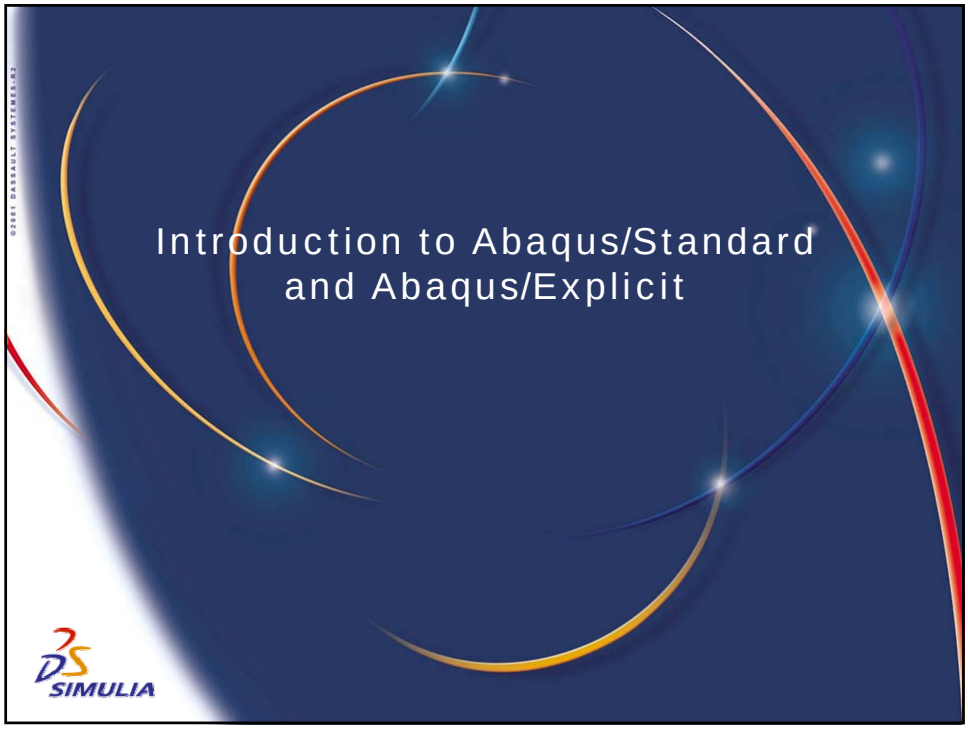
Abaqus/Standard 和 Abaqus/Explicit

基础教程讲义

目录

Introduction……1
第一讲：定义 Abaqus 模型……5
第二讲：线性静态分析……43
第三讲：Abaqus/Standard 中的非线性分析……59
第四讲：Abaqus 中的多步骤分析……84
第五讲：Abaqus 中的接触建模……95
第六讲：Abaqus/Standard 中的接触问题……138
第七讲：动力学简介……148
第八讲：使用 Abaqus/Explicit……167
第九讲：Abaqus/Explicit 中的接触问题……177
第十讲：Abaqus/Explicit 中的准静态分析……190
第十一讲：合并 Abaqus/Standard 和 Abaqus/Explicit……212
附录 1：单元选择标准……225
Workshop Preliminaries……252
习题1：基本的输入输出……255
习题2：悬臂梁的线性静力学分析……277
习题3：斜板的非线性静力学分析……297
习题4：接触分析……308
习题5：Abaqus/Standard中密封圈的CLD分析……324
习题6：动力学……352
习题7：使用Abaqus/Explicit进行接触分析……370
习题8：准静态和回弹分析……386



Day 1

- **Lecture 1** Defining an Abaqus Model
- **Workshop 1** Basic Input and Output
- **Lecture 2** Linear Static Analysis
- **Workshop 2** Linear Static Analysis of a Cantilever Beam: Multiple Load Cases
- **Lecture 3** Nonlinear Analysis in Abaqus/Standard
- **Workshop 3** Nonlinear Statics
- **Lecture 4** Multistep Analysis in Abaqus

© Dassault Systèmes, 2008

Day 2

- Lecture 5 Constraints and Contact
- Workshop 4 Simple Contact
- Lecture 6 Contact Issues Specific to Abaqus/Standard
- Workshop 5 Seal Contact
- Lecture 7 Introduction to Dynamics
- Workshop 6 Dynamics

© Dassault Systèmes, 2008

Day 3

- Lecture 8 Using Abaqus/Explicit
- Workshop 6 Dynamics (continued)
- Lecture 9 Contact Issues Specific to Abaqus/Explicit
- Workshop 7 Contact with Abaqus/Explicit
- Lecture 10 Quasi-Static Analysis in Abaqus/Explicit
- Lecture 11 Combining Abaqus/Standard and Abaqus/Explicit
- Workshop 8 (Optional) Quasi-Static and Import Analysis

© Dassault Systèmes, 2008

Additional Material

- **Appendix 1 Element Selection Criteria**

© Dassault Systèmes, 2008

Legal Notices

The Abaqus Software described in this documentation is available only under license from Dassault Systèmes and its subsidiary and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

© Dassault Systèmes, 2008.

Printed in the United States of America

Abaqus, the 3DS logo, SIMULIA and CATIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus Version 6.8 Release Notes and the notices at: http://www.simulia.com/products/products_legal.html.

© Dassault Systèmes, 2008

Revision Status

Lecture 1	3/08	Updated for V6.8
Lecture 2	3/08	Updated for V6.8
Lecture 3	3/08	Updated for V6.8
Lecture 4	3/08	Updated for V6.8
Lecture 5	3/08	Updated for V6.8
Lecture 6	3/08	Updated for V6.8
Lecture 7	3/08	Updated for V6.8
Lecture 8	3/08	Updated for V6.8
Lecture 9	3/08	Updated for V6.8
Lecture 10	3/08	Updated for V6.8
Lecture 11	3/08	Updated for V6.8
Appendix 1	3/08	Updated for V6.8
Workshop 1	3/08	Updated for V6.8
Workshop 2	3/08	Updated for V6.8
Workshop 3	3/08	Updated for V6.8
Workshop 4	3/08	Updated for V6.8
Workshop 5	3/08	Updated for V6.8
Workshop 6	3/08	Updated for V6.8
Workshop 7	3/08	Updated for V6.8
Workshop 8	3/08	Updated for V6.8

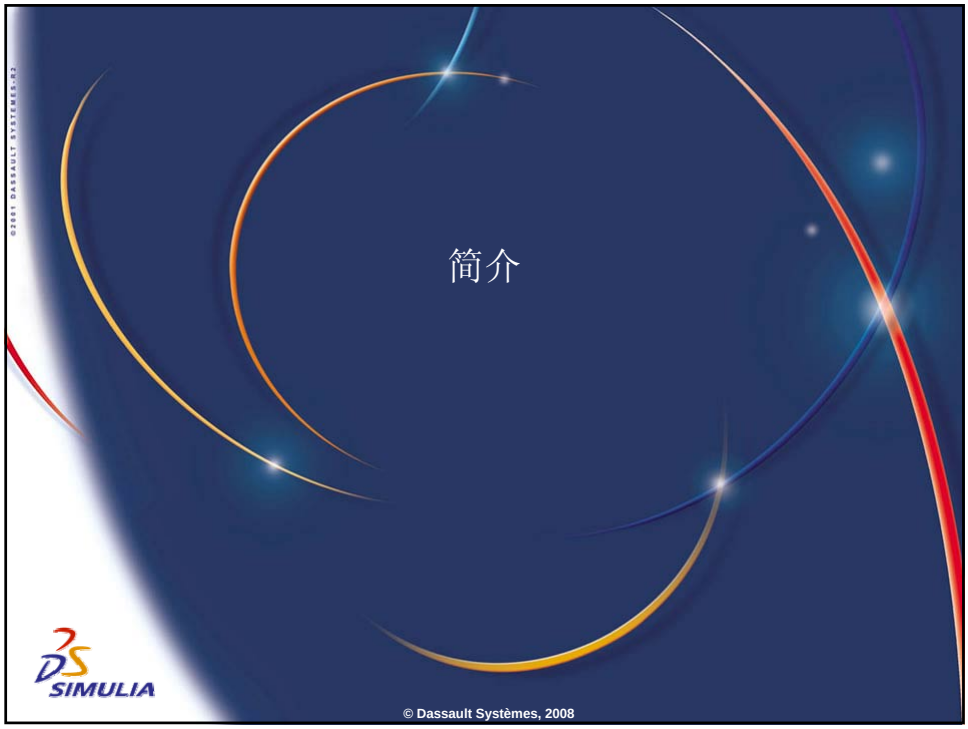
Answers 1	3/08	Updated for V6.8
Answers 3	3/08	Updated for V6.8
Answers 4	3/08	Updated for V6.8
Answers 5	3/08	Updated for V6.8
Answers 6	3/08	Updated for V6.8
Answers 7	3/08	Updated for V6.8
Answers 8	3/08	Updated for V6.8

© Dassault Systèmes, 2008

概述

- 简介
- Abaqus模型的组件
- Abaqus输入文件的细节
- Abaqus输入文件惯例
- Abaqus输出
- 例子：悬臂梁模型
- 部件和装配件（可选）

© Dassault Systèmes, 2008



SIMULIA

- **SIMULIA是达索的注册商标，专注于提供模拟现实世界仿真技术的解决方案**
  - Unified FEA 统一的有限元
  - Multiphysics 多物理场分析
  - SLM 仿真生命周期管理
- **总部位于Providence, RI, USA**
  - R&D centers in Providence and in Suresnes, France

© Dassault Systèmes, 2008

简介

SIMULIA Headquarters: Providence, Rhode Island

Offices:			
USA:	California	Indiana	Michigan
	Ohio	Rhode Island	Texas
Overseas:	Australia	Austria	China
	Finland	France	Germany (2)
	India	Italy	Japan (2)
	Korea	Netherlands	Sweden
	UK (2)		
Representatives:			
Overseas:	Argentina	Brazil	Czech Republic
	Malaysia	New Zealand	Poland
	Russia	Singapore	South Africa
	Spain	Taiwan	Turkey

© Dassault Systèmes, 2008

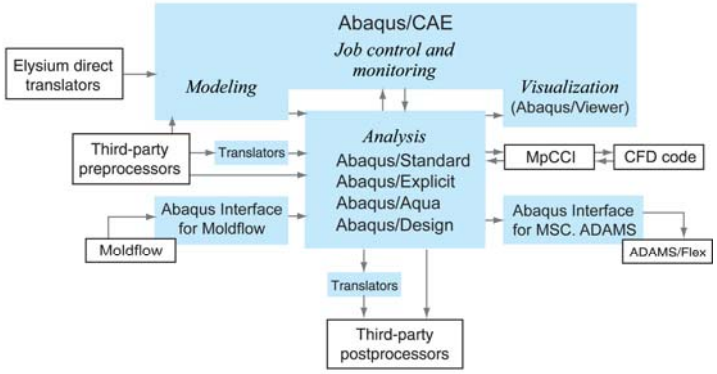
简介

- **课程预备知识**
  - 本课将介绍Abaqus/Standard和Abaqus/Explicit；假定学员具有有限元分析的基础知识。
  - 本课的目的是让学员快速运用Abaqus工作，并介绍相关的概念—本课并不覆盖 Abaqus所有的细节。
  - 根据本课中的主题，还有几个附加信息：
    - SIMULIA网站（ **www.simulia.com** ）。
    - Abaqus文档—在用户手册中包括所有的使用细节。
    - Abaqus广泛的讲稿库（ **www.simulia.com**可以找到讲稿的列表）。

© Dassault Systèmes, 2008

简介

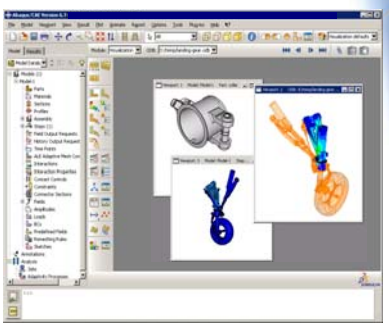
Abaqus是一组有限元分析模块



© Dassault Systèmes, 2008

简介

- **Abaqus/CAE**
  - **C**omplete **A**baqus **E**nvironment完整的Abaqus环境，可以用于建模、管理和监控Abaqus分析过程，并进行结果的可视化处理。
  - 直观的、一致的用户界面。
  - 与普通的CAD系统相似， Abaqus也是基于部件和部件实例装配件的概念。
  - 可以在Abaqus/CAE中创建部件，或从其它系统中导入几何体（在Abaqus/CAE中分网）或网格。
  - 创建部件是内建的、基于特征的、参数化的建模系统。

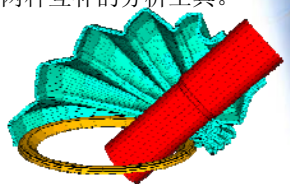


Abaqus/CAE 主界面

© Dassault Systèmes, 2008

简介

- **求解器模块**
  - Abaqus/Standard和Abaqus/Explicit为用户提供两种互补的分析工具。
  - **Abaqus/Standard的功能：**
  - 一般分析功能
    - 静态的应力/位移分析：
      - 率无关响应
      - 率相关（粘弹性/蠕变/粘塑性）响应
    - 瞬态动力学应力/位移分析
    - 瞬态或稳态热传导分析
    - 瞬态或稳态质量扩散分析

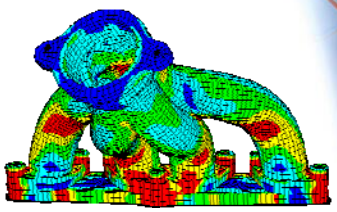


Articulation of an automotive boot seal

© Dassault Systèmes, 2008

简介

- 稳态传输分析
- 耦合问题：
  - 热力（顺序或完全耦合）
  - 热电
  - 孔隙流-固
  - 应力-质量扩散（顺序耦合）
  - 压电分析（只有线性）
  - 声-固

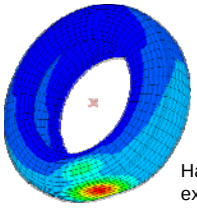


排气管的热应力分析

© Dassault Systèmes, 2008

简介

- **线性摄动分析**
  - **静力/位移分析：**
    - 线性静力位移分析
    - 特征值屈曲载荷预计
  - **动力学应力/位移分析：**
    - 固有频率和固有模态提取
    - 基于模态叠加的瞬态响应
    - 谐波载荷引起的稳态响应谱分析
      - 包括可选的“子空间投影”法，用于分析具有频率相关属性（比如阻尼）的大的模型
    - 响应谱分析
    - 随机载荷引起的动态响应分析



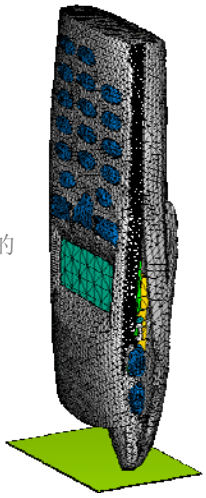
Harmonic excitation of a tire

© Dassault Systèmes, 2008

简介

Abaqus/Explicit’s 功能:

- 高速动力学
- 准静态分析
- 多物理场
  - 热力耦合分析
    - 完全耦合：包含力学和热学响应的显式方法
    - 可以包含绝热升温效应
  - 声固耦合分析
  - 欧拉-拉格朗日耦合分析 (CEL)
  - 流固耦合分析FSI



手机跌落试验

© Dassault Systèmes, 2008

简介

- **Abaqus/Standard和Abaqus/Explicit的比较**

Abaqus/Standard

- 通用的有限元程序
  - 非线性问题需要迭代
- 在结构模拟过程中，可以求解真正的静态平衡问题
- 提供多种分析不同类型问题的功能
  - 非结构问题
  - 耦合或非耦合的问题

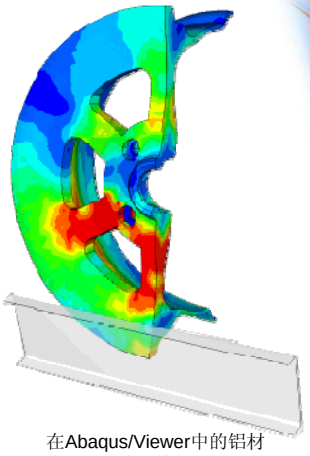
Abaqus/Explicit

- 通用的显式有限元程序
  - 求解过程无需迭代
- 适于求解高度不连续和高速的动力学问题
- 耦合场分析包括：
  - 热力耦合
  - 声固耦合
  - CEL

© Dassault Systèmes, 2008

简介

- **交互式后处理**
  - Abaqus/Viewer是Abaqus/CAE的后处理模块。
    - 在Abaqus/CAE中可用，或者可以作为单独的产品使用
    - 无论模型是否由Abaqus/CAE创建，都可以进行Abaqus结果的后处理
    - 可以为大的模型提供有效的后处理

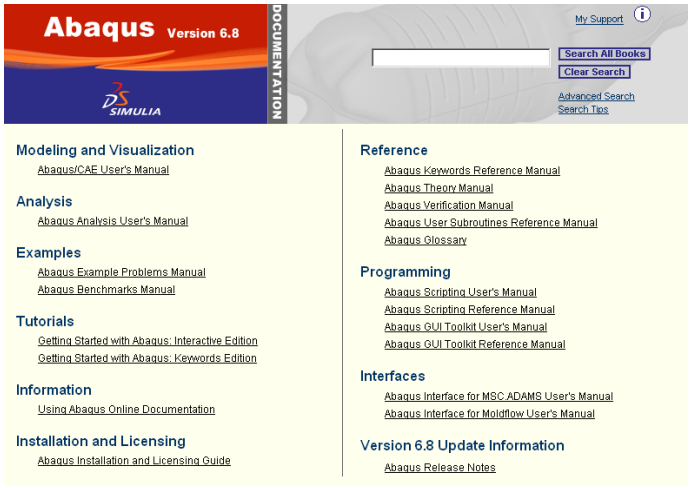


在Abaqus/Viewer中的铝材轮子与路边碰撞的云图绘制

© Dassault Systèmes, 2008

简介

- **文档**



© Dassault Systèmes, 2008

简介

- **主要参考资料**
  - 除非额外声明，所有的文档都可用，包括在线方式和打印文本
  - **Abaqus Analysis User's Manual**
  - **Abaqus/CAE User's Manual** (只有在线方式)
  - **Abaqus Example Problems Manual**
  - **Abaqus Benchmarks Manual** (只有在线方式)
  - **Abaqus Verification Manual** (只有在线方式)
  - **Abaqus Keywords Reference Manual**
  - **Abaqus User Subroutines Reference Manual** (只有在线方式)
  - **Abaqus Theory Manual** (只有在线方式)

© Dassault Systèmes, 2008

简介

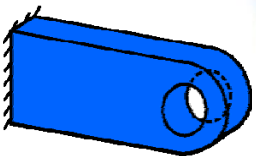
- **附加的参考资料**
- **Abaqus安装手册**
  - (安装指导)
- **Abaqus新版本发布信息**
  - (自从上一次发布后的变化)
- **不同主题的高级讲稿 (只有打印版本)**
- **入门手册**
  - *Getting Started with Abaqus: Interactive Edition*
  - *Getting Started with Abaqus: Keywords Edition* (只有在线版本)
- **设计手册**
  - (脚本和GUI工具箱手册)
- **SIMULIA主页**

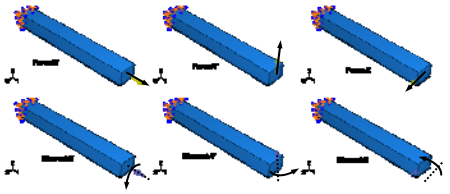
  `www.simulia.com`

© Dassault Systèmes, 2008

简介

- **本教程的内容**
  - 分析模块和交互式后处理简介。
  - 利用Abaqus解决不同结构分析问题的细节：
  - **线性静态分析**
  - **习题 1: 基本的输入和输出**—分析耳片的连接力
  - **习题 2: 悬臂梁的线性静态分析**—多工况

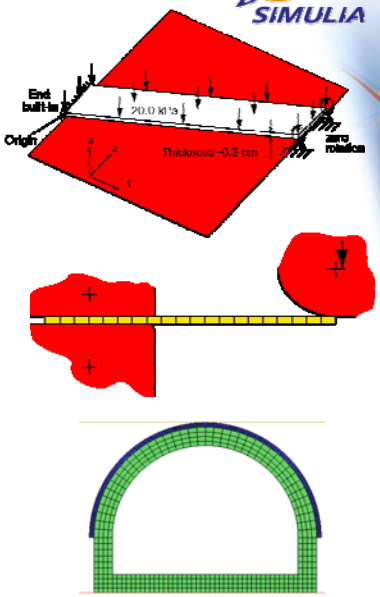




© Dassault Systèmes, 2008

简介

- **非线性有限元分析**
- **习题 3: 非线性静态**—斜板的大变形分析
- **利用多个分析步模拟**
- **多体接触**
- **习题 4: 接触**—平夹具的接触分析和刚体冲模的位移分析
- **习题5：密封圈接触**—橡胶密封圈的压缩分析

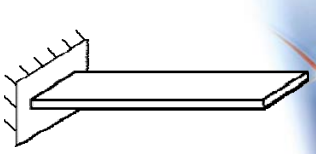


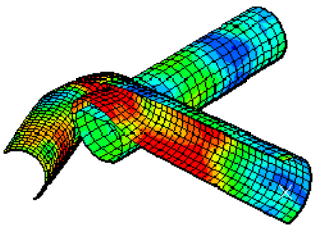
© Dassault Systèmes, 2008

简介

线性和非线性动态分析

- **习题6: 动力学**—悬臂梁的频率分析和显式、隐式自由振动分析
- **Abaqus/Explicit**中的高速动力学分析
- **习题7: 接触分析**
  Abaqus/Explicit— 管道拍打

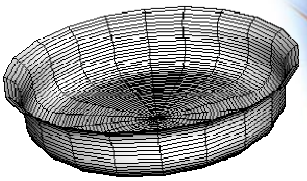




© Dassault Systèmes, 2008

简介

- **Abaqus/Standard和Abaqus/Explicit准静态问题的合并分析**
- 习题8（可选）：准静态和导入分析—罐底的深冲压，包括回弹
- 非结构应用—比如热传导、土壤固结和声学分析。
  - 所有的Abaqus分析技术使用同样的框架。
  - 本课中的知识将对其它应用的学习起到辅助作用。



© Dassault Systèmes, 2008

Abaqus模型的组件

© Dassault Systèmes, 2008

Abaqus模型的组件

- Abaqus的分析模块以批处理的方式运行。分析模块的基本输入为输入文件。在输入文件中包含单元、材料、过程和载荷库等选项。
- 这些选项可以以任意合理的方式组合，所以可以为多种问题建模。
- 输入文件被分为两个部分：模型数据和历程数据。

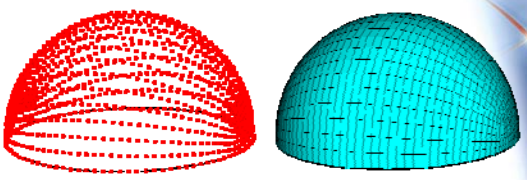
模型数据	几何选项—节点、单元
	材料选项
	其它模型选项
历程数据	过程选项
	载荷选项
	输出选项

© Dassault Systèmes, 2008

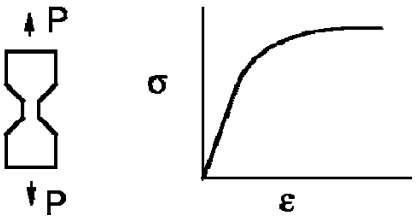
Abaqus模型的组件

- **模型数据—定义物理模型**

几何体离散模型—节点、单元



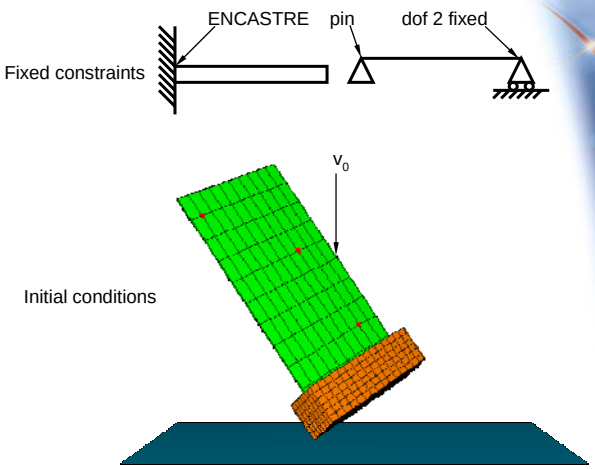
材料属性



© Dassault Systèmes, 2008

Components of an Abaqus Model

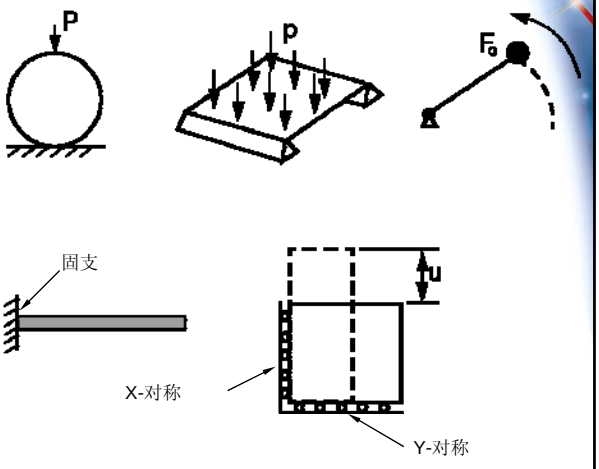
- **Model data**



© Dassault Systèmes, 2008

Abaqus模型的组件

- **历程数据—指定模型的动作**
  - 分析类型—静态、动态、土壤、热传导等等。
  - 载荷
  - 指定约束
  - 输出需求— 应力、应变、反力和接触压力等等。



© Dassault Systèmes, 2008

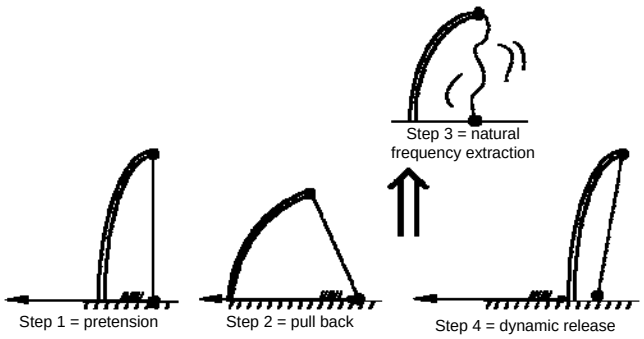
Abaqus模型的组件

- 分析历程通过划分成若干个分析步实现
  - 为方便起见，可以将分析历程进一步细分为分析步。
  - 不同的分析步包含不同的分析过程 — 例如，动态分析步后为静态分析步。
  - 通用分析步和线性摄动分析步的区别：
    - *General* steps(通用分析步)定义的是一个接一个顺序的分析流程
      - 前一个通用分析步的结束状态是后一个通用分析步的初始状态。通用分析步是历程相关的。
    - *Linear perturbation* steps(线性摄动分析步)分析"基础状态"基础上的线性响应，而基础状态是前溯最近的general step(通用分析步)。
    - 下一个分析步和*Linear perturbation steps*是没有关系的。

© Dassault Systèmes, 2008

Abaqus模型的组件

- **例题: 弓和箭模拟**



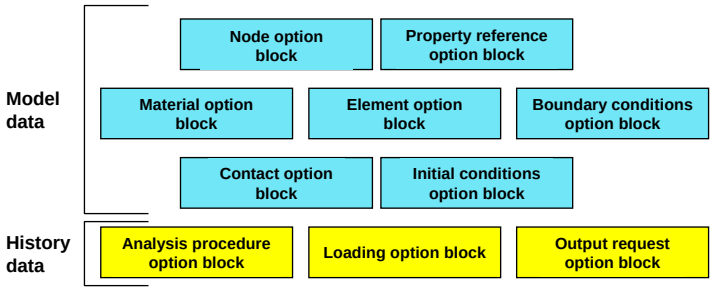
Step 1: 弓弦安装和预拉伸
Step 2: 弓弦后拉
Step 3: Linear perturbation step(线性摄动分析步)提取系统固有频率—对后续分析步没有影响
Step 4: 释放弓箭

© Dassault Systèmes, 2008

Abaqus输入文件的细节

SIMULIA

- **可选的数据块**
  - 在可选的数据块中定义了所有的数据，它描述了模型定义的具体方面，比如单元定义等等。同时，选项块数据创建了模型。



© Dassault Systèmes, 2008

Abaqus输入文件的细节

- 每个可选的数据块以关键字行（ keyword line）开头（第一个字符为*）。
- 如果需要，数据行(Data lines)将跟在关键字行(keyword line)之后。
- 注释行以**开头，可以放在数据文件的任意位置。
- 所有的输入行长度限制在256 个字符以内。
- 变量名限制在80个字符以内，且必须以字母开头。例如下面的变量名是合法的：

- **nodes_at_the_top_of_the_block_next_to_the_gasket**

© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **关键字行**
  - 以一个*号和关键字开头，后面跟着选项名。
  - 可以包含若干必需的和可选的参数以及它们的值，它们以逗号隔开。
    - 例子：材料选项数据块可以定义一组材料属性。

MATERIAL, NAME=material name*

keyword　parameter　parameter value

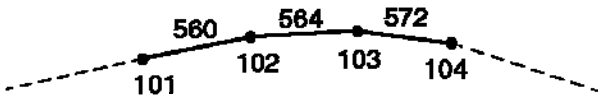
材料选项数据块的第一行

© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **数据行**
  - 为给定选项定义批量数据；例如，单元定义。
  - 关键字行可以包含许多与之相关的数据行。
    - 例如：可以利用单元选项数据块，通过指定单元类型、单元号和节点连接关系定义单元。

```
*ELEMENT, TYPE=B21   ← 关键字行
560, 101, 102
564, 102, 103        ← 数据行
572, 103, 104        ← 节点号（对于梁B21单元）
.                    ← 单元号
.
```

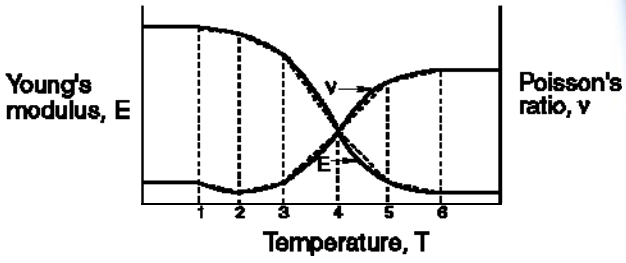


© Dassault Systèmes, 2008

Abaqus输入文件的细节

- 例如：弹性材料选项数据块定义了弹性模型的类型和弹性材料属性。

```
*ELASTIC, TYPE=ISOTROPIC   ← 关键字行
200.0E4, 0.3, 20.0
150.0E3, 0.35, 400.0       ← 数据行
.                          ← 温度
.                          ← 泊松比
                           ← 杨氏模量
```



© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **可选数据块的顺序**
  - 每个选项数据块要么属于模型数据(Model Data)，要么属于历程数据(History Data)，详见用户手册。
  - 在模型数据(model data)或历程数据(history data)内部，选项数据块的顺序和位置是任意的，除以下几个少数的特列：
    - 例如：
      - *HEADING 必须放在数据文件的第一行。
      - *ELASTIC，*DENSITY和 *PLASTIC是*MATERIAL的子选项。因此，它们必须直接跟在*MATERIAL数据块后。子选项没有自己的名字参考。
      - *STATIC，*DYNAMIC和*FREQUENCY必须在 *STEP数据块之后，用于指定分析步对应的分析类型。

© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **节点集和单元集**
  - 用于有效的交叉索引。
  - 允许一次引用整个集，而不是单个的引用节点或单元。

例子：节点集

```
*NODE, NSET=TOPNODES
101, 0.345, 0.679, 0.223
102, 0.331, 0.699, 0.234
.
.
*BOUNDARY, TYPE=DISPLACEMENT
TOPNODES, YSYMM
```

节点集**TOPNODES**饱含以下节点 101,102, ...

边界条件被应用到节点集**TOPNODES**中的所有节点。

© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **例子：单元集**

```
*ELEMENT, TYPE=B21, ELSET=SEATPOST
560, 101, 102
564, 102, 103
.
.
*BEAM SECTION, SECTION=PIPE, MATERIAL=STEEL,
ELSET=SEATPOST
0.12, 0.004
```

单元集**SEATPOST** 包括
单元560, 564, ...

梁的截面属性应用到单元集
SEATPOST中的所有单元。

壁厚

管子半径

© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **从其它文件引用数据**
  - Abaqus从另外的数据文件(*include* file)读数据与它利用输入文件直接读取数据文件在效果上是一致的。
  - 引用数据文件(include file)可以包含输入文件的任何内容，它本身也可以进一步引用其它数据文件。
  - 引用数据文件的格式必须与直接数据文件的格式必须保持一致。

© Dassault Systèmes, 2008

Abaqus输入文件的细节

- **例如：输入文件中引用包含文件**

```
*HEADING
*INCLUDE, INPUT=node_and_element_numbers.txt
.
.
```

- 包含文件中的内容 **node_and_element_numbers.txt**:

```
*NODE, NSET=TOPNODES
101, 0.345, 0.679, 0.223
102, 0.331, 0.699, 0.234
*ELEMENT, TYPE=B21, ELSET=SEATPOST
560, 101, 102
564, 102, 103
```

© Dassault Systèmes, 2008

Abaqus输入文件惯例

© Dassault Systèmes, 2008

Abaqus输入文件惯例

- **单位**
  - Abaqus并没有内部指定的单位规则
  - 用户需要自己去统一单位
    - 例如：
      - N, kg, m, s

        或
      - N, 10^3 kg, mm, s

        etc.

Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	lbf s^2 /in
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	$tonne/mm^3$	$slug/ft^3$	lbf s^2 /in^4

常用的单位系统

© Dassault Systèmes, 2008

Abaqus 输入文件惯例

- **时间度量**
  - Abaqus跟踪分析的总时间和每个分析步时间。
  - 对于一些分析过程，时间是有物理含义的，比如瞬态动力学问题。
  - 对于一些分析过程，时间是没有物理含义的。
    - 在率无关、静态过程中，时间是增量载荷的一种方便的、单调递增的一种度量。

© Dassault Systèmes, 2008

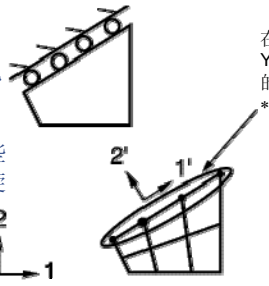
Abaqus 输入文件惯例

- **坐标系**
  - 对于初始节点位置的输入。
    - 默认坐标系为直角笛卡儿坐标系。
    - 可以利用*SYSTEM或*NODE，SYSTEM=[RECTANGULAR | CYLINDRICAL | SPHERICAL]指定其它的坐标系。
    - 因为局部坐标系下定义的节点坐标被自动转换到总体直角坐标系，所以局部坐标系的输入不影响载荷或输出。

© Dassault Systèmes, 2008

Abaqus 输入文件惯例

- 对于节点载荷、边界条件、初始条件和输出。
  - 默认为直角笛卡儿坐标系。
  - 使用*TRANSFORM选项指定其它的坐标系。
  - 在大位移分析问题中，这些坐标系的方向不随着材料旋转。
  - 例如：斜边上的边界条件。



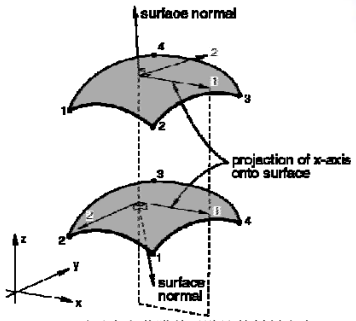
在这些节点上使用YSYMM对称边界条件的同时，使用*TRANSFORM

© Dassault Systèmes, 2008

Abaqus 输入文件惯例

- 关于材料点方向（与每个单元的材料或积分点相关的方向）。
  - 影响输入：各项异性材料方向。
  - 影响输出：应力/应变输出方向。
  - 默认值取决于单元类型。
  - 实体单元使用全局的直角笛卡儿坐标系。
  - 壳和薄膜单元使用全局笛卡儿坐标系到表面的投影。

实体单元的默认材料方向

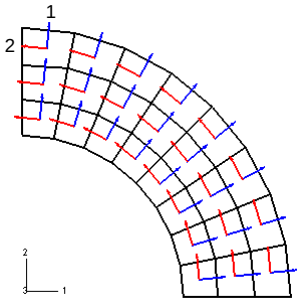


对于壳和薄膜单元默认的材料方向

© Dassault Systèmes, 2008

Abaqus 输入文件惯例

- 使用*ORIENTATION选项指定局部材料坐标系
- 在大位移分析中，局部材料坐标系的方向随着材料一起旋转

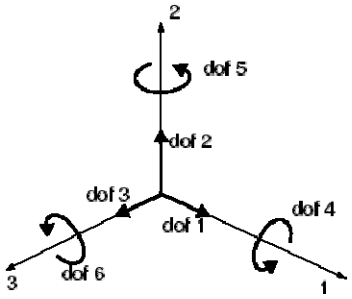


© Dassault Systèmes, 2008

Abaqus 输入文件惯例

- **自由度**
  - 节点的基本求解变量。
  - 有效的节点自由度取决于单元类型。
  - 自由度以数字为标号： 1=*x*-displacement, 2=*y*-displacement, ..., 11=temperature

1 Translation in the 1-direction.
2 Translation in the 2-direction.
3 Translation in the 3-direction.
4 Rotation about the 1-direction.
5 Rotation about the 2-direction.
6 Rotation about the 3-direction.



© Dassault Systèmes, 2008

Abaqus输出

© Dassault Systèmes, 2008

Abaqus输出

- **输出**
  - 四种输出类型：
    - 利用*OUTPUT选项和相关的子选项，可以将中性二进制输出内容写到输出数据库 (**.odb**)文件。
    - 打印输出可以写到打印输出 (**.dat**)文件。
      - 只在Abaqus/Standard中可用。
    - 为了进行重启动分析，可以利用*RESTART选项，将重启动输出写到重启动(**.res**)文件(第四讲将继续讨论)。
    - 可以写出结果(**.fil**)文件输出，用于第三方后处理软件。

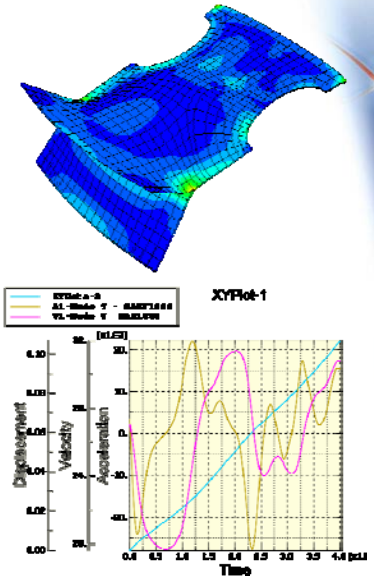
© Dassault Systèmes, 2008

Abaqus输出

- **输出到输出数据库文件**
  - The output database file is used by Abaqus/Viewer将使用输出数据库文件。在Python和C++中，保留了应用程序接口，可以用于外部后处理(比如，在Abaqus/Viewer中添加显示数据)
  - 两种类型的输出数据：场和历程数据
    - 场数据用于模型绘图 （变形图、云图等）和*X–Y* 绘图

      ***OUTPUT, FIELD**
    - 历程数据用于*X–Y* 绘图

      ***OUTPUT, HISTORY**



© Dassault Systèmes, 2008

Abaqus输出

- **控制输出的方法**
  - **控制场变量输出**
    - 通过控制增量步的数目 **(Abaqus/Standard only)**
      - `*OUTPUT, FIELD, FREQUENCY=`*n* — Every *n* increments
    - 通过控制间隔的数目
      - `*OUTPUT, FIELD, NUMBER INTERVAL=`*n* — At *n* evenly spaced time intervals
    - 通过控制时间间隔
      - `*OUTPUT, FIELD, TIME INTERVAL=`*x* — Every *x* units of time
    - 通过控制时间点
      - `*OUTPUT, FIELD, TIME POINTS=t_out` — At user-specified time points
      - `*TIME POINTS, name = t_out`

© Dassault Systèmes, 2008

Abaqus输出

- 历程变量的输出控制:
  - 通过控制增量步的数目
    - `*OUTPUT, HISTORY, FREQUENCY=`*n*
  - 通过控制间隔的数目**(Abaqus/Standard only)**
    - `*OUTPUT, HISTORY, NUMBER INTERVAL=`*n*
  - 通过控制时间间隔
    - `*OUTPUT, HISTORY, TIME INTERVAL=`*x*
  - 通过控制时间点**(Abaqus/Standard only)**
    - `*OUTPUT, HISTORY, TIME POINTS=t_out`
    - `*TIME POINTS, name=t_out`

© Dassault Systèmes, 2008

Abaqus输出

- **定义输出请求到ODB文件**
  - 如果在模型中没有定义输出请求，输出取决于环境文件的设置 (**abaqus_v6.env**)：
    - **odb_output_by_default=ON**: 预选择的输出被写入到ODB文件
      - 这个是默认的设置；输出的内容取决于分析类型
    - **odb_output_by_default=OFF**: 没有任何ODB产生
  - 如果使用下列的任何子选项，默认的输出将被覆盖:
    ***NODE OUTPUT**
    ***ELEMENT OUTPUT**
    ***ENERGY OUTPUT**
    ***CONTACT OUTPUT**
    ***INCREMENTATION OUTPUT** (Abaqus/Explicit only)

© Dassault Systèmes, 2008

Abaqus输出

- **预选择的ODB输出**
  - 预选择的输出取决于分析类型
  - 举例，对于静力分析：
    - 默认的场变量输出请求有：
      - Stresses – **S**
      - Total Strains – **E** (or logarithmic strain **LE** if NLGEOM is active)
      - Plastic Strains – **PE**, **PEEQ**, and **PEMAG**
      - Displacements and Rotations – **U**
      - Reaction Forces and Moments– **RF**
      - Concentrated (applied) Forces and Moments – **CF**
      - Contact Stresses – **CSTRESS**
      - Contact Displacements – **CDISP**
    - 默认的历史变量输出请求包含所有能量
  - 对于其他的分析类型，可查看Abaqus Analysis User's Manual

© Dassault Systèmes, 2008

Abaqus输出

- **输出到打印输出文件**
  - VARIABLE=PRESELECT选项是默认的场变量和历史变量输出的选项，这些选项允许将表格数据写到ASCII文件，该文件可以用文本编辑器读入。
  - 这些选项只有在Abaqus/Standard中可用。
  - 句法：

```
*NODE PRINT
*EL PRINT
*ENERGY PRINT
```

© Dassault Systèmes, 2008

Abaqus输出

- **输出到重启动文件**
  - 如果分析过早的停止，可以利用重启动数据继续之前的分析。重启动模拟可以起始于某个中间的点，不必重复任何计算。

```
*RESTART, WRITE
```

    - 该选项在第四讲中有进一步的讨论。
- **输出到结果文件**
  - 结果文件可以用于第三方后处理。

```
*FILE OUTPUT  (该选项只有在Abaqus/Explicit需要)
*NODE FILE
*EL FILE
*ENERGY FILE
```

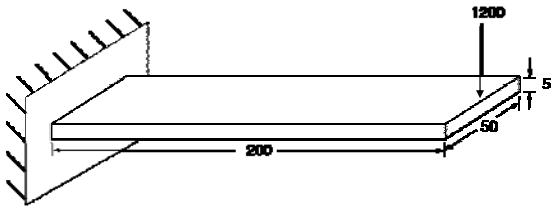
选择指定的输出变量

© Dassault Systèmes, 2008

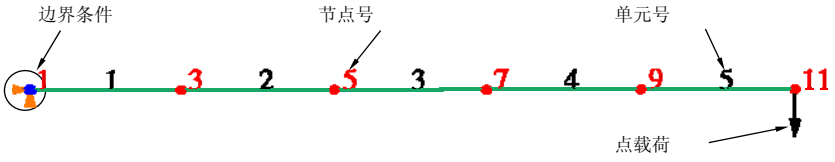
例子：悬臂梁模型

© Dassault Systèmes, 2008

例子：悬臂梁模型



使用梁单元的有限元模型



© Dassault Systèmes, 2008

例子：悬臂梁模型

- **带有注释的Abaqus输入文件**
  - 模型数据

该行将会出现在每个输出页上

```
*HEADING
CANTILEVER BEAM EXAMPLE
UNITS IN MM, N, MPa
*NODE
1,  0.0, 0.0
.
.
11, 200.0, 0.0
*NSET, NSET=END
11,
*ELEMENT, TYPE=B21, ELSET=BEAMS
1, 1, 3
.
.
5, 9, 11
*BEAM SECTION, SECTION=RECT, ELSET=BEAMS, MATERIAL=MAT1
50.0, 5.0
** Material from XXX testing lab
*MATERIAL, NAME=MAT1
*ELASTIC
2.0E5, 0.3
*BOUNDARY
1, ENCASTRE
```

选项块标题

节点选项块

节点集定义

单元选项块

属性引用选项块

注释行

弹性选项块

材料选项块

固定边界条件选项块

© Dassault Systèmes, 2008

例子：悬臂梁模型

- 历程数据

```
*STEP
APPLY POINT LOAD
*STATIC
*CLOAD
11, 2, -1200.0
*OUTPUT, FIELD, FREQUENCY=10
*ELEMENT OUTPUT, VARIABLE=PRESELECT
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=END
U
*EL PRINT, FREQUENCY=10
S, E
*NODE FILE, FREQUENCY=5
U
*END STEP
```

历程数据以第一个***STEP** 选项开始

历程数据***END STEP**选项结束。

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **使用集名引用属性**

```
*ELEMENT, TYPE=B21, ELSET=BEAMS
1, 1, 3
*BEAM SECTION, SECTION=RECT, ELSET=BEAMS, MATERIAL=MAT1
50.0, 5.0
*MATERIAL, NAME=MAT1
*ELASTIC
2.0E5, 0.3
```

  - *BEAM SECTION将单元集BEAMS和材料MAT1建立联系。
  - 选项还可以提供几何信息。上面的情况中，横截面类型为长方形**(RECT)**；宽度为50.0；高度为 5.0。
  - 模型中所有的单元必须有合适的属性引用。实体单元引用*SOLID SECTION；壳单元引用*SHELL SECTION；等等。

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **材料数据**

```
*MATERIAL, NAME=MAT1        <-- 材料名
*ELASTIC
2.0E5, 0.3                  <-- 泊松比
^
弹性模量
```

  - 定义各向同性线弹性材料
  - Abaqus认为*MATERIAL之后的选项是同一材料选项数据块的一部分，除非在*MATERIAL之后出现了非材料属性选项，比如*NODE选项。
  - *ELASTIC选项被称为子选项，它必须同*MATERIAL选项一同使用。

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **固支边界条件**

```
*BOUNDARY
1, 1, 6
```

自由度范围或BC类型（简支、固支、对称、反对称）
节点或节点集

  - 为有效自由度施加固定边界条件约束。
  - 预定义的非零边界条件只可以出现在历程数据中。
  - Abaqus只激活节点必要的自由度。因此，二维问题，只有1，2，6自由度为有效自由度，下面的输入数据是等效的：

```
1, 1, 2
1, 6, 6
```

或

```
1, 1, 6
```

批处理前处理器对于非活动的自由度将给出警告信息。

或

```
1, ENCASTRE
```

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **历程定义**

```
*STEP
APPLY POINT LOAD
*STATIC
```

以历程数据开头
该行出现在结果的每一行
指定静态分析过程

  - *STEP选项数据块可以包括任意长度的标题。
  - 在*STEP之后的第一个选项必须是过程定义。

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **载荷**
  - 在全局2方向定义集中载荷：

```
*CLOAD
11, 2, -1200.0
```

节点或节点集 ← 11
自由度 ← 2
大小 ← -1200.0

  - 有多种可用的分布载荷，包括表面压力、体力、集中力和柯氏力等等。

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **输出需求**

```
*OUTPUT, FIELD, FREQUENCY=10
*ELEMENT OUTPUT, VARIABLE=PRESELECT
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=END
U,
```

← 输出到输出数据库文件

  - 上面语句中， 要求输出预先选定的、常用的场变量输出集。
  - 还为前面定义的节点集END定义了位移的历程输出。
    - 因为历程输出需要相对高的频率，所以集应该尽量小。
  - 每个输出需求包括FREQUENCY参数。如果分析需要多个增量，FREQENCY参数指定了结果的写出频率。

© Dassault Systèmes, 2008

例子：悬臂梁模型

```
*EL PRINT, FREQUENCY=10
S, E
*NODE FILE, FREQUENCY=5
U
```

打印输出到数据文件

输出到结果文件

- 利用*EL PRINT选项，将表格输出打印到数据 (**.dat**)文件，用于直观的检查。
  - 上面语句中，要求输出应力(S)和应变 (E)组件。
- 使用*NODE FILE选项，将二进制输出写到Abaqus结果文件，用户其它后处理器的后处理。
  - 上面语句中，要求输出位移(U)组件。

© Dassault Systèmes, 2008

例子：悬臂梁模型

- **分析步结束**

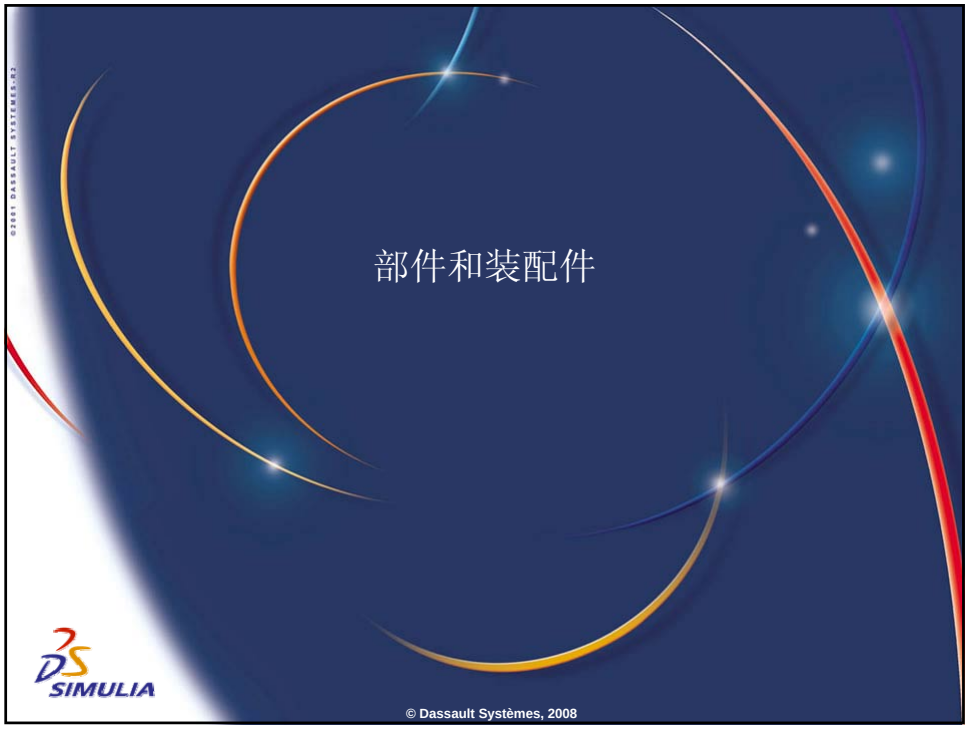
```
*END STEP
```

结束分析步

- 每个分析步以*END STEP选项结束。
- 对于最后的分析步，最后的选项是输入文件中的*END STEP选项。

Video Clip

© Dassault Systèmes, 2008



部件和装配件

SIMULIA

- **输入文件可以以部件、部件实例和装配件定义。**
  - 在输入文件中Abaqus模型同Abaqus/CAE中使用同样的术语。
  - Abaqus以后的改进将进一步利用部件和装配件的优势。
    - 为引用模型的不同区域提供一种内在的方法。用户不再需要定义分开的集。
    - 允许重新使用部件定义，对于创建大型、复杂的模型将非常有用。
    - 标签—节点和单元编号、集名—只需在它们定义的等级内唯一。

© Dassault Systèmes, 2008

部件和装配件

- **定义部件**
  - 使用*PART和*END PART选项定义部件。它们必须出现在装配件定义之外。每个部件必须有唯一的名字。
- **定义部件实例**
  - 在装配件定义范围内，使用*INSTANCE和*END INSTANCE选项定义部件实例。每个部件实例必须有唯一的名字。
- **定义装配件**
  - 使用*ASSEMBLY和*END ASSEMBLY选项定义装配件。在一个模型中只能定义一个装配件。
  - 附加的集和表面定义，连同约束和刚体定义，必须出现在装配件定义范围之内。

© Dassault Systèmes, 2008

部件和装配件

- **装配件输入文件例子**

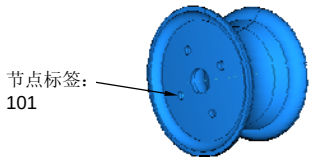
```
*HEADING
...
*PART, NAME=Tire
  Node, element, section, set, and surface definitions
*END PART
*PART, NAME=Rim
  Node, element, section, set, and surface definitions
*END PART
...
*ASSEMBLY, NAME=Tire_and_rim
   *INSTANCE, NAME=I_Tire, PART=Tire
     <positioning data>
     set and surface definitions (optional)
   *END INSTANCE
   *INSTANCE, NAME=I_Rim, PART=Rim
     <positioning data>
     set and surface definitions (optional)
   *END INSTANCE
   Additional set and surface definitions
   *NSET, NSET=Output
    I_Tire.514, I_Tire.520
    I_Rim.101, I_Rim.102
*END ASSEMBLY
```

```
...
*MATERIAL, NAME=Rubber
*AMPLITUDE
*INITIAL CONDITIONS
*PHYSICAL CONSTANTS
...
*STEP
*STATIC
   *BOUNDARY
    Tire_and_rim.I_Rim.101, 1, 3, 0.0
   *CLOAD
    Tire_and_rim.I_Tire.514, 2, 1000.0
   *OUTPUT, HISTORY, FREQUENCY=10
   *NODE OUTPUT, NSET=Tire_and_rim.Output
    RF, CF
*END STEP
```

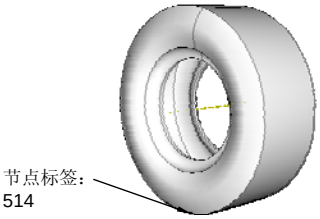
© Dassault Systèmes, 2008

部件和装配件

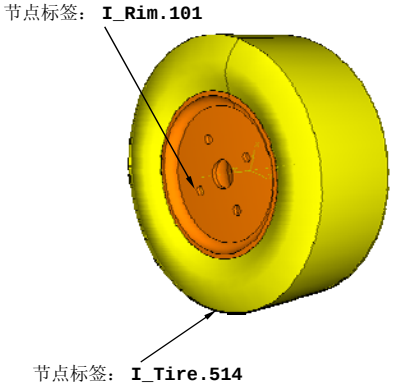
- **部件和装配件的节点标签**



部件：轮缘



部件：轮胎



装配件： **Tire_and_rim**

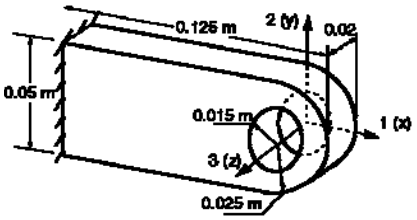
© Dassault Systèmes, 2008

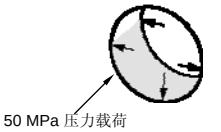
习题1: 基本输入和输出

© Dassault Systèmes, 2008

习题1: 基本输入和输出

- **习题任务**
  1. 使用Abaqus的一些工具程序。
  2. 打开在线文档，并查找有用的信息。
  3. 使用在线文档确定不同选项的句法。
  4. 为已有的输入文件添加细节，完成连接耳片模型。





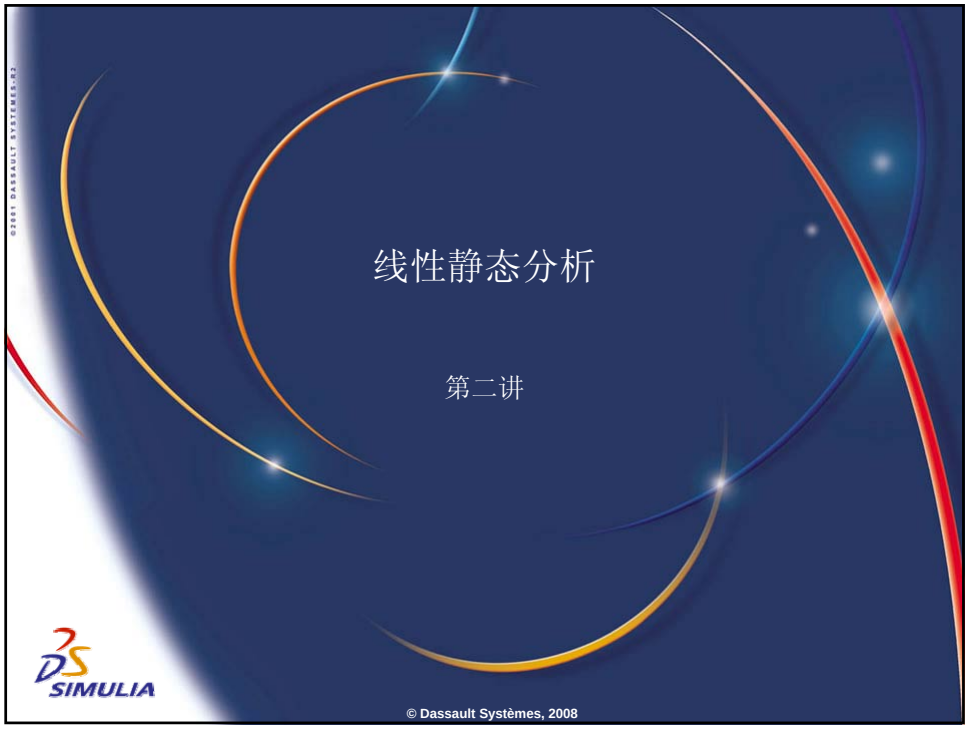
© Dassault Systèmes, 2008

习题1: 基本输入和输出

5. 以不同的方式提交分析作业（只是数据检查、完全分析、交互式和批处理提交作业）。
6. 使用Abaqus/Viewer察看结果。
7. 熟悉打印输出文件的内容。
8. 修改模型，理解结果的变化。

Video Clip

© Dassault Systèmes, 2008


线性静态分析
第二讲
SIMULIA
© Dassault Systèmes, 2008

概述
SIMULIA
• 线性和非线性过程
• 线性静态分析和多工况
• 多工况应用
• 例子
Introduction to Abaqus/Standard and Abaqus/Explicit
© Dassault Systèmes, 2008

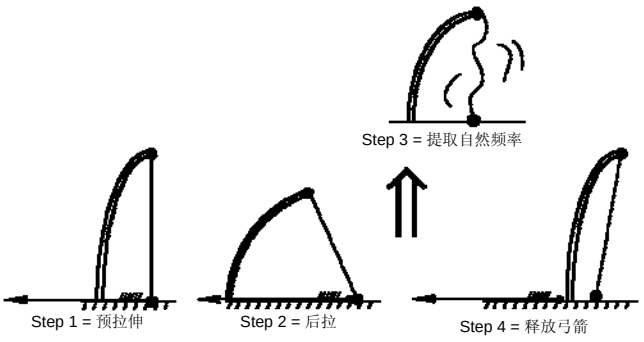
线性和非线性过程

© Dassault Systèmes, 2008

线性和非线性过程

- **Abaqus/Standard中有两种类型的分析步：**
  - **通用分析步：**响应可以是线性或非线性的。
  - **线性摄动分析步：**响应只能是线性的。

Video Clip



© Dassault Systèmes, 2008

线性和非线性过程

- 线性分析是基状态（初始构型或当前构型）的线性摄动。
- 基状态之前的响应可以是非线性的。
  - 但是，模型必须是静态平衡的(在进行线性摄动分析之前，只有先利用*STATIC分析步达到静力平衡，才可以应用*DYNAMIC选项）。
- 在摄动分析步之后，可以继续进行非线性分析步。
- 在Abaqus/Explicit中，只有通用分析步。

© Dassault Systèmes, 2008

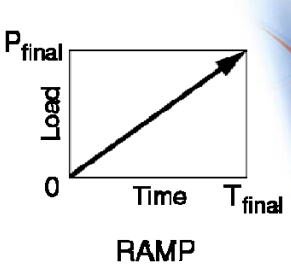
线性和非线性过程

- 在Abaqus/Standard中，纯的线性摄动过程包括：
  - *BUCKLE (特征值屈曲分析)
  - *FREQUENCY (特征值提取)
  - *MODAL DYNAMIC (瞬态线性动力学分析)
  - *RANDOM RESPONSE (随机激励的结构响应)
  - *RESPONSE SPECTRUM (动载荷的峰值线性响应)
  - *STEADY STATE DYNAMICS (谐波激励下的稳态动力学响应)
- *STATIC分析步可以是线性摄动或通用过程。
  - 下面有详细的讨论。

© Dassault Systèmes, 2008

线性和非线性过程

- **默认的幅值引用**
  - 不同的分析过程有不同的默认值
  - 对于没有自然时间度量的过程
    AMPLITUDE=RAMP



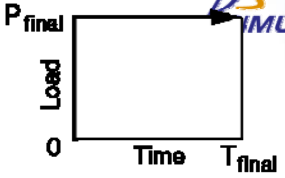
RAMP

*STATIC
*HEAT TRANSFER, STEADY STATE
*COUPLED TEMPERATURE-DISPLACEMENT, STEADY STATE
*SOILS, STEADY STATE
*COUPLED THERMAL-ELECTRICAL, STEADY STATE
*STEADY STATE TRANSPORT

© Dassault Systèmes, 2008

线性和非线性过程

- 对于有自然时间度量的过程
  AMPLITUDE=STEP



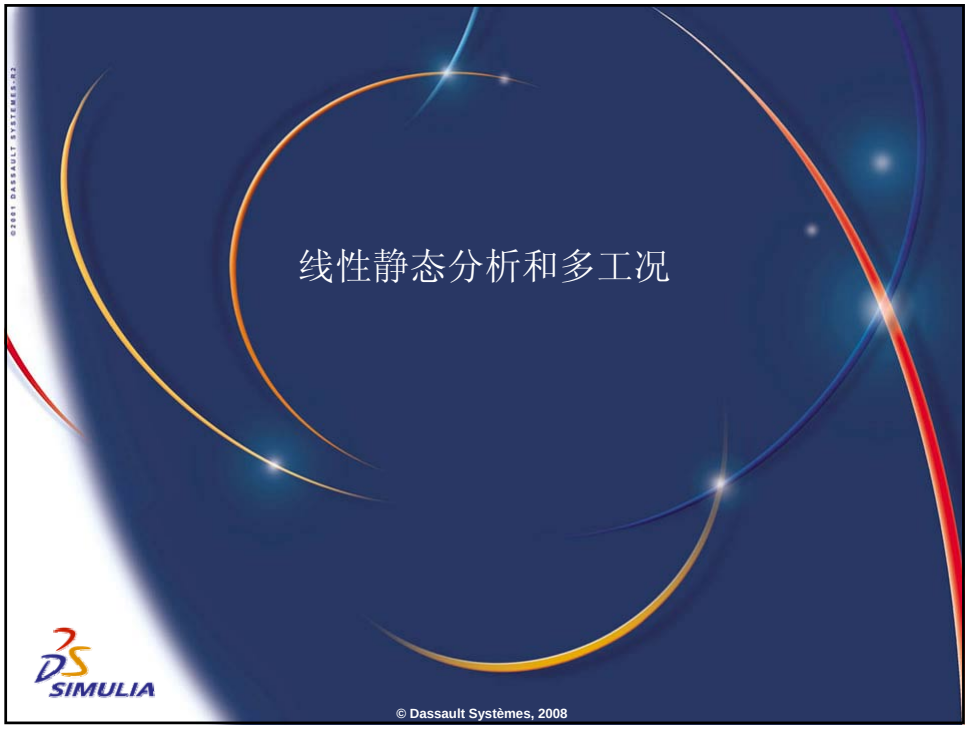
STEP

*DYNAMIC
*VISCO
*HEAT TRANSFER (瞬态)
*COUPLED TEMPERATURE-DISPLACEMENT (瞬态)
*DYNAMIC TEMPERATURE-DISPLACEMENT, EXPLICIT
*COUPLED THERMAL-ELECTRICAL (瞬态)
*SOILS, CONSOLIDATION
*STEADY STATE DYNAMICS
*RANDOM RESPONSE
*MODAL DYNAMIC

注释：频域过程—幅值引用定义了载荷和频率之间的关系。

- 在显式动力学过程(*DYNAMIC, EXPLICIT)中，预定的非零边界条件必须引用幅值选项。

© Dassault Systèmes, 2008



线性静态分析和多工况

SIMULIA

- **静态分析是唯一既可以包含通用分析步，又可以包含摄动分析步的过程：**

```
*STEP
*STATIC
```

或

```
*STEP, PERTURBATION
*STATIC
```

  - 当研究结构在不同载荷集和边界条件集的线性响应时，在单个摄动分析步中使用多工况分析比多摄动步分析更方便。
    - 载荷工况包括载荷和边界条件。

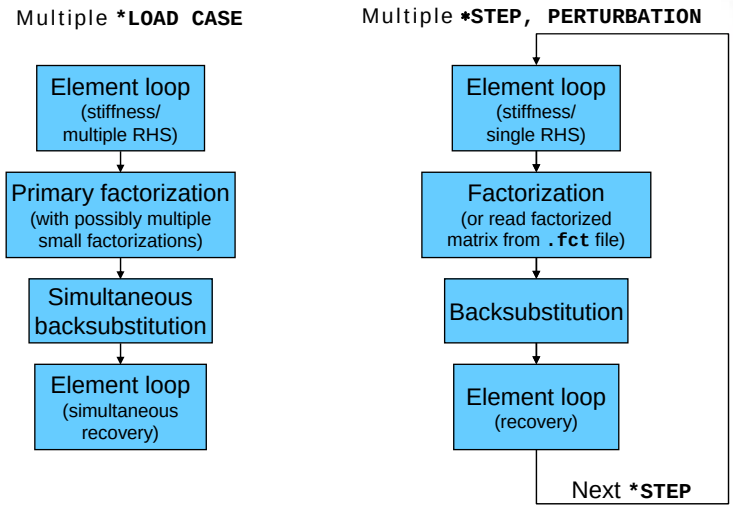
© Dassault Systèmes, 2008

线性静态分析和多工况

- 当被分析的组件承受多种不同类型的载荷时，多工况分析将非常有用。
  - 在许多工业领域非常常见。
  - 例如，飞机在起飞、爬升、巡航、俯冲、着陆和滑行过程中经历的不同载荷。
    - 每个工况独立施加。
    - 如果在载荷历程中的所有阶段假定结构的刚度为常数（线性假设），多工况分析可以辅助确定载荷包络线。

© Dassault Systèmes, 2008

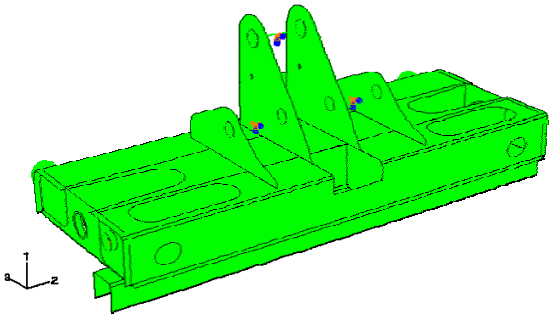
线性静态分析和多工况



© Dassault Systèmes, 2008

线性静态分析和多工况

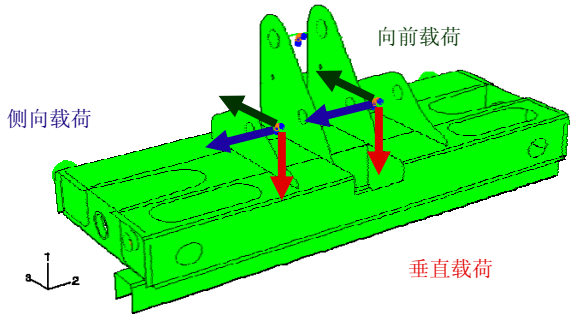
- **例子：农用工具**
  - 下图是农用工具，它通过三点挂钩连接到拖拉机，并拖在拖拉机之后。
  - 挂钩的目的是用于向工具传递拖动载荷，但是不允许工具浮动，或工具的运动独立于拖拉机。



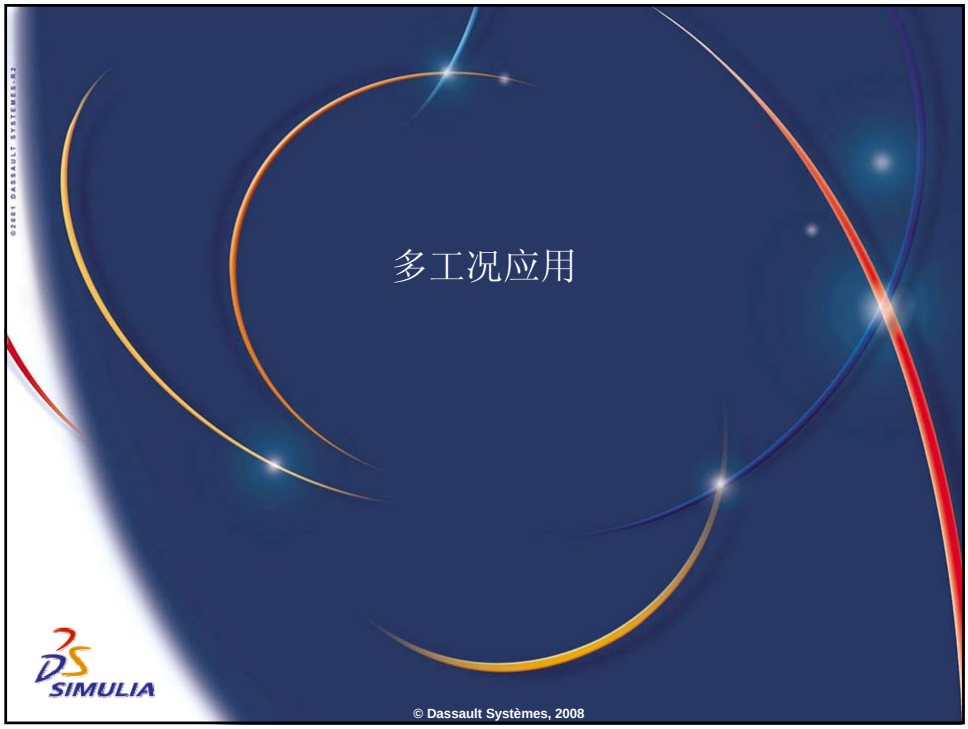
© Dassault Systèmes, 2008

线性静态分析和多工况

- **三个工况**
  - 连接件非常柔软，工具上的载荷没有明确指出，但是可能是多种不同类型载荷的组合。



© Dassault Systèmes, 2008



多工况应用

SIMULIA

- **下面的过程支持多工况分析：**

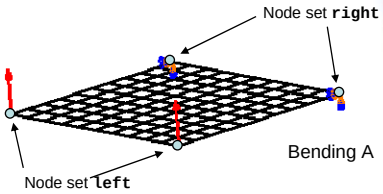
```
*STEP, PERTURBATION
*STATIC

*STEADY STATE DYNAMICS, DIRECT
```

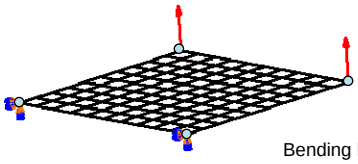
- **工况中可以包含下面的载荷类型：**
  - 集中力和分布力
  - 边界条件(不同的工况可以有不同的边界条件)
  - 惯性解除

© Dassault Systèmes, 2008

多工况应用

```
...
*Step, perturbation
*Static
*Load Case, name="Bending A"
*Boundary
 right, 1, 6
*Cload
 left, 3, 1.
*End Load Case
*Load Case, name="Bending B"
*Boundary
 left, 1, 6
*Cload
 right, 3, 1.
*End Load Case
*End Step
```





© Dassault Systèmes, 2008

多工况应用

- **附加规则**
  - 工况名(***LOAD CASE, name=***name*)必须唯一
  - 在工况之外指定的载荷选项对所有工况都有效
  - 基状态边界条件传播到所有的工况
  - 使用**OP=NEW**的规则
    - 如果在工况步内的任意位置使用该选项，则必须在相应的分析步的每个工况中都要使用该选项
    - 如果在工况步中使用任何的*BOUNDARY，则所有从历史步中传播的边界条件将被删除
  - 不同*LOAD CASE选项定义的载荷或者边界条件不会相互影响

© Dassault Systèmes, 2008

多工况应用

- **为不同的工况改变边界条件**
  - 当只是改变边界条件的大小时，对求解性能不会造成影响。
  - 从工况到工况之间，尽量减小边界条件位置变化的数量。
    - 如果改变位置的边界条件的分布和数量变化很大，多工况分析可能会明显的慢于等效的多步骤分析（与问题非常相关）。
    - 如果对此怀疑，运行datacheck分析（多步骤 vs 多工况），比较求解器中的求解器信息数据（.dat）文件（比如，内存需求，浮点操作的数量，等等）。

© Dassault Systèmes, 2008

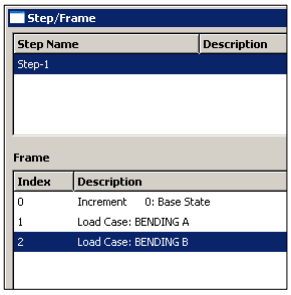
多工况应用

- **问题大小**
  - 合并的自由度个数和工况个数决定了“问题大小”。
  - 多工况分析可能需要更多的：
    - 内存（相对于等效的多步骤分析）。在回代的过程中，方程的右手边必须保留在内存中。
    - 硬盘空间（单元和节点数据库）。
  - 在可能的情况下，设置：
    - **`standard_memory`** 为最小的 I/O (请察看数据文件)。
    - **`standard_memory_policy`** 为 **`MAXIMUM`**。
  - 如果必要，将不同的工况散布到几个分析步中，用以减小每个分析步中内存和硬盘的使用量。
    - 最坏情况：回到多摄动步分析（在这里，还可以比较数据文件中的求解器信息）。

© Dassault Systèmes, 2008

多工况应用

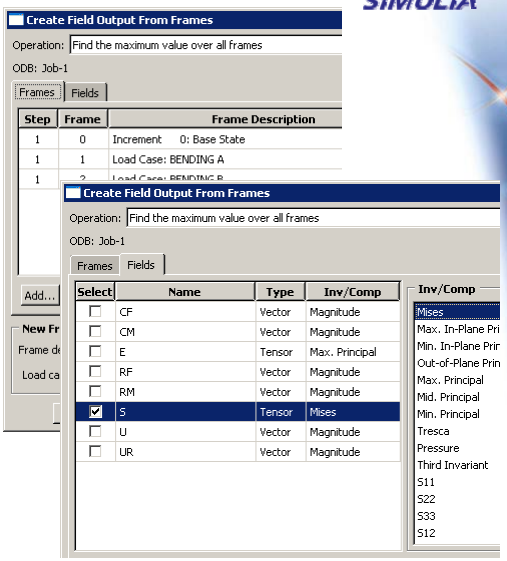
- **输出**
  - 按每个分析步输出(不是每个工况)
  - 可以输出到输出数据库(**.odb**)和打印输出文件(**.dat**)
  - 对于输出数据库文件：
    - 工况的所有输出变量被映射到帧
      - 同增量映射到帧类似
    - 帧包含工况名
    - 只有场输出(没有历程输出)



© Dassault Systèmes, 2008

多工况应用

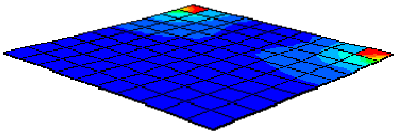
- **Abaqus/Viewer的后处理**
  - 支持在整个帧中进行操作
  - 对于选定的帧，可以创建：
    - 线性合并(比如，工况的线性组合)
    - 最大/最小包络线(比如，在所有工况中找出最大应力)



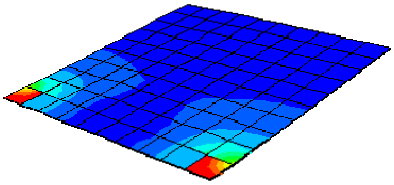
© Dassault Systèmes, 2008

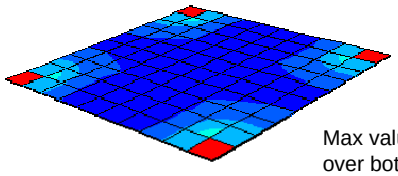
多工况应用

Mises stress: Bending A



Mises stress: Bending B





Max value of Mises stress over both frames

© Dassault Systèmes, 2008

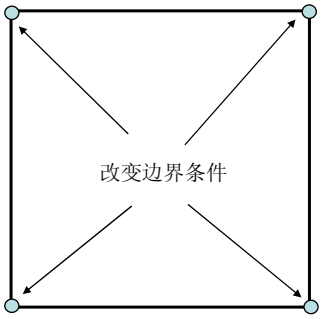
例子

© Dassault Systèmes, 2008

例子

- **方板基准程序**

模型	# 节点/边	# 变量 (# DOF)
1	101	61206
2	201	242406
3	501	1506006
4	751	3384006

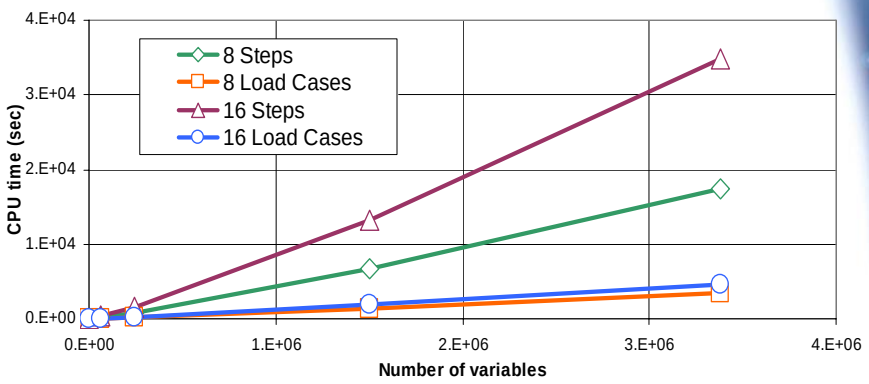


  - 工况的个数： 8 和 16
  - **`*Static, perturbation`**
  - 改变角点处边界条件的位置
  - 默认输出

© Dassault Systèmes, 2008

例子

- **执行结果：全部CPU时间**



© Dassault Systèmes, 2008

例子

- **执行： 751 × 751 模型的细节**

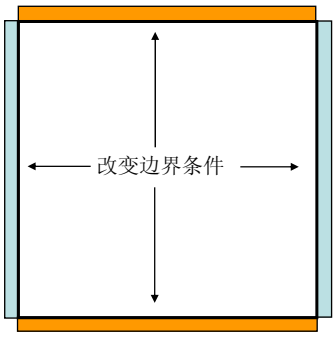
相对 CPU 时间 – 3.4 M variable case

	8 Steps/8 Load cases	16 Steps/16 Load cases
Solver	7.52	14.3
Total	5.04	7.48

© Dassault Systèmes, 2008

例子

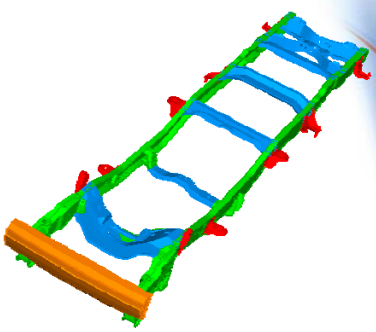
- **修改 501 × 501 模型**
  - 8 工况
  - 在每个工况中改变对边的边界条件
  - 相对全部 CPU 时间： ~0.153
    (多工况 ~6.6× 慢!)
  - 观察边界条件数量和位置的变化！



© Dassault Systèmes, 2008

例子

- **底盘-车架的机动性分析：**
  - 变量的个数： 534,000
  - 方程的个数： 483,000
  - 工况的个数： 60
  - **`*Steady-state dynamics, direct`**
    (10个频率点)
  - 输出： U (输出数据库)



CPU 时间 (sec)

	60 steps (projected based on 1 step)	60 load cases
Solver	1290 × 60 = 77,400	1990 **(39× faster)**
Total	1965 × 60 = 117,600	11,600 **(10× faster)**

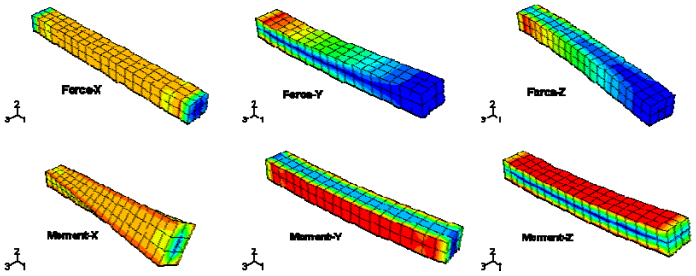
© Dassault Systèmes, 2008

习题2: 悬臂梁的线性静态分析

© Dassault Systèmes, 2008

习题2: 悬臂梁的线性静态分析

- **习题任务**
  1. 使用多工况分析的办法，分析梁的弯曲响应。
  2. 比较使用单个分析步的多工况分析和多个分析步的单工况分析，得到的求解和分析时间的不同。



© Dassault Systèmes, 2008

概述

- 非线性结构力学
- 求解平衡方程
- 非线性输入文件
- 非线性悬臂梁分析的输出
- 习题3：非线性静力学

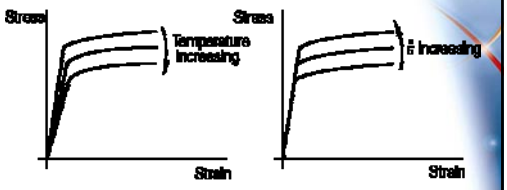
© Dassault Systèmes, 2008

非线性结构力学

© Dassault Systèmes, 2008

非线性结构力学

- **非线性的来源**
  - **材料** 非线性：
    - 非线性弹性塑性
    - 材料损伤
    - 失效机制
    - 等等。
  - 注意：如果预定义了温度或场变量，则材料与温度或场变量的相关性不引入非线性。

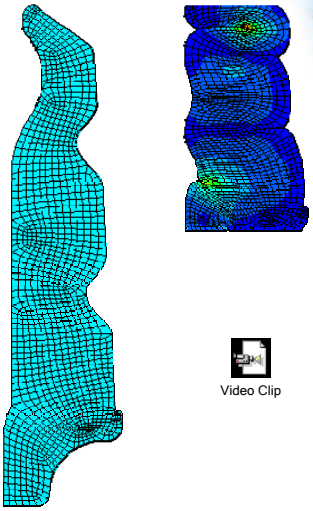


一些材料非线性的例子

© Dassault Systèmes, 2008

非线性结构力学

- **边界条件** 非线性：
  - 接触问题
    - 在分析过程中边界条件变化。
    - 严重不连续形式的非线性。



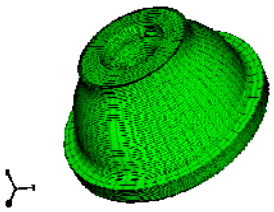
自接触的例子：例子手册1.1.16，
压缩减震器

Video Clip

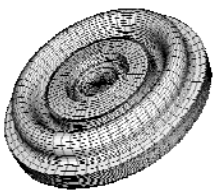
© Dassault Systèmes, 2008

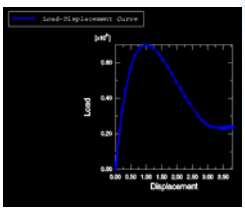
非线性结构力学

- **几何** 非线性：
  - 大绕度和大变形
  - 大旋转
  - 结构不稳定（屈曲）
  - 预载荷效应



Video Clip





几何非线性例子：聚合物键盘罩

© Dassault Systèmes, 2008

求解平衡方程

© Dassault Systèmes, 2008

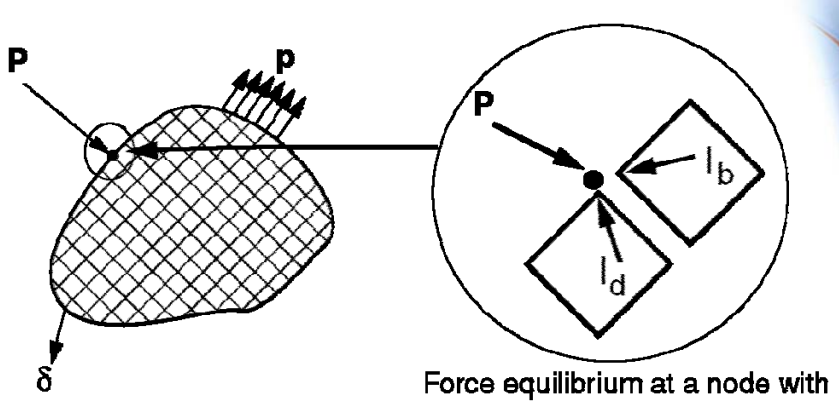
求解平衡方程

- 典型的非线性问题具有所有三种形式的非线性。
  - 在方程中必须包括非线性项。
  - 一般的，每个自由度的非线性方程是耦合的。
- 静态平衡的基本表达式为：由单元应力引起的加在节点上的内力I，与外力 P，必须平衡，即：

$$P(u) - I(u) = 0 \qquad \text{(Eq. 3.1)}$$

© Dassault Systèmes, 2008

求解平衡方程



Force equilibrium at a node with internal and external forces

© Dassault Systèmes, 2008

求解平衡方程

- 为了求解非线性平衡问题，在Abaqus/Standard中使用基于牛顿-拉普森技术的增量迭代法。
- 假定前一步载荷增量的解，u_0，为已知。
- 假定在第i 次迭代之后，得到近似解u_i 。设c_{i+1}为离散平衡方程的精确解和当前解之差（ Equation 3.1 ），所以有：

$$P(u_i + c_{i+1}) - I(u_i + c_{i+1}) = 0. \qquad \text{(Eq. 3.2)}$$

- 将方程3.2的左手边在近似解u_i 附近，以泰勒级数方式展开，可以得到

$$P(u_i) - I(u_i) + \left(\frac{\partial P(u_i)}{\partial u} - \frac{\partial I(u_i)}{\partial u} \right) c_{i+1} + ... = 0. \qquad \text{(Eq. 3.3)}$$

© Dassault Systèmes, 2008

求解平衡方程

- 略去高阶项，方程可以写为

$$K_i c_{i+1} = P(u_i) - I(u_i),$$

其中 $K_i = \frac{\partial I(u_i)}{\partial u} - \frac{\partial P(u_i)}{\partial u}$ 为切线刚度。

- 解的下一次近似为

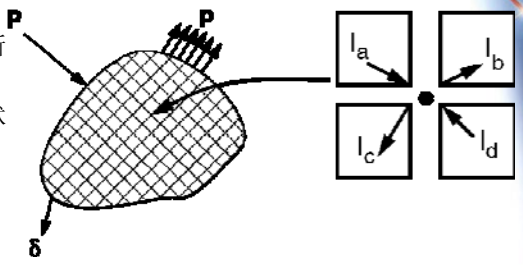
$$u_{i+1} = u_i + c_{i+1}.$$

- 注意：如果载荷与位移相关（比如，旋转的表面压力），刚度矩阵中包含载荷刚度的贡献。

© Dassault Systèmes, 2008

求解平衡方程

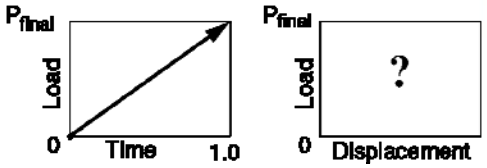
- **网格中的静力平衡**
  1. 施加规定的“载荷增量”。
  2. 迭代，直到每个节点的所有节点力的和非常小。
  3. 满足平衡后，更新模型状态。
  4. 回到第1步，施加下一个载荷增量。



© Dassault Systèmes, 2008

求解平衡方程

- **单自由度例子**
  - 非线性弹簧
  - 求解$u(P)$ *或* $P(u)$—典型的，在分析步中，载荷从0递增到P_{FINAL}。
  - 时间经常从0变到1。

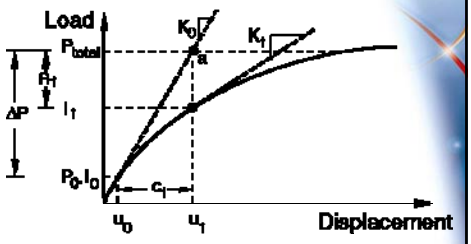


© Dassault Systèmes, 2008

求解平衡方程

- **牛顿-拉普森求解技术**
- **第一次迭代 (i=1)**
  - 假定前面收敛增量步的解u_0，P_0，为已知的。
  - 在当前增量步中，将一个小的增量ΔP，载荷施加到结构上。
  - Abaqus基于u_0处的切线刚度K_0确定位移修正c_1；前一增量步结束时，总载荷P_{TOTAL}和内力间的关系为：

$$I_0 : K_0 c_1 = P_{TOTAL} - I_0.$$



© Dassault Systèmes, 2008

求解平衡方程

- Abaqus更新模型的状态为u_1，形成K_1并计算I_1。
- 总载荷P_{TOTAL}与内力I_1的差称为残差，R_1: $R_1 = P_{TOTAL} - I_1$.
- 如果R_1在模型的每个自由度上都非常小 (在容差范围之内)，结构就是平衡的。
  - 默认的容差R_1必须小于结构对时间平均力的0.5%。
  - Abaqus自动计算时间平均力。
- 如果迭代不能得到收敛的解，Abaqus执行另外的迭代，以找到收敛的解。

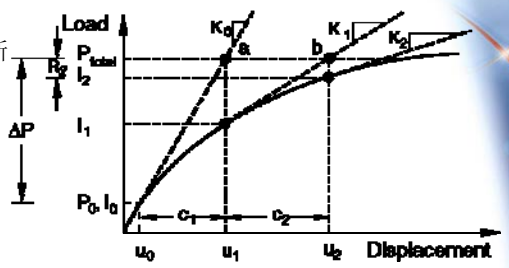
© Dassault Systèmes, 2008

求解平衡方程

- **第二次迭代 (i=2)**
  - 基于更新的刚度K_1，计算新的位移纠正c_2，并且

$$I_1 : K_1 c_2 = P_{TOTAL} - I_1.$$

  - 把新的残差R_2与容差进行比较，察看在u_2处是否得到收敛解，。



© Dassault Systèmes, 2008

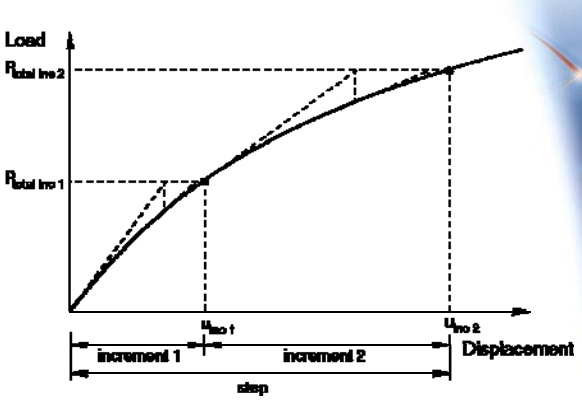
求解平衡方程

- 该过程将一直重复，直到力的残差在允许的容差之内。每次迭代i需要：
  1. 形成切线刚度K_i。
  2. 求解系统方程组，得到位移修正c_{i+1}。
     - 修正位移的估计值： $u_{i+1} = u_i + c_{i+1}$。
  3. 基于u_{i+1} 计算内力向量I_{i+1}。
  4. 进行平衡收敛判断：
     - 是否R_{i+1} 在容差之内?
     - 是否 $c_{i+1} << \sum_{j=1}^{\#iter} c_j$?

