surface**.
 - Leave the name of the load as default.

- Ensure that this load is being added to the same load set as the previous load.
- Using the selection tool, select the surface as shown in the following figure.

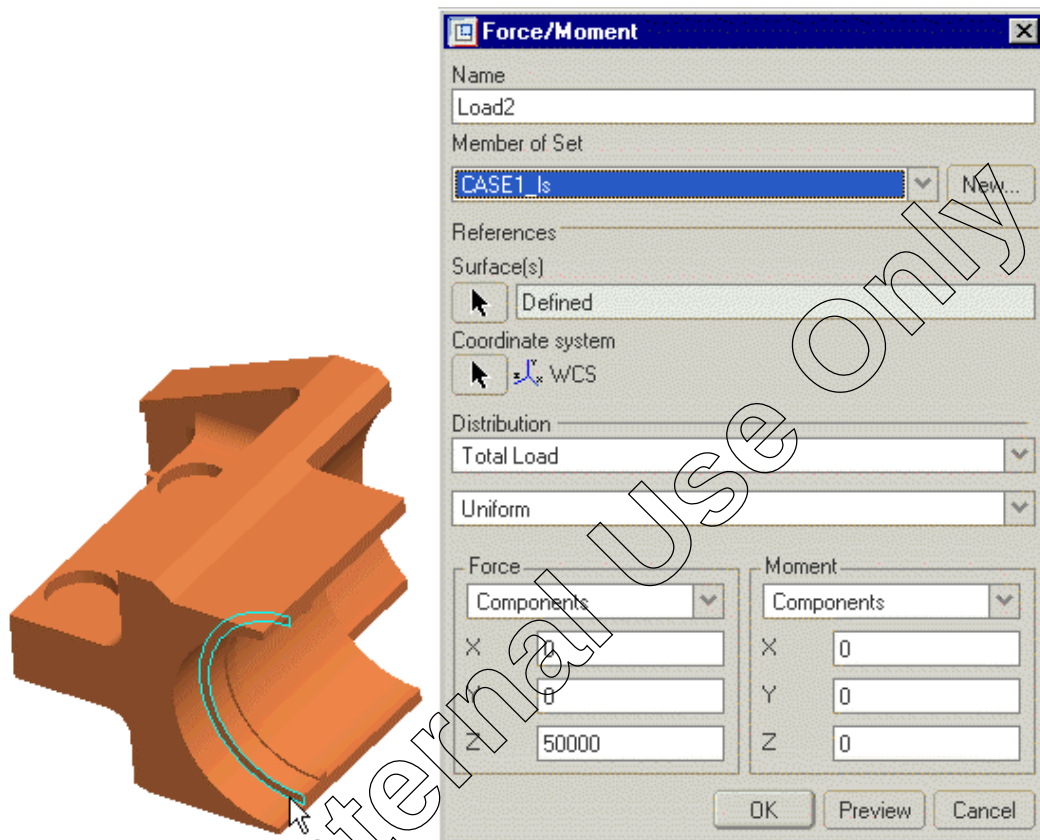


Figure 3: Surface Load in the Z-direction

4. The model should appear as shown in the following figure.

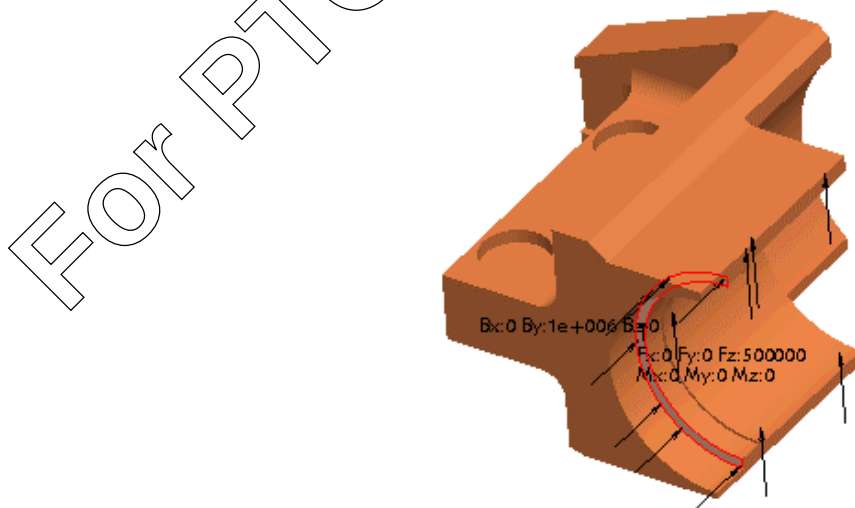


Figure 4: Finished Loads

Step 4. Create three constraints that belong to the same constraint set.

- Create a sliding constraint on the pin hole surfaces. Constrain the bottom flat surface of all three pin holes. Fix the Y-direction translation and free the other DOFs.
- Create another sliding constraint, make sure to use the same constraint set. Select the flat surface to constrain. Fix the Z-direction translation and free the other DOFs.
- Create a symmetric constraint, make sure to use the same constraint set. Select the plane of symmetry. Fix the X-direction translation and the Y and Z-direction rotation. Free the other DOFs.

1. Click **Model > Constraints > New > Surface** and create a sliding constraint on the pin hole surfaces by fixing Y-translation and freeing all other DOFs.

- Select the surfaces as shown in the following figure.

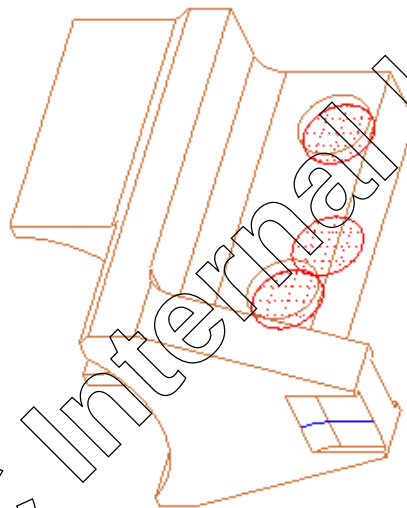


Figure 5: Surfaces for the first constraint

- Click **OK**.

Note:

A sliding constraint allows a constrained surface to slide in its own plane but not perpendicular to it. Therefore, the translation normal to the plane is fixed.

2. Click **Model > Constraints > New > Surface** and create another sliding constraint. Fix the Z-translation and free all other DOFs.

- Select the flat surface to constrain as shown in the following figure.
- Ensure that the constraint is part of the same constraint set.

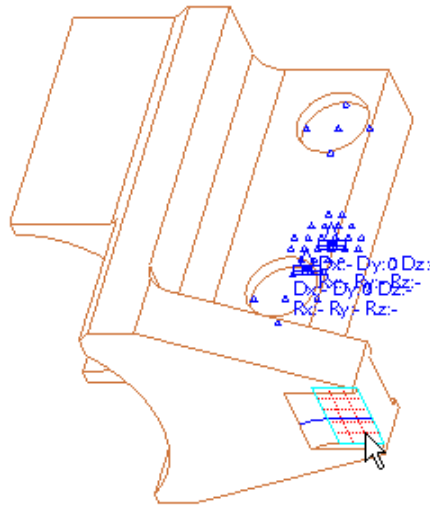


Figure 6: Surfaces for the second constraint

- Click **OK**.
3. Click **Model > Constraints > New > Surface** and create yet another constraint. Fix the X-direction translation and the Y and Z-direction rotation. Free the other DOFs.
- Select the plane of symmetry surface as shown in the following figure.
 - Ensure that the constraint is part of the same constraint set.

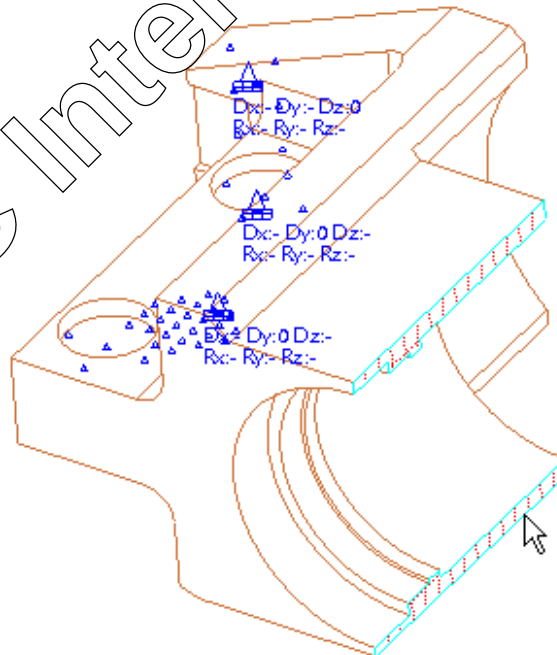


Figure 7: Surfaces for the third constraint

- Click **OK > Done/Return**.

Step 5. Define and run the analysis.

- Create a static SPA analysis named **mill_static**. Accept the other default options.
- Run the analysis and monitor the process.
- Review the summary file.

1. In the MEC STRUCT menu click **Analyses/Studies**.

2. Click **File > New Static** to create a new static SPA analysis named **mill_static**. Accept the other default options.

3. Run the analysis and monitor the process.

4. Review the summary file. Note the following information.

- Total Elapsed Time _____
- Total CPU Time _____
- How many elements are in the model? _____
- What is the error of the solution? _____ %.
- Measure: max_disp_mag _____
- Measure: max_stress_vm _____

Step 6. Create result windows and View the results.

- Create two result windows, a repeated displacement animation fringe plot over the entire model and repeated von Mises stress animation fringe plot over the entire model.
- Show both result windows.

1. Create two result windows:

- Animated fringe plot of displacement magnitude over the entire model.
- Animated fringe plot of von Mises stress over the entire model.

2. Show both result windows. The result windows should appear as shown in the following figure. Switch to the appropriate orientations to visually verify the effects of the boundary conditions.

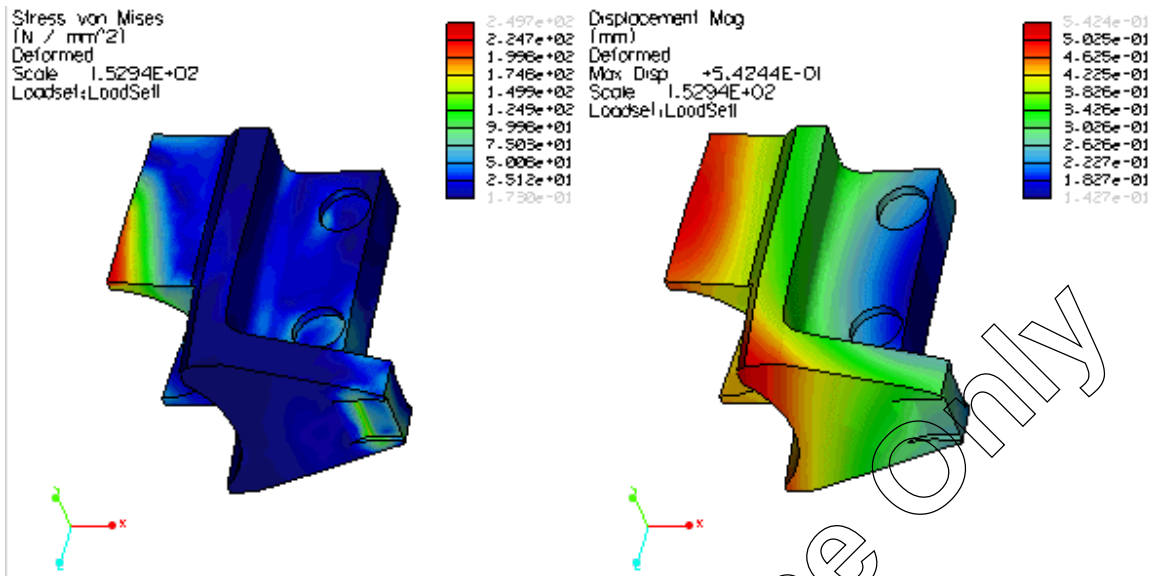


Figure 8: Stress and Displacement Results

Note:

In the stress results window, notice the "hot" spots (in red) near the symmetric boundary.

Step 7. Focus on the "hot" spots (von Mises stress higher than 100 MPa).

- If the yield stress of steel is 200 MPa and you want to maintain a safety factor of 2, no stress in the model should exceed 100 MPa. Edit the legend values and change the fringe legend so that any stress above 100 MPa (N/mm²) is indicated in red.
1. In the stress window, edit the legend values and change the fringe legend so that any stress above 100 MPa (N/mm²) is indicated in red by selecting Edit > Legend Value and clicking the top value. Change the highest value in the legend to **100**.

Note:

If the yield stress of steel is 200 MPa and you want to maintain a safety factor of 2, no stress in the model should exceed 100 MPa.

2. Exit the result window. Do not save.
3. Save the model.
4. Close the window and erase all models from session.

Exercise 2: Frying Pan Thermal and Structural Analysis

Introduction

In this exercise, you create a static analysis to see if the frying pan assembly can sustain its own weight then create a thermal analysis to investigate how the heat is conducted through the handle.

Objectives

After successfully completing this exercise, you will know how to:

- Create design study to run simultaneously a thermal and a static analysis.
- Analyze a pan assembly for heat distribution and strength.

Scenario

In this exercise, you will use Mechanica to ensure that a frying pan distributes heat evenly and is structurally strong.

The model consists of two simple parts—the pan itself and the handle. The bolts have been removed to simplify the assembly. You will perform two types of analysis:

- A thermal analysis will simulate the pan sitting on a hot stove.
- A static analysis will determine if the handle is strong enough to withstand the weight of the pan.

Step 1. Retrieve the pan model and create the datum curves for region creation.

- Open PAN.PRT. Make sure that unit system is millimeter Newton Seconds (mmNs).
- Create a sketched circular datum curves on the bottom of the pan. Select the bottom surface of the pan for sketching plane and any orientation reference. Sketch a 177.8 mm diameter circle concentric to the pan.
- Create another circular datum curve with a 127 mm diameter using the same procedures.

In order to model the heat load from the stove, a surface region needs to be created in the pan model.

1. Open PAN.PRT.
2. Create a sketched circular datum curves on the bottom of the pan. Select the bottom surface of the pan for sketching plane and any orientation reference.
 - Sketch a 177.8 mm diameter circle concentric to the pan.
 - Complete the curve creation.

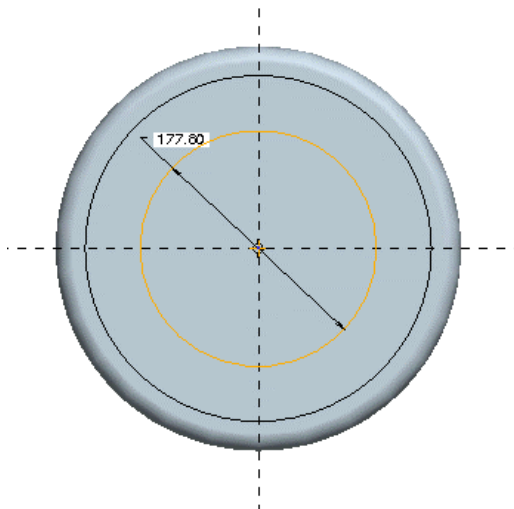


Figure 9: First datum curve sketch

3. Create another circular datum curve with a 127 mm diameter using the same procedures. The model should appear as shown in the following figure.



Figure 10: Bottom of The Pan With Two Datum Curves

Note:

The datum curves will be used for regions and must be applied to the part, not to the assembly. In addition, the curves must be created as separate features or the region creation will fail.

Step 2. Create two surface regions in Mechanica.

- Enter Mechanica Structure. Create a first surface region. Select the outer datum curve for the splitting curve and the bottom surface of the pan to split.
- Create another surface region. This time select the inner curve and the bottom inner surface of the pan. The bottom, inner surface is split into two surfaces (for a total of 3 surfaces on the bottom of the pan).
- Switch to the standard application. Save and close the window.

1. Enter Mechanica Structure.
2. Click **Model > Features > Surf Region > Create > Select > Done** to create the first surface region. Select the outer datum curve for the splitting curve and the bottom surface of the pan to split.
3. Create another surface region. This time select the inner curve and the bottom inner surface of the pan. The bottom, inner surface is split into two surfaces (for a total of 3 surfaces on the bottom of the pan).
4. Save the model.
5. Close the window.

Step 3. Retrieve the handle model and create surface regions in it.

Retrieve the PAN_HANDLE.PRT and review it. A curve is to be used for a surface region. Enter Mechanica and create a region on each of the four sides of the handle.

- Open PAN_HANDLE.PRT. Make sure that unit system is mmNs. A curve is already created in the model.
- Enter Mechanica and create surface regions to split the top handle surface into two surface regions. Select the datum curve for the splitting curve and select the top, bottom, and side surfaces of the handle for surfaces to split.
- Show how to verify that the regions are created properly. Switch to the standard application. Save and close the window.

1. Open PAN_HANDLE.PRT. Note that a curve has already been created in the model.
2. Enable Mechanica Structure.
3. Click **Model > Features > Surf Region > Create > Select > Done** and create surface regions by selecting the datum curve as the splitting curve and select the top, bottom, and side surfaces of the handle for surfaces to split.
4. The four surfaces are split into two surfaces each, for a total of eight surfaces.
5. Verify that four regions have been created using **Show All**.
6. Save the model.
7. Close the window.

Step 4. Retrieve the frying pan assembly model and create surface regions in it.

- Open the frying_pan.asm. Make sure that unit system is mmNs.
- Enter Mechanica Thermal. Create a surface region. Select the pan, when prompted for component. Notice that you can create a surface region from the assembly in the part. Select the rectangular datum curve, when prompted for curves and select the outer surface of the pan that intersects the handle.

1. Open the FRYING_PAN.ASM.
2. Enable Mechanica Thermal.
3. Click **Model > Features > Surf Region > Create > Select > Done** and create a surface region.
 - Select PAN.PRT, when prompted for component. Notice that you can create a surface region from the assembly on the part.
 - Select the rectangular datum curve, when prompted for curves and select the outer surface of PAN.PRT that intersects PAN_HANDLE.PRT.

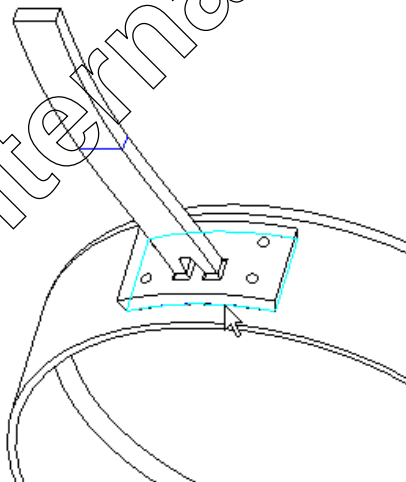


Figure 11: Curve for surface region

Step 5. Assign the material properties.

Assign AL6061 to the pan and STEEL to the handle part.

1. Assign AL6061 to PAN.PRT and STEEL to PAN_HANDLE.PRT.

Note:

Aluminum has more than 4 times the thermal conductivity of steel, making a good material for cooking. On the other hand, steel has lower thermal conductivity, making it a good candidate for the handle because of the high thermal resistance.

Step 6. Simulate the cooking heat with the appropriate boundary conditions.

- Simulate the cooking heat from the gas stove (12,500 BTU/hr = 3,661,000 mW) with a heat load. Create a surface heat load. Enter **cooking_heat** as the name and new load set named **cooking_heat** as well. Select total load and enter 3661000 for value. For the surface select the ring shape surface region.
1. Simulate the cooking heat from the gas stove (12,500 BTU/hr = 3,661,000 mW) with a heat load.
 - Click **Model > Heat Loads > New > Surface** to create a surface heat load.
 - Enter **cooking_heat** as the name and new load set named **cooking_heat** as well.
 - Enter a value of **3661000** for the total load.
 - Using the selection tool, select the ring shape surface region to apply the load.

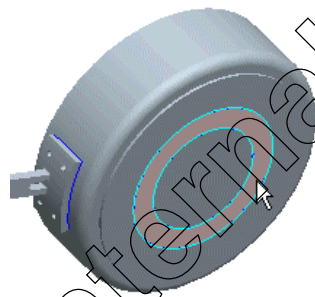


Figure 12: Heat load surface

- Click **OK**.

Step 7. Simulate the heat loss due to ambience air with convection boundary conditions.

- Create a convective boundary condition. Name it **cool_air** and place in a new constraint set named **all_convections**.
 - Select the following surfaces: The narrow top surface, Internal and external surfaces of the round and internal and external surfaces of side. When selecting the external side surfaces, the rectangular region where the pan intersects the handle is excluded.
 - Enter 0.4 for the convection coefficient and 25 for the bulk temperature.
1. Click **Model > Bndry Conds > New > Conv Cond** and create a convective boundary condition, named **cool_air**.
 2. Create a new constraint set named **all_convections**.
 3. Select the following surfaces as shown in the following figure:

- The narrow top surface.
- Internal and external surfaces of the round. When selecting the external side surfaces, the rectangular region where the pan intersects the handle is excluded.
- Internal and external surfaces of side.

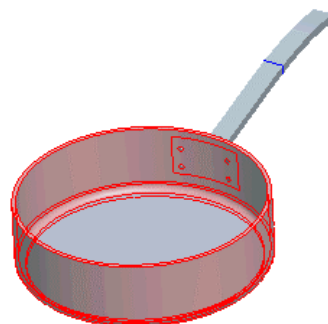


Figure 13: Air Convection surfaces

4. Enter **0.4** for the Convection Coefficient and **25** for the Bulk Temperature.
5. Click **OK**.

Step 8. Simulate the heat loss due on the cooking surface with a convection boundary condition.

- Create a convective boundary condition. Name it **cooked_food** and place in the **all_convections** set.
- Select the interior bottom surface of the pan where the food is cooked and enter 0.5 for the convection coefficient and 100 for the bulk temperature.

1. Create a convective boundary condition. Name it **cooked_food**.
2. Place it in the constraint set **all_convections**.
3. Select the interior bottom surface of the pan where the food is cooked.

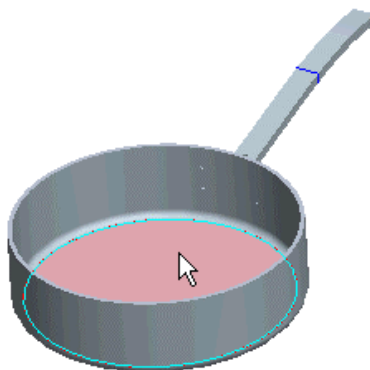


Figure 14: Cooked food convection surface

4. Enter **0.5** for the Convection Coefficient and **100** for the Bulk Temperature.
5. Click **OK**.

Step 9. Simulate the heat loss through the handle surfaces not held by the cook's hand with a convection boundary condition.

- Create a convective boundary condition. Name it **handle_heat_loss** and place in the **all_convections** set.
- Select the handle surfaces that are not touched by hand and enter 0.35 for the convection coefficient and 25 for the bulk temperature.

1. Create a convective boundary condition named **handle_heat_loss**.
2. Place it in the constraint set **all_convections**.
3. Select the handle surfaces that are not touched by hand, as shown in the following figure.

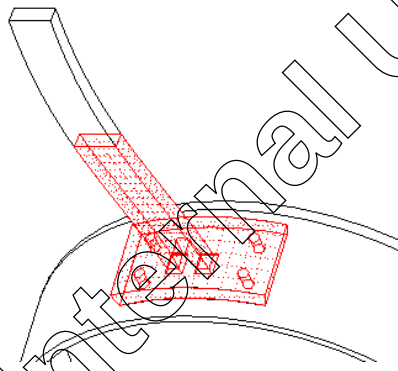


Figure 15: Cooked food convection surface

4. Enter **0.35** for the Convection Coefficient and **25** for the Bulk Temperature.
5. Click **OK**.

Step 10. Create a steady-state thermal analysis.

- Create steady thermal analysis. Name the analysis **pan_therm**. Make sure you select the correct constraint set **all_convections** and the correct load set **cooking_heat**. Accept all other default options.

1. Create steady thermal analysis named **pan_therm**.
 - Click **Analyses/Studies** in the MEC THERMAL menu.
 - Click **File > New Steady State Thermal**.
2. Make sure you select the constraint set **all_convections** and the correct load set **cooking_heat**. Accept all other default options.

