Design standards and ANSYS: is a full integration possible?

22. CADFEM ANSYS Simulation Conference, 2017-06-14
Agenda

1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
DAES expertise

CFD - Fluid Dynamics

Thermal analysis

Structural analysis

Dynamic - Explicit

Seismic

Functional Analyses

Safety Analyses

Member of the TechNet Alliance

DEAS ID: 001324 v2
Rigorous Design methodology based on System Engineering methods:

- QA Management ISO 9001-2015 certified
- CAD software: SolidWorks
- Design standards (RCC-MRx, EN13445, DIN…) constraints taken into account during all project phases
- Knowledge about machining issues
- Expertise in irradiated material properties

DAES is participating in the sub-committee for Design rules of RCC-MRx
Scientific/Engineering software (ANSYS) add-ons development, material model implementation:

- **Pre-processing tools**
  - SpaceClaim add-ons for screws (surface imprint)
  - SpaceClaim add-ons for welds according to DVS1608
  - Workbench add-ons for p+ heat deposition in thin structure

- **Post-Processing tools**
  - Workbench add-ons for RCC-MRx usage factor calculation
  - Workbench add-ons for RCC-M usage factor calculation
Agenda

1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
Working with standards

• A standard is:
  – A set or rules to fulfill requirements
  – In accordance with the (local) regulation
  – More than a guideline: it can be mandatory
  – Usually verbose (bad for engineers)
  – Always blurry (bad for everyone, even for those who wrote the standard)
• Usual flow*:

- To be compliant at the pre-processing phase
  - Standard analysis: what shall be considered
  - Calculation in accordance with standard
  - Usual post-processing

- To be compliant at the Post-processing phase
  - Calculation (sometimes in accordance with standard)
  - Export stress/strain/...
  - Post-processing (excel, python...)
  - Optional: back to FE

- To be compliant all the way!! (should be the right way)

- No real integration

*Fully subjective view based on my personal experience
1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
ANSYS and standards, current status

- ANSYS is a **general** FE tool
  - Not limited to 1 industry
  - Can’t cope with all existing standards
  - Gap between expectations and provided tools
- ANSYS has already **most** of the needed tool
  - Similar tools/methods for related standards
  - Can be used for almost all calculations in accordance with standards
  - Gap for some calculation requiring additional tools
- ANSYS is **easy to use** (Workbench, AIM)
  - Anyone can use ANSYS
  - Only few understand standards
  - Gap between knowledge
Agenda

1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
Working with ANSYS and standards
Example 1

• Context
  – Customer: CEA
  – thick shell steel cover
  – Temperature: 550°C
  – Pressure: external, 180 bar
  – Plasticity to be considered
  – Calculation performed with different FE software leading to different results (at least due to “following pressure”)

• Standard: RCC-MRx
  – Pressure vessel for nuclear application
  – (more or less) Similar rules as in ASME

• Goal: Buckling analysis of thick shells subjected to external pressure

• Solution: your brain
Working with ANSYS and standards
Example 1

• What says the standard?
  – § A7 3200

The organization of the analysis, schematically represented in Figure A7.3212 is as follows:

Step 1. By a bifurcation analysis on the nominal geometry, calculate the bifurcation modes of this structure (A7.3221).
Step 2. Determine the "modified" geometry (A7.3222).
Step 3. Carry out an elastoplastic analysis (A7.3223) on this "modified" geometry.
Step 4. In accordance with A7.3224, check that for the multiplication coefficients required in RB 3271, the structure remains stable.

- Initially, the equilibrium states of the structure subjected to proportional loadings (\(\lambda L\)) marked by a loading parameter \(\lambda\) are calculated. This calculation, made on the nominal geometry, should take account of geometrical non-linearities and plasticity.

- For each loading value \(\lambda L\), the coefficient \(k\) by which the corresponding stress state should be multiplied in order to get bifurcation, must be calculated. If the stress state is "plastic", calculate this coefficient by replacing Young’s modulus by a modified modulus, at each point which has become "plastic", in order to take account of the plasticity. It is recommended that the tangent modulus at the point concerned be adopted as the modified modulus. Other values may nonetheless be used provided that the choice can be justified.

