Table of Contents

Figures Table

Bestpfe.com

1 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 according to Chandra U., Ahmed A. (2002). Since the residual stresses can increase or decrease the fatigue life of a component, an interest on its consideration during the design process has grown in the industry of casted parts. Scientific information supporting the validity of such interest is offered in Gustafsson E., Hofwing M., Stromberg N. (2007).

Considerable differences, when residual stresses are included or not in shape optimization processes of castings, has been presented in Chandra U., Ahmed A. (2002),
Modelling for casting and solidification processing,
Marcel Dekker, New York

Gustafsson E., Stromberg N. (2006). Differences between results of residual stresses obtained from the commercial softwares Abaqus and Magmasoft are also presented in Gustafsson E., Hofwing M., Stromberg N. (2007).

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, both in Abaqus and Magmasoft including the steps for the geometry preparation, mesh generation and results comparison using ProEngineer, Hypermesh and Matlab respectively. The results obtained from the two solvers are also compared and the theoretical fundamentals are given.

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.

An academic problem is set using a simple geometry to implement and explain the procedure. Then, residual stresses are calculated on the truck Hub part provided by Volvo 3P, and finally the same simulation is performed on a topologically optimized version of the mentioned part.

1.1 Background

The thermal analysis

The governing equation for the thermal analysis is the classical heat equation

$$\rho \frac{\partial H}{\partial T}\frac{\partial T}{\partial t} = div[k\nabla T] \tag{1.1}$$

Where ρ , H and k are temperature dependent and represent density, enthalpy and thermal conductivity respectively. T is the temperature and t is the time.

The stress analysis

The equilibrium equation for the residual stress analysis is

$$div[\sigma] = 0 \tag{1.2}$$

Where σ is the stress tensor.

The yield surface equation for the J_2-plasticity model reads

$$f(\sigma, T) = \sqrt{3J_2} - h\varepsilon^{-p} - \sigma_y \le 0 \tag{1.3}$$

Where J_2 is the second invariant of the deviatoric stress tensor, h is the temperature dependent hardening parameter, ε^{-p} is the equivalent plastic strain and σ_y is the temperature dependent yield strength.

BESTPFE.COM
List of research project topics and materials

1.2 Purpose and aims

1-Compare the residual stress development of parts subjected and not subjected to topology optimization processes.

2-Present a methodology to perform numerical simulations of residual stresses.

3-Compare solutions obtained from the FE solver Abaqus and the FD solver Magmasoft.

1.3 Delimits

For the purpose of our work, a general understanding of the Finite Element and the Finite Difference formulations are sufficient. This work does not present the mathematical details of the FE or the FD method.

No details about the topology optimization process are intended to be provided in this work. The topologically optimized version of the part provided by Volvo 3P was given.

2 Theoretical background

2.1 The Thermal Analysis

This chapter aims to provide basic information related with the simulation of solidification in castings about heat transfer mechanisms, material properties, boundary conditions, the heat conduction equation and the numerical methods.

2.1.1 Heat transference

When a system is at a different temperature than its surroundings, the Nature tries to reach thermal equilibrium. To do so, as the second law of thermodynamics explains, the thermal energy always moves from the system of higher temperature to the system of lower temperature.

This transfer of thermal energy occurs due to one or a combination of the three basic heat transport mechanisms: Conduction, Convection and Radiation.

2.1.1.1 Conduction

Is the transference of heat through direct molecular communication, i.e. by physical contact of the particles within a medium or between mediums. It takes place in gases, liquids and solids. In conduction, there is no flow of any of the material mediums.

The governing equation for conduction is called the Fourier's law of heat conduction and it express that the heat flow per unit area is proportional to the normal temperature gradient, where the proportionality constant is the thermal conductivity:

$$q = -kA\frac{\partial T}{\partial x} \tag{2.1}$$

Where q is the heat flux perpendicular to a surface of area A, [W]; A is the surface area through which the heat flow occurs, [m^2] ; k is the thermal conductivity, [W/(mK)]; T is the temperature, [K] or [°C]; and x is the perpendicular distance to the surface traveled by the heat flux.

2.1.1.2 Convection

Is the heat transfer by mass motion of a fluid when the heated fluid moves away from the heat source. It combines conduction with the effect of a current of fluid that moves its heated particles to cooler areas and replace them by cooler ones. The flow can be either due to buoyancy forces (natural convection) or due to artificially induced currents (forced convection).

The equation that represents convection comes from the Newton's law of cooling and is of the form:

$$q = -hA(T_\infty - T_s) \tag{2.2}$$

Where h is the convective heat transfer coefficient [W/(m^{2}K)]; T_∞ is the temperature of the cooling fluid; and T_s is the temperature of the surface of the body.

2.1.1.3 Radiation

In general, radiation is energy in the form of waves or moving subatomic particles. Among the radiation types, we are specifically interested in the Thermal radiation. Thermal radiation is heat transfer by the emission of electromagnetic waves from the surface of an object due to temperature differences which carry energy away from the emitting object.

The basic relationship governing radiation from hot objects is called the Stefan-Boltzmann law:

$$q = \varepsilon\sigma A\left(T_1^{\,4} - T_2^{\,4}\right) \tag{2.3}$$

Where ε is the coefficient of emissivity (=1 for ideal radiator); σ is the Stefan-Boltzmann constant of proportionality (5.669E-8 [W/(m2K4)]); A is the radiating surface area; T_1 is the temperature of the radiator; and T_2 is the temperature of the surroundings.

The three of the previously mentioned heat transport mechanisms can be expressed by the model law that state that a flux is proportional to a difference in driving potential divided by a resistance, in our case:

$$q = -\frac{\Delta T}{R_{th}} \tag{2.4}$$

Being the Thermal Resistance (R_{th}) for each one of them as follow:

$$R_{th}^{cond} = \frac{\Delta x}{kA} \tag{2.5}$$

$$R_{th}^{conv} = \frac{1}{h_{conv} A} \tag{2.6}$$

$$R_{th}^{rad} = \frac{1}{h_{rad} A} \tag{2.7}$$

Where h_{rad} is:

$$h_{rad} = \varepsilon\sigma\left(T_1^3 + T_1^2 T_2 + T_1 T_2^2 + T_2^3\right) \tag{2.8}$$

2.1.2 Material properties

2.1.2.1 Thermal conductivity (k)

Is the ability of a material to conduct heat. It is defined as the quantity of heat, ΔQ, transmitted during a period of time Δt through a thickness L, in a direction normal to a surface of area A, due to a temperature difference ΔT, under steady state conditions and when the heat transfer is dependent only on the temperature gradient.

$$k = \frac{\Delta Q}{\Delta t} \times \frac{L}{A \times \Delta T} \qquad \text{[W/mK]} \tag{2.9}$$

2.1.2.2 Density (ρ)

Indicate the mass per unit volume of a material.

$$\rho = \frac{m}{V} \qquad \text{[Kg/m}^3\text{]} \tag{2.10}$$

Density is a temperature and pressure dependent material property. In solids and liquids is just slightly affected by these factors but in gases is strongly dependent in both of them.

2.1.2.3 Specific Heat (c_v and c_p)

In general, is the measure of the heat energy required to increase the temperature of a unit quantity of a substance by a defined temperature step. For example, how much heat must be added to increase the temperature of one gram of water by one Celsius degree.

The specific heat is defined at constant pressure (C_p [J/Kg°K]) and a constant volume (C_v [J/m^3°K]). In gases, C_p and C_v have important differences but, since for most solids and liquids C_p and C_v are equal, in casting processes for simplicity, we call specific heat C_p.

$$C_p \equiv \frac{\partial H}{\partial T} \qquad \text{[J/Kg°K]} \tag{2.11}$$

Where H is the enthalpy per unit mass.

2.1.2.4 Latent Heat (L)

Is the amount of energy in the form of heat that is released or absorbed by a substance during a change of phase (solid, liquid, or gas)

$$L = \frac{Q}{m} \qquad [\text{J/Kg}] \tag{2.12}$$

Where Q is the amount of energy needed to change the phase of the substance; m is the mass of the substance and L correspond to the specific latent heat of the particular substance.

2.1.2.5 Thermal diffusivity (α)

Is the ratio of the thermal conductivity to the volumetric heat capacity of the material.

$$\alpha = \frac{k}{\rho c_p} \tag{2.13}$$

Where ρc_p represents the volumetric heat capacity [J/m^{3}K].

Mediums with high thermal diffusivity reach thermal equilibrium rapidly with their surroundings due to their capacity of fast heat transfer compared with their mass.

2.1.3 Initial and Boundary conditions

Initial conditions and boundary conditions are needed, together with the heat conduction equation, to fully define a transient thermal problem. If the given problem is in steady-state, there is no necessity to define initial conditions

The initial conditions represent the initial temperature distribution throughout the body. In casting processes, the initial condition is assumed to be constant throughout the mould, is also assumed to be constant for the melt in the mould filling simulation, where the temperature will be a superheating temperature. For the solidification simulation, the initial condition is given by the temperature field immediately after filling.

For simplicity, when there is no interest in the mould filling simulation, a constant temperature throughout the melt after filling can be assumed and the superheating temperature of the melt can be used as initial condition for the solidification simulation.

Next, five types of boundary conditions relevant for the modeling of casting processes are introduced together with their mathematical representation:

2.1.3.1 Prescribed boundary temperature

$$T(P,t) = \overline{T}(P,t) \tag{2.14}$$

Where P is a position on the surface (in 1-D described just by the x value), t is time and "¯" denotes prescribed.

2.1.3.2 Perfectly insulated (adiabatic) boundary

An adiabatic boundary has no heat flux across.

$$\frac{\partial T}{\partial n}(P,t) = 0 \tag{2.15}$$

Where n is the outward pointing normal to the surface at point P.

Another way to define this boundary condition is setting the heat transfer coefficient h to zero in Newton's law.

2.1.3.3 Convection boundary condition

The heat flux across the bounding surface is proportional to the difference between the temperatures of the surface $T(P,t)$ and the surrounding $T_\infty(t)$ cooling medium. It is defined by the Newton's convective law of cooling:

$$k\frac{\partial T}{\partial n}(P,t)=h\left(T_\infty(t)-T(P,t)\right) \tag{2.16}$$

As mentioned for equation (2.2), h is the convective heat transfer coefficient.

2.1.3.4 Radiation boundary condition

When a boundary surface receives heat by radiation, the following expression applies:

$$k\frac{\partial T}{\partial n}(P,t)=h_{rad}\left(T_\infty(t)-T(P,t)\right) \tag{2.17}$$

Where

$$h_{rad}=\varepsilon\sigma\left(T_1^3+T_1^2T_2+T_1T_2^2+T_2^3\right) \tag{2.18}$$

Which, for simplicity, assumes h_{rad} = constant. This is used when the time step of the analysis is so small that the temperatures may be assumed constant during the time step.

2.1.3.5 Internal boundary (two solids bodies in contact) condition

If we assume perfect thermal contact, the heat leaving one body must be equal to that entering the other. In which case, for a point P in the contact surface:

$$T_1(P,t)=T_2(P,t) \tag{2.19}$$

$$k_1\frac{\partial T_1}{\partial n}(P,t)=k_2\frac{\partial T_2}{\partial n}(P,t) \tag{2.20}$$

The subscripts 1 and 2 refer to the two bodies.

Bestpfe.com

2.1.4 The Heat Conduction Equation

2.1.4.1 1-D transient (time dependent) heat conduction equation

$$\rho C_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x} \right) + \dot{Q}'''_{gen} \qquad (2.21)$$

Where $\dot{Q}'''_{gen}$ is the internal generation of heat per unit time per unit volume present within the body.

2.1.4.2 1-D steady-state heat conduction equation

Since the steady-state is independent of time, is defined as:

$$\frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x} \right) + \dot{Q}'''_{gen} = 0 \qquad (2.22)$$

2.1.4.3 The 3-D transient Heat Conduction Equation

For casting processes, it represents the basis of all heat conduction calculations. Is the general form of the heat conduction equation and is as follows:

$$\rho C_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y}\left(k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z}\left(k \frac{\partial T}{\partial z} \right) + \dot{Q}'''_{gen} \qquad (2.23)$$

If we consider no $\dot{Q}'''_{gen}$ and replace C_p by its equivalent value $\frac{\partial H}{\partial T}$ we get:

$$\rho \frac{\partial H}{\partial T} \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y}\left(k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z}\left(k \frac{\partial T}{\partial z} \right) \qquad (2.24)$$

Which is the same equation presented in section 1.1 as the classical heat equation:

$$\rho \frac{\partial H}{\partial T} \frac{\partial T}{\partial t} = div[k \nabla T] \qquad (2.25)$$

2.1.5 Numerical solutions

The purpose of the numerical solution of partial differential equations is to determine the value of the dependent variable at various predefined points (nodal points). The values of the dependent variable will always be the primary unknowns. The resulting equation systems in the primary unknowns are written so that for each nodal point in the calculation domain there is an equation for every dependent variable. These equations are referred to as discretization equations.

The calculation domain is divided into sub-domains called cells, elements or control volumes with the intention of identify the dependent variable in a smaller area as a function of the values in the nodal points. In this way, different profiles can be applied to each sub-domain allowing more suitable sub-domains for the actual problem.

The most commonly used numerical methods in casting simulations are the Finite Differential Method (FDM), the Finite Element Method (FEM) and the Finite Volume Method (FVM). The differences between them are mainly in the profile assumptions for the cells, elements or control volumes and in the methods of deriving the discretization equations. Nevertheless, they have also much in common, for instance the all need a geometry definition, which describe the calculation domain, appropriate material data, definition of initial and boundary conditions, they all use solvers of linear algebraic equations to perform the calculations and they all use a postprocessor to present the results.

According to Abaqus (2007),
Analysis user's manual version 6.7-1
ABAQUS, Inc, Providence, RI, USA

Becker, A. A. (2004), some of the main features of the FE and the FD methods are:

2.1.5.1 Finite element method

1-The solution domain is divided into a grid of finite segments or elements.

2-The governing partial differential equations are solved for each element in mesh.

3-The elements are assembled together and the continuity requirements and equilibrium conditions are satisfied with adjacent elements.

4- A unique solution can be obtained to the whole system of linear algebraic equations once the boundary conditions are satisfied.

5-The solution matrix is populated with relatively few non-zero coefficients.

6-The FE method is suitable for the analysis of complex geometries and is not difficult to modify the element size in particular regions.

BESTPFE.COM
List of research project topics and materials

2.1.5.2 Finite difference method

1-The solution domain is divided into a grid of cells or elements

2-The derivatives in the governing partial differential equations are converted into finite difference equations.

3-These finite difference approximations are applied to each interior point so that the displacement of each node is a function of the displacements at the other nodes connected to it.

4- A unique solution can be obtained to the whole system of linear algebraic equations once the boundary conditions are satisfied.

5-The solution matrix is banded

6- The FD method is not suitable for very complex geometries and is difficult to change the element size in particular regions

6- The FD method is not as popular for stress analysis problems as for heat transfer and fluid flow problems.

The approximation quality of the FE method is better, but it comes with a greatest computer calculation time price.

2.2 The Stress Analysis

2.2.1 Residual Stresses

Residual stresses are tensions or compressions that exist in the bulk of a material without applying an external load.

In a casting process, while cooling, residual stresses are induced due to temperature gradients across the whole casting, mechanical constraints given by the mold during the shrinkage of the metal and volumetric change and transformation plasticity related to the solid-state phase transformation. Hence, residual stresses are a function of the shape of the casting and the cooling rate of the casting process.

Compressive residual stresses are desirable in a component as they improve the fatigue life and reduce the stress corrosion cracking tendency since they also offer resistance to crack propagation.

2.2.2 Elasticity

Within a certain limit, when a load is applied, a component undergoes a deformation that is recovered when the load is released. This behavior of a material is known as elasticity and its limit is known as the elastic limit.

The measure of the elastic behavior of a material is known as Young's modulus. This modulus is experimentally determined as the slope of the stress-strain curve obtained during tensile tests carried out on samples of a given material.

2.2.2.1 Elastic strain

According to the Hook's law, within the elastic limit of a material the stress is proportional to the strain. This strain is known as elastic strain (ε^{el}) and is expressed as:

$$\varepsilon^{el} = \sigma / E \qquad (2.26)$$

2.2.2.2 Thermal strain

When a metal body is heated or cooled it expand or contract if it's free to deform. The amount of deformation that the body undergoes is then proportional to the rise or fall down of the temperature.

This give place to the mathematical expression for the thermal strain:

$$\varepsilon^{\mathrm{th}} = \int_{T1}^{T2} \alpha(T)dT \tag{2.27}$$

This thermal deformation can be a contraction or an expansion and can result in deformation only or stresses only, if the body is free to contract or is totally constrained respectively or it can result in a combination of deformation and stresses which is the most common situation in reality for castings.

2.2.3 Plasticity

When the load applied to a material produce a deformation over the elastic limit, is said that the material is in the plastic region, where any experimented deformation is permanent. The transition from elastic to plastic behavior is called yield and the stress that correspond to this transition is called yield stress.

At elevated temperatures, the metals undergo such irreversible deformations and in casting processes, which occurs over a large temperature range, this inelastic or plastic behavior becomes important.

In plasticity, is not possible to define the stresses as functions of the strains on total form (so Hooke's law, which is a total constitutive law, does not apply). Instead, it is possible to express the *changes* in stresses as *changes* in strains (which is known as an incremental constitutive law).

2.2.3.1 Ideal Plasticity

Ideal Plasticity is the simplest approximation to the inelastic behavior of a material. It assumes that the yield stress σ_y is constant independently of the mechanical strain (see *Figure 2.1*).

Notice that we said mechanical strain and not total strain. The mechanical strain is equal to the elastic strain plus the plastic strain:

$$\varepsilon^{mech} = \varepsilon^{el} + \varepsilon^{pl} \tag{2.28}$$

And the total strain (the one see from outside the component) is the sum of the mechanical and the thermal strains and therefore the strain for the thermo-elasto-plastic case:

$$\varepsilon^{total} = \varepsilon^{mech} + \varepsilon^{th} = \varepsilon^{el} + \varepsilon^{pl} + \varepsilon^{th} \tag{2.29}$$

The importance of this differentiation and the fact that ideal plasticity is defined with respect to the mechanical strain is because even if the total strain of a fully constrained component subjected to a thermal gradient is zero, the mechanical strain is not and is this strain the one to be used in the stress-strain plot.

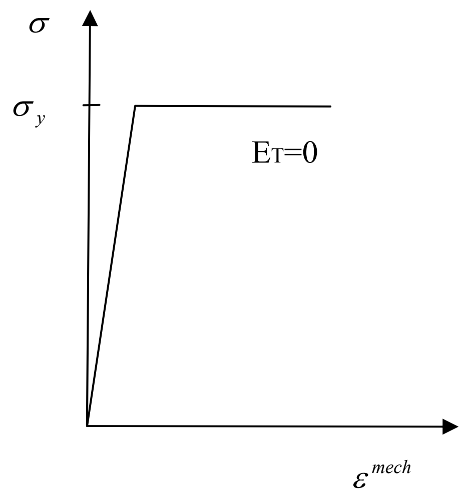


Figure 2.1. Ideal Plasticity

If the event happens at constant temperature, the mechanical strain is equal to the total strain.

2.2.3.2 Hardening

A better approximation to the real behavior of metallic materials is this concept of hardening because most of them instead of behave ideal plastically exhibit increasing stress with increasing plastic deformation, which the hardening approach assumes to have a linear relation. Hence is known as strain hardening or linear hardening.

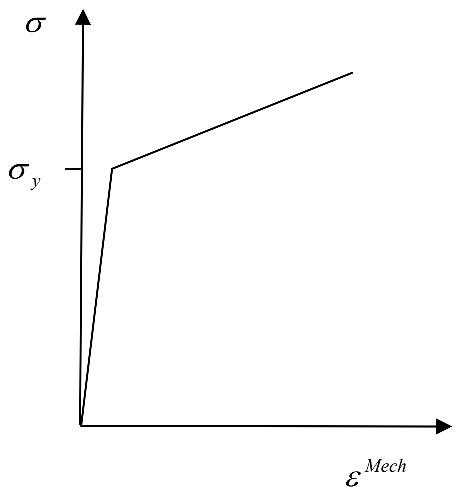


Figure 2.2. Linear hardening

2.2.3.3 Temperature Dependent Yield Stress

The yield stress is also a function of the temperature and this dependence is very important in casting processes for the presence of plastic deformations:

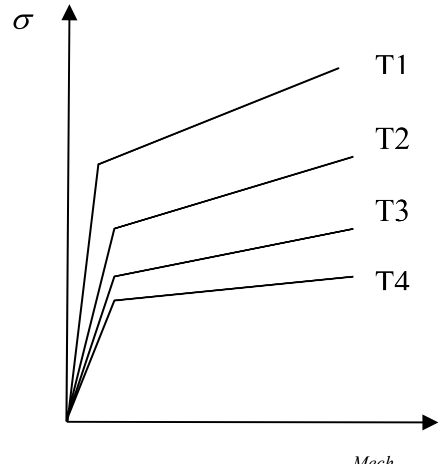


Figure 2.3. Stress-train curves at different temperatures, linear hardening approach.

Taking the temperature dependence into consideration, the two main dependences of yield stress for the plasticity in a casting process are:

1. Temperature
2. Plastic strain

This can be expressed mathematically as:

$$\sigma_y = \sigma_y(T, \varepsilon^{pl}) \tag{2.30}$$

2.2.3.4 Plasticity Material models

The residual stresses are calculated by a quasi-static rate independent elasto-plastic analysis and the majority of the plasticity models use "incremental" theories where the mechanical strain rate is divided into an elastic part and a plastic part.

Those incremental plasticity models are expressed in terms of

1. Yield function
2. Flow rule
3. Hardening law

2.2.3.4.1 The Yield Surface

The definition of a yield surface is very useful in the multi-dimensional formulation of the plasticity theory. Is a general way to define the yield criterion by means of a yield function, *f*, for the material.

The yield surface encompasses the elastic region of the material behavior which means that the state of the stresses while inside the surface is elastic. Since the yield function is defined to be zero in the plastic state, the yield point is reached when the stresses reach the surface, and outside the surface the material behavior becomes plastic.

2.2.3.4.2 Illustration for a Simple Mathematical Model

The classical rate independent (ideal) plasticity model can be illustrated with a simple model of a one dimensional mechanical device. This device exhibit the notion of irreversible response and consists of a spring with a spring constant E and coulomb friction element with constant $\sigma_y > 0$. This device is assumed to have unit length and unit area initially:

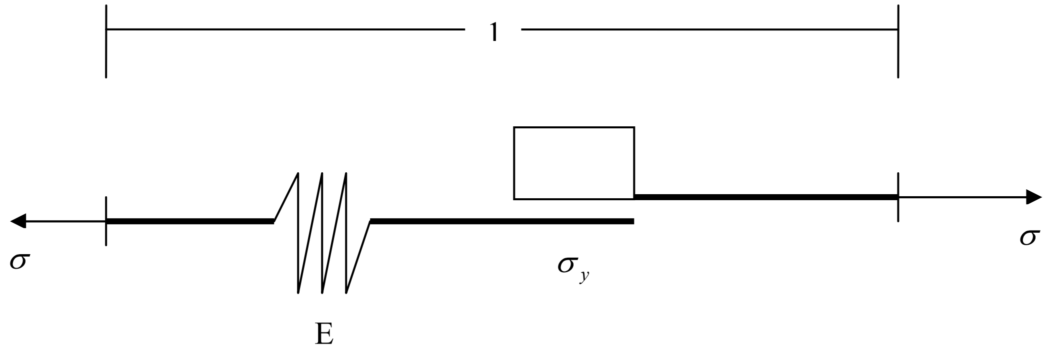


Figure 2.4. One dimensional frictional device representing ideal plasticity

σ represents the stress or force applied to the model, σ_y is the flow stress of the friction device and ε represents the total strain or change in length.

The total strain is decomposed into elastic and plastic strain.

$$\varepsilon = \varepsilon^{el} + \varepsilon^{pl} \tag{2.31}$$

By equilibrium conditions and using eq. (2.31) the elastic stress-strain relationship is given by:

$$\sigma = E\varepsilon^{el} = E\left(\varepsilon - \varepsilon^{p}\right) \tag{2.32}$$

The yield condition is defined from the assumption that the absolute value of the stress in the frictional device cannot be greater than $\sigma_y>0$:

$$f(\sigma) = |\sigma| - \sigma_y \leq 0 \tag{2.33}$$

If $f(\sigma)<0$, the $\dot{\varepsilon}^{p}$ is zero and the instantaneous response of the device is elastic.

If $f(\sigma)=0$, the frictional device slip with constant slip rate in the direction of the applied stress. The following expression describes the flow rule:

$$\dot{\varepsilon}^{p} = \gamma \frac{\partial f}{\partial \sigma} \tag{2.34}$$

Where γ represents the slip rate and is ≥ 0.

The conditions that the stresses must be admissible and the plastic flow can take place just on the yield surface are known as Kuhn-Tucker complimentary conditions and mathematically look like:

$$\gamma \geq 0\,, \qquad f(\sigma) \leq 0\,, \qquad \gamma f(\sigma) = 0 \tag{2.35}$$

A final condition known as consistency condition must be stated, that is:

$$\gamma \dot{f}(\sigma) = 0 \quad (\text{if } f(\sigma) = 0) \tag{2.36}$$

As mentioned before, this mathematical model corresponds to *ideal plasticity*. The constitutive model to account for isotropic linear *hardening* effects has the following differences:

A hardening law:

$$\dot{\alpha} = \gamma \tag{2.37}$$

The Yield condition changes to:

$$f(\sigma,\alpha) = |\sigma| - (\sigma_y + K\alpha) \leq 0\,, \qquad \text{where} \quad \alpha \geq 0\,; \quad \sigma_y > 0\,; \quad K \geq 0 \tag{2.38}$$

Here K is the plastic modulus and α is a function of the amount of plastic flow (slip) known as an internal hardening variable.

The Kuhn-Tucker complementary conditions are now:

$$\gamma \geq 0\,, \qquad f(\sigma,\alpha) \leq 0\,, \qquad \gamma f(\sigma,\alpha) = 0 \tag{2.39}$$

The flow rule is;

$$\dot{\varepsilon}^{p} = \gamma \frac{\partial f}{\partial \sigma} \tag{2.40}$$

And $\gamma \geq 0$ is determined by the following consistency condition:

$$\gamma \dot{f}(\sigma,\alpha) = 0 \quad \text{if} \quad f(\sigma,\alpha) = 0 \tag{2.41}$$

Bestpfe.com

2.2.3.5 J2-Plasticty model

The yield function, which defines the elastic range of the material behavior and when plasticity begins, is governed by the second invariant of the deviatoric stress tensor. This is known as J_2 flow theory and is given as:

$$J_2 = \frac{1}{2} s_{ij} s_{ji} \tag{2.42}$$

The J_2 flow theory makes the yield function independent of hydrostatic pressure.

2.2.3.5.1 J2-Plasticity model - Constitutive Laws

The equilibrium equation of a quasi-static mechanical problem is:

$$div\sigma = 0 \tag{2.43}$$

Where σ is the stress tensor.

The total strain is governed by

$$\varepsilon = \frac{1}{2}\left[\nabla u + (\nabla u)^T\right] \tag{2.44}$$

Where u represents the displacement.

The constitutive law is

$$\sigma = D\left(\varepsilon - \varepsilon^p - \varepsilon^{th}\right) \tag{2.45}$$

Where *D=D(T)* is the temperature dependent elastic tensor.

The thermal strain is of the form

$$\varepsilon^{th} = \alpha I (T - T_{ref}) \tag{2.46}$$

Where $\alpha = \alpha(T)$ is the thermal expansion parameter.

The yield function is defined as

$$f(\sigma,T) = \sqrt{3J_2} - H\overline{\varepsilon^p} + Y_0 \le 0 \tag{2.47}$$

Where H=H (T) is the linear hardening parameter,

The flow rule is given by

$$\dot{\varepsilon}^p = \gamma \frac{\partial f}{\partial \sigma} \tag{2.48}$$

Where γ is the plastic multiplier and which value is govern by the Karush-Kuhn-Tucker conditions $\gamma \ge 0$, $f(\sigma,T) \le 0$ and $\gamma f(\sigma,T) = 0$

$$\bar{\varepsilon}^p = \int_0^t \sqrt{\frac{2}{3}} \left\| \dot{\varepsilon}^p \right\| dt \tag{2.49}$$

BESTPFE.COM
List of research project topics and materials

3 Implementation

3.1 Process Summary

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

1. Mesh the part
2. Define the material properties
    a. CASTING
        i. Density
        ii. Conductivity
        iii. Specific Heat
        iv. Latent Heat
        v. Liquidus Temperature
        vi. Solidus Temperature
    b. SAND MOLD
        i. Density
        ii. Conductivity
        iii. Specific Heat
3. Define the initial boundary conditions
    a. Initial temperature of the casting
    b. Initial temperature of the mold
4. Define the shake-out event
    a. By time or temperature

5. Define the interactive boundary conditions before shake-out (Only in Abaqus)

    a. Conduction. Between the external surface of the casting and the surface of the mold cavity

    b. Convection. Between the external surface of the mold and the ambient

    c. Radiation. Between the external surface of the mold and the ambient

6. Define the interactive boundary conditions after shake-out (Only in Abaqus)

    a. Convection. Between the external surface of the casting and the ambient

    b. Radiation. Between the external surface of the casting and the ambient

The Stress simulation

7. Use the same mesh used in the thermal simulation for the casting (the mold is not present in our stress analysis)

8. Define the material properties

    a. Expansion Coefficient

    b. Young's Modulus

    c. Poisson's Ratio

    d. Hardening Coefficient (Only in Magmasoft)

    e. Plasticity (Only in Abaqus)

        i. Yield Stress

        ii. Plastic Strain

9. Define the initial boundary condition

    a. Initial temperature of the casting (as in the thermal analysis)

10. Load the nodal thermal history generated in the thermal simulation as a predefined temperature field (Only in Abaqus)

11. Define the mechanical boundary conditions (Only in Abaqus)

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

3.2 Procedure

The simulation procedure steps are explained in details in the Appendix sections 10.1 and 10.2 according to the following diagram:

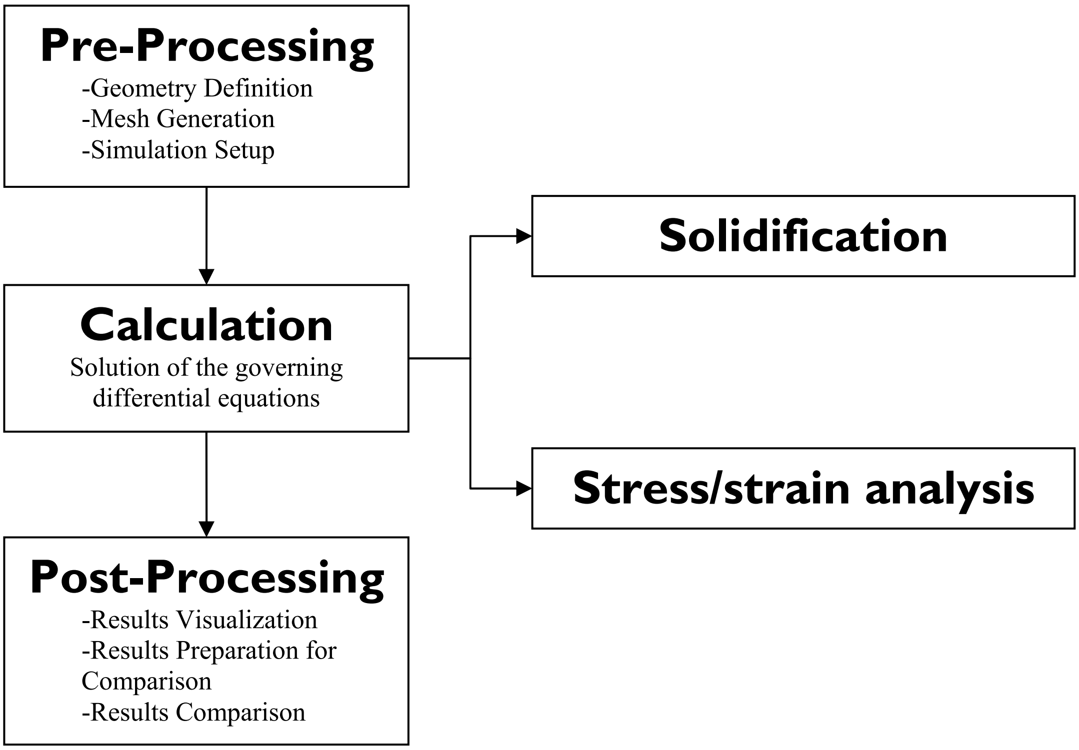


Figure 3.1. Steps sequence for residual stress analysis

In sections 10.1 and 10.2, to focus on the methods and to minimize geometry related problems, we go through the whole process using a simple geometry, specifically a cylinder. The Results Comparison method is treated apart in the Appendix section 10.3

All the needed steps to perform our simulations in Abaqus and Magmasoft are presented in a sequence format so the reader can use them as a **step by step guide of a residual stress simulation**.

The casting material for all the models is grey iron, and our mold material is based in the "Coldbox" sand material defined in the Magmasoft database (for details on the material data refer to the Appendix section 10.4).

4 Cylinder Results

4.1 Geometry

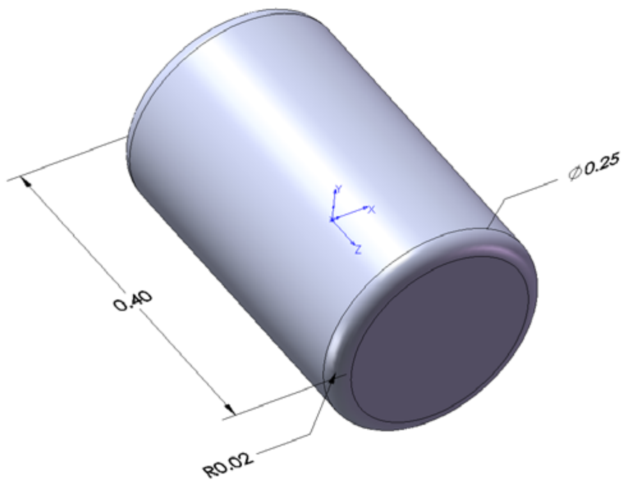


Figure 4.1. The Cylinder part. Units: Meters

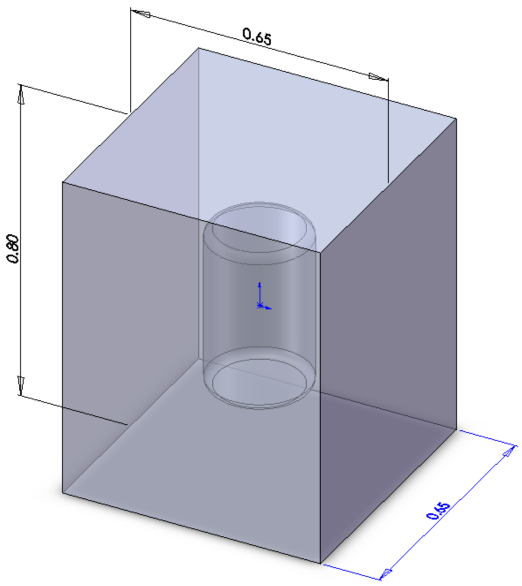


Figure 4.2. The Mold part. Units: Meters

4.2 Mesh

	Cylinder		Mold	
	Elements	Nodes	Elements	Nodes
Abaqus	42461	9231	107464	21829
Magmasoft	10768	10768	46352	46352

Notice that Magmasoft uses the Control Volume Finite Difference Method so for each element there is only one node, which is positioned in the center of the element. This justifies the fact of having the same number of nodes as elements in Magmasoft. We tried to match the number of nodes for the casting in both softwares. Still, it is difficult to control the number of elements assigned to the casting and the mold in Magmasoft therefore the difference. For details about how to mesh the parts in Abaqus and Magmasoft, refer to sections 10.1.1.2 and 10.2.1.2 respectively.

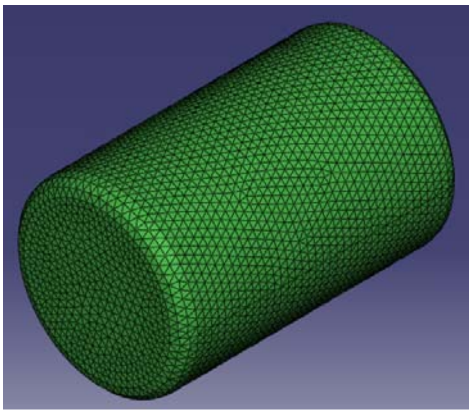


Figure 4.3. Hypermesh mesh used in Abaqus

Figure 4.4. Magmasoft mesh

4.3 Boundary Conditions

4.3.1 Thermal boundary conditions

4.3.1.1 Before Shake-out

Conduction. Between the external surface of the casting and the surface of the mold cavity. For details on how to set this type of boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under "9-Interaction Properties Definition" and "10-Interactions Definitions"). For details on how to set this boundary condition in Magmasoft, refer to section 10.2.1.3 (instruction for *Figure 10.63*).

Convection. Between the external surface of the mold and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

Radiation. Between the external surface of the mold and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

4.3.1.2 After shake out

Convection. Between the external surface of the casting and the ambient. A temperature dependent convective heat transfer coefficient property was defined in Abaqus. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.2 (under "9-Interaction Properties Definition" and "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

Radiation. Between the external surface of the casting and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.2 (under "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

4.3.2 Mechanical boundary conditions

4.3.2.1 Stress analysis step

The user does not participate in the definition of boundary conditions for the stress analysis in Magmasoft. It is an automatic procedure. Therefore we only present our Abaqus approach.

The task is to restrain the rigid body translations and rotations in X, Y and Z, but allow the body to deform naturally (to shrink, basically). In the cylinder model the 6 degrees of freedom has been constrained as follow:

Translations in X, Y and Z

In the flat face of the Cylinder lying in Z=0, a node in Y=0 is constrained in X, Y and Z. See *Figure 4.5.* Notice that in the picture, X is the horizontal axis, Y the vertical axis, and the Z axis is perpendicular to the paper.

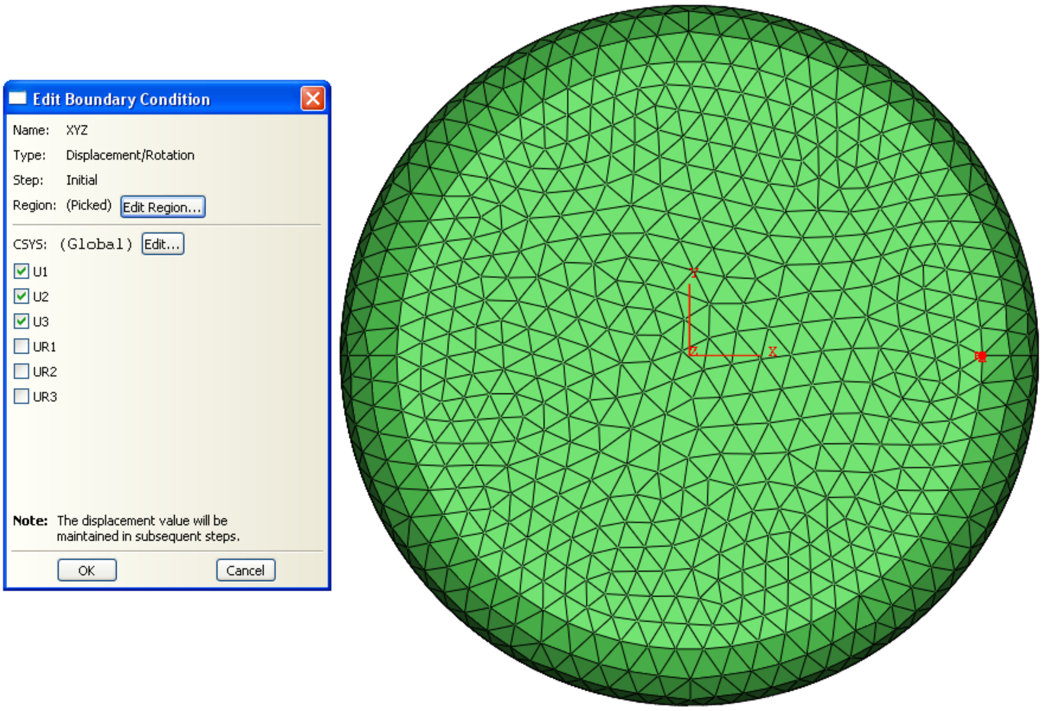


Figure 4.5. Constraining the rigid body translations in X, Y and Z in the Cylinder

Three rigid body translations have been constrained by fixing the point in the space. As a result the part would shrink toward the point. However, the body could still pivot in our fixed node, so now the rotations have to be constrained.

Rotations in the Y and Z axis

A node also lying in Z=0 and Y=0 but in the opposite side of the part with respect to the fixed node is constrained in Y and Z. See *Figure 4.6.*

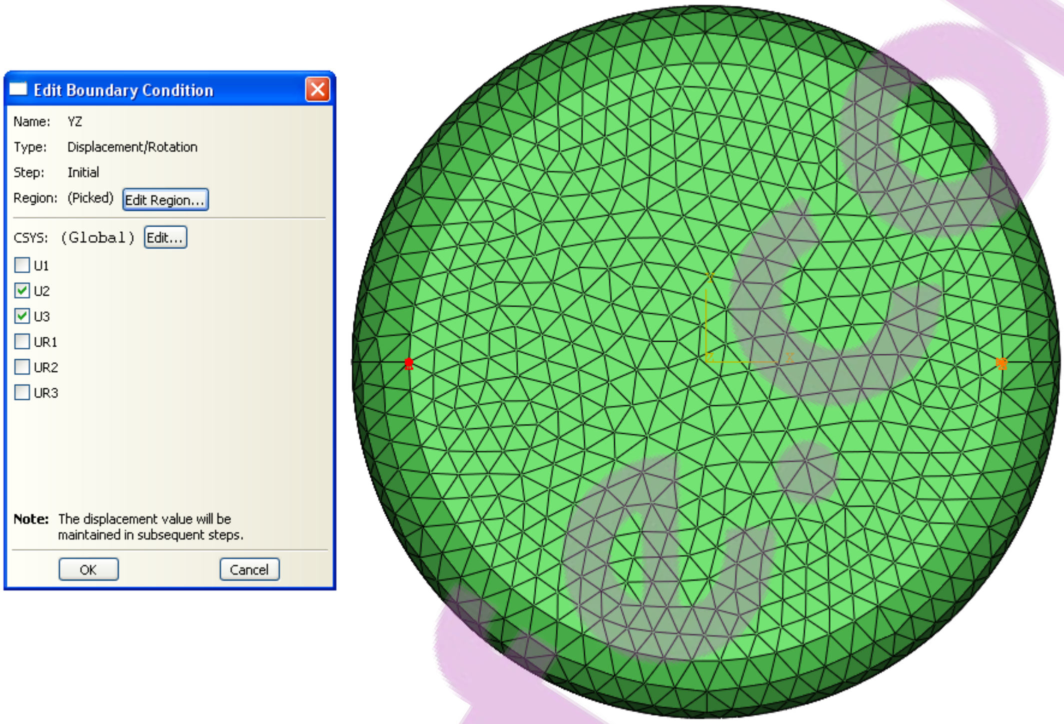


Figure 4.6. Constraining the rotations in Y and Z in a single node selection (in red) for the Cylinder model

The constraint in Y avoids the rotation in Z and the constraint in Z avoids the rotation in Y. The reason why the node is left free to move in X is because that is the correct contraction direction toward the totally fixed node.

Bestofee.com

Rotation in the X axis

A node aligned in the Z axis with the totally fixed one (that is with the same X coordinate and Y=0) lying in the opposite flat face of the Cylinder is constrained in X and Y. See *Figure 4.7*. In this way the displacement of that node can just happen in Z, ensuring that the length axes of the body will remain parallel to the Z axis.

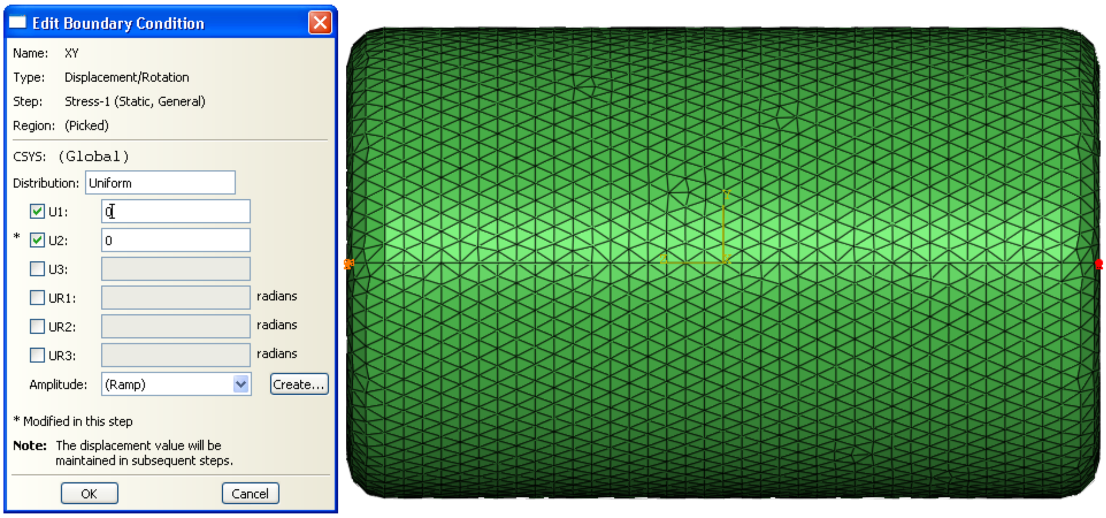


Figure 4.7. Constraining the rotations in X in a single node selection (in red) for the Cylinder model

4.4 Cooling curves

The following results presented in *Figure 4.8* were obtained from the central point of the geometry of the whole Cylinder.

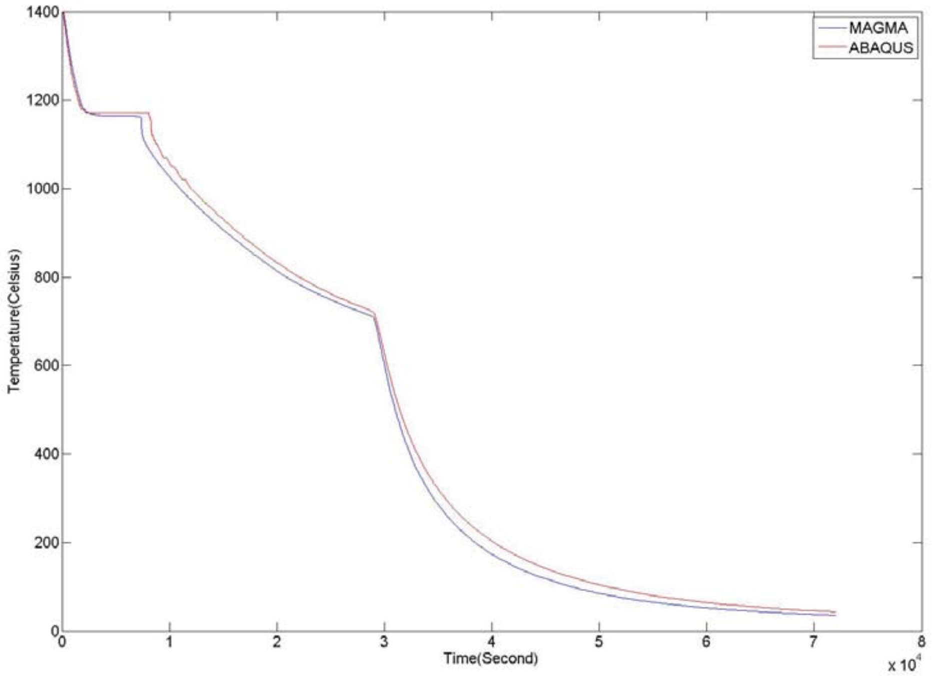
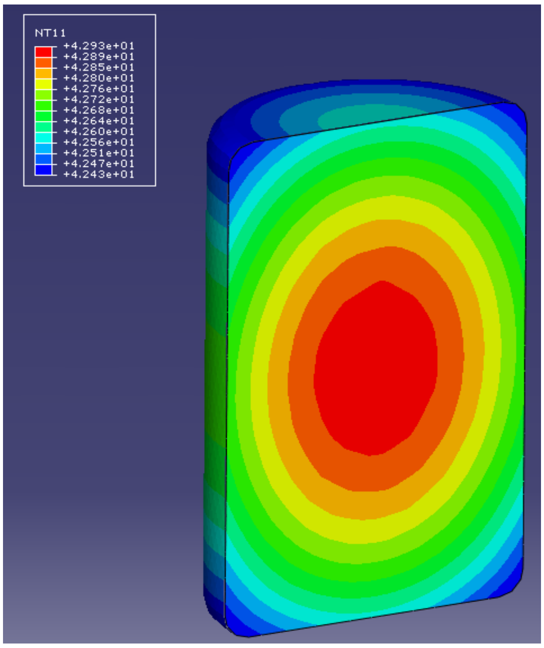


Figure 4.8. Abaqus vs. Magmasoft cooling curves for the Cylinder model

BESTPFE.COM
List of research project topics and materials

4.5 Thermal color spectrums



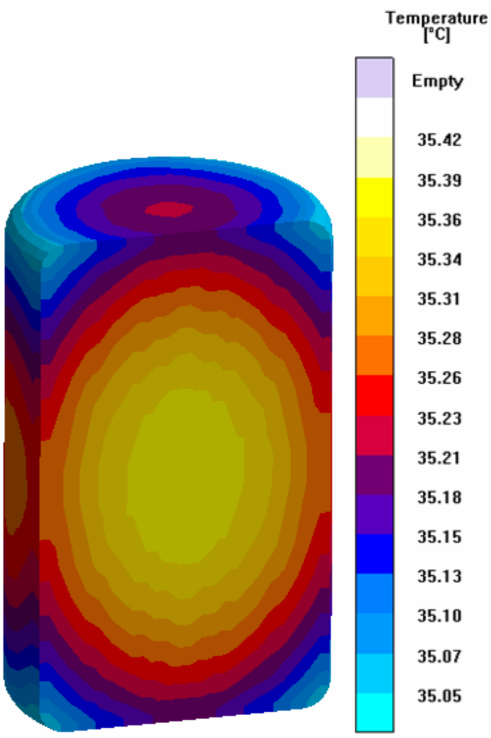


Figure 4.9. Abaqus (top) and Magmasoft (bottom) thermal color spectrums of the last step after shake out of the Cylinder model.

4.6 Stress curves

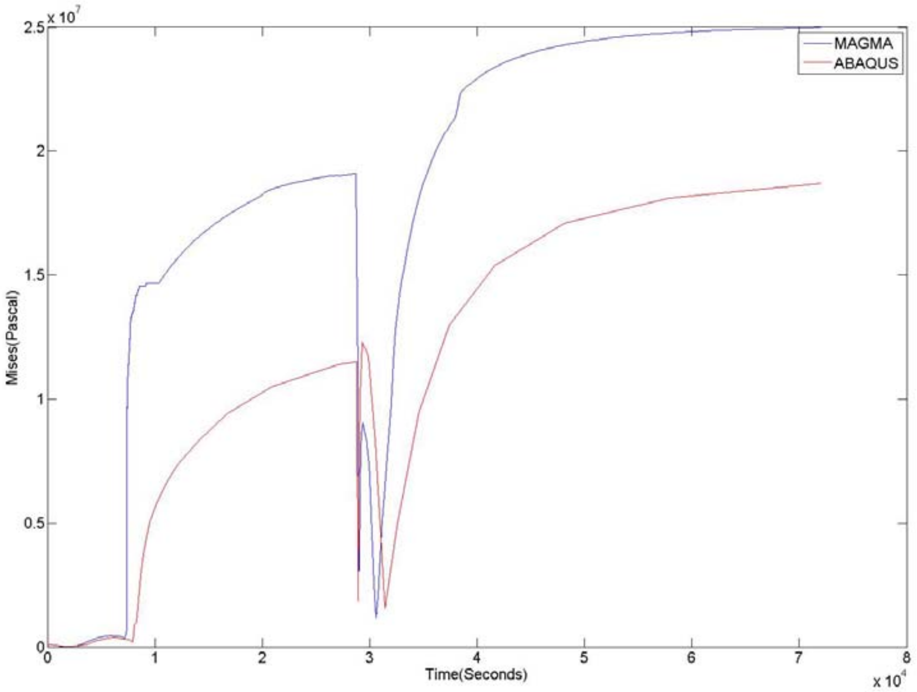


Figure 4.10. Abaqus vs. Magma Von Mises curves for the Cylinder model

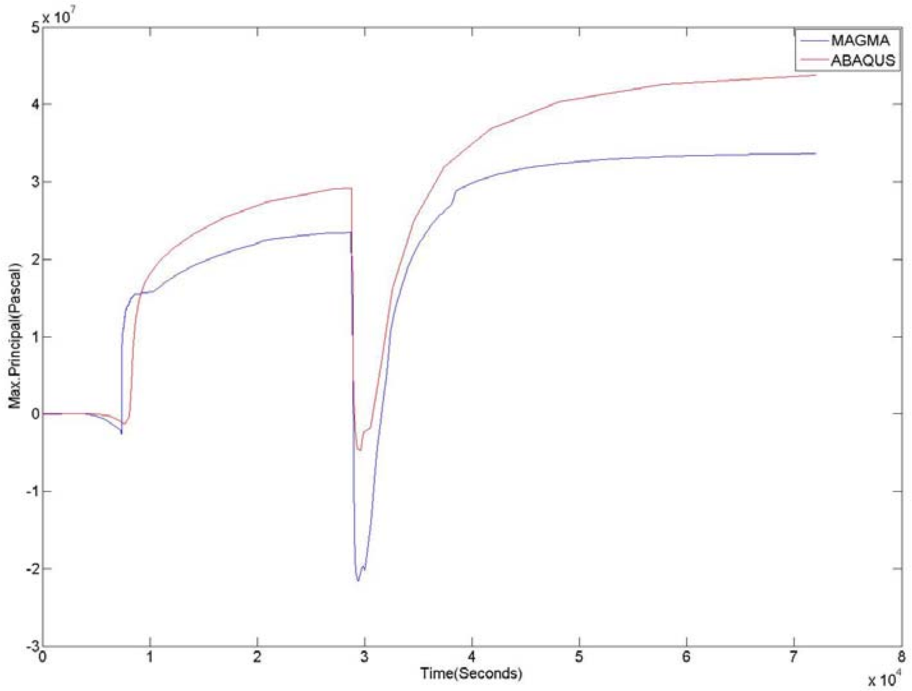


Figure 4.11. Abaqus vs. Magma Maximum Principal stresses for the Cylinder model

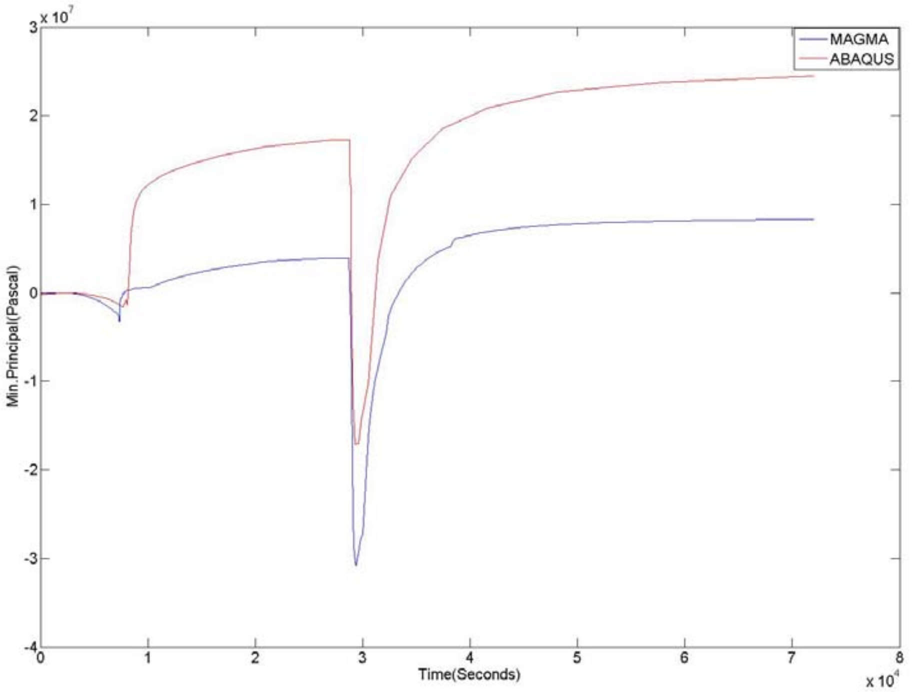


Figure 4.12. Abaqus vs. Magma Minimum Principal stresses for the Cylinder model

The stress results presented in *Figure 4.10*, *Figure 4.11* and *Figure 4.12* were obtained from the central point of the geometry of the whole Cylinder.

