International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 3, July 2013

Research Paper

PRETENDING THROUGH ANSYS OF RESIDUAL STRESSES IN CAST IRON TRACTOR HUB

Mohd Rehan[1]*, Sachin Chaturvedi[2] and Mohd Parvez[1]

*Corresponding Author: **Mohd Rehan,** ✉ discoverrehan@gmail.com

This work presents a study and implementation of the pretending of residual stresses in molding. The objects of study are a cast iron tractor Hub part (provided by the company ESCORT, Faridabad) and an optimized version of the Hub resulting from the application of a topology optimization process. The models are solved through an uncoupled thermo-mechanical solidification analysis, performed both in the FE commercial software Ansys and the results. The results shows that the part subjected to the topology optimization process develop less residual stresses than its original Hub.

Keywords: ANSYS, Residul stresses

INTRODUCTION

During the solidification process of castings, residual stresses are developed due to temperature gradients between different parts of the casting, mechanical constraints imposed by the mold during shrinkage of the cast metal, and volumetric change and transformation plasticity associated with the solid state phase transformation.

This work presents a comparison of residual stress development between parts that has and has not undergone topology optimization processes. As well, we provide a detailed procedure to carry out residual stress simulations in Abaqus and the steps for the geometry preparation, mesh generation and results comparison using Ansys.

The residual stresses are calculated using an uncoupled thermo-mechanical solidification analysis. A thermal analysis is performed first and then, the thermal history is read into a quasi-static mechanical analysis to calculate the residual stresses, using a J2-plasticity model.

[1] Department of Mechanical Engineering, Al-Falah School of Engineering and Technology, Dhauj, Faridabad, India.
[2] Department of Mechanical Engineering, Brown Hills College of Engineering and Technology, Dhauj, Faridabad, India.

An academic problem is set using a simple geometry to implement and explain the procedure. Then, residual stresses are calculated on the tractor Hub part provided by ESCORT, Faridabad, and finally the same simulation is performed on a topologically optimized version of the mentioned part.

PURPOSE

Compare the residual stress development of parts subjected and not subjected to topology optimization processes. Present a methodology to perform numerical simulations of residual stresses. Compare solutions obtained from the FE solver Abaqus.

SIMULATION THROUGH SOFTWARE'S OF CYLINDER AND TRACTOR HUB

Next we present a general list of what have to be set to perform a residual stress simulation in a problem like ours. The following summary corresponds to an uncoupled thermo-mechanical analysis as described in the Introduction chapter of this work.

The Thermal Simulation

- Mesh the part
- Define the material properties
  - Casting: Density, Conductivity, Specific Heat, Latent Heat, Liquidus Temperature, Solidus Temperature.
  - Sand Mold: Density, Conductivity, Specific Heat.
- Define the initial boundary conditions
  - Initial temperature of the casting
  - Initial temperature of the mold
- Define the interactive boundary conditions
  - Conduction: Between the external surface of the casting and the surface of the mold cavity.
  - Convection: Between the external surface of the mold and the ambient.
  - Radiation: Between the external surface of the mold and the ambient.

The Stress Simulation

- Use the same mesh used in the thermal simulation for the casting (the mold is not present in our stress analysis)
- Define the material properties (Expansion Coefficient: Young's Modulus, Poisson's Ratio, Plasticity, Yield Stress, Plastic Strain)
- Define the initial boundary condition.
- Initial temperature of the casting (as in the thermal analysis)
- Load the nodal thermal history generated in the thermal simulation as a predefined temperature field.
- Define the mechanical boundary conditions.

Constrain the rigid body translations and rotations in X, Y and Z, but allow the body to deform.

