ANSYS/LS-Dyna Customization in Workbench Using ACT

Steven Hale
Senior Engineering Manager
CAE Associates, Inc.

June 13, 2013

© 2013 CAE Associates
CAE Associates

- CAE Associates is an engineering services firm specializing in all aspects of engineering simulation.
  - Incorporated in 1981 as an engineering consulting firm, specializing in finite element analysis.
  - One of the original 4 ANSYS Channel Partners, since 1985.
  - Built upon these foundations to become a quality provider of engineering simulation products and services to hundreds of corporations, both large and small.
CAE Associates

- 3 Major Areas of Focus:
  - Engineering Consulting
    - Provide solutions to complex structural, thermal, fluid, and multiphysics challenges.
  - Sales and Technical Support of the ANSYS Suite of Simulation Software
    - CT, LI, Northern NJ, NY Metro, Western MA
  - ANSYS FEA and CFD Training
    - Application specific
    - Coaching and mentoring
Areas of Expertise

- Structural Analysis
  - Composites
  - Fatigue
  - Implicit and Explicit Dynamics
- Thermal Analysis
- CFD
  - Aerodynamics
  - Turbo machinery, Propulsion
  - Coupled Thermal-Flow
  - Fluid-Structure Interaction
  - Chemical reacting flows
- ANSYS Customization
Our Approach

- Maximize the value of simulation to our clients by ensuring they get the best combination of engineering software and services to achieve their objectives.

- Provide the highest level of expertise every step of the way to ensure a fast, successful implementation, as well as ongoing support.
Introduction

- The Workbench/LS-Dyna ACT Extension is a tool that adds considerable functionality and usability to the existing Workbench/LS-Dyna tool: Explicit Dynamics (LS-Dyna Export).
  - Combines the power of the LS-DYNA solver with the ease-of-use of the ANSYS Workbench environment.
  - LS-Dyna is a very popular commercial explicit dynamics solver.
    - Very robust and fast for solving high-energy transient events.
    - Applications include impact, crash, drop tests, manufacturing, buckling, etc.
Outline

- What is ACT and what can it do?
- ACT implementation, development, and licensing
- ACT extensions available for download
- Workbench/LS-Dyna ACT extension and demo
What is ACT?

- Since ANSYS Workbench is a general purpose FE element tool, it does not always provide the most direct method for analyzing specific situations.

- ACT (Application Customization Toolkit) is a new capability in WB that allows you to add custom features to the Workbench Mechanical interface.
  - Custom features can be specific to your industry or to a particular model.
  - No need to compile external code or link with ANSYS libraries.

- ACT extensions can relieve some of the pain and difficulties in specific modeling areas
  - Eliminate the need for command blocks.
  - Automate certain hand calculations.
  - Easily incorporate previously-developed APDL macros into WB Mechanical.
What Can ACT Do?

- ACT customizations can include things like:
  - Specialized loads
    - Examples: Acoustic pressure, Rigid body force (LS-Dyna)
  - Element types and options not directly available in WB
    - Examples: Acoustic elements, LS-Dyna shell/solid formulations
  - Specialized post-processing
    - Examples: Acoustic pressure, Energy time-history results (LS-Dyna)
  - FE model information and access
    - Examples: Node and element #s and display
  - Use of MAPDL macros
    - Example: Menus that call and send input parameters to a custom MAPDL convection load macro
  - Use of 3rd party solver
    - Example: LS-Dyna
ACT Implementation

- ACT customizations can show up as new tool bar buttons

- Or as new menus
ACT Implementation

- These custom buttons function just like regular Workbench tools
  - Items are added to the tree and settings can be modified in the details pane.
  - Scoping is the same, specifying load direction is the same, etc.
  - No need to learn a new methodology.
ACT Development

- ACT Extension files are created by combining XML files and Python Scripts.
  
  - XML files define the menus, buttons, icons, and details information.
  
  - Python scripts perform the operations in Mechanical
    - Main function – Extract the data from the user-entered details and write items to the input file (ANSYS DS.dat file or LS-Dyna .k file).
ACT Licensing

- To use ACT extensions no additional licensing is required!
  - To install a compiled ACT extension file simply go to Extensions > Install Extension and browse for the .WBEX file
  - To use the extension in a particular project, go to Extensions > Manage Extensions and check off the extensions to use.