Comments

These results show that the Cylinder model develop more stresses in tension than in compression.

4.7 Stress color spectrums

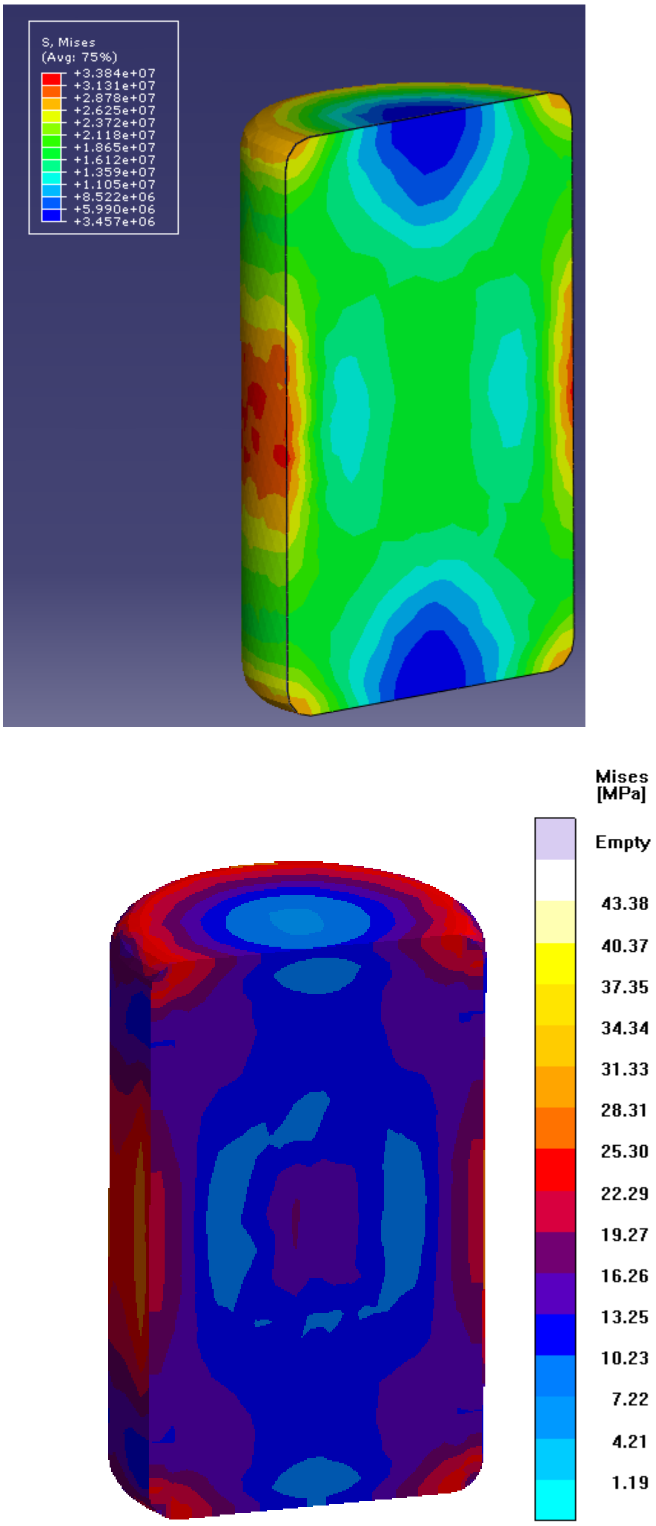
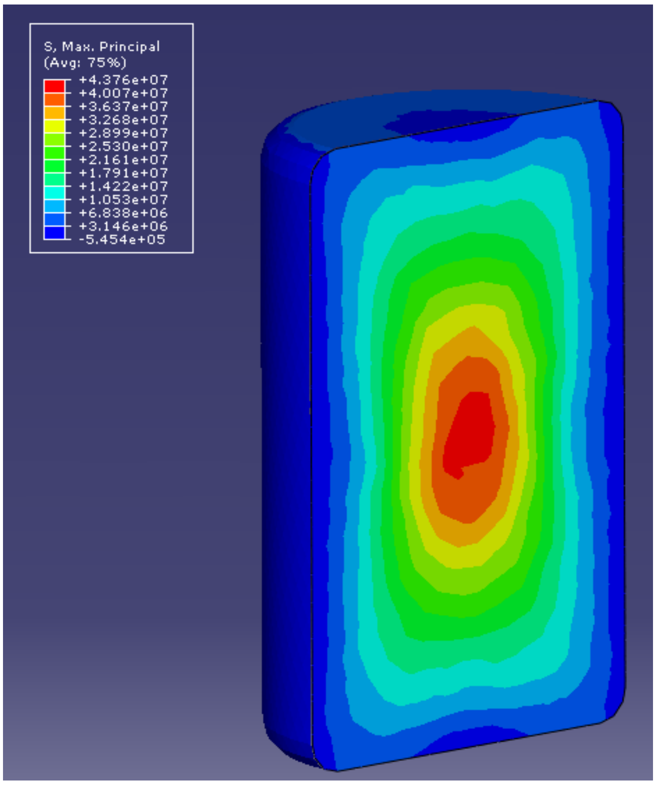


Figure 4.13. Abaqus (top) and Magmasoft (bottom) color spectrums for the ***Mises*** *results of the Cylinder model.*



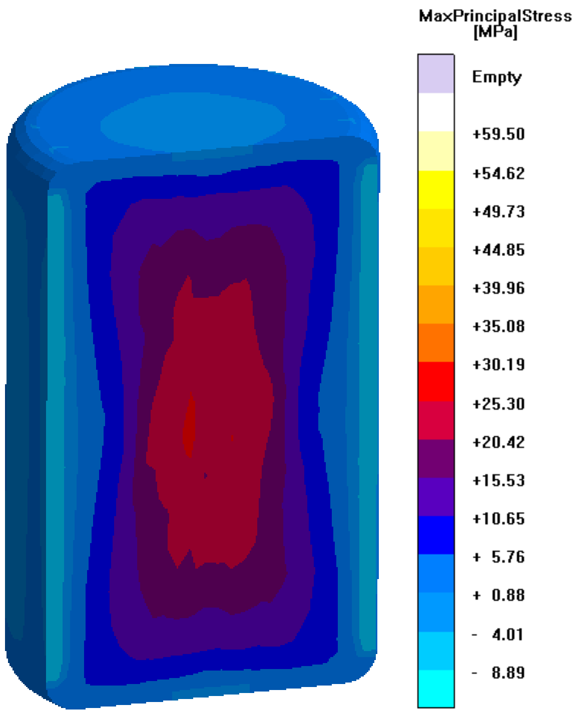
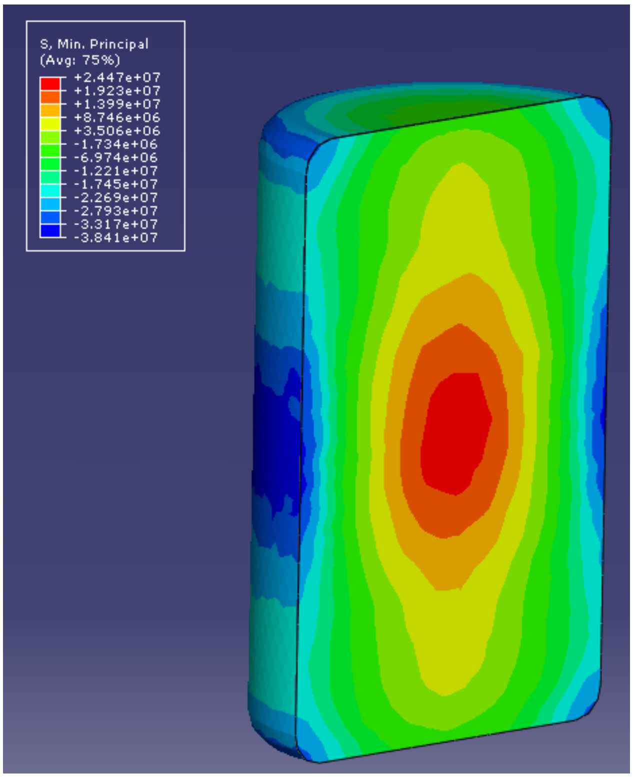


Figure 4.14. Abaqus (top) and Magmasoft (bottom) color spectrums for the residual ***Maximum Principal*** *stresses of the Cylinder model.*



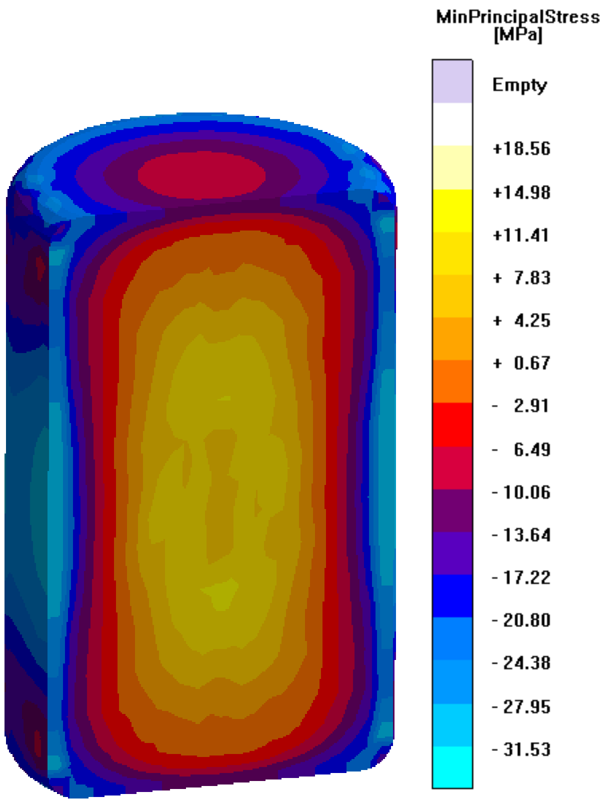


Figure 4.15. Abaqus (top) and Magmasoft (bottom) color spectrums for the residual ***Min. Principal stresses*** *of the Cylinder model.*

4.8 Simulation time for the Cylinder

	Thermal	Stress	Total
Abaqus	7hrs. 12min.	24min.	7hrs. 36 min.
Magmasoft			03hrs. 16min.

5 Original Hub Results

5.1 Geometry

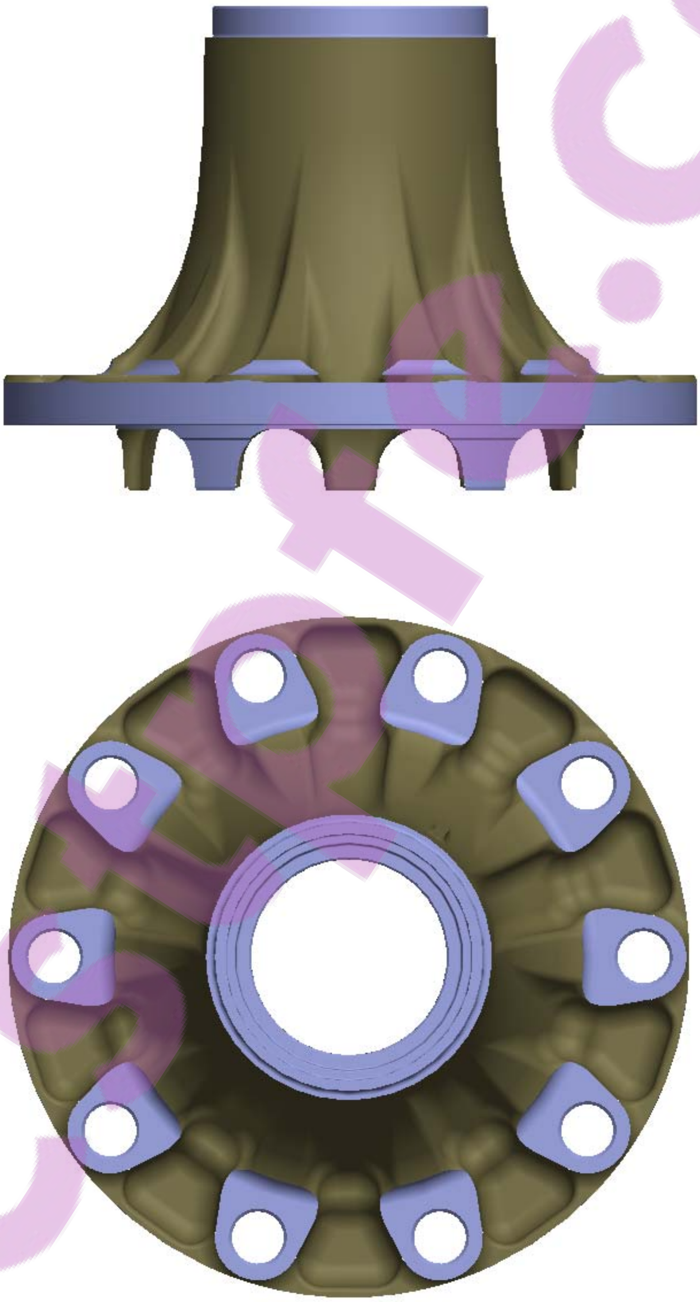


Figure 5.1. Front and Top view of the Original Hub model

Bestpfe.com

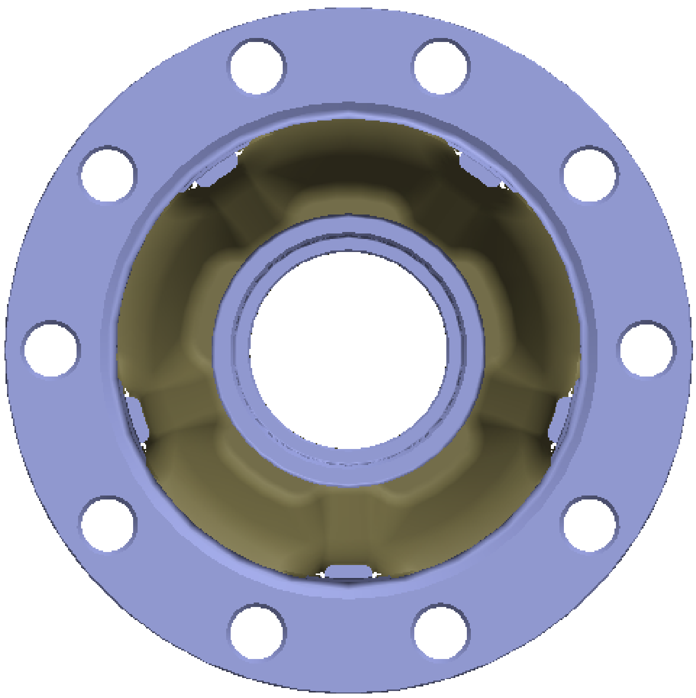


Figure 5.2. Bottom view of the Original Hub model

The mold of the original Hub is just a box with the Hub cavity in its center. The external dimensions are 0.7x0.7x0.65. See *Figure 5.3.*

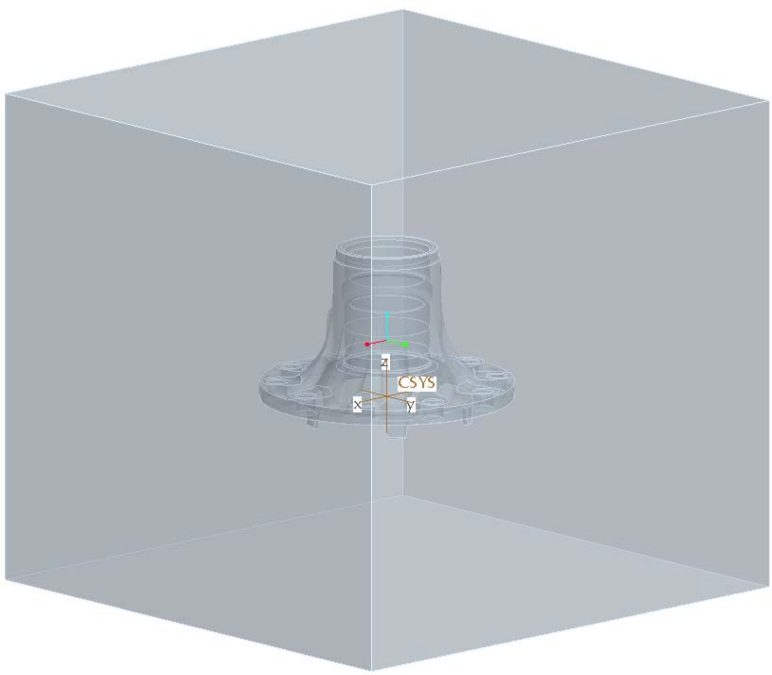


Figure 5.3. The Hub Mold part. Units: Meters

5.2 Thermal and stress curves points placement

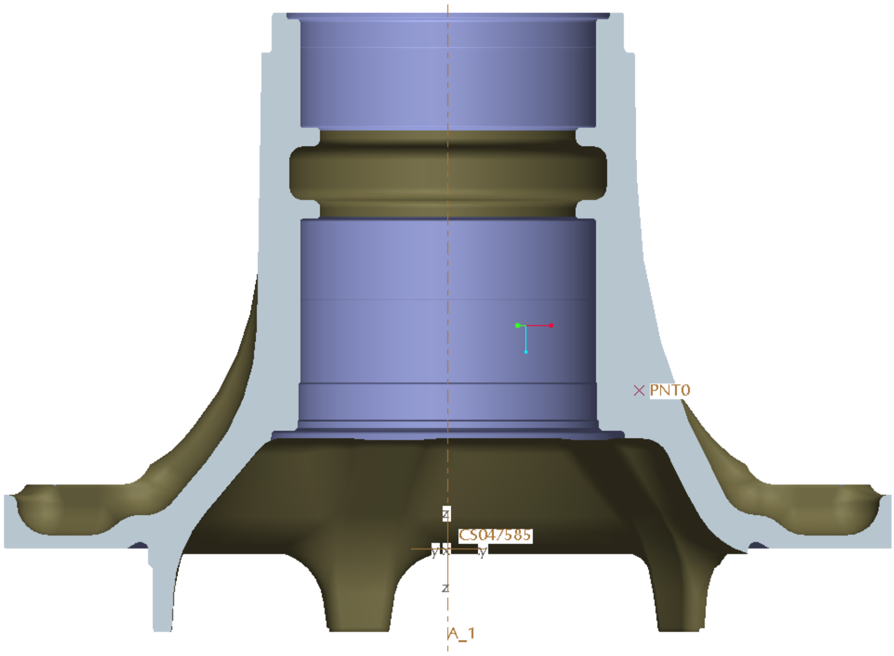
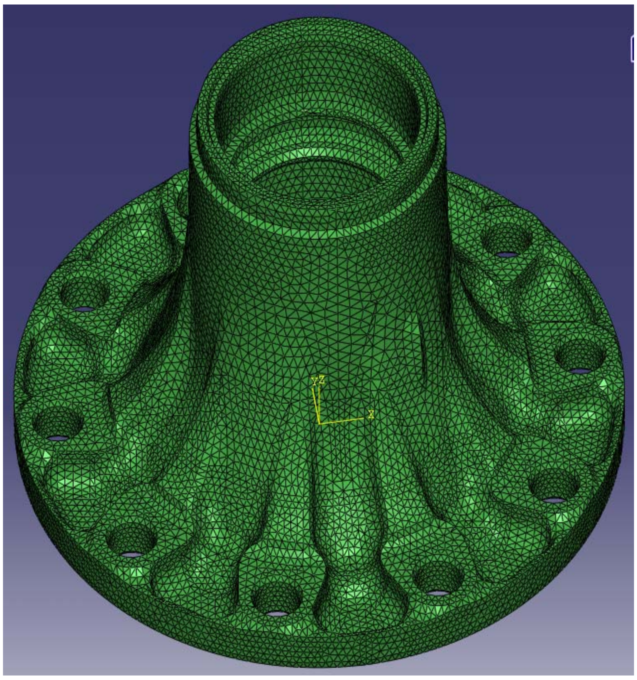


Figure 5.4. Location of the cooling and stress points of the Hub part

5.3 Mesh

	Original Hub		Mold	
	Elements	Nodes	Elements	Nodes
Abaqus	142337	35599	387923	77829
Magmasoft	36064	36064	7963936	7963936

BESTPFE.COM
List of research project topics and materials



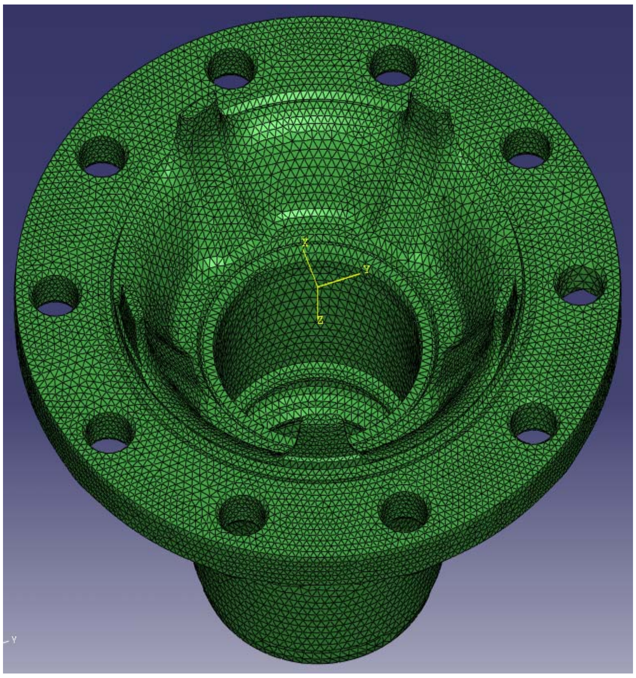


Figure 5.5. Mesh of the Hub model used in Abaqus

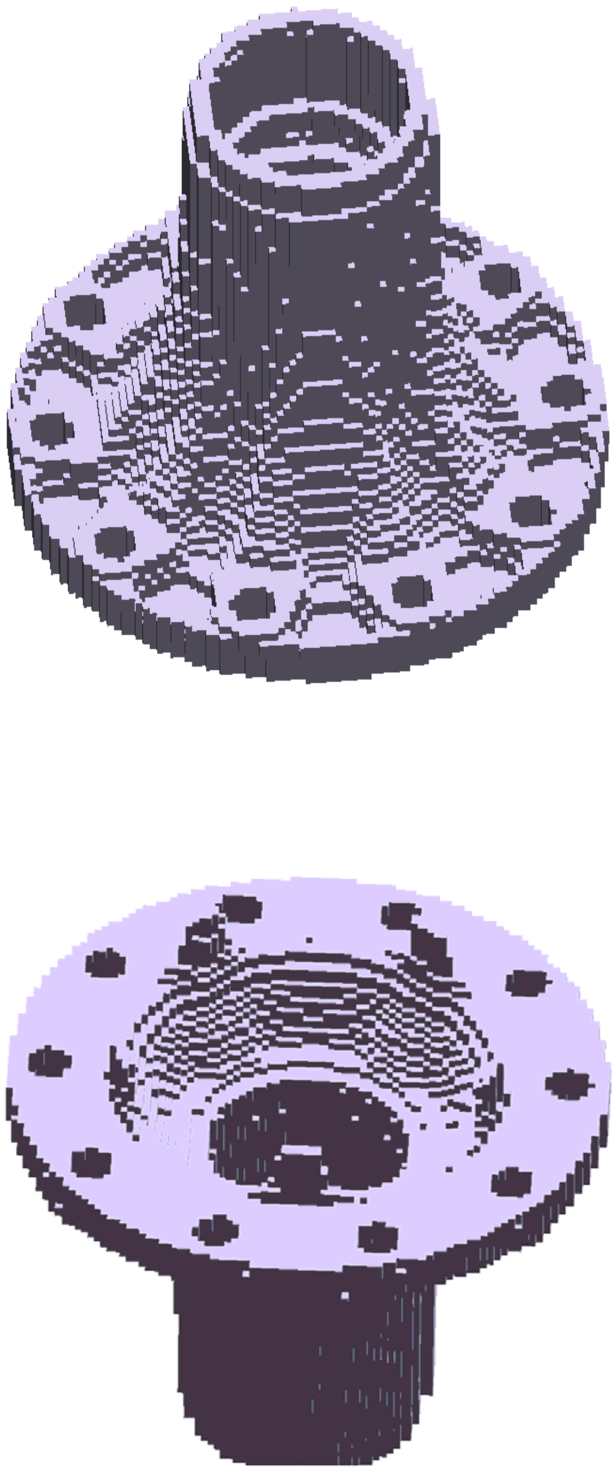


Figure 5.6. Magmasoft mesh of the Hub model

5.4 Boundary Conditions

5.4.1 Thermal boundary conditions

5.4.1.1 Before Shake-out

Conduction. Between the external surface of the casting and the surface of the mold cavity. For details on how to set this type of boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under “9-Interaction Properties Definition” and “10-Interactions Definitions”). For details on how to set this boundary condition in Magmasoft, refer to section 10.2.1.3 (instruction for *Figure 10.63*).

Convection. Between the external surface of the mold and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under “10-Interactions Definitions”). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

Radiation. Between the external surface of the mold and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under “10-Interactions Definitions”). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

5.4.1.2 After shake out

Convection. Between the external surface of the casting and the ambient. A temperature dependent convective heat transfer coefficient property was defined in Abaqus. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.2 (under “9-Interaction Properties Definition” and “10-Interactions Definitions”). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

Radiation. Between the external surface of the casting and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.2 (under “10-Interactions Definitions”). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

5.4.2 Mechanical boundary conditions

5.4.2.1 Stress analysis step

As mentioned in the Cylinder model results, the user does not participate directly in the definition of boundary conditions for the stress analysis in Magmasoft. It is an automatic procedure. Therefore just the Abaqus approach is presented.

The task is to constrain the rigid body translations and rotations in X, Y and Z, but allow the body to deform. In the Original Hub model the 6 degrees of freedom has been constrained as follow:

Translations in X, Y and Z

In the top flat surface of the Original Hub, a node in X=0 is constrained in X, Y and Z. See *Figure 5.7.* Notice that in the picture, X is the horizontal axis, Y the vertical axis, and the Z axis is perpendicular to the paper.

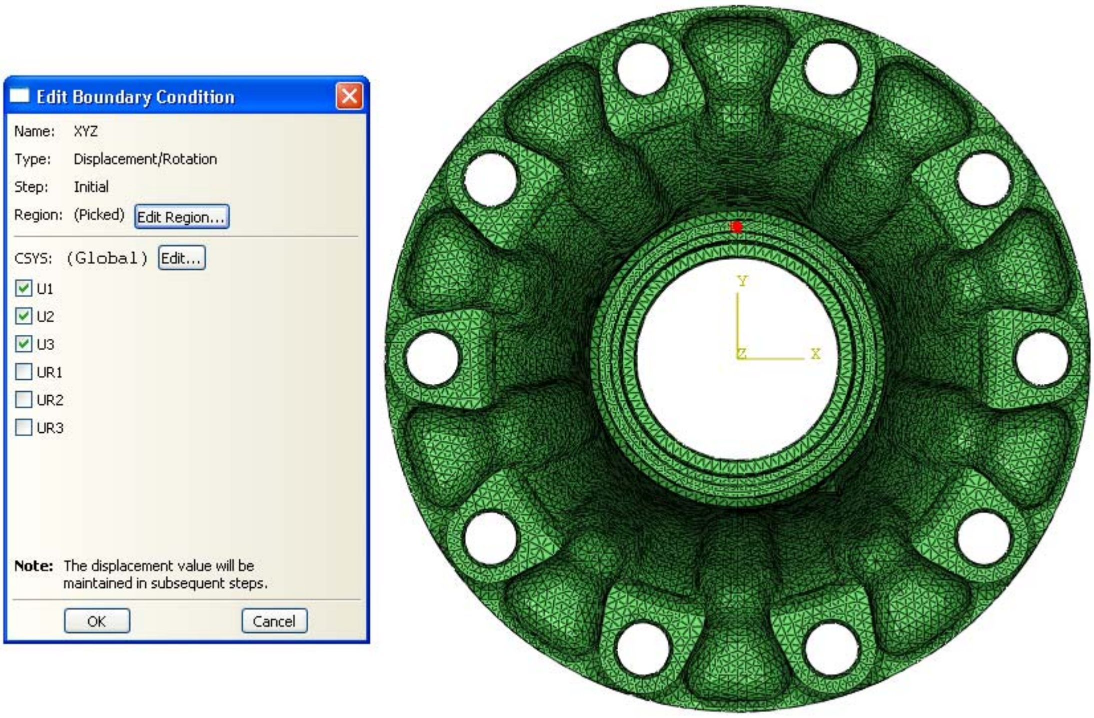


Figure 5.7. Constraining the rigid body translations in X, Y and Z in the Optimized Hub

By fixing this node in the space, three rigid body translations are constrained. Consequently, the part would shrink toward this node. Now, the remaining task is to constraint the three degrees of freedom corresponding to the rigid body rotations.

Rotations in the X and Z axis

A node also in the top flat surface of the Original Hub (with the same Z coordinate), and in X=0 is constrained in X, and Z. See *Figure 5.8.*

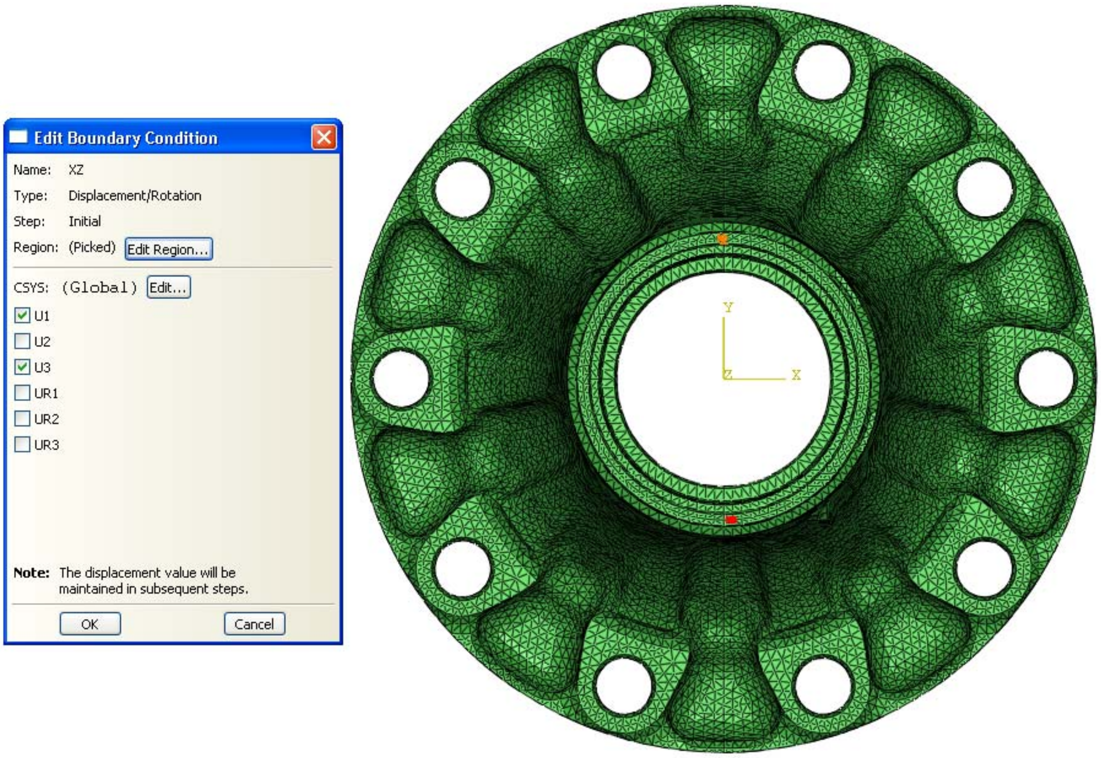


Figure 5.8. Constraining the rotations in the X and Z axes in the Original Hub

The constraint in X avoids the rotation in the Z axis and the constraint in Z avoids the rotation in the X axis. Notice that the node is free to move in the Y direction so the part is still able to deform normally.

Rotation in the Y axis

A node in X=0 and vertically aligned with the totally fixed node (same Y coordinate) but in a different Z coordinate, in this case in the flat surface of the end of the cylindrical section of the hub, is constrained in X and Y. See *Figure 5.9*. In this way the rotation in the X axis is restrained and all vertical axes of the part are fixed to remain parallel to the Z axes.

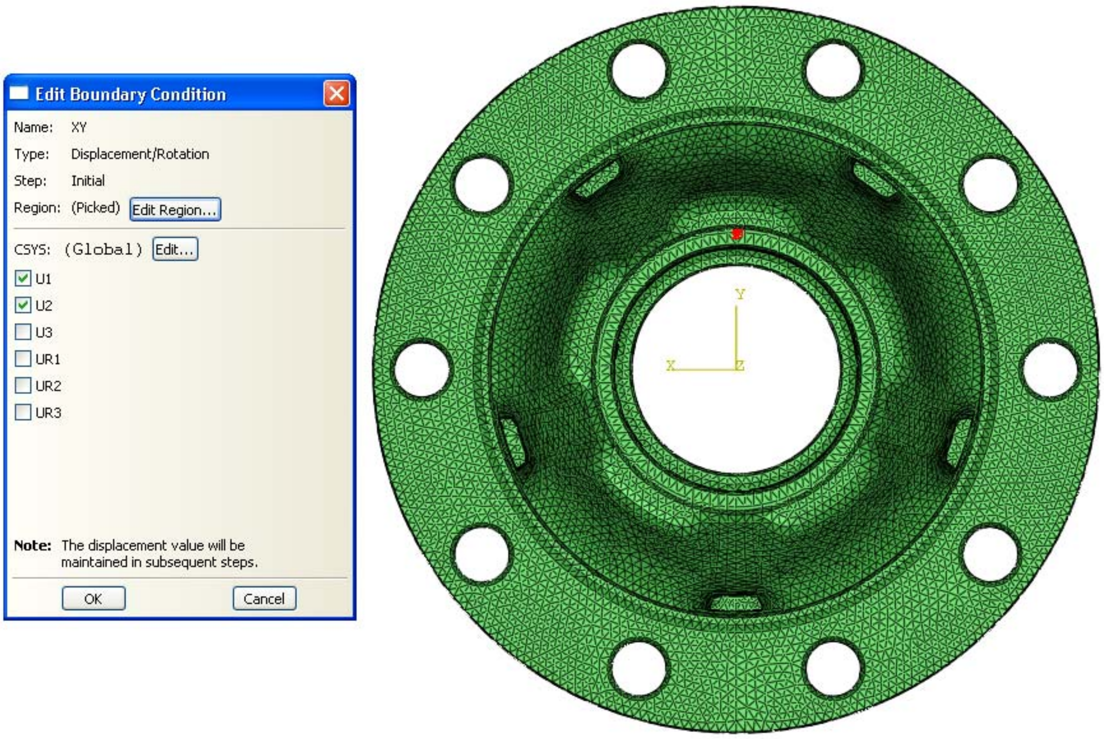


Figure 5.9. Constraining the rotation in the X axis in the Original Hub

5.5 Cooling curves for the Original Hub

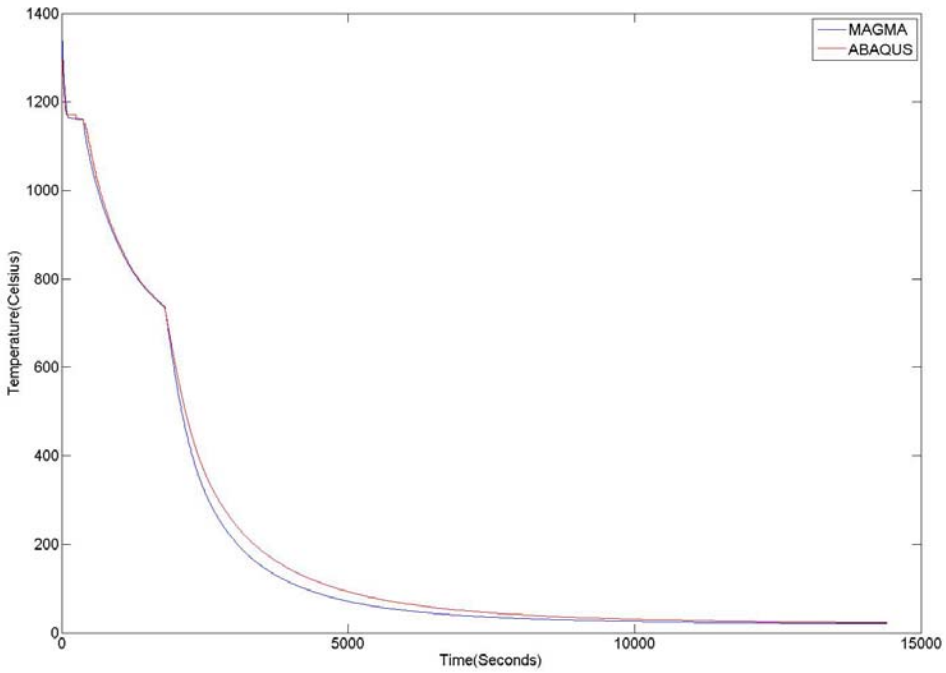


Figure 5.10. Abaqus vs. Magmasoft cooling curves for the Hub model

The results presented in *Figure 5.10* were obtained from PNT0 which location is shown is *Figure 5.4*.

5.6 Thermal color spectrums

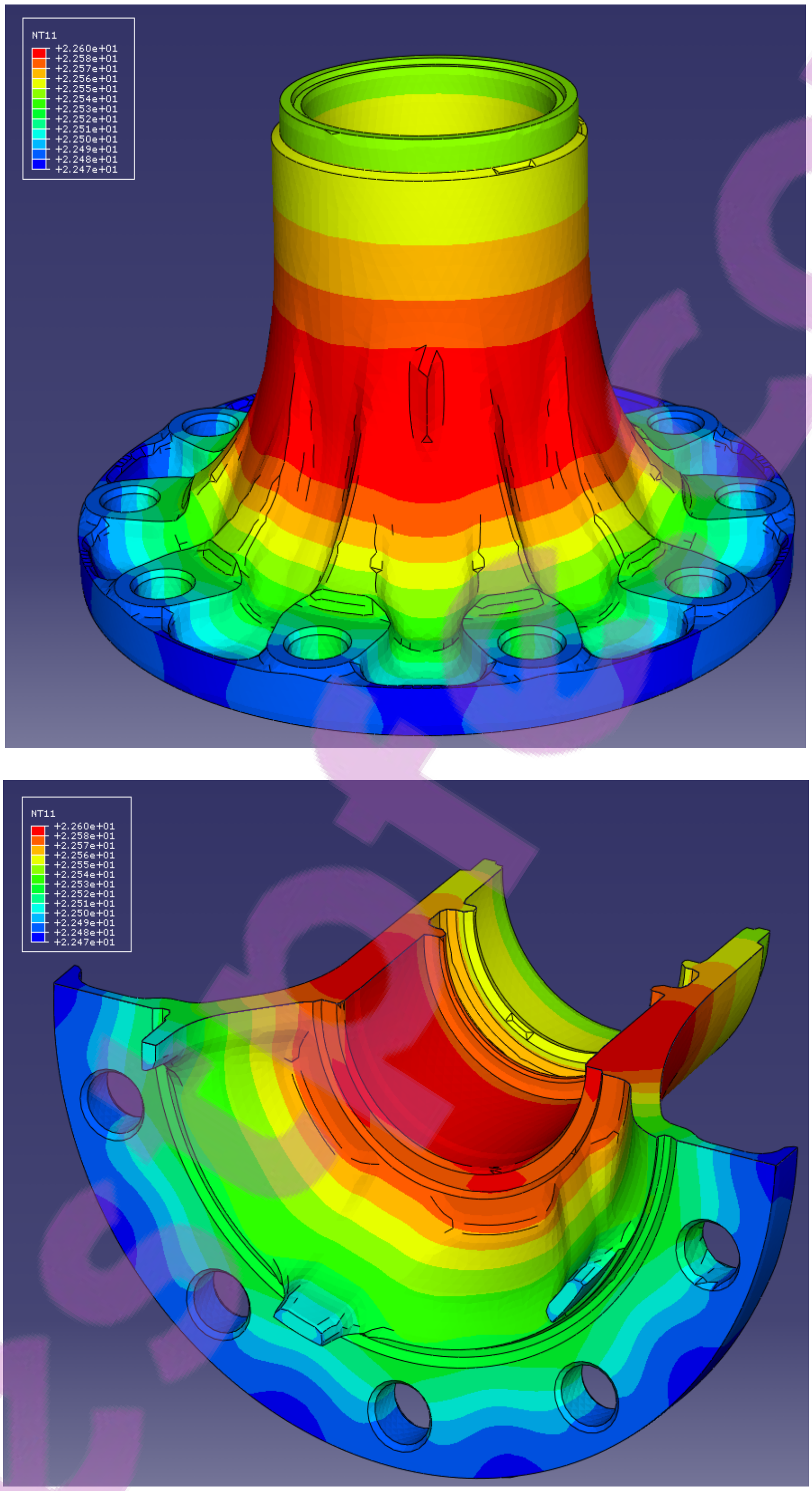


Figure 5.11. Abaqus thermal color spectrums for the last step after shake-out of the Original Hub model

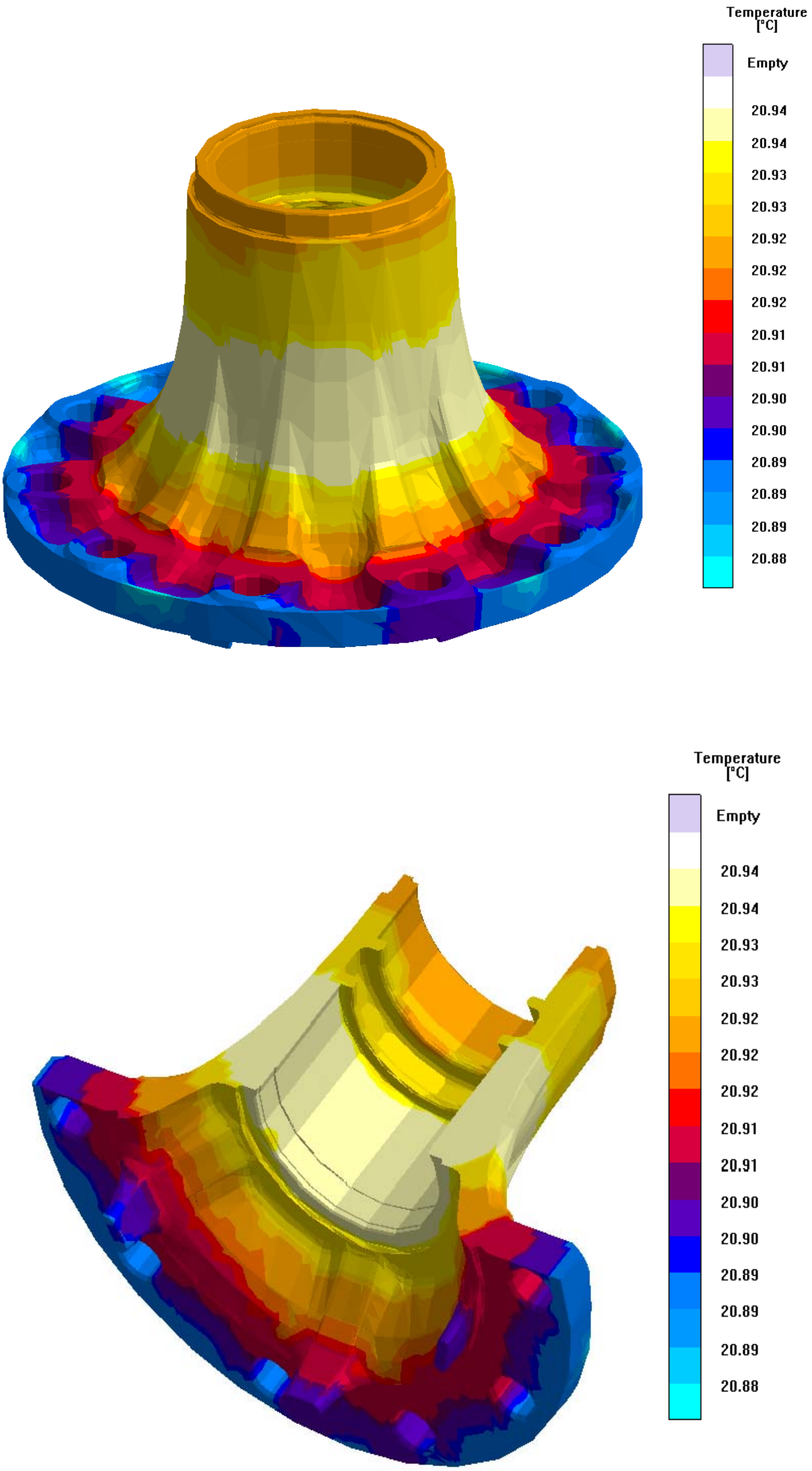


Figure 5.12. Magmasoft thermal color spectrums for the last step after shake-out of the Original Hub model

5.7 Stress curves for the Original Hub

Using PNT0 as reference point as shown in *Figure 5.4*:

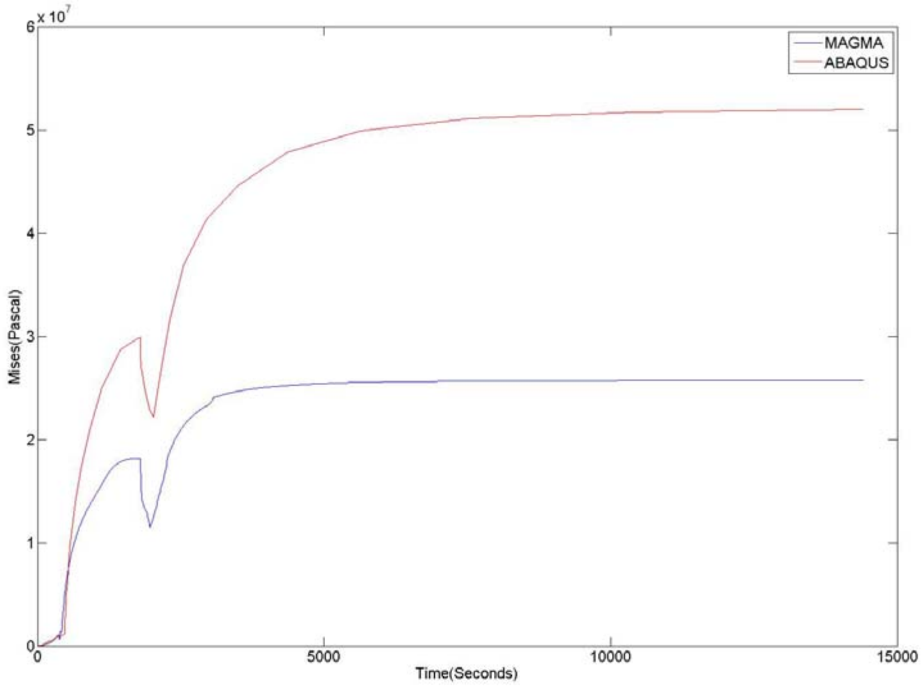


Figure 5.13. Von Mises curves from PNT0 of the Original Hub.

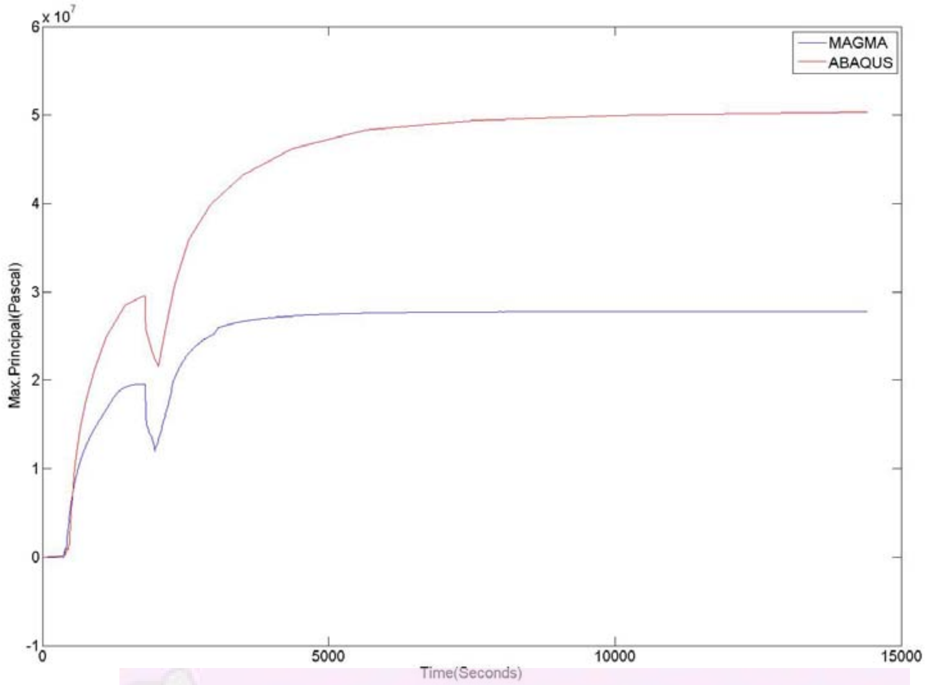


Figure 5.14. Maximum Principal stresses from PNT0 of the Original Hub.

BESTPFE.COM
List of research project topics and materials

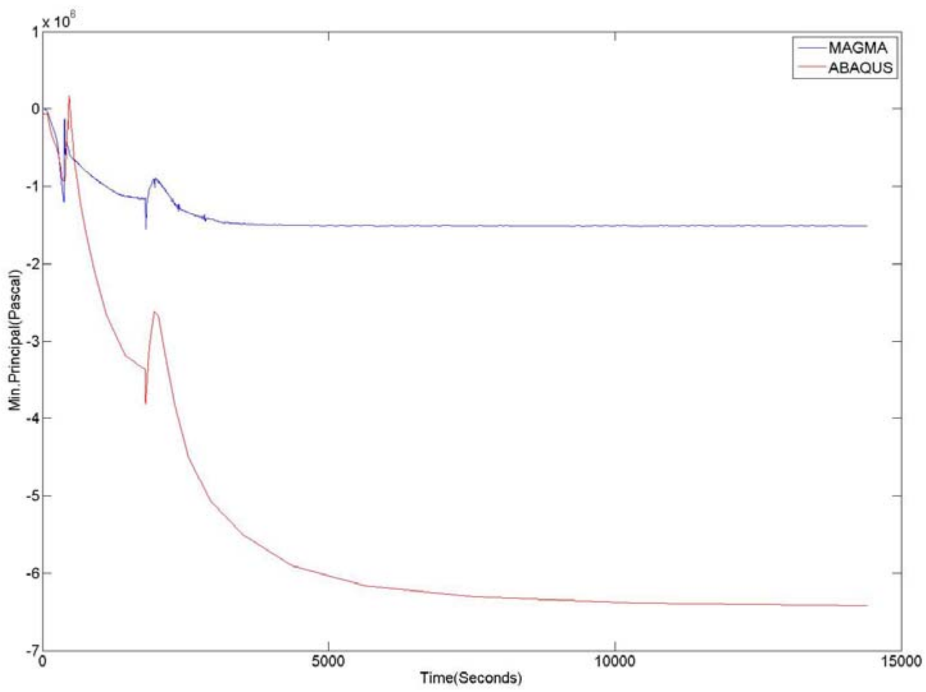
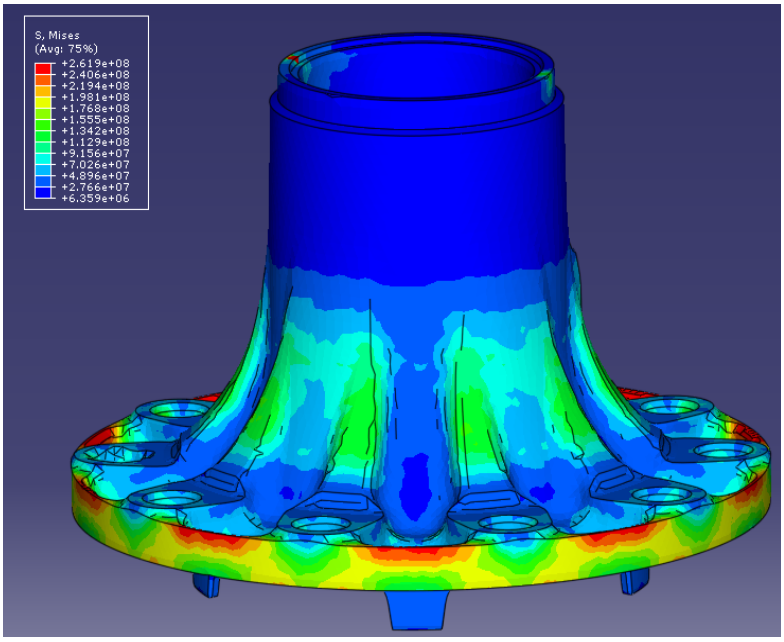


Figure 5.15. Minimum Principal stresses from PNT0 of the Original Hub.

5.8 Stress color spectrums



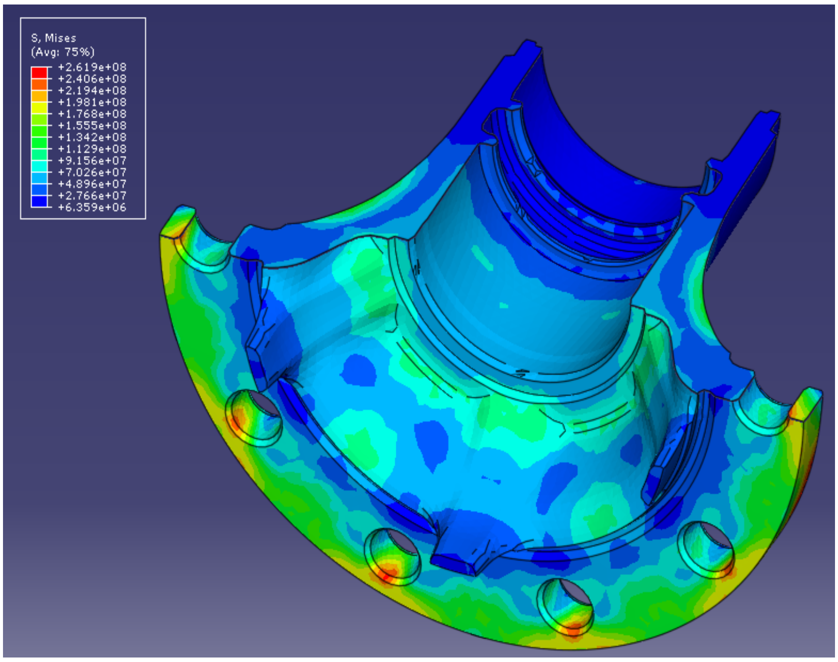


Figure 5.16. Abaqus color spectrums for the Mises results of the Original Hub.

