Customization and Automation of Engineering Simulation Workflow

Mai Doan, Application Engineer
Mai.Doan@ansys.com
Agenda

1. Need for customization
2. Different Customization Methods
3. Application Customization Toolkit (ACT) and Examples
4. Other Mechanical Customization possibilities
5. Other Fluid Customization possibilities
6. Other possibilities (Common)
Workflow Streamlining Using Design Points and Optimization

All of this and much more.... without any customization!
Need for Customization

Capture the existing simulation process
Make repetitive operations automatic
Integrate CAE with other (or in-house) tools
Make the technology available to a wider group (non CAE experts)

....
Native and Data-Integrated Applications

Native applications
• Built entirely on WB2 Framework
• Embedded within the “Workbench” window
• Project Schematic, Design Exploration, Engineering Data
• Fully supported by Workbench scripting
• Scripting language: Python

Data-integrated applications
− Share data and parameters with Workbench, native applications, and other data-integrated applications
− Independent UI, window
− E.g., Mechanical, Mechanical APDL, CFX, FLUENT, DesignModeler
− Scripting Language: JScript, Scheme, APDL ...
Workbench Scripting Overview

Application-level Scripting
• For task automation at the application level
  – Mechanical, DM, Meshing: JScript
  – CFX: CCL
  – FLUENT: Scheme
  – MAPDL: APDL

Workbench Scripting
• For task automation at project level
  – Creating project, performing parameters simulations, optimization etc.
• Works “hand-in-hand” with scripting in DIAs
  – Can embed JScript, CCL, Scheme, APDL
Different Customization Methods

Application Customization Toolkit (ACT)
• Create new load/BC/results
• Integrate an external solver in Mechanical

Jscript (for DM, Meshing, Mechanical etc.)
• Task automation within the various applications
• Jscript add-in to add functionality in the application

Wizards (for DM, Meshing, Mechanical etc.)
• Creating a customized workflow within the DIA

Python Journaling
• For task automation at WB level (applicable for all applications available on WB)
  – Creating Project, Performing Parameters simulations etc.
• Can use Jscript macros for DM, Meshing, Mechanical etc.
• Can use Scheme (Fluent), CCL (CFX), VBScript (Maxwell etc.)

External Connection Add-in
• Making external applications (not integrated with WB) to participate in workflow through parameters
• Python scripting can be integrated to add functionality to WB

C# Add-in using Software Development Kit (SDK)
• Integrate external applications in workflow through custom systems
• WB GUI customization (addition of buttons, menu etc.)
An Introduction to Application Customization Toolkit (ACT) and Examples
What is ACT

It is a Toolkit to customize Applications in Workbench

• In R14, ACT allows customization in the Mechanical application (Beta)
  – Define customized boundary conditions and post-processing
  – Replace command snippets with interactive objects
  – Do much more than what the standard features in Mechanical
    • Access node/element information, material, solution, results data ...
    • Perform: Crack propagation, Sub-modeling, MEMS, CMS (using ROM)
  – Even replace the APDL solver with your custom solver!

• ACT would be available for more applications in future

• ACT provides an option to migrate APDL users to Mechanical
  – Allow reusing legacy APDL developments in user-friendly way
  – Gets the best of the two worlds!

• Developing ACT solutions is fast and efficient
  – One doesn’t have to be a developer
  – Simple ACT solutions can be developed in minutes
  – ACT solutions are future ready!
Examples – What can be done?