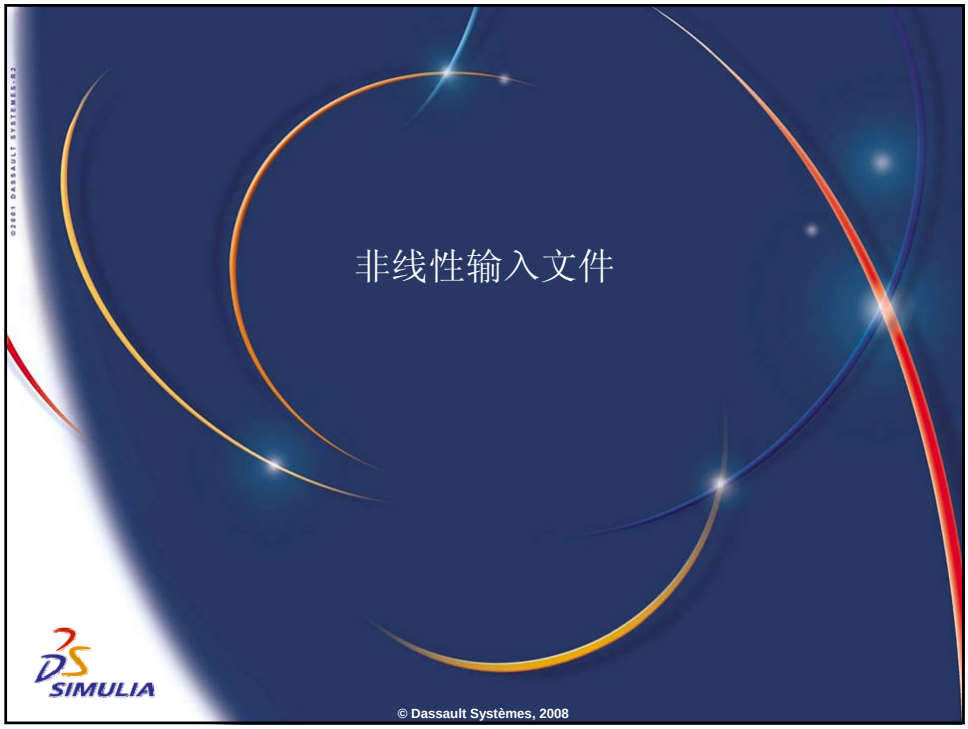
© Dassault Systèmes, 2008

求解平衡方程

- 一般的，每个分析步(*STEP)需要几个增量步。



© Dassault Systèmes, 2008

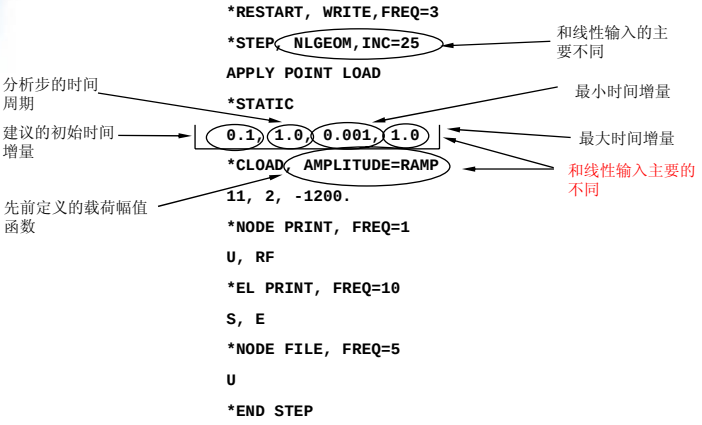


非线性输入文件

```
*HEADING
 CANTILEVER BEAM EXAMPLE--LARGE DISPLACEMENT
*NODE
1,  0., 0.
11, 200., 0.
*NGEN
1, 11, 1
*ELEMENT, TYPE=B21
1, 1, 3
*ELGEN, ELSET=BEAMS
1, 5, 2, 1
*BEAM SECTION, SECTION=RECT, ELSET=BEAMS, MATERIAL=MAT1
50., 5.
*MATERIAL, NAME=MAT1
*ELASTIC
2.E5, .3
*BOUNDARY
1, 1, 6
*AMPLITUDE, NAME=RAMP
0.0, 0.0, 0.5, 0.3, 1.0, 1.0
```


© Dassault Systèmes, 2008

非线性输入文件



© Dassault Systèmes, 2008

非线性输入文件

- **分析步和过程输入**

  ***STEP, NLGEOM, INC=25**

  - NLGEOM: 包括所有由以下原因引起的几何非线性效应：
    - 大挠度、大旋转、大变形。
    - 预载荷（初始应力）。
    - 载荷刚度。
  - 如果上面列出的项不重要，应用NLGEOM选项得到的结果同没有应用NLGEOM选项得到的结果类似，但是求解的费用更高。
  - INC=25: 在本例中允许的最大增量为25：
    - 如果在施加全部载荷之前达到了最大增量数量，程序将会中止。
    - 保证程序的运行时间不会太长—用户可以重新启动分析。
    - 默认值为100。

© Dassault Systèmes, 2008

非线性输入文件

- 对于所有的瞬态过程，都有类似的时间增量，它们是
  - *STATIC
  - *DYNAMIC
  - *HEAT TRANSFER
  - *VISCO
  - *COUPLED TEMPERATURE-DISPLACEMENT
  - *SOILS
  - *MODAL DYNAMIC (只允许固定时间增量)
  - *COUPLED THERMAL-ELECTRIC
- 物理（真实）时间或名义时间的度量取决于过程和时间相关过程或率相关行为。

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

- **状态(.sta) 文件**
  - 总结分析的过程—显示计算过程中使用的自动时间增量。
  - 在作业运行的同时，可以检查状态文件。
  - 在每个成功的增量之后，记录一行。

```
SUMMARY OF JOB INFORMATION:
STEP INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
             DISCON ITERS ITERS  TIME/    TIME/LPF    TIME/LPF    MONITOR RIKS
             ITERS               FREQ

  1    1   1     0     3     3  0.100      0.100      0.1000
  1    2   1     0     2     2  0.200      0.200      0.1000
  1    3   1     0     2     2  0.350      0.350      0.1500
  1    4   1     0     2     2  0.575      0.575      0.2250
  1    5   1     0     3     3  0.913      0.913      0.3375
  1    6   1     0     2     2   1.00       1.00      0.08750
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

- **自动的时间增量**
  - 试探算法（基于多年的经验）控制时间积分的精度。
  - 在静力学分析中，基于迭代次数达到收敛。
    - 容易得到收敛解（比最大允许迭代数量少很多）：
      ⇒增加迭代步长
    - 不容易收敛或发散：
      ⇒减小增量步长
    - 否则：
      ⇒保持同样的增量步长
  - 自动的时间增量步长控制工作很好。如果没有特殊的原因，用户不要改变它。

    提示： 对于高度非线性问题，推荐初始的时间增量为总时间增量的一小部分（比如10%）。

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

- **信息 (.msg) 文件**
  - 包括：
    - 所有的收敛控制：
      - 利用*CONTROLS选项覆盖默认值—不是经常需要
    - 关于具体模型特征的细节：
      - 非默认模型特征
      - 使用NLGEOM参数
      - 重启动文件的写出频率
    - 所有的迭代细节

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

- 求解器信息：
  - **数值奇异：** 这表明在线性方程组的求解过程中造成了大量数位的丢失，以至于结果是不可靠的。最常见的原因是静力应力分析过程中没有约束的刚体模式。
  - **零主元：** 在线性方程组求解过程中，当没有相应的刚 度抵抗所施加的力时，会出现这样的信息。最常见的原因是没有约束的刚体模式和过度约束的自由度。
  - **负特征值：** 负特征值表明刚度矩阵是非正定的；比如， 超过结构的屈曲载荷。

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

- 有用的修理故障信息：
  - 最大残差的位置
  - 过度变形的位置
  - 接触条件变化的位置

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
                        S T E P      1    S T A T I C   A N A L Y S I S


              SMALL DISPLACEMENT ANALYSIS

       AUTOMATIC TIME CONTROL WITH -
            A SUGGESTED INITIAL TIME INCREMENT OF                0.100
            AND A TOTAL TIME PERIOD OF                           1.00
            THE MINIMUM TIME INCREMENT ALLOWED IS               1.000E-03
            THE MAXIMUM TIME INCREMENT ALLOWED IS                1.00

    CONVERGENCE TOLERANCE PARAMETERS FOR FORCE
       CRITERION FOR RESIDUAL FORCE     FOR A NONLINEAR PROBLEM          5.000E-03
       CRITERION FOR DISP.    CORRECTION IN A NONLINEAR PROBLEM          1.000E-02
       INITIAL VALUE OF TIME AVERAGE FORCE                               1.000E-02
       AVERAGE FORCE     IS TIME AVERAGE FORCE
       ALTERNATE CRIT. FOR RESIDUAL FORCE     FOR A NONLINEAR PROBLEM    2.000E-02
    CRITERION FOR ZERO FORCE     RELATIVE TO TIME AVRG. FORCE             1.000E-05
       CRITERION FOR RESIDUAL FORCE     WHEN THERE IS ZERO FLUX          1.000E-05
       CRITERION FOR DISP.    CORRECTION WHEN THERE IS ZERO FLUX         1.000E-03
       CRITERION FOR RESIDUAL FORCE     FOR A LINEAR INCREMENT           1.000E-08

       FIELD CONVERSION RATIO                                             1.00
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
CONVERGENCE TOLERANCE PARAMETERS FOR MOMENT
     CRITERION FOR RESIDUAL MOMENT    FOR A NONLINEAR PROBLEM          5.000E-03
     CRITERION FOR ROTATION CORRECTION IN A NONLINEAR PROBLEM          1.000E-02
     INITIAL VALUE OF TIME AVERAGE MOMENT                             1.000E-02
     AVERAGE MOMENT    IS TIME AVERAGE MOMENT
     ALTERNATE CRIT. FOR RESIDUAL MOMENT    FOR A NONLINEAR PROBLEM    2.000E-02
     CRITERION FOR ZERO MOMENT    RELATIVE TO TIME AVRG. MOMENT        1.000E-05
     CRITERION FOR RESIDUAL MOMENT    WHEN THERE IS ZERO FLUX          1.000E-05
     CRITERION FOR ROTATION CORRECTION WHEN THERE IS ZERO FLUX          1.000E-03
     CRITERION FOR RESIDUAL MOMENT    FOR A LINEAR INCREMENT           1.000E-08
     FIELD CONVERSION RATIO                                            1.00


     VOLUMETRIC STRAIN COMPATIBILITY TOLERANCE FOR HYBRID SOLIDS       1.000E-05
     AXIAL STRAIN COMPATIBILITY TOLERANCE FOR HYBRID BEAMS             1.000E-05
     TRANS. SHEAR STRAIN COMPATIBILITY TOLERANCE FOR HYBRID BEAMS      1.000E-05
     SOFT CONTACT CONSTRAINT COMPATIBILITY TOLERANCE FOR P>P0          5.000E-03
     SOFT CONTACT CONSTRAINT COMPATIBILITY TOLERANCE FOR P=0.0         0.100
     DISPLACEMENT COMPATIBILITY TOLERANCE FOR DCOUP ELEMENTS           1.000E-05
     ROTATION COMPATIBILITY TOLERANCE FOR DCOUP ELEMENTS               1.000E-05
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
TIME INCREMENTATION CONTROL PARAMETERS:
     FIRST EQUILIBRIUM ITERATION FOR CONSECUTIVE DIVERGENCE CHECK          4
     EQUILIBRIUM ITERATION AT WHICH LOG. CONVERGENCE RATE CHECK BEGINS     8
     EQUILIBRIUM ITERATION AFTER WHICH ALTERNATE RESIDUAL IS USED          9
     MAXIMUM EQUILIBRIUM ITERATIONS ALLOWED                               16
     EQUILIBRIUM ITERATION COUNT FOR CUT-BACK IN NEXT INCREMENT           10
     MAXIMUM EQUILIB. ITERS IN TWO INCREMENTS FOR TIME INCREMENT INCREASE   4
     MAXIMUM ITERATIONS FOR SEVERE DISCONTINUITIES                        12
     MAXIMUM CUT-BACKS ALLOWED IN AN INCREMENT                             5
     MAXIMUM DISCON. ITERS IN TWO INCREMENTS FOR TIME INCREMENT INCREASE    6
     CUT-BACK FACTOR AFTER DIVERGENCE                                 0.2500
     CUT-BACK FACTOR FOR TOO SLOW CONVERGENCE                         0.5000
     CUT-BACK FACTOR AFTER TOO MANY EQUILIBRIUM ITERATIONS            0.7500
     CUT-BACK FACTOR AFTER TOO MANY SEVERE DISCONTINUITY ITERATIONS   0.2500
     CUT-BACK FACTOR AFTER PROBLEMS IN ELEMENT ASSEMBLY               0.2500
     INCREASE FACTOR AFTER TWO INCREMENTS THAT CONVERGE QUICKLY        1.500
     MAX. TIME INCREMENT INCREASE FACTOR ALLOWED                       1.500
     MAX. TIME INCREMENT INCREASE FACTOR ALLOWED (DYNAMICS)            1.250
     MAX. TIME INCREMENT INCREASE FACTOR ALLOWED (DIFFUSION)           2.000
     MINIMUM TIME INCREMENT RATIO FOR EXTRAPOLATION TO OCCUR          0.1000
     MAX. RATIO OF TIME INCREMENT TO STABILITY LIMIT                   1.000

     FRACTION OF STABILITY LIMIT FOR NEW TIME INCREMENT               0.9500
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
PRINT OF INCREMENT NUMBER, TIME, ETC., EVERY    1  INCREMENTS

   RESTART FILE WILL BE WRITTEN EVERY     3  INCREMENTS

   THE MAXIMUM NUMBER OF INCREMENTS IN THIS STEP IS                    25

        LARGE DISPLACEMENT THEORY WILL BE USED

   LINEAR EXTRAPOLATION WILL BE USED

   CHARACTERISTIC ELEMENT LENGTH      40.0

   DETAILED OUTPUT OF DIAGNOSTICS TO DATABASE REQUESTED

   PRINT OF INCREMENT NUMBER, TIME, ETC., TO THE MESSAGE FILE EVERY     1  INCREMENTS

   EQUATIONS ARE BEING REORDERED TO MINIMIZE WAVEFRONT

   COLLECTING MODEL CONSTRAINT INFORMATION FOR OVERCONSTRAINT CHECKS

   COLLECTING STEP CONSTRAINT INFORMATION FOR OVERCONSTRAINT CHECKS
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
INCREMENT     1 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.100

               CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                      1.251E+03   TIME AVG. FORCE       1.251E+03
 LARGEST RESIDUAL FORCE            -4.637E+03   AT NODE         11    DOF  1
 LARGEST INCREMENT OF DISP.         -1.84       AT NODE         11    DOF  2
 LARGEST CORRECTION TO DISP.        -1.84       AT NODE         11    DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                     7.200E+03   TIME AVG. MOMENT      7.200E+03
 LARGEST RESIDUAL MOMENT             28.8       AT NODE          9    DOF  6
 LARGEST INCREMENT OF ROTATION     -1.382E-02   AT NODE         11    DOF  6
 LARGEST CORRECTION TO ROTATION    -1.382E-02   AT NODE         11    DOF  6
          ROTATION CORRECTION TOO LARGE COMPARED TO ROTATION INCREMENT .

               CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

 AVERAGE FORCE                       37.8       TIME AVG. FORCE        37.8
 LARGEST RESIDUAL FORCE             0.215       AT NODE         11    DOF  1
 LARGEST INCREMENT OF DISP.         -1.84       AT NODE         11    DOF  2
 LARGEST CORRECTION TO DISP.       -1.007E-02   AT NODE         11    DOF  1
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                     7.200E+03   TIME AVG. MOMENT      7.200E+03
 LARGEST RESIDUAL MOMENT           -0.346       AT NODE          5    DOF  6
 LARGEST INCREMENT OF ROTATION     -1.382E-02   AT NODE         11    DOF  6
 LARGEST CORRECTION TO ROTATION     5.898E-07   AT NODE         11    DOF  6
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```

× 0.005 = 6.25

× 0.005 = 36

× .005 = 0.2

× 0.005 = 36

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     3

AVERAGE FORCE                        37.7       TIME AVG. FORCE        37.7
LARGEST RESIDUAL FORCE           -2.281E-06   AT NODE         11   DOF  1
LARGEST INCREMENT OF DISP.          -1.84     AT NODE         11   DOF  2
LARGEST CORRECTION TO DISP.      3.349E-05    AT NODE         11   DOF  2
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                    7.200E+03   TIME AVG. MOMENT     7.200E+03
LARGEST RESIDUAL MOMENT           1.523E-05   AT NODE          7   DOF  6
LARGEST INCREMENT OF ROTATION    -1.382E-02   AT NODE         11   DOF  6
LARGEST CORRECTION TO ROTATION    3.637E-07   AT NODE         11   DOF  6
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED

ITERATION SUMMARY FOR THE INCREMENT:   3 TOTAL ITERATIONS, OF WHICH
  0 ARE SEVERE DISCONTINUITY ITERATIONS AND  3 ARE EQUILIBRIUM ITERATIONS.

TIME INCREMENT COMPLETED  0.100    ,  FRACTION OF STEP COMPLETED  0.100
STEP TIME COMPLETED       0.100    ,  TOTAL TIME COMPLETED        0.100
```

× 0.005

0.2

× 0.005

36

4次或更少次数的迭代（如果这样的情况再次出现，Δt可以增加）

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
INCREMENT     2 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.100

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                        75.6       TIME AVG. FORCE        56.7
 LARGEST RESIDUAL FORCE              0.861       AT NODE         11   DOF  1
 LARGEST INCREMENT OF DISP.          -1.84       AT NODE         11   DOF  2
 LARGEST CORRECTION TO DISP.      -2.013E-02     AT NODE         11   DOF  1
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                    1.440E+04   TIME AVG. MOMENT     1.080E+04
 LARGEST RESIDUAL MOMENT             -1.38     AT NODE          5   DOF  6
 LARGEST INCREMENT OF ROTATION    -1.382E-02   AT NODE         11   DOF  6
 LARGEST CORRECTION TO ROTATION    3.914E-06   AT NODE         11   DOF  6
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```

没有增加

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
             CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

AVERAGE FORCE                        144.       TIME AVG. FORCE        90.9
LARGEST RESIDUAL FORCE            -6.928E-05    AT NODE          11   DOF  1
LARGEST INCREMENT OF DISP.         -1.84        AT NODE          11   DOF  2
LARGEST CORRECTION TO DISP.        1.701E-04    AT NODE          11   DOF  2
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                     1.600E+04    TIME AVG. MOMENT      1.160E+04
LARGEST RESIDUAL MOMENT            1.218E-04    AT NODE           7   DOF  6
LARGEST INCREMENT OF ROTATION     -1.382E-02    AT NODE          11   DOF  6
LARGEST CORRECTION TO ROTATION     1.804E-06    AT NODE          11   DOF  6
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
         TIME INCREMENT MAY NOW INCREASE TO   0.150

ITERATION SUMMARY FOR THE INCREMENT:   2 TOTAL ITERATIONS, OF WHICH
  0 ARE SEVERE DISCONTINUITY ITERATIONS AND  2 ARE EQUILIBRIUM ITERATIONS.

TIME INCREMENT COMPLETED  0.100    ,  FRACTION OF STEP COMPLETED  0.200
STEP TIME COMPLETED       0.200    ,  TOTAL TIME COMPLETED        0.200
```

4个连续增量步的迭代次数等于或小于4：$\Delta t = 1.5\Delta t_{old}$

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
INCREMENT     3 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.150

             CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                        133.       TIME AVG. FORCE        105.
 LARGEST RESIDUAL FORCE                3.02      AT NODE          11   DOF  1
 LARGEST INCREMENT OF DISP.           -2.75      AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.       -3.764E-02    AT NODE          11   DOF  1
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                     2.518E+04    TIME AVG. MOMENT      1.613E+04
 LARGEST RESIDUAL MOMENT              -4.47      AT NODE           5   DOF  6
 LARGEST INCREMENT OF ROTATION     -2.071E-02    AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION     1.722E-05    AT NODE          11   DOF  6
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```

$\Delta t = 1.5\Delta t_{old}$

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

AVERAGE FORCE                        252.       TIME AVG. FORCE        145.
LARGEST RESIDUAL FORCE            -7.965E-04    AT NODE          11   DOF  1
LARGEST INCREMENT OF DISP.         -2.75        AT NODE          11   DOF  2
LARGEST CORRECTION TO DISP.        5.629E-04    AT NODE          11   DOF  2
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                     2.798E+04    TIME AVG. MOMENT      1.706E+04
LARGEST RESIDUAL MOMENT            7.461E-04    AT NODE           7   DOF  6
LARGEST INCREMENT OF ROTATION     -2.070E-02    AT NODE          11   DOF  6
LARGEST CORRECTION TO ROTATION     5.967E-06    AT NODE          11   DOF  6
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
         TIME INCREMENT MAY NOW INCREASE TO   0.225

ITERATION SUMMARY FOR THE INCREMENT:   2 TOTAL ITERATIONS, OF WHICH
  0 ARE SEVERE DISCONTINUITY ITERATIONS AND  2 ARE EQUILIBRIUM ITERATIONS.

TIME INCREMENT COMPLETED  0.150    ,  FRACTION OF STEP COMPLETED  0.350
STEP TIME COMPLETED       0.350    ,  TOTAL TIME COMPLETED        0.350

    RESTART INFORMATION WRITTEN IN STEP   1  AFTER INCREMENT     3
```

4 or fewer

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
INCREMENT     4 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.225

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

AVERAGE FORCE                      1.528E+03    TIME AVG. FORCE        490.
LARGEST RESIDUAL FORCE            -4.550E+03    AT NODE          11   DOF  1
LARGEST INCREMENT OF DISP.         -5.95        AT NODE          11   DOF  2
LARGEST CORRECTION TO DISP.        -1.82        AT NODE          11   DOF  2
         FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

AVERAGE MOMENT                     4.853E+04    TIME AVG. MOMENT      2.493E+04
LARGEST RESIDUAL MOMENT            -344.        AT NODE           9   DOF  6
LARGEST INCREMENT OF ROTATION     -4.477E-02    AT NODE          11   DOF  6
LARGEST CORRECTION TO ROTATION    -1.371E-02    AT NODE          11   DOF  6
          MOMENT    EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.
```

$\Delta t = 1.5\Delta t_{old}$

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

AVERAGE FORCE                        281.       TIME AVG. FORCE        179.
LARGEST RESIDUAL FORCE              0.349       AT NODE          11   DOF  2
LARGEST INCREMENT OF DISP.          -5.94       AT NODE          11   DOF  2
LARGEST CORRECTION TO DISP.       -9.348E-03    AT NODE          11   DOF  1
         THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                     4.847E+04    TIME AVG. MOMENT      2.491E+04
LARGEST RESIDUAL MOMENT             -2.26       AT NODE           5   DOF  6
LARGEST INCREMENT OF ROTATION     -4.471E-02    AT NODE          11   DOF  6
LARGEST CORRECTION TO ROTATION     5.353E-05    AT NODE          11   DOF  6
         THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
         TIME INCREMENT MAY NOW INCREASE TO   0.338

ITERATION SUMMARY FOR THE INCREMENT:   2 TOTAL ITERATIONS, OF WHICH
  0 ARE SEVERE DISCONTINUITY ITERATIONS AND  2 ARE EQUILIBRIUM ITERATIONS.

TIME INCREMENT COMPLETED  0.225    ,  FRACTION OF STEP COMPLETED  0.575
STEP TIME COMPLETED       0.575    ,  TOTAL TIME COMPLETED        0.575
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
INCREMENT     5 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  0.338 <-- Δt = 1.5Δt_old

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                       1.248E+04   TIME AVG. FORCE       2.638E+03
 LARGEST RESIDUAL FORCE             -3.911E+04   AT NODE          11   DOF  1
 LARGEST INCREMENT OF DISP.          -14.2       AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.         -5.31       AT NODE          11   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                      1.049E+05   TIME AVG. MOMENT      4.090E+04
 LARGEST RESIDUAL MOMENT            -4.323E+03   AT NODE           9   DOF  6
 LARGEST INCREMENT OF ROTATION       -0.107      AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION     -4.037E-02   AT NODE          11   DOF  6
          MOMENT    EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

              CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

 AVERAGE FORCE                        556.       TIME AVG. FORCE         254.
 LARGEST RESIDUAL FORCE              16.6        AT NODE          11   DOF  1
 LARGEST INCREMENT OF DISP.          -14.2       AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.       -8.119E-02    AT NODE          11   DOF  1
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                      1.044E+05   TIME AVG. MOMENT      4.080E+04
 LARGEST RESIDUAL MOMENT             -42.5       AT NODE           5   DOF  6
 LARGEST INCREMENT OF ROTATION       -0.107      AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION      1.095E-04   AT NODE          11   DOF  6
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
                CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     3

 AVERAGE FORCE                        559.       TIME AVG. FORCE        255.
 LARGEST RESIDUAL FORCE              -28.9       AT NODE          11   DOF  1
 LARGEST INCREMENT OF DISP.          -14.1       AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.         0.130       AT NODE          11   DOF  2
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                      1.153E+05   TIME AVG. MOMENT      4.299E+04
 LARGEST RESIDUAL MOMENT             3.833E-02   AT NODE           5   DOF  6
 LARGEST INCREMENT OF ROTATION      -0.106       AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION      1.112E-03   AT NODE          11   DOF  6
 ESTIMATE OF ROTATION CORRECTION    -1.004E-06
 MOMENT    EQUILIB. ACCEPTED BASED ON SMALL RESIDUAL AND ESTIMATED CORRECTION

                CONVERGENCE CHECKS FOR EQUILIBRIU

 AVERAGE FORCE                       1.053E+03   TIME AVG. FORCE        354.
 LARGEST RESIDUAL FORCE              1.092E-03   AT NODE          11   DOF  2
 LARGEST INCREMENT OF DISP.          -14.1       AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.        -2.092E-04   AT NODE          11   DOF  2
          THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

 AVERAGE MOMENT                      1.153E+05   TIME AVG. MOMENT      4.299E+04
 LARGEST RESIDUAL MOMENT            -2.910E-02   AT NODE           7   DOF  6
 LARGEST INCREMENT OF ROTATION      -0.106       AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION     -1.875E-06   AT NODE          11   DOF  6
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED

 ITERATION SUMMARY FOR THE INCREMENT:   3 TOTAL ITERATIONS, OF WHICH
   0 ARE SEVERE DISCONTINUITY ITERATIONS AND  3 ARE EQUILIBRIUM ITERATIONS.

 TIME INCREMENT COMPLETED  0.338    ,  FRACTION OF STEP COMPLETED  0.913
 STEP TIME COMPLETED       0.913    ,  TOTAL TIME COMPLETED        0.913
```

残余应力在容差之内，但是转动位移纠正太大。估计下次迭代的转动位移纠正会比较小。

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

```
INCREMENT     6 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  8.750E-02

                CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     1

 AVERAGE FORCE                        641.       TIME AVG. FORCE        402.
 LARGEST RESIDUAL FORCE               74.0       AT NODE          11   DOF  1
 LARGEST INCREMENT OF DISP.          -3.55       AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.        -0.180       AT NODE          11   DOF  1
          FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

 AVERAGE MOMENT                      1.179E+05   TIME AVG. MOMENT      5.547E+04
 LARGEST RESIDUAL MOMENT             -99.4       AT NODE           5   DOF  6
 LARGEST INCREMENT OF ROTATION      -2.702E-02   AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION      5.186E-04   AT NODE          11   DOF  6
 ESTIMATE OF ROTATION CORRECTION    -1.594E-05
 MOMENT    EQUILIB. ACCEPTED BASED ON SMALL RESIDUAL AND ESTIMATED CORRECTION

                CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION     2

 AVERAGE FORCE                        695.       TIME AVG. FORCE        411.
 LARGEST RESIDUAL FORCE              -0.505      AT NODE          11   DOF  1
 LARGEST INCREMENT OF DISP.          -3.53       AT NODE          11   DOF  2
 LARGEST CORRECTION TO DISP.         1.386E-02   AT NODE          11   DOF  2
          THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

 AVERAGE MOMENT                      1.309E+05   TIME AVG. MOMENT      5.764E+04
 LARGEST RESIDUAL MOMENT             8.716E-02   AT NODE           7   DOF  6
 LARGEST INCREMENT OF ROTATION      -2.687E-02   AT NODE          11   DOF  6
 LARGEST CORRECTION TO ROTATION      1.493E-04   AT NODE          11   DOF  6
          THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```


© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

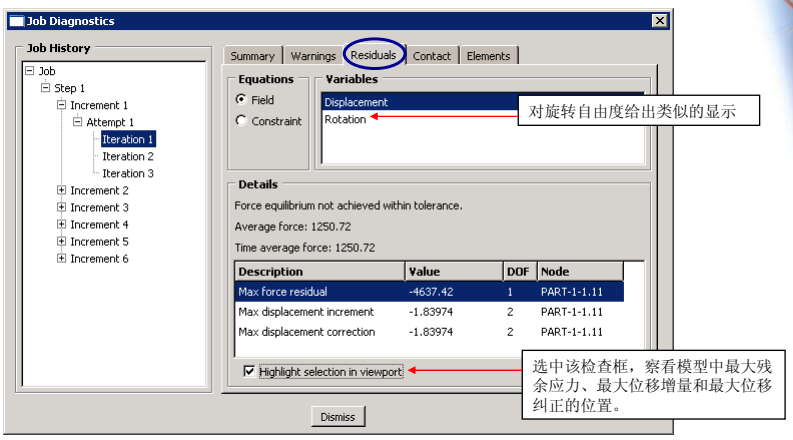
```
 ITERATION SUMMARY FOR THE INCREMENT:   2 TOTAL ITERATIONS, OF WHICH
   0 ARE SEVERE DISCONTINUITY ITERATIONS AND  2 ARE EQUILIBRIUM ITERATIONS.

 TIME INCREMENT COMPLETED  8.750E-02,  FRACTION OF STEP COMPLETED   1.00
 STEP TIME COMPLETED        1.00    ,  TOTAL TIME COMPLETED         1.00

     RESTART INFORMATION WRITTEN IN STEP   1  AFTER INCREMENT     6


         THE ANALYSIS HAS BEEN COMPLETED


     ANALYSIS SUMMARY:
     TOTAL OF          6  INCREMENTS
                       0  CUTBACKS IN AUTOMATIC INCREMENTATION
                      15  ITERATIONS INCLUDING CONTACT ITERATIONS IF PRESENT
                      15  PASSES THROUGH THE EQUATION SOLVER OF WHICH
                      15  INVOLVE MATRIX DECOMPOSITION, INCLUDING
                       0  DECOMPOSITION(S) OF THE MASS MATRIX
                       1  REORDERING OF EQUATIONS TO MINIMIZE WAVEFRONT
                       0  ADDITIONAL RESIDUAL EVALUATIONS FOR LINE SEARCHES
                       0  ADDITIONAL OPERATOR EVALUATIONS FOR LINE SEARCHES
                       3  WARNING MESSAGES DURING USER INPUT PROCESSING
                       0  WARNING MESSAGES DURING ANALYSIS
                       0  ANALYSIS WARNINGS ARE NUMERICAL PROBLEM MESSAGES
                       0  ANALYSIS WARNINGS ARE NEGATIVE EIGENVALUE MESSAGES
                       0  ERROR MESSAGES
```

在此处察看警告和错误信息。请察看信息文件或数据文件，确定引起这些信息的原因。

© Dassault Systèmes, 2008

非线性悬臂梁分析的输出

- **Abaqus/Viewer中的可视化诊断**



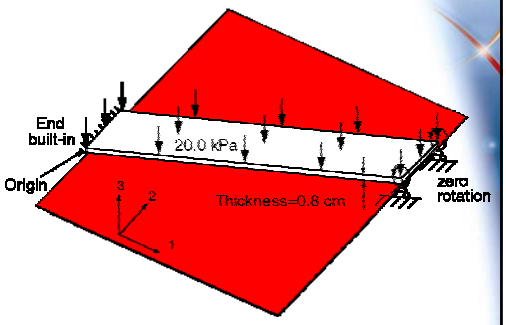
© Dassault Systèmes, 2008

习题3: 非线性静力学

© Dassault Systèmes, 2008

习题3: 非线性静力学

- **习题任务**
  1. 根据平板倾斜的角度，定义另外的材料方向。
  2. 在考虑或不考虑几何非线性效应的条件下，分析斜板的变形。
  3. 在材料定义中包含塑性。
  4. 利用Abaqus/Viewer察看结果。

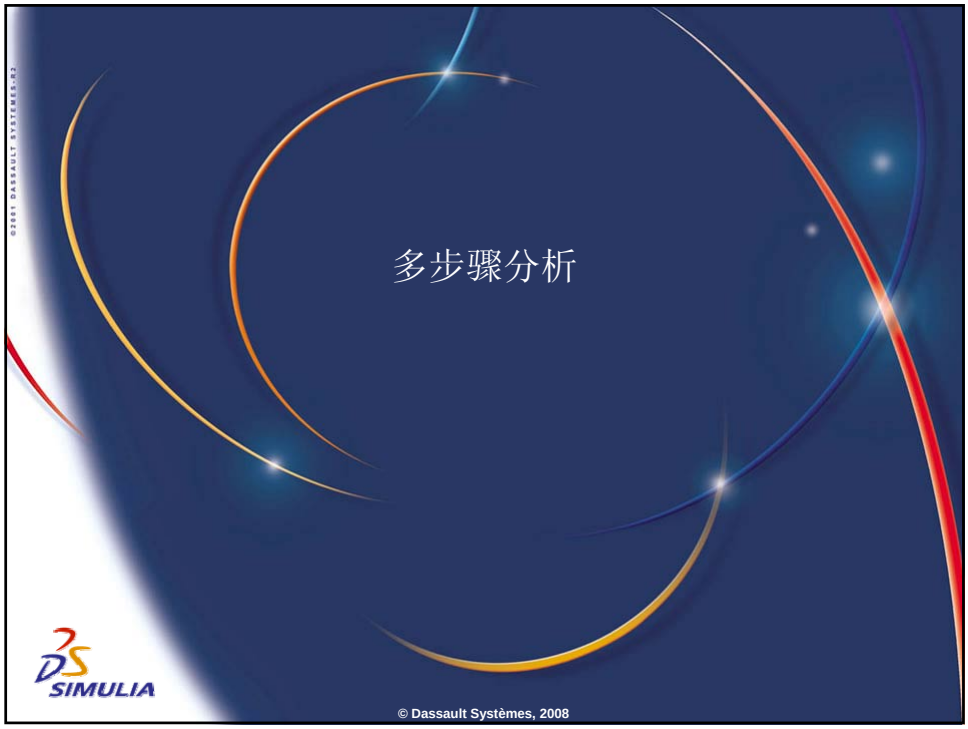


Video Clip

© Dassault Systèmes, 2008

Abaqus中的多步骤分析
第四讲
SIMULIA
© Dassault Systèmes, 2008

L4.2
概述
• 多步骤分析
• Abaqus中的重启动分析
SIMULIA
Introduction to Abaqus/Standard and Abaqus/Explicit
© Dassault Systèmes, 2008



多步骤分析

- 一般的，将分析过程分解为多个分析步进行会比较方便。这样，载荷或边界条件可以在分析步中施加，或者可以修改输出需求。
  - 一般需要几个通用分析步。
  - 通用分析步中间可以插入摄动分析步。

© Dassault Systèmes, 2008

多步骤分析

- **可能的分析步顺序**
  - 通用分析步可以是通用分析步
    - 第一个通用分析步的结束条件是第二个通用分析步的起始条件
  - 通用分析步可以是线性摄动分析步
    - 紧接着的通用分析步忽略先前的线性摄动响应
  - 线性摄动分析步可以是线性摄动分析步
    - 是一系列独立的分析步
    - 一些有先后顺序 (比如频率提取必须在modal dynamics之前)
  - 线性摄动分析步可以是通用分析步
    - 第一个通用分析步的结束条件是第二个通用分析步的起始条件

© Dassault Systèmes, 2008

多步骤分析

- **通用分析步之后可以是另一个通用分析步**
  - 第一个分析步的结束条件是第二个分析步的起始条件
  - 载荷一般作为全部载荷

© Dassault Systèmes, 2008

多步骤分析

- 通用分析步之后可以是摄动分析步：
  - 摄动分析步的结果是基状态的扰动。
    - 如果使用NLGEOM参数，通用分析步的结束构型为基状态。
    - 如果没有使用NLGEOM参数，初始构型为基状态。
    - 特征值屈曲分析 (*BUCKLE)例外：
      - 即使没有使用NLGEOM参数，屈曲分析的基状态也会包含前面通用分析步的应力影响。
  - 最近的通用分析步的接触状态被强制施加。
  - 摄动分析步施加的载荷被认为是摄动载荷。

Video Clip

© Dassault Systèmes, 2008

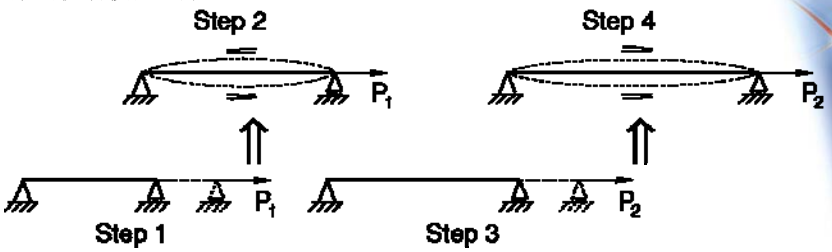
多步骤分析

- **在摄动分析步之后继续通用分析步**
  - 最近通用分析步的结束条件（如果有）是下一个通用分析步的起始条件
  - 忽略扰动响应

© Dassault Systèmes, 2008

多步骤分析

- **索的振动模拟—方法 1**

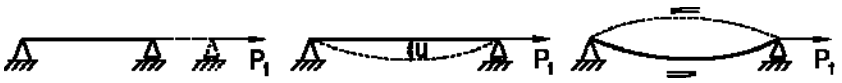


分析步	动作	分析步类型
1	拉伸	具有NLGEOM参数的通用分析步
2	提取频率	在第一个分析步的结束条件（基状态）的基础之上进行线性摄动分析
3	继续拉伸	在第一个分析步的结束条件（上一个非线性分析步）的基础上继续通用分析步
4	再一次提取频率	在第三个分析步的结束条件（新的基状态）的基础之上进行线性摄动分析步

© Dassault Systèmes, 2008

多步骤分析

- **索的振动模拟—方法 2**
  - 可以将历程修改为一系列分开的通用分析步，这样可以得到系统的最低特征频率：



分析步	动作
1	拉伸
2	偏置中点
3	释放，观察索的振动

© Dassault Systèmes, 2008

多步骤分析

•历程定义：

```
*STEP, NLGEOM
 STEP 1: STRETCH CABLE
*STATIC
*CLOAD
 13, 1, 500.
*RESTART, WRITE
*NODE FILE
U
*EL PRINT
S, MISES, E
*NODE PRINT
U, RF, CF
*END STEP
**
*STEP, NLGEOM
 STEP 2: DEFLECT MIDPOINT
*STATIC
.1, 1.
*BOUNDARY, OP=MOD
7, 2, 2, -1.
*END STEP
```

除非故意修改或删除，该载荷在全部分析过程中保持不变。

中点的偏置被添加到模型定义中指定的边界条件上。

© Dassault Systèmes, 2008

多步骤分析

```
**
*STEP, NLGEOM, INC=200
STEP 3:RELEASE & SEE VIBRATE
*DYNAMIC
** use fixed time incs for
** this example
** dtinit, ttot, dtmin, dtmax
.0002, .04
*BOUNDARY, OP=NEW
1, 1, 2
13, 2
*PRINT, FREQUENCY=100
*EL PRINT, FREQUENCY=0
*NODE PRINT, FREQUENCY=0
*END STEP
```

All previously specified boundary conditions are removed, and the pin and roller conditions are redefined. The midpoint deflection is removed since it is not redefined.

© Dassault Systèmes, 2008

多步骤分析

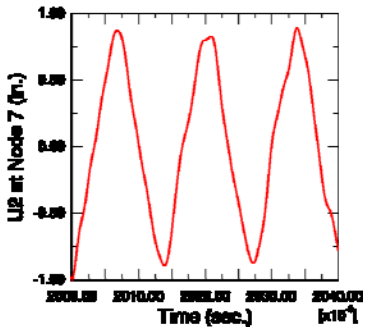
•状态文件 (*jobid*`.sta`)总结分析的增量信息。

```
SUMMARY OF JOB INFORMATION:
STEP INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF    DOF     IF
             DISCON ITERS ITERS   TIME/    TIME/LPF   TIME/LPF  MONITOR RIKS
              ITERS               FREQ
   1    1   1     0     2     2   1.00       1.00      1.000
   2    1   1     0     2     2   1.10      0.100      0.1000
   2    2   1     0     1     1   1.20      0.200      0.1000
   2    3   1     0     1     1   1.35      0.350      0.1500
   2    4   1     0     1     1   1.58      0.575      0.2250
   2    5   1     0     1     1   1.91      0.913      0.3375
   2    6   1     0     1     1   2.00       1.00      8.7500E-02
   3    1   1     0     1     1   2.00      2.000E-04  2.0000E-04
   3    2   1     0     1     1   2.00      4.000E-04  2.0000E-04
   3    3   1     0     1     1   2.00      6.000E-04  2.0000E-04
  ---------------------------------------------------------------
    .
    .
    .
  ---------------------------------------------------------------
   3  195   1     0     1     1   2.04      3.900E-02  2.0000E-04
   3  196   1     0     1     1   2.04      3.920E-02  2.0000E-04
   3  197   1     0     1     1   2.04      3.940E-02  2.0000E-04
   3  198   1     0     1     1   2.04      3.960E-02  2.0000E-04
   3  199   1     0     1     1   2.04      3.980E-02  2.0000E-04
   3  200   1     0     1     1   2.04      4.000E-02  2.0000E-04
```


© Dassault Systèmes, 2008

多步骤分析

- 可以在Abaqus/Viewer中将第三个分析步中的动力学响应绘制为 *X*–*Y* 图。



© Dassault Systèmes, 2008

Abaqus中的重启动分析

SIMULIA

- **重启动文件用于：**
  - 启动在分析过程中终止的分析。
    - 作业可能因为以下的原因中止：
      - 达到分析步指定的最大增量数量。
      - 没有足够的硬盘空间，或计算机失效。
      - 作业不能收敛。
    - 在以下情况之后，需要继续分析作业：
      - 在特殊的点检查结果。
      - 修改历程：分析内容、载荷、输出或控制信息。
  - 在Abaqus/Standard和 Abaqus/Explicit之间传递数据。

© Dassault Systèmes, 2008

Abaqus中的重启动分析

- **Abaqus/Standard中的重启动句法：**

```
*RESTART, READ, STEP= , INC= , END STEP, WRITE, FREQUENCY= , OVERLAY
```

• READ, STEP, 和 INC	从前面分析的指定分析步和指定增量步读取重启动数据。（默认值为从上一个可用的重启动数据读取数据。）
• END STEP	在读取重启动数据时使用；后面有详细介绍。
• WRITE 和 FREQUENCY	在分析过程中，控制重启动数据写出的频率。如果WRITE被指定，重启动数据在分析步结束时被写出。
• OVERLAY	让Abaqus只写出最后一组重启动数据。（在每个分析步中将只有一组重启动数据。）

© Dassault Systèmes, 2008

Abaqus中的重启动分析

- **Abaqus/Explicit中的重启动句法：**
  - 在特定的分析中，Abaqus/Explicit的重启动文件允许分析在某个指定的点完成（重启动输出的时间间隔），并且在之后的分析中重新启动并继续分析。
    - 如果需要重新启动Abaqus/Explicit 模拟，需要状态文件和初始重启动文件等信息。
  - Abaqus/Explicit重启动分析的句法与Abaqus/Standard稍有不同：

```
*RESTART, READ, STEP=P, INTERVAL=Q
```

  - 上面的例子中，在分析步*P*完成*Q*个时间间隔后，分析重新启动。

© Dassault Systèmes, 2008

Abaqus中的重启动分析

- **提交重启动作业：**

abaqus job=*job-name* **oldjob=***oldjob-name*

job=*job-name*: name of the new input file

oldjob=*oldjob-name*: name of the restart file created by the previous run

  - 在重启动分析中，可以改变下面的模型数据：
    - 幅值定义
    - 节点集
    - 单元集

© Dassault Systèmes, 2008

Abaqus中的重启动分析

- **索的振动模拟—方法3: 重启动分析**
  - 进行索振动模拟的另外的方法就是执行重启动分析。

分析作业 1	分析步 1	拉伸
		写出重启动文件
分析作业 2	分析步 2	读入重启动信息
		偏移索的中点
	分析步 3	释放中点，并研究索的振动

© Dassault Systèmes, 2008

Abaqus中的重启动分析

```
*HEADING
READ SOLUTION AT END OF STEP 1 AND CONTINUE
THE VIBRATING CABLE SIMULATION
*RESTART, READ, STEP=1,INC=1
**
*STEP, NLGEOM
 STEP 2: DEFLECT MIDPOINT
*STATIC
.1, 1.
*BOUNDARY, OP=MOD
7, 2, 2, -1.
*END STEP
**
*STEP, NLGEOM, INC=200
STEP 3: RELEASE & SEE VIBRATE
*DYNAMIC
.0002, .04
*BOUNDARY, OP=NEW
1,  1, 2
13, 2
*END STEP
```

从原始分析生成的重启动文件读入模型定义和分析步1的数据。

© Dassault Systèmes, 2008

Abaqus中的重启动分析

- **管理复杂的分析**
  - Abaqus中灵活的重启动功能为管理复杂的分析提供了方便。但是有三个重要的规则：
    1. **不能为重启动文件继续添加内容。**
       - Abaqus总是从旧的重启动文件读入，并将结果写到新的重启动文件。
       - 如果包括一系列重启动分析，将生成几个重启动文件。
       - 同重启动文件一样，输出数据库 (**.odb**)文件也不能继续添加内容。
         - 每个重启动分析都有它自己的输出数据库文件。
       - 重启动分析的结果 (**.fil**)文件包含前面分析的结果和当前分析的结果（默认情况）。

© Dassault Systèmes, 2008

Abaqus中的重启动分析

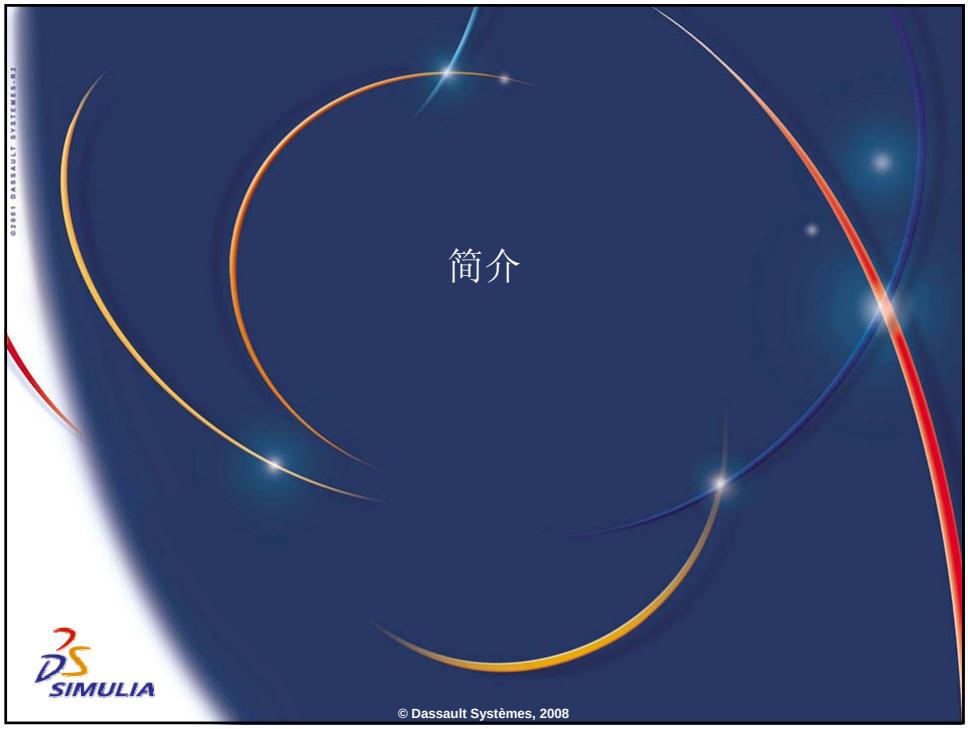
2. **除非在新的分析步中明确的作出修改，在重启动分析中，前面分析的所有输出需求和载荷仍然有效。**
3. **如果Abaqus从没有完成的分析重新启动，在开始新的分析步之前，默认情况下，它将首先尝试完成前面分析过程中已经开始分析、但是没有完成的分析步。**
   - Abaqus将只完成前面分析中已经开始分析，但是还没有完成的分析步。
     - Abaqus将不会尝试原有分析过程任何其它的分析步。
     - 如果需要执行前面分析中还没有开始分析的分析步，必须把它们添加到重启动分析的输入文件中。
   - 使用END STEP参数，可以终止重启动文件从它读入的分析步继续分析，而直接从新定义的分析步继续分析。
   - 使用END STEP参数，可以继续分析因为达到最大增量数而终止的分析。

© Dassault Systèmes, 2008

概述

- 简介
- Tie约束
- 刚体
- 接触
- 定义双面接触
- 定义面的规则
- 局部表面行为
- 接触问题中点的相对滑动
- 接触问题中调整初始节点位置
- 接触输出
- 习题4：简单接触

© Dassault Systèmes, 2008



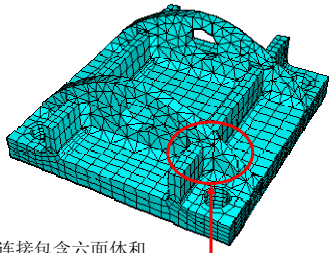
简介

- **什么是约束？**
  - 约束允许模型在节点之间传递运动关系
  - 这些关联在模型中的自由度定义
    - 相反的，装配约束仅仅是定义各部件之间的初始位置
  - 例如:
    - 绑定约束Tie
    - 多点约束MPC
    - 壳-实体的耦合Shell-to-solid coupling
    - 刚体约束等

© Dassault Systèmes, 2008

简介

- **绑定约束**
  - 允许将两个区域绑定，即使两个区域的网格不协调

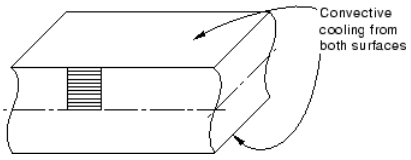


绑定约束用于连接包含六面体和四面体单元的网格

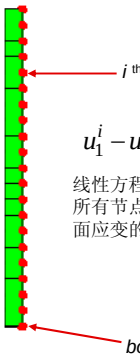
© Dassault Systèmes, 2008

简介

- **多点约束 (MPCs)**
  - 节点之间可以说是线性或非线性的约束
  - 线性方程是MPC的一种形式



无限大平板的淬火问题



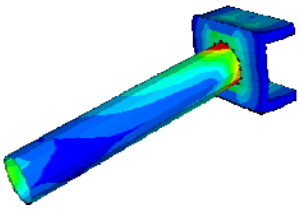
$$u_1^i - u_1^{bot} = 0$$

线性方程约束用于有右边所有节点，以模拟广义平面应变的条件

© Dassault Systèmes, 2008

简介

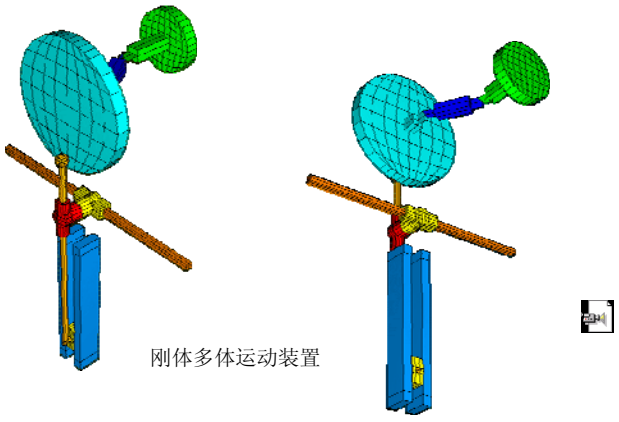
- **壳-实体的耦合**
  - 将壳的边与实体的面的运动耦合



© Dassault Systèmes, 2008

简介

- **刚体约束**
  - 允许将装配体中的部分区域的运动约束到一个参考点上



刚体多体运动装置

© Dassault Systèmes, 2008

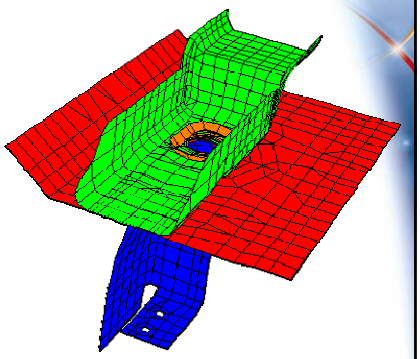
简介

- **什么是接触?**
  - 当两个实体接触时，力通过它们的公共表面传递。
    - 在某些情况下，只传递垂直接触表面的力。
    - 如果存在摩擦，沿接触表面的切向传递有限的切向力。
      - 沿接触表面，摩擦力导致剪切应力。
    - 一般目标： 确定接触面积和传递的应力。
  - 接触是严重不连续形式的非线性行为。
    - 或者施加约束（表面不可以互相穿透），或者忽略约束.

© Dassault Systèmes, 2008

简介

- **为什么要定义接触?**
  - 除非用户指定可能会接触的表面和/或节点，Abaqus现在还不能检测接触。

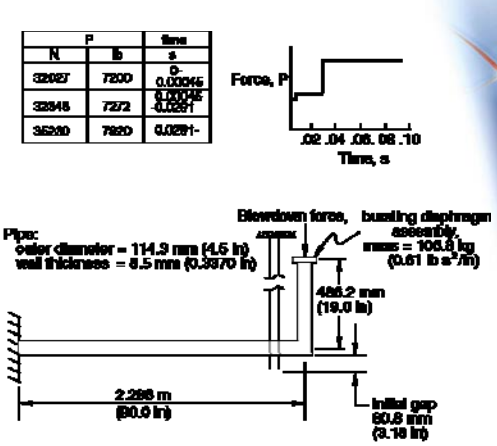


例子选自 “Submodeling of a stacked sheet metal assembly,” Abaqus例子手册1.1.19节

© Dassault Systèmes, 2008

简介

- **接触例子**
- **间隙接触**
  - 用节点与节点接触的方式为点接触建模。



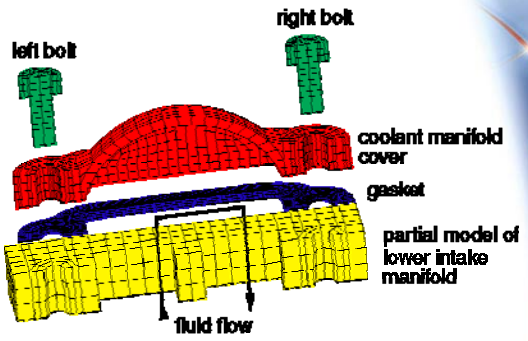
“Detroit Edison pipe whip experiment,”Abaqus例子手册 2.1.2节

© Dassault Systèmes, 2008

简介

赫兹接触

- 接触面之间的相对位移很小。
- 分布表面的接触。
- 典型例子：轴承设计、硬垫圈、过盈配合.



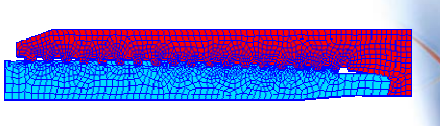
例子选自 “Coolant manifold cover gasketed joint,” Abaqus例子手册5.1.4 节

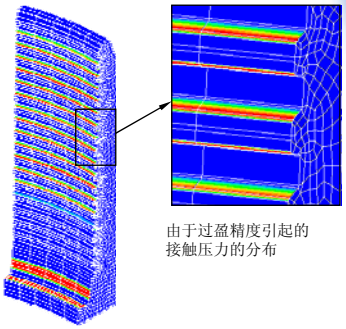
© Dassault Systèmes, 2008

简介

变形体之间的大滑动接触

- 这是最一般类型的接触。
- 例子：螺纹连接。
  - 这些问题一般包含初始的过盈配合（因为锥形的螺纹），并且具有相似强度的材料之间为有限滑动.





由于过盈精度引起的接触压力的分布

Video Clip

“Axisymmetric analysis of a threaded connection,” 例子手册中 1.1.20

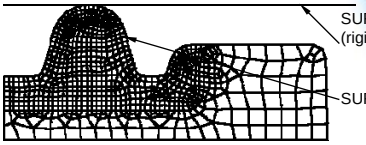
© Dassault Systèmes, 2008

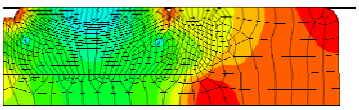
简介

自接触

- 单个表面与它自身接触称为自接触。在Abaqus的二维和三维模型中可用。
- 在分析过中，当表面严重变形时，使用自接触将非常方便。对于某些问题，在分析之前不可能确定单个的接触区域，或者确定接触区域是非常困难的。
- 把单个接触表面作为接触对定义自接触，而不是通过两个不同的表面定义.

Video Clip





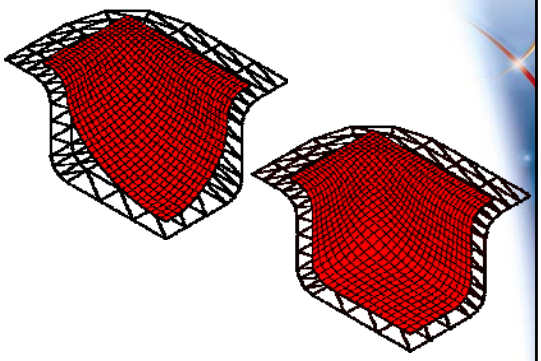
最小主应力云图

例子：橡胶垫片的压缩 (选自 “Self-contact in rubber/foam components: rubber gasket,” Abaqus例子手册1.1.18 节)

© Dassault Systèmes, 2008

简介

- **变形体与刚体的接触**
  - 表面之间的有限滑动（大位移）.
  - 变形组件之间的有限应变.
  - 典型例子:
    - 橡胶密封件
    - 马路上的轮胎
    - 海床底部的管道
    - 成型模拟（刚体砧/模具、可变形组件）



例子：金属成型模拟

Video Clip

“Superplastic forming of a rectangular box,” 例子手册 1.3.2

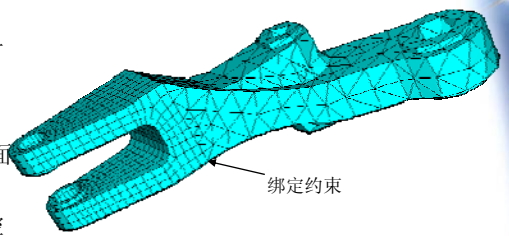
© Dassault Systèmes, 2008

绑定约束

© Dassault Systèmes, 2008

绑定约束

- 在Abaqus中，通过捆绑约束定义完全的约束行为。
- 捆绑约束可以以简单的方式，将表面永久的捆绑在一起。
  - 容易进行网格过渡。
- 使用主-从公式定义基于表面的约束。
- 该约束防止从属表面和主控表面分离或产生相对滑动



© Dassault Systèmes, 2008

绑定约束

- 句法：

```
*TIE, NAME=name, ADJUST=[YES | NO],
[POSITION TOLERANCE | TIE NSET]
SLAVE, MASTER
```

  - POSITION TOLERANCE参数定义被绑定从属表面节点与主控表面间距的容差。在此容差范围之内的从属表面上的节点将被绑定。
    - 如果从属表面上的节点与主控表面的间距大于该距离，从属表面上的节点将不被捆绑。
  - 另外，可以使用TIE NSET参数，将包含从属表面节点的节点集绑定到主控表面。
    - 如果节点在从属表面上，但不在该节点集中，这些节点将不被绑定.

© Dassault Systèmes, 2008

绑定约束

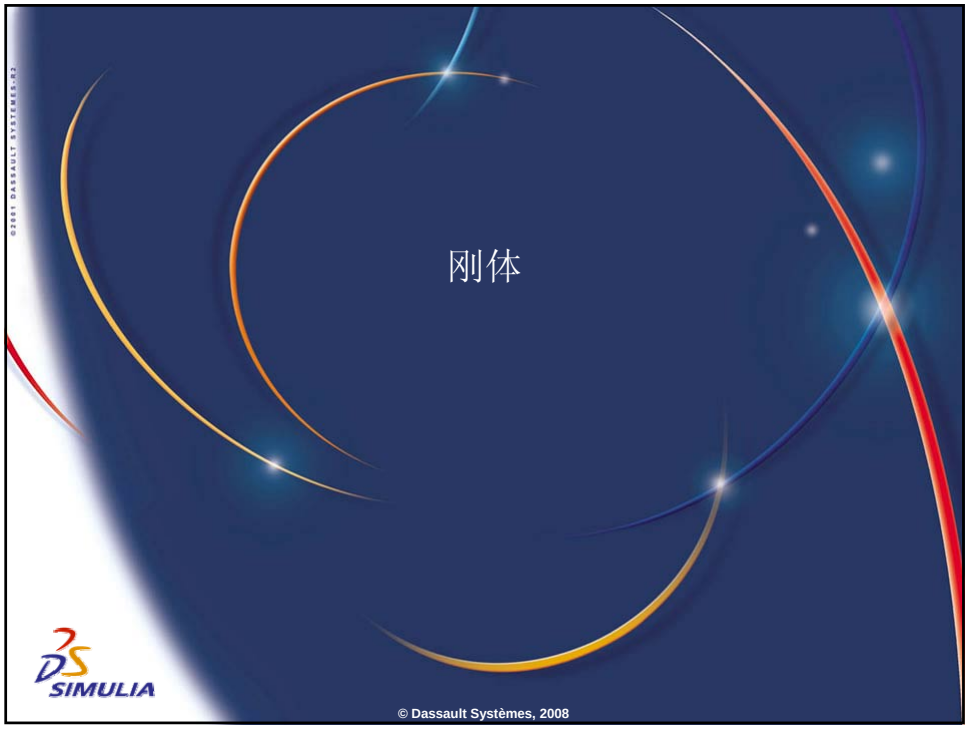
- ADJUST参数是可选的。
  - 设置该参数为YES，在初始构型中，将移动主控表面中所有的从属节点（由可选的POSITION TOLERANCE参数选项定义的距离），并不产生任何附加应变。
- 从属节点的状态（张开或接触）将打印在输出 (**.dat**)文件中。
  - 在打印输出文件中，对于捆绑约束中没有与主控表面接触的从属节点将给出警告信息。
- 默认情况下，平动和转动自由度将都被约束。
  - 如果不需要约束旋转自由度，可以使用NO ROTATION参数.

© Dassault Systèmes, 2008

绑定约束

- 对捆绑约束中的从属节点不要使用边界条件、约束方程或多点约束。这将引起从属节点的过渡约束，并导致分析过程中的错误。
  - 症状：
    - Abaqus/Standard信息(**.msg**)文件中零主元警告
    - Abaqus/Explicit中变形波速错误

© Dassault Systèmes, 2008



刚体

- Abaqus具有一般刚体的功能
  - 刚体是节点和单元的集合，刚体的运动由一个节点，即它的参考节点控制
  - 任何实体或部分实体都可以被定义为刚体
  - 刚体可以承受任意大的刚体运动
- 刚体的计算效率较高
  - 不超过六个自由度，就可以完全描述刚体的运动。
  - 对于组成刚体的单元，不进行单元计算
- 对于两个相互接触的物体，如果其中一个物体比另外的物体刚硬许多，可以将较为刚硬的物体指定为刚体；例如，金属成型过程中的砧

© Dassault Systèmes, 2008

刚体

- 三种方式定义刚体:
  - 定义离散刚体和使用*RIGID BODY选项将单元分派为刚体.
    - 任何形式的几何外形
  - 定义解析刚体表面
    - 有限种类的几何外形
  - 用户子程序**RSURFU** (Abaqus/Standard only)

© Dassault Systèmes, 2008

刚体

- 刚体定义最多可以包括:
  - 1个单元集
  - 1个捆绑节点集
  - 1个销节点集
  - 1解析表面
  - 1参考节点
- 每个刚体定义必须唯一。
  - 刚体定义中不可以共享节点、单元或参考节点.

© Dassault Systèmes, 2008

刚体

- **离散刚体**
  - 大多数单元类型可以作为刚体的一部分.
    - 比如，如果使用*RIGID BODY选项将单元分派为刚体，实体单元和刚体单元可以模拟同样的效果。
    - 包含实体单元定义刚体的例子：

```
*ELEMENT, TYPE=C3D8R, ELSET=SOLID1
...
*SOLID SECTION, ELSET=SOLID1, MATERIAL=STEEL
*MATERIAL, NAME=STEEL
*ELASTIC
200.0E9, 0.3
*DENSITY
7800.0,
*RIGID BODY, REF NODE=10000, ELSET=SOLID1
```


© Dassault Systèmes, 2008

刚体

- **销节点 vs. 捆绑节点**
  - 每个刚体从属节点可以被指定为以下两种类型之一：销节点或捆绑节点
  - 当刚体中包含单元时，可以定义附加的节点集，为可变形单元提供更多的连接点.

```
*RIGID BODY, REF NODE=10000, PIN NSET= , TIE NSET=
```


© Dassault Systèmes, 2008

刚体

- 销节点只有平动自由度同刚体相关联.
  - 通过销节点连接的刚体和变形体单元之间只传递位移和力.
- 捆绑节点的平动和转动自由度均同刚体相关联.
  - 通过捆绑节点连接的刚体和变形体单元之间，除了传递位移和力之外，还传递转动和弯矩.

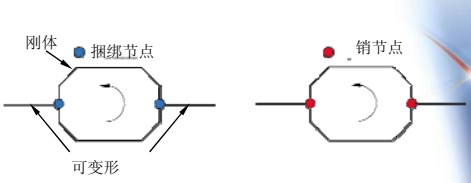
© Dassault Systèmes, 2008

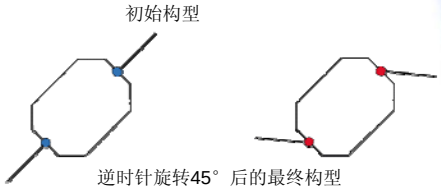
刚体

- 如果节点与多个类型的单元相连，默认情况下为“捆绑”约束。比如，如果节点与CPE3和B21 单元连接，默认情况下，该节点为捆绑节点。
- 可以通过在销节点集或捆绑节点集包含同样的节点，覆盖默认的节点类型。

  RIGID BODY, REF NODE=node*, **ELSET=***element set*, **PIN NSET=***node set*, **TIE NSET=***node set*

  thickness





Video Clip

© Dassault Systèmes, 2008

刚体

- **解析刚体表面**
  - 使用*SURFACE选项，有三种可用的解析表面：
    - 使用TYPE=SEGMENTS定义二维刚体表面。
    - 使用TYPE=CYLINDER定义三维刚体表面，在垂直于平面方向无限延伸。
    - 使用TYPE=REVOLUTION定义旋转的三维表面.

© Dassault Systèmes, 2008

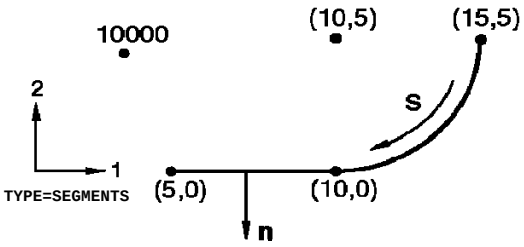
刚体

- 解析刚体表面不能自动平滑。然而，平滑表面的接触计算将更加容易。
  - 使用FILLET RADIUS参数提供导角半径，用于平滑解析刚体表面的各个部分。
- 使用*RIGID BODY选项将刚体指定为表面，并分派参考节点.

© Dassault Systèmes, 2008

刚体

- 解析刚体表面的输入句法:



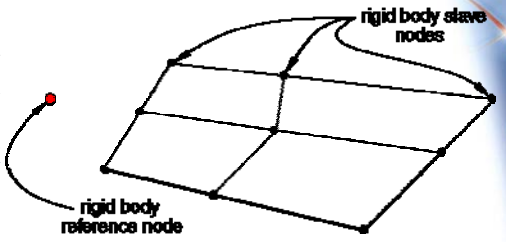

```
*SURFACE, TYPE=SEGMENTS, NAME=SRIGID
START, 15.0, 5.0
CIRCL, 10.0, 0.0, 10.0, 5.0
LINE, 5.0, 0.0
*RIGID BODY, ANALYTICAL SURFACE=SRIGID, REF NODE=10000
```

通过定义s，利用线段的顺序定义法向n。 $n = z \times s$，其中z 为平行于z-轴的单位向量，接触发生在n方向。

© Dassault Systèmes, 2008

刚体

- **刚体的运动**
  - 刚体的运动由刚体参考节点的运动控制—可以是边界条件，也可以是施加在刚体上的力。
    - 形成刚体的其它节点被称为“刚体从属节点”。



形成刚体的节点

© Dassault Systèmes, 2008

刚体

- **刚体参考点的位置**
  - 用户可以将刚体的参考节点放在模型中的任何位置。
  - 如果在分析过程中，刚体在所施加载荷下自由移动，参考节点的位置是非常重要的；这样的情况下，参考节点应该放在刚体的质心。
  - Abaqus可以自动计算质心，并重新确定参考节点的位置.

© Dassault Systèmes, 2008

刚体

- Abaqus通过组成刚体单元的质量分布确定质心
  - 如果参考节点被重新放置于刚体的质心，在打印输出文件的结尾处，参考节点的新坐标将被输出。
  - 句法:

```
*RIGID BODY, REF NODE=node, ELSET=element set,
POSITION=CENTER OF MASS
```


© Dassault Systèmes, 2008

刚体

- **刚体的惯性属性**
  - 基于单元的贡献，可以计算划分网格刚体的质量和转动惯量；或者可以使用刚体从属节点或刚体参考节点定义的MASS和ROTATYI单元指定质量和转动惯量。
    - 每个刚体的质量、质心和关于质心的惯性矩将出现在打印输出 (**.dat**) 文件中.

© Dassault Systèmes, 2008

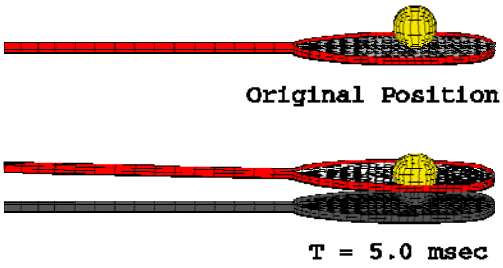
刚体

- 出于确认的目的，可以指定部分的模型为刚体，借以帮助分析。
  - 比如，在复杂模型中，不能预测所有潜在的接触条件。可以把距离所关心区域较远的区域的单元作为刚体，这样在建立模型时，可以加速运行时间。
  - 当用户满意模型和接触对的定义之后，可以将刚体定义删除，使用精确的可变形单元对模型进行描述.

© Dassault Systèmes, 2008

刚体

- 以刚体作为验证目的的例子



Video Clip

网球拍撞击网球

最关心的是球与线之间的相互作用。因为框非常刚硬，出于计算效率的考虑，开始时把框模拟为刚体。在球与线之间的分析完成之后，将刚体定义删除，使球拍成为可变形体

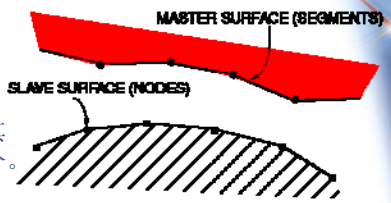
© Dassault Systèmes, 2008

接触

© Dassault Systèmes, 2008

接触

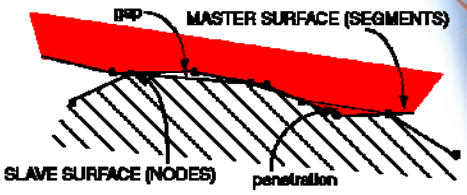
- **Abaqus/Standard 中处理接触问题需要定义接触对**
  - 双面接触对
  - “点对面” 是默认的约束处理方法:
    - 表面上的节点（从属表面）接触其它表面（主控表面）离散化的部分。
    - 严格的主从格式
  - 另外一种是面对面约束方法



© Dassault Systèmes, 2008

接触

- **Abaqus/Standard中使用的严格的主/从计算公式具有许多假设**
  - 从属表面的节点不能穿透到主控表面。
  - 主控表面的节点可以穿透到从属表面



  - 接触方向通常是主面的法向
    - 沿着主面法向检查接触状态
    - 法向接触力通过法向传递。
    - 摩擦力通过接触面的切向传递.

© Dassault Systèmes, 2008

接触

- **Abaqus/Explicit 提供两种方式定义接触**
  - 通用接触
    - 一般在模型中存在多个部件或者复杂的拓扑结构情况下使用
    - 自动定义相互作用面
  - 双面接触
    - Explicit中一般使用平衡主从接触定义.
      - 接触约束施加两次并平均，在第二施加接触约束时候互换主从面
      - 尽可能的减少接触对之间的倾彻.

© Dassault Systèmes, 2008

接触

- **接触中壳厚度的考虑**
  - 默认的，除了finite-sliding, node-to-surface 接触，Abaqus在接触计算中考虑壳的厚度.
  - 为了忽略壳厚度的影响，可在*CONTACT PAIR选项中使用NO THICKNESS参数

© Dassault Systèmes, 2008

定义双面接触

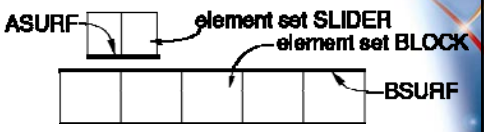
- **定义接触的三个步骤:**
  1. 基于表面下面的单元、解析几何体、节点定义表面.
  2. 定义接触对.
  3. 定义接触属性：摩擦、软化层等.

© Dassault Systèmes, 2008

定义双面接触

- 输入文件中，完整定义接触句法的例子：

```
*SURFACE, NAME=ASURF
SLIDER, S1
*SURFACE, NAME=BSURF
BLOCK, S3
*CONTACT PAIR, INTERACTION=FRIC1
ASURF, BSURF
*SURFACE INTERACTION, NAME=FRIC1
1.0,
*FRICTION
0.4,
```

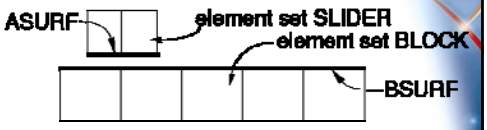


下面将详细的解释这些选项.

© Dassault Systèmes, 2008

定义双面接触

- **定义表面**
  - 利用*SURFACE选项定义表面。
  - 利用每个单元集的表面标识符指定面。
  - 可以用单元集的名字或单元号指定表面。

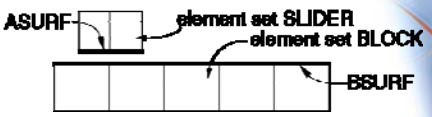



```
*SURFACE, NAME=ASURF
SLIDER, S1        ←——— 接触发生在单元集SLIDER的底部 (S1)
*SURFACE, NAME=BSURF
BLOCK, S3         ←——— 接触发生在单元集的BLOCK顶部 (S3)
```


© Dassault Systèmes, 2008

定义双面接触

- **定义可以接触的表面对**
  - 定义表面之后，就可以定义“接触对”。
  - 在分析过程中，接触对指定了两个相互接触的表面。
    - 在Abaqus/Standard中，第一个表面为从属表面，第二个表面为主控表面
    - 在Abaqus/Explicit中，表面的顺序一般不影响接触计算.