Step 11. Simulate the structural aspect of the pan model. Define the constraint (where the cook is holding the handle) and the load (gravity).

- Enter Mechanics Structure. Define a surface constraint that simulates the holding hand. Select the four surface regions held by hand. Fix X, Y and Z-direction translations and free the other DOFs.
- Create a gravity load. Enter -9810 for Z.

1. Enable Mechanics Structure.
2. Define a surface constraint that simulates the holding hand.
3. Select the four surface regions held by hand.

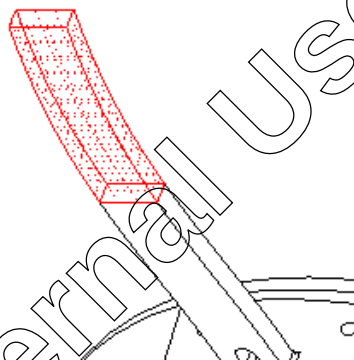


Figure 16: Constraint surfaces

4. Fix X, Y and Z-direction translations and free the other DOFs.
5. Create a gravity load.
6. Enter **-9810** for Z.

Step 12. Create a static structural analysis.

- Create a static analysis. Name the analysis pan_static.
- Make sure you select the correct constraint and load sets and accept all other default options.

1. Create a static analysis.

- Click **Analyses/Studies** in the MEC STRUCT menu.
- Click **File > New Static**.
- Name the analysis **pan_static**.

2. Make sure you select the correct constraint and load sets and accept all other default options.

Step 13. Create a design study that includes the structural and thermal analyses.

- Create a standard design study. Name it **pan_study**. Select both the **pan_therm** and the **pan_static**.

1. Create a standard design study named **pan_study**.

- Click **Analyses/Studies** in the MEC STRUCT menu.
- Click **File > New Design Study**.
- Select both the **pan_therm** and the **pan_static**.
- Complete the design study setup.

Step 14. Run the design study and review the summary file.

- Run the design study and monitor the process.
- Review the summary file.

1. Run the design study and monitor the process using the summary file. Note the following information when the analysis is finished.

- Total Elapsed Time _____
- Total CPU Time _____
- How many elements are in the model? _____
- What is the error of the solution? _____ %
- What is the maximum displacement magnitude? _____ mm
- What is the maximum von Mises stress? _____ N/mm²
- What is the maximum temperature? _____ °C

Step 15. Create result windows to display the results.

- Create an animated result window of displacement magnitude over the model.
- Create an animated fringe plot to display von Mises stress. As expected, high stresses occur at the junction between the handle and the mounting plate. However, since these stress values are much lower than the material's yield strength, these stresses are not a big concern.
- Create a result window to display temperature distribution. Notice that the temperature is evenly distributed around the thick aluminum bottom.
- Exit the Results window. Save and erase the model.

1. Create an animated result window to view the displacement magnitude over the entire model.

2. Create an animated fringe plot to display von Mises stress over the entire model. As expected, high stresses occur at the junction between the handle and the mounting plate.

However, since these stress values are much lower than the material's yield strength, these stresses are not a big concern.

3. Create a result window to display temperature distribution. Notice that the temperature is evenly distributed around the thick aluminum bottom.
4. Exit the Results window. Do not save.
5. Save the model.
6. Close the window and erase all models from session.

For PTC Internal Use Only

Exercise 3: Analyzing the Turning Fork

Introduction

In this exercise, you will run a modal analysis and compute the first modal frequency. Then an optimization study is used to tune a fork to the musical note "G", which vibrates at 384 Hz.

Objectives

After successfully completing this exercise, you will know how to:

- Create a modal analysis.
- Design a tuning fork for the musical note "G".

Scenario

Tuning forks produce a sound by vibrating at specific resonant frequencies. If the resonant frequency is within the audible range of 20 Hz to 20,000 Hz, then the frequency can be classified as a musical note. For example, the note, middle-C, vibrates at 256 Hz, whereas the note, E, vibrates at a frequency of 320 Hz. Piano tuners and guitar players use tuning forks to tune their instruments to certain musical notes.

In this exercise, you tune a fork to the musical note "G", which vibrates at 384 Hz.

Step 1. Open the FORK.PRT and check the units.

- Open the part FORK.PRT. Make sure the units are in mm, Newtons, and seconds.
1. Open FORK.PRT.
 2. Review the units and make sure the units are in mm, Newtons, and seconds.

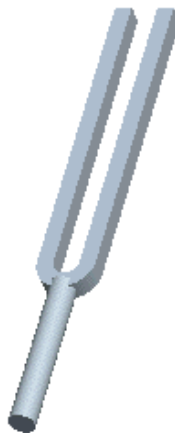


Figure 17: The tuning fork

Step 2. Enter Mechanica and assign the material properties.

- Enter Mechanica Structure and apply steel material properties to the tuning fork part.
1. Enable Mechanica Structure
 2. Apply STEEL material properties to FORK.PRT.

Step 3. Add constraints on the handle.

- Fully constrain the handle surface of the fork (the cylindrical protrusion).
1. Click **Model > Constraints > New > Surface** and create a new constraint.
 2. Fully constrain the handle surface of FORK.PRT (the cylindrical protrusion).

Step 4. Create and run a modal analysis.

- Create a modal analysis. Name it **fork_mode** and change the number of modes to **1**, because we are interested in the lowest, or primary mode. Accept all other defaults.
 - Run the analysis.
1. Create a modal analysis.
 - Click **Analyses/Studies** in the MEC STRUCT menu.
 - Click **File > New Modal**.
 - Name the analysis **fork_mode**.
 - Change the number of modes to **1**, to analyze the lowest, or primary mode.
 2. Accept all other defaults.
 3. Run the analysis.

Step 5. Review the summary file.

- Monitor the process and review the summary file.
1. Verify the accuracy of the model and note the following:
 - Total Elapsed Time _____
 - Total CPU Time _____
 - How many elements are in the model? _____
 - What is the error of your solution? _____ %
 - What is the modal frequency? _____ Hz. You have determined that this frequency is approximately G# (G-sharp) which is close, but half a note from your desired frequency of G.

Step 6. Define a design parameter. The length of the tuning fork will dictate its resonant frequency.

- Create a design parameter to vary the length of the fork. Pick the 91.24 mm dimension of the fork. Name the parameter **Fork_length** and define the range from a minimum of 90 to a maximum of 110.

1. Create a design parameter to vary the length of the fork.

- Click **Model > Dsgn Controls > Design Params.**
- Click **Create**.
- Select the fork feature and select the 91.24 mm dimension as shown in the following figure.

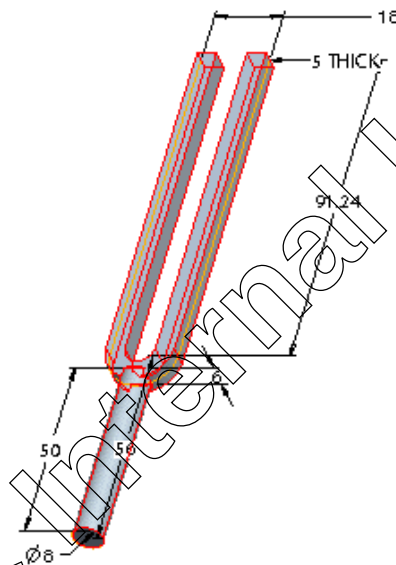


Figure 18: Design Parameter On The Tuning Fork Length

- Enter **Fork_length** as the new name for design parameter.
- Define the range from a minimum of **90** to a maximum of **110**.
- Click **Accept > Done**.

Step 7. Create an optimization study to optimize the model. To achieve the desired note of G, the resonant frequency must be 384 Hz. This will be accomplished by varying the length of the fork.

- Create an optimization design study. Name the study **fork_opt**. Clear the Goal check box and define the Limits on Measures and select modal frequency.
- Set modal_frequency equal to **384**. Select the Fork_length parameter in the parameters section and accept the default interval.
- Run the optimization.

1. Create an optimization design study named **fork_opt**.
 - Click **Analyses/Studies** in the MEC STRUCT menu.
 - Click **File > New Design Study**.
 - Select the type as Optimization from the drop-down list.
 - Clear the Goal check box.
 - Define the Limits on Measures and select modal frequency.
 - Set modal_frequency equal to **384**.
 - Select the Fork_length parameter in the parameters section and accept the default interval.
2. The dialog box should appear as shown in the following figure.

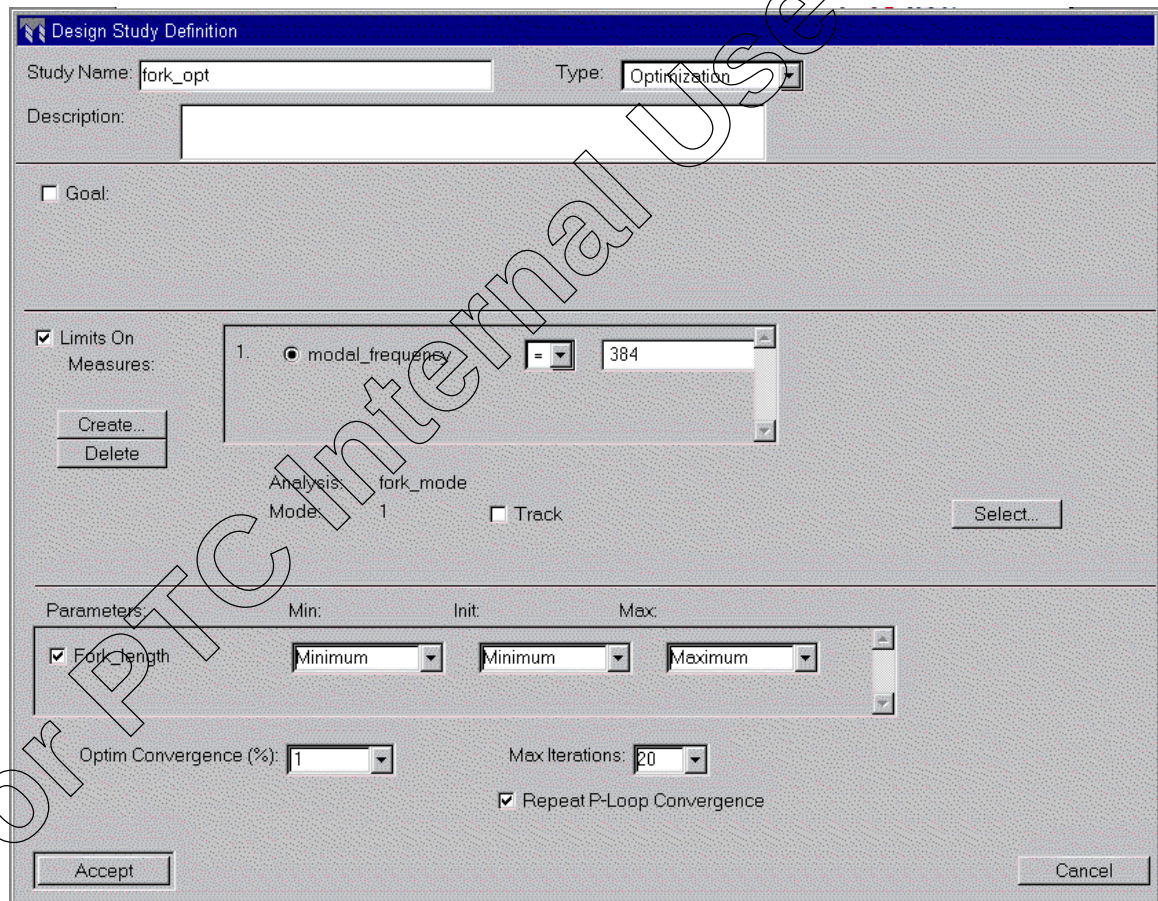


Figure 19: Optimization dialog box

3. Click **Accept**.
4. Run the optimization.

Step 8. Verify the results of the optimization.

- Monitor the run and review the report file and the results.

1. Monitor the run and review the report file and the results.

- Enter the total Elapsed time. _____ sec
- What is the final value of design parameter? _____ mm
- What is the optimized modal frequency? _____ Hz
- Is this within the 1% specified in the optimization convergence? _____

Step 9. Update the model using the optimized design parameter.

- Review the optimization history, when prompted and answer **yes** to all questions.
- When you have completed this process, the fork will be tuned to G.
- Create a plot of the modal frequency vs. optimization pass.
- Exit the result interface. Switch to the standard application. Save and erase the model.

1. Review the optimization history of the fork_opt study from design control.

- Click **Model > Dsgn Controls > Optimize Hist > Search Study > List**.
- Select the study **fork_opt**.
- When prompted and answer **Y** to all questions. When you have completed this process, the fork will be tuned to G and the fork looks like the following figure.

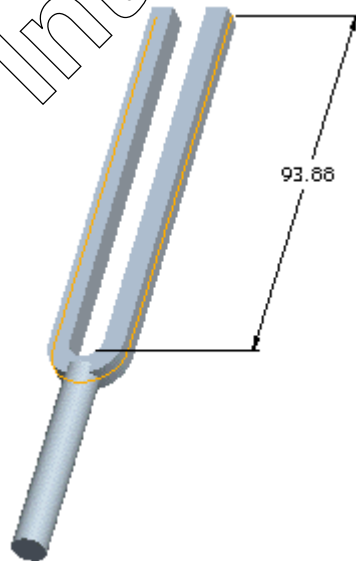


Figure 20: The optimized fork

2. Create a plot of the modal frequency vs. optimization pass.

- Click **Results** and create a new result window.

- Configure the result window dialog box as shown in the following figure.

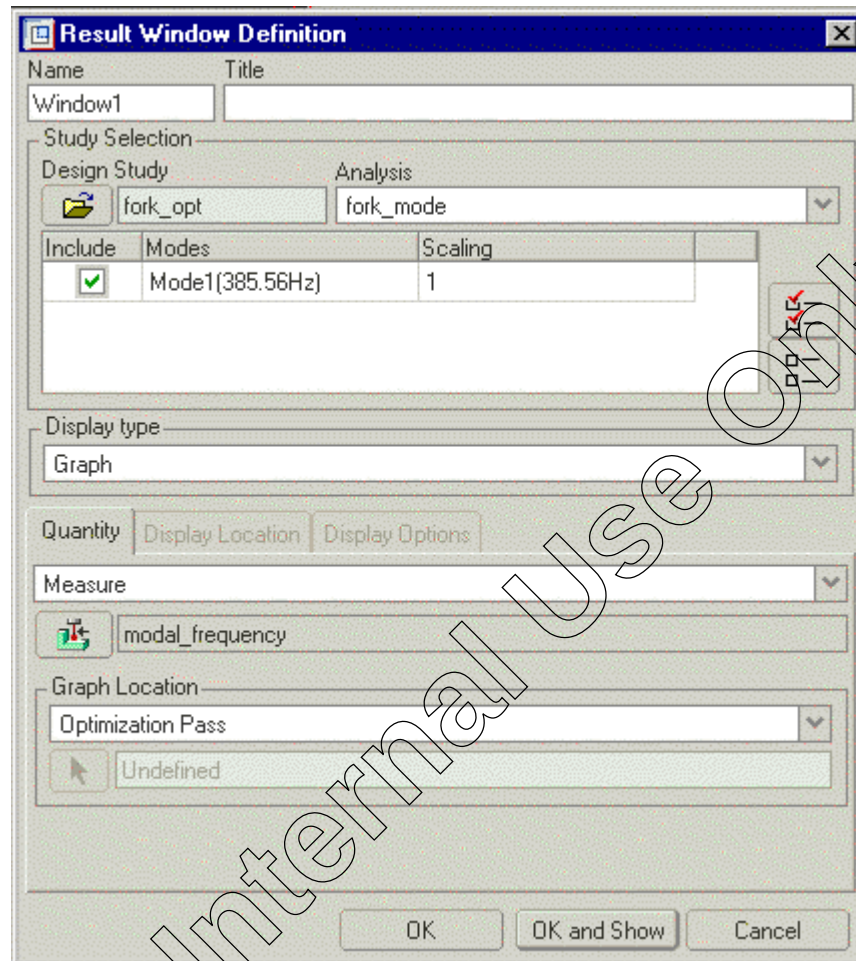


Figure 21: Result Window Definition

- Click **OK and Show**. The result plot should look like the following figure.

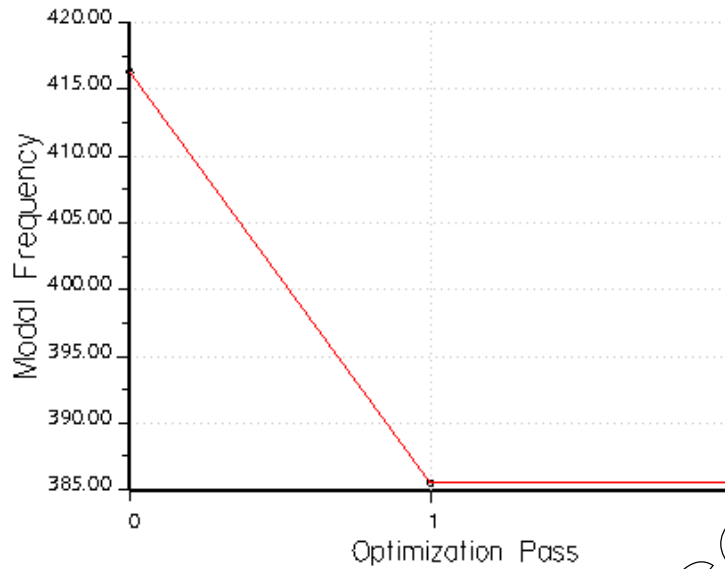


Figure 22: Modal Frequency vs. Optimization Pass

3. Exit the Results window. Do not save.
4. Save the model.
5. Close the window and erase all models from session.

Summary

After completing this module, you will be able to:

- Describe the basic steps for building a Pro/ENGINEER Wildfire model for analysis.
- Set up and run a thermal analysis and a structural analysis, simultaneously.
- Set up and run a modal analysis and an optimization design study.

For PTC Internal Use Only

For PTC Internal Use Only

Running Sensitivity Studies

Introduction

So far, you have learned how to model and analyze a part in Mechanica. In this lesson, you will learn the tools that enable you to determine how sensitive measures such as stress, displacement, and mass are to changes in model parameters. You will also learn in a later module how to optimize your model by setting goals and limits on measures, while varying model parameters.

Sensitivity studies enable you to more fully understand the effects of varying the design parameters on your model. You can use these studies to determine how sensitive a particular quantity, such as von Mises stress, is to variations of a particular dimension parameter.

Running Global Sensitivity Studies

Global sensitivity studies are used to generate a picture of how measures respond to changing a design parameter over a specified range (usually a large range, hence the name global). Global sensitivity studies provide the big picture and will be most useful in understanding the overall effect of a given parameter.

Running a global sensitivity study helps you understand in detail how changes are affecting your part. You will then use this information to set up the optimization study. Normally, you would run global sensitivity studies on all the parameters that survived the local sensitivity study, but in the interest of time, you will run global sensitivity studies for limited parameters.

In the exercises that follow, you will run a global sensitivity on the slot length parameter to get a better understanding of its impact on your model. However, by doing so, you can only determine the optimal value for a given parameter if it were varied alone. But would this optimal parameter value be the same if other design parameters were changed simultaneously? The answer is most likely no. However, this does give you a good starting position for the optimizer.

Running Local Sensitivity Studies

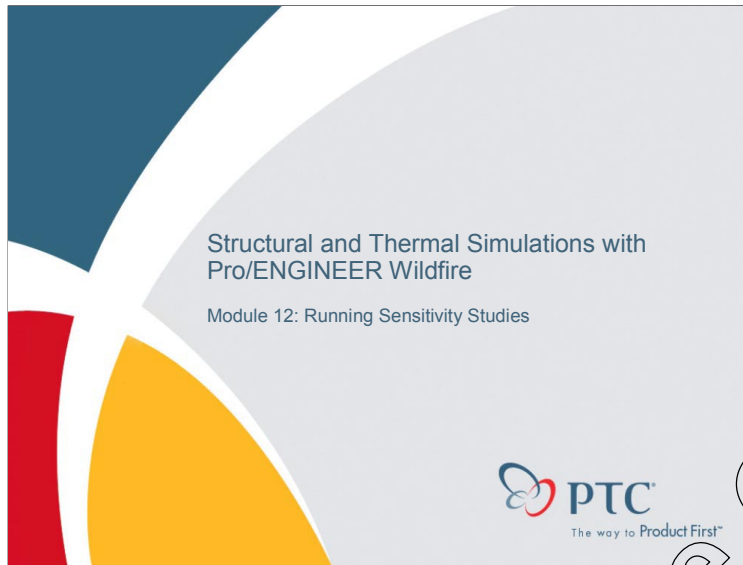
Local Sensitivity Studies are used to calculate the sensitivity of your model's measures to perturbations (very slight changes) in parameters.

Mechanica uses local sensitivity to perturb a dimension parameter by 1% to estimate the derivative of your measure with respect to the parameter. Thus, for a given location in parameter space, you may take a "snapshot" of the sensitivity of your model subject to all parameters. If parameter A induces a slope of your measure of 100, while parameter B induces a slope of your measure of 0.01, then you may postulate that changing the parameter A will have a bigger effect than working with parameter B. Therefore, you use local sensitivity to narrow your selection of dimension parameters to the most important ones.

Objectives

After completing this module, you will be able to:

- Describe the purpose of design parameters.
- Create design parameters and measures.
- Set up and run a local and global sensitivity studies.
- Review sensitivity plots.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

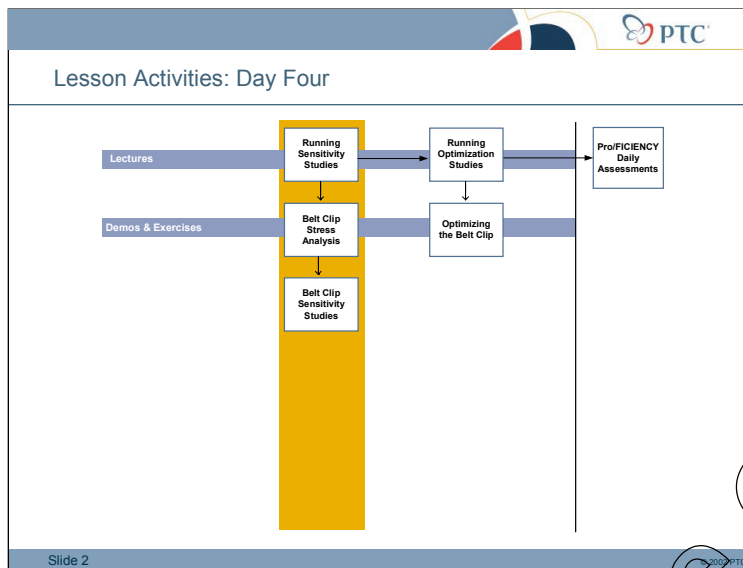
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 30 min
- Demos: 45 min
- Labs: 120 min
- Total: 3 hours 15 min



Objectives

After completing this module, you should be able to:

- Describe the purpose of design parameters.
- Create design parameters and measures.
- Set up and run a local and global sensitivity studies.
- Review sensitivity plots.

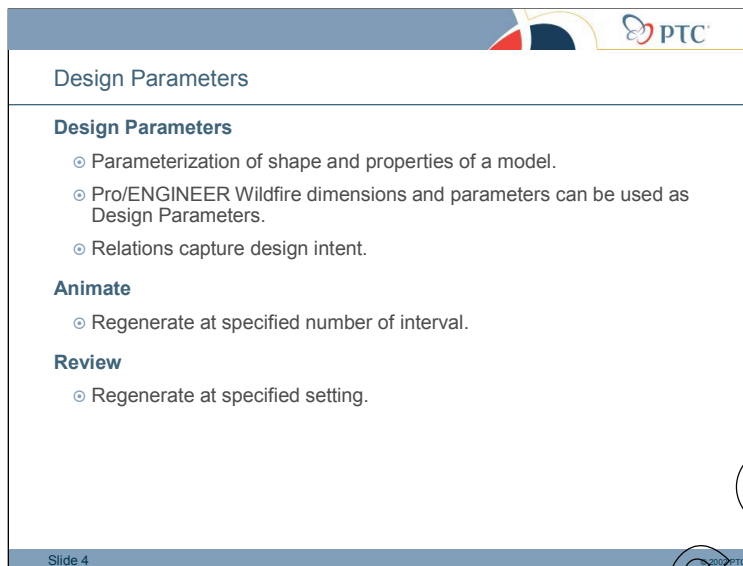
Slide 3

Objectives

So far, you have learned how to model and analyze a part in Mechanica. In this lesson, you will learn the tools that enable you to determine how sensitive measures such as stress, displacement, and mass are to changes in model parameters. You will also learn in a later module how to optimize your model by setting goals and limits on measures, while varying model parameters.

