Aspects of pre/post processing a detailed wing FEM with Femap

FEMAP SYMPOSIUM
January 17, Vero Beach, FL

Eric Gustafson, Senior Stress Analyst, SDA – eric@structures.aero
Alex Skavdahl, Stress Analyst, SDA – askavdahl@structures.aero
Aspects of pre/post processing a detailed wing FEM with Femap

- geometry preparation
- meshing practices
- Nastran’s advanced non-linear solver
- the use of built-in tools and development of new ones to interpret results
What are some typical problems faced when generating a wing FEM (or any FEM)?

- Highly detailed geometry from designers is not amenable to direct meshing
- Choices of meshing approach
- A great many unique properties to be assigned
- Handling loads
- How to interpret vast amounts of data quickly for effective decision-making
Rapid wing model pre-processing

Information captured from NX models via Teamcenter (A Siemens product)

- Geometry model
  - Midsurfaces built and assembled from original CAD
- Material and dimension data
  - Documented for each major primary structure component
- < 1 day to gather all necessary information to begin FEM pre-processing
- 1 engineer created properties for all unique zones and created a surface ID vs property ID list
This build is far too complicated for auto-midsurfacing

Simple part CAD
- Automatic midsurfacing is useful for solid monolithic geometry with constant thickness

Aerospace CAD
- Many more solids than what is needed by analyst
- The task of meshing must be bounded with some early geometry preparation
- Early decisions on modeling approach for features
Surface preparation and mesh process

1. Raw geometry from design CAD imported and organized into layers
2. For each surface
   1. **Geometry → Midsurface → Single**
3. All surfaces intersected and combined with **Geometry → Surface → NonManifold Add**
4. Surfaces protruding OML are eliminated
5. Features are imprinted and then cut
   1. E.g. Access, routing, and lightening holes, mouse holes, rib stiffeners

**Diagram:**
- Two adjacent surfaces with two overlapping curves will lead to coincident nodes.
- If the two adjacent surfaces are part of a NonManifold solid, there will be no overlapping curves and thus no coincident nodes.
Surface prep and mesh process
Surface prep and mesh process
Surface prep and mesh process

Geometry → Surface → NonManifold Add operation then performed to merge the surfaces onto the same solid. Surface ID is tracked in Excel file for property assignment.
Organization is key

Layer 1: Upper skin
Layer 2: Ribs
Layer 3: Spanwise
Layer 4: Lower skin

We prefer layers AND groups, the former for quick access to entities representing internal structure, and the latter for creating custom run decks.
Rapid wing model pre-processing

**Surface Object Properties API Tool**

**Purpose**
Rapidly assign property IDs to surfaces.

**Input**
List of surface ID vs property ID.

**Output**
Assigns property ID to surface ID. Also reads size, type, layer, CG location, property ID information.
Rapid Model Generation

- 719 surfaces part of nonmanifold geometry
  - edges curves are shared between adjacent surfaces
- 2 man-weeks to process entire wing
- 212k elements
  - Element count driven by having 4 thru stringer height
  - Mostly 2D plate elements
  - All mesh linked to geometry, eliminates costly manual mesh update operations
- Applying properties was trivial and done programmatically

Final set of surfaces
Rapid Model Generation

- 719 surfaces part of nonmanifold geometry
  - edges curves are shared between adjacent surfaces
- 2 man-weeks to process entire wing
- 212k elements
  - Element count driven by having 4 thru stringer height
  - Mostly 2D plate elements
  - All mesh linked to geometry, eliminates costly manual mesh update operations
- Applying properties was trivial and done programmatically

Final set of surfaces (upper skin removed)
Rapid Model Generation

- 719 surfaces part of nonmanifold geometry
  - edges curves are shared between adjacent surfaces
- **2 man-weeks** to process entire wing
- 212k elements
  - Element count driven by having 4 thru stringer height
  - Mostly 2D plate elements
  - All mesh linked to geometry, eliminates costly manual mesh update operations
- Applying properties was trivial and done programmatically