✓ Node coupling
✓ Sub-modeling simulation in Mechanical
✓ Acoustics simulation in Mechanical
✓ Custom result – “Contact Force Vectors”
✓ 2D Convection Load extension
✓ Clamp and Displace extension
✓ Third party solver connection
Node Coupling

New extension for specific load creation

The Coupling object
Sub-modeling extension

Sub-modeling in Mechanical between 2 structural analysis

- Interpolation of displacement (CBDOF) and temperature (BFINT).
- Choose 2D-2D, 3D-3D or 2D axi-3D sub-modeling
- Choose Solid-solid & Shell-shell key or Solid-shell key
- Tabular data for multi-stepping

The sub-modeling extension was developed as an example just to show the possibility. Since, sub-modeling is scheduled to be released natively in Mechanical at R14.5, the extension is not being developed further or distributed.
Acoustics simulation in Mechanical using ACT

Acoustics features in Mechanical without any command objects
– Define acoustics elements, real constants & material properties
– Apply acoustics boundary conditions & loads
– Plot Far/Near field & Time/Frequency results
– Postprocess Pressure & SPL
Example simulations

- Speaker

- Underwater
Visualize the Contact Force Vectors

- Contact Force is not available for visualization in Mechanical
- In MAPDL, we have access to the contact force components
- Using APDL commands, we can get all the force components
- Using ACT, we can visualize the contact force vectors
Visualize the Contact Force Vectors

The Minimum and Maximum values of the result can be defined as “parameter” to perform DP and DX analysis.
• Apply a convective boundary condition to the blade surface in a 2D axi-symmetric analysis

• Instead, expose the same functionality via ACT
! APDL_script_for_convection.inp

/prep7
thickness = 0.001
film_coefficient = 200.
temperature = 120
cmsel,s,component

*GET,n_el,ELEM,0,num,max
*GET,mat1, ELEM,n_el,ATTR,MAT

et,100,152
keyop,100,8,2.
et,1001,131
keyo,1001,3,2
sectype,1001,shell
secdata,thickness,mat1
secoff,mid

emodif,all,type,1001
emodif,all,secnum,1001
type,100
esurf
fini
alls
/solu
esel,s,type,,100
nsle
sf,all,conv,film_coefficient,temperature
allsel, all
Clamp and Displace extension

Load Step-1: Apply the pressure → Allow the body to bend
Load Step-2: Clamp the top surface in its deformed state
Load Step-3: Apply a deformation to the clamped surface
Solution using Command Snippet

Definition based on named selection previously defined

cmsel,s,clamp,node
*GET,ND_CNT,NODE,0,COUNT
N_idx=0
*D0,lp_idx,1,ND_CNT
N_idx=NDNEXT(N_idx)

*Get,UX_ND,NODE,N_idx,U,X
*Get,UY_ND,NODE,N_idx,U,Y

!DOF constrains applied in ND:
D,N_idx,UX,UX_ND
D,N_idx,UY,UY_ND

*ENDDO
ALLSEL

LS 2

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Target
Mechanical APDL

Details of "Commands (APDL)"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
2.
Target
Mechanical APDL

Details of "Commands (APDL) 2"

File
File Name
File Status
Definition

Suppressed
No
Step Selection Mode
By Number
Step Number
3.
Solution using ACT

User inputs in the APDL command snippets are translated to ACT properties related to the newly integrated ACT load:

- The location provided by the named selection is now defined based on a scoping method compatible with both named selection and direct geometry selection.

- The load step number for clamping is available from a drop-down menu initialized with the number of steps already defined in the Analysis Settings object.

- The X Displacement value for the next load step is defined by a new property declared as a length. This makes this property always consistent with the current unit system activated in Mechanical.
Stress Limit Damage extension

Post-process the maximum admissible criteria based on temperature dependent properties in Mechanical

The relevant material property are retrieved directly from EDA

The material property can also be defined by the user directly in Mechanical (user defined property)
Stress Limit Damage extension
FEInfo extension

E:1: (0.958, 0.958, 0.0417)
E:2: (0.875, 0.958, 0.0417)
E:3: (0.792, 0.958, 0.0417)
Tosca integration in Workbench

Tosca structure is a Non-parametric optimization solver (topological optimization).

Initial design

Optimal result

Optimal design

Verification
Using ACT extensions
Using the ACT extension for a Project

The “Extensions” option is available in the menu bar of the project page.

Extension Manager to Load / Unload available extensions.