Sensitivity studies enable you to more fully understand the effects of varying the design parameters on your model. You can use these studies to determine how sensitive a particular quantity, such as von Mises stress, is to variations of a particular dimension parameter. Mechanica provides the following sensitivity capabilities:

- Global Sensitivity Studies
- Local Sensitivity Studies



Design Parameters

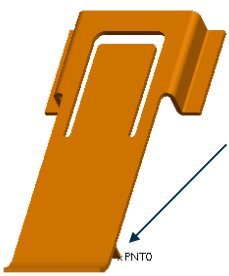
- Design Parameters are used to define what can change in a model when running Sensitivity and Optimization Studies.
- Design Parameters are used **ONLY** in Integrated Mode and Linked Mode. They allow the parameterization of any Pro/ENGINEER Wildfire dimension (including section dimension) as well as the use Pro/ENGINEER Wildfire parameters. The following command sequence is used: *Structure > Model > Dsgn Controls > Design Params*.
- To accomplish the desired changes on the geometry in Integrated Mode, we need to do the following:
 - Define a relation in Pro/ENGINEER Wildfire, relating "d8" to the length of the plate (trans_curve) Define a Design Parameter called "trans_curve", assigning minimum and maximum values of 4 and 6 to that dimension.
 - Define a Design Parameter on called "radius_change", assigning minimum and maximum values of 0.5 and 1.5 to that dimension.
 - When defining a design parameter, the **Type** (Dimension, Pro/ENGINEER Wildfire Parameter, Section Dimension) must be selected first, along with the feature dimension (click on **Select**).

PTC

Design Parameters (cont.)

Measures

- Additional probes in the model
 - **Quantity**
 - **Component**
 - **Spatial Evaluation**
 - At a point
 - Maximum
 - Minimum
 - **Time/Freq Evaluation**
 - At each step
 - Maximum
 - Minimum
 - At time



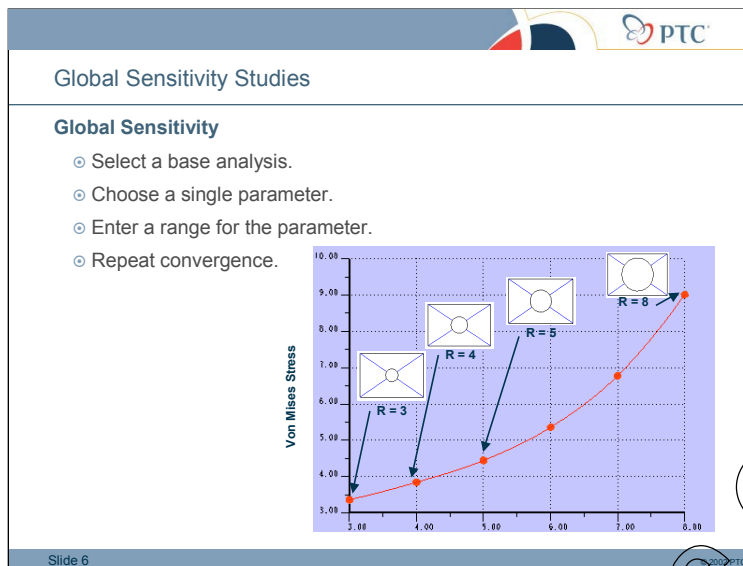
Displacement Magnitude at a point

PNT0

Slide 5

Measures

Identify Measures (default or user defined) to be used to control the optimization goal and limits (for example: minimize mass while keeping stress at a particular location below a given value).



Global Sensitivity

Global sensitivity studies are used to generate a picture of how measures respond to changing a design parameter over a specified range (usually a large range, hence the name global). Global sensitivity studies provide the big picture and will be most useful in understanding the overall effect of a given parameter.

Running a global sensitivity study helps you understand in detail how changes are affecting your part. You will then use this information to set up the optimization study. Normally you would run global sensitivity studies on all the parameters that survived the local sensitivity study, but in the interest of time, you will run global sensitivity studies for limited parameters.

A global sensitivity study allows the user to obtain information on the way a particular quantity is affected by a parameter over a range of values. The following inputs are required to define a global sensitivity analysis:

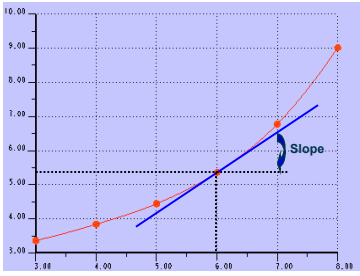
- **Study name:** Mechanica will use this name to create the directory in which results are stored.
- **Type:** Select "Global Sensitivity" from the design study types available.
- **Description:** Descriptions will save you time when retrieving old analysis files or performing disk maintenance operations.
- **Base analysis:** This is the analysis to be performed for each value of the parameter
- **Parameters:** Select a single parameter from the list. Selecting several parameters removes the ability to isolate the effect of one parameter on the quantity of interest. If several parameters are selected, they are incremented (from minimum to maximum) together.
- **Range:** Enter a range of *values* (Integrated Mode) or *percentages* (Independent mode) through which the parameter will vary.
- **Number of intervals:** This is the number of intervals (between the minimum and maximum value) for which you want to run the analysis.
- **Repeat P-Loop convergence:** When this option is selected, Mechanica performs the convergence algorithm at every value of the parameter. Otherwise, edge orders (p-levels) determined for the model at the initial parameter value are used.
- **Element update options (Independent Mode only):** The mesh has to adapt to changing geometry. This can be done in two ways:
 - **Smooth:** When Smooth is selected, AutoGEM checks the validity of elements and attempts to fix invalid elements by moving points, while retaining the original set of elements. Validity is defined by the AutoGEM settings limits (face angles, edge angles, etc.).
 - **Regenerate:** if parameters impose geometric changes on the mesh that are "beyond repair" then new elements must be generated.
 - Determining which option should be used can be done using Shape-Review and Shape-Animate.



Local Sensitivity Studies

Local Sensitivity

- Select a base analysis.
- Select all the parameters to study.
- Enter settings for the parameters.
- Analysis conducted with all parameters at specified settings.
- Analysis conducted for each parameter at +1% from specified setting.
- Slopes are calculated in all directions.



• Slope of the global curve at the current variable setting
 • Each variable changed separately
 • Results are ONLY ACCURATE LOCALLY

Slide 7
© 2007 PTC

Local Sensitivity

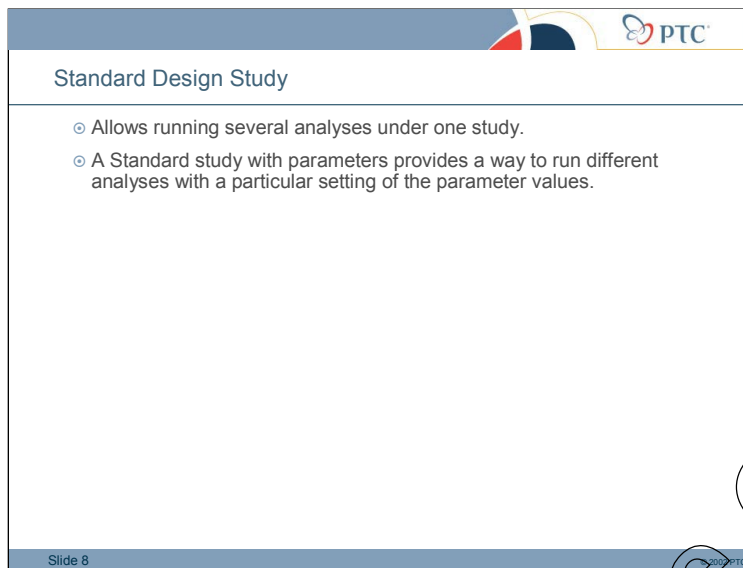
Local Sensitivity Studies are used to calculate the sensitivity of your model's measures to perturbations (very slight changes) in parameters.

Mechanica uses local sensitivity to perturb a dimension parameter by 1% to estimate the derivative of your measure with respect to the parameter. Thus, for a given location in parameter space, you may take a "snapshot" of the sensitivity of your model subject to all parameters. If parameter A induces a slope of your measure of 100, while parameter B induces a slope of your measure of 0.01, then you may postulate that changing the parameter A will have a bigger effect than working with parameter B. Therefore, you use local sensitivity to narrow your selection of dimension parameters to the most important ones.

- Local Sensitivity design studies are used to find out which design variables have the most influence on the design.
- The process of a Local Sensitivity Analysis is defined below:
 1. The base analysis is performed at the specified parameter settings.
 2. The analysis is then repeated with each parameter set a +/- 1% from its original value while all other parameters remain unchanged (this is also known as a perturbation analysis). The 1% change is based on 1% of the difference between the minimum and maximum design variable settings. If the current value is set at the maximum of the range, the perturbation is set at 1% towards the minimum setting, otherwise the perturbation is set at 1% towards the maximum.
 3. Sensitivity slopes are obtained by comparing results of steps 1 and 2.
 4. Slopes are displayed graphically by extending the line in both directions. The graphs can be compared in order to determine which design variables have the most influence.

The following inputs are used to define a local sensitivity study:

- **Study name:** Mechanica will use this name to create the directory in which results will be stored.
- **Type:** Select "Local Sensitivity" from the design study types available .
- **Description:** These descriptions are used for keeping track of analysis performed. These will save you time when retrieving old analysis files or performing disk maintenance operations.
- **Base analysis:** This is the analysis to be performed for different values of the parameters considered in sensitivity study.
- **Parameters:** Select parameters to be compared in terms of their impact on the quantity of interest (for example: Sensitivity of Maximum von Mises stress to changes in radius, trans_curve).
- **Setting:** Enter a *value* (about which the local sensitivity analysis is to be performed).
- **Element update options (Independent Mode only):** See discussion in section on "Global Sensitivity"



Design Study

A standard design study can be used to run a base analysis at specific parameter values. This allows the user to run consequential scenarios without modifying actual part dimensions. The following inputs are used to define a standard design study:

- **Study name:** Mechanica will use this name to create the directory in which results will be stored.
- **Type:** Select "Standard" from the design study types available.
- **Description:** These descriptions are used for keeping track of analysis performed. These will save you time when retrieving old analysis files or performing disk maintenance operations.
- **Base analysis:** Select one or more analysis to be performed with the particular set of parameters.
- **Parameters:** Select parameters to be used.
- **Settings:** Specify values for parameters.



Demonstrations

- Setup a stress analysis using the belt clip.
- Setup a local and global sensitivity study for the belt clip.
- Create design parameters for the PCB assembly.
- Create and run local and global sensitivity studies for the PCB assembly.

Slide 9

© 2022 PTC

Demonstrations

In this demonstration, I will:

- Setup a stress analysis using the belt clip.
- Setup a local and global sensitivity study for the belt clip.
- Create design parameters for the PCB assembly.
- Create and run local and global sensitivity studies for the PCB assembly.

You will repeat the same steps in the lab exercises.



Summary

After completing this module, you should be able to:

- ◉ Describe the purpose of design parameters.
- ◉ Create design parameters and measures.
- ◉ Set up and run a local and global sensitivity studies.
- ◉ Review sensitivity plots.

Slide 10

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 12 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Belt Clip Stress Analysis

Introduction

In this exercise, you will setup a static analysis for the clip belt and run it. Then create design parameters and a deflection measure.

Objectives

After successfully completing this exercise, you will know how to:

- Setup a static analysis.
- Define dimension design parameters and measures.

Scenario

The belt clip model you will be optimizing fits onto a portable electronic device. The part is inserted and released from its latch by applying a 2.5 N force downward on the tab. The catch that protrudes down from the curved tip moves up and must clear a latch that is 7 mm high, but the maximum deflection cannot exceed 9 mm. In addition, the stresses must be kept to a minimum without increasing the weight of the part.

First, you will create a base static analysis to compute the stress and strain distribution over the entire model. Then you will capture the model variations by defining design parameters.

Step 1. Open CLIP.PRT model and build the simulation model by cutting away half of the symmetric model.

- Check that units are mm, Newtons and Seconds.

1. Set the working directory to C:\users\student\mechanica_330\module_12_13.
2. Open CLIP.PRT.

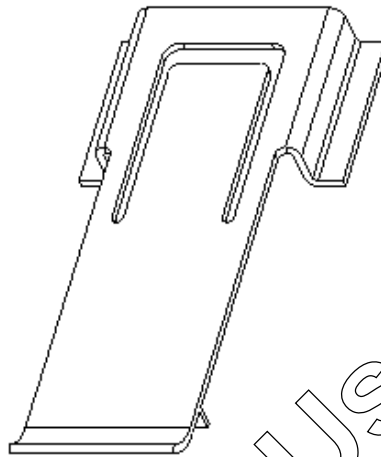


Figure 1: CLIP.PRT

3. Cut away half of CLIP.PRT.
 - Select DTM2 as the Sketching Plane and an appropriate surface as the sketcher orientation reference.
 - Sketch a rectangle to remove the left half of the clip and complete the feature as shown in the following figure.

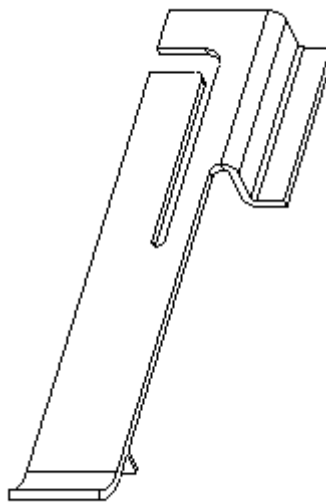


Figure 2: The Symmetric Clip Cut in Half

Step 2. Create a datum point for the measure.

You want to make sure the catch will displace enough to clear the latch on the mating part. You will now add a measure to track the displacement of a point on the tip of the catch, but first you need to create the datum point at the point of interest.

1. Create a datum point as shown in the following figure. Select the vertex of the catch to create the point, as shown.

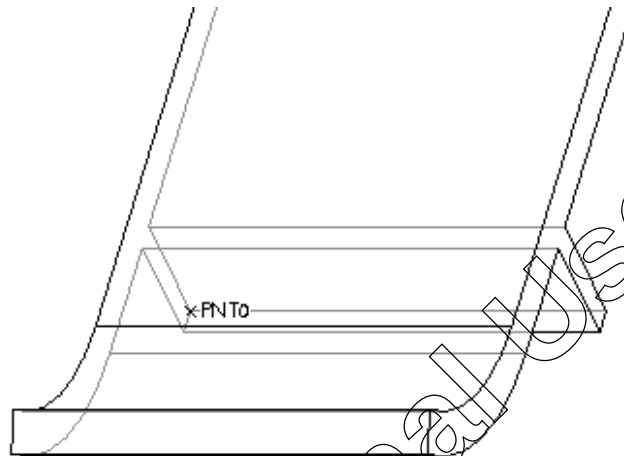


Figure 3: Datum point at the tip of the catch.

Step 3. Create a surface region in Mechanica to apply the load.

- Enter Mechanica Structure and create a surface region by sketching the curve.
- Select the top surface of the tab as the sketching and DTM3 as the reference plane. Specify the appropriate sketching references and draw a horizontal line across the tab.
- Dimension the curve 6.35 from the edge of the tab.
- Select the top surface of the clip to split, where the sketch resides.
- Verify the region

1. Enable Mechanica Structure.
2. Click **Model > Features > Surf Region > Create > Sketch > Done** and create a surface region by sketching the curve.
3. Select the top surface of the tab as the sketching plane, select **Bottom** and select **DTM3** as the reference plane.

4. Specify the appropriate sketching references and draw a horizontal line across the tab as shown in the following figure.
5. Dimension the curve 6.35 from the end of the tab.

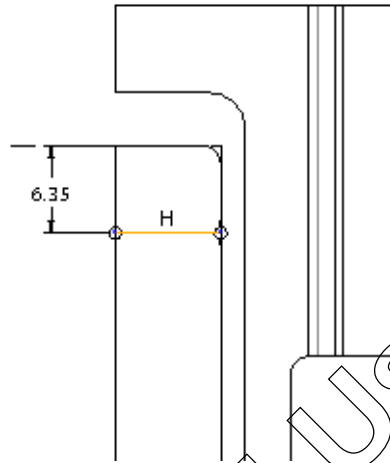


Figure 4: Curve for Surface region

6. Select the top surface of the clip to be split by the surfacing region.
7. Verify the split surface region using **Show All**.

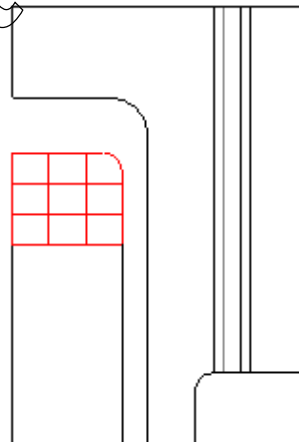


Figure 5: Surface Region

Note:

A finger pressing on a localized area represented by a region will apply the load on the tab.

Step 4. Create the shell mid-surfaces.

- Create the surface pairs using Idealization, Shell, Midsurfaces and Auto Detect and test the compression.
1. Create the surface pairs and test the compression.
 - Click **Idealization > Shell > Midsurfaces > Auto Detect > Compress > Shell Only > Show Compress.**
 2. The compressed model should look like the following figure.

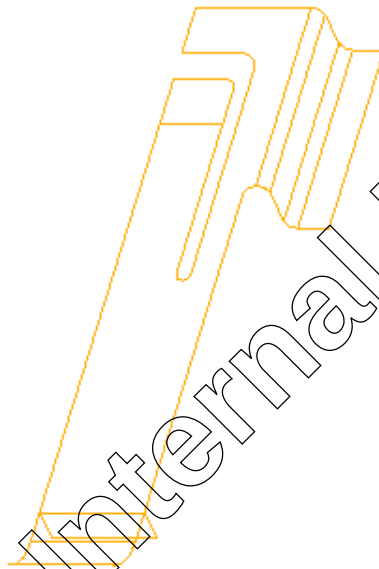


Figure 6: The Compressed clip model

3. Click **Done > Done/Return > Done/Return.**

Step 5. Set the material properties.

- Assign NYLON material properties to the clip. Edit the NYLON properties and assign **85 N/mm²** for the Tensile Yield Stress as a failure criteria
1. Assign NYLON material properties to the clip.
 2. Edit the NYLON properties and assign values for failure criteria.
 - Click the Failure Criteria tab.
 - Select the **Distortion Energy** (von Mises) option from the drop-down list.
 - Enter **85 N/mm²** for the Tensile Yield Stress.
 - Click **OK** and close the Materials dialog box.

Note:

The Tensile Yield Stress strength entered does not affect the calculations that Mechanica makes during the analysis. The purpose of this value is to provide a more convenient option for plotting stress results.

Step 6. AutoGEM the model and check the shell elements created.

- AutoGEM the model and show the elements.
1. Create solid elements in the model using AutoGEM.
 - Click **AutoGEM** in the STRC MODEL menu.
 - Click **Create** to create the elements.
 2. View the elements and shrink them by 10%
 - Click **View > Simulation Display**.
 - In the Mesh tab select the check box Shrink Elements and edit the value to **10**.

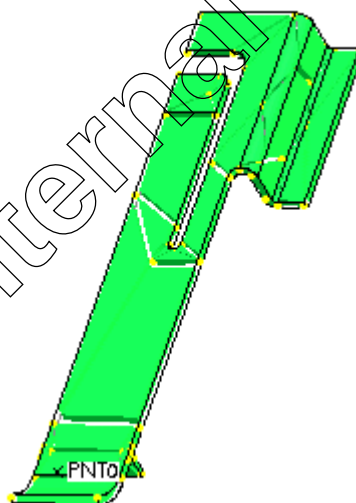


Figure 7: Shell elements

- Click **OK**.
3. Click **Close** and save the mesh.

Step 7. Define the constraints.

- Apply a symmetry constraint to the model. Select the 5 edges. Constraint the X translation, Y and Z rotations on the plane of symmetry. Free the other DOFs.
- Create a second constraint. The clip is glued to the appliance at the flange location. Fix all DOFs of the top or bottom surface of the flange.

1. Apply a symmetry constraint to the model.

- Click **Model > Constraints > New > Edge/Curve**.
- Select the 5 edges shown in the following figure.
- Constraint the translation in X-direction, and rotation in Y and Z-directions on the edges. Free the other DOFs.

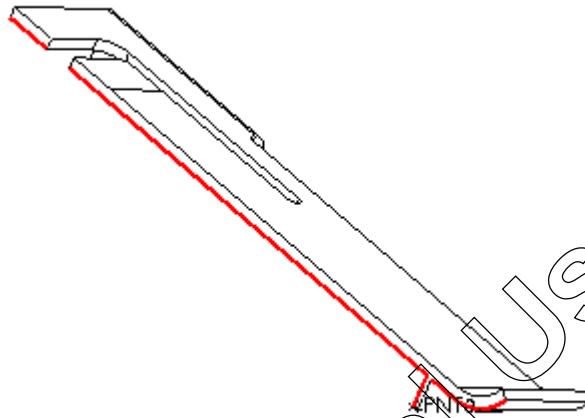


Figure 8: Symmetry constraint

2. Create a second constraint. The clip is glued to the appliance at the flange location.

- Click **Model > Constraints > New > Surface**.
- Fix all DOFs of the top surface of the flange, shown in the following figure.

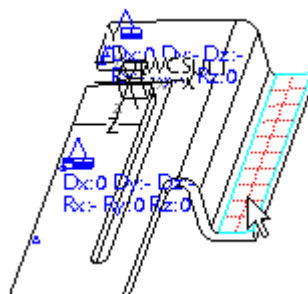


Figure 9: Flange constraint

Step 8. Define a surface load to simulate a finger pushing on the tab.

- Apply a **1.25 N** surface load (half of the total load, due to the symmetry), in the negative Y-direction on the created region

1. Define a surface load.

- Click **Model > Loads > New > Surface**.
- Apply a **1.25 N** surface load (half of the total load, due to the symmetry), in the negative Y-direction on the created region. After the load is applied, the model should appear as shown in the following figure.

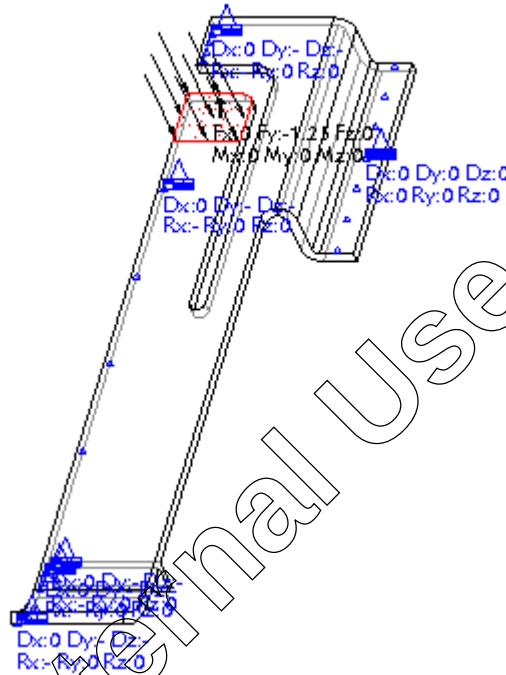


Figure 10: Apply a surface load.

Step 9. Define the measures.

- Create a displacement measure of the point PNT0 created earlier. Name the measure **Y_disp_clip_tip**.

1. Create a displacement measure using PNT0.
2. Name the measure **Y_disp_clip_tip**.

Note:

The tip of the catch must clear a 7 mm latch. You will want to track the displacement at the bottom of the catch with a measure.

Step 10. Create and run a static analysis.

- Create a static analysis named **clip_static** and run it.

1. Create a new static analysis.

- In the MEC STRUCT menu click **Analyses/Studies**.
- Click **File > New Static**.
- Name it **clip_static**.
- Complete the analysis creation.

2. Run the static analysis.

Step 11. Checking the results.