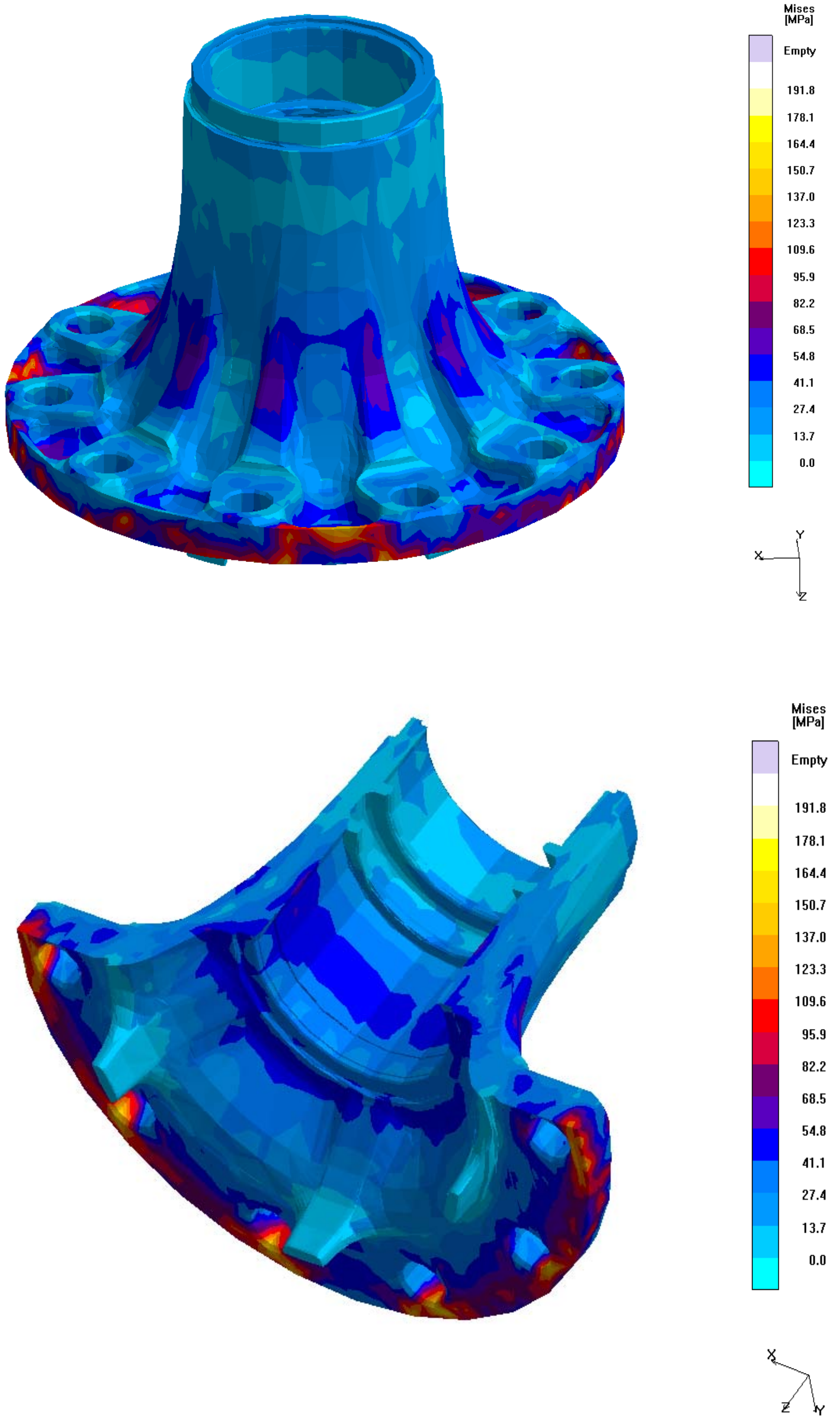
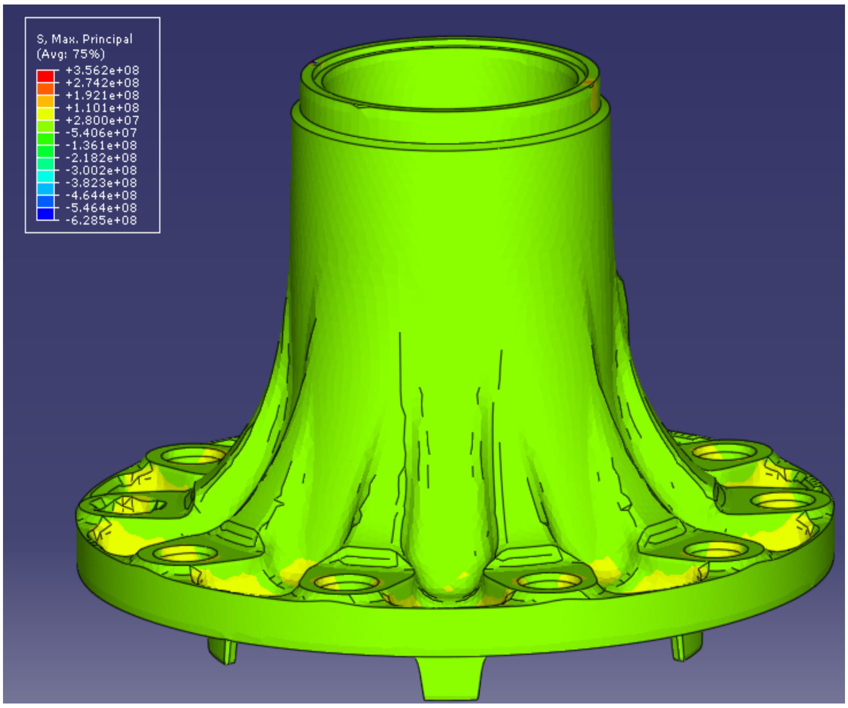


Figure 5.17. Magmasoft color spectrums for the Mises results of the Original Hub.



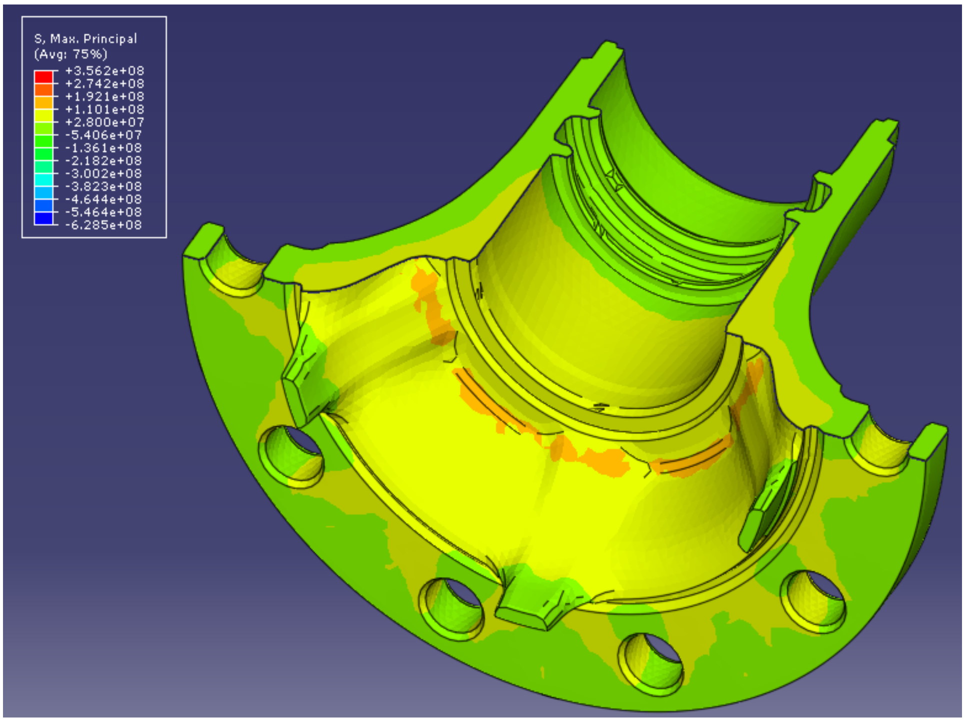


Figure 5.18. Abaqus color spectrums for the residual Max. Principal stresses of the Original Hub.

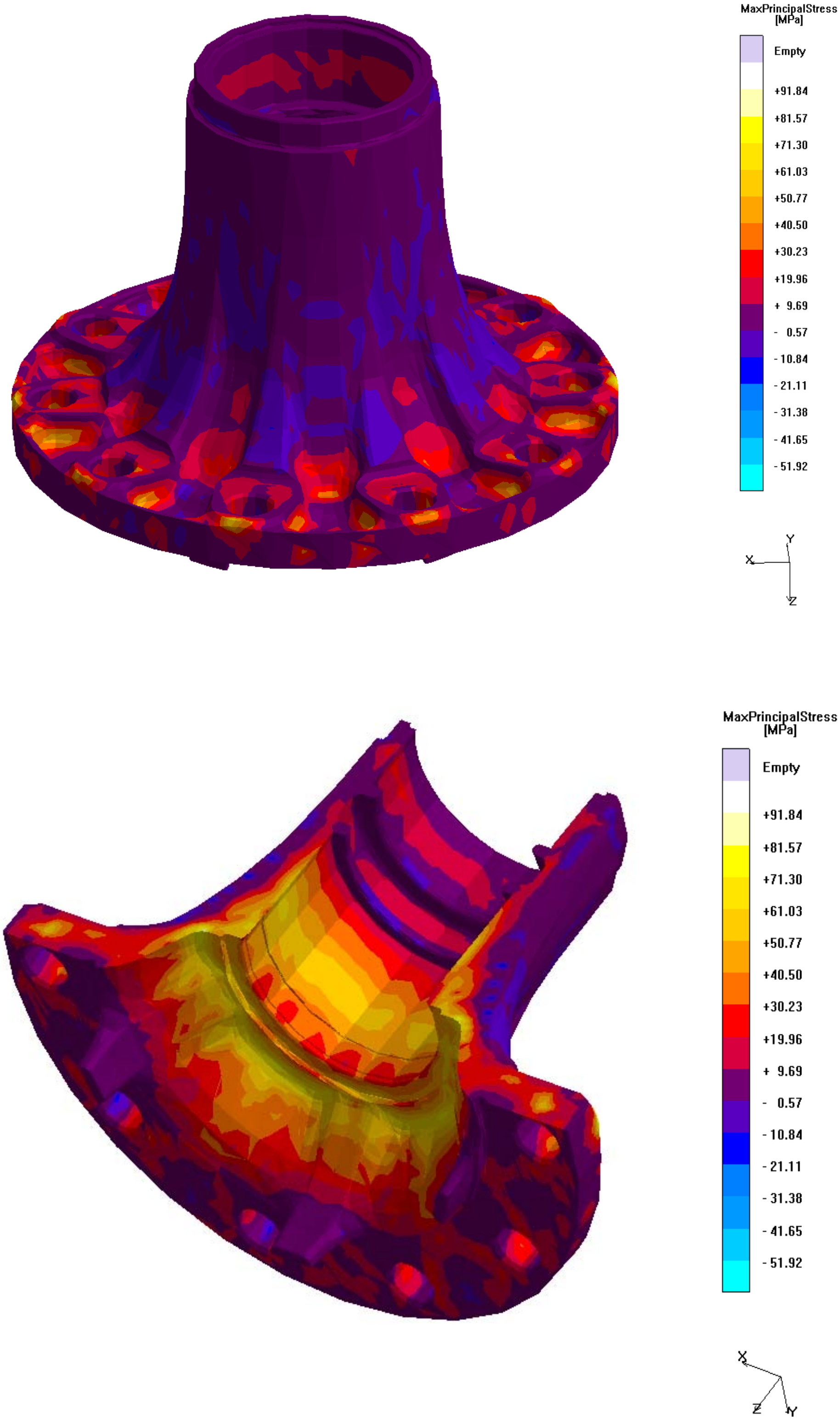
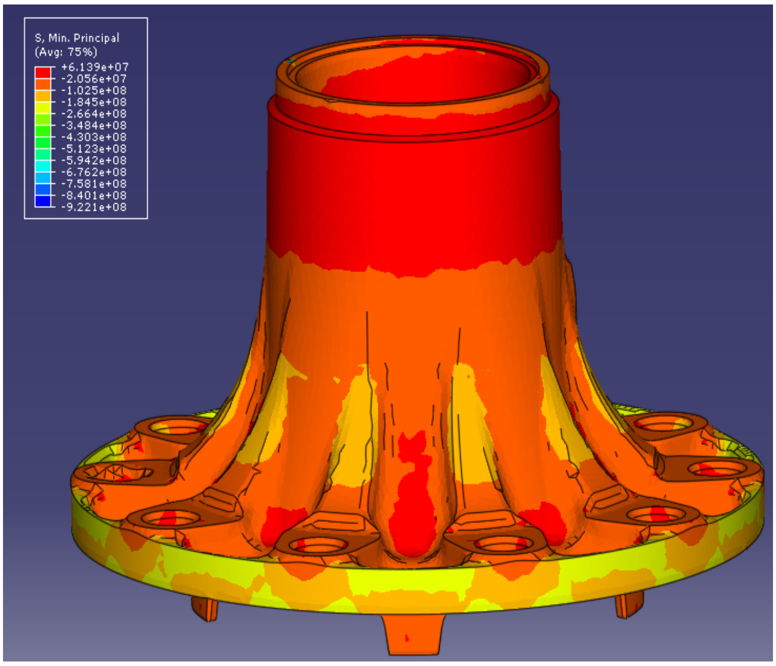


Figure 5.19. Magmasoft color spectrums for the residual Max. Principal stresses of the Original Hub.



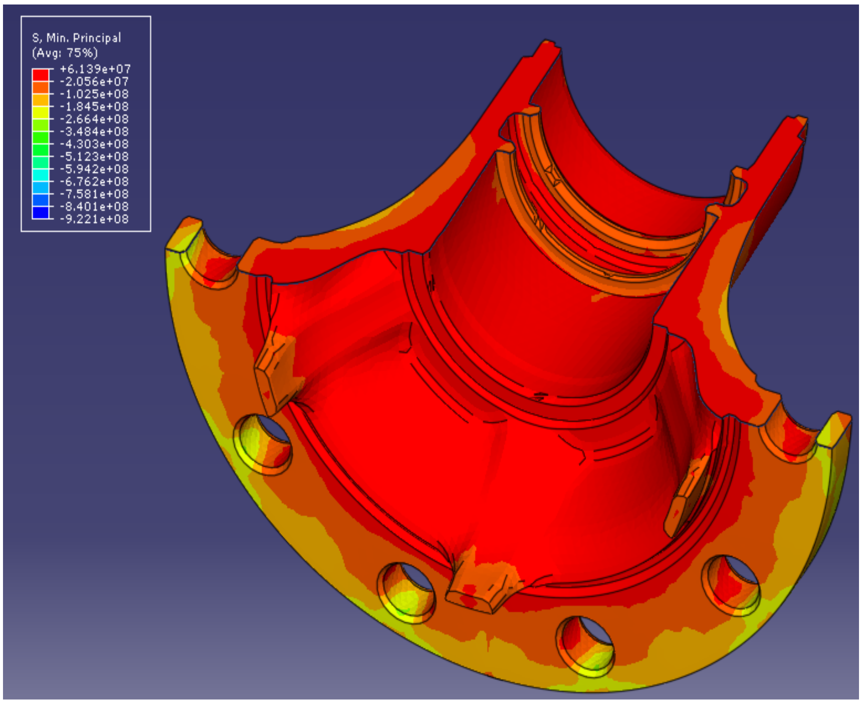
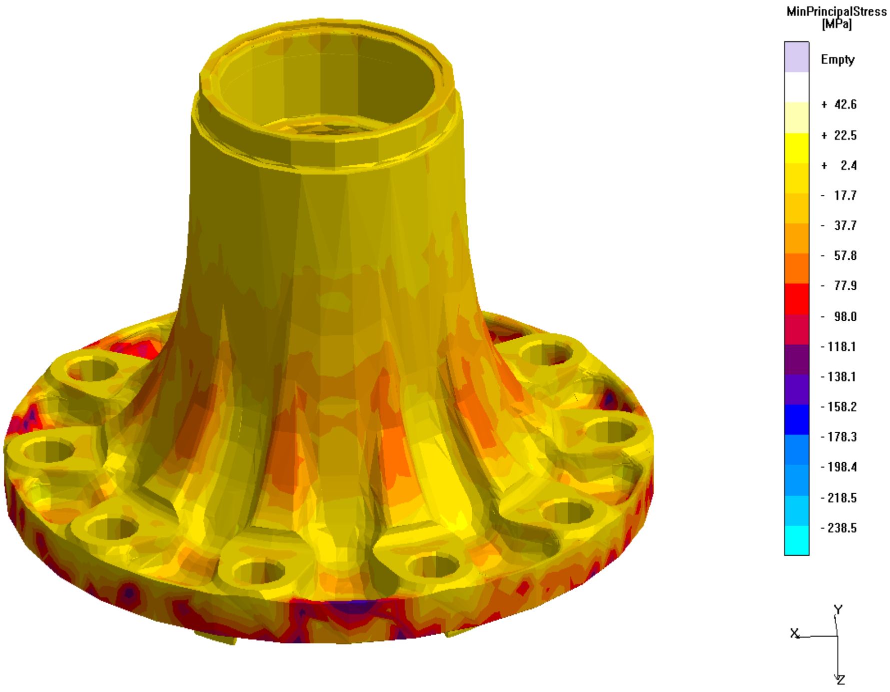


Figure 5.20. Abaqus color spectrums for the residual Min. Principal stresses of the Original Hub.



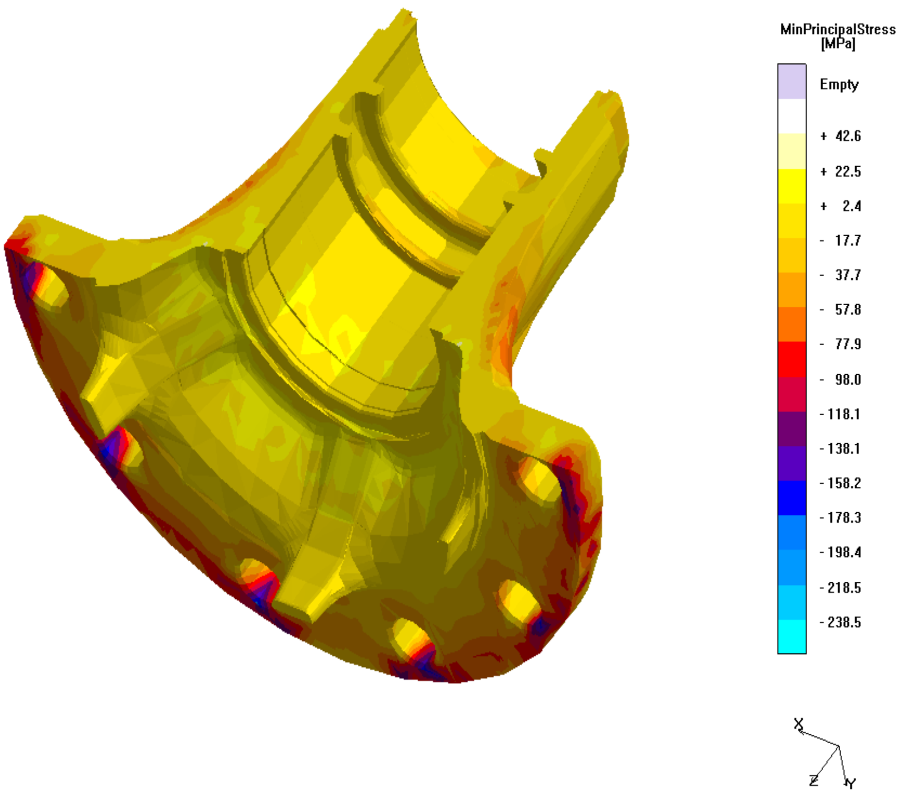


Figure 5.21. Magmasoft color spectrums for the residual Min. Principal stresses of the Original Hub.

5.9 Simulation time of the Original Hub

	Thermal	Stress	Total
Abaqus	52hrs. 6min.	10hrs. 50min.	62hrs. 56min.
Magmasoft			08hr. 05min.

Bestpfe.com

6 Optimized Hub Results

6.1 Geometry

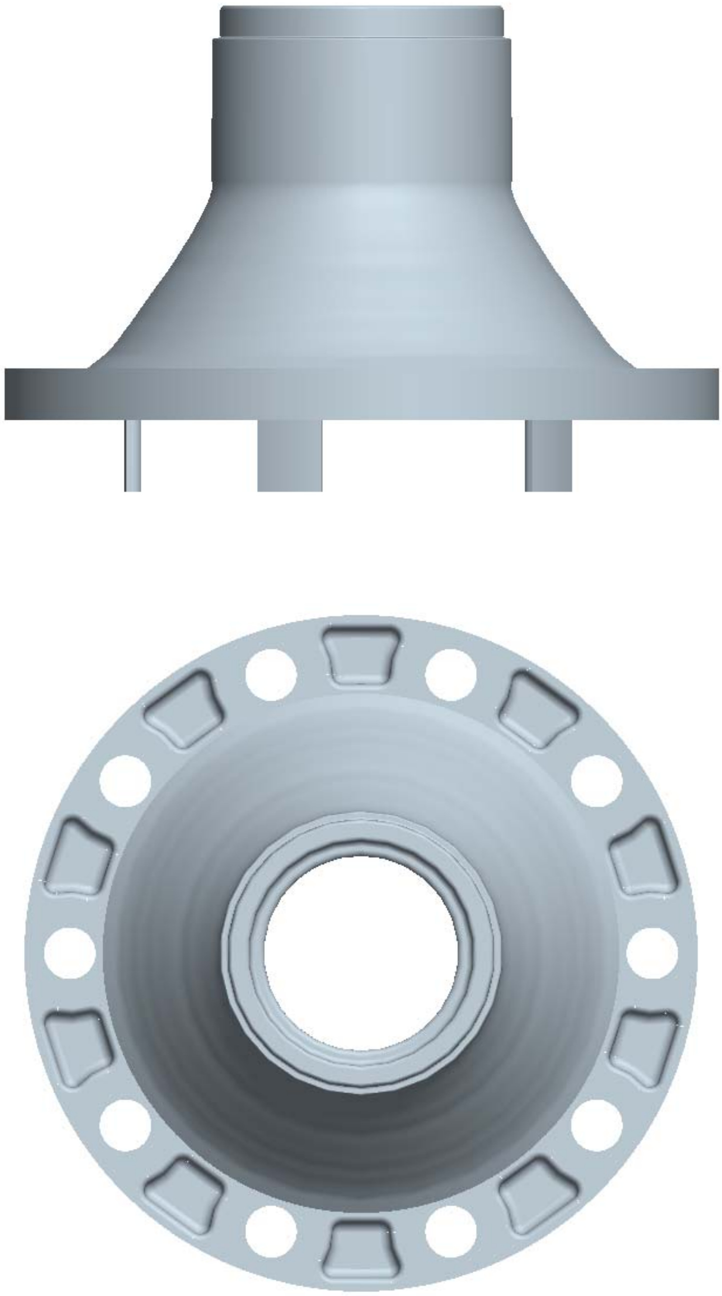


Figure 6.1. Front and Top view of the Optimized Hub model

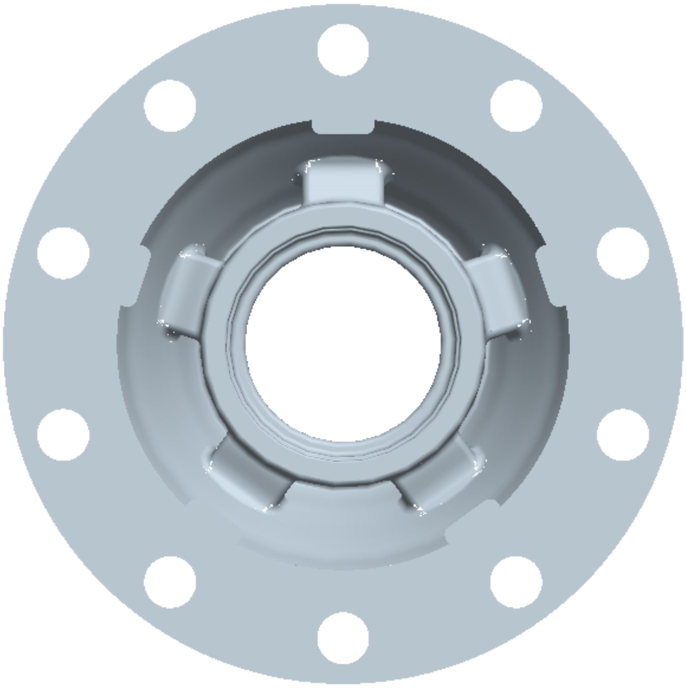
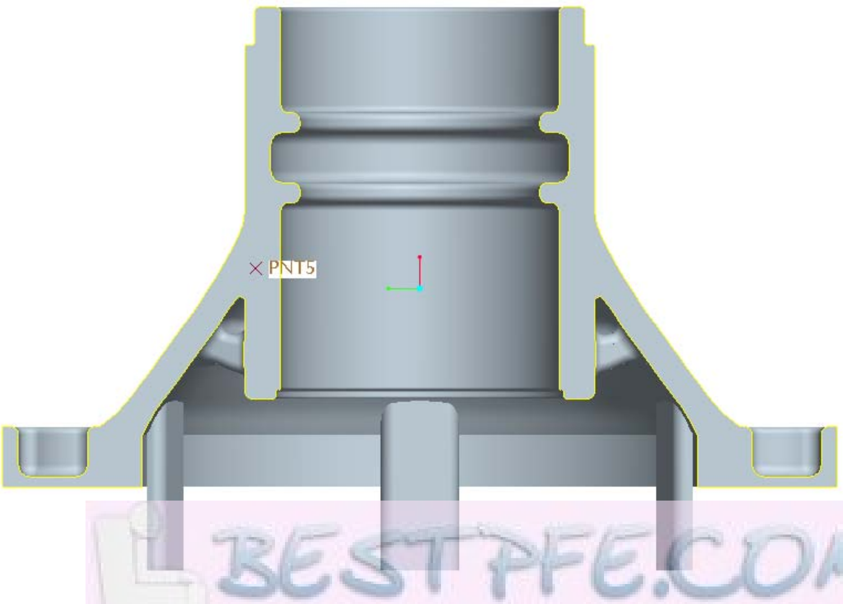


Figure 6.2. Bottom view of the Optimized Hub model

The Optimized Hub mold has the same dimensions as the Original Hub mold as defined in section 5.1

6.2 Thermal and stress curves points placement



BESTPFE.COM
List of research project topics and materials

Figure 6.3. Location of cooling and stress points 5 and 6 for the Optimized Hub.

6.3 Mesh

	Optimized Hub		Mold	
	Elements	Nodes	Elements	Nodes
Abaqus	252767	61039	570286	115089
Magmasoft	61068	61068	1188932	1188932

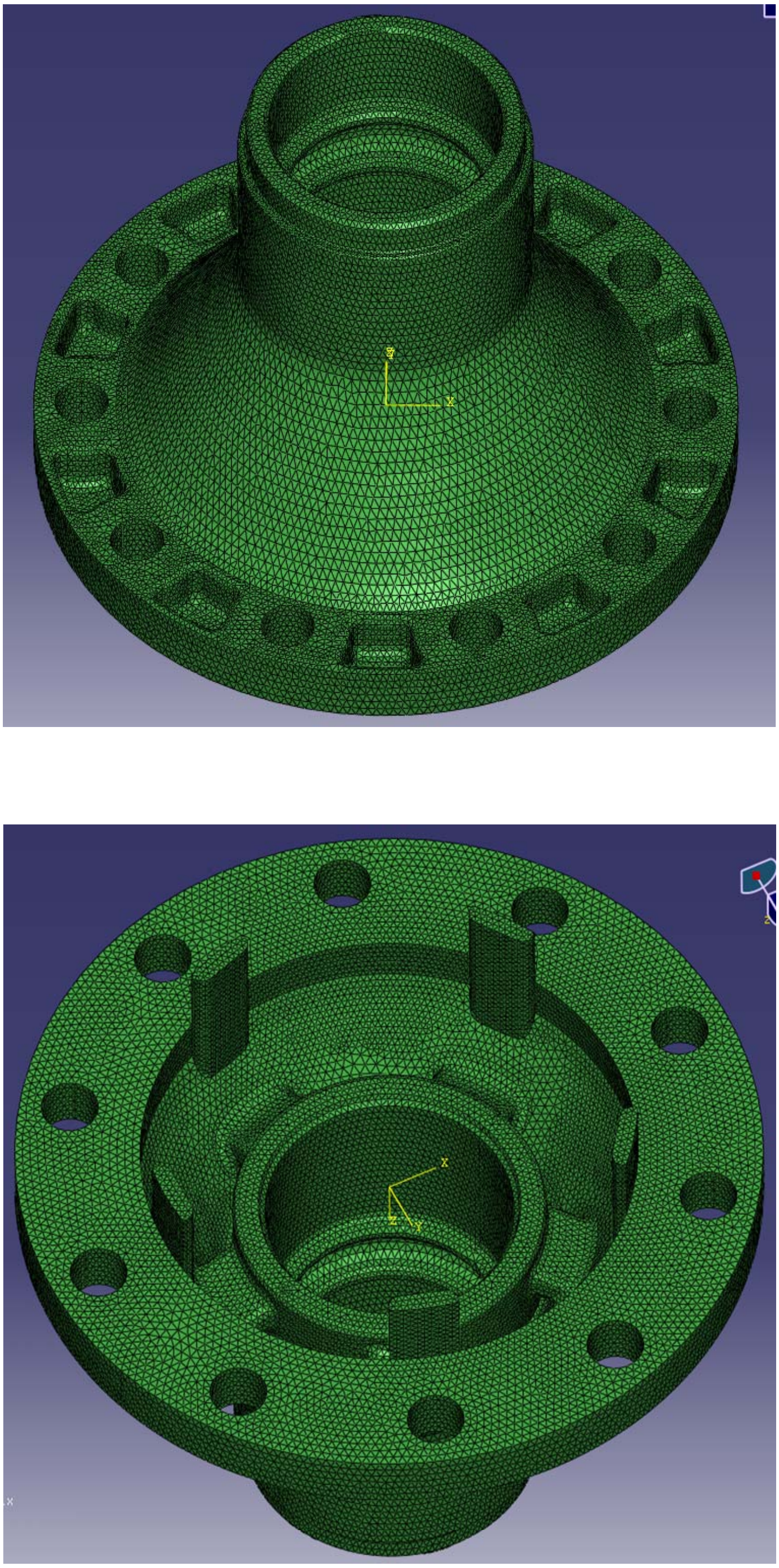


Figure 6.4. Mesh of the Optimized Hub model used in Abaqus

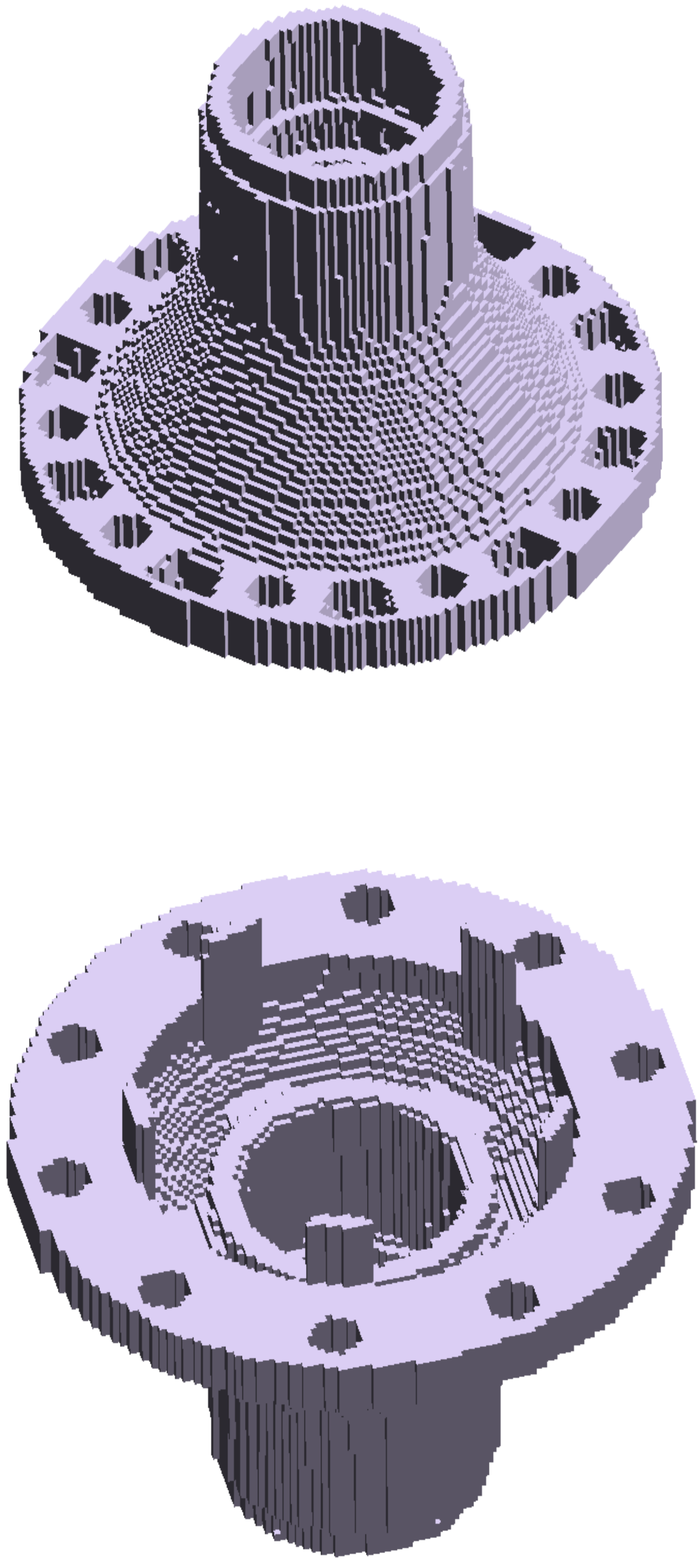


Figure 6.5. Magma mesh of the Optimized Hub model.

6.4 Boundary Conditions

6.4.1 Thermal boundary conditions

6.4.1.1 Before Shake-out

Conduction. Between the external surface of the casting and the surface of the mold cavity. For details on how to set this type of boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under "9-Interaction Properties Definition" and "10-Interactions Definitions"). For details on how to set this boundary condition in Magmasoft, refer to section 10.2.1.3 (instruction for *Figure 10.63*).

Convection. Between the external surface of the mold and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

Radiation. Between the external surface of the mold and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.1 (under "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

6.4.1.2 After shake out

Convection. Between the external surface of the casting and the ambient. A temperature dependent convective heat transfer coefficient property was defined in Abaqus. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.2 (under "9-Interaction Properties Definition" and "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

Radiation. Between the external surface of the casting and the ambient. For details on how to set this boundary condition in Abaqus, refer to section 10.1.1.3.1.2 (under "10-Interactions Definitions"). In Magmasoft this condition is defined automatically so the user has no participation in the setting.

6.4.2 Mechanical boundary conditions

6.4.2.1 Stress analysis step

We must mention again that the user does not participate directly in the definition of boundary conditions for the stress analysis in Magmasoft. Therefore, we present just our Abaqus approach.

As we have established in the Cylinder and Original Hub results, the task here also is to constrain the rigid body translations and rotations in X, Y and Z, but allow the body to deform. In the Optimized Hub model the 6 degrees of freedom has been constrained as follow:

Translations in X, Y and Z

In the flat surface of the upper inner ring of the cylindrical section, a node in Y=0 is constrained in X, Y and Z. See *Figure 6.6.* Notice that in the picture, X is the horizontal axis, Y the vertical axis, and the Z axis is perpendicular to the paper.

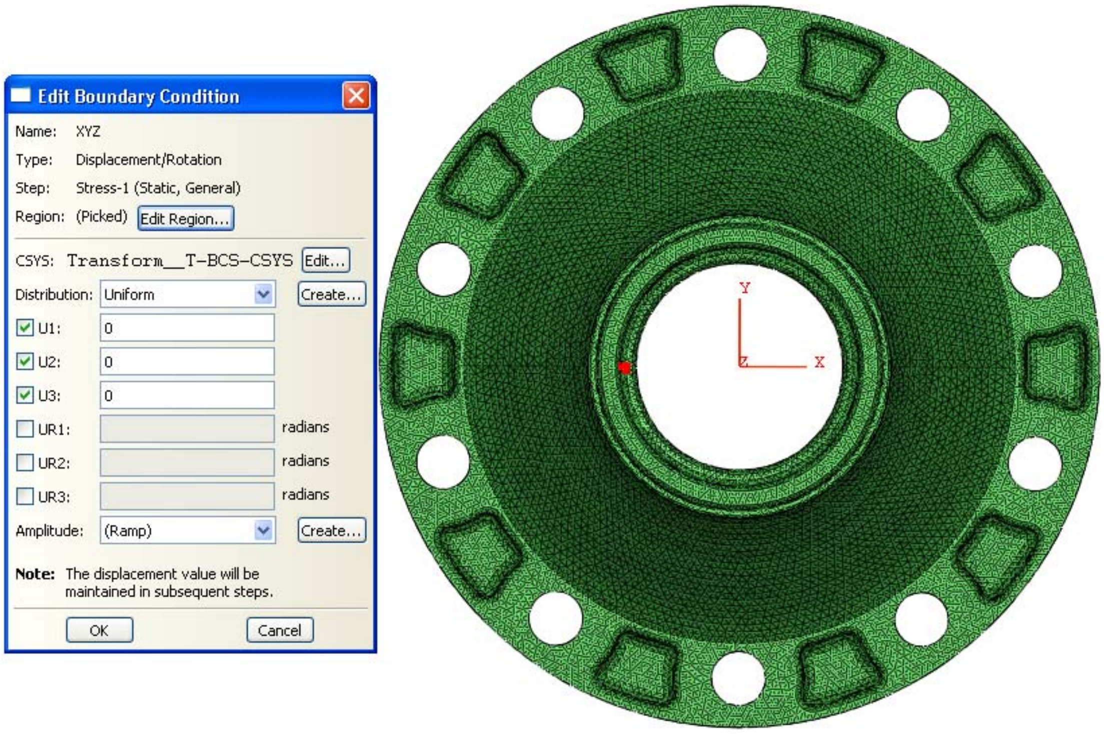


Figure 6.6. Constraining the rigid body translations in X, Y and Z in the Optimized Hub

By fixing this node in the space, three rigid body translations are constrained. Consequently, the part would shrink toward this node. Now, the remaining task is to constraint the three degrees of freedom corresponding to the rigid body rotations.

Rotations in the Y and Z axes

A node at the same Z level of the fully constrained one and also in Y=0 is fixed in Y and Z. See *Figure 6.7*. The node is free to move in the X direction so it can follow a correct shrinking trajectory.

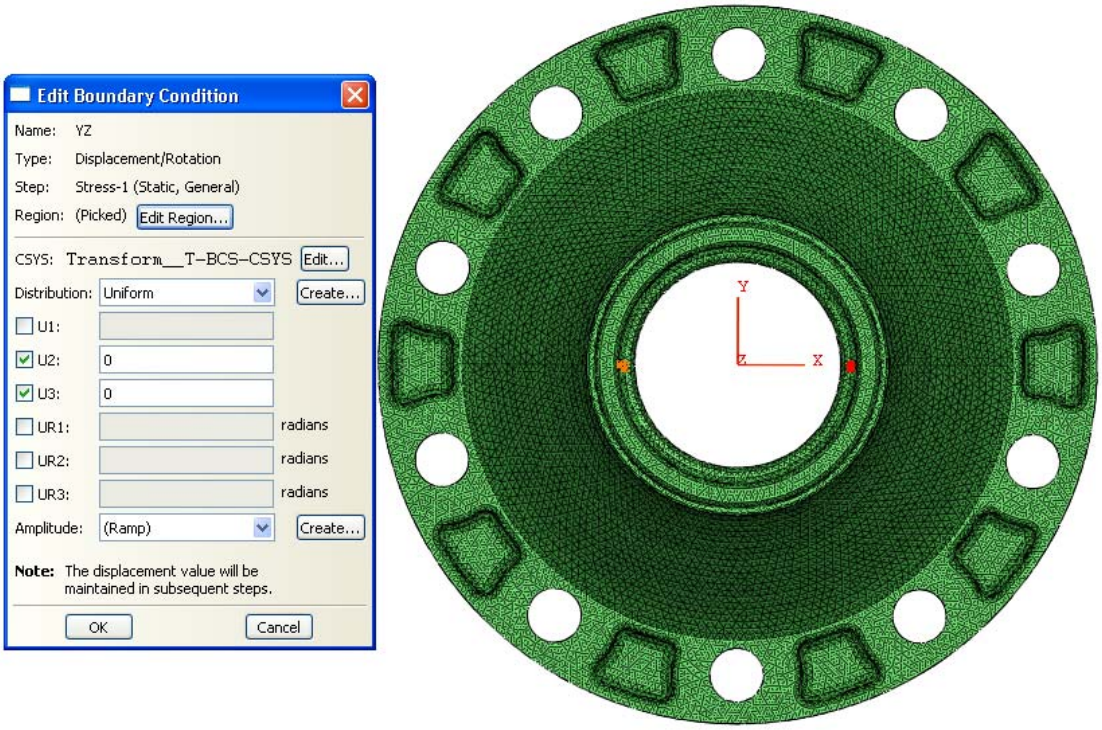


Figure 6.7. Constraining the rotations in the Y and Z axes in the Optimized Hub

Rotation in the X axis

A node in Y=0, with the same X coordinate that the totally fixed node but in a different Z coordinate, in this case in the bottom flat surface of the lower inner ring of the cylindrical section, is constrained in X and Y. See *Figure 6.8*. In this way the rotation in the X axis is restrained and all vertical axes of the part are fixed to remain parallel to the Z axes.

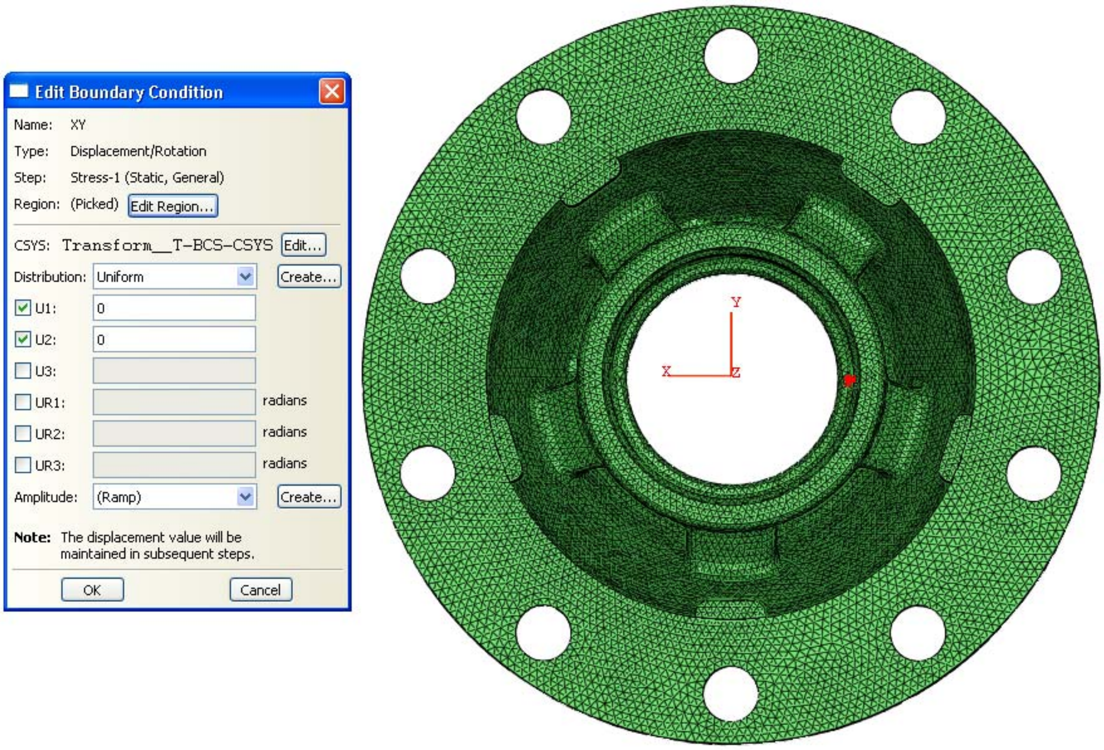


Figure 6.8. Constraining the rotation in the X axis in the Optimized Hub

6.5 Cooling curves for the Optimized Hub

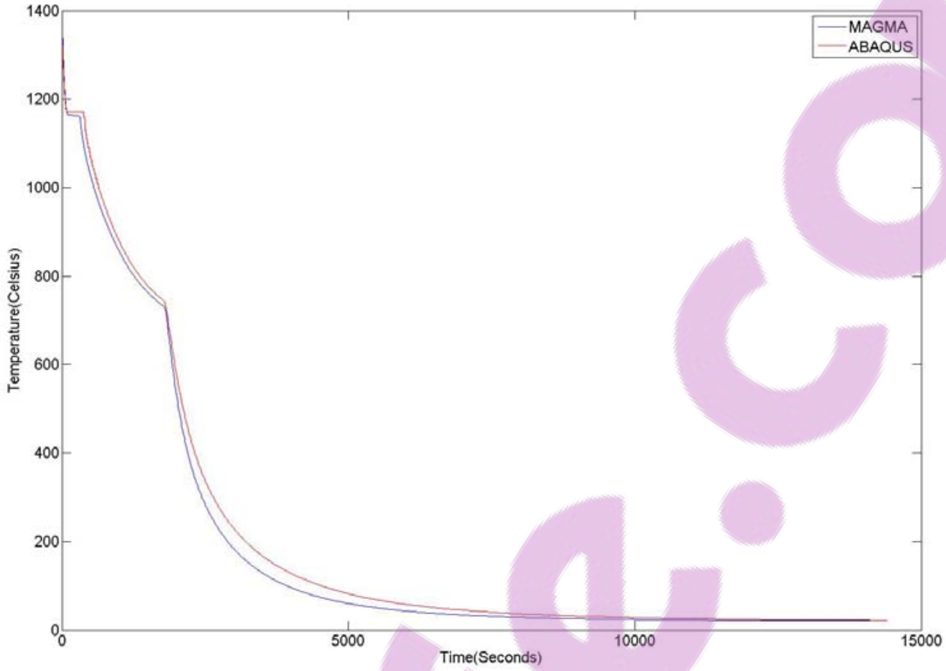
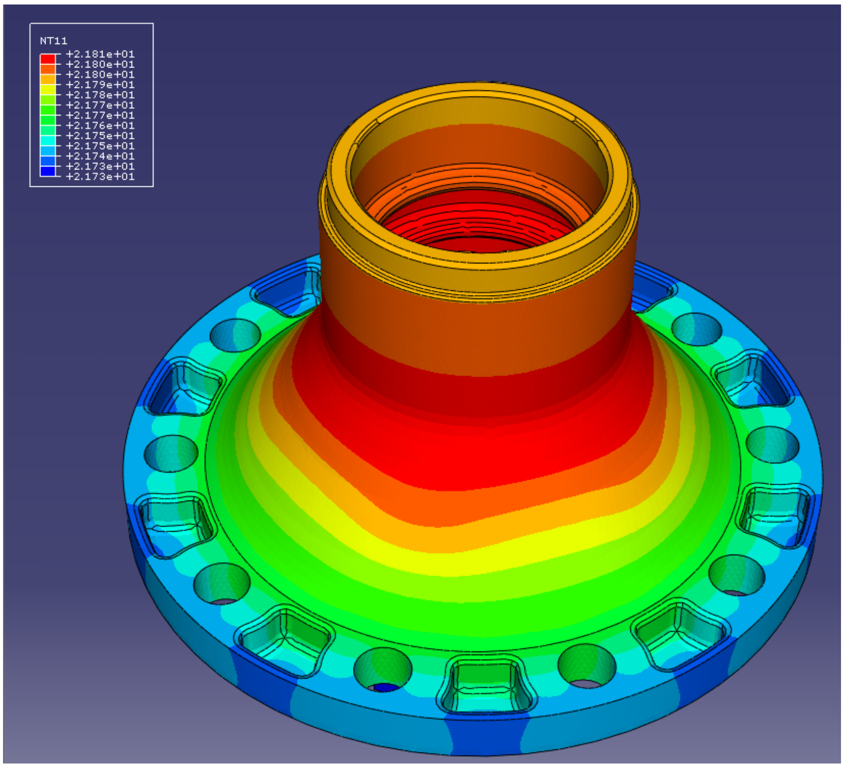


Figure 6.9. Abaqus vs. Magmasoft cooling curves for the Optimized Hub model

The reference point for the curves in *Figure 6.9* is PNT5, which location is shown in *Figure 6.3.*

Bestpfe.com

6.6 Thermal color spectrums



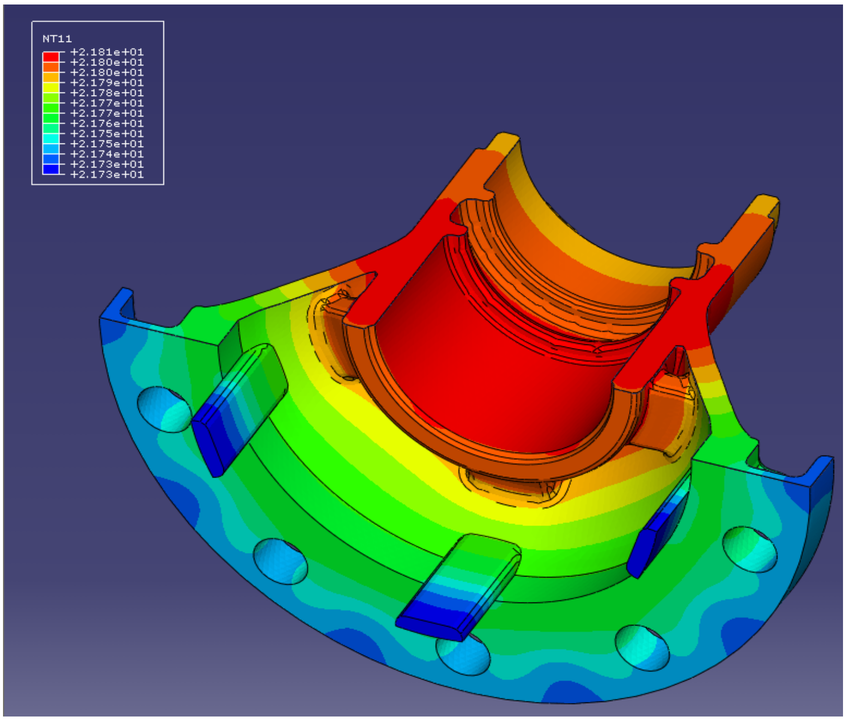
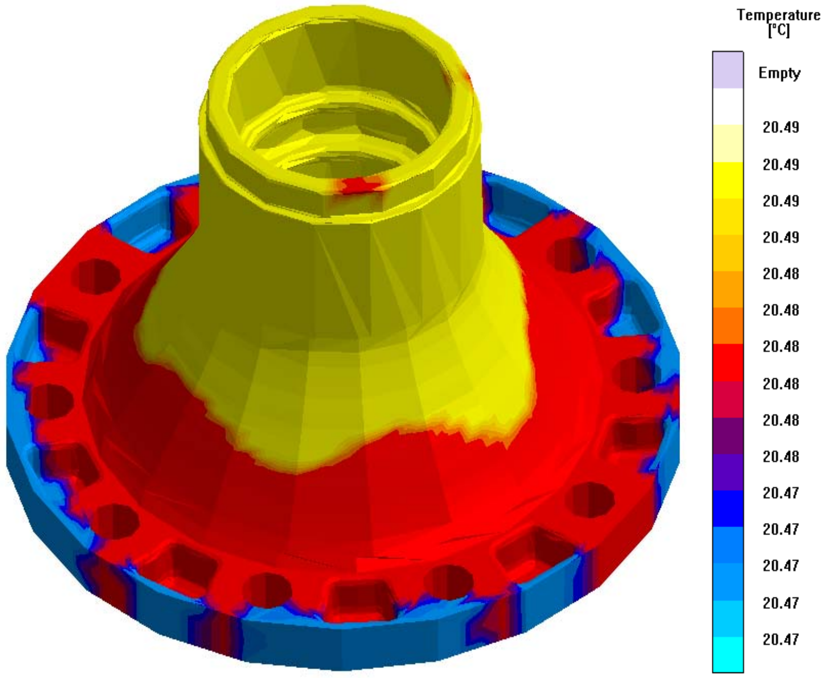


Figure 6.10. Abaqus thermal color spectrums for the last step after shake-out of the Optimized Hub model



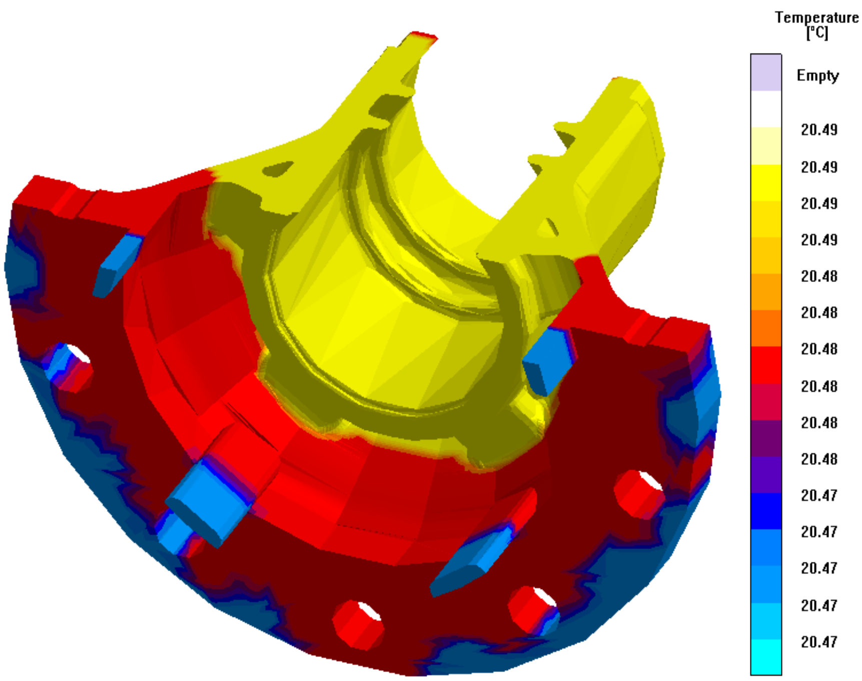


Figure 6.11. Magmasoft thermal color spectrums for the last step after shake-out of the Optimized Hub model

BESTPFE.COM
List of research project topics and materials

6.7 Stress curves for the Optimized Hub

Using PNT5 as reference point as shown in *Figure 6.3*:

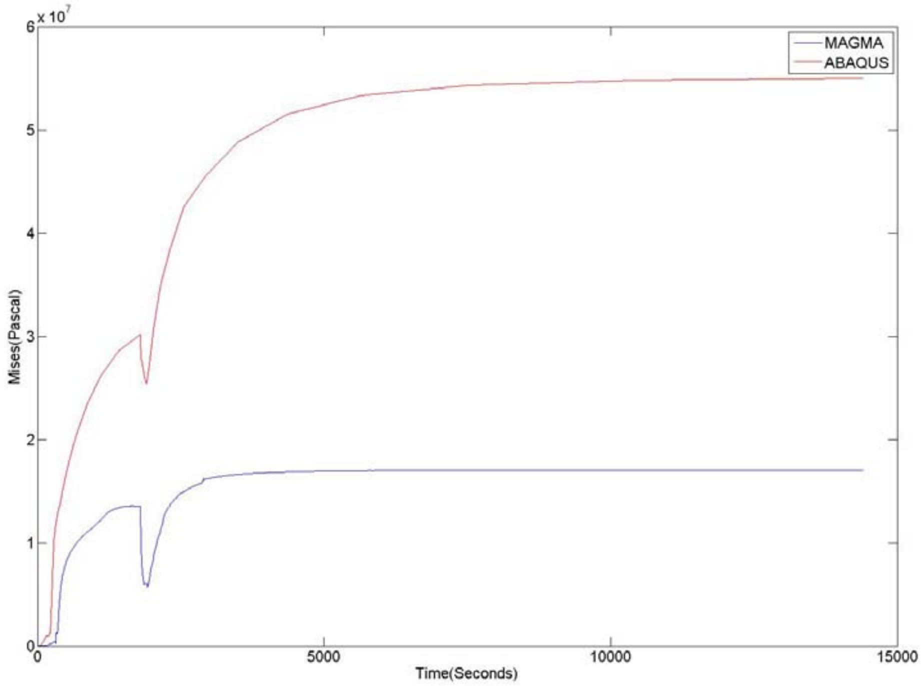


Figure 6.12. Von Mises curves from PNT5 of the Optimized Hub

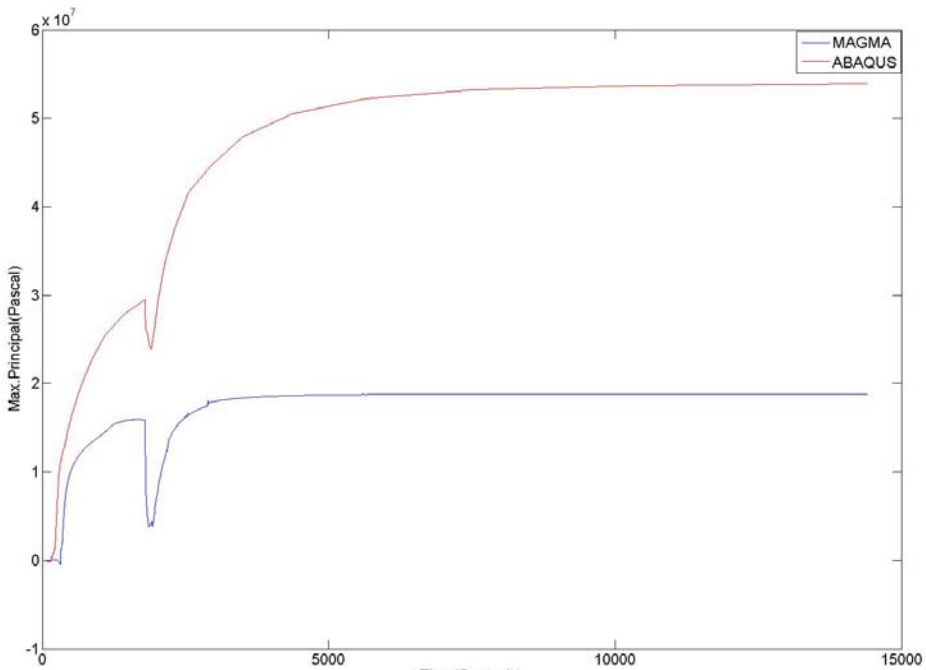


Figure 6.13. Maximum Principal stresses from PNT5 of the Optimized Hub

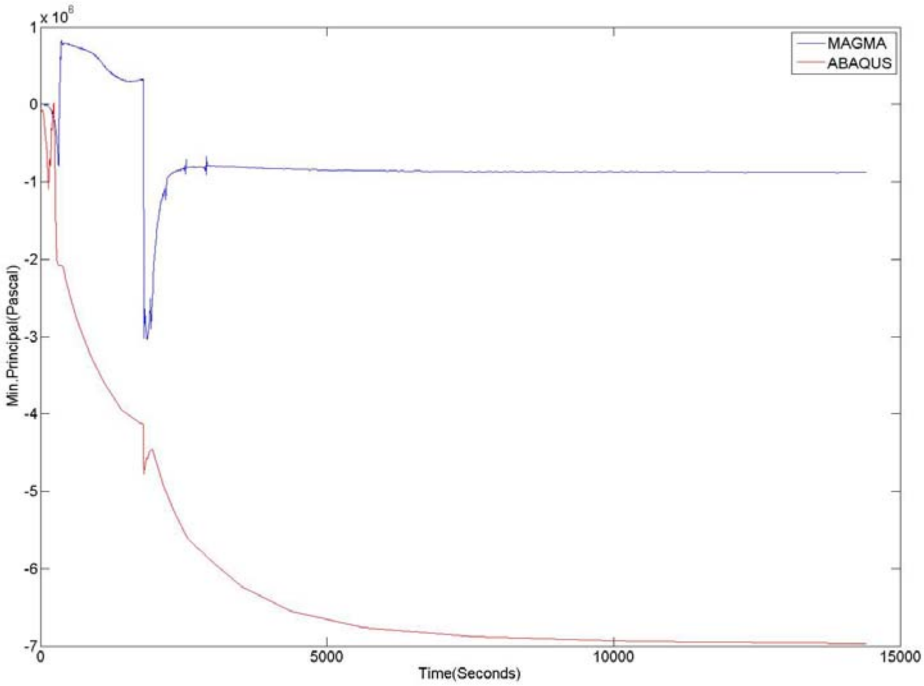
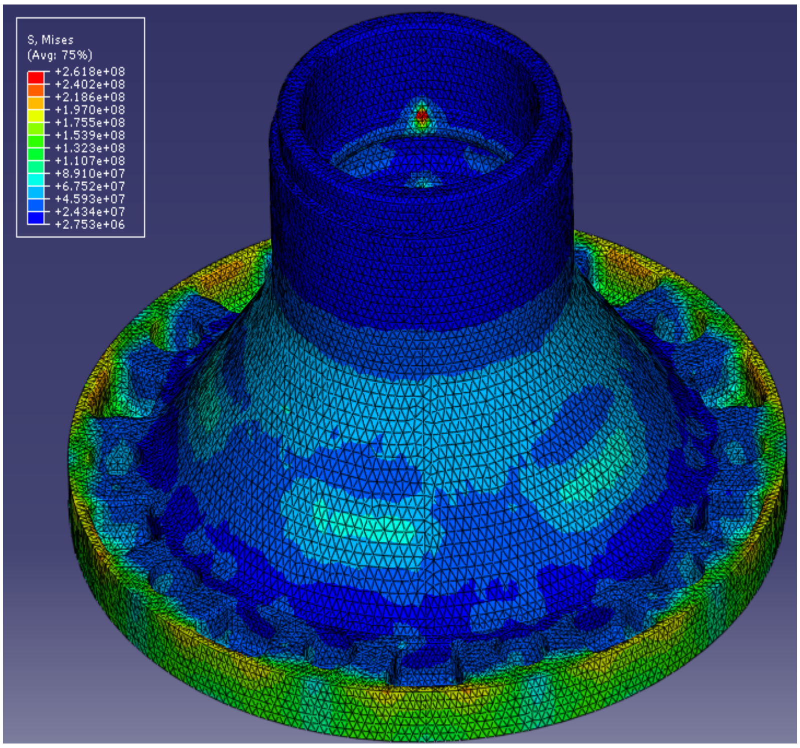


Figure 6.14. Minimum Principal stresses from PNT5 of the Optimized Hub

6.8 Stress color spectrums



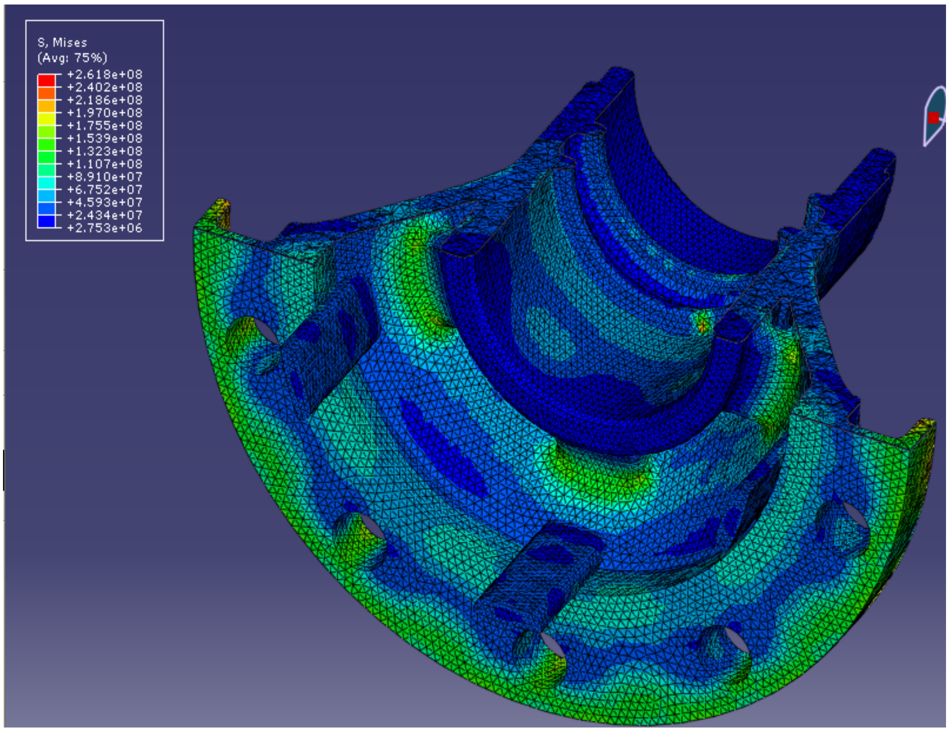
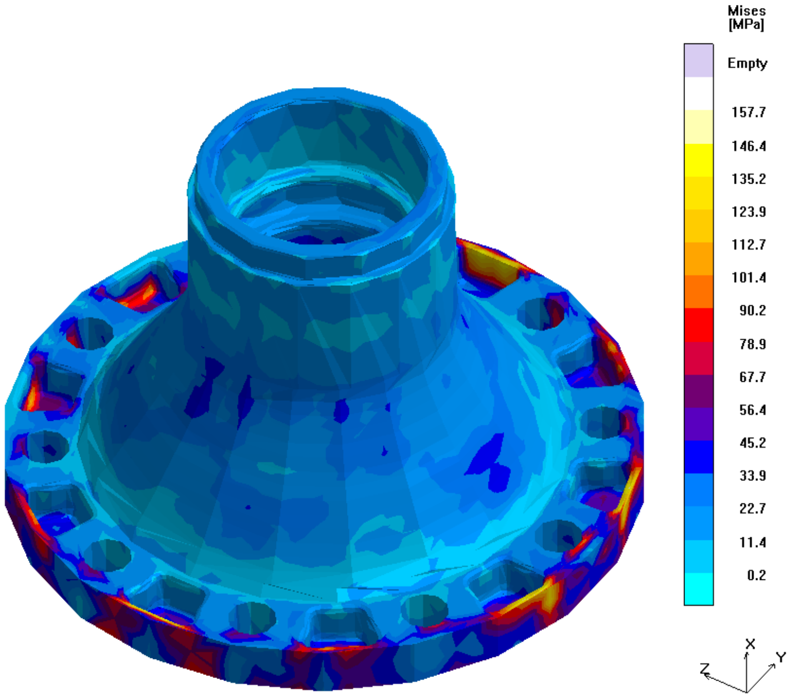


Figure 6.15. Abaqus color spectrums for the Mises results of the Optimized Hub.



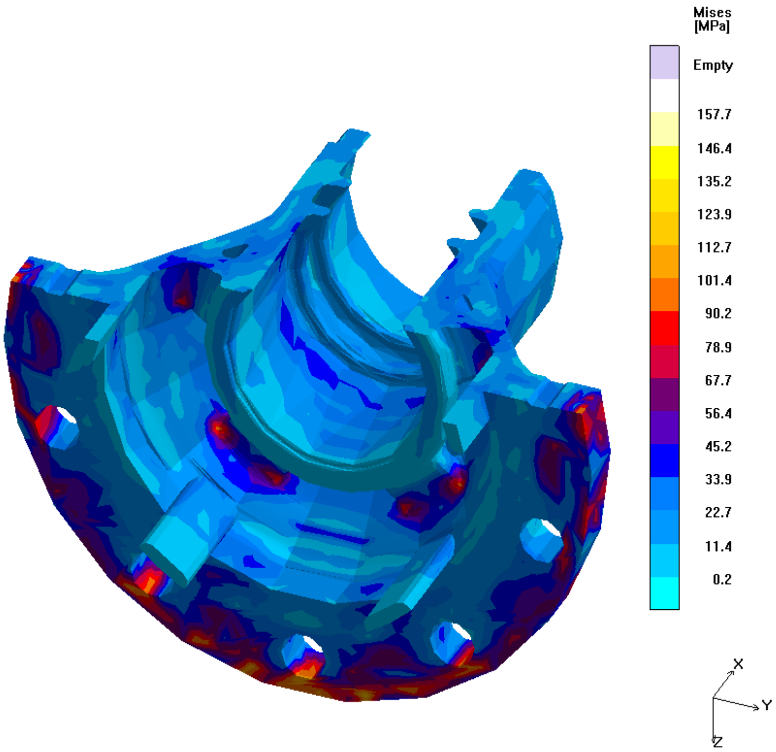
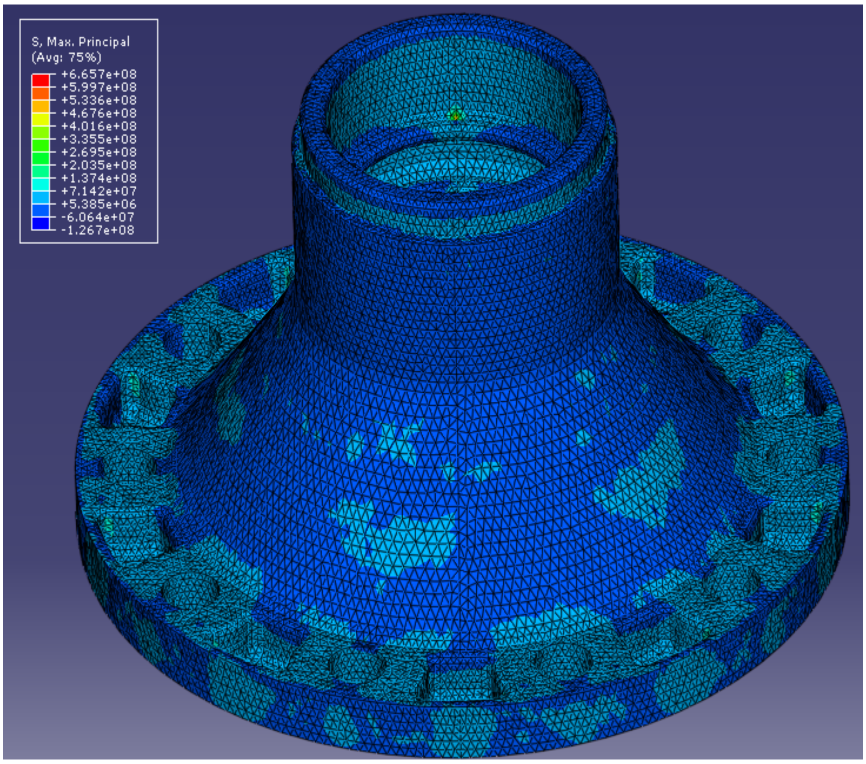


Figure 6.16. Magmasoft color spectrums for the Mises results of the Optimized Hub.



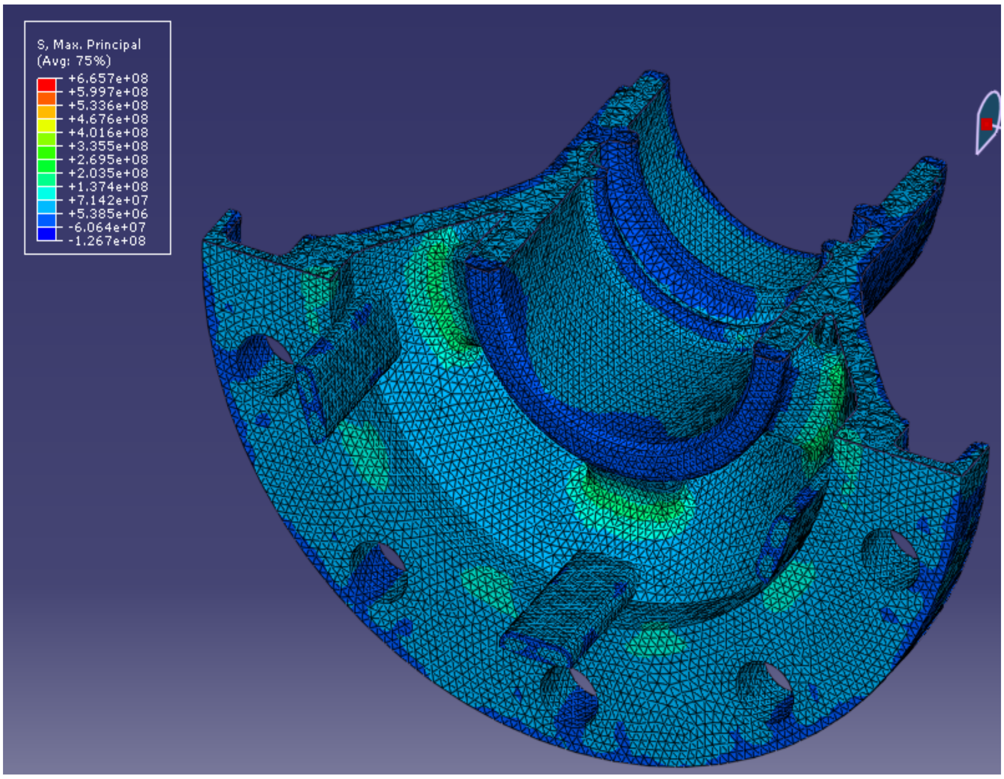


Figure 6.17. Abaqus color spectrums for the residual Max. Principal stresses of the Optimized Hub.

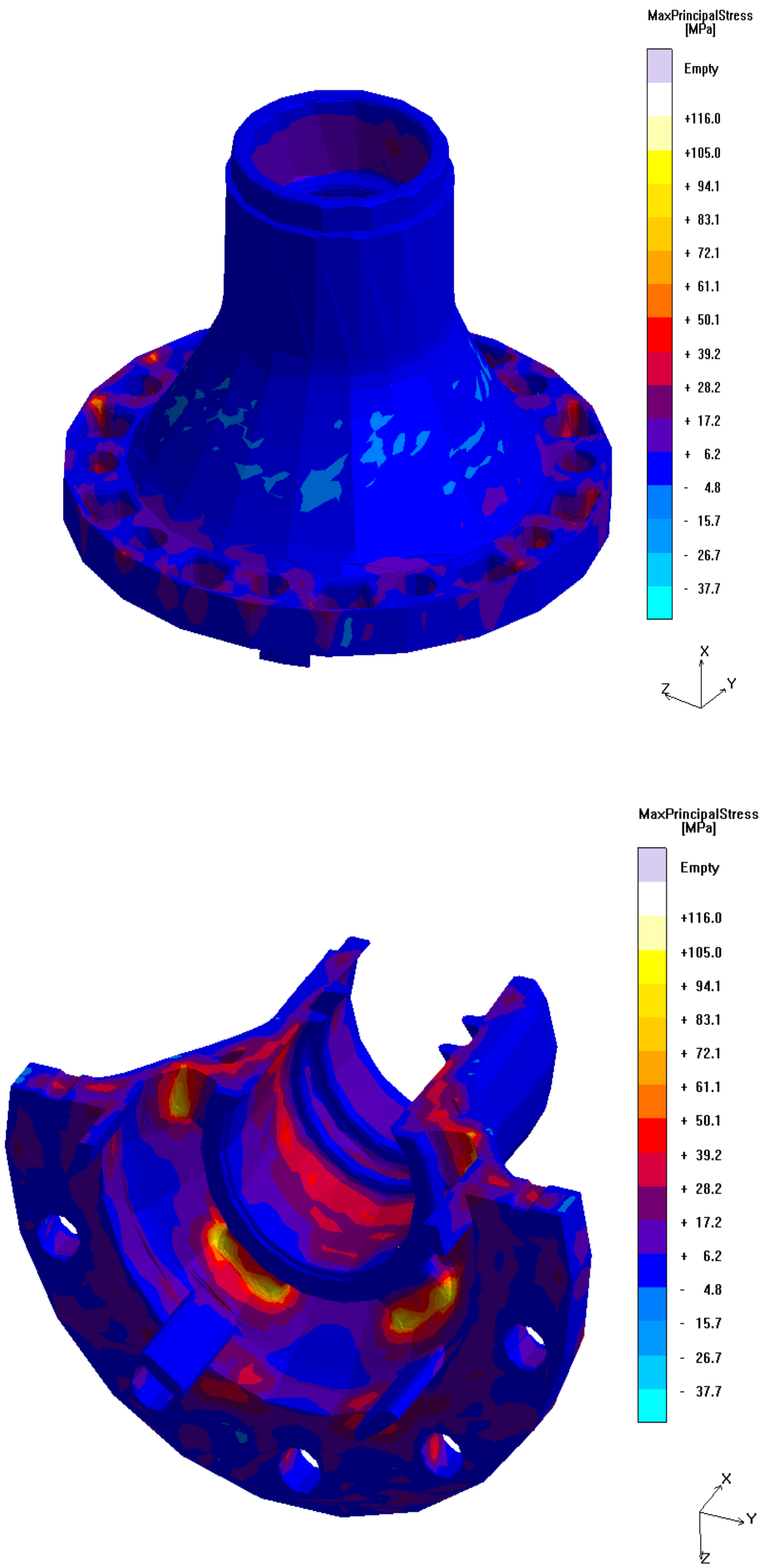
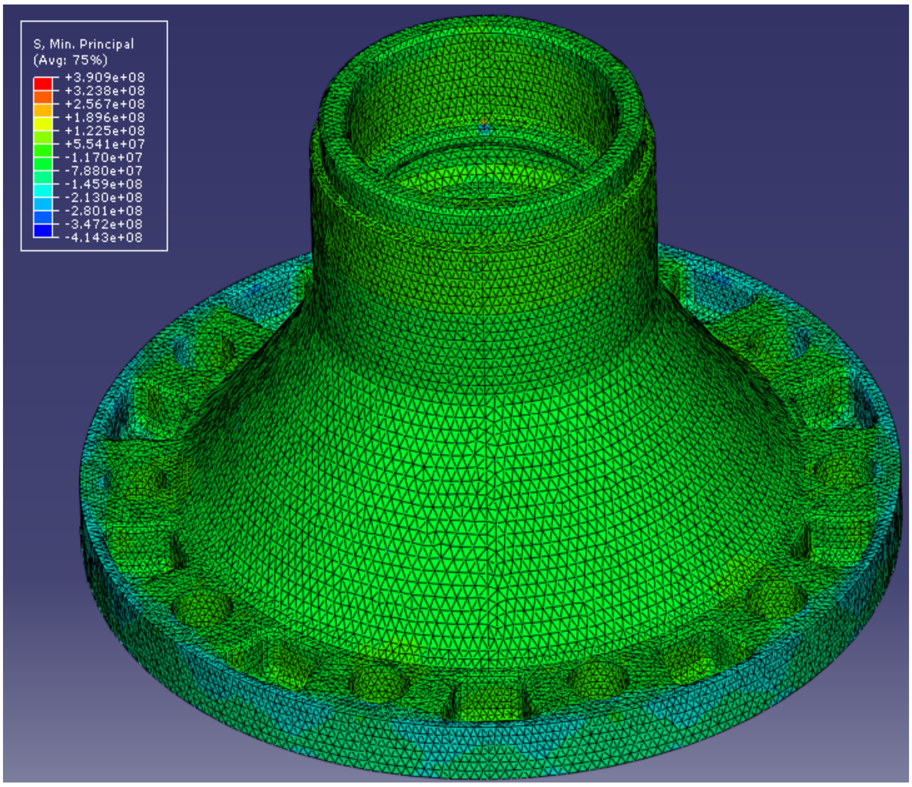


Figure 6.18. Magmasoft color spectrums for the residual Max. Principal stresses of the Optimized Hub.



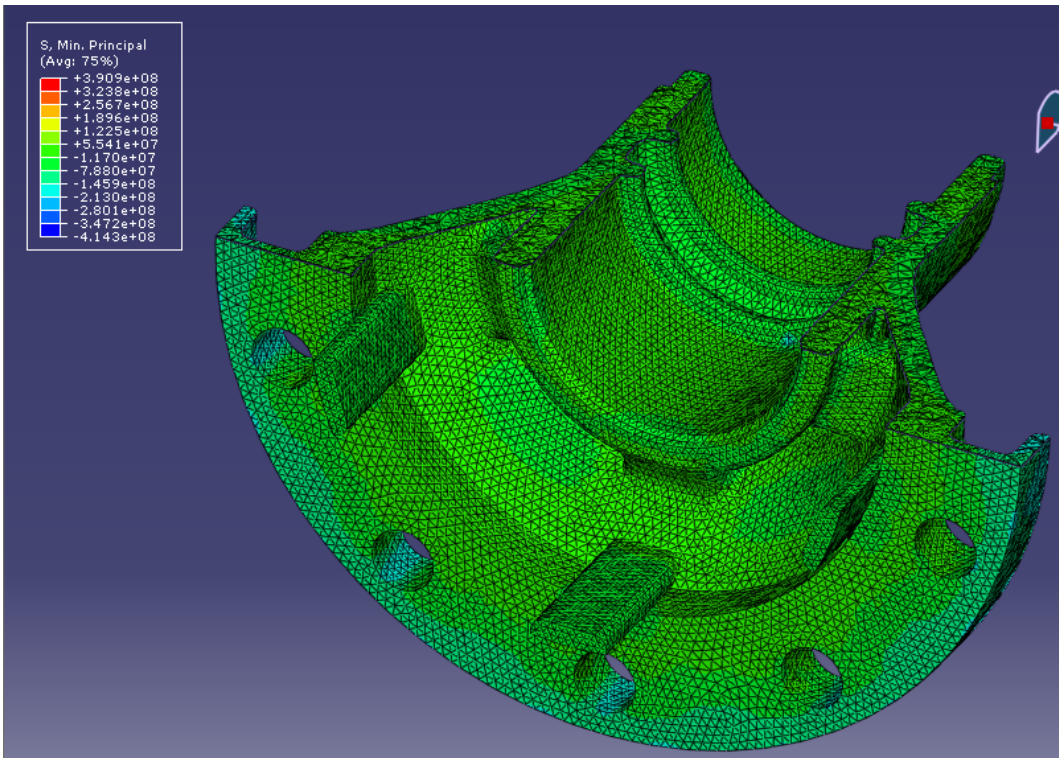


Figure 6.19. Abaqus color spectrums for the residual Min. Principal stresses of the Optimized Hub.

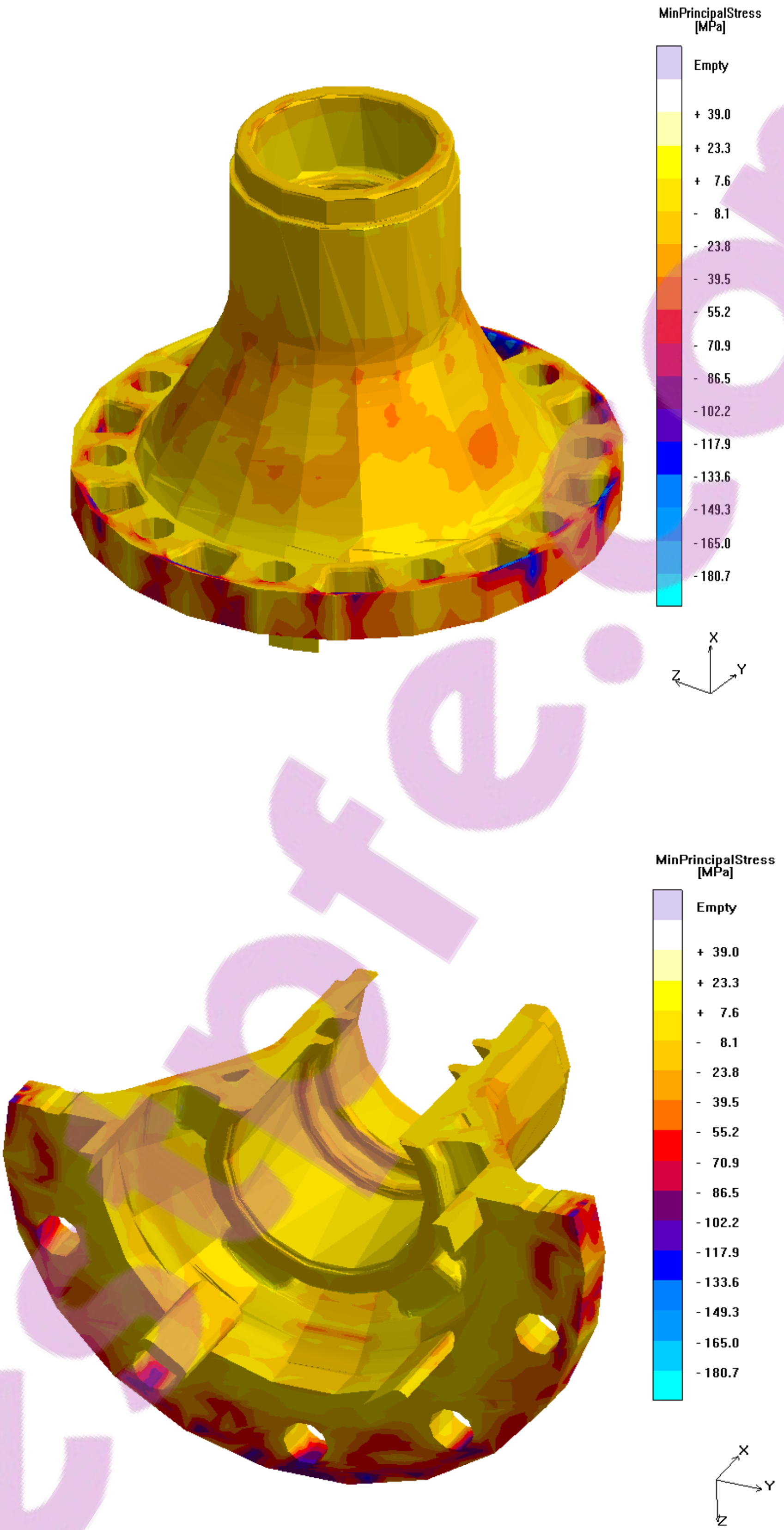


Figure 6.20. Magmasoft color spectrums for the residual Min. Principal stresses of the Optimized Hub.

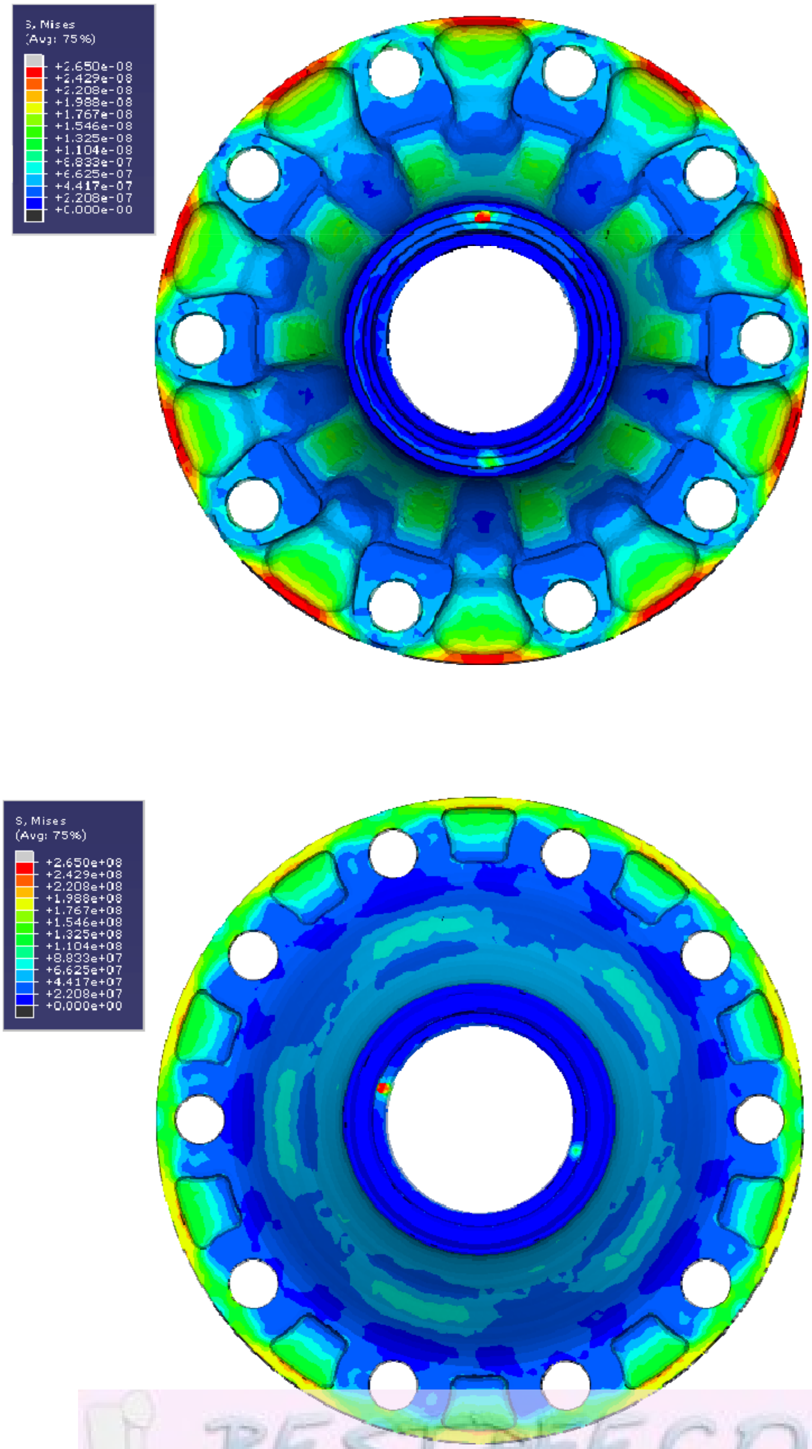
Bearfe.com

6.9 Simulation time of the Optimized Hub

	Thermal	Stress	Total
Abaqus	99hrs. 32min.	20hrs. 25min.	119hrs. 57min.
Magmasoft			16hr. 00min.

7 Original and Optimized Hub Comparison

7.1 Mises



BESTPFE.COM
List of research project topics and materials

Figure 7.1. Original Hub (top) and Optimized Hub (bottom) Mises comparison. Top view.

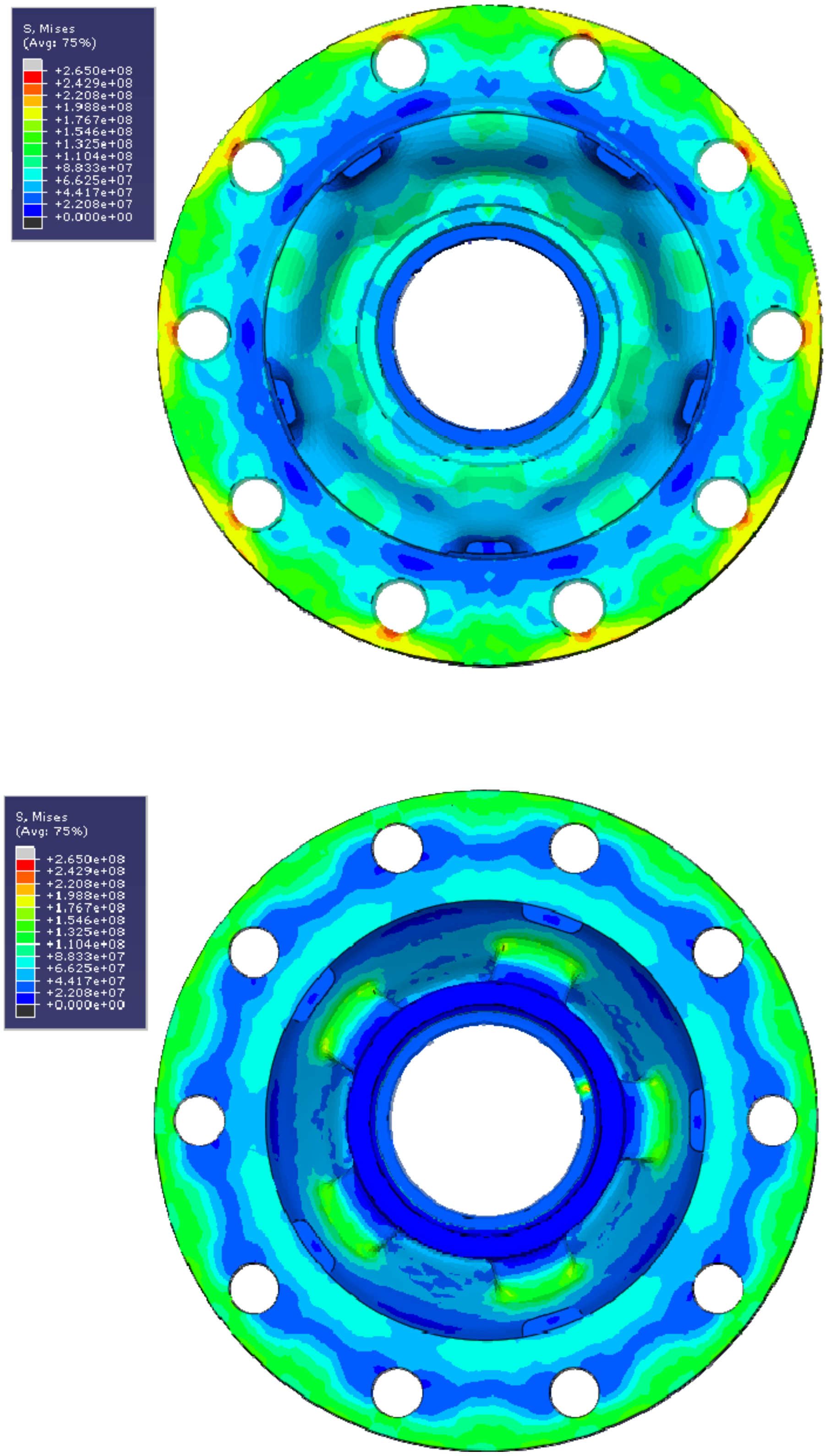


Figure 7.2. Original Hub (top) and Optimized Hub (bottom) Mises comparison. Bottom view.

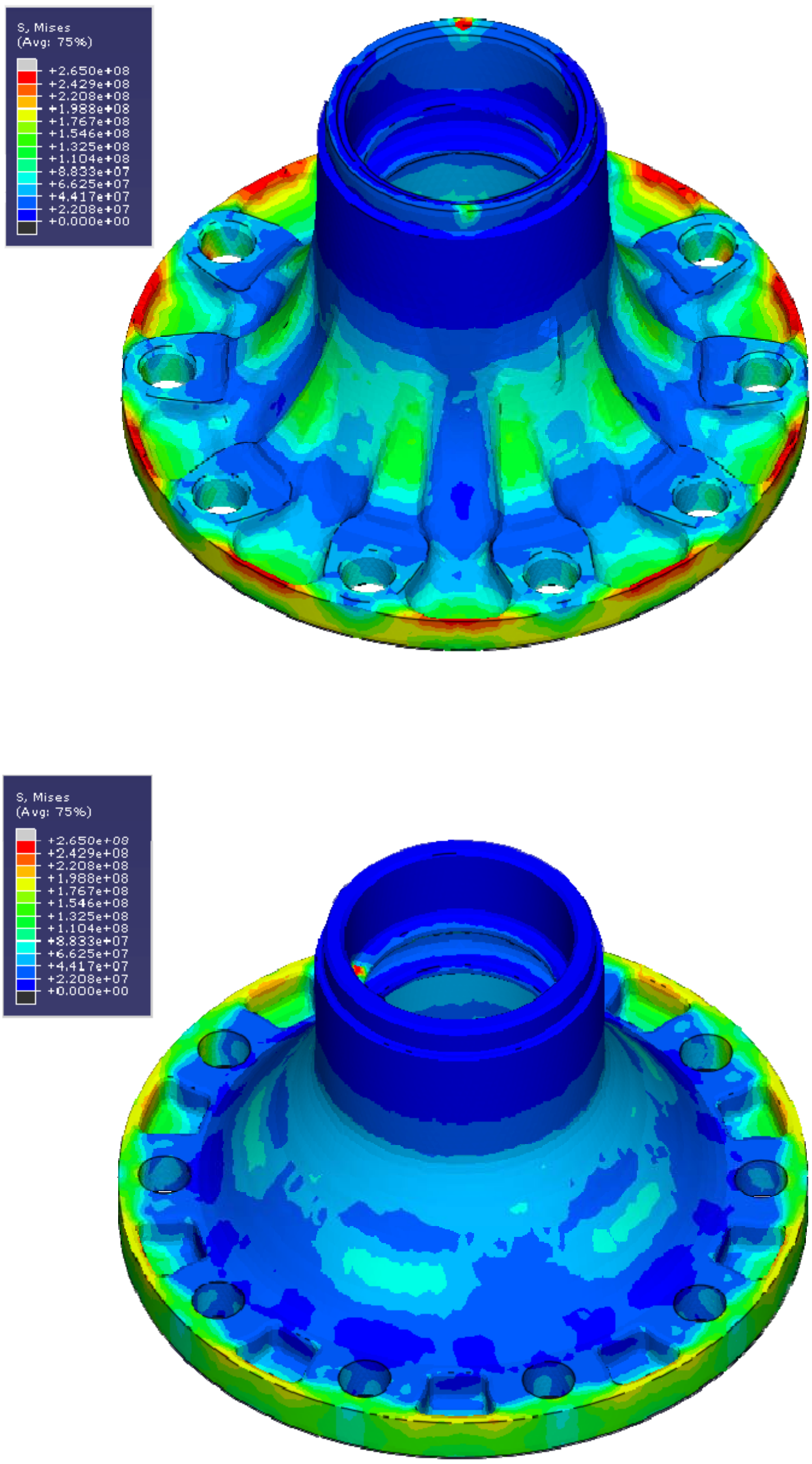


Figure 7.3. Original Hub (top) and Optimized Hub (bottom) Mises comparison. Inclined view.

7.2 Maximum Principal Stress

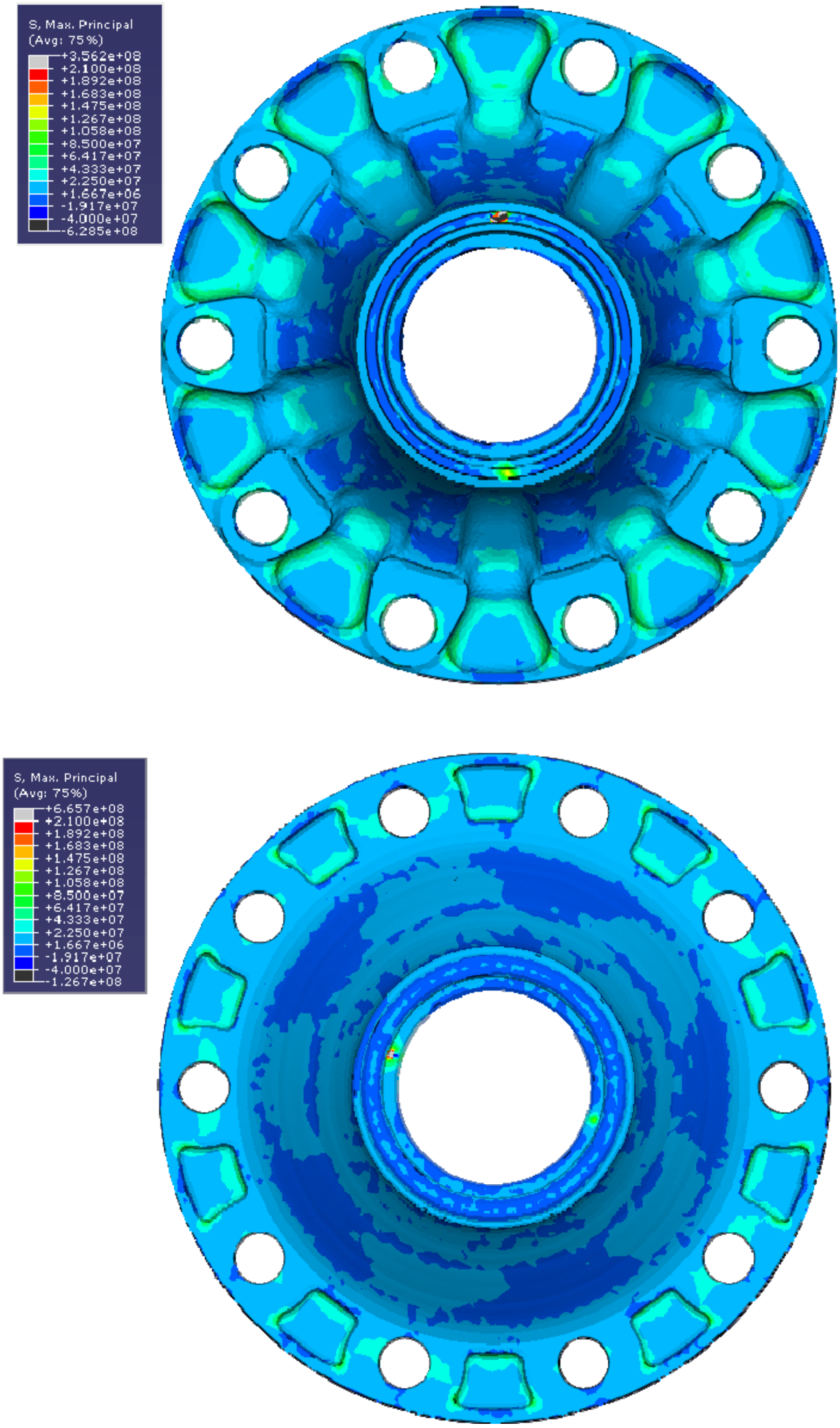


Figure 7.4. Original Hub (top) and Optimized Hub (bottom) Maximum Principal Stress comparison. Top view.

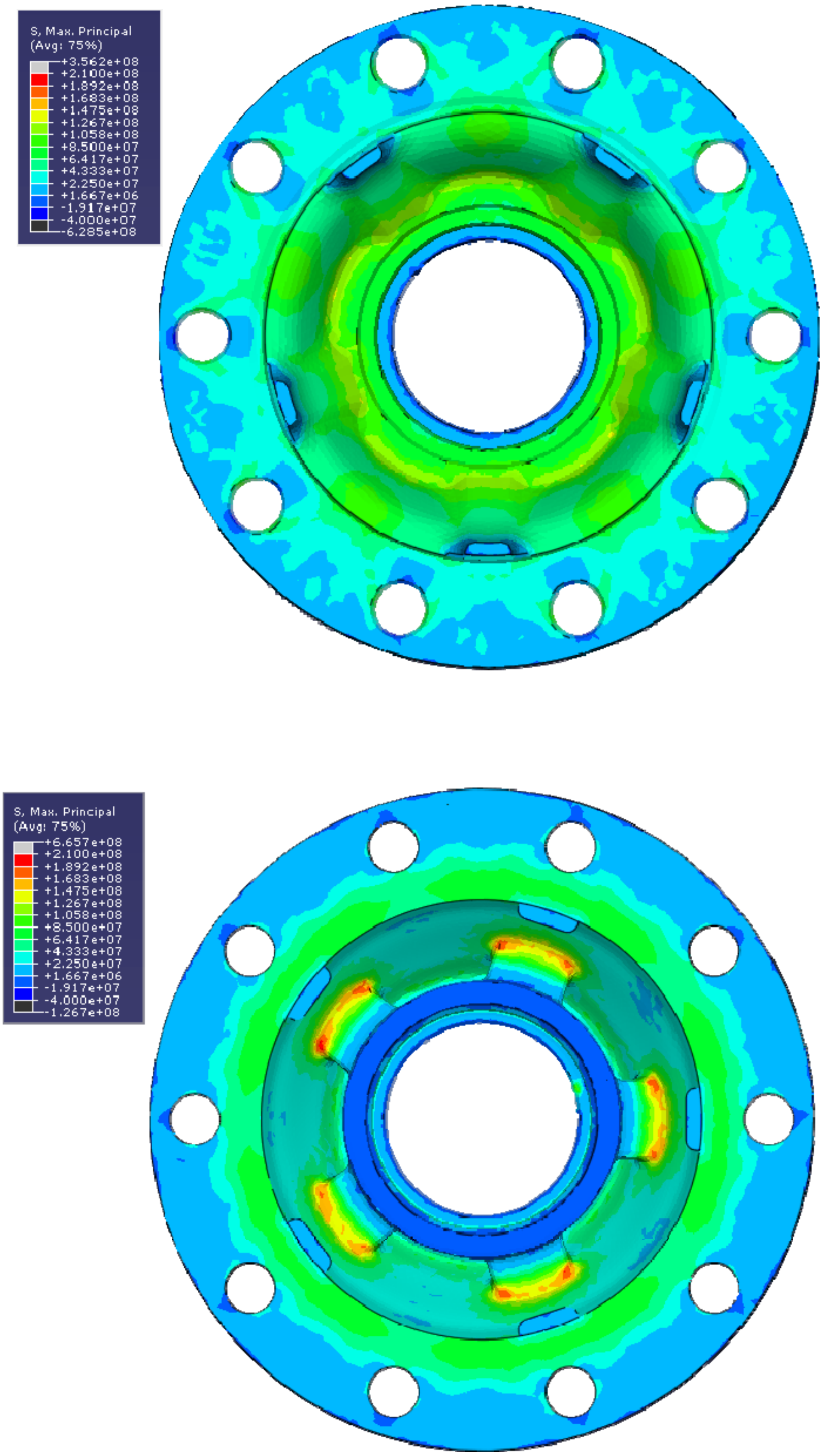


Figure 7.5. Original Hub (top) and Optimized Hub (bottom) Maximum Principal Stress comparison. Bottom view.

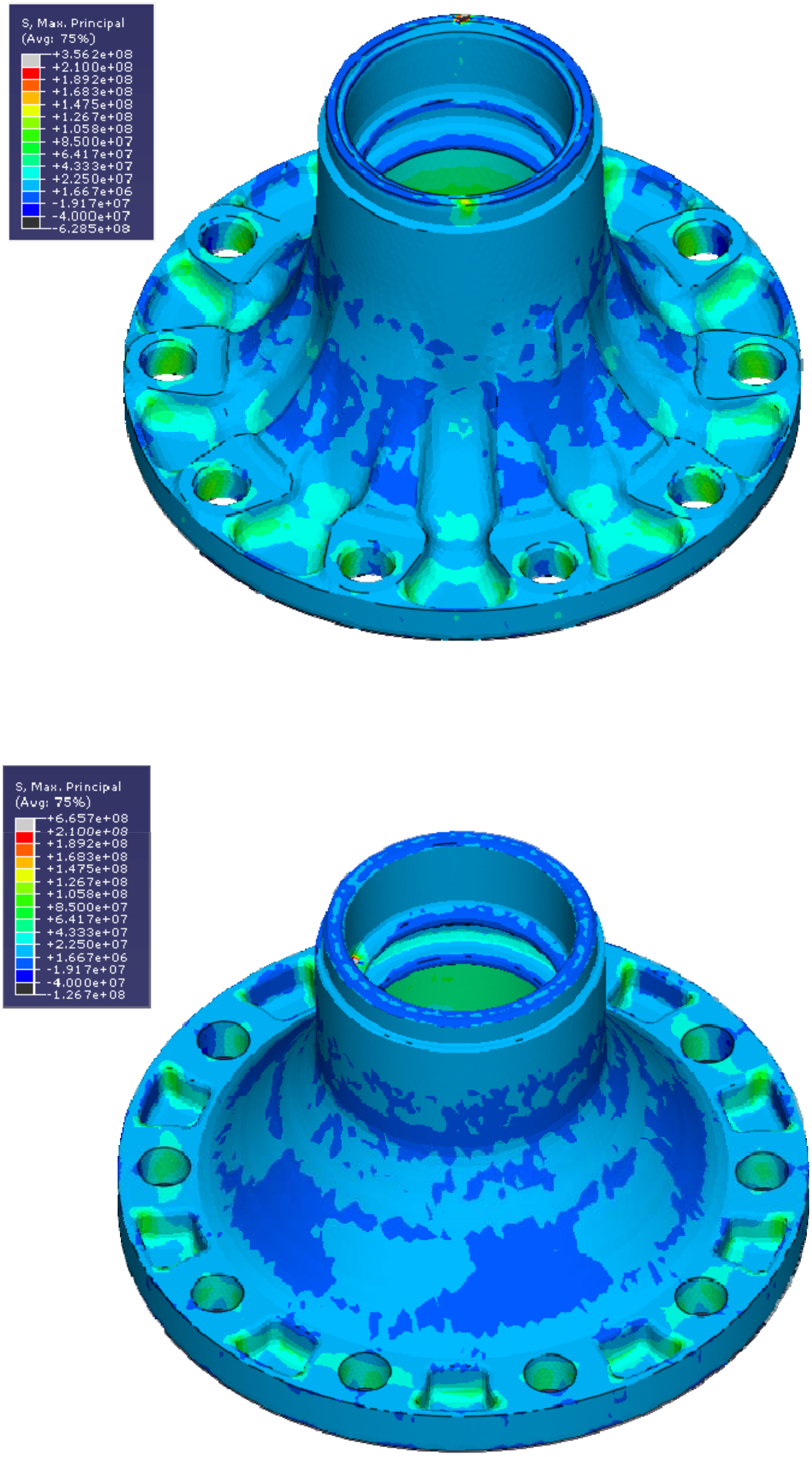


Figure 7.6. Original Hub (top) and Optimized Hub (bottom) Maximum Principal Stress comparison. Inclined view

7.3 Minimum Principal Stress

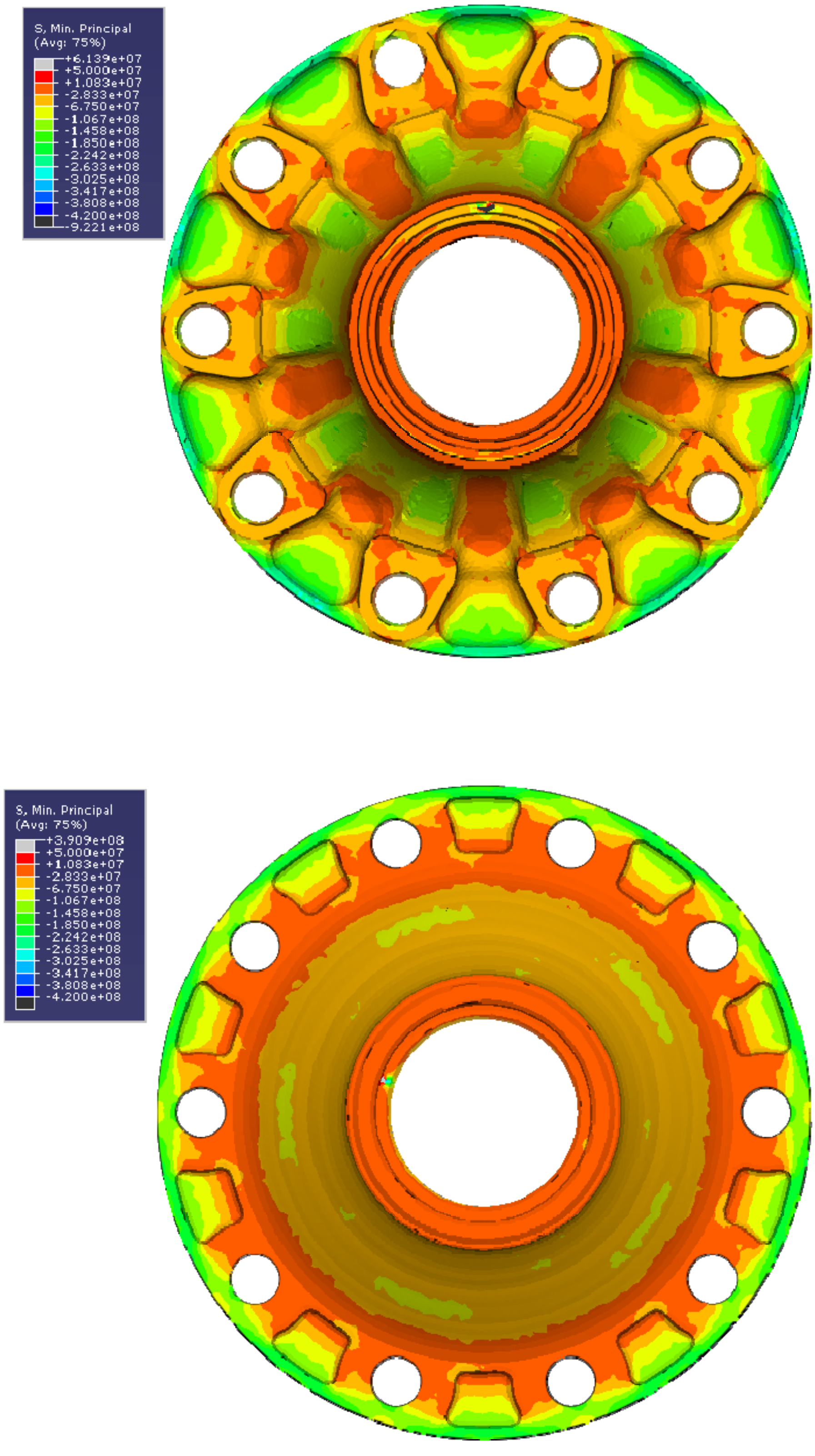


Figure 7.7. Original Hub (top) and Optimized Hub (bottom) Minimum Principal Stress comparison. Top view.

