Flexible Multibody Simulation Models For Load and Durability Calculation

Stefan Dietz
What is Multibody System Simulation?

- Multibody Simulation (MBS) is used to simulate, predict and optimize the motion of any mechanical system.
- Component loads can be used as input to durability calculation.
Flexible Bodies - A Variety of Choices

Small Deformation
- automated setup possible
- stress & fatigue of components
- noise & vibration
- detailed fluid structure interaction

Large Deformation
- reduced order representation based on Abaqus models
- composite beam technology

General Physics
- Abaqus Standard & Explicit
- collision analyses
- plastic deformation

Reduced order models

Abaqus

Co simulation

logical scale

physical scale
1) Prepare flexible body with Abaqus

2) Run Simpack time integration

3) Export Component Loads to Abaqus

4) Pass Loads to Abaqus

5) Pass stress calculated with Abaqus to fe-safe

- Nonlinear structural behavior may be taken into account
- Often quasi static analysis (eigen dynamics gets lost)
- Longer runtime due to FE-analyses of the unreduced FE-model

Loads Data Generation Vs. Modal Stress Recovery
Loads Data Generation Vs. Modal Stress Recovery

Modal Stress Recovery

1) Prepare flexible body with Abaqus
   - Slightly different reduction
   - Export of stress/strain recovery data

2) Run Simpack time integration
   - Request output of fe-safe data

3) Pass Simpack results to fe-safe
   - .txt
   - .ldf
   - .stlx

4) Stress recovery in fe-safe
   - Fast computation of stresses for scenarios having a huge number of output time points
   - Limited to linear structural behavior

How accurate is modal stress recovery?
Flexible Bodies in Multibody System Models

- Reduced order model (ROM) representations of FE-models are essential for multibody simulation
- How can we retain the essential information of a high fidelity FE-model in a straightforward way?

$$\mathbf{u}(\mathbf{x}, t) = \sum \Psi_i(\mathbf{x}) q_i(t)$$

- Selecting modes means to setup the ROM's deformation space
How Do We Setup the ROM‘s Deformation Space?

<table>
<thead>
<tr>
<th>FE Properties</th>
<th>Mass Properties</th>
<th>Position</th>
<th>Modes</th>
<th>Options</th>
<th>Loads</th>
<th>Training</th>
</tr>
</thead>
</table>

Eigenmode: f-min, n-modes

Interface modes: IRM

- FE-model
- MBS with eigenmodes
- MBS with eigenmodes + interface modes

frequency response [m]

f_max/1.5

f_max

frequency [Hz]
How Do We Setup the ROM's Deformation Space?

3 coupling points * 6 directions at each point = 18 unit loads

- As soon as Interface Modes are activated Simpack automatically adds an appropriate deformation shape to the flex. body representation
- While we add/remove force elements, constraints or joints Simpack updates the set of interface modes accordingly
Stress Recovery From ROM’s – Input to Durability

- Comparison of Different Methods

<table>
<thead>
<tr>
<th>Number of modes</th>
<th>10</th>
<th>15</th>
<th>20</th>
<th>25</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max Frequency [kHz]</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>65.2*</td>
<td>105.3 *</td>
<td>232.9</td>
<td>257.1</td>
<td></td>
</tr>
<tr>
<td>Normal Stress X</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>14.508</td>
<td>18.602</td>
<td>0.082</td>
<td>0.056</td>
<td></td>
</tr>
<tr>
<td>Y</td>
<td>70.674</td>
<td>78.889</td>
<td>1.327</td>
<td>1.029</td>
</tr>
<tr>
<td>Z</td>
<td>346.708</td>
<td>320.970</td>
<td>2.250</td>
<td>1.612</td>
</tr>
<tr>
<td>Shear Stress XY</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>80.358</td>
<td>29.283</td>
<td>4.678</td>
<td>3.315</td>
<td></td>
</tr>
<tr>
<td>YZ</td>
<td>571.447</td>
<td>484.255</td>
<td>9.623</td>
<td>7.150</td>
</tr>
<tr>
<td>XZ</td>
<td>80.250</td>
<td>29.205</td>
<td>4.592</td>
<td>3.232</td>
</tr>
</tbody>
</table>

- Simpack’s combination of eigenmodes and interface modes yields accurate stress with the least number of modes

* error in % at the critical spot
Stress Recovery & Durability

- Interface mode+eigenmode based substructuring in Abaqus
- All data (including strain + stress recovery matrices) is transferred through the FBI file
  - Greatly simplifies the entire workflow
- FBI file directly generated by Abaqus

Abaqus  ➔  Simpack
Stress Recovery & Durability

- Input to fe-safe can be generated during the measurement run

- fe-safe reads modal stress from Abaqus via Simpack and time signals from Simpack generated “history”

Abaqus → Simpack → fe-safe
Stress Recovery & Durability

Simpack generates for fe-safe:
- .txt: time history of substructure coordinates
- .ldf: assignment to stress recovery columns
- .stlx: project settings file
Stress Recovery & Durability

Postprocessing of Stress Results in Abaqus

Fe-safe

Fe-safe results go back to Simpack
Stress Recovery & Durability

Workflow Automation

Workflow automation in Isight – In future with Simpack
POWER’BY this will be done with Process Composer

- Rpm range
- Simpack
- Simpack Time Integration
- Fe-safe

FKM utilization factors

DOE

Analysis of a Crankshaft
Flexible Multibody Simulation Models For Load and Durability Calculation

Conclusions

► Component Load Generation with Simpack
  ▶ Useful for nonlinear FE-Model
  ▶ When stress for a very limited number of time steps is to be investigated
  ▶ Longer runtimes for very large numbers of time steps
  ▶ Currently quasi-static

► Modal Stress Recovery
  ▶ Efficient, i.e. short runtimes
  ▶ Accurate if reduction is carried out appropriately (see templates in the Simpack documentation)
  ▶ Suitable for long time histories
  ▶ Limited to linear structural behavior
IF WE™