```
*CONTACT PAIR, INTERACTION=FRIC1
ASURF, BSURF
```


© Dassault Systèmes, 2008

定义双面接触

- **定义表面接触属性**
  - *SURFACE INTERACTION选项定义表面接触属性。
    - 定义表面行为属性，比如摩擦。
    - 对于二维问题，定义接触面在垂直平面方向的厚度。
    - 即使没有声明附加属性，在Abaqus/Standard中，该选项也是必须的。
      - 在Abaqus/Explicit中，该选项是可选的.

© Dassault Systèmes, 2008

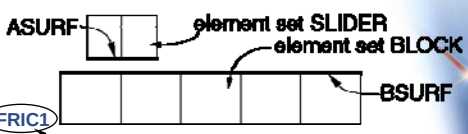
定义双面接触

- *CONTACT PAIR选项通过*SURFACE INTERACTION选项中的名字引用接触属性。

```
*CONTACT PAIR, INTERACTION=FRIC1
ASURF, BSURF
*SURFACE INTERACTION, NAME=FRIC1
1.0,
*FRICTION
0.4,
```

垂直平面方向的厚度

*SURFACE INTERACTION的子选项，列出表面的基本属性



© Dassault Systèmes, 2008

定义面的规则

© Dassault Systèmes, 2008

定义面的规则

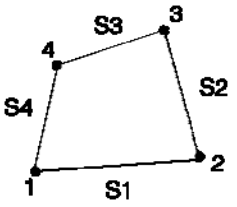
- 在变形体或划分网格的刚体上，使用*SURFACE, TYPE=ELEMENT选项定义表面。
  - 通过指定单元表面标识符定义表面。
    - 或者
  - 让Abaqus自动确定连续体单元的“自由表面”。
- 使用*SURFACE, TYPE=[SEGMENTS | CYLINDER | REVOLUTION] 选项和*RIGID BODY选项定义解析刚体表面。
- 使用*SURFACE, TYPE=NODE选项，定义可能接触的节点表面.

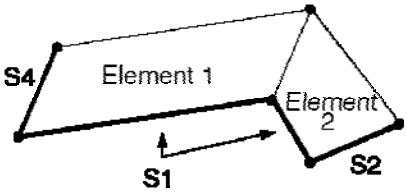
© Dassault Systèmes, 2008

定义面的规则

- **在实体单元上定义表面**
- **使用表面标识符**
  - 例子： 4-节点四面体单元 (CPE4, CAX4, 等等)

```
*SURFACE, NAME=EXAMPLE1
1, S4
1, S1
2, S1
2, S2
...
```





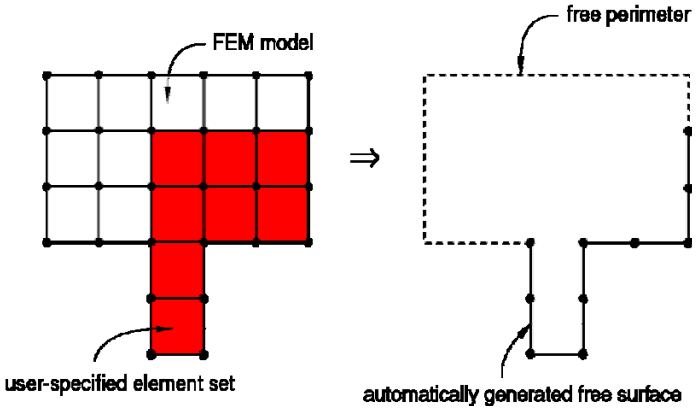
© Dassault Systèmes, 2008

定义面的规则

- **使用自动的表面定义**

```
*SURFACE, NAME=EXAMPLE2
ELSET1,
```

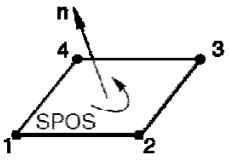
没有面的标识符



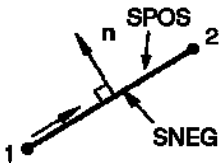
© Dassault Systèmes, 2008

定义面的规则

- **在结构单元上定义表面 (壳、薄膜、刚体、梁)**
  - 结构单元的法向表示预期的接触方向。
  - 法向基于单元局部的节点编号。
    - 正的法线方向在 = SPOS 表面。
    - 负的法线方向在 = SNEG 表面。



壳和薄膜
(S4R,S8R,M3D4,等)



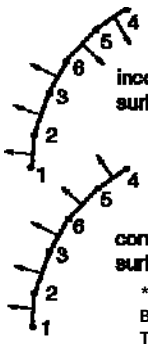
2-D 桁架和梁
(B21,T2D2,等)

© Dassault Systèmes, 2008

定义面的规则

- 在所定义的表面集之内，表面法向的定义必需一致.

```
*ELEMENT, TYPE=B21, ELSET=BOTTOM
10, 1, 2
11, 2, 3
12, 3, 6
*ELEMENT, TYPE=B21, ELSET=TOP
20, 4, 5
21, 5, 6
*ELSET, ELSET=BEAMS
BOTTOM, TOP
```



inconsistent surface normals

```
*SURFACE, NAME=SURF1
BEAMS, SPOS
```


consistent surface normals

```
*SURFACE, NAME=SURF1
BOTTOM, SPOS
TOP, SNEG
```


© Dassault Systèmes, 2008

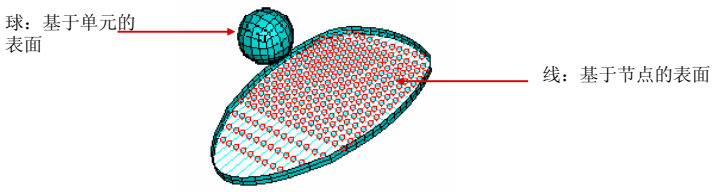
定义面的规则

- **基于节点的表面**
  - 为接触定义点的另一种方法。
    - 与指定单元表面定义接触表面的方法不同，基于节点的表面只包含节点
    - 基于节点的表面一般被用作从属表面.

© Dassault Systèmes, 2008

定义面的规则

- 基于节点表面的例子




```
*SURFACE, TYPE=NODE, NAME=STRINGS
STRINGS,
*CONTACT PAIR, INTERACTION=SMOOTH
STRINGS, BALL
```

定义包含接触节点的表面

前面定义的表面

© Dassault Systèmes, 2008

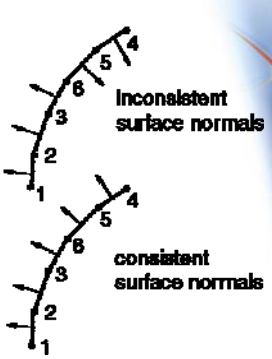
定义面的规则

- **表面定义的限制**
  - 接触表面下所有的单元必须协调。它们必须：
    - 维数相同（二或三维）。
      - 对于二维表面：所有单元为平面的或轴对称的（不可以两种都有）。
    - 插值阶数相同（一或二阶）。
    - 所有单元为可变形的或所有单元为刚体（不可以两种都有）.

© Dassault Systèmes, 2008

定义面的规则

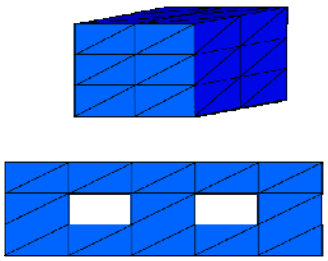
- **附加的限制**
  - 表面法向
    - 主表面法向必须一致
    - 主表面法向必须指向从面.
      - 否则将会出现收敛问题
  - 刚性表面
    - 刚体表面必须是主面.



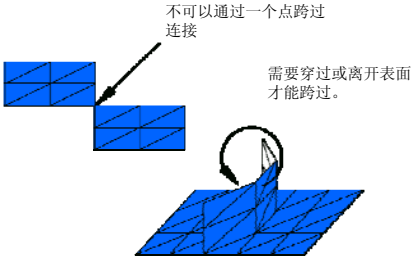
© Dassault Systèmes, 2008

定义面的规则

- 对于Abaqus/Standard中的主控表面和Abaqus/Explicit中的所有表面，还有附加的限制：
  - 在不离开表面、穿过表面或通过单个点穿越表面的前提下，必须可以连接表面中的两个任意的点.

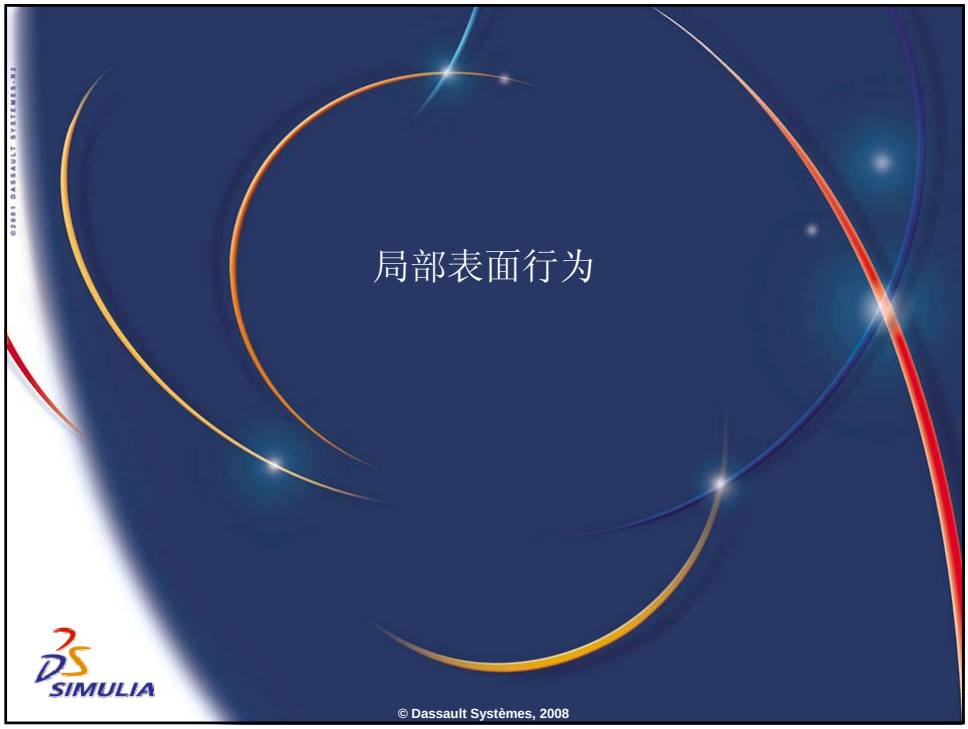


合理的主表面



不合理的3-D主表面

© Dassault Systèmes, 2008



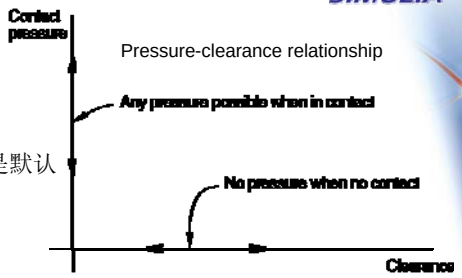
局部表面行为

- 一般的接触建模包括不同方向的接触相互作用：
  - 主控表面的法向
  - 表面的切向
- 其他的接触属性包括接触阻尼和几何属性等.

© Dassault Systèmes, 2008

局部表面行为

- **接触法向行为**



- **硬接触**
  - 在所有接触问题中，“硬” 接触是默认的局部行为。
  - 接触约束可以通过
    - 默认方法：

      Lagrange Multipliers for Abaqus/Standard

      Precise kinematic compliance for Abaqus/Explicit (two-surface contact only)

    - 罚函数方法

      ```
      *surface interaction, name=...
      *surface behavior, penalty
      ```
      (Abaqus/Standard)

      ```
      *contact pair, mechanical=penalty
      ```
      (Abaqus/Explicit)

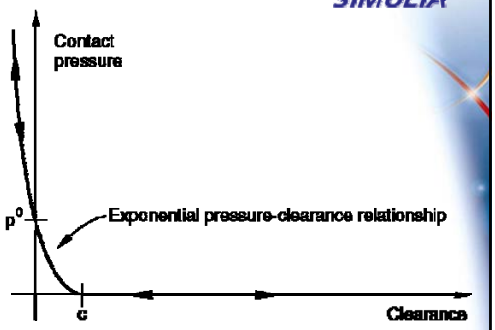
    - 增广拉格朗日

      ```
      *surface behavior, augmented lagrange
      ```
      (Abaqus/Standard only)

© Dassault Systèmes, 2008

局部表面行为

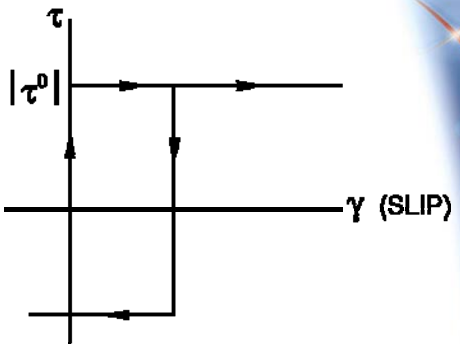
- **除硬接触外，其它的方法**
  - 作为*SURFACE INTERACTION选项的子选项*SURFACE BEHAVIOR选项用于指定：
    - 软接触（指数或表格方式表达的压力-间距关系）
    - 没有分离的接触
  - 其它选项：
    - 间距相关的粘性阻尼 (*CONTACT DAMPING)。
    - 带有过盈或拉伸接触力的接触 (*CONTACT CONTROLS; 只有Abaqus/Standard可用)。



© Dassault Systèmes, 2008

局部表面行为

- **接触的切向行为**
  - 两个实体之间的接触面可能存在*摩擦*剪切应力τ。
  - 如果剪切应力达到某个临界值，实体之间可能会发生相对滑动；否则它们粘在一起。



© Dassault Systèmes, 2008

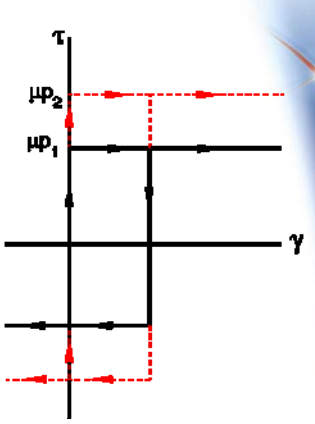
局部表面行为

- 摩擦可以引起高度的非线性效应。
  - 很难获得解。
  - 除非物理上必要，尽量不要使用。
- 摩擦是非保守的。
  - 在Abaqus/Standard中，摩擦将导致系统方程的不对称。 对于较大的摩擦系数μ (μ >0.2)，*STEP, UNSYMM=YES选项将自动启用。
  - 使用UNSYMM=NO将减缓收敛速度，但是解将是正确的（如果得到）。它可能会使用更少的磁盘空间。
  - 在Abaqus/Explicit中，过大的摩擦系数将不会引起类似的问题，因为不需要求解系统方程组.

© Dassault Systèmes, 2008

局部表面行为

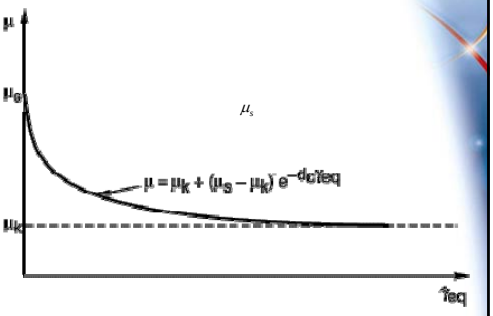
- 默认情况下，Abaqus使用库伦摩擦模型。
  - 临界摩擦应力取决于接触压力： $\tau cr = \mu p$。
  - 基本句法：
    - ***FRICTION**
    - μ
  - 摩擦系数μ可以是相对滑动速度、压力、温度和场变量的函数$(\dot{\gamma}_{eq}, p, \theta, f_i)$.
  - 出于计算的考虑， Abaqus/Standard中默认的摩擦模型对理想行为作一定的近似：在达到不可恢复滑动发生之前，允许小量的弹性滑动。



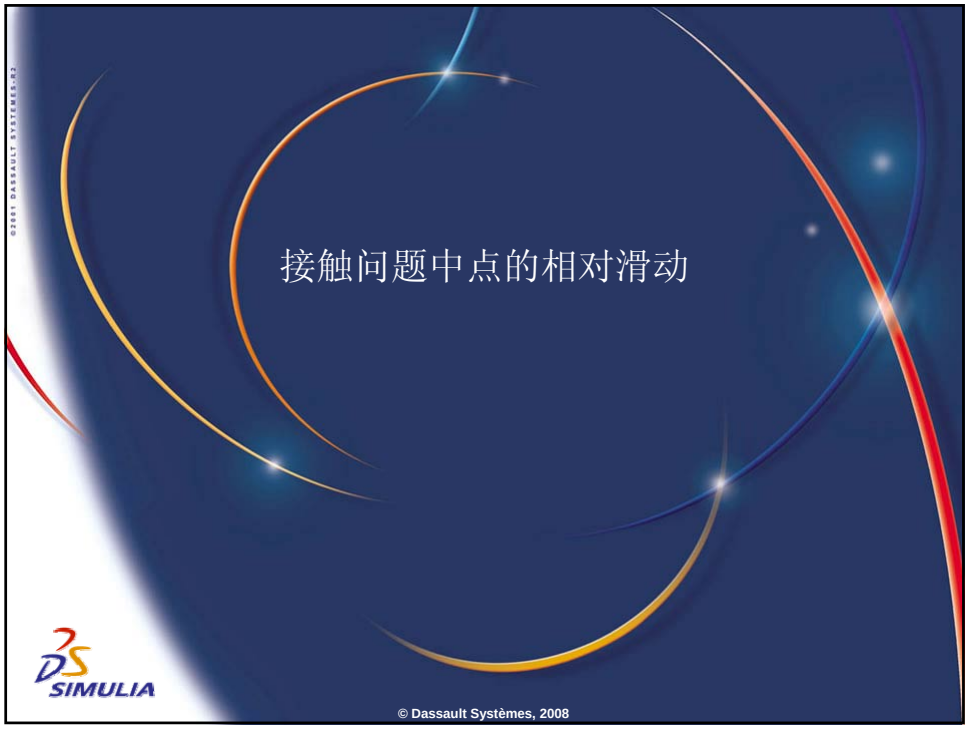
© Dassault Systèmes, 2008

局部表面行为

- 可以定义混合的静动摩擦模型。
  - 作为滑动速度 $\dot{\gamma}$, 函数的摩擦系数，从μ_s（静摩擦系数）到μ_k（动摩擦系数）以指数方式衰退。
- 还有其它可用的摩擦行为，包括用户自定义的摩擦模型。：
- (在Abaqus/Standard中为**FRIC**在Abaqus/Explicit中为**VFRIC**)。



© Dassault Systèmes, 2008



接触问题中点的相对滑动

- **两种滑动距离选项：**
  - 有限滑动：
    默认为有限滑动，有限滑动是最通用的滑动方式。
    在接触表面之间，允许任意大的滑动和旋转。

    ```
    *CONTACT PAIR
    ```

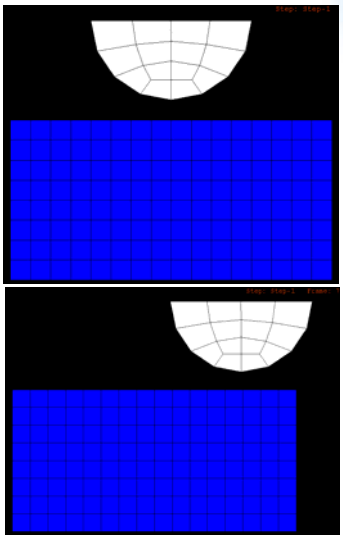
  - 小滑动：
    在接触表面之间，允许小的相对滑动。
    只要接触表面之间没有大的相对移动，允许接触表面之间有大的转动。
    相比有限滑动，具有较小的计算费用

    ```
    *CONTACT PAIR,
    SMALL SLIDING
    ```


© Dassault Systèmes, 2008

接触问题中点的相对滑动

- **下面的例子将说明有限滑动和小滑动的区别。**
  - 考虑右图的模型。
  - 刚体冲头产生水平位移，并保持图形中的间距。然后，向下推向变形体。
  - 如果使用有限滑动，在保持间距时，没有接触发生（与期望的一样）

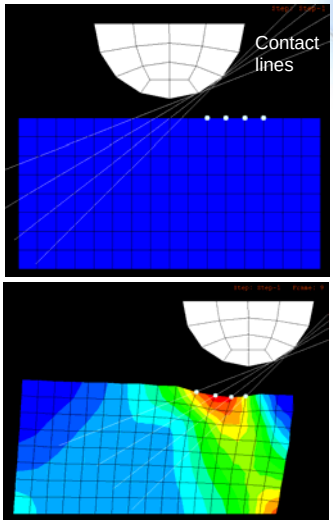


Finite sliding

© Dassault Systèmes, 2008

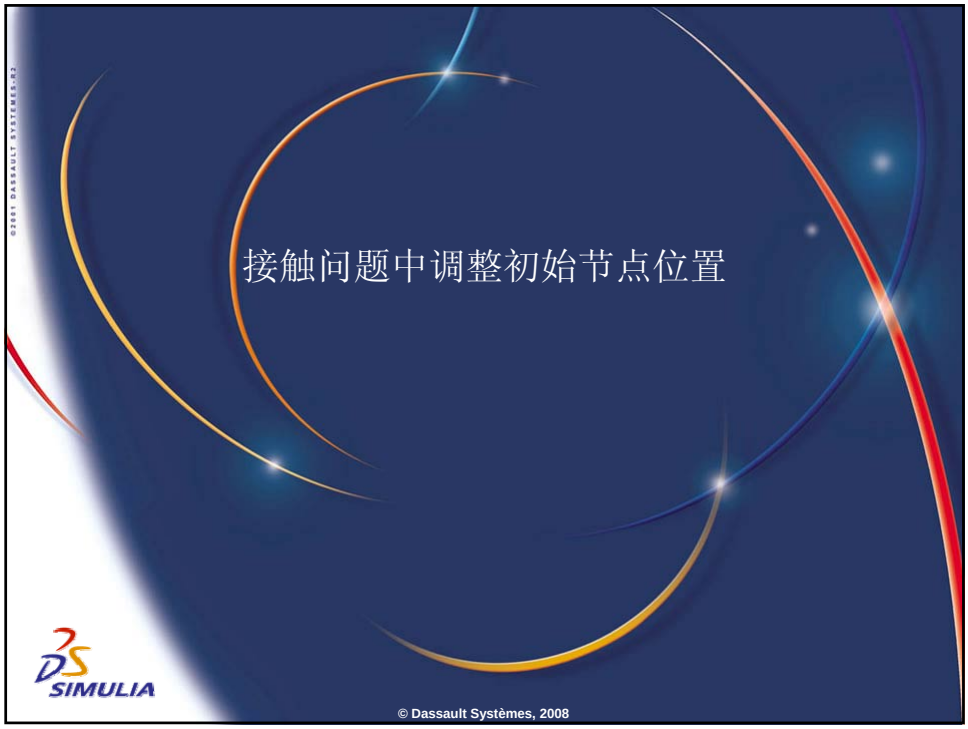
接触问题中点的相对滑动

- 考虑小滑动的情况。
- 如果是小滑动，接触行为由局部接触平面（三维）或线（二维/轴对称）控制。
- 如右图所示，局部接触线被高亮显式。
- 在冲头水平移动的过程中，尽管冲头和变形物体之间还存在物理间距，右图中高亮显式的从属节点根据局部接触线仍然建立接触关系！



Small sliding

© Dassault Systèmes, 2008



接触问题中调整初始节点位置

SIMULIA

- 在分析之前，可以调整接触面中节点的初始位置，并且不产生任何的应力和应变。
- Abaqus/Standard允许用户通过指定绝对距离或节点集的方式调整节点位置。
  - 在分析过程中，由于残留的初始过盈引起从属节点的应变将被施加在接触面上。

```
*CONTACT PAIR, ADJUST=[distance | node set name]
```

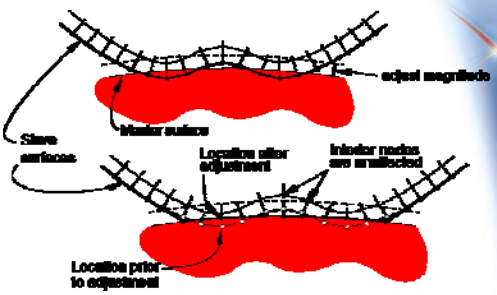

© Dassault Systèmes, 2008

接触问题中调整初始节点位置

- **指定需要调整的绝对距离：**

```
*CONTACT PAIR,
INTERACTION=FRIC1,
ADJUST=a
```

  - 在调整范围内，初始张开的从属节点将被移到主控表面上。
  - 调整距离沿主控表面的法向度量。
  - 所有初始过盈的从属节点被重新放在主控表面上

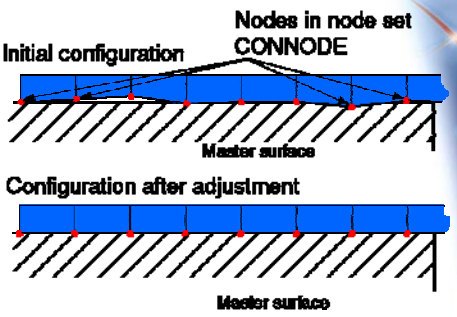


© Dassault Systèmes, 2008

接触问题中调整初始节点位置

- **指定需要调整的从属节点集：**
  - 在分析过程中，不在节点集中并带有初始过盈的从属节点将保持过盈，并产生应变。这些应变将被施加在接触面上。
  - 指定 **ADJUST** node set的例子：

```
*NSET, NSET=CONNODE,
  GENERATE
1, 8, 1
*CONTACT PAIR,
  INTERACTION=RIG,
  ADJUST=CONNODE
```



© Dassault Systèmes, 2008

接触问题中调整初始节点位置

- Abaqus/Explicit不允许接触表面的初始过盈。
  - 在分析之前，接触面上的节点将被自动调整，以删除任何的初始过盈。在随后的分析步中，这样的调整将引起应变。
- 大量的调整将引起初始单元形状的严重扭曲。
  - 如果发现这样的错误信息，最好需要运行**datacheck**分析，并利用Abaqus/Viewer查找问题.

© Dassault Systèmes, 2008

接触输出

© Dassault Systèmes, 2008

接触输出

- **在使用*CONTACT OUTPUT、 *CONTACT FILE或*CONTACT PRINT选项之后，所有接触表面的接触信息都是可用的。**
  - 对于Abaqus/Standard和Abaqus/Explicit：
    - 接触应力：CSTRESS (接触压力CPRESS和摩擦剪切应力CSHEAR1和 CSHEAR2)
  - 对于Abaqus/Standard，还可以得到：
    - 接触位移： CDISP (接触张开距离COPEN和相对切向运动CSLIP1和CSLIP2)
  - 对于Abaqus/Explicit，还可以得到：
    - 滑动速度： FSLIPR
    - 累计的滑动距离：FSLIP

© Dassault Systèmes, 2008

接触输出

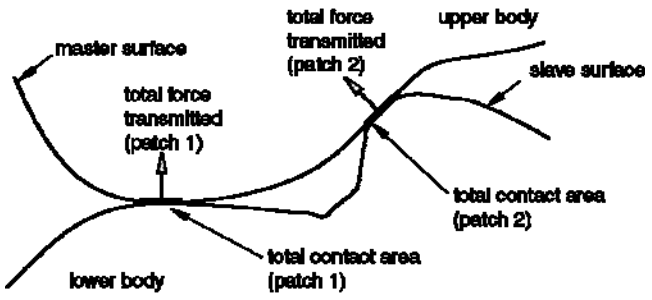
- **接触输出包括：**

输出变量	描述
CAREA	接触中从属表面的全部面积
CFN CFS	由从属表面上的接触压力和摩擦剪应力各自引起的全部力向量
CMN CMS	由从属表面上的接触压力和摩擦剪应力各自引起的、关于原点的全部弯矩向量
CFT	CFN 和 CFS的向量和
CMT	CMN 和 CFS的向量和
XN	点的坐标，从属表面上的接触压力对该点的全部弯矩为零
XS	点的坐标，从属表面上的摩擦力对该点的全部弯矩为零
XT	点的坐标，从属表面上的接触压力和摩擦力对该点的全部弯矩为零

© Dassault Systèmes, 2008

接触输出

- 例如：两个表面在两个位置接触



total force = total force of patch 1 + total force of patch 2
total area = total area of patch 1 + total area of patch 2
total moment = total moment of patch 1 + total moment of patch 2

© Dassault Systèmes, 2008

接触输出

- 与Abaqus/Standard中接触分析相关的，用于生成与打印输出相关的两个其它选项为：

*PREPRINT, CONTACT=YES:

- 在预处理过程中，控制打印到输出文件的输出
- 给出内部生成的接触单元的细节
- 推荐用于小滑动接触问题，用于验证主-从节点的相互作用
- 用于检测表面定义和相互接触的正确性

*PRINT, CONTACT=YES:

- 在分析过程中，用于控制信息文件的输出
- 给出迭代过程的细节
- 用于分析在接触计算过程中出现的数值问题

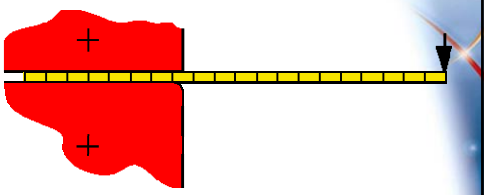
© Dassault Systèmes, 2008

习题4: 简单接触

© Dassault Systèmes, 2008

习题4: 简单接触

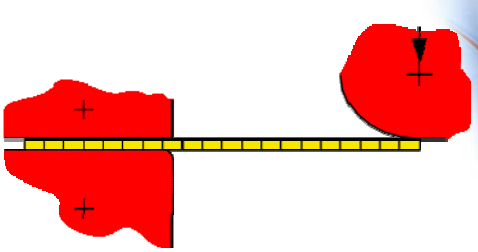
- **习题任务**
  1. 通过刚体表面约束悬臂梁一端的底部，并进行分析。
  2. 为下部刚体表面添加导角，减小接触侵彻。
  3. 添加上部刚体表面，对梁进一步施加约束。



© Dassault Systèmes, 2008

习题4: 简单接触

4. 添加刚体表面，移动梁的另一端。



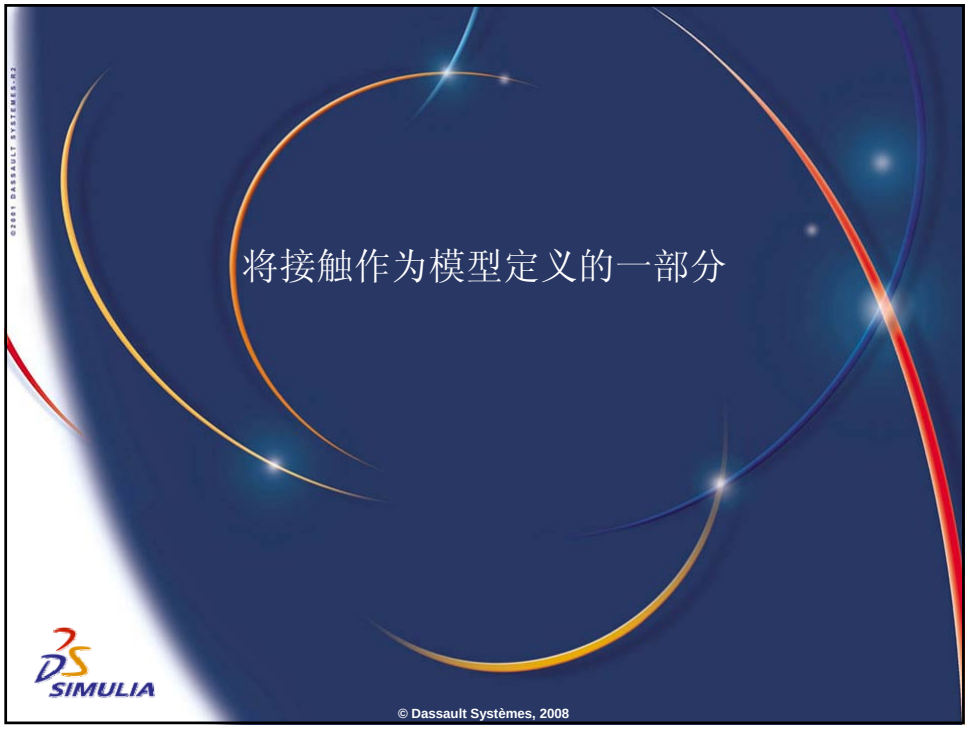
Video Clip

© Dassault Systèmes, 2008

简介

- 将接触作为模型定义的一部分
- 主/从方法中网格密度的考虑
- Abaqus/Standard中的接触逻辑
- 习题5：密封接触

© Dassault Systèmes, 2008



将接触作为模型定义的一部分

SIMULIA

- 在Abaqus/Standard 整个contact定义作为模型数据必须在step之前
- 接触对在分析历程中可以使用*MODEL CHANGE选项激活或者不激活

```
*MODEL CHANGE, TYPE=CONTACT PAIR, [ADD | REMOVE]
surface_1, surface_2
```

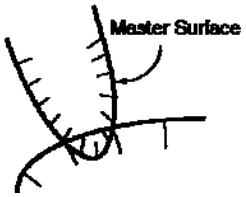

© Dassault Systèmes, 2008

在严格的Master/Slave定义中要考虑网格密度

© Dassault Systèmes, 2008

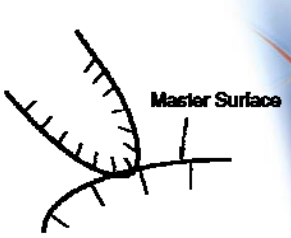
在严格的Master/Slave定义中要考虑网格密度

- 从属表面应该是网格划分得更精细的表面.
- 如果网格密度相近似，从属表面应由更为柔软的材料组成.



Incorrect

Master surface placed on fine mesh ⇒
Gross penetration into slave surface.



Correct

Master surface placed on coarse mesh ⇒
Minimal penetration into slave surface.

Video Clip

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **对于点接触需要施加约束**
  - 有多种不同的约束施加方法.
  - 缺省情况下, Abaqus/Standard使用拉格朗日多项式方法.
  - 对于每个潜在的接触点，接触条件通过单个的、非线性的、不等式约束描述：

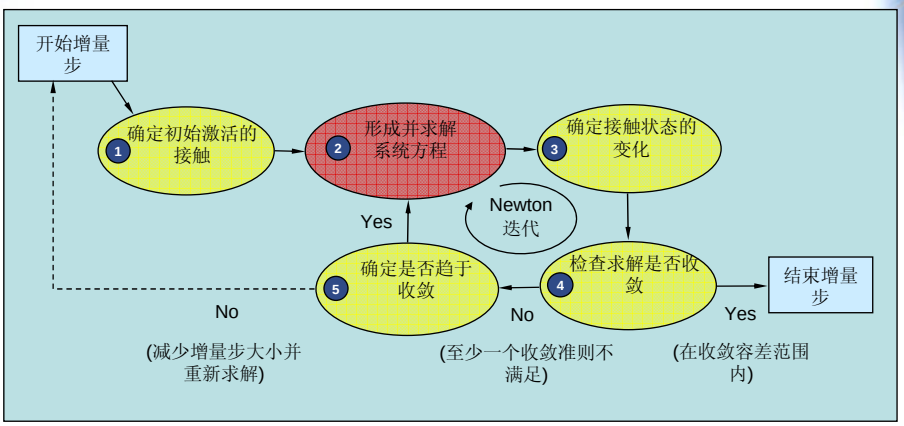
$$h(u^1, u^2, u^3, ...) \le 0,$$

其中h 为“侵彻”， u^N 为自由度

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **一个增量步内状态的原理图（默认状态）**



© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **默认的接触算法（参考前页的流程图）**

1. 确定每个点当前的接触状态（接触或张开）
   - 对于一个分析步的第一个增量步，基于初始模型的状态；否则，基于求解的外插
2. 计算刚度，施加相应的接触约束的影响。形成系统方程并求解。
3. 接触压力和间隙是否与接触状态一致。
   - 接触状态改变（张开/闭合或粘结/滑移）通常导致系统方程的重大变化。
   - 由接触状态改变产生的迭代被标记为严重不连续迭代(SDIs)

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

4. 是否达到收敛？
   - 默认的， Abaqus量化与SDIs关联的接触不相容性。那些关于接触状态的不相容性必须十分小才能达到收敛。
   - 也必须确保力的剩余误差和求解修正很小才能达到平衡
   - 如果接触状态和平衡条件满足各自的收敛准则，这个增量步完成
5. 如果没有达到收敛，它可能会收敛吗？
   - Abaqus估算趋势，例如连续迭代中接触状态改变的次数，以确定是否继续迭代或减小增量步长重算
   - 如果可能收敛，更新基于3的接触条件和刚度，重新求解系统方程；否则，利用更小的增量步重试。

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **接触打印输出例子**
  - 参考: Example Problem 1.3.4, “Deep drawing of a cylindrical cup”
  - 状态 (**.sta**) 文件:

```
SUMMARY OF JOB INFORMATION:
 MONITOR NODE:     200  DOF:  2
 STEP  INC ATT SEVERE EQUIL TOTAL  TOTAL      STEP       INC OF       DOF    IF
               DISCON ITERS ITERS  TIME/     TIME/LPF   TIME/LPF    MONITOR RIKS
               ITERS              FREQ
   1     1   1     1     1     2  1.00        1.00       1.000       0.000
   2     1   1     0     1     1  2.00        1.00       1.000       0.000
   3     1   1    10     0    10  2.01        0.0100     0.01000    -0.000600
   3     2   1     7     1     8  2.02        0.0200     0.01000    -0.00120
   3     3   1U   10     0    10  2.02        0.0200     0.01500    -0.00120
   3     3   2     5     0     5  2.02        0.0238     0.003750   -0.00142
   3     4   1     3     1     4  2.03        0.0294     0.005625   -0.00176
   3     5   1     2     3     5  2.04        0.0378     0.008438   -0.00227
   3     6   1     4     1     5  2.05        0.0505     0.01266    -0.00303
   3     7   1     4     1     5  2.07        0.0695     0.01898    -0.00417
   3     8   1     6     1     7  2.10        0.0979     0.02848    -0.00588
   3     9   1U    8     0     8  2.10        0.0979     0.04271    -0.00588
   3     9   2     4     3     7  2.11        0.109      0.01068    -0.00652
   3    10   1     2     3     5  2.12        0.125      0.01602    -0.00748
   3    11   1     4     2     6  2.15        0.149      0.02403    -0.00892
   .
   .
   .
```


© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **Message文件, Step 3, Increment 6:**

```
INCREMENT     6 STARTS. ATTEMPT NUMBER  1, TIME INCREMENT  1.266E-02
 CONTACT PAIR (ASURF,BSURF) NODE 167 IS NOW SLIPPING.
 CONTACT PAIR (ASURF,BSURF) NODE 171 IS NOW SLIPPING.
 :
 :
 :
 :
 CONTACT PAIR (ASURF,BSURF) NODE 153 OPENS. CONTACT PRESSURE/FORCE IS -845822..
 CONTACT PAIR (ASURF,BSURF) NODE 161 OPENS. CONTACT PRESSURE/FORCE IS -1.50656E+006.
 CONTACT PAIR (ASURF,BSURF) NODE 163 OPENS. CONTACT PRESSURE/FORCE IS -108355..
 CONTACT PAIR (ASURF,BSURF) NODE 165 OPENS. CONTACT PRESSURE/FORCE IS -620880..
 CONTACT PAIR (CSURF,DSURF) NODE 363 OPENS. CONTACT PRESSURE/FORCE IS -3.5893E+006.
 CONTACT PAIR (ESURF,FSURF) NODE 309 IS NOW SLIPPING.
                6 SEVERE DISCONTINUITIES OCCURRED DURING THIS ITERATION.
                5 POINTS CHANGED FROM CLOSED TO OPEN
Due to slip reversal ⟶ 1 POINTS CHANGED FROM STICKING TO SLIPPING
```

从节点滑移：如果应用Lagrange摩擦或发生滑移反转，将产生SDIs。

*PRINT, CONTACT=YES，将细节打印输出。
(对于错误信息的处理非常有用)

假定的接触状态中不相容的被检测到 → SDI

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **Message文件, Step 3, Increment 6 (cont'd):**

```
          CONVERGENCE CHECKS FOR SEVERE DISCONTINUITY ITERATION     1

  MAX. PENETRATION ERROR -8.16193E-009 AT NODE 331 OF CONTACT PAIR (ESURF,FSURF)
  MAX. CONTACT FORCE ERROR -4369.44 AT NODE 363 OF CONTACT PAIR (CSURF,DSURF)
        THE ESTIMATED CONTACT FORCE ERROR IS LARGER THAN THE TIME-AVERAGED FORCE.

AVERAGE FORCE                    5.393E+03   TIME AVG. FORCE       3.147E+03
LARGEST RESIDUAL FORCE          -1.110E+04   AT NODE        333   DOF  2
LARGEST INCREMENT OF DISP.      -7.782E-04   AT NODE        329   DOF  2
LARGEST CORRECTION TO DISP.     -1.737E-05   AT NODE        337   DOF  2
        FORCE     EQUILIBRIUM NOT ACHIEVED WITHIN TOLERANCE.

AVERAGE MOMENT                     114.      TIME AVG. MOMENT       89.8
ALL MOMENT    RESIDUALS ARE ZERO
LARGEST INCREMENT OF ROTATION    1.853E-33   AT NODE        100   DOF  6
LARGEST CORRECTION TO ROTATION   6.489E-34   AT NODE        300   DOF  6
        THE MOMENT    EQUILIBRIUM EQUATIONS HAVE CONVERGED
```

接触状态的收敛性检查

平衡的收敛性检查

不仅是接触的不相容太大，而且是力的平衡也没有达到

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- 需要四次额外的迭代；前三次都是SDIs（包含接触不相容）。
- 在最后的迭代中接触和平衡检查都通过，迭代收敛。

```
          CONVERGENCE CHECKS FOR SEVERE DISCONTINUITY ITERATION     2 ...
          CONVERGENCE CHECKS FOR SEVERE DISCONTINUITY ITERATION     3 ...
          CONVERGENCE CHECKS FOR SEVERE DISCONTINUITY ITERATION     4 ...
          CONVERGENCE CHECKS FOR EQUILIBRIUM ITERATION      1

  MAX. PENETRATION ERROR -1.38869E-014 AT NODE 331 OF CONTACT PAIR (ESURF,FSURF
  MAX. CONTACT FORCE ERROR -0.00111133 AT NODE 331 OF CONTACT PAIR (ESURF,FSURF
        THE CONTACT CONSTRAINTS HAVE CONVERGED.

AVERAGE FORCE                     5.244E+03   TIME AVG. FORCE       3.123E+03
LARGEST RESIDUAL FORCE             -9.24      AT NODE        367   DOF  1
LARGEST INCREMENT OF DISP.        -7.809E-04  AT NODE        129   DOF  2
LARGEST CORRECTION TO DISP.        4.229E-08  AT NODE        137   DOF  2
        THE FORCE     EQUILIBRIUM EQUATIONS HAVE CONVERGED

AVERAGE MOMENT                      109.      TIME AVG. MOMENT       89.0
ALL MOMENT    RESIDUALS ARE ZERO
LARGEST INCREMENT OF ROTATION      1.925E-33  AT NODE        100   DOF  6
LARGEST CORRECTION TO ROTATION     2.049E-35  AT NODE        100   DOF  6
        THE MOMENT    EQUILIBRIUM RESPONSE WAS LINEAR IN THIS INCREMENT
```

No SDIs in this iteration

© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

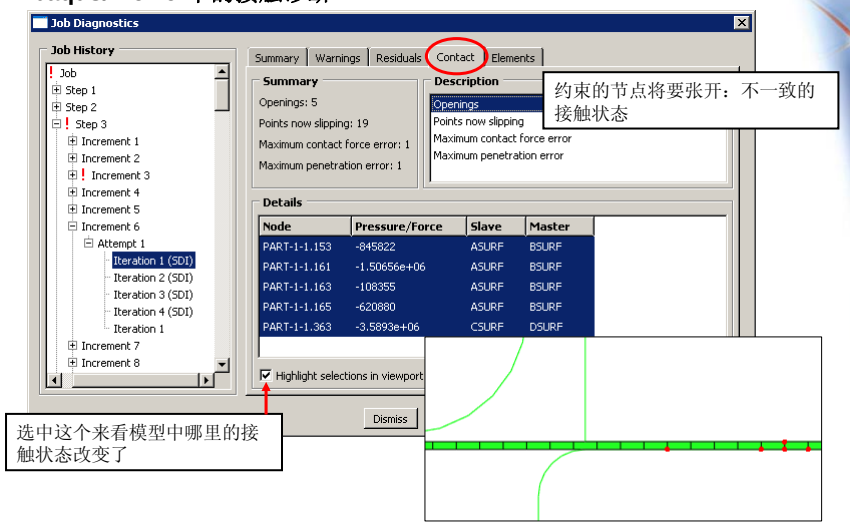
- 增量摘要:

```
ITERATION SUMMARY FOR THE INCREMENT:   5 TOTAL ITERATIONS, OF WHICH
  4 ARE SEVERE DISCONTINUITY ITERATIONS AND  1 ARE EQUILIBRIUM ITERATIONS.

CURRENT VALUE OF MONITOR NODE        200 D.O.F.  2 IS  -3.028E-03

TIME INCREMENT COMPLETED  1.266E-02,  FRACTION OF STEP COMPLETED  5.047E-02
STEP TIME COMPLETED       5.047E-02,  TOTAL TIME COMPLETED         2.05
```


© Dassault Systèmes, 2008

在 Abaqus/Standard中CONTACT规则

- **Abaqus/Viewer中的接触诊断**



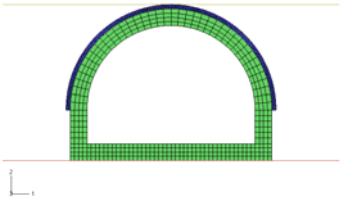
© Dassault Systèmes, 2008

习题5：密封圈接触

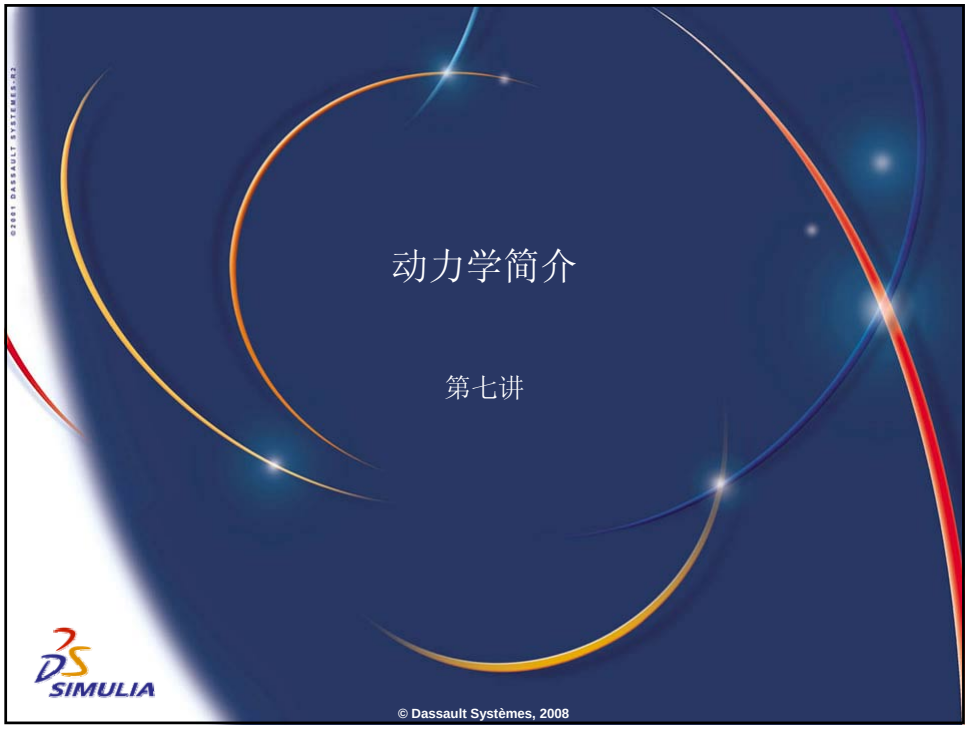
© Dassault Systèmes, 2008

习题5：密封圈接触

- **习题任务**
  1. 评估超弹材料模型
  2. 定义面-面接触
  3. 应用边界条件
  4. 完成大位移分析
  5. 显示结果



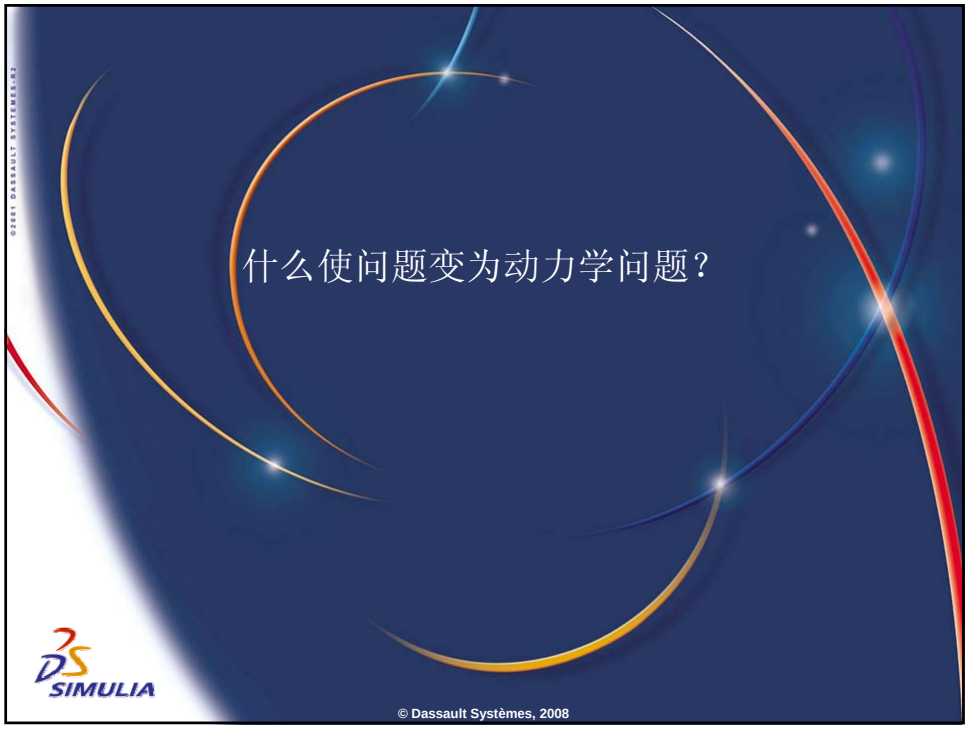
© Dassault Systèmes, 2008



概述

- 什么使问题变为动力学问题？
- 动力学问题方程组
- 线性动力学
- 非线性动力学
- Abaqus/Explicit和Abaqus/Standard比较
- 非线性动力学例子
- 习题6：动力学

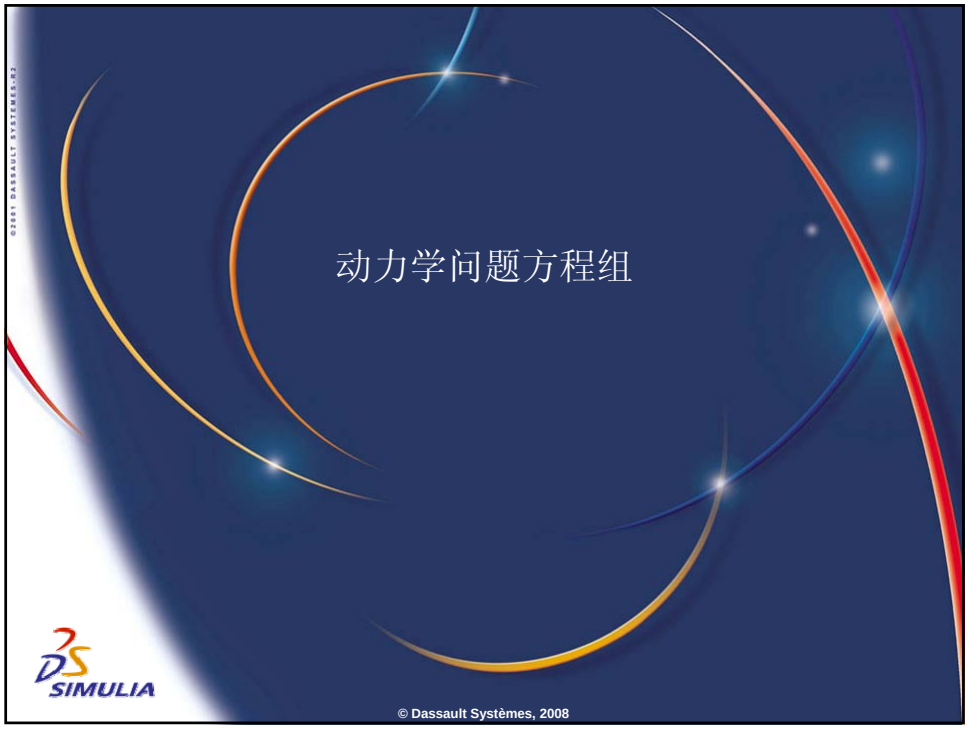
© Dassault Systèmes, 2008



什么使问题变为动力学问题？

- **当惯性力非常大，并且随时间变化较快时，问题将变成动力学问题。**
  - 在结构中，惯性力与质量和加速度成比例。
  - 求解动力学问题需要将运动方程组对时间进行积分。
  - 如果载荷变化得足够慢，惯性力足够小，响应可以认为是准静态的。
  - 有时，虽然有较大的惯性载荷，但是仍然可以作为静态问题处理，因为载荷随时间变化较慢（比如，离心载荷）。
  - 有时，动力学振动问题可以通过频域的办法进行有效的研究。

© Dassault Systèmes, 2008



动力学问题方程组

- **动力平衡**
  - 为了方便，将动力学问题方程组中的惯性力独立于其它力：

$$M\ddot{u} + I - P = 0$$

  - **假设：**
    - M (质量矩阵) 对时间为常数。
    - I 和 P 依赖于节点位移和速度，但是与高阶时间导数无关。
      - 这样，系统关于时间是二阶的，并且阻尼/耗散包含在 I 和 P中。
  - 如果 $I = Ku + C\dot{u}$, 其中 K (刚度) 和 C (阻尼) 为常数，问题就是线性的。

© Dassault Systèmes, 2008

动力学问题方程组

- 这些是最一般的平衡方程。
  - 它们可以应用到任何力学系统，并可以包含所有的非线性行为。
  - 当方程的第一项—惯性力或动态力—足够小，方程组退化为静力平衡形式.

© Dassault Systèmes, 2008

线性动力学

© Dassault Systèmes, 2008

线性动力学

- **自然频率分析**
  - 如果需要研究结构系统的振动特征，经常由自然频率（或特征值）着手。
  - Abaqus/Standard中的*FREQUENCY过程用于提取无阻尼系统的特征值：

$$M\ddot{u} + Ku = 0.$$

  - 特征值和模态形状描述了结构的自由振动或频率成分。
  - 如果使用几何非线性，在计算特征值之前施加的预载将影响计算结果。
    - 如果在*STEP选项中使用NLGEOM参数，Abaqus将计入包括预载等几何非线性效应的影响（预载对K有贡献）。

© Dassault Systèmes, 2008

线性动力学

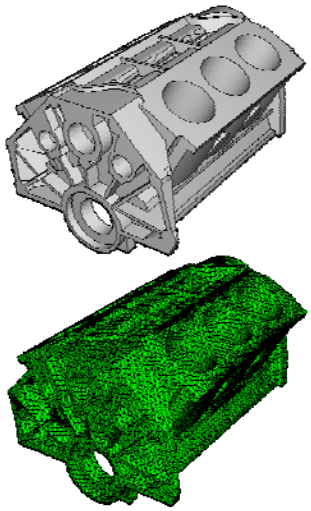
- 对于对称实特征值提取问题，有三种特征值求解器：
  - 自动多层次子结构法（AMS）
    - 当需要大量的特征值时AMS是最高效的求解器
    - AMS是Standard求解器中增加的方法
  - Lanczos（默认）
    - 对于很大的模型，使用SIM结构
  - 子空间迭代求解器
- 结构可以是无约束的，也可以是有约束的。
  - 如果被约束，可以包含预载的影响。

*SIM结构是一个高效的线性动力学结构

© Dassault Systèmes, 2008

线性动力学

- **例子：发动机台的频率提取**
  - 以10节点四面体单元建模 (C3D10)。
  - 线弹性材料模型。
    - 钢材
  - 结构没有约束.



© Dassault Systèmes, 2008

线性动力学

- **分析步定义**

自然频率提取过程

调用基于SIM结构

```
*STEP
*FREQUENCY,EIGENSOLVER=LANCZOS, SIM
100, 1., ,
```

提取模态数量

调用AMS求解器时设置使之等于**AMS**

指定最小频率以排除刚体模态

最大频率为空以保证提取到100阶模态

```
*OUTPUT, FIELD
*NODE OUTPUT
U
*END STEP
```

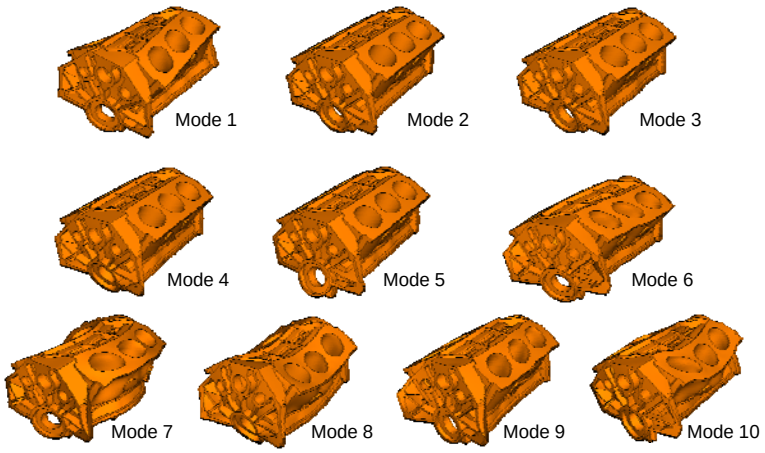
输出节点位移，用于察看模态形状

注意：不需要指定模态数；指定最大频率来代替

© Dassault Systèmes, 2008

线性动力学

- **前10阶非刚体模式的特征值：**



© Dassault Systèmes, 2008

线性动力学

- **模态叠加**
  - 在几种不同的基于模态的求解过程中，结构的特征值可以用于研究结构的动态相应：
    - 模态动力学
      - 在时域内计算结构的线性动力学响应。
      - 可以使用直接积分法。
    - 稳态动力学
      - 计算由谐波激励引起的动态响应。
      - 可以使用直接积分法。
    - 响应谱分析
      - 计算运动过程中的峰值响应。
    - 随机响应分析
      - 计算随机连续激励的响应。

© Dassault Systèmes, 2008

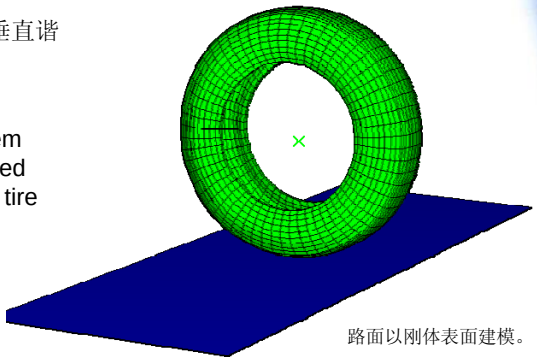
线性动力学

- **稳态动力学**
  - 当具有阻尼的结构承受谐波激励时，它的瞬态响应将很快消失，这些瞬态响应不是关心的重点。
  - 最终，结构达到反映谐波响应特征的稳态状态。
  - 当载荷是谐波激励时，**STEADY STATE DYNAMICS** 过程用于求解运动的线性动力学方程。
  - 稳态动力学分析有以下三种选项：
    - **直接**
    - **基于模态**
    - **子空间投影**

© Dassault Systèmes, 2008

线性动力学

- **例子：轮胎的谐波激励**
  - 研究轮胎的频率响应。
  - 施加在马路参考点的垂直谐波载荷引起的激励。
  - 轮胎的边缘是固定的。
  - 参考: Example Problem 3.1.3, “Subspace-based steady-state dynamic tire analysis.”



路面以刚体表面建模。

© Dassault Systèmes, 2008

线性动力学

- 分析步定义

```
*STEP, NLGEOM=YES
*STATIC
*BOUNDARY
 RIM,  1, 3
 ROAD, 1, 2
 ROAD, 4, 6
*DSLOAD
 INSIDE, P, 200.E3
*CLOAD
 ROAD, 3, 3300.
*END STEP
```

静态预载荷
(“footprint” step)

© Dassault Systèmes, 2008

线性动力学

提取频率

```
*STEP
*FREQUENCY,EIGENSOLVER=LANCZOS
20
*END STEP
```

基于子空间的稳态动力学过程

```
*STEP,NLGEOM=YES
*STEADY STATE DYNAMICS,
SUBSPACE PROJECTION=ALL FREQUENCIES,
INTERVAL=EIGENFREQUENCY, FREQUENCY SCALE=LINEAR
 80,130,3
*CLOAD
  ROAD,3,200.
*END STEP
```

频率范围

载荷是完全同相的：

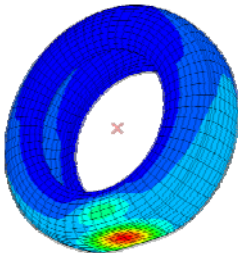
$$F_z \sim \cos \omega t$$

© Dassault Systèmes, 2008

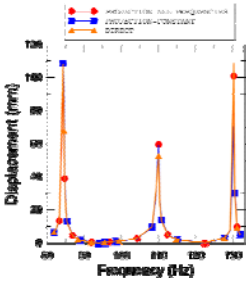
线性动力学

- 结果

Video Clip



位移云图



马路参考点的垂直位移

© Dassault Systèmes, 2008

非线性动力学

© Dassault Systèmes, 2008

非线性动力学

- **运动学方程组的时间积分**
  - 非线性动力学问题需要对运动方程进行直接积分。
  - 空间离散（有限元近似）把偏微分方程组描述的动力学平衡方程变成一组时域内耦合的、非线性、常微分方程组。
  - 利用时间积分求解系统的常微分方程组。
    - 时域内积分这些方程组的方法将区分Abaqus/Explicit和Abaqus/Standard。
    - Abaqus/Standard使用Newmark积分方法，Abaqus/Explicit使用中心差分法。

© Dassault Systèmes, 2008

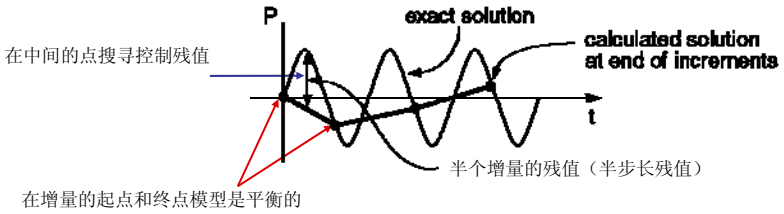
非线性动力学

- Abaqus/Standard
  - 使用二阶精度的、隐式方法：Hilber-Hughes-Taylor (HHT)规则。
    - 这是一般化的Newmark方法。
  - 二阶精度方法意味着该方法能够对常加速度进行精确积分。
  - 该方法是无条件稳定的：可以使用任意的时间增量，并且解仍然是有界的。
- Abaqus/Explicit
  - 使用二阶精度的、显示积分方法。
  - 该方法是条件稳定的—只有在时间增量小于一定的临界值时，才能给出有界的解。

© Dassault Systèmes, 2008

非线性动力学

- **自动时间增量**
  - **Abaqus/Standard**
    - 在增量的中点，求解残值（又称为半步长残值或HAFTOL）的大小控制时间增量的大小。



    - 利用在增量起点和终点的解计算 $\Delta t/2$ 处的残值（计算费用不高）。

© Dassault Systèmes, 2008

非线性动力学

- 在隐式动力学分析中，*DYNAMIC选项的HAFTOL参数控制自动时间增量的求解精度。
- 下面的建议可以得到适当的精度：
  - 对于非耗散系统，使用HAFTOL = F to $10F$，其中 F 为模型中的典型节点力。
  - 对于耗散系统，使用HAFTOL = $10F$ to $100F$。
  - 如果需要更高的精度，需要更小的值（尤其是关心高频响应）。

© Dassault Systèmes, 2008

非线性动力学

- **Abaqus/Explicit**
  - 通过稳定时间增量控制时间增量。
    - 只有时间增量小于临界时间增量或稳定时间增量时，显式动力学分析才能给出有界的解。
  - 模型中的最高阶特征值ω_{max}和最高模态的临界阻尼部分ξ决定了稳定极限：

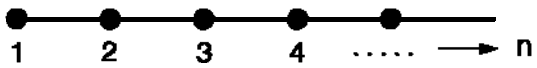
$$\Delta t_{min} \leq \frac{2}{\omega_{max}}\left(\sqrt{1+\xi^2}-\xi\right).$$

    - 阻尼将减小稳定时间增量！
  - 由于计算系统的最高阶特征值ω_{max}是不可行的，所以使用容易计算的保守的估计。

© Dassault Systèmes, 2008

非线性动力学

- 通过一维问题，可以很容易的解释稳定时间增量的概念。



一维问题

- 稳定时间增量是膨胀波通过模型中任何单元的最小时间。
  - 膨胀波包括体积膨胀和收缩。

© Dassault Systèmes, 2008

非线性动力学

- 对于一维问题，膨胀波速c_d可以表示为

$$c_d = \sqrt{\frac{E}{\rho}},$$

其中 E 为杨氏模量， ρ 为当前材料密度。

- 基于当前的几何构型，模型中每个单元的特征长度为 L^e。

© Dassault Systèmes, 2008

非线性动力学

- 这样，稳定时间增量可以表示为

$$\Delta t = \frac{L^e}{c_d}.$$

- 减小 L^e 和/或增加 c_d 将会减小稳定时间增量。
  - 减小单元尺寸将减小 L^e。
  - 增加材料刚度将增加c_d。
  - 减小材料的压缩性将增加c_d。
  - 减小材料密度将增加c_d。
- Abaqus/Explicit在求解过程中监控有限元模型，以确定稳定时间增量。

© Dassault Systèmes, 2008

Abaqus/Explicit和Abaqus/Standard比较

SIMULIA

Abaqus/Standard

- 没有稳定时间增量的限制：完成指定分析，一般需要较少的时间增量。
- 因为每个增量步都需要求解线性方程组，所以每个增量步的费用非常昂贵。

Abaqus/Explicit

- 有稳定时间增量的限制： 完成指定分析，一般需要较多的时间增量。
- 因为不需要求解线性方程组，所以每个增量步的费用较低。
  - 所有的计算费用与单元计算相关（形成和组装 *I* ）。

© Dassault Systèmes, 2008

Abaqus/Explicit和Abaqus/Standard比较

Abaqus/Standard

- 适合于模拟与模型的振动频率相比研究响应周期较长的问题。
  - 由于时间增量的限制，使用显式动力学求解效率很低。
- 用于具有适度非线性的问题，其中非线性是平滑的（比如，塑性）。
  - 如果是平滑的非线性响应，Abaqus/Standard需要较少次数的迭代找到收敛解。

Abaqus/Explicit

- 适合于模拟高速动力学问题
  - 需要较少的时间增量；隐式动力学效率较低。
- 对于包含不连续的非线性问题，一般更加可靠。
  - 接触行为是不连续的，并且包含碰撞。对于隐式时间积分，接触和碰撞将产生求解问题。
  - 其它不连续行为包括屈曲和材料失效。

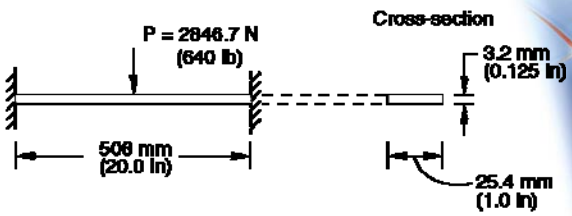
© Dassault Systèmes, 2008

非线性动力学例子

© Dassault Systèmes, 2008

非线性动力学例子

参考：Benchmark Problem 1.3.2, “Double cantilever elastic beam under point load”



Material: elastic

Young's modulus = 206.8 GPa (30×10^6 lb/in^2)

density = 2714 kg/m^3 (2.54×10^{-4} lb s^2/in^4)

© Dassault Systèmes, 2008

非线性动力学例子

Abaqus/Standard输入文件

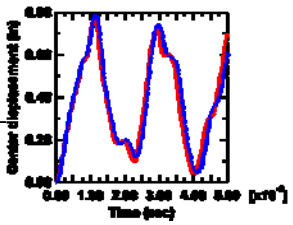
```
*HEADING
 NON-LINEAR ELASTIC BEAM
*NODE
 1, 0.
 6, 10.
*NGEN
 1, 6
*NSET, NSET=NFIL
 6,
*ELEMENT, TYPE=B23
 1, 1, 2
*ELGEN, ELSET=BEAM
 1, 5
*BEAM SECTION, ELSET=BEAM,
 SECTION=RECT, MATERIAL=A1
 1., .125
 0., 0., -1.
 3
*MATERIAL, NAME=A1
*ELASTIC
 30.E6,
*DENSITY
 2.5362E-4,
*RESTART, WRITE, FREQUENCY=10
```

```
*STEP, INC=400, NLGEOM
*DYNAMIC, HAFTOL=10.
 25.E-6, 5.E-3
*BOUNDARY
 1, 1, 2
 1, 6
 6, 1
 6, 6
*CLOAD
 6, 2, 320.
*END STEP
```

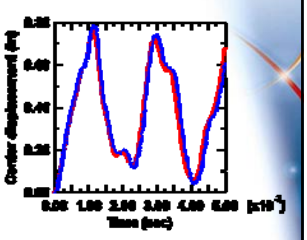

© Dassault Systèmes, 2008

非线性动力学例子

- **比较Abaqus/Standard和Abaqus/Explicit结果**
  - 如果在隐式分析中给出较小的HAFTOL值，显式和隐式求解器得到的解非常的一致。
  - 如果HAFTOL的值较大，隐式的结果将大幅退化。

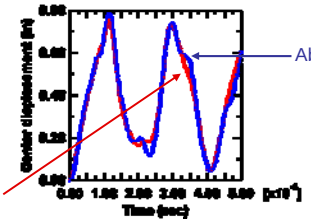


HAFTOL = 10



HAFTOL = 100

Video Clip



HAFTOL = 1000

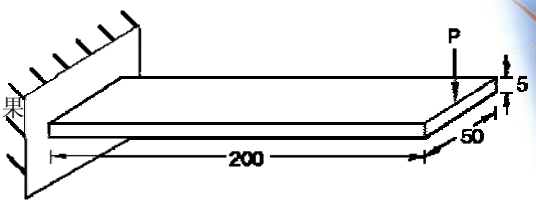
© Dassault Systèmes, 2008

习题6: 动力学

© Dassault Systèmes, 2008

习题6: 动力学

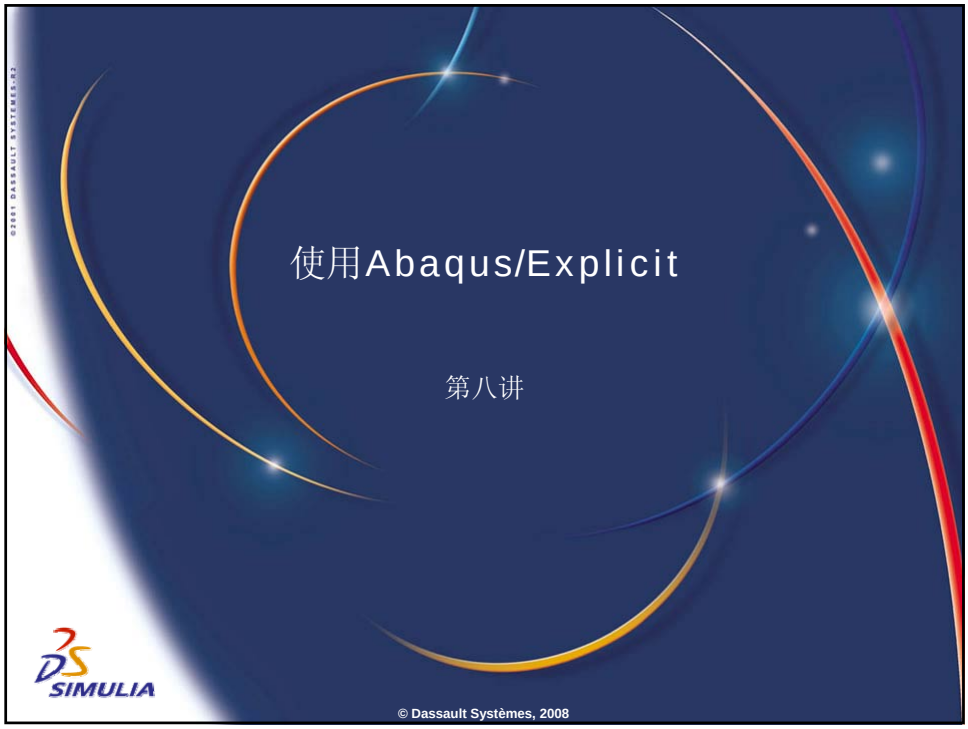
- **习题任务**
  1. 修改已有的输入文件，并提取结构的自然频率。
  2. 察看相关的频率输出结果
  3. 在Abaqus/Viewer观察模态。



  4. 修改输入文件，并使用隐式动力学方法对结构进行自由振动分析。比较两种分析的结构。
  5. 通过重启动分析，扩展求解周期。

Video Clip

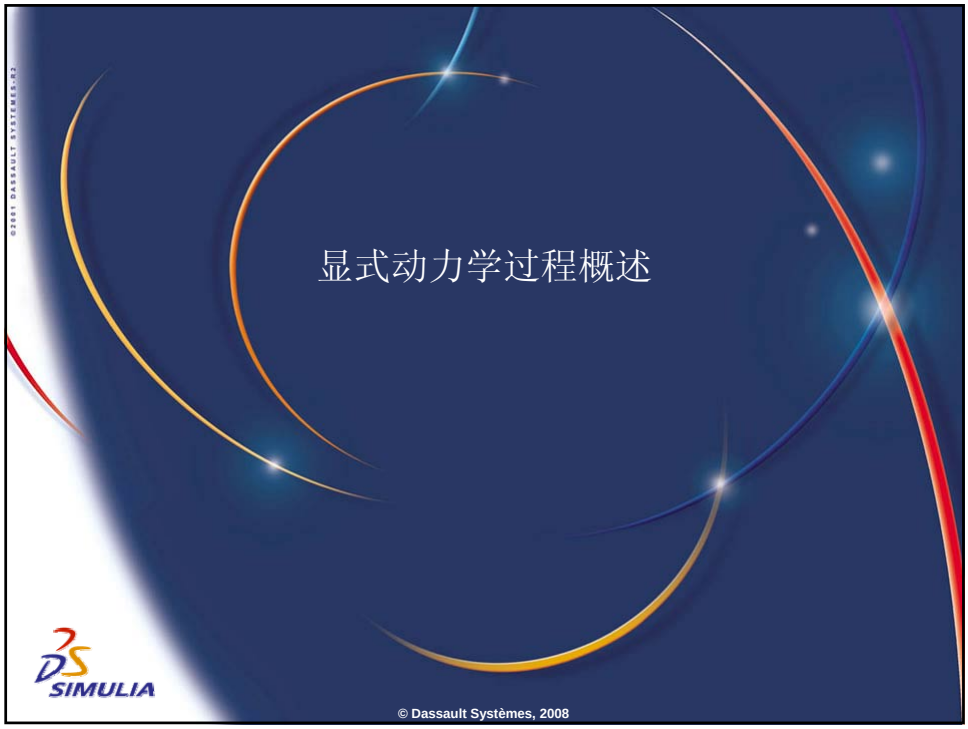
© Dassault Systèmes, 2008



概述

- 显式动力学过程概述
- Abaqus/Explicit句法 Syntax
- 刚体
- 习题6：动力学（继续）

© Dassault Systèmes, 2008



显式动力学过程概述

- **显示动力学求解器与隐式求解器，比如Abaqus/Standard，是互为补充的。从用户的角度出发，隐式与显式方法显著的区别为：**
  - 显式方法需要小的时间增量。
    - 只与模型的最高自然频率相关。
    - 与载荷类型和载荷持续时间无关。
    - 一般的，增量步的数量级为10,000到1,000,000个增量，但是每个增量步内的计算费用相对较小。

© Dassault Systèmes, 2008

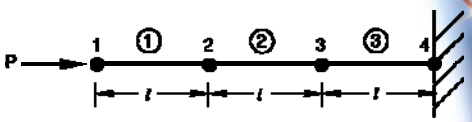
显式动力学过程概述

- 隐式方法对于时间增量没有内在的限制。
  - 增量大小一般由精度和收敛性控制。
  - 同显式方法相比，隐式模拟中的增量步成数量级的少。
  - 然而，因为必须在每个增量步内求解系统总体方程组，隐式方法在每个增量步中的费用比显式方法大很多。
- 知道两种方法的特征之后，在实际分析过程中就可以为所分析的问题选择合适的方法。

© Dassault Systèmes, 2008

显式动力学过程概述

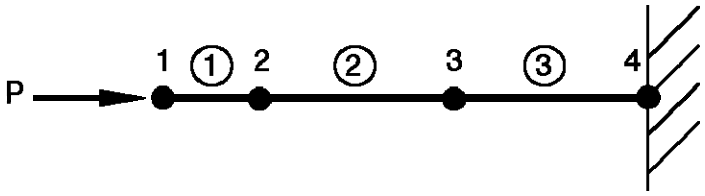
- **应力波的传播**
  - 应力波传播的例子说明了显式动力学方法的求解过程：没有迭代，或求解线性方程组。
  - 考虑应力波沿着三个杆单元传播问题。在时间增加的过程中，研究杆的状态。
    - 质量被集中到节点。



杆的初始构型，自由端有一个集中力P

© Dassault Systèmes, 2008

显式动力学过程概述



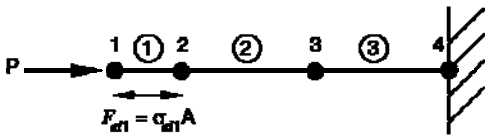
$$\ddot{u}_1 = \frac{P}{M_1} \Rightarrow \dot{u}_1 = \int \ddot{u}_1 dt \Rightarrow \dot{\varepsilon}_{el1} = \frac{-\dot{u}_1}{l} \Rightarrow d\varepsilon_{el1} = \int \dot{\varepsilon}_{el1} dt$$

$$\Rightarrow \varepsilon_{el1} = \varepsilon_0 + d\varepsilon_{el1} \Rightarrow \sigma_{el1} = E\varepsilon_{el1}$$

第一个增量步结束时的构型

© Dassault Systèmes, 2008

显式动力学过程概述

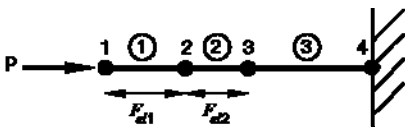


$$\ddot{u}_1 = \frac{P - F_{el1}}{M_1} \Rightarrow \dot{u}_1 = \dot{u}_1^{old} + \int \ddot{u}_1 dt \qquad \dot{\varepsilon}_{el1} = \frac{\dot{u}_2 - \dot{u}_1}{l} \Rightarrow d\varepsilon_{el1} = \int \dot{\varepsilon}_{el1} dt$$

$$\Rightarrow \varepsilon_{el1} = \varepsilon_1 + d\varepsilon_{el1}$$

$$\ddot{u}_2 = \frac{F_{el1}}{M_2} \Rightarrow \dot{u}_2 = \int \ddot{u}_2 dt \qquad \Rightarrow \sigma_{el1} = E\varepsilon_{el1}$$

第二个增量步开始时杆的构型



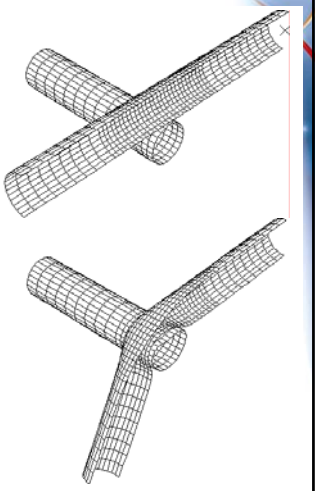
第三个增量步开始时杆的构型

© Dassault Systèmes, 2008

适于用Abaqus/Explicit求解的例子

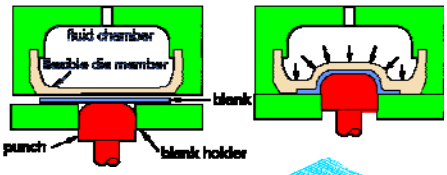
- **管道缠绕**
  - 该例说明了电厂中高压线路开裂之后，管道和管道之间的碰撞问题。
  - 流体的突然释放导致管道的一部分绕着它的支撑旋转，并撞到相邻的管道。

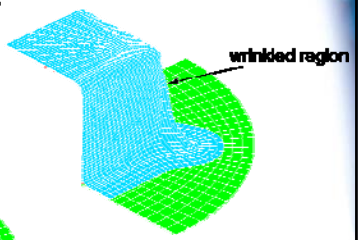
Video Clip

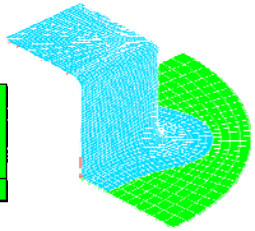


© Dassault Systèmes, 2008

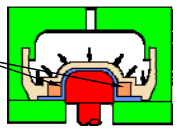
- **液压成型**
  - 利用流体压力使组件成型。
  - Abaqus/Explicit将捕获多余毛坯材料不稳定的起皱现象。







引入冲压罩，用于减小起皱效应。



© Dassault Systèmes, 2008

Abaqus/Explicit句法

SIMULIA

- Abaqus/Explicit模型的基本输入结构和选项同Abaqus/Standard类似。
  - 这样，用户可以将Abaqus/Standard中掌握的内容应用到Abaqus/Explicit中。
- 当输入文件中包含*DYNAMIC, EXPLICIT过程选项时，运行Abaqus/Explicit分析。
- 在大多数的Abaqus/Explicit分析中，用户只需提供总时间步长。时间增量的大小由程序自动确定，这样总可以满足稳定时间增量的限制。

