Week 12 - Lecture Mechanical Event Simulation

ME 24-688 Introduction to CAD/CAE Tools

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



ME 24-688 Introduction to CAD/CAE Tools

Mechanical Event Simulation (MES)

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



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

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



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



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.



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 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.

Creating 1 Surface Boundary Condition Object						
Constrained DOFs Tx Ty Ty Tz Rx Ry Rz	Predefined Fixed Free No Translation No Rotation	X Symmetry Y Symmetry Z Symmetry	X Antisymmetric Y Antisymmetric Z Antisymmetric			
Coordinate System:	Global (Default)					
Description						
OK Cancel						

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)

ME 24-688 Introduction to CAD/CAE Tools

Guided Lab Project 1

Guides instructions for transferring loads from a Dynamic Simulation to a linear FEA.



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





ME 24-688 Introduction to CAD/CAE Tools

Demo Topics

ME 24-688 Introduction to CAD/CAE Tools

Exporting Dynamic Simulation Results

Y	Time (s)	Force (Revoluti	Force (Welding:	
Ø	1.45000	161.21000	80.39210	80
	1.46667	161.20900	80.39160	80
	1.48333	161.19900	80.38640	80
	1.50000	161.17923	80.37650	80



FEA Load-Bearing Faces Selection				
Parts Selected for FEA				
① Second Arm: 1				
۰				
Joints to complete: Load bearing				
Revolution:3 (Main Arm asm: 1, Second Arm: 1) Welding:4 (Pin: 1, Second Arm: 1)				
Automatic Face Selection				
OK Cancel				

Create Simulation with Motion Loads

Create New Simulation	
Name: Design Objective:	Second Arm Analysis Single Point
Simulation Type Model Static Analysis	ate te Rigid Body Modes
Separate Stresses Motion Loads Anal Part Second Arm: 1	Across Contact Surfaces ysis Time Step t: 1.45

ME 24-688 Introduction to CAD/CAE Tools

Pin and Universal Joints

Mesh Dr	aw Setup	o Ana	ilysis Sela
			f_x
Generate 3D Mesh 3D Mesh Settings	Generate 2D Mesh	Joint	Inventor Parameters
Mesh 🔻			CAD Ad

Create Joint						
Participating surfaces						
			Add			
			Remove			
Joint Pin Joint (li	nes to axis endpo	ints)	-			
Pin Joint (lir Aut <mark>Universal J</mark>	nes to axis endpo pint (lines to axis	ints) midpoint)				
🔘 Manual axis/ce	nter-point specifi	cation				
	X (in)	Y (in)	Z (in)			
Axis endpoint 1	0	0	0			
Axis endpoint 2	0	0	0			
	ОК	Cancel	Help			

Truss Elements

Element Definition - Truss		? ×
General Advanced		and the second se
General Settings		
Material model	Linear 👻	
Cross-sectional area	1 in ²	
Dashpot Coefficient	0 Ibf·s/in	
Analysis Type	Large Displacement	
		Reset From Model
OK Cancel	Help	Reset From Default

Beam Elements

Element Definition - E	Beam	? ×
General Thermal	Advanced	R
General Settings		Territori
Material model	Isotropic	
Section Type	Pre-defined	
Stress Update Me	ethod Generalized Mid-Point	
Analysis Trans		
Analysis Type		
Layer A	Sectional Properties (irr²) J1 (in^4) I2 (in^4) I3 (in^4) S2 (irr²) S3 (irr²) Sa2 (78539: 0.09817468 0.04908734 0.04908734 0.0981744 0.0981744 0.696	
	•	
Ітроп	Cross-Section Libranes	Reset From Model
	OK Cancel Help	Reset From Default

Rotational Prescribed Displacement

Setup	Analysis	Selection	View	Tools	Getting St	arted	(Creating 2 Nodal Prescrib	ed Displacement	? X
ontact	≕ ?	\bigcirc	"⇒¤ Pres	cribed D	isplacement	+		Туре		
	General Pi Constraint	n Constrain	t			Fo		Translation	Rotation	<u> </u>
		Con	straints 🔻					Magnitude		
								1		revolutions
Analysis P	arameters - MES: Pre	scribed Motion				? x		Coordinate System		
Active F	Ranges	Birth	and Death Time					Global (Default)		
	Index Birth Time (s 1 0) Death Time (s) 1	Rebirth Inde 0	ex				Direction		
								🔶 🍥 Scalar X 🛛 X 🔍 0)	
								Scalar Y Y 0)	
	Add Row	Delete Row			Import	Exporte				
ОК	Cancel	Help	Re	set From Mode	Reset F	rom Default	ļ			

Part Mesh Settings



Draw Line

MP	Mesh	Draw	Setup	Analysis	Selection
Line	Circle	- Construe	🔶 ction Vert	ex	ngle 🔪
		Dra	aw 🕶		

Actuator Elements

Element Definition - Actuator		? ×
Type of actuation	Distance (displacement)	
Displacement Rotation C	onstraints Advanced	· · · ·
Time-Dependent Specified I Specified length (load) cu Load Curve Multiplier	Length urve number 1 1 in	
ОК	Cancel Help	Reset From Model Reset From Default