Analyzing Plastic Parts with Finite Element Analysis

Patrick Cunningham
October 14, 2014
Agenda

- 8:00-8:30 Welcome and General Introduction
- 8:30-9:15 Introduction to DatapointLabs – Hubert Lobo
- 9:15-10:15 Keynote: Plastics and Simulation – Hubert Lobo – DatapointLabs
- 10:15-10:30 Break
- 10:30-11:00 Keynote: Continued
- 11:00-11:45 Introduction to MatWeb – Barbara Hudock – MatWeb
- 11:45-12:30 ANSYS Demonstration – Plastic and Simulation – Patrick Cunningham – CAE Associates
- 12:30-1:30 Lunch
- 1:30-3:00 Keynote Part 2: Plastics and Simulation – Hubert Lobo – DatapointLabs
- 3:00-3:15 Break
- 3:30-3:45 Case Study – Including Strain Rate Effects – Steve Hale – CAE Associates
- 3:45-4:00 Case Study – Time Dependant Response of Plastics – Mike Bak – CAE Associates
- 4:00-4:15 Concluding Remarks
What is FEA?

- Finite Element Analysis is a way to simulate loading conditions on a design and determine the design’s response to those conditions.
- The design is modeled using discrete building blocks called elements.
- Coefficients that describe the response of the material used are assigned to each element.
- The “sum” of the response of all elements in the model gives the total response of the design.

**Physical System**

**F.E. Model**
Modeling Plastic Parts with FEA

- Consider a “highly sophisticated inter-locking brick system” like the one shown.
- Using FEA we can:
  - Determine the amount of force required to push Lego Man’s foot onto a brick.
  - Determine the deformation and resulting stress in both parts.
  - Evaluate the sensitivity of the design to manufacturing variances.
  - Evaluate the sensitivity of the design to material property variances.
  - Evaluate the response of the design under different environmental conditions that can effect the material response.
- All this can be done with FEA provided that you have appropriate material properties assigned to the elements.
Modeling Plastic Parts with FEA

- Step 1: Geometry and mesh generation.
  Most geometry comes from CAD tools like SolidWorks, ProEngineering, Inventor, etc. The geometry is imported into the ANSYS environment from the CAD program where the finite element mesh is generated.
Step 2: Material Properties

The material properties are used to calculate the stiffness of the parts in the analysis as well as the relationship between deformation and stress. The results of the finite element model are only as good as the material properties provided (Garbage In, Garbage Out).
Obtaining Material Properties

- Finding the properties for the materials used in your design is often one of the biggest challenges of the modeling process.
- Matweb.com has an extensive database of materials that is easily searchable.
- With an ANSYS/Matweb partner license you can download material files that can be imported directly into ANSYS Workbench.
Obtaining Material Properties

- Once the materials are downloaded and imported into ANSYS Workbench, materials are assigned to each body.
Step 3: Analysis Setup

Inside ANSYS the required connections (contact regions) and boundary conditions are defined. Symmetry on the block and a fixed support on the hole at the top of the leg is defined.
Modeling Plastic Parts with FEA

- Step 4: Loading

The loading for this model consists of a specified displacement of .0669” (1.7mm) pushing the bloc up into the leg.
Step 5 – Evaluating Results

The displacement load reports a reaction force (the force required to press the block into the leg cavity). With linear elastic materials and a coefficient of friction of 0.1 the force required around .33#. 

![Image of FEA model](image-url)
Modeling Plastic Parts with FEA

- The Von Mises stress in the leg when the block is fully pressed in is
- Comparing to the tensile yield stress of the material (provided with the Matweb material data) yields a safety factor around 0.97 for the leg and 1.4 for the block.
Modeling Plastic Parts with FEA

- However, when we look more closely at the result history we can see that there is a localized peak stress that occurs when the block first contacts the leg.
- This peak stress value is well in excess of the elastic limit of the material.
Including Nonlinear Material Effects

- Once the yield stress has been exceeded in a ductile material unrecoverable deformation and strain will occur (plasticity).
- The material tends to soften and the load is redistributed to the surrounding region.
- Once this occurs the relationship between stress and strain changes and a nonlinear material model is needed.
DatapointLabs Matereality database can provide nonlinear material properties in a format that is readable to the ANSYS Program.
Obtaining Nonlinear Material Properties

- There are two options for using advanced material models from Datapoint Labs:
  1. Request the data in an XML format to be imported directly into the Workbench Engineering Data page.
  2. Use a command block to overwrite the default material setting in Workbench using traditional APDL commands. The command block attribute parameter “matid” makes this easy to accomplish for each body.
Nonlinear Results

- Allowing the material to deform plastically redistributes the load over a larger area.
- The redistribution of load lowers the peak stress observed in the linear model.
Nonlinear Results

- The plastic strain region experiences non-recoverable (permanent) deformation.
- This change on the shape of the part will effect the pressure distribution the next time the leg is pressed onto the block. This may be a life limiting effect on the product.
Summary

- The best finite element model is only as good as the material properties used.
- The ANSYS can accommodate a wide range of material models including:
  - Nonlinear elasticity
  - Hyperelasticity
  - Elastic/plastic response
  - Viscoplasticity
  - Creep
  - Temperature dependency
- One of the biggest challenges in finite element modeling is obtaining representative material properties.
- User defined material libraries are easily defined in ANSYS.