*Final mesh (no element edges shown)*
Rapid Model Generation

Final mesh (detail)
Rapid Model Generation

Final mesh (no upper skin)
Boundary Conditions

- Originally constrained using CBUSH elements @ root connections
  - Over represented inboard stiffness – poor correlation
- Connected wing to portion of fuselage in loads FEM
  - Better representation of inboard stiffness – better correlation
  - Wing connected using File -> Merge command
File -> Merge is a useful tool for combining 2 separate FEMAP windows
Fuel Pressure Loads

- Needed to spread fuel pressure to various bays shown below
- Typically a job for Femap’s Data Surface tool
  - BUT, no easy way to connect bays to elements
- A custom API was developed taking advantage of still ASSOCIATED mesh / geometry

FEMAP geometric surfaces mapped to their corresponding fuel bays
Tabulated Fuel Bay, Pressure, and Surfaces
Custom FEMAP / Excel API to spread Pressure loads to elements on surfaces
Fuel Pressure Loads – Load Spreading Interface

Coordinate System
Initial Pressure
Delta Pressure
Pressure to Apply
FEMAP Surface IDs

<table>
<thead>
<tr>
<th>Bay</th>
<th>Ccys</th>
<th>P_0</th>
<th>Del_P</th>
<th>Pressure</th>
<th>SIDs</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>1</td>
<td>2.042</td>
<td>35</td>
<td>536</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2</td>
<td>1.933</td>
<td>76</td>
<td>80</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>1.823</td>
<td>177</td>
<td>180</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>4</td>
<td>1.305</td>
<td>705</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>5</td>
<td>1.522</td>
<td>223</td>
<td>225</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>6</td>
<td>1.718</td>
<td>224</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>7</td>
<td>1.240</td>
<td>704</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>8</td>
<td>1.443</td>
<td>271</td>
<td>273</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>9</td>
<td>1.626</td>
<td>265</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>10</td>
<td>1.761</td>
<td>274</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>11</td>
<td>1.889</td>
<td>268</td>
<td>272</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>12</td>
<td>1.170</td>
<td>703</td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>13</td>
<td>1.343</td>
<td>329</td>
<td>330</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>14</td>
<td>1.528</td>
<td>328</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Pseudo-Code:
- Create a Load Set
- For each Fuel Bay
  - Create Load Definition
  - Assign load parameters
    - Coordinate System
    - Pressure value
    - Equation
  - Apply load to elements on selected surfaces
- ~1hr to map surfaces to bays
- 1 time process
- <1 day to Run all cases
FEMAP Symposium 2017

Nonlinear Solution Approach - Motivation

• Why run NL?
  • Linear cannot capture proper internal loading for ribs (Brazier effect)
  • Want to verify behavior of structure up to ultimate is stable analytically and by comparison to test
  • If any instabilities encountered, can show them to match predictions, be absorbable, or critical. Can gain insight into where and why they occurred.
  • Can match strain predictions for cert and/or determine reserve strength
Nonlinear Solution Approach - Types

<table>
<thead>
<tr>
<th>Nastran SOL</th>
<th>Femap “Analysis Type”</th>
<th>Exec</th>
</tr>
</thead>
<tbody>
<tr>
<td>106</td>
<td>10..Nonlinear Static</td>
<td>106</td>
</tr>
<tr>
<td>129</td>
<td>12..Nonlinear Transient Response</td>
<td>129</td>
</tr>
<tr>
<td>601</td>
<td>22..Advanced Nonlinear Static</td>
<td>601,106</td>
</tr>
<tr>
<td>601</td>
<td>23..Advanced Nonlinear Transient</td>
<td>601,129</td>
</tr>
<tr>
<td>701</td>
<td>24..Advanced Nonlinear Explicit</td>
<td>701</td>
</tr>
</tbody>
</table>

