Application Customization Toolkit

Jim Kosloski
What is ACT?

- ANSYS ACT is the unified and consistent tool for the customization and expansion of ANSYS products.
- Using ACT, you can create vertical apps or “extensions” to tailor ANSYS Workbench, Mechanical, Fluent, and Electronics Desktop and other products to meet your application-specific and multi-physics simulation needs.
- Many Apps already exist from ANSYS and from CAEA
- The use of binary extensions does not require a license
- Scripting and compiling extensions is now included in the Enterprise license.
What Can ACT Do?

- ACT provides 3 main resources to the user
  - Feature Creation
  - Process Compression
  - Workflow Integration
ACT Implementation

- Custom ACT features function just like regular Workbench tools, items are added to the tree and the settings can be modified in the details pane:
- Little or no instructions are needed to use ACT extensions
  - If you know how to use WB, you can use the extensions.
ACT Development

- ACT Extensions are created by a combination of XML files and Python Scripts.

**ACT Extension**

- **XML Definition File**
  - Defines and configures the extension.
  - Defines context, custom GUI, & callbacks to invoke IronPython functions.

- **IronPython Script**
  - Defines functions that respond to user interactions/GUI events.
  - Implements the behavior of the extension.
ACT in v17

- New ACT start page makes it easy to install and manage apps:
ACT v17 Improvements

- New access and documentation for almost the entire Mechanical tree structure.

- In previous versions ACT was limited to adding new features (specialized loads, postprocessing).

- Now ACT can be used to automate existing functionality:
  - Add mesh controls
  - Define coordinate systems
  - Add any existing load/boundary conditions.
Act wizards are used to build an entire analysis system. They can run at the project page or inside of applications or combination of both.
ACT Extensions

- ANSYS Inc has many pre-developed ACT extensions available for your use.
- Available on the customer portal
CAEA has developed several extensions as well.

- **CSVPlOT**: Plot data in WB from a comma separated values file
- **PLSER**: Plot mesh discretization error measure of SMXB, SDSG in WB
  - **Mechanical**
- **SPRING6DOF**: Insert a 6 DOF spring between 2 remote points in WB
  - **Mechanical**
- **UPCOORD**: Provide the functionality of the APDL UPCOORD command in WB
  - **Mechanical**
    - **Now available directly in Mechanical no app needed.**
- **RotateView**: Rotate view by specified angle about screen normal.

All of these are available in the Resource Library section of our website:

- [www.caeai.com/resources](http://www.caeai.com/resources)

At CAE Associates we have the expertise to develop ACT extensions to help you improve your efficiency in modeling and performing complex analyses in WB.

- Please contact us if you would like to discuss the development an ACT extension.
Topological Optimization using Mechanical

Pat Cunningham
- ANSYS topological optimization ACT extension for Workbench
  - The extension provides an easy-to-use interface which allows the user to setup structural optimization problems, post-process them, and export the data within ANSYS Mechanical.
ANSYS Topological Optimization

- Topology optimization: Basic steps
  - Drag and drop the ANSYS Topology Optimization tool from the Toolbox onto the Model cell of a static analysis system.

- In ANSYS Mechanical, add design optimization controls through the *ANSYS Topology Optimization* toolbar. This toolbar allows the user to:
  - Define Optimization Objectives
  - Define Optimization Constraints
  - Specify Manufacturing constraints
Topology optimization: Basic steps

After the topology optimization is solved, the user can post-process the results to show optimized topology density isosurface plots.

Export the optimized structure to a STL file.
  • Can generate solid geometry in SpaceClaim
1. Install the ACT extension
2. Activate the extension.
ANSYS Topological Optimization Procedure

3. Drag and drop a Static Structural analysis system into the project.
4. Drag and drop the ANSYS Topology Optimization extension onto the Model row of the Static system.
5. Attach geometry and open Mechanical.
6. Set up the static analysis in the standard fashion (assign materials, connections, mesh controls, loading and supports).

7. Solve and verify the results.
8. Set up the Topological Optimization using toolbar:
   – Use the Design Space pulldown to define the Design Region and any Exclusion Regions.
8. Set up the Topological Optimization using toolbar (continued):
   - Define any manufacturing constraints (size minimums, symmetry).
8. Set up the Topological Optimization using toolbar (continued):
   - Define an optimization objective and constraints.
9. Solve
10. Use The Capped IsoSurface display to view the resulting shape.
11. Export the mass optimized shape to an STL file.
12. Import the STL data into ANSYS SpaceClaim to reverse engineer the geometry.
ANSYS Topological Optimization Procedure

- Reverse Engineering in SpaceClaim:
  - Create a plane through the center of the part.
  - Repair > Fit Curves to the cross section.
ANSYS Topological Optimization Procedure

- Reverse Engineering in SpaceClaim:
  - Use Repair to Fix Curves or replace with sketch entities.
- Create a surface by filling the edges.
- Delete unwanted surfaces.
Select the outer perimeter and use the Pull feature with “Copy Edge” to create the outer flange surface.
ANSYS Topological Optimization Procedure

- Pull the outer flange 6 mm and the inner face 3 mm.
- Turn on the faceted mesh view to compare.
Create fillets with the Pull feature.
Add flanges using the sketch and pull features.
Bring the reverse engineered geometry into Mechanical and analyze.
ANSYS Topological Optimization Procedure

- Any further design adjustments required are easily accommodated by the ANSYS SpaceClaim modeler.