TryItNow!
Step by Step Walkthrough: Automotive Fuse

ANSYS designed this TryItNow! experience to give you a quick access to ANSYS AIM simulations on the cloud. This AIM demonstration shows a steady-state thermal-electric-stress analysis of a fuse assembly. The simulation demonstrates the ease of setting up and solving a nonlinear multiphysics simulation. Default settings are used to highlight ease-of-use and to demonstrate how AIM automates many of the steps required for multiphysics simulation.

1 Select Structural Template
Select the Electric Conduction template from the Study menu in AIM

2 Template Settings
• Select Structural and Thermal as additional physics
• Select “Create Simulation Process”
• Select the TryItNow-AutomotiveFuse, then geometry file Fuse.sat to execute the template.

3 Display
• Activate Translucent Display to visualize the internal components of the assembly.

4 Review the Physics panel
• Select the Physics Tasks and review the Physics panel. The initial settings are automated based on the selected template. In each section you can review the setup and modify it based on your own requirements.
• The material assignments, interface conditions and solver options can all be reviewed.
• Next click on “Material Assignments” to specify the materials for the fuse assembly.

Ready to learn more about ANSYS AIM? CONTACT US
5 Specify Aluminum for fuse
   • Selection filters enable you to switch between selecting points, lines, surfaces or bodies.
   • Turn on the Body selection filter.

   • Select the fuse body, right click and add Material Assignment from Physics Modeling.

   • Select Aluminum Alloy for the fuse material.

6 Specify Polyethylene for Cover
   • Use 'Ctrl' to select the two cover bodies and right click and add another Material Assignment from Physics Modeling.
   • Select Polyethylene for the cover material. You can start typing in the name to quickly find the material.

7 Remove the default assignment
   • Select the Structural Steel Assignment from the Fix menu.

   • Delete the assignment.

8 Specify Current of 30 Amps
   • Change the selection filter back to Face Selection

   • Click on the Z-axis in the Triad to align the view.
   • Hold down ‘Shift’ to box select the terminal faces.
   • Right-click to add a Current Boundary Condition of 30 A.
9 Specify Ground Voltage of 0 Volt
- Hold down ‘Shift’ to box select the terminal faces.
- Right-click to add a Voltage Boundary Condition of 0 V.

10 Specify Temperature of 30°C
- Hold down ‘Shift’ to box select faces of both terminals
- Right-click to add a Temperature Boundary Condition of 30°C.

11 Specify Support
- Use ‘Ctrl’ to multi-select the two bottom faces, then right-click to add a Support condition

12 Update Results
- Click on Results task in the workflow view.
- Right-click to select Update to generate the mesh, execute the solution, and calculate results.

13 Post-process Current Density
- Select pre-defined Current Density results from the Search and Select Bar.
- Right-click to animate results.

Ready to learn more about ANSYS AIM? CONTACT US
Add additional Results and Evaluate
• Select fuse element and use right-click context menu to add results for Temperature and Equivalent Stress.

• Right-click to Evaluate Results.

• Switch between currently defined results from the Search and Select Bar.

• Right-click to animate results.

ANSYS, Inc. is one of the world’s leading engineering simulation software providers. Its technology has enabled customers to predict with accuracy that their product designs will thrive in the real world. The company offers a common platform of fully integrated multiphysics software tools designed to optimize product development processes for a wide range of industries, including aerospace, automotive, civil engineering, consumer products, chemical process, electronics, environmental, healthcare, marine, power, sports and others. Applied to design concept, final-stage testing, validation and trouble-shooting existing designs, software from ANSYS can significantly speed design and development times, reduce costs, and provide insight and understanding into product and process performance. Visit www.ansys.com for more information.