- Examine the summary file and the key statistics regarding the model, such as the number of elements, the elapsed time, and required disk resources.
1. Examine the summary file and the key statistics regarding the model, such as the number of elements, the elapsed time, and required disk resources.
 2. Look for the maximum von Mises and principal stresses, maximum displacement, resultant loads, and error estimates.
 3. Did the run complete without any errors? _____
 4. What is the value for Y_disp_clip_tip? _____ mm

Step 12. Create and display results windows.

- Create two animated result windows to display the displacement fringe plot and the von Mises Stress fringe plot.
1. Create an animated result window to display the displacement fringe plot.
 2. Create an animated result window to display the von Mises Stress fringe plot.
 3. Notice that the high stress area is near the inner tab fillet area.

Step 13. Create a Failure Criteria results window.

- Create a failure index result window.
1. Create a failure index result window.
 2. The results indicate that the failure index is below 1.0 for the entire model. This means there is no areas in the model where the stresses will surpass yield stress of the material.
 3. (Optional) Set the color legend to display the area that is above 1 in red. The model should have no red spots.
 4. Exit the result window interface.

Step 14. Create design parameters. Design Parameters change the shape of the model within a specified range during a sensitivity or optimization design study.

- Create a dimension design parameter. Select the R0.889 dimension. This dimension controls the width of the slot.
- Change the name to slot_radius and enter 0.8 for the minimum and 1.5 for the maximum.
- Repeat this process to create the design parameters shown in Table 1.

1. Create a design parameter.

- Click **Model > Dsgn Controls > Design Params.**
- Use **Dimension** type and click on the slot feature to show the dimension.
- The dimensions associated with the slot feature will then appear. Select the R0.889 dimension. This dimension controls the width of the slot.
- Change the name to slot_radius and enter **0.8** for the minimum and **1.5** for the maximum.

2. Repeat this process to create the design parameters shown in the following table.

Table 1: Design Parameters

Dimension	Minimum	Maximum
slot_length, d32 (orig=31.75)	19.05	44.45
slot_width, d50 (orig=7.62)	2.54	7.62
body_width, d51 (orig=20.32)	17.78	25.4

Step 15. Shape Animate the clip.

- Shape animate the model with all four of the design parameters selected, and set the number of intervals to 2.

Note:

Use Shape Animate to vary the parameters across different ranges and in different combinations, to anticipate problems that might arise during the optimization process, and to make sure the parameter ranges are causing the part to change as intended.

Also, use Shape Review to review the model at specific settings for each parameter and to identify conflicting design changes within dimension ranges.

1. Animate the shape of the model with change in the dimensions as per the parameters defined.

- Click **Model > Dsgn Controls > Shape Animate.**
- Select on all four of the design parameters, and set the number of intervals to 2.

2. The clip is regenerated at the initial and final configuration shown in the following figure.

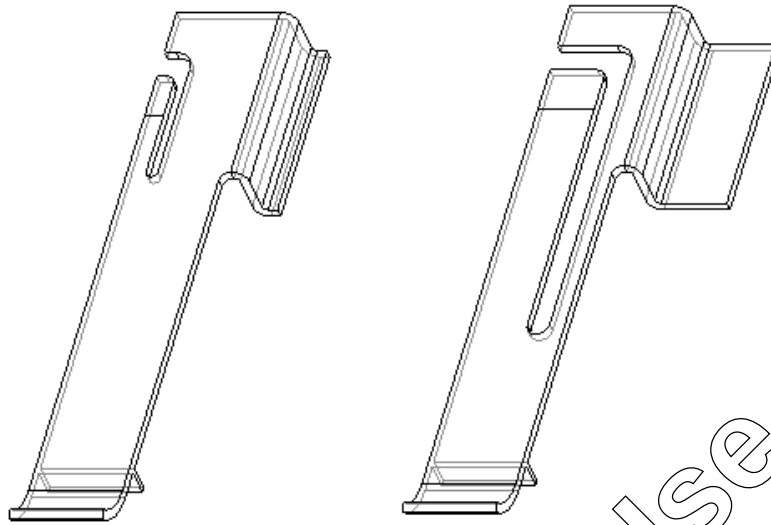


Figure 11: Shape Animation

3. Restore the model to its original shape.

Exercise 2: Belt Clip Sensitivity Studies

Introduction

In this exercise, you will setup two design studies, a global sensitivity study and a local sensitivity study.

Objectives

After successfully completing this exercise, you will know how to:

- Setup a local sensitivity study.
- Setup a global sensitivity study.

Scenario

After defining design parameters, local and global sensitivity studies are created and run. The effects of varying the design parameters on specific aspects of the model are explored.

Step 1. Create a local sensitivity study.

- Create a local sensitivity study named **clip_ls**.
- Verify that clip_static is selected and set all the parameters to vary and their values as given.

1. Create a local sensitivity study named **clip_ls**.

- In the MEC STRUCT menu click **Analyses/Studies**.
- Click **File > New Design Study**.
- Select **Local Sensitivity** as the type from the drop-down list.
- Verify that clip_static is selected.

2. Set all the parameters to vary and set their values as shown in the following figure.

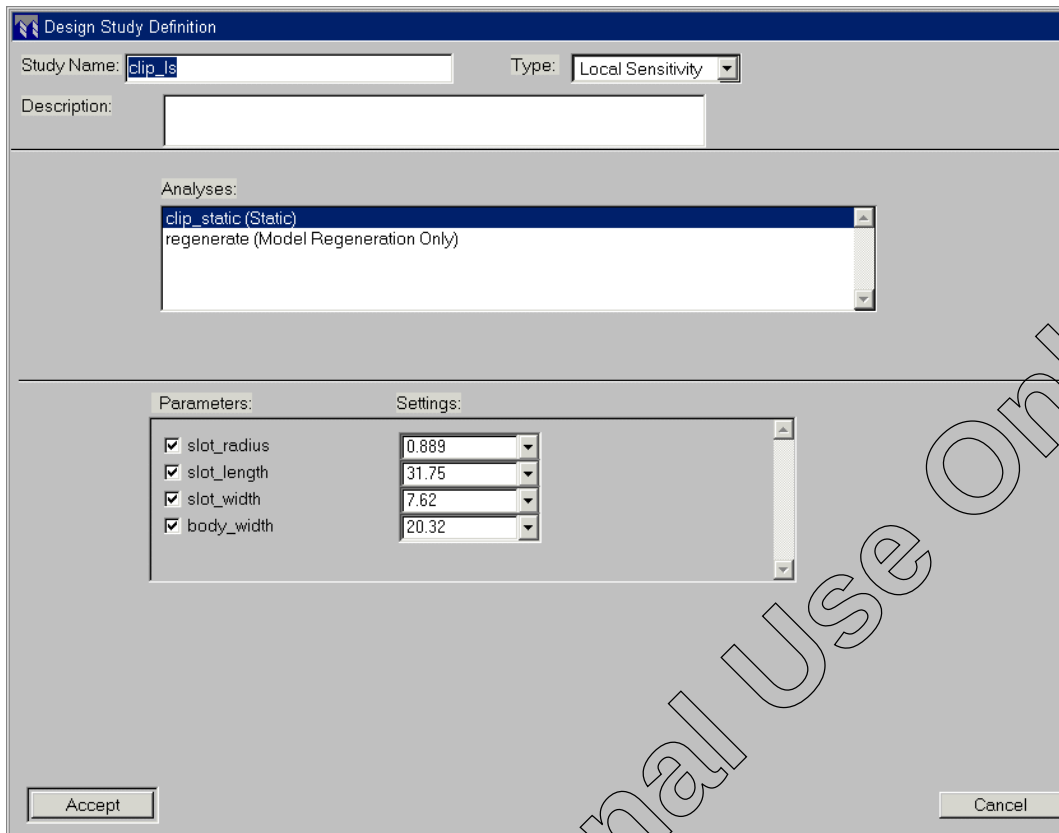


Figure 12: Local sensitivity design study dialog box

3. Click **Accept**.

Step 2. Run the local sensitivity design study.

- Before running the local sensitivity design study, check to reuse elements from an existing study by selecting **clip_static** as the analysis under Settings.
- Run the local sensitivity study. Monitor the process through the summary file.

1. Before running the local sensitivity design study, check to reuse elements from an existing study by selecting **clip_static** as the analysis under Settings.
2. Run the local sensitivity study.
3. Monitor the process through the summary file.
4. Pay attention to the run times, convergence behavior, and information dealing specifically with local sensitivity.

Step 3. Review local sensitivity results. Create four result windows to compare how much of an effect each of the four parameters is having on maximum von Mises Stress.

- Create a new result window. Name the window **slot_rad_stress**.
- Graph **max_stress_vm** measure vs. the design variable **slot_radius**.
- Create three more graphs named **slot_wide_stress**, **slot_len_stress** and **body_wide_stress**. Keep the **max_stress_vm** as the MEASURE.
- Select **slot_width**, **slot_length**, and **body_width** as the design variable respectively. Display all four windows.
- By default, the four windows will all have different Y-axis scales. To compare the four parameters, you need to have all four curves with the same Y-axis scale. Change the Y-axis minimum and maximum to 0 and 120 respectively on all the graphs.

1. Click **Results** and create a new result window.
2. Name the window **slot_rad_stress**.
3. Select the clip_Is results directory.
4. Graph **max_stress_vm** measure vs. the design variable **slot_radius**.
5. Accept and show the result window.
6. Copy the result window to create three additional windows as shown in the table below.

Table 2. Window Parameters

Window Name	Parameters
slot_wide_stress	slot_width
slot_len_stress	slot_length
body_wide_stress	body_width

7. Display all four windows.
8. By default, the four windows will all have different Y-axis scales. To compare the four parameters, you need to have all four curves with the same Y-axis scale. Change the Y-axis minimum and maximum to 0 and 120 respectively on all the graphs.

Note:

The von Mises stress should always be non-negative. The local sensitivity plot extrapolates the von Mises stress over a parameter range, using the sensitivity at a specific parameter value. Therefore, the graphs may indicate “negative” von Mises stress.

Step 4. Create four windows to see the effect of the parameters on the displacement measure Y_disp_clip_tip.

- Repeat the steps above to create four more results window to graph Y_disp_clip_tip vs design parameters.

- Copy the **body_wide_stress** window to **body_wide_y_disp**.
- Change the MEASURE from **max_stress_vm** to **Y_disp_clip_tip**.
- Blank all the von Mises stress windows.
- Copy the result window to create three additional windows as shown in the table below.

Table 3. Window Parameters

Window Name	Parameters
slot_wide_y_disp	slot_width
slot_len_y_disp	slot_length
slot_rad_y_disp	slot_radius

- Scale all the Y-axis to be from **-4** to **14**.

Step 5. Review the plots.

- Review the plots.

- Review the plots and determine the following information.

- Which parameter has the greatest effect on the displacement of the tip?

- Which parameter has the least, or perhaps no effect? _____

- The following observations can be made:

- The tip displacement is most sensitive to slot length and slot width.
- Slot radius appears to have less effect.
- Body width has no effect at all.

- The following conclusions are made:

- The slot length and slot width have a big impact on the maximum von Mises stress and clip tip displacement.
- Slot radius has less effect.
- Body width has no effect on the maximum von Mises stress and clip tip displacement.
- Since body width has no effect on any quantity of interest, it will be excluded from future studies.

Note:

The calculated slopes are strictly relevant to the single geometry state called for in the Local Sensitivity Study definition dialog box. The slopes will be different for different current values of your parameters.

Step 6. Define and run a global sensitivity design study to study the effect of slot_length over its range.

- Create a global sensitivity study named clip_gs.
- Verify that clip_static is selected and select the slot_length only to vary from min to max.
- Run the study and monitor the run.

1. Create a global sensitivity study.

- In the MEC STRUCT menu click **Analyses/Studies**.
- Click **File > New Design Study**.
- Select **Global Sensitivity** as the type from the drop-down list.
- Name it **clip_gs**.
- Verify that clip_static is selected.
- Select **slot_length** and unselect all other parameters.
- Change the NUMBER OF INTERVALS to **4** and verify that **Repeat P-Loop Convergence** is selected. The dialog box should appear as shown in the following figure.

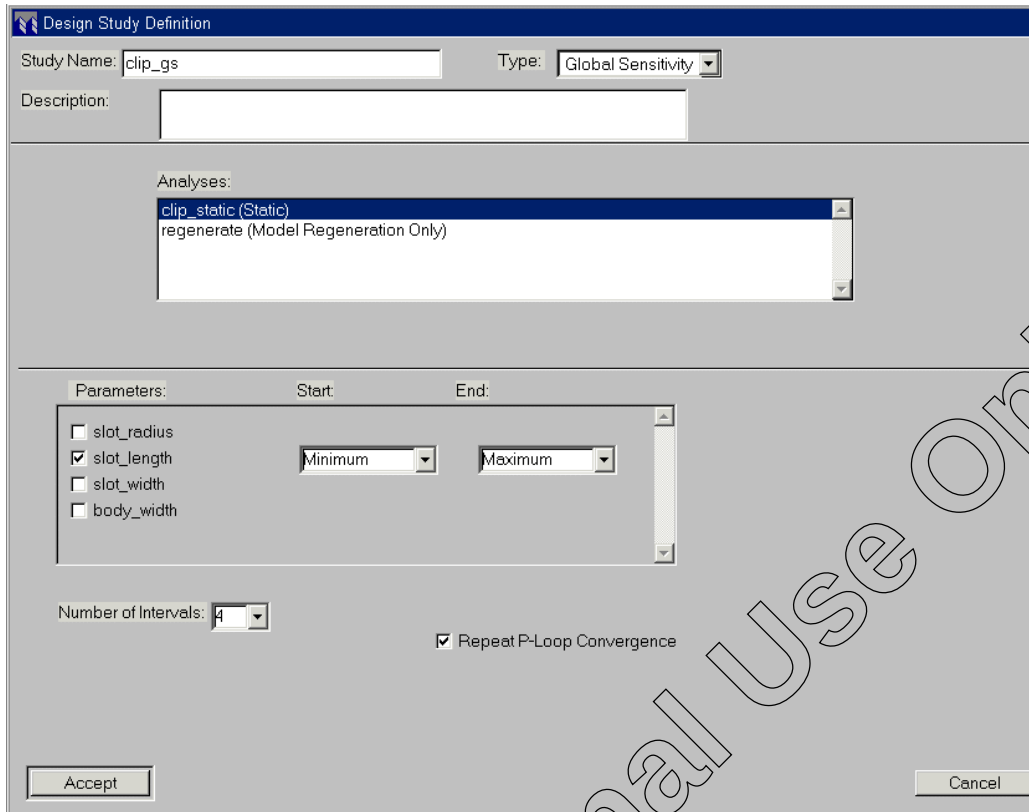


Figure 13: Global sensitivity design study dialog box

- Click **Accept**.

2. Run the global sensitivity study **clip_gs**. Monitor the process using the Summary file.

Step 7. Review the global sensitivity results. Look at sensitivity plots of mass, von Mises Stress, and tip displacement versus the slot length parameter.

- Create a results window, named **slot_len_mass**, referencing the **clip_gs** design study.
- Graph **total_mass** for the measure vs. **slot_length** for the design variable.
- Create and show three more windows by copying the **slot_len_mass** window. Use the same design variable **slot_length** and the names and measures as follow:

Name: **slot_len_vm**; MEASURE: **max_stress_vm**

Name: **slot_len_disp**; MEASURE: **Y_disp_clip_tip**

Name: **slot_len_maxd**; MEASURE: **max_disp_mag**

1. Create a results window named **slot_len_mass**, referencing the **clip_gs** design study.
2. Graph **total_mass** for the measure vs. **slot_length** for the design variable.
3. Copy the result window to create three additional windows as shown in the table below.

Table 4. Window Parameters

Window Name	Parameters
slot_len_vm	max_stress_vm
slot_len_disp	Y_disp_clip_tip
slot_len_maxd	max_disp_mag

Step 8. Review the plots.

- The following observations can be made from the result windows:
 - Increasing the slot length decreases mass.
 - Increasing the slot length increases maximum displacement and the clip tip deflection.
 - Increasing the slot length increases the maximum stress.
- The observations above will help us prepare the optimization, where our goal is to find the value for the slot_length that decreases stress, and mass, and results in a clip tip deflection between 7 mm and 9 mm.
- Exit the result window. Do not save.
- Save the model.
- Close the window and erase all models from session.

Summary

After successfully completing this module, you should know how to:

- Describe the purpose of design parameters.
- Create design parameters and measures.
- Set up and run a local and global sensitivity studies.
- Review sensitivity plots.

For PTC Internal Use Only

Running Optimization Studies

Introduction

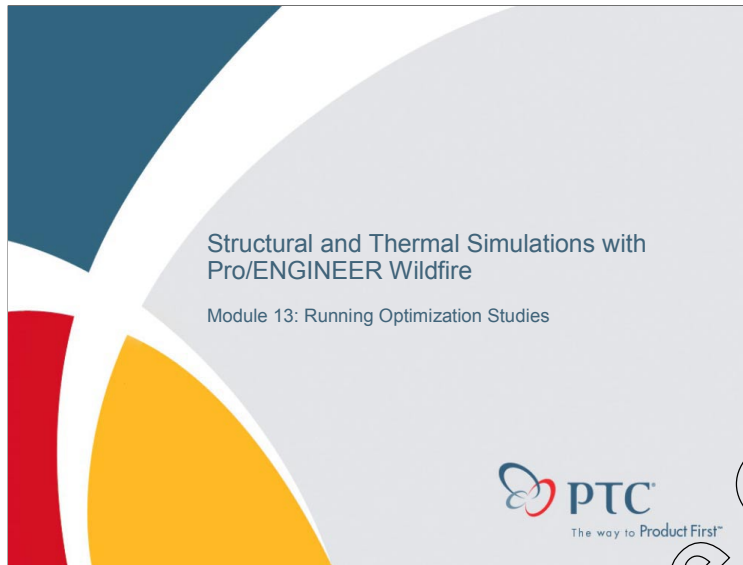
An optimization study adjusts one or more parameters to best achieve a specified goal or to test feasibility of a design, while respecting specified limits.

For the previous clip model, you will run an optimization study to find an optimized set of dimensions for your model. The goal is to reduce the stress in this model while increasing the maximum deflection to a value greater than 7 mm, but less than 9 mm. You will also attempt to reduce the weight of the current design.

Objectives

After completing this module, you will be able to:

- Setup and run optimization studies.
- View the optimized model.



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

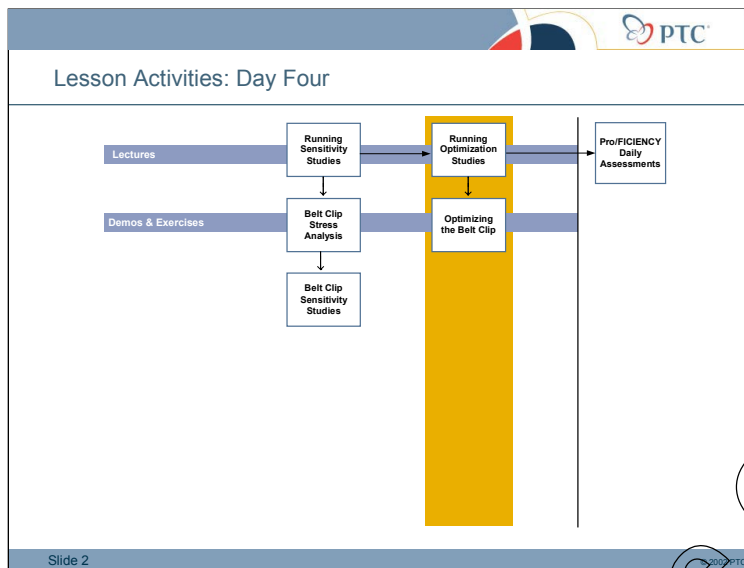
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

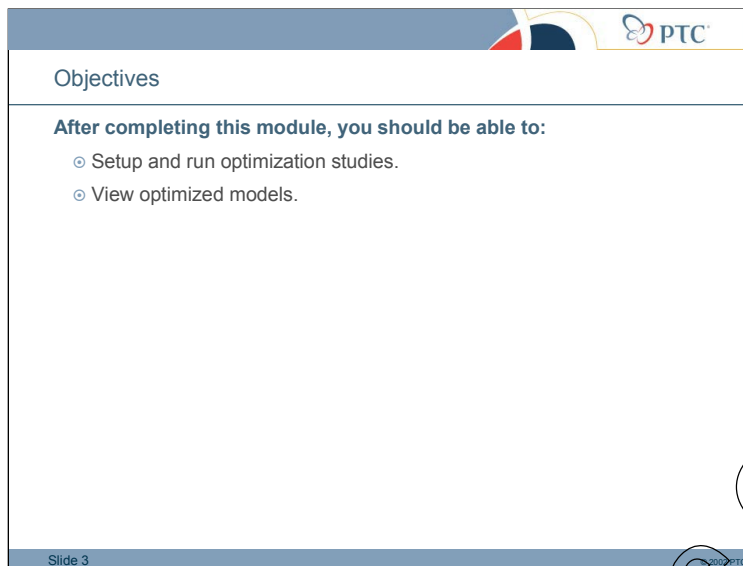
Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 30 min
- Demos: 45 min
- Labs: 120 min
- Total: 3 hours 15 min



Objectives

Running Optimizations

An optimization study adjusts one or more parameters to best achieve a specified goal or to test feasibility of a design, while respecting specified limits.

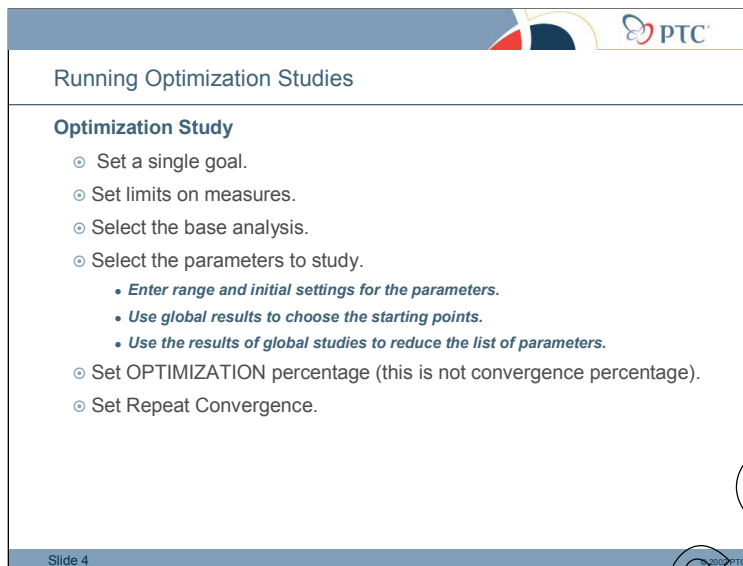
To create an optimization study, you define the following components:

- **Goal** – You select a measure to minimize or maximize as the study's goal.
- **Limits** – You define limits on one or more measures that Mechanica cannot violate during the optimization.
- **Parameters** – You select one or more design parameters you want Mechanica to adjust to achieve the goal. You will also define a range and initial value for each parameter.

The goal and limits are each optional, but you must have at least one goal or one limit.

Mechanica adjusts the model's parameters in a series of iterations through which it tries to move closer to the goal while satisfying any limits. If you have no goal, Mechanica simply tries to satisfy your limits. An optimization with no goal is sometimes called a feasibility study. If you do not define a goal, you must define limits. Without a goal, Mechanica searches for the first feasible design that satisfies the limits you define.

