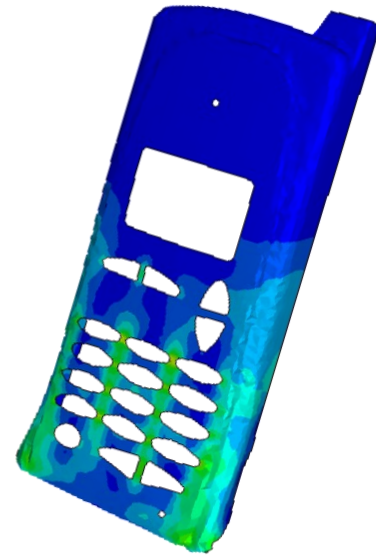


Warping and Residual Stress Analysis using the Abaqus Interface for Moldflow

Summary

Residual stresses may be introduced into plastic parts produced by the injection molding process. As a result, the part may warp or experience a reduction in strength. The design of an injection molded product can be improved if the effect of residual stresses on the final shape and performance of the product are predicted accurately. Abaqus and Moldflow can be used for this purpose. The residual stresses generated by the solidification of the plastic material are computed by Moldflow and transferred to Abaqus using the Abaqus Interface for Moldflow. The component can then be structurally analyzed with Abaqus to determine warpage and/or response to in-service loading. In this Technology Brief, this methodology is demonstrated with two case studies.



Background

The quality of a product manufactured by the injection molding process can be sensitive to residual stress. Excessive residual stress can lead to warping, shrinkage, or unexpected failures when the part enters service. Further, product assembly may be hindered or prevented if warping exceeds a given tolerance. In order to improve the designs of injection molded products, it is important to understand the magnitude, distribution, and effects of process induced residual stresses.

Moldflow is mold design software that can be used to estimate the residual stresses in a molded product. In order to determine how these residual stresses affect the final shape and performance of the product, a detailed structural analysis that accounts for the residual stresses is necessary. Abaqus can be used for this purpose, and the Abaqus Interface for Moldflow provides for the transfer of stress results from a Moldflow analysis to an Abaqus analysis.

In this Technology Brief we present two case studies in the structural analysis of molded products. The first case study presents a plastic chair model and the second case study involves the analysis of a cellular phone cover.

Key Abaqus Features and Benefits

- Mechanical property, finite element mesh, and residual stress data can be easily transferred from Moldflow to Abaqus
- The inclusion of residual processing stresses in an Abaqus analysis allows for more accurate simulation of the injection molded product

Analysis Approach

The procedure for performing a warping and stress analysis on a molded product begins with a simulation of the injection molding process using Moldflow. The Moldflow analysis results include descriptions of the material properties and the distribution of residual stress in the solidified part. The Abaqus Interface for Moldflow is then used to translate this data into a form that can be used by Abaqus; specifically, the interface generates files that contain mesh information, residual stress results, and material properties of the plastic. These data are then used in subsequent Abaqus analyses for modeling warping and the effects of residual stresses.

The discretized models of the molded chair and cellular phone cover are shown in Figure 1. For both models, the Moldflow analyses used shell elements with 21 layers through the thickness. The warping simulations were performed using the static analysis capability in Abaqus/Standard.

Contour plots of Mises stress from the Abaqus/Standard warping analyses are shown for the chair and phone cover models in Figures 4 and 5. It can be clearly seen that, with the warping deflection comes a reduction in the magnitude of Mises stress stored in both parts.



Figure 1: Chair and phone cover meshes.

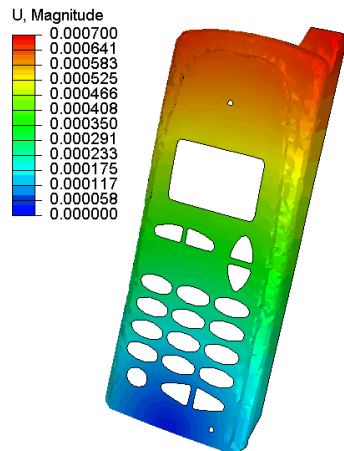


Figure 3: Warping displacements [m] for the phone cover model.

