Direct Coupled Thermal-Structural Analysis in ANSYS WorkBench

Roberto Silva
ESSS
TOPICS

• Multiphysics Coupling
• Thermal-structural coupling
• ANSYS coupled field elements
• Analysis procedure
  – Material definition
  – Meshing
  – Loads and boundary conditions
MULTIPHYSICS COUPLING

- In real-life scenarios, multiple physics interact simultaneously.
MULTIPHYSICS COUPLING

• Usually, physics coupling is ignored or simplified.
  – Simulation engineers are usually single-physics.
  – Coupled analyses are more computationally intensive.

• However, coupled analyses provide more realistic results.

• ANSYS WorkBench is designed to make it easier to simulate multiphysics coupling.
MULTIPHYSICS COUPLING

ANSYS Mechanical

Structural

Electromag

ANSYS Maxwell / HFSS

Thermal

ANSYS CFX / Fluent

Fluid Dyn.

ANSYS Mechanical

ANSYS CFX / Fluent
- Thermal-structural coupling can be modeled in the same ANSYS Mechanical solver.

Mechanical properties based on temperature

Thermal strain

Heat generated by plastic strain

Heat generated by friction
• Some examples:

- Thermal expansion of rails due to Sun exposure
- Heat generated in brake disc
- Friction stir weld (FSW) procedure
Regarding coupling methodology:

1. **One direction only**
   - Coupling is considered in one direction only.
   - This is usually solved by **sequential 1-way coupling**.

2. **Both directions**
   - Coupling is considered in both directions.
   - This can be modeled with **2-way or direct coupling**.
• Sequential coupling
THERMAL STRUCTURAL COUPLING

• Direct coupling
THERMAL STRUCTURAL COUPLING

- 1-way thermal to structural coupling can be easily defined in WorkBench.
  - Just need to connect the simulation systems.
• However, 1-way structural to thermal coupling is not possible in ANSYS.
  – It’s not possible to do this…
THERMAL STRUCTURAL COUPLING

• Direct coupling is available in ANSYS, but not in the WorkBench interface.
  – This system does not exist yet!
To represent direct coupling, APDL commands should be used.
- User must select coupled-field elements.

1-way structural to thermal coupling is usually represented by direct coupling as well.
- It’s easier than export the deformed mesh and results from the structural analysis to the thermal analysis.
COUPLED-FIELD ELEMENTS

- ANSYS includes the following coupled elements:

  SOLID5
  PLANE13
  SOLID98
  PLANE223
  SOLID226
  SOLID227

Current Technology Elements
COUPLED-FIELD ELEMENTS

• Coupled-field elements can include several DOFs, and the associated couplings between them.

\[
\begin{align*}
\{M\}\ddot{\{u\}} + \{C\}\dot{\{u\}} + \{K\}\{u\} &= \{F\} & \text{Structural solution} \\
\{C^t\}\ddot{\{T\}} + \{K^t\}\{T\} &= \{Q\} & \text{Thermal solution}
\end{align*}
\]

- **Strong Coupling**

\[
\begin{align*}
\begin{bmatrix} M & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \ddot{u} \\ \ddot{T} \end{bmatrix} + \begin{bmatrix} C & 0 \\ C^t & K^t \end{bmatrix} \begin{bmatrix} \dot{u} \\ \dot{T} \end{bmatrix} + \begin{bmatrix} K & K^{ut} \\ 0 & K^t \end{bmatrix} \begin{bmatrix} u \\ T \end{bmatrix} &= \begin{bmatrix} F \\ Q \end{bmatrix}
\end{align*}
\]

- **Weak Coupling**

\[
\begin{align*}
\begin{bmatrix} M & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} \ddot{u} \\ \ddot{T} \end{bmatrix} + \begin{bmatrix} C & 0 \\ C^t & 0 \end{bmatrix} \begin{bmatrix} \dot{u} \\ \dot{T} \end{bmatrix} + \begin{bmatrix} K & 0 \\ 0 & K^t \end{bmatrix} \begin{bmatrix} u \\ T \end{bmatrix} &= \begin{bmatrix} F + F^{th} \\ Q + Q^{ted} \end{bmatrix}
\end{align*}
\]
ANALYSIS PROCEDURE

Prep

- Geometry
- Material
- Mesh
- Loads and BC’s

Solu

- Analysis Settings
- Convergence

Post

- Results

Those are the most important features for a direct-coupled analysis
ANALYSIS PROCEDURE

• Example:
  – Steel axissymmetric pipe with fins

Convection
\[ h = 200 \text{ W/m}^2 \text{ °C} \]
\[ T_{amb} = 20 \text{ °C} \]

Temperature = 200 °C
Pressure = 5 MPa

Symmetry
ANALYSIS PROCEDURE

• Which system should be used?