- Implicit vs Explicit
  - All solutions undertaken were IMPLICIT as this is a static event with a slowly loaded structure
  - Explicit is better for dynamic events but has restrictions
    - Time step requirements would require an inordinate amount of run time.
    - No pcomps
    - Interesting feature unexplored is implicit/explicit switching
  - 106 vs 601
    - Adv NX Nastran based on ADINA code, is complementary to regular NX Nastran
    - 601 solver has more robust solution methods for getting through much more severe behavior such as large strains (>10%), contact, post-buckling, element birth/death, etc.
Highly NL solutions can be plagued with convergence issues.

Lack of convergence arises when externally applied forces are not balanced by nodal forces within the convergence tolerances.

This can stem from the solution “getting ahead of itself”. Most often occurs as a result of large jumps in stiffness, such as when a skin or stringer component buckles. User may wish to use smaller loading increments, or have the solution automatically decrease magnitude of load step.
NL Solution Approach – SOL 601 Parameters

PARAM, LGDISP, 1
- Enabled on the NASTRAN Bulk Data Options analysis dialog
- For large displacements (geometric NL) effects and resulting follower forces
- FORCE and MOMENT cannot be follower forces (use FORCE1 or FORCE2 for point forces or alternatively, PLOAD pressure cards)

NXSTRAT “strategy parameters”
- This card is unique to the advanced solver, and it’s functionality is similar to the NLPARM card with 106.

Matrix Stabilization (MSTAB)
- Scales diagonal stiffness terms to prevent pivot ratio problems. Acts as if weak springs were attached to all DOF, but has no detrimental effect on solution.

Load change with deformation (LOADOPT)
- Pressure/ distributed loading is deformation dependent.
- Applicable when LGDISP = 1.
NL Solution Approach - Iteration and Convergence Parameters

**Auto increment (AUTO)**
- 1..On = Automatic time stepping (ATS) enabled. If no convergence for user-defined time step, program automatically subdivides time step until convergence attained.
- Time is analogous to load level

**Continue if Non-Positive Definite (NPOSIT)**
- If diagonal element in stiffness matrix is 0 or negative (indicating non-invertible), program will assign large value to that element (as if a stiff spring was affixed to the DOF it represents) and continue execution.

**Max iterations/ step (MAXITE)**
- Sets maximum number of iterations to be used per time step

**Line search (LSEARCH)**
- Can assist convergence of first few iterations of a time step when current displacements are far from the converged solution.

**Low speed dynamics (ATSLOWS)**
- Brings mass and damping effects into the otherwise static analysis. Allows for solutions with rigid-body modes and local snap-through or buckling instabilities.

*Remaining options left to defaults*
NL Solution Approach - Verifying accuracy of solution

- Accuracy indicator history printed to solution F06 file
- Want magnitude of convergence-assisting effects to be low relative to external forces acting on the model

Solution Accuracy Indicators for Low Speed Dynamics, Contact Damping, Shell Drilling and Stiffness Stabilization

<table>
<thead>
<tr>
<th>EXTERNAL FORCES</th>
<th>DRILLING FORCES</th>
<th>DAMPING FORCES</th>
<th>INERTIA FORCES</th>
<th>CONTACT DAMP. FORCES</th>
<th>STIFFNESS STABIL.</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.711E+02</td>
<td>2.144E-03</td>
<td>1.480E+00</td>
<td>1.422E+00</td>
<td>--</td>
<td>8.323E-02</td>
</tr>
<tr>
<td>% of ext. forces</td>
<td>0.00</td>
<td>0.40</td>
<td>0.38</td>
<td>--</td>
<td>0.02</td>
</tr>
</tbody>
</table>
Quick Demonstration – Navigating Models

- Since internal structure can be obscured, any number of visibility settings (layers, groups, property enabling/disabling) can be used to expose necessary model parts.