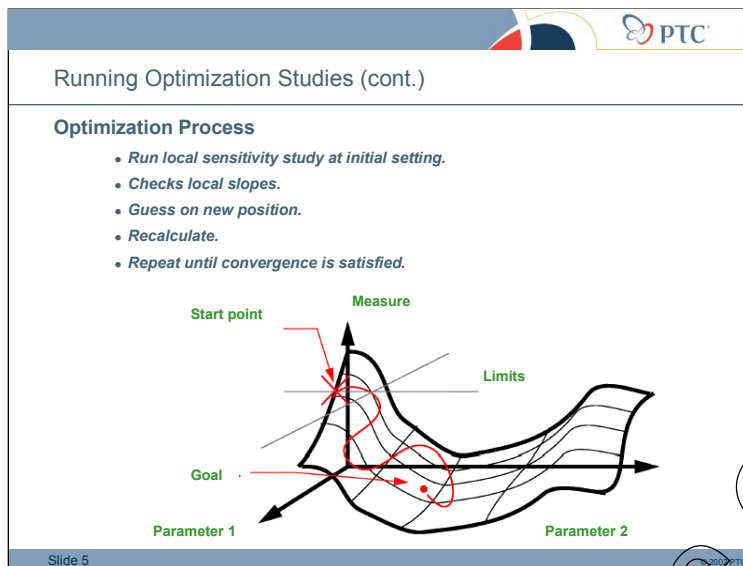
When defining a goal and limits, you can select measures associated with different analysis types. You can set up an optimization that would perform any of Mechanica's analysis types except motion and contact. For example, you could optimize the clip for stress, displacement, natural frequencies (modes), and temperature simultaneously.



Running Optimizations (cont.)

Additional components:

- **Base Analysis:** Select the analysis to be performed. Note that optimizations can be performed on several analyses of the same part (for example: modal, static and thermal).
- **Range and Initial setting:** This defines the size of the solution space Mechanica will be working with. For best results, we want to limit this solution space (for example: only use the necessary range for each parameter). The initial setting for each parameter is also important. Use results from global sensitivity studies to determine where to start your analysis. Note that initial conditions will greatly influence the runtime.
- **Optimization convergence:** This is the tolerance percentage used to determine if the goal/limits have been reached. This is not to be confused with the base analysis convergence percentage which will dictate when to terminate the p-pass algorithm.
- **Max Iterations:** If Mechanica can not converge to an optimum set of parameters, the optimizer algorithm will stop after 20 iterations. This usually points to a problem with the way the optimization study has been defined (Incompatible goal and limits, too many parameters, too large a solution space or poor initial values). Setting the "Max Iteration" value to a larger number will not solve these problems. Only when you consider that the optimizer is on the right track and wish to push it further, should that number be changed.
- **Element Update Options and Repeat Convergence.** See discussion on "Global Sensitivity Analysis".



Optimization Process

- This is a schematic representation of how an Optimization study works. The analogy used is that of a blind bug or blind-folded skier on top of a mountain.
- Consider the surface generated by the variation of a particular measure (say stress) as a function of 2 parameters (say hole diameter and thickness). This is the solution space. Note that limits have precedence over the optimization goal (in the example at hand, there is not much use for a very light but flimsy part).
- The goal is to get to the bottom of the hill. The problem is that we do not know what that surface looks like. We will therefore proceed with the following method:
 1. Test the local slopes to see which way is down. We will perform a local sensitivity study for each parameter and then decide in which direction we want to increment these parameters.
 2. Once limits are satisfied, we can guess a new position based on the information obtained from sensitivity studies.
 3. We now perform the base analysis at these parameter settings (Standard Study) and extract all necessary values of measures used to evaluate if our optimization goal or limits have been reached. If neither our goal or the maximum number of iterations has been reached, then iterate to step (1) above.



Demonstrations

- ◉ Setup and run an optimization study for a belt clip.
- ◉ View the optimized belt clip.
- ◉ Define and run an optimization study for the CPU assembly.

Slide 6

Demonstrations

In this demonstration, I will:

- Create an optimization study and run it.
- View the results and check the optimized model.

You will repeat the same steps in the lab exercises.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 7

Precision Learning

- Learn: by listening to lectures, watching demos, and completing lab exercises.
- Assess: your progress with Pro/FICIENCY assessment questions.
- Improve: The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Note to Instructors:

Getting Started

- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Exams

For each course, Pro/FICIENCY exams are based upon the topics covered each day.

How to use it

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After completing this module, you should be able to:

- ◉ Setup and run optimization studies.
- ◉ View optimized models.

Slide 8

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 13 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Optimizing the Belt Clip

Objectives

After successfully completing this exercise, you will know how to:

- Setup and run optimization studies
- View the optimized model.

Scenario

Continuing with the previous scenarios, we run an optimization study to find a set of dimensions that optimize the model. The goal is to reduce the stress in this model while increasing the maximum deflection to a value greater than 7 mm, but less than 9 mm. We will also attempt to reduce the weight of the current design.

Step 1. Define an optimization.

- Create an optimization design study named **clip_opt**. Minimize max_stress_vm and set the limits: Y_disp_clip_tip > 7, Y_disp_clip_tip < 9, and total_mass < 2.2e-6.
- The parameters that will be used in the optimization are: slot_radius, slot_length and slot_width. Set the parameter initial values to the middle one, except for the slot_width set it to 7.62.
- Set the convergence to **2%** with a max number of iterations of **20**. Verify that **Repeat P-Loop Convergence** is selected.

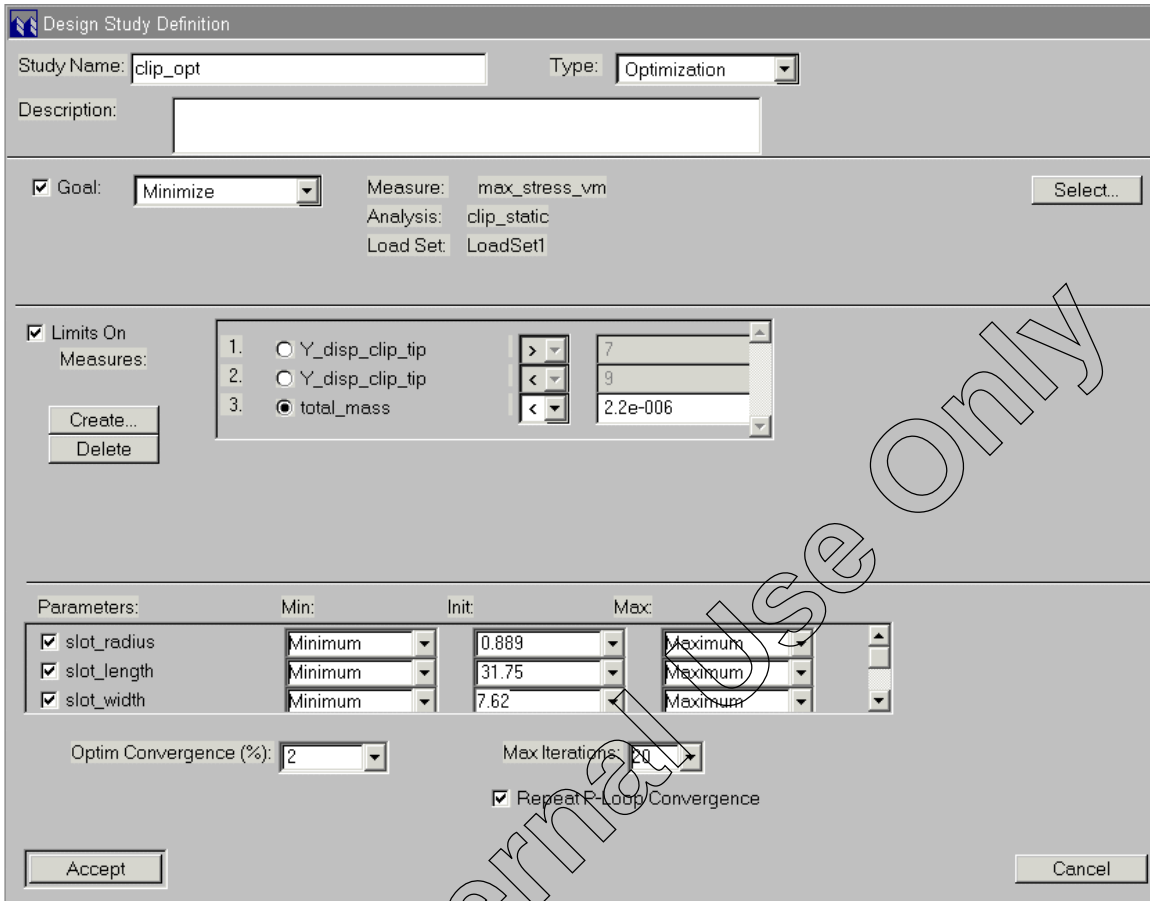
1. Set the working directory to C:\users\student\mechanica_330\module_12_13.

2. Open CLIP.PRT.
3. Enable Mechanica Structure.
4. Click **Analyses/Studies > File > New Design Study** and create an optimization design study.
 - Select **Optimization** as the type from the drop-down list.
 - Name the optimization design study **clip_opt**.
 - As a goal minimize **max_stress_vm**.
 - As for the limits, set: $Y\_disp\_clip\_tip > 7$, $Y\_disp\_clip\_tip < 9$, and $total\_mass < 2.2e-6$.

Note:

When the unit system is mm N sec, the mass unit is metric tonnes.

- The parameters that will be used in the optimization are: slot_radius, slot_length and slot_width.
- Unselect the check box body_width, since the parameter body_width does not impact stress or displacement as seen in the local sensitivity analysis. You may have to scroll through the parameter list.
- Using the Init list, set the parameter initial values to the middle value for all parameters, except the slot_width. Set the slot_width to **7.62**.
- Set the Optim Convergence to **2%** and the Max Iterations to **20**. Verify that **Repeat P-Loop Convergence** is selected, as shown in the following figure.



Design Study Definition

Study Name: Type:

Description:

☒ Goal: Measure:

Analysis:

Load Set:

☒ Limits On Measures:

1.	☐ Y_disp_clip_tip	7
2.	☐ Y_disp_clip_tip	9
3.	☒ total_mass	2.2e-006

Parameters:	Min:	Init:	Max:
☒ slot_radius	Minimum	0.889	Maximum
☒ slot_length	Minimum	31.75	Maximum
☒ slot_width	Minimum	7.62	Maximum

Optim Convergence (%): Max Iterations:

☒ Repeat P-Loop Convergence

Figure 1: Optimization study definition dialog box

Step 2. Run the optimization.

Note to the instructors:

Instead of running the analysis you can show the report file from the already run analysis. The analysis results are saved in C:\users\student\mechanica_330\module_12_13\results\clip_opt.

Note:

The analysis will take a considerable amount of time to complete. The time taken to run the analysis is dependent on the complexity of the model and the hardware resources available.

Do not run the analysis you created. The results from the clip_opt analysis are available for review.

1. Open the report files located in
C:\users\student\mechanica_330\module_12_13\results\clip_opt with notepad and examine the run.

Note:

You may run the analysis in your spare time. Ensure that the RAM allocation is set to half of the system RAM, in the analysis settings. You can monitor the run by checking the status, and view the element creation progress.

Step 3. Create four result windows, showing the optimization history of displacement, tip displacement, von Mises stress, and mass.

- Use the results provided to create result windows. Create the following result windows to display the measure variation vs. Iteration during the optimization.

NAME **vm_opt_pass**; MEASURE: **max_stress_vm**

NAME **disp_opt_pass**; MEASURE: **Y_disp_clip_tip**

NAME **mass_opt_pass**; MEASURE: **total_mass**

NAME **maxd_opt_pass**; MEASURE: **max_disp_mag**

- Examine the windows.

- Create the following result windows using the table below to display the measure variation vs. Iteration during the optimization.

Table 1. Window Parameters

Window Name	Measure
vm_opt_pass	max_stress_vm
disp_opt_pass	Y_disp_clip_tip
mass_opt_pass	total_mass
maxd_opt_pass	max_disp_mag

- Examine the windows:
 - The results show the measure variation during the optimization. Notice the maximum von Mises stress changes.
 - The optimized model satisfies all constraints.
 - As Mechanica approaches the optimal shape, the values for stress, displacement, and mass stop fluctuating; and, as soon as all of the values are within 2% of the goal and limits, the optimization is complete.

Step 4. Create result windows for von Mises and displacement stress fringe animations.

- Create von Mises stress fringes without animation. Display the maximum von Mises stress on the model.
- Create an animated displacement result window. Review the results window carefully.

- Create von Mises stress fringes without animation.

2. Display the maximum von Mises stress on the model.
3. Create an animated displacement result window.
4. Exit the result window. Do not save.

Step 5. Update the current part design dimensions to those of the optimized design.

- Review the optimization history for clip_opt.
 - Leave the model at the optimized shape. Compare the original design to the new one.
 - Switch to the standard application. Save and erase the model.
1. Review the optimization history for clip_opt using Design Controls and Optimization History.
 2. Mechanica begins an animation of the shape changes attempted during the optimization.
 3. Leave the model at the optimized shape. Compare the original design to the new one. In the original design, the stresses were higher, and it was heavier. The new design has sufficient displacement, less stress, and is lighter.
 4. Save the model.
 5. Close the window and erase all models from session.

Summary

After completing this module, you will be able to:

- Setup and run optimization studies.
- View the optimized model.

For PTC Internal Use Only

Applying Mechanics Analysis Concepts

Introduction

This module will provide design studies in varying degrees of difficulty. You can choose a design study that reflects your area of interest. Design studies cover the following topics:

Contact Functionality

You can use contact analysis to solve design problems where parts come into contact, but the exact regions that touch are unknown. Examples of these parts are rollers pressing against each other, press fits, rotating gears or shaft assemblies. Contact analysis is applicable whenever the model stiffness or load path changes as a function of the applied load.

Before running a contact analysis, you first must define contact regions. Contact regions indicate which curves or surfaces may touch during a contact analysis. Mechanica ignores contact regions for analysis types other than contact. Additionally, contact regions are frictionless and will only transfer normal forces.

When running a contact analysis, Mechanica calculates the detailed stress gradients and deformations found in contact regions that in some cases may cause the part failure. Determining stresses and deformations in a contact analysis is a non-linear problem requiring an iterative solution scheme. The non-linearity is due to the contact area varying non-linearly with the applied load.

Mechanica uses advanced P-element technology and geometric elements with a penalty solution method for achieving a converged contact solution. This procedure is fully adaptive and automated and requires no artificial constraints such as the gap elements used in conventional finite element codes.

Running Contact Analyses

Contact analysis in Mechanica is considered frictionless; the interface in Contact is assumed to be perfectly lubricated.

When using contact analysis on your own models, you should be aware of the following limitations:

- You cannot use enforced displacements on contact regions.
- You can only run standard or global sensitivity design studies and optimization with a contact analysis; local sensitivity design studies cannot be run.
- You cannot combine results from two load sets.
- The contact solution is computed for solid elements only. Mechanica will not create contact regions on surfaces with shell or beam elements.

Mechanica Structure handles moderate displacements and rotations. These displacements and rotations are small enough so that the Small Displacement Theory is adequate for element calculations, but large enough that significant rotation of element normal or significant relative tangential motion of the two surfaces of contact occurs. For 3-D models, the allowable inter-penetration tolerance is one-half the minimum “length” of the two surfaces defining a contact region, where “length” represents the square root of the surface area.

Defining Contact Regions

You can define the contact regions manually or automatically. However, if you use the automatic option, and the assembly is complex (many possible contact surfaces), some contact regions that are not needed may be defined. To prevent longer run times, you should delete the excess contact icons. For a complex model, this can be cumbersome, thus, use this option with care.

Defining Contact Analysis Measures

There are two default global measures that are calculated for every contact analysis:

- **contact_area:** Sums the total area of all the contact regions in the model.
- **contact_max_pres:** Tracks the maximum pressure in any of the defined contact regions.

You can also set up user-defined measures to track five attributes of a single contact region. These are very helpful when your assembly contains multiple contact regions:

- Contact area
- Maximum contact pressure
- Average contact pressure
- Contact load
- Force

Setting Contact Analysis Options

The following options are available when defining contact analyses.

Specifying Load Increments

For contact analyses, you can enter a value for the Number of Load Increments, which specifies the steps at which Mechanica calculates results. This enables you to see how measures vary with the load. This is similar to performing a global sensitivity with a design parameter on the load (if that were possible in Structure).

You will use the default of 1 for number of load increments. This option tells Mechanica how to apply the loads or enforced displacements to the model incrementally. You can set the number of load increments to more than 1 if you want to view any contact measures as a function of the applied load. For example, use a value larger than 1 if you need a plot of contact area vs. applied load.

In conventional FEA (h-codes), the contact solution accuracy is dependent on the load increment. Mechanica, on the other hand, gives you solution accuracy regardless of what load increment you specify.

Refining the Localized Mesh

Localized Mesh Refinement is available for achieving accurate contact pressures in the model.

If Localized Mesh Refinement (often called h/p adaptive refinement) is selected, the mesh will automatically be refined near the contact region to improve the pressure accuracy. However, Localized Mesh Refinement does increase analysis run time.

Contact analyses are run for a variety of reasons. One reason to define a contact analysis is to calculate stresses at an interface of two parts that are contacting each other. Another reason to run a contact analysis is to accurately simulate the load transfer between parts, where the maximum stress in the model is not at the contact zone. For this type of analysis, localized mesh refinement is not needed, and should be turned off to save run time.

Transient Thermal Analysis

For engineering processes that involve heating and cooling, the transitional period of time is of great interest. The analysis must be modified to take into account the change in internal energy of the body with time. In Mechanica, this type of data is obtained through a Transient Thermal analysis.

Fundamentals

When evaluating a design, you may need to investigate what will take place between material bodies as a result of a temperature difference. This energy

transfer between two bodies (commonly referred to as heat) may be considered in steady state or in a transient state.

Thermodynamics deals with systems in equilibrium - it may not be used to predict how fast a change will take place. Systems in equilibrium are considered to be in steady state.

Heat transfer is used to predict the rate at which this exchange will take place under specified conditions. Bodies not in equilibrium are considered to be in a transient phase.

Transient Heat Transfer

When bodies are suddenly subjected to a change in environment, some time may elapse before an equilibrium temperature condition will exist in the body. Until then, the body is in an unsteady state.

Many engineering problems are concerned only with the steady-state heat transfer of a part or assembly. This type of information can be obtained by running a Steady-State Thermal analysis in Mechanical.

However, for engineering processes that involve heating and cooling, this transitional period of time is of great interest as well. The analysis must be modified to take into account the change in internal energy of the body with time. In Mechanical, this type of data is obtained through a Transient Thermal analysis.

Whenever there exists a temperature difference in a medium or between media, heat transfer must occur. This exercise deals with two modes of heat transfer: conduction and convection.

Thermal Conduction

Conduction is the heat transfer mode that occurs when a temperature gradient exists in a stationary solid or fluid. Conduction is the transfer of heat through materials without net mass motion of the material.

The general equation for heat transfer by conduction is:

$$q = -kA \frac{dT}{dx} \quad \text{where:}$$

q is the rate of heat flow in x -direction

k is the thermal conductivity

A is the area normal to x -direction through which heat flows

$\frac{dT}{dx}$ is the temperature gradient in the x -direction

dT is the temperature change in the x -direction and dx is increment in length in x -direction

Thermal Convection

Convection is the heat transfer mode that occurs between a surface and moving fluid when they are at different temperatures. It is the process in which thermal energy is transferred between a solid and a fluid flowing past it. The general equation for heat transfer by convection is:

$$q = hA \Delta T$$

where:

q is the rate of heat flow via convection

h is the heat-transfer coefficient (also known as the film coefficient)

A is the surface area through which heat flows (convection area)

$\Delta T = T_w - T_\infty$, (temperature potential for heat flow away from surface)

T_w is the wall (surface) temperature and T_∞ is the bulk temperature (average fluid temperature far away from wall)

Defining the Measures

The Transient Thermal analysis has three default measures that are evaluated over the model at each time step. They are:

- min_dyn_temperature
- max_dyn_temperature
- max_flux_mag

Any information that you want graphed against time, other than these measures, has to be specified as a user-defined measure before an analysis is run.

The options for Quantity are:

- Temperature
- Heat Flux
- Temperature Gradient
- Driven Pro Parameter
- Time

The options for Spatial Variation are:

- At Point
- Max Over Model
- Range Over Model
- Min Over Model
- Max Abs Over Model

The options for Time Evaluation are:

- At Each Time Step
- Maximum
- Minimum
- Maximum Absolute
- At Time

Dynamic Time Analysis

A special kind of dynamic loading better known as impulsive can be analyzed in the time analysis. Impulsive loads are generally relatively short on duration, therefore, they are assumed to have an infinite period. These types of loads are frequently important in the design of certain classes of structural systems (for example, vehicles such as trucks or automobiles). In the case of impulsive loading, damping has much less importance in controlling the maximum response of a structure. This phenomenon occurs because the maximum response is reached in a very short time before the damping force can absorb much energy from the structure.

Dynamic Time Analysis can be used to obtain the steady state as well as the transient response of the system. For most case, the maximum response of the system is the most important factor. The maximum response of the system will occur during the time the load is applied.

Dynamic Frequency Analysis

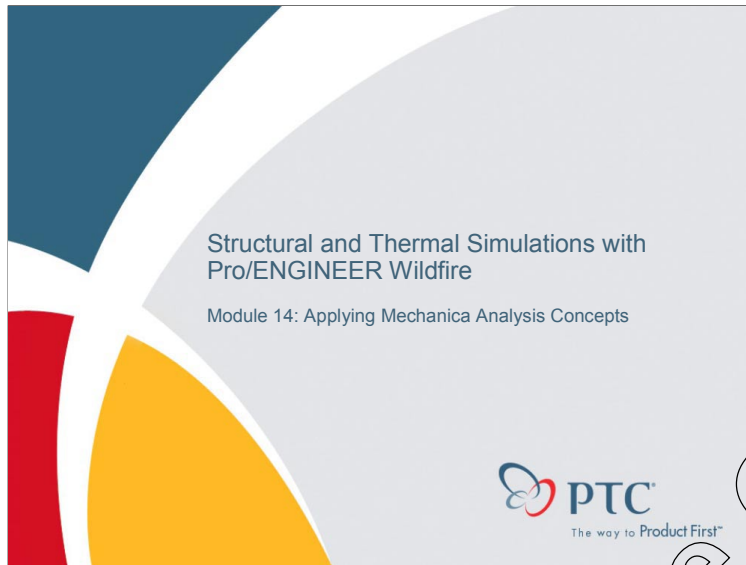
There are some types of load condition, which make an analysis more convenient to perform in the frequency domain. The frequency domain is superior to the time domain approach when the forcing load is periodic. A periodic loading exhibits the same time variation successively for a large number of cycles. The simple periodic loading has a sinusoidal variation also known as simple harmonic. Simple harmonic loading is characteristic of unbalanced-mass effects in rotating machinery. Other example of periodic loading are those caused by hydrodynamic pressure generated by a propeller at the stern of a ship or by inertia effects in reciprocating machinery, which are frequently more complex. However, by means of a Fourier analysis any periodic loading can be represented as the sum of a series of simple harmonic components.

Objectives

After completing this module, you will be able to:

- Create spot-welding connections.
- Create setup contact connections surfaces and setup contact analysis.
- Create time dependent boundary conditions and setup a transient thermal analysis.
- Create a large deformation analysis.
- Create a dynamic time analysis.
- Create a dynamic frequency analysis.

For PTC Internal Use Only



Instructor Preparation

(minimum skills listed; additional experience recommended)

Before teaching *Mechanica*, you **must** have read and **thoroughly understand** the following materials:

General Information

- Review Pro/ENGINEER Wildfire primer site at <http://rdweb.ptc.com/primer/>

Wildfire Documentation

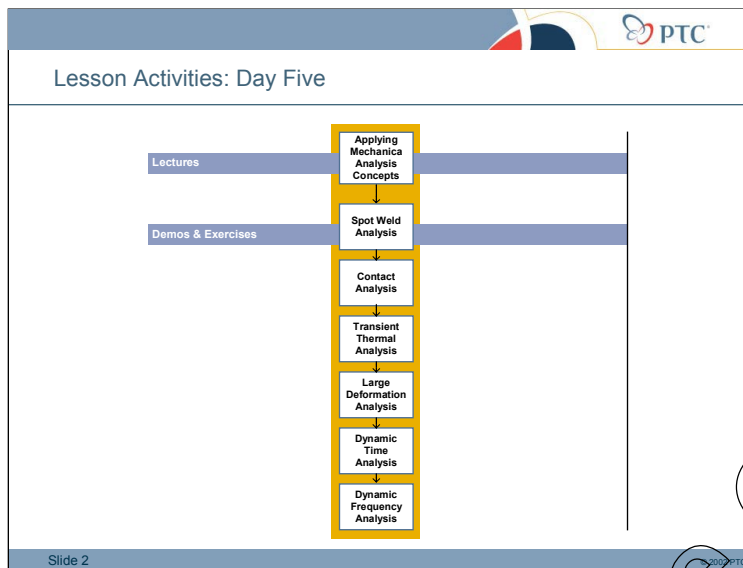
- Review Pro/ENGINEER Wildfire documentation in Windmill, located in */GS Training Materials/Domain Knowledge/Create/ProENGINEER/Core Concepts*

Pro/ENGINEER Wildfire

- Review Pro/ENGINEER Wildfire training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T994-330-Instructor_Kit-EN*

Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE.