```
*STEP
*DYNAMIC, EXPLICIT
, 70.E-3
```

- 对特殊情况，可以人为的控制时间增量大小。

© Dassault Systèmes, 2008

Abaqus/Explicit句法

- 与Abaqus/Standard不同，在默认情况下，Abaqus/Explicit使用有限应变、大位移、大旋转公式。
  - 在*STEP选项中，不需要NLGEOM参数。
  - 可以通过使用NLGEOM=NO参数，进行几何线性分析（小变形分析）。
- 显式动力学过程需要使用单元的集中质量矩阵。
- 因为使用Abaqus/Explicit时求解效率一般是非常重要的因素，所以可用的单元只有一阶减缩积分单元。
  - 例外：修正的三角形和四面体单元 (CPS6M, CPE6M, C3D10M)、二阶梁单元 (B22 和 B32)、全积分膜单元（M3D4）、全积分壳单元（ S4, S4T ）、全积分一阶六面体单元(C3D8, C3D8I, C3D8T)。

© Dassault Systèmes, 2008

Abaqus/Explicit句法

- **Abaqus/Explicit独有的一些选项**
  - 下面的模型定义和历程选项只对Abaqus/Explicit有效（没有列出输出和控制选项）：

分析过程	*DYNAMIC, EXPLICIT: 该过程指定动力学分析步Abaqus/Explicit将作为求解器。
	*DYNAMIC TEMPERATURE-DISPLACEMENT, EXPLICIT: 该过程指定热固耦合分析步。
	*ANNEAL: 该过程将所有的节点速度和状态变量，比如应力和塑性应变设置为零。

© Dassault Systèmes, 2008

Abaqus/Explicit句法

材料

*EOS:材料的状态方程可以用于模拟流体动力学 (爆炸)材料, 或几乎不可压流体。

*DAMAGE INITIATION and *DAMAGE EVOLUTION: 这些材料选项模拟材料中的渐进的损伤和材料的失效。

* EXTREME VALUE: 该选项指定在每个增量步需要监控的关键变量的极值。

*TRACER PARTICLE: 该选项定义跟踪粒子，用于在自适应网格域跟踪材料点。

© Dassault Systèmes, 2008

刚体

© Dassault Systèmes, 2008

刚体

- 第五讲曾经提到过，Abaqus具有通用刚体功能。
- 在Abaqus/Explicit中使用该功能时，需要考虑以下两点。
  - 刚体中的单元不影响稳定时间增量。
  - 在*RIGID BODY选项中，可以定义刚体单元的厚度和密度。

    RIGID BODY, REF NODE=node*, **ELSET=***element set*, **DENSITY=#** *thickness*
    - 在*RIGID BODY选项之后的数据行中，可以定义刚体的常数厚度。
    - 在*RIGID BODY选项中，可以使用NODAL THICKNESS参数指定厚度变化的刚体。

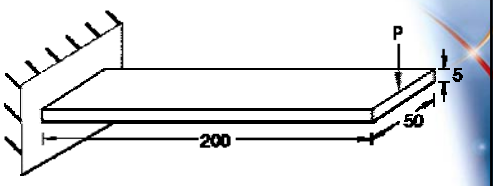
© Dassault Systèmes, 2008

习题6: 动力学（继续）

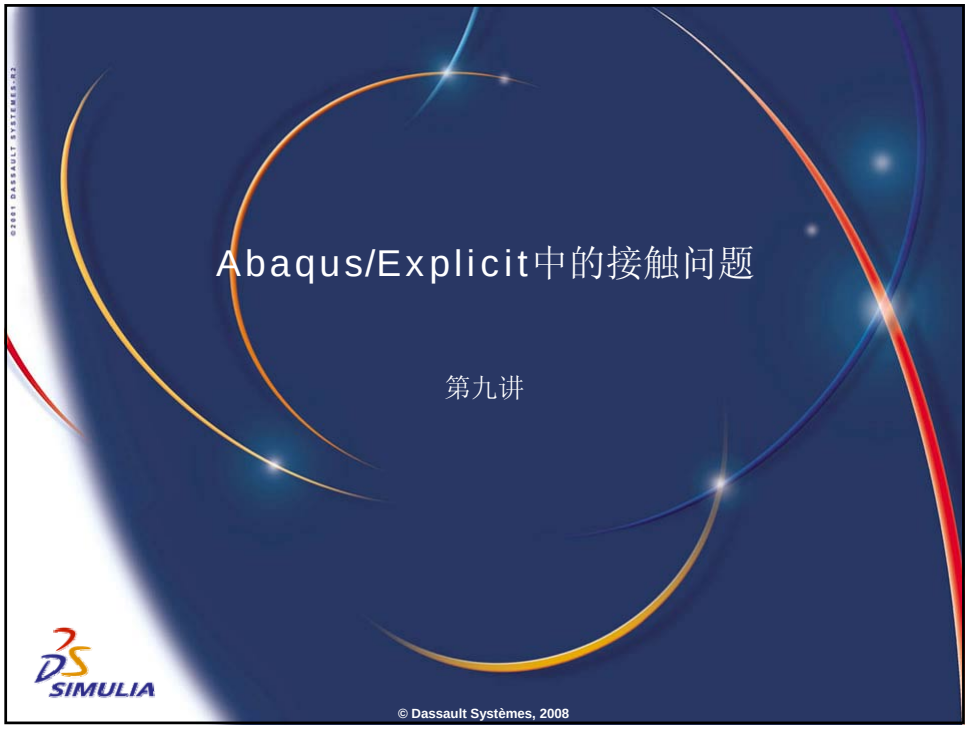
© Dassault Systèmes, 2008

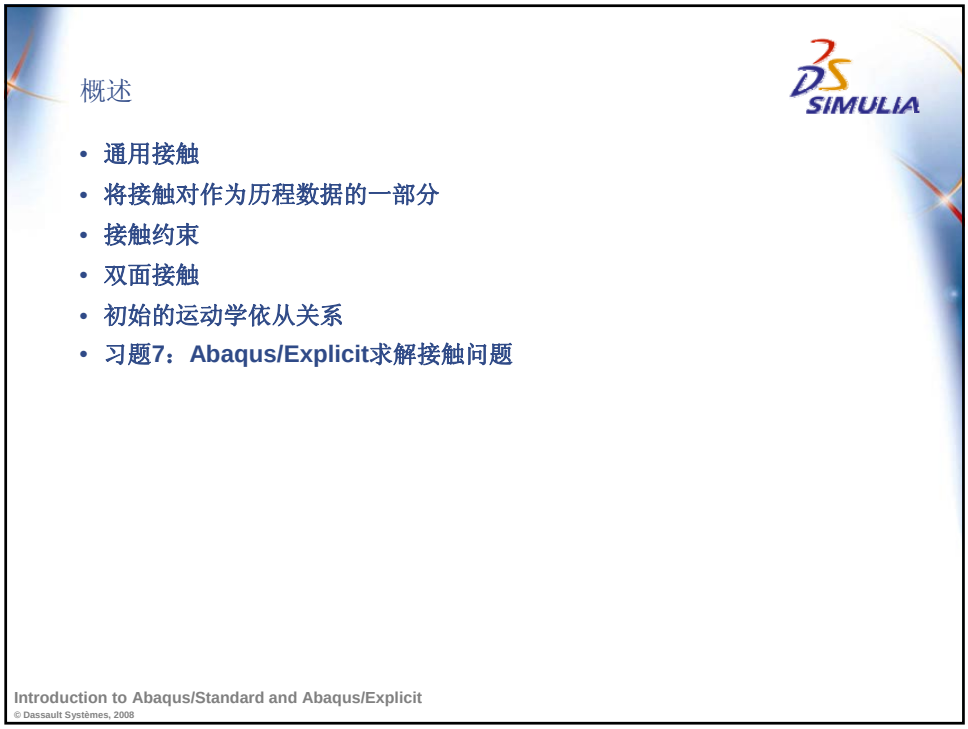
习题6: 动力学（继续）

- **习题任务**
  1. 使用显式动力学方法运行自由振动分析。
  2. 比较三种振动分析方法的结果：频率提取、隐式动力学分析和显式动力学分析。



© Dassault Systèmes, 2008


Abaqus/Explicit中的接触问题
第九讲
SIMULIA
© Dassault Systèmes, 2008

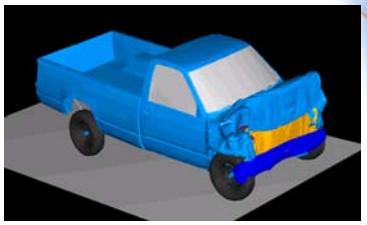

概述
SIMULIA
通用接触
将接触对作为历程数据的一部分
接触约束
双面接触
初始的运动学依从关系
习题7：Abaqus/Explicit求解接触问题
Introduction to Abaqus/Standard and Abaqus/Explicit
© Dassault Systèmes, 2008

通用接触

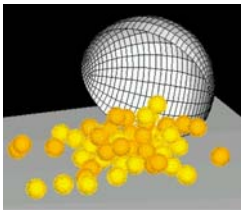
© Dassault Systèmes, 2008

通用接触

- **通用接触用于为多组件，并具有复杂拓扑关系的模型建模**
- **容易使用**
  - “自动接触”
  - 节省生成模型的时间
- **通用接触算法一般比双面接触算法快**



Video Clip



在“Abaqus/Explicit: Advanced Topics”课程中将详细讨论通用接触。

© Dassault Systèmes, 2008

通用接触

- **常用的、最简单的接触定义：**

```
*CONTACT
*CONTACT INCLUSIONS, ALL ELEMENT BASED
```

  - 整个模型的“自动接触”
- **分析中的接触定义逐渐变得更加详细**
  - 全局/局部摩擦系数和其它接触属性
  - 用户控制的接触厚度（尤其是壳）
  - 以接触对的方式制定接触域（而不是ALL ELEMENT BASED）
  - 特征边条件

© Dassault Systèmes, 2008

通用接触

- **主选项**

```
*CONTACT
```

- **子选项**
  - 常用的选项：

```
*CONTACT INCLUSIONS
*CONTACT PROPERTY ASSIGNMENT
```

  - 不常用的选项：

```
*SURFACE PROPERTY ASSIGNMENT
*CONTACT EXCLUSIONS
```

  - 很少用的选项：

```
*CONTACT FORMULATION
*CONTACT CONTROLS ASSIGNMENT
```


© Dassault Systèmes, 2008

通用接触

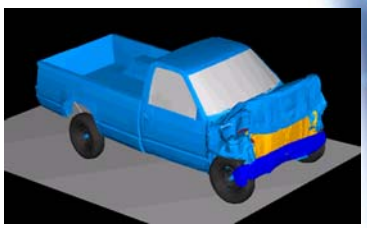
- **可以为指定的接触区域独立的分配接触属性**

```
*CONTACT
*CONTACT INCLUSIONS, ALL ELEMENT BASED
*CONTACT PROPERTY ASSIGNMENT
 , , prop_1                          (以全局的方式重新制定属性)
alum_surf, steel_surf, prop_2        (局部修改)
alum_surf, alum_surf, prop_3         (局部修改)
```


© Dassault Systèmes, 2008

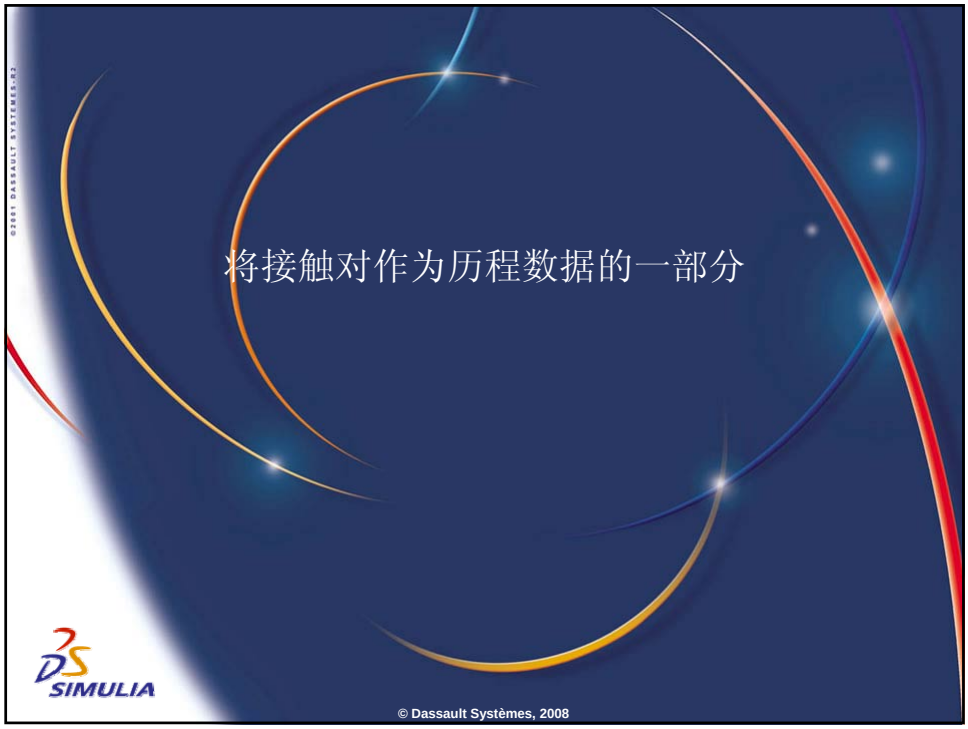
通用接触

- **例子：雪佛兰卡车 (55,000 单元)**
  - 本例为雪佛兰汽车与护栏碰撞。
  - 这是公开的模型：
    - FHWA/NHTSA National Crash Analysis Center 网址 (**http://www.ncac.gwu.edu/archives/model/index.html**)
  - 模型的细化程度非常高；比如，它包含引擎罩内引擎的组件。
  - 如果使用接触对，将很难定义所有可能的接触。



Video Clip Video Clip Video Clip

© Dassault Systèmes, 2008



将接触对作为部分的历程数据

SIMULIA

•在输入文件中，对于Abaqus/Explicit，将接触对定义作为历程数据的一部分。

```
*HEADING
.
.
*STEP
*DYNAMIC, EXPLICIT
, 200E-3
*CONTACT PAIR
ASURF, BSURF
.
.
.
*STEP
*DYNAMIC, EXPLICIT
, 200E-3
*CONTACT PAIR
ASURF, DSURF
```

可以根据需要，以分析步为基础，定义接触对、或删除接触对。

© Dassault Systèmes, 2008

将接触对作为部分的历程数据

- ***CONTACT PAIR选项可以包含OP参数，OP参数可以的值为ADD或DELETE。**
  - 例子：

```
*STEP
*DYNAMIC, EXPLICIT
.
.
*CONTACT PAIR
ASURF, BSURF
*END STEP
*STEP
.
.
*CONTACT PAIR, OP=DELETE
ASURF, BSURF
*CONTACT PAIR, OP=ADD
BSURF, CSURF
*END STEP
```

删除包含表面ASURF和BSURF的接触对。

添加包含表面BSURF和CSURF的接触对。

© Dassault Systèmes, 2008

实施接触约束

© Dassault Systèmes, 2008

实施接触约束

- **可以用下面的算法之一实施接触约束：**
  - 运动依从（只有接触对算法可用）
  - 罚函数
- **多数情况下，运动依从算法和罚函数算法将得到几乎一致的结果。然而，在某些情况下，一种方法可能比另一种方法更可取。**

© Dassault Systèmes, 2008

实施接触约束

- **运动依从接触**
  - 默认的运动接触公式达到的计算精度与接触条件相一致。
  - 在多数情况下，它工作得很好。但是在某些情况下，如抖动接触，使用罚函数接触会更容易得到收敛的解。
  - 不能为刚体-刚体接触建模。
  - 只对**接触对**算法可用。

© Dassault Systèmes, 2008

实施接触约束

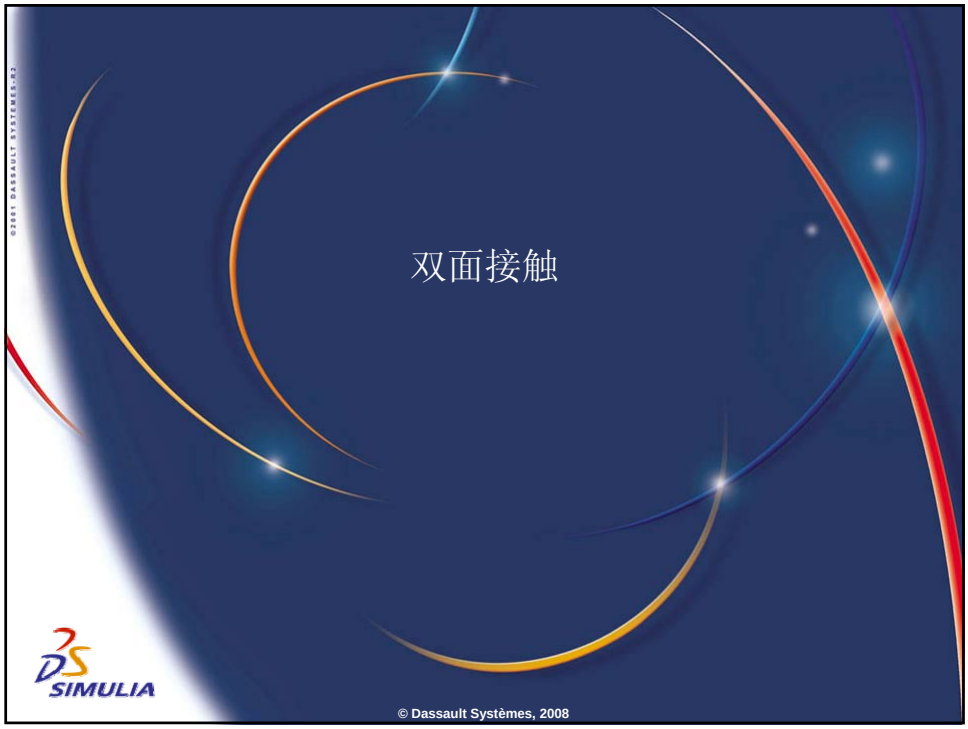
- **罚函数接触**
  - 罚函数接触算法中接触约束的严格性低于运动学算法。
  - 罚函数算法可以处理更一般类型的接触；比如，刚体之间的接触。
  - 因为罚函数算法在模型中引入附加刚度，该附加刚度将影响稳定时间增量。
  - 通用接触算法中使用罚函数接触约束。
  - 如果需要在接触对算法中使用罚函数接触约束：
    - 在*CONTACT PAIR选项中，设置MECHANICAL CONSTRAINT参数为 PENALTY。

© Dassault Systèmes, 2008

实施接触约束

- Abaqus/Explicit自动计算由与接触力相关的侵彻距离引入的“弹簧”刚度或“罚”刚度。但是必须考虑相关的影响：
  - 应该尽量减少对稳定时间增量的影响。
  - 在所有的分析中，允许的侵彻不可以太大。
- 对于**接触对**算法：
  - 通过在*CONTACT CONTROLS选项指定SCALE PENALTY参数，用户可以缩放默认的罚刚度。
- 对于**通用接触**算法：
  - 可以使用*CONTACT CONTROLS ASSIGNMENT, TYPE=SCALE PENALTY 选项缩放罚刚度。

© Dassault Systèmes, 2008



双面接触

- **由壳单元、薄膜单元和刚体单元定义的单个表面可以包含这些单元的顶面（SPOS）和底面（SNEG）。**
  - 为使用**接触对**算法，Abaqus/Explicit可以自动生成双边表面。
    - 为了定义双边表面，在*SURFACE选项忽略表面标识符。
    - 不需要一致单元法向。
  - **通用接触**算法自动使用双边表面。

© Dassault Systèmes, 2008

双面接触

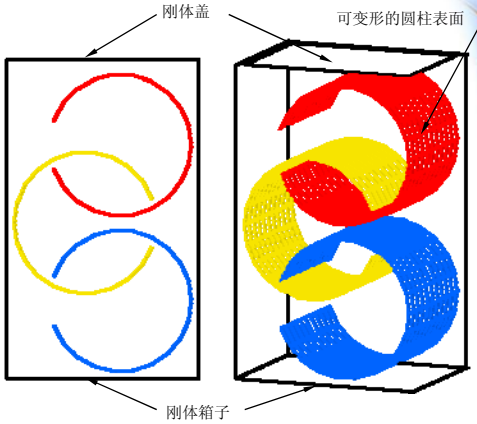
- **接触可能发生在形成双面表面单元的任何表面。**
  - 比如，在分析开始时，从属节点在双边表面的一侧；在分析过程中，从属节点跨过表面的边，到达双边表面的另一侧。
  - 在这样的情况下，需要定义双边表面。
  - 使用双边接触的附加计算费用是最小的。

© Dassault Systèmes, 2008

双面接触

- **例子：互相嵌入圆柱壳的压缩**

Video Clip



前视图　　斜视图（箱子的正面和侧面被删除）

© Dassault Systèmes, 2008

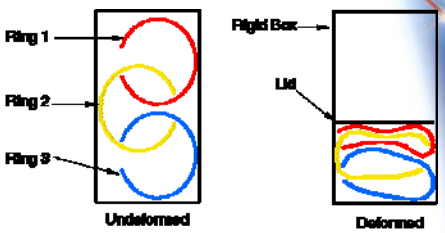
双面接触

通用接触

```
*HEADING
:
*CONTACT
*CONTACT INCLUSIONS,
 ALL ELEMENT BASED
:
*STEP
*DYNAMIC, EXPLICIT
:
*END STEP
```

接触对

```
*HEADING
:
*SURFACE, NAME=RING1
RING1
*SURFACE, NAME=RING2
RING2
*SURFACE, NAME=RING3
RING3
*SURFACE, NAME=BOX
BOX
*SURFACE, NAME=LID
LID
:
*STEP
*DYNAMIC, EXPLICIT
:
*CONTACT PAIR
RING1, RING2
RING1, RING3
RING2, RING3
RING1, BOX
RING2, BOX
RING3, BOX
RING1, LID
:
*END STEP
```



双面接触允许简单的定义复杂的接触条件

因为在视图窗口中不显示壳的厚度，所以在变形构型的接触表面之间有缝隙（可以在接触计算中取消壳的厚度）

© Dassault Systèmes, 2008

初始的运动学依从关系

© Dassault Systèmes, 2008

初始的运动学依从关系

- **Abaqus/Explicit不允许接触表面的初始过盈**
  - 接触表面的节点将被调整，删除分析之前的初始过盈：
    - 只移动接触表面的节点。
    - 对于分析过程第一个分析步定义的接触对，由于调整表面引起的位移不产生初始应变或应力。
  - 在随后的分析步中：
    - 对于接触对算法，调整将产生应变。
    - 对于通用接触算法，忽略初始过盈。

© Dassault Systèmes, 2008

初始的运动学依从关系

- 如果接触对是平衡的主/从接触对，两个表面都将被调整。
- 使用*DIAGNOSTICS选项，关于初始过盈的细节信息可以写到信息(`.msg`)文件。

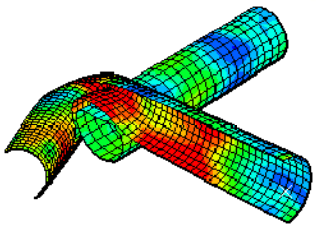
© Dassault Systèmes, 2008

习题7: Abaqus/Explicit求解接触问题

© Dassault Systèmes, 2008

习题 7: Abaqus/Explicit求解接触问题

- **习题任务**
  - 定义两个管的通用接触
  - 指定边界条件和初始速度
  - 进行冲击分析
  - 查看变形形状和能量历程

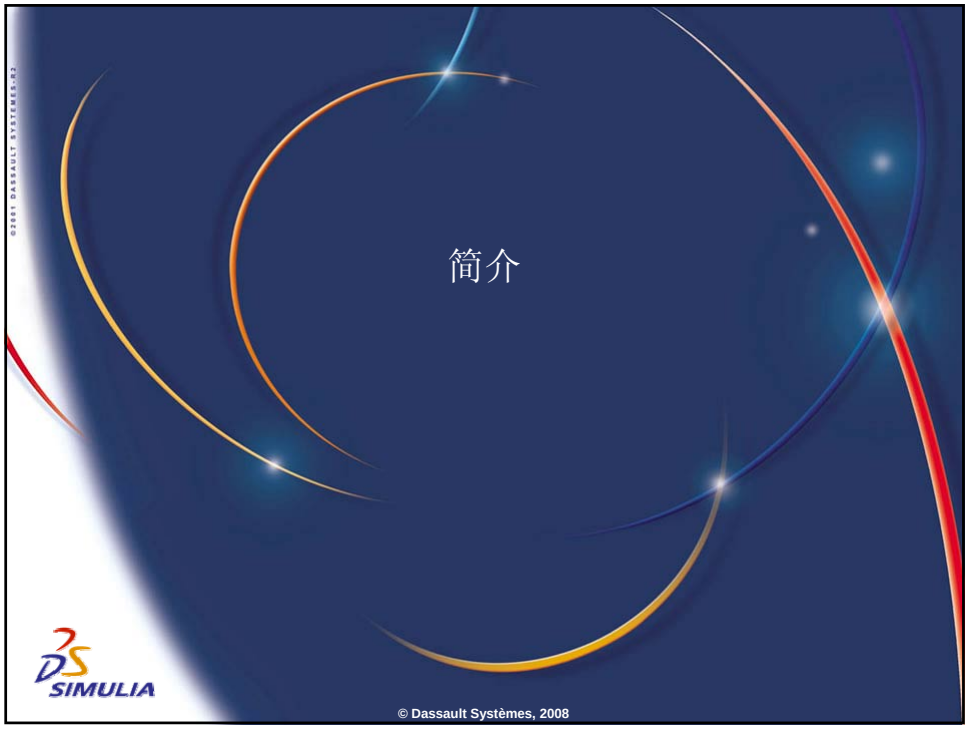


© Dassault Systèmes, 2008

概述

- 简介
- 载荷速率
- 能量平衡
- 质量缩放
- 自适应网格
- 总结

© Dassault Systèmes, 2008



简介

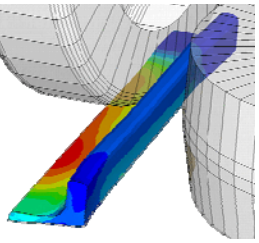
- **显式方法是真正的动力学过程。它最初用于模拟高速碰撞问题。**
  - 它用于求解结构的动力平衡状态。
  - 在求解过程中，惯性力起到决定性的作用。
  - 非平衡力以应力波的方式在相邻单元之间传播。
  - 稳定时间增量一般较小。

© Dassault Systèmes, 2008

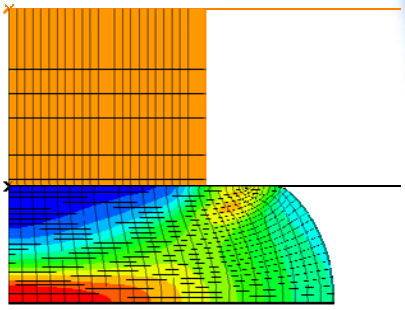
简介

- **显式动力学方法还可以模拟准静态问题，比如金属成型过程，但是需要特殊的考虑：**
  - 如果以自然时间周期计算，用显式动力学方法求解准静态问题是不切实际的。一般需要上百万的时间增量。

Video Clip



对称I型截面的轧制



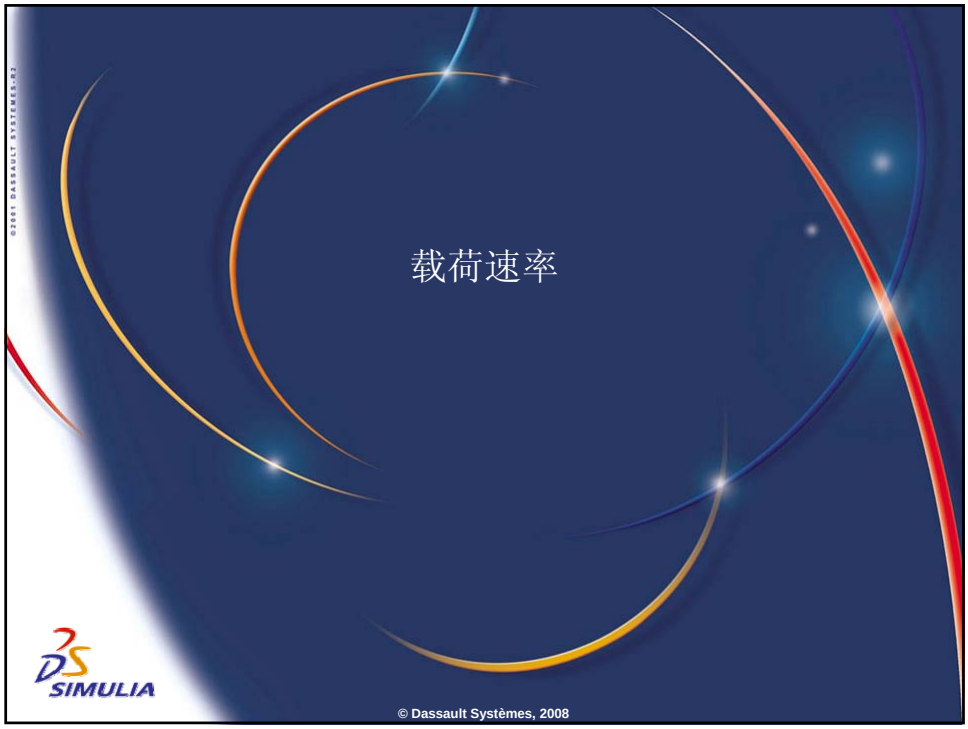
圆柱钢坯的镦锻

© Dassault Systèmes, 2008

简介

- 为节省计算时间，可以在模拟过程中人为的增加轧制过程的速度。
- 在增加轧制速度之后，静平衡问题演化为动平衡问题。惯性力的影响将会增加。
- 准静态分析的目标就是：在惯性力的影响较小的前提之下，尽量缩短计算的时间周期。

© Dassault Systèmes, 2008



载荷速率

SIMULIA

- **在模拟过程中，人为的增加准静态成型过程的速度是必要的，它可以让求解过程更经济。**
- **但是，在不使结果退化的前提下，究竟可以把速度提高多少呢？**
  - 比如，金属成型过程中，典型的工具速度大约为1 m/s的数量级。
  - 这个波速与金属中的典型波速相比是非常小的（钢中的波速为5000 m/s）。
- **一般推荐的载荷速率为材料中波速的1%。**

© Dassault Systèmes, 2008

载荷速率

- 推荐的方法：
  - 以不同的速率多次模拟（比如，工具的速度为100, 50, 5 m/s）。
  - 因为以低的载荷速率进行分析的时间比较长，所以以从高的载荷速度到低的载荷速度进行分析。
  - 检查结果（变形形状、应力、应变、能量），分析不同载荷速率对结果的影响。
    - 在显式板金成型模拟过程中，过大的工具速度将抑制起皱现象，并激起非真实的局部拉伸。
    - 在屈曲成型过程中，过大的工具速度将引起“喷注”效应—水动力学响应（下页中有图形）。

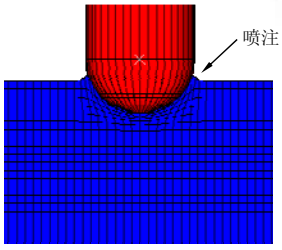
© Dassault Systèmes, 2008

载荷速率

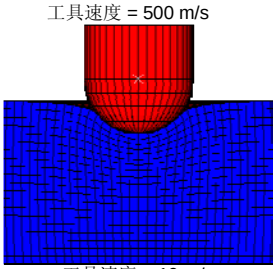
喷注

- 考虑下面的屈曲成型过程（轴对称模型180° 的截面）。
- 当工具速度非常大时，产生高度局部化的变形（喷注）。

Video Clip



工具速度 = 500 m/s

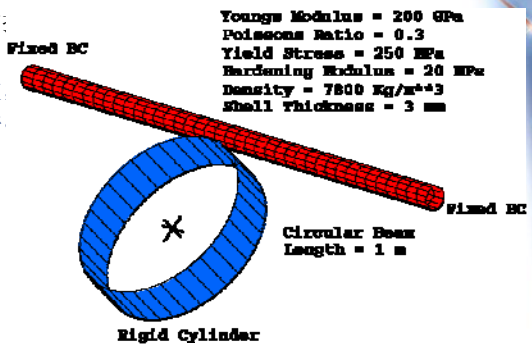

工具速度 = 10 m/s

工具速度对变形形状的影响

© Dassault Systèmes, 2008

载荷速率

- **例子：板金**
  - 右图为汽车门标准门梁的击测试的简化模型。
    - 圆梁在每个端点固定接触刚体圆柱后变形
    - 测试为准静态的。

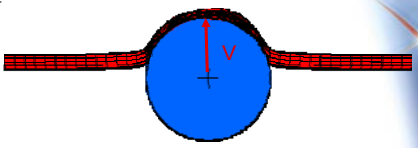


刚体圆柱与可变形梁的碰撞

© Dassault Systèmes, 2008

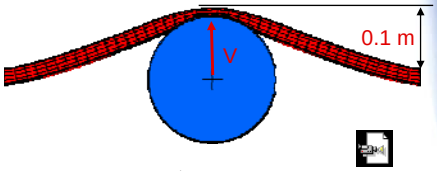
载荷速率

- 如果碰撞速度很高，400 m/sec，变形高度局部化，梁没有结构响应。
- 静态测试中的主要响应为梁的一阶模态。该模态的频率用于预计碰撞速度。
  - 一阶频率大约为250 Hz。
  - 碰撞在4微秒内完成。
  - 利用25 m/sec的速度碰撞，圆柱在4微秒内向梁推动0.1 m。



速度 400 m/s:

局部效应



速度 25 m/s:

Video Clip

好的全局结果

© Dassault Systèmes, 2008

载荷速率

- **为什么25 m/sec的速度是合适的？**
  - 一阶频率 (f) 大约为 250 Hz。
  - 相应的周期为 t=0.004 秒。
  - 在此周期内，刚体圆柱被推向梁 d = 0.1 m。
    - 这样，估计的速度 v 为 $v = d/t$ = 0.1/0.004 = 25 m/sec。
  - 金属波速为5000 m/sec，所以碰撞速度25 m/sec为波速的 0.5% 。
    - 碰撞速度应该小于材料波速的1%。
  - 在分析步内，以光滑缓坡的方式，把碰撞速度从零增加到所施加的碰撞速度可以得到更加精确的解。

© Dassault Systèmes, 2008

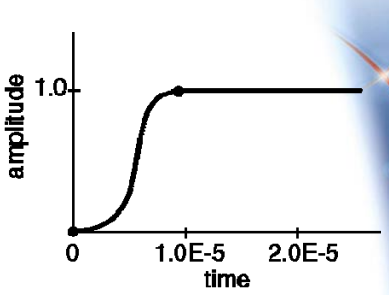
载荷速率

- **SMOOTH STEP幅值曲线**
  - 通过逐步施加载荷的方式，可以提高准静态解的精度：
    - 工具中的常速度条件导致金属毛坯遭受突然的冲击载荷。
    - 这将导致应力波穿过毛坯，产生不希望发生的结果。
    - 以逐步降低（增大）的方式，使工具速度从零逐步增加，将减小这些不利的效应。
    - 出于同样的原因，在分析结束移除工具的过程中，也以斜坡的方式将工具速度逐步降为零。

© Dassault Systèmes, 2008

载荷速率

- SMOOTH STEP幅值定义两个幅值之间以5阶多项式过渡。比如，在过渡开始和结束时一阶和二阶时间导数为零。
- 在使用SMOOTH STEP定义位移时间历程时，每个指定的幅值处的速度和加速度为零。




```
*AMPLITUDE, NAME=SSTEP, DEFINITION=SMOOTH STEP
0.0, 0.0, 1.0E-5, 1.0
*BOUNDARY, TYPE=DISPLACEMENT, AMP=SSTEP
12, 2, 2, 2.5
```


© Dassault Systèmes, 2008

能量平衡

© Dassault Systèmes, 2008

能量平衡

- **能量平衡方程可以用于帮助评估计算结果是否为合理的准静态响应。**
  - 在Abaqus/Explicit中，能量平衡可以写为

$$E_{KE} + E_I + E_V + E_{FD} - E_W - \underbrace{E_{PW} - E_{CW} - E_{MW}} = E_{TOT} = constant,$$

由接触和约束罚函数产生，并驱动由于质量缩放而产生的附加质量

其中

E_{KE} 为动能。

E_I 为内能 （包括弹性应变能、塑性应变能和与沙漏控制相关的伪能）。

E_V 为粘性机制耗散的能量。

E_{FD} 为摩擦耗散的能量。

E_W 为外力功

E_{TOT} 为系统的总能量。

© Dassault Systèmes, 2008

能量平衡

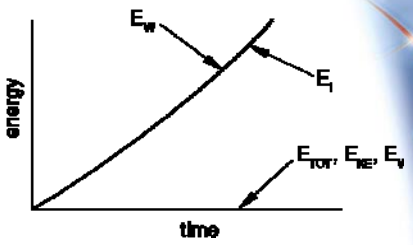
- **考虑单轴拉伸试样的拉伸测试**
  - 如果实际测试是准静态的，拉伸试件的外力功等于试件的内能。

单轴拉伸测试

© Dassault Systèmes, 2008

能量平衡

- **准静态测试的能量历程显示在右图中：**
  - 惯性力是可以忽略的。
  - 测试试件的材料速度是很小的。
  - 动能是可以忽略的。
- **当测试的速度增加以后：**
  - 试件的响应偏离静态、趋于动态。
  - 因此，材料速度和动能更加明显。



准静态拉伸测试的能量历程

© Dassault Systèmes, 2008

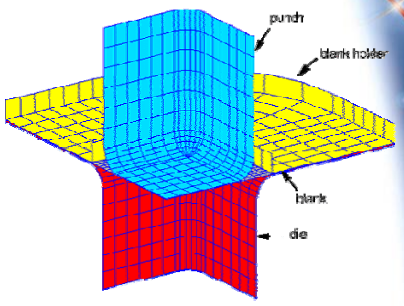
能量平衡

- **因此，能量检查为Abaqus/Explicit金属成型过程的结果是否反应了准静态解提供了另外的评估方法。**
  - 在主要的成型过程中，变形材料的动能不可以超过内能的一小部分。
    - 这个小部分一般为1–5%。
      - 因为毛坯在产生显著变形之前将被移动，因此在成型过程的早期，一般很难达到这个值。
      - 使用光滑幅值曲线将改进早期响应。
    - 不关心工具的动能。
      - 从模型的全部动能中减掉工具的动能，或者限制变形组件的能量输出。

© Dassault Systèmes, 2008

能量平衡

- **例子：圆柱杯的深冲压**
  - 右图为有限元模型的1/4模型。
  - 在所有的接触面中定义摩擦：
    - 冲头和毛坯： $\mu = 0.25$。
    - 砧和毛坯： $\mu = 0.125$。
    - 毛坯夹具和毛坯： $\mu = 0$。
  - 通过为毛坯夹具施加22.87 KN的向下力模拟深冲压过程，并将冲头向下移动36 mm。



圆柱杯深冲压的初始构型

© Dassault Systèmes, 2008

能量平衡

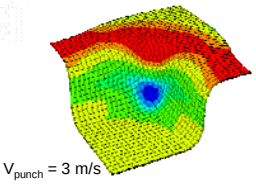
- 尝试三种不同的冲压速度：
  - 3 m/s
  - 30 m/s
  - 150 m/s
- 下表中总结了每个圆柱杯深冲压的计算费用：

冲压速度 (m/s)	时间增量	正则化的CPU时间
3 (1X)	27929	1.0
30 (10X)	2704	0.097
150 (50X)	529	0.019

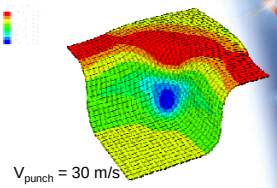
© Dassault Systèmes, 2008

能量平衡

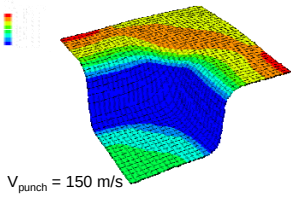
- 最终构型中毛坯厚度云图
- 过大的冲压速度导致结果与实际的物理现象不符。
- 尽管计算费用相差10倍，以30 m/s和3 m/s的速度冲压的结果非常接近。



V_{punch} = 3 m/s



V_{punch} = 30 m/s



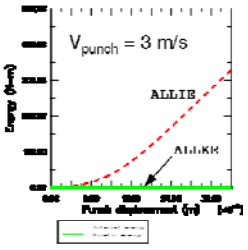
V_{punch} = 150 m/s

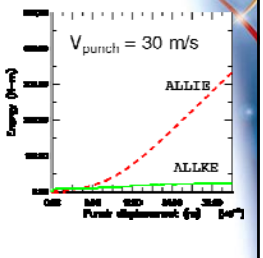
Video Clip

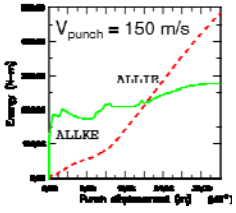
© Dassault Systèmes, 2008

能量平衡

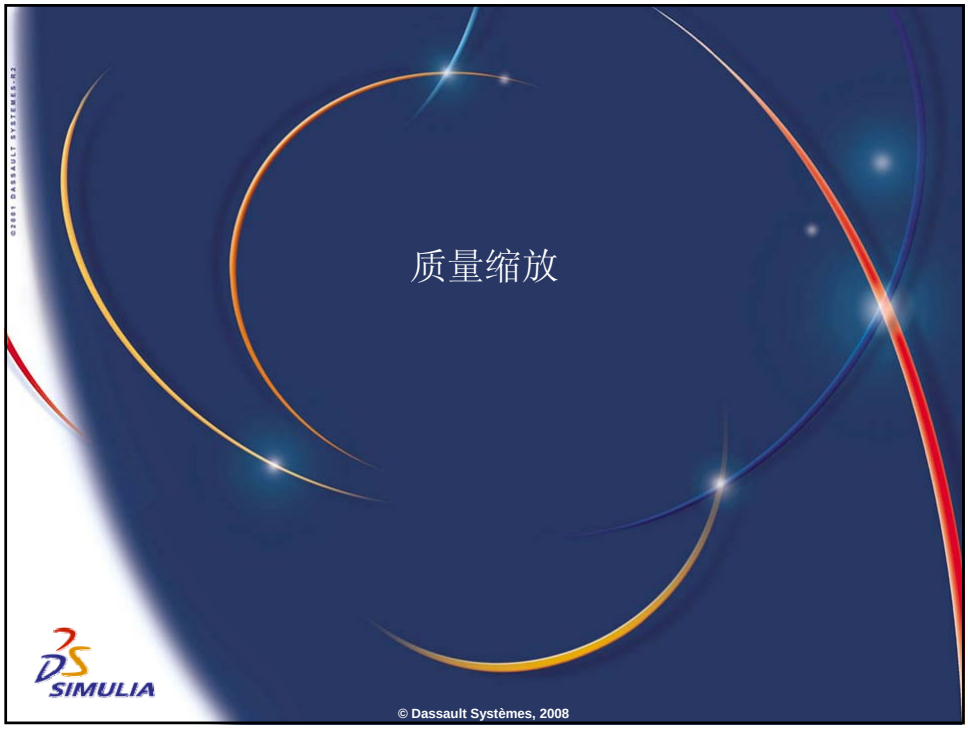
- 比较内能和动能
- 当冲压速度为150 m/s时，毛坯中的动能与内能相比占很大的比例。
- 当冲压速度为3 m/s和30 m/s时，在成型过程中，动能与内能相比只占很小的一部分。







© Dassault Systèmes, 2008



质量缩放

SIMULIA

- **人为的增加成型速度可以提高解的经济性。同时，材料应变率以同样的速度增加。**
  - 如果材料对于应变率是不敏感的，这是不相关的。
  - 如果模型中考虑应变率敏感性，将导致错误的结果。
  - 如果考虑率相关性，一般需要用自然时间周期模拟成型过程。
    - 可以通过质量缩放实现这样的功能。

© Dassault Systèmes, 2008

质量缩放

- 显式动力学过程稳定极限的估计公式为

$$\Delta t = \left(\frac{L^e}{c_d} \right),$$

其中 L^e 是最小的特征单元长度， c_d 是材料的膨胀波速。

- 泊松比为
- 材料的膨胀波速为

$$c_d = \sqrt{\frac{E}{\rho}},$$

其中 E 为杨氏模量， ρ 为材料密度。

© Dassault Systèmes, 2008

质量缩放

- 如果以 f^2 的方式人为的增加材料密度：
  - 膨胀波速以 f 的方式减小。
  - 稳定时间增量以 f 的方式增加。
- 通过质量缩放的方式人为的增加稳定时间，使得可以以自然时间周期分析成型过程。
- 人为的增加工具速度之后，质量缩放对惯性效应具有同样的影响。过多的质量缩放将导致非真实的解。
- 如果质量缩放用于完全的动态条件下，总质量的变化应该尽量小（小于1%）。

© Dassault Systèmes, 2008

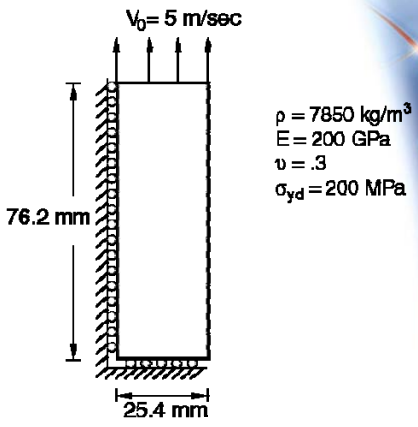
质量缩放

- 利用*FIXED MASS SCALING选项，可以进行质量缩放。
  *FIXED MASS SCALING在分析步开始时施加质量缩放。
- 句法：
  FIXED MASS SCALING, ELSET=name*, **FACTOR=** f^2
  - 在相应的单元集中，每个单元的密度以 f^2的方式增加，因此以f 的方式增加稳定时间增量。

© Dassault Systèmes, 2008

质量缩放

- **质量缩放例子：**
  - 右图为低碳钢平面应变试件的拉伸测试。
  - 由于对称的原因，只选取模型的1/4。

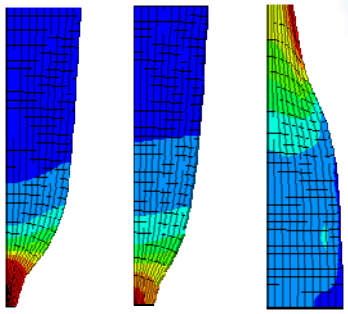


单轴拉伸测试

© Dassault Systèmes, 2008

质量缩放

- **图形显示了三种分析的不同结果（PEEQ云图）。**
  - 右图中左边的结果和中间的结果几乎相同。
    - 中间的结果与左边的结果相比，只需要1/5的计算时间。
  - 与原始的静态解相比，右边的结果基本是没有意义的。



质量缩放因子 **1** **25** **10000**

Video Clip

变化实际应变速率

© Dassault Systèmes, 2008

网格自适应

© Dassault Systèmes, 2008

网格自适应

- **目的**
  - 在许多非线性模拟过程中，结构或加工过程中的材料承受非常大的变形。
    - 这些变形将扭曲有限元网格，过渡扭曲的网格将不能得到准确的解，或由于数值原因导致分析提前终止。
    - 在这些模拟过程中，必须使用网格自适应工具周期性的减小网格的扭曲。
  - 注意：在本次讲座中，我们只讨论Explicit中的ALE自适应网格，关于Standard中的自适应网格格式将在以后的讲座中提到。

© Dassault Systèmes, 2008

网格自适应

- 网格自适应对很多问题都很有用：
  - 对于大变形的瞬态问题，可以作为连续的自适应网格工具使用。比如：
    - 动力冲击
    - 侵彻
    - 晃动
    - 锻造
  - 可以作为一种求解技术，模拟稳态过程，比如：
    - 挤压
    - 轧制
  - 作为一种工具，可以分析稳态过程中的瞬时状态。

© Dassault Systèmes, 2008

网格自适应

- **网格自适应基础**
  - 在Abaqus/Explicit中，自适应网格功能通过任意的拉格朗日-欧拉（ALE）方法实现。自适应网格功能的基本特征为：
    - 网格被定期平滑，用于减小单元扭曲，并维持良好的单元长宽比。
    - 保持网格原有的拓扑关系—单元号、节点号和它们的连接关系并不改变。
    - 可以分析拉格朗日（瞬态）问题和欧拉（稳态）问题。

© Dassault Systèmes, 2008

网格自适应

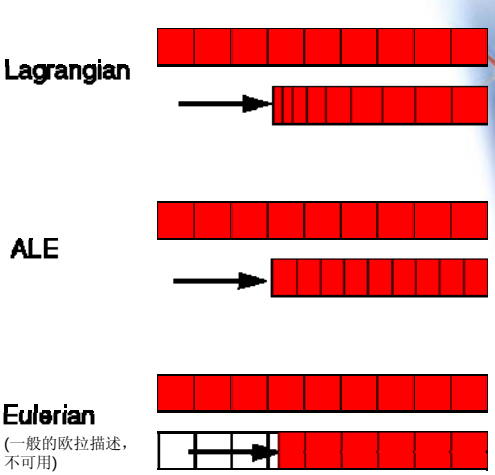
•任意的拉格朗日-欧拉 (ALE)方法

Lagrangian description	节点同材料点同步移动。 这种方法容易跟踪自由表面，并施加边界条件。 如果出现高的应变梯度，网格将出现扭曲。
Eulerian description	材料在网格中流动时，节点固定不动。 这种方法很难跟踪自由表面。 如果网格是固定的，没有网格扭曲。 当前欧拉描述的实施是有限的。
ALE	在必要的地方（在自由边界），网格运动约束于材料运动，但是，其它的材料运动和网格运动是独立的。

© Dassault Systèmes, 2008

网格自适应

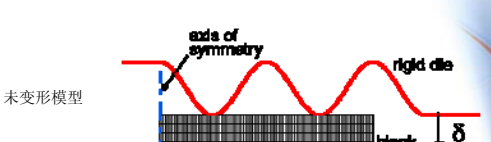
- 不同方法中网格和材料的运动



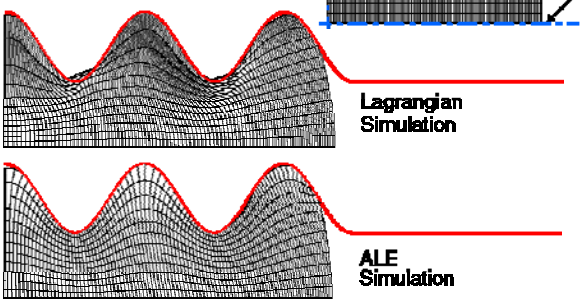
© Dassault Systèmes, 2008

网格自适应

- 轴对称成型问题的ALE模拟



Video Clip

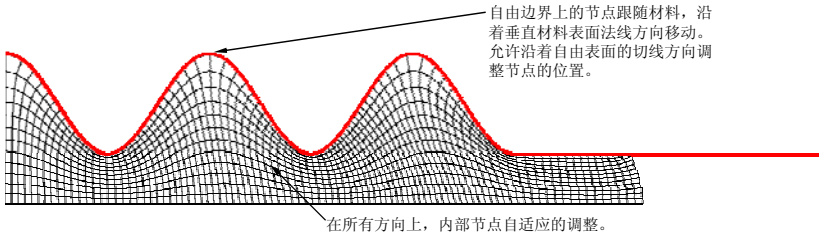


砧移动70%的网格变形

© Dassault Systèmes, 2008

网格自适应

- 使用自适应网格技术，可以在整个分析过程中保持高质量的网格。



ALE 模拟：砧移动100%时的变形网格

© Dassault Systèmes, 2008

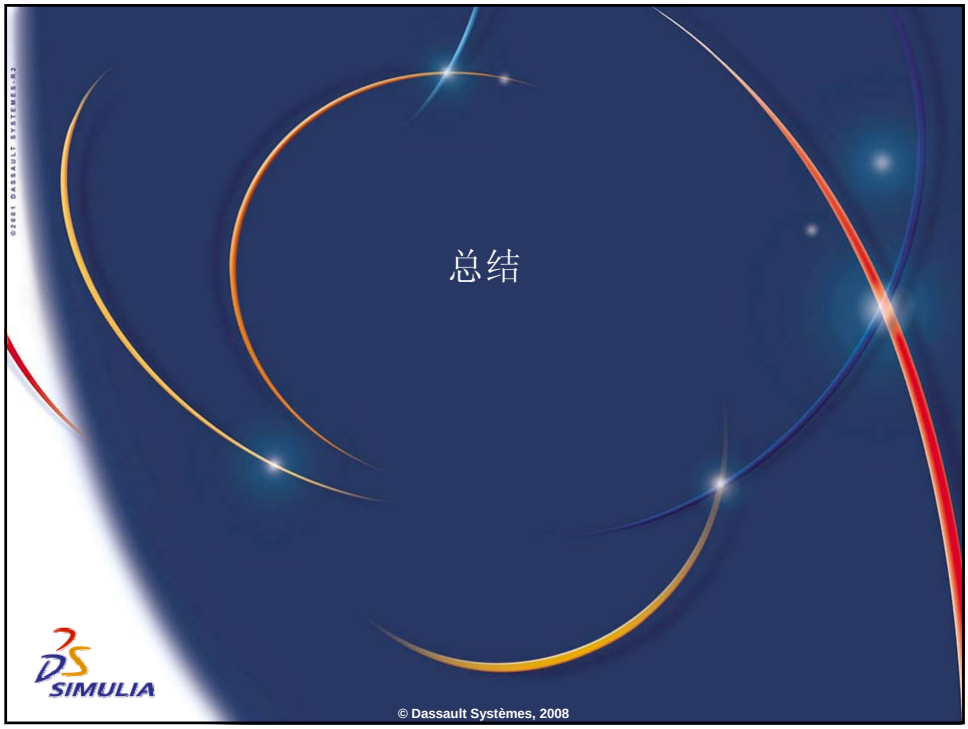
网格自适应

- 在瞬态（拉格朗日-类型）问题中，比如锻造模拟，只需要很少的附加输入就可以激活自适应网格功能。

```
*HEADING
....
*ELSET, ELSET=BLANK
....
*STEP
*DYNAMIC, EXPLICIT
....
*ADAPTIVE MESH, ELSET=BLANK [,FREQUENCY=…,MESH SWEEPS=…]
....
*END STEP
```

- 对于一阶减缩积分实体单元可以使用适应性网格。
  - 模型中可以存在其它单元类型。

© Dassault Systèmes, 2008



总结

- 过大的载荷速率将产生带有显著惯性效应的结果。
- 一般的建议为限制载荷施加的速率，比如，工具速度小于材料波速的1%。
- 以斜坡的方式，从零增加载荷速率，也可以提高准静态响应的准确性。
  - 使用SMOOTH STEP幅值定义
- 质量缩放可以用于率相关材料行为，允许以自然时间周期进行模拟。
- 能量平衡可以用于评估计算结果：对于施加的载荷是否给出了结构的准静态响应。

© Dassault Systèmes, 2008

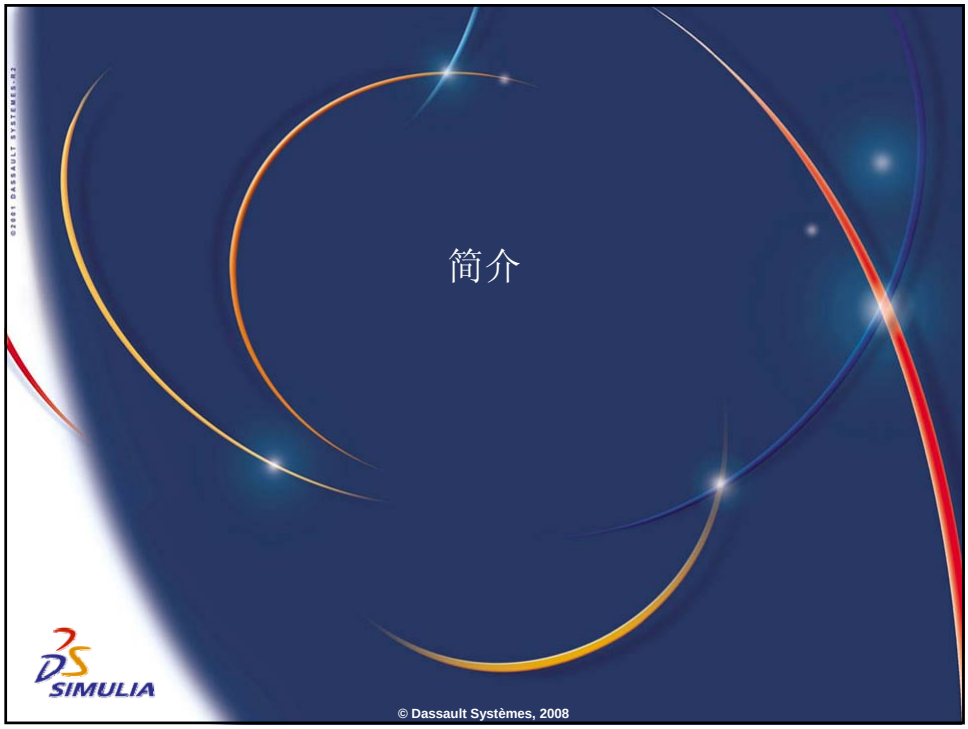
总结

- 因为结果取决于加工过程的速度（通过质量缩放调整真实的或人为的速度），所以必须保证人为的加速加工过程不生成非真实的结果。
  - 如果需要确认Abaqus/Explicit的结果是否是真实的，可以将问题简化，再利用Abaqus/Standard做静态分析，比较分析的结果。出于比较的目的，创建简化测试模型的最简单方法为以二维的方式定义所研究问题的一部分。
  - 最简单的途径就是建立一个该问题的二维的计算模型进行试算。
- 自适应网格技术可以在大变形的过程中保持很高质量的网格。

© Dassault Systèmes, 2008

合并Abaqus/Standard和Abaqus/Explicit
第十一讲
SIMULIA
© Dassault Systèmes, 2008

概述
SIMULIA
• 简介
• Abaqus中的用法
• 利用Abaqus/Standard计算回弹
• 习题8（可选）：准静态问题和导入分析
Introduction to Abaqus/Standard and Abaqus/Explicit
© Dassault Systèmes, 2008



简介

SIMULIA

- **Abaqus提供在Abaqus/Explicit和Abaqus/Standard之间传递变形网格和相关状态的功能。**
- **该功能提供非常大的灵活性，比如，模拟金属成型过程中的回弹。**
- **变形的模型可以把模型的变形状态从Abaqus/Explicit导入到Abaqus/Standard，比如：**
  - 得到动态分析之后的最终静态构型。
  - 在金属成型之后模拟回弹。
  - 在变形的部件中进行特征值提取或屈曲分析。
  - 更加有效的模拟刚体工具的运动。

© Dassault Systèmes, 2008

简介

- **可以将Abaqus/Standard的模型变形导入到Abaqus/Explicit，比如：**
  - 模拟回弹过程完成之后的其它成型分析步。
  - 模拟模型经过热处理过程，并冷却后的成型过程（热应力在Abaqus/Standard中计算）。
  - 在Abaqus/Standard进行分析具有较高效率的分析之后，继续新的模拟过程。
  - 在Abaqus/Standard中进行稳态旋转分析之后，利用Abaqus/Explicit在崎岖的路面上模拟瞬态的旋转过程。

© Dassault Systèmes, 2008

简介

- **导入总结**
  - 在不同分析模块之间导入结果的主要需求为导入材料状态和节点位置的能力。
  - 下面的表格总结了导入能力：

可以被导入的内容	需要重新制定的内容	不能被导入的内容
材料状态*	边界条件	一些其它材料*
节点位置	载荷	
单元、单元集	接触条件	
节点、节点集	输出需求	
温度	多点约束	
	节点的转换	
	幅值定义	

* 查看下一页

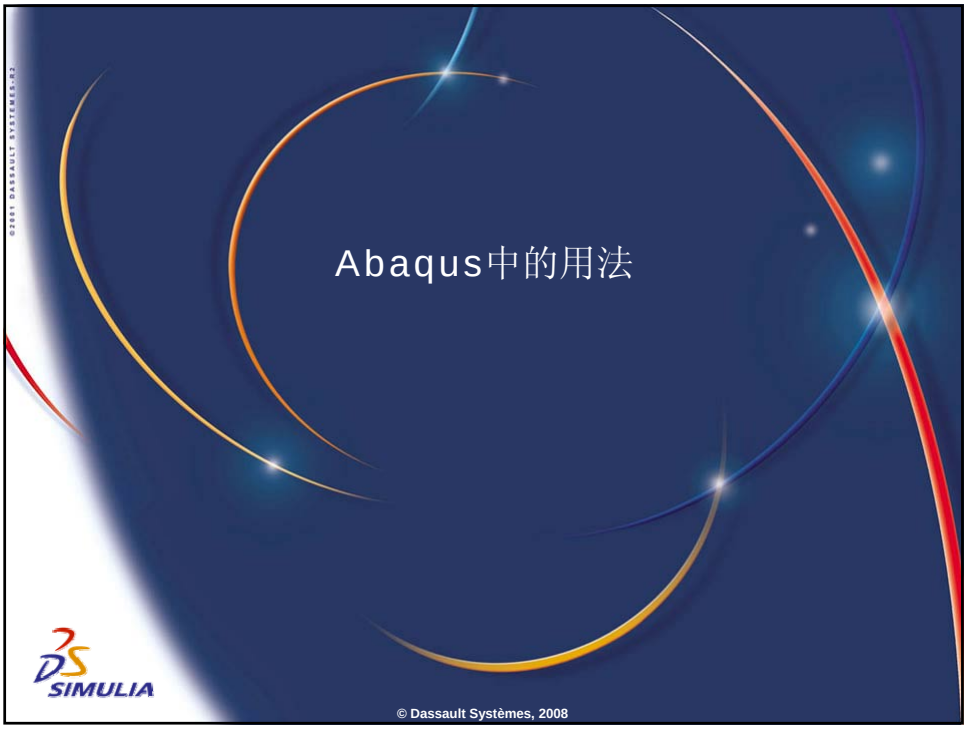
© Dassault Systèmes, 2008

简介

- **导入材料状态注释**
  - 只有以下的材料状态
    - linear elastic,
    - hyperelastic,
    - Mullins effect,
    - hyperfoam,
    - Mises plasticity (including the kinematic hardening models),
    - viscoelastic, and
    - damage for cohesive elements material models, or
    - materials defined in user subroutines UMAT and VUMAT

  可以导入进行进一步分析。

© Dassault Systèmes, 2008



Abaqus中的用法

- **运行导入分析需要下列信息：**
  - 包含模型当前状态的重启动(**.res**)文件。
    - 如果从 Abaqus/Standard导入到Abaqus/Explicit, 还需要分析的数据库(**.mdl** and **.stt**), 部件(**.prt**), 和输出数据库(**.odb**)文件。
    - 如果从 Abaqus/Explicit导入到Abaqus/Standard, 还需要状态 (**.abq**), 分析数据库(**.stt**), 组装(**.pac**), 部件(**.prt**),和输出数据库 (**.odb**)文件。
    - 上面提到的附加文件都会默认生成，在计划执行一个重启动分析前不要删除它们。
  - 用于下一次分析的新的输入(**.inp**)文件包括：
    - 紧跟在*HEADING选项之后的*IMPORT选项
    - 任何附加的模型数据
    - 下阶段模拟所需的历程数据

© Dassault Systèmes, 2008

Abaqus中的用法

- **执行导入分析**
  - 如果需要在Abaqus/Explicit和Abaqus/Standard之间传递模型状态，在用Abaqus/Explicit或Abaqus/Standard进行分析的过程中，必须写出重启动文件。
  - 使用Abaqus驱动器，设置**job**参数为新的作业名，设置**oldjob**参数为前面分析步中与重启动文件相关的作业名。

    abaqus job=*new_jobname* **oldjob=***previous_jobname*

© Dassault Systèmes, 2008

Abaqus中的用法

- **指定导入的分析增量步**
  - 将模型从Abaqus/Explicit导入到Abaqus/Standard：
    - 导入*IMPORT选项指定了被导入重启动文件的模型状态的STEP和INTERVAL：

      IMPORT, STEP=step number*, **INTERVAL=***interval number*
  - 将模型从Abaqus/Standard导入到Abaqus/Explicit：
    - 导入*IMPORT选项指定了被导入重启动文件的模型状态的STEP和INCREMENT：

      IMPORT, STEP=step number*, **INC=***increment number*

© Dassault Systèmes, 2008

Abaqus中的用法

- **单元和节点**
  - 在*IMPORT选项的数据行指定需要导入的单元集。

    IMPORT, STEP=step number*, **INTERVAL=***interval number*
    elset_1, *elset_2*, *elset_3*
    - 在*IMPORT选项的数据行中指定的每个单元集名必须出现在原始分析的截面定义（比如，*SOLID SECITON）选项中。
  - 如果UPDATE=YES，壳和薄膜的当前厚度被自动导入，并作为单元的初始厚度。
  - 与被导入单元连接的所有节点都被导入。
  - 可以在新的分析中定义附加的节点和单元。

© Dassault Systèmes, 2008

Abaqus中的用法

- **材料状态和参考构型**
  - 默认情况下，在导入分析中，材料状态被导入 （*IMPORT选项中STATE=YES）。
  - 如果需要不重新设置参考构型而继续分析，在*IMPORT选项中设置UPDATE=NO：

```
*IMPORT, UPDATE=NO
```

  - 在某些情况下，需要获得回弹位移和回弹分析开始时相对于几何体的应变(在回弹分析步开始时设置为零)。在*IMPORT选项中设置UPDATE=YES：

```
*IMPORT, UPDATE=YES
```

  - 因为参考构型不一致，所以如果需要进行附加成型过程的分析，不应该使用UPDATE=YES。

© Dassault Systèmes, 2008

Abaqus中的用法

- STATE和UPDATE参数的其它组合也是可用的，但是此处不再讨论。
- NLGEOM的设置被导入，并成为新分析的设置。

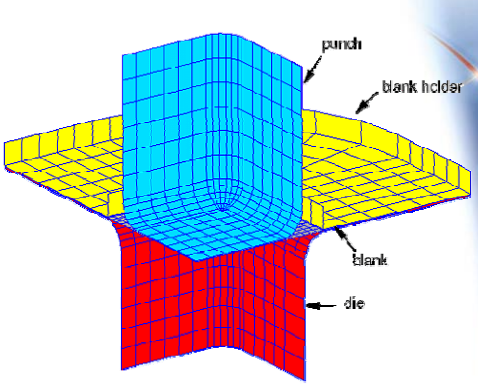
© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

- **右图中毛坯图形，在板筋成型过程中经历大的变形。**
  - 成型过程完成并撤去限制工具之后，毛坯将恢复它的弹性变形。
  - 回弹现象可能会导致成型产品不可预计的翘曲。
    - 必需设计合适的成型工具补偿回弹效应。



© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

- **与板筋成型过程相关的回弹计算：**
  - 一般的，利用Abaqus/Explicit模拟成型过程，因为它在分析此类问题具有较高的效率。
  - 在成型过程的末状态，毛坯的变形网格和它相关的材料状态被导入到Abaqus/Standard，用于回弹分析。
  - 在Abaqus/Standard中，回弹分析的效率高于Abaqus/Explicit。
  - 如果在*IMPORT选项中设置UPDATE=NO，则Abaqus/Standard计算的位移为成型过程和回弹分析的位移之和。

© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

- **平衡**
  - 导入变形的毛坯和它的当前状态到Abaqus/Standard之后，模型不是静力平衡的。 在Abaqus/Explicit中存在的，但是Abaqus/Standard不存在的动态力、接触力和边界条件，造成了静力不平衡：

    动态力：
    - 成型过程用动力学方法模拟，所以变形的毛坯是动力学平衡状态。模型中存在惯性力和阻尼力。
    - 在准静态成型模拟过程中，动力学平衡状态与静力学平衡状态非常接近。

    边界条件和接触条件：
    - 接触力不能导入。
    - 在导入分析中，可以修改边界条件。

© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

- **在回弹分析过程中得到静态平衡**
  - 当变形的毛坯带着材料状态导入到Abaqus/Standard后，Abaqus/Standard自动施加一组“伪”内力，用于平衡导入的应力。因此，在分析开始时，系统是静力平衡的。
  - 伪内力在回弹计算的分析步中逐步消失。
  - 当这些应力删除之后，作为内力重新分布的结果，毛坯进一步变形（一般称为回弹）。
  - 完全删除伪应力，或初始非平衡力之后，达到了回弹之后的最终构型。

© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

- **例子：圆柱罐的回弹计算**
  - Abaqus/Standard中下面的输入用于定义回弹分析：