Results and Discussion

The deformed shapes of the chair and cell phone cover models after the warping analyses in Abaqus/Standard are shown in Figures 2 and 3.

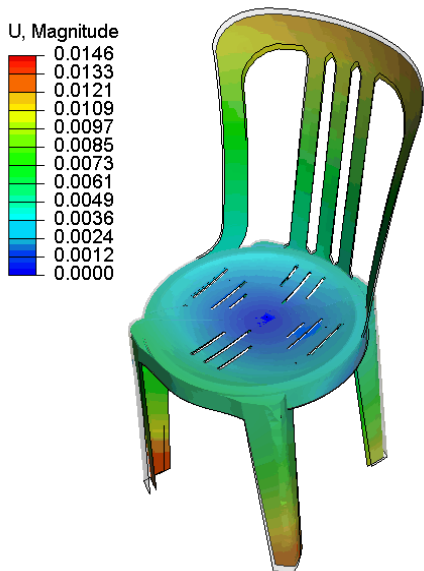


Figure 2: Warping displacements [m] for the chair model.

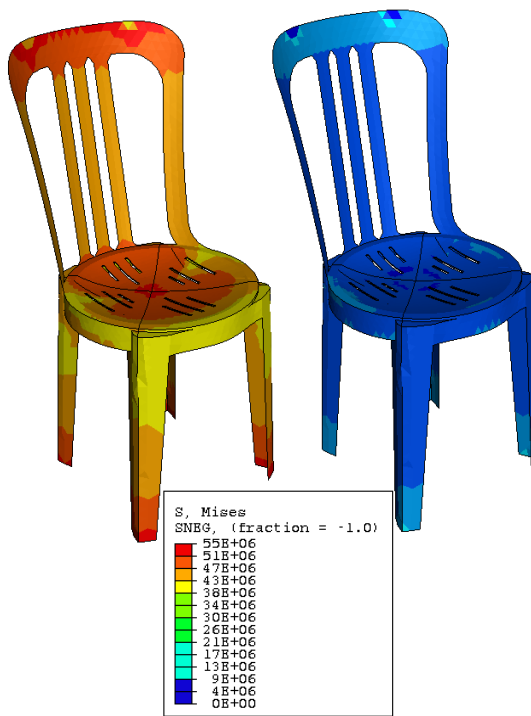


Figure 4: Mises stress [Pa] for the chair model: Before warping (Left) and after warping (Right).

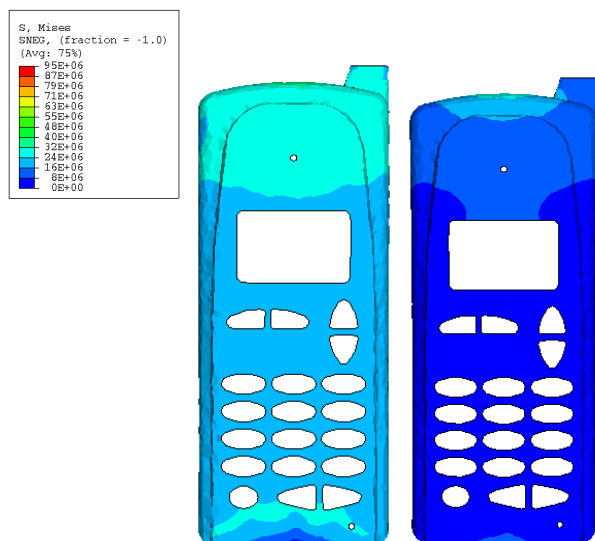


Figure 5: Mises stress [Pa] for the phone cover model:
Before warping (Left) and after warping (Right).

Conclusion

Abaqus provides a powerful capability for performing detailed structural analyses. Moldflow provides the ability to calculate residual stresses and material properties for an injection molded product. By providing the means for transferring Moldflow results into an Abaqus analysis, the Abaqus Interface for Moldflow allows for a more accurate and efficient design process.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.11 documentation reference:

- Abaqus Interface for Moldflow User's Manual.

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include ABAQUS, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright © 2007 Dassault Systèmes