- Review Structural and Thermal Simulation with Pro/ENGINEER WILDFIRE training material in Windmill, located in */GS Ed Svcs Operations/GS Education Library/Instructor Materials/Instructor Kits/EN/T981-330-Instructor_Kit-EN*



Duration

- Lecture: 30 min
- Demos: 60 min
- Labs: 120 min
- Total: 3 hours 30 min



Objectives

After completing this module, you should be able to:

- Create spot-welding connections.
- Create setup contact connections surfaces and setup contact analysis.
- Create a time dependent loads and setup a transient thermal analysis.
- Create a large deformation analysis.
- Create a dynamic time analysis.
- Create a dynamic frequency analysis.

Slide 3

Objectives

This module will provide design studies in varying degrees of difficulty. You can choose a design study that reflects your area of interest. Design studies cover the following topics:

- Static.
- Thermal.
- Contact.
- Large deformations.
- Vibrations.



Demonstrations

Examples

- ⦿ Apply Mechanics analysis concepts to various Structure and Thermal designs.
 - **Spot Weld Analysis**
 - Creating spot welds.
 - **Contact Analysis**
 - Defining contact regions and measures.
 - Defining a static analysis with contact regions.
 - Specifying load increments.
 - **Transient Thermal Analysis**
 - Defining time dependent boundary condition.
 - **Large Deformation Analysis**
 - **Dynamic Time Analysis**
 - Defining measures.
 - **Dynamic Frequency Analysis**

Slide 4

Demonstrations

In this demonstration, I will:

- Various Structure and Thermal Analyses.

You will have the opportunity to work on several real life models of your choosing.

PTC

Daily Skill Checks

Evaluate your progress:

- Achieve the course objectives.
- Use Pro/FICIENCY assessment questions.
- Apply Precision Learning.



Slide 5

Precision Learning

- Learn: by listening to lectures, watching demos, and completing lab exercises.
- Assess: your progress with Pro/FICIENCY assessment questions.
- Improve: The next day the instructor reviews the exam results of the group and reviews those topics that received the fewest correct answers.

Note to Instructors:

Getting Started

- Before lunch on the first day of class, set up the customer accounts.
- When the customers are returning from lunch, refer them to the new Appendix.
- Have them take the sample exam.
- Review the results of the group and use as an icebreaker.

Daily Exams

For each course, Pro/FICIENCY exams are based upon the topics covered each day.

How to use it

- Use the customer accounts already setup for the sample exam.
- At the end of each day the customers take the 10 question exam relating to that days' topics.
- The next morning, review the results of the group.
- Review those topics with the class that obtained the most incorrect answers.



Summary

After completing this module, you should be able to:

- Create spot-welding connections.
- Create setup contact connections surfaces and setup contact analysis.
- Create a time dependent loads and setup a transient thermal analysis.
- Create a large deformation analysis.
- Create a dynamic time analysis.
- Create a dynamic frequency analysis.

Slide 6

Summary

There are no instructor notes on this slide.

For PTC Internal Use Only

Module 14 Lab Exercises

Demonstration Instructions

Preparation

Complete the following tasks before running this demo for customers:

- Practice running the demo so you can easily complete it.
- Check for and review the errata sheet for this course.
- Use Pro/ENGINEER Wildfire build code 2003250 or later.
- Use Mechanica build code 2003250 or later.
- Download and install the class files promech_wildfire_330.tar.gz as described in the classroom setup notes.

Exercise 1: Spot Weld Analysis

Introduction

In this exercise, we will create spot weld connections on assembly, and run a static analysis to explore the transfer of stress through these connections.

Objectives

After successfully completing this exercise, you will know how to:

- Create spot-welding connections.

Scenario

In this exercise, you will create spot welds. A spot weld consists of a beam that is “fused” to the surface over an area defined by the diameter of the weld. You can use spot welds to model welded structures and bolted connections. When you use spots welds, you specify both the diameter of the weld and the material. The length of the weld is determined by the location of the fusion points on each surface. Spot welds can be used in both part mode and assembly mode, but patterns and zero-length spot welds are not supported.

Imagine the following scenario. You are the lead engineer for a company that makes motor-controlled robots for picking up heavy objects. Due to marketing demands, engineering has decided to use a heavier, more powerful motor. The current motor rests in a u-channel that is spot welded to an electronics barrier shelf. This shelf then slides into the framework of a larger assembly. The team you are working with is worried that the added weight of the new motor will cause excessive deflection in the u-channel (the new motor weighs 1000 N). A deflection of more than 2.54 mm would cause the motor to come in contact with the lower level of the assembly. For this reason, they want to move to a continuous seam weld instead of spot welds. From a labor

point of view, that option is very expensive and your boss wants you to make sure it is absolutely necessary. The motor has a 152 mm x 406.4 mm base attached to the bottom of the U-Channel. The four spot welds are placed at 152.4 mm intervals between them. Because the parts are thin, you will define the assembly as a shell model. The spot welds are 25.4 mm diameter and 6.35 mm long.

Step 1. Open SPOT_WELD.ASM model and test the shell compression.

- Open the SPOT_WELD.ASM model from the correct directory. Note the inits.
 - Enter Mechanics Structure and test shell compression. Notice that the mid-surfaces have not been defined.
1. Set the working directory to C:\users\student\mechanica_330\module_14.
 2. Open SPOT_WELD.ASM. The units are mmNs.
 3. Notice the there is a 6.35 mm gap between the parts for the spot welds.

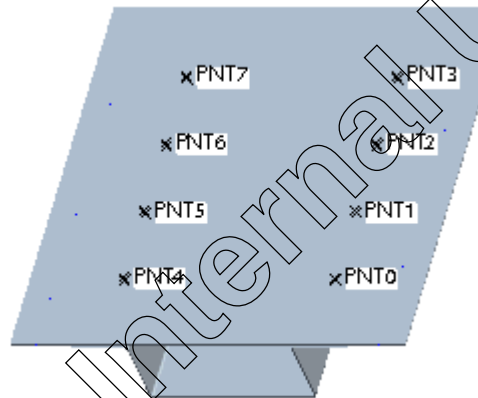


Figure 1: SPOT_WELD.ASM

4. Enable Mechanics Structure.
5. Click **Model > Idealizations > Shells > Midsurfaces > Compress > Shells Only** and test the shell compression. Notice that the mid-surfaces have not been defined.

Step 2. Open channel and shelf models and define shell mid-surfaces.

- Open the part U_CHANNEL.PRT and enter Mechanics Structure.
 - Define shell mid-surface using auto-detect and test compression.
 - Repeat the same procedure to compress the Shelf.prt.
1. Open U_CHANNEL.PRT.
 2. Enable Mechanics Structure.
 3. Define shell mid-surface using auto-detect and test compression.

- Click **Model > Idealizations > Shells > Midsurfaces > Auto Detect > Compress > Shells Only > Show Compress**.
- Click **Done > Done/Return**.

4. Save the model.

5. Close the window.

6. Repeat the same procedure to compress SHELF.PRT.

Step 3. Idealize the assembly as a shell model and assign material properties.

- Switch to the assembly window and compress the surface pairs.
- Assign STEEL as the material for both parts.

1. Activate the model SPOT_WELD.ASM.

2. Compress the surface pairs.

- Click **Model > Idealizations > Shells > Midsurfaces > Compress > Shells Only**.
- Click **Done > Done/Return > Done/Return**.

3. Assign the material **STEEL** to U_CHANNEL.PRT and SHELF.PRT.

Step 4. Create the spot welds.

- Create spot weld connections. For the first surface, select the bottom surface of the shelf to be welded and for the second surface, select the top right flange surfaces of the U channel.
- For the points, select the 4 datum points on the right side of the shelf.
- Enter 24.5 for the diameter and assign STEEL as the material for the spot welds.
- Repeat the steps above to create spot welds on the left-hand side.

1. Create spot weld connections.

- Click **Model > Connections > Spot Welds > Create**.
- For the first surface, select the bottom surface of SHELF.PRT to be welded.
- For the second surface, select the top right flange surfaces of U_CHANNEL.PRT.
- Select the datum point sets PNT0, PNT1, PNT2 and PNT3 on the right side of SHELF.PRT, using a region selection. Click **OK**.

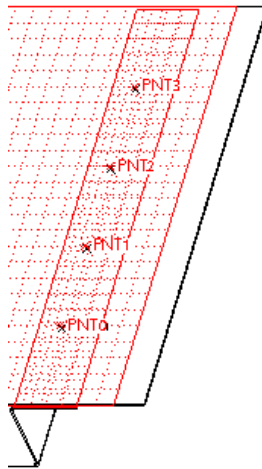


Figure 2: Spot weld references

- Enter **24.5** for the diameter and assign STEEL as the material for the spot welds.

2. Repeat the steps above to create spot welds on the left-hand side.

Step 5. Define the boundary conditions. The surface regions for the boundary conditions have already been defined for your convenience.

- Create a surface constraint to represent the slot that clamps the edges of the shelf. Name the constraint Slot and select the two surface regions that represent the clamp area on the shelf part, as shown below. Fix all 6 DOFs.
- Create a surface load to simulate the motor. Name the load motor on the rectangular surface region on the bottom of the U channel. Enter -1000 as the Force component in the Y direction.

1. Create a surface constraint to represent the slot that clamps the edges of the shelf.

- Click **Model > Constraints > New > Surface**.
- Name the constraint **Slot** and select the two surface regions that represent the clamp area on SHELF.PRT as shown in the following figure.
- Fix all 6 DOFs.

2. Create a surface load to simulate the motor named **motor**.

- Click **Model > Loads > New > Surface**.
- Select the rectangular surface region on the bottom of U_CHANNEL.PRT.
- Enter **-1000** as the Force component in the Y-direction.

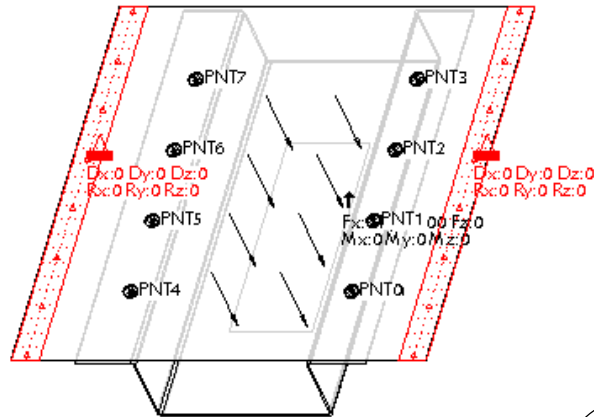


Figure 3: Assembly with all the boundary conditions and spot welds defined

Step 6. Create and run a static analysis.

- Define and run a static analysis named **spot_weld** and accept all the defaults.
1. Click **Analyses/Studies > File > New Static** and define a static analysis named **spot_weld** and accept all the defaults.
 2. Run the **spot_weld** analysis. A confirmation window warns that there are originally 2 disjoint bodies in the model. Confirm to proceed. They should be properly connected by spot welds.

Note:

Depending on the Mechanical software build code, the warning message will not be displayed.

Step 7. Display and interpret the results.

- Create and show displacement a von Mises stress result window. Note the maximums in the model.
1. Create and show a displacement result window. Note the maximum displacement in the model.

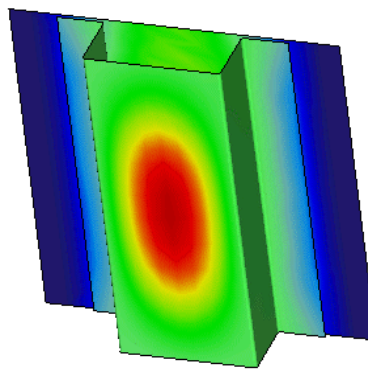


Figure 4: Displacement Fringe Plot

2. Create and show a von Mises stress result window. Note the maximum stress in the model.

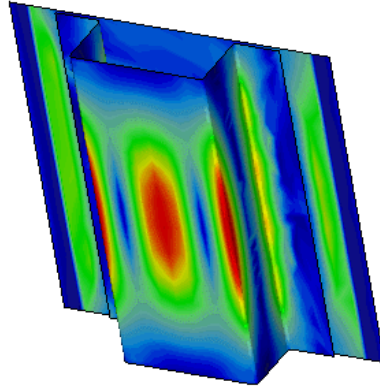


Figure 5: von Mises Fringe Plot

3. Do the spot welds create any excessive stress concentrations on the shelf?
4. Exit the results window. Do not save.
5. Save the model.
6. Close the window and erase all models from session.

Exercise 2: Contact Analysis

Objectives

After successfully completing this exercise, you will know how to:

- Create setup contact connections surfaces.
- Setup contact analysis.

Scenario

In this exercise, a contact region is created and an analysis with contact surfaces is conducted. An analysis with contacts is used to solve design problems where parts come into contact, but the exact regions that touch are unknown. Examples of these parts are rollers pressing against each other, press fits, rotating gears or shaft assemblies. Contact analysis is applicable whenever the model stiffness or load path changes as a function of the applied load.

As an engineer you designed a latch mechanism used to secure a small electronic box on an airplane. When the front compartment door to this box is closed, the latch is subjected to a horizontal force of 400 N. Further movement is prevented when it comes into contact with a pin that is completely fixed to the interior skin of the aircraft. You are concerned about stresses on the mounting surface of the pin due to this contact. In addition, you want to check for any unreasonable stresses in the latch or pin due to the contact pressure.

Step 1. Open the LATCH.ASM model and enter Mechanica Structure.

- Open LATCH.ASM, check that the units are mmNs. Enter Mechanica Structure.

1. Open LATCH.ASM and check that the units are mmNs.
2. Enable Mechanica Structure.



Figure 6: Latch assembly

Step 2. Assign STEEL as material for both parts.

- Assign STEEL material for both parts

1. Assign the material **STEEL** to CATCH.PRT and GROUND.PRT models.

Step 3. Constrain LATCH.ASM.

- Create a constraint and constraint the bottom of the latch as shown below so that the latch can only move freely in the x-direction.
 - Create another constraint and fully constrain the back of the pin as shown below.
1. Create a surface constraint. Constraint the bottom surface of the latch as shown in the following figure so that LATCH.PRT can only move freely in the x-direction.
 - Click **Model > Constraints > New > Surface**.
 - Free translation in x-direction and fix all other DOFs.

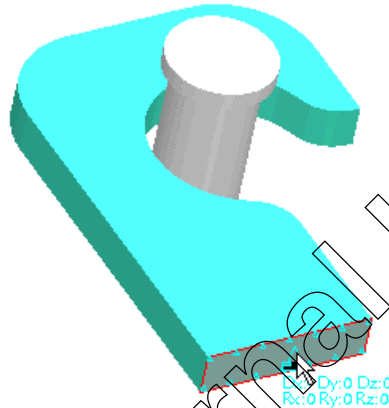


Figure 7: Latch Surface Constraint

2. Create another surface constraint under the same constraint set name. Constrain the rear surface of the pin as shown in the following figure.
 - Click **Model > Constraints > New > Surface**.
 - Fix all the six DOFs.

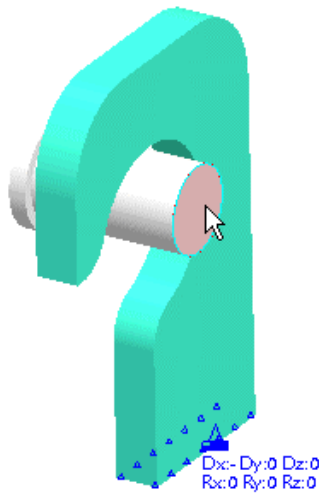


Figure 8: Pin Surface Constraint

Step 4. Apply the load.

- Apply a load of 400 N in the x direction.
1. Apply a surface load of 400 N in the X-direction as shown in the following figure.
 - Click **Model > Loads > New > Surface**.
 - Select the surface as shown in the following figure.
 - Enter **400** as the Force component in the X-direction.

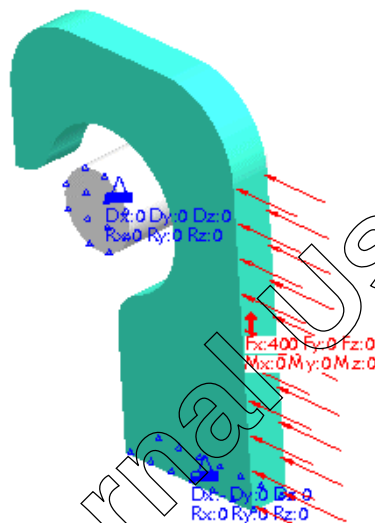


Figure 9: Surface Load

Step 5. Define the contact regions.

- Create a contact connection between the surfaces that may be in contact as shown below. Notice that the contact icon appears.
1. Create a contact connection.
 - Click **Model > Connection > Contacts > Create > Face/Surface**.
 - Select the surfaces that may be in contact as shown in the following figure. Notice that the contact icon appears.
 - Review the contact region. Review the contact connection, the system highlight surfaces in red.

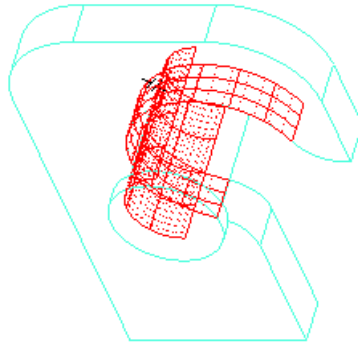


Figure 10: Contact Region Surfaces

Step 6. Review the measures.

- Set up any relevant measures you need for this analysis.
1. Click **Model > Measures** and review the contact measures that are automatically defined. You can define other relevant measure you need for the analysis.

Step 7. Define the contact analysis and run it.

- Create a static analysis named `contact_spa` and include contact regions. For convergence method select Single-Pass Adaptive and select Localized Mesh Refinement.
 - Run the `contact_spa` analysis.
 - Monitor the process using the summary file.
1. Click **Analyses/Studies > File > New Static** and create a static analysis named **contact_spa**.
 - Select the check box **Include Contact Regions**.
 - For convergence method select Single-Pass Adaptive.
 - Select the check box Localized Mesh Refinement.
 2. Run the **contact_spa** analysis. Confirm when prompted for error checking.
 3. Confirm when system warns that there 2 disjoint bodies in the model. Since there is a contact region defined between the 2 bodies, the model is sufficiently constrained.

Note:

Depending on the Mechanica software build code, the warning message will not be displayed.

4. Monitor the process using the summary file. Notice that the Mechanica recognizes the contact area is small and automatically refines the mesh in that area.

Step 8. View and interpret the results.

- Create and display an animated displacement fringe plot.
1. Create and display an animated displacement fringe plot. Verify that the loading is causing the latch to come into contact with the pin. Note that the latch may appear to pass through the pin. This happens because the magnitude of the displacement is magnified (scale factor).

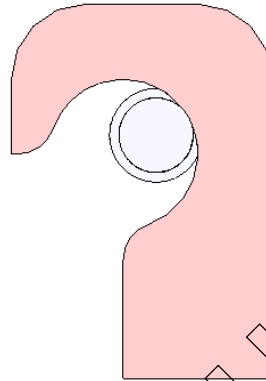


Figure 11: Displacement Result Window

- Create and display a non-animated von Mises stress fringe plot.
2. Create and display a non-animated von Mises stress fringe plot. Edit legend scale for clearer display as necessary.

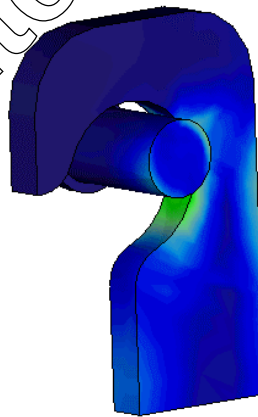


Figure 12: von Mises Stress Fringe Plot

3. Notice that the contact area is very small. Is this an effective latch mechanism? From the stress level, you can conclude that the latch mechanism is over designed.

Step 9. Take a closer look at the contact area and pressure on the latch.

- Create a 30% depth cutting surface on XY plane to look at the inside of the model.
1. Create a cutting surface to look at the inside of the model.

2. Accept the default WCS as well as the XY plane. Enter 30% for the cutting depth.
3. Dynamically modify the cutting plane.

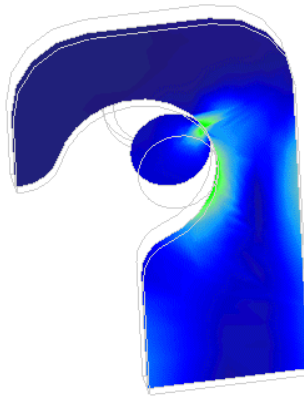


Figure 13: Stress Inside Of The Model Using The Cutting Plane

4. Exit the results window. Do not save.
5. Save the model.
6. Close the window and erase all models from session.

Exercise 3: Transient Thermal Analysis

Objectives

After successfully completing this exercise, you will know how to:

- Create a time dependent loads.
- Setup a transient thermal analysis.

Scenario

In this exercise, an unsteady thermal analysis is created and run. For engineering processes that involve heating and cooling, the transitional period of time is of great interest. The analysis must be modified to take into account the change in internal energy of the body with time. In Mechanica, this type of data is obtained through a Transient Thermal analysis.

You will be working on a rocket-engine nozzle for a prototype shuttle between Earth and the new Space Station. At the time of ignition, this nozzle will be subjected to hot gases at 1648.8 °C. The gases combine and reach their maximum temperature over time. Your job as engineer is to determine if the rocket's engine is on too long before launch, thus, over-heating the nozzle. The engine is on for 100 seconds before lift off. There is an electronic warning switch located on the inner surface of the nozzle that will alert launch control if the nozzle temperatures are higher than 1371°C. In addition, you need to ensure that the temperature gradients in the coating are not too large during the initial start-up. There can be no more than a 400°C difference between the inner and outer surfaces of the nozzle. Finally, the thermal protection engineers would like to correlate data with your analysis and have asked you to provide complete temperature profiles for two points during the count-down; one half-way through ignition (50 seconds), and one right before lift-off (100 seconds). You will use Mechanica to determine:

- How long after start up will it take for the temperature on the ceramic coating's surface to reach 1371°C?
- If there are any large thermal gradients during ignition that might cause the ceramic coating to crack?
- What the temperature profile looks like halfway through ignition and right before lift-off?

Considerations for Transient Thermal Analyses

Some issues to consider when using Transient Thermal Analysis in Mechanica are as follows:

- The model cannot contain multi-point constraints.
- It is available for 3D models with isotropic elements only (no shells/beams).
- It is available for standard analyses only (no sensitivity or optimization).

The time dependence is a multiplier function, so the user can only enter loads that also have a spatial variation as the product of two functions. For example, $f(x,y,z)$, $f(\text{time})$ not $f(x,y,z,\text{time})$

Step 1. Open NOZZLE.PRT and assign material properties.

- Open NOZZLE.PRT and enter the Mechanica Thermal.
- Create a new material named CERAMIC using the information below and assign it to the part

1. Open NOZZLE.PRT. Note that the unit system is mmNs.

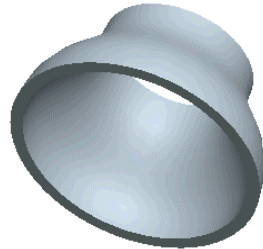


Figure 14: NOZZLE.PRT

2. Enable Mechanica Thermal.

3. Create a new material named **CERAMIC** using the information below and assign it to NOZZLE.PRT.

- The material properties of ceramic and the convection coefficient are as follows:

Table 1: Material Properties

Thermal Conductivity	$k=10,781 \text{ W/m}^\circ\text{C}$
Specific Heat Capacity	$c=1.624 \text{ kJ/kg }^\circ\text{C}$
Density	$\rho=4.52 (10^3) \text{ kg/m}^3$
Young's Modulus	$E = 400,000 \text{ N/mm}^2$
Convection Coefficient	$h = 4 (10^3) \text{ kW/m}^2^\circ\text{C}$

- The above properties can be converted as follows:

Table 2: Converted Properties

Thermal Conductivity	$k = 10781 \text{ N/sec C}$
Specific Heat Capacity	$c = 1624 \text{ m}^2/\text{sec}^2\text{C}$
Density	$\rho = 4520 \text{ kg/m}^3$
Young's Modulus	$E = 400,000 \text{ N/mm}^2$
Convection Coefficient	$h = 4000 \text{ mW/mm}^2\text{C}$

Step 2. Apply the convection boundary conditions with a time dependent bulk temperature

- Create a time dependent convective surface boundary condition named flame on the inside nozzle surface.
- Enter **4000** for the convection coefficient and **1** for the Bulk Temperature.
- For the table import the convection.txt located in the current directory. Review the graph.