```
*HEADING
Springback Calculation
*IMPORT, STEP=1, INTERVAL=10, UPDATE=YES, STATE=YES
BLANK,
*STEP, NLGEOM
*STATIC
0.1, 1.
*BOUNDARY
NodeX, XSYMM
NodeY, YSYMM
 1, 3, 3, 0.0
*RESTART, WRITE, FREQUENCY=5
*EL PRINT, ELSET=BLANK, FREQUENCY=99
S,
*END STEP
```

必需有足够的边界条件防止刚体运动

© Dassault Systèmes, 2008

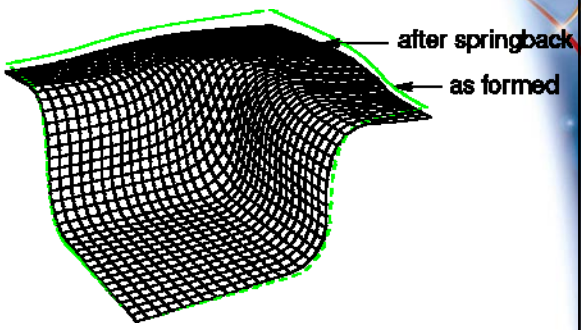
利用Abaqus/Standard计算回弹

- 单元集**BLANK是导入到** Abaqus/Standard中唯一的单元集。
- 节点集**NodeX和NodeY**被导入，在随后的分析中用于定义对称边界条件。
- 限制节点1沿z方向的位移，防止变形毛坯的刚体运动。
- 在*STEP选项中必须使用NLGEOM参数，因为Abaqus/Explicit在默认情况下包含几何非线性效应。
- *STATIC过程以增量的方式进行。初始的非平衡力随着时间增量的增加逐渐减小。

© Dassault Systèmes, 2008

利用Abaqus/Standard计算回弹

- 右图为回弹之后的构型。为了可视化的目的，将位移的放大因子设为10。



回弹后变形的构型

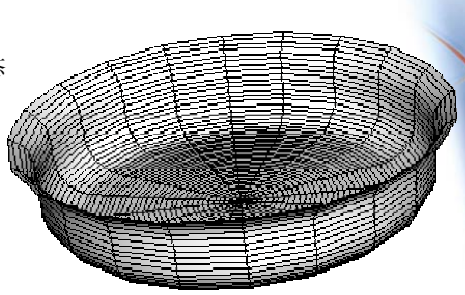
© Dassault Systèmes, 2008

习题8（可选）：准静态问题和导入分析

© Dassault Systèmes, 2008

习题8（可选）：准静态问题和导入分析

- **习题任务**
  1. 进行频率提取分析，为准静态过程确定合适的分析时间。
  2. 完成刚体工具的几何体定义，并包含接触和材料定义。



© Dassault Systèmes, 2008

习题8（可选）：准静态问题和导入分析

3. 利用SMOOTH STEP的幅值定义，改进准静态行为。
4. 在不造成结果退化的前提下，引入质量缩放，以减少分析时间。
5. 运行分析，并确定结果是否可以接受。
6. 将结果导入Abaqus/Standard进行分析，确定回弹。

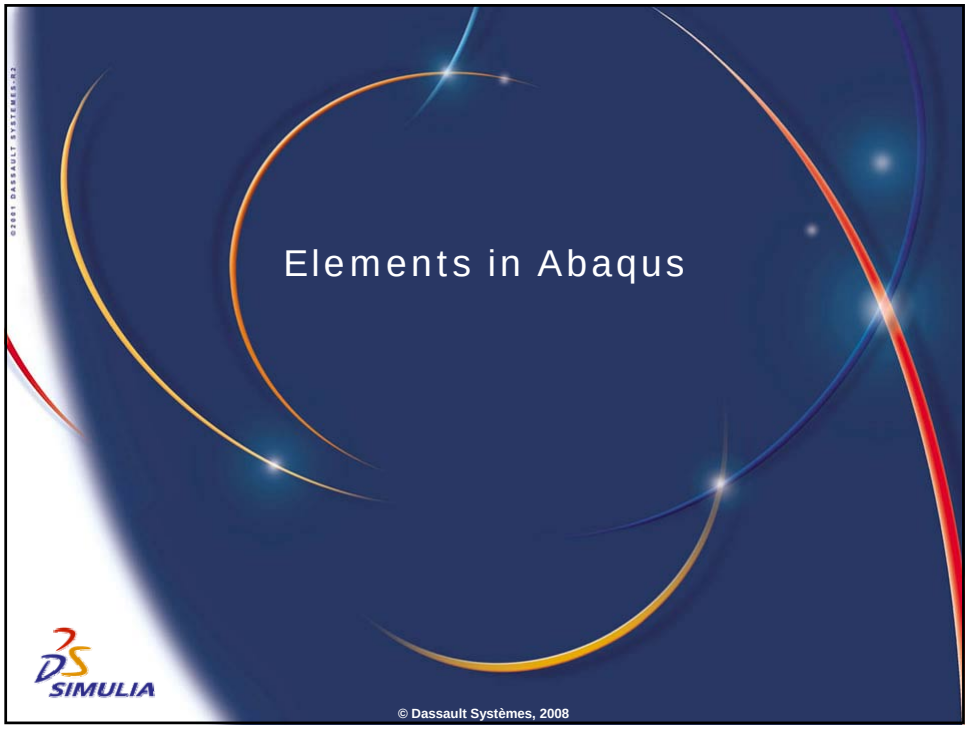
Video Clip

© Dassault Systèmes, 2008

Overview

- Elements in Abaqus
- Structural Elements (Shells and Beams) vs. Continuum Elements
- Modeling Bending Using Continuum Elements
- Stress Concentrations
- Contact
- Incompressible Materials
- Mesh Generation
- Solid Element Selection Summary

© Dassault Systèmes, 2008



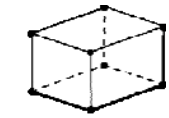
Elements in Abaqus

- **The wide range of elements in the Abaqus element library provides flexibility in modeling different geometries and structures.**
  - Each element can be characterized by considering the following:
    - Family
    - Number of nodes
    - Degrees of freedom
    - Formulation
    - Integration

© Dassault Systèmes, 2008

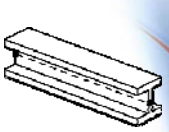
Elements in Abaqus

- **Family**
  - A family of finite elements is the broadest category used to classify elements.
  - Elements in the same family share many basic features.
  - There are many variations within a family.



continuum (solid elements)

shell elements

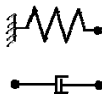


beam elements

rigid elements

membrane elements

infinite elements



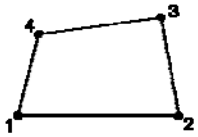
special-purpose elements like springs, dashpots, and masses

truss elements

© Dassault Systèmes, 2008

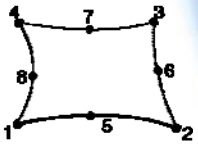
Elements in Abaqus

- **Number of nodes (interpolation)**
  - An element's number of nodes determines how the nodal degrees of freedom will be interpolated over the domain of the element.
  - Abaqus includes elements with both first- and second-order interpolation.



CPE4

First-order interpolation



CPE8

Second-order interpolation

© Dassault Systèmes, 2008

Elements in Abaqus

- **Degrees of freedom**
  - The primary variables that exist at the nodes of an element are the *degrees of freedom* in the finite element analysis.
  - Examples of degrees of freedom are:
    - Displacements
    - Rotations
    - Temperature
    - Electrical potential
  - Some elements have internal degrees of freedom that are not associated with the user-defined nodes.

© Dassault Systèmes, 2008

Elements in Abaqus

- **Formulation**
  - The mathematical formulation used to describe the behavior of an element is another broad category that is used to classify elements.
  - Examples of different element formulations:

Plane strain	Small-strain shells
Plane stress	Finite-strain shells
Hybrid elements	Thick shells
Incompatible-mode elements	Thin shells

© Dassault Systèmes, 2008

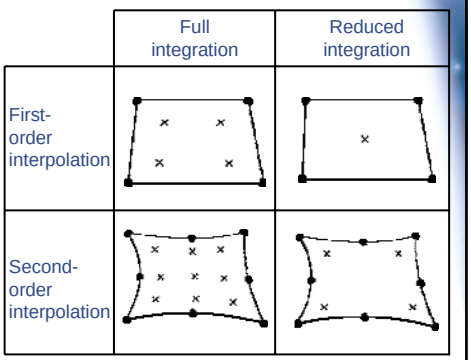
Elements in Abaqus

- **Integration**
  - The stiffness and mass of an element are calculated numerically at sampling points called “integration points” within the element.
  - The numerical algorithm used to integrate these variables influences how an element behaves.

© Dassault Systèmes, 2008

Elements in Abaqus

- Abaqus includes elements with both “full” and “reduced” integration.
  - **Full integration:**
    - The minimum integration order required for exact integration of the strain energy for an undistorted element with linear material properties.
  - **Reduced integration:**
    - The integration rule that is one order less than the full integration rule.

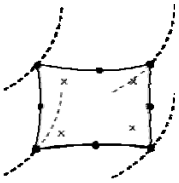


© Dassault Systèmes, 2008

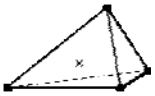
Elements in Abaqus

- **Element naming conventions: examples**

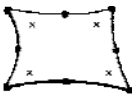
B21: **B**eam, **2**-D, **1**st-order interpolation



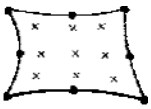
CAX8R: **C**ontinuum, **AX**isymmetric, **8**-node, **R**educed integration



DC3D4: **D**iffusion (heat transfer), **C**ontinuum, **3**-**D**, **4**-node



S8RT: **S**hell, **8**-node, **R**educed integration, **T**emperature



CPE8PH: **C**ontinuum, **P**lane strain, **8**-node, **P**ore pressure, **H**ybrid

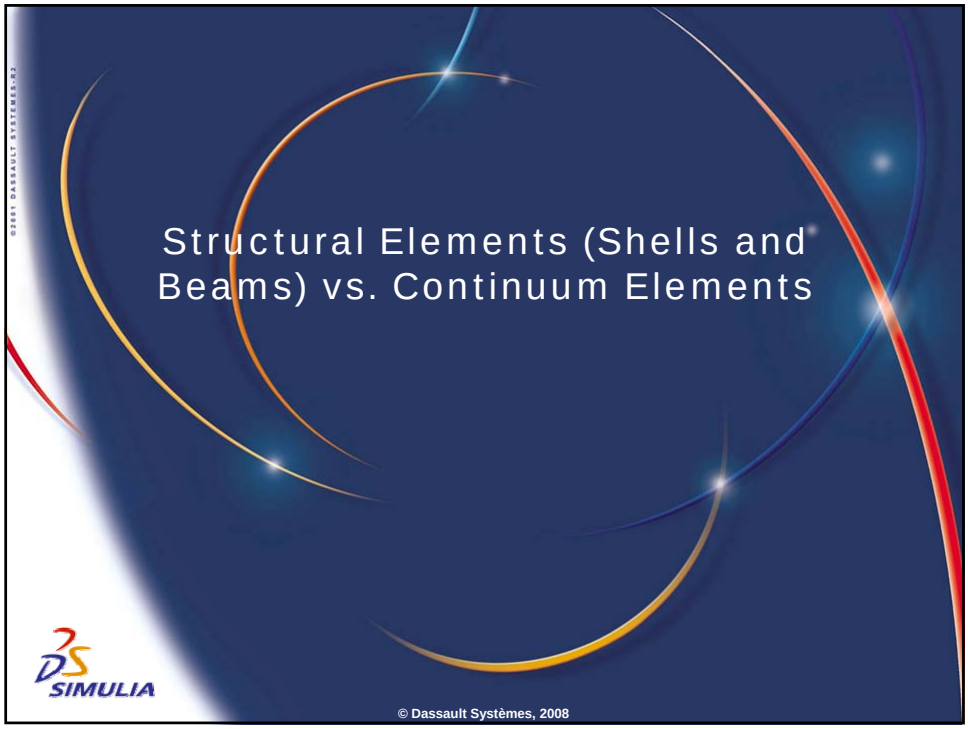
DC1D2E: **D**iffusion (heat transfer), **C**ontinuum, **1**-**D**, **2**-node, **E**lectrical

© Dassault Systèmes, 2008

Elements in Abaqus

- **Comparing Abaqus/Standard and Abaqus/Explicit element libraries**
  - Both programs have essentially the same element families: continuum, shell, beam, etc.
  - Abaqus/Standard includes elements for many analysis types besides stress analysis: heat transfer, soils consolidation, acoustics, etc.
    - Acoustic elements are also available in Abaqus/Explicit.
  - Abaqus/Standard includes many more variations within each element family.
  - Abaqus/Explicit includes mostly first-order reduced-integration elements.
    - Exceptions: second-order triangular and tetrahedral elements; second-order beam elements; first-order fully-integrated brick (including incompatible mode version), shell, and membrane elements.
  - Many of the same general element selection guidelines apply to both programs.

© Dassault Systèmes, 2008



Structural Elements (Shells and Beams) vs. Continuum Elements

- **Continuum (solid) element models can be large and expensive, particularly in three-dimensional problems.**
- **If appropriate, structural elements (shells and beams) should be used for a more economical solution.**
  - A structural element model typically requires far fewer elements than a comparable continuum element model.
- **For structural elements to produce acceptable results, the shell thickness or the beam cross-section dimensions should be less than 1/10 of a typical global structural dimension, such as:**
  - The distance between supports or point loads
  - The distance between gross changes in cross section
  - The wavelength of the highest vibration mode

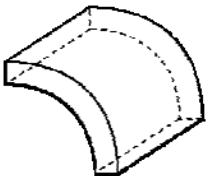
© Dassault Systèmes, 2008

A1.15

Structural Elements (Shells and Beams) vs. Continuum Elements

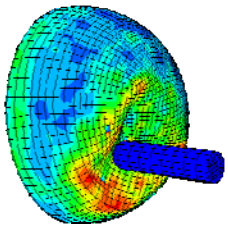
- **Shell elements**
  - Shell elements approximate a three-dimensional continuum with a surface model.
    - Model bending and in-plane deformations efficiently.
  - If a detailed analysis of a region is needed, a local three-dimensional continuum model can be included using multi-point constraints or submodeling.

Video Clip



3-D continuum

surface model



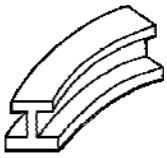
shell model of a hemispherical dome subjected to a projectile impact

© Dassault Systèmes, 2008

A1.16

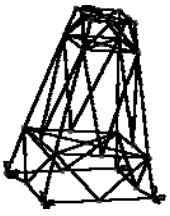
Structural Elements (Shells and Beams) vs. Continuum Elements

- **Beam elements**
  - Beam elements approximate a three-dimensional continuum with a line model.
    - Model bending, torsion, and axial forces efficiently.
    - Many different cross-section shapes are available.
    - Cross-section properties can also be specified by providing engineering constants.



3-D continuum

line model



framed structure modeled using beam elements

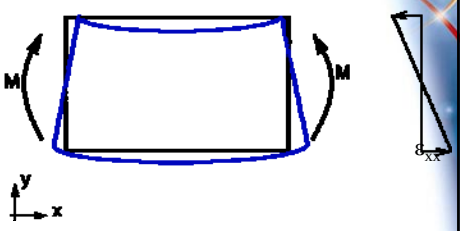
© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

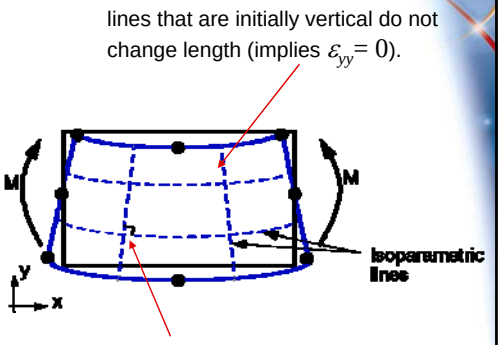
- **Physical characteristics of pure bending**
  - This is the assumed behavior of the material that finite elements attempt to model.
    - Plane cross-sections remain plane throughout the deformation.
    - The axial strain ε_{xx} varies linearly through the thickness.
    - The strain in the thickness direction ε_{yy} is zero if $\nu = 0$.
    - No membrane shear strain.
      - Implies that lines parallel to the beam axis lie on a circular arc.



© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

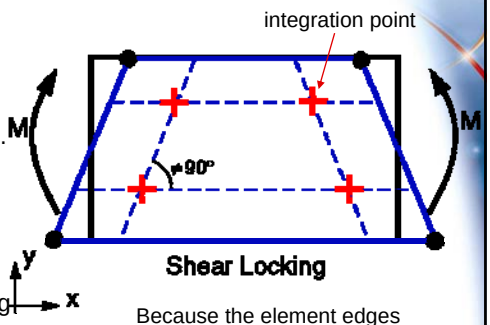
- **Modeling bending using second-order solid elements (CPE8, C3D20R, …)**
  - Second-order full- and reduced-integration solid elements model bending accurately:
  - The axial strain equals the change in length of the initially horizontal lines.
  - The thickness strain is zero.
  - The shear strain is zero.



© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

- **Modeling bending using first-order fully integrated solid elements (CPS4, CPE4, C3D8)**
  - These elements detect shear strains at the integration points.
    - Nonphysical; present solely because of the element formulation used.
  - Overly stiff behavior results from energy going into shearing the element rather than bending it (called “shear locking”).

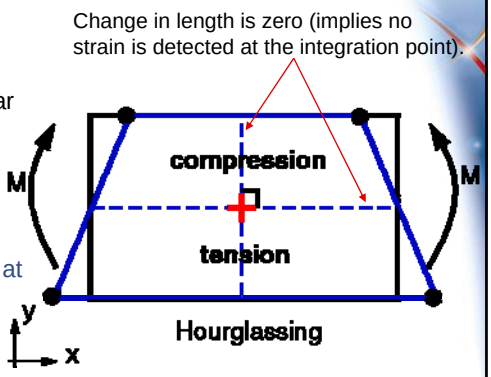


Do not use these elements in regions dominated by bending!

© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

- **Modeling bending using first-order reduced-integration elements (CPE4R, ...)**
  - These elements eliminate shear locking.
  - However, *hourglassing* is a concern when using these elements.
    - Only one integration point at the centroid.
    - A single element through the thickness does not detect strain in bending.
    - Deformation is a zero-energy mode (deformation but no strain; called "hourglassing").

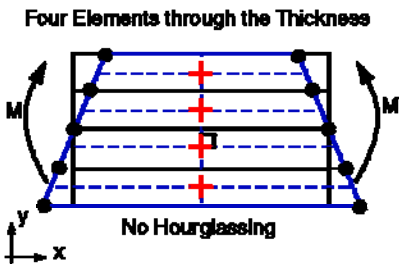


Bending behavior for a single first-order reduced-integration element

© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

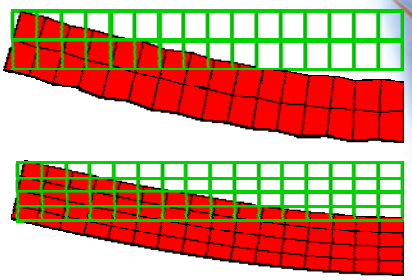
- Hourglassing can propagate easily through a mesh of first-order reduced-integration elements, causing unreliable results.
- Hourglassing is not a problem if you use multiple elements—at least four through the thickness.
  - Each element captures either compressive or tensile axial strains but not both.
  - The axial strains are measured correctly.
  - The thickness and shear strains are zero.
  - Cheap and effective elements.



© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

- **Detecting and controlling hourglassing**
  - Hourglassing can usually be seen in deformed shape plots.
    - Example: Coarse and medium meshes of a simply supported beam with a center point load.
  - Abaqus has built-in hourglass controls that limit the problems caused by hourglassing.
    - Verify that the artificial energy used to control hourglassing is small (<1%) relative to the internal energy.

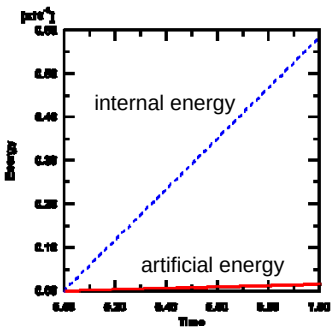


Same load and displacement magnification (1000×)

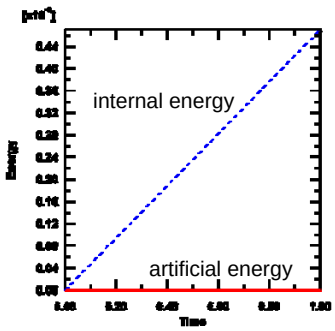
© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

- Use the *X*–*Y* plotting capability in Abaqus/Viewer to compare the energies graphically.



Two elements through the thickness: Ratio of artificial to internal energy is 2%



Four elements through the thickness: Ratio of artificial to internal energy is 0.1%

© Dassault Systèmes, 2008

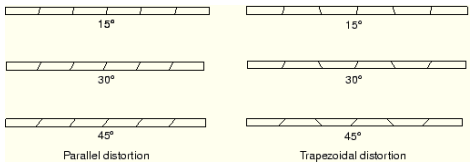
Modeling Bending Using Continuum Elements

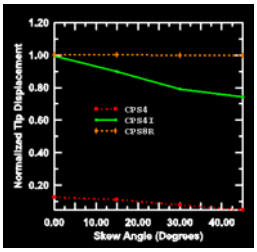
- **Modeling bending using incompatible mode elements (CPS4I, …)**
  - Perhaps the most cost-effective solid continuum elements for bending-dominated problems.
  - Compromise in cost between the first- and second-order reduced-integration elements, with many of the advantages of both.
    - Model shear behavior correctly—no shear strains in pure bending.
    - Model bending with only one element through the thickness.
    - No hourglass modes, and work well in plasticity and contact problems.
  - The advantages over reduced-integration first-order elements are reduced if the elements are severely distorted; however, all elements perform less accurately if severely distorted.

© Dassault Systèmes, 2008

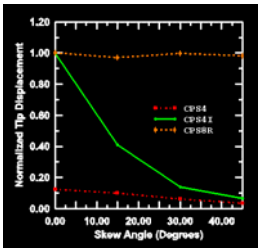
Modeling Bending Using Continuum Elements

- **Example: Cantilever beam with distorted elements**





Parallel distortion



Trapezoidal distortion

© Dassault Systèmes, 2008

Modeling Bending Using Continuum Elements

- **Summary**

Element type	ε_{xx}	ε_{yy}	ε_{xy}	Notes
Physical behavior	$\neq 0$	0	0	
Second-order	$\neq 0$	0	0	OK
First-order, full integration	$\neq 0$	$\neq 0$	$\neq 0$	Shear locking
First-order, reduced integration	0	0	0	Hourglassing if too few elements through thickness
	$\neq 0$	0	0	OK if enough elements through the thickness
Incompatible mode	$\neq 0$	0	0	OK if not overly distorted

© Dassault Systèmes, 2008

Stress Concentrations

© Dassault Systèmes, 2008

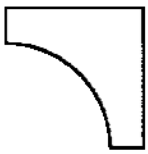
Stress Concentrations

- **Second-order elements clearly outperform first-order elements in problems with stress concentrations and are ideally suited for the analysis of (stationary) cracks.**
  - Both fully integrated and reduced-integration elements work well.
  - Reduced-integration elements tend to be somewhat more efficient—results are often as good or better than full integration at lower computational cost.

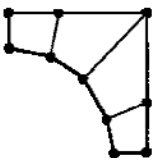
© Dassault Systèmes, 2008

Stress Concentrations

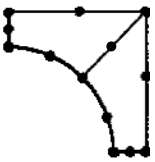
- Second-order elements capture geometric features, such as curved edges, with fewer elements than first-order elements.



Physical model



Model with first-order elements—element faces are straight line segments.

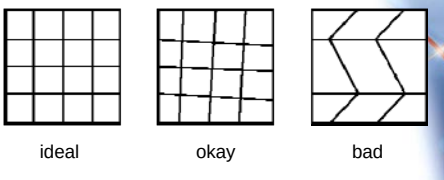


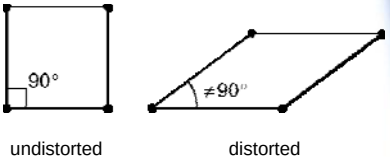
Model with second-order elements—element faces are quadratic curves.

© Dassault Systèmes, 2008

Stress Concentrations

- Both first- and second-order quads and bricks become less accurate when their initial shape is distorted.
  - First-order elements are known to be less sensitive to distortion than second-order elements and, thus, are a better choice in problems where significant mesh distortion is expected.
- Second-order triangles and tetrahedra are less sensitive to initial element shape than most other elements; however, well-shaped elements provide better results.

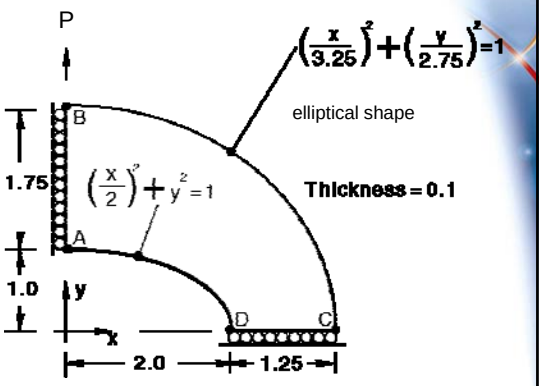




© Dassault Systèmes, 2008

Stress Concentrations

- A typical stress concentration problem, a NAFEMS benchmark problem, is shown at right. The analysis results obtained with different element types follow.



© Dassault Systèmes, 2008

Stress Concentrations

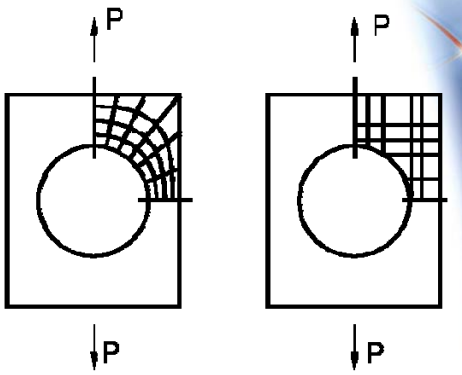
- First-order elements (including incompatible mode elements) are relatively poor in the study of stress concentration problems.

Element type	σ_{yy} at D (Target=100.0)	
	Coarse mesh	Fine mesh
CPS3	55.06	76.87
CPS4	71.98	91.2
CPS4I	63.45	84.37
CPS4R	43.67	60.6
CPS6	96.12	101.4
CPS8	91.2	100.12
CPS8R	92.56	97.16

© Dassault Systèmes, 2008

Stress Concentrations

- Second-order elements such as CPS6, CPS8, and CPS8R give much better results.
- Well-shaped, second-order, reduced-integration quadrilaterals and hexahedra can provide high accuracy in stress concentration regions.
  - Distorted elements reduce the accuracy in these regions.



Distortion in low strain gradient area

Distortion in high strain gradient area

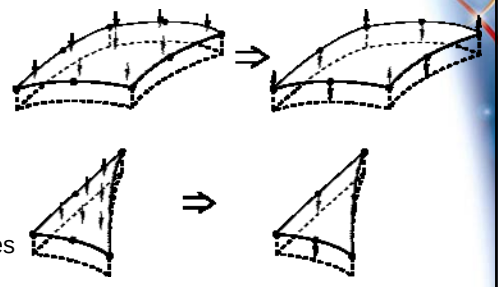
© Dassault Systèmes, 2008

Contact

© Dassault Systèmes, 2008

Contact

- **Almost all element types are formulated to work well in contact problems using the (default) “hard” contact formulation, with the following exceptions:**
  - Second-order quad/hex elements
  - “Regular” second-order tri/tet (as opposed to “modified” ones whose names end with the letter “M”), second-order wedge, and 6-node shell and membrane elements.
- **Convergence difficulties may arise with these elements when using “hard” contact (use penalty contact instead).**



For the second-order tet, wedge, and 6-node shell and membrane elements, the directions of the consistent nodal forces resulting from a pressure load are not uniform with “classical” hard contact; use penalty-based contact instead.

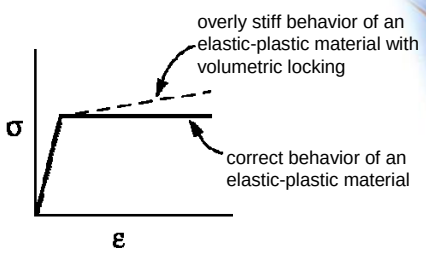
© Dassault Systèmes, 2008

Incompressible Materials

© Dassault Systèmes, 2008

Incompressible Materials

- **Many nonlinear problems involve incompressible materials ($\nu = 0.5$) and nearly incompressible materials ($\nu > 0.475$).**
  - Rubber
  - Metals at large plastic strains
- Conventional finite element meshes often exhibit overly stiff behavior due to *volumetric locking*, which is most severe when these materials are highly confined.



Example of the effect of volumetric locking

© Dassault Systèmes, 2008

Incompressible Materials

- For an incompressible material each integration point's volume must remain almost constant. This overconstrains the kinematically admissible displacement field and causes volumetric locking
  - For example, in a refined three-dimensional mesh of 8-node hexahedra, there is—on average—1 node with 3 degrees of freedom per element.
  - The volume at each integration point must remain fixed.
  - Fully integrated hexahedra use 8 integration points per element; thus, in this example, we have as many as 8 constraints per element, but only 3 degrees of freedom are available to satisfy these constraints.
  - The mesh is overconstrained—it "**locks**."
- Volumetric locking is most pronounced in fully integrated elements.
- Reduced-integration elements have fewer volumetric constraints.
  - Reduced integration effectively eliminates volumetric locking in many problems with nearly incompressible material.

© Dassault Systèmes, 2008

Incompressible Materials

- Fully incompressible materials modeled with solid elements must use the "hybrid" formulation (elements whose names end with the letter "H").
  - In this formulation the pressure stress is treated as an independently interpolated basic solution variable, coupled to the displacement solution through the constitutive theory.
  - Hybrid elements introduce more variables into the problem to alleviate the volumetric locking problem. The extra variables also make them more expensive.
  - The Abaqus element library includes hybrid versions of all continuum elements (except plane stress elements, where this is not needed).

© Dassault Systèmes, 2008

Incompressible Materials

- Hybrid elements are only necessary for:
  - All meshes with strictly incompressible materials, such as rubber.
  - Refined meshes of reduced-integration elements that still show volumetric locking problems. Such problems are possible with elastic-plastic materials strained far into the plastic range.
- Even with hybrid elements a mesh of first-order triangles and tetrahedra is overconstrained when modeling fully incompressible materials. Hence, these elements are recommended only for use as “fillers” in quadrilateral or brick-type meshes with such material.

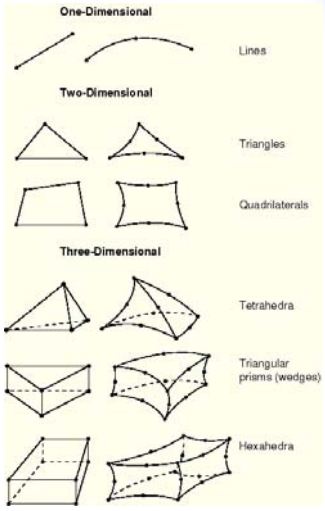
© Dassault Systèmes, 2008

Mesh Generation

© Dassault Systèmes, 2008

Mesh Generation

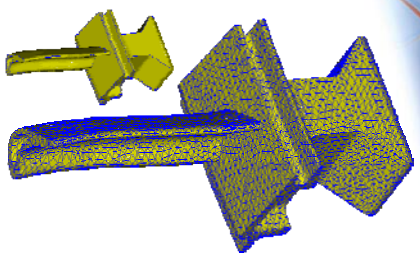
- **Mesh module**
  - Elements are generated in the Mesh module of Abaqus/CAE.
  - Meshes containing the element shapes shown at right can be generated.
  - Most elements in Abaqus are topologically equivalent to these shapes.
    - For example, CPE4 (stress), DC2D4 (heat transfer), and AC2D4 (acoustics) are topologically equivalent to a linear quadrilateral.



© Dassault Systèmes, 2008

Mesh Generation

- **Quad/hex vs. tri/tet elements**
  - Of particular importance when generating a mesh is the decision regarding whether to use quad/hex or tri/tet elements.
  - Quad/hex elements should be used wherever possible.
    - They give the best results for the minimum cost.
    - When modeling complex geometries, however, the analyst often has little choice but to mesh with triangular and tetrahedral elements.



Turbine blade with platform modeled with tetrahedral elements

© Dassault Systèmes, 2008

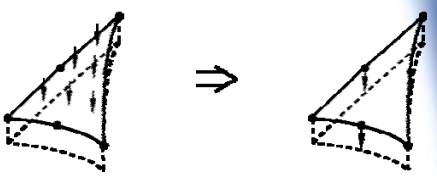
Mesh Generation

- First-order tri/tet elements (CPE3, CPS3, CAX3, C3D4, C3D6) are poor elements; they have the following problems:
  - Poor convergence rate.
    - They typically require very fine meshes to produce good results.
  - Volumetric locking with incompressible or nearly incompressible materials, even using the “hybrid” formulation.
- These elements should be used only as fillers in regions far from any areas where accurate results are needed.

© Dassault Systèmes, 2008

Mesh Generation

- “Regular” second-order tet, second-order wedge, and 6-node shell and membrane elements (C3D10, C3D15, STRI65, M3D6) should not be used to model contact unless a **penalty**-based contact formulation is used.
  - Under uniform pressure the contact forces are significantly different at the corner and midside nodes with “classical” hard contact.
- Second-order triangles (CAX6, CPE6, CPS6) may show a noisy contact distribution and may cause convergence difficulties.



Equivalent nodal forces created by uniform pressure on the face of a regular second-order tetrahedral element (C3D10) with “classical” hard contact

© Dassault Systèmes, 2008

Mesh Generation

- **Modified** second-order tri/tet elements (C3D10M, etc.) alleviate the problems of other tri/tet elements.
  - Good convergence rate—close to convergence rate of second-order quad/hex elements.
  - Minimal shear or volumetric locking.
    - Can be used to model incompressible or nearly incompressible materials in the hybrid formulation (C3D10MH).
  - These elements are robust during finite deformation.
  - Uniform contact pressure allows these elements to model contact accurately.

Use them!

© Dassault Systèmes, 2008

Mesh Generation

- **Mesh refinement and convergence**
  - Use a sufficiently refined mesh to ensure that the results from your Abaqus simulation are adequate.
    - Coarse meshes tend to yield inaccurate results.
    - The computer resources required to run your job increase with the level of mesh refinement.
  - It is rarely necessary to use a uniformly refined mesh throughout the structure being analyzed.
    - Use a fine mesh only in areas of high gradients and a coarser mesh in areas of low gradients.
  - Can often predict regions of high gradients before generating the mesh.
    - Use hand calculations, experience, etc.
    - Alternatively, you can use coarse mesh results to identify high gradient regions.

© Dassault Systèmes, 2008

Mesh Generation

- Some recommendations:
  - Minimize mesh distortion as much as possible.
  - A minimum of four quadratic elements per 90^o should be used around a circular hole.
  - A minimum of four elements should be used through the thickness of a structure if first-order, reduced integration solid elements are used to model bending.
  - Other guidelines can be developed based on experience with a given class of problem.

© Dassault Systèmes, 2008

Mesh Generation

- It is good practice to perform a mesh convergence study.
  - Simulate the problem using progressively finer meshes, and compare the results.
    - The mesh density can be changed very easily using Abaqus/CAE since the definition of the analysis model is based on the geometry of the structure.
  - When two meshes yield nearly identical results, the results are said to have “converged.”
    - This provides increased confidence in your results.

© Dassault Systèmes, 2008

Solid Element Selection Summary

SIMULIA

Class of problem	Best element choice	Avoid using
General contact between deformable bodies	First-order quad/hex	Second-order quad/hex
Contact with bending	Incompatible mode	First-order fully integrated quad/hex or second-order quad/hex
Bending (no contact)	Second-order quad/hex	First-order fully integrated quad/hex
Stress concentration	Second-order	First-order
Nearly incompressible (ν>0.475 or large strain plasticity e_{pl}>10%)	First-order elements or second-order reduced integration elements	Second-order fully integrated

© Dassault Systèmes, 2008

Solid Element Selection Summary

Class of problem	Best element choice	Avoid using
Completely incompressible (rubber $\nu = 0.5$)	Hybrid quad/hex, first-order if large deformations are anticipated	
Bulk metal forming (high mesh distortion)	First-order reduced integration quad/hex	Second-order quad/hex
Complicated model geometry (linear material, no contact)	Second-order quad/hex if possible (if not overly distorted) or second-order tet/tri (because of meshing difficulties)	
Complicated model geometry (nonlinear problem or contact)	First-order quad/hex if possible (if not overly distorted) or modified second-order tet/tri (because of meshing difficulties)	
Natural frequency (linear dynamics)	Second-order	
Nonlinear dynamic (impact)	First-order	Second-order

© Dassault Systèmes, 2008

Workshop Preliminaries

Setting up the workshop directories and files

If you are taking a public seminar, the steps in the following section have already been done for you: skip to **Basic Operating System Commands, (p. WP.2)**. If everyone in your group is familiar with the operating system, skip directly to the workshops.

The workshop files are included on the Abaqus release CD. If you have problems finding the files or setting up the directories, ask your systems manager for help.

Note for systems managers: If you are setting up these directories and files for someone else, please make sure that there are appropriate privileges on the directories and files so that the user can write to the files and create new files in the directories.

Workshop file setup

(Note: UNIX is case-sensitive. Therefore, lowercase and uppercase letters must be typed as they are shown or listed.)

1. Find out where the Abaqus release is installed by typing

   UNIX and Windows NT: **`abq`***xxx* **`whereami`**

   where **`abq`***xxx* is the name of the Abaqus execution procedure on your system. It can be defined to have a different name. For example, the command for the 6.8–1 version might be aliased to **`abq681`**.

   This command will give the full path to the directory where Abaqus is installed, referred to here as *abaqus_dir*.

2. Extract all the workshop files from the course tar file by typing

   UNIX: **`abq`***xxx* **`perl`** *abaqus_dir*`/samples/course_setup.pl`

   Windows NT: **`abq`***xxx* **`perl`** *abaqus_dir*`\samples\course_setup.pl`

   Note that if you have Perl and the compilers already installed on your machine, you may simply type:

   UNIX: *abaqus_dir*`/samples/course_setup.pl`

   Windows NT: *abaqus_dir*`\samples\course_setup.pl`

3. The script will install the files into the current working directory. You will be asked to verify this and to choose which files you wish to install. Choose "**`y`**" for the appropriate lecture series when prompted. Once you have selected the lecture series, type "**`q`**" to skip the remaining lectures and to proceed with the installation of the chosen workshops.

© Dassault Systèmes, 2008

Basic operating system commands

(You can skip this section and go directly to the workshops if everyone in your group is familiar with the operating system.)

Note: The following commands are limited to those necessary for doing the workshop exercises.

Working with directories

1. Start in the current working directory. List the directory contents by typing

   UNIX: **`ls`**

   Windows NT: **`dir`**

   Both subdirectories and files will be listed. On some systems the file type (directory, executable, etc.) will be indicated by a symbol.
2. Change directories to a workshop subdirectory by typing

   Both UNIX and Windows NT: **`cd`** *dir_name*
3. To list with a long format showing sizes, dates, and file, type

   UNIX: **`ls -l`**

   Windows NT: **`dir`**
4. Return to your home directory:

   UNIX: **`cd`**

   Windows NT: **`cd`** *home-dir*

   List the directory contents to verify that you are back in your home directory.
5. Change to the workshop subdirectory again.
6. The **`*`** is a wildcard character and can be used to do a partial listing. For example, list only Abaqus input files by typing

   UNIX: **`ls *.inp`**

   Windows NT: **`dir *.inp`**

Working with files

Use one of these files, *filename*`.inp`, to perform the following tasks:

1. Copy *filename*`.inp` to a file with the name **`newcopy.inp`** by typing

   UNIX: **`cp`** *filename*`.inp newcopy.inp`

   Windows NT: **`copy`** *filename*`.inp newcopy.inp`
2. Rename (or move) this new file to **`newname.inp`** by typing

   UNIX: **`mv newcopy.inp newname.inp`**

   Windows NT: **`rename newcopy.inp newname.inp`**

   (Be careful when using **`cp`** and **`mv`** since UNIX will overwrite existing files without warning.)

© Dassault Systèmes, 2008

3. Delete this file by typing
   UNIX: **`rm newname.inp`**
   Windows NT: **`erase newname.inp`**
4. View the contents of the files *filename*`.inp` by typing
   UNIX: **`more`** *filename*`.inp`
   Windows NT: **`type`** *filename*`.inp | more`
   This step will scroll through the file one page at a time.

Now you are ready to start the workshops.

Workshop 1

Basic Input and Output

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Learn to use Abaqus utilities and documentation.
- Understand the basic structure of an Abaqus/CAE model, and be able to make simple modifications to it.
- Learn how to perform a **datacheck** analysis and how to submit an analysis job in Abaqus/CAE.
- Gain familiarity with the Visualization module.
- Explore the structure and contents of the printed output (**`.dat`**) file.

Abaqus utilities and documentation

Abaqus provides various utilities for obtaining information on usage, system configuration, example problems, and environment settings for the analysis package.

1. At the prompt, enter the command

   `abaqus information=system`

   to obtain information on the system.

   Note that **`abaqus`** is a generic command that may have been renamed on your system. For example, if more than one version is installed on the system, the command might include the version number, as in **`abq681`**. In the remainder of this workshop as well as all subsequent workshops, use the appropriate command for your system.

Question W1–1: What is the processor on your machine?

Question W1–2: What is the operating system (OS) level?

2. Open the online documentation with the command

   `abaqus doc`

   Open the Abaqus Analysis User's Manual, and search for the string **DSLOAD** to find information on the *DSLOAD option. You can find information related to the data line syntax in the Abaqus Keywords Reference Manual (use the hyperlink for the *DSLOAD option, or open the Keywords Manual directly). The online documentation graphical user interface is shown in Figure W1–1.

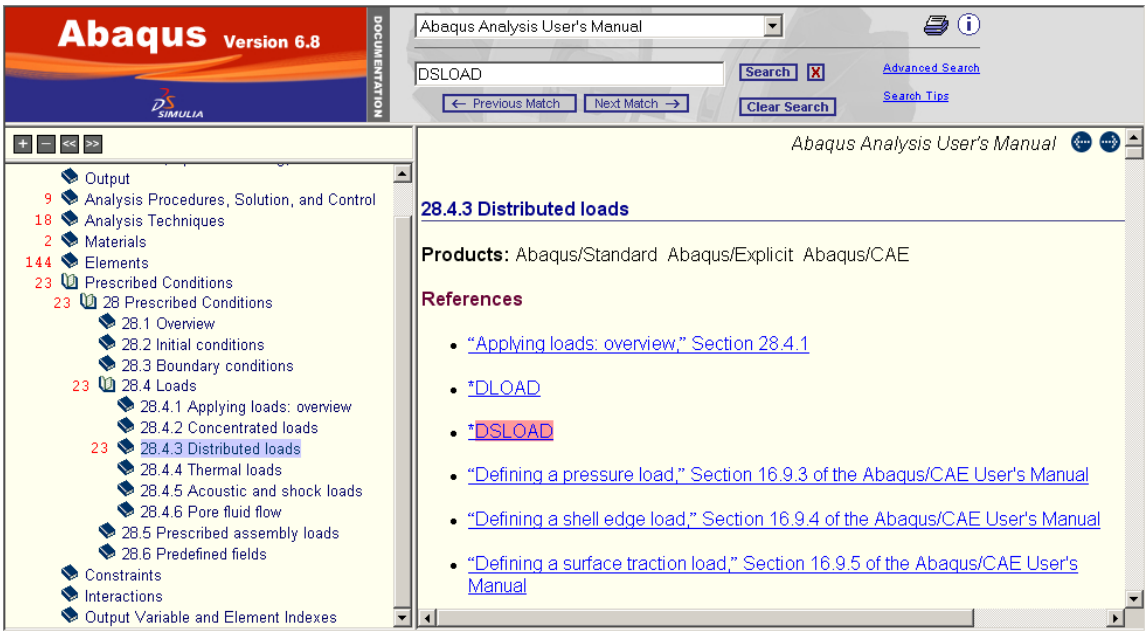


Figure W1–1. Online documentation

3. Open the online Abaqus Example Problems Manual. Search for **`plate buckling`** to find example problems that discuss plate buckling.

Question W1–3: What are the three example problems that fit the search criteria?

4. Go to Example Problem 1.1.14 in the online Abaqus Example Problems Manual. In the left panel of the window, display the subtopics of the problem and click **Input files**. In the right panel of the window, the list of input files associated with this problem appears. You can select any input filename from the list; a separate window will open containing that file.

5. All example problem input files are included in the Abaqus release and can be obtained using the **abaqus fetch** utility. In your terminal window, enter

   `abaqus fetch job=damagefailcomplate_cps4`

   at the command line prompt.

6. Use the online documentation to determine the input syntax for some options.

   A keyword line starts with an asterisk (*) followed directly by the keyword option. Parameters and their associated values appear on the keyword line, separated by commas. Many options require data lines, which follow directly after their associated keyword line and contain the data specified in the Abaqus Keywords Reference Manual for each option. Data items are separated by commas. Refer to the discussions of keyword line and data line syntax in Lecture 1, as necessary.

   Question W1–4: How would you run a script from within the Abaqus/CAE environment?

   Hint: Search for "run script" in the Abaqus/CAE User's Manual

   Question W1–5: In the space provided, write which **Category** option you would choose to define a displacement/rotation boundary condition in Abaqus/CAE.

   Hint: Search displacement/rotation boundary condition in the Abaqus/CAE User's Manual.

© Dassault Systèmes, 2008

Analyzing a connecting lug

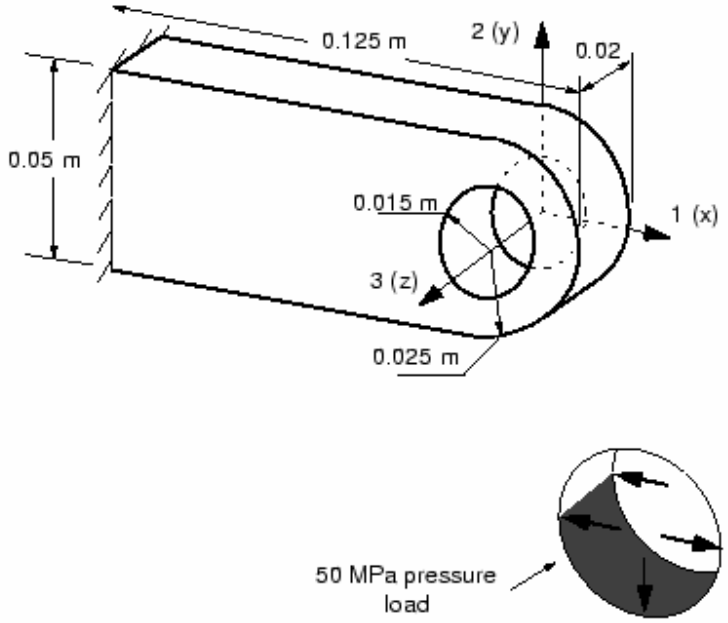


Figure W1–2. Sketch of the connecting lug

In this workshop you will model the connecting lug shown in Figure W1–2. The lug is welded to a massive structure at one end, so we assume that this end is fixed. The other end contains a hole through which a bolt is placed when the lug is in service. You have to calculate the deflection of the lug when a load of 30 kN is applied to the bolt along the negative 2-direction.

To model this problem, you will use three-dimensional continuum elements and perform a linear analysis with elastic materials. You will model the load transmitted to the lug through the bolt as a uniform pressure load applied to the bottom half of the hole, as shown in Figure W1–2. In this workshop SI units (N, m, and s) will be used.

Preliminaries

1. Enter the working directory for this workshop:
   `../abaqus_solvers/interactive/workshop1`
2. Run the script **`ws_solver_lug.py`** using the following command:
   `abaqus cae startup=ws_solver_lug.py`

The above command creates an Abaqus/CAE database named **`Lug.cae`** in the current directory. The geometry, mesh, and step definitions for the lug are included in a model named **`standard`**.

© Dassault Systèmes, 2008

Before completing the model, view the contents of the model using the Model Tree on the left hand side of the main window.

Question W1–6: How many steps are there in this analysis?

Use the **Query information** tool (or select **Tools→Query** from the main menu bar) to query element information of the lug. Switch to the Mesh module and click . In the **Query** dialog box, select **Element** in the **General Queries** field. Select one element of the lug in the viewport. Read the query results reported in the message area at the bottom of the main window.

Question W1–7: What element type is used to model the lug?

Completing the model

You will now add the material definition, and create the boundary conditions and the pressure load to complete the lug model.

1. Note that a dummy material named **`Steel`** has already been created and assigned to the part **Lug**. Add the steel material properties to this material.
   a. In the Model Tree, expand the **Materials** container and double-click **Steel**.
      The material editor appears.
   b. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic**. Enter the following elastic material properties: Elastic modulus E = 200.E9 Pa and Poisson's ratio ν = 0.3.

   Question W1–8: Do you need to define a density to complete the material definition? Material density is necessary for what types of analyses?

2. In the Model Tree, double-click the **BCs** container to create a ENCASTRE boundary condition named on the flat end as highlighted in Figure W1–3.
   a. In the **Create Boundary Condition** dialog box, name the boundary condition **`Fix left end`**, choose the category **Mechanical** and the type **Symmetry/Antisymmetry/Encastre**, and click **Continue**.
   b. Select the flat end of the lug as shown in Figure W1–3. Use **[Shift]+Click** to select both regions. Adjust your view, if necessary, to see the model geometry more clearly.
   c. Click mouse button 2 in the viewport or click **Done** in the prompt area to confirm the selections.
      The boundary condition editor appears.
   d. The **Edit Boundary Condition** dialog box, choose **ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0)** and click **OK** to exit the boundary condition editor.

© Dassault Systèmes, 2008

The arrow symbols appear on the flat end indicating the constrained degrees of freedom. The ENCASTRE boundary condition constrains all active structural degrees of freedom.

Question W1–9: How else could you define a completely constrained boundary condition?

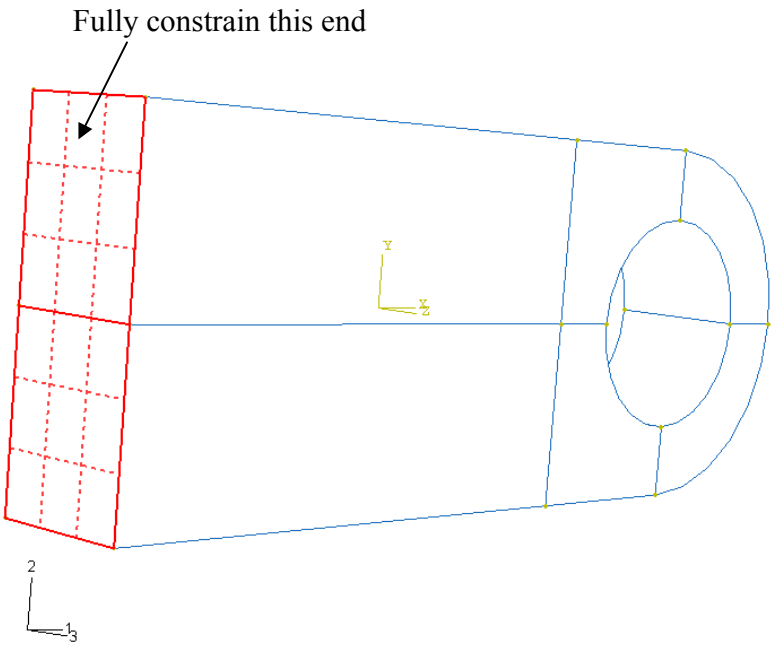


Figure W1–3. Region for fully constrained boundary condition

3. In the Model Tree, double-click the **Loads** container to create a distributed pressure load with a magnitude of 50 MPa on the highlighted surfaces shown in Figure W1–4.
   a. In the **Create Load** dialog box, name the load **`Pressure Load`**, select the step **LugLoad**, choose the category **Mechanical** and the type **Pressure**, and click **Continue**.
   b. Select the surfaces highlighted in Figure W1–4.
   c. Click mouse button 2 in the viewport or click **Done** in the prompt area to confirm the selections.

      The load editor appears.
   d. In the **Edit Load** dialog box, accept the **Uniform** distribution, enter a value of **`50E6`** for the **Magnitude**, and click **OK** to exit the load editor.

© Dassault Systèmes, 2008

Note that the magnitude of the applied uniform pressure is determined by dividing the total load by the projected horizontal area of the hole, where $\frac{30\text{kN}}{2\times 0.015\text{m}\times 0.02\text{m}} = 50\text{MPa}$.

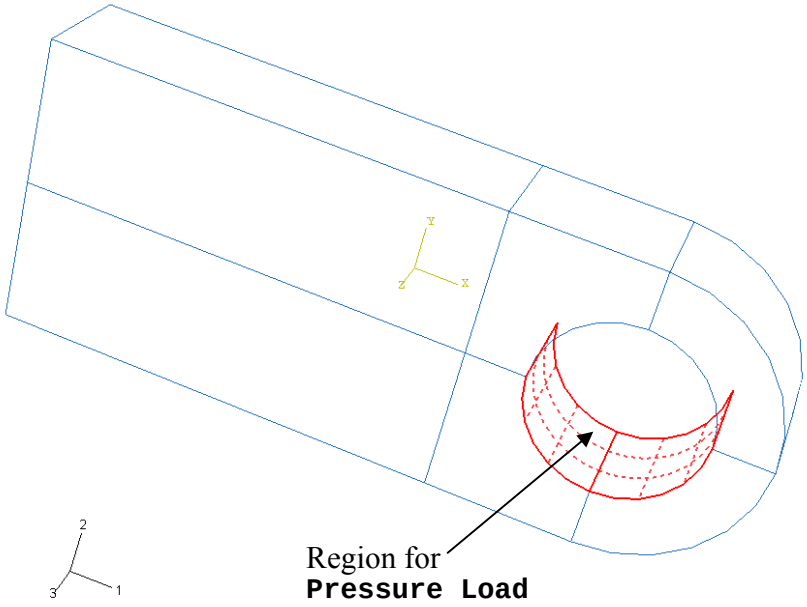


Figure W1–4. Region for distributed pressure load

Submitting a datacheck analysis

You will first perform a **datacheck** analysis and then a full analysis.

1. In the Model Tree, double-click the **Jobs** container. In the **Create Job** dialog box, name the job **`lug`** and click **Continue**. In the **Edit Job** dialog box, accept all default settings and click **OK** to exit the job editor.
2. Save your model database.
3. In the Model Tree, expand the **Jobs** container. Click mouse button 3 on the job **lug** and select **Data Check** from the menu that appears.
4. Click mouse button 3 on the job **lug** and select **Monitor** from the menu that appears to monitor any warnings or errors that may occur during the **datacheck** analysis.
5. In your present working directory, open the data file (**`lug.dat`**) using any text editor. Search for the string **P R O B L E M** to see the summary of the problem size. Include spaces between the letters of the search string.

   Question W1–10: How many elements are there in the model? How many variables are there?

Running a full analysis

1. In the Model Tree, expand the **Jobs** container. Click mouse button 3 on the job **lug** and select **Submit** from the menu that appears.

   If Abaqus/CAE asks if you want to overwrite old job files, click **OK**. This means that output files with the same job name that exist from a previous analysis will be overwritten.
2. Monitor the job's progress.

Postprocessing the results

When the analysis is complete, use the following procedure to view the analysis results in the Visualization module:

1. In the Model Tree, click mouse button 3 on the job **lug** and select **Results** from the menu that appears to open the file **`lug.odb`** in the Visualization module.
2. When the output database is opened in the Visualization module, the undeformed model shape is displayed by default. To change the plot mode, you can use either the **Plot** menu or the toolbox icons displayed on the left side of the viewport (see Figure W1–5). You can identify the function of each tool in the toolbox by positioning your cursor above the icon for that tool; a label for the icon pop-ups describing its function.

1. To plot the deformed shape, click the **Plot Deformed Shape** tool in the toolbox or select **Plot→Deformed Shape** from the main menu bar.

3. Open the **Common Plot Options** dialog box by clicking in the toolbox. Turn on the node and element numbers, and make the nodes visible.
4. Use the display option tools (see Figure W1–5) to switch to hidden line, filled, or wireframe display.

© Dassault Systèmes, 2008

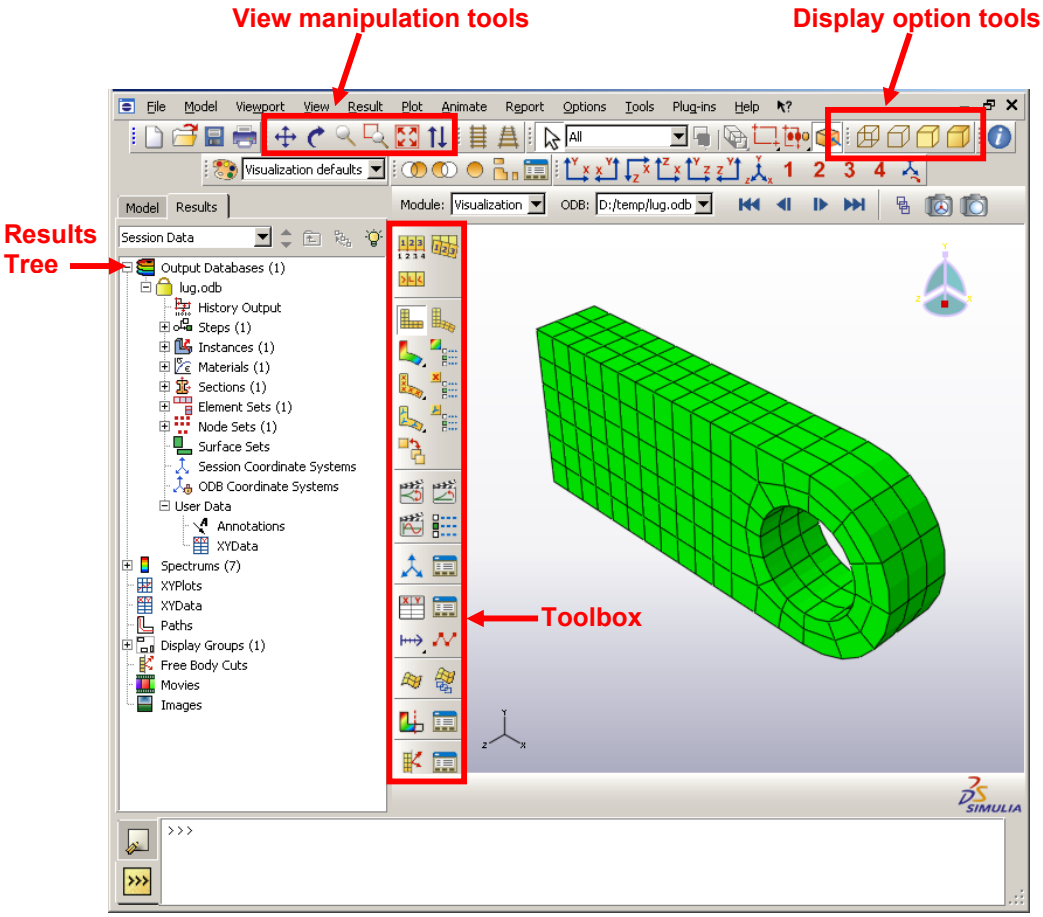


Figure W1–5. Abaqus/Viewer main window

5. Note the displacement magnification factor shown in the bottom of the title. By default, Abaqus/CAE automatically scales the displacement according to the maximum model dimensions for a small-displacement analysis. Displacements are scaled so that the deformed shape will be clear. For a large-displacement analysis the scale factor is 1.0 by default.

   Set the displacement magnification factor to 1.0 so that you can see the actual displacement, and redraw the displaced shape plot.

   Hint: You will have to use the **Common Plot Options** dialog box.

6. Create a contour plot of the Mises stress by clicking the **Plot Contours on Deformed Shape** tool .

7. Frequently users want to remove all annotations that are written on the plots, especially when they are creating hard-copy images or animations. From the main menu bar, select **Viewport→Viewport Annotation Options** to suppress the annotations used in the plots.

© Dassault Systèmes, 2008

The annotations are divided into three categories: legend, title block, and state block. Each category can be controlled separately. The title block contains information about which Abaqus version was used and when the analysis was performed. The state block contains the step title, the increment and step time of the data being displayed, and information on the variable and magnification factor used to calculate the shape of the model.

8. Probe the displacement of the nodes around the hole in the lug.

   a. Click the **Query information** tool . In the **Query** dialog box that appears, select **Probe Values** in the **Visualization Module Queries** field.

   b. In the **Probe Values** dialog box that appears, click **Field Output** to change the default field output variable into the displacement component U2.

   c. In the **Field Output** dialog box that appears, select **U** as the output variable and **U2** as the component and click **OK** to save the selection and exit the **Field Output** dialog box.

   d. In the **Probe Values** field in the **Probe Values** dialog box, select **Probe: Nodes** and toggle on **U,U2**.

   e. Select a node in viewport to obtain its displacement along the 2-direction. Click on a node to query its displacement value along the 2 direction.

9. Use a similar procedure to probe the Mises stress in the elements around the hole in the lug.

Modifying the model and understanding changes in the results

1. Switch to the Load module.
2. Reduce the amplitude of the distributed pressure load to 25 MPa.
3. Create a new job named **`lugmod`** and submit the analysis.
4. View the results in the Visualization module.

   Question W1–11: How have the displacement and stress results changed after the load reduction? Do the results reflect the reduction in loading?

Note: A script that creates the complete model described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_solver_lug_answer.py` and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 1

Basic Input and Output

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Learn to use Abaqus utilities and documentation.
- Understand the basic structure of an input file, and be able to make simple modifications to it.
- Learn how to perform a **datacheck** analysis and how to submit an analysis job using the Abaqus driver.
- Gain familiarity with Abaqus/Viewer.
- Explore the structure and contents of the data (`.dat`) and log (`.log`) files.

Abaqus utilities and documentation

Abaqus provides various utilities for obtaining information on usage, system configuration, example problems, and environment settings for the analysis package.

1. At the prompt, enter the command

   abaqus `information=system`

   to obtain information on the system.

   Note that ***abaqus*** is a generic command that may have been renamed on your system. For example, if more than one version is installed on the system, the command might include the version number, as in `abq681`. In the remainder of this workshop as well as all subsequent workshops, use the appropriate command for your system.

   Question W1–1: What is the processor on your machine?

Question W1–2: What is the operating system (OS) level?

2. Open the online documentation with the command

   ***abaqus* doc**

   Open the Abaqus Analysis User's Manual, and search for the string **DSLOAD** to find information on the *DSLOAD option. You can find information related to the data line syntax in the Abaqus Keywords Reference Manual (use the hyperlink for the *DSLOAD option, or open the Keywords Manual directly). The online documentation graphical user interface is shown in Figure W1–1.

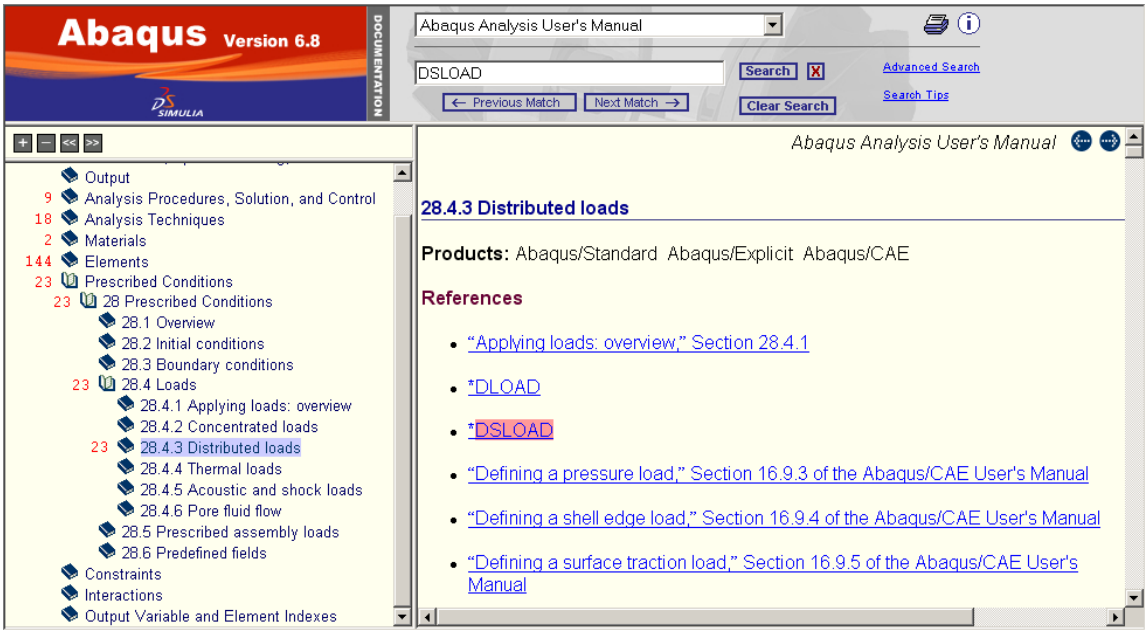


Figure W1–1. Online documentation

3. Open the online Abaqus Example Problems Manual. Search for **`plate buckling`** to find example problems that discuss plate buckling.

   Question W1–3: What are the three example problems that fit the search criteria?

4. Go to Example Problem 1.1.14 in the online Abaqus Example Problems Manual. In the left panel of the window, display the subtopics of the problem and click **Input files**. In the right panel of the window, the list of input files associated with this problem appears. You can select any input filename from the list; a separate window will open containing that file.
5. All example problem input files are included in the Abaqus release and can be obtained using the **abaqus fetch** utility. In your terminal window, enter

   ***abaqus* `fetch job=damagefailcomplate_cps4`**

   at the command line prompt.

© Dassault Systèmes, 2008

6. Use the online documentation to determine the input syntax for some options.

   A keyword line starts with an asterisk (*) followed directly by the keyword option. Parameters and their associated values appear on the keyword line, separated by commas. Many options require data lines, which follow directly after their associated keyword line and contain the data specified in the Abaqus Keywords Reference Manual for each option. Data items are separated by commas. Refer to the discussions of keyword line and data line syntax in Lecture 1, as necessary.

   Question W1–4: In the space provided, write the input you would use to define a node set called **TOP_NODES** that contains previously defined nodes 21, 22, 23, and node set **TOP_LEFT**.

   Hint: Use the information on the *NSET option in the Abaqus Keywords Reference Manual to determine the necessary parameter and data line.

   ***NSET,**

© Dassault Systèmes, 2008

Question W1–5: In the space provided, write the input you would use to define a velocity boundary condition on a node set named **NALL** using the direct format. The velocity is 7.0 m/s in the 2-direction. Will this option appear in the model data or the history data portion of the input file?

Hint: Use the information on the *BOUNDARY option in the Abaqus Keywords Reference Manual, including the reference to the "Boundary Conditions" Section of the Abaqus Analysis User's Manual, to determine the appropriate syntax.

Question W1–6: (Optional) In the space provided, write the input you would use to define the *BEAM SECTION option for beam elements in element set **ELBEAMS** referring to a material named **STEEL**. The beam has a rectangular cross-section with a height of 0.5 m and a width of 0.2 m.

Hint: This option requires one data line for the beam section geometric data. Follow the hyperlink to the beam cross-section library and the rectangular section to determine the appropriate data line input.

```
*BEAM SECTION,
```

© Dassault Systèmes, 2008

Analyzing a connecting lug

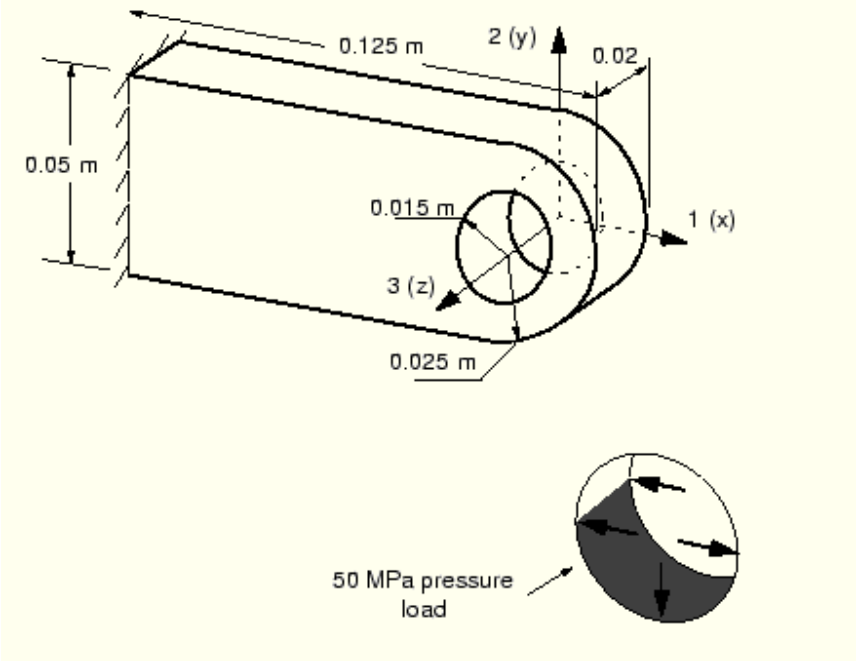


Figure W1–2. Sketch of the connecting lug

In this workshop you will model the connecting lug shown in Figure W1–2. The lug is welded to a massive structure at one end, so we assume that this end is fixed. The other end contains a hole through which a bolt is placed when the lug is in service. You have to calculate the deflection of the lug when a load of 30kN is applied to the bolt in the –2 direction.

To model this problem, you will use three-dimensional continuum elements and perform a linear analysis with elastic materials. You will model the load transmitted to the lug through the bolt as a uniform pressure load applied to the bottom half of the hole, as shown in Figure W1–2. In this workshop SI units (N, m, and s) will be used.

Creating the input file

1. Change to the **`../abaqus_solvers/keywords/workshop1`** directory.
2. View the contents of **`w_lug.inp`**. The model and history data are incomplete, and no mesh or loading is defined.

   Question W1–7: What is the first option in the model data? What is the last option in the model data?

   Question W1–8: What is the first option in the history data?

   Question W1–9: How many steps are there in this analysis?

© Dassault Systèmes, 2008

3. View the files **w_lug_nodes.inp** and **w_lug_elem.inp**. Boundary conditions and loads will be defined using the node and element sets defined in these files.

   Question W1–10: What type of elements are used to model the lug?

4. Edit the input file to set the INPUT parameter on the *INCLUDE options to read the appropriate node and element data files.
5. Complete the *MATERIAL option block by defining an elastic material with elastic modulus E = 200 GPa and Poisson's ratio ν = 0.3. The complete material block should appear as follows:

```
*MATERIAL
*ELASTIC
200E9, 0.3
```

   Question W1–11: Do you need to define a density to complete the material definition? Material density is necessary for what types of analyses?

The boundary conditions and the loads cannot be defined without knowledge of the node and element sets and surfaces. Figure W1–3 shows the various sets and surfaces.

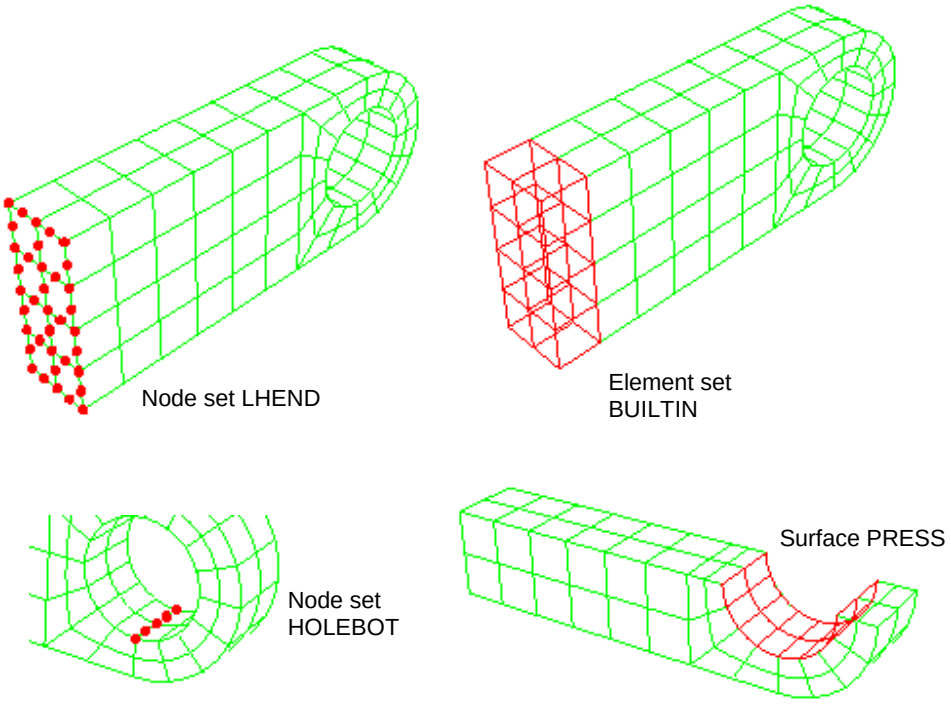


Figure W1–3. Useful sets and surfaces

6. Boundary conditions are applied using the ∗BOUNDARY option. Use the online documentation to obtain a description of the option. The left end of the lug is fixed. Thus, constrain degrees of freedom 1 through 6 of all nodes in node set **LHEND** by entering

```
*BOUNDARY
LHEND, 1, 6
```

Question W1–12: How else could you define a completely constrained boundary condition?

7. Distributed loads are applied to surfaces using the *DSLOAD option. In this problem, the load should be applied to the surface named **PRESS** (which covers the bottom region of the hole). The option to specify the distributed (pressure) load on this surface is

```
*DSLOAD
PRESS, P, 50.E6
```

The magnitude of the applied uniform pressure is 50 MPa. We determined the load magnitude by dividing the total load by the projected horizontal area of the hole, where $\frac{30\text{kN}}{2\times 0.015\text{m}\times 0.02\text{m}} = 50\text{MPa}$.

8. Add a restart output request and printed output requests to the step using the ∗RESTART, ∗EL PRINT, and ∗NODE PRINT options. Abaqus includes a large amount of printed output by default. Requesting printed output of specific variables allows you to limit the volume of output to the data (**.dat**) file. Request printed data output of nodal displacements for node set **HOLEBOT** and reaction forces for node set **LHEND** (including the total force). In addition, request output for stresses in element set **BUILTIN**.

You can do this by entering

```
*RESTART, WRITE
*NODE PRINT, NSET=HOLEBOT
U2
*NODE PRINT, NSET=LHEND, TOTAL=YES, SUMMARY=NO
RF
*EL PRINT, ELSET=BUILTIN
S, MISES
```

Default output requests for the output database are made automatically, and they will be sufficient for this workshop.

© Dassault Systèmes, 2008

Submitting a datacheck analysis

1. Submit the job for a **datacheck** analysis by entering the command

   abaqus **`datacheck job=w_lug interactive`**

   at the prompt. The **`interactive`** parameter will cause all log file output to print to the screen.

2. View the data file (**`w_lug.dat`**) in a text editor.

   Question W1–13: What version of Abaqus are you using?

3. Search for the strings "**WARNING**" and "**ERROR**" to find any warning and error messages. These messages will indicate whether anything unusual was encountered during the **datacheck** analysis (keep in mind that your editor may be case-sensitive for searching).

   Question W1–14: What warning messages did you get? Do they require changes to the input file, or can you ignore them?

4. Search for the string "**P R O B L E M**" to see the summary of the problem size. Include spaces between the letters of the search string.

   Question W1–15: How many elements are there in the model? How many variables are there?

Running a complete analysis

1. Submit **`w_lug.inp`** as an Abaqus job in interactive mode by typing

   abaqus **`job=w_lug interactive`**

   at the prompt.

   If the driver asks if you want to overwrite old job files, type "**`y`**." This means that output files with the same job name that exist from a previous analysis will be overwritten.

2. Now resubmit the job in background mode by typing

   abaqus **`job=w_lug`**

   at the prompt.

   The log file output will be saved in **`w_lug.log`** instead of printing to the screen. You can open **`w_lug.log`** in a text editor and view its contents.

3. You can also let the Abaqus driver prompt you for the necessary job information by typing

   abaqus

   at the prompt.

   Specify **`w_lug`** at the prompt for the job identifier, enter **[RETURN]** at the prompt for user subroutines (since there are none for this job), and type "**`y`**" to overwrite

the files from the last run with the same name. Doing so will submit the analysis job in background mode.

4. List all files with **`w_lug`** as the root of the file name (using a “long” format on Unix systems):

   `dir w_lug.*` (NT)

   `ls -l w_lug.*` (Unix)

   Note the files that were created by Abaqus. We will take a closer look at the printed output file (**`w_lug.dat`**) later in this workshop.

Results visualization in Abaqus/Viewer

1. To run Abaqus/Viewer and load the output database for the lug analysis, type

   `abaqus` **`viewer odb=w_lug`**

   at the prompt.

   Note: The file name extension (**`.odb`**) is not needed.

   If an output database is not specified on the command line, you can select **File→Open** from the main menu bar in Abaqus/Viewer to access the **Open Database** dialog box, as shown in Figure W1–4. Select the file **`w_lug.odb`** from the output database list.

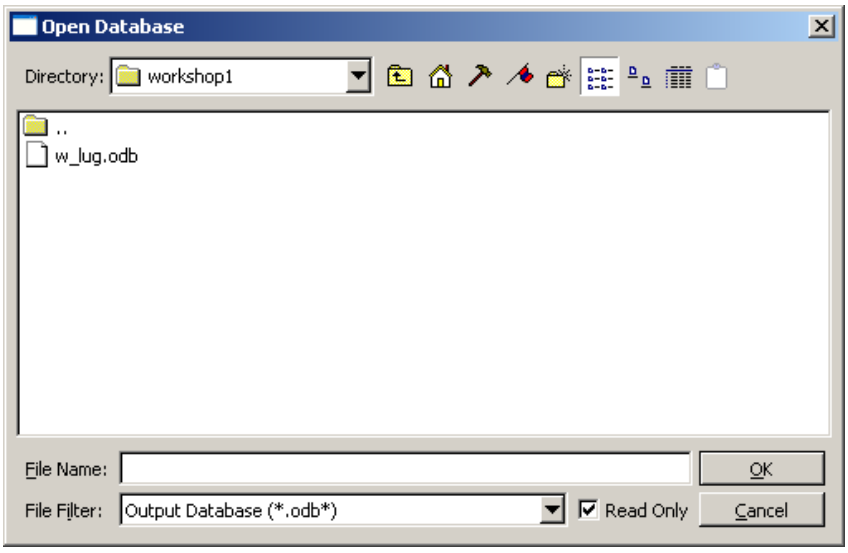


Figure W1–4. **Open Database** dialog box

2. When Abaqus/Viewer opens the output database, the undeformed model shape will be displayed. To change the plot mode, you can use either the **Plot** menu or the toolbox icons displayed on the left side of the viewport (see Figure W1–5). You can identify the function of each tool in the toolbox by positioning your cursor above the icon for that tool. A label for the icon will pop up describing its function.

© Dassault Systèmes, 2008

3. To plot the deformed shape, click the **Plot Deformed Shape** tool in the toolbox or select **Plot→Deformed Shape** from the main menu bar.

4. Open the **Common Plot Options** dialog box by clicking in the toolbox. Turn on the node and element numbers, and make the nodes visible.

5. Use the display option tools to switch to hidden line, filled, or wireframe display.

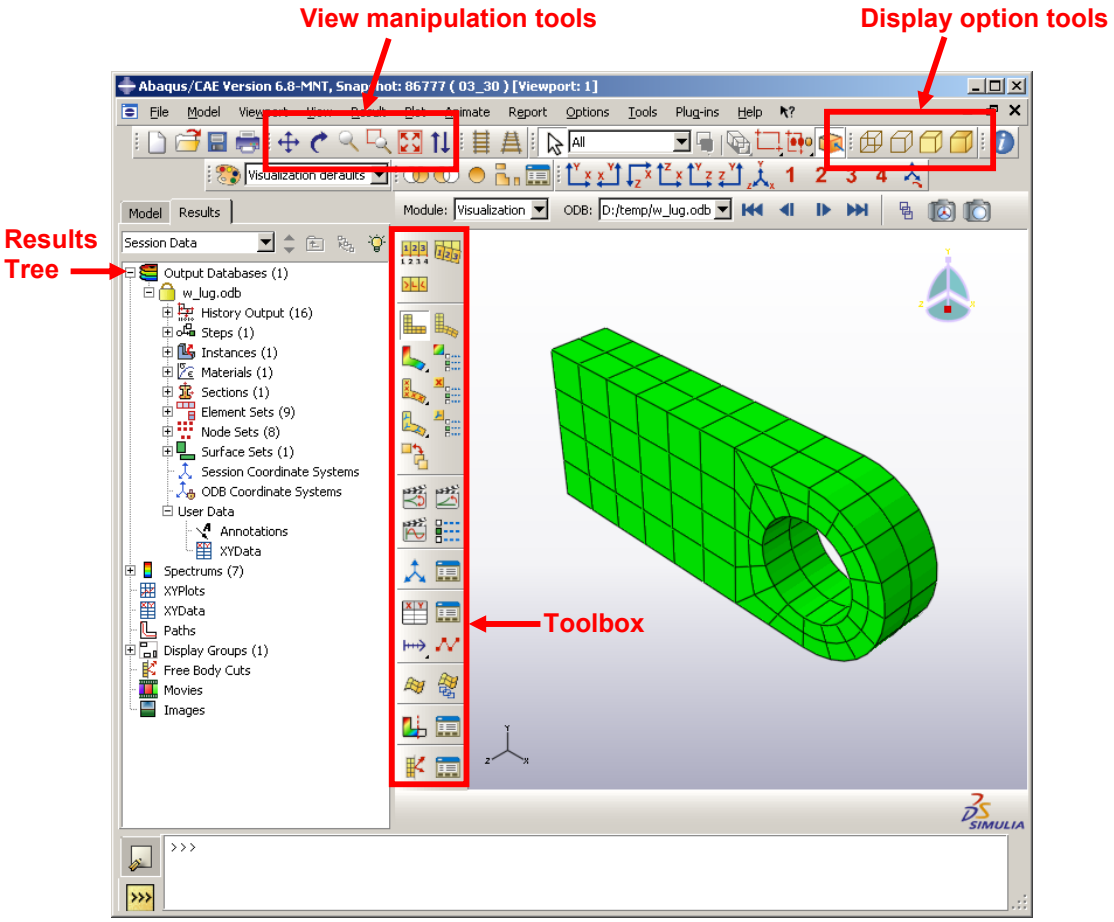


Figure W1–5. Abaqus/Viewer main window

6. Note the displacement magnification factor shown in the bottom of the title.

   By default, Abaqus/Viewer automatically scales the displacement according to the maximum model dimensions for a small-displacement analysis. Displacements are scaled so that the deformed shape will be clear. For a large-displacement analysis the scale factor is 1.0 by default. Set the displacement magnification factor to 1.0 so that you can see the actual displacement, and redraw the displaced shape plot.

   Hint: You will have to use the **Common Plot Options** dialog box.

© Dassault Systèmes, 2008

7. Create a contour plot of the Mises stress by clicking the **Plot Contours on Deformed Shape** tool in the toolbox.
8. Frequently users want to remove all annotations that are written on the plots, especially when they are creating hard-copy images or animations. From the main menu bar, select **Viewport→Viewport Annotation Options** to suppress the annotations used in the plots.

   The annotations are divided into three categories: legend, title block, and state block. Each category can be controlled separately. The title block contains information about which Abaqus version was used and when the analysis was performed. The state block contains the step title (which is the text provided on the data line of the *STEP option), the increment and step time of the data being displayed, and information on the variable and magnification factor used to calculate the shape of the model.
9. From the main menu bar, select **File→Exit** to exit from Abaqus/Viewer.

Viewing the printed output file

Open the printed output file **`w_lug.dat`** in the text editor of your choice.

1. Look at the input echo near the top of the file. Below this you will find the section titled "**OPTIONS BEING PROCESSED**." This is the first place any warning or error messages will appear.
2. A summary of model data follows. Here you can check that Abaqus has correctly interpreted your model definition.

   Question W1–16: Which elements are in element set **`HOLEIN`**?
3. Next you will find the summary of history data for each step. Search for the strings "**B O U N D A R Y**" and "**D I S T R I B U T E D**" to verify that the boundary conditions and distributed loads have been interpreted correctly. Include spaces between the letters of the search string. To start a search through the entire file, go to the top of the file (some editors will wrap to the top of the file upon reaching the end).
4. The next section in the data file is the results section. The tables are printed according to the various output requests.

   Search for the strings "**N O D E**" and "**E L E M E N T**" to find the tables that contain the output requested. The maximum deflection and peak stress are reported at the ends of the respective tables.

   Question W1–17: What are the maximum direct stresses in the 1- and 2-directions (i.e., σ_{11} and σ_{22})?

   (**Hint**: The maximum direct stresses will occur in element set **`BUILTIN`**.)

   Question W1–18: What is the deflection of node 20001 in node set **`HOLEBOT`** in the 2-direction?

© Dassault Systèmes, 2008

5. Search for the string "**TOTAL**" to find the sum of the reaction forces in the 2-direction.

   Question W1–19: What is the net reaction force in the 2-direction at the nodes in node set **LHEND**? Is this equal to the applied load?

   Question W1–20: Why is the sum of the reaction forces at the nodes in node set **LHEND** in the horizontal direction (1-direction) zero?

Modifying the model and understanding changes in the results

1. Open the input file **w_lug.inp** in the text editor.
2. Reduce the distributed pressure load to 25 MPa.
3. Save the modified file to a new file named **w_lugmod.inp**.
4. Submit the new input file as an Abaqus job.
5. Look at the output database file in Abaqus/Viewer.

   Question W1–21: What is the deflection of node 20001 in node set **HOLEBOT**? Do the results reflect the reduction in loading?

Note: A complete input file is available for your convenience. You may consult this file if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input file is named

w_lug_complete.inp

and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 2

Linear Static Analysis of a Cantilever Beam: Multiple Load Cases

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Introduction

In this workshop you will become familiar with using load cases in a linear static analysis. You will model a cantilever beam using the same geometry created in Workshop 1. The left end of the beam is encastred while a series of loads are applied to the free end. Six load cases are considered: unit forces in the global *X*-, *Y*-, and *Z*-directions and well as unit moments about the global *X*-, *Y*-, and *Z*-directions. The model is shown in Figure W2–1. You will solve the problem using a single perturbation step with six load cases and (optionally) using six perturbation steps with a single load case in each step.

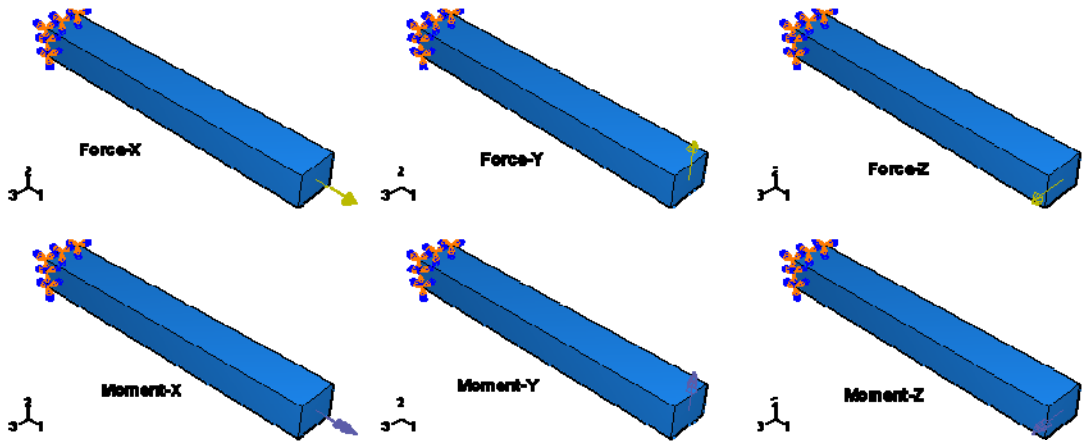


Figure W2–1. Cantilever beam model

Preliminaries

1. Enter the working directory for this workshop
   `../abaqus_solvers/interactive/workshop2`
2. Run the script **`ws_solver_load_cases.py`** using the following command:
   `abaqus cae startup=ws_solver_load_cases.py`.

The above command creates an Abaqus/CAE database named **`Beam.cae`** in the current directory. The geometry, mesh and boundary condition definitions for the beam are included in the model named **`LoadCases`**. You will add the step, load, and load case definitions to complete the model.

Defining a linear perturbation static step

1. In the Model Tree, double-click the **Steps** container.
2. In the **Create Step** dialog box, name the step **`BeamLoadCases`**, choose the **Linear perturbation** procedure type, and select **Static, Linear perturbation** from the list of procedures, and click **Continue**.

   The step editor appears.
3. In the **Basic** tabbed page of the step editor, type **`Six load cases applied to right end of beam`** in the **Description** field.
4. Click **OK** to create the step and to exit the step editor.

Defining loads and load cases

As indicated in Figure W2–1, we wish to apply forces and moments to the right end of the beam. However, the beam is modeled with solid C3D8I elements which possess only displacement degrees of freedom. Thus, only forces may be directly applied to the model. Rather than applying force couples to the model, we will apply concentrated moments to the end of the beam. To this end, all loads will be transmitted to the beam through a rigid body constraint. This approach is adopted to take advantage of the fact that the rigid body reference node possesses six degrees of freedom in three-dimensions: 3 translations and 3 rotations and thus allows direct application of concentrated moments. Rigid bodies and constraints will be discussed further in Lecture 5.

Note that a rigid body constraint named **`Constraint-1`** has been created to constrain the free end of the beam with a predefined reference point named **`RP-1`**; therefore, the forces and moments which you will specify on **`RP-1`** will be transmitted to the beam through this rigid body constraint (see Figure W2–2).

© Dassault Systèmes, 2008

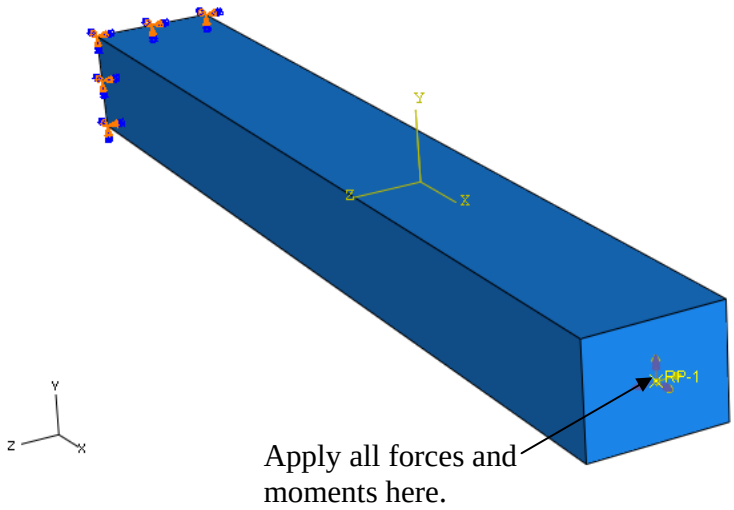


Figure W2–2. Rigid body reference point

To define loads:

1. In the Model Tree, double-click the **Loads** container.
2. In the **Create Load** dialog box, name the load **`Force-X`**, select the step **BeamLoadCases**, choose the category **Mechanical** and the type **Concentrated force**, and click **Continue**.
5. Select the reference point **RP-1** as the point to which the load will be applied.
6. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selection.
7. In the **Edit Load** dialog box, enter a value of **`1.0`** for **CF1**.
8. Click **OK** to complete the load definition.
9. Using a similar procedure, create two additional **Concentrated force** loads named **`Force-Y`** and **`Force-Z`** and three **Moment** loads named **`Moment-X`**, **`Moment-Y`**, and **`Moment-Z`**, with the definitions as listed in Table W2–1.

   Tip: To define the additional forces, simply copy **Force-X** into a new name and edit its definition; to define the moments, first create **Moment-X** and then copy/edit it to define the additional loads.

   Abaqus/CAE displays arrows at the reference point indicating the loads applied to the model.

© Dassault Systèmes, 2008

Table W2–1. Load definitions

Load name	Definition
Force-X	**CF1** = **1.0**
Force-Y	**CF2** = **1.0**
Force-Z	**CF3** = **1.0**
Moment-X	**CM1** = **1.0**
Moment-Y	**CM2** = **1.0**
Moment-Z	**CM3** = **1.0**

To define load cases:

1. In the Model Tree, expand the branch of the step **BeamLoadCases** underneath the **Steps** container and double-click **Load Cases** to create a load case in the step.
2. In the **Create Load Case** dialog box, name the load case **LC-Force-X**, accept **BeamLoadCases** as the step, and click **Continue**.

   The load case editor appears.
3. Click **Add** at the bottom of the **Edit Load Case** dialog box.
4. In the **Load Selection** dialog box that appears, select **Force-X** and click **OK** to confirm the selection and to return to the load case editor.
5. Click **OK** to exit the **Edit Load Case** dialog box.
6. Create five additional load cases: one for each of the remaining loads. Name the load cases **LC-Force-Y**, **LC-Force-Z**, **LC-Moment-X**, **LC-Moment-Y**, and **LC-Moment-Z** and add the corresponding load to each.

   Tip: Copy/edit **LC-Force-X** to define the additional load cases.

   Note that the fixed-end boundary conditions were defined in the initial step, and as such, are active in each load case of the analysis step.

Creating and submitting the analysis job

To create and submit the analysis job:

1. Create a job named **Loadcases** for this linear static perturbation analysis.

   Tip: To create a job, double-click **Jobs** in the Model Tree.
2. Save your model database file and submit the job for analysis. In the Model Tree, click mouse button 3 on the job name and select **Submit** from the menu that appears. From the same menu, you can select **Monitor** to monitor the job's progress.

Viewing the analysis results

When the job is complete, click mouse button 3 on the job **Loadcases** in the Model Tree and select **Results** from the menu that appears.

© Dassault Systèmes, 2008

Abaqus/CAE switches to the Visualization module and opens the output database **`Loadcases.odb`**. Examine the results of the analysis. Note that load case output is stored in separate frames in the output database. Use the **Frame Selector** (click in the context bar) to choose which load case is displayed (alternatively, open the **Step/Frame** dialog box by selecting **Result→Step/Frame**). Figure W2–3 shows contour plots of the Mises stress for each of the load cases.

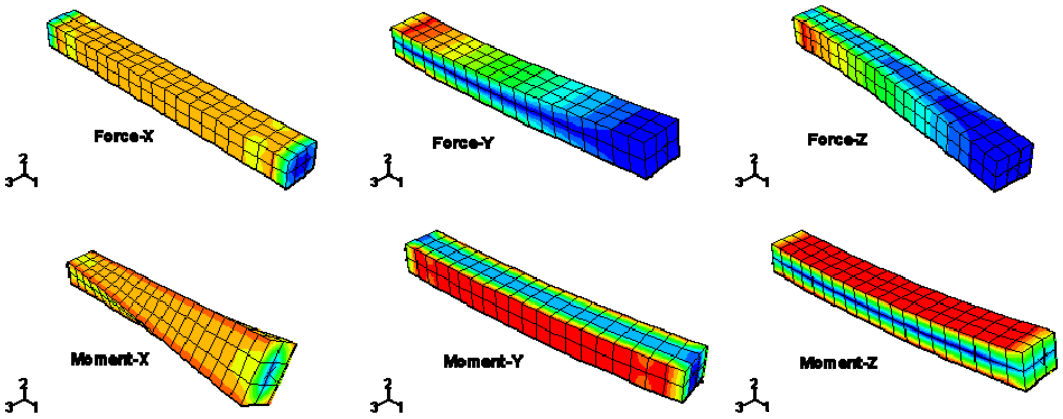


Figure W2–3. Mises stress contours

© Dassault Systèmes, 2008

Combing results from the load cases and envelope plots

You will now linearly combine the results of each load case to plot the stress and deformation in the beam under a given load combination. Recall that each load case is based on a unit load; the results of each load case will be scaled relative those obtained for **LC-Force-Y** when combining the data.

1. From the main menu bar, select **Tools→Create Field Output→From Frames**.
2. In the dialog box that appears, accept **Sum values over all frames** as the operation.
3. In the **Frames** tabbed page, click **Add**. In the **Add Frames** dialog box that appears, choose **Step-1** as the step from which to obtain the data. Click **Select All** and then click **OK** to close the dialog box.
4. Remove the initial frame; for the remaining frames, enter the scale factors shown in Figure W2–4.

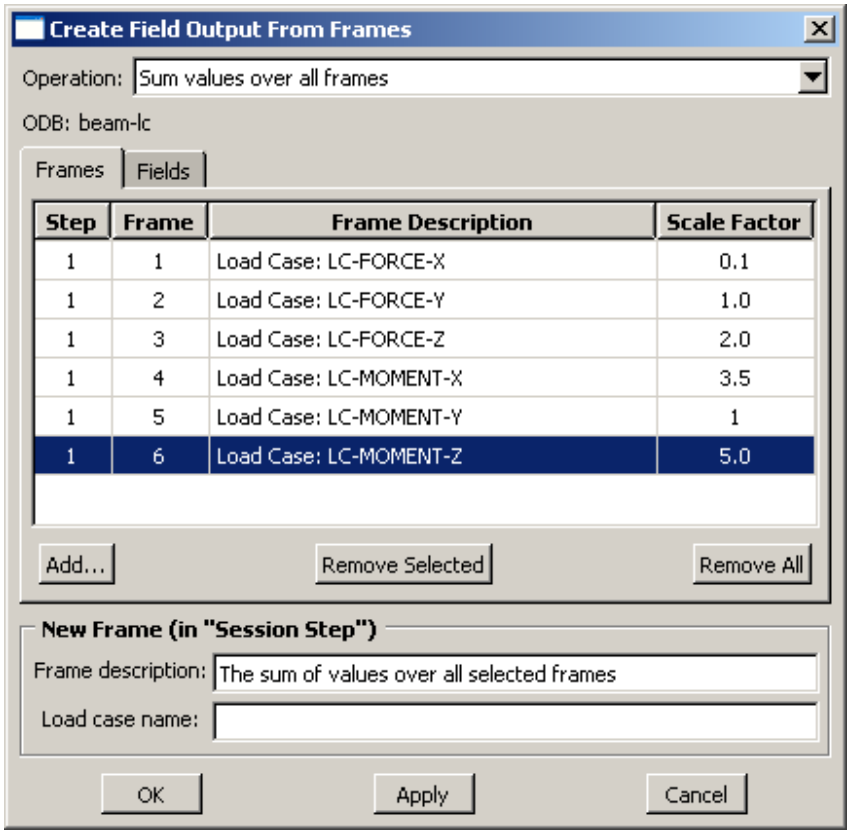


Figure W2–4 Scale factors for linear combination of load cases.

5. Switch to the **Fields** tabbed page to examine the data that will be combined. Accept the default selection (all available field data) and click **OK** to close the dialog box.
6. From the main menu bar, select **Result→Step/Frames**.

© Dassault Systèmes, 2008

7. In the **Step/Frame** dialog box, select **Session Step** as the active step for output and click **OK**.
8. Plot the Mises stress as shown in Figure W2–5. Note that this figure has been customized to overlay the undeformed model shape on the contour plot and a deformation scale factor of **5e4** has been used.

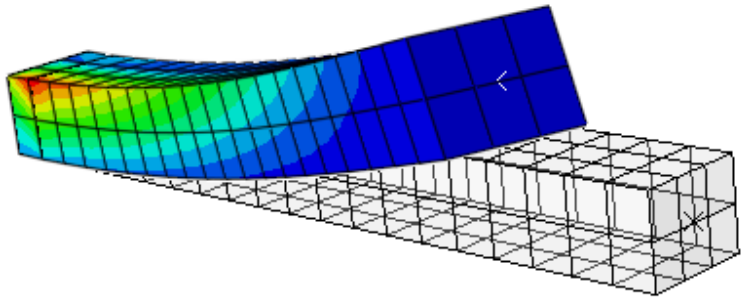


Figure W2–5 Mises stress due to combined loading.

9. Now create an envelope plot of the maximum stress in the beam:
    a. From the main menu bar, select **Tools→Create Field Output→From Frames**.
    b. In the dialog box that appears, select **Find the maximum value over all frames** as the operation.
    c. In the **Frames** tabbed page, click **Add**. In the **Add Frames** dialog box that appears, choose **Step-1** as the step from which to obtain the data. Select all but the initial frame then click **OK** to close the dialog box.
    d. Switch to the **Fields** tabbed page. Unselect all output and then select only **S**.
    e. Click **OK** to close the dialog box.
    f. From the main menu bar, select **Result→Step/Frames**.
    g. In the **Step/Frame** dialog box, select **Session Step** as the active step for output and **The maxmum value over all selected frames** as the frame, as shown in Figure W2–6.

© Dassault Systèmes, 2008

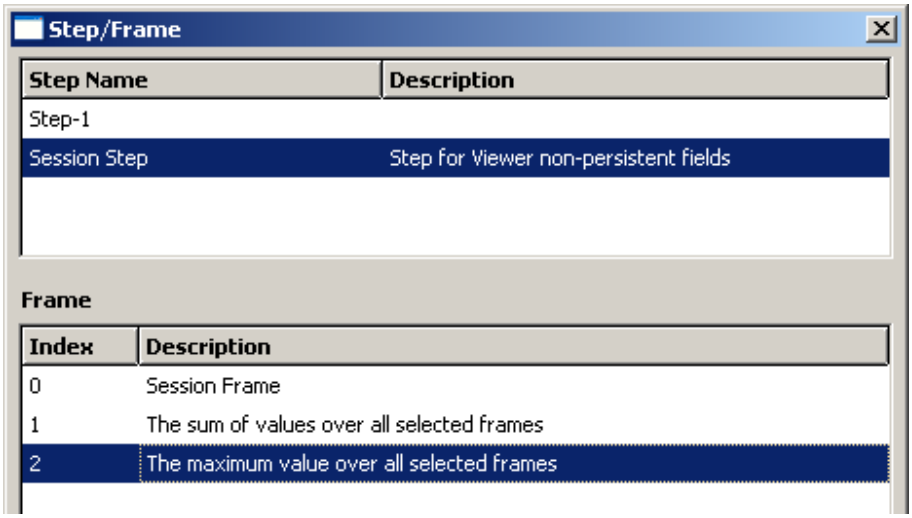


Figure W2–6 Frame selection for envelope plot.

h. Plot the Mises stress as shown in Figure W2–7. Note that this figure has been customized to overlay the undeformed model shape on the contour plot and a deformation scale factor of **5e4** has been used.

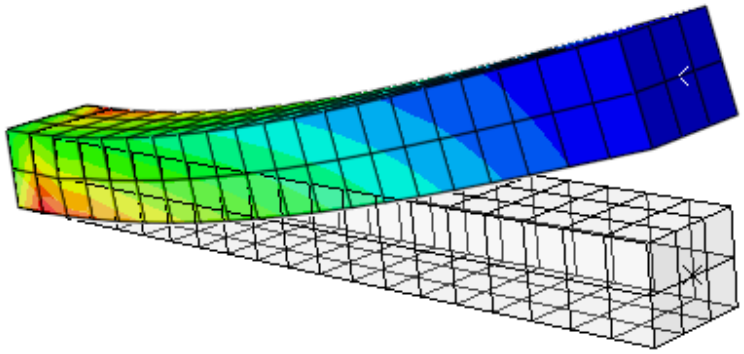


Figure W2–7 Envelope plot of maximum Mises stress.

© Dassault Systèmes, 2008

Using Multiple Perturbation Steps (Optional)

Now perform the same analysis using multiple perturbation steps rather than multiple load cases.

1. In the Model Tree, click mouse button 3 on the model **LoadCases** and select **Copy Model** from the menu that appears. Name the new model **`MultiSteps`**.
2. For the model **MultiSteps**, delete the step **BeamLoadCases** but create six new linear perturbation static steps named **`Step-FX`**, **`Step-FY`**, **`Step-FZ`**, **`Step-MX`**, **`Step-MY`**, and **`Step-MZ`**.

   Note that all of the loads and load cases will be deleted when you delete the step **BeamLoadCases**.
3. In the Model Tree, double-click the **Loads** container for the model **MultiSteps** and define a concentrated force load called **`Force-X`** in the step **Step-FX** with **CF1**=**`1.0`** at the reference point.
4. Similarly, create loads named **`Force-Y`**, **`Force-Z`**, **`Moment-X`**, **`Moment-Y`**, and **`Moment-Z`** in steps **Step-FY**, **Step-FZ**, **Step-MX**, **Step-MY**, and **Step-MZ**, respectively. Here **CF2**=**`1.0`**, **CF3**=**`1.0`**, **CM1**=**`1.0`**, **CM2**=**`1.0`**, and **CM3**=**`1.0`** at the reference point in the respective loads.

   Note that the fixed-end boundary conditions were defined in the initial step, and therefore, are active in each analysis step.
5. Create a new job named **`MultiSteps`** for the model **MultiSteps** and make sure to select the new model for the source. Submit the new job for analysis and monitor the job's status.
6. When the job is complete, open the output database **`MultiSteps.odb`** in the Visualization module and compare the results obtained using both modeling approaches. You will find that the results are identical.

Comparing solution times

Next, open the message (**`.msg`**) file for each job in a text editor. Scroll to the bottom of the file and compare the solution times. You will notice that the multiple step analysis required approximately twice as much CPU time as the multiple load case analysis. For a small model such as this one, the overall analysis time is small so speeding up the analysis by a factor of two may not appear significant. However, it is clear that for large jobs, the speedup offered by multiple load cases will play a significant role in reducing the time required to obtain a solution for a given problem.

© Dassault Systèmes, 2008

Multiple load case analysis:

```
ANALYSIS SUMMARY:
TOTAL OF          1  INCREMENTS
                  0  CUTBACKS IN AUTOMATIC INCREMENTATION
                  1  ITERATIONS
                  1  PASSES THROUGH THE EQUATION SOLVER OF WHICH
              :
              :

THE SPARSE SOLVER HAS BEEN USED FOR THIS ANALYSIS.