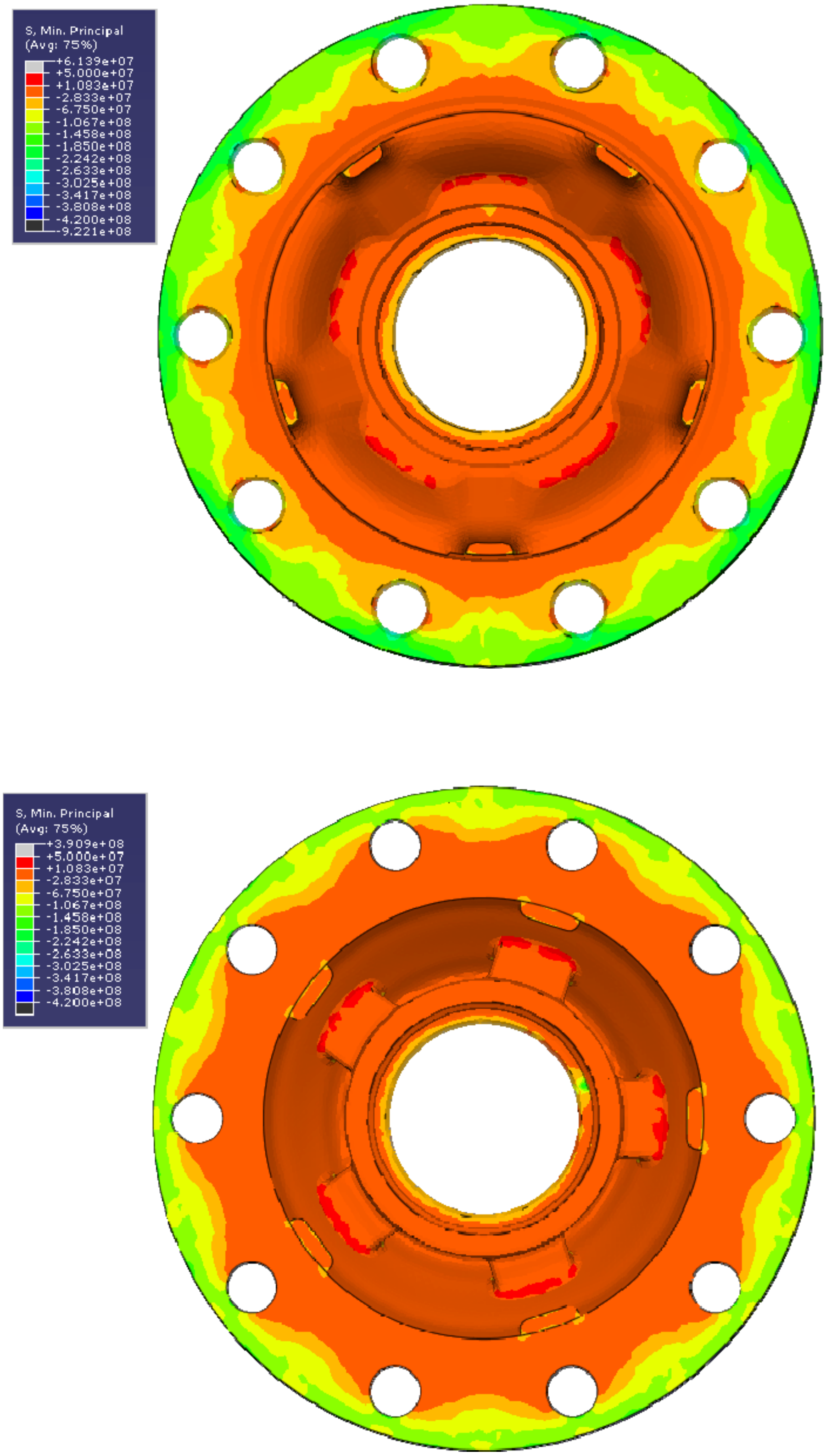


Figure 7.8. Original Hub (top) and Optimized Hub (bottom) Minimum Principal Stress comparison. Bottom view

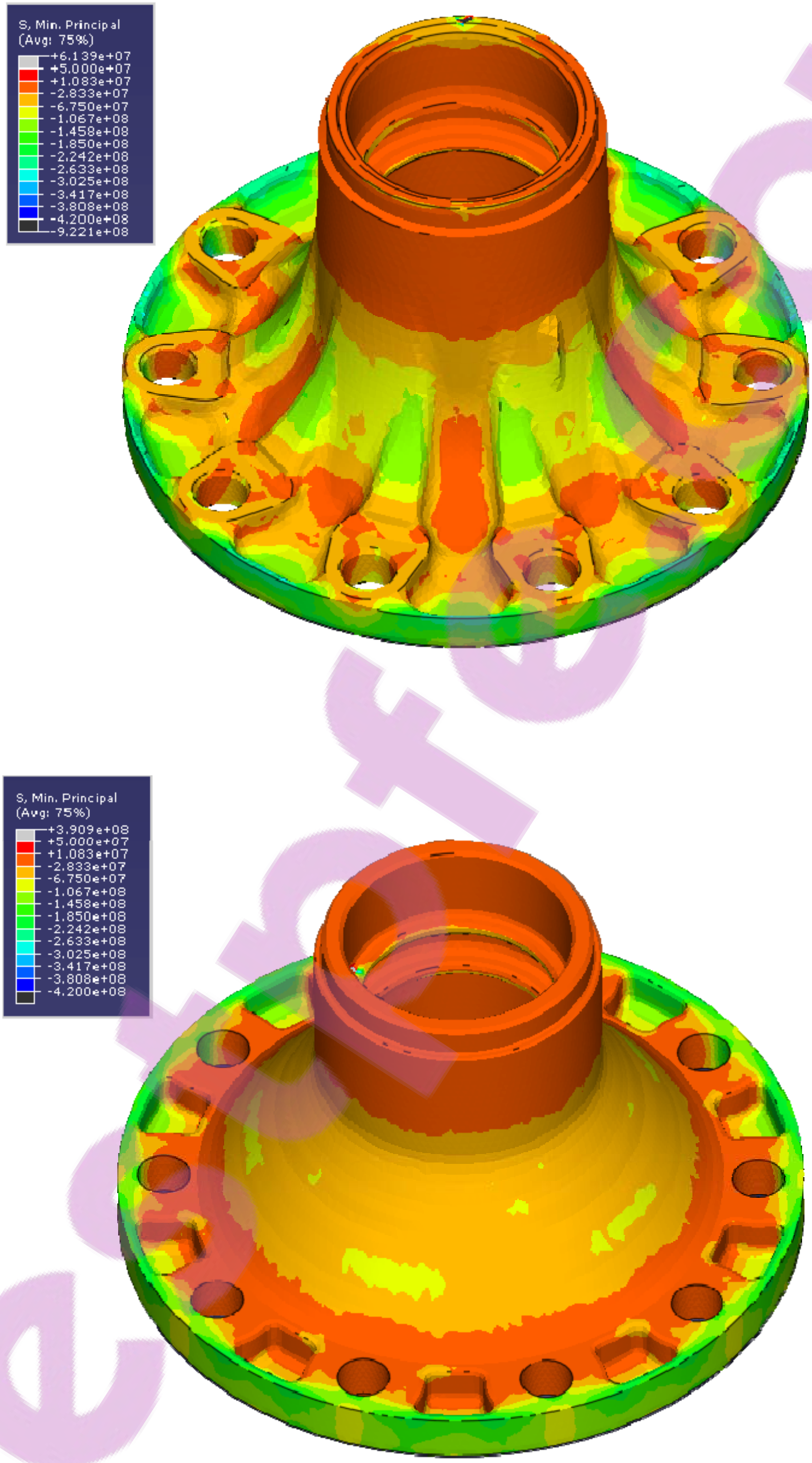


Figure 7.9. Original Hub (top) and Optimized Hub (bottom) Minimum Principal Stress comparison. Inclined view

8 Conclusions and discussions

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.

9 References

Abaqus (2007),
Abaqus/CAE User's Manual version 6.7-1
ABAQUS, Inc, Providence, RI, USA

Abaqus (2007),
Analysis user's manual version 6.7-1
ABAQUS, Inc, Providence, RI, USA

Becker, A. A. (2004),
An introductory guide to finite element analysis
London: Professional Engineering, ISBN 1860584101

Chandra U., Ahmed A. (2002),
Modelling for casting and solidification processing,
Marcel Dekker, New York

Gustafsson E., Stromberg N. (2006),
Optimization of Casting by using response surface methodology
SweCast AB, Jonkoping, Sweden

Gustafsson E., Hofwing M., Stromberg N. (2007),
Simulation and Measurement of Residual Stresses in a Stress Lattice
SweCast AB & Department of Mechanical Engineering, University of Jonkoping

Hyperworks (2006),
Hypermesh User's guide, from Hyperworks 8.0SR1
Altair Engineering Inc., Troy, MI, USA

Jesper Hattel (2005),
Fundamentals of Numerical Modeling of Casting Processes
Polyteknisk Forlag, Kgs. Lyngby, ISBN 8750209698

MAGMA Online Help documentation,
MAGMASOFT version 4.4,
MAGMA GmbH, Aachen, Germany

Magmasoft (2000),
MAGMASOFT version 4.4, MAGMAstress Module manual,
MAGMA GmbH, Aachen, Germany

MATLAB (2006)
MATLAB Help documentation version R2006a
The Mathworks, Inc. USA

BESTPFE.COM
List of research project topics and materials

Simo J.C., Hughes T.J.R. (1998),
Computational Inelasticity,
Springer, New York

Wikipedia, free encyclopedia (2008) http://en.wikipedia.org (Acc.21/01/2008)

10 Appendix

10.1 Abaqus Implementation

The same general steps presented in *Figure 3.1* will now be followed for the Abaqus implementation of our residual stress analysis. For reference purposes, the figure is presented again:

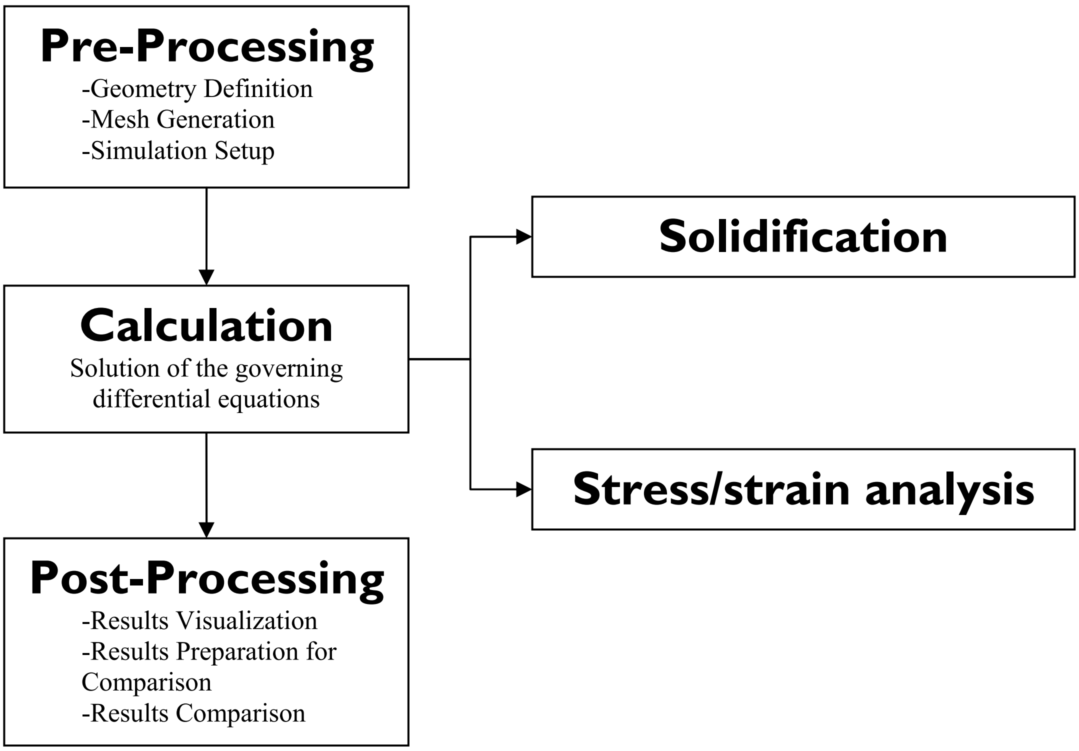


Figure 10.1. Steps sequence for the residual stress analysis

10.1.1 Pre-Processing

10.1.1.1 Geometry Definition

A cylinder of length=0.4m diameter=0.25m and a 0.02m radius at each end will be used for the detailed description of the entire process, see *Figure 10.2*:

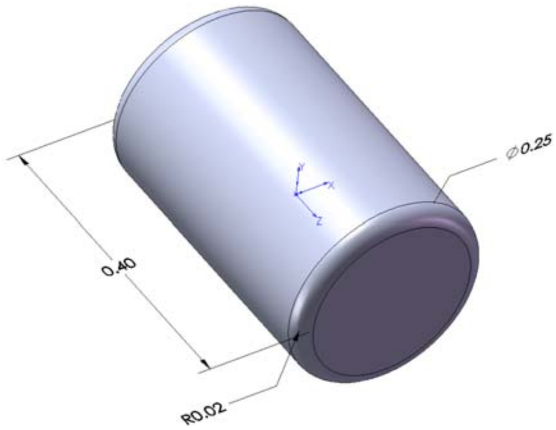


Figure 10.2. The Cylinder part. Units: Meters

The mold for the cylinder is simply a box with the cylinder cavity in the center, its general dimensions in meters are 0.65 x 0.65 x 0.80, see *Figure 10.3*:

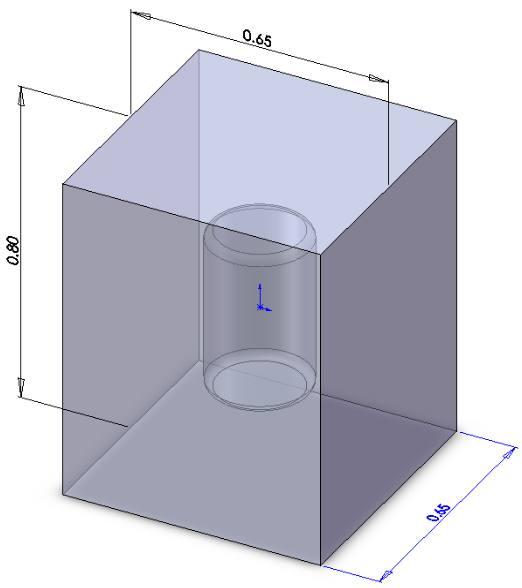


Figure 10.3. The Cylinder Mold part. Units: Meters

Both Abaqus and Magmasoft have CAD capabilities for us to create the cylinder and its mold, but instead, we decide to use the specialized CAD software ProEngineer Wildfire 3 to create and edit these geometries, not only because we receive the other models (the Hub and the optimized Hub) as ProEngineer .part files but also because is a good practice to use the best characteristics of every software at hand if the effectiveness is not compromised.

The cylinder is a solid part created by a simple revolve feature.

The mold general shape is an extrusion of a rectangular section. To create the cavity of the mold, the cylinder is removed form the mold with a Boolean operation. In ProEngineer is as follow:

> Note: For simplicity, create the cylinder and the mold in the same relative position with respect to their coordinate systems so if you make the coordinate systems coincide when assembling the parts they will get in the correct position without needing any further adjustment.

With the mold part active go to Insert → Shared Data → Merge/Inheritance → Select the "Remove material" button → Select the "Open a model which geometry will be copied" button → Open the cylinder → Select its coordinate system, then the mold coordinate system → Press the  button.

Now the cylinder and the mold are ready to be exported.

For the implementation of the analysis in Abaqus, the meshes are created in Hypermesh. In this case, the Mold will be exported as a step (.stp) file. Just the surfaces and the coordinate system must be saved to the .stl. In Hypermesh, the mesh for the Cylinder will be obtained from the Mold one so is not necessary to export the Cylinder.

Exporting the mold as a step file:

File menu → Save a Copy → Type: Step → Name it "Mold_Surface" → OK → Check the box for the Surfaces → Ensure the other check boxes are unchecked → Press the  button → Select the Mold coordinate system → OK

For the implementation in Magmasoft, the mold is not necessary as a external CAD file so just the Cylinder will be exported. This time, is an STL (.stl) file:

File menu → Save a Copy → Type: STL → Name it "Magma_Cylinder" → OK → Press the  button → Select the Mold coordinate system → Set the rest of the window as *Figure 10.4*.

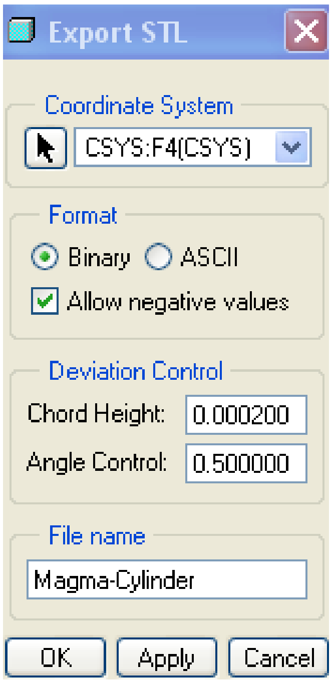


Figure 10.4. STL export window, ProEngineer.

10.1.1.2 Mesh Generation

The mesh of the parts is created in Hypermesh 8.0 and exported as an Abaqus .inp file containing just nodes and volume elements.

The mold surface is imported and meshed in Hypermesh as a surface mesh. To produce matching nodes in the interface between the casting and the mold, the surface mesh for the mold is duplicated to another file. In that duplicate file, the external surfaces of the mold are deleted remaining just the surface of the cavity, already meshed, which will be used as the surface of the cylinder. From each file a volume mesh is generated and saved to an .inp file.

Note: This version of Hypermesh doesn't have the "undo" option, so if you may need to go back to certain point in your work progress, you can create a save as of the model in the desired point.

Start Hypermesh and select the Abaqus User profile as the profile option.

Start Hypermesh → Preferences menu → User Profiles → Application: Hypermesh → Check the Abaqus option → OK

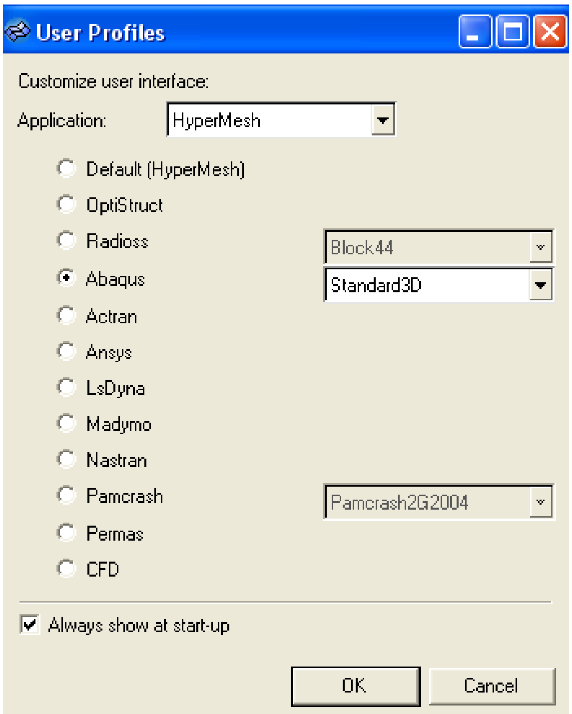


Figure 10.5. User Profiles window

10.1.1.2.1 Importing the Geometry

File menu → Import → Geometry → Step → Browse the "Mold_Surface.stl" part → Open

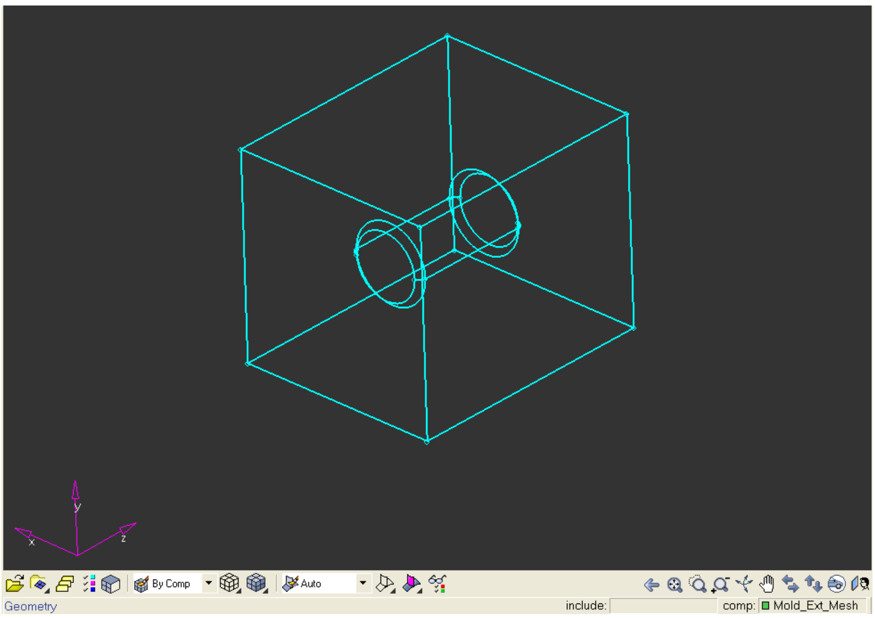


Figure 10.6. Wireframe appearance of the .stl geometry of the mold in Hypermesh

Save the model as “Mold_Surf_Mesh.hm”:

File menu → Save → File name: “Mold_Surf_Mesh.hm” → Save

10.1.1.2.2 Geometry Cleanup

They may be small gaps between surfaces. These gaps turn the geometry into a not closed volume representing a problem to create the volume mesh that we are aiming for. To repair this gaps we used the “Auto Cleanup” option:

Geometry menu → Auto Cleanup → Click surfs → Select “All” → Set the target element size (0.001)* → autocleanup → return

*This value is modified with the edit parameters... button. For details on the effect of those parameter see Hyperworks (2006).

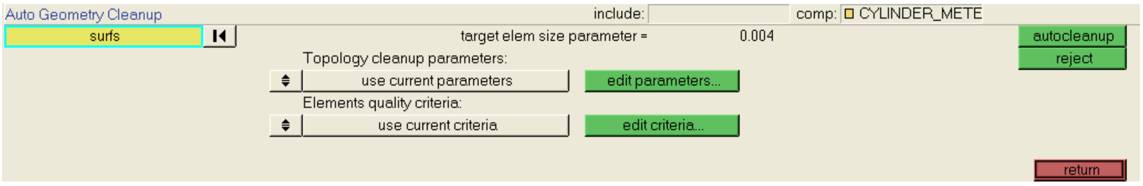


Figure 10.7. General appearance of the Auto Cleanup panel

10.1.1.2.3 Organizing the model

Hypermesh group the entities of the models into “collectors”, this collectors allow us to handle the collected data as a unit. Is convenient to separate the surfaces to be meshed into logical groups (e.g. external surfaces and cavity surfaces) because it does not only simplify the visualization of the model (since collectors can be hidden or displayed) but also the selection of large groups of surfaces as usually casted industrial parts have.

All entities must belong to a collector. If we have not created one before the import of the geometry, Hypermesh create it automatically, which means that all our surfaces get automatically grouped in one collector when we first import the part. This collector gets the name of the imported file without the extension which is replaced “.prt” if is not originally a .prt file.

In the case of this simple model we will keep the surfaces of the mold in one collector and will create two more collectors to separate the mesh of the external surfaces and the mesh of the cavity surfaces.

Create a collector as follow:

Organize menu → Collectors → Check the "create" option → Click the arrow for the collector type and select "Component" → Name it "Ext_Mesh" → Assign a color if desired → Create

In the same way, create a collector named "Cavity_Mesh" and another one called "Cavity_Surface". The existent collectors can be seen inside the Components item in the Model Browser tab. If the Model Browser is not displayed go to the View menu and select Model Browser.

Rename the firstly created collector (the one with the name of the .step file from ProEngineer) as "Ext_Surfaces"

Right click the collector named "**MOLD_SURFACE.PRT**" → Rename → Name it "Ext_Surfaces" → Return

The collectors list should look as

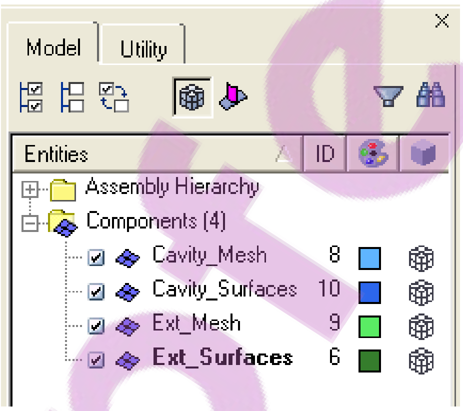


Figure 10.8. Collectors of the Mold_Surf_Mesh.hm model in Hypermesh

To move an entity into a collector we proceed as follow:

Organize menu → Entities (or Shift+F11) → Click the  button and specify the type of entity to select from the list → Select the desired entity or entities → Click the 

button and select the collector to which you want to assign the entity → Move

Accordingly, move the corresponding entities to each one of the collectors previously created.

The collectors can be hidden or displayed using the "Display" panel which can be accessed from the View menu (Shortcut: d key):

View menu → Display → Collectors

For details on how to use the Display panel, refer to Hyperworks (2006).

Bestpfe.com

Notice that for the imported .step geometry of our mold, Hypermesh do not distinguish between the external surfaces and the surfaces of the cavity even if they are isolated one from the other. Instead it considers them as one single surface, which represents an obstacle to separate the external from the cavity ones into different collectors.

To go around this problem, the surfaces are copied (not moved) into another collector; then the fist collector is hidden, next, the surfaces that are not wanted to be in the new collector (e.g. the external ones) are deleted. Finally, the opposite is done in the first collector, so the new collector is hidden and the first collector is displayed; there the surfaces to be deleted are the equivalent ones to those left to remain in the new collector (e.g. the cavity ones).

10.1.1.2.4 Meshing the surfaces

By default, Hypermesh try to adapt the sizes of the elements in the surface being meshed, to that of the surrounding meshes in the proximity of the union between the surfaces. Taking that into account, a surface needed to keep a constant element size is better to be meshed first, because the mesh that adapt to the surrounding ones don't keep a constant element size (unless you assign the same size to the elements of all surfaces involved).

The surface meshes were created using the Automesh function (shortcut: F12 key). The external surface of our mold was meshed with the following procedure:

> Right click the "Ext_Mesh" collector in the Model Browser → Make Current → Mesh menu → Automesh → Select "surfs" (surfaces) as the entity type → Select the surfaces to mesh* → set the rest of the window as in *Figure 10.9* → Mesh → Return

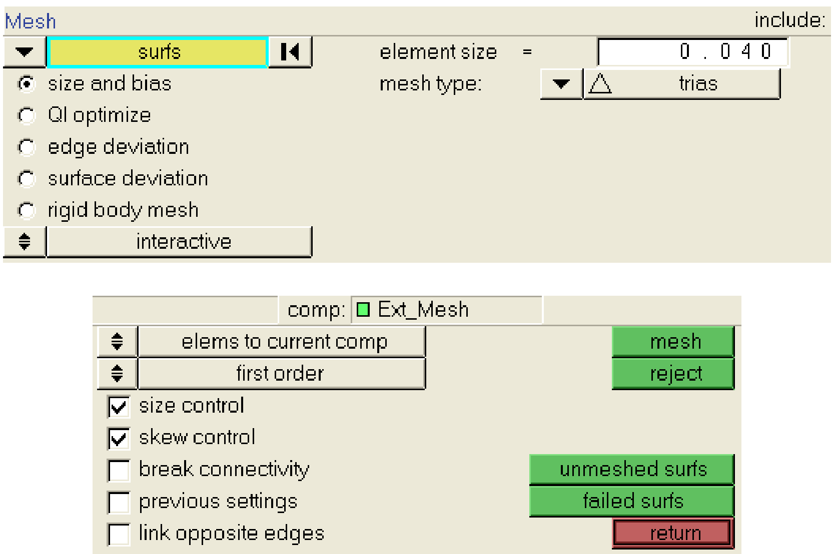


Figure 10.9. Automesh panel (is divided in two for display purposes).

**By clicking the* surfs *button a list of selection criterias is displayed. To take advantage of the collector choose "by collector", then check the check box of the "Ext_Surfaces" collector and press the* select *button.*

Similarly, mesh the surfaces in the "Cavity_Surfaces" collector with an element size of 0.01. Remember to "Make Current" the "Cavity Mesh" collector and to select elems to current comp in the Automesh panel so the elements get automatically collected in the appropriate collector.

The surface mesh for the mold is now ready.

Save the file:

File menu → Save

Note: Hypermesh do not recognize CTRL+S so use the Save option in the File menu.

The model should look as:

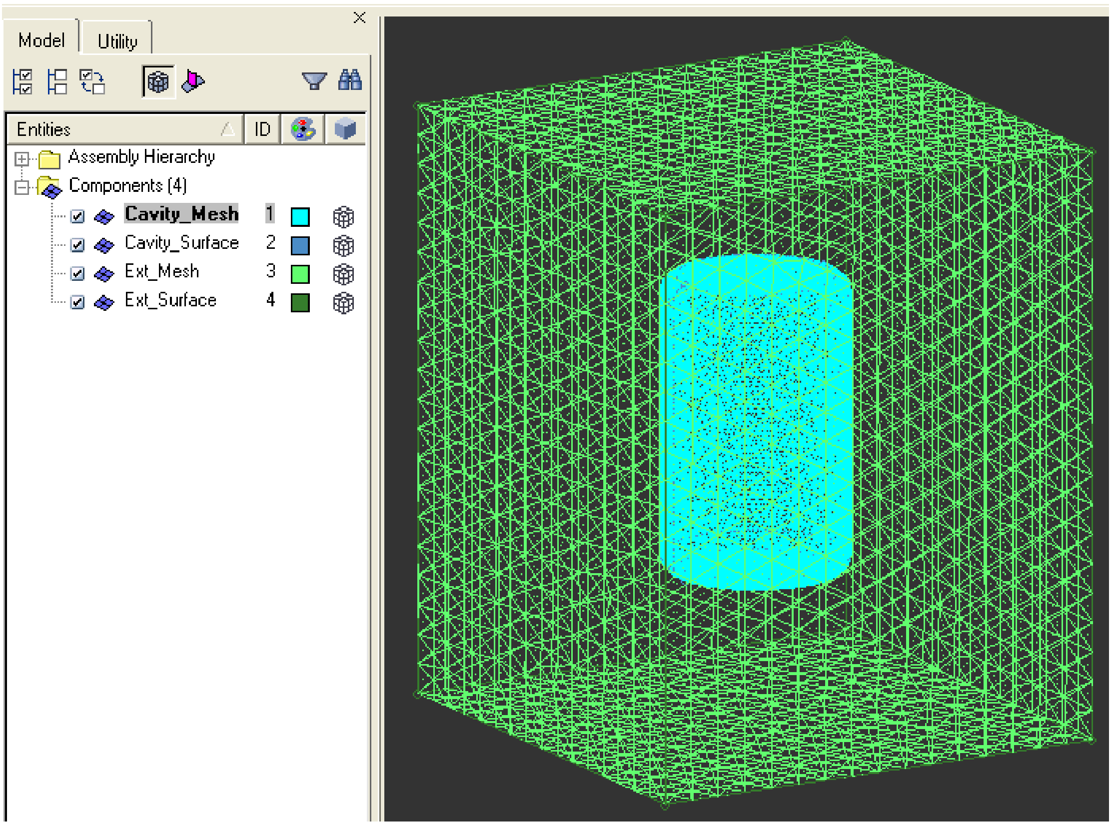


Figure 10.10. Mold_Surf_Mesh.hm model

Create a "Save As" copy of the file and name it "Cylinder_Surface_Mesh.hm".

File menu → Save As → File name: "Cylinder_Surf_Mesh.hm" → Save

BEST PFE.COM
List of research project topics and materials

Notice that when you make a "Save As" copy, the copy opens immediately, closing the file that originate it. That means you should be now seeing the "**Cylinder_Surf_Mesh.hm**" model **in which we will continue working now**.

We want to have the same mesh in the interface between the casting and the mold. Therefore, we will keep the mesh that correspond to the cavity mesh of the mold and use it as the surface mesh of the casting. Since we are now working in the "Cylinder_Surf_Mesh.hm", the external surface of the mold and the corresponding elements are not needed and must be deleted:

Right click the "Ext_Mesh" collector in the Model Browser → Delete → Ok

Right click the "Ext_Suefaces" collector in the Model Browser → Delete → Ok

If you are not using collectors, delete the surfaces using the "Delete" option in the Edit menu (shortcut: F2), check the "delete associated elems" check box to delete surfaces and elements at once.

The cylinder surface mesh is ready now. It should look as:

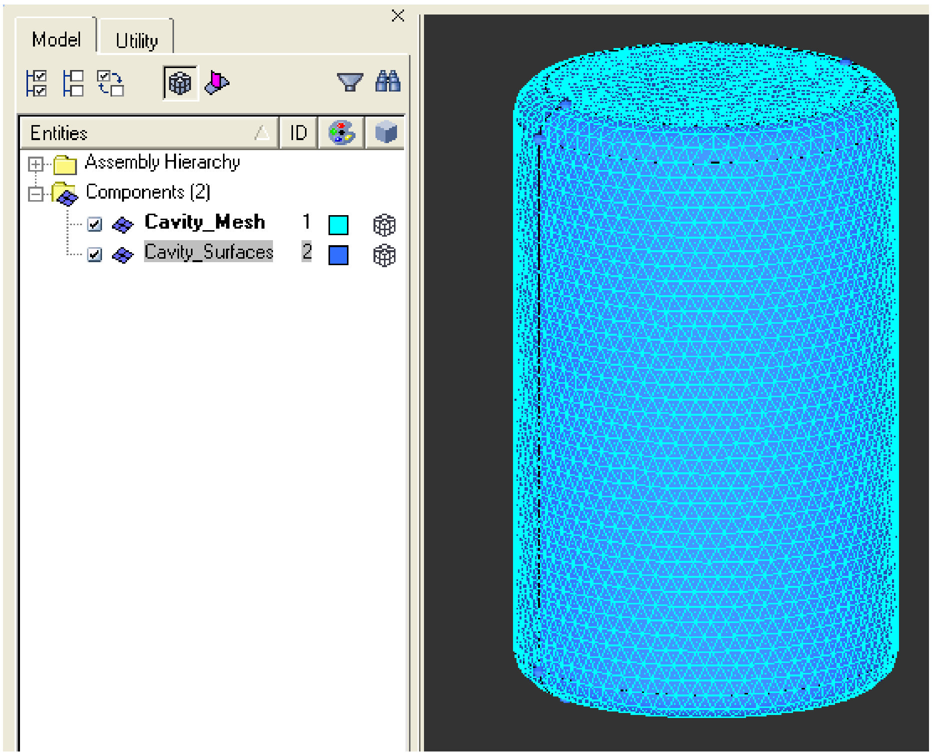


Figure 10.11. Cylinder_Surf_Mesh.hm model

Save the file:

File menu → Save

Note: Remember that Hypermesh do not recognize CTRL+S so use the Save option in the File menu.

The next step is to create a volume mesh from each one of the surface meshes.

10.1.1.2.5 Meshing the volumes

The Cylinder:

Create a "Save As" copy of the "Cylinder_Surface_Mesh.hm" file and name it "Cylinder_Volume_Mesh.hm".

File menu → Save As → File name: "Cylinder_Volume_Mesh.hm" → Save

The model displayed is now "Cylinder_Volume_Mesh.hm" in which a volume mesh with tetrahedral elements will now be created.

Create a new collector called "Volume_Mesh"as explained in section 10.1.1.2.3 to store the volume mesh. Activate it ("Make Current").

Mesh menu → Tetramesh → Select "volume tetra" as the mesh type → Set the "enclosed volume" option to "surfs"→ Select the surfaces of the cylinder* → set the rest of the panel as in *Figure 10.12* → mesh → return

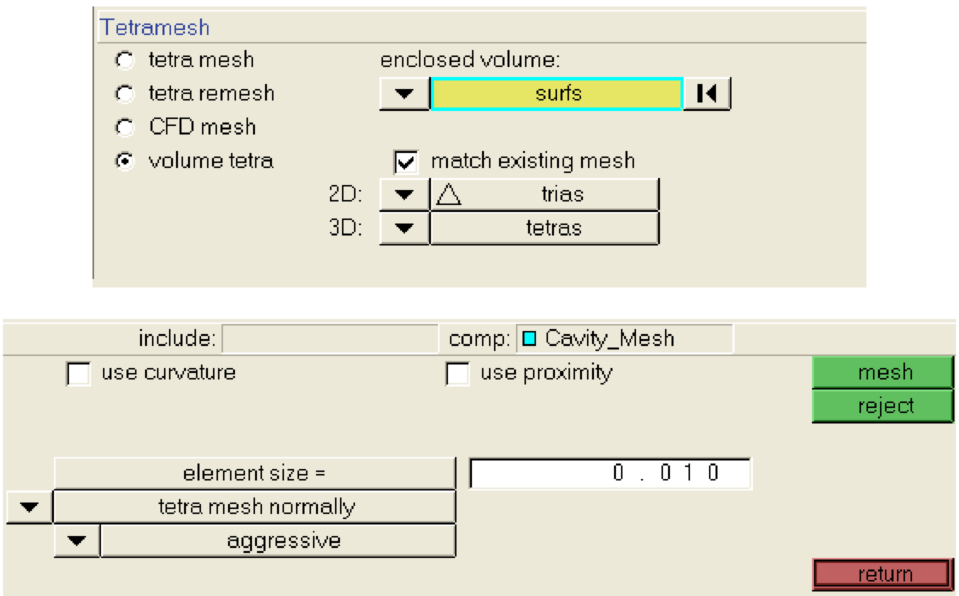


Figure 10.12. Tetramesh panel setting for the "Cylinder_Volume_Mesh.hm" model

**By clicking one surface, Hypermesh automatically select all the surfaces that together with the selected one enclose a volume.*

The volume mesh for the Cylinder is ready.

Save the file:

File menu → Save

Optional: To see the appearance of the elements inside the cylinder, the "Mask" option can be used. It allows us to hide selected elements from the display. For details about how to use the "Mask" option refer to the Hypermesh help documentation.

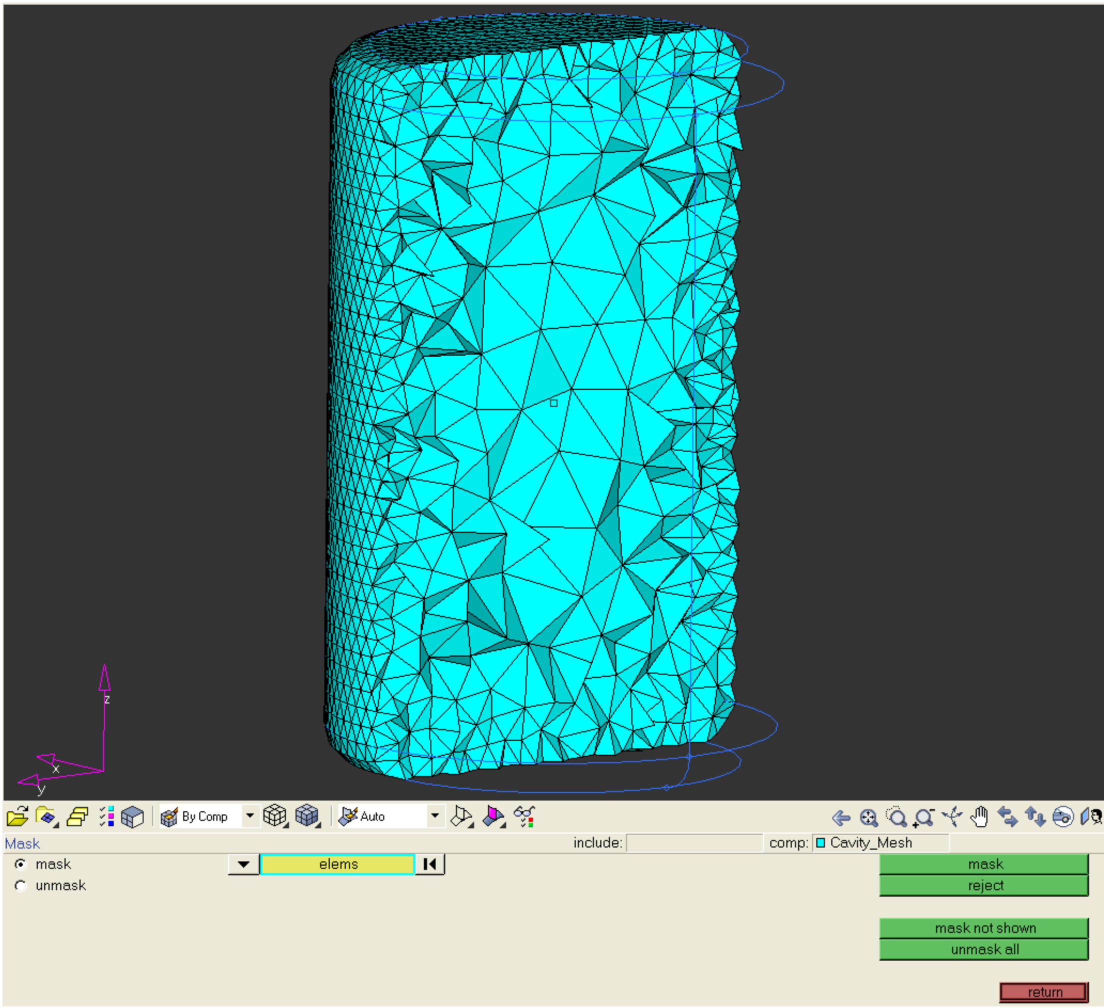


Figure 10.13. A masked view of the Cylinder model where inner elements can be seen

The Mold:

Open the Mold_Surf_Mesh.hm model → File menu → Save As → Name it: "Mold_Volume_Mesh.hm" → Save

Now the active file should be the Mold_Volume_Mesh.hm model.

The option "tetra mesh" is used here instead of "volume tetra" as in the Cylinder volume mesh. Volume tetra just allows the selection of one single closed volume, identified by the surface that enclose it and while meshing it ignore other closed volumes that could be inside the selected one.

This work for the cylinder, but for the mold we want to mesh a volume between two closed volumes, i.e. the cavity volume and the external walls of the mold. In this case, the tetra mesh option is used, which identify closed volumes by the surface elements that enclose them and allow multiple selections. This option meshes the volume in between the cavity and the mold external walls. I our case we can select all the elements but if it would be necessary, the surface elements can be selected by collectors or individually.

For the mold also, create a new collector called "Volume_Mesh" as explained in section 10.1.1.2.3 to store the volume mesh. Activate it ("Make Current").

Mesh menu → Tetramesh → Select "tetra mesh" as the mesh type → Set the "float trias/quads" option to "elems"→ Click elems → Select "All" → set the rest of the panel as in *Figure 10.14* → mesh → return

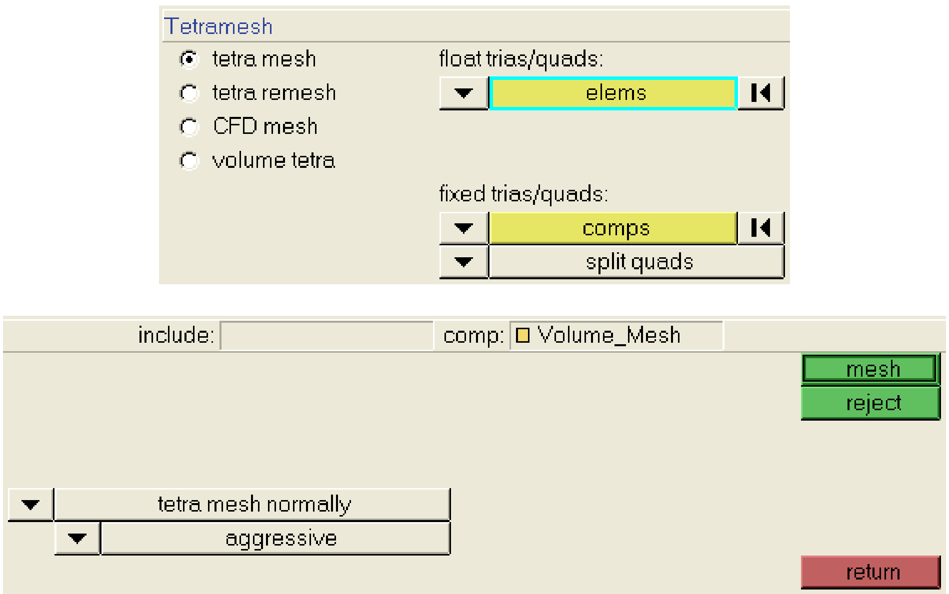


Figure 10.14. Tetramesh panel setting for the "Mold_Volume_Mesh.hm" model

The volume mesh for the mold is ready.

Save the file:

File menu → Save

Optional: The "Mask" option can be used again to see the appearance of the inner mesh of the model.

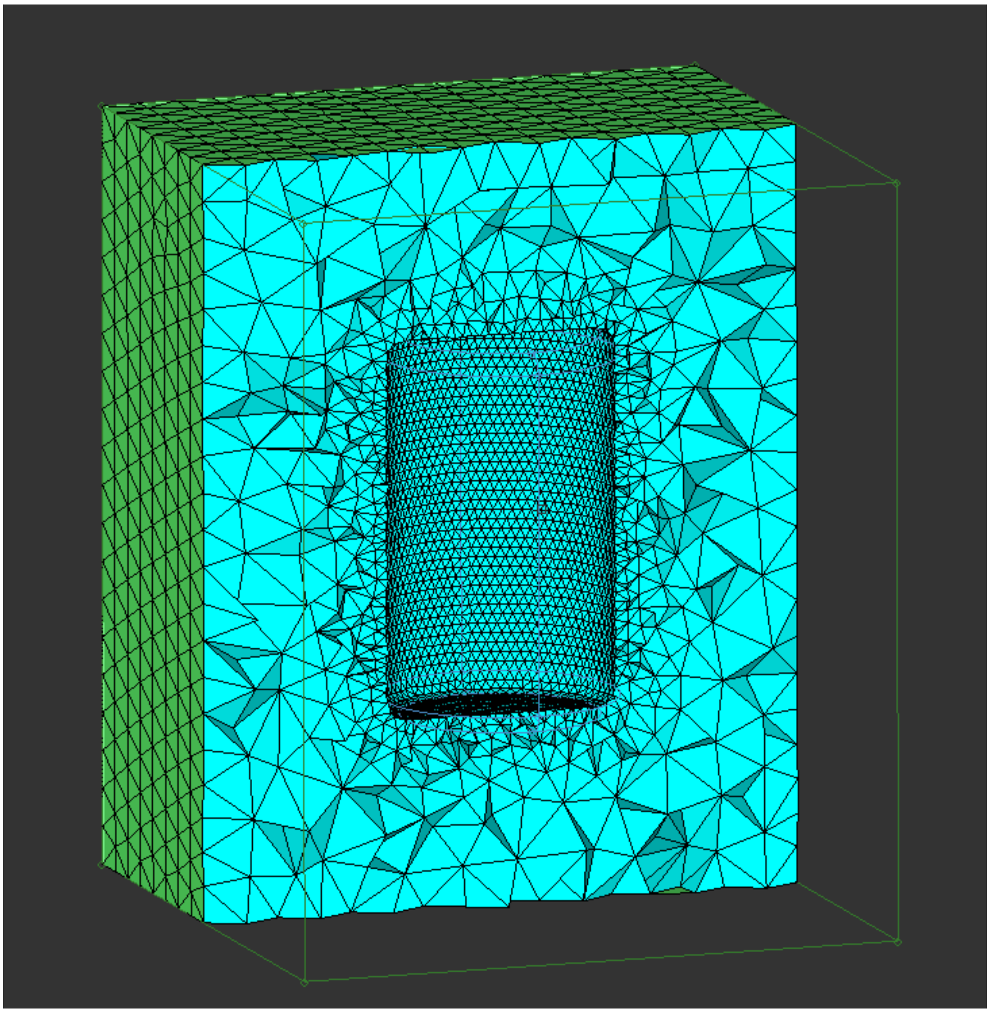


Figure 10.15. A masked view of the Mold model where inner elements can be seen

10.1.1.2.6 Exporting the meshes to Abaqus

The volume meshes for the Mold and the Cylinder were exported as separate .inp files that just contain a collection of nodes and elements.

These .inp files were then imported into Abaqus CAE as new models, but since the model just contains the geometry, the part was copied to a previously prepared model that includes properties as material, section and interaction. This last process would be equivalent to copy the nodes and elements from the .inp file generated in Hypermesh and replace the nodes and elements directly on an .inp file generated by Abaqus containing the rest of the necessary keywords. The details about our approach to import the models into Abaqus will be discussed in section 10.1.1.3.

Exporting the Mold mesh:

Confirm you are working in the "Mold_Volume_Mesh.hm" model.

If the User Profile has been set to "Abaqus" as illustrated in *Figure 10.5*, the Utility browser should look like in *Figure 10.16*:

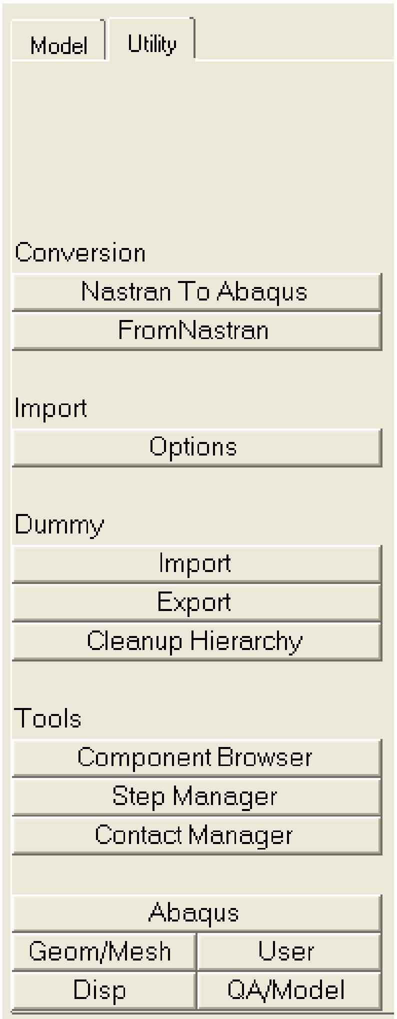


Figure 10.16. Utility browser appearance for the Abaqus User Profile

Procedure:

Click the “Export” button in the Utility Browser → Press the  button in the “Export Abaqus deck” window → Browse a destination folder and name the file “Mold.inp” → Save → For the Export option in the “Export Abaqus deck” window select “all” (

) → Ok

The .inp file of the Mold mesh is now exported and ready to be used in Abaqus.

Save the file:

File menu → Save

Exporting the Cylinder mesh:

Open the “Cylinder_Volume_Mesh.hm” model → Click the “Export” button in the Utility Browser → Press the  button in the “Export Abaqus deck” window → Browse a destination folder and name the file “Cylinder.inp” → Save → For the Export option in the “Export Abaqus deck” window select “all” (

) → Ok

The .inp file of the Cylinder mesh is now exported and ready to be used in Abaqus.

Save the file:

File menu → Save

10.1.1.3 Abaqus Simulation Setup

Here a step by step procedure to setup and run first the thermal simulation and then the stress simulation in Abaqus/CAE V6.7 is presented and commented.

Setup Overview:

1- Importing the models

2- Materials definition

3- Sections definition

4- Sections assignment

5- Mesh element type

6- Assembly

7- Steps definition

8- Predefined Fields definition

9- Interaction Properties definition

10- Interactions definition

11- Boundary Conditions

12- Predefined Field Requests

13- Job creation

10.1.1.3.1 The Thermal Simulation

We will include a shake-out process in our thermal simulation, which means that before completion of the cooling, the casting is removed from the mold and is left to cool down until room temperature exposed to the ambient. This implies that the mold must be present in the simulation corresponding to the before shake-out (BSO) period, and must not be in the simulation of the after shake-out (ASO) period. Therefore, in this steps-guide the thermal simulation will be found divided in two models: before shake-out, to be called (I-BSO), and after shake-out (II-ASO).

Bestpfe.com

10.1.1.3.1.1 Before Shake-Out model

1- Importing the models

Open Abaqus/CAE → Create Model Database

An empty model is automatically created.

Expand the Model tree (left side of the Abaqus/CAE user interface under the Model tab)→ Right click the empty model (Model-1) → Rename… → Rename it as "I-BSO".

Now we will import the mesh of the cylinder and the mesh of the mold created with Hypermesh in section 10.1.1.2 as two independent new models.

First the .inp file of the cylinder

File menu→ Import → Model → "Cylinder.inp" → Ok

Now the .inp file of the mold

File menu→ Import → Model → "Mold.inp" → Ok

These models just contain a Part. Copy the Part object from the Cylinder model to the model I-BSO, where the simulation will be set, as follow:

Model menu→ Copy Objects → From model: Cylinder → To model: I-BSO → Click the arrow next to the Parts object category → Check the box next the part name (Part-I) → Apply

With the Copy Objects dialog box still open, copy the Part object from the Mold model to I-BSO:

From model: Mold → To model: I-BSO → Click the arrow next to the Parts object category → Check the box next the part name (Part-I) → Ok

Confirm that the parts are in the model I-BSO and rename them as "Cylinder" and "Mold" respectively.

Delete the two imported models.

The Model tree should look like:

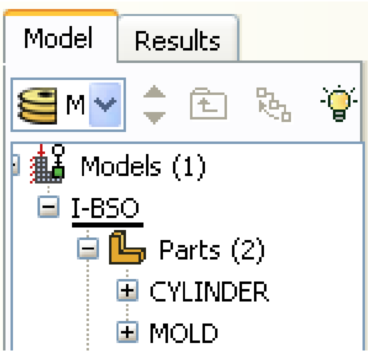


Figure 10.17. Imported CAD files in Abaqus

2- Materials definition

The material properties to be defined for the casting part (CYLINDER) in the "I-BSO" model are:

-Density

-Conductivity

-Specific Heat

-Latent Heat (constant)

The properties to be defined for the mold part (MOLD) are:

-Density

-Conductivity

-Specific Heat

The values of our material data can be found in the Appendix section 10.4. However, in this section we present the curves of the previously mentioned temperature dependent material properties.

"Abaqus/CAE does not use specific units, but the units must be self-consistent throughout the model", "which means that derived units of the chosen system can be expressed in terms of the fundamental units without conversion factors" see Abaqus (2007).

BESTPFE.COM
List of research project topics and materials

An example of a self-consistent set of units is the International System of units (SI), which fundamental units are length in meters (m), mass in kilograms (kg), time in seconds (s), temperature in degrees Kelvin (K), and electric current in Amperes (A). Derived units as Newton (N), Joule (J) or Coulomb (C) must be expressed in terms of the fundamental ones.

The geometries for this model where created in meters, so the materials properties values must be consequent with it.

Creating the material for the Cylinder part:

Right click the Materials container in the Model tree → Create → Name it "CAST-MAT" → Under the General material editor menu select Density → Under the Thermal material editor menu select Conductivity, Specific Heat and Latent Heat.

