Week 10 - Lecture
Nonlinear Structural Analysis
Product Lifecycle – Week 10

Goals of Simulation

- Validate Designs
- Predict Product Performance
- Optimize Designs
Replicate the Physical World

The more closely a simulation conforms to what happens in the physical world, the more value it has to offer.

– Model Geometry
– Analysis Type
– Materials
– Boundary Conditions
– Loading Conditions
Materials

• The available material selection is vast and changes everyday.
• The material properties and material model determine the behavior under operating conditions.
Force Estimation Methods

- **Experience**
  - Engineers reply on past experience from similar projects to estimate forces.

- **Rigid-Body Dynamics**
  - Leverage the 3D design data to perform a motion simulation using rigid bodies to gain insights into force values.

- **Physical Experimentation**
  - Experiment with prototypes or past products to obtain accurate force values.
Linear vs. Nonlinear Analysis

• Linear
  – Structure returns to original form
  – No changes in loading direction or magnitude
  – Material properties do not change
  – Small deformation and strain

• Nonlinear Focus for this week
  – Geometry changes resulting in stiffness change
  – Material deformation that may not return to original form
  – Supports changes in load direction and constraint locations
  – Support of nonlinear load curves
Nonlinear Analysis

• Design Engineer Belief
  – Too Complicated
  – Time Consuming
  – Specialized software packages

• Reality
  – Nonlinearities are common.
  – Advances in FEA Software
  – Advances in Computing Hardware
Types of Nonlinearity

• Nonlinear Material
  – Materials that do not have a complete linear stress strain curve as seen in plastic and rubber materials for example.

• Nonlinear Geometry
  – The changing shape of a model when large deformations exists provide nonlinear changes in the components stiffness.

• Nonlinear Boundary Condition
  – Boundary conditions that involve components in contact with one another often produce disproportionate changes in deformation.

• Nonlinear Loading Condition
  – Loading changes over time.
Materials Types

\[ \text{Stress} = \frac{\text{Force}}{\text{Area}} \]

\[ \text{Strain} = \frac{\text{Change in Length}}{\text{Original Length}} \]
Polymeric Material Example

Stress = \frac{\text{Force}}{\text{Area}}

\text{Strain} = \frac{\text{Change in Length}}{\text{Original Length}}

Elastic Region

Plastic Region

Yield Limit

Proportional Limit

Young’s Modulus

Failure
Nonlinear Boundary Condition

- Component contacts produce stresses and friction that result in disproportionate changes in deformation.
Nonlinear Geometry

- Changing shape triggers nonlinear changes in stiffness of the part model.
Event Simulation

- Event simulation allows for the entire event of a condition not just a static solution.
- In mechanical engineering the static case often does not dictate the design.
- Designs must consider the worst-case scenario which often occurs after loading.
Event Simulation Example

Linear static solution would have the entire load carried by vertical member.
Event Simulation Example

With an event simulation the horizontal members would carry some of the load.
Nonlinear Example

• Unbending and bending a paperclip back into position has nonlinear material and geometry effects.
Everyday Nonlinear Requirements
Autodesk Simulation

Autodesk Simulation helps designers and engineers make decisions earlier and predict product performance.

- Linear and Nonlinear Static Stress
- Fatigue Analysis
- Linear Dynamic
- Mechanical Event
- Heat Transfer
- CFD
- Multiphysics
Element Types

The following are some of the element types available in Autodesk Simulation

- Beam
- Spring
- Brick
- Shell
- Truss
- Special (Actuator)
- Special (Membrane)
- Special (Slider)
Load Curves

Load curves control the load value through a multiplier over a time period. Each specific load can follow a different load curve.
Six Flags Theme Parks - Case Study

Challenge
Maintain and renovate hundreds of thrill rides with minimal downtime.

Results

- New, in-house design of anti-rollback mechanism to modernize a wooden roller coaster, the Texas Giant

- Better collaboration with fabricators through use of high-quality drawings and documentation

- Improve quality by increasing reliability of high-wear parts.

“Our primary concern is safety. Inventor and Autodesk Simulation allow us to study real-world loading scenarios so we can be sure we have appropriate safety factors in our designs.”

