Chapter 8: Analysis Setup

Setting up Loading Conditions
Formatting models for Analysis
Analysis Setup: What is it?

- Definition of all information for an analysis besides the mesh
  - Specification of solver to be used
  - Creation materials, properties, etc.
  - Assignment of a solver specific format to HyperMesh entities
  - Creation boundary conditions (constraints, loads, contacts, etc.)
  - Definition of other required information (solution requests, general run parameters, etc.)
Analysis Setup: HyperMesh Capabilities

• HM is a “solver neutral” pre-processor
  • Works with many different solvers
  • Can convert between supported solvers
  • Capable of assembly from input files of different solvers
  • Can be customized to support other solver codes

• Can set up many types of analysis
  • Structural (Stress, NVH, Durability, Non-Linear Structural)
    • Radioss (Linear), Abaqus, Nastran, Ansys, Marc, nSOFT
  • Manufacturing (Flow / Mold-Filling, Extrusion)
    • Moldflow, CMold, HyperExtrude
  • Safety (Impact / Crash, Occupant Safety)
    • Dyna, Pamcrash, Radioss, Madymo
  • Optimization (Topology, Topography, Shape, Size / Gauge)
    • OptiStruct, Nastran
HyperMesh interacts with many solvers
  - Each solver has its own unique formats, terminology, etc.
  - Example: compare nodes and elements in Abaqus and OptiStruct / Nastran
    - 3 nodes
    - 2 quad elements
    - Format / structure is obviously different

<table>
<thead>
<tr>
<th>Radios (Linear)</th>
<th>Abaqus</th>
</tr>
</thead>
<tbody>
<tr>
<td>GRID 1</td>
<td>*NODE</td>
</tr>
<tr>
<td>GRID 2</td>
<td>1, 0.0  , 1.0 , 0.0</td>
</tr>
<tr>
<td>GRID 3</td>
<td>2, 0.0  , 0.0 , 0.0</td>
</tr>
<tr>
<td>CQUAD4</td>
<td>3, 1.0  , 0.0 , 0.0</td>
</tr>
<tr>
<td>CQUAD4</td>
<td>*ELEMENT,TYPE=S4,ELSET=part_1</td>
</tr>
<tr>
<td>1, 1, 1, 2</td>
<td>1, 1, 2, 3, 4</td>
</tr>
<tr>
<td>2, 3, 4</td>
<td>2, 3, 4, 5, 6</td>
</tr>
</tbody>
</table>

Radios (Linear) and Abaqus examples are shown for comparison. The format and structure of the input data are different even though the geometry is the same.
Solver Formats: HyperMesh “Templates”

- HyperMesh can interact with different solvers by using “templates”
  - The selected template tells HyperMesh what solver the model is for
  - The template also tells HyperMesh how entities are formatted for that solver
    - Each entity may have several available formats for that solver
    - Each format has fields that make up its definition
    - These fields may need to have values entered by the user
  - Example: a component for Radioss (Linear) can be a PSHELL or PSOLID format
    - PSHELL: defines shell elements, ID = 1, material = 1, thickness 5.0
      - PSHELL
        | PSHELL | 1 | 15.0 | 1 | 1 | 0.0 |
    - PSOLID: defines solid elements, ID = 2, material = 1
      - PSOLID
        | PSOLID | 2 | 1   | 0 |
Solver Formats: Solver Formats for Loads

- Loads format is specified by setting a “load type”

<table>
<thead>
<tr>
<th>HyperMesh Load Configuration:</th>
<th>Example solver keywords available as element types:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Radioss (Linear)</td>
</tr>
<tr>
<td>Constraint</td>
<td>SPC, SPCD, ASET, ASET1, SUPPORT, QSET1, etc.</td>
</tr>
<tr>
<td>Force</td>
<td>FORCE</td>
</tr>
<tr>
<td>Pressure</td>
<td>PLOAD, PLOAD2, PLOAD4, QBDY1</td>
</tr>
</tbody>
</table>
Solver Formats: Tools

- **Preferences > User Profiles...**
  - Loads the appropriate template for that solver
    - Template can also be set manually
      - *File > Load > Solver Template*
  - Sets the *File > Import > Solver Deck File type* field to the appropriate solver
  - Loads a macro menu with tools specific to working with that solver
  - Customizes the HyperMesh menu
    - Removes panels that are not used with that solver
    - Removes controls inside a panel that are not used with that solver
    - Renames some panels & controls in panels to match solver terminology
**Solver Formats: Tools**

- **Collectors** pull-down and **collectors** panels
  - **Create**
    - Assign a card image to the collector being created
    - Edit the card image fields if desired
    - Assign a material to the collector being created
  - **Update**
    - Assign a material to an existing collector
    - Assign and/or edit a card image of an existing collector

- **Elem types > load types** panels
  - Set a “current element / load type” for an element / load configuration
  - Any new elements / loads of that configuration created will have that element / load type
  - Change the element / load type of existing elements / loads
Solver Formats: Tools

- **Collectors > Card Edit** or toolbar >  
  - View / edit the card image of any entity in the model  
  - Includes entities that are not collectors (nodes, elements, loads, etc.)

