ANSYS Workbench for Process Compression and Scalability

Jiaping Zhang, Technical Service Engineer,
Ansys Inc. Houston Office
Jiaping.Zhang@Ansys.com
Agenda

1. Ansys Workbench & Mechanical: An Introduction
2. 6 Steps to a Successful FEA Simulation
3. Physics Coupling and External Data Import
4. ACT Preview: Customizing the User Interface
1. **Ansys Workbench & Mechanical: An Introduction**

2. **6 Steps to a Successful FEA Simulation**

3. **Physics Coupling and External Data Import**

4. **ACT Preview: Customizing the User Interface**
Productivity Challenge

- Shorten setup time for a single simulation
- Reduce solving time
- Simplify results analysis
- Increase simulations in fixed time

Boost Productivity

HPC+GPU

Setup Solving Results Variations

One simulation

Setup Solving Results Variations

N simulations
Workbench: Shorten Setup Time

Multiphysics Workflow

Automatic Contact

CAD& Parametric

Automatic Meshing
Workbench: Reduce Solving Time

- Distributed ANSYS+GPU (Graphics Processing Unit) Acceleration

**Solder Joint Creep Analysis**
- 4M DOF

**Mold**

**PCB**

**Solder balls**

**R14 Distributed ANSYS w/wo GPU**

- Without GPU: 1.7x, 1.9x, 3.2x
- With GPU: 3.4x, 4.4x

**Total Speedup**
- 16 cores
- 32 cores
- 64 cores
Workbench: Simplify Postprocessing
Workbench: Allows Variations

Single Parameter

Response Surface

Goal Driven Optimization

Sensitivity Study
Agenda

1. Ansys Workbench & Mechanical: An Introduction
2. 6 Steps to a Successful FEA Simulation
3. Physics Coupling and External Data Import
4. ACT Preview: Customizing the User Interface
Step 1: Define the Simulation Workflow
Step 2: Define Geometry and Materials I
Step 2: Define Geometry and Materials II

Material Library

Define Properties

Material Curve Fitting
Step 3: Define Connections between Bodies
Automatic Contacts Detection

409 Parts, 967 Contacts

Contact Bodies: Part 3
Target Bodies: Part 6

- **Definition**
  - Type: Frictional
  - Friction Coefficient: 0.1
  - Scope Mode: Automatic
  - Behavior: Program Controlled
  - Suppressed: No

- **Advanced**
  - Formulation: Program Controlled
  - Detection Method: Program Controlled
  - Interface Treatment: Add Offset, No Ramping
  - Offset: 0 mm
  - Normal Stiffness: Program Controlled
  - Update Stiffness: Program Controlled
  - Stabilization Damping Factor: 0
  - Pinball Region: Program Controlled
  - Time Step Controls: None
More Connections are Available

- Beam
- Spot Welds
- Joint
- Spring
- Mesh connection
Step 4: Mesh the Model

[Diagram showing the ANSYS project tree with the Mesh node expanded, including Mesh, Named Selections, Static Structural (A5), Analysis Settings, Pressure, Force, Fixed Support, Elastic Support, Commands (MAPDL), Solution (A6), and Solution Information nodes.]
Global Mesh Control

Curvature “On”

Proximity “On”

Generate Mesh

Details of “Mesh”

- Use Advanced Size Function: On: Proximity and Curvature
- Relevance Center: Coarse
- Initial Size Seed: Active Assembly
- Smoothing: Medium
- Transition: Fast
- Span Angle Center: Coarse
More Local Controls are Available
Adaptive Mesh Refinement for Convergence
Step 5: Define Loads and Boundary Conditions
 Loads and Boundary Condition Options

- Inertial
  - Acceleration
  - Standard Earth Gravity
  - Rotational Velocity

- Loads
  - Pressure
    - Hydrostatic Pressure
  - Force
  - Remote Force
  - Bearing Load
    - Bolt Pretension
    - Moment
  - Generalized Plane Strain
  - Line Pressure
  - Thermal Condition
  - Joint Load
  - Fluid Solid Interface
  - Detonation Point

- Supports
  - Fixed Support
  - Displacement
    - Velocity
    - Impedance Boundary
  - Frictionless Support
    - Compression Only Support
    - Cylindrical Support
    - Simply Supported
    - Fixed Rotation
  - Elastic Support
    - Coupling
    - Constraint Equation

- Imported Loads
Applying Loads to Nodes

“Nodal orientation” allows users to assign a coordinate system to node or nodal sets.

Direct FE loads and boundary conditions can be applied to selected nodes, whose direction is defined by “Nodal orientation.”

Nodes are oriented in cylindrical system for loads and boundary condition definitions.
Step 6: Understanding and Verifying Results
Thoroughly Investigate Your Results
Check The Quality of Your Results

- Applied Load
- Reactive Force
Create a Project Report

**TABLE 15**
Structural Steel > Constants

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>0.2838 lbm in^-3</td>
</tr>
<tr>
<td>Coefficient of Thermal Expansion</td>
<td>6.6687e-006 F^-1</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>0.10366 BTU lbm^-1 F^-1</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>8.0817e-004 BTU s^-1 in^-1 F^-1</td>
</tr>
<tr>
<td>Resistivity</td>
<td>8.5235 ohm cmil in^-1</td>
</tr>
</tbody>
</table>

**TABLE 16**
Structural Steel > Compressive Ultimate Strength

| Compressive Ultimate Strength psi | 0 |

**TABLE 17**
Structural Steel > Compressive Yield Strength

| Compressive Yield Strength psi | 36259 |

**FIGURE 2**
Model (A4) > Static Structural (A5) > Solution (A6) > Total Deformation > Figure
Enhance Simulation Using Command Objects
Command Object Example: Contact Setting