- We tended to use the cross section. You can use the alt+scroll wheel when this is enabled to quick translate the cutting plane in the normal direction.

- Since this project was started, **Draw/Erase** has been added as a new feature in Femap v11.3.
Running a large amount of load cases

- Successfully ran 43 load cases in roughly 1 week
  - Load cases correspond directly to FAR certification
  - Average run time: ~6 hrs on Intel Core i7 32GB RAM custom built workstation
  - > 7 days computer run time for all cases

- NX Nastran Advanced nonlinear
  - With parallel processing feature, simply set number of CPUs to use in the Femap analysis setup dialog for solution options
    - Most beneficial running 4-8 processors (disable HT for best performance is the official Siemens recommendation)
  - Memory (NXNA_MEMORY)
    - Avoid over-allocating RAM so that O/S has room for file I/O
      - No more than 80%-90% of your physical RAM
      - Siemens normally recommends no more than 50%
    - 32 GB is the norm at SDA, some machines 64 GB

- All about book keeping (and time)
  - Figuring out what worked, what did not, and WHY
  - Figuring out options to push for convergence
Post-processing vast NL results

- General Process:
  - Envelope all results
    - Gives look at most critical conditions
    - Shows a good starting place
  - FEMAP’s PostProcessing Toolbox is a great tool for visualizing results
  - Criteria options allow for visualization within a user specified range
  - QUICKLY look through results, isolate possible problem areas
Post-processing vast NL results

**NL Data Charting API tool**

**Purpose**
Have Excel file accept lists of elements and output vectors to enable rapid charting of NL predictions to compare to strain gage readings from test.

**Inputs**
Elements and output vectors, in simple column format. Better traceability/repeatability for reporting down the road as inputs are stored in separate file.

**Output**
Charts within Femap, one for each element that contains plot of OV and OV2. These can then be further manipulating by adding to the baseline script code or through the Femap charting GUI.
Post-processing vast NL results

Script written with simple FEM first

Top and bottom nonlinear strains successfully charted for all requested elements WITHOUT manual entry

Output Set: Internalize Set 7 - Case 1 Time 10.
Deformed(0.00524): Total Translation
Elemental Contour: Nonlinear Plate Top Minor Stress
Contour double: Nonlinear Plate Bot Minor Stress
Post-processing vast NL results

Script written directly in Excel

Pseudocode

- Connect to Femap
- For each row (each element)
  - Create chart
  - Create data series for OV
  - Create data series for OV2

```
Dim app As Object
Set app = GetObject(, "femap.model")
Dim dsTop As Object
Set dsTop = app.fmDataset
Dim dsBot As Object
Set dsBot = app.fmDataset

Dim chrt As Object
Set chrt = app.fmChart

With dsTop
    .SeriesType = FCHD_TYPE_SETVAL
    .OutputSetAll = True
    .OutputVector = ActiveSheet.Cells(i, 4).Value
    .Location = elemid
    .Put (dsTop.NextEmptyID)
End With

With dsBot
    .SeriesType = FCHD_TYPE_SETVAL
    .OutputSetAll = True
    .OutputVector = ActiveSheet.Cells(i, 5).Value
    .Location = elemid
    .Put (dsBot.NextEmptyID)
End With

chrt.Title = ActiveSheet.Cells(i, 3)

rc = chrt.AddDataSeries(-1 * dsTop.ID)
rc = chrt.AddDataSeries(-1 * dsBot.ID)
rc = chrt.Put(chrt.NextEmptyID)
```
Example

Results for element 82291
FEMAP has many great tools and features, but 2 limitations resulted in custom built tools:

1. SOL 601 does not output any inter-laminar information
2. No quick way to check against an allowable that varies with span

Custom user APIs were written to calculate and summarize the required information using available output.
Post-processing vast NL results – Custom Built Tools

- Advanced Non-Linear does not output inter-laminar shear for a composite or multilayer element, but we still need a way to determine the interlaminar Failure Index for bonded structure.