Mike Neuzil
Corporate Engineer
Six Flags Theme Parks, Inc.
United States
Howden - Case Study

Challenge
Reduce the time taken to produce tenders with technically and economically competitive proposals, while taking account of increasingly detailed customer requirements

Results
• Substantial reduction in total design time.
• Faster response to customer design requirements.
• Simulation allows investigation and validation of new design approaches.
• Weight reduction through design optimization brings reduced costs.

“Flexibility, adaptability and reliability are the key benefits that we enjoy with Autodesk Simulation products.”

Laurent Tisserand
Technical Director
Howden
France
Computer-Cluster Projects (CP10)
Guided Lab Project 1

Guides instructions for completing a static stress linear simulation in Autodesk Simulation.
Guided Lab Project 2

Guided instructions for completing a mechanical event nonlinear analysis in Autodesk Simulation.
Guided Lab Project 3

Guided instructions for completing a snap fit nonlinear analysis on a medical device in Autodesk Simulation.
Problem Set Assignment

Analyze the electrical switch snap fit mounting to determine force required and design quality.
Demo Topics
User Interface

1. Setup tab
2. Stress Analysis browser
3. Graphical display
Mesh Panel and Settings

![Mesh Panel and Settings](image)
Simulation Browser

1. Multiple simulations
2. Analysis Type
3. Constraints and Loads
4. Part
5. Element Type
6. Element Definition
7. Material
8. Surfaces
9. Contacts
Assign Materials
Assign Constraints

![Constraint Assignments](image1)

![Constraint Settings](image2)
Assign Loads

![Assigning Loads in CAD/CAE Tools](image)

- **Magnitude**: 5
- **Direction**: Normal
- **Time vs. Multiplier Graph**
- **Load Case / Load Curve**: 1
- **Description**: Text field for description

**Multiplier Table Editor**
- **Import CSV**
- **Export CSV**
- **Equation Editor**
- **Print Table**
- **Print Plot**

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Multiplier</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.5</td>
<td>0.06667</td>
</tr>
<tr>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>2.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**Options**
- Follows moving surface
- Vector Selector
Contacts

![Image of CAD/CAE tool settings]

- **Surface-to-Surface Contact**
- **Impact Planes**

**Analysis Parameters - Controls and Parameters for Contact Pair**

- **Parameters**
  - Contact problem type: Low Speed Contact (Press-Fit)
  - Contact method: Frictionless Contact
  - Contact type: Automatic

- **Modeling Friction**
  - Static friction coefficient: 0
  - Sliding friction coefficient: 0
  - Tangential stiffness ratio: 1

- **Tied Contact Options**
- **Slide / No Bounce Contact Options**
  - No bounce
  - No slide
Analysis Parameters

- Duration
- Capture Rate
- Load Curves
Selection Panel

1 Selection Shape
2 Select Type
3 Subentites of Selection
Run Simulation

MES with Nonlinear Material Models

**Analysis Description**
- MES

**Analysis Duration**
- Total: 2 sec
- Extend Total By: 0 sec

**Analysis Configuration**
- Target Computer: Computer
- Monitor Rate: 2 sec
- Number of threads/cores: Automatic

**Analysis Information**
- 3-D surface contact element calculation (Initial), number of elements = 1788
- Nonzero entries initially in upper triangle matrix = 348088
- BCSLIB-EXT Sparse Solver Solution

<table>
<thead>
<tr>
<th>Time</th>
<th>Percent</th>
<th>DT</th>
<th>L Iter.</th>
<th>Residual</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000000</td>
<td>0.00</td>
<td>0.000000</td>
<td>1</td>
<td>0.0000E+00</td>
</tr>
<tr>
<td>0.050000</td>
<td>2.50</td>
<td>0.050000</td>
<td>1</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.050000</td>
<td>2.50</td>
<td>0.050000</td>
<td>1</td>
<td>5.0000E-03</td>
</tr>
<tr>
<td>0.100000</td>
<td>5.00</td>
<td>0.050000</td>
<td>1</td>
<td>1.0000E+00</td>
</tr>
<tr>
<td>0.100000</td>
<td>5.00</td>
<td>0.050000</td>
<td>1</td>
<td>5.0000E-03</td>
</tr>
</tbody>
</table>
Results User Interface

1. Results Contours tab
2. Stress Analysis browser
3. Legend
4. Graphical display

The image shows a graphical representation of stress analysis results with a mesh model and various analysis options available in the user interface. The stress analysis browser and legend are also visible, providing detailed visualizations and data for analysis.
Results Contours / Results Inquire
Graph Results