- **Model Browser**  
  - Right click a collector and select **edit card**  
  - View / edit the card image of the selected collector

- **Preferences > graphics**  
  - **template labels type**  
    - Activate the graphic displayed names of entities in solver (template) terminology rather than HyperMesh (solver neutral) terminology  
    - Helps keep track of what is in the model
Solver Formats: Tools

- **Solver Browser**
  - Displays solver-based cards in a tree format
  - Uses organization & structure of the represented solver
  - Performs basic actions involving cards
    - Create new cards
    - Delete existing cards
    - Edit attributes of existing cards
  - Display controlled in the menu bar:
    View > Browsers > HyperMesh > Solver
Solver Formats: Tools

- **Summary** panel
  - Displays a text window with various types of information about the model
  - Helps to review the model and make sure all information has been entered properly

<table>
<thead>
<tr>
<th>Summary Type</th>
<th>Included Information</th>
</tr>
</thead>
<tbody>
<tr>
<td>Components</td>
<td>Component Name, ID, Material Name, Thickness, Mass,</td>
</tr>
<tr>
<td></td>
<td>#Elements</td>
</tr>
<tr>
<td>Center of Gravity</td>
<td>Component Name, ID, Mass, X, Y, Z</td>
</tr>
<tr>
<td>Elements</td>
<td>Type of Elements, Element Configurations</td>
</tr>
<tr>
<td>Error Checks</td>
<td>Load Collector, Load Steps, Components</td>
</tr>
<tr>
<td>Loads</td>
<td>Load Collector, ID, FX, FY, FZ, Magnitude</td>
</tr>
<tr>
<td>Moment of Inertia</td>
<td>Moment of Inertia</td>
</tr>
</tbody>
</table>
Solver Formats: Process

1. Create the entities needed for your model
   - Keep in mind what is needed for the solver and analysis being used
   - Entities need to be properly organized in collectors
     - All entities in a collector share the same attributes

2. Load the proper card image or type where needed
   - Generally use the Setup/collectors, elem type, or load type panel
   - Elements and loads will always have a type
   - Sometimes collectors may not need a card image
3. Enter values in the card images as required
   • Use Collectors > Card Editor panels to check card images of all collectors
   • Some card images require other entities to be selected as a reference
     • Example: Dyna requires a component’s card image to point to a property collector for thickness information, etc.
   • The goal in formatting for analysis is:
     • All entities have the proper formats (card image / type)
     • Card images of all entities have necessary information entered
   • Understand the details of how HyperMesh interacts with your solver
     • Refer to the External Interfacing portion of online help for details
     • Altair has training classes for interfacing with some solvers
     • Contact Altair support for additional questions
Boundary Conditions: Supported Entity Types

- **FE Loading**
  - *Loads* (constraint, force, pressure, moment, temperature, flux, velocity, acceleration)
  - *Equations* (mathematical link between nodes)

- **Contacts**
  - *Group* (defines contact between entities)
  - *Contact Surfs* (defines a list of entities that can be used as master or slave in a group)

- **Output Requests**
  - *Loadsteps* (combinations of load collectors)
  - *Output Blocks* (request output from an analysis for certain entities)

- **Control cards** (job-level, global parameters for the analysis)
Boundary Conditions: Tools

- Analysis page
  - Analysis page is devoted to setting up analyses

<table>
<thead>
<tr>
<th>vectors</th>
<th>load types</th>
<th>interfaces</th>
<th>control cards</th>
</tr>
</thead>
<tbody>
<tr>
<td>systems</td>
<td>constraints</td>
<td>rigid walls</td>
<td>output block</td>
</tr>
<tr>
<td></td>
<td>equations</td>
<td>entity sets</td>
<td>load steps</td>
</tr>
<tr>
<td></td>
<td>forces</td>
<td>blocks</td>
<td></td>
</tr>
<tr>
<td></td>
<td>moments</td>
<td>contact surfs</td>
<td>optimization</td>
</tr>
<tr>
<td></td>
<td>pressures</td>
<td>bodies</td>
<td>Radioss</td>
</tr>
<tr>
<td></td>
<td></td>
<td>nsm</td>
<td>OptiStruct</td>
</tr>
</tbody>
</table>

- User profile macro menus
  - User profiles add macro menu pages with tools specific to that solver
    - **Abaqus** – Step Manager, Contact Manager, Component Browser
    - **Ansys** – Contact Wizard, Component Manager, etc.
    - **LS-Dyna** – Name Mapping, Constrained Rigid Body, Content Table, etc.
    - **Nastran** – Subcase Manager, Part Info, 1D Property Table, etc.
    - **Radioss Bulk / OptiStruct** – Subcase Manager, Component Table, etc.
    - **Radioss Block** – D01 Tool, Sections, Component List, Material table, etc.
    - etc.
Boundary Conditions: Loads on Geometry

- Loads can be created on geometry as well as FE entities
  - Set the entity selector to a geometry entity
  - Create the load
  - Create the mesh
  - Use the load on geom panel to map the loads from the geometry to the elements

Create load on geometry
Create the mesh
Map the load to the mesh