View log file to review messages generated from the extensions.
Using ACT in Mechanical

- Command line editor
- Functions list
- Info window
- Output window

Refresh to reload the extensions
ACT Extension Basics

• An ACT extension consists of
  – **XML file**
    • Configures the UI content: buttons in the toolbar and objects in tree
    • Defines the extensions properties
    • Configures behaviors for custom loads and results.
    • Binds application events to IronPython script functions.
  – **IronPython script file**
    • Implements the extension functionalities: retrieving mesh data, writing in the ds.dat file, computing a customized result...
    • Event driven
      – Functions are invoked by application generated events
    • Supports access to external libraries
    • The script file is typically placed in a folder in the same name as the XML file

• One may have additional files/folders to organize the content better
  – E.g. a separate folder for images, other resources etc.
Extension structure

Customization Toolkit
- Mechanical toolbar
- Pre-Processing object
- Post-Processing object

Python scripts

UI

WB Project/Mechanical
- Materials
- Geometry
- Mesh
- Simulation data
- Results
- Events

XML definition
Other possibilities (Mechanical)
Automation in Mechanical: Jscript macro

- Objectives: Apply Transient Force & Moment Loads from Excel
  - Transient Force and Moment load consists of thousands (30,000+) of data points
  - Applying these loads manually with so many inputs is impossible

- Solution:
  - A new script is written for this automation
  - This solution works fine on all versions on Mechanical (R11 to R14)
Automation in Mechanical

- Objectives: Automate Contact Generation and Naming from Excel
  - Defining various contacts from Named Selections
  - Rename the contacts appropriately: ContactType – sourceNS -to- targetNS
  - Keep the generated contacts in a group for easy identification
  - Have an Excel file for interactive and easy input

- Solution:
  - A script is written for this automation
  - A macro enabled Excel file helps user to provide correct inputs
  - An additional script is also provided to rename contacts appropriately (not generated from Excel file)
WB simulations from MS Excel

- WB simulations can be performed with user inputs in Excel

<table>
<thead>
<tr>
<th>Inputs</th>
<th>Study-1</th>
<th>Study-2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radius (in)</td>
<td>20</td>
<td>15</td>
</tr>
<tr>
<td>Depth (in)</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Force (lbf)</td>
<td>200</td>
<td>200</td>
</tr>
</tbody>
</table>

Both the Design Points are solved, Project is saved.
Max deformation values from those analysis are updated in Excel

Results:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS Mechanical Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max Deformation (in)</td>
<td>Study-1: 1.667, Study-2: 1.201</td>
</tr>
</tbody>
</table>

- WB Python journal is used for getting inputs from MS Excel, performing the simulation and publishing results back to MS Excel
- Similarly other simulations (FLUENT, HFSS etc.) can be done
Customization possibilities
(Fluids examples)
WB simulations from MS Excel

• WB simulations can be performed with user inputs in Excel

<table>
<thead>
<tr>
<th>Inputs</th>
<th>Dp1</th>
<th>Dp2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Catalyst length (mm)x10</td>
<td>20</td>
<td>40</td>
</tr>
<tr>
<td>Inlet mass flow rate (kg/s)</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>Dynamic viscosity (Pa s)</td>
<td>1.79E-05</td>
<td>1.79E-05</td>
</tr>
<tr>
<td>1/Permeability Constant (m^2)</td>
<td>2.00E-08</td>
<td>2.00E-08</td>
</tr>
<tr>
<td>Density (kg/m^3)</td>
<td>1.225</td>
<td>1.225</td>
</tr>
<tr>
<td>Cross-sectional Area (m^2)</td>
<td>2.41E-02</td>
<td>2.41E-02</td>
</tr>
</tbody>
</table>

Results:

<table>
<thead>
<tr>
<th></th>
<th>ANSYS FLUENT Solution</th>
<th>Analytical Solution</th>
<th>Relative Error, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure drop, Pa</td>
<td>Dp1: 6.068</td>
<td>Dp2: 12.13</td>
<td>Dp1: 6.07</td>
</tr>
<tr>
<td></td>
<td>Dp2: 12.14</td>
<td></td>
<td>Dp2: 0.0</td>
</tr>
</tbody>
</table>