1. Create a convective surface boundary condition named **f_lame**.
 - Click **Model > Bndry Conds > New > Conv Cond > Surface**.
 - Edit the name to **f_lame**.
 - Accept the default constraint set and select the inside nozzle surface.

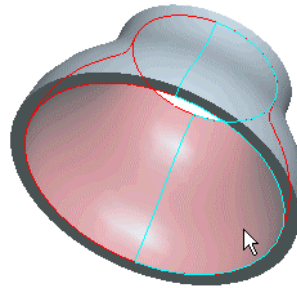


Figure 15: Convective boundary condition surfaces

- Enter **4000** for the Convection Coefficient and **1** for the Bulk Temperature.
- Select the check box **Time Dependant**.
- Click **f(x)** and select **Table** as type from the drop-down list.
- Click **Import** and select **convection.txt** located in the current directory.
- Click **Review > Graph** to review the graph.

Note:

In this case, the Bulk Temperature is only a factor. The actual time dependent temperature is defined in the next step.

- Close the Graphtool, function and the Convection Condition windows.

Step 3. Define and run a Transient Thermal analysis.

- Create a transient thermal analysis named **nozzle_transient**. Enter **-17.7** as the initial temperature.
- Set the analysis to calculate Heat Flux. Set 3 Master Intervals. For Interval 0, enter **0** seconds; for interval 1, enter **50** seconds; for interval 2, enter **100** seconds; for interval 3, enter **120** seconds. Check the Full Results box next to each interval.
- Run the analysis. Monitor the process using the summary file.

1. Create a transient thermal analysis named **nozzle_transient**.
 - Click **Analyses/Studies > File > New Transient Thermal**.
 - Name the analysis **nozzle_transient**.
 - Select the default constraint set.

- Enter **-17.7** as the temperature.
- In the Output tab, select the check box Heat Flux.
- Select the option User-defined Output Intervals from the Output Intervals drop-down list. Change the Master Intervals value to **3**.
- Click **User-defined Steps**, and enter **0** seconds for Interval 0; enter **50** seconds, for interval 1; enter **100** seconds, for interval 2; enter **120** seconds, for interval 3.
- Select the check box **Full Results** next to each interval.
- Click **OK**.

2. Run the analysis. Monitor the process using the summary file.

Step 4. Review the results.

- Create a graph for max_dyn_temp measure vs. time.
 - Create a temperature gradient fringe plot at 50 and 100 seconds.
 - Create a temperature distribution fringe plot at 50 and 100 seconds.
1. Create a graph for **max_dyn_temp** measure vs. time. The result window should appear as shown in the following figure.
 2. Using the following graph:
 - Determine at what time the nozzle reaches a temperature of 1371 °C: _____.
 - Is it prior to lift-off (100 seconds)? _____
 3. Create a temperature gradient fringe plot at 50 and 100 seconds.
 - Does the temperature gradient exceed 400°C/sec anywhere in the model?
 4. Create a temperature distribution fringe plot at 50 and 100 seconds. This fringe plot could be used to correlate to experimental data.
 5. Exit the results window. Do not save.
 6. Save the model.
 7. Close the window and erase all models from session.

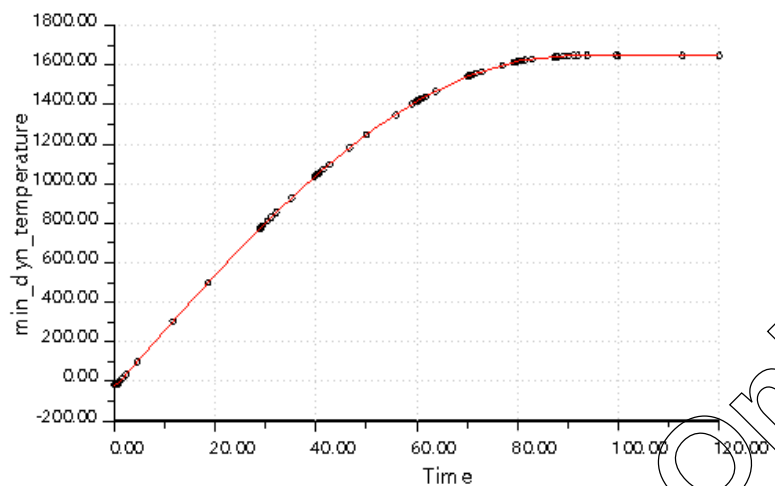


Figure 16: Maximum Dynamic Temperature vs. Time

Exercise 4: Large Deformation Analysis

Objectives

After successfully completing this exercise, you will know how to:

- Create a large deformation analysis.

Scenario

In this exercise, an analysis with large deformation is created and run for an implant part for an eye. As an engineer, you are concerned about stresses and displacement around the tips of the implant.

Step 1. Retrieve EYE_IMPLANT.PRT.

- Retrieve the EYE_IMPLANT.PRT located in the current working directory. The model is cut in half to take advantage of its symmetry.
1. Open EYE_IMPLANT.PRT.
 2. The model is cut in half to take advantage of its symmetry. Investigate the last cut by redefining it.



Figure 17: Symmetrical implant part

Step 2. Assign the material properties.

- Enter Mechanics Structure and assign Nylon for the material library to the part.
1. Enable Mechanics Structure.
 2. Assign **Nylon** for the material library to the part.

Step 3. Create a constraint for symmetry.

- Create a new surface constraint named symmetry on the cut surface. Change the coordinate system to CS1 as shown below.
- Fix Z Translation and free for all other DOF's.

1. Create a new surface constraint named **symmetry**.

- Click **Model > Constraints > New > Surface**.
- Name the constraint **symmetry**.
- Select the cut surface as shown in the following figure.

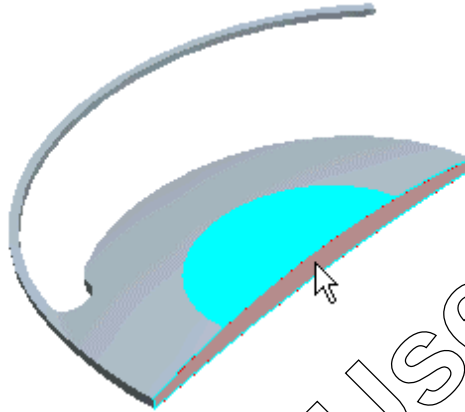


Figure 18: Surface for symmetry constraint

- Change the coordinate system to CS1 as shown in the following figure.

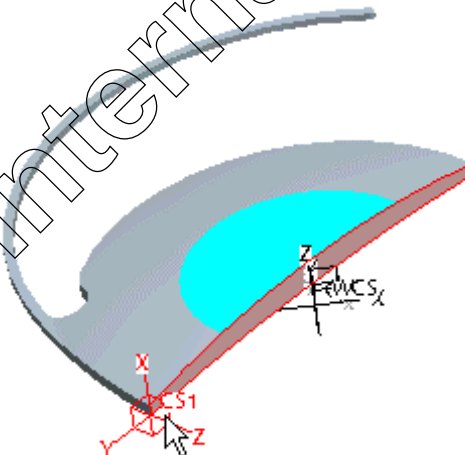


Figure 19: Coordinate system for the symmetry constraint

- Fix translation in Z-direction and rotation in X and Y-direction.
- The data dialog box should appear as shown in the following figure.

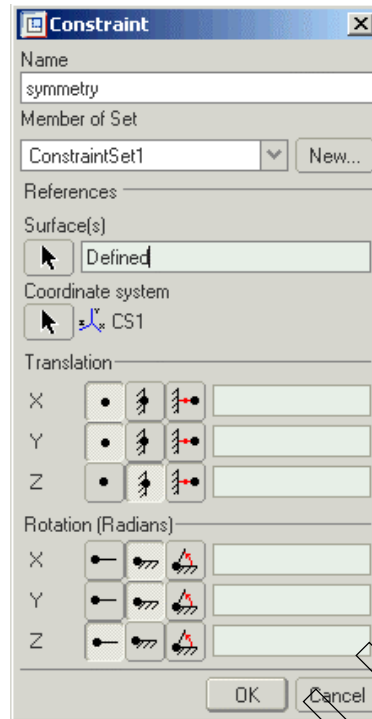


Figure 20: Symmetry constraint data dialog box

- Click **OK**.

Step 4. Create a constraint on bottom of the implant.

- Create another surface constraint on the bottom surface of the model as shown below, named eyeball. Change the coordinate system to the World Coordinate System.
- Fix X, Y, Z Translation and free for all the DOF's.

1. Create another surface constraint named **eyeball** on the bottom surface of the model as shown in the following figure.

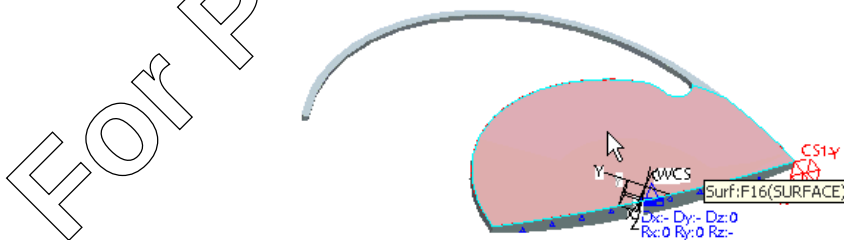


Figure 21: Eyeball surface for constraint

- Change the coordinate system to the World Coordinate System.
- Fix all the DOF's.
- The data dialog box should appear as shown in the following figure.

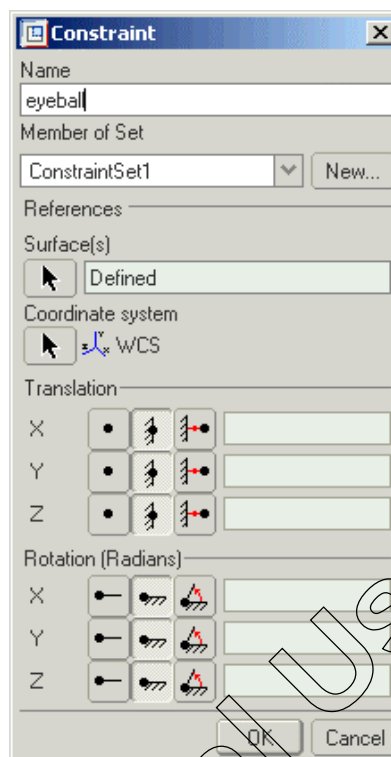


Figure 22: Eyeball constraint data dialog box

- Click **OK**.

Step 5. Create a load at the end of the long segment.

- Create a surface load on the surface region shown. Select the world coordinate system WC.
- As distribution, select Interpolated Over Entity and add the points PNT0, PNT1, PNT2. Enter 1 for the second point and preview the load. Enter -0.0008 as the Y force component.

1. Create a surface load on the surface region shown.

- Click **Model > Loads > New > Surface**.



Figure 23: Tip load surface

2. Select the world coordinate system WC.
3. Select **Interpolated Over Entity** as the distribution.
4. Define the interpolation points as follows:

- Turn on the datum point display as necessary.
- Add the points PNT0, PNT1, PNT2, as shown in the following figure.

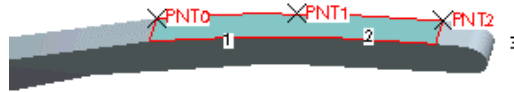


Figure 24: Interpolation points

5. Enter **1** for the second point and preview the load.
6. Enter **-0.0008** as the Y force component. The Force/Moment data dialog box should appear as shown in the following figure.

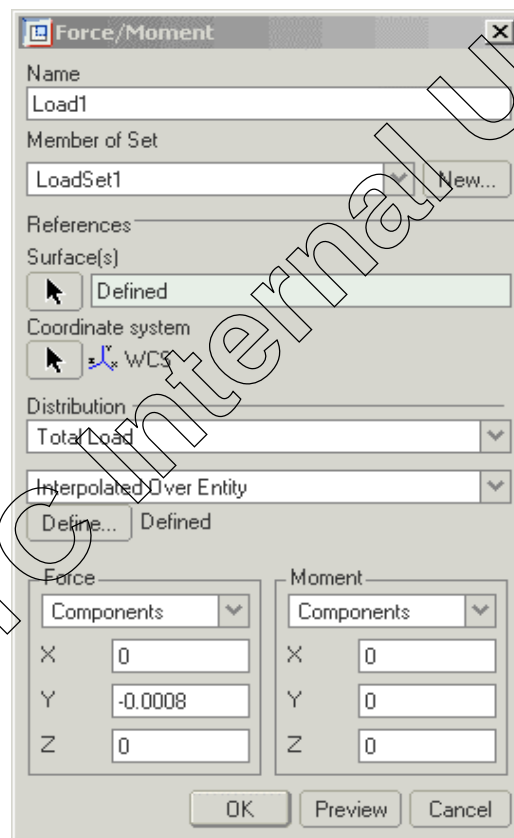


Figure 25: Tip load data dialog box

Step 6. Define a static analysis with large deformations.

- Create a static analysis named large_def. Accept the default constraint set and the load set.
- Check the Calculate Large Deformations and accept the default Single-Pass Adaptive.
- Examine the options under Load Intervals tab. Accept the default interval 1.

1. Create a static analysis named **large_def**.
2. Accept the default constraint set and the load set.
3. Select the check box **Calculate Large Deformations**.
4. Accept the default Single-Pass Adaptive.
5. Examine the options under Load Intervals tab. You can use the Number of Intervals to run the analysis at certain load intervals. To save time, accept the default interval **1**.

Step 7. Run the analysis.

- Run the analysis just created and monitor the analysis process.
1. Run the analysis just created.
 2. Monitor the analysis process. When the analysis run is finished, close the summary window.

Step 8. Create the result windows.

- Create a fringe plot of displacement magnitude and another of von Mises stress.
1. Create result window to display a fringe plot of displacement magnitude over the model from large_def results.
 2. Create another result window to display a fringe plot of von Mises stress over the model from large_def results.

Step 9. Display the result window and examine the results.

1. Display both result windows.

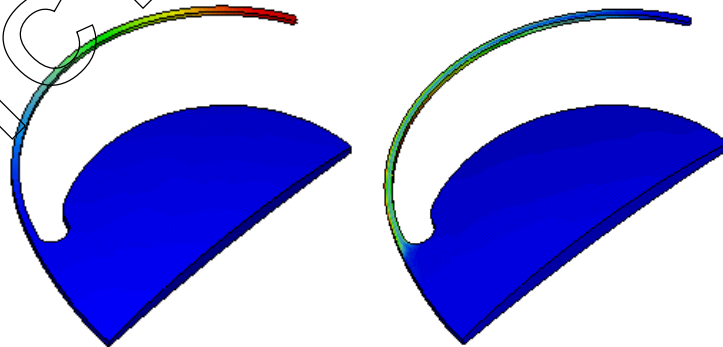


Figure 26: Displacement and von Mises stress fringe plots

2. Close the result window without saving.
3. Save the model.
4. Close the window and erase all models from session.

Exercise 5: Dynamic Time Analysis

Objectives

After successfully completing this exercise, you will know how to:

- Create a time dependent excitation.
- Create a dynamic time analysis.

Scenario

In this exercise, vibration studies on an exhaust part are conducted. A dynamic time analysis is used to solve the model's response to a time varying input.

The model used is an exhaust pipe. As an engineer your job is to ensure the following:

- The design will withstand a 10ms, 20g half-sine shock encountered

Step 1. Open the EX_PIPE.PRT, assign material properties and define model idealizations.

- Open the part EX_PIPE and enter Mechanica structure.
- Assign SS (stainless steel) to the part and define the shell pairs using auto-detect.

1. Open EX_PIPE.PRT.
2. Enable Mechanica structure.



Figure 27: Exhaust part

3. Assign SS (stainless steel) to EX_PIPE.PRT.
4. Define the shell pairs using Auto detect.
 - Click **Model > Idealizations > Shells > Midsurfaces > Auto Detect > Done/Return.**

Step 2. Adding constraints.

- Create a constraint to apply a symmetry constraint to the curves defining either the inside or the outside of the exhaust in the symmetrical cutting plane. The symmetrical cutting plane is the XY plane.
- Finish constraining the model by fixing the surface, where the exhaust pipe attaches to the cylinder head, in the TransX and TransY directions.

1. Create an edge constraint to apply a symmetry constraint to the curves defining either the inside or the outside of the exhaust in the symmetrical cutting plane. The symmetrical cutting plane is the XY plane.

- Click **Model > Constraints > New > Edge/Curve**.

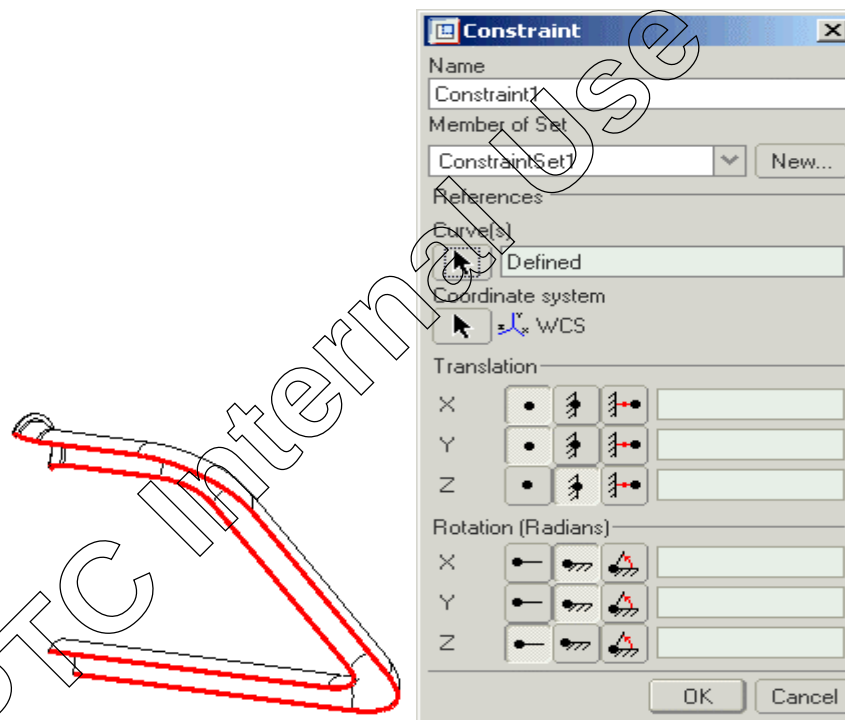


Figure 28: Edges for Symmetry Constraint and the Constraint Definition

- Set up the Constraint dialog box as shown in the previous figure.
2. Finish constraining the model by fixing the surface as shown in the following figure, where the exhaust pipe attaches to the cylinder head. Be sure to add it to the same constraint set.
 - Click **Model > Constraints > New > Surface**.
 - Select the surface as shown in the following figure.
 - Fix the translation in the X and Y-directions.

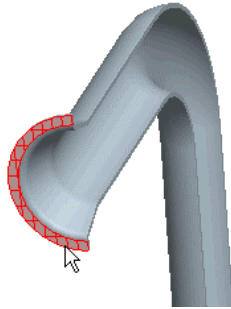


Figure 29: Constraint surface

Step 3. Create and run a model analysis.

- Create a modal analysis named **p_modal**. Use four modes only. Run the analysis.

Note:

Since Mechanica uses a modal analysis to solve all dynamic problems, the quality of the results are directly related to the accuracy of the modal analysis results.

1. Create a modal analysis named **p_modal** as shown in the following figure.

- Click **Analyses/Studies > File > New Modal**.
- Enter value **4** for the Number of Modes.

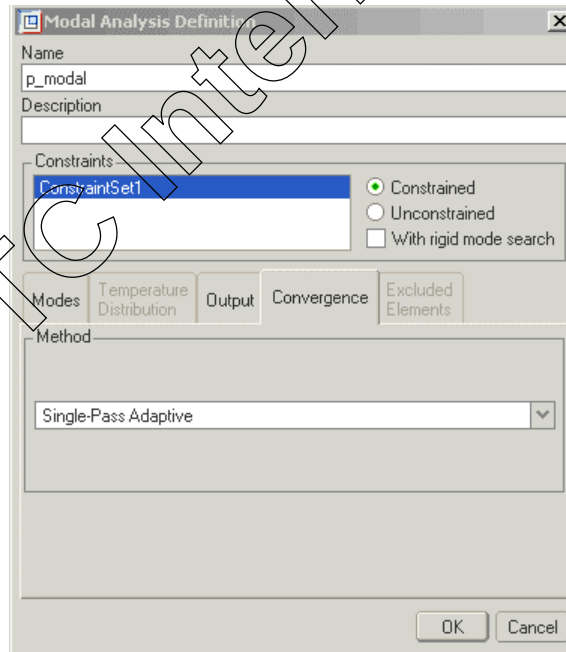


Figure 30: Modal analysis dialog box

- Click **OK**.
2. Run the analysis.

Step 4. Create measures.

- Create a datum point on vertex at the tip of the pipe. Create the measures as listed.

Note:

For this model, you are concerned about displacements and stresses. In addition, you may want to know what the acceleration is at the exit tip of the exhaust pipe.

1. Create a datum point on vertex at the tip of the pipe as shown in the following figure.

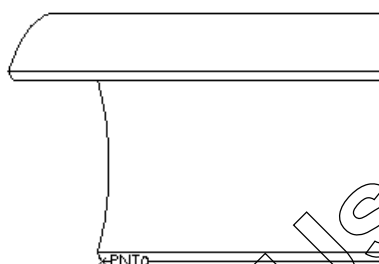


Figure 31: Datum point for acceleration at tip

2. Create the measures listed in Table 3. If you want, you can create additional measures:

Table 3: Measures

Measure	Quantity	Component	Spatial Eval	Time/Freq Eval
disp_y_max	Displacement	Y	Maximum Over Model	At Each Step
stress_vm_max	Stress	von Mises	Maximum Over Model	At Each Step
accel_y_tip	Acceleration	Y	At a Point	At Each Step

Step 5. Create a dynamic time analysis.

- Create a dynamic time analysis named PIPE_DYNAMIC_TIME.

1. Create a dynamic time analysis named **pipe_dynamic_time**.

- Click **Analyses/Studies > File > New Dynamic > Time**.
- Name the analysis **pipe_dynamic_time**.
- Set up the analysis as shown in the following figures.