Fill in the appropriate data. For reference, see the Appendix section 10.4

→OK

Creating the material for the Mold part:

Right click the Materials container in the Model tree → Create → Name it "MOLD-MAT" → Select Density, Conductivity and Specific Heat → Fill in the right data → OK

The curves for the Cylinder temperature dependent material data are:

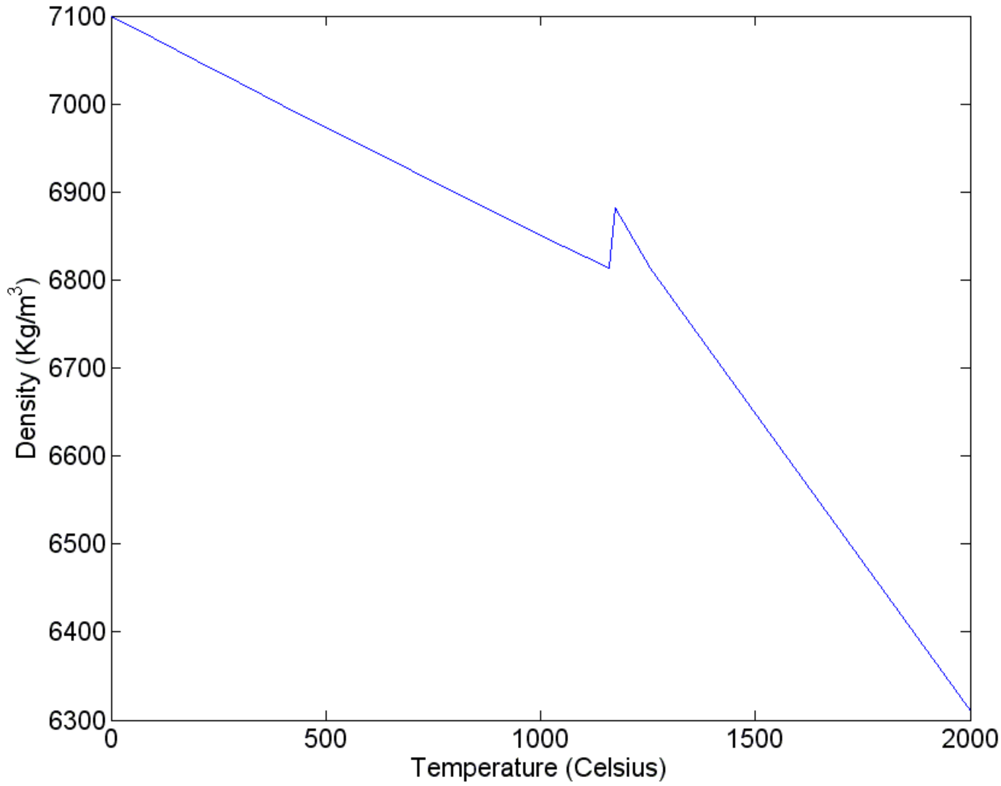


Figure 10.18. Density material data curve for the Cylinder part

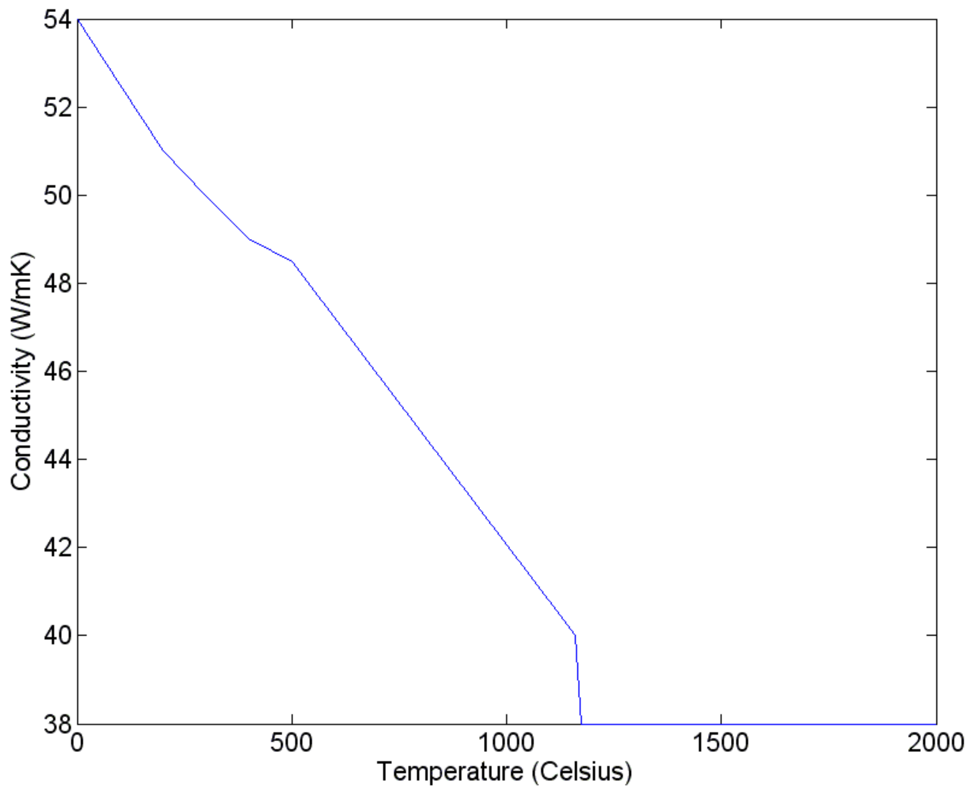


Figure 10.19. Conductivity material data curve for the Cylinder part

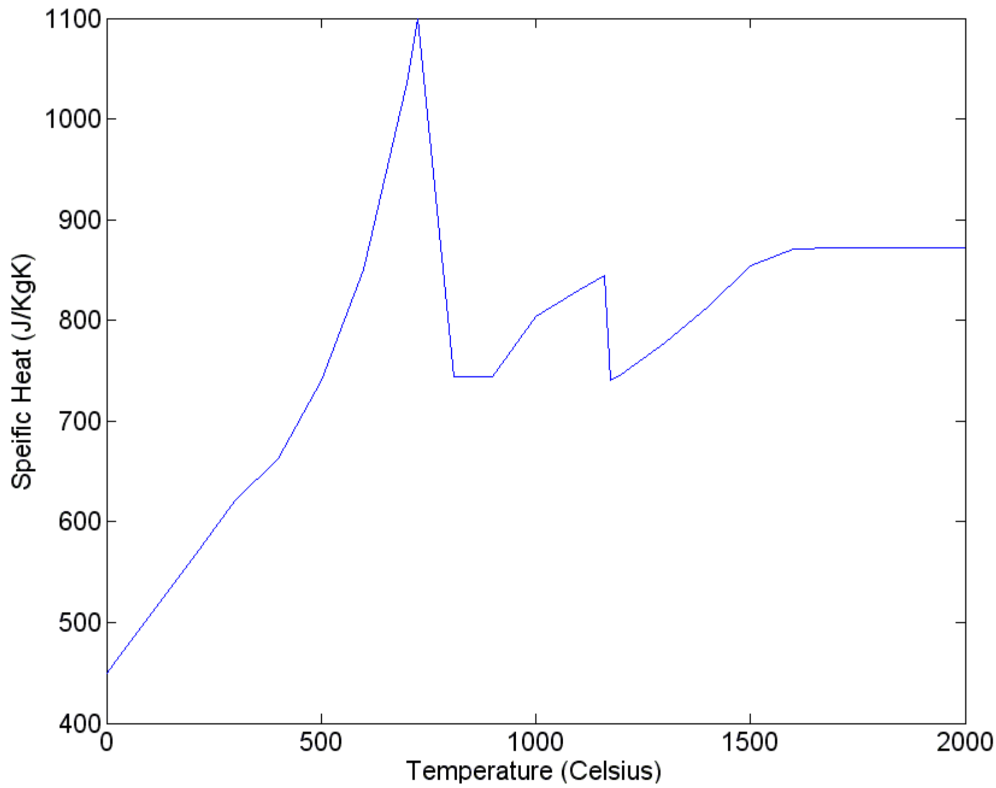


Figure 10.20. Specific Heat material data curve for the Cylinder part

Our curves for the MOLD material data are:

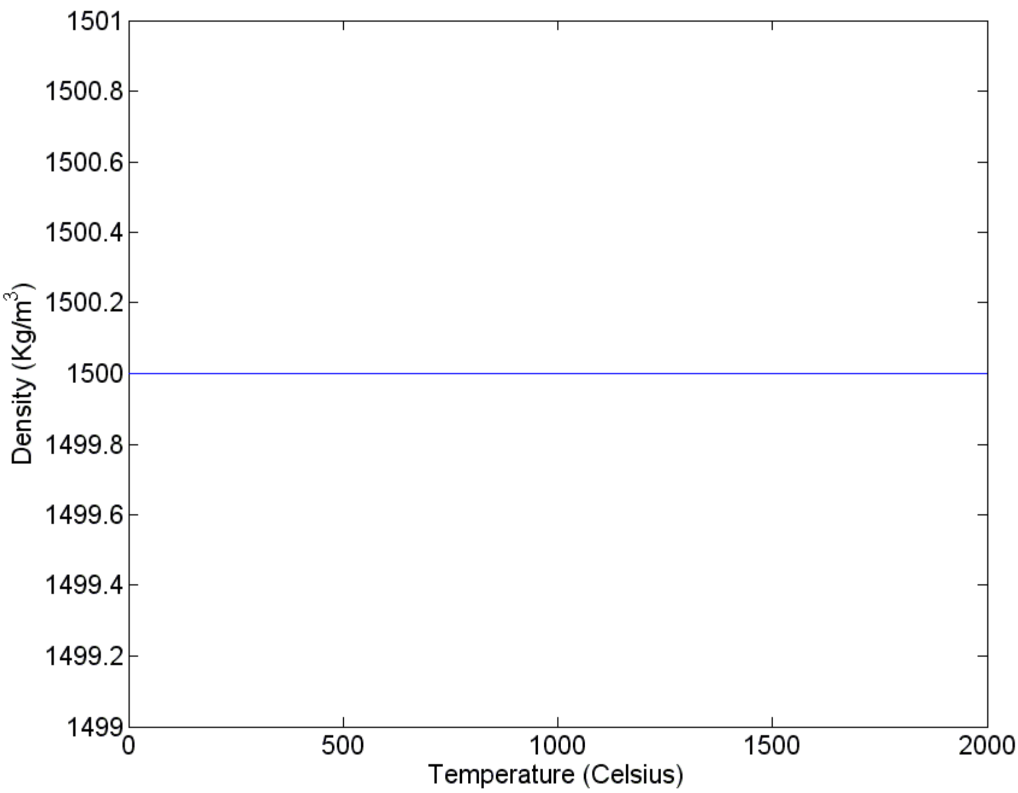


Figure 10.21. Density material data curve for the Mold part

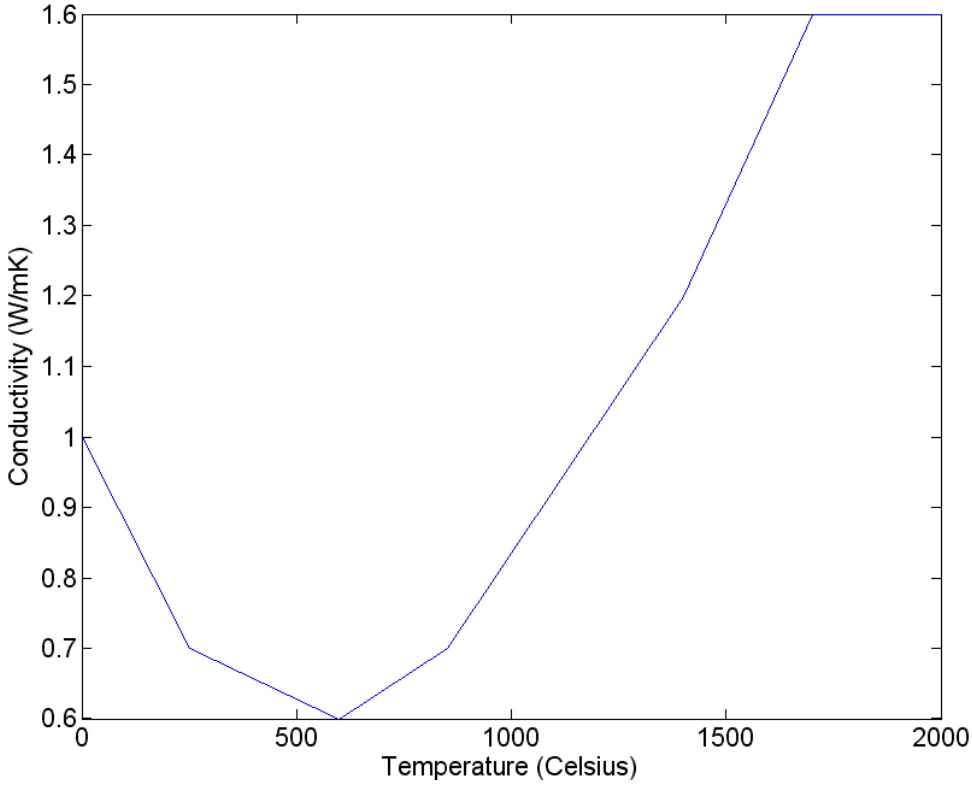


Figure 10.22. Conductivity material data curve for the Mold part

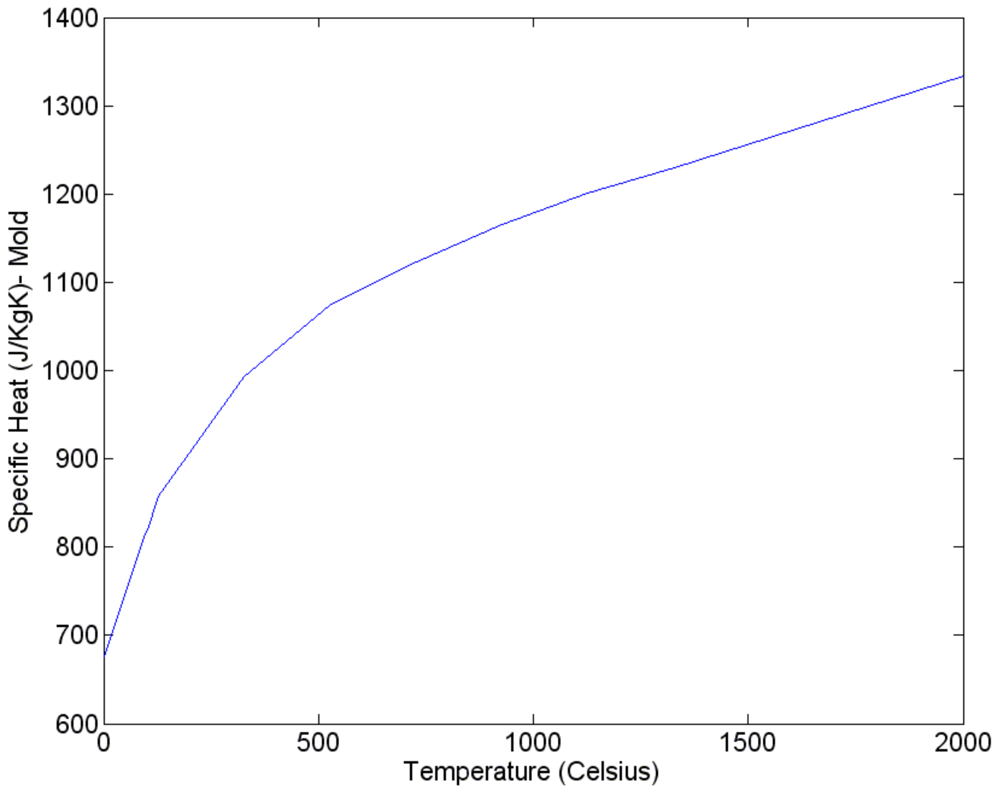


Figure 10.23. Specific Heat material data curve for the Mold part

3- Section definition

Create two Solid, Homogeneous sections with the corresponding material for the Cylinder and the Mold respectively.

Cylinder section:

Right click Sections in the Model tree → Create → Name it "CAST-SEC" → Category: Solid → Type: Homogeneous → Continue → Material: CAST-MAT → OK

Mold section:

Right click Sections in the Model tree → Create → Name it "MOLD-SEC" → Category: Solid → Type: Homogeneous → Continue → Material: MOLD-MAT → OK

4- Section assignment

Assign the respective section to the whole geometry of the Cylinder and the Mold.

Cylinder section assignment:

Expand the Cylinder part in the Model tree → Right click the Sections Assignment collector → Create

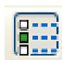
If the "Selection option tools" are not displayed, press  and ensure that the "Select from all entities" option () is selected since it allow to select elements from both outside and inside a part.

Figure 10.24. Selection option tools.

→ Select the whole geometry → Done → Section: CAST-SEC → OK → Done

Similarly, the section assignment for the Mold is performed as follow:

> Expand the Mold part in the Model tree → Right click the Sections Assignment collector → Create → Select the whole geometry → Done → Section: MOLD-SEC → OK → Done

5- Mesh Element Type

Assign the Heat Transfer element type family to both parts. A DC3D4 element type will be automatically selected.

For the Cylinder:

> Right click the Mesh item under the Cylinder part in the Model tree → Switch Context → Mesh menu → Element Type → Select the whole geometry → Done → Element Library: Standard → Geometric Order: Linear → Family: Heat Transfer → OK → Done

For the Mold:

> Right click the Mesh item under the Mold part in the Model tree → Switch Context → Mesh menu → Element Type → Select the whole geometry → Done → Element Library: Standard → Geometric Order: Linear → Family: Heat Transfer → OK → Done

So far, the Model tree should look as:

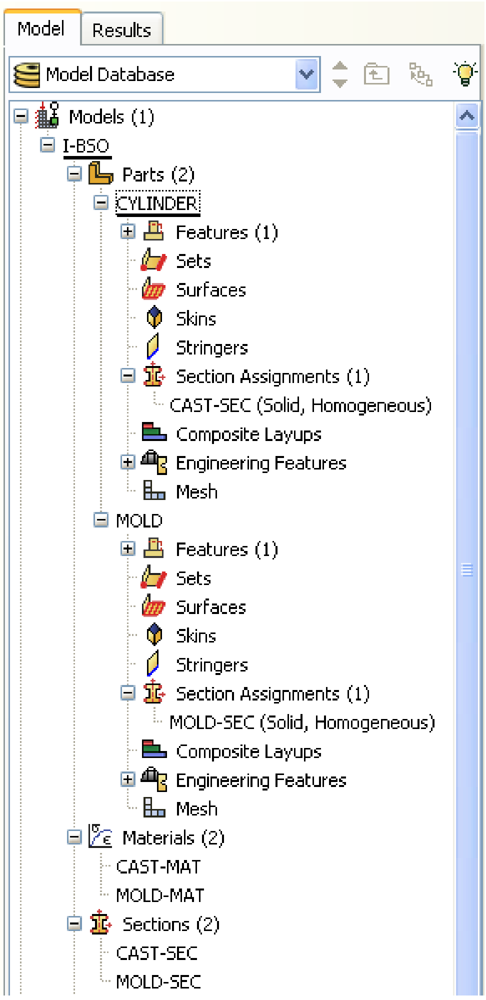


Figure 10.25. Model tree after completing the first 5 steps of the setup

6- Assembly

Since the parts where made with respect to the same coordinate system, when they get inserted in the assembly, they will be automatically in the correct relative position and no further positioning operations will be needed.

Note: If instead of importing the parts, they would have been created in Abaqus, is in the Assembly module where a Boolean operation should be performed to produce the mold cavity.

Expand the Assembly item in the Model tree → Right click the Instances collector → Create → Select both parts holding the Shift key → OK

7- Steps definition

An Initial Step is created by default. The initial conditions and the contact interaction between the casting and the mold will later be defined on it.

However, a General – Heat transfer step must be added, the before shake-out step. Here the mold-ambient interactions are described and the output data that we are interested in is specified. The duration of this step define the time that the casting remains in the mold prior shake-out.

Right click the Steps collector → Create → Name it "Before Shake-Out" → Procedure type: General → Heat transfer → Continue →

Time period (seconds): 28800 →

Incrementation Type: Automatic →

Maximum number of increments: 28800 →

Initial Increment size: 10 →

Minimum Increment size: 1E-12 →

Maximum Increment size: 28800 →

Max. allowable temperature change per increment: 10 →

Max. allowable emissivity change per increment: 0.1 → OK

8- Predefined Fields definition

We assume a homogeneous initial temperature of 1400°C for the casting and 20°C for the mold.

Cylinder initial temperature:

Right click the Predefined Fields collector → Create → Name it "Cast-Initial-Temp" → Step: Initial → Category: Other → Type: Temperature → Continue → Select the whole geometry of the Cylinder → Done → Distribution: Direct specification → Section variation: Constant through region → Magnitude: 1400

Mold initial temperature:

Right click the Predefined Fields collector → Create → Name it "Mold-Initial-Temp" → Step: Initial → Category: Other → Type: Temperature → Continue → Select the whole geometry of the Mold → Done → Distribution: Direct specification → Section variation: Constant through region → Magnitude: 20

Bestpfe.com

9- Interaction Properties definition

The interaction property to be defined describes the heat transfer coefficient (HTC) between the Cylinder and the Mold. We used a constant HTC of 1000 in Magmasoft; to use a similar HTC in Abaqus, the interaction is defined through a clearance dependent data, set as in *Figure 10.26*.

> Right click the Interaction Properties collector → Create → Name it "CAST-MOLD-CONTACT-INTERACTION-PROPERTY" → Type: Contact → Continue → Thermal menu → Thermal Conductance → Check "Use only clearance-dependency data" → Fill in appropriate data (see the Appendix section) → OK

The "Edit Contact Property" window may look as:

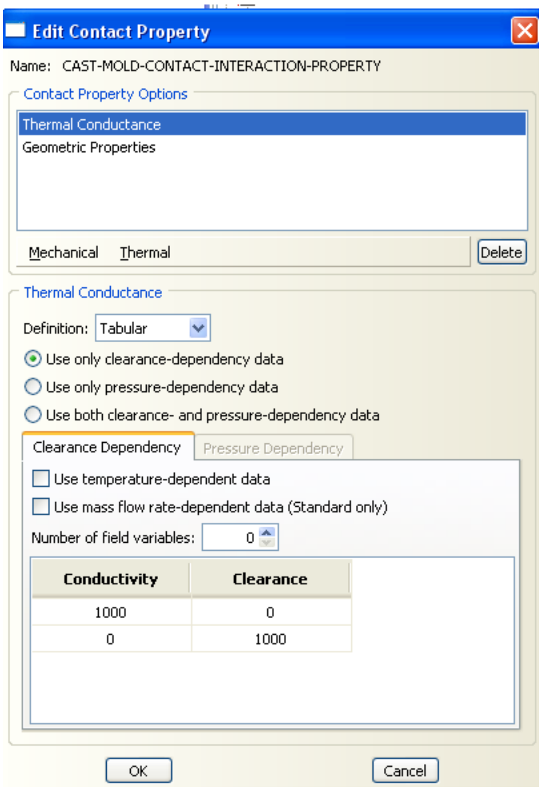


Figure 10.26. HTC – Conduction interaction property between the cast and the mold

10- Interactions definition

The conduction between the casting and the mold is described by means of a surface-to-surface contact interaction using the previously created property in the step 9. The interaction is assigned to the Initial step.

Another approach consist in simulate perfect conduction by means of a TIE Constraint, but the contact interaction produce better comparisons between Abaqus and Magmasoft by allowing us to set the same value of heat transfer coefficient (HTC) (for details, see the Appendix section 10.7).

The contact interaction give the possibility of run the Before and After Shake-Out simulations as consecutive steps in the same model by suppressing the interaction in the After Shake-Out step, which can not be done with the tie constraint. However this alternative will have the mold present in the after shake-out step even if is not interacting with the casting, which means that the solver will continue calculating the cooling of the mold, consuming (unnecessarily) precious processing capacity of the computer and slowing down the calculation.

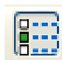
Therefore, even with the contact interaction, we choose to run the Before and After Shake-Out steps in different models, where the thermal history in the output database of the Before Shake-Out is read as initial temperature field in the After Shake-Out step. In the latest case the mold is totally removed from the After Shake-Out simulation.

A convective interaction between the mold and the ambient is defined through a Surface film condition interaction. The interaction is assigned to the Before-Shake-Out step.

A radiation interaction between the mold and the ambient is defined through a Surface radiation to ambient interaction. The interaction is assigned to the Before-Shake-Out step.

Conduction (Cylinder-Mold):

Right click the Interaction collector → Create → Name it "CAST-MOLD-CONTACT-INTERACTION" → Step: Initial → Type: Surface-to-surface contact → Continue → Select the external surface of the Cylinder for the Master surface

If the "Selection option tools" are not displayed, press  and ensure that the "Select from exterior entities" option () is activated.

→ Done → Slave type: Surface → Select the surface of the cavity of the Mold → Done → Define the Edit Interaction window as in *Figure 10.27*:

BESTPFE.COM
List of research project topics and materials

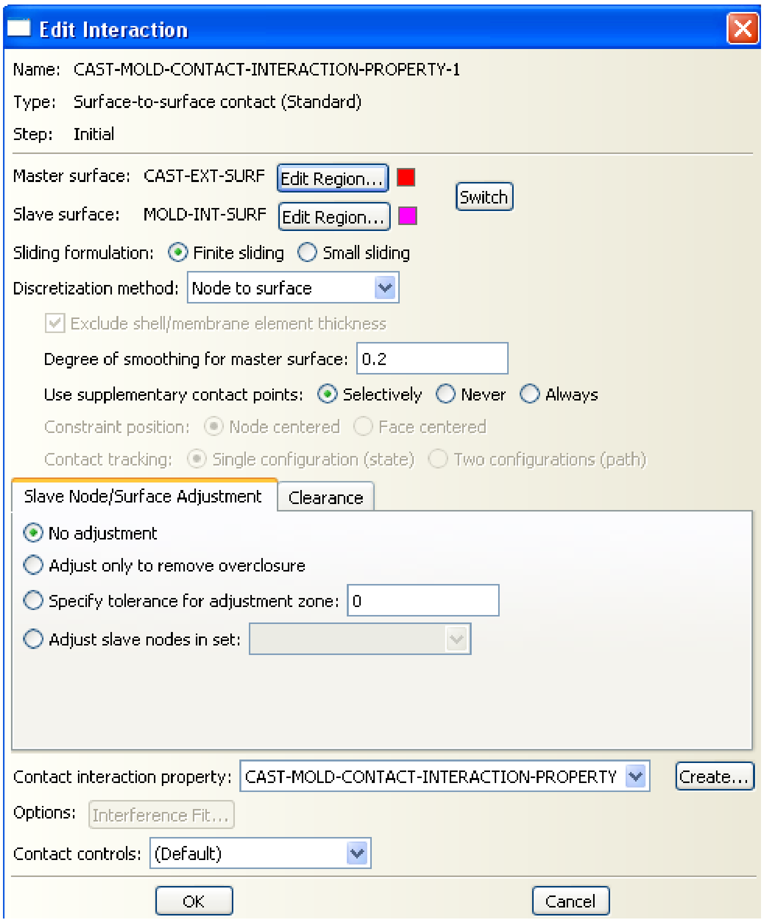


Figure 10.27. Conductive interaction between the cast and the mold

Convection (Mold-Ambient):

Right click the Interaction collector → Create → Name it “MOLD-AMBIENT-CONVECTION” → Step: Before-Shake-Out → Type: Surface film condition → Continue → Select the external surface of the Mold → Done → Define the Edit Interaction window as:

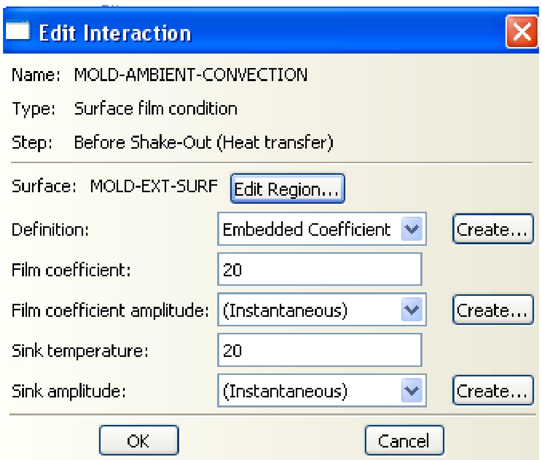


Figure 10.28. Convective interaction between the mold and the ambient

Radiation (Mold-Ambient):

Right click the Interaction collector → Create → Name it "MOLD-AMBIENT-RADIATION" → Step: Before-Shake-Out → Type: Surface radiation to ambient → Continue → Select the external surface of the Mold → Done → Define the Edit Interaction window as:

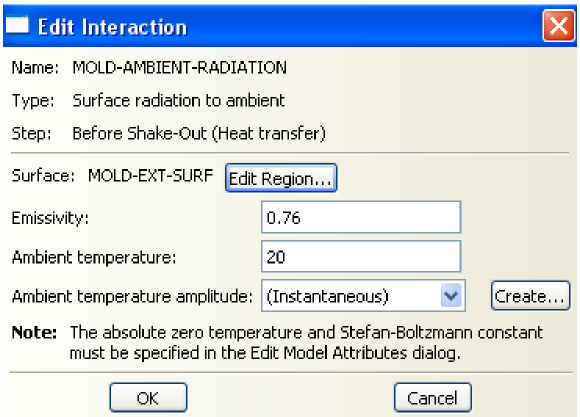


Figure 10.29. Radiation interaction between the mold and the ambient

11- Boundary Conditions

No mechanical boundary conditions are needed to be specified in the thermal problem.

12- Predefined Field Requests

In this step the nodal thermal history is requested to be written in the output database (.odb file).

Right click the Field Output Requests collector → Create → Name it "Nodal-Thermal-History" → Step: Before-Shake-Out → Continue → Set the Edit Field Output Request window as in *Figure 10.30*:

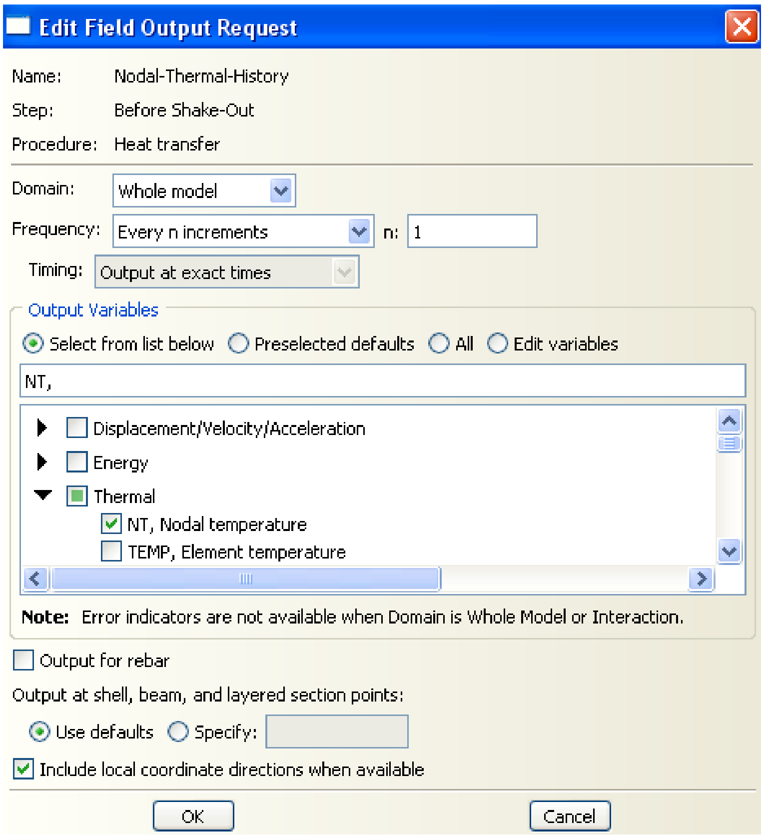


Figure 10.30. Field Output Request of the Nodal Thermal History.

Note: Is not necessary to create any History Output Request.

13- Job creation

When a Job is created, an input file (.inp) for the FE solver is written. This file will not be read until the Job is submitted for calculation. To create the Job, do as follow:

> Expand the "Analysis" item in the Model tree → Right click the "Jobs" collector → Create → Name it "BSO-R1" → Source: Model → In the displayed models list select "I-BSO" → Continue → Write a short description of the analysis if desired → Job Type: Full analysis → OK

Now the Job is ready to be submitted for calculation in the solver.

Note that the results obtained after calculation of this I-BSO model are needed for the setup and further run of the second part of the thermal analysis which now follows. For details about how to run the analysis refer to section 10.1.2.

10.1.1.3.1.2 After Shake-Out model

The After Shake-Out model is created from a modified duplicate of the Before Shake-Out one. In this copy, we will remove the Mold from the Assembly module to represent the shake-out. As long as is not in the assembly, it will not affect the simulation, but to avoid confusion and to simplify the .inp file, the Mold and the related information (material, section, sets, etc) will be removed from the entire model.

In this model, the initial temperature field of the casting, will be read from the thermal history stored in the *BSO-R1.odb* file of the Before Shake-Out analysis.

The cylinder will be cooled down by means of a convective and a radiate interaction with the ambient applied to the whole external surface of the cylinder.

The same 13 steps as in the Before Shake-Out model will be followed and just the differences will be detailed.

-Copying the model

Right click the I-BSO item in the model tree. → Copy Model → Name it "II-ASO" → OK

-Removing the Mold and related information from the model

Part

Expand the new II-ASO model in the model tree → Expand its Parts collector → Right click the MOLD part → Delete → Yes

Material

Expand the Materials collector of the II-ASO model → Right click MOLD-MAT → Delete → Yes

Section

Expand the Sections collector of the II-ASO model → Right click MOLD-SEC → Delete → Yes

Assembly instance

Expand the Assembly item of the II-ASO model → Expand the Instances collector →Right click MOLD-1 → Delete → Yes

Interaction

Expand the Interactions collector of the II-ASO model → Right click CAST-MOLD-CONTACT-INTERACTION → Delete → Yes

Interaction Property

Expand the Interactions Properties collector of the II-ASO model → Right click CAST-MOLD-CONTACT-INTERACTION-PROPERTY → Delete → Yes

Predefined Fields

Expand the Predefined Fields collector of the II-ASO model → Right click Mold-Initial-Temp → Delete → Yes

1- Importing the models

The part file of the Cylinder is copied together with the rest of the model, therefore there is no need to import or create any part file.

2- Materials definition

No modifications needed

3- Sections definition

No modifications needed

4- Sections assignment

No modifications needed

5- Mesh element type

No modifications needed

6- Assembly

After deletions of the MOLD instance no more changes are needed in the assembly.

7- Steps definition

The Initial step will be altered later through a modification to the Predefined Fields.

The Before Shake-Out step will be removed and the After Shake-Out step will be created.

Expand the Steps collector of the "II-ASO" model → Right click the "Before Shake-Out" step → Delete → Yes

Right click the Steps collector → Create → Name it "After Shake-Out" → Procedure type: General → Heat transfer → Continue →

Time period (seconds): 43200 →

Incrementation Type: Automatic →

Maximum number of increments: 43200 →

Initial Increment size: 10 →

Minimum Increment size: 1E-12 →

Maximum Increment size: 43200 →

Max. allowable temperature change per increment: 1 →

Max. allowable emissivity change per increment: 0.1 → OK

8- Predefined Fields definition

The Cast-Initial-Temp field will be modified to be read from the .odb file of the BSO-R1 analysis.

Expand the Predefined Fields collector of the "II-ASO" model → Right click "Cast-Initial-Temperature" → Edit → Distribution: From results or output database file → Press "Select" button for the File name → Browse and select the BSO-R1.odb file → OK → Step: 1 → Increment: insert the number of the last increment of the BSO-R1 analysis → Interpolation: Compatible → OK

Note: To find the number of the last increment of the BSO-R1 analysis go to the folder where the analysis was saved and open the BSO-R1.sta file in a text editor. The number in the last line of the "INC" column (second from left to right) corresponds to the last increment.

9- Interaction Properties definition

A convective interaction property will be defined with temperature dependent data for the heat transfer coefficient between the casting and the ambient.

Right click the Interaction Properties collector of the "II-ASO" model → Create → Name it "Conv-HTC" → Type: Film condition → Continue → Check the "Use temperature dependent data" check box → Fill in appropriate data → OK

Our interaction property data looks like:

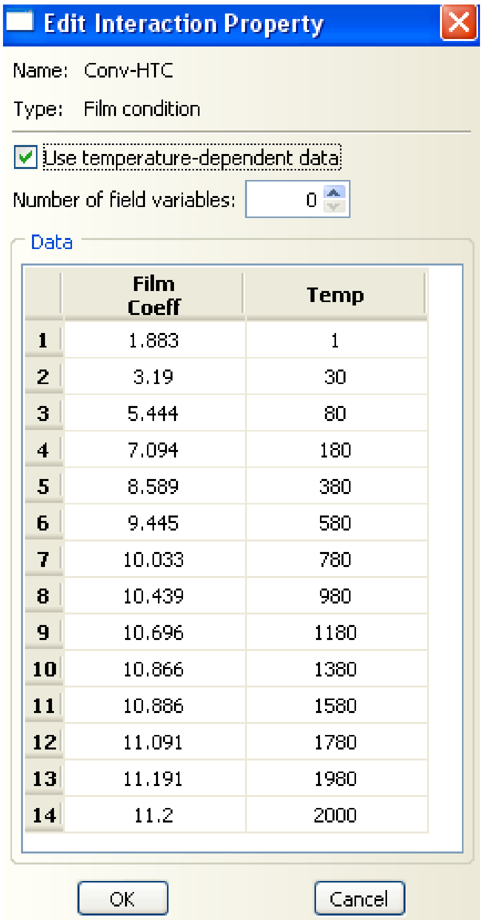


Figure 10.31. Convective interaction property between the casting and the ambient.

10- Interactions definition

Convection between the casting and the ambient

Right click the Interaction collector of the "II-ASO" model → Create → Name it "Cast-Ambient-Convection" → Step: After Shake-Out → Type: Surface film condition → Continue → Select the whole external surface of the casting → Done → Set the "Edit Interaction" window as follow:

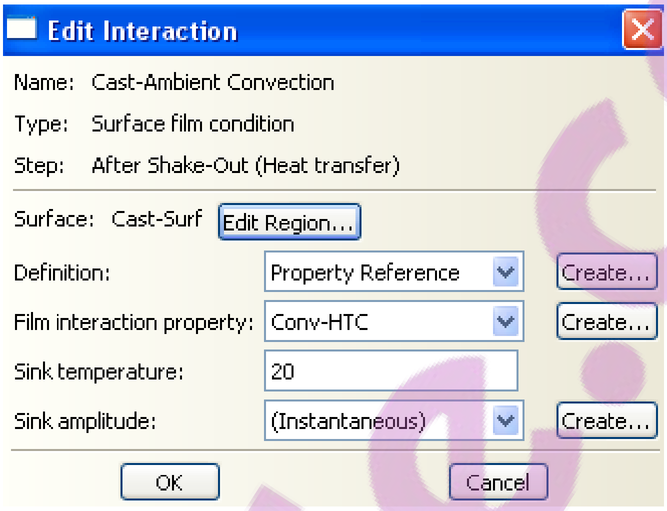


Figure 10.32. Cast-Ambient-Convection interaction

Radiation between the casting and the ambient

Right click the Interaction collector of the "II-ASO" model → Create → Name it "Cast-Ambient-Radiation" → Step: After Shake-Out → Type: Surface radiation to ambient → Continue → Select the whole external surface of the casting → Done → Set the "Edit Interaction" window as follow:

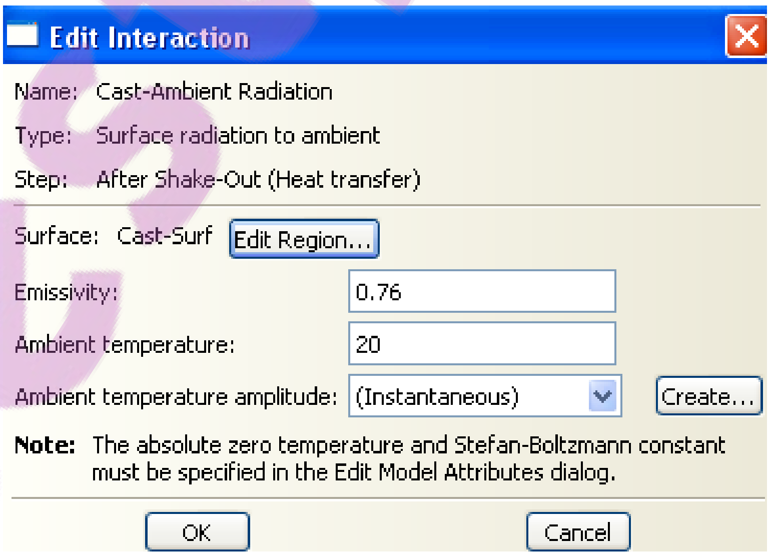


Figure 10.33. Cast-Ambient-Radiation interaction

11- Boundary Conditions

No mechanical boundary conditions are needed to be specified in the thermal problem.

12- Predefined Field Requests

Remains unchanged

13- Job creation

Expand the “Analysis” item in the Model tree → Right click the “Jobs” collector → Create → Name it “ASO-R1” → Source: Model → In the displayed models list select “II-ASO” → Continue → Write a short description of the analysis if desired → Job Type: Full analysis → OK

10.1.1.3.2 The Stress Simulation

The residual stress analysis is performed in a single model. The model is a modified copy of the After Shake-Out model to utilize the same part file and other unchanged data like the section and the material.

Note that is a duplicate of the II-ASO model and not of the II-BSO. That is because we are not interested in the mold part in this analysis neither.

The stresses before and after shake-out are solved in two consecutive steps.

The nodal thermal history is used as an external force to solve stress calculations. Therefore, the corresponding temperature field must be read into each step of the analysis.

In this model, the initial temperature field, assigned to the Initial step, must be the same as the equivalent in the I-BSO model. In the step for the stresses before shake-out, the thermal history from the *BSO-R1.odb* must be read. As well, the thermal history from the *ASO-R1.odb* must be read into the step for the stresses after shake-out.

-Copying the model

Right click the II-ASO item in the model tree. → Copy Model → Name it "III-STRESS" → OK

1- Import the CAD files

Since is a copy of the II-ASO, the model already contains the part file.

2- Materials definition

The material properties to be defined in stress analysis for the Cylinder are:

-Elasticity (Temperature dependent Young's Modulus and Poisson's Ratio)

-Thermal Expansion Coefficient (a pertinent conversion from the Magmasoft data was performed (for details see the Appendix section 10.6)

-Plasticity

The curves for the used material data are:

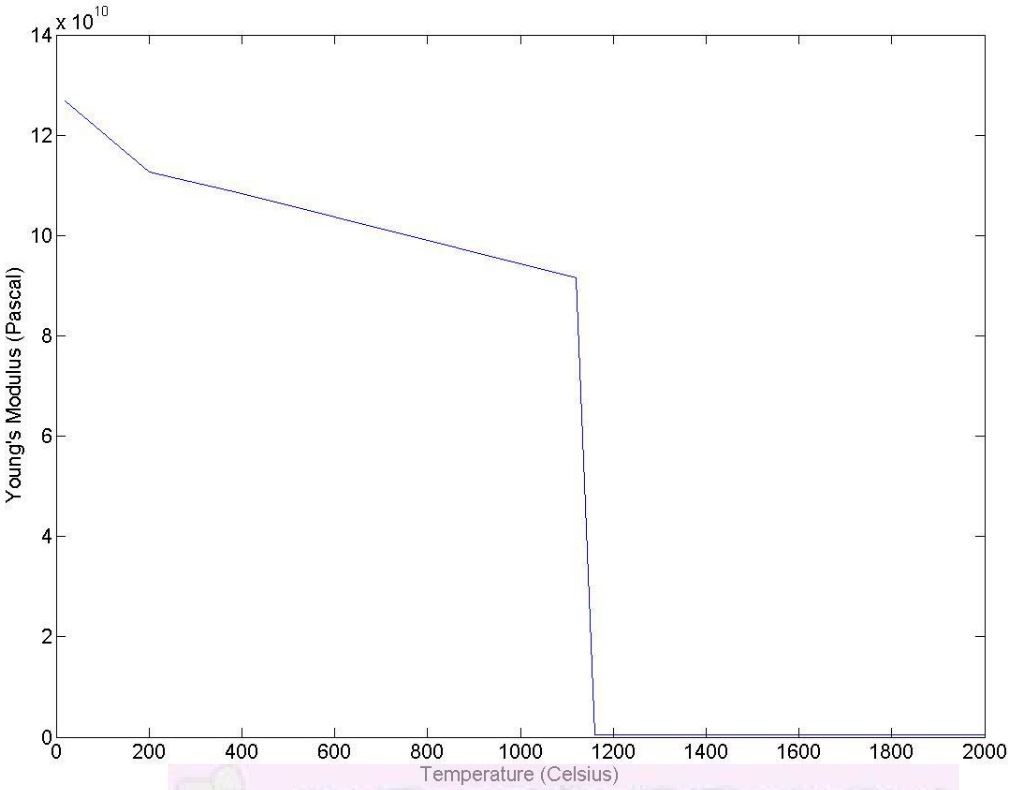


Figure 10.34. Young Modulus material data curve for the Cylinder part

BESTPFE.COM
List of research project topics and materials

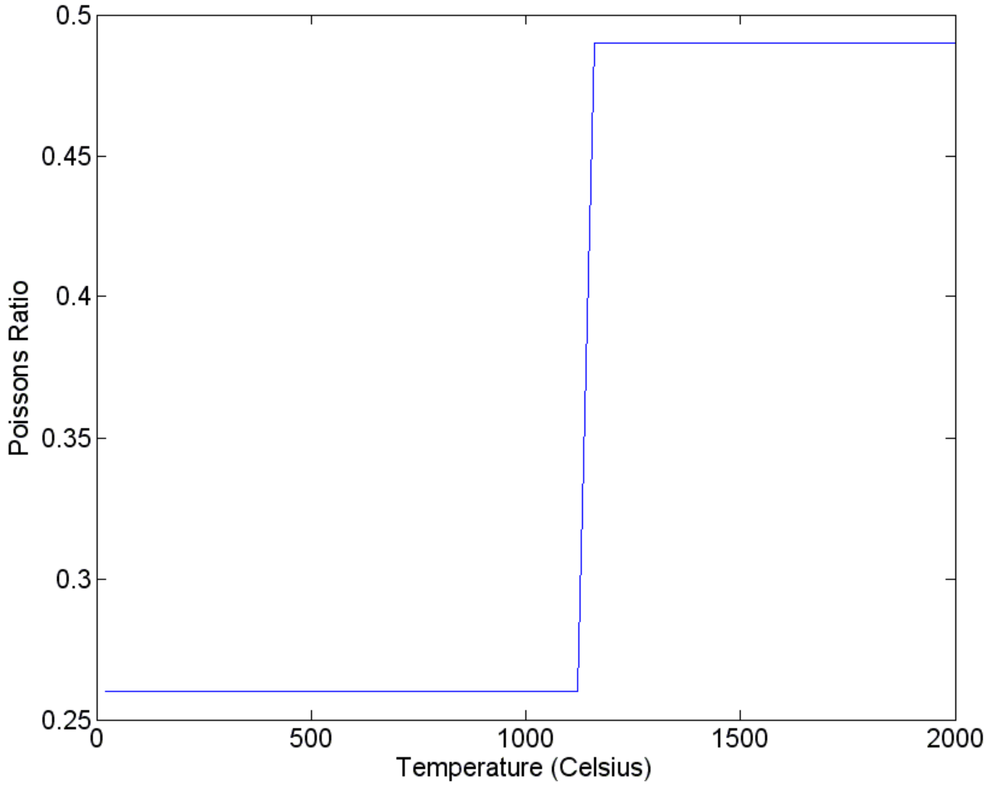


Figure 10.35. Poisson's Ratio material data curve for the Cylinder part

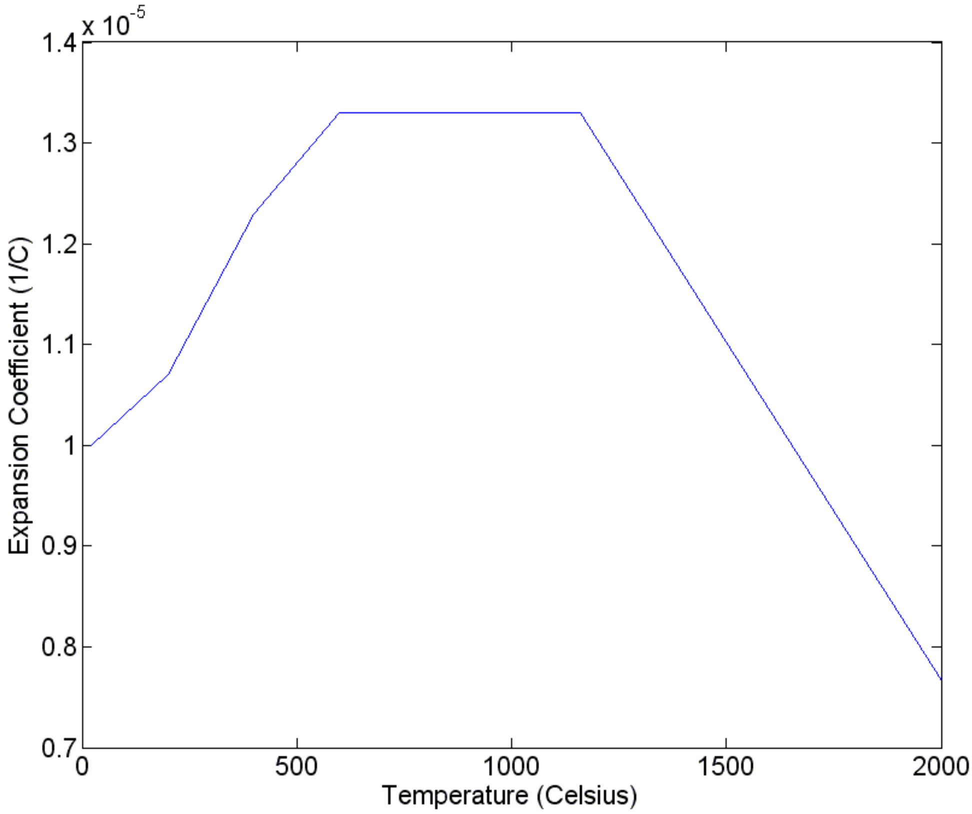


Figure 10.36. Thermal Expansion Coefficient material data curve for the Cylinder part

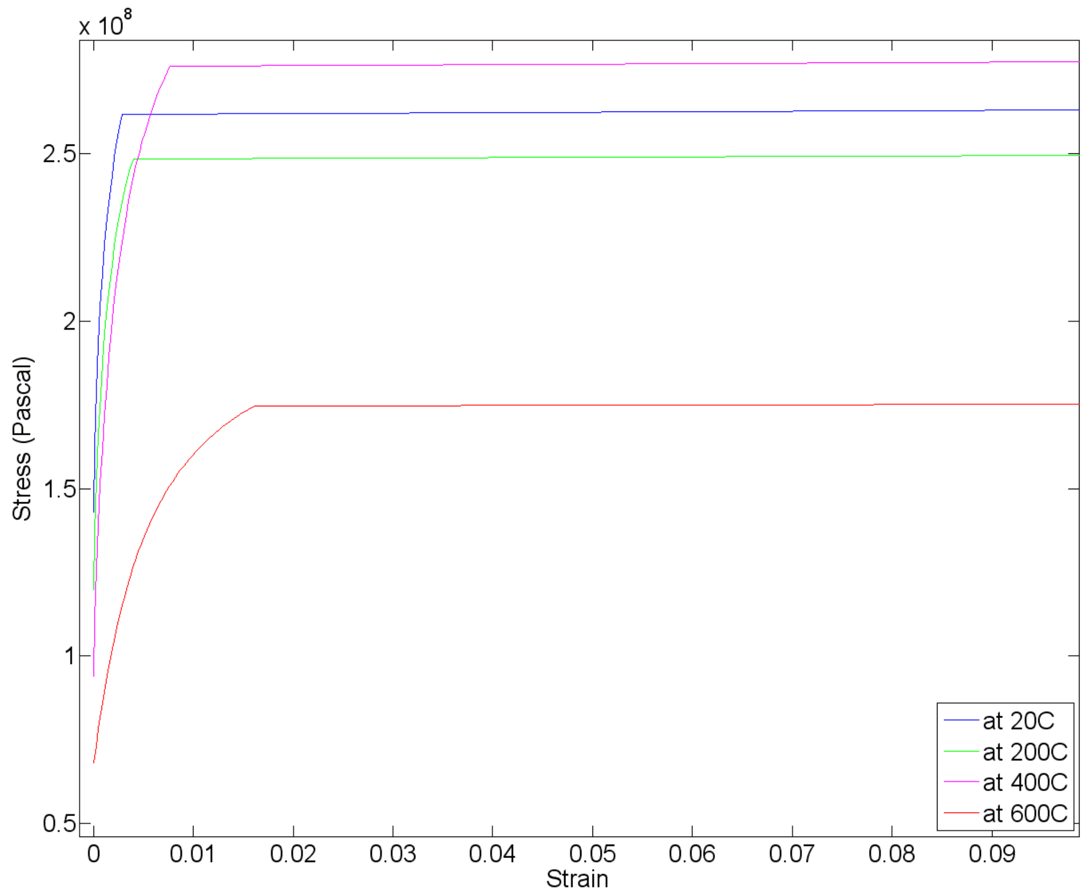


Figure 10.37. Plasticity material data curve for the Cylinder part

3- Sections definition

No modifications needed

4- Sections assignment

No modifications needed

5- Mesh element type

The element type must be changed to 3D-STRESS

Expand the Parts collector in the III-STERSS model → Expand the CYLINDER part item → Right click the Mesh item → Switch Context → Mesh menu → Element Type → Select the whole geometry → Done → Element Library: Standard → Geometric Order: Linear → Family: 3D-STRESS → OK → Done

6- Assembly

No modifications needed

7- Steps definition

The Initial step will be altered later through a modification on the Predefined Fields.

Two new general-static steps will be created to define the analysis of the stresses before and after shake-out. In each one of them, the corresponding .odb file from the thermal analysis must be read into.

The After Shake-Out step must be deleted.

Expand the Steps collector of the III-STRESS model → Right click the "After Shake-Out" step → Delete → Yes

Stress step before shake-out

Right click the Steps collector → Create → Name it "Stress-BSO" → Procedure type: General → From the list select "Static, General" → Continue →

Time period (seconds): 28800 (the same as in the I-BSO model) →

Incrementation Type: Automatic →

Maximum number of increments: 28800 →

Initial Increment size: 10 →

Minimum Increment size: 1E-12 →

Maximum Increment size: 10000 →

→ OK

Stress step after shake-out

Right click the Steps collector → Create → Name it "Stress-ASO" → Procedure type: General → From the list select "Static, General" → Continue →

Time period (seconds): 43200 (the same as in the II-ASO model) →

Incrementation Type: Automatic →

Maximum number of increments: 43200 →

Initial Increment size: 10 →

Minimum Increment size: 1E-12 →

Maximum Increment size: 10000 →

→ OK

8- Predefined Fields definition

The Cast-Initial-Temp field will be modified to be constant and of the same value as in the I-BSO model

Expand the Predefined Fields collector of the "III-STRESS" model → Right click "Cast-Initial-Temperature" → Edit → Distribution: Direct specification → Magnitude: 1400 → OK

9- Interaction Properties definition

There are no interaction properties involved in this analysis. Consequently, the "Conv-HTC" interaction property must be deleted

Expand the Interaction Properties collector of the "III-STRESS" model → Right click "Conv-HTC" → Delete → OK

10- Interactions definition

There are no interaction properties involved in this analysis. Therefore, the "Cast-Ambient-Convection" and the "Cast-Ambient-Radiation" interactions must be deleted

Expand the Interactions collector of the "III-STRESS" model → Right click "Cast-Ambient-Convection"→ Delete → OK

Expand the Interactions collector of the "III-STRESS" model → Right click "Cast-Ambient-Radiation"→ Delete → OK

11- Boundary Conditions

The Cylinder must be constrained in such a way that **no rigid body motions could happen**. Then the translations and the rotations in X, Y and Z ust be constrained. However, it must be able to shrink. To do so, three nodes will be fixed. First, in one of the flat end faces of the cylinder a node will be totally constrained:

Right click the BCs collector of the “III-STRESS” model → Create → Name it “U-123” → Step: Initial → Category: Mechanical → Type: Displacement/Rotation → Continue → Select a node as in *Figure 10.38*→ Done → Check the check boxes for U1, U2 and U3 → OK

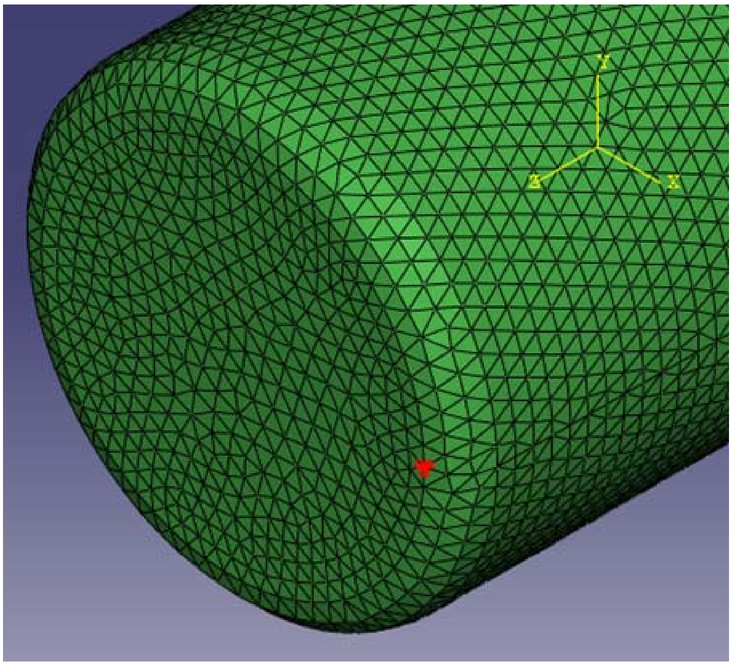


Figure 10.38. Totally constrained node. Boundary Conditions, Stress Analysis

The Edit Boundary Condition window should look as:

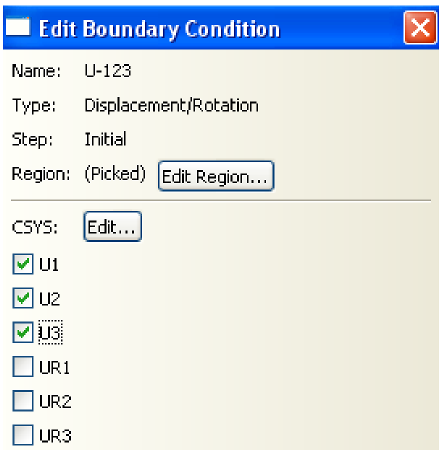


Figure 10.39. Edit Boundary Conditions window for a fully constrained node.