JOB TIME SUMMARY
  USER TIME (SEC)      =  0.50000
  SYSTEM TIME (SEC)    =  0.30000
  TOTAL CPU TIME (SEC) =  0.80000
  WALLCLOCK TIME (SEC) =        2
```

Multiple perturbation step analysis:

```
ANALYSIS SUMMARY:
TOTAL OF          6  INCREMENTS
                  0  CUTBACKS IN AUTOMATIC INCREMENTATION
                  6  ITERATIONS
                  6  PASSES THROUGH THE EQUATION SOLVER OF WHICH
              :
              :

THE SPARSE SOLVER HAS BEEN USED FOR THIS ANALYSIS.


JOB TIME SUMMARY
  USER TIME (SEC)      =   1.2000
  SYSTEM TIME (SEC)    =  0.50000
  TOTAL CPU TIME (SEC) =   1.7000
  WALLCLOCK TIME (SEC) =        2
```

Note: A script that creates the complete models described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_solver_load_cases_answer.py` and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 2

Linear Static Analysis of a Cantilever Beam: Multiple Load Cases

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Introduction

In this workshop you will become familiar with using load cases in a linear static analysis. You will model a cantilever beam using the same geometry created in Workshop 1. The left end of the beam is encastred while a series of loads are applied to the free end. Six load cases are considered: unit forces in the global *X*-, *Y*-, and *Z*-directions and well as unit moments about the global *X*-, *Y*-, and *Z*-directions. The model is shown in Figure W2–1. You will solve the problem using a single perturbation step with six load cases and (optionally) using six perturbation steps with a single load case in each step.

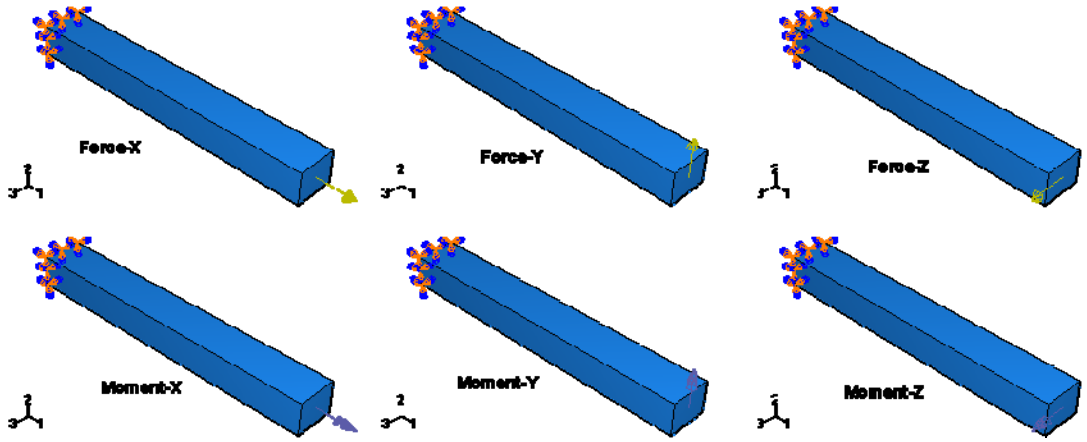


Figure W2–1. Cantilever beam model

© Dassault Systèmes, 2008

As indicated in Figure W2–1, we wish to apply forces and moments to the right end of the beam. However, the beam is modeled with solid C3D8I elements, which possess only displacement degrees of freedom. Thus, only forces may be directly applied to the nodes of the model. Rather than applying force couples to the model, we will apply concentrated moments to the end of the beam. To this end, all loads will be transmitted to the beam through a rigid body constraint. This approach is adopted to take advantage of the fact that the rigid body reference node possesses six degrees of freedom in three-dimensions: 3 translations and 3 rotations and thus allows direct application of concentrated moments. Rigid bodies and constraints will be discussed further in Lecture 5.

Defining loads and load cases in the input file

1. Change to the **`../abaqus_solvers/keywords/workshop2`** directory.
2. Open the file **`w_beam_loadcase.inp`** in a text editor. The file includes all the model data required for this problem: node, element, and set definitions; material and section properties; and fixed boundary conditions. The history data (i.e., step definition) is incomplete.
3. Complete the step definition by defining the loads and load cases. The loads will be applied in the form of concentrated forces and moments via the **`*CLOAD`** option to the rigid body reference node. This node is contained in node set **`refPt`**. For example, for the force acting along the axial direction of the beam (i.e., the *X*-direction), the following load case may be defined:

```
*Load Case, name=Force-X
*Cload
 refPt, 1, 1.0
*End Load Case
```

© Dassault Systèmes, 2008

4. The complete step definition should resemble the following:

```
*Step, name=BeamLoadCases, perturbation
*Static
**
*Load Case, name=Force-X
*Cload
 refPt, 1, 1.0
*End Load Case
**
*Load Case, name=Force-Y
*Cload
 refPt, 2, 1.0
*End Load Case
**
*Load Case, name=Force-Z
*Cload
 refPt, 3, 1.0
*End Load Case
**
*Load Case, name=Moment-X
*Cload
 refPt, 4, 1.0
*End Load Case
**
*Load Case, name=Moment-Y
*Cload
 refPt, 5, 1.0
*End Load Case
**
*Load Case, name=Moment-Z
*Cload
 refPt, 6, 1.0
*End Load Case
**
*End Step
```

   Note that the fixed-end boundary conditions have been defined as part of the model data, and as such, are active in **each** load case.
5. Save the input file.
6. Submit the job for analysis by entering the following command at your system prompt:

   abaqus **job=w_beam_loadcase**
7. Monitor the status of the job by looking at the log (**.log**) or status (**.sta**) files.

© Dassault Systèmes, 2008

Viewing the analysis results

1. When the job has completed successfully, start a session of Abaqus/Viewer by entering the following command at your system prompt:

   ***abaqus* viewer odb=w_beam_loadcase**

   Abaqus/Viewer opens the output database file **w_beam_loadcase.odb** created by the job and displays the undeformed model shape. Examine the results of the analysis. Note that load case output is stored in separate frames in the output database file. Use the **Frame Selector** (click in the context bar) to choose which load case is displayed (alternatively, open the **Step/Frame** dialog box by selecting **Result→Step/Frame**). Figure W2–2, for example, shows contour plots of the Mises stress for each of the load cases.

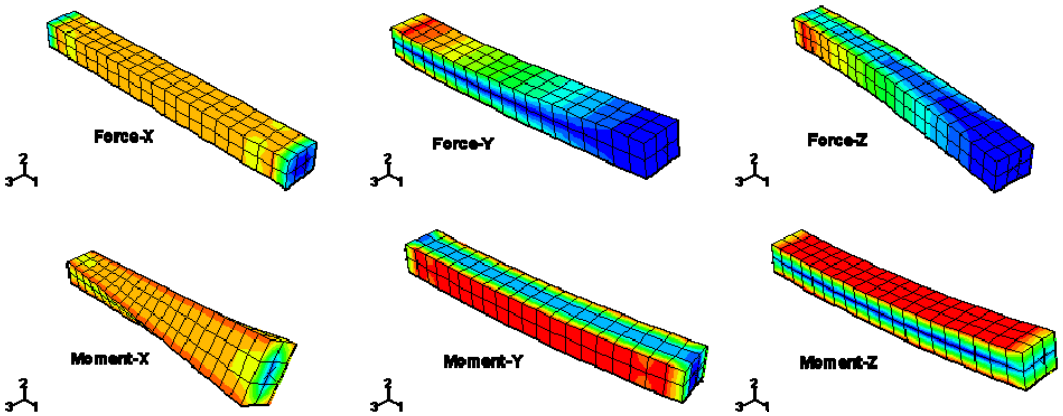


Figure W2–2. Mises stress contours

© Dassault Systèmes, 2008

Combing results from the load cases and envelope plots

You will now linearly combine the results of each load case to plot the stress and deformation in the beam under a given load combination. Recall that each load case is based on a unit load; the results of each load case will be scaled relative those obtained for **LC-Force-Y** when combining the data.

1. From the main menu bar, select **Tools→Create Field Output→From Frames**.
2. In the dialog box that appears, accept **Sum values over all frames** as the operation.
3. In the **Frames** tabbed page, click **Add**. In the **Add Frames** dialog box that appears, choose **Step-1** as the step from which to obtain the data. Click **Select All** and then click **OK** to close the dialog box.
4. Remove the initial frame; for the remaining frames, enter the scale factors shown in Figure W2–3.

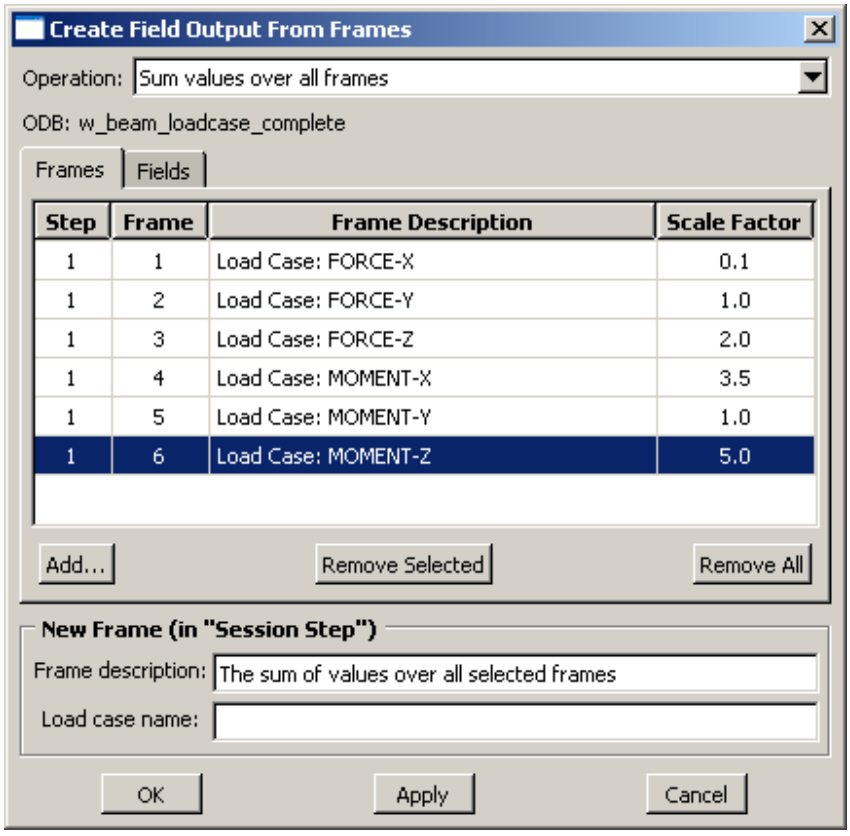


Figure W2–3 Scale factors for linear combination of load cases.

5. Switch to the **Fields** tabbed page to examine the data that will be combined. Accept the default selection (all available field data) and click **OK** to close the dialog box.
6. From the main menu bar, select **Result→Step/Frames**.

© Dassault Systèmes, 2008

7. In the **Step/Frame** dialog box, select **Session Step** as the active step for output and click **OK**.
8. Plot the Mises stress as shown in Figure W2–4. Note that this figure has been customized to overlay the undeformed model shape on the contour plot and a deformation scale factor of **5e4** has been used.

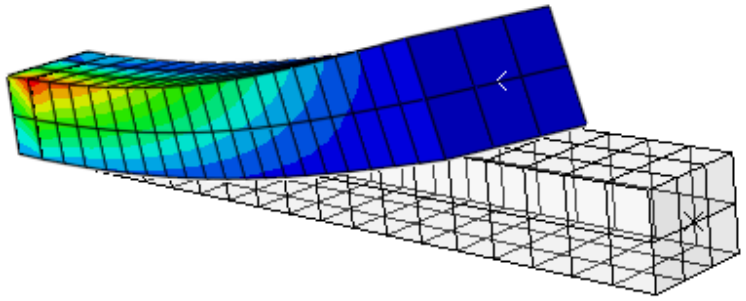


Figure W2–4 Mises stress due to combined loading.

9. Now create an envelope plot of the maximum stress in the beam:
    a. From the main menu bar, select **Tools→Create Field Output→From Frames**.
    b. In the dialog box that appears, select **Find the maximum value over all frames** as the operation.
    c. In the **Frames** tabbed page, click **Add**. In the **Add Frames** dialog box that appears, choose **Step-1** as the step from which to obtain the data. Select all but the initial frame then click **OK** to close the dialog box.
    d. Switch to the **Fields** tabbed page. Unselect all output and then select only **S**.
    e. Click **OK** to close the dialog box.
    f. From the main menu bar, select **Result→Step/Frames**.
    g. In the **Step/Frame** dialog box, select **Session Step** as the active step for output and **The maxmum value over all selected frames** as the frame, as shown in Figure W2–5.

© Dassault Systèmes, 2008

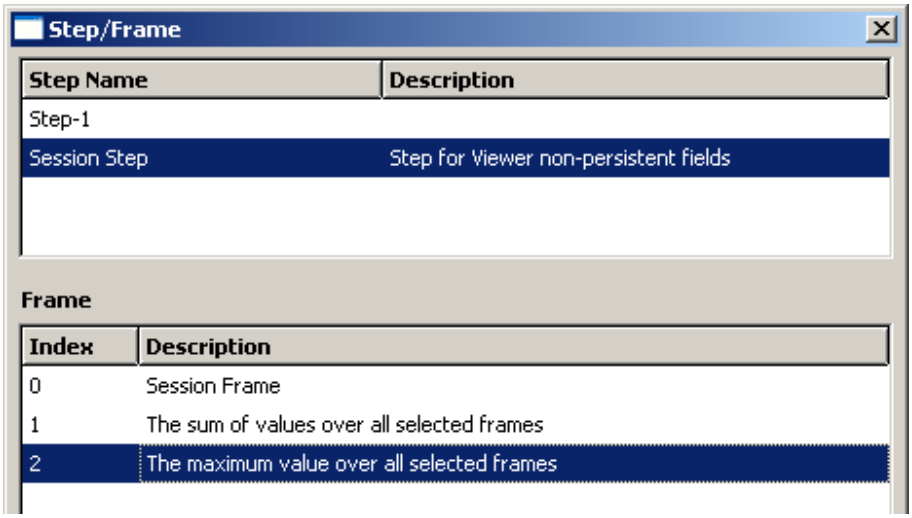


Figure W2–5 Frame selection for envelope plot.

h. Plot the Mises stress as shown in Figure W2–6. Note that this figure has been customized to overlay the undeformed model shape on the contour plot and a deformation scale factor of **`5e4`** has been used.

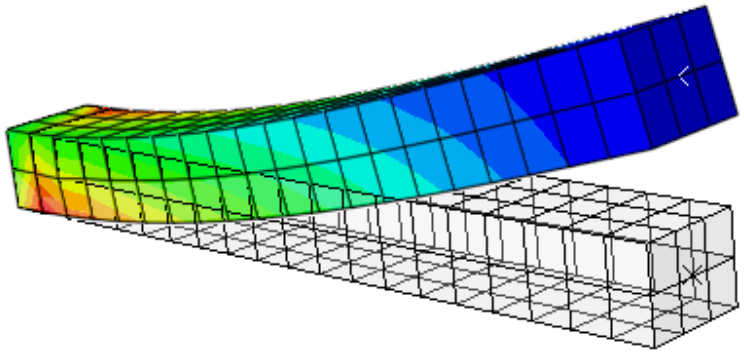


Figure W2–6 Envelope plot of maximum Mises stress.

Using Multiple Perturbation Steps (Optional)

Now perform the same analysis using multiple perturbation steps rather than multiple load cases.

1. Open the file **`w_beam_multstep.inp`** in a text editor. As before, the file includes all the model data required for this problem: node, element, and set definitions; material and section properties; and fixed boundary conditions. The history data (i.e., step definition) is incomplete.
2. Complete the history data by defining the steps. As before, the loads will be applied in the form of concentrated forces and moments via the **`*CLOAD`** option to the rigid body reference node. This node is contained in node set **`refPt`**. For example, for the force acting along the axial direction of the beam (i.e., the *X*-direction), the following step may be defined:

© Dassault Systèmes, 2008

```
*Step, name=Force-X, perturbation
*Static
*Cload
 refPt, 1, 1.
*End Step
```

3. The complete set of step definitions should resemble the following:

```
*Step, name=Force-X, perturbation
*Static
*Cload
 refPt, 1, 1.0
*End Step
**
*Step, name=Force-Y, perturbation
*Static
*Cload
 refPt, 2, 1.0
*End Step
**
*Step, name=Force-Z, perturbation
*Static
*Cload
 refPt, 3, 1.0
*End Step
**
*Step, name=Moment-X, perturbation
*Static
*Cload
 refPt, 4, 1.0
*End Step
**
*Step, name=Moment-Y, perturbation
*Static
*Cload
 refPt, 5, 1.0
*End Step
**
*Step, name=Moment-Z, perturbation
*Static
*Cload
 refPt, 6, 1.0
*End Step
```

   Note that the fixed-end boundary conditions have been defined as part of the model data, and as such, are active in **each** step.
4. Save the input file.

© Dassault Systèmes, 2008

5. Submit the job for analysis by entering the following command at your system prompt:

   abaqus **job=w_beam_multstep**
6. Monitor the status of the job by looking at the log (**.log**) or status (**.sta**) files.
7. When the job has completed successfully, open the output database **w_beam_multstep.odb** created by the job in Abaqus/Viewer and compare the results obtained using both modeling approaches. You will find that the results are identical.

Comparing solution times

Next, open the message (**.msg**) file for each job in a text editor. Scroll to the bottom of the file and compare the solution times. You will notice that the multiple step analysis required approximately twice as much CPU time as the multiple load case analysis. For a small model such as this one, the overall analysis time is small so speeding up the analysis by a factor of two may not appear significant. However, it is clear that for large jobs, the speedup offered by multiple load cases will play a significant role in reducing the time required to obtain a solution for a given problem.

© Dassault Systèmes, 2008

Multiple load case analysis:

```
ANALYSIS SUMMARY:
TOTAL OF          1  INCREMENTS
                  0  CUTBACKS IN AUTOMATIC INCREMENTATION
                  1  ITERATIONS
                  1  PASSES THROUGH THE EQUATION SOLVER OF WHICH
              :
              :

THE SPARSE SOLVER HAS BEEN USED FOR THIS ANALYSIS.


JOB TIME SUMMARY
  USER TIME (SEC)      =  0.20000
  SYSTEM TIME (SEC)    =  0.60000
  TOTAL CPU TIME (SEC) =  0.80000
  WALLCLOCK TIME (SEC) =          1
```

Multiple perturbation step analysis:

```
ANALYSIS SUMMARY:
TOTAL OF          6  INCREMENTS
                  0  CUTBACKS IN AUTOMATIC INCREMENTATION
                  6  ITERATIONS
                  6  PASSES THROUGH THE EQUATION SOLVER OF WHICH
              :
              :

THE SPARSE SOLVER HAS BEEN USED FOR THIS ANALYSIS.


JOB TIME SUMMARY
  USER TIME (SEC)      =   0.6000
  SYSTEM TIME (SEC)    =  1.50000
  TOTAL CPU TIME (SEC) =   2.1000
  WALLCLOCK TIME (SEC) =          2
```

Note: Complete input files are available for your convenience. You may consult these files if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input files are named

`w_beam_loadcase_complete.inp`

`w_beam_multstep_complete.inp`

and are available using the Abaqus fetch utility.

Workshop 3

Nonlinear Statics

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define alternate nodal and material directions.
- Include nonlinear geometric effects by adding the NLGEOM parameter.
- Include nonlinear material effects by defining plastic material behavior.
- Become familiar with the output for an incremental analysis.

Introduction

In this workshop you will model the plate shown in Figure W3–1. It is skewed at 30° to the global *X*-axis, built-in at one end, and constrained to move on rails parallel to the plate axis at the other end. You will determine the midspan deflection when the plate carries a uniform pressure. You will modify the model to include alternate nodal and material directions as well as nonlinear effects.

© Dassault Systèmes, 2008

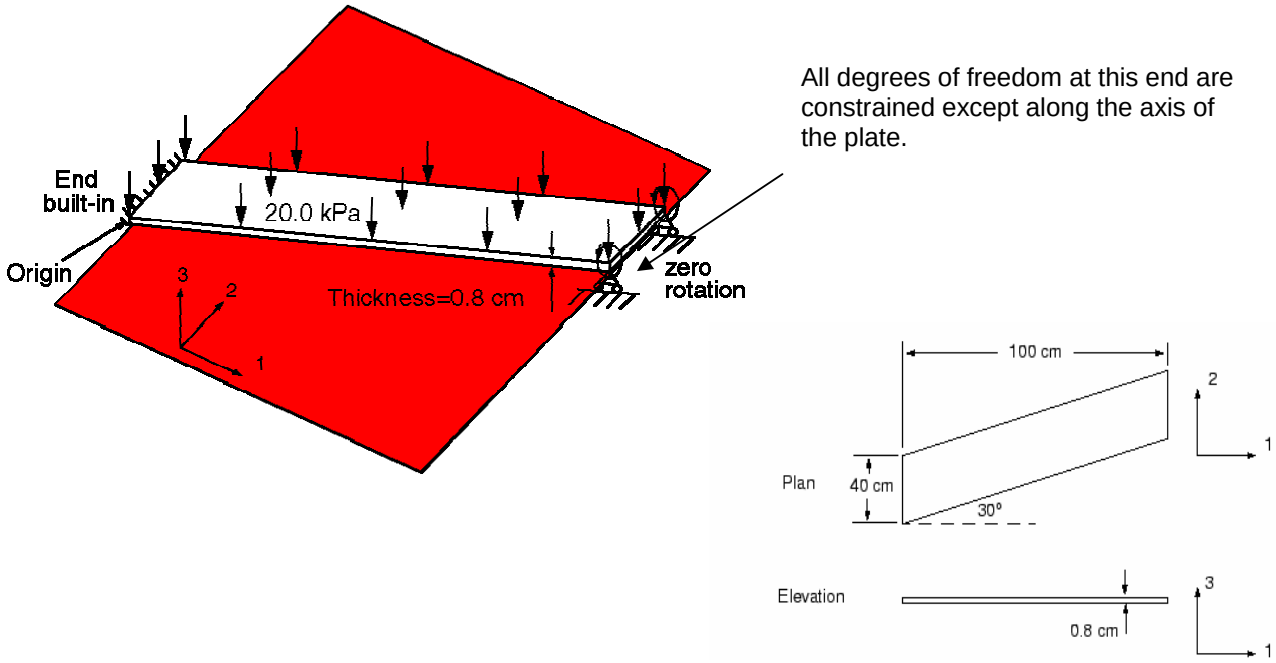


Figure W3–1. Sketch of skewed plate

Preliminaries

1. Enter the working directory for this workshop
   `../abaqus_solvers/interactive/workshop3`
2. Run the script **`ws_solver_skew_plate.py`** using the following command:
   `abaqus` **`cae startup=ws_solver_skew_plate.py`**.

The above command creates an Abaqus/CAE database named **`SkewPlate.cae`** in the current directory. A model named **`linear`** includes the geometry, mesh and material definitions for the plate. You will add the necessary data to complete the linear analysis model and then perform the simulation considering geometrically nonlinear effects.

Defining the local material directions

The orientation of the structure in the global coordinate system is shown in Figure W3–1. The global Cartesian coordinate system defines the default material directions, but the plate is skewed relative to this system. It will not be easy to interpret the results of the simulation if you use the default material directions because the direct stress in the material 1-direction (i.e., global X-direction), σ_{11}, will contain contributions from both the axial stress, produced by the bending of the plate, and the stress transverse to the axis of the plate. It will be easier to interpret the results if the material directions are aligned with the axis of the plate and the transverse direction. Therefore, a local

© Dassault Systèmes, 2008

rectangular coordinate system is needed in which the local *x*-direction lies along the axis of the plate (i.e., at 30° to the global *X*-axis) and the local *y*-direction is also in the plane of the plate.

You will define the datum coordinate system (CSYS) and then assign the material orientation.

1. Switch to the Property module and define a rectangular datum coordinate system as shown in Figure W3–2 using the **Create Datum CSYS: 2 Lines** tool .
   a. Note the small black triangles at the base of the toolbox icons. These triangles indicate the presence of hidden icons that can be revealed. Click the **Create Datum CSYS: 3 Points** tool but do not release the mouse button. When additional icons appear, release the mouse button.
   b. Select the **Create Datum CSYS: 2 Lines** tool . It appears in the toolbox with a white background indicating that you selected it.
   c. In the **Create Datum CSYS** dialog box, name the datum CSYS **Skew**, select the **Rectangular** coordinate system type, and click **Continue**. Make the next two selections as indicated in Figure W3–2.

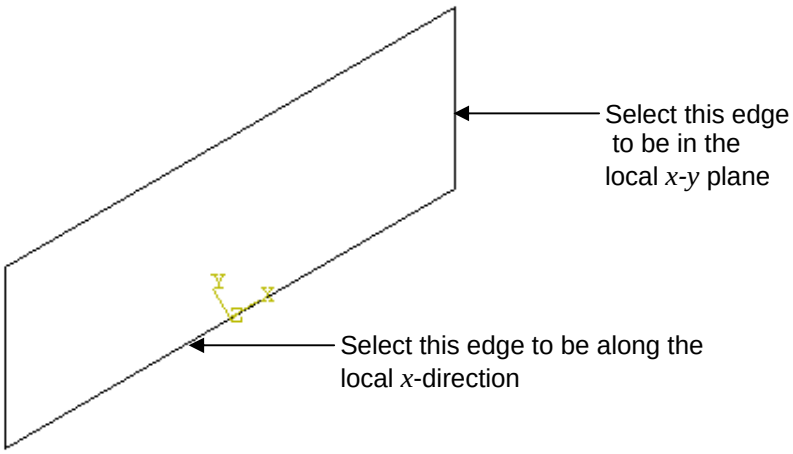


Figure W3–2. Datum coordinate system used to define local directions

2. Use the **Assign Material Orientation** tool to assign the material orientations to the plate.
   a. Click the **Assign Beam Orientation** tool but do not release mouse button 1. When additional icons appear, release the mouse button. Select the **Assign Material Orientation** tool
   b. Select the entire part as the region to be assigned a local material orientation.

© Dassault Systèmes, 2008

c. Click mouse button 2 in the viewport or click **Done** in the prompt area to confirm the selection.

d. Click **Datum CSYS List** in the prompt area, to select the datum CSYS **`Skew`**.

e. In the **Datum CSYS List** dialog box, select **skew** and click **OK**. In the material orientation editor, select **Axis-3** for the direction of the approximate shell normal. No additional rotation is needed about this axis.

f. Click **OK** to confirm the input.

Tip: To verify that the local material directions have been assigned correctly, select **Tools→Query** from the main menu bar and perform a property query on the material orientations.

Once the part has been meshed and elements have been created in the model, all element variables will be defined in this local coordinate system.

Prescribing boundary conditions and applied loads

As shown in Figure W3–1, the left end of the plate is completely fixed; the right end is constrained to move on rails that are parallel to the axis of the plate. Since the latter boundary condition direction does not coincide with the global axes, you must define a local coordinate system that has an axis aligned with the plate. You can use the datum coordinate system that you created earlier to define the local directions.

1. In the Model Tree, double-click the **BCs** container and define a **Displacement/Rotation** mechanical boundary condition named **`Rail boundary condition`** in the **`Apply Pressure`** step.

   In this example you will assign boundary conditions to sets rather than to regions selected directly in the viewport. Thus, when prompted for the regions to which the boundary condition will be applied, click **Sets** in the prompt area of the viewport.

2. From the **Region Selection** dialog box that appears, select the set **Plate-1.EndB**. Toggle on **Highlight selections in viewport** to make sure the correct set is selected. The right edge of the plate should be highlighted. Click **Continue**.

3. In the **Edit Boundary Condition** dialog box, click **Edit** to specify the local coordinate system in which the boundary condition will be applied. In the viewport, select the datum CSYS **`Plate-1.Skew`**. The local *x*-direction is aligned with the plate axis.

   Note that **`Plate-1.Skew`** is the assembly-level datum CSYS generated by the part-level datum CSYS **`Skew`**.

4. In the **Edit Boundary Condition** dialog box, fix all degrees of freedom except for **U1** by toggling them on and entering a value of **`0`** for each.

   The right edge of the plate is now constrained to move only in the direction of the plate axis. Once the plate has been meshed and nodes have been generated in the model, all printed nodal output quantities associated with this region

(displacements, velocities, reaction forces, etc.) will be written in this local coordinate system.

5. Create another boundary condition named **`Fix left end`** to fix all degrees of freedom at the left edge of the plate (set **Plate-1.EndA**). Use the default global directions for this boundary condition.
6. Define a uniform pressure load named **`Pressure`** across the top of the shell in the **Apply Pressure** step. Select both regions of the part using **[Shift]+Click**, and choose the top side of the shell (**Brown**) as the surface to which the pressure load will be applied. You may need to rotate the view to more clearly distinguish the top side of the plate. Specify a load magnitude of **`2.0E4`** Pa.

Running the job and postprocessing the results

1. Create a job named **`SkewPlate`** with the following description: **`Linear Elastic Skew Plate, 20 kPa Load`**.
2. Save your model database file.
3. Submit the job for analysis and monitor the solution progress.

   When the analysis is complete, use the following procedure to postprocess the analysis results.
4. In the Model Tree, click mouse button 3 on the job **SkewPlate** and select **Results** from the menu that appears to open the file **`SkewPlate.odb`** in the Visualization module.
5. Click the **Plot Deformed Shape** tool to plot the deformed shape.
6. Use the the **Query information** tool to probe the value of the midspan deformation.
   a. In the **Query** dialog box, select **Probe Values** in the **Visualization Module Queries** field and click **Apply**.
   b. Change the displayed field variable to the displacement along the 3-direction. In the **Probe Values** dialog box, click **Field Output** to change the default field variable to U3. In the **Field Output** dialog box that appears, select **U** as the output variable and **U3** as the component and click **OK**.
   c. Under the **Probe Values** field in the **Probe Values** dialog box, select the **Nodes** probe and toggle on **U,U3**.
   d. Click on a node (as indicated in Figure W3–3) along the midespan to probe its displacement along the 3-direction. Enter this value in the “Linear” column of Table W3–1.

© Dassault Systèmes, 2008

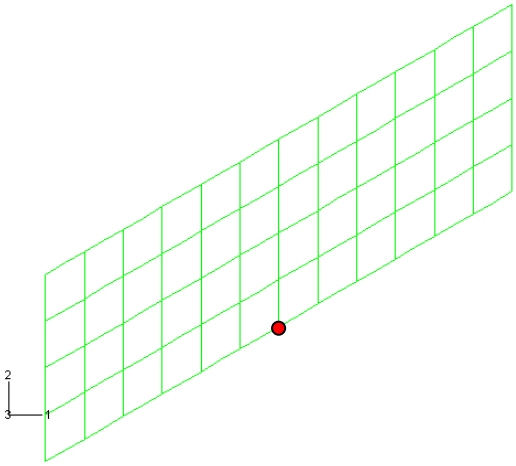


Figure W3–3. Midspan node

Adding geometric nonlinearity

Now perform the simulation considering geometrically nonlinear effects. Copy the model named **linear** to a new model named **`nonlinear`**. You will add geometric nonlinearity into the model **nonlinear**; the changes required for this model are described next.

1. In the Model Tree, double-click **Apply Pressure** underneath the **Steps** container of the model **nonlinear** to edit the step definition. In the **Basic** tabbed page of the **Edit Step** dialog box, toggle on **Nlgeom** to include geometric nonlinearity effects and set the time period for the step to **`1.0`**; in the **Incrementation** tabbed page, set the initial increment size to **`0.1`**. Note that the default maximum number of increments is **`100`**; Abaqus may use fewer increments than this upper limit, but it will stop the analysis if it needs more.

   You may wish to change the description of the step to reflect that it is now a nonlinear analysis step.

2. Create a job named **`NlSkewPlate`** for the model nonlinear and give it the description **`Nonlinear Elastic Skew Plate`**. Save your model database file.

3. Submit the job for analysis and monitor the solution progress.

   The **Job Monitor** is particularly useful in nonlinear analyses. It gives a brief summary of the automatic time incrementation used in the analysis for each increment. The information is written as soon as the increment is completed, so you can monitor the analysis as it is running. This facility is useful in large, complex problems. The information given in the **Job Monitor** is the same as that given in the status file (**`NlSkewPlate.sta`**).

4. When the job is complete, open the output database **`NlSkewPlate.odb`** in the Visualization module and plot the deformed model shape.

5. Query the vetical displacement (U3) of the same midspan node as discribed earlier and enter the displacement result in the “Linear” column of Table W3–1.

© Dassault Systèmes, 2008

Table W3–1. Midspan displacements

Load (kPa)	Linear (m)	Nlgeom (m)
20		
60		

6. Triple the pressure in both the linear and nonlinear analysis models. Create new jobs and run each of these analyses
7. Upon job completion, look at the results and enter the vertical displacement of the same node in Table W3-1.

Question W3–1: How does tripling the load affect the midspan displacement in the linear analyses?

Question W3–2: How do the results of the nonlinear analyses compare to each other and to those from the linear analyses?

Adding Plasticity

You will now include another source of nonlinearity: plasticity. The plastic material option requires true stress and logarithmic plastic strain data. The data shown in Figure W3–4 are plotted using this strain measure. You will edit the model **nonlinear** to add the material plasticity behavior.

1. In the Model Tree, expand the **Materials** container and double-click **Steel**. In the **Edit Material** dialog box, add plasticity by choosing **Mechanical→Plasticity→ Plastic**. Enter the data lines corresponding to points A and B on the stress-strain curve as shown in Figure W3–4.

   The Young's modulus for this material is 30E9 Pa.

   Hint: The total strain ε_{tot} at any point on the curve is equal to the sum of the elastic strain ε_{el} and plastic strain ε_{pl}. The elastic strain at any point on the curve can be evaluated from Young's modulus and the true stress: $\varepsilon_{el} = \sigma / E$. Use the following relationship to determine the plastic strains to include on the plastic option:

   $$\varepsilon_{pl} = \varepsilon_{tot} - \varepsilon_{el} = \varepsilon_{tot} - \sigma / E.$$

Question W3–3: Why is the second entry on the first data line of the plasticity option equal to 0.0?

2. Change the pressure to 10 kPa. In the Model Tree, double-click **Pressure** underneath the **Loads** container. In the **Edit Load** dialog box that appears, enter a value of **`10000`** for **Magnitude**.
3. Create a new job named **`PlSkewPlate`** and give it the description **`Nonlinear Plastic Skew Plate`**. Save your model database file.
4. Submit the job for analysis and monitor the solution progress.

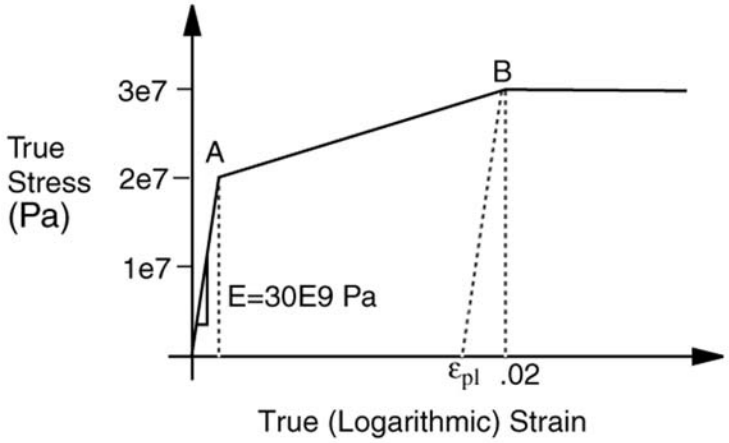


Figure W3–4. Stress versus strain curve

Postprocessing an incremental analysis

When the job is complete, visualize the output database **`PlSkewPlate.odb`** in the Visualization module.

1. By default, the last increment of the last step is selected. Use the **Frame Selector** in the context bar to select other steps or increments; alternatively, use the **Step/Frame** dialog box (**Result→Step/Frame**).
2. Use the view manipulation tools to position the model as you wish. Turn perspective on or off by clicking the **Turn Perspective On** tool or the **Turn Perspective Off** tool in the toolbar.
3. Plot the deformed shape by clicking the **Plot Deformed Shape** tool .

   A sample deformed shape plot is shown in Figure W3–5. Your plot may look different if you have positioned your model differently

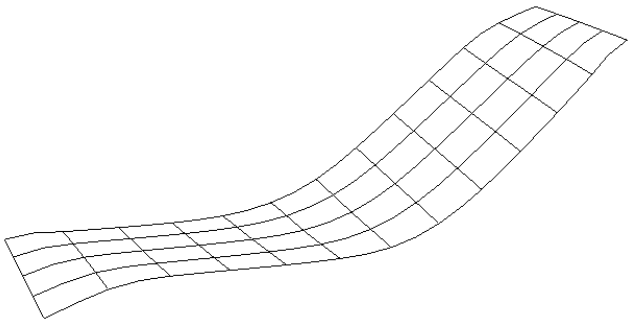


Figure W3–5. Final deformed shape

4. Create a contour plot of variable S11 by following this procedure:

   a. Click the **Plot Contours** tool in the toolbox.

   b. Select **Result→Field Output**.

   c. In the **Field Output** dialog box, select **S11** as the stress component.

   d. Click **Section Points** to select a section point.

   e. In the **Section Points** dialog box that appears, select **Top and bottom** as the active locations and click **OK**.

      Your contour plot should look similar to Figure W3–6. Abaqus plots the contours of the Mises stress on both the top and bottom faces of each shell element. To see this more clearly, rotate the model in the viewport.

© Dassault Systèmes, 2008

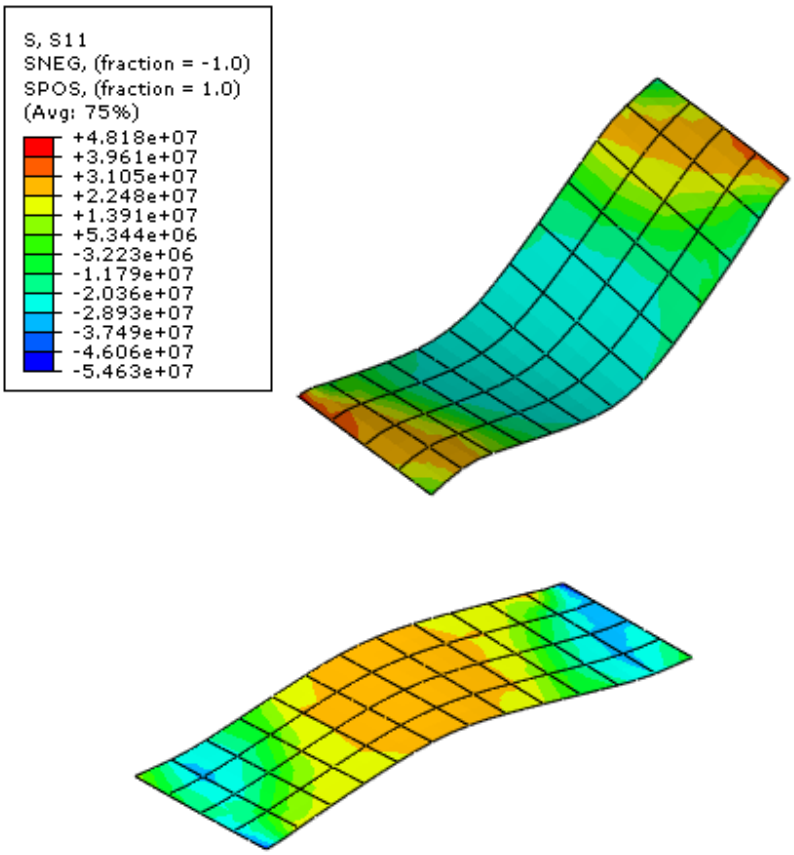


Figure W3–6. Contour plot of S11: SPOS, top image; SNEG, bottom image

Question W3–4: Where do the peak displacements and stresses occur in the model?

5. Click the **Animate: Time History** tool to animate the results.

   You can stop the animation and move between frames and steps by using the arrow buttons in the context bar.

6. Create a displacement history plot of U3 of the midspan node you tracked in the previous analyses:
   a. In the Results Tree, expand the **History Output** container underneath the output database named **`PlSkewPlate.odb`**.
   b. Click **History Output** and press **F2**; filter the container according to **`*U3*`**.
   c. Double-click the data object for the node tracked in the previous analyses.

      Your plot should look similar to Figure W3–7. Note this figure has been customized.

© Dassault Systèmes, 2008

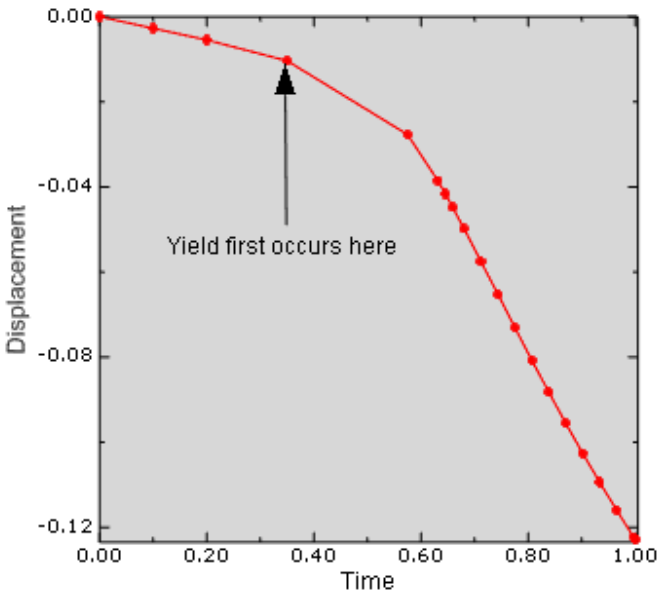


Figure W3–7. History of displacement at the midspan

Note: A script that creates the complete models described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_solver_skew_plate_answer.py` and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 3

Nonlinear Statics

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define alternate nodal and material directions.
- Include nonlinear geometric effects by adding the NLGEOM parameter.
- Include nonlinear material effects by defining plastic material behavior.
- Become familiar with the output for an incremental analysis.

Introduction

In this workshop you will model the plate shown in Figure W3–1. It is skewed at 30° to the global 1-axis, built-in at one end, and constrained to move on rails parallel to the plate axis at the other end. You will determine the midspan deflection when the plate carries a uniform pressure.

You will modify the input file that models this problem to include alternate nodal and material directions as well as nonlinear effects.

Alternate nodal and material directions

1. Change to the **`../abaqus_solvers/keywords/workshop3`** directory, and open the file **`w_skew_plate_linear.inp`** in an editor. You will need to specify local nodal and material directions by following the steps given below.
2. The right end of the plate is constrained to move parallel to an axis that is skewed relative to the global axes. Thus, the nodes at this end of the plate must be transformed into a local coordinate system that is aligned with the plate. The following ∗TRANSFORM option block defines a local coordinate system, x',

© Dassault Systèmes, 2008

y', z', by specifying points ***a*** and ***b,*** as shown in Figure W3–2 (see the Abaqus Analysis User's Manual for a detailed explanation of the data line).

```
*TRANSFORM, NSET=ENDB, TYPE=R
0.1, 0.0577, 0.0,  -0.0577, 0.1, 0.0
```

x, *y*, *z* of point ***a*** *x*, *y*, *z* of point ***b***

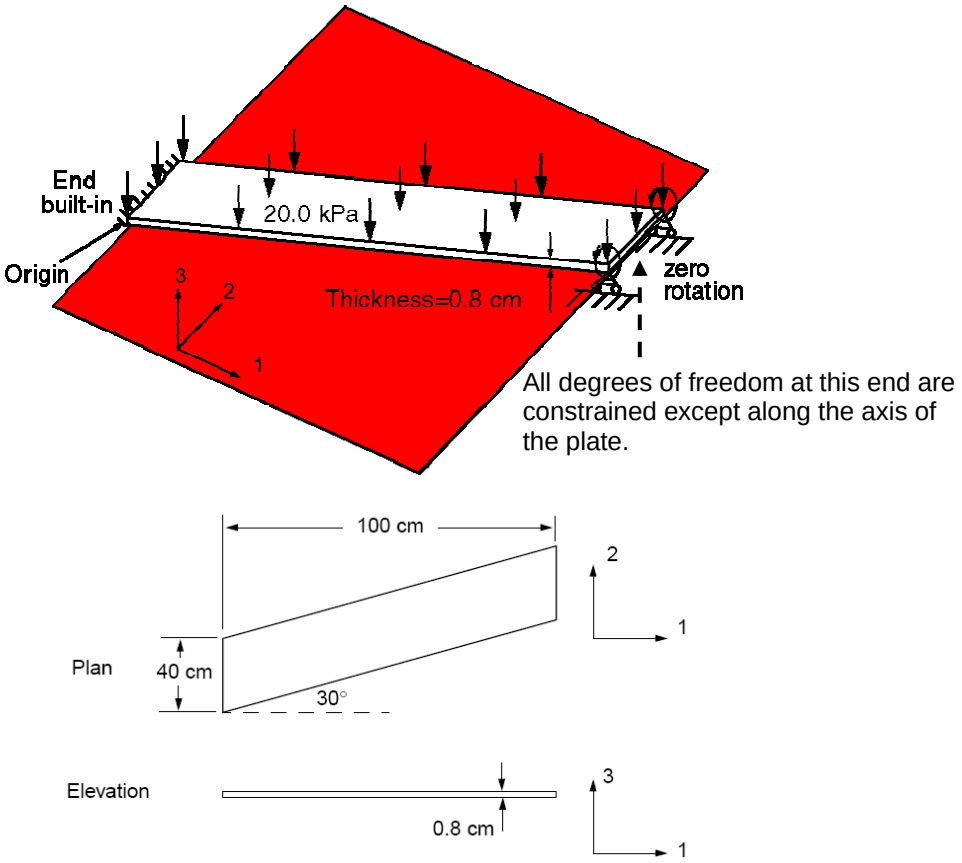


Figure W3–1. Sketch of the skewed plate

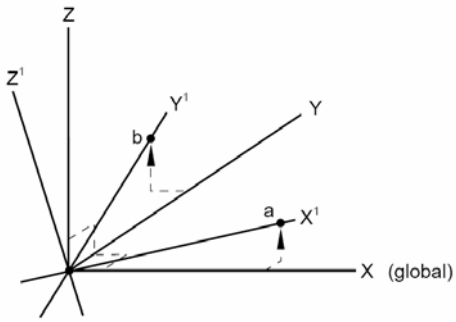


Figure W3–2. Rectangular Cartesian transformation

© Dassault Systèmes, 2008

3. The default material directions in this model are aligned with the global axes. In this default system the direct stress in the material 1-direction, σ_{11}, will contain contributions from both the axial stress (produced by the bending of the plate) and the stress transverse to the axis of the plate. The results will be easier to interpret if the material directions are aligned with the axis of the plate and the transverse direction.

   These local material directions can be defined with the following ∗ORIENTATION option. The first data line defines a local coordinate system by specifying points ***a*** and ***b,*** as shown in Figure W3–2. The second data line defines an additional rotation of 0.0° about the 3-axis (see the online User's Manual for detailed explanations of the data lines).

   ```
   *ORIENTATION, NAME=SKEW, SYSTEM=RECTANGULAR
   0.1, 0.0577, 0.0, -0.0577, 0.1, 0.0
   3, 0.0
   ```

   This option acts independently of the ∗TRANSFORM option.

4. Run the analysis. To submit this job, you must enter

   abaqus **job=w_skew_plate_linear**

   at the prompt.

5. When the analysis is complete, open the data (**.dat**) file and find the value of the vertical displacement (degree of freedom 3) at the midspan (node 357). Enter this value in the "Linear" column of Table W3–1.

Geometric Nonlinearity

1. Copy the input file to a new file called **w_skew_plate_nonlin.inp**, and make the following changes to account for geometric nonlinearity:
2. Set NLGEOM = YES on the ∗STEP option. This parameter indicates that geometric nonlinearity will be accounted for during the step.
3. Set the initial time increment to 0.1 and the total time to 1.0 on the data line following the ∗STATIC option.

   "Time" in a static analysis is just a convenient way to measure the progress of an incremental solution unless rate-dependent behavior is involved. The beginning of the step definition should look something like this:

   ```
   *STEP, NLGEOM=YES
   *STATIC
   0.1, 1.0
   ```

   Run the new analysis, and enter the vertical displacement (degree of freedom 3) of node 357 in the "NLGEOM" column of Table W3–1.

© Dassault Systèmes, 2008

Table W3–1. Midspan displacements

Load (kPa)	Linear (m)	NLEOM (m)
20		
60		

4. Triple the load in both the linear and nonlinear analysis input files, rerun each of these analyses, and enter the vertical displacement of node 357 from each analysis in Table W3–1. The pressure loading is applied normal to the shell surface with the ∗DLOAD option.

Question W3–1: How does tripling the load affect the midspan displacement in the linear analyses?

Question W3–2: How do the results of the nonlinear analyses compare to each other and to those from the linear analyses?

Plasticity

1. You will now include another source of nonlinearity: plasticity. The ∗PLASTIC option requires true stress and logarithmic plastic strain data. The data shown in Figure W3–3 are plotted using this strain measure.

   In the material block of the input file **`w_skew_plate_nonlin.inp`** add the ∗PLASTIC option and enter the data lines corresponding to points A and B on the stress-strain curve shown in Figure W3–3.

   The Young's modulus for this material is 30E9 Pa.

   Hint: The total stain ε_{tot} at any point on the curve is equal to the sum of the elastic strain ε_{el} and plastic strain ε_{pl}. The elastic strain at any point on the curve can be evaluated from Young's modulus and the true stress: $\varepsilon_{el} = \sigma / E$. Use the following relationship to determine the plastic strains:

$$\varepsilon_{pl} = \varepsilon_{tot} - \varepsilon_{el} = \varepsilon_{tot} - \sigma / E .$$

   Add the ∗PLASTIC option underneath *MATERIAL to complete the material block. The complete material option block is given below:

```
*MATERIAL, NAME=MAT1
*ELASTIC
 3.0E10,0.3
*PLASTIC
 2.E7, 0.0
 3.E7, 0.019
```

© Dassault Systèmes, 2008

Question W3–3: Why is the second entry on the first data line of the ∗PLASTIC option equal to 0.0?

2. Change the pressure to 10 kPa.

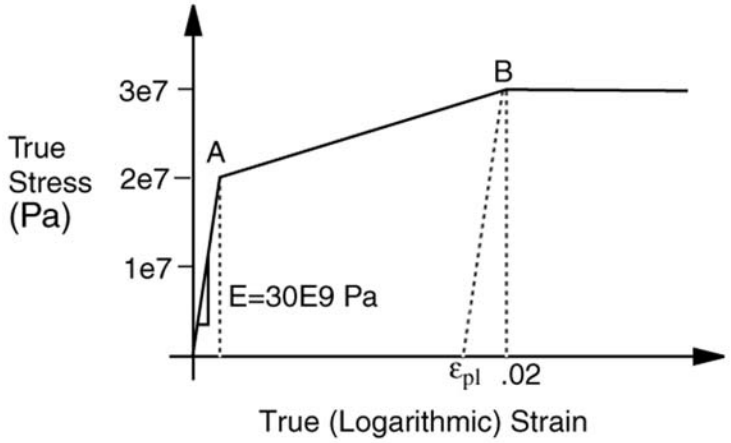


Figure W3–3. Stress versus strain curve

3. Make the following additional changes:
   a. Modify the ∗RESTART option to write restart output every 10th increment. Set the FREQUENCY parameter equal to 10.
   b. You will use Abaqus/Viewer to postprocess the results. To create a more readable printed output (**`.dat`**) file, set the output frequency to this file to every 100 increments. Specifying a frequency larger than or equal to the maximum number of increments ensures that output to the data file is written only at the end of the last increment of the step.
   c. It is useful to be able to check the progress of an analysis by monitoring the value of one degree of freedom. To do so, add the ∗MONITOR option to the history section of the input file. Set the value of the NODE parameter to 357, and set the value of the DOF parameter to 3.
4. Run the analysis. While the job is running, you can check on the progress of the analysis by looking at the status (**`.sta`**) file. The “DOF MONITOR” column should show the value of the midspan displacement.

Postprocessing an Incremental Analysis in Abaqus/Viewer

1. Start Abaqus/Viewer by entering the following command at the prompt:
   `abaqus` **`viewer`**

   Open the appropriate output database by selecting **File→Open** from the main menu bar. Select the file **`w_skew_plate_nonlin.odb`**, and click **OK**.

2. By default, the last increment of the last step is selected. Use the **Frame Selector** in the context bar to select other steps or increments; alternatively, use the **Step/Frame** dialog box (**Result→Step/Frame**).
3. Use the view manipulation tools to position the model as you wish. Turn perspective on or off by clicking the **Turn Perspective On** tool or the **Turn Perspective Off** tool in the toolbar.
4. Plot the deformed shape by clicking the **Plot Deformed Shape** tool .

   A sample deformed shape plot is shown in Figure W3–4. Your plot may look different if you have positioned your model differently

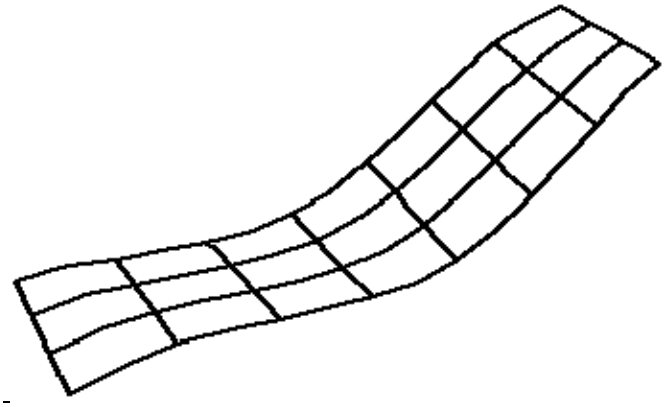


Figure W3–4. Final deformed shape

5. Create a contour plot of variable S11 by following this procedure:

   a. Click the **Plot Contours** tool in the toolbox.
   b. Select **Result→Field Output**.
   c. In the **Field Output** dialog box, select **S11** as the stress component.
   d. Click **Section Points** to select a section point.
   e. In the **Section Points** dialog box that appears, select **Top and bottom** as the active locations and click **OK**.

      Your contour plot should look similar to Figure W3–5. Abaqus plots the contours of the Mises stress on both the top and bottom faces of each shell element. To see this more clearly, rotate the model in the viewport.

© Dassault Systèmes, 2008

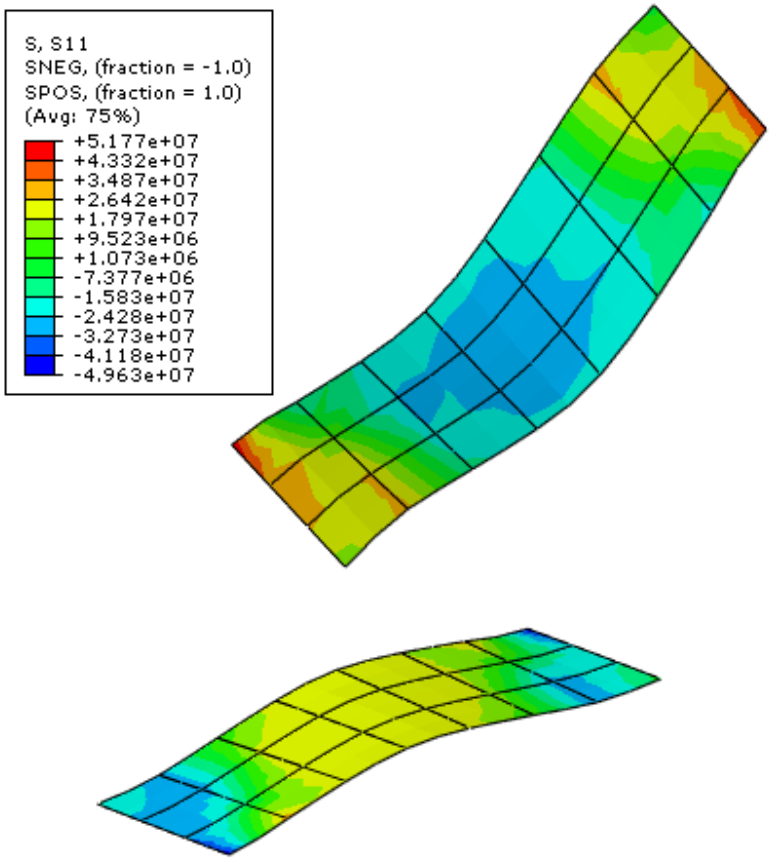


Figure W3–5. Contour plot of S11: SPOS, top image; SNEG, bottom image

Question W3–4: Where do the peak displacements and stresses occur in the model?

6. Click the **Animate: Time History** tool to animate the results.

   You can stop the animation and move between frames and steps by using the arrow buttons in the context bar.

7. Use the following procedure to create a history plot of displacement U3 for node 357:

   a. In the Results Tree, expand the **History Output** container underneath the output database named **`w_skew_plate_nonlin.odb`**.

   b. Click **History Output** and press **F2**; filter the container according to **`*U3*`**.

   c. Double-click the data object for node 357.

      Your plot should look similar to Figure W3–6. Note this figure has been customized.

© Dassault Systèmes, 2008

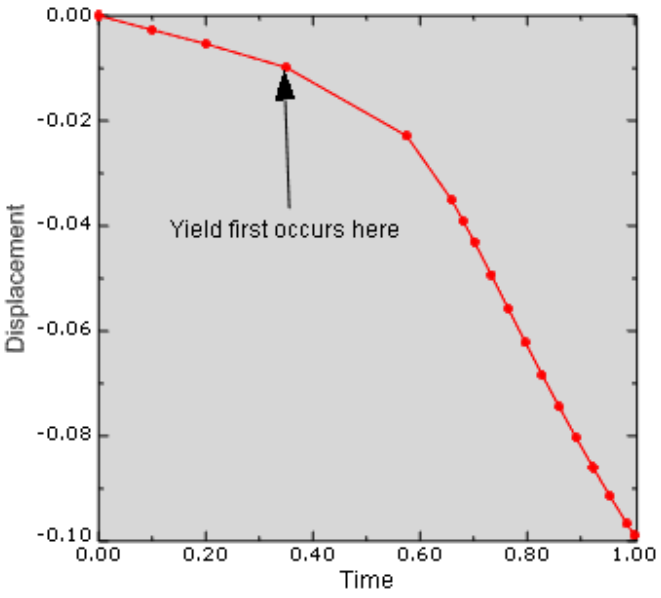


Figure W3–6. History of displacement at the midspan

Note: Complete input files are available for your convenience. You may consult these files if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input files are named

`w_skew_plate_linear_complete.inp`

`w_skew_plate_nonlin_complete.inp`

and are available using the Abaqus fetch utility.

Workshop 4

Contact

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Learn how to define a rigid surface.
- Learn important techniques and options for analyzing contact problems.
- Use the job diagnostics tool to diagnose the contact simulation.

Introduction

In this workshop you will analyze contact between a plate and rigid surfaces. The plate is modeled with CPE4I solid, plane strain elements, which model bending behavior accurately with one element through the thickness of the plate. The rigid surfaces are modeled as analytical rigid surfaces. In the first part of the workshop you will analyze contact between the plate and a single lower surface as the right-hand edge of the plate is displaced downward and the other end is fixed. In the second part the plate is held fixed between two rigid surfaces as the right-hand edge is displaced downward, as shown in Figure W4–1. In the third part contact with a rigid surface replaces the prescribed edge displacement to deflect the right-hand edge downward.

© Dassault Systèmes, 2008

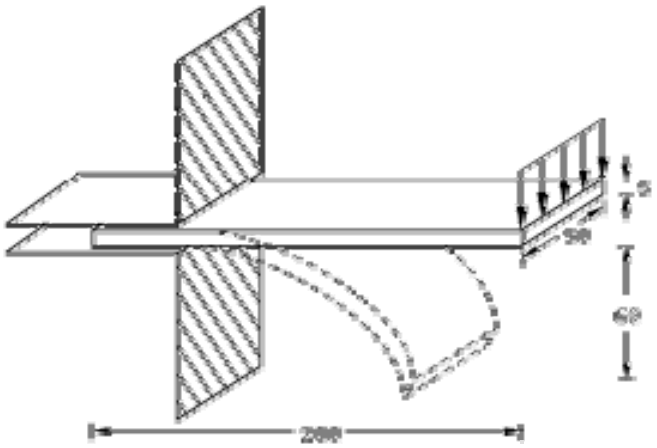


Figure W4–1. Plate constrained by rigid surfaces at one end and displaced downward at the other

Preliminaries

1. Enter the working directory for this workshop
   `../abaqus_solvers/interactive/workshop4`
2. Run the script **`ws_solver_contact.py`** using the following command:
   `abaqus cae startup=ws_solver_contact.py`.

The above command creates an Abaqus/CAE database named **`Contact.cae`** in the current directory. The model named **`contact`** includes a complete model in which a plate is deflected to contact with a single lower rigid surface. In this model, a prescribed displacement of 60.0 units along the negative 2-direction is applied to the right-hand edge of the plate to lead to the deflection, as shown in Figure W4–1. You will add a request to write the contact diagnostic information to the output database, visualize the analysis results, and later modify the model to perform the subsequent analyses.

Contact with a lower rigid surface

1. Review the predefined contact definitions—interactions and interaction properties. These options are used to define contact in Abaqus/CAE.

   Question W4–1: Review the rigid surface definition. Would the rigid surface behave differently if its reference node had been defined 5 units to the left?

2. Create a job named **`contact_10`** and submit it for analysis; monitor the status of the job. The severe discontinuity iterations are caused by changes in contact. Before proceeding, save your model database.

© Dassault Systèmes, 2008

3. When the job is complete, visualize the results:
   a. Enter the Visualization module and open the file **`contact_10.odb`**.
   b. Plot the deformed model shape.

      The plot shows that the plate has lifted off the rigid surface to the left of the corner.
   c. From the main menu bar, select **Tools→Job Diagnostics** to look at the contact diagnostic information.
   d. In the **Job Diagnostics** dialog box that appears, expand the branch in the Job History tree to look at the information for the first iteration of the first attempt of the first increment in the analysis, as shown in Figure W4–2. Note that **SDI** means "severe discontinuity iteration."

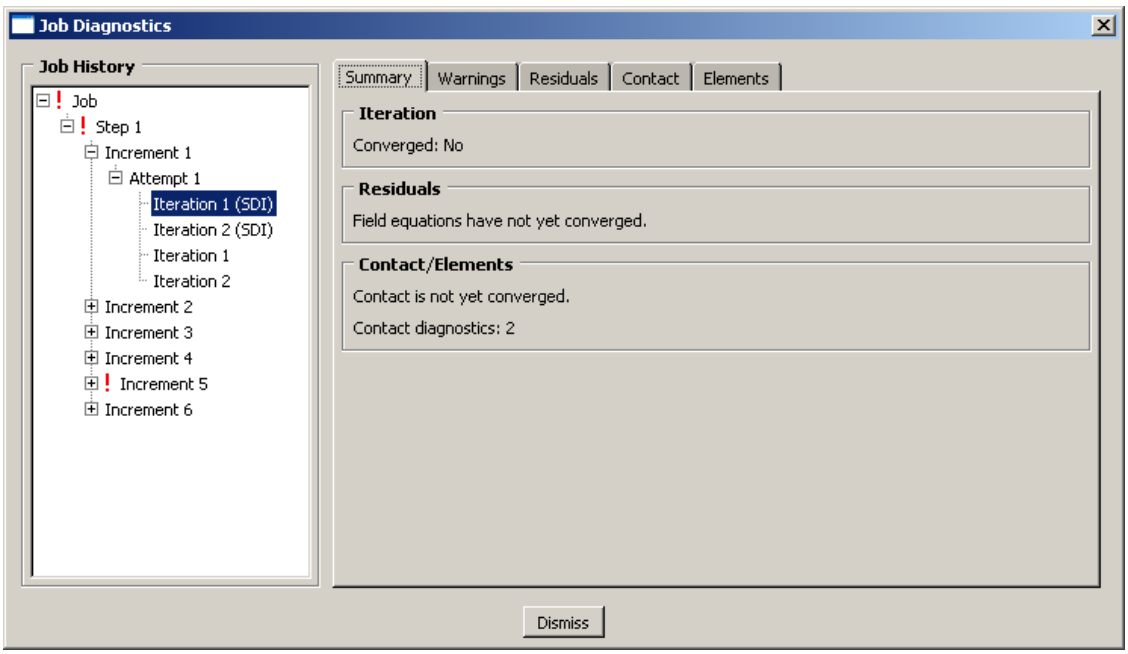


Figure W4–2. Summary information for the fist iteration of the fist attempt of the first increment

   e. In the **Job Diagnostics** dialog box, open the **Contact** tabbed page (see Figure W4–3). In the first sever discontinuity iteration of the first increment, 1 node on the plate experience contact opening, indicating its contact status should be changed. To see where the node is located on the model, toggle on **Highlight selections in viewport**.

© Dassault Systèmes, 2008

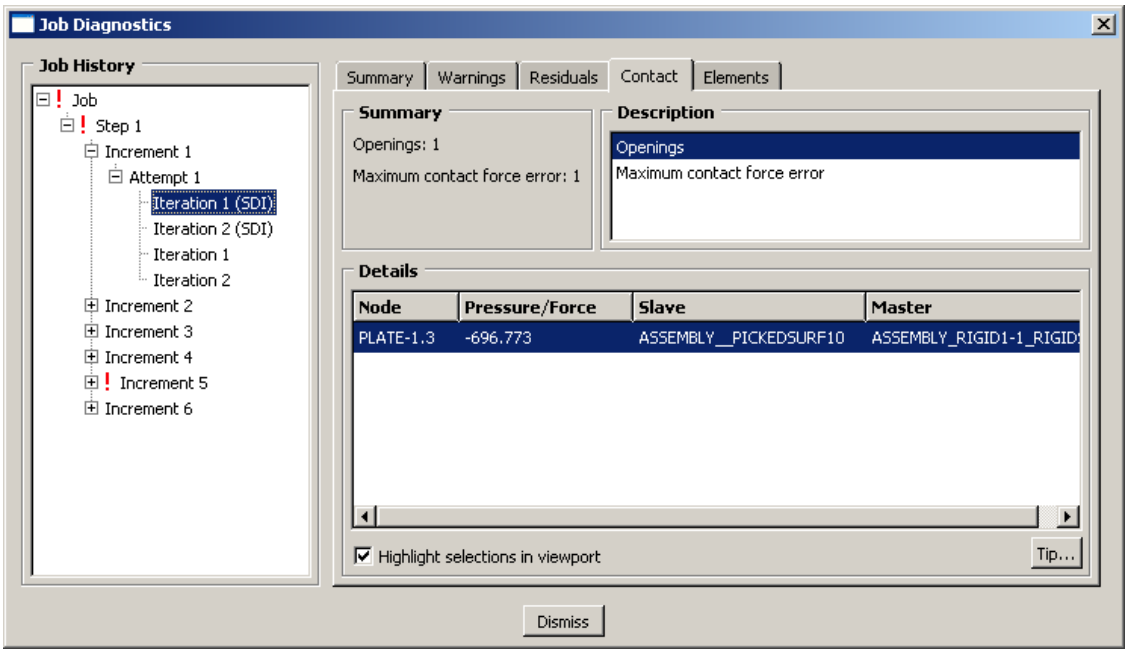


Figure W4–3. Contact overclosure in the first severe discontinuity iteration

f. Use the similar procedure to review the contact status in other severe discontinuity iterations. While the rigid surface was defined to be touching the deformable body exactly, there may be slight overclosures as a result of numerical round-off.

Question W4–2: Is the separation of the sheet from the rigid surface noted in contact diagnostics? (Hint: use the **Job Diagnostics** tool to highlight nodes that separate.)

Question W4–3: Why does the die “penetrate” the deformable plate? How could the model be modified to prevent or minimize this effect?

Modifications to address contact penetration

It is quite clear that the mesh is inadequate for the model. The mesh allows too much penetration of the rigid surface and cannot bend easily around the corner. Therefore, we will double the number of elements for the deformable part.

1. In the Model tree, expand the branch of the part named **Plate** underneath the **Parts** container. Double-click **Mesh** in the list under the **Plate** branch (see Figure W4–4).

   Note that Abaqus/CAE switches to the Mesh module.

© Dassault Systèmes, 2008

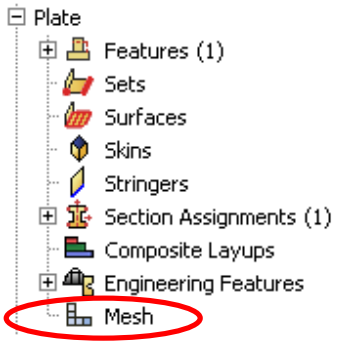


Figure W4–4. Branch of the part **Plate**

2. Use the **Seed Part** tool in the toolbox to re-seed the part with a global element size of **`10`**. Abaqus/CAE will issue a warning that the original mesh will be invalid by the seeding changes. Select **Delete Meshes**.
3. Use the **Mesh Part** tool to re-mesh the part based on the new seed size.

   Note that the predefined interaction definitions remain unchanged since they are based on the part geometry and not the part mesh.
4. Create a new job called **`contact_20`**.

   Tip: In the Model Tree, double-click the **Jobs** container.
5. Save your model database, submit the job for analysis, and monitor the job's process.
6. When the job is complete, enter the Visualization module and open the output database file **`contact_20.odb`**.
7. Plot the deformed shape.

   You will notice that the rigid surface still penetrates the plate. Such penetration could prevent the plate from sliding around the corner. Sharp corners should generally be avoided in contact models for this reason.
8. To make the corner more rounded, add a fillet to the analytical rigid part.
   a. In the Model Tree, expand the **Features** container underneath the branch of the part **RIGID1** and double-click **2D Analytical rigid shell-1**.

      The feature editor appears.
   b. In the **Edit Feature** dialog box, click **Edit Section Sketch**.
   c. In the sketcher, use the **Create Fillet: Between 2 Curves** tool to add a fillet of radius **`5.0`** in the analytical rigid surface. Click mouse button 2 in the viewport or click **Done** in the prompt area to exit the sketcher.
   d. In the **Edit Feature** dialog box, select **OK** to regenerate the feature.
9. Save your model database and re-run the job **contact_20**.
10. When the job is complete, enter the Visualization module and open the output database file **`contact_20.odb`**. Plot the deformed shape of the modified model.

© Dassault Systèmes, 2008

Zoom in on the corner of the rigid surface. You will see that this smoothing does not totally prevent penetration. The slave nodes are constrained against penetrating the master surface, but the segments forming the slave surface are not constrained against penetration by the master surface.

Adding an upper rigid surface

To prevent the separation of the plate from the rigid surface, add another rigid surface on top of the plate. Use the following steps to add a second rigid surface, as shown in Figure W4–5.

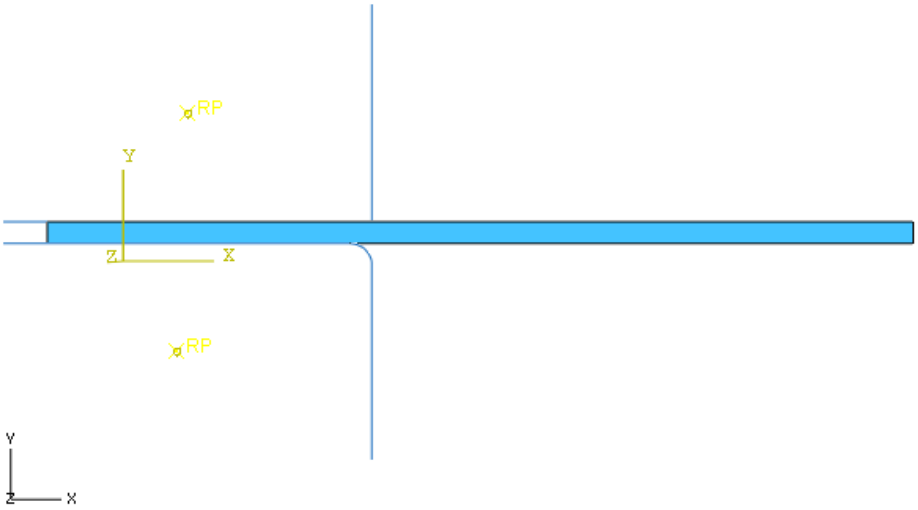


Figure W4–5. Plate with the upper and lower rigid surfaces

1. In the Model Tree, double-click **Instances** underneath the **Assembly** container to create an instance of the analytical rigid part **RIGID2**. Note that Abaqus/CAE switches to the Assembly module. If necessary, use the **Create Constraint: Edge to Edge** tool (click to expand the list of tools) to position the instance, as shown in Figure W4–5.
2. Create a contact interaction between the second rigid surface and the plate. In the Model Tree, double-click the **Interactions** container. In the **Create Interaction** dialog box, select **Surface-to-surface contact (Standard)** and click **Continue**. Select the second rigid surface as the master surface, making sure to choose the side facing the plate. Select the top surface of the plate as the slave surface. Accept all defaults in the interaction editor.
3. Create an **ENCASTRE** boundary condition to fix all degrees of freedom of the reference node of the part **RIGID2**.

   Tip: In the Model Tree, double-click the **BCs** container.
4. Create a new job named **`contact_20_mod1`**.

   Tip: In the Model Tree, double-click the **Jobs** container.

© Dassault Systèmes, 2008

5. Save your model database, submit the job for analysis and monitor the job's process.
6. When the job is complete, enter the Visualization module and open the file **`contact_20_mod1.odb`**. Plot the deformed shape.
   a. Is the reference node constrained correctly?
   b. Is the slave surface located at the correct location on the deformable body?
   c. Have you specified the element side information correctly?

   It is a common error to define the wrong surface for an analytical rigid surface. For example, one might define the top of the analytical rigid surface **RIGID2** as the master surface in the contact pair. Therefore, since the deformable body is underneath the rigid surface, Abaqus will immediately detect significant overclosures and try to move the deformable body to the outside of the rigid surface, causing severe element distortion and other problems. (If you specified the rigid surface correctly, you might wish to try reversing the side to see the messages printed to the message file.)

   Plotting the normals is useful for checking that the rigid surface is defined correctly. The normals of the master and slave contact surfaces should point toward one another.
7. Normals can be displayed on either the undeformed or the deformed model. To plot the normal in the undeformed state:
   a. Plot the undeformed model shape.
   b. Open the **Common Plot Options** dialog box by clicking in the toolbox.
   c. In the dialog box, switch to the **Normals** tabbed page.

      Using the options on this page, you can switch on display of normals on either elements or surfaces, set the color of the normals, etc.

      Are the deformable bodies properly constrained?

Using a third rigid surface to deform the plate

In this section you will create a third analytical rigid surface and use it to apply the displacement to the right-hand edge through contact.

1. Create an instance of the analytical rigid part **RIGID3** in the assembly. If necessary, use the **Translate Instance** tool to position the instance as shown in Figure W4–6. The beginning of the curvature should coincide with the upper-right endpoint of the plate.

© Dassault Systèmes, 2008

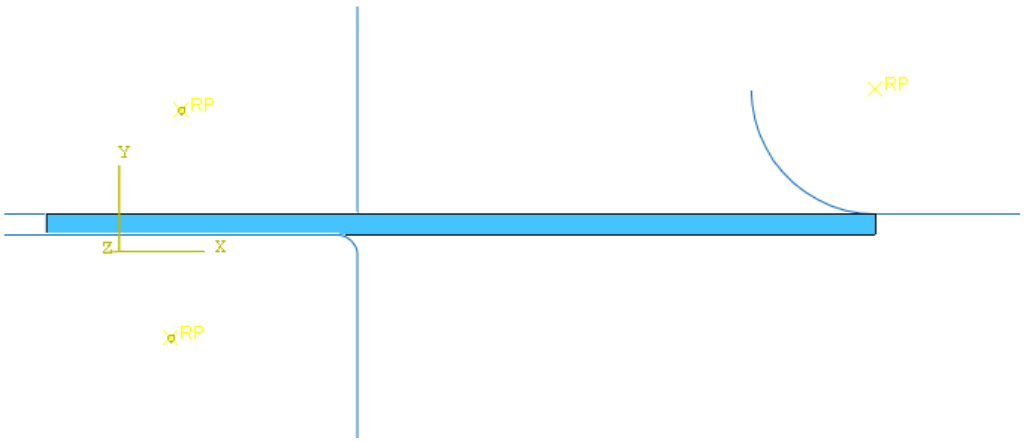


Figure W4–6. Plate with three rigid surfaces

The extension on the right side of the part **RIGID3** ensures that the deformable body does not move beyond the edge of the rigid surface. Given the configuration of the model and the loading, it is unlikely that the plate would move beyond the right edge of the rigid surface. However, it is generally good practice to extend rigid surfaces well past possible contact regions when possible.

2. Using a similar procedure as described earlier, define a contact interaction between **RIGID3** and the plate.
3. Edit the displacement boundary condition named **BC-3** to apply the displacement on the reference node of **RIGID3**.
   a. In the Model Tree, expand the **BCs** container and double-click **BC-3**.
   b. In the boundary condition editor, click **Edit Region** to select the reference node of **RIGID3**.
   c. In addition to the prescribed (non-zero) value for **U2**, also fix degrees of freedom **U1** and **UR3**.
4. Create a new job named **`contact_20_mod2`**.

   Tip: In the Model Tree, double-click the **Jobs** container.
5. Save your model database, submit the job for analysis and monitor the job's process.
6. When the job is complete, enter the Visualization module and open the file **`contact_20_mod2.odb`**. Plot the deformed shape and compare with the results from the previous analyses.

Note: A script that creates the complete models is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_solver_contact_answer.py` and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 4

Contact

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Learn how to specify the input for a simple rigid surface.
- Learn important techniques and options for analyzing contact problems.
- Understand the printed output for contact problems in the status (`.sta`), data (`.dat`), and message (`.msg`) files.

Introduction

In this workshop you will analyze contact between a plate and rigid surfaces. The plate is modeled with CPE4I solid, plane strain elements, which model bending behavior accurately with one element through the thickness of the plate. The rigid surfaces are modeled as analytical rigid surfaces. In the first part of the workshop you will analyze contact between the plate and a single lower surface as the right-hand edge of the plate is displaced downward and the other end is fixed. In the second part the plate is held fixed between two rigid surfaces as the right-hand edge is displaced downward, as shown in Figure W4–1. In the third part contact with a rigid surface replaces the prescribed edge displacement to deflect the right-hand edge downward.

© Dassault Systèmes, 2008

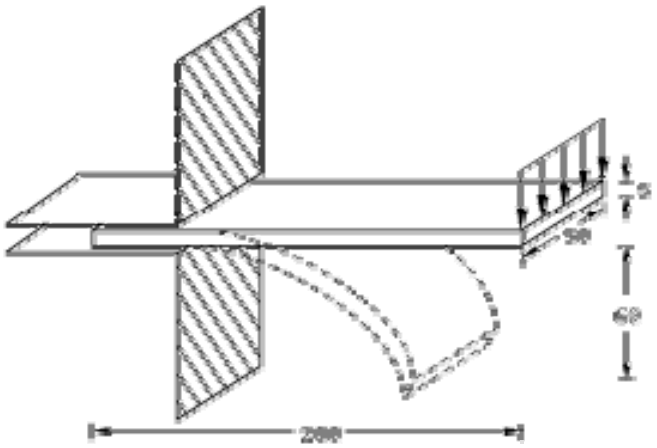


Figure W4–1. Plate constrained by rigid surfaces
at one end and displaced downward at the other

Contact with a lower rigid surface

1. Change to the **`../abaqus_solvers/keywords/workshop4`** subdirectory.
2. Look at input file **`w_contact_10.inp`**.

   Question W4–1: Note the definition of the rigid surface. Would the rigid surface behave differently if its reference node (901) had been defined at coordinates (30.0, 20.0)?

   All the printout options are in the first step definition. The ∗PRINT option with the CONTACT=YES parameter is useful for contact problems. The ∗PRINT option controls output to the message file. The CONTACT parameter on the ∗PRINT option is used to track the development of the solution in contact problems. Details about which points are contacting or separating in every “severe discontinuity iteration” are available in the message file.

   A prescribed displacement of –60.0 units is applied to the right-hand edge of the plate to deflect it in the 2-direction.
3. Run the Abaqus job using this file.
4. Open the status file in an editor. You can look at this file before the analysis is complete. The “severe discontinuity iterations” are caused by changes in contact.
5. When the job is finished, open the data file. Scan through the file, and look at the model interpretation sections as well as any warning or error messages at the end of the file.
6. Open the message file.
   a. The “SLAVE SURFACE” status printout is the result of the ∗PRINT, CONTACT=YES option. While the rigid surface was defined to be

© Dassault Systèmes, 2008

touching the deformable body exactly, there may be slight overclosures as a result of numerical round-off.

b. There is additional information concerning the cause of the “severe discontinuity iterations.”

7. Visualize results:

   a. Open the output database for the analysis in Abaqus/Viewer.

   b. Plot the deformed model shape.

      The plot shows that the plate has lifted off the rigid surface to the left of the corner.

Question W4–2: Is the separation of the sheet from the rigid surface noted in the message file? (Hint: search for the word “OPENING.”) Alternatively, use the **Job Diagnostics** tool in Abaqus/Viewer to highlight nodes that separate.

Question W4–3: Why does the die “penetrate” the deformable sheet? How could the model be modified to prevent or minimize this effect?

Modifications to address contact penetration

It is quite clear that the mesh is inadequate for the model. The mesh allows too much penetration of the rigid surface and cannot bend easily around the corner. Input file **`w_contact_20.inp`** has double the number of elements for the deformable surface.

1. Look at the file with the editor.

   The node definitions are the same as for the less refined mesh, but the element definitions use the even-numbered nodes that were not used before.

   The ∗SURFACE and ∗CONTACT PAIR options remain unchanged.

2. Run this analysis, and look at the printed output files. View the deformed model shape in Abaqus/Viewer.
3. You will notice that the rigid surface still penetrates the plate. Such penetration could prevent the sheet from sliding around the corner. Sharp corners should generally be avoided in contact models for this reason. To make the corner more rounded, add FILLET RADIUS=5.0 to the ∗SURFACE option and rerun the model.
4. Plot the deformed model shape of the modified model in Abaqus/Viewer. Zoom in on the corner of the rigid surface to see that the smoothing does not totally prevent penetration. The slave nodes are constrained against penetrating the master surface, but the segments forming the slave surface are not constrained against penetration by the master surface.

© Dassault Systèmes, 2008

Adding an upper rigid surface

To prevent the separation of the sheet from the rigid surface, add another rigid surface on top of the sheet. Refer to the online User's Manual as needed. Use the following steps to add a second rigid surface that looks like the plot shown in Figure W4–2.

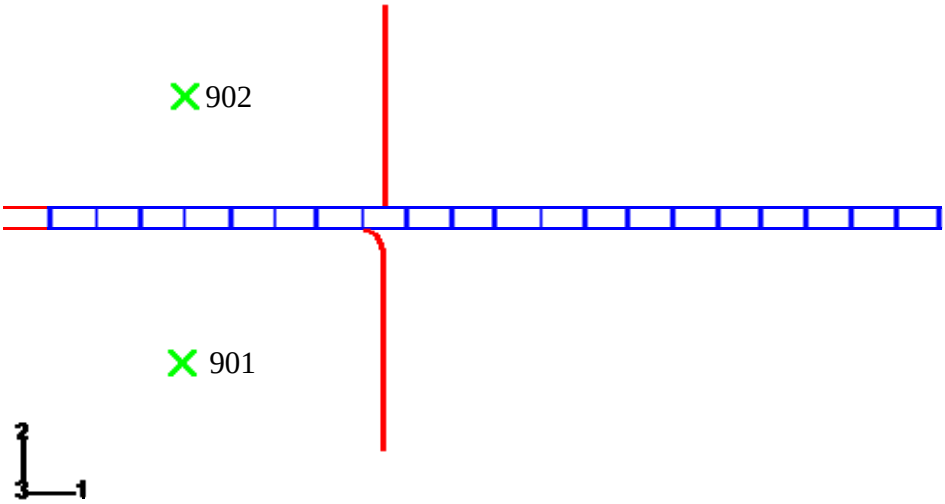


Figure W4–2. Plate with upper and lower rigid surfaces

1. Add the nodal definition for the reference node 902 for the new rigid surface.
2. Add a ∗SURFACE option for the second die.
3. Add a ∗RIGID BODY option for the second die.
4. Add a ∗SURFACE option for a new slave surface on the top of the sheet.
5. Add a ∗CONTACT PAIR definition for the new contact surface.
6. Add a ∗SURFACE INTERACTION definition, and specify the same thickness for the contact pair as the thickness given on the ∗SOLID SECTION option for the deformable sheet.
7. Fix all degrees of freedom for the reference node using the ∗BOUNDARY option in the model definition section.

© Dassault Systèmes, 2008

8. Run this analysis, and look at the printed output files. View the deformed model shape in Abaqus/Viewer.

   Is the reference node constrained correctly?

   Is the slave surface located at the correct location on the deformable body? Have you specified the element side information correctly?

   Are the points for the analytical rigid surface definition given in the correct order?

9. The order in which the points are specified defines the outward normal. A common input error is to define a rigid surface backward so that it is "inside out." For example, in this problem if the second rigid surface is specified by giving points from left to right, the outward normal is upward. Therefore, since the deformable body is underneath the rigid surface, Abaqus will immediately detect significant overclosures and try to move the deformable body to the outside of the rigid surface, causing severe element distortion and other problems. (If you specified the rigid surface correctly, you might wish to try reversing the order of the points defining the line segments just to see what sort of messages are printed in the message file.) Plotting the normals in a postprocessor is useful for checking that the rigid surface is defined correctly. The normals of the master and slave contact surfaces should point toward one another.

   Normals can be displayed in Abaqus/Viewer on either the undeformed or the deformed model. To plot the normal in the undeformed state:

   a. Plot the undeformed model shape.

   b. Open the **Common Plot Options** dialog box by clicking in the toolbox.

   c. In the dialog box, switch to the **Normals** tabbed page.

      Using the options on this page, you can switch on display of normals on either elements or surfaces, set the color of the normals, etc.

      Are the deformable bodies properly constrained?

For this problem, one end of the deformable body is clamped. A common problem for contact models is that inadequately constrained deformable bodies cause numerical singularity problems because of rigid body motion. This is especially true at the beginning of the analysis if you are relying on the contacting surfaces or bodies to prevent rigid body motion; before contact is established, the rigid body modes may not be constrained.

© Dassault Systèmes, 2008

Using a third rigid surface to deform the plate

In this section you will define a third analytical rigid surface and use it to apply the displacement to the right-hand edge through contact.

1. Define a third analytical rigid surface as shown in Figure W4–3 using the procedure described in the previous section.

   The rigid surface should have a circular segment with a radius of 30.0 and a small straight segment extending to the right at the bottom.

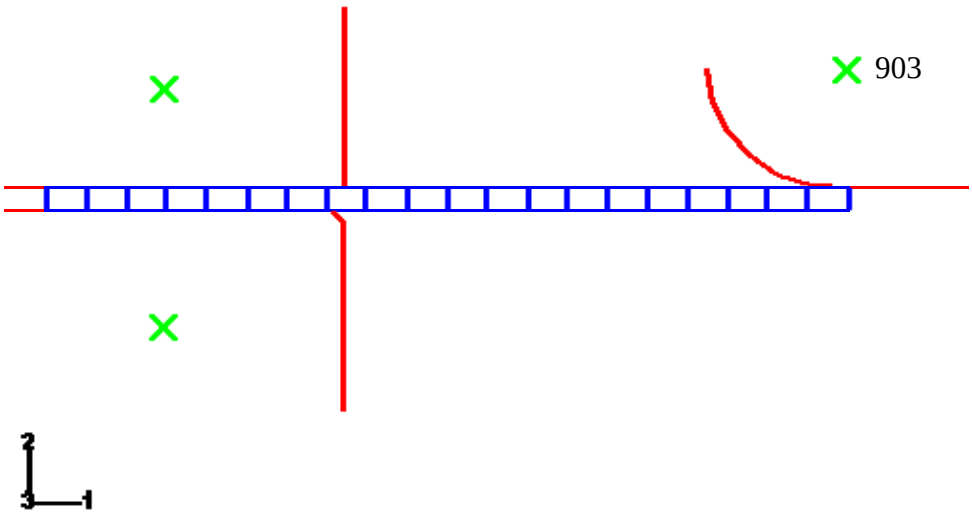


Figure W4–3. Plate with three rigid surfaces

   This extension ensures that the deformable body does not move beyond the edge of the rigid surface. Given the configuration of the model and the loading, it is unlikely that the deformable body would move beyond the right edge of the rigid surface. However, it is generally good practice to extend rigid surfaces well past possible contact regions when possible. You do not need to define a new slave surface on the top of the plate because it has already been defined.

2. Constrain the appropriate degrees of freedom of the new reference node as before, including the rotational degree of freedom 6. Do not use the type boundary condition ENCASTRE to constrain the reference node because you will need to prescribe displacements in the 2-direction.
3. Delete the displacement boundary condition that was applied to the right-hand edge of the plate. Then, define a similar boundary condition on the reference node of the analytical rigid surface: 60.0 units in the negative 2-direction.

© Dassault Systèmes, 2008

Tip: If you constrain the reference node in the model data using a node set name, you must apply the motion in the step definition using the node set name, as shown below:

```
*BOUNDARY
REF3, 1, 6
...
*STEP
...
*BOUNDARY
REF3, 2, 2, -60.0
```

The default parameter on the *BOUNDARY option is OP=MOD, which specifies that the boundary conditions will be "modified." Only displacement degree of freedom 2 of the reference node will be changed; all other boundary conditions from the model definition will remain.

4. Run this analysis, and look at the printed output files. View the deformed model shape in Abaqus/Viewer.

Note: Complete input files are available for your convenience. You may consult these files if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input files are named

`w_contact_20_fillet_complete.inp`

`w_contact_20_mod1_complete.inp`

`w_contact_20_mod2_complete.inp`

and are available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 5

CLD Analysis of a Seal using Abaqus/Standard

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Evaluate a hyperelastic material.
- Define surface-to-surface contact and self-contact.
- Perform a large displacement analysis with Abaqus/Standard.
- Use the Visualization module to create a compression load-deflection curve.

Introduction

In this workshop, a compression analysis of a rubber seal is performed to determine the seal's performance. The goal is to determine the seal's compression load-deflection (CLD) curve, deformation and stresses. The analysis will be performed using Abaqus/Standard.

As shown in Figure W5–1, the top outer surface of the seal is covered with a polymer layer, and the seal is compressed between two rigid surfaces (the upper one is displaced along the negative *Y*-direction; the lower one is fixed). During compression, the cover contacts the top rigid surface; the outer surface of the seal is in contact with the cover and the bottom rigid surface; in addition the inner surface of the seal may come into contact with itself.

© Dassault Systèmes, 2008

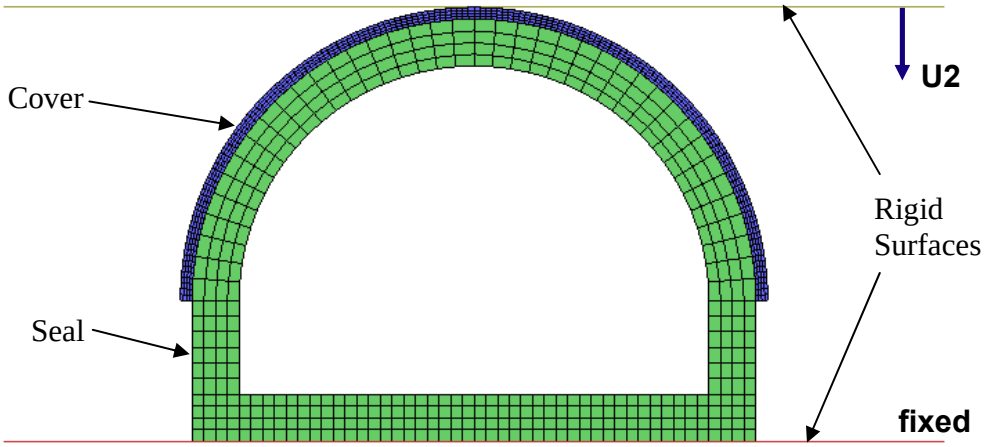


Figure W5–1. Seal model: meshed assembly

Preliminaries

1. Enter the working directory for this workshop
   `../abaqus_solvers/interactive/workshop5`
2. Run the script **`ws_solver_seal.py`** using the following command:
   `abaqus` **`cae startup=ws_solver_seal.py`**.

The above command creates an Abaqus/CAE database named **`seal.cae`** in the current directory. The geometry, mesh, and material definitions are included in the model named **`Seal`**. You will first perform a material evaluation to evaluate the stability of the hyperelastic material model, add the necessary data to complete the model, run the job, and finally postprocess the results.

Evaluating Material

It is important to determine whether the material model of the seal will be stable during the analysis. Before completing the model, evaluate the material definition used for the seal.

1. Review the material definition. In the Model Tree, double-click **Santoprene** underneath the **Materials** container. It is a hyperelastic material with a first-order polynomial strain energy potential. The coefficients are already chosen for the analysis.
2. Evaluate the material definition. Abaqus/CAE provides a convenient **Evaluate** option that allows you to view the behavior predicted by a hyperelastic material by performing standard tests to choose a suitable material formulation. You will use this option to view the behavior predicted by the material **Santoprene**.

© Dassault Systèmes, 2008

a. From the main menu bar in the Property module, choose **Material→Evaluate→Santoprene**.

b. The **Evaluate Material** dialog box appears. Notice that you can choose either the **Coefficients** or **Test data** source for evaluating the material. Typically the test data are used to define a material model; you can use the **Evaluate** option to view the predicted behavior and adjust the material definition as necessary. In this workshop you will only evaluate the stability of the material model for the given coefficients.

c. In the **Evaluate Material** dialog box, accept all defaults and click **OK**. Abaqus/CAE creates and submits a job to perform the standard tests using the material **Santoprene**; at the same time, Abaqus/CAE switches to the Visualization module and displays the evaluation results when the job is complete. Figure W5–2 shows the **Material Parameters and Stability Limit Information** dialog box; Figure W5–3 shows three stress vs. strain plots from uniaxial, biaxial, and planar tests.

Question W5–1: What do the plots indicate about the stability of the material?

Based on these results, you can have confidence that your material will remain stable.

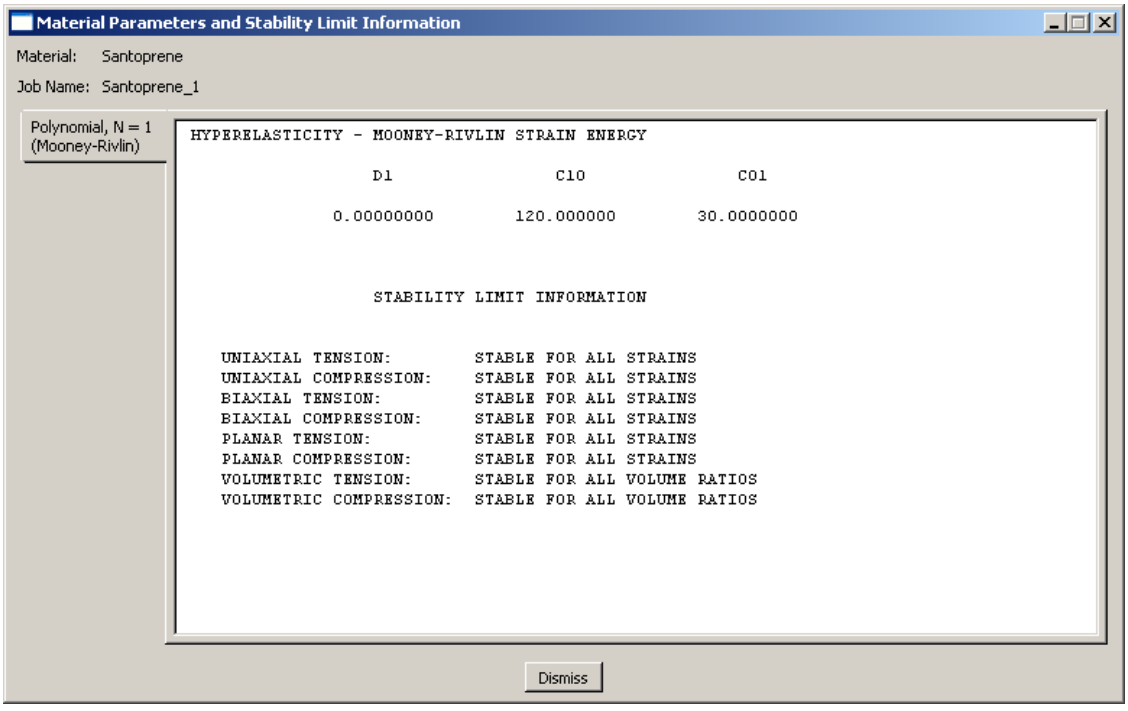


Figure W5–2. Material parameters and stability limit information

© Dassault Systèmes, 2008

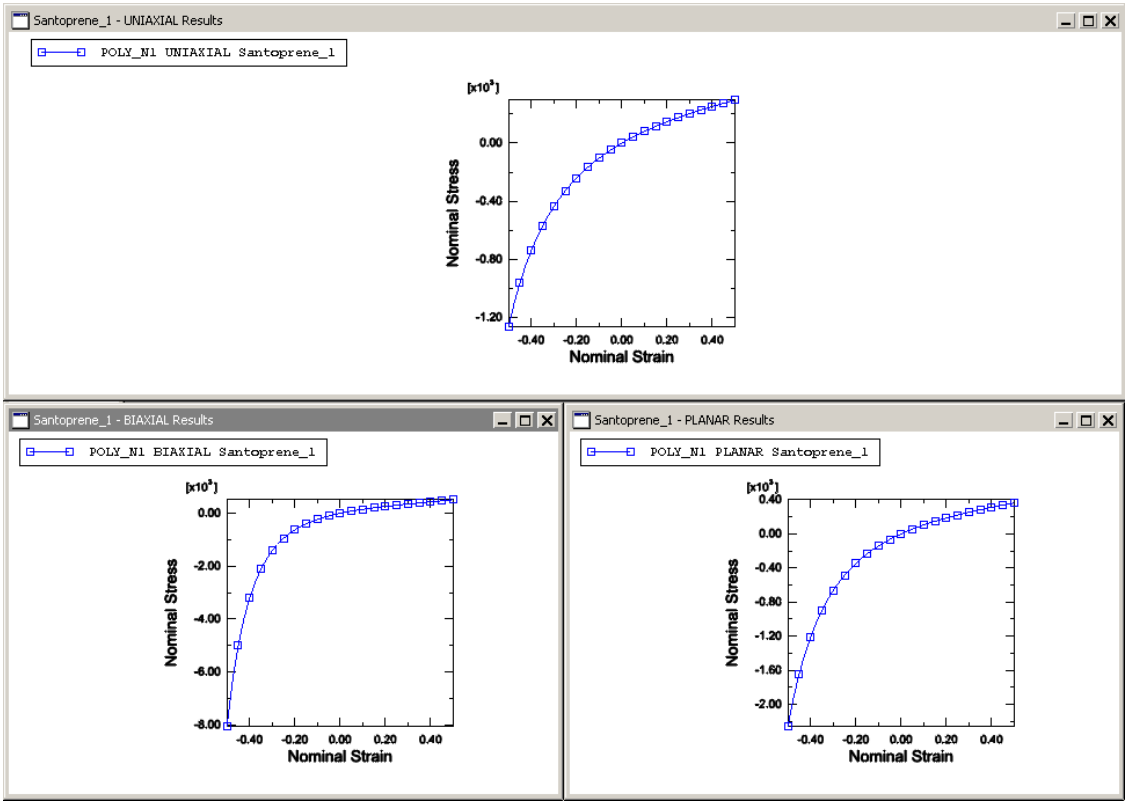


Figure W5–3. Material evaluation results for uniaxial, biaxial, and planar tests

After evaluating the material, you will now complete the model definition. Close the viewports and dialog box displaying the material evaluation results, if necessary, to view the model for the subsequent procedure.

Defining the step and contact interactions

1. In the Model Tree, double-click the **Steps** container to create a static, general step named **PushDown** with a time period of **1**. Turn on **Nlgeom**. In the **Incrementation** tabbed page of the step editor, enter a value of **0.005** for **Initial Increment size** and **200** for **Maximum number of increments**.
2. Define a surface-to-surface contact interaction between the seal and the bottom rigid surface.
    a. In the Model Tree, double-click the **Interactions** container. In the **Create Interaction** dialog box, name the interaction **BotSeal** and select the step **PushDown** and **Surface-to-surface contact (Standard)**. Click **Continue**.
    b. You will be prompted to select a master surface. In the prompt area, click **Surfaces**. In the **Region Selection** dialog box that appears, select the predefined surface **Bottom** and toggle on **Highlight selections in viewport** to view this surface. Click **Continue**.

© Dassault Systèmes, 2008

c. In the prompt area, select **Surface** as the slave surface type. In the **Region Selection** dialog box that appears, select the predefined surface **SealOuter** and visualize this surface. Click **Continue**.

The interaction editor appears.

d. In the **Edit Interaction** dialog box, enter a value of **`0.001`** for **Specify tolerance for adjustment zone** and accept all other defaults. Click **OK** to exit the interaction editor.

Note that Abaqus/CAE automatically assigns the predefined (also the only available) interaction property **frictionless** to this interaction.

3. Using a similar procedure, define the following surface-to-surface contact pairs as listed in Table W5–1 with the interaction property **frictionless** and the adjustment value of **`0.001`**.

Table W5–1. Contact pairs

Interaction Name	Master Surface	Slave Surface
`TopCover`	`Top`	`Cover`
`SealCover`	`SealOuter`	`Cover`

Question W5–2: In the interaction **`SealCover`**, why do we choose **`SealOuter`** as the master surface?

4. Create a self-contact interaction for the inner surface of the seal.

a. In the Model Tree, double-click the **Interactions** container. In the **Create Interaction** dialog box, name the interaction **`SealSelf`** and select the step **`PushDown`** and **Self-contact (Standard)**. Click **Continue**.

b. In the **Region Selection** dialog box that appears, select the predefined surface **SealInner** and visualize this surface. Click **Continue**.

The interaction editor appears.

c. In the **Edit Interaction** dialog box, accept all defaults and click **OK**.

Defining boundary conditions and output requests

Asymmetric lateral sliding of the model is prevented by constraining the seal and the cover along their vertical symmetry axes in the *X*-direction. The bottom rigid surface is fixed, and a displacement of –6 units is applied to the top rigid surface along the *Y*-direction to compress the seal between the two surfaces. To complete these boundary conditions:

1. In the Model Tree, double-click the **BCs** container to create a **Displacement/Rotation** type boundary condition named **`Fix1`** in the step **PushDown**. When prompted to select the region, click **Sets** in the prompt area (if necessary). In the **Region Selection** dialog box, select the predefined set **Fix1**,

toggle on **Highlight selections in viewport** to visualize the selection, and click **Continue**. In the **Edit Boundary Condition** dialog box, toggle on **U1**, accept the default value of **0**, and click **OK**.

2. Create a **Symmetry/Antisymmetric/Encastre** type boundary condition named **FixBot** to encastre the predefined set **BotRP** (the reference node of the bottom rigid surface).
3. Create a **Displacement/Rotation** type boundary condition named **PushDown** in the step **PushDown** to define the displacement of the top rigid surface. Select the predefined set **TopRP** (the reference node of the top rigid surface). Specify a value of **0** for **U1** and **UR3**, and **-6** for **U2**.
4. Edit the field output request named **F-Output-1** to include the nominal strain, **NE**.
5. Create a new history output request in the step **PushDown** for the set **TopRP** to write the history of the variables **Displacements: U** and **Forces: RF** to the output database file.

Running the job and postprocessing the results

1. Create a job named **seal** for the model **Seal**.
2. Save your model database, submit the job for analysis, and monitor the job's process.

   When the job is complete, open the output database file **seal.odb** in the Visualization module and postprocess the results.
3. Plot the undeformed and the deformed model shapes. To distinguish between the different instances, color code the model based on instances.

   Tip: From the toolbar, select **Part instances** from the color-coding pull down menu, as shown in Figure W5–4 (or use the **Color Code Dialog** tool to customize the color for each instance).

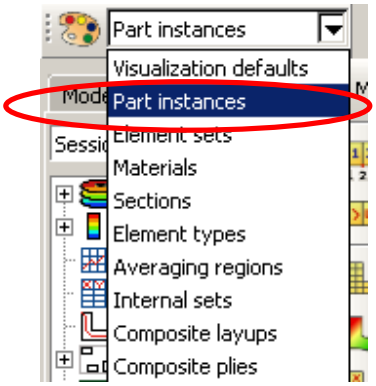


Figure W5–4. Color-coding pull down menu

4. Use the **Animate: Time History** tool to animate the deformation history.

© Dassault Systèmes, 2008

5. Display only the seal. In the Results Tree, expand the **Instances** container underneath the output database file named **`seal.odb`**. Click mouse button 3 on the instance **SEAL-1** and select **Replace** from the menu that appears.

   Abaqus/CAE now displays only this instance.

6. Contour the Mises stress of the seal on the deformed shape. If necessary, use the frame selector in the context bar to select the final increment.

   The contour plot is shown in Figure W5–5.

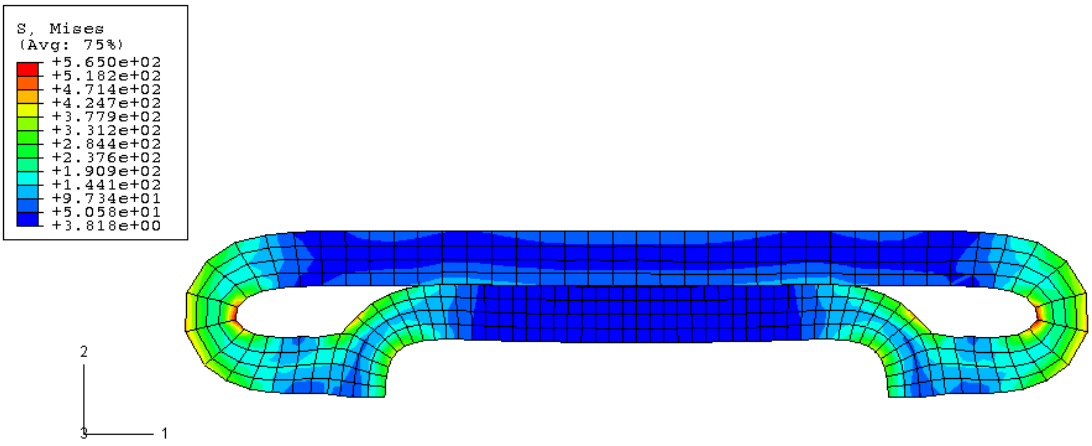


Figure W5–5. Mises contour plot

7. Contour the minimum and maximum principal nominal strains. Elastic strains can be very high for hyperelastic materials. Because of this, the linear elastic material model is not used because it is not appropriate for elastic strains greater than approximately 5%.
8. Display the reaction force history at the reference node of the top rigid surface: In the Results Tree, expand the **History Output** container underneath the output database file named **`seal.odb`** and double-click **Reaction force: RF2 PI: TOP-1 Node 1 in NSET TOPRP**.
9. You will now create the CLD curve.
   a. In the **History Output** container, click mouse button 3 on **Reaction force: RF2 PI: TOP-1 Node 1 in NSET TOPRP** and select **Save As** from the menu that appears. Save the data as **`Force`**.
   b. Click mouse button 3 on **Spatial displacement: U2 PI: TOP-1 Node 1 in NSET TOPRP** and select **Save As** from the menu that appears. Save the data as **`Disp`**.
   c. In the Results Tree, double-click **XYData**. In the **Create XY Data** dialog box that appears, select the **Operate on XY data** source and click **Continue**.

      The **Operate on XY Data** dialog box appears.

© Dassault Systèmes, 2008

d. From the **Operators** listed in the **Operate on XY Data** dialog box, select **combine(X, X)** and then **abs(A)**. Note that the **abs(A)** operator is used to obtain the absolute values. In the **XY Data** field, double-click the curve **Disp**. The current expression reads **`combine(abs("Disp"))`**. Move the cursor before the far-right bracket, enter a comma, and then select the operator **abs(A)**. In the **XY Data** field, double-click the curve **Force**. The final expression reads **`combine(abs("Disp"), abs("Force") )`**. Click **Plot Expression** to plot this expression. Save this plot as **`CLD`**.

10. Customize the plot as follows:

a. Double-click anywhere on the chart to open the **Chart Options** dialog box.

- In the **Grid Display** tabbed page, toggle on the major *X*- and *Y*-grid lines. Set the line color to blue and the line style to dashed.
- Change the fill color using the following **RGB** values: red: 175; green: 250; blue: 185.
- In the **Grid Area** tabbed page, select **Square** as the size and drag the slider to **80**. From the list of auto-alignments, choose the one that places the chart in the center of the viewport

b. Double-click the legend to open the **Chart Legend Options** dialog box.

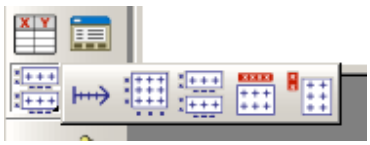
- In the **Contents** tabbed page, click to increase the legend text font size to **`10`**.
- In the **Area** tabbed page, toggle on **Inset**.
- Toggle on **Fill** to flood the legend with a white background.
- In the viewport, drag the legend over the chart.

c. Double-click either axis to open the **Axis Options** dialog box.

- In the **X Axis** region of the dialog box, select the displacement axis.
- In the **Scale** tabbed page, place **4** major tick marks on the *X*-axis at (use the **By count** method).
- In the **Title** tabbed page, change the *X*-axis title to **`Displacement (inch)`**.
- In the **Y Axis** region of the dialog box, select the force axis.
- In the **Scale** tabbed page, specify that the *Y*-axis should extend from **`0`** (the *Y*-axis minimum) to **`250`** (the *Y*-axis maximum).
- Increase the number of *Y*-axis minor tick marks per increment to **4**.
- In the **Title** tabbed page, change the *Y*-axis title to **`Force (lbf)`**.
- In the **Axes** tabbed page, change the font size for both axes to **`10`**.

© Dassault Systèmes, 2008

d. Expand the list of plot option icons in the toolbox:



e. Examine the remaining options. Add the following plot title: **`CLD Diagram`**. Double-click the plot title to open the **Plot Title Options** dialog box.

- In the **Title** tabbed page, click to change the legend text style to bold.
- In the **Area** tabbed page, toggle on **Inset**.
- In the viewport, drag the plot title above the chart.

f. Click in the toolbox to open the **Curve Options** dialog box. Change the legend text to **`Top Surface Ref Point`** and toggle on **Show symbol**. Set the color for both the line and symbols to red. Use large filled circles for the symbols. Reposition the legend as necessary.

The final plot appears as shown in Figure W5–6.

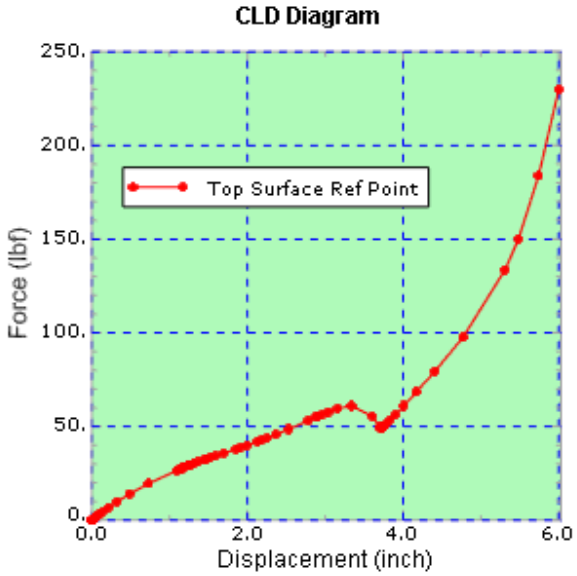


Figure W5–6. Compression load deflection diagram

Question W5-3: What does the inverted peak near 4 inches of deflection represent?

© Dassault Systèmes, 2008

Note: A script that creates the complete seal model is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_solver_seal_answer.py` and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 5

CLD Analysis of a Seal using Abaqus/Standard

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Evaluate a hyperelastic material.
- Define surface-to-surface contact and self-contact.
- Perform a large displacement analysis with Abaqus/Standard.
- Use Abaqus/Viewer to create a compression load-deflection curve.

Introduction

In this workshop, a compression analysis of a rubber seal is performed to determine the seal's performance. The goal is to determine the seal's compression load-deflection (CLD) curve, deformation and stresses. The analysis will be performed using Abaqus/Standard.

As shown in Figure W5–1, the top outer surface of the seal is covered with a polymer layer, and the seal is compressed between two rigid surfaces (the upper one is displaced along the negative 2-direction; the lower one is fixed). During compression, the cover contacts the top rigid surface; the outer surface of the seal is in contact with the cover and the bottom rigid surface; in addition the inner surface of the seal may come into contact with itself.

© Dassault Systèmes, 2008

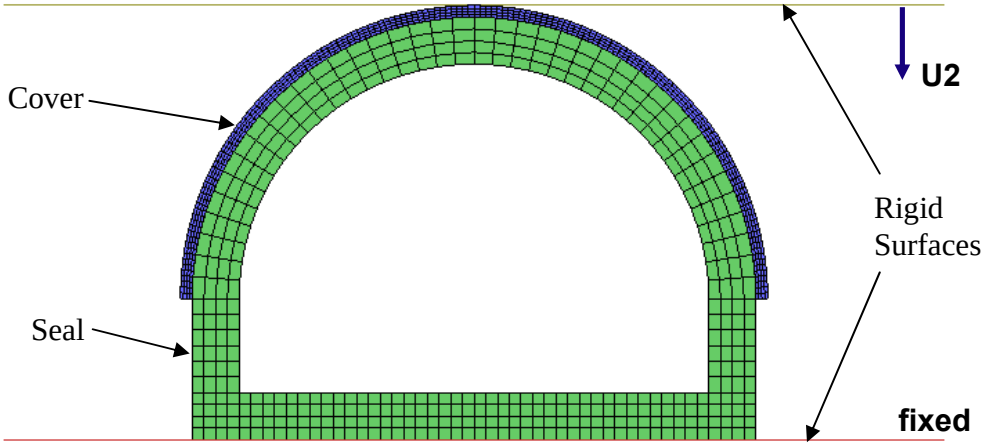


Figure W5–1. Seal model

Seal analysis

1. Change to the **`../abaqus_solvers/keywords/workshop5`** directory.
2. Open the input file **`w_seal.inp`**, which already contains the nodes, elements, and material model data for the analysis. You will first use Abaqus/CAE functionality to evaluate the stability of the hyperelastic material model and then edit the input file to include the contact, step and boundary condition definitions.

Material Evaluation

It is important to determine whether the material model of the seal will be stable during the analysis. Before completing the input file, evaluate the material definition that is used for the seal.

1. Use your text editor to review the supplied workshop model contained in the file **`w_seal.inp`**.
2. The material named **`SANTOPRENE`** is used for the seal. Locate the **`*MATERIAL, NAME=SANTOPRENE`** option. It is a hyperelastic material with a first order polynomial strain energy potential. The coefficients are already specified for the analysis.
3. Evaluate the material definition. Abaqus/CAE provides a convenient **Evaluate** option that allows you to view the behavior predicted by a hyperelastic material by performing standard tests to choose a suitable material formulation. You will use this option to view the behavior predicted by the material **`SANTOPRENE`**.
    a. Start a session of AQUS/CAE using the following command at the command prompt:

    `abaqus` **`cae`**

    In the **Start Session** dialog box, select **Create Model Database**.

© Dassault Systèmes, 2008

b. In the Model Tree, double-click the **Materials** container to create a material definition as specified in the input file. In the **Edit Material** dialog box, name the material **Santoprene**; from the menu bar, select **Mechanical→Elasticity→Hyperelastic**; in the **Hyperelastic** field, select the **Polynomial** strain energy potential and the **Coefficients** input source, accept a strain energy potential order of 1, and enter the values of the coefficients (defined in the input file) as shown in Figure W5–2. Click **OK** to save the material definition and exit the material editor.

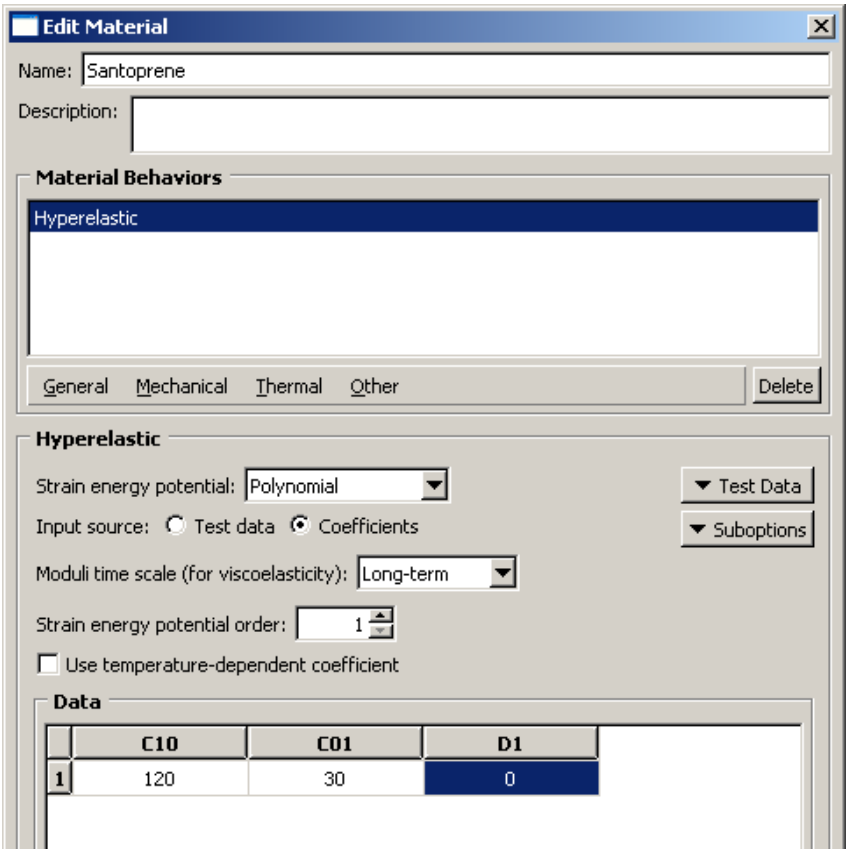


Figure W5–2. Material editor

c. From the main menu bar in the Property module, select **Material→Evaluate→Santoprene**.

d. The **Evaluate Material** dialog box appears. Notice that you can choose either the **Coefficients** or **Test data** source for evaluating the material. Typically the test data are used to define a material model; you can use the **Evaluate** option to view the predicted behavior and adjust the material definition as necessary. In this workshop you will only evaluate the stability of the material model for the given coefficients.

e. In the **Evaluate Material** dialog box, accept all defaults and click **OK**. Abaqus/CAE creates and submits a job to perform the standard tests using the material **Santoprene**; at the same time, Abaqus/CAE switches to the

© Dassault Systèmes, 2008

Visualization module and displays the evaluation results when the job is complete. Figure W5–3 shows the **Material Parameters and Stability Limit Information** dialog box; Figure W5–4 shows three stress vs. strain plots from uniaxial, biaxial, and planar tests.

Question W5–1: What do the plots indicate about the stability of the material?

Based on these results, you can have confidence that your material will remain stable.

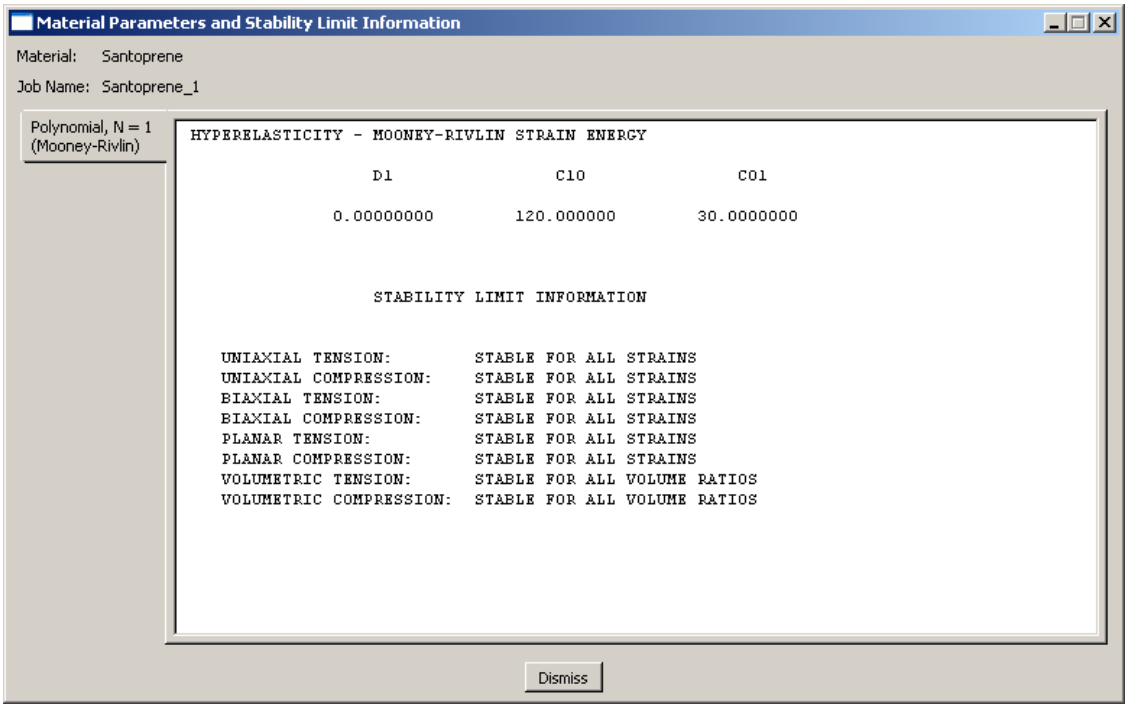


Figure W5–3. Material parameters and stability limit information

© Dassault Systèmes, 2008

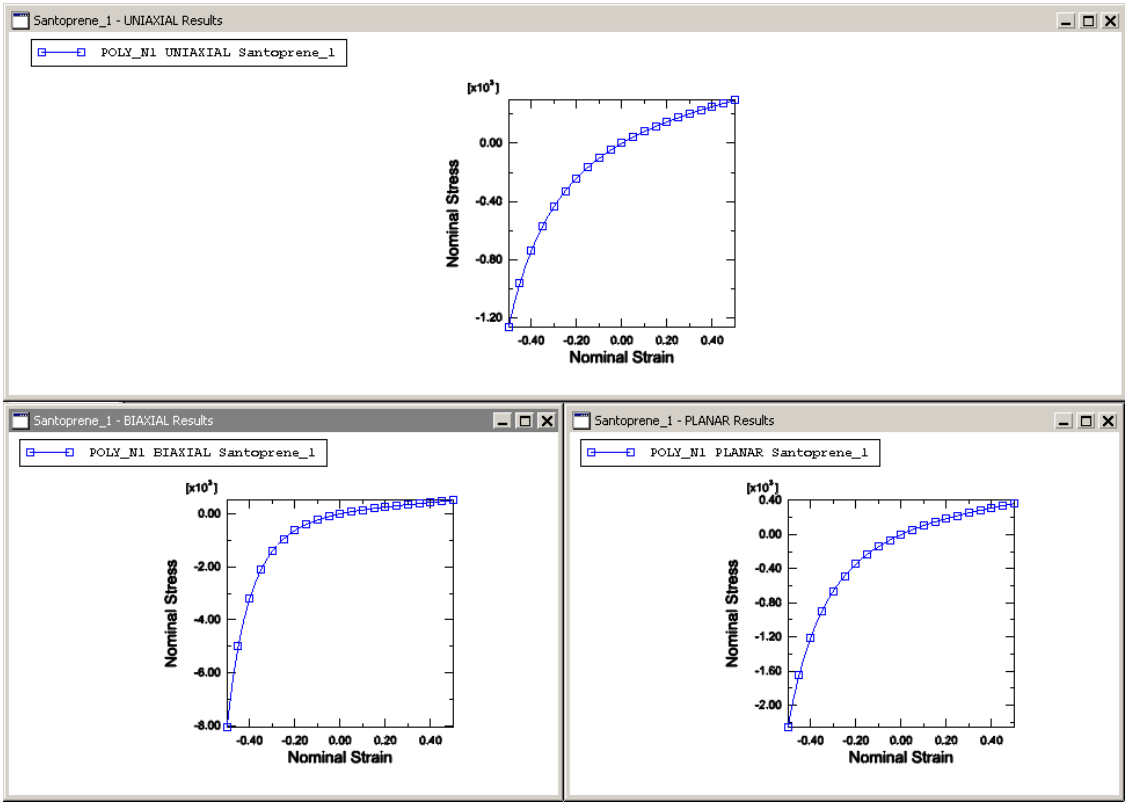


Figure W5–4. Material evaluation results for uniaxial, biaxial, and planar tests

After evaluating the material, you can exit Abaqus/CAE and will now complete the model definition.

Contact interactions

1. Open the input file **`w_seal.inp`** in a text editor.
2. Define contact pairs as listed in Table W5–1. The surfaces which will be used in the contact pair definitions are shown in Figure W5–5. The required option is:

   ```
   *CONTACT PAIR, INTERACTION=frictionless, ADJUST=0.001
   Cover, SealOuter
   Cover, Top
   SealOuter, Bottom
   ```

   Note that the interaction property named **`frictionless`** has already been defined in the input file. Locate the **`*SURFACE INTERACTION, NAME=frictionless`** option to review its definition.

© Dassault Systèmes, 2008

Table W5–1. Contact pairs

Slave Surface	Master Surface
Cover	**SealOuter**
Cover	**Top**
SealOuter	**Bottom**

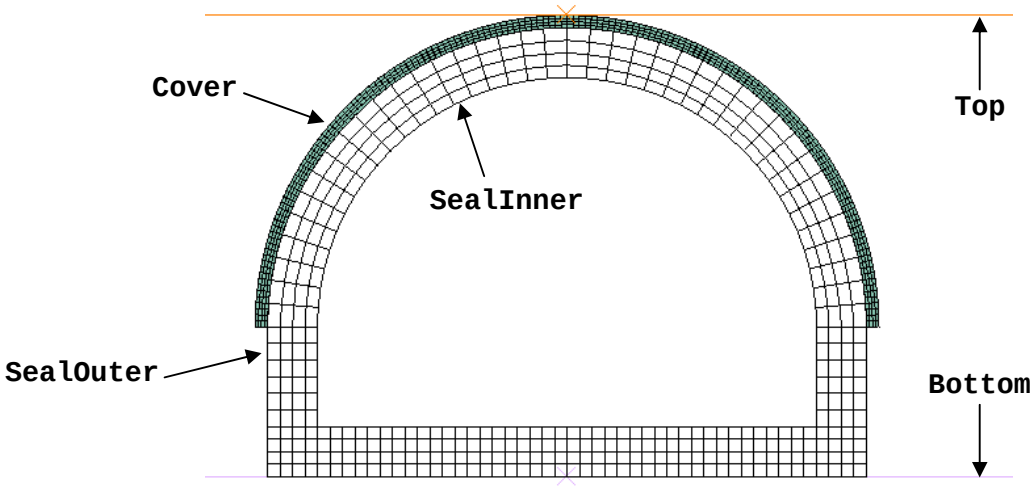


Figure W5–5. Contact surfaces

3. Define a self-contact definition for the inner surface of the seal:

```
*CONTACT PAIR, INTERACTION=frictionless
SealInner,
```

Question W5–2: In the interaction between the seal and the cover, why do we choose **SealOuter** as the master surface?

Step definition

1. Define a general static step considering geometric nonlinearity. Set the initial time increment size to 0.5% of the total time period. The following option defines the procedure:

```
*STEP, NLGEOM=YES
*STATIC
0.005, 1.
```

© Dassault Systèmes, 2008

Boundary conditions and history output requests

1. Asymmetric lateral sliding of the model is prevented by constraining the seal and the cover along their vertical symmetry axes in the *X*-direction. The bottom rigid surface is fixed, and a displacement of –6 units is applied to the top rigid surface along the *Y*-direction to compress the seal between the two surfaces. The node sets on which the boundary conditions will be defined are shown in Figure W5–6. The following option completes these boundary conditions:

```
*BOUNDARY
Fix1, 1, 1
BotRP, ENCASTRE
TopRP, 1, 1
TopRP, 2, 2, -6.
TopRP, 6, 6
```

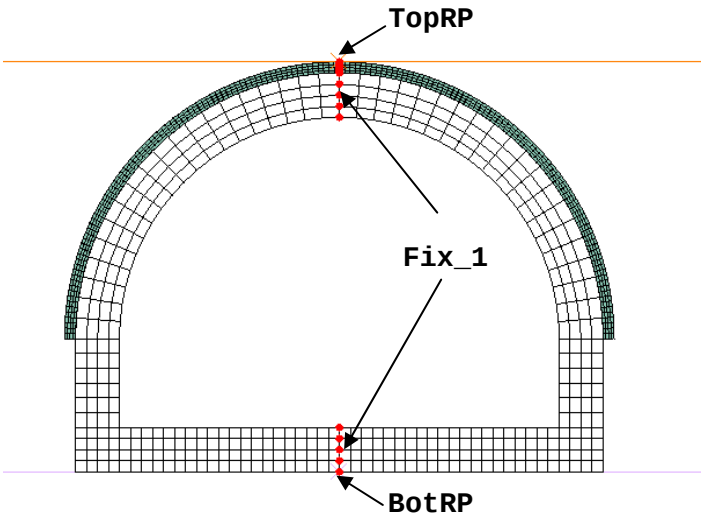


Figure W5–6: Node Sets

2. The preselected default field output does not include the nominal strain **NE**; to visualize the nominal strain in Abaqus/Viewer, you will write additional field output to the output database file. Locate the ***OUTPUT, FIELD, VARIABLE=PRESELECT** option and add the following sub-option:

```
*ELEMENT OUTPUT
NE,
```

3. Add a history output request to write the history of **RF2** and **U2** for the set **TopRP** to the output database file. The required option is:

```
*OUTPUT, HISTORY
*NODE OUTPUT, NSET=TopRP
RF2, U2
```

4. Save all the changes and close the input file.

© Dassault Systèmes, 2008

Running the job and visualizing the results:

Run the analysis using the following command:

```
abaqus job=w_seal
```

When the job is complete, use the following procedure to visualize the results using Abaqus/Viewer:

1. Start Abaqus/Viewer and open the file **w_seal.odb**:

   ```
   abaqus viewer odb=w_seal.odb
   ```

2. Plot the undeformed and the deformed model shapes. To distinguish between the different parts, color code the model based on section assignments.

   Tip: From the toolbar, select **Sections** from the color-coding pull down menu, as shown in Figure W5–7 (or use the **Color Code Dialog** tool to customize the color for each section).

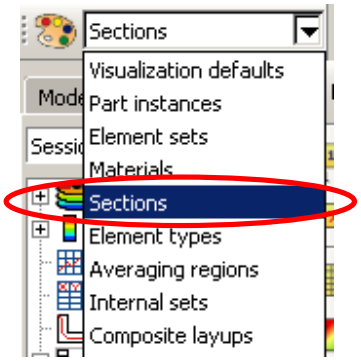


Figure W5–7. Color-coding pull down menu

3. Use the **Animate: Time History** tool to animate the deformation history.
4. Display only the seal. In the Results Tree, expand the **Sections** container underneath the output database file named **w_seal.odb**. Click mouse button 3 on the section **PART-1-1.ALL_SEAL_ELEMS.Section-ALL_SEAL_ELEMS** and select **Replace** from the menu that appears.

   Abaqus/CAE now displays only the elements associated with the seal.
5. Contour the Mises stress of the seal on the deformed shape. If necessary, use the frame selector in the context bar to select the last increment.

   The contour plot is shown in Figure W5–8.

© Dassault Systèmes, 2008

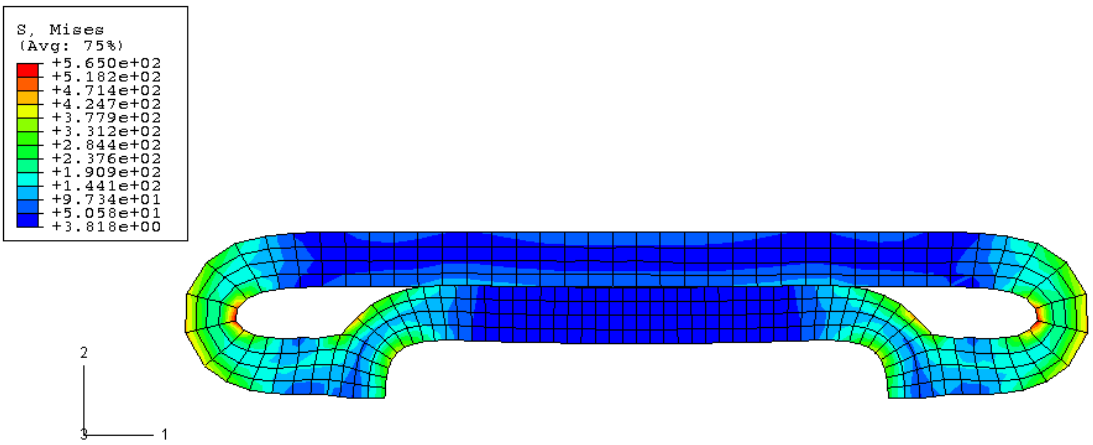


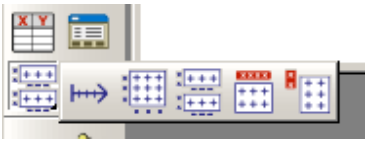
Figure W5–8. Mises contour plot

6. Contour the minimum and maximum principal nominal strains. Elastic strains can be very high for hyperelastic materials. Because of this, the linear elastic material model is not used because it is not appropriate for elastic strains greater than approximately 5%.
7. Display the reaction force history at the reference node of the top rigid surface: In the Results Tree, expand the **History Output** container underneath the output database file named **`w_seal.odb`** and double-click **Reaction force: RF2 PI: Part-1-1 Node 1307 in NSET TOPRP** to display the reaction force history at the reference node of the top rigid surface.
8. You will now create the CLD curve.
    a. In the **History Output** container, click mouse button 3 on **Reaction force: RF2 PI: Part-1-1 Node 1307 in NSET TOPRP** and select **Save As** from the menu that appears. Save the data as **`Force`**.
    b. Click mouse button 3 on **Spatial displacement: U2 PI: Part-1-1 Node 1307 in NSET TOPRP** and select **Save As** from the menu that appears. Save the data as **`Disp`**.
    c. In the Results Tree, double-click **XYData**. In the **Create XY Data** dialog box, select **Operate on XY data** as the source and click **Continue**.

    The **Operate on XY Data** dialog box appears.
    d. From the **Operators** listed in the **Operate on XY Data** dialog box, select **combine(X, X)** and then **abs(A)**. Note that the **abs(A)** operator is used to obtain the absolute values. In the **XY Data** field, double-click the curve **Disp**. The current expression reads **`combine(abs("Disp"))`**. Move the cursor before the far-right bracket, enter a comma, and then select the operator **abs(A)**. In the **XY Data** field, double-click the curve **Force**. The final expression reads **`combine(abs("Disp"), abs("Force") )`**. Click **Plot Expression** to plot this expression. Save this plot as **`CLD`**.

© Dassault Systèmes, 2008

9. Customize the plot as follows:
   a. Double-click anywhere on the chart to open the **Chart Options** dialog box.
      - In the **Grid Display** tabbed page, toggle on the major *X*- and *Y*-grid lines. Set the line color to blue and the line style to dashed.
      - Change the fill color using the following **RGB** values: red: 175; green: 250; blue: 185.
      - In the **Grid Area** tabbed page, select **Square** as the size and drag the slider to **80**. From the list of auto-alignments, choose the one that places the chart in the center of the viewport
   b. Double-click the legend to open the **Chart Legend Options** dialog box.
      - In the **Contents** tabbed page, click to increase the legend text font size to **10**.
      - In the **Area** tabbed page, toggle on **Inset**.
      - Toggle on **Fill** to flood the legend with a white background.
      - In the viewport, drag the legend over the chart.
   c. Double-click either axis to open the **Axis Options** dialog box.
      - In the **X Axis** region of the dialog box, select the displacement axis.
      - In the **Scale** tabbed page, place **4** major tick marks on the *X*-axis at (use the **By count** method).
      - In the **Title** tabbed page, change the *X*-axis title to **Displacement (inch)**.
      - In the **Y Axis** region of the dialog box, select the force axis.
      - In the **Scale** tabbed page, specify that the *Y*-axis should extend from **0** (the *Y*-axis minimum) to **250** (the *Y*-axis maximum).
      - Increase the number of *Y*-axis minor tick marks per increment to **4**.
      - In the **Title** tabbed page, change the *Y*-axis title to **Force (lbf)**.
      - In the **Axes** tabbed page, change the font size for both axes to **10**.
   d. Expand the list of plot option icons in the toolbox:



e. Examine the remaining options. Add the following plot title: **`CLD Diagram`**. Double-click the plot title to open the **Plot Title Options** dialog box.

- In the **Title** tabbed page, click to change the legend text style to bold.
- In the **Area** tabbed page, toggle on **Inset**.
- In the viewport, drag the plot title above the chart.

f. Click in the toolbox to open the **Curve Options** dialog box. Change the legend text to **`Top Surface Ref Point`** and toggle on **Show symbol**. Set the color for both the line and symbols to red. Use large filled circles for the symbols. Reposition the legend as necessary.

The final plot appears as shown in Figure W5–9.

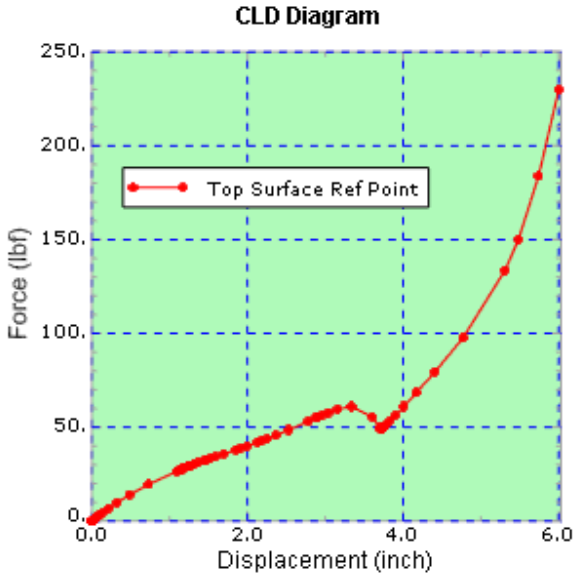


Figure W5–9. Compression load deflection diagram

Question W5-3: What does the inverted peak near 4 inches of deflection represent?

Note: A complete input file is available for your convenience. You may consult this file if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input file is named

`w_seal_complete.inp`

and is available using the Abaqus fetch utility.

Workshop 6

Dynamics

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Become familiar with the Abaqus/CAE procedures for frequency extraction, implicit dynamic, and explicit dynamic analyses.
- Become more familiar with monitoring job status.
- Learn how to plot eigenmodes and create history plots using Abaqus/CAE.
- Learn how to perform a "restart" analysis.

Introduction

In this workshop the dynamic response of the cantilever beam shown in Figure W6–1 is investigated. The beam is modeled using B21 beam elements. A frequency extraction is performed to determine the 10 lowest vibration modes of the beam. The problem is also solved in both Abaqus/Standard and Abaqus/Explicit by performing direct integration dynamic analyses to simulate the vibration of the beam upon removal of the tip load.

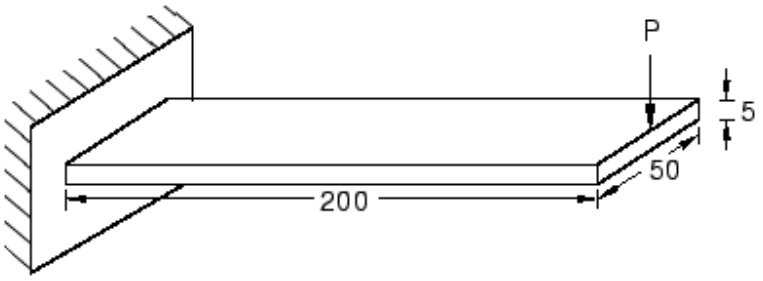


Figure W6–1. Problem description

© Dassault Systèmes, 2008

Preliminaries

1. Enter the working directory for this workshop
   `../abaqus_solvers/interactive/workshop6`
2. Run the script **`ws_solver_beam.py`** using the following command:
   `abaqus cae startup=ws_solver_beam.py`.

The above command creates an Abaqus/CAE database named **`Beam.cae`** in the current directory. The model named **`static`** includes the beam model for a static, general analysis. The beam element type B21 is used in the model. You will edit this model to perform the frequency extraction, implicit dynamic, restart, and explicit dynamic analyses.

Part 1: Frequency extraction analysis

You will first perform a frequency extraction analysis to determine the 10 lowest eigenmodes of the structure. Before continuing, make a copy of the model **`static`**.

1. Copy the model named **`static`** to a model named **`frequency`**. Make the following changes to the model **`frequency`**.

   Tip: To copy the model, click mouse button 3 on the model name in the Model Tree and select **Copy Model** in the menu that appears.
2. Note that the Model Tree can be simplified to display only the model that you are working on by clicking mouse button 3 on the model name and selecting **Set As Root** from the menu that appears. To return the Model Tree to the default view, click the **Set Root to Model Database** tool on the top of the Model Tree. If necessary, set this model to be the root of the tree.
3. Add a density of 2.3E−6 unit into the beam material definition named **`MATEA`**. In the Model Tree, expand the **Materials** container and double-click the material **MATEA**. In the material editor, select **General→Density** from the menu bar; enter the value **`2.3E-6`** for **Mass Density** in the **Density** field.
4. The frequency analysis procedure will be used instead of the general static one. Thus, replace the general static step **Displace** with a frequency step.
   a. In the Model Tree, expand the **Steps** container and click mouse button 3 on the step **Displace** and select **Replace** from the menu that appears.
   b. In the **Replace Step** dialog box, select **Linear perturbation** as the procedure type and **Frequency** from the list of available perturbation steps. Click **Continue**.
   c. In the step editor, accept the default **Lanczos** eigensolver and enter a value of **`10`** for **Number of eigenvalues requested**. Click **OK** to save the change and exit the step editor.
   d. Rename the step **`Frequency`**.
5. Create a job named **`frequency`** for the model **frequency**.

© Dassault Systèmes, 2008

Note: If necessary, return the Model Tree to the default view to display the **Jobs** container.

6. Save your model database, submit the job for analysis, and monitor the job's process.

Visualizing results

When the analysis is complete, use the following procedure to view the eigenmodes and eigenvalues from the frequency analysis in the Visualization module:

1. In the Model Tree, click mouse button 3 on the job **frequency** and select **Results** from the menu that appears to open the file **`frequency.odb`** in the Visualization module.
2. Plot the first eigenmode (plot the deformed model shape and use the **Frame Selector** or the **Step/Frame** dialog box to choose the frame corresponding to Mode 1).
3. Click the **Common Plot Options** tool in the toolbox.

   The **Common Plot Options** dialog box opens.
4. From the **Labels** page, toggle on the node labels (numbers) and node symbols and click **Apply**.
5. Using the arrow keys in the context bar, select different mode shapes. Note that the node symbols help you visualize the extensional modes.

Question W6–1: Are there modes of the physical system that cannot be captured by your model because of limitations in element type or mesh? (Remember that the elements are planar and the mesh is somewhat coarse.)

Question W6–2: Do any of the mode shapes for your model look non-physical?

Part 2: Implicit dynamic analysis

You will now investigate the free vibration of the beam upon removal of the tip load, using an implicit dynamic procedure (Abaqus/Standard).

1. Copy the model named **`static`** to a model named **`implicit`**. Make the following changes to the model **`implicit`**. If necessary, set this model to be the root of the tree.
2. Use the procedure described earlier to add a density of 2.3E−6 units to the material **MATEA**.
3. Add an additional dynamic, implicit step after the static, general step **Displace**.
   a. In the Model Tree, double-click the **Steps** container.

b. In the **Create Step** dialog box, name the step **`Release`**, select **Dynamic, Implicit** from the list of available **General** procedure types, and click **Continue**.

c. In the **Edit Step** dialog box, accept the default step time **`1`**. In the **Incrementation** tabbed page, choose **Fixed** incrementation, enter a value of **`10`** for the **Maximum number of increments** and **`0.01`** for the **Increment size**. Click **OK** to save the data and exit the step editor.

4. Deactivate the load in the step **`Release`**. In the Model Tree, expand the branch of the load **DisplaceTip** underneath the **Loads** container, as shown in Figure W6–2a. Click mouse button 3 on **Release (propagated)** under the **States** sub-container and select **Deactivate** from the menu that appears. Note that **Release (propagated)** is changed into **Release (Inactive)**, as shown in Figure W6–2b, to indicate that the load is deactivated in this step.

(a) (b)

Figure W6–2. Loads container in the Model Tree

5. Create a geometry-based set containing the vertex at the loaded end of the beam. This set will be used to request history output and monitor the progress of the analysis.

a. In the Model Tree, expand the **Assembly** container and double-click **Sets** underneath it.

b. In the **Create Set** dialog box, name the set **`TIP`** and click **Continue**.

c. Select the vertex at the right end of the beam as indicated in Figure W6–3.

d. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selection.

Figure W6–3. Node set

© Dassault Systèmes, 2008

6. Add a history output request to write the displacement history for the set **TIP** to the output database file.
   a. In the Model Tree, double-click the **History Output Requests** container. In the **Create History** dialog box, select the step **Displace** and click **Continue**.
   b. In the **Edit History Output Request** dialog box, select the domain **Set** and the set **TIP**.
   c. Expand the **Displacement/Velocity/Acceleration** branch in the **Output Variables** field and toggle on **U, Translations and rotations**.
   d. Click **OK** to exit the history output editor.
7. It is useful to be able to monitor the progress of an analysis by tracking the value of one degree of freedom.
   a. From the main menu bar of the Step module, select **Output→DOF Monitor** to open the **DOF Monitor** dialog box.
   b. Activate the stippled entries by toggling on **Monitor a degree of freedom throughout the analysis**.
   c. Click **Edit** to select the set **TIP** as the **Region**.

      Tip: Click **Points** in the prompt area to select the set **TIP** from the **Region Selection** dialog box.
   d. Enter **2** as the **Degree of freedom**.
   e. Click **OK** to exit the **DOF Monitor** dialog box.
8. A restart file should be saved so that the data for the last increment of each step are available for continuing ("restarting") the analysis from that point without having to re-run the first part of the analysis.
   a. From the main menu bar of the Step module, select **Output→Restart Requests**.
   b. In the **Edit Restart Requests** dialog box, specify a restart frequency of **1** and toggle on **Overlay** for each step, as shown in Figure W6–4.

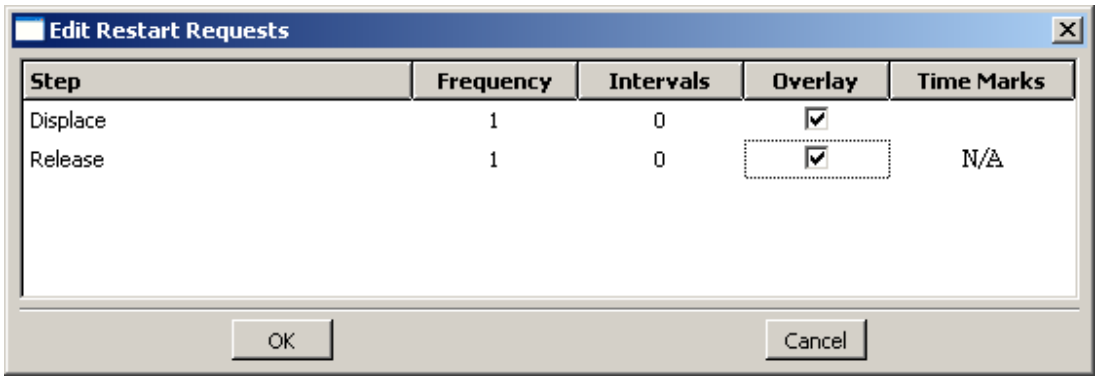


Figure W6–4. **Edit Restart Requests** dialog box

9. Create a job named **implicit** for the model **implicit**.

© Dassault Systèmes, 2008

10. Save your model database, submit the job for analysis, and monitor the job's process.

    You will see that the job will exit with an error message in the **implicit Monitor** dialog box. This message may refer you in turn to the message file. In this case the message is simply that the analysis did not go to the end of the step because the number of allowable increments is only 10.

    Note that the message file also contains information concerning the solution progress.

Visualizing results

1. Open the file **`implicit.odb`** in the Visualization module.
2. Plot the history of the displacement component U2 at the tip node. In the Results Tree, expand the **History Output** container underneath the output database named **`implicit.odb`** and double-click **Spatial displacement: U2 at Node xxx in NSET TIP**.

   You will see that less than one full period of oscillation is plotted.

Part 3: Restart analysis

The remainder of the implicit dynamics analysis will now be completed using the restart analysis capability.

1. Copy the model named **`implicit`** to a model named **`implicit-restart`**. Make the following changes to the model **`implicit-restart`**. If necessary, set this model to be the root of the tree.
2. Create an additional dynamic, implicit static step named **`release2`** for the restart analysis. Because the previous analysis terminated after 10 increments and ran for only 0.1 seconds, set the step time period to **`0.9`**. Use the fixed incrementation with an initial increment size of **`0.01`** and maximum increment number of **`200`**.
3. Click mouse button 3 on the model **implicit-restart** and select **Edit Attributes** from the menu that appears. In the **Edit Model Attributes** dialog box, specify the values shown in Figure W6–5.

   Note that, if you did not select the option **and terminate the step at this point**, Abaqus would attempt to complete the previous step instead of continuing directly to the new step.

© Dassault Systèmes, 2008

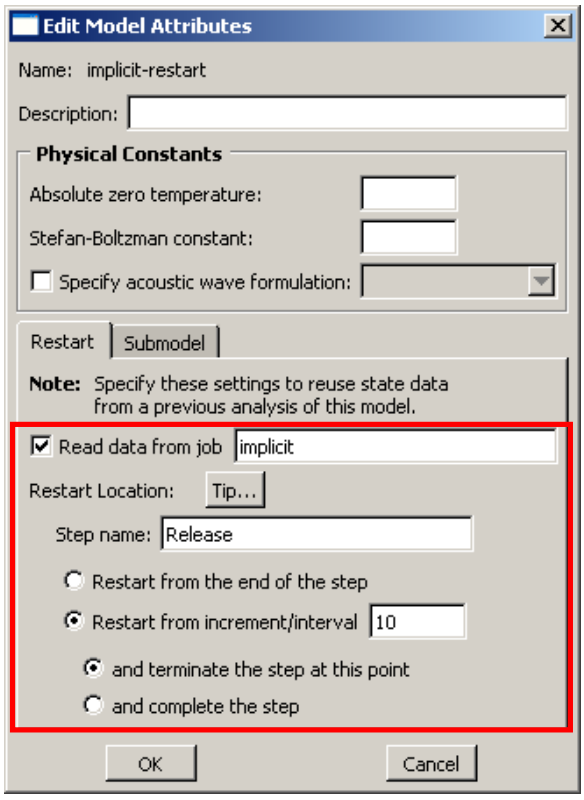


Figure W6–5. **Edit Model Attributes** dialog box

4. Create a job named **`restart`** for the model **implicit-restart**. In the **Edit Job** dialog box, set the job type to **Restart**.
5. Save your model database, submit the job for analysis, and monitor the job's process.

Visualizing results

1. Open the file **`restart.odb`** in the Visualization module.
2. Using the procedure described earlier, plot the history of the displacement component U2 at the node in the node set **`TIP`**.

   Notice that the plot traces the displacement history of the node in the restart analysis. Since the restart analysis is a continuation of an earlier job, it is often useful to view results from the entire (original and restarted) analysis. Thus, to create a plot of displacement history from these two analyses, follow the steps outlined below:

   a. Save the above history plot. In the Results Tree, click mouse button 3 on **Spatial displacement: U2 at Node xxx in NSET TIP** in the **History Output** container underneath the output database named **`restart.odb`** and select **Save As** from the menu that appears. Name the plot **RESTART**.

© Dassault Systèmes, 2008

b. Save the displacement history at the same node obtained in the previous analysis. In the Results Tree, expand the **History Output** container underneath the output database named **`implicit.odb`** and click mouse button 3 on **Spatial displacement: U2 at Node xxx in NSET TIP**. Select **Save As** from the menu that appears and name the plot **`ORIGINAL`**.

c. View these two plots in a single *X*–*Y* plot. In the Results Tree, expand the **XYData** container, select plots **`RESART`** and **`ORIGINAL`** and click mouse button 3. From the menu that appears, select **Plot** to plot the U2 history for the entire simulation.

Tip: Use **[Ctrl]+click** for multiple selections.

3. From the graph, find the period and calculate the frequency of the vibration.

Question W6–3: How does this compare with the frequency calculated in the eigenvalue analysis?

4. Animate the deformed shape.

a. Switch back to the output database file **`restart.odb`**.

b. Click the **Plot Deformed Shape** tool in the toolbox.

c. Click the **Animate: Time History** tool to animate the deformed shape.

d. Click **Animation Options** to modify the speed of the animation.

Note: Before continuing with this workshop, complete Lecture 8, *Using Abaqus/Explicit*.

Part 4: Explicit dynamic analysis

You will now investigate the vibration of the beam using an explicit dynamics procedure (Abaqus/Explicit) and compare the results to those obtained with an implicit dynamics procedure (Abaqus/Standard).

1. Copy the model named **`implicit`** to a model named **`explicit`**. Make the following changes to the model **`explicit`**. If necessary, set this model to be the root of the tree.
2. Delete the second analysis step (**Release**).
3. Replace the remaining static step with a dynamic, explicit step with a total step time of 1.0. Turn **NIgeom** off.

Question W6–4: Why is **NIgeom** turned off in this analysis?

4. Create an amplitude curve.

a. In the Model Tree, double-click the **Amplitudes** container.

b. In the **Create Amplitude** dialog box, name the amplitude curve **Ramp** and accept **Tabular** for the type. Click **Continue**.

c. In the **Edit Amplitude** dialog box, enter the values as shown in Figure W6–6.

d. Click **OK** to exit the amplitude editor.

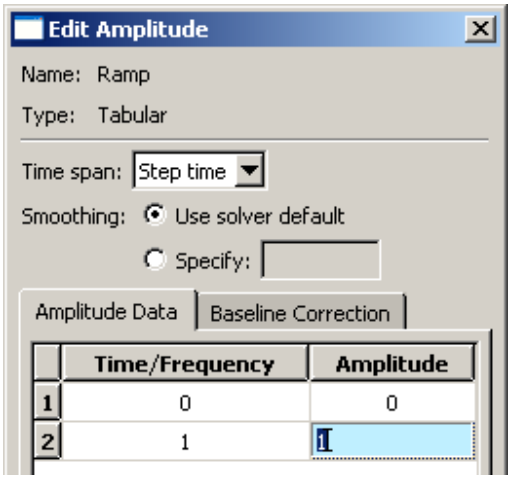


Figure W6–6. **Edit Amplitude** dialog box

5. Add an amplitude reference to the predefined load **DisplaceTip**.

   a. In the Model Tree, expand the branch of **States** underneath the load **DisplaceTip** in the **Loads** container and double-click **Displace (Created)**.

   b. In the load editor, choose the amplitude **Ramp** for **Amplitude**. Click **OK** to save the change and exit the load editor.

   This amplitude reference will linearly ramp the load from zero to the specified value over the course of the step. By loading the structure slowly enough, we can simulate a quasi-static response. Simulating quasi-static processes with explicit dynamic methods is discussed in more detail in Lecture 10.

6. Create a second dynamic, explicit step named **Release** with a total step time of 1.0 unit and **Nlgeom** turned off.

7. Edit the displacement history output request to that the data are written every **0.005** units of time.

8. Deactivate the load **DisplaceTip** in the step **Release**.

9. Create a job named **explicit** for the model **explicit**.

10. Save your model database, submit the job for analysis and monitor the job's process.

© Dassault Systèmes, 2008

Visualizing results

1. Open the file **`explicit.odb`** in the Visualization module.
2. Plot the history of U2 at the tip node.
3. Compare the results obtained with Abaqus/Standard against the results obtained with Abaqus/Explicit by plotting both results on the same plot. Your results should look similar to those shown in Figure W6–7.

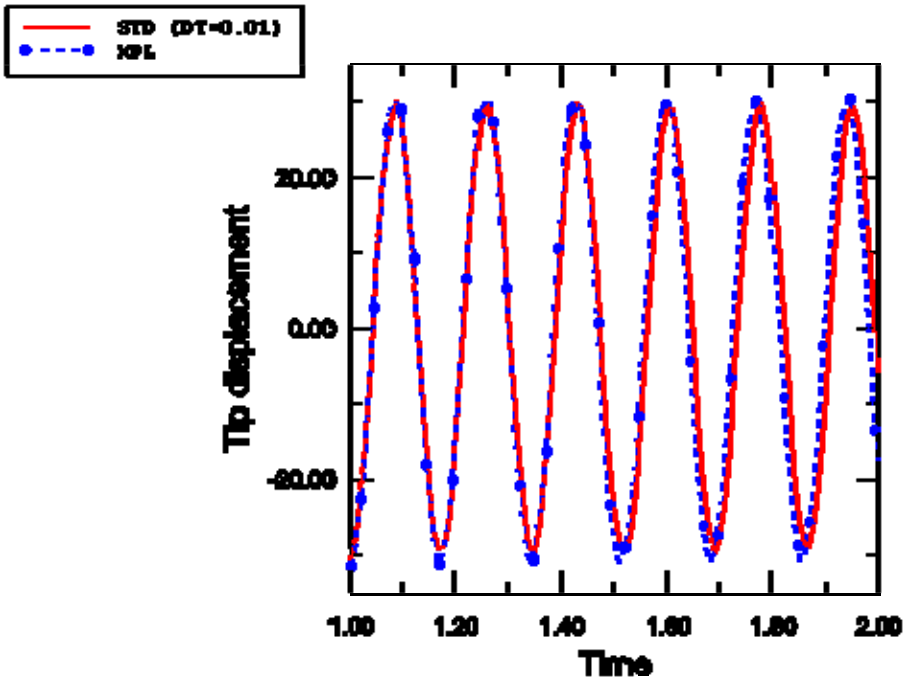


Figure W6–7. Comparison of the tip node displacement history

Question W6–5: How do the results compare with one another? What factors contribute to the discrepancies in the solutions?

4. Compare the internal and kinetic energy content of the model by plotting these quantities on the same plot. Do this only for the Abaqus/Explicit results.

Question W6–6: Was the loading rate small enough to ensure a quasi-static response in the first step?

Note: A script that creates the complete model described in these instructions is available for your convenience. Run this script if you encounter difficulties following the instructions outlined here or if you wish to check your work. The script is named ws_solver_beam_answer.py and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 6

Dynamics

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Become familiar with the input and output for frequency extraction, implicit dynamic, and explicit dynamic analyses.
- Become more familiar with the status (`.sta`) and message (`.msg`) files.
- Learn how to plot eigenmodes and create history plots using Abaqus/Viewer.
- Learn how to perform a "restart" analysis.

Introduction

In this workshop the dynamic response of the cantilever beam shown in Figure W6–1 is investigated. The beam is modeled using B21 beam elements. A frequency extraction is performed to determine the 10 lowest vibration modes of the beam. The problem is also solved in both Abaqus/Standard and Abaqus/Explicit by performing direct integration dynamic analyses to simulate the vibration of the beam upon removal of the tip load.

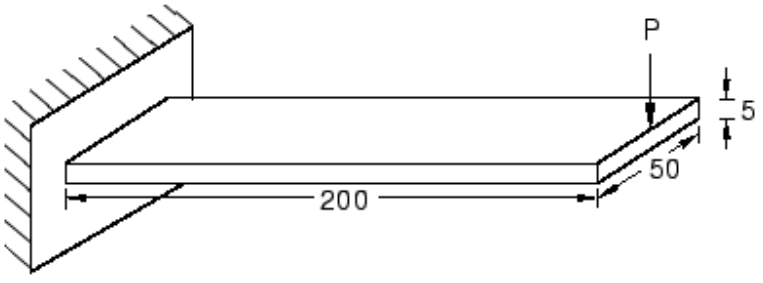


Figure W6–1. Problem description

© Dassault Systèmes, 2008

Frequency extraction analysis

Change to the **../abaqus_solvers/keywords/workshop6** directory, and copy **w_beam.inp** to a new file named **w_beam_freq.inp**.

Input specification

1. Make the following changes to **w_beam_freq.inp**. Refer to the online documentation as necessary.
   a. Include a density of 2.3E−6 in the material definition. Add the following option block below the ∗MATERIAL option:

      ```
      *DENSITY
      2.3E-6,
      ```

   b. Change the procedure type from ∗STATIC to ∗FREQUENCY, and select the Lanczos eigensolver. Request 10 modes. The finished option block should look like the following:

      ```
      *FREQUENCY, EIGENSOLVER=LANCZOS
      10,
      ```

   c. Remove the loading, but retain the boundary conditions (one end built-in).
   d. Limit the printed node output to the data (.**dat**) file by restricting output to a node set:
      i. Create a node set called **TIP** that contains the node at the loaded end of the beam.

      ```
      *NSET, NSET=TIP
      11,
      ```

      ii. Add the following output requests within the first step:

      ```
      *NODE PRINT, NSET=TIP
      U,
      *OUTPUT, FIELD, FREQUENCY=999
      *NODE OUTPUT
      U,
      ```

2. Submit the frequency extraction analysis as an Abaqus job.
3. After the analysis has completed, check the printed output file and make any necessary corrections to the input.

Examining the eigenmodes and eigenvalues

1. Open the printed output file in an editor of your choice.
2. Search for the second occurrence of "**E I G E N**" to find the beginning of the analysis results. The first table gives the eigenvalue output. Find the frequency (cycles/time) for the lowest mode.

© Dassault Systèmes, 2008

3. Visualize results:
   a. Start Abaqus/Viewer, and open the output database associated with this analysis.
   b. Plot the first eigenmode (plot the deformed model shape and use the **Frame Selector** or the **Step/Frame** dialog box to choose the frame corresponding to Mode 1).
   c. Click the **Common Plot Options** tool in the toolbox.
      The **Common Plot Options** dialog box opens.
   d. From the **Labels** page, toggle on the node labels (numbers) and node symbols and click **Apply**.
   e. Using the arrow keys in the context bar, select different mode shapes. Note that the node symbols help you visualize the extensional modes.

Question W6–1: Are there modes of the physical system that cannot be captured by your model because of limitations in element type or mesh? (Remember that the elements are planar and the mesh is somewhat coarse.)

Question W6–2: Do any of the mode shapes for your model look nonphysical?

Implicit dynamic analysis

We now investigate the free vibration of the beam upon removal of the tip load.

Input specification

1. Copy **`w_beam.inp`** to a new file named **`w_beam_dynam.inp`**.

   Use the following steps to modify the file so that the tip of the model is loaded and then released and allowed to vibrate freely:

   a. Include a density of 2.3E–6 in the material definition.
   b. Add a second step to the analysis history with a ∗DYNAMIC procedure type. Give a suggested time increment of 0.01 and a time period of 1.0. Specifiy 10 as the maximum number of increments using the INC parameter on the ∗STEP option.
   c. Remove the tip load in the dynamic step by specifying ∗CLOAD, OP=NEW. This option removes all existing concentrated loads.
   d. Request that the tip displacement be written to the output database (**`.odb`**) file. As before, use the ∗NSET option to define a node set called **`TIP`** containing the node at the loaded end of the beam. Add the following output requests to the input file:

```
*OUTPUT, HISTORY, FREQUENCY=1
*NODE OUTPUT, NSET=TIP
U,
```

© Dassault Systèmes, 2008

e. Request that the displacements for all the nodes in the model be written as field data to the output database (**.odb**) file.

```
*OUTPUT, FIELD, FREQUENCY=1
*NODE OUTPUT
U,
```

f. It is useful to be able to monitor the progress of an analysis by noting the value of one degree of freedom. To do so, add the following option to the first analysis step:

```
*MONITOR, NODE=11, DOF=2
```

g. A restart file should be saved so that the data for the last increment of each step are available for continuing ("restarting") the analysis from that point without having to rerun the first part of the analysis. To obtain the data, add the OVERLAY parameter to the *RESTART option.

2. Submit the new analysis as an Abaqus job.

   While the job is running, you can check on the progress of the analysis by looking at the status file.

Checking the output files and visualizing results

1. Check the bottom of the printed output file to see if there are warning or error messages. These messages may refer you in turn to the message file. In this case the message is simply that the analysis did not go to the end of the step because the number of allowable increments is only 10.
2. Note that the message file also contains information concerning the solution progress.
3. Open **w_beam_dynam.odb** in Abaqus/Viewer.
4. Create a history plot of the displacement of the tip node. You will see that less than one full period of oscillation is plotted.

   Tip: In the Results Tree, expand the **History Output** container underneath the output database named **w_beam_dynam.odb**. Double-click the curve corresponding to the variable U2 for the tip node.

Restart analysis

In this section you will "restart" the analysis to continue it without starting over from the beginning.

1. Create a new file called **w_beam_dynamres.inp** in the text editor, and add the *HEADING option to the top of the file.
2. The *RESTART option is the only other option needed in the model data of the input file. Add the following option after *HEADING:

   ```
   *RESTART, READ, STEP=2, INC=10, END STEP, WRITE, OVERLAY
   ```

   This option specifies that the analysis is continued from Step 2, increment 10. The END STEP parameter terminates the current step at this point. If this parameter is

omitted, the 10 increment limit for the step will again prevent the analysis from continuing. The WRITE parameter writes restart output at the default frequency of 1 (every increment). The OVERLAY parameter causes restart data from each increment to overwrite data from the previous increment so that only the last increment is retained.

3. Specify the history data:
   a. Begin the history data with the *STEP option. Set the INC parameter to 200 to ensure that the analysis will not stop before the end of the step due to the increment limit.
   b. Specify a *DYNAMIC procedure with the time period set to the additional time (from the end of the second step) required for the analysis to complete a total time of 1.0.

      No loading is necessary since this is a continuation of the previous free vibration analysis.
4. Use the following command to submit this job:

   `abaqus job=w_beam_dynamres oldjob=w_beam_dynam`

Results visualization with Abaqus/Viewer

1. Open **`w_beam_dynamres.odb`** in Abaqus/Viewer.
2. Plot the displacement history of U2 at node 11.

   Notice that the plot traces the displacement history of the node in the restart analysis. Since the restart analysis is a continuation of an earlier job, it is often useful to view results from the entire (original and restarted) analysis. Thus, to create a plot of displacement history from these two analyses, follow the steps outlined below:
   a. Save the current plot as **`RESTART`**.

      Tip: Click mouse button 3 on the curve in the Results Tree and select **Save As** from the menu that appears.
   b. Open **`w_beam_dynam.odb`** in Abaqus/Viewer. Save the plot of U2 at node 11. Name this plot **`ORIGINAL`**.
   c. In the Results Tree, expand the **XYData** container.

      Notice that the curves **`ORIGINAL`** and **`RESTART`** are listed underneath.
   d. Select **`ORIGINAL`** and **`RESTART`** and click mouse button 3. From the menu that appears, select **Plot** to create the plot of displacement U2 at node 11 for the entire simulation.
3. From the graph, find the period and calculate the frequency of the vibration.

Question W6–3: How does this compare with the frequency calculated in the eigenvalue analysis?

© Dassault Systèmes, 2008

4. Animate the results.
   a. Switch back to the output database file **`w_beam_dynamres.odb`**.
   b. Click the **Plot Deformed Shape** tool in the toolbox.
   c. Click the **Animate: Time History** tool to animate the deformed shape.
   d. Click **Animation Options** to modify the speed of the animation.

Note: Before continuing with this workshop, complete Lecture 8, *Using Abaqus/Explicit.*

Explicit dynamic analysis

We now investigate the vibration of the beam using an explicit dynamic procedure (Abaqus/Explicit) and compare the results to those obtained with an implicit dynamic procedure (Abaqus/Standard).

Input specification

1. Copy **`w_beam.inp`** to a new file named **`w_beam_xdynam.inp`**. Use the following steps to modify the file so that the tip of the model is loaded and then released and allowed to vibrate freely:
   a. Include a density of 2.3E–6 in the material definition.
   b. Define an amplitude function named RAMP by adding the following to the input file:

      ```
      *AMPLITUDE, NAME=RAMP
      0.0, 0.0, 1.0, 1.0
      ```

   c. Modify the step definition by setting the parameter NLGEOM=NO on the ∗STEP option, and change the procedure option for the step to ∗DYNAMIC, EXPLICIT. Specify a total step time of 1.0 s, as shown below:

      ```
      *DYNAMIC, EXPLICIT
      ,1.0
      ```

   Question W6–4: Why is the NLGEOM parameter set to NO in this analysis?

   d. Add the AMPLITUDE parameter to the ∗CLOAD option, and set it equal to RAMP.

      This will linearly ramp the load from zero to the specified value over the course of the step. By loading the structure slowly enough, we can simulate a quasi-static response. Simulating quasi-static processes with explicit dynamic methods is discussed in more detail in Lecture 10.

© Dassault Systèmes, 2008

e. Request that the tip displacement be written to the output database file. As before, use the ∗NSET option to define a node set called **TIP** containing the node at the loaded end of the beam. In addition, write the whole model internal and kinetic energies to these files. The following output requests should be added to the input file:

```
*OUTPUT, HISTORY, TIME INTERVAL=0.005
*NODE OUTPUT, NSET=TIP
U,
*ENERGY OUTPUT
ALLIE, ALLKE
```

f. Request that the displacements for all the nodes in the model be written as field data to the output database (**.odb**) file.