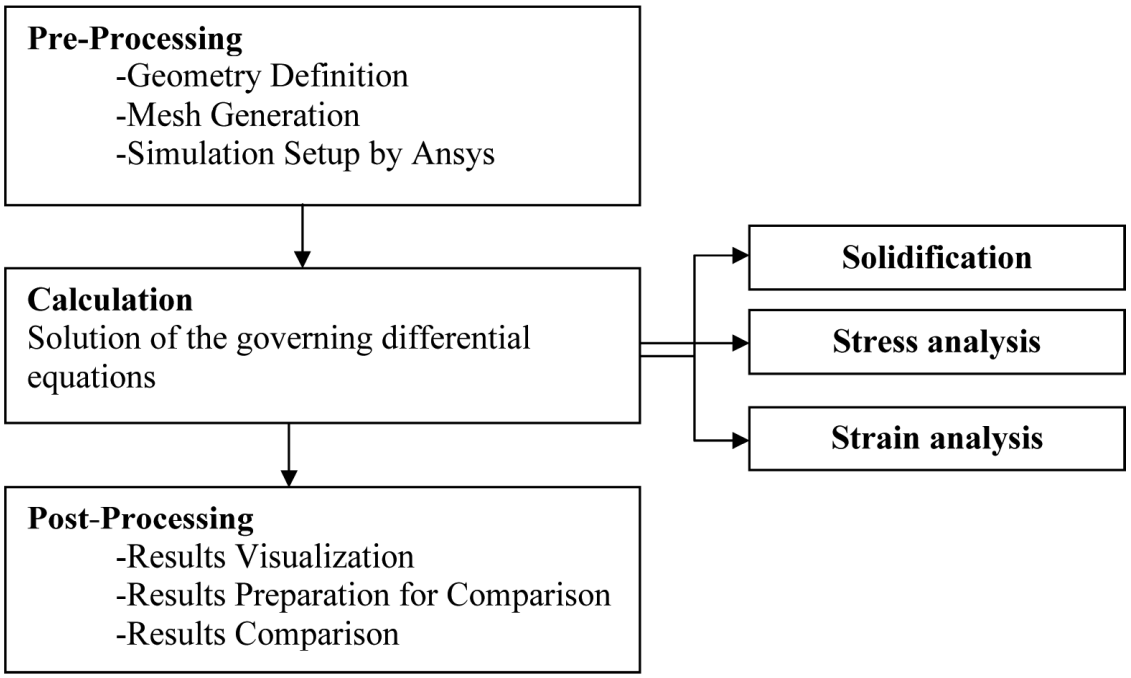
PROCEDURE

The simulation procedure steps are the following diagram:

ABAQUS SIMULATION SETUP

Here a step by step procedure to setup and run first the thermal simulation and then the stress simulation in Abaqus is presented and commented.

Figure 1: The Simulation Procedure



- Importing the models
- Materials definition
- Sections definition
- Sections assignment 5- Mesh element type 6- Assembly
- Steps definition
- Predefined Fields definition
- Interaction Properties definition
- Interactions definition
- Boundary Conditions
- Predefined Field Requests
- Job creation

RESULTS

Original Hub and Optimized Hub Von – Mises comparison.

