Dynamic and Impact Analysis of Aerospace Vehicles Using ABAQUS/Explicit

Presented at the
2004 FEMCI Workshop
NASA/GSFC, Greenbelt, MD

Kyle C. Indermuehle
Product Management – Aerospace Applications
ATA Engineering -- ABAQUS
Aerospace Vehicles are Complex Systems With Numerous Different Analyses that Need to Be Performed

• Aerospace systems are exposed to various loading conditions that all need to be fully analyzed
  – Static, dynamic, thermal, acoustic, operational

• Typical satellite analyses include:
  – Dynamic analysis for shipping, launch, and operation
  – Detailed component stress and margin calculations
  – Mechanism analysis for deployment of solar panels and reflector
  – Thermal analysis for in-orbit operation

• These analyses are typically performed in the linear domain
  – Often the modal domain for dynamic problems

• But, there are some load cases that cannot be analyzed linearly…
Impact is a Highly Nonlinear, Dynamic Event

• Example problem: Satellite impact with ground

**Engineering Challenges**

• Complicated, nonlinear, dynamic event with many contact regions and damage / failure

**Goals of analysis**

• Determine forces on satellite caused by impact
• Determine if components / joints failed and correlate to actual results
• Determine peak component accelerations
Disclaimer

• This paper is a discussion of methodology and the latest simulation capabilities

• The impact event is used simply as an example
  – Simulation is based solely on information implied from the image in AWST
  – A simple, generic satellite is used for the analysis

• The methodology and capabilities discussed have been used on other similar analyses
**Methodology for Impact Analysis is to Start Simple and Add Increasing Complexity**

**Methodology**
- Perform simple rigid body / mechanism dynamic analysis
  - Allows for quick insight into event
- Perform flexible body impact dynamic analysis
  - More accurate simulation of true event
- Perform flexible body impact analysis with failure models
  - Accounting for joint / material failure further increases accuracy of simulation

**Workflow**
- Translate NASTRAN FEM to ABAQUS
- Perform rigid body / mechanism dynamic analysis
- Perform flexible body dynamic analysis using ABAQUS /Explicit
- Perform flexible body analysis with failure using ABAQUS /Explicit
NASTRAN Models Can Easily Be Translated into ABAQUS Using ABAQUS fromnastran Utility

NASTRAN Bulk Data processed: (no errors in translation)

CBAR, CONM2, CORD2R, CQUAD4, CTRIA3, GRID, MAT1, MAT2, PBAR, PSHELL, RBAR, RBE2, RBE3

<table>
<thead>
<tr>
<th>Mode</th>
<th>ABAQUS</th>
<th>Nastran CQUAD4</th>
<th>Difference</th>
<th>Nastran CQUADR</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.00</td>
<td>10.66</td>
<td>-3.2%</td>
<td>10.82</td>
<td>-1.7%</td>
</tr>
<tr>
<td>2</td>
<td>11.05</td>
<td>10.71</td>
<td>-3.2%</td>
<td>10.86</td>
<td>-1.7%</td>
</tr>
<tr>
<td>3</td>
<td>17.39</td>
<td>16.43</td>
<td>-5.9%</td>
<td>16.63</td>
<td>-4.6%</td>
</tr>
<tr>
<td>4</td>
<td>18.53</td>
<td>17.56</td>
<td>-5.5%</td>
<td>17.69</td>
<td>-4.7%</td>
</tr>
<tr>
<td>5</td>
<td>28.42</td>
<td>28.16</td>
<td>-0.9%</td>
<td>28.31</td>
<td>-0.4%</td>
</tr>
<tr>
<td>6</td>
<td>28.54</td>
<td>28.29</td>
<td>-0.9%</td>
<td>28.44</td>
<td>-0.4%</td>
</tr>
<tr>
<td>7</td>
<td>33.08</td>
<td>31.33</td>
<td>-5.6%</td>
<td>32.30</td>
<td>-2.4%</td>
</tr>
<tr>
<td>8</td>
<td>33.36</td>
<td>31.38</td>
<td>-6.3%</td>
<td>32.63</td>
<td>-2.2%</td>
</tr>
</tbody>
</table>
Impact Analysis:  Step 1—Prepare Model