- The elastoplastic bifurcation load (\(\lambda_{\text{cr}} L\)) is the load at which the multiplication coefficient \(k\) is equal to 1; the deformation pattern (eigenmode) of this mode is the bifurcation load to be taken into consideration.
Working with ANSYS and standards
Example 1

- Translation in ANSYS language
  - Steps to perform:
    - Plastic analysis of the structure (incl. large deformation) up to 3 times the nominal load
    - For each sub-step of interest, perform a “restart” to create a linear perturbation. “Tangent” option shall be activated
    - Linear buckling analysis using linear perturbation
• Solution process:
  – Full post-buckling analysis to check mesh/elements/keyoptions vs. example from literature
  – Additional buckling analysis on simplified model to check mesh and elements (incl. keyoptions) in buckling analysis
  – Full model calculation
Working with ANSYS and standards
Example 1

- Literature example description

Vuong-Dieu Trinh.
Formulation, développement et validation d’éléments finis de type coques volumiques sous-intégrés stabilisés utilisables pour des problèmes à cinématique et comportement non linéaires. Sciences de l’ingénieur.
Arts et Métiers ParisTech, 2009
Working with ANSYS and standards
Example 1

- Post-buckling analysis: elastic material
  - Comparison with measurement from literature
  - Test on:
    - Mesh size (not shown)
    - Element type (SOLID 185, 186, 187 and SOLSH 190)
    - Element technology (keyoptions)
  - Arc-length method used
Working with ANSYS and standards

Example 1

• Post-buckling analysis: conclusion
  – ANSYS is able to properly calculate buckling and post-buckling behaviour
  – For elastic material: Solid186/187 recommended
  – For plastic material: Solid 185 with enhanced strain formulation or Solid187, mixed U/P formulation if large plastic strain expected are recommended
Working with ANSYS and standards
Example 1

- Simplified model and standard method
  - Full standard methodology implemented
  - Elements chosen from previous analysis
  - Bad behavior of element at the fillet (whatever the mesh) in buckling analysis
Working with ANSYS and standards

Example 1

• Simplified model and standard method
  – Elastic analysis

<table>
<thead>
<tr>
<th>Element</th>
<th>Formulation</th>
<th>1&lt;sup&gt;st&lt;/sup&gt; critical load factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>SOLID186</td>
<td>Reduced integration</td>
<td>Numerical issue</td>
</tr>
<tr>
<td>SOLID186</td>
<td>Full integration</td>
<td>102.5</td>
</tr>
<tr>
<td>SOLID185</td>
<td>Reduced integration</td>
<td>Numerical issue</td>
</tr>
<tr>
<td>SOLID185</td>
<td>Full integration</td>
<td>102.2</td>
</tr>
<tr>
<td>SOLID185</td>
<td>Enhanced strain</td>
<td>Numerical issue</td>
</tr>
<tr>
<td>SOLID185</td>
<td>Simplified enhanced strain</td>
<td>Numerical issue</td>
</tr>
<tr>
<td>SOLID187</td>
<td>-</td>
<td>102.4</td>
</tr>
</tbody>
</table>
Working with ANSYS and standards
Example 1

- Simplified model and standard method
  - Plastic analysis

<table>
<thead>
<tr>
<th>Nr. of element</th>
<th>Solid186</th>
<th>Solid185</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Nr. of num. mode</td>
<td>1&lt;sup&gt;st&lt;/sup&gt; real mode</td>
</tr>
<tr>
<td>50</td>
<td>5</td>
<td>-</td>
</tr>
<tr>
<td>30</td>
<td>5</td>
<td>-</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>0</td>
<td>43.6</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>43.6</td>
</tr>
</tbody>
</table>
Working with ANSYS and standards
Example 1

• Simplified model: Conclusion:
  – Full integration only for buckling analysis
  – Mesh not the driving parameter
  – SOLID185 more robust than SOLID186
  – SOLID187 to be investigated, but cannot be excluded (easier mesh)

• Element choice for full analysis:
  – **Linear SOLID185 with full integration**, min. 10 elements in the fillet
Example 1

- Full model calculation

- Pressure on the exterior faces

- Force (X,Y,Z) calculated from pressure

- Meshing: specific procedure & conformal meshing – only SOLID185

- Bottom faces fixed in all directions
Working with ANSYS and standards
Example 1

- Full model calculation

Equ. stress

Mode 1
Grey: plastic region

Mode 2

Mode 3

Mode 4

Mode 5

Non symmetric behaviour and mode swap visible
Working with ANSYS and standards
Example 1

• Conclusion on study:
  – Plasticity has a large impact on calculated critical loads: much lower than elastic model ➔ shall be considered!
  – Not studied yet to fulfill the standard:
    • Boundary conditions and geometry defects: ANSYS procedures already implemented (modal analysis and upgeom)

• Conclusion on ANSYS/Standard integration
  – ANSYS provides all necessary tools 😊
    • Linear perturbation with tangent material data
    • Comparison with other FE codes and tests could help checking the accuracy
    • Still some parameters based on engineering feelings to be defined
  – Standard methodology not implemented as a Wizard, but ANSYS documentation fully compliant with standard 😊
  – With knowledge (standard and ANSYS), this kind of analysis can be considered as integrated 😐
Agenda

1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
Working with ANSYS and standards
Example 2

• Context
  – Customer: under NDA (as usual...)
  – Pressure vessel calculation
  – Temperature: -° C
  – Pressure: - bar
  – Fatigue, Thermal ratchetting assessment...