In the same face, another node, aligned with the first one in the direction of one of the axis parallels to the face (in this case the x axis), will be constrained in the other two directions that are not aligned, allowing contraction. In our study case the node will be free in X and fixed in Y and Z:

Right click the BCs collector of the "III-STRESS" model → Create → Name it "U-23" → Step: Initial → Category: Mechanical → Type: Displacement/Rotation → Continue → Select a node as the one with red marks in *Figure 10.40* → Done → Check the check boxes for U2 and U3 → OK

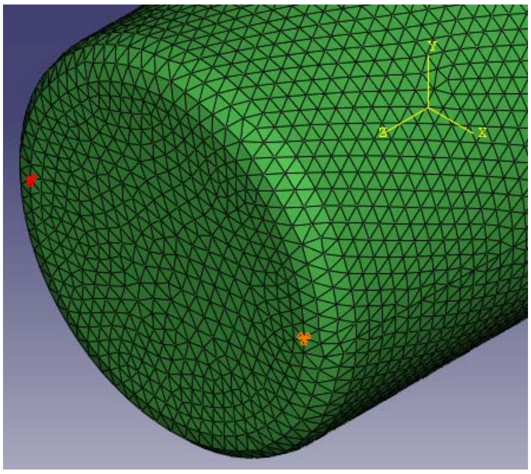


Figure 10.40. Semi-fixed node (red) aligned in the x direction with the totally fixed one

In the opposite end face, a node aligned with the first one in the axis of the length (in this case the Z axis) will be constrained in the other two axes.

Right click the BCs collector of the "III-STRESS" model → Create → Name it "U-12" → Step: Initial → Category: Mechanical → Type: Displacement/Rotation → Continue → Select a node as the one with red marks in *Figure 41* → Done → Check the check boxes for U1 and U2 → OK

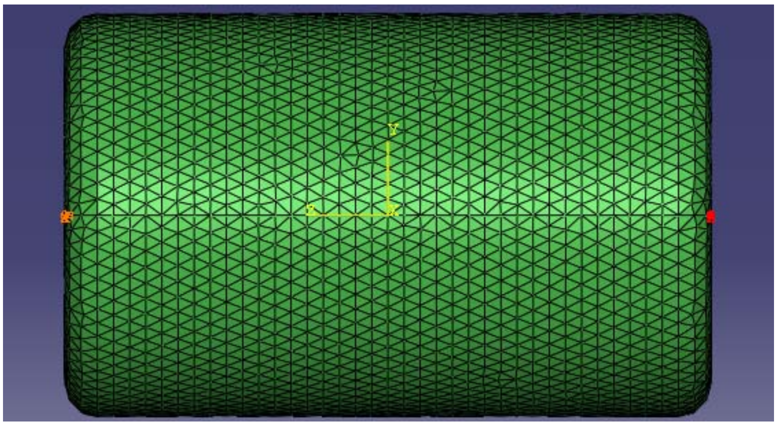


Figure 10.41. Semi-fixed node aligned in z with the totally constrained one

12- Predefined Field Requests

The Stress components and invariants, the Equivalent plastic strain and the Translations and rotations will be requested to be written into the output database.

The existent "Nodal-Thermal-History" field output request must be deleted from this model.

Note: No History Output Request is necessary

Expand the Field Output Requests collector → Right click the "Nodal-Thermal-History" item → Delete → Yes

Right click the Field Output Requests collector → Create → Name it "III-Stress-Output" → Step: Before-Shake-Out → Continue → Set the Edit Field Output Request window as:

Note: When we select the step to which the output request will be applied, the request gets automatically propagated to the following steps.

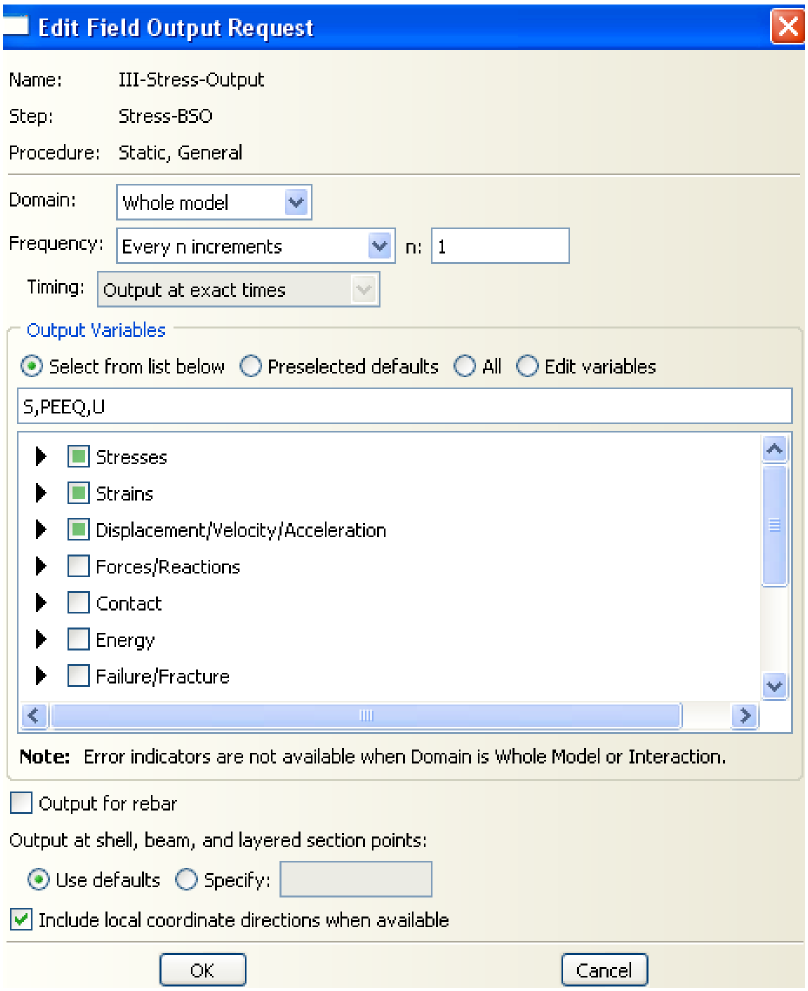
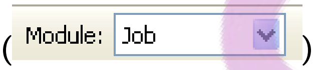


Figure 10.42. Field output request configuration for the III-STRESS model

13- Job Creation

Expand the "Analysis" item in the Models tree → Right click the "Jobs" collector → Create → Name it "STRESS-R1" → Source: Model → In the displayed models list select "III-STRESS" → Continue → Write a short description of the analysis if desired → Job Type: Full analysis → OK

Or

Select the "Job" Module in the context bar (

) → Jobs menu → Create→ Name it "STRESS-R1" → Source: Model → In the displayed models list select "III-STRESS" → Continue → Write a short description of the analysis if desired → Job Type: Full analysis → OK

10.1.2 Calculation

When submitting a Job for analysis, an input (.inp) file is automatically written. The only participation of the user in this phase is in the actual submitting of the Job since the calculation itself is performed automatically by the solver.

To submit a Job for analysis:

Expand the "Analysis" item in the Models tree → Expand the "Jobs" collector → Right click the job to be submitted, e.g. "BSO-R1" → Submit

Or

Select the "Job" Module in the context bar (

) → Jobs menu → Submit → Select the job to be submitted

The status of the Job is always presented next to the Job name in the Analysis tree. For example, if the calculation is running in the solver, the message "running" will appear between parentheses as follow:

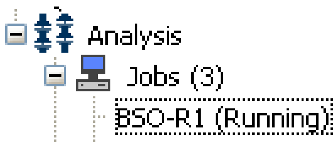


Figure 10.43. Job status

Bestpfe.com

To monitor the progress of an analysis job:

Expand the "Analysis" item in the Models tree → Expand the "Jobs" collector → Right click the job that is running → Monitor

Or

Select the "Job" Module in the context bar → Jobs menu → Monitor → Select the job to be monitored

10.1.3 Post-Processing

10.1.3.1 Results Visualization

In this step of the process we will obtain a graphical representation of the results through colored spectrums applied to the model, and curves which values can be written to text files for further comparisons.

The results are presented into the Visualization module and read from the .odb file. They can be read at any moment of the calculation since each time an increment is completed, Abaqus write the results to the .odb being accessed and an update can be performed.

First we must load our .odb file into the Visualization module.

10.1.3.1.1 Loading the Output Data Base

Expand the "Analysis" item in the Models tree → Expand the "Jobs" collector → Right click the job you want to see results from (e.g. BSO-R1)→ Results

Or

Select the "Visualization " Module in the context bar → File menu → Open → Browse and select the .odb file from which you want to see results → OK

The .odb is placed in the Results tree under the Output Databases collector:

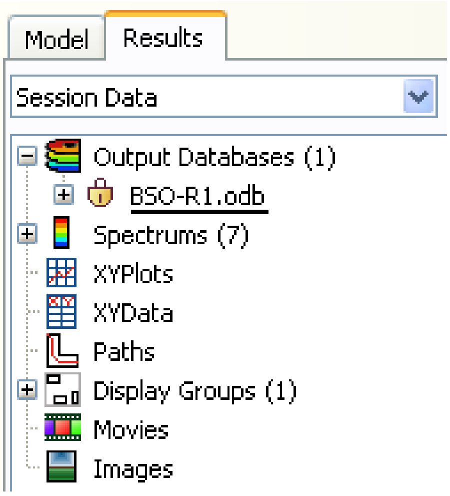
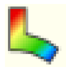


Figure 10.44. An .odb in the Results tree

They are different types of plots that allow us to see results (for details, see Abaqus/CAE User's Manual). We chose the "Plot Contours on Deformed Shape" option (). This option, represent the values of our selected analysis variable as colored faces.

It can be selected from the toolbox:

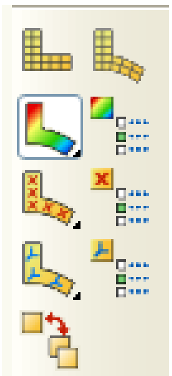


Figure 10.45. "Plot Contours on Deformed Shape" button (selected)

Or from the menu:

Plot menu → Contour → On Deformed Shape

BESTPFE.COM
List of research project topics and materials

10.1.3.1.2 Cut Sections

We can display a cut section of our model using the “View Cut Manager” tool from the toolbox:

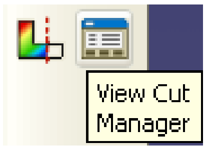


Figure 10.46. “View Cut Manager” button (selected)

Or from the menu:

Tools menu → View Cut → Manager

From the View Cut Manager window (see *Figure 10.47)*, we can select one of the tree default planar cut sections or we can create our own (for details, see the Abaqus/CAE User’s Manual). As we adjust the cut, it is applied in real time to the model. The settings and cut selection is memorized and it can be activated or deactivated with the “Activate/Deactivate View Cut” tool () in the toolbox.

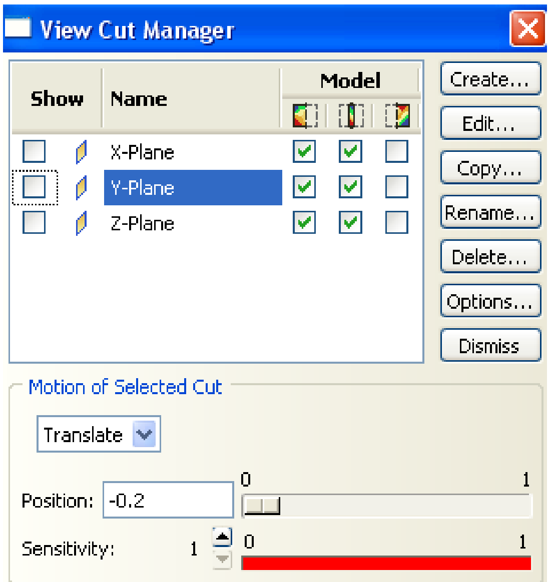


Figure 10.47. View Cut Manager window

10.1.3.1.3 Removing a part from the viewport

When we have more than one part in a model, as in the I-BSO where we have the Mold and the Cylinder, we may face the necessity of remove one part from the viewport of the Visualization module to have a better view of the results of another one. A way to achieve it is:

Expand the Output Databases collector in the Results tree → Expand the .odb item of your analysis Job (e.g. BSO-R1.odb) → Expand the Instances collector → Right click the part you want to remove (e.g. MOLD-1) → Remove

This is a Boolean operation applied just to the viewport, which means that the part is not deleted, is just not displayed.

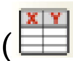
If is necessary to display the part again, the same previous procedure will do it by selecting "Add" instead of "Remove" at the end.

This procedure is a shortcut of the capabilities of the "Display Group" option, for more information see Abaqus/CAE User's Manual.

10.1.3.1.4 Creating X-Y Curves

We plot the nodal history of an analysis variable by using the XY Data tool.

To plot the thermal history of a node in our cylinder from the BSO-R1 analysis for example, proceed as follow:

Press the "Create XY Data" button from the tool box () → Source: ODB field output → Open the Variables tab → Position: Unique Nodal → Check the checkbox for NT11: Nodal temperature → Open the Elements/Nodes tab → Select the node/s from which you want to see results using one of the listed methods → Plot

The XY Data create option can also be accessed from:

Tools menu → XY Data → Create

To confirm that the displacement boundary conditions where satisfied in the stress analysis (STRESS-R1), we plot the displacement results in the fixed directions of the nodes where we applied the constraints as follows:

Press the "Create XY Data" button from the tool box → Source: ODB field output → Open the Variables tab → Position: Unique Nodal → Click the arrow next to U: Spatial displacement → Select the degrees of freedom you want to check in your node/s → Open the Elements/Nodes tab → Select the node/s from which you want to see results using one of the listed methods → Plot

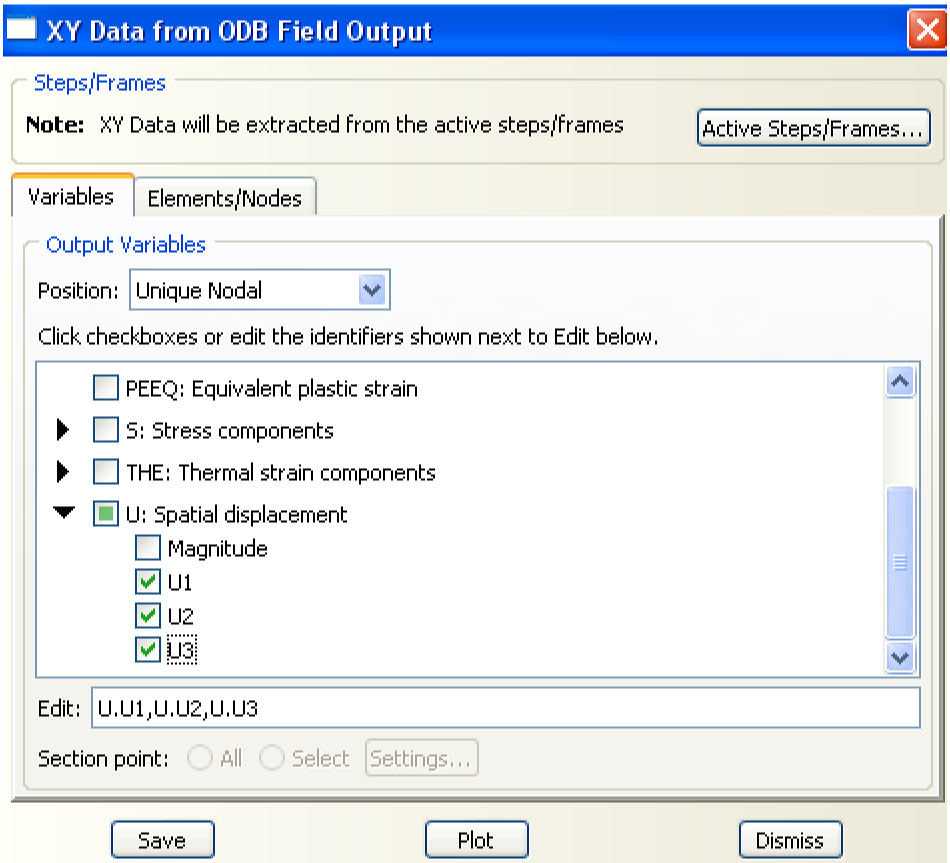


Figure 10.48. Variables tab in the XY Data from ODB Output window

10.1.3.2 Results Preparation for Comparison

When we need to compare Abaqus results with Magmasoft results, the approach consist in export the curves of the Abaqus analysis results to Abaqus Report files (.rpt) from where the X-Y data of the curves can be obtained in a table form at that can be used in a software as Matlab, where the actual comparison of the data from both sources can performed.

If instead, the results to compare belong to one or more Abaqus .odb files, the comparison can be made in the same Visualization module through the “XY Data Manager”.

In section 10.1.3.1 we saw how to plot a curve. We can create an X-Y data report of the curve being displayed in the viewport, but also, instead of plotting the curves, we can save them to the “XY Data Manager” without needing to see them and then select from a list which curve/s you want to export to the report file. If you want to compare curves from two different .odb files, is mandatory to save them first prior to the combination of the results.

Note: The saved curves just remain in memory during the session.

10.1.3.2.1 Saving a curve

The setup of the curve must be carried out as explained in section 10.1.3.1 but at the end, we just press the "**Save**" button from the "XY Data from ODB Field Output" window (see *Figure 10.49*). Is not relevant if the curve have been plotted or not.

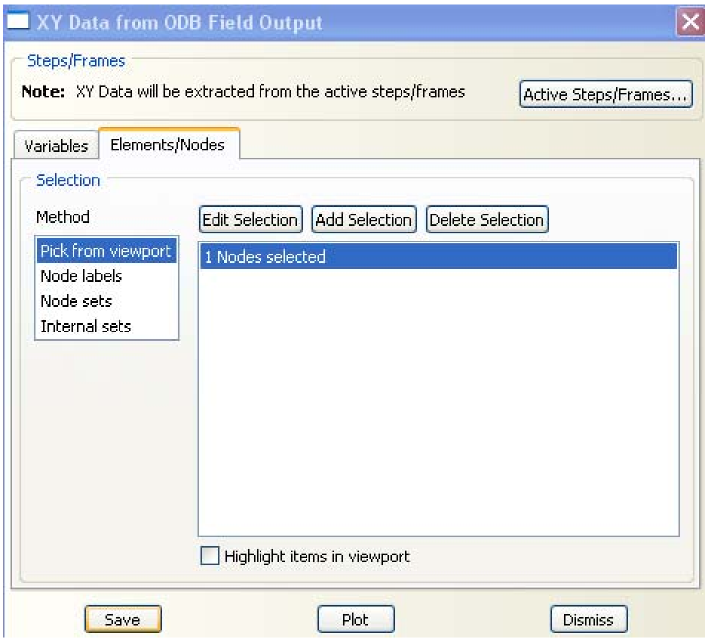


Figure 10.49. Elements/Nodes tab in the XY Data from ODB Output window

10.1.3.2.2 Exporting the curves

Report menu → XY → Select from the list one or more curves to export to the .rpt file → Switch to the Setup tab → Assign a mane → Select a destination folder with the Select... button → Set desired options → Press Apply to create the file and continue exporting results, or OK to create it and finish

Note: The "Append to file" button allows writing more than one result to the same .rpt file. When exporting a result to an existent report file, if "Append to file" is checked the result will follow the previous one in the .rpt, if is not checked the result will overwrite the previous one.

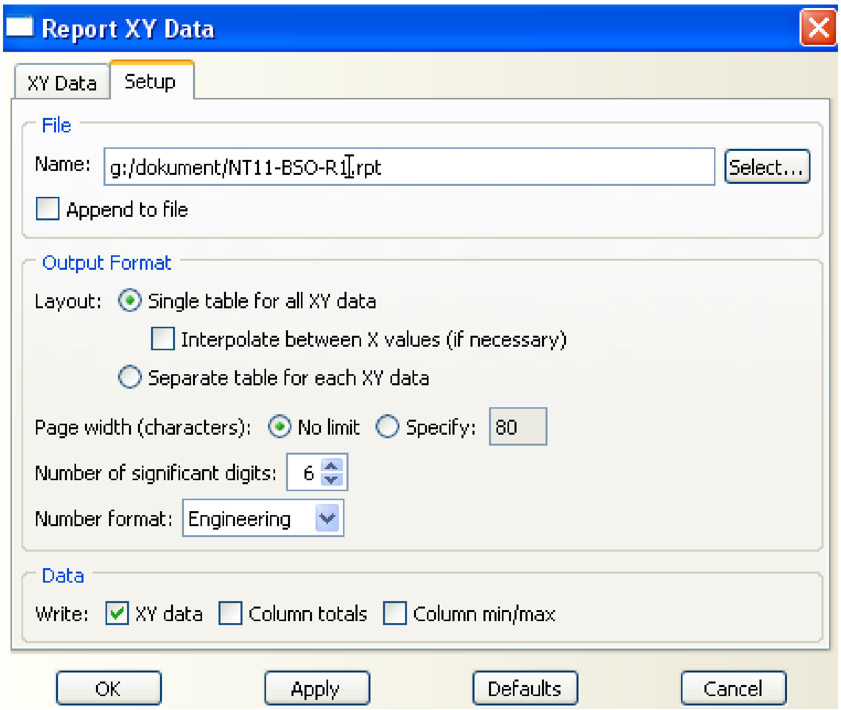


Figure 10.50. Report XY Data window

10.2 Magmasoft Implementation

The same general steps presented in *Figure 3.1* that where followed for the Abaqus implementation will be followed for the Magmasoft one. For reference purposes, the figure is presented again:

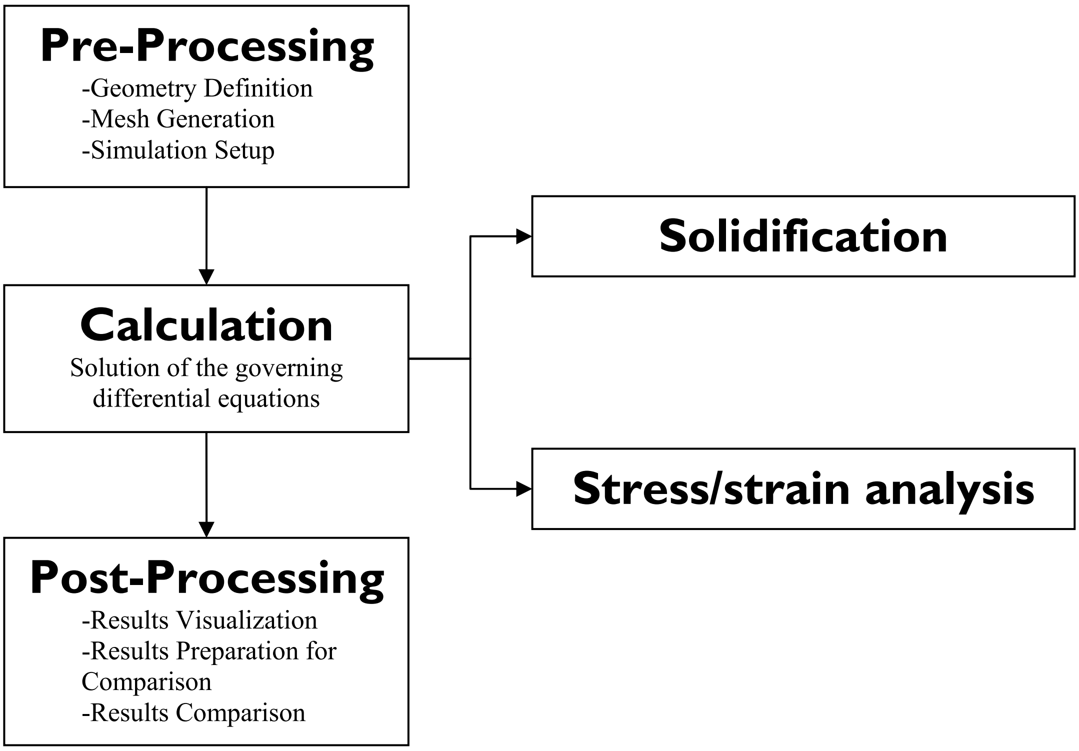


Figure 10.51. Steps sequence for the residual stress analysis

10.2.1 Pre-Processing

10.2.1.1 Geometry Definition

The mesh to be used will be generated in Magmasoft and not in Hypermesh as we did in the Abaqus implementation. In this case, the model to be imported is the model of the casting. The mold is created directly in the Magmasoft preprocessor.

A material must be assigned to the geometries during the Geometry Definition step. However, the details about the material data are to be edited later during the Simulation step.

Overview

1- Importing the model

2- Assigning a material type to the Cylinder

3- Creating the mold and assigning a material type

5- Placing cooling curves points and stress curves points

1- Importing the model

Create a new Project:

Project menu → Create Project → Project Mode: Shape Casting / Batch Production → Name it "Magma-Cylinder" → Press the Return key* → OK

*When running in Windows, Magmasoft usually don't recognize the Enter key of the numeric key pad, so use the Return key instead.

Since the mesh will be generated in Magmasoft, the .stl file created in ProEngineer is directly imported into the Magmasoft preprocessor. The model to be imported is the "Magma_Cylinder.stl" created in ProEngineer:

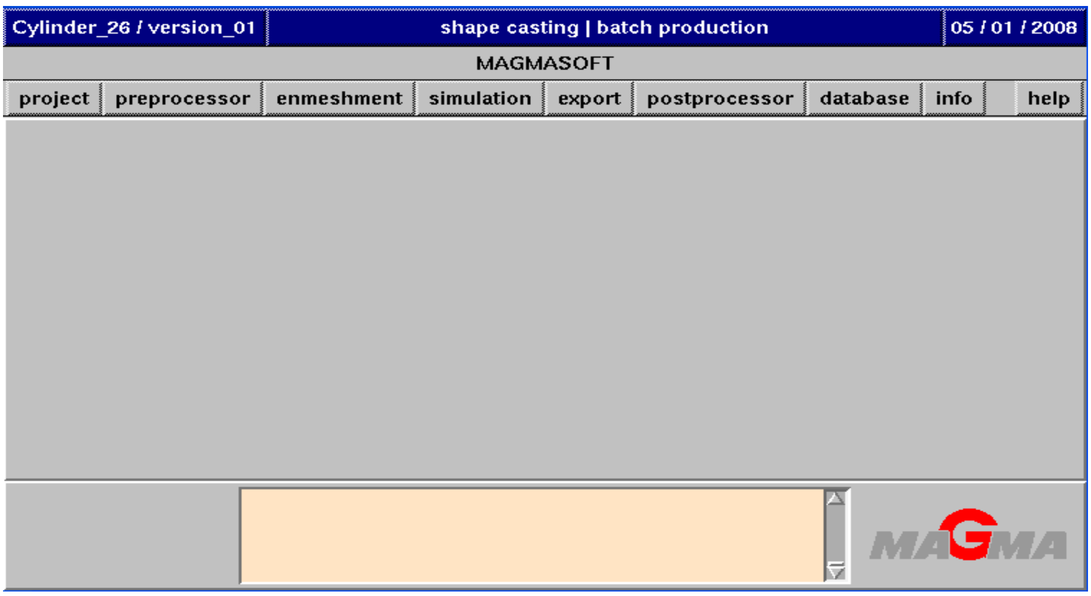


Figure 10.52. Typical appearance of the Magmasoft main interface

Click the preprocessor button on the main window → File menu → LOAD SLA → Browse the "Magma_Cylinder.stl" model → Open → Ok

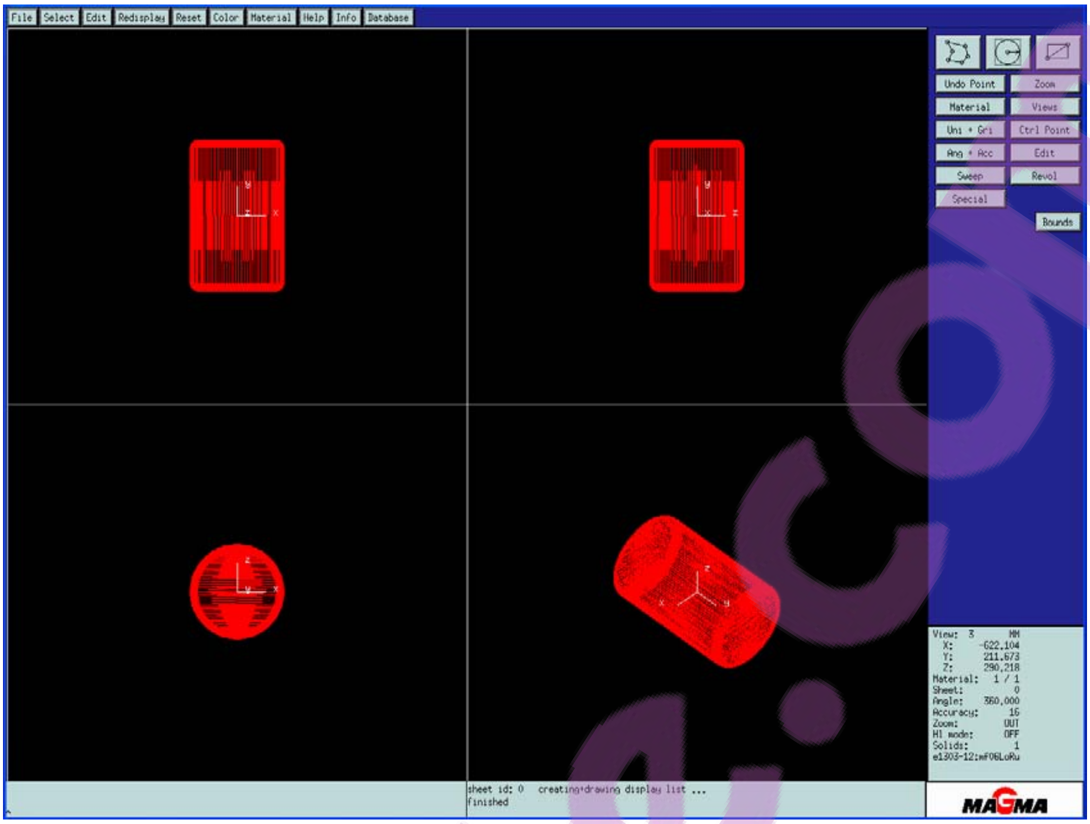


Figure 10.53. Magma_Cylinder.stl file imported into the Magmasoft preprocessor

2- Assigning a material type to the Cylinder

Select menu → Volume → Select the volume "SLA.Magma_Cylinder" from the list → Return → Material → CAST → CHANGE MAT

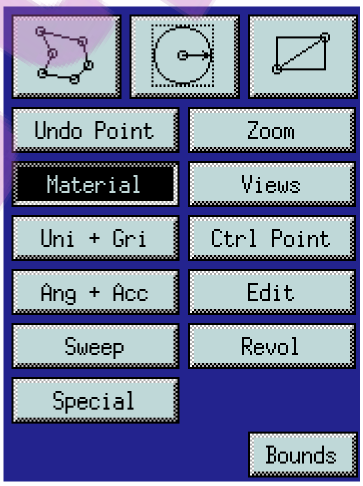


Figure 10.54. Location of the Material button in the Preprocessor interface

Bestpfe.com

3- Creating the mold and assigning a material type

The material type that happens to be selected before the creation of a new feature is the one to be assigned to it. Therefore, to not need to change the material type after creating the mold box, the right material type (sand mold for us) must be selected first.

Material → SANDM

There is more than one way to define the mold box (see for example the “begin box” and the “set cube” commands in the Magmasoft Online Help documentation); we do it with the “set cube” command, defining a body diagonal by introducing the coordinates of two opposite corners via the keyboard.

Set Cube x1 y1 z1 x2 y2 z2 → Return

Example:

```
set cube 0 0 0 100 100 100
```

→ Return

This results in a cube with a corner in the origin.

Magmasoft automates the boolean operation that subtract the casting part from the mold box creating the cavity, but the decision of which volume must be subtracted from which one is made by the order of the volumes in the volumes list.

In this automatic cavity creation, a volume that appear later in the list is removed from the volume that appear earlier in the list. Since our first volume was the Magma_Cylinder it appears first in the list, but we need the cylinder to be removed from the mold and not the other way around. Therefore, the mold volume must be placed before the casting in the list of volumes.

Select menu → Volume → Select the volume corresponding to the mold → Click the upper entity selector button (above the “Move Before” and “Move After” buttons)→ Select the volume corresponding to the casting → Click the lower entity selector button (at this point it must look like *Figure 10.55*)→ Move Before **→** Return

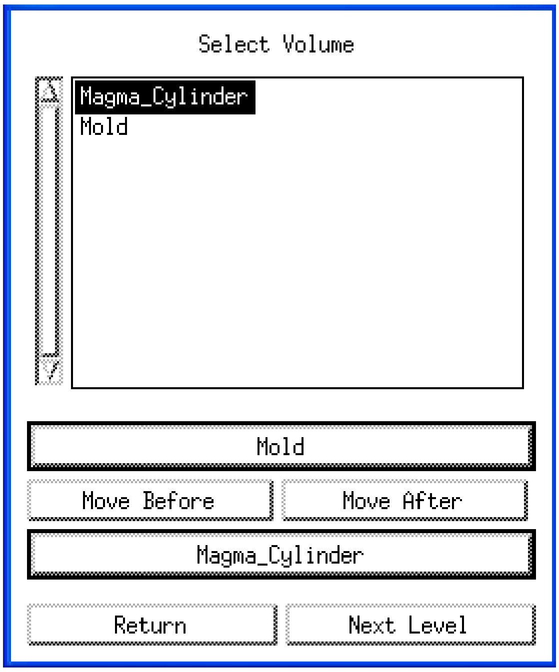


Figure 10.55. Entity Selections windows with the volumes selected prior organizing

Save your work

File menu → Save Active → Name it "M-Cylinder" → Save

BESTPFE.COM
List of research project topics and materials

10.2.1.2 Mesh Generation

First exit the Preprocessor

File menu → Exit

Click the **enmeshment** button in the Magmasoft main interface → Choose the automatic method → Assign the desired number of elements for the whole model (casting and mold) → **generate** → **dismiss**

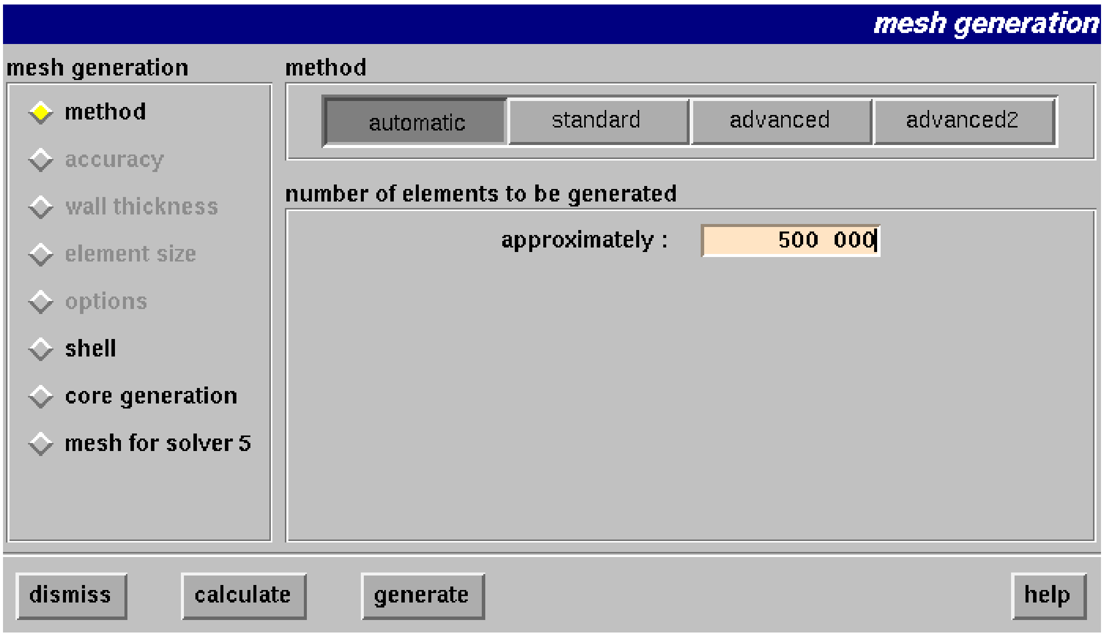


Figure 10.56. Magmasoft mesh generation window

Note: Since Magmasoft use a Control Volume Finite Difference formulation, where there is just one node in the center of each element, the number of elements are equivalent to the number of nodes in Abaqus.

10.2.1.3 Magmasoft Simulation Setup

Click the simulation button in the Magmasoft main interface → Set the "process mode" window as *Figure 10.57* → Ok

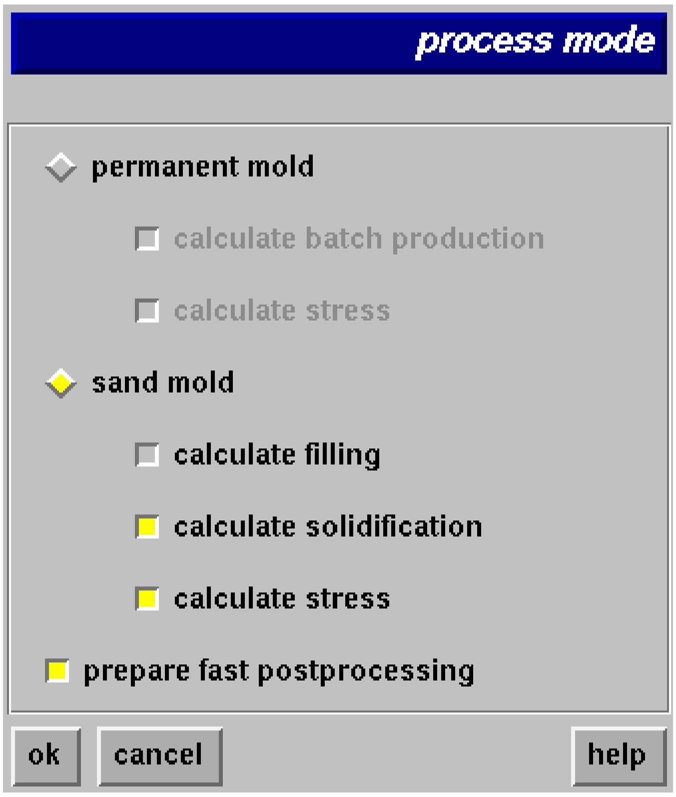


Figure 10.57. Process mode window

The next step consists in assign a material to the casting and the mold. Since our material data differ from the one in the Magmasoft database, we will copy a material for the casting, specifically GL-150, and for the mold, COLDBOX, from the Magma database to the Project database where they will be modified and from where they will be used.

In the "material definitions" window select the "Cast Alloy" material class as in *Figure 10.58*→ select data → Database: Project → Group: Cast Alloy → MAGMAdata ... (see *Figure 10.59*)→ Import menu → From MAGMA → From the Choice list select: GL-150 → > → Choice: COLDBOX → > → Import

Now the materials are in the Project database.

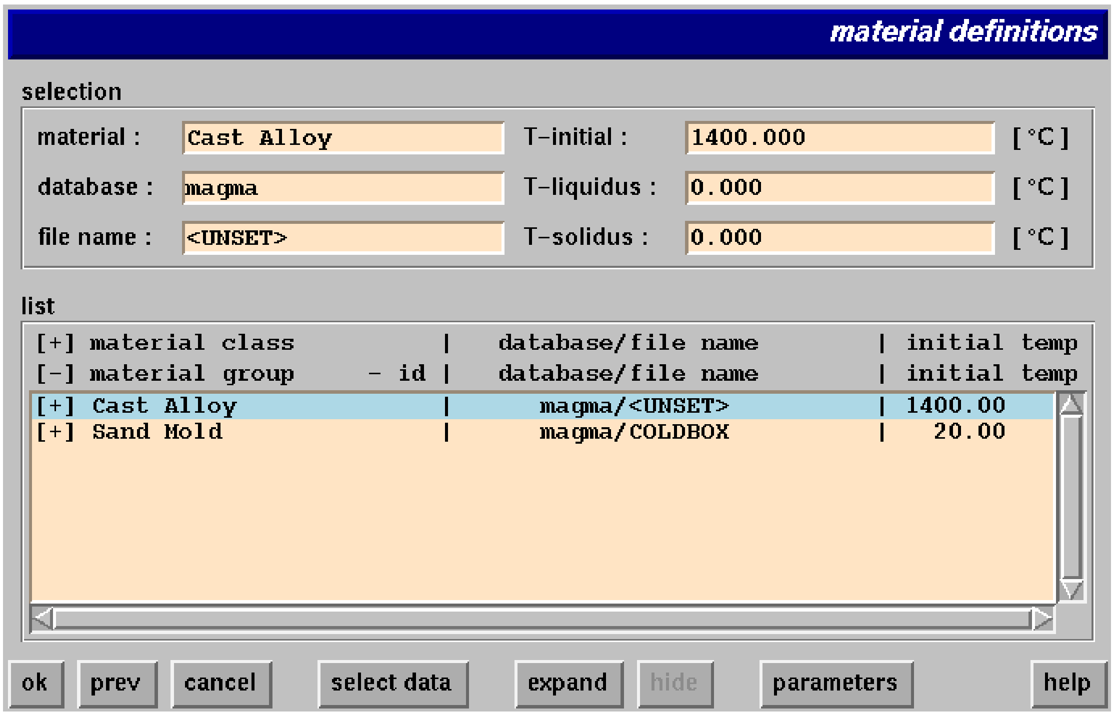


Figure 10.58. The "material definitions" window

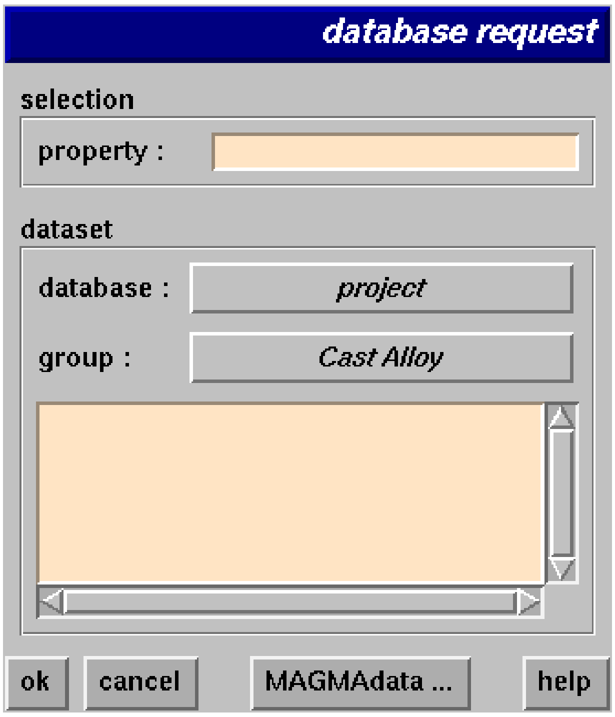


Figure 10.59. Database request window

The material data can be edited from the "MAGMAdata" window for the project database that automatically appears after importing the materials. If you can not see the MAGMAdata window, it can be accessed from the **database** button in the Magmasoft main interface and then select Project from the Database menu.

To edit the casting material:

> Select GJL-150 from the MAGMAdata window as in *Figure 10.60* → **Edit** → From the Edit menu select the parameter to be edited and input the correct data → Data menu → Save → Close → Database menu → Quit

For details about the correct material data, see the Appendix section 10.4.

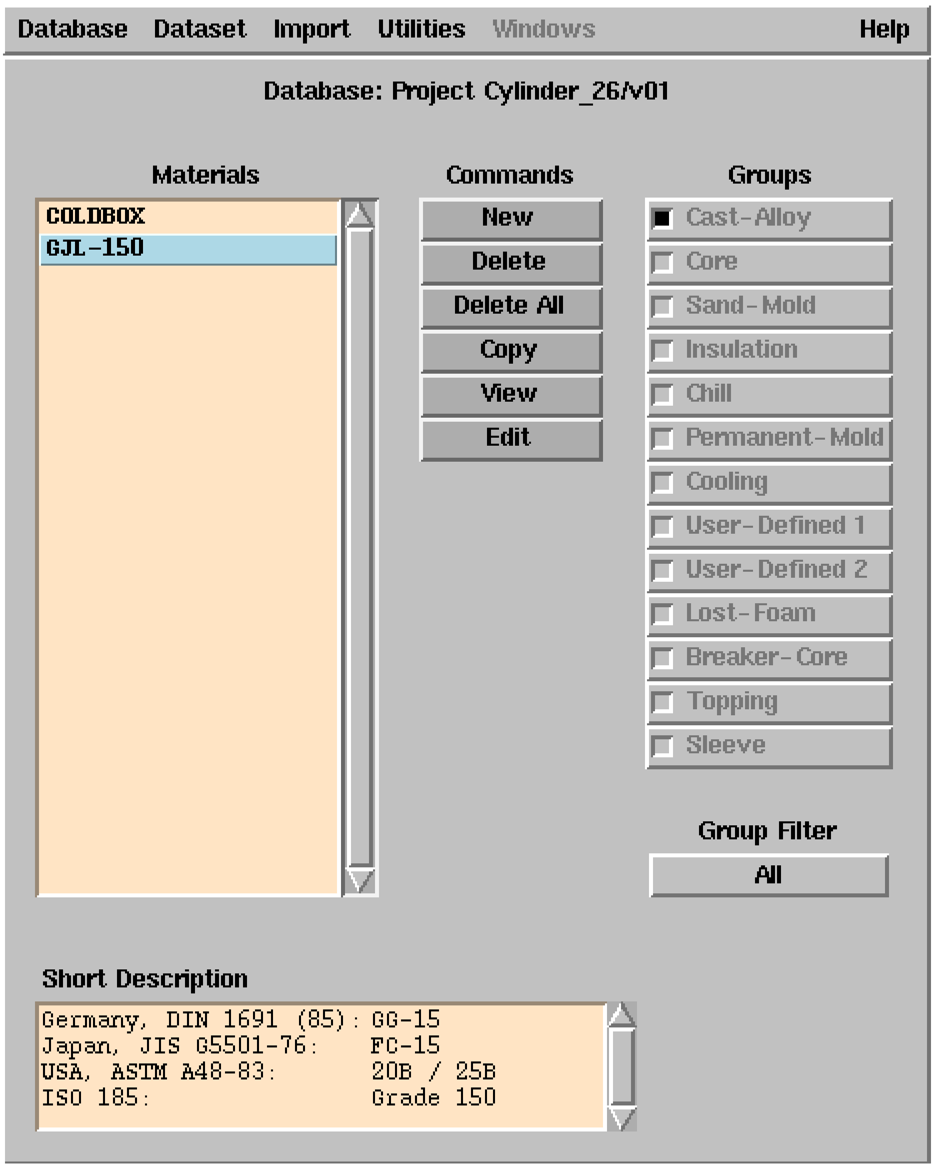


Figure 10.60. MAGMAdata window for the project database

Notice that Magmasoft tells us the units being used for each material data. See for example *Figure 10.61*.

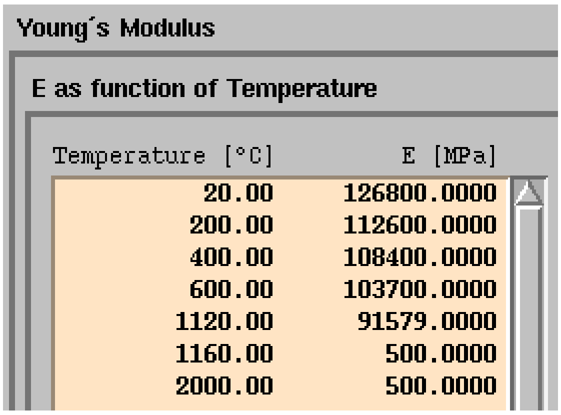


Figure 10.61. Default Young's Modulus for the GJL-150 material

The database request window should now look like:

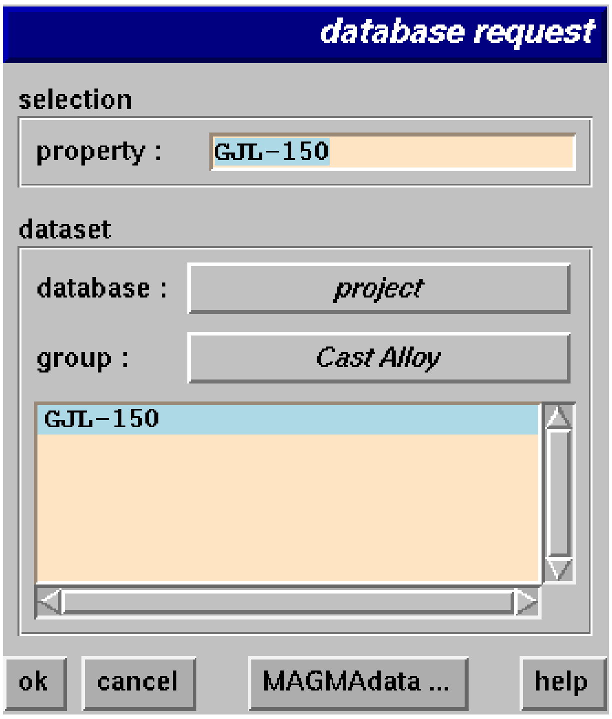


Figure 10.62. GJL-150 material selected from the project database in the database request window

Press the ok button in the database request window

Similarly, assign and modify the material for the mold as follow:

In the "material definitions" window select the "Sand Mold" material class shown in *Figure 10.58*→ select data → Database: Project → Group: Sand Mold → MAGMAdata ... → Select COLDBOX from the MAGMAdata window shown *Figure 10.60* → Edit → From the Edit menu select the parameter to be edited and input the correct data → Data menu → Save → Close → Database menu → Quit → ok

Press ok in the material definitions window.

Let's now select the conductive heat transfer coefficient between the casting and the mold:

From the "heat transfer definitions" window, select the "Cast Alloy" material class as in *Figure 10.63* → select data → Database: MAGMA → Group: Constant → Select C1000.0 from the list → ok → Press ok in the heat transfer definitions window.

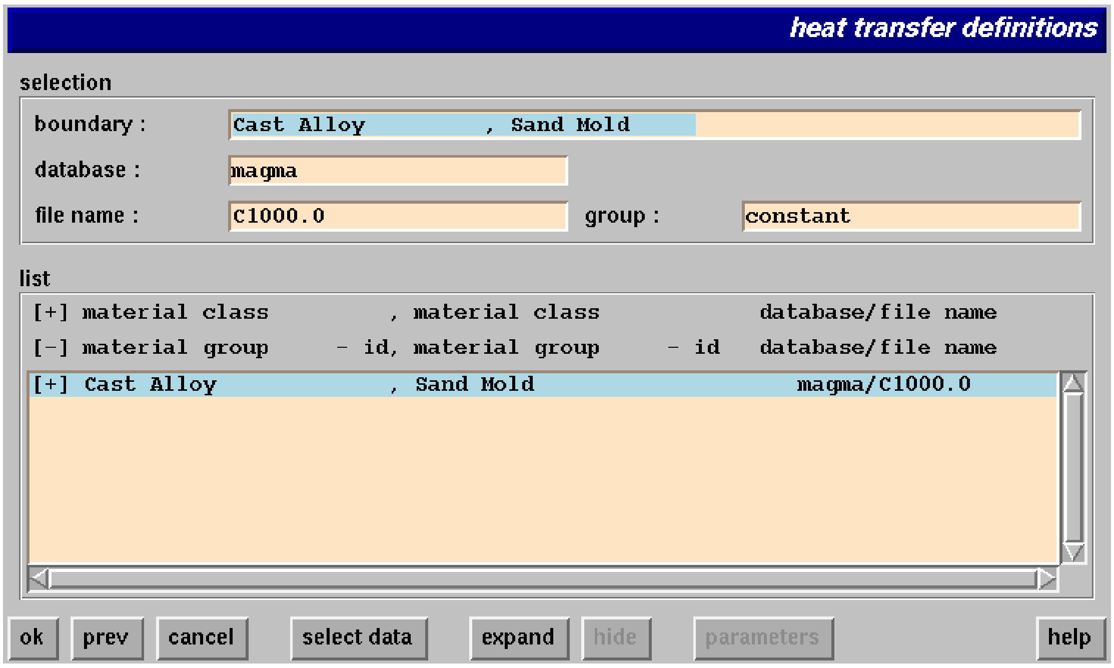


Figure 10.63. Heat transfer definitions window

Set the “Options” window as in *Figure 10.64* → **parameters...**

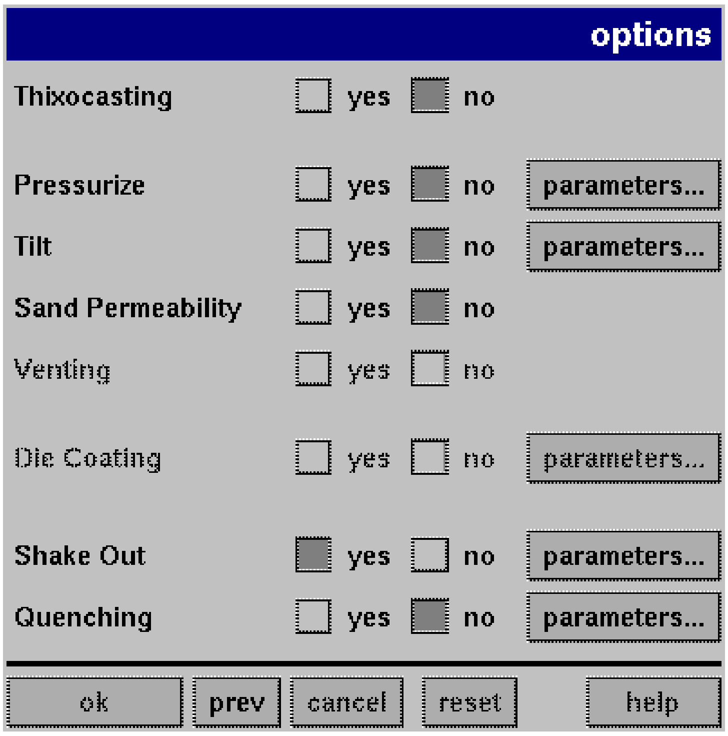


Figure 10.64. Options window

Select “Sand Mold” from the identifier list in the “shake out definitions” window → **options**

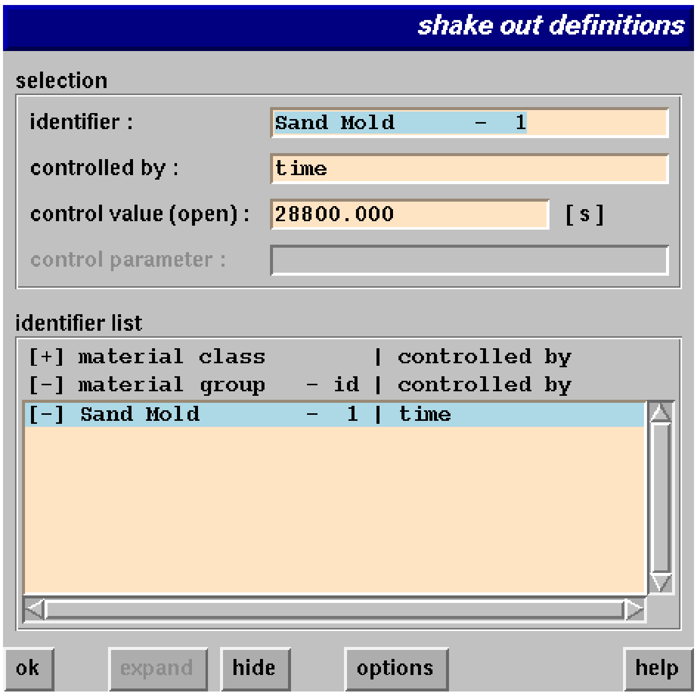


Figure 10.65. Shake out definitions window

Controlled by: Time → Control value (open): 28800 → Ok → Ok → Ok.