** Rigid ground
*NODE, NSET=ALLNODES, SYSTEM=R
  58179, 10.000000E+01,-2.500000E+02,-2.200000E+02
  58180, 10.000000E+01, 5.000000E+01,-2.200000E+02
  58181, 10.000000E+01,-2.500000E+02, 8.000000E+01
  58182, 10.000000E+01, 5.0000000E+01, 8.0000000E+01
*ELEMENT, TYPE=S4R, ELSET=ground
  56720, 58179, 58180, 58182, 58181
*rigid body, ref node = 58180, elset=ground, position=input

Definition of contact surfaces

*surface, name=ssat, type=node
  sat_box
*surface, name=sground, type=element
  ground, neg
*contact pair, cpset=ct_sat, interaction=sat_int, mechanical constraint=penalty
  ssat, sground
*surface interaction, name=sat_int
*friction
  0.8,
*contact damping, definition=damping coefficient
  0.05,
*contact controls, cpset=ct_sat
Impact Analysis: Step 2—Define Event Excitation

Simulation is of rotation of mounting plate

```plaintext
*step
  *dynamic, explicit
    , 10.0
  ** fix ground, dolly, and mount
  *boundary
    58180,1,6,0.0
    48480,1,6,0.0
    48326,1,5,0.0
  **define rotation
  *boundary, amplitude=rotation, type=displacement
    48326, 6, 6, 1.0
  *amplitude, name=rotation, definition=tabular
    0.0, 0.0, 3.0, 0.4
  *dload
    all_massive_elements, grav, 386.088, 1.0, 0.0, 0.0
*end step
```
Impact Analysis: Step 3—Run Simulation

- Model is ready to run
  - Full FEM translated from NASTRAN
    - Assumption is that this is a legacy model
  - Rigid ground, dolly, and mount defined
  - Contact surfaces and surface friction defined
  - Event excitation defined

- Initial estimation of stable time increment is 1e-8 seconds
  - For 10-second simulation this means 1e9 time steps
  - Run will take 4+ hours
Impact Analysis: Step 3a—Run Rigid Body Simulation

• Define satellite as a rigid body
  
  *rigid body, ref node=27329, elset=sat

• Define /Explicit analysis
  
  *dynamic, explicit, direct user control
  
  0.01, 10.0

• Run rigid body simulation
  
  – Time increment is now 1e-2 seconds
  – For 10-second simulation this means 1e3 time steps
  – Run will take 1 minute

  abaqus job=rigid_body double
Impact Analysis: Step 3a—Postprocess Rigid Body Simulation

- Rigid body analysis provides quick insight into the event
  - *RIGID BODY makes FE mesh a rigid body
  - Fast run time (1 minute for this model on a laptop)
    - Easy to verify and debug model
  - Provides insight such as displacement, acceleration, contact forces
Impact Analysis: Step 3b—Run Flexible Body Simulation

• Remove rigid body definition for satellite
  
  **rigid body, ref node=27329, elset=sat

• Define ABAQUS/Explicit analysis
  
  *dynamic, explicit
  
  , 10.0

• Run flexible body simulation
  
  – Time increment is now 1e-8 seconds
  – For 10-second simulation this means 1e9 time steps
  – Run will take 4+ hours
  – Parallel processing can be used to reduce run to 2 hours

  abaqus job=rigid_body cpus=2
Impact Analysis: Step 3b—Postprocess Flexible Body Simulation

- Flexible body simulation
  - Same model; run file as rigid body analysis, just removed *RIGID BODY from input file
  - Analysis time now over 4 hours on 3 GHz PC for half of the event
    - Time reduced to 2 hours using parallel processing
  - Can recover displacement, acceleration, contact forces, element forces and stresses
Impact Analysis: Step 3c—Run Flexible Body Simulation with Component Failures

- Add **CONNECTOR FAILURE** to connector definitions

```
**connector element connection 27252 to 127252 (antenna mass joint)
*element, type=conn3d2, elset=antenna_joint
  127252, 127252, 27252
*connector section, elset=antenna_joint, behavior=antenna_behav
  weld
*connector behavior, name=antenna_behav
*connector failure, component=1, release=all
  ,,-5000,5000
*connector failure, component=2, release=all
  ,,-5000,5000
*connector failure, component=3, release=all
  ,,-5000,5000
*connector failure, component=4, release=all
  ,,-5000,5000
*connector failure, component=5, release=all
  ,,-5000,5000
*connector failure, component=6, release=all
  ,,-5000,5000
```
Impact Analysis: Step 3c—Postprocess Flexible Body Simulation with Component Failures