Figure 2: Original Hub (Top) and Optimized Hub (Bottom) – Top View

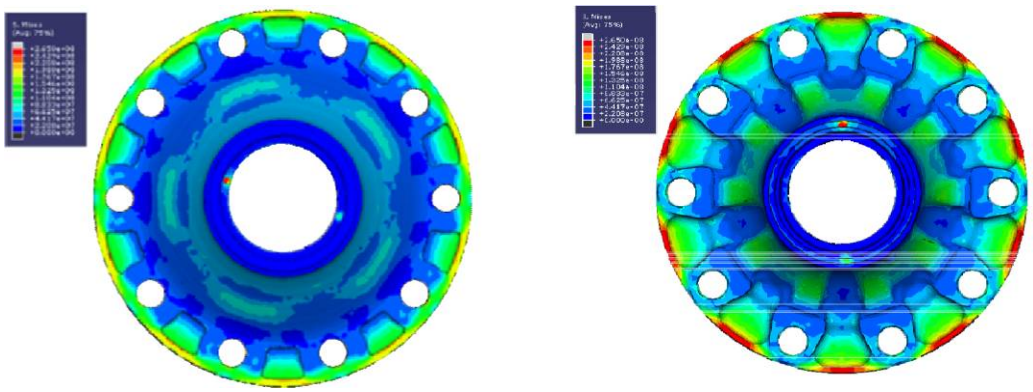


Figure 3: Original Hub (Top) and Optimized Hub (Bottom) – Bottom View

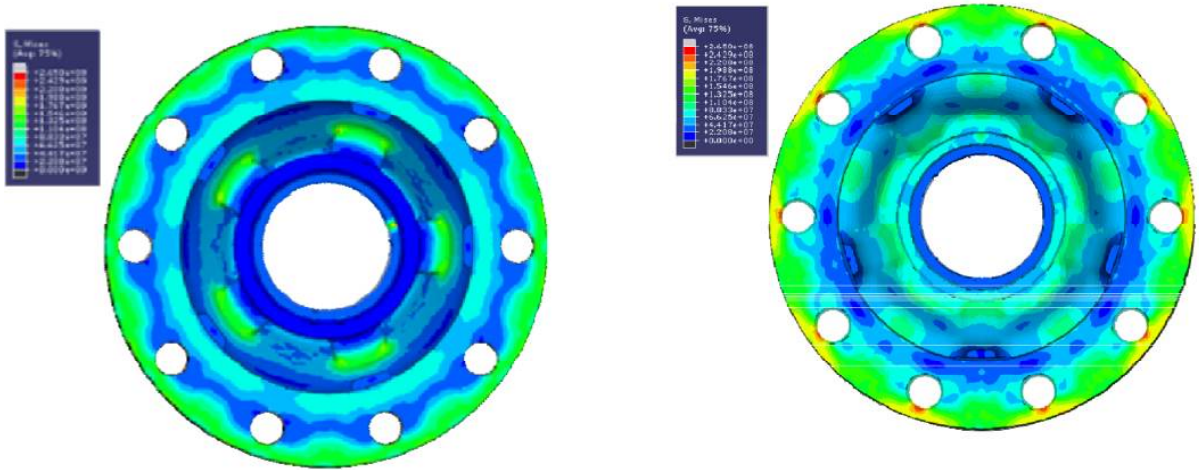
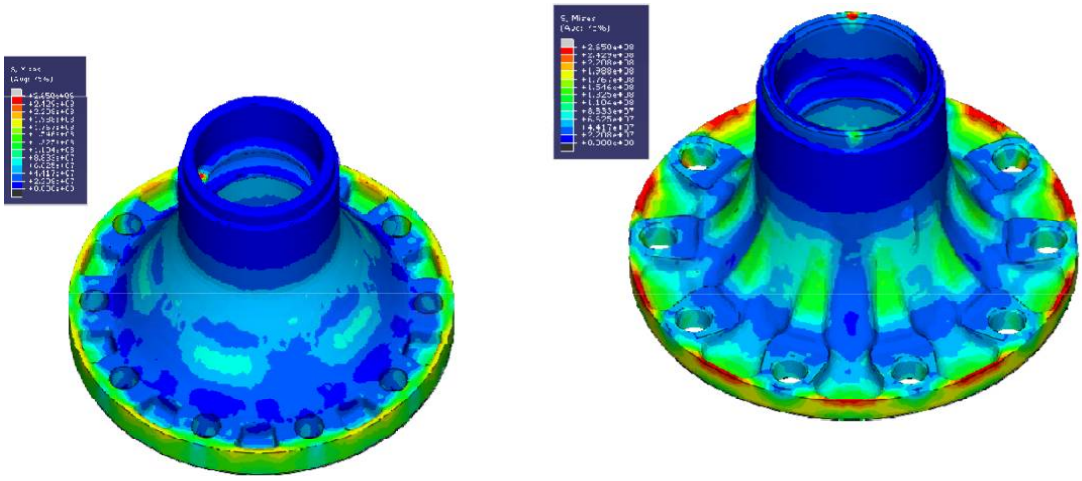


Figure 4: Original Hub (Top) and Optimized Hub (Bottom) – Inclined View



CONCLUSION

The implementation of residual stress analysis during the design of castings can lead to important improvements on the mechanical behavior of the final parts on aspects as crucial as fatigue life. Therefore, we strongly recommend the use of this type of numerical simulations as part of the design routine of casted parts.

As well, the difference in the residual stress development of parts that has and has not undergone topology optimization procedures, suggest the benefits of the inclusion of shape optimization in the design process.

BIBLIOGRAPHY

1. Abaqus 2010 Software
2. ANSYS Software
3. MATLAB 2009
4. Wikipedia, free encyclopedia (2012) http://en.wikipedia.org