A 1,000 liter water container is extrusion blow molded. The container is placed in a steel cage to reduce the amount of deformation when filled. The blow molding process is simulated using ANSYS Polyflow. The initial thickness distribution of the parison is controlled resulting a weight reduction by 10% and reduction in maximum principle stress by 17%. The thickness distribution obtained from the blow molding simulation is mapped onto a static structural FEA model using ANSYS Mechanical to predict the part performance under hydrostatic loading.

**Design Challenge**
- From the Manufacturing (Blow Molding) Process:
  - Is the part completely formed?
  - Are there any defects (knit lines)?
  - What is the final material/thickness distribution?
- Resultant Part Performance (FEA Testing):
  - Will the final part have the required properties?
    - When stacked on top of each other, will the part have enough “top load” strength (linear, non-linear buckling)?
    - If dropped during use or transportation, how will it perform? (drop test)
    - Other performance characteristics (side squeeze, puncture resistance, hydrostatic loading)

**ANSYS Simulation-Driven Design Flow**

**Final Material Distribution**

**Mapped Thickness Variation**

**Different (Independent) Meshes**

**Hydrostatic Loading [1 ton of water]**

**Total Deformation**

**Total (von Mises) Stresses**

**Initial Thickness Optimization**

**Improvement in Final Thickness**

**Total (von Mises) Stresses Comparison**

17% reduction in max stress