- Failure models include
  - Force overload, peak displacement, material plasticity, laminate failure, ABAQUS user subroutines
- Simulation accurately reflects the change in the structure
Comparison of Rigid Body, Flexible, and Flexible with Failure Shows Increased Accuracy

• Comparison of responses shows flexible body model has responses 10% higher than rigid body

Time response

Shock response spectra
Methodology Used for Analysis of Satellite Impact Was to Start Simple and Add Increasing Complexity

• Methodology
  – Use existing loads / dynamics model for analysis
    • Can translate from NASTRAN using fromnastran utility
  – Define impact analysis
    • Define ground, dolly, and mount (rigid)
    • Define contact surfaces
  – Perform simple rigid body/mechanism dynamic analysis
    • Allows for quick insight into event
  – Perform flexible body impact dynamic analysis
    • More accurate simulation of true event
  – Perform flexible body impact analysis with failure models
    • Accounting for joint/material failure further increases accuracy of simulation
Simulation of Satellite Impact for Varying Model Fidelity Allows Progressive Accuracy and Insight into Event

**Engineering Challenges**
- Complicated, nonlinear, dynamic event with many contact regions and damage / failure

**Simulation Capabilities**
- Able to use existing FE model for analysis (typically a Nastran CLA model)
- Easily change from flexible to rigid body analysis
- Robust, general contact algorithm
- Nonlinear material properties and failure criteria
- Parallel processing to reduced run time
- Mechanism – flexible body co-simulation
Current Software Technology Provides the Capability to Perform Multiple Simulations in One Toolkit

• **Unified FEA**
  – Fewer software products needed
  – More and smarter reuse of models and results
  – Better technical solution through coupled analysis
  – Reduced data management

• **One FE model and one software code to perform**
  – Dynamic analysis
  – Nonlinear static analysis
  – Mechanism simulations
  – Impact/crash
  – Structural-thermal coupled problems
Satellite Unified FEA

- Global dynamic, component-level stress, mechanism, and impact analysis can all be performed using
  - One code—ABAQUS
  - One FE model—With minor changes (*RIGID BODY, *SUBMODEL, *COMPONENT FAILURE)
Satellite Dynamic Analysis

• Example problem: Analysis of launch vehicle loads on satellite

**Engineering challenges**
- Solving for modes of a complicated often large FE model
- Definition of dynamic environment
- Output of many responses
- Graphically viewing responses

**ABAQUS solutions**
- Efficient Lanczos solver
- Straight forward definition of excitation environment
- ELSET and NSET definition for groups of output entities
- Postprocessing is easy using ABAQUS/Viewer

![Frequency Response Function (FRF)](image)
Satellite Component Analysis

• Example problem: Stress analysis of brackets using Submodeling

**Engineering challenges**
- Multiple static load cases
- Possible material nonlinearities
- Thermal loads
- Easy, visual postprocessing of results

**ABAQUS solutions**
- Submodeling capability for easy analysis
- Can efficiently analyze many load cases using perturbation analysis
- Advanced FEA capabilities include material nonlinearity and nonlinear geometry effects
Satellite Mechanism Analysis

• Example problem: Deployment of solar panels

**Engineering challenges**
- Mechanism analysis
- Need to understand forces and stresses due to deployment
- Flexibility of panels is important to analysis—rigid body simulation is not sufficient

**ABAQUS solutions**
- ABAQUS can solve the coupled mechanism-flexible body problem, including nonlinear effects
Satellite Mechanism Analysis

• Animation of deployment
Dynamic and Impact Analysis of Aerospace Vehicles using ABAQUS/Explicit

Presented at the 2004 FEMCI Workshop
NASA/GSFC, Greenbelt, MD

Kyle C. Indermuehle  ATA Engineering / ABAQUS  858.792.3958
Mike Sasdelli  ABAQUS East  410.420.8587