Frictional Surface

Set shear stress limit

rmodif,cid,9,120  !define TAU(MAX) as 0.6*SY
Apply Constraint and Loading on this surface

Load step 2: Clamp top surface

Load Step 3: Apply vertical loading to clamped surface
Command Object Example: PostProcessing
Agenda

1. Ansys Workbench & Mechanical: An Introduction
2. 6 Steps to a Successful FEA Simulation
3. Physics Coupling and External Data Import
4. ACT Preview: Customizing the User Interface
Coupling Physics Approach Reality

Reduce engineering assumptions by coupling multiple physics

Increase productivity by including all physics in unified environment

Rely more on high fidelity virtual prototypes

Depend less on costly physical testing
ANSYS Solution for Multiphysics

Electromagnetic Simulation
- Low Frequency
  - Maxwell Simulator
- High Frequency
  - HFSS
  - Slwave

Mechanical Simulation
- Implicit
  - ANSYS Mechanical
- Explicit
  - ANSYS Explicit
  - ANSYS AUTODYN
  - ANSYS LS-DYNA

Computational Fluid Dynamics (CFD)
- Electronics cooling
  - ANSYS Icepak
- General CFD
  - ANSYS CFD

Fluid Dynamics

Structural Mechanics

Electromagnetics
Fluid-Structure Coupling
Electromagnetic-Structure Coupling

Magnetics forces will induce stresses (Example: Motor)

Resistive losses will cause thermal stress (Example: Satellite Dish Antenna)
Electromagnetic-Fluid-Structure Coupling

• Maxwell+Fluent
  – One-way and two-way $\beta$ coupling
• Combine with 1-way FSI
External Data Mapping: Motivation

Exchange files are frequently encountered to transfer quantities from one simulation to another.

Efficient mapping of point cloud data is required to account for misalignment, non matching units or scaling issues.
Supported Data Types

<table>
<thead>
<tr>
<th>Column</th>
<th>Data Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Not Used</td>
</tr>
<tr>
<td>2</td>
<td>A</td>
</tr>
</tbody>
</table>

- Not Used
- X Coordinate
- Y Coordinate
- Z Coordinate
- Temperature
- Pressure
- Heat Transfer Coefficient
- Heat Flux
- Heat Generation
- Thickness
- Displacement (Beta)
Importing Multiple Files

Multiple files can be imported for **transient analyses** or to handle different data to be mapped on **multiple bodies**.
Validating the Mapped Data

Visual tools have been implemented to control how well the data has been mapped onto the target structure.

Both the size of the spheres and their color provide indication of the mapping quality.
Agenda

1. Ansys Workbench & Mechanical: An Introduction

2. 6 Steps to a Successful FEA Simulation

3. Physics Coupling and External Data Import

4. ACT Preview: Customizing the User Interface
ACT Introduction

Application Customization Toolkit (New in R14.0)

- **Encapsulate APDL macros**: Allows re-use of legacy APDL-scripts and encourages migration from MAPDL to Mechanical via “encapsulated macros”
- **MAPDL exposure**: Fills the gap between MAPDL solver capabilities and their exposure in ANSYS Mechanical
- **New pre-processing features** *(custom loads and boundary conditions)*
- **New post-processing features** *(custom results)*
- **3rd party/in-house solver integration** *(Mechanical GUI)*
Simple ACT Example

Opportunity to migrate existing process automation from MAPDL to ANSYS Mechanical at “low cost”
Customize Toolbar Using XML

XML definition:

```xml
<load internalName="Convection on Blade" caption="Convection on Blade" icon="Convection" issupport="false" isload="true">
    <version>1</version>
    <callbacks>
        <onsolve>Convection_Blade_Computation</onsolve>
    </callbacks>
    <details>
        <property internalName="Geometry" dataType="string" control="scoping"/>
        <property internalName="Thickness" caption="Thickness" dataType="string" control="text"/>
        <property internalName="Film Coefficient" caption="Film Coefficient" dataType="string" control="text"/>
        <property internalName="Ambient Temperature" caption="Ambient Temperature" dataType="string" control="text"/>
    </details>
</load>
```
Define Internal Process Using Python

Python script:

```python
# Get the scoped geometry:
propGeo = result.GetDPropertyFromName("Geometry")
refIds = propGeo.Value

# Get the related mesh and create the component:
for refId in refIds:
    meshRegion = mesh.MeshRegion(refId)
elementIds = meshRegion.Elements
eid = aap.mesh.element[elementIds[0]].Id
f.write("*get,ntyp,ELEM,"+eid.ToString()+",ATTR,TYPE\n")
f.write("esel,s,type,,ntyp \n cm,component,ELEM")

# Get properties from the details view:
propThick = load.GetDPropertyFromName("Thickness")
thickness = propThick.Value
propCoef = load.GetDPropertyFromName("Film Coefficient")
film_coefficient = propCoef.Value
propTemp = load.GetDPropertyFromName("Ambient Temperature")
temperature = propTemp.Value

# Insert the parameters for the APDL commands:
f.write("thickness="+thickness.ToString()+"\n")
f.write("film_coefficient="+film_coefficient.ToString()+"\n")
f.write("temperature="+temperature.ToString()+"\n")

# Reuse the legacy APDL macros:
f.write("/input,APDL_script_for_convection.inp\n")
```
3rd Party/In-house Solver Integration

ACT allows partners/customers to seamlessly integrate into Mechanical

Non parametric optimization solver (topological optimization)
Concluding Remarks

Workbench compresses and scales your simulation via:

- Compress setup process
- Simplify physics coupling and data import
- Allows User Customization
Thank you!