- To develop and compile ACT extensions an additional license (ANSYS SDK license) is required.
There are many pre-developed free ACT extensions available for your use. Some examples include:

- **3D_Surface_Effect Extension R145 v1**
  Create a 3D surface effect using SURF154 elements

- **Acoustics Extension R145 v6**
  Expose 3D acoustics solver capabilities

- **BeamEndRelease Extension R145 v1**
  Expose the end release feature for beam elements and enable advanced graphic post-processing for beam results

- **Convection Extension R145 v1**
  Expose convection with pilot node capability

- **FE Info Extension R145 v4**
  Expose node and element related information

- **FSI Transient R145 v1**
  Map temperature and pressure loads (from a CFD calculation) to a multi-step Mechanical analysis for transient one-way FSI

- **MatChange R145 v1**
  Change material ID to user specified value for the selected bodies

- **Morphing2D Extension R145 v1**
  Perform a set of morphing capabilities on 2D models

- **Non Linear Results Info Extension R145 v1**
  Enable tracking for non-linear solutions (contact & Newton-Raphson residuals)

- **Piezo Extension R145 v2**
  Expose piezo-electric solver capabilities

- **Submodeling Extension R145 v1**
  Sub-modeling for shell-to-solid (R14.5 native implementation already supports solid-to-solid sub-modeling)

- **Workbench LS-DYNA R145 v2**
  Fully integrated access to ANSYS LS-DYNA with all the power of Workbench through the Mechanical GUI
Current State of LS-Dyna in Workbench

- Explicit Dynamics (LS-Dyna Export) utility
  - Limited access to LS-Dyna functionality
  - Cannot solve in Mechanical
  - Cannot post-process in Mechanical
  - Other limitations:
    - Limited shell and solid section types
    - No eroding contact
    - Only global damping and hourglass controls
      - Cannot apply different values/types to different parts
    - Cannot create point masses, springs, or dampers
    - Cannot apply certain load types to rigid bodies
    - Cannot track contact or reaction forces
Workbench/LS-Dyna ACT Extension

- Workbench LS-DYNA is an ACT-developed user environment that adds considerable functionality and usability.
  - Used for preprocessing, solving, and post-processing LS-DYNA models.
  - New at Version 14.5.
  - Requires an ANSYS LS-DYNA license.
Main Features

- New functionality in some existing menus plus ACT add-on menus

LS-Dyna ACT toolbar

- Rigid body loads and constraints
- Bonded (Tied), frictional, eroding, and single-surface contacts
- Point masses, beam connectors, springs, and dampers
- Access additional LS-Dyna solid and shell formulations
- Access most LS-Dyna hourglass controls
- Result tracking for high-frequency output at specific nodes and for contact forces
- Comprehensive post processing
  - Do not need to learn LS-PrePost
Demo: Pliers Drop Test

- Pliers dropped from 20 ft. onto a hard surface
  - Goal: Obtain max. tensile stress history in the bottom clamp
    - Impact velocity = 430 in/s
    - Can model all parts as rigid except for the bottom clamp
    - Can use bonded contact as shown and frictional contact at all other joints
Demo: Pliers Drop Test

- Assign:
  - Impact velocity
  - Constrain impact surface
    - Rigid body constraint
  - Insert a body contact tracker
    - Allows for graphing contact/impact forces
  - Analysis settings
    - Transient time (0.01 seconds)
    - Mass scaling to speed up the run
    - Number of CPUs
    - Number of output time pts to write
Demo: Pliers Drop Test

- Instructions
  - Change all bodies except for “Clamp bot” to rigid
  - Connections
    - Delete the bonded contacts at all locations except “Bonded – Clamp top to Arm top”
    - Add 2 manual bonded contacts between the top of the CenterLink and Arm top

- Keep Body Interaction contact active (and frictional)
  - Set the friction coefficient to 0.2
Demo: Pliers Drop Test

- **Mesh**
  - Physics Preference = Explicit
  - Advanced Size Function = Off
  - Element Size = 0.04 in.

- **Initial Conditions**
  - Velocity (430 in/s normal to the floor surface)

- **Analysis Settings:**
  - End Time = 0.01 sec.
  - Automatic Mass Scaling = Yes
    - Time step Size = 1.e-7
  - Number of CPUs = 2
  - Solver Controls:
    - Unit System = Bin
  - Output Controls:
    - Calculate Results At = Equally Spaced Points
      - Value = 80
    - Time History Output Controls:
      - Calculate Results At = Equally Spaced Points
        - Value = 200
Demo: Pliers Drop Test

- Rigid Body Constraint
  - Apply to the floor (surface body)

- Other items not required for this demo but may want to show:
  - Analysis Settings: Many different hourglass controls, Damping
  - Can insert pressure, force, displacement, velocity, or acceleration vs. time

- Solution:
  - Insert Total Deformation
  - Maximum Principal Stress scoped to the bottom clamp
  - ASCII > Global Data (Add Kinetic and Internal Energy)
    - Use Worksheet to plot
  - ASCII > Contact Force
    - Contact = Body Interaction
    - Scope to the floor body
Demo: Pliers Drop Test

- Maximum principal stress in the bottom clamp
Demo: Pliers Drop Test
Conclusions

- The Workbench/LS-Dyna ACT extension allows you to rapidly set up models in Mechanical for solving high-energy dynamic analyses in LS-Dyna.
  - Provides much more functionality than the current Workbench/LS-Dyna tool.
  - Allows post-processing in Mechanical (or LS-Prepost if you prefer).
  - Can leverage all of the power of Workbench for geometry processing, parameterization, meshing, and load application.