Week 12 - Lecture
Mechanical Event Simulation
Lecture Topics

• Mechanical Event Simulation Overview
• Additional Element Types
• Joint Component Description
• General Constraint Refresh
• Mesh Control
Force Estimation Methods

• Experience
  – Engineers reply on past experience from similar projects to estimate forces. Often results in an over designed product.

• 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.
Let’s Keep Exploring Simulation
Mechanical Event Simulation (MES) combines kinematic, rigid, and flexible-body dynamics, and nonlinear stress analysis.

- Events with Large Deformations
- Nonlinear Material Properties
- Kinematic Motion
- Forces Caused by Motion
- Stress Results / Motion Forces
MES Use Cases

- Complete FEA on part throughout complete loading cycle.
- Conduct a impact test on a product.
- Validate the motion of flexible body components.
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
MES High Level Process

• Setting up the Model
  – Setup the complete physical event within the model using loads, boundary conditions, contact, and etc.

• Analyzing the Model
  – Solve the simulation frame by frame for the duration of the event.  (Note: Computer Resources Required)

• Results Evaluation
  – Examine the results of the analysis throughout the timeline of the study.
What is Being Introduced in MES

- Additional Element Types
- Joint Components
- Load Curves
- Gravity
Additional Element Types

The following additional element types apply to the MES environment.

– Beam / Truss
– Spring
– Actuator
– Pulley
– Slider
– 3-D Kinematic
3D Kinematic Element

3D Kinematic elements do not experience strains and as a result do not report or calculate stresses.

- Elements Have Mass
- Apply Loads
- Experience Motion
- Acts like a Rigid Body
Spring Element

A spring element can connect two nodes on parts with a defined stiffness value.

- Axial or Rotational
- Displays as a Line or 3D Object
Slider Element

The slider element is used to translation a single node along a defined axis.

– Requires 3 Connection Nodes
– Node 3 Slides between Node 1 and 2
– Two Lines must be Connect and Parallel
– Slider Elements can Attach to Beam, Truss, 2D, Brick
**Actuator Element**

An actuator element is a two-node element that can either change length or rotate during an analysis.

- Uses a load curve to control the action.
Pulley Element

The pulley element is used to represent simple pulleys to rotate objects in most cases.

- Consists of Three Nodes (Driver, Pivot, and Slack)
- Can Stay Stationary
- Can Move with Other Parts
Joint Component

Joint components are used to simulate joints and pinned connections for rotational purposes.

- **Pin Joint**
  - Lines to Axis Endpoints

- **Universal Joint**
  - Lines to Axis Midpoint
Pin Joints

• Used instead of modeling a true 3D pin part to allow rotation about the defined axis.

• Creates truss elements that use pinned connections to allow the bodies to rotate.
Universal Joint

• Creates a part that allows the model to rotate about the axis and swivel about the center point of the axis.

• All of the space truss elements connect to the center.
General Constraint Boundary Condition

The General Constraint establishes boundary conditions to constrain DOF’s.

Note:
Similar to the Autodesk Inventor Professional standard joint features in Dynamic Simulation.
Model Mesh Control

- CAD model mesh options can be controlled at a part level override.
- This provides the ability to streamline the mesh for the most optimal results for larger assembly based models.
Computer-Cluster Projects (CP12)
Guided Lab Project 1

Guides instructions for transferring loads from a Dynamic Simulation to a linear FEA.
Guided Lab Project 2

Guided instructions for simulating a piston and crank using Mechanical Event Simulation (MES) with nonlinear materials.
Guided Lab Project 3

Guided instructions for completing a MES analysis with additional element types and joints.
Problem Set Assignment

Conceptual Design Validation of Crank Slider Mechanism
Demo Topics
Exporting Dynamic Simulation Results

- Time (s) | Force (Revolution) | Force (Welding) |
- 1.45000  | 161.21000         | 80.39210       |
- 1.46667  | 161.20900         | 80.39160       |
- 1.48333  | 161.19900         | 80.38640       |
- 1.50000  | 161.17923         | 80.37650       

- Second Arm: 1
- Revolution: 3 (Main Arm asm: 1, Second Arm: 1)
- Welding: 4 (Pin: 1, Second Arm: 1)
- Welding: 5 (Upper_Pin_asm: 1, Second Arm: 1)
Create Simulation with Motion Loads

Create New Simulation

Name: Second Arm Analysis
Design Objective: Single Point

Simulation Type

Select: Static Analysis

- Detect and Eliminate Rigid Body Modes

- Separate Stresses Across Contact Surfaces

- Motion Loads Analysis

Model State

Part: Second Arm: 1
Time Step: t: 1.45
Pin and Universal Joints
Truss Elements

Element Definition - Truss

General Settings
- Material model: Linear
- Cross-sectional area: 1 in²
- Dashpot Coefficient: 0 lbf s/in
- Analysis Type: Large Displacement
Beam Elements

![Element Definition - Beam](image)

- **Material model**: Isotropic
- **Section Type**: Pre-defined
- **Stress Update Method**: Generalized Mid-Point
- **Parameter for Generalized Mid-point**: 1
- **Analysis Type**: Large Displacement

<table>
<thead>
<tr>
<th>Layer</th>
<th>A (in²)</th>
<th>J1 (in⁴)</th>
<th>I2 (in⁴)</th>
<th>I3 (in⁴)</th>
<th>S2 (in²)</th>
<th>S3 (in²)</th>
<th>S_{a2} (in²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.78539</td>
<td>0.0981746</td>
<td>0.0490873</td>
<td>0.0490873</td>
<td>0.0981746</td>
<td>0.0981746</td>
<td>0.696</td>
</tr>
</tbody>
</table>

[Image of the software interface showing beam element properties and calculations.]
Rotational Prescribed Displacement

Type: Rotation
Magnitude: 1 revolution
Coordinate System: Global (Default)
Direction: Scalar X
Scalar X: 0
Scalar Y: 0
Scalar Z: 0

Analysis Parameters - Prescribed Motion

Index | Birth Time (s) | Death Time (s) | Rebirth Index
--- | --- | --- | ---
1 | 0 | 1 | 3

Add Row | Delete Row | Import | Export
OK | Cancel | Help | Reset From Model | Reset From Default
Part Mesh Settings

Mesh size
- Size: 4 in
- Type: Absolute mesh size

Retries
- Number of retries: 6
- Retry reduction factor: 0.75

Generate 2nd order elements
Draw Line
Actuator Elements

![Element Definition - Actuator](image-url)

- **Type of actuation**: Distance (displacement)
- **Specified length (load) curve number**: 1
- **Load Curve Multiplier**: 1