• It is recommended to use a Structural system.
  – Thermal setup is easier to implement with APDL.
MATERIAL DEFINITION

- Toggling Engineering Data filter off, all properties are available.

![Outline of Schematic A2: Engineering Data](image)

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>7.85E-06</td>
<td>kg mm^-3</td>
</tr>
<tr>
<td>Isotropic Secant Coefficient of Thermal Expansion</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Isotropic Elasticity</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Alternating Stress Mean Stress</td>
<td></td>
<td>Tabular</td>
</tr>
<tr>
<td>Strain-Life Parameters</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tensile Yield Strength</td>
<td>250</td>
<td>MPa</td>
</tr>
<tr>
<td>Compressive Yield Strength</td>
<td>250</td>
<td>MPa</td>
</tr>
<tr>
<td>Tensile Ultimate Strength</td>
<td>460</td>
<td>MPa</td>
</tr>
<tr>
<td>Compressive Ultimate Strength</td>
<td>0</td>
<td>MPa</td>
</tr>
<tr>
<td>Isotropic Thermal Conductivity</td>
<td>0.0605</td>
<td>W mm^-1 C^-1</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>4.34E+05</td>
<td>mJ kg^-1 C^-1</td>
</tr>
</tbody>
</table>
MESHING

• An APDL command is used to change element type.
  – Element must be chosen accordingly to mesh geometry!

  `ET, matid, PLANE223`  \(\rightarrow\)  *This changes element type*

  `KEYOPT, matid, 1, 11`  \(\rightarrow\)  *This defines thermal-structural behavior*

  `KEYOPT, matid, 3, 1`  \(\rightarrow\)  *This redefined axissymetric behavior*
LOADS AND BOUNDARY CONDITIONS

• Structural loads and BC’s are applied as usual.

• For thermal loads, APDL commands are needed.

• Thermal loads must be applied on nodes and elements, via Named Selections.

• Be careful with !
  – It defines zero value to all DOFs, including temperature!
Named Selections can be defined with geometry…

… or by direct nodal selection

Elements can be selected based on nodal selection, using APDL command ESLN
LOADS AND BOUNDARY CONDITIONS

• Load definition via APDL
  – Refer to ANSYS documentation for more information.

  - **Temperature**
    - **APDL command = D**
  - **Convection**
    - **APDL commands = SF, SFE**
  - **Radiation**
    - **APDL command = F**
  - **Heat Flow**
  - **Perfectly Insulated**
  - **Heat Flux**
  - **Internal Heat Generation**
    - **APDL commands = BF, BFE**
LOADS AND BOUNDARY CONDITIONS

D, temp_face, TEMP, 200 → Defines temperature
SF, conv_face, CONV, 200, 20 → Defines convection
ALLSEL → Select all entities
LOADS AND BOUNDARY CONDITIONS

• A note about units:
  – It is highly recommended to use **metric system**. This avoids uncommon units defined by the ANSYS solver when using other systems.
  – To be sure, define solver units manually.

![Analysis Data Management Table]

<table>
<thead>
<tr>
<th>Analysis Data Management</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver Files Directory</td>
<td>E:\Apresentações\UG...</td>
</tr>
<tr>
<td>Future Analysis</td>
<td>None</td>
</tr>
<tr>
<td>Scratch Solver Files Directory</td>
<td></td>
</tr>
<tr>
<td>Save MAPDL db</td>
<td>No</td>
</tr>
<tr>
<td>Delete Unneeded Files</td>
<td>Yes</td>
</tr>
<tr>
<td>Nonlinear Solution</td>
<td>No</td>
</tr>
<tr>
<td>Solver Units</td>
<td>Manual</td>
</tr>
<tr>
<td>Solver Unit System</td>
<td>mks</td>
</tr>
</tbody>
</table>
SOLUTION

• Analysis can be solved as usual.

• If non-linear behavior is expected, heat flow convergence can be monitored in WorkBench.
RESULTS

• Thermal results can be plotted with the User-defined Result.
  – Tip: select solution and click Worksheet.
FINAL REMARKS

• Multiphysics simulations can provide more precise results, evaluating how different phenomena interact.

• Thermal-structural coupling can be solved using the same ANSYS Mechanical solver.

• Using APDL commands, direct coupling can be easily implemented in the WorkBench interface.