```
*OUTPUT, FIELD
*NODE OUTPUT
U,
```

g. Add a second analysis step with a ∗DYNAMIC, EXPLICIT procedure type. Set NLGEOM=NO on the ∗STEP option, and specify a total step time of 1.0.

h. Remove the load by specifying a ∗CLOAD option with the OP=NEW parameter:

```
*CLOAD, OP=NEW
```

i. A restart file should be saved so that the data for the last increment of each step are available for continuing (“restarting”) the analysis from that point without having to rerun the first part of the analysis. To obtain the data, add the OVERLAY parameter to the ∗RESTART option

2. Submit the new analysis as an Abaqus job.

   You can check on the progress of the analysis by looking at the status file if the job is running in background mode. If the job is running interactively, the contents of the status file are printed to the screen.

Checking the output files and postprocessing

1. Check the bottom of the printed output file to see if there are warning or error messages. These messages may refer you to the status file.

   Note that the status file also contains information concerning the progress of the calculations, provided the job was run in background mode.

2. Open **w_beam_xdynam.odb** in Abaqus/Viewer.
3. Plot the tip node’s displacement history, and compare the results obtained with Abaqus/Standard against the results obtained with Abaqus/Explicit by plotting both results on the same plot. Your results should look similar to those shown in Figure W6–2.

© Dassault Systèmes, 2008

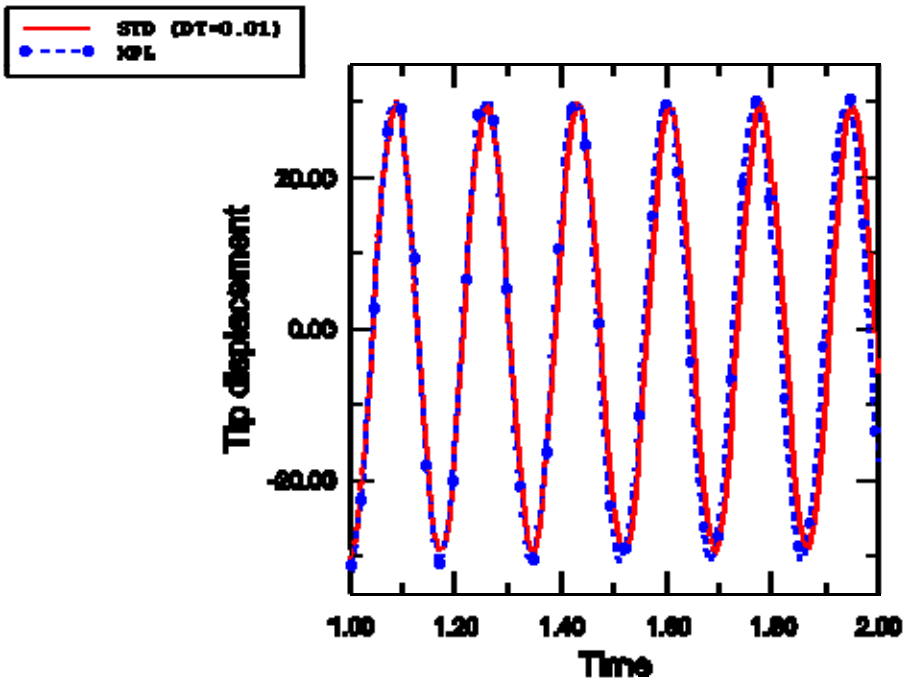


Figure W6–2. Comparison of the tip node displacement history

Question W6–5: How do the results compare with one another? What factors contribute to the discrepancies in the solutions?

4. Compare the internal and kinetic energy content of the model by plotting these quantities on the same plot. Do this only for the Abaqus/Explicit results.

Question W6–6: Was the loading rate small enough to ensure a quasi-static response in the first step?

Note: Complete input files are available for your convenience. You may consult these files if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input files are named

`w_beam_freq_complete.inp`

`w_beam_dynam_complete.inp`

`w_beam_dynamres_complete.inp`

`w_beam_xdynam_complete.inp`

and are available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 7

Contact with Abaqus/Explicit

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define a rigid body constraint.
- Define a general contact interaction.
- Apply boundary and initial conditions.
- Perform an impact analysis.
- Use Abaqus/Viewer to view results.

Introduction

This workshop involves the simulation of a pipe-on-pipe impact resulting from the rupture of a high-pressure line in a power plant. It is assumed that a sudden release of fluid could cause one segment of the pipe to rotate about its support and strike a neighboring pipe. The goal of the analysis is to determine strain and stress conditions in both pipes and their deformed shapes. The simulation will be performed using Abaqus/Explicit.

© Dassault Systèmes, 2008

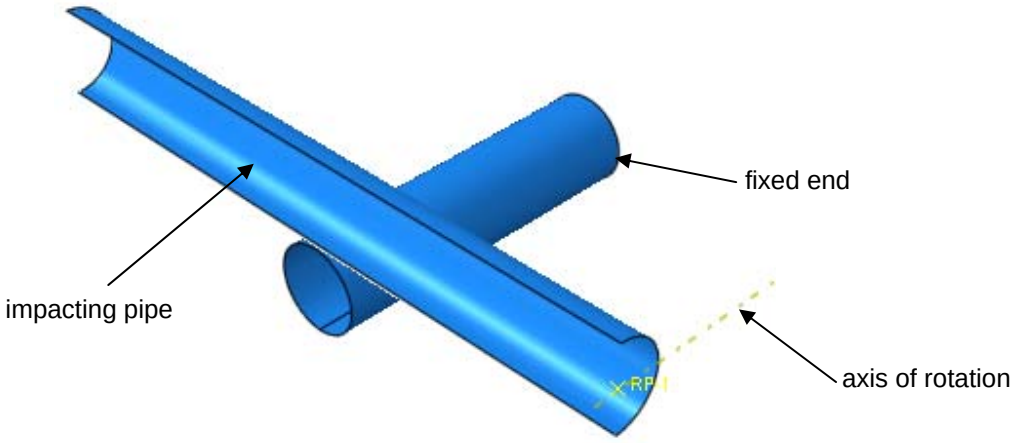


Figure W7–1. Pipe whip model assembly

Both pipes have a mean diameter of 6.5 inches with a 0.432 inch wall thickness and a span of 50 inches between supports. The fixed pipe is assumed to be fully restrained at both ends, while the impacting pipe is allowed to rotate about a fixed pivot located at one of its ends, with the other end free. We exploit the symmetry of the structure and the loading and thus model only the geometry on one side of the central symmetry plane, as shown in Figure W7–1.

Preliminaries

1. Enter the working directory for this workshop:
   `../abaqus_solvers/interactive/workshop7`
2. Run the script **`ws_solver_pipe_whip.py`** using the following command:
   `abaqus cae startup=ws_solver_pipe_whip.py`

The above command creates an Abaqus/CAE database named **`pipeWhip.cae`** in the current directory. A model named **`contact`** consists of the geometry and mesh definitions for the pipes. You will add necessary data to complete the model for the impact analysis.

Defining material and section properties

Both pipes are made of steel. A Von Mises elastic, perfectly plastic material model is used, with a yield stress of 45,000 psi.

1. In the Model Tree, double-click **Materials** to create a material named **`Steel`** with the following properties:

Modulus of elasticity:	30.0E6 psi
Poisson's ratio:	0.3
Yield Stress:	45.0E3 psi
Density:	7.324E–4 lb-sec^2/in^4

© Dassault Systèmes, 2008

Question W7–1: Why is density required in the material model definition? Can you comment on the units of density used in this problem?

2. In the Model Tree, double-click **Sections** to create a homogeneous shell section named `PipeSection`. In the **Basic** tabbed page of the **Edit Section** dialog box, select **During analysis** for the section integration, specify a shell thickness of `0.432` in, select the **Gauss** thickness integration rule, and view and accept all other default settings. Click **OK** to exit the section editor.
3. In the Model Tree, expand the branch of each part underneath the **Parts** container and double-click **Section Assignments** to assign this shell section to both parts.

Question W7–2: Why are only three integration points used through the thickness?

Defining rigid body constraint

You will define a rigid body constraint between the nodes at the pivot end of the impacting pipe and the reference point, as shown in Figure W7–2.

1. In the Model Tree, double-click **Constraints**.
2. In the **Create Constraint** dialog box, select **Rigid body** as the constraint type and click **Continue**.
3. In the **Edit Constraint** dialog box, select the region type **Tie (nodes)** and click **Edit** in the right side of the dialog box.
4. Select the edge(s) shown in Figure W7–2 as the tie region for the rigid body.
5. Similarly, select the reference point **RP-1** in the viewport as the rigid body reference point.
6. In the **Edit Constraint** dialog box, click **OK** to apply the constraint.

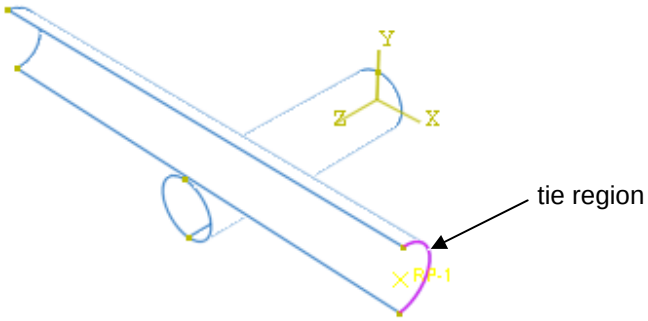


Figure W7–2. Rigid body constraint

Question W7–3: In order to drive both the translations and rotations of the pipe edge nodes, what type of node set needs to be used in the rigid body constraint?

© Dassault Systèmes, 2008

Defining step and output requests

Because of the high-speed nature of the event, the simulation is performed using a single explicit dynamics step.

1. In the Model Tree, double-click **Steps** to create a **Dynamic, Explicit** step with a time period of `0.015` seconds. Accept all defaults for the time incrementation and other parameters.
2. In the Model Tree, expand the **Field Output Requests** container and double-click **F-Output-1**. In the **Edit Field Output Requests** dialog box, review the preselected field output variables. Change the frequency at which the output is written to `12` evenly spaced time intervals.
3. In the Model Tree, double-click **History Output Requests** to create a history output request for reaction forces at the constrained end of the fixed pipe. In the **Edit History Output Request** dialog box:
    a. Select **Set** in the **Domain** field and select `RefPt` from the **Set** drop down list. Note that the set `RefPt` contains the reference point.
    b. Request history output at `100` evenly spaced time intervals during the analysis.
    c. From the list of available output variables, click the arrow next to **Forces/Reactions** and toggle on **RF, Reaction forces and moments** from the list that appears.
    d. Click **OK**.

Defining contact interaction

1. In the Model Tree, double-click **Interaction Properties**.
2. In the **Create Interaction Property** dialog box, accept **Contact** as the interaction type and click **Continue**.
3. In the **Edit Contact Property** dialog box, select **Mechanical→Tangential Behavior** and choose the **Penalty** friction formulation. Specify a friction coefficient of `0.2`. Click **OK** to close the dialog box.
4. In the Model Tree, double-click **Interactions**.
5. In the **Create Interaction** dialog box, accept **Step-1** as the step in which the interaction will be created and **General contact (Explicit)** as the interaction type. Click **Continue**.
6. In the **Edit Interaction** dialog box, accept the **All* with self** contact domain. Click **OK** to close the dialog box.

Question W7–4: Could you define contact in the same way if this problem were being solved in Abaqus/Standard?

© Dassault Systèmes, 2008

Defining Initial conditions

The impacting pipe is given an initial angular velocity of 75 radians/sec about its supported (pinned) end.

Question W7–5: How can you use the coordinates of the reference point to define the axis of rotation?

1. Perform a **Point/Node** query ( ) to determine the coordinates of two end points on the axis of rotation at the pivot end of impacting pipe, as shown in Figure W7–3.

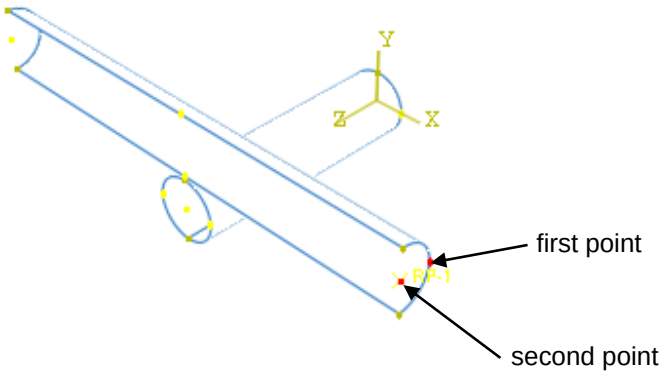


Figure W7–3. Points on axis of rotation

The coordinates will be printed out to the message area as shown in Figure W7–4.

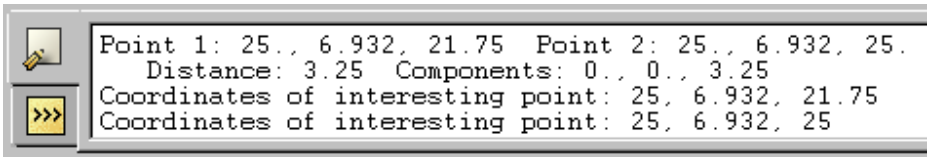


Figure W7–4. Point coordinates

2. In the Model Tree, double-click **Predefined Fields**.
3. In the **Create Predefined Field** dialog box, select the **Initial** step, the **Mechanical** category, and the **Velocity** type. Click **Continue** to proceed.
4. Select the impacting pipe as the region to which the initial velocity will be assigned, and click **Done**.
5. In the **Edit Predefined Field** dialog box, change the field definition to **Rotational only**. Enter a value of `75` for the **Angular velocity**. Use the coordinates of the first point indicated in Figure W7–3 to define **Axis point 1** and the coordinates of

© Dassault Systèmes, 2008

the second point indicated in Figure W7–3 to define **Axis point 2**. The initial velocity is displayed on the appropriate nodes.

Tip: Copy and paste the coordinates from the message area into the dialog box.

Question W7–6: What keyword was added to the input file when you created the angular velocity field? Search the Abaqus Keywords Reference Manual and read the documentation on this keyword.

Hint: You can see how Abaqus/CAE creates the input file for a given model by selecting **Model→Edit Keywords** from the main menu bar and viewing its contents. In order to find what keyword was added in a given step, check the keyword editor before and after the step in Abaqus/CAE and note the changes.

Defining boundary conditions

The edges located on the symmetry plane must be given appropriate symmetry boundary conditions. One end of the impacting pipe and both ends of the fixed pipe are fully restrained.

1. In the Model Tree, double-click **BCs**.
2. In the **Create Boundary Condition** dialog box, accept **Symmetry/ Antisymmetry/Encastre** as the boundary condition type and click **Continue** to create the boundary conditions shown in Figure W7–5.
   - Symmetry boundary conditions: Select the edges shown in Figure W7–5; and in the **Edit Boundary Condition** dialog box, choose the **ZSYMM (U3=UR1=UR2=0)** boundary condition.
   - Fully constrained boundary conditions: Select the edge shown in Figure W7–5; and in the **Edit Boundary Condition** dialog box, choose the **ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0)** boundary condition.
   - Pinned Boundary condition: Select **RP-1** in the viewport. In the **Edit Boundary Condition** dialog box, choose the **PINNED (U1=U2=U3=0)** boundary condition.

© Dassault Systèmes, 2008

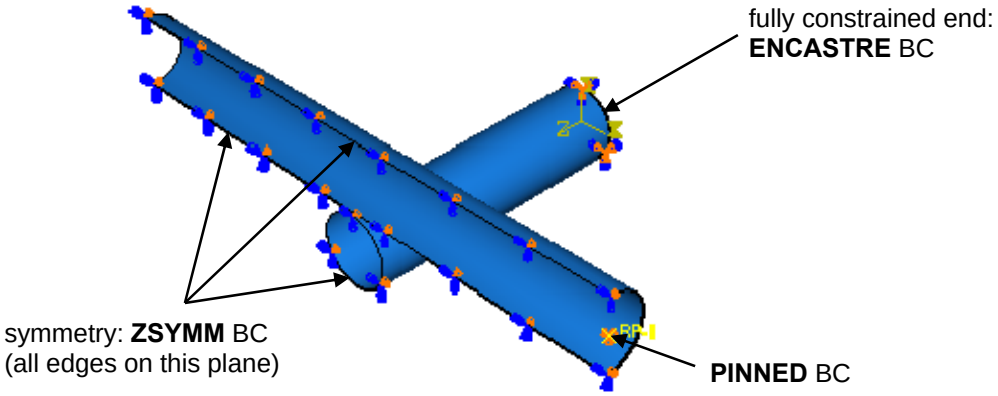


Figure W7–5. Boundary conditions

Question W7–7: Would the results of this analysis differ if both halves of the pipe were modeled instead of using symmetry boundary conditions?

Running the job and postprocessing the results

1. Save your model database file.
2. A named **`pipe-whip`** has been already been created for you. Submit the job for analysis, and monitor its progress.
3. When the analysis has completed, open the output database file **`pipe-whip.odb`** in the Visualization module.
4. Plot the undeformed and the deformed model shapes. Use the **Color Code Dialog** tool to customize the color for each instance, as shown in Figure W7–6.

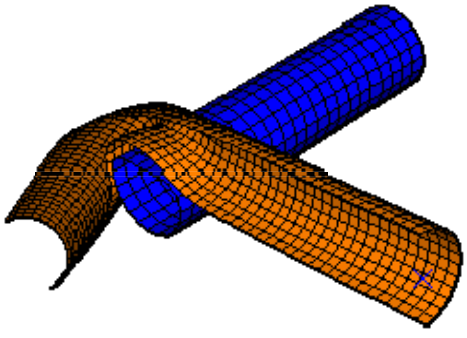


Figure W7–6. Deformed model shape

© Dassault Systèmes, 2008

5. Use the **Animate: Time History** tool to animate the deformation history.
6. Contour the Mises stress and equivalent plastic strain (PEEQ) on the deformed shape, as shown in Figure W7–7.

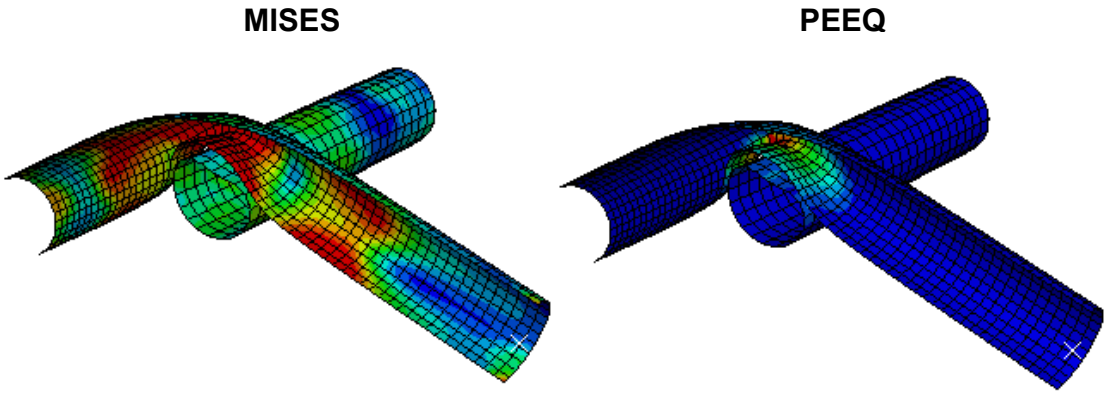


Figure W7–7. Contour plots

7. Create *X*–*Y* plots of the model's kinetic energy (ALLKE), internal energy (ALLIE), and plastic dissipated energy (ALLPD). The energy plot is shown in Figure W7–8. Note this figure has been customized for clarity.

   Tip: Expand the **History Output** container in the Results Tree and select the three curves noted above. Click mouse button 3 and select **Plot** from the menu that appears.

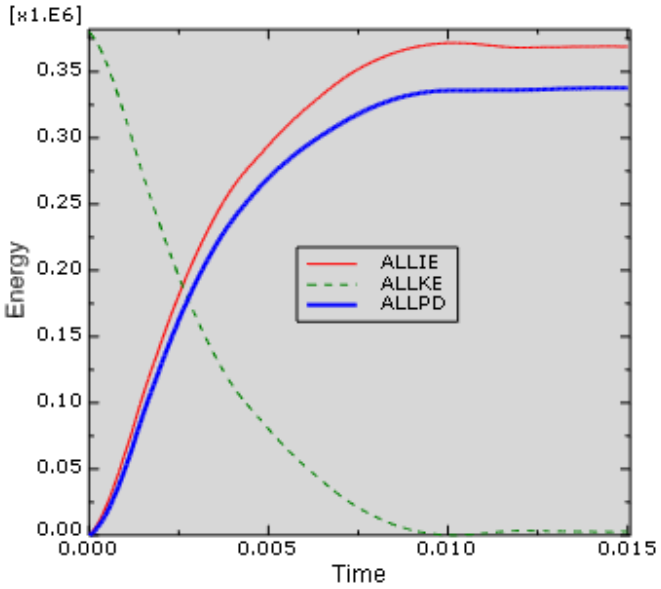


Figure W7–8. Energy histories

Question W7–8: What do the energy history plots indicate?

8. Select the pinned node reaction force components RF1, RF2, and RF3. The reaction force plot is shown in Figure W7–9. Note this figure has been customized for clarity.

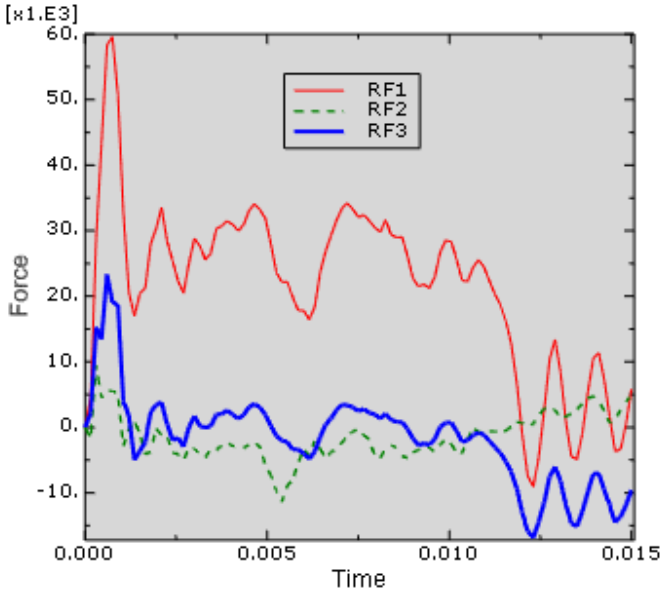


Figure W7–9. Reaction force histories

Note: A script that creates the complete pipe assembly model is available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script is named `ws_solver_pipe_whip_answer.py` and is available using the Abaqus fetch utility.

Workshop 7

Contact with Abaqus/Explicit

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Define a rigid body constraint.
- Define a general contact interaction.
- Apply boundary and initial conditions.
- Perform an impact analysis.
- Use Abaqus/Viewer to view results.

Introduction

This workshop involves the simulation of a pipe-on-pipe impact resulting from the rupture of a high-pressure line in a power plant. It is assumed that a sudden release of fluid could cause one segment of the pipe to rotate about its support and strike a neighboring pipe. The goal of the analysis is to determine strain and stress conditions in both pipes and their deformed shapes. The simulation will be performed using Abaqus/Explicit.

This workshop is based on "Pipe whip simulation," Section 1.3.9 of the Abaqus Benchmarks Manual.

© Dassault Systèmes, 2008

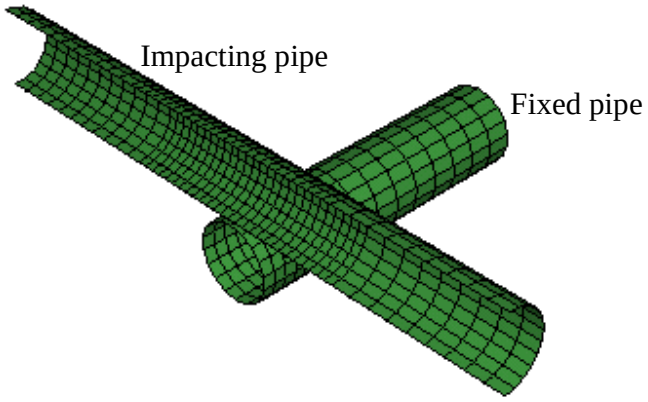


Figure W7–1. Pipe whip model: finite element mesh

Both pipes have a mean diameter of 6.5 inches with a 0.432 inch wall thickness and a span of 50 inches between supports. The fixed pipe is assumed to be fully restrained at both ends, while the impacting pipe is allowed to rotate about a fixed pivot located at one of its ends, with the other end free. We exploit the symmetry of the structure and the loading and, thus, model only the geometry on one side of the central symmetry plane, as shown in Figure W7–1.

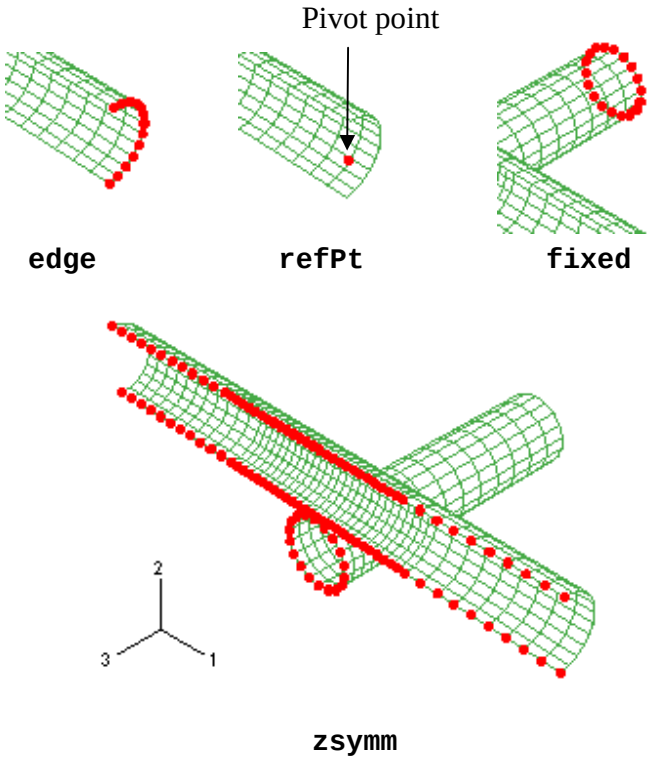


Figure W7–2. Node sets

Impact analysis

The model geometry, material properties, and loading history for the impact analysis are already defined and can be found in **`../abaqus_solvers/keywords/workshop7/w_pipe_whip.inp`**. You will have to edit the input file to include the material properties, rigid body constraint, contact interaction, initial conditions, boundary conditions, step definition, and output requests. Predefined sets are included to ease your work. These are shown in Figure W7–2.

Material and section properties

1. Both pipes are made of steel. A von Mises elastic, perfectly plastic material model is used. Create a material named **`Steel`** with the following properties:

Modulus of elasticity:	**`30E6`** psi
Poisson's ratio:	**`0.3`**
Yield Stress:	**`45.0E3`** psi
Density:	**`7.324E-4`** lb-sec^2/in^4

   Question W7–1: Why is density required in the material model definition? Can you comment on the units of density used in this problem?

2. Assign shell section properties to each pipe. Each pipe is 0.432 inches thick. Use Gauss integration with 3 points through the thickness for each section property. The elements of the impacting pipe are contained in element set **`pipe-impacting`**, while the elements of the fixed pipe are in element set **`pipe-fixed`**.

   Question W7–2: Why are only three integration points used through the thickness?

Rigid body constraint

Define a rigid body constraint between the nodes at the pivot end of the impacting pipe (node set **`edge`**) and the rigid body reference point (node set **`refPt`**). Both the translations and rotations of the pipe nodes are controlled by the rigid body constraint.

Question W7–3: In order to drive both the translations and rotations of the pipe edge nodes, what type of node set needs to be used in the rigid body constraint?

Contact interaction

Define general contact between the two pipes. Assume frictional contact with a coefficient of friction equal to 0.2.

Question W7–4: Are the contact constraints part of the model or history data?

© Dassault Systèmes, 2008

Initial conditions

The impacting pipe has an initial angular velocity of 75 radians/sec about its supported (pinned) end. Assign a rotating velocity initial condition to all the nodes in the impacting pipe (node set **`pipe-impacting`**).

The rotation is about the positive *Z*-direction passing though the rigid body reference point. The coordinates of the reference point are 25.0, 6.932, 25.0.

Question W7–5: How can you use the coordinates of the reference point to define the axis of rotation?

Boundary conditions

The edges located on the symmetry plane (node set **`zsymm`**) must be given appropriate symmetry boundary conditions. One end of the fixed pipe is fully restrained (node set **`fixed`**). The rigid body reference point (node set **`refPt`**) is free to rotate about its position.

Question W7–6: Are the boundary conditions part of the model or history data in an Abaqus/Explicit analysis?

Step definition and output requests

Because of the high-speed nature of the event, the simulation is performed using a single explicit dynamics step.

1. Create an explicit dynamics step with a time period of 0.015 seconds.
2. Write preselected field output to the output database at 12 equally spaced intervals.
3. Request reaction force history output at the constrained end of the impacting pipe. Write the data to the output database at 100 evenly spaced time intervals during the analysis.
4. Request preselected history output at the default number of intervals.

Save the input file, and run the impact analysis by entering the following command at the prompt:

```
abaqus job=w_pipe_whip
```

Visualization

1. Once the analysis completes successfully, open the output database file in Abaqus/Viewer.
2. Plot the undeformed and the deformed model shapes. From the main menu bar, select **Tools→Color Code** (or click in the toolbar) and assign different colors to the two pipes (you can distinguish between them using section assignments), as shown in Figure W7–3.

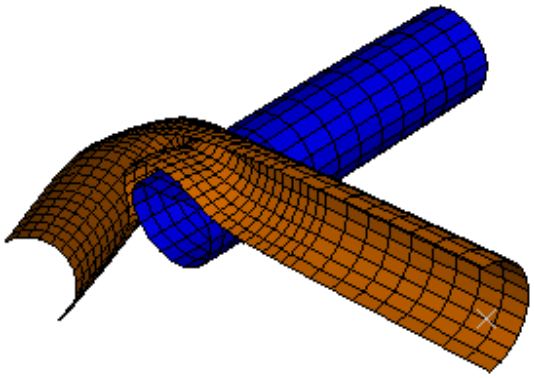


Figure W7–3. Deformed model shape

3. Use the **Animate: Time History** tool to animate the deformation history.
4. Contour the Mises stress and equivalent plastic strain (PEEQ) on the deformed shape, as shown in Figure W7–4.

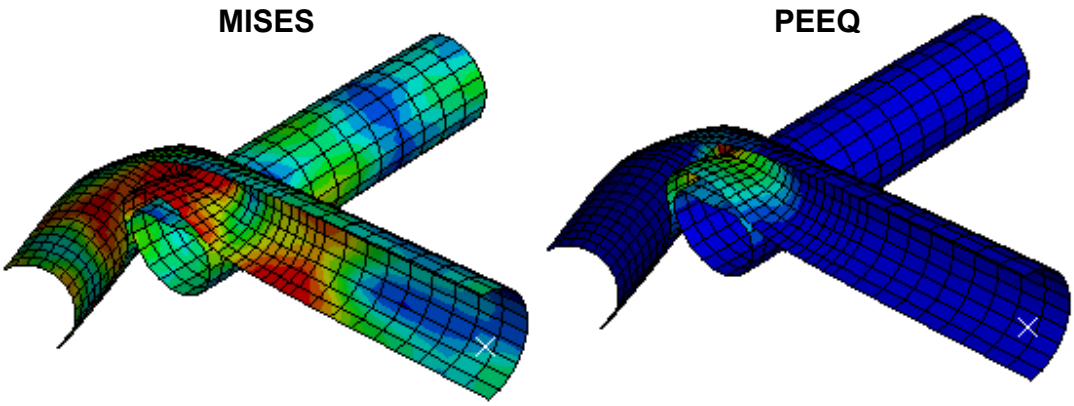


Figure W7–4. Contour plots

5. Create *X*–*Y* plots of the model's kinetic energy (ALLKE), internal energy (ALLIE), and dissipated energy (ALLPD). The energy plot is shown in Figure W7–5. Note this figure has been customized for clarity.

   Tip: Expand the **History Output** container in the Results Tree and select the three curves noted above. Click mouse button 3 and select **Plot** from the menu that appears.

© Dassault Systèmes, 2008

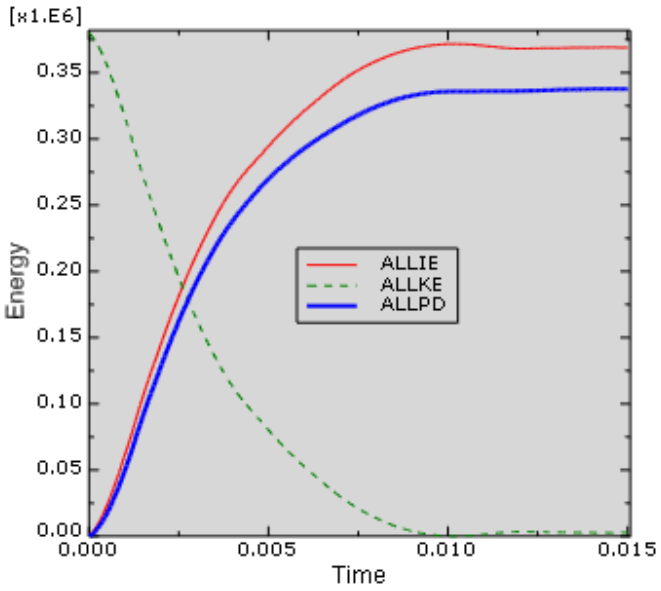


Figure W7–5. Energy histories

Question W7–7: What do the energy history plots indicate?

© Dassault Systèmes, 2008

6. Select and plot the pinned node reaction force components RF1, RF2, and RF3. The curves appear in Figure W7–6. Note this figure has been customized for clarity.

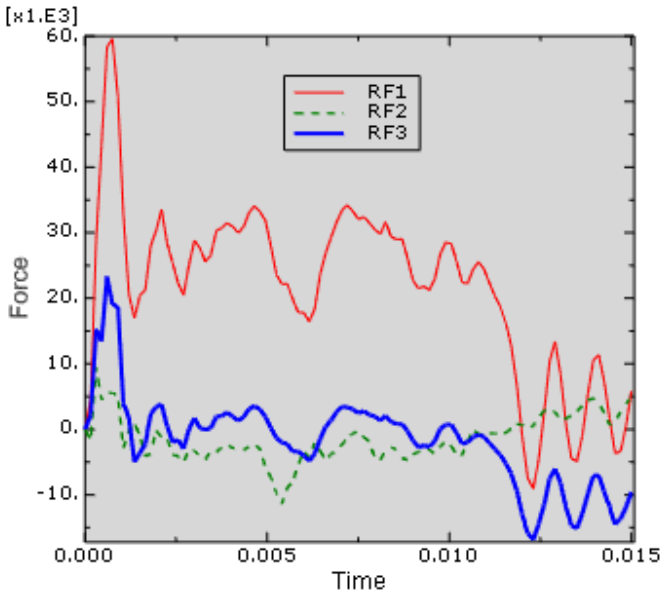


Figure W7–6. Reaction force histories

Note: A complete input file is available for your convenience. You may consult this file if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input file is named

`w_pipe_whip_complete.inp`

and is available using the Abaqus fetch utility.

© Dassault Systèmes, 2008

Workshop 8

Quasi-Static and Import Analysis

Interactive Version

Note: This workshop provides instructions in terms of the Abaqus GUI interface. If you wish to use the Abaqus Keywords interface instead, please see the "Keywords" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Approximate a quasi-static solution using Abaqus/Explicit.
- Understand the effects of mass scaling.
- Transfer results between Abaqus/Explicit and Abaqus/Standard.

Introduction

In this workshop you will examine the deep drawing of a can bottom. A one-stage forming process is simulated in Abaqus/Explicit; the springback analysis is performed in Abaqus/Standard. The final deformed shape of the can bottom is shown in Figure W8–1. The import capability is used to transfer the results between Abaqus/Explicit and Abaqus/Standard.

One of the advantages of using Abaqus/Explicit for metal forming simulations is that, in general, Abaqus/Explicit resolves complicated contact conditions more readily than Abaqus/Standard.

© Dassault Systèmes, 2008

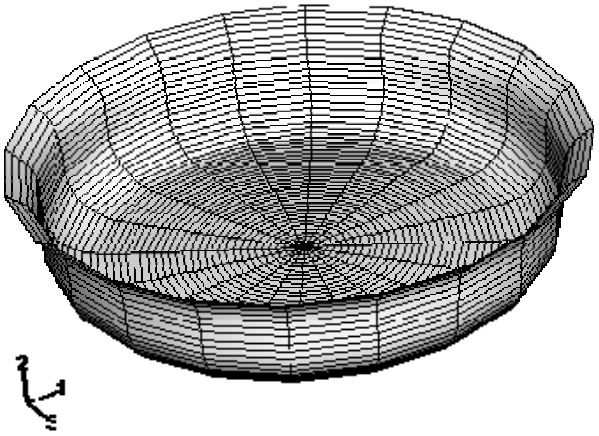


Figure W8–1. Final deformed shape

Preliminaries

1. Enter the working directory for this workshop:
   `../abaqus_solvers/interactive/workshop8`
2. Run the script **`ws_solver_can_bottom.py`** using the following command:
   `abaqus cae startup=ws_solver_can_bottom.py`

The above command creates an Abaqus/CAE database named **`canBottom.cae`** in the current directory. It includes two models. The one named **`frequency`** will be used to determine the first eigenmode of the blank to establish the step time for the subsequent Abaqus/Explicit analysis. The one named **`stamp`** will initially be used to perform the metal forming analysis and will later be edited for the springback analysis. Figure W8–2 shows the components of the model—the punch, the die, and the blank—in their initial positions. The blank is modeled using axisymmetric shell elements (SAX1). The shell reference surface lies at the shell midsurface.

© Dassault Systèmes, 2008

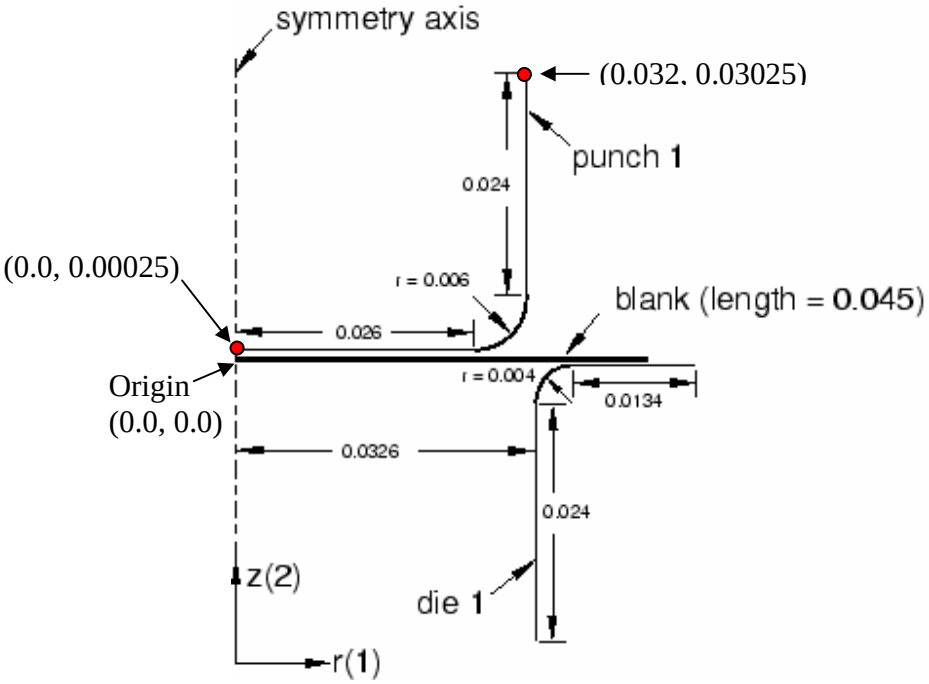


Figure W8–2. Model geometry

Part 1: Establishing the Abaqus/Explicit analysis time

In this section you will determine the first eigenmode of the blank and use it to establish the step time for the subsequent Abaqus/Explicit analysis.

1. Using the Model Tree, review the model definitions of the model **frequency**.

   Question W8–1: What analysis procedure is used in this model?

   Question W8–2: In Abaqus a distinction is made between linear perturbation analysis steps and general analysis steps. What type of procedure is the analysis procedure in this model?

   Question W8–3: In an analysis with more than one step in the same model, what influence does the result of a linear perturbation step have on the base state of the model for the following analysis step?

2. Create a job named **`Frequency`** for the model **frequency**.
3. Save your model database file, submit the job for analysis, and monitor its progress.
4. When the analysis is complete, open the output database file **`frequency.odb`** in the Visualization module.

© Dassault Systèmes, 2008

5. Plot the deformed model shape. The deformed shape for the first eigenmode will be displayed in the viewport. The corresponding eigenvalue will be reported in the state block.

   The fundamental frequency, *f*, of the blank is 304 Hz, corresponding to a time period of 0.0033 s ($T = 1/f$). This time period provides a lower bound on the step time for the first forming stage. Choosing the step time to be 10 times the time period of the fundamental natural frequency, or 0.033 s, should ensure a quality quasi-static solution. This time period corresponds to a constant punch velocity of 0.45 m/s, which is typical for metal forming.

Part 2: Metal Forming Analysis

You will now complete the model **stamp** to perform the metal forming analysis using ABAUS/Explicit. Make the following changes to the model **stamp**.

Completing the assembly

In this section you will complete the assembly definition of the can bottom forming model (Figure W8–2) by instancing the part representing the punch (**PUNCH1**).

1. Make current the model **stamp**. If necessary, set this model to be the root of the tree.
2. In the Model Tree, expand the **Assembly** container and double-click **Instances** to create an instance of the analytical rigid part **PUNCH1**. In the **Create Instance** dialog box, select part **PUNCH1**, accept other defaults setting, and click **OK**. Make sure to offset the punch from the blank by the half thickness of the blank (0.00025 m) using the **Translate Instance** tool in the toolbox. The viewport displays the assembly with the geometry as shown in Figure 8–2.

Defining displacement history output

In this section you will add a history output request to write the displacement history at the reference point of the punch to the output database file.

1. Create a set including the reference point associated with the punch. In the Model Tree, expand the branch of the part **PUNCH1** underneath the **Parts** container and double-click **Sets**. Name the set **PunchRP**. From the viewport, select the reference point **RP** of the part **PUNCH1**.
2. In the Model Tree, double-click **History Output Requests** to create an additional history output request to output the displacement history for the set **PUNCH1-1.PunchRP**. Note that **PUNCH1-1.PunchRP** is an assembly-level set generated from the previously-created part-level set **PunchRP** by Abaqus/CAE.

Defining contact

In this section you will define contact of the blank with the die and the punch.

1. Define a contact property. In the Model Tree, double-click **Interaction Properties**. In the **Create Interaction Property** dialog box, select the type **Contact** and click **Continue**. From the menu bar of the contact property editor, select **Mechanical→Tangential Behavior**. Select the **Penalty** friction formulation and enter **`0.1`** for the friction coefficient. Click **OK** to exit the contact property editor.
2. Define a surface-to-surface contact interaction between the blank and die.
    a. In the Model Tree, double-click the **Interactions** container. In the **Create Interaction** dialog box, name the interaction **`blank_die`**, select **Step-1** as the step and the **Surface-to-surface contact (Explicit)** type, and click **Continue**.
    b. You will be prompted to select the first surface. In the viewport, select the die.
    c. Click mouse button 2 in the viewport or click **Done** in the prompt area to confirm the selection.
    d. You will be prompted to choose a side of the edge. Choose the side facing the blank by selecting the corresponding color, **Magenta** or **Yellow**, in the prompt area.
    e. In the prompt area, select **Surface** as the second surface type. In the viewport, select the blank.
    f. Click mouse button 2 in the viewport or click **Done** in the prompt area to confirm the selection.
    g. Again, you will be prompted to choose a side of the edge. Choose the side facing the die.

       The interaction editor appears.
    h. In the **Edit Interaction** dialog box, view and accept the default setting. Click **OK** to create the interaction and exit the interaction editor.

       Note that Abaqus/CAE automatically assigns the previously-created interaction property to this interaction.
3. Using a similar procedure, define an additional surface-to-surface contact interaction named **`blank_punch`** between the blank and punch.

Question W8–4: What effect will an increase in friction have on the solution?

Defining material

In this section you will add the material definition of the blank. The blank material is steel with Young's modulus E =210E9 Pa, Poisson's ratio ν =0.3, and density ρ =7800 kg/m^3. Figure W8–3 shows the nominal plasticity material data for the blank as tabulated in Table W8–1.

© Dassault Systèmes, 2008

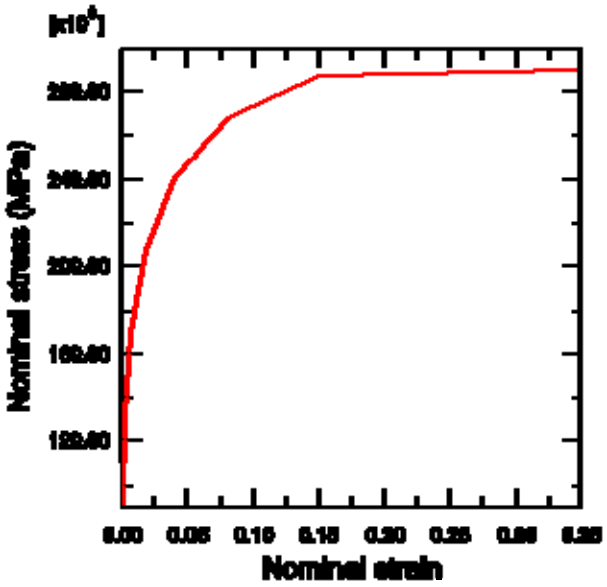


Figure W8–3. Nominal stress vs. nominal strain

Question W8–5: When entering plasticity data into the material model, what are the stress and strain measures that Abaqus uses?

Table W8–1. Nominal stress vs. nominal strain

Nominal stress (Pa)	Nominal strain
90.96×10^6	4.334×10^{-4}
130.71×10^6	2.216×10^{-3}
169.75×10^6	7.331×10^{-3}
207.08×10^6	1.888×10^{-2}
240.99×10^6	4.153×10^{-2}
268.89×10^6	8.218×10^{-2}
287.59×10^6	1.509×10^{-1}
290.57×10^6	3.456×10^{-1}

Table W8–2. True stress vs. log plastic strain

True stress (Pa)	Log plastic strain
91×10^6	0.0
131×10^6	0.159×10^{-2}
171×10^6	0.649×10^{-2}
211×10^6	0.177×10^{-1}
251×10^6	0.395×10^{-1}
291×10^6	0.776×10^{-1}
331×10^6	0.139
391×10^6	0.295

© Dassault Systèmes, 2008

Table W8–2 lists the corresponding true stress and logarithmic strain values. These values were obtained using the following relationships:

$$\sigma = \sigma_{nom}\left(1+\varepsilon_{nom}\right)$$

$$\varepsilon = 1\mathrm{n}\left(1+\varepsilon_{nom}\right)$$

$$\varepsilon_{pl} = \varepsilon_{tot} - \varepsilon_{el} = \varepsilon_{tot} - \sigma / E$$

These equations are valid for isotropic materials and establish the relationships between the true stress and strain measures (used in Abaqus) and the nominal stress and strain measures.

Note that a dummy material named **`Steel`** has been created and assigned to the part **BLANK**. You will need to add the material properties:

1. In the Model Tree, expand the **Materials** container and double-click **Steel**.
2. From the menu bar of the material editor, select **Mechanical→Elasticity→Elastic** and enter a value of **`210e9`** for **Young's modulus** and **`0.3`** for **Poisson's ratio**; select **Mechanical→Plasticity→Plastic** and enter the data provided in Table W8–2 for **Yield Stress** and **Plastic Strain**; and select **General→Density** and enter a value of **`7800`** for **Mass Density**.
3. To reduce high-frequency noise in the solution (caused primarily by the oscillations of the blank's free end) add stiffness proportional damping to the material definition of the blank. It is best to use the smallest amount of damping possible to obtain the desired solution since increasing the stiffness damping decreases the stable time increment and, thus, increases the computer time. To avoid a dramatic drop in the stable time increment, the stiffness proportional damping factor β_R should be less than, or of the same order of magnitude as, the initial stable time increment without damping. We choose a damping factor of $\beta_R = 1\times10^{-7}$.

   From the material editor's menu bar, select **Mechanical→ Damping** and enter a value of **`1.e-7`** in the **Beta** field.
4. Click **OK** to save the data and exit the material editor.

Question W8–6: What effects would a higher damping coefficient have?

Defining displacement boundary condition

To form the can bottom, we will displace the punch by moving its rigid body reference point 0.015 m in the negative 2-direction. The punch displacement will be applied in the form of a displacement boundary condition. Because Abaqus/Explicit does not permit displacement discontinuities, prescribed displacements must refer to an amplitude definition. Figure W8–4 shows the desired displacement behavior for the punch. Note that this curve is smooth in its first and second derivatives.

© Dassault Systèmes, 2008

Question W8–7: What is the slope of the curve at the beginning and end, and why is this important?

1. In the Model Tree, double-click **Amplitudes** to define an amplitude curve corresponding to Figure W8–4. In the **Create Amplitude** dialog box, name the amplitude **FORM1**, choose the **Smooth step** type, and click **Continue**. In the **Edit Amplitude** dialog box, enter the data pair **0, 0** for the first row and **0.033, 1** for the second row, and click **OK** to exit the amplitude editor.
2. In the Model Tree, double-click **BCs** to create a **Displacement/Rotation** boundary condition named **PunchMove** in **Step-1** to move the reference point associated with the punch in the 2-direction by –0.015 m. In the **Edit Boundary Condition** dialog box, toggle on **U1**, **U2**, and **UR3**; enter a value of **-0.0015** for **U2** and **0** for **U1** and **UR3**; and choose the amplitude curve **FORM1**.

   The amplitude values will be multiplied by the displacement you define in the boundary condition.

Question W8–8: How would the results change if a linear amplitude definition was used instead?

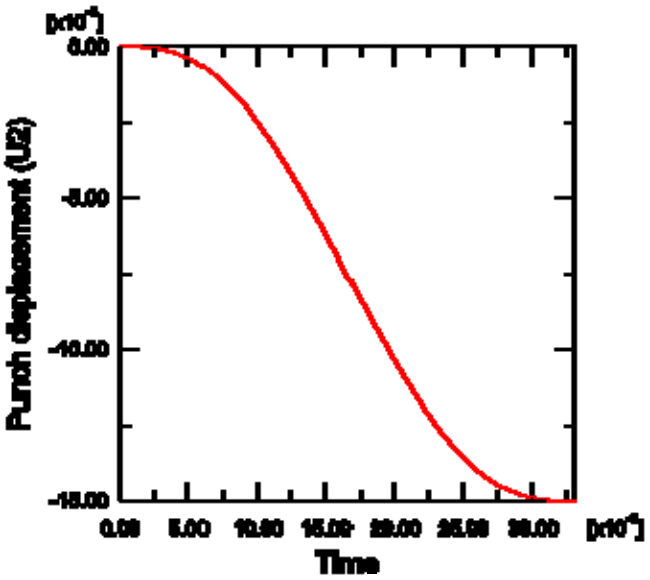


Figure W8–4. Displacement curve of punch

© Dassault Systèmes, 2008

Speeding up the analysis

In general, quasi-static processes cannot be modeled in their natural time scale in Abaqus/Explicit since a large number of time increments would be required (recall that time increments in Abaqus/Explicit are generally very small). Thus, it is sometimes necessary to increase the speed of the simulation artificially to reduce the computational cost. One method to reduce the cost of the analysis is to use mass scaling.

While various forms of mass scaling are available in Abaqus/Explicit, we will concentrate on fixed mass scaling in this workshop and will implement it using the fixed mass scaling option available in the step editor. The reason for choosing fixed mass scaling is that it provides a simple means to modify the mass properties of a quasi-static model at the beginning of the analysis. It is also computationally less expensive than variable mass scaling, because the mass is scaled only once at the beginning of the step.

1. In the Model Tree, expand the **Steps** container and double-click **Step-1** to edit this step definition to include mass scaling.
2. In the **Edit Step** dialog box, click the **Mass scaling** tab.
3. In the **Mass scaling** tabbed page, choose **Use scaling definitions below** and click **Create**.
4. In the **Edit mass scaling** dialog box that appears, accept all defaults and enter **`10`** in the **Scale by factor** field. Click **OK** to save the data and exit the mass scaling editor.
5. Click **OK** to save the changes and exit the step editor.

Question W8–9: How do you determine if an analysis that includes mass scaling produces acceptable results?

Question W8–10: How does mass scaling affect the solution time?

Running job and postprocessing results

1. Create a job named **`draw_bot`** for the model **Stamp**.
2. Save your model database file, submit the job for analysis, and monitor its progress.
3. When the analysis is complete, open the output database file **`draw_bot.odb`** in the Visualization module.
4. Display the curves of internal and kinetic energy (i.e., ALLIE and ALLKE) in the same plot by selecting them from the Results Tree (underneath the **History Output** container). Use the **XY Curve Options** tool in the toolbox to display the curve symbols. You should see a plot similar to Figure W8–5. Note this figure has been customized for clarity.

   Tip: Use **[Ctrl]+Click** for multiple selections.

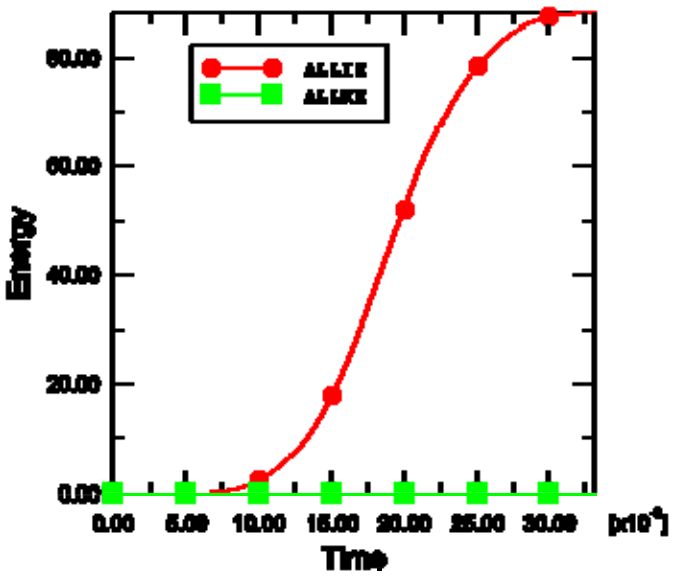


Figure W8–5. Internal and kinetic energy

5. Certain elements have hourglass modes that affect their behavior. Hourglass modes are modes of deformation that do not cause any strains at the integration points. An indication of whether hourglassing has an effect on the solution is the artificial energy, variable ALLAE. Plot the artificial energy and the internal energy, variable ALLIE, on the same plot. The artificial energy should always be much less than the internal energy (say less than 0.5%).

   Question W8–11: What elements are used to model the blank, and does this element type have an hourglass deformation mode?

6. Display the deformed shape of the blank only. In the Results Tree, expand the **Instances** container underneath the output database **`draw_bot.odb`**. Click mouse button 3 on the instance **BLANK-1** and select **Replace** from the menu that appears.
7. Expand the displayed area to 180^{o} by selecting **View→ODB Display Options** from the main menu bar. In the **Sweep/Extrude** tabbed page in the **ODB Display Options** dialog box, toggle on **Sweep elements** and accept the default settings. You should see a shape similar to that in Figure W8–6.

© Dassault Systèmes, 2008

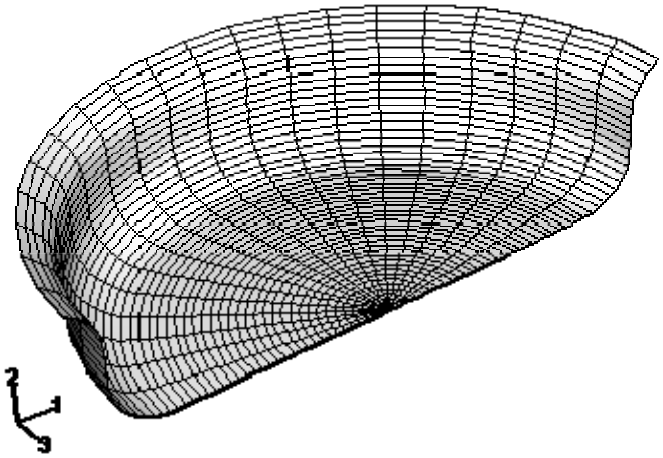


Figure W8–6. 180° expanded deformed shape

8. Contour the Mises stress distribution of the 180° model using the **Plot Contours on Deformed Shape** tool in the toolbox. To select other variables for contouring, use the **Result→Field Output** menu from the main menu bar.
9. Plot the punch displacement history (U2 for the node set **PUNCHRP**) shown in Figure W8–4 by double-clicking **Spatial displacement: U2 PI: PUNCH1-1 NODE xxx in NSET PUNCHRP** under the **History Output** container in the Results Tree.

Part 3: Springback analysis

In the manufacturing process the part is removed after the forming has been completed and the material is free to springback into an unconstrained state. To understand the final shape after this physical effect, we perform a springback analysis in Abaqus/Standard.

1. Copy the model **stamp** to a model named **springback**. Make all subsequent model changes to the model **springback**.
2. Since only the blank needs to be imported, delete the following features from the model **springback**:
    a. Part instances **DIE1** and **PUNCH1**.
    b. All contact interactions and properties.
    c. Boundary conditions **FixDie** and **PunchMove**.
    d. History output request for **PunchRP**.
3. Replace the dynamic, explicit step with a general, static step. Set the time period to **1** and the initial increment to **0.1**, and include the effects of geometric nonlinearity. Rename the step **springback**.

4. Define an initial state.
   a. In the Model Tree, double-click **Predefined Fields**. In the **Create Predefined Field** dialog box, select **Initial** as the step, **Other** as the category, and **Initial State** as the type. Click **Continue**.
   b. Select the blank as the instance to assign the initial state.
   c. In the **Edit Predefined Field** dialog box that appears, enter the job name **`draw_bot`**, accept all other default settings, and click **OK**. This definition will allow the state of the model—stresses, strains, etc.—to be imported. By not updating the reference configuration, the springback displacements will be referred to the original undeformed configuration.
5. Note that the **XSYMM** boundary condition **BlankSymm** specified on the set **BSYM** constrains rigid body motions in the **U1** and **UR3** directions of the blank. Thus, you need an additional boundary condition to prevent rigid body motion along **U2**. In what follows you will fix the node at its final position at the end of the forming stage.

   In the Model Tree, double-click **BCs** to apply a **Displacement/Rotation** boundary condition to the set **BSYM** in **Step-1**. In the **Edit Boundary Condition** dialog box, choose the **Fixed at Current Position** method and fix **U2**.
6. Create a job named **`springback`** for the model **springback**.
7. Save your model database file, submit the job for analysis, and monitor its progress.

Question W8–12: Why is it advantageous to use Abaqus/Standard for the springback analysis?

Postprocessing

1. When the analysis is complete, open the output database file **`springback.odb`** in the Visualization module.
2. Contour the Mises stress distribution of the 180^o model.
3. Plot the final deformed shape for the model **`springback`**.
4. Plot the springback and formed shapes together. (First toggle off the **Sweep elements** option.)

   By not updating the reference configuration, the formed shape is stored in frame 0 of the output database. You must use overlay plots to superimpose the images:
   a. From the main menu bar, select **View→Overlay Plot**.
   b. Use the **Frame Selector** or the arrows in the context bar to select frame 0.
   c. In the **Overlay Plot Layer Manager**, click **Create**. Name the layer **`formed`**.

© Dassault Systèmes, 2008

d. Use the **Frame Selector** to select the last frame.

e. Use the **Common Plot Options** tool to change the fill color of the elements to blue.

f. In the **Overlay Plot Layer Manager**, click **Create**. Name the layer **`springback`**.

g. In the **Overlay Plot Layer Manager**, click **Plot Overlay**.

h. Zoom in to examine the shape differences more closely.

If you had updated the reference configuration, the formed shape is treated as the undeformed shape of the import analysis model (recall that when the reference configuration is updated, the end state of the previous analysis becomes the reference configuration of the import analysis; the reference configuration is considered the undeformed shape):

a. In the toolbox, click the **Allow Multiple Plot States** tool .

b. In the toolbox, click both the **Plot Undeformed Shape** and **Plot Deformed Shape** tools .

c. Use the **Common Plot Options** tool to increase the deformation scale factor so that the differences between the formed and springback shapes are clearly visible.

Note: A scripts that creates the complete stamping model are available for your convenience. Run this script if you encounter difficulties following the instructions or if you wish to check your work. The script named `ws_solver_can_bottow_answer.py` is available using the Abaqus fetch utility.

Workshop 8

Quasi-Static and Import Analysis

Keywords Version

Note: This workshop provides instructions in terms of the Abaqus Keywords interface. If you wish to use the Abaqus GUI interface instead, please see the "Interactive" version of these instructions.

Please complete either the Keywords or Interactive version of this workshop.

Goals

- Approximate a quasi-static solution using Abaqus/Explicit.
- Understand the effects of mass scaling.
- Transfer results between Abaqus/Explicit and Abaqus/Standard.

Introduction

In this workshop you will examine the deep drawing of a can bottom. A one-stage forming process is simulated in Abaqus/Explicit; the springback analysis is performed in Abaqus/Standard. The final deformed shape of the can bottom is shown in Figure W8–1. The import capability is used to transfer the results between Abaqus/Explicit and Abaqus/Standard.

One of the advantages of using Abaqus/Explicit for metal forming simulations is that, in general, Abaqus/Explicit resolves complicated contact conditions more readily than Abaqus/Standard.

© Dassault Systèmes, 2008

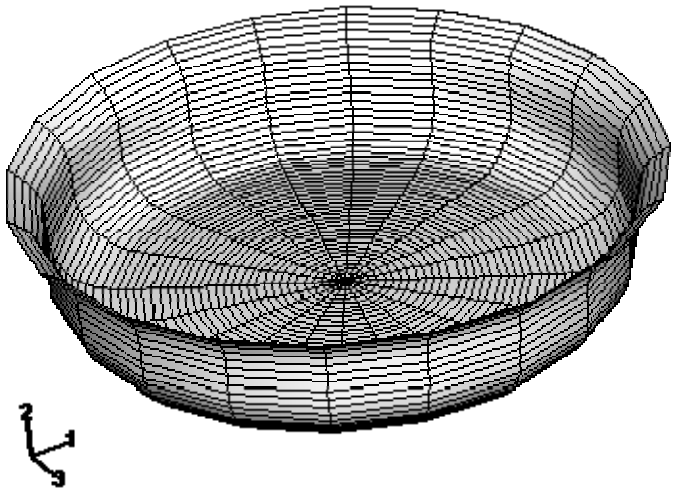


Figure W8–1. Final deformed shape

Change to the **`../abaqus_solvers/keywords/workshop8`** directory.

Establishing the Abaqus/Explicit analysis time

In this section you will determine the first eigenmode of the blank and use it to establish the step time for the subsequent Abaqus/Explicit analysis.

1. Open the file **`w_draw_freq.inp`**, and examine its contents to help you answer the following questions:

   Question W8–1: What analysis procedure is used in this input file?

   Question W8–2: In Abaqus a distinction is made between linear perturbation analysis steps and general analysis steps. What type of procedure is the analysis procedure in this input file?

   Question W8–3: In an analysis with more than one step in the same input file, what influence does the result of a linear perturbation step have on the base state of the model for the following analysis step?

2. Run the job by entering the following command:

   ```
   abaqus job=w_draw_freq
   ```

Plot the first eigenmode in Abaqus/Viewer. The fundamental frequency, *f*, of the blank is 304 Hz, corresponding to a time period of 0.0033 s ($T = 1/f$). This time period provides

a lower bound on the step time for the first forming stage. Choosing the step time to be 10 times the time period of the fundamental natural frequency, or 0.033 s, should ensure a quality quasi-static solution. This time period corresponds to a constant punch velocity of 0.45 m/s, which is typical for metal forming.

Geometry definition

In this section you will complete the geometry definition of the can forming model by defining the punch as an analytical rigid surface.

Figure W8–2 shows the components of the model—the punch, the die, and the blank—in their initial positions. The blank is modeled using axisymmetric shell elements (SAX1). The shell reference surface lies at the shell midsurface.

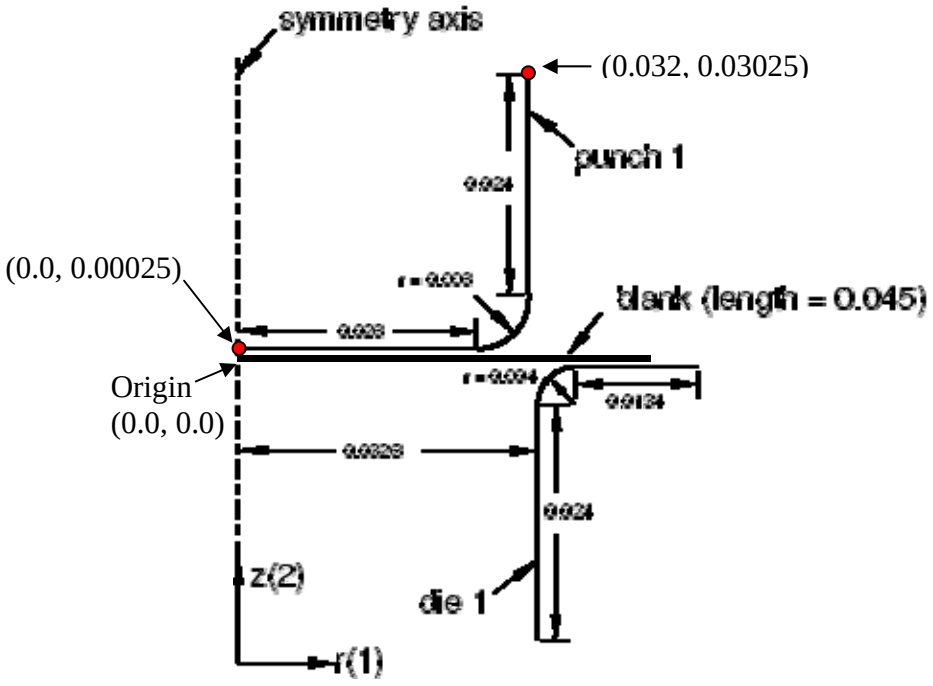


Figure W8–2. Model geometry

1. Open the file **`w_draw_bot.inp`** in an editor, and define punch 1 as an analytical rigid surface (see Figure W8–2 for the relevant dimensions).

   Use the definition of die 1 in the input file as an example of the input for an analytical rigid surface. The end point for punch 1 lies on the symmetry axis, a distance of half the blank thickness above the shell midsurface. Give the rigid surface the name **`PUNCH1`**, and use node 1001 as the rigid body reference node. The *RIGID BODY option has been defined already.

Question W8–4: How does the order of the line segments affect the ability of Abaqus to resolve the contact condition?

2. Define the surfaces, the contact pairs, and the surface interaction for the complete model using the ∗SURFACE, ∗CONTACT PAIR, and ∗SURFACE INTERACTION options. The blank is defined such that the element normal direction points toward the punch. The friction coefficient between the rigid tools and the blank is 0.1.

Question W8–5: What effect will an increase in friction have on the solution?

Question W8–6: In Abaqus the input data are classified as either model or history data. What type of data is the contact pair definition in Abaqus/Explicit? What type of data is the contact pair definition in Abaqus/Standard?

Material definition

In this section you will add the entire material definition to the input file.

The material is steel with Young's modulus E =210E9 Pa, Poisson's ratio ν =0.3, and density ρ =7800 kg/m^3. Figure W8–3 shows the nominal plasticity material data for the blank as tabulated in Table W8–1.

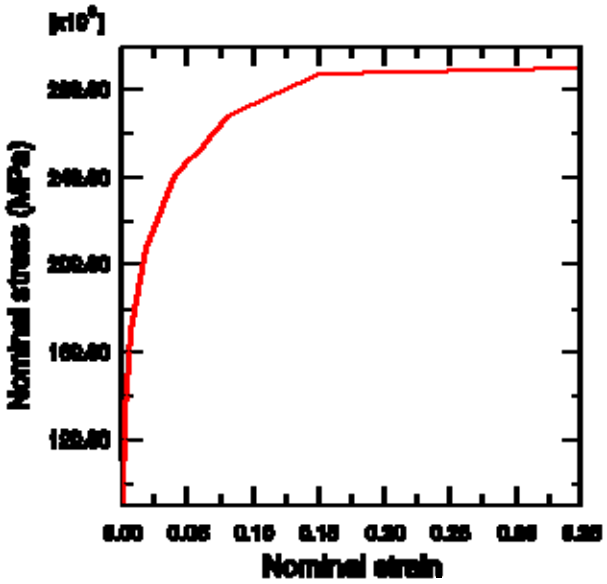


Figure W8–3. Nominal stress vs. nominal strain

© Dassault Systèmes, 2008

Question W8–7: When entering plasticity data with the ∗PLASTIC option, what are the stress and strain measures that Abaqus uses?

© Dassault Systèmes, 2008

Table W8–1

Nominal stress (Pa)	Nominal strain
90.96×10^6	4.334×10^{-4}
130.71×10^6	2.216×10^{-3}
169.75×10^6	7.331×10^{-3}
207.08×10^6	1.888×10^{-2}
240.99×10^6	4.153×10^{-2}
268.89×10^6	8.218×10^{-2}
287.59×10^6	1.509×10^{-1}
290.57×10^6	3.456×10^{-1}

Table W8–2

True stress (Pa)	Log plastic strain
91×10^6	0.0
131×10^6	0.159×10^{-2}
171×10^6	0.649×10^{-2}
211×10^6	0.177×10^{-1}
251×10^6	0.395×10^{-1}
291×10^6	0.776×10^{-1}
331×10^6	0.139
391×10^6	0.295

Table W8–2 lists the corresponding true stress and logarithmic strain values. These values were obtained using the following relationships:

$$\sigma = \sigma_{nom}\left(1+\varepsilon_{nom}\right)$$

$$\varepsilon = 1\mathrm{n}\left(1+\varepsilon_{nom}\right)$$

$$\varepsilon_{pl} = \varepsilon_{tot} - \varepsilon_{el} = \varepsilon_{tot} - \sigma / E$$

These equations are valid for isotropic materials and establish the relationships between the true stress and strain measures (used in Abaqus) and the nominal stress and strain measures.

1. Complete the material definition, and name the material **STEEL**. Use the ∗ELASTIC option to enter Young's modulus and Poisson's ratio and the ∗PLASTIC option to enter the material data in Table W8–2.

   Tip: Both of these options must be grouped under the *MATERIAL option.

2. To reduce high-frequency noise in the solution (caused primarily by the oscillations of the blank's free end), add stiffness proportional damping to the material definition of the blank. It is best to use the smallest amount of damping possible to obtain the desired solution since increasing the stiffness damping decreases the stable time increment and, thus, increases the computer time. To avoid a dramatic drop in the stable time increment, the stiffness proportional

damping factor β_R should be less than, or of the same order of magnitude as, the initial stable time increment without damping. We choose a damping factor of $\beta_R = 1\times10^{-7}$, which is included by using the ∗DAMPING, BETA=1.E–7 material option.

Question W8–8: What effects would a higher damping coefficient have?

Amplitude definition

To form the can bottom, we will displace the punch by moving its rigid body reference node 0.015 m in the negative 2-direction. The punch displacement will be applied in the form of a displacement boundary condition. Because Abaqus/Explicit does not permit displacement discontinuities, prescribed displacements must refer to an amplitude definition. In this section you will add the amplitude definition to the input file. Figure W8–4 shows the desired displacement behavior for the punch.

Question W8–9: What is the slope of the curve at the beginning and end, and why is this important?

1. Define the amplitude curve corresponding to Figure W8–4. The curve shown in Figure W8–4 is smooth in its first and second derivatives and is defined by using the DEFINITION=SMOOTH STEP parameter with the ∗AMPLITUDE option. Define the punch displacement amplitude, and name the amplitude FORM1.

Question W8–10: How would the results change if a linear amplitude definition was used instead?

2. Note that in the input file there is a boundary condition that refers to the amplitude definition (FORM1) just completed.

Speeding up the analysis

In general, quasi-static processes cannot be modeled in their natural time scale in Abaqus/Explicit since a large number of time increments would be required. (Recall that time increments in Abaqus/Explicit are generally very small). Thus, it is sometimes necessary to increase the speed of the simulation artificially to reduce the computational cost. One method to reduce the cost of the analysis is to use mass scaling.

© Dassault Systèmes, 2008

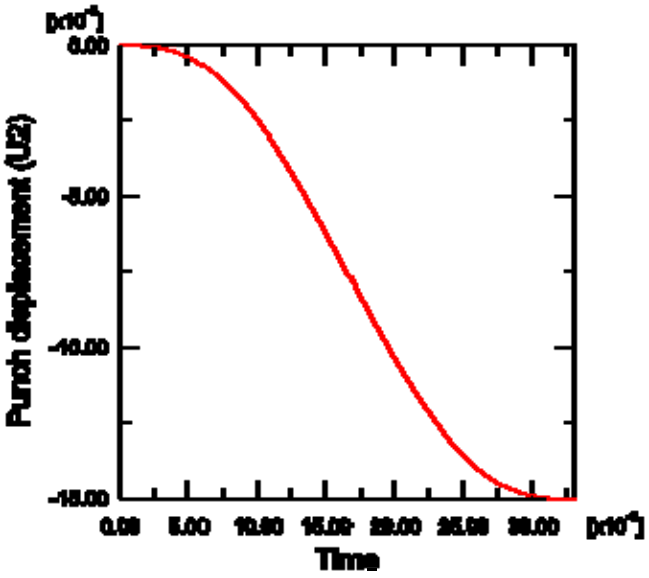


Figure W8–4. Displacement curve of punch

While various forms of mass scaling are available in Abaqus/Explicit, we will concentrate on fixed mass scaling in this workshop and will implement it using the *FIXED MASS SCALING option. The reason for choosing fixed mass scaling is that it provides a simple means to modify the mass properties of a quasi-static model at the beginning of the analysis. It is also computationally less expensive than variable mass scaling, because the mass is scaled only once at the beginning of the step.

1. Specify a mass scaling factor of 10 by setting the FACTOR parameter on the *FIXED MASS SCALING option, and complete the mass scaling definition in the input file.

Question W8–11: How do you determine if an analysis that includes mass scaling produces acceptable results?

Question W8–12: How does mass scaling affect the solution time?

Analysis and results evaluation

1. Run the analysis with the input file **`w_draw_bot.inp`**.
2. Monitor the progress of the solution in the status file.
3. Open the output database **`w_draw_bot.odb`** in Abaqus/Viewer.
4. Display the curves for internal and kinetic energy (variables ALLIE and ALLKE, respectively) in the same plot by selecting them from the Results Tree

© Dassault Systèmes, 2008

(underneath the **History Output** container). To display the curve symbols, use the **XY Curve Options** tool in the toolbox. You should see a plot similar to Figure W8–5. Note this figure has been customized for clarity.

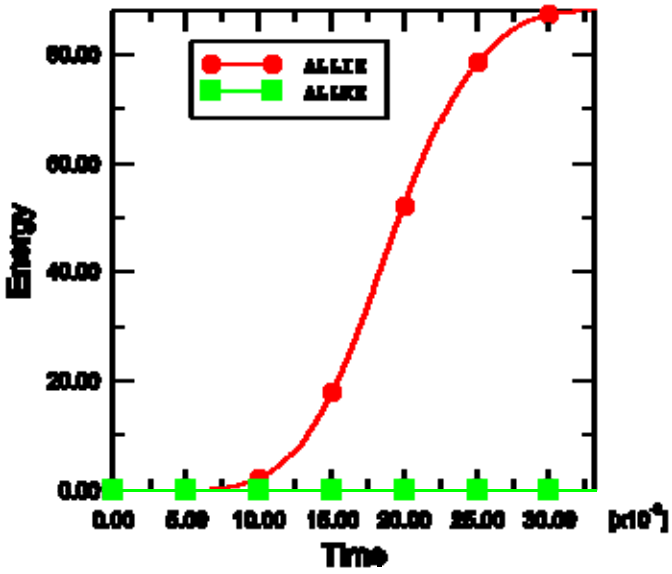


Figure W8–5. Internal and kinetic energy

5. Certain elements have hourglass modes that affect their behavior. Hourglass modes are modes of deformation that do not cause any strains at the integration points. An indication of whether hourglassing has an effect on the solution is the artificial energy, variable ALLAE. Plot the artificial energy and the internal energy, variable ALLIE, on the same plot. The artificial energy should always be much less than the internal energy (say less than 0.5%).

   Question W8–13: What elements are used to model the blank, and does this element type have an hourglass deformation mode?

6. Display only the deformed shape of the blank:
   a. Expand the **Materials** container in the Results Tree and click mouse button 3 on **STEEL**.
   b. From the menu that appears, select **Replace**.
7. Expand the displayed area to 180°:
   a. Select **View→ODB Display Options** from the main menu.
   b. In the **Sweep/Extrude** tabbed page of the **ODB Display Options** dialog box, toggle on **Sweep elements**.

      You should see a shape similar to that in Figure W8–6.

© Dassault Systèmes, 2008

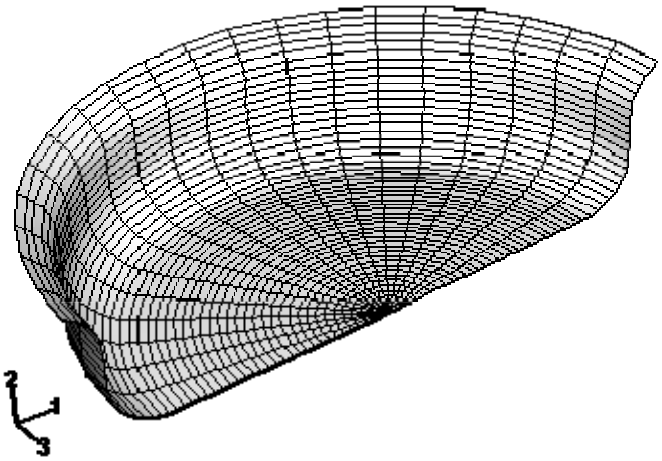


Figure W8–6. 180° expanded deformed shape

c. Contour the Mises stress distribution of the 180° model using the **Plot Contours** tool in the toolbox; to select other variables for contouring, use the **Result→Field Output** menu from the main menu bar.

d. Plot the punch displacement shown in Figure W8–4 by double-clicking the U2 curve for node 1001 in the Results Tree (underneath the **History Output** container).

Springback analysis

In the manufacturing process the part is removed after the forming has been completed and the material is free to springback into an unconstrained state. To understand the final shape after this physical effect, we perform a springback analysis in Abaqus/Standard.

1. Open the file **`w_draw_bot_spring.inp`** in an editor, and import the blank from the end of the **`w_draw_bot`** analysis. Use the STATE=YES parameter on the ∗IMPORT option to import the material state of the elements.

   Question W8–14: To what value should the UPDATE parameter on the ∗IMPORT option be set if the total Mises stresses are to be plotted at the end of the springback analysis?

   Question W8–15: Where do you find the information to define the STEP and INTERVAL parameters on the ∗IMPORT option?

2. The boundary conditions are not imported and must be respecified. In addition, it is necessary to fix a single point, such as node set **`BSYM`**, in the 2-direction to prevent rigid body motion. It is important to use the FIXED parameter on the *BOUNDARY option so that **`BSYM`** is fixed at its final position at the end of the forming stage.

© Dassault Systèmes, 2008

Question W8–16: Why is it advantageous to choose Abaqus/Standard for the springback analysis?

Analysis and postprocessing

1. Run the analysis by entering the following command:

   ```
   abaqus job=w_draw_bot_spring oldjob=w_draw_bot
   ```

2. Open the output database **`w_draw_bot_spring.odb`** in Abaqus/Viewer.
3. Contour the Mises stress distribution of the 180^o model.
4. Plot the final deformed model shape, as shown in Figure W8–1.
5. Plot the springback and formed shapes together. (First toggle off the **Sweep elements** option.)

   If you used UPDATE=NO, the formed shape is stored in frame 0 of the output database. You must use overlay plots to superimpose the images in this case:

   a. Select **View→Overlay Plot** from the main menu bar.

   b. Use the **Frame Selector** or the arrows in the context bar to select frame 0.

   c. In the **Overlay Plot Layer Manager**, click **Create**. Name the layer **`formed`**.

   d. Use the **Frame Selector** or the arrows in the context bar to select frame 1.

   e. Use the **Common Plot Options** tool to change the fill color of the elements to blue.

   f. In the **Overlay Plot Layer Manager**, click **Create**. Name the layer **`springback`**.

   g. In the **Overlay Plot Layer Manager**, click **Plot Overlay**.

   h. Zoom in to examine the shape differences more closely.

   If you used UPDATE=YES, the formed shape is treated as the undeformed shape of the import analysis model (recall that when UPDATE=YES, the end state of the previous analysis becomes the reference configuration of the import analysis; the reference configuration is considered the undeformed shape):

   a. In the toolbox, click the **Allow Multiple Plot States** tool .

   b. In the toolbox, click both the **Plot Undeformed Shape** and **Plot Deformed Shape** tools .

   c. Use the **Common Plot Options** tool to increase the deformation scale factor so that the differences between the formed and springback shapes are clearly visible.

© Dassault Systèmes, 2008

Note: Complete input files are available for your convenience. You may consult these files if you encounter difficulties following the instructions outlined here or if you wish to check your work. The input files are named

`w_draw_bot_complete.inp`

`w_draw_bot_spring_complete.inp`

and are available using the Abaqus fetch utility.