• Standard: RCC-M
  – Pressure vessel for nuclear application
  – (more or less) Similar to ASME

• Goal: Post-processing

• Solution: ACS
Example 2

• Standard approach: example for thermal ratchetting
  – Input parameter:
    • Maximum membrane stress $\sigma_m$
    • Yield stress $S_y$
  – Calculation of $x = \frac{\sigma_m}{S_y}$
  – Calculation of $y$:
    • Linear profile over thickness:
      – $0 < x < 0.5$: $y = \frac{1}{x}$
      – $0.5 < x < 1$: $y = 4 \times (1 - x)$
    • Parabolic profile over thickness:
      – $0.615 < x < 1$: $y = 5.2 \times (1 - x)$
      – $x < 0.615$: $y$ to be interpolated:

<table>
<thead>
<tr>
<th>$x$</th>
<th>0.3</th>
<th>0.4</th>
<th>0.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>$y$</td>
<td>4.65</td>
<td>3.55</td>
<td>2.7</td>
</tr>
</tbody>
</table>

– Calculation of $\sigma_\theta = S_y \cdot y$
– Comparison with thermal stress amplitude variation
Working with ANSYS and standards

Example 2

• Solution process:
  – Step 1: Understand the design standard
  – Step 2: Make sure we understand the design standard
  – Step 3: Build a simple model, define the features to add
  – Step 4: Create an ACS app
  – Step 5: Test the app, write the documentation
  – Step 6: between myself and bartender
Working with ANSYS and standards
Example 2

• Step 1, 2: not detailed.
• Step 3:
  – model not detailed
  – Features to add:
    • Results only
    • 4 analysis types (1 detailed)
    • Path to be defined automatically or manually, on all element types (shell, solid)
    • Results to be available from other analysis (same model)
    • Profile to be chosen. Both can be calculated at the same time.
    • Criteria to be adapted depending on event number
Working with ANSYS and standards

Example 2

• Step 4:
  – .xml file to define interface.
  – IO table to define parameters to be used
  – Split of general/specific scripts for reuse

• Results only
• 4 analysis types
### Step 4

- Path to be defined automatically or manually, on all element types (shell, solid)
- Results to be available from other analysis (same model)
- Profile to be chosen. Both can be calculated at the same time.
- Criteria to be adapted depending on event number

Python coding to be done at that stage...
• Step 5:
  – Nice pictures are not enough: results shall be checked for all features, all options, at various locations
• Conclusion
  – Using ACS, one can easily add in ANSYS all necessary tools 😊
  – “Easily” was a bit exaggerated… 😐
  – With knowledge (standard and ANSYS), this kind of postprocessing can be fully integrated within Workbench 😐
1. DAES presentation
2. Working with standards
3. ANSYS and standards, current status
4. Working with ANSYS and standards:
   a) Example 1: Thick plates buckling
   b) Example 2: RCC-M(Rx) and ANSYS
5. Conclusion
Conclusion

• Design standard analysis procedure can be fully integrated *
  – At least, based on my knowledge
  – If not available, ACS/APDL/UDF can help

• Design standard post-processing procedure can be fully integrated *
  – Can only help avoiding uninteresting work*

• Can’t replace standard knowledge.

• With a bit (ok, a lot) of work, all gaps can be filled!

*: one more subjective opinion
Thank you...

Cyril Kharoua: cyril.kharoua@daes.pro
Phone: +41 22 792 82 25
mobile: +41 79 967 79 35

Francois Plewinski: francois.plewinski@daes.pro
Phone: +41 22 792 82 26
mobile: +41 79 967 79 40

Pascal Sabbagh: pascal.sabbagh@daes.pro
Phone: +41 22 792 82 24
mobile: +41 79 967 79 24

www.daes.pro

DAES SA
Chemin Gérard-de-Ternier, 14
1213 Petit-Lancy
Switzerland

Incorporated under the laws of Geneva, Switzerland / Capital 100’000 CHF / CH-660.0.276.001-2 / TVA: CHE-101:828.611
2017-06-14