- The only TRANSVERSE information we have from Advanced Nonlinear is transverse stress per ply.

- Inter-laminar Shear is calculated using $V \times Q / l \times t$.

- Custom API written to calculate and create output for all laminate bonded structure for each load case.
General Process:

- Use transverse shear STRESS output \( \tau_{xz} / \tau_{yz} \) for each ply in a laminate to calculate the transverse shear FORCE \( V_{xz} / V_{yz} \) for an element.

- Use the calculated shear force to calculate the interlaminar shear stresses using \( \tau = \frac{VQ}{It} \).

- Calculate a bond-line failure index: \( F.I. = \frac{\sqrt{\tau_{zx}^2 + \tau_{zy}^2}}{\tau_{allowable}} \).

- Cycle through all elements and print the results.
## Sample Output: Maximum Interlaminar Shear Failure Index

<table>
<thead>
<tr>
<th>V: 1</th>
<th>L: 1</th>
<th>C: 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4647</td>
<td>0.6741</td>
<td>1.3892</td>
</tr>
<tr>
<td>0.2562</td>
<td>0.1578</td>
<td>0.2258</td>
</tr>
<tr>
<td>0.3197</td>
<td>0.2049</td>
<td>0.0994</td>
</tr>
<tr>
<td>0.252</td>
<td>0.07538</td>
<td>0.147</td>
</tr>
<tr>
<td>0.2662</td>
<td>0.05988</td>
<td>0.161</td>
</tr>
<tr>
<td>0.3103</td>
<td>0.06321</td>
<td>0.1296</td>
</tr>
<tr>
<td>0.5406</td>
<td>0.3308</td>
<td>0.2039</td>
</tr>
<tr>
<td>0.9673</td>
<td>1.2141</td>
<td>0.837</td>
</tr>
<tr>
<td>2.4427</td>
<td>8.1082</td>
<td>7.3376</td>
</tr>
</tbody>
</table>

Output Set: Case 1 Time 10.
Criteria: Maximum Failure Index
Post-processing vast NL results – Custom Built Tools

Problem: How does one QUICKLY write a M.S. against an allowable that varies with span?

Solution: Programmatically survey model so that allowable can be set based on location

- Stress Survey APIs were written to write an element based M.S. for each load case
  - Ultimate
  - Yield
  - Shear
  - Crippling

- A new FEMAP “Result Vector” is created with M.S. information and a summary was printed to a .csv file
Post-processing vast NL results – Custom Built Tools

Process:

User Inputs:
- Elements & Output Sets to Survey
- Location to save Margin Summary
- Allowable*

For each Output Set:
- Cycles through every element
- Determines allowable based on element location*
- Compares NL element stress or strain to allowable and writes a M.S.

Calculating & printing a M.S. from FEMAP/NASTRAN Output

Outputs:
- FEMAP “Output Vector” of Element M.S. for each Output Set
- M.S. Summary .csv
- Peak Element Stress & Lowest M.S. from Surveyed elements / output sets
Post-processing vast NL results – Custom Built Tools

Example FEMAP Output

Output Set: Case 10 Time 10.
Criteria: Min Yield M.S. Laminate

Example Summary
Conclusion:
Good correlation from 0-100% load
Test Correlations

Elastically buckled skin pockets apparent prior to ultimate load in FEA and test.
Animation
Animation
In summary, we discussed...

- geometry preparation
- meshing practices
- Nastran’s advanced non-linear solver
- the use of built-in tools and development of new ones to interpret results

Any questions?
Aspects of pre/post processing a detailed wing FEM with Femap

Eric Gustafson
Senior Stress Analyst, SDA
eric@structures.aero
703-673-1128

Alex Skavdahl
Stress Analyst, SDA
askavdahl@structures.aero
703-673-1125

FEMAP Symposium 2016
October 11, 2016
Vero Beach, Florida