• WB Python journal is used for getting inputs from MS Excel, performing the simulation and publishing results back to MS Excel
• Similarly other simulations (Mechanical, HFSS etc.) can be done
Customizing Mixing Simulation in WB

1. Quantitative Data

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid Density</td>
<td>9.962e+02 [kg m^-3]</td>
</tr>
<tr>
<td>RPM</td>
<td>1.000e+02</td>
</tr>
<tr>
<td>RPS</td>
<td>1.667e+00</td>
</tr>
<tr>
<td>Rad/sec</td>
<td>1.047e+01 [radian s^-1]</td>
</tr>
</tbody>
</table>

Geometry Parameters are exposed in WB for design point studies.

Different analyses can be combined easily.

An unified customized report can be generated.

- Generate the Mixing Tank interactively.
- Geometry Parameters are exposed in WB for design point studies.
- Different analyses can be combined easily.
- An unified customized report can be generated.
Using the Excel Component in WB

- L = 30
- d = 3

\[ 0.05 \times L = 1.5 \rightarrow \text{constrained } d = 1.5 \]

\[ d \text{ is constrained only if } d > 0.05L \]

(d and L are now dependent)
Custom GUI on WB

- A custom GUI opens that allows a Pipe Flow CFD Simulation.
- Provide the dimensions of the pipe.
- Provide the inputs.
- Validate the flow condition.
- Provide the inputs.
- Generate Mesh. A suitable mesh with appropriate Y+ is generated.
- View Mesh.
- Provide solver controls.
- Provide some inputs for extracting results (radial profiles).
- Iterate. The residuals are displayed as the simulation is progressing.
- Some standard results are displayed for a quick check.
- Plots generated are available for display.
- Open a standard report (with plots, contours, vectors etc.).
- Open CFD Post for an interactive post-processing.
Customizing Mixing Simulation in WB

1. Quantitative Data

<table>
<thead>
<tr>
<th>Table 1. Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fluid Density</strong></td>
</tr>
<tr>
<td><strong>RPM</strong></td>
</tr>
<tr>
<td><strong>RPS</strong></td>
</tr>
<tr>
<td><strong>Rad/sec</strong></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2. Moment, Power and Power Number Data</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Boundary</strong></td>
</tr>
<tr>
<td>-------------</td>
</tr>
<tr>
<td>Impeller_1</td>
</tr>
<tr>
<td>Impeller_2</td>
</tr>
<tr>
<td>Total</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3. Flow Number data</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Boundary</strong></td>
</tr>
<tr>
<td>---------------</td>
</tr>
<tr>
<td>Impeller_1</td>
</tr>
<tr>
<td>Impeller_2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 4. CFD based Mixing Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Variables</strong></td>
</tr>
<tr>
<td>Total Volume (Liters)</td>
</tr>
<tr>
<td>Speed (RPM)</td>
</tr>
<tr>
<td>Speed (RPS)</td>
</tr>
<tr>
<td>Torque (N-m)</td>
</tr>
<tr>
<td>Total Power (W)</td>
</tr>
<tr>
<td>Total Power (hp)</td>
</tr>
<tr>
<td>Power Per Unit Volume (W/m3)</td>
</tr>
<tr>
<td>Total Power Number (Po)</td>
</tr>
</tbody>
</table>

- Generate the Mixing Tank interactively
- Geometry Parameters are exposed in WB for design point studies
- Different analyses can be combined easily
- An unified customized report can be generated
Other possibilities (Common)
External Application Integration

- External Connection
  - Allow external applications to participate in the workflow defined through the project schematic

- Allow custom Toolbar button and menus
External Application Integration

Example:
- nCode is integrated in Workbench Project Schematic using C# add-in (SDK)

Typical Usage:
- Integrate in-house/third-party codes deep in WB workflow
- Create Customized GUI on WB
Conclusion

• ANSYS provides multiple methods to extend and customize Workbench
• Power and complexity range from basic scripting to full programming
• Choose method that best fits your needs
THANK YOU!