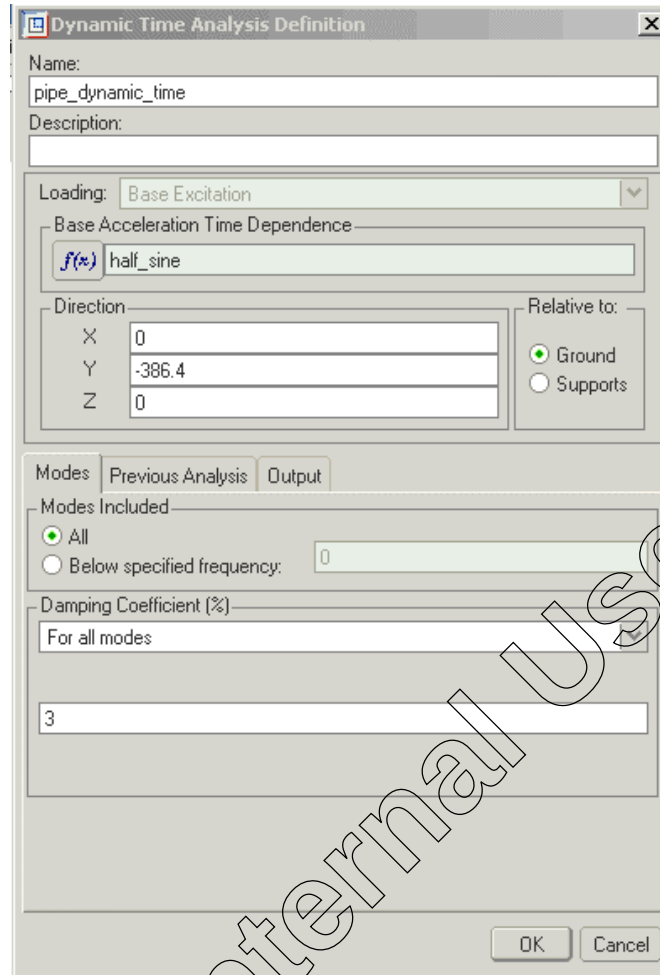


Figure 32: Dynamic time analysis dialog box

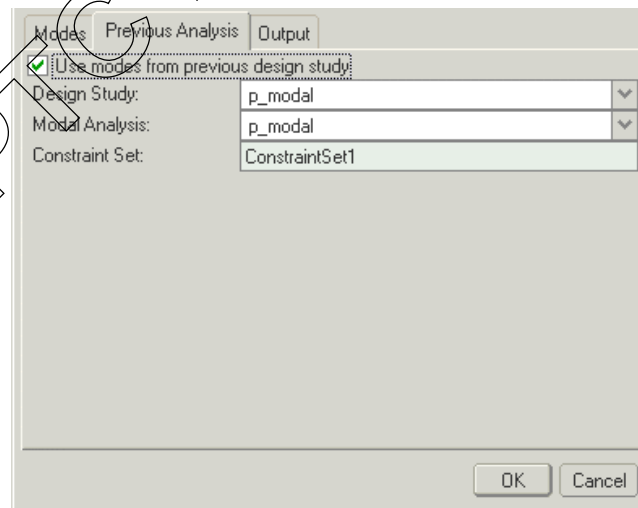


Figure 33: Setting the Dynamic time analysis

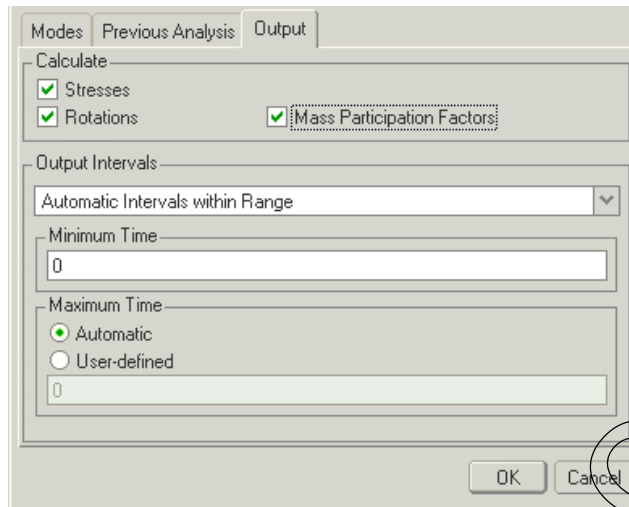


Figure 34: Setting the Dynamic time analysis

2. For the Base Acceleration Time Dependence, create a new time function called **half_sine** and enter the expression as symbolic:
 - **`if(time<0.01,20*sin(pi*time/0.01),0)`**
3. Use the plotting utility to graph the half-sine input using a lower limit of **0**, and an upper limit of **0.025**. Your plot should look like the following graph:

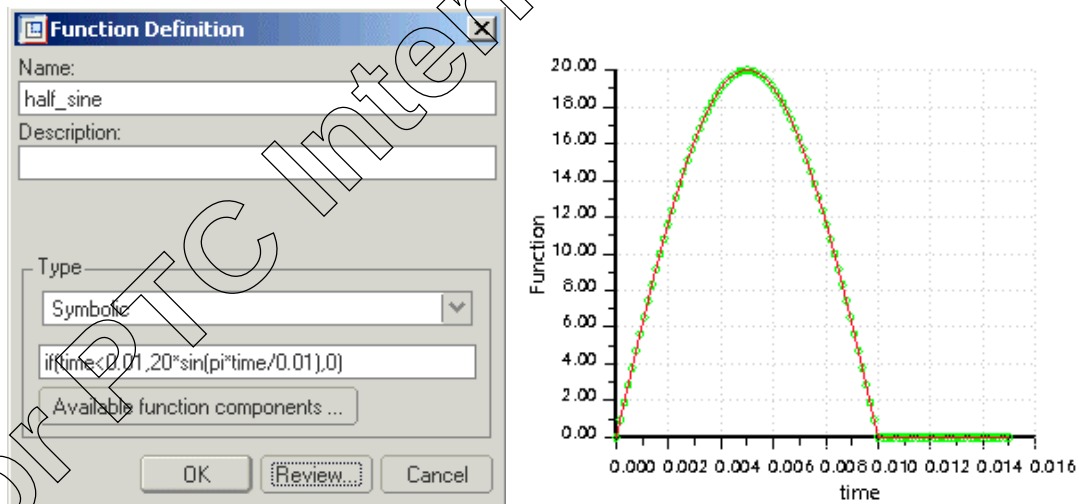


Figure 35: Input function

4. Select **p_modal** as the modal analysis.
5. Select the check box Mass Participation Factor so Mechanica will calculate it.
6. Leave the time step intervals in Auto mode. This instructs the computer to pick a time stepping interval based on the applied load and modal frequencies.

Note:

Some experts advocate formulas for individual mode damping, while others are proponents of frequency-dependent functions. Typically, a simple uniform percentage is used.

7. Assuming little flexibility in the bolted interface between the cylinder head and exhaust flange, use 3% critical damping.

Step 6. Run the dynamic time and explore the results.

- Run `pipe_dynamic_time` and monitor the run using the summary and the log file file.
- Set up three graphs, one for each measure, as a function of time.

1. Run `pipe_dynamic_time`.
2. Monitor the run using the summary and the log file.
3. When the run has completed, first check for the total mass participation to make sure you captured enough modes. Remember this is the second check you should do to access the quality of your solution. You can find the total mass participation on the summary file. Make sure that the total mass participation is equal 80% or above.

Note:

The mass participation factor (m.p.f) is reflective of the mass that participates in a particular mode. The sum of the mass participation factors tends towards the sum of the model mass (i.e. adding up all the individual mass participation factors, for a large enough set of modes, should be some large percentage, say 50-90%, of the square of the mass). The effective mass (e.m) is the ratio of the mass participation factor over the total mass squared, and is expressed as a percentage. The total effective mass (t.e.m.) is the sum of the individual effective masses of all the modes up to and including the current mode, and is also expressed as a percentage.

The estimated % that the total effective mass should reach is the industry guideline of 80%. If the reported percentage is less than 80%, the user should request more modes in the modal analysis.

However, it is possible that the load is being applied such that only high frequency modes (e.g. modes 72 to 75) are excited. Therefore, it is still possible that the mass participation percentage will be low if the analysis includes a low number of modes (i.e. 20). In this case, four modes were more than enough to reach the 80% total mass mark. If, by the end of mode four, the total mass was still below 80%, you would need to redefine the modal analysis `p_modes` to include more modes, run the modal analysis again, and iterate until the shock response yielded 80% total mass or better.

4. Now you are ready to look at some results using the measures you created earlier. More specifically, set up three graphs, one for each measure, as a function of time. `Stress_vm_max` vs. time is shown here.

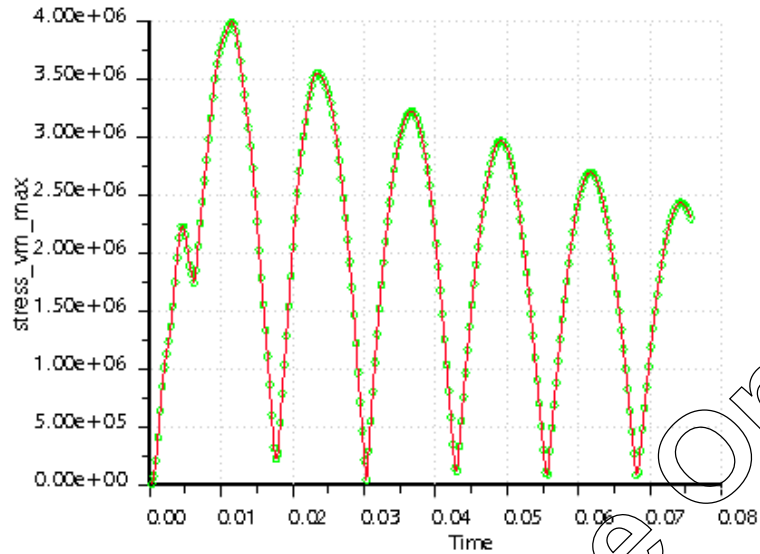


Figure 36: von Mises stress vs. time

5. You can determine at what time the stress, displacement and acceleration peaks occur. These are the times at which you will want to have full results calculated. Review the graph and extract the times corresponding to the first three peaks.

Step 7. Create another dynamic time analysis. This time extract full results at those three times.

- Create a dynamic time analysis named **pipe_dynamic_time_final** and change the parameters.

1. Create a dynamic time analysis named **pipe_dynamic_time_final**.
2. Change the parameters to appear as shown in the following figure.

Dynamic Time Analysis Definition

Name: pipe_dynamic_time_final

Description:

Loading: Base Excitation

Base Acceleration Time Dependence

$f(\%)$ half_sine

Direction

X	0
Y	-386.4
Z	0

Relative to:

☒ Ground
☐ Supports

Modes Previous Analysis Output

Modes Included

☐ All
☒ Below specified frequency: 400

Damping Coefficient (%)

For all modes

3

OK Cancel

Modes Previous Analysis Output

Calculate

☒ Stresses
☒ Rotations
☒ Mass Participation Factors

Output Intervals

User defined Output Intervals

Number of Master Intervals 3

0.	0	☐ Full results
1.	0.0115	☒ Full results
2.	0.0235	☒ Full results
3.	0.0365	☒ Full results

Measures Output Intervals per Master Interval 10

User-defined Steps
Space Equally

OK Cancel

Figure 37: Setting up the Dynamic time Analysis

Step 8. Create a fringe plot to display von Mises over the model.

- Create a fringe plot to display von Mises stress over the model at the first time (0.0115sec).

1. Create a fringe plot to display von Mises stress over the model at the first time (0.0115sec).

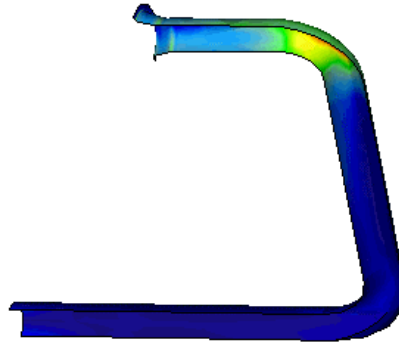


Figure 38: von Mises fringe plot

Exercise 6: Dynamic Frequency Analysis

Objectives

After successfully completing this exercise, you will know how to:

- Create a frequency dependent excitation.
- Create a dynamic frequency analysis.

Scenario

In this exercise, you will run dynamic frequency analysis to explore the frequency response of a model.

The model used is the same as in the previous exercise. You will explore how the vibration frequencies of the engine affect the design.

Step 1. Create the Dynamic Frequency Analysis and run it.

- The dynamic frequency analysis you will define is very similar to the dynamic time analysis you defined in the previous exercise. Create a new dynamic frequency analysis named `pipe_dynamic_frequency`. Run the analysis.

Note:

The dynamic frequency analysis you will define is very similar to the dynamic time analysis you defined in the previous exercise.

1. Create a new dynamic frequency analysis named **`pipe_dynamic_frequency`**.

- Click **Analyses/Studies > File > New Dynamic > Frequency**.
- Set up the analysis as shown in the following figures.

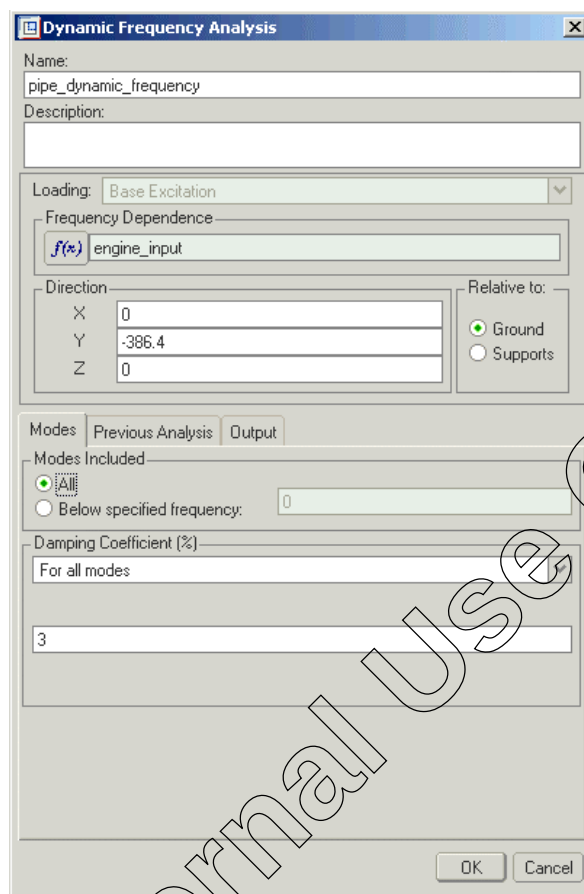


Figure 39: Setting Dynamic frequency Analysis

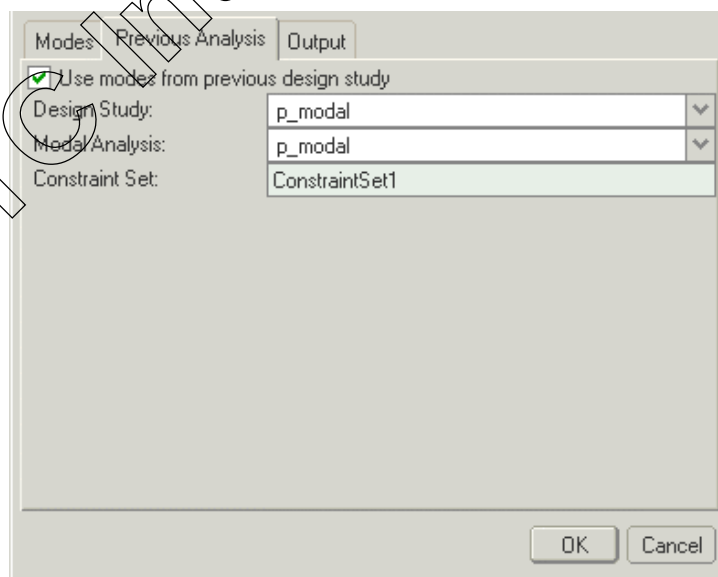


Figure 40: Dynamic frequency dialog box

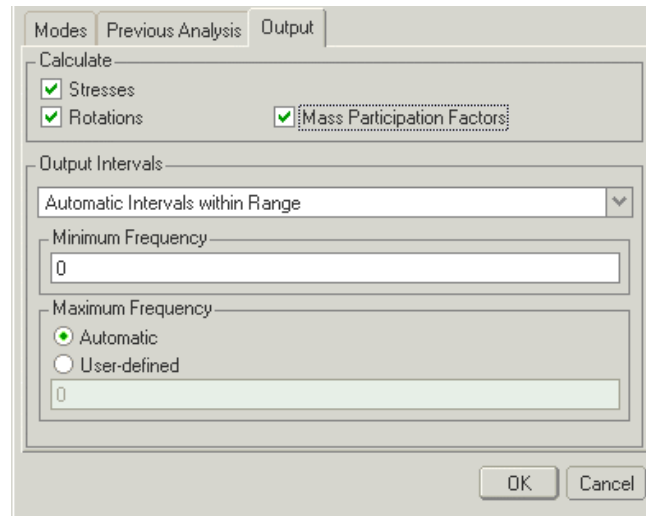


Figure 41: Dynamic frequency dialog box

2. For the frequency dependent input, create a new table input called **engine_input** as shown below.

Table 4: Frequency input

Frequency	Function Value
2	0.3
120	6
270	2
376	11
640	2.5
770	8
1170	1.5
1360	1.5
1920	4
2880	0.4
3200	0.4

Function Definition

Name: engine_input

Description:

Type: Table

frequency Value

1	2	0.3
2	120	6
3	270	2
4	376	11
5	640	2.5

Linear Linear

OK Review... Cancel

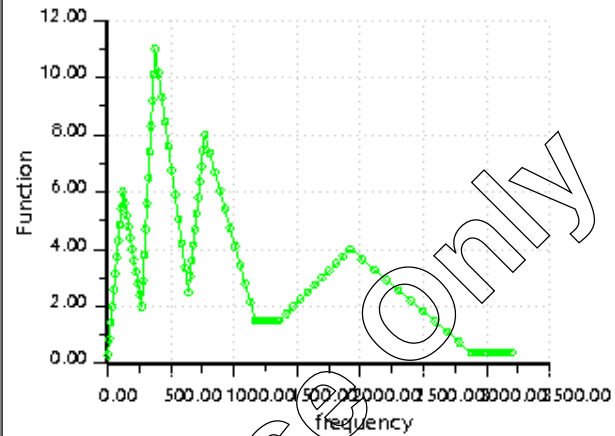


Figure 42: Function Definition for Frequency Dependence input

3. Run the analysis.

Step 2. Explore the results.

- When the analysis is complete, create the three graphs, one for each measure, as a function of frequency. The stress_vm_max vs. frequency graph is shown below:

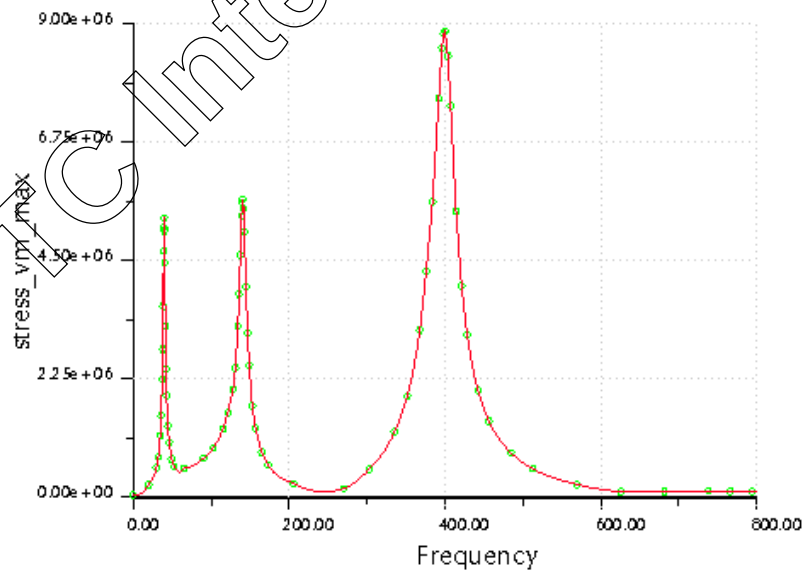


Figure 43: von Mises stress vs. frequency

- Determine the frequencies corresponding to the three peaks.

Step 3. Create another dynamic frequency analysis with intermediate full results at the peaks.

- Create a dynamic frequency analysis called **pipe_dynamic_frequency_final**. Set up the analysis and run the analysis.
- Display the final von Mises stress fringe from the pipe_dynamic_frequency_final analysis at 400.1Hz.

1. Create a dynamic frequency analysis called **pipe_dynamic_frequency_final**.
2. Set up the analysis as follows:

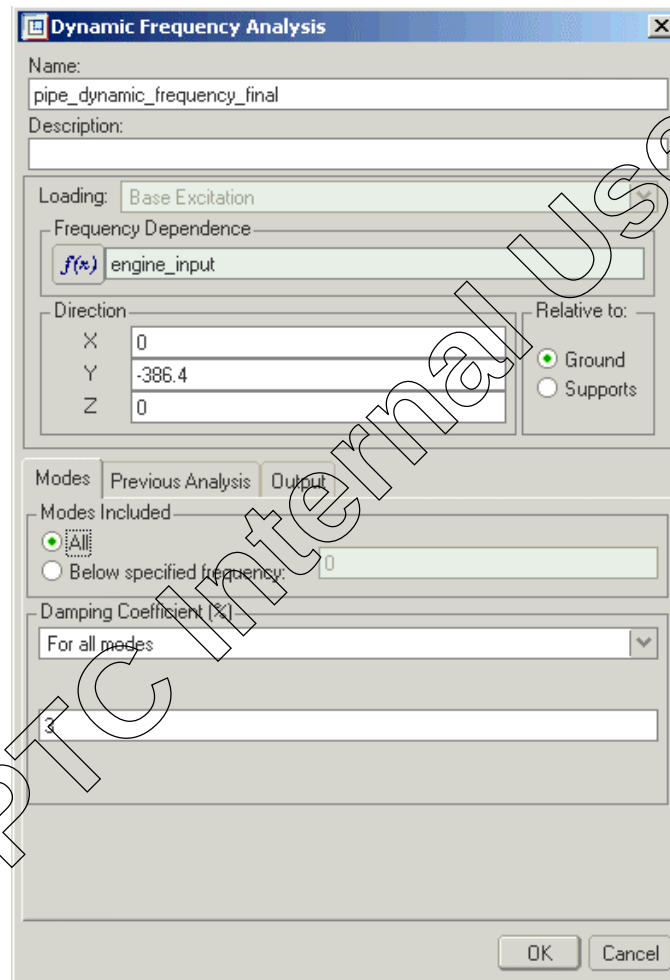


Figure 44: Setting the Dynamic Frequency Analysis

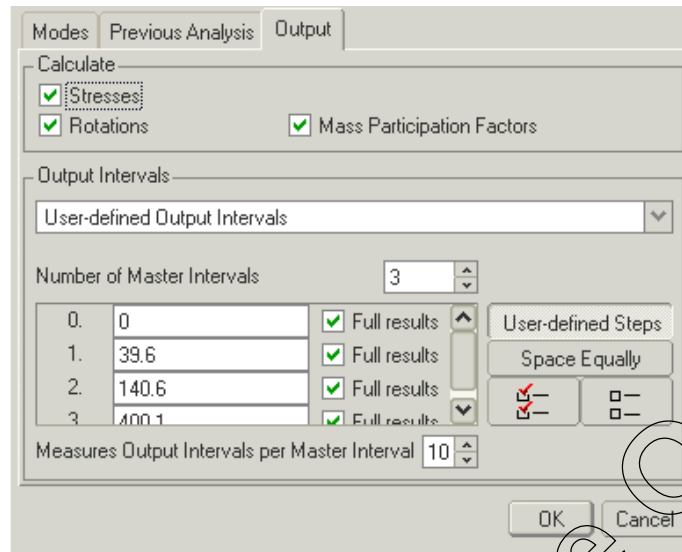


Figure 45: Dynamic Frequency Analysis Dialog Box

3. Run the analysis.
4. Display the final von Mises stress fringe from the pipe_dynamic_frequency_final analysis at 400.1Hz, as shown below.

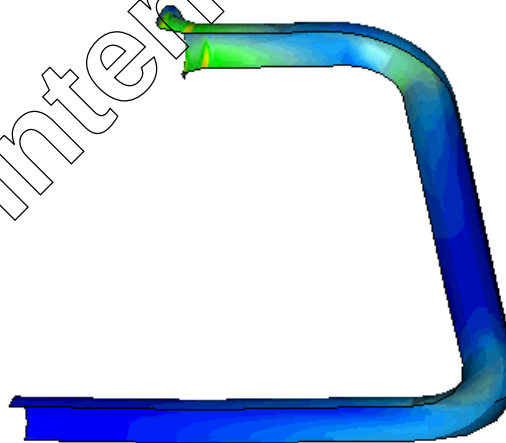


Figure 46: von Mises stress at 400.1 Hz

5. Exit the results window. Do not save.
6. Save the model.
7. Close the window and erase all models from session.

Summary

After completing this module, you will be able to:

- Create spot-welding connections.
- Create setup contact connections surfaces and setup contact analysis.
- Create time dependent boundary conditions and setup a transient thermal analysis.
- Create a large deformation analysis.
- Create a dynamic time analysis.
- Create a dynamic frequency analysis.

For PTC Internal Use Only