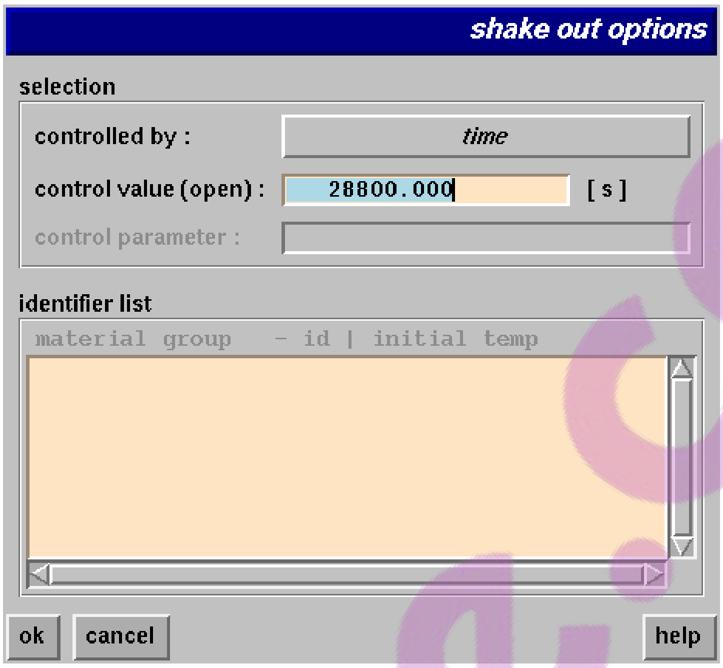


Figure 10.66. Shake out options window

In the "solidification definitions" window select edit → Select "delete all" → Yes → In the "storing data definitions" window set the "input data" option as time → To save results each X number of time steps, the request must be written in the input field in the form "From(time in seconds) To(time in seconds) X(increment)", see *Figure 10.67* → Press the Return key → Ok

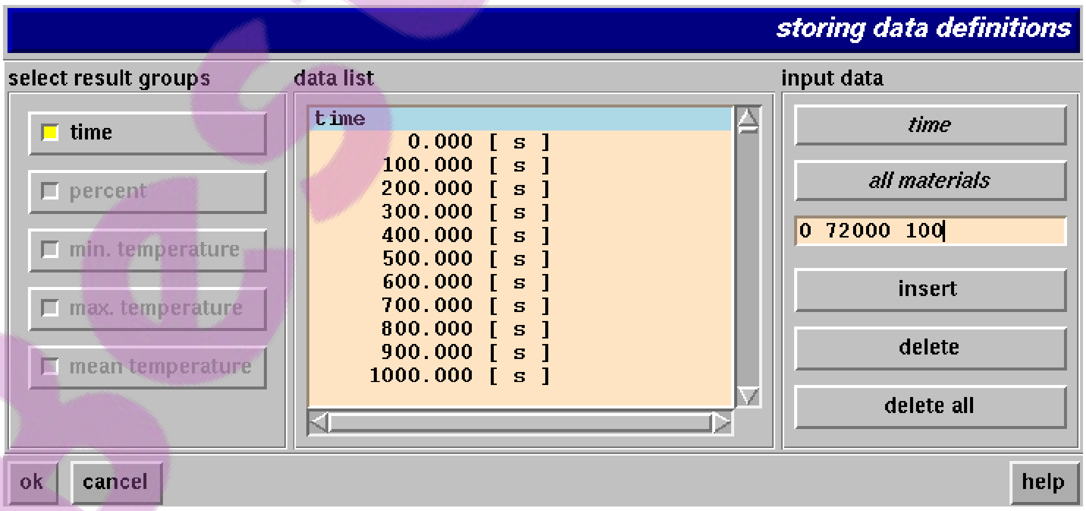


Figure 10.67. Storing data definitions window

We are not interested in the feeding simulation, then is necessary to ensure that "calculate feeding" is switched to NO in the "solidification definitions" window as in *Figure 10.68*

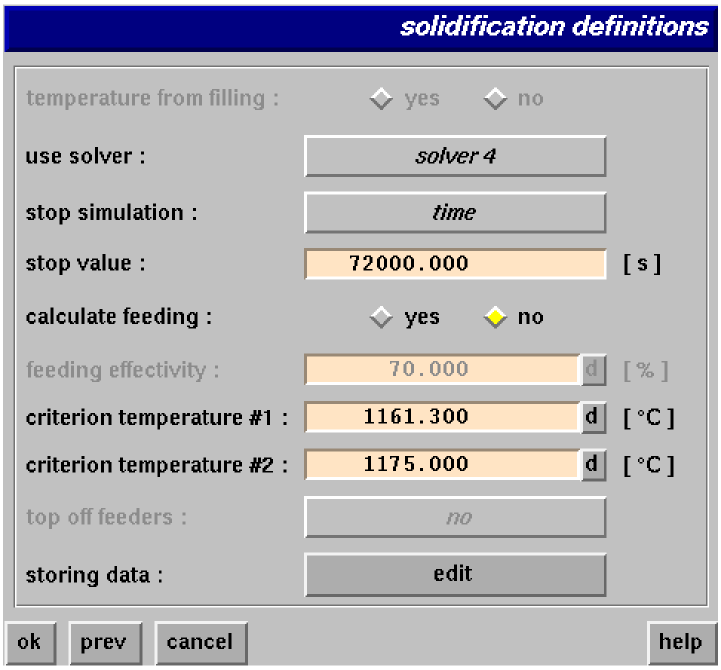


Figure 10.68. Solidification definitions window

Set all the options to NO in the "stress simulation options" window as in *Figure 10.69* → Ok

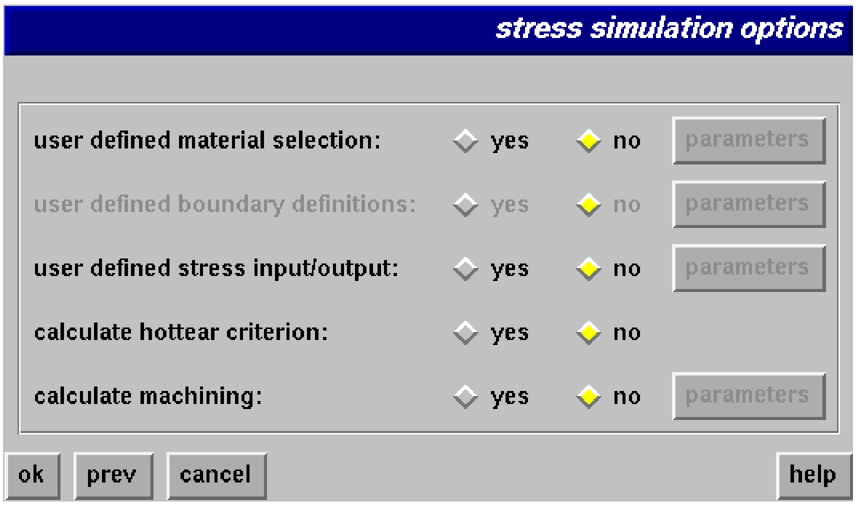


Figure 10.69. Stress simulation options window

In the "fast postprocessing preparation" window check the *solidififcation*, *stress*, *add on mesh* and *criteria(fill, solid)* options as in *Figure 10.70* → Ok

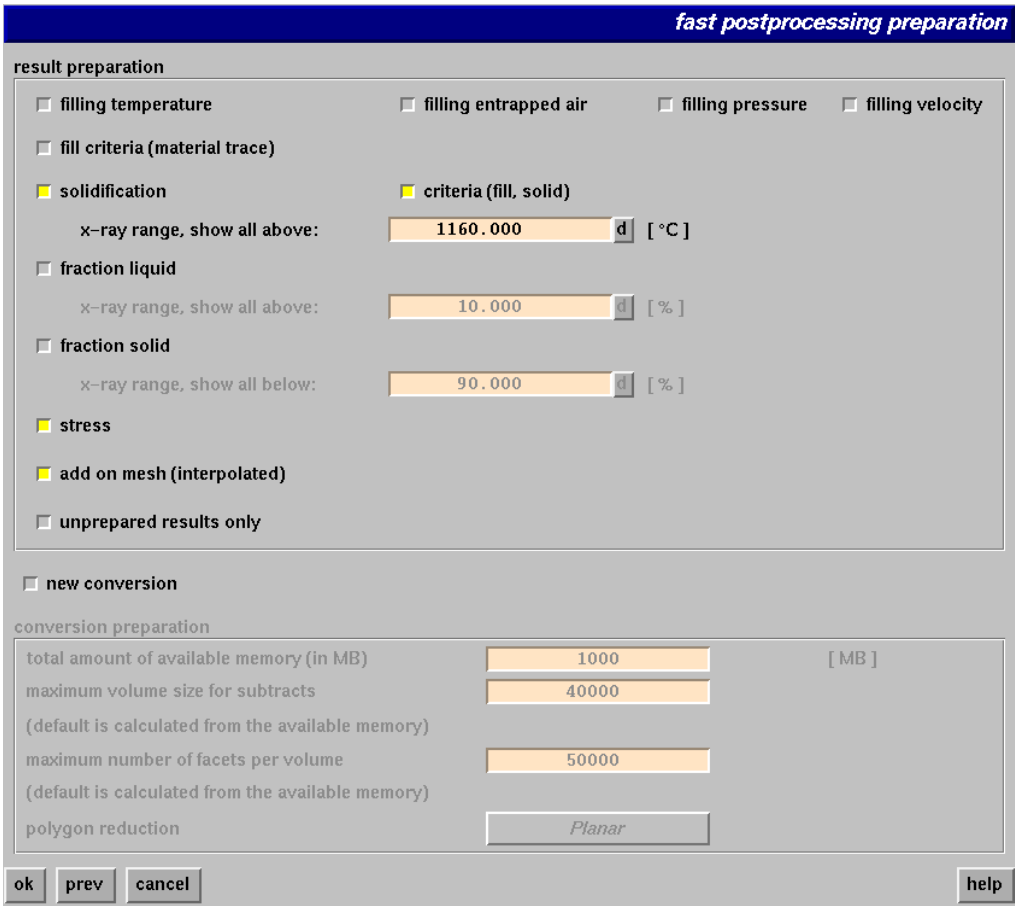


Figure 10.70. Fast postprocessing preparation window

10.2.2 Calculation

The calculation is performed automatically by the solver, is just start the simulation what is needed from the user.

After selecting **ok** in the "fast postprocessing preparation" window, the "online job simulation control" window appears.

To begin the calculations of the analysis press the **start** button.

Once the calculation is finished press **dismiss**.

BESTPFE.COM
List of research project topics and materials

10.2.3 Post-Processing

10.2.3.1 Results Visualization

The graphical representations of the results are accessed through the Postprocessor module. Concerning our interest, they can be colored spectrums applied to the model or curves of the history of a variable in a point specified in the Preprocessor. This curves values can also be written to text files for further comparisons.

To access the post processing module:

Press the postprocessor button in the Magmasoft main interface → On geometry

The fist time that you choose **Postprocessor → On geometry** for the current project version, Magmasoft initiates a geometry conversion process to the ACIS® format and the “ACIS(R) converter window” appears. Here you must enter parameters for the control of the conversion process. The size of the geometry defines the duration of this process.

For our case, set the “ACIS(R) converter window” as in *Figure 10.71* → OK → Next

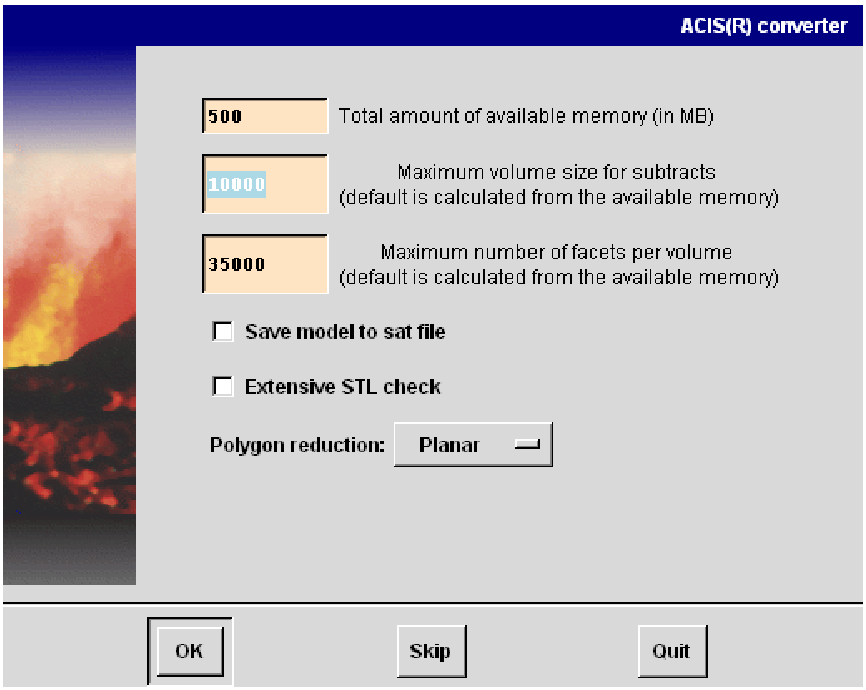


Figure 10.71. ACIS® converter window

For details about this conversion parameters see the MAGMA Online Help documentation,.

The next time that the Postprocessor get opened, it will jump to the Postprocessor main interface. For our project, it opens by default as *Figure 10.72*

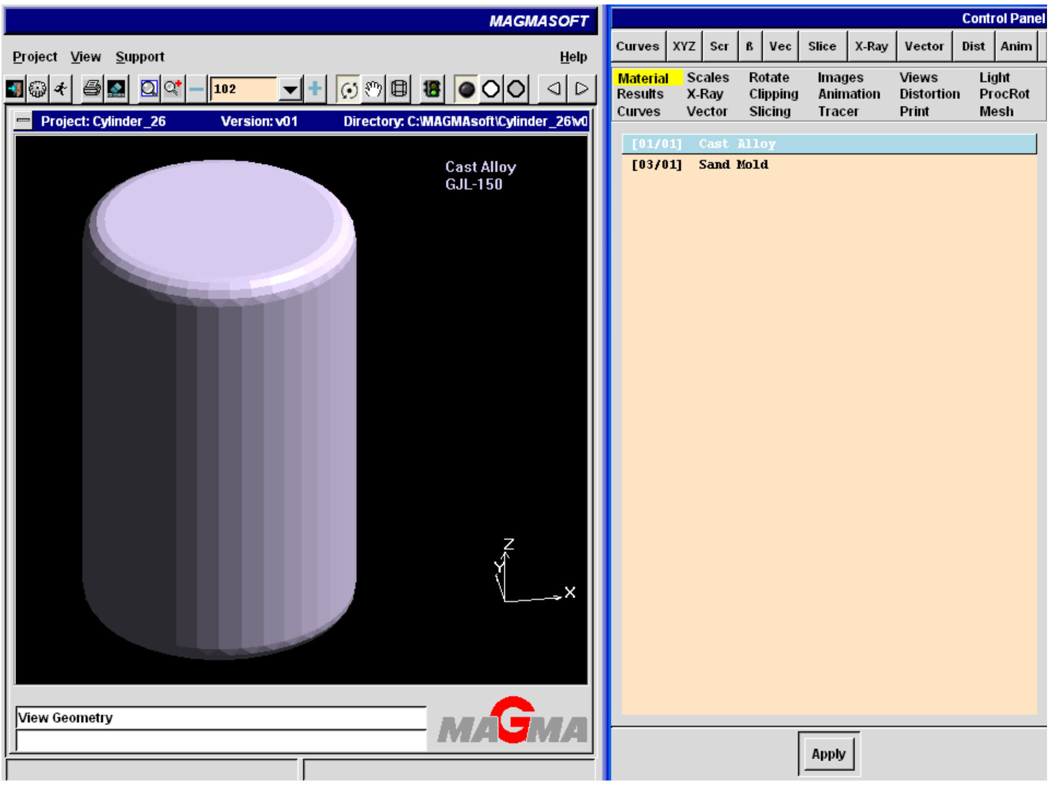


Figure 10.72. Postprocessor main interface

To see the thermal results:

Select "Results" from the Postprocessor's Control Panel window → From the Results list double click "Solidification" → double click "Temperature" → From the list of thermal results that appear, double click the one that correspond to the time that you are interested in

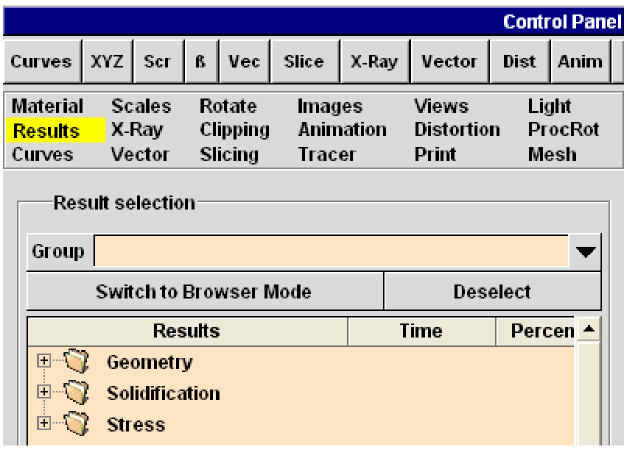


Figure 10.73. Results tab selected in the Postprocessor's Control Panel window

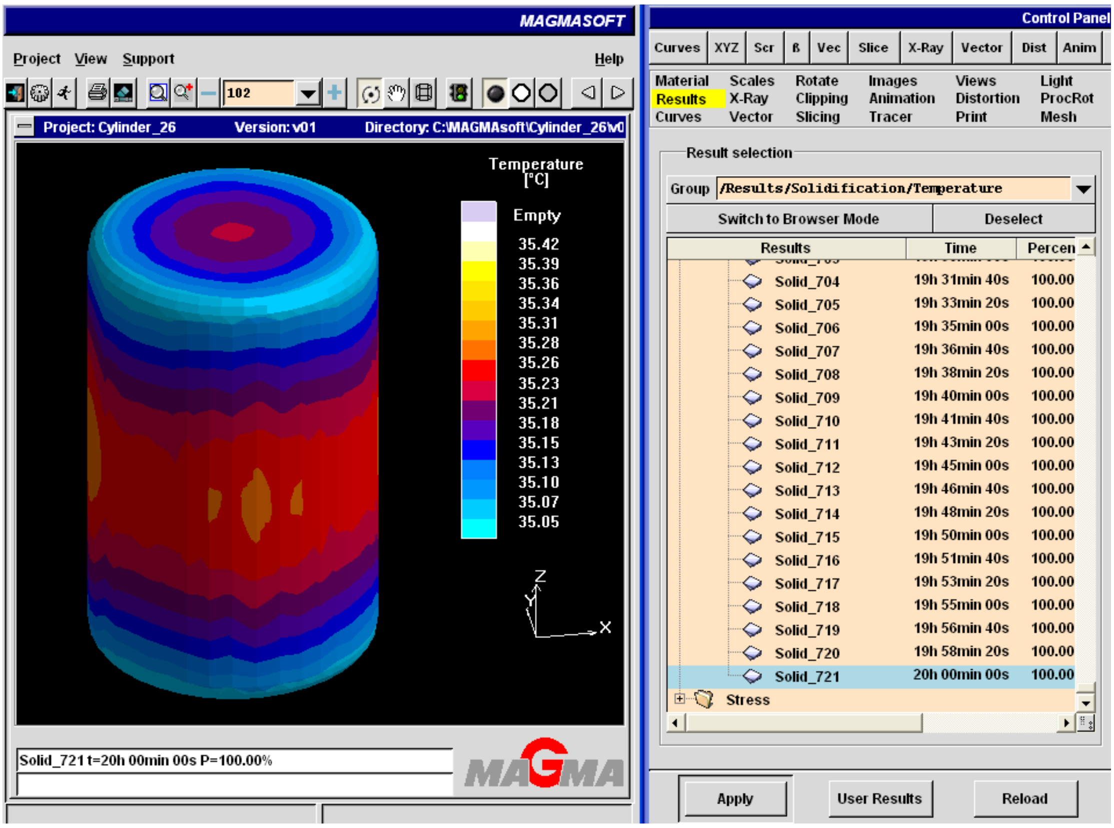


Figure 10.74. Temperature field result displayed in the Postprocessor's main window

Similarly, to see the stress results:

> Select "Results" from the Postprocessor's Control Panel window → From the Results list double click "Stress" → Under the Stress item, double click "Stress" → From the list of stress results that appear, double click the one that correspond to the stress type and time that you are interested in

10.2.3.1.1 Creating a cut view:

> Select "Slice" from the Postprocessor's Control Panel window → Check the check box for the "Activate / deactivate slicing dialog" →Choose a Slice direction → The cut is displayed in the main window, so move the slide bar to adjust the cut

The slide bar can be moved with the mouse or the left and right arrow keys of the keyboard.

The cut view remain is applied to all the results that you display in the main window until the "Activate / deactivate slicing dialog" check box get unchecked again.

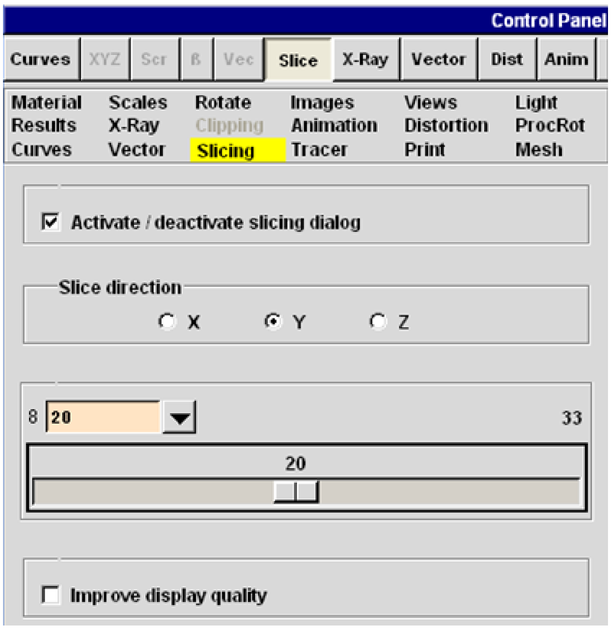


Figure 10.75. Cut view setting with the Slice functionality

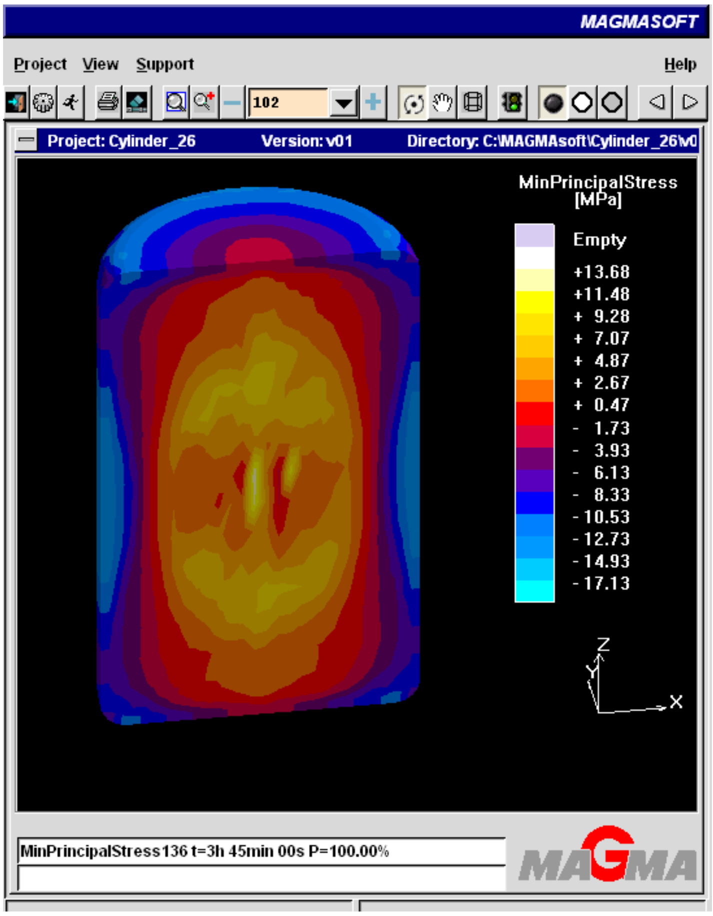


Figure 10.76. Cut view displayed in the main window

10.2.3.1.2 The curves:

In the Postprocessor we can just see the results history curves of the cooling and stress points which positions where defined in the Preprocessor module. Therefore no creation of curves from new points in the part is possible in the Postprocessor.

To see the cooling curves:

> Select "Curves" from the Postprocessor's Control Panel window → From the Results group list select "Solidification" → From the Curves list, select the curve corresponding to the cooling point that you are interested in

The curve will be displayed in the main window as in *Figure 10.78*

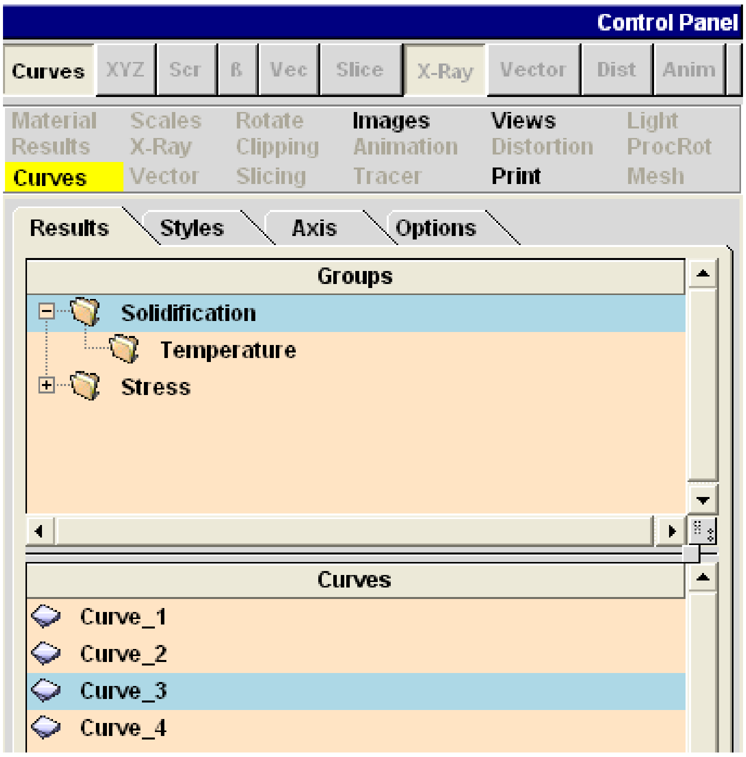


Figure 10.77. Cooling curve selected for display in the Control Panel window

To see the stress curves:

> Select "Curves" from the Postprocessor's Control Panel window → From the Results group list expand the "Stress" item → From the list displayed under Stress, select all the results you want to see for the "to be selected" stress point → From the Curves list, select the curve corresponding to the stress point that you are interested in.

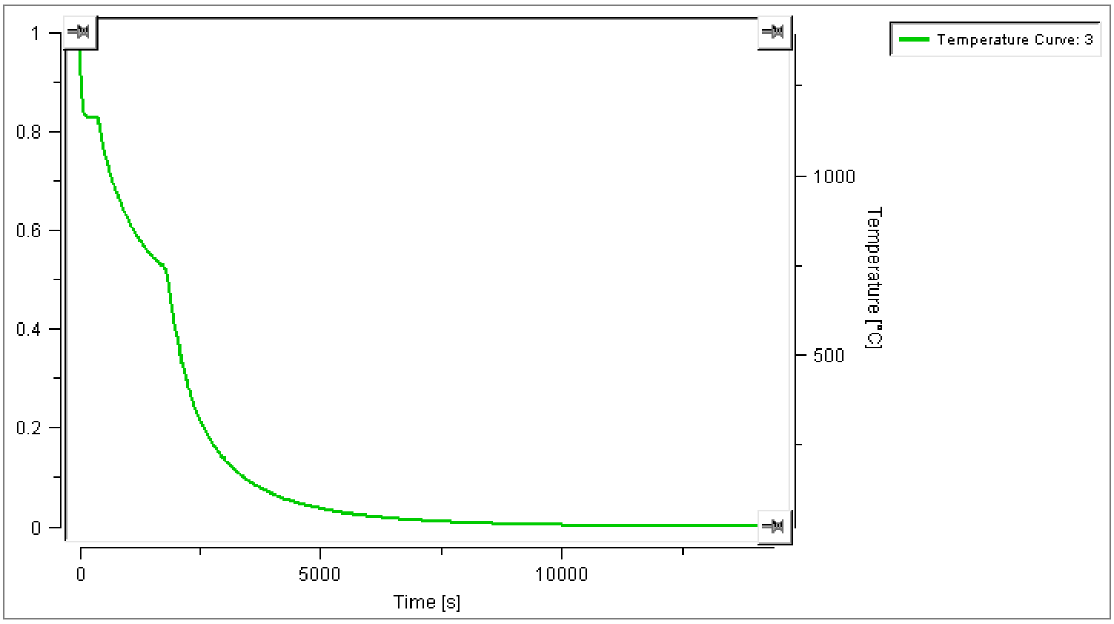


Figure 10.78. Cooling curve display in the Postprocessor main window

To see the position of the cooling and stress points in the Postprocessor:

> Select "Results" from the Postprocessor's Control Panel window → From the Results list expand the "Geometry" item → Double click "Cooling_points" or "Stress_points" as needed → The points will appear in the main window

Notice that the points are displayed over the actual mesh of the part.

10.2.3.2 Results Preparation for Comparison

The approach consist in export the thermal and the stress curves from the Magmasoft Postprocessor to text files (.txt) from where the X-Y data of the curves can be obtained in a table form at that can be used in a software as Matlab, where plots of the Abaqus and Magma curves are combined to perform the actual comparison.

10.2.3.2.1 Exporting a curve:

With the curve you want to export displayed in the main window do as follow

> Go to the "Options" tab as in *Figure 10.79* → On the "Spread sheet file" subdivision of the Options tab, assign a filename to the curve → Press the "Write File" button

The curve is exported as a text file (.txt) to the folder of the current project version

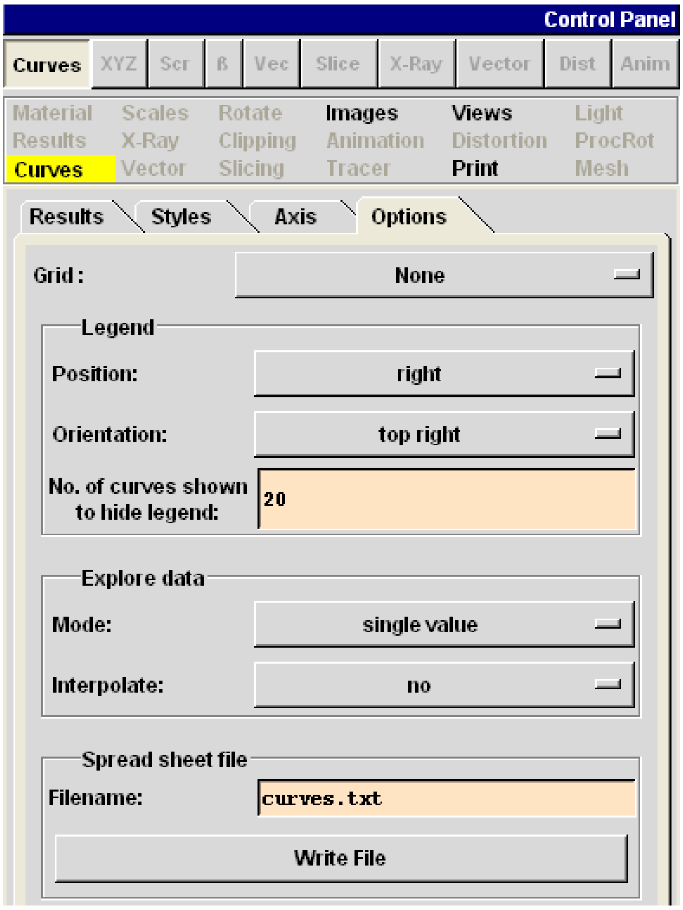


Figure 10.79. Exporting the curves from the Curve's Options tab

10.3 Results Comparison Approach

The comparison consists in combining the plots of the result curves from the different models. We do so in Matlab, but first the results are loaded in Excel where we adjust all the data to have the same units and solve other problems explained in this section. We compare results generated from the same point in the geometry of the models involved.

Matlab do not recognize the report (.rpt) files generated by Abaqus and therefore we make then pass by Excel. However, it does recognize the text (.txt) files from Magmasoft but we chose to load them in Excel also just to store all the results in the .xls format.

At this point we assume that all the curves to be combined and/or compared have been exported as suggested in this report from their sources.

Overview

Thermal results comparison approach

1-Combining the Abaqus thermal results

1.1-Loading the Abaqus .rpt files into Excel

1.2-Combining the Before and After Shake-Out .rpt files

2-Loading the Magma .txt file into Excel

3-Setting the Matlab M-File

4-Plotting the comparison

5-Exporting the comparison image

Stress results comparison approach

1-Loading the Abaqus .rpt file into Excel

2-Loading the Magma .txt file into Excel

3-Modifying the units of the Magma XY data

4-Setting the Matlab M-File

5-Plotting the comparison

6-Exporting the comparison image

Bestpfe.com

10.3.1 Thermal Results Comparison Approach

10.3.1.1 Combining the Abaqus thermal results

As we mention in section 10.1.3.2 if all the results to be compared are from Abaqus we could compare them in the Abaqus Visualization module. But there is a drawback for this approach from our Abaqus implementation procedure and that is that the After Shake-Out simulation results are apart (in a different .odb file) from the Before Shake-Out simulation results. That means that when plotting curves from the After Shake-Out .odb file, the time axis will start from zero instead as from the last time of the Before Shake-Out simulation. So even if we can create a graphic that shows both curves in the Visualization module, they will not be shown one after the other but instead superposed.

To create a continuous curve of the Abaqus thermal results, the .rpt files are imported into Excel and then combined:

10.3.1.1.1 Loading the Abaqus .rpt files into Excel

Open Microsoft Excel → File menu → Open → Files of type: All files → Browse the Abaqus .rpt file → Open → In the "Text Import Wizard Step 1 of 3" window select "Fixed width" (see *Figure 10.80*) → Next →

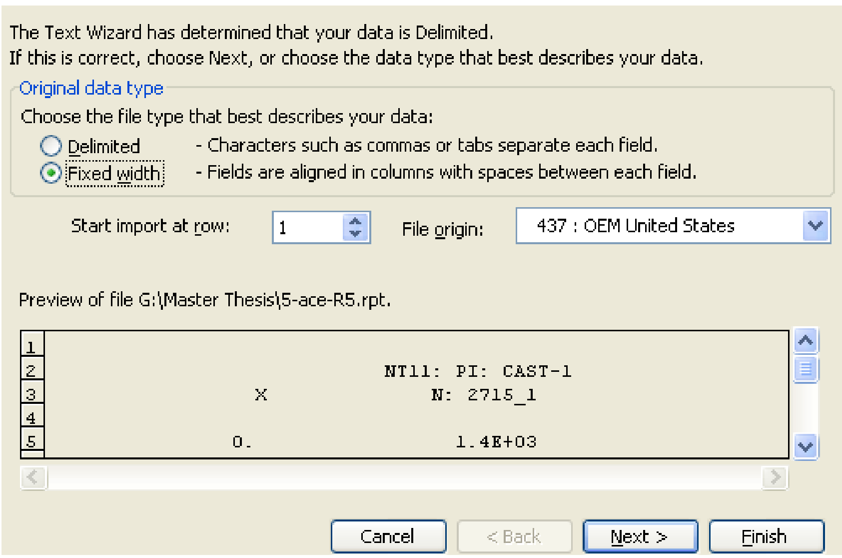


Figure 10.80. Importing an .rpt file into Excel. Text Import Wizard step 1 of 3 window

→ In the "Text Import Wizard Step 2 of 3" window adjust the vertical separation line to properly divide the two data columns (see *Figure 10.81*) → Next →

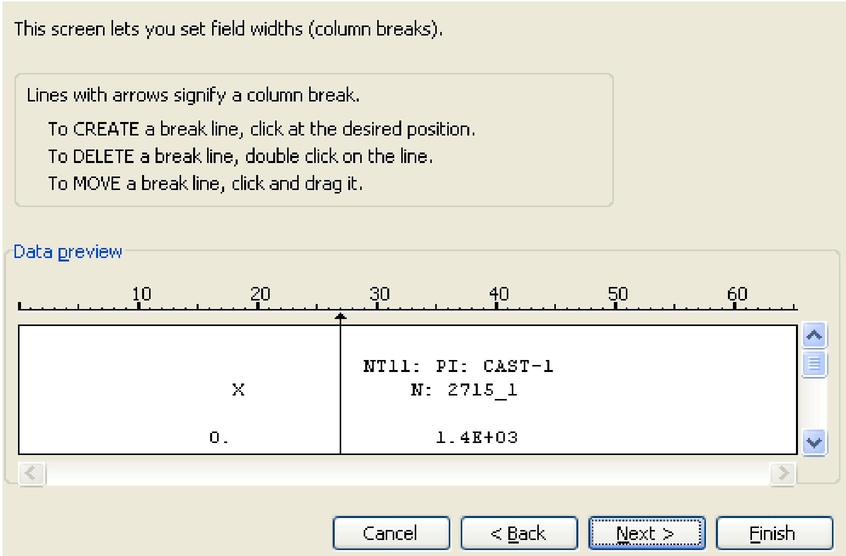


Figure 10.81. Importing an .rpt file into Excel. Text Import Wizard step 2 of 3 window

→ In the "Text Import Wizard Step 3 of 3" window select Colum data format: General (see *Figure 10.82*)→ Confirm the columns are correctly separated → Press Finish

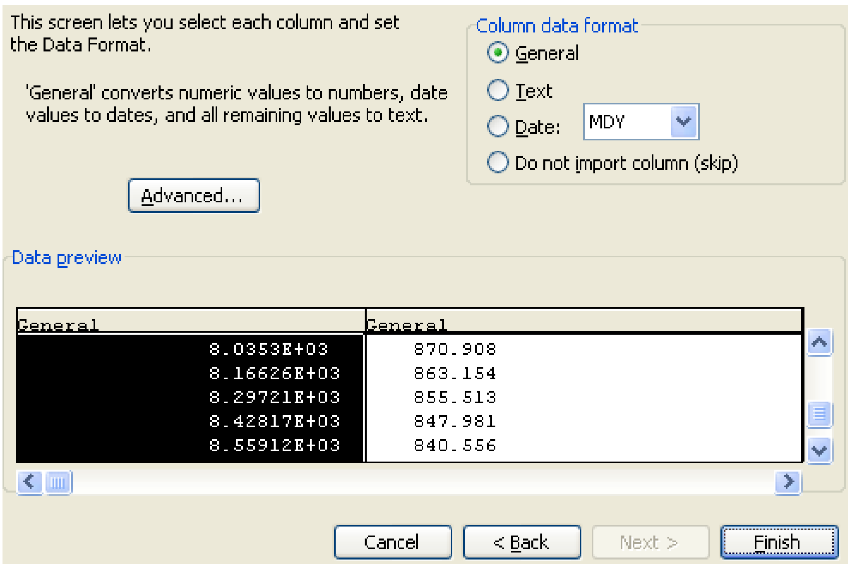


Figure 10.82. Importing an .rpt file into Excel. Text Import Wizard step 3 of 3 window

BESTPFE.COM
List of research project topics and materials

10.3.1.1.2 Combining the Before and After Shake-Out Abaqus .rpt files

Add the last value of the time column of the Before Shake-Out XY data to each value of the time column of the After Shake-Out XY data → Copy the After Shake-Out XY data after the Before Shake-Out one in their respective columns

Now all the time and temperature data is together in one column each from where we will create the corresponding vectors in Matlab. We could combine the data and plot it in Abaqus, but to maintain the same graphic style all the curves results are presented from Matlab.

10.3.1.2 Loading the Magma .txt file into Excel

Open Microsoft Excel → File menu → Open → Files of type: Text files → Browse the Magma .txt file → Open → In the "Text Import Wizard Step 1 of 3" window select "Delimited" (see *Figure 10.83*) → Next →

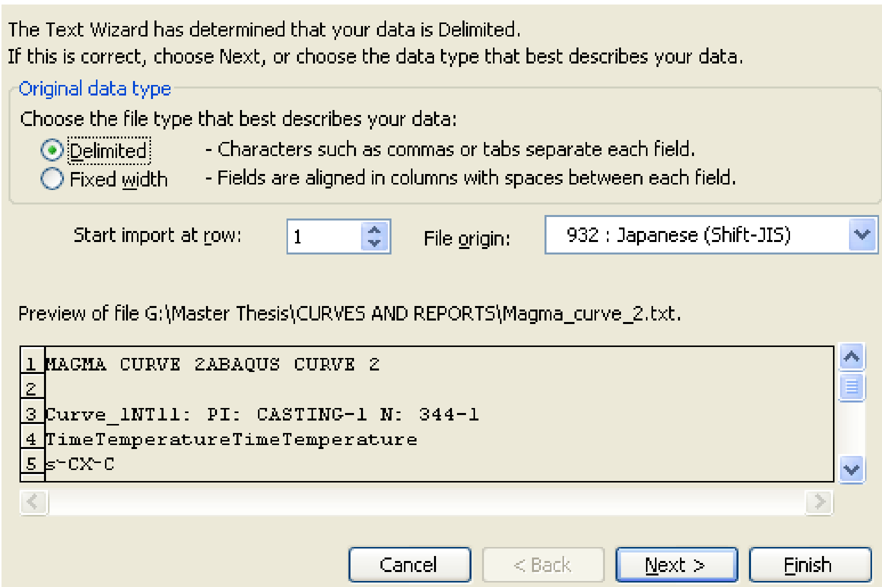


Figure 10.83. Importing an .txt file into Excel. Text Import Wizard step 1 of 3 window

→ In the "Text Import Wizard Step 2 of 3" window check the "Tab" check box (see *Figure 10.84*) → Next →

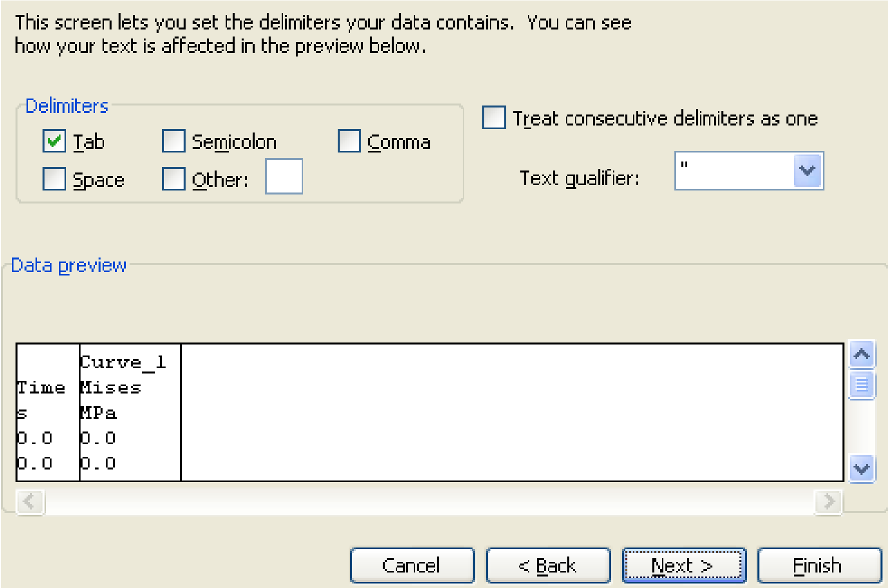


Figure 10.84. Importing an .txt file into Excel. Text Import Wizard step 2 of 3 window

→ In the "Text Import Wizard Step 3 of 3" window select Colum data format: General (see *Figure 10.85*) → Confirm the columns are correctly separated → Press 

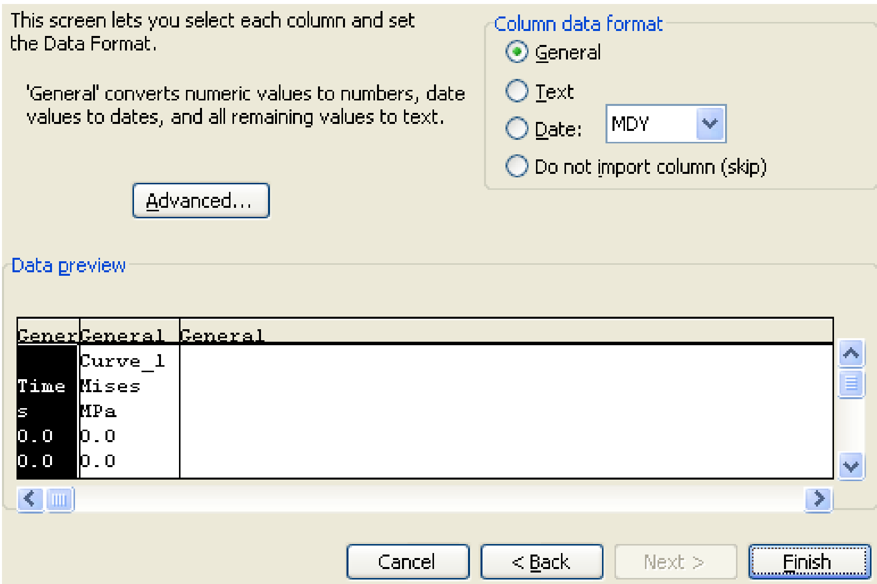


Figure 10.85. Importing an .txt file into Excel. Text Import Wizard step 3 of 3 window

10.3.1.3 Setting up the Matlab M-File

In a Matlab M-File, we will create two vectors per curve to be compared, containing each one of them the data from one of the columns (variables) of the XY data for the curve. Then, the codes to plot the curves in the same graphic will be written.

We will always use the time in the X axis of the graphic.

Create the M-File:

Open Matlab → File menu → New → M-File

Note: We recommend to comment the M-Files (simply write "%" at the beginning of a comment line so it get ignored in the code) for ease of understanding for any reader. Also is useful to separate the vectors in "cells" (Cell menu / Insert cell divider) to simplify the finding of the variables later on (you can use the "Go To" function in the "Go" menu). However, is not mandatory.

The codes:

Write a skeleton code as in *Figure 10.86*:

```
%% MAGMASOFT Thermal Results
%Magma Time (s)
A=[];

%%
%Magma Temperature (C)
B=[];

%% ABAQUS Thermal Results
%Abaqus Time (s)
C=[];

%%
%Abaqus Temperature (C)
D=[];

%% Plot Setting
figure(1)
plot(A,B,'b')
hold on
plot(C,D,'r')
xlabel('Time (s)')
ylabel('Temperature (C)')
title('Thermal Results Comparison')
legend('Magmasoft','Abaqus')
```

Figure 10.86. Template code for the thermal comparison

In *Figure 10.86* we presented our template code for the thermal comparison. As you can see, all the vectors (A, B, C and D) are empty so they have to be populated with the respective information from Excel.

In the "Plot Setting" section in *Figure 10.86*, the first line of code (`figure(1)`) was written to assign a name ("1" in this case) to the graphic to be created. This name will not be plotted. With the second line (`plot(A,B,'b')`) we specify that we want to plot the A vector in the X axis against the B vector in the Y axis and that we want the color of this curve to be blue (`'b'`). The third line (`hold on`) is the one that produce the combination of the curves by including in the same graphic all plots specified after "hold on" and before a "hold off". For more information about Matlab commands see the MATLAB (2006).

Populating the vectors:

Go to Excel → Select all the numerical values from the time column in the Magma data imported from the .txt file → Copy (CTRL+C) → Go to Matlab → Paste them (CTRL+V) in the "A" vector between the "[ ]" signs

In the same way, copy - paste the information of the temperature column into the "B" vector.

Similarly, populate the two Abaqus results vectors "C" and "D".

Only for illustration purposes, in *Figure 10.87* we present the "A" vector populated with values from 0-60 in steps of 10 for you to have an idea about how do a populated vector look like.

```
%% MAGMASOFT Thermal Results
%Magma Time (s)
A=[0
    10
    20
    30
    40
    50
    60];
```

Figure 10.87. Example of a populated vector

10.3.1.4 Plotting the comparison

Once all the vectors contain their respective information and the codes for the plot have been written, simply do as follow:

Press the "Evaluate entire file" () button

10.3.1.5 Exporting the comparison image

A new window per "figure ()" appear with the vectors plotted on it.

On the window corresponding to the Figure you want to export:

File menu → Save as → Browse a destination for the file → Name it → In the "Save as type:" option, choose an appropriate format → Save

10.3.2 Stress results comparison approach

10.3.2.1.1 Loading the Abaqus .rpt files into Excel

Open Excel → File menu → Open → Files of type: All files → Browse the Abaqus .rpt file for the stress result→ Open → In the "Text Import Wizard Step 1 of 3" window select "Fixed width" → Next → In the "Text Import Wizard Step 2 of 3" window adjust the vertical separation line to properly divide the two data columns → Next → In the "Text Import Wizard Step 3 of 3" window select Colum data format: General → Confirm the columns are correctly separated → Press 

10.3.2.2 Loading the Magma .txt file into Excel

Open Excel → File menu → Open → Files of type: Text files → Browse the Magma .txt file → Open → In the "Text Import Wizard Step 1 of 3" window select "Delimited" → Next → In the "Text Import Wizard Step 2 of 3" window check the "Tab" check box → Next → In the "Text Import Wizard Step 3 of 3" window select Colum data format: General → Confirm the columns are correctly separated → Press 

10.3.2.3 Modifying the units of the Magma XY data

Our stress results from Magmasoft are given in Mega Pascals (MPa) while the Abaqus stress results are given in Pascals (Pa). We choose to adapt the Magma units to the Abaqus ones; consequently we have to multiply each value of the stress column of the Magma .txt file by a coefficient of 1.00E+06.

Go to Excel → Create a column for the coefficient containing the value 1.00E+06 in as many cells as stress values exist in the file → Create another column for the stress with new units, where the value of each cell would be equal to the value in the same row of the original data multiplied by the coefficient (e.g. in *Figure 10.88* the value of the cell F6 would be "=B6*D6")

Ensure that the first numerical data from each column is in the same row.

	A	B	C	D	E	F
1		Curve_1				
2	Time	Mises				
3	s	MPa		Coefficient		New Mises
4	0	0		1.00E+06		0.000000
5	0	0		1.00E+06		0.000000
6	0.002	0.000141		1.00E+06		141.488000
7	100	6.36077		1.00E+06		6360770.000000
8	200	10.5996		1.00E+06		10599600.000000
9	300	12.7192		1.00E+06		12719200.000000
10	400	13.4409		1.00E+06		13440900.000000
11	500	13.479		1.00E+06		13479000.000000
12	600	13.2322		1.00E+06		13232200.000000
13	700	12.867		1.00E+06		12867000.000000
14	800	12.4483		1.00E+06		12448300.000000
15	900	11.993		1.00E+06		11993000.000000
16	1000	11.5349		1.00E+06		11534900.000000
17	1100	11.0946		1.00E+06		11094600.000000
18	1200	10.6816		1.00E+06		10681600.000000
19	1300	10.2933		1.00E+06		10293300.000000
20	1400	9.93637		1.00E+06		9936370.000000
21	1500	9.59507		1.00E+06		9595070.000000
22	1600	9.28254		1.00E+06		9282540.000000
23	1700	8.99214		1.00E+06		8992140.000000
24	1800	8.71725		1.00E+06		8717250.000000
25	1900	8.46273		1.00E+06		8462730.000000
26	2000	8.22375		1.00E+06		8223750.000000
27	2100	7.9947		1.00E+06		7994700.000000
28	2200	7.77988		1.00E+06		7779880.000000
29	2300	7.57117		1.00E+06		7571170.000000
30	2400	7.37467		1.00E+06		7374670.000000

Figure 10.88. Changing Magmasoft stress curve results from MPa to Pa.

10.3.2.4 Setting the Matlab M-File

Proceed as in section 10.3.1.3.

10.3.2.5 Plotting the comparison

Proceed as in section 10.3.1.4.

10.3.2.6 Exporting the comparison image

Proceed as in section 10.3.1.5.

10.4 Material Data

10.4.1 Thermal Material Data

Note: Apart from the Latent Heat, which just differs in the units, the rest of the thermal material data is common between Abaqus and Magmasoft.

CASTING

Density

[kg/m3]	Temp. [C]
7100	1
7074.5	100
7049.1001	200
7023.7998	300
6998.6001	400
6850.6001	1000
6814	1160
6882	1173
6813.8799	1255
6745.25	1355
6310.1802	2000

Conductivity

[W/mK]	Temp. [C]
54	1
52.5	100
51	200
50	300
49	400
48.5	500
40	1160
38	1173
38	2000

Specific Heat

[J/KgK]	Temp. [C]
450	1
467	30
506	100
563	200
621	300
663	400
741	500
851	600
1036	700
1100	725
744	810
744	900
804	1000
830	1100
844	1160
740	1173
747	1200
778	1300
813	1400
854	1500
871	1600
872	1700
872	2000

General Parameters

ABAQUS	
Latent Heat [J/Kg]	230000
Liquidus Temp [C]	1173
Solidus Temp [C]	1160

MAGMASOFT	
Latent Heat [KJ/Kg]	230
Liquidus Temp [C]	1173
Solidus Temp [C]	1160

The graphics some of the casting thermal material data can be found in:

Density - *Figure 10.18*; Conductivity - *Figure 10.19*; Specific Heat - *Figure 10.20*

Bestpfe.com

MOLD

Density

[kg/m3]	Temp. [C]
1500	1
1500	2000

Conductivity

[W/mK]	Temp. [C]
1	1
0.7	250
0.6	600
0.7	850
1.2	1400
1.6	1700
1.6	2000

Specific Heat

[J/KgK]	Temp. [C]
676	1
816	98
820	101
858	127
993	327
1074	527
1123	727
1166	927
1201	1127
1230	1327
1333.33	2000

The graphics of the mold thermal material data can be found in:

Density - *Figure 10.21*; Conductivity - *Figure 10.22*; Specific Heat - *Figure 10.23*

10.4.2 Stress Material Data

CAST (there is no mold in our stress analysis)

ABAQUS	MAGMASOFT

Expansion Coefficient

ABAQUS:

Expansion Coeff. / C	Temp. [C]
1.00E-05	20
1.07E-05	200
1.23E-05	400
1.33E-05	600
1.33E-05	1120
1.33E-05	1160
7.67E-06	2000

MAGMASOFT:

Expansion Coeff. / C	Temp. [C]
1.07E-05	20
1.07E-05	199
1.37E-05	201
1.37E-05	399
1.52E-05	401
1.52E-05	599
1.33E-05	601
1.33E-05	1159
1.00E-10	1161
1.00E-10	2000

Young's Modulus

ABAQUS:

E [Pa]	Temp. [C]
126800000000	20
112600000000	200
108400000000	400
103700000000	600
91579000000	1120
500000000	1160
500000000	2000

MAGMASOFT:

E [MPa]	Temp. [C]
126800	20
112600	200
108400	400
103700	600
91579	1120
500	1160
500	2000

Poisson's Ratio

ABAQUS:

μ	Temp. [C]
0.26	20
0.26	200
0.26	400
0.26	600
0.26	1120
0.49	1160
0.49	2000

MAGMASOFT:

μ	Temp. [C]
0.26	20
0.26	200
0.26	400
0.26	600
0.26	1120
0.49	1160
0.49	2000

Hardening Coefficient

ONLY FOR MAGMA

n	Temp. [C]
4.64	20
4.24	200
3.32	400
5.44	600
100.00	1120
100.00	1160
100.00	2000

BEST PFE.COM
List of research project topics and materials

Plasticity Data		
ONLY FOR ABAQUS		
Yield Stress [Pa]	Plastic Strain [µ]	Temp. [C]
1.429590E+08	**0.000000E+00**	**2.00E+01**
1.623300E+08	0.000139366	2.00E+01
1.793920E+08	0.000297755	2.00E+01
1.930600E+08	0.00048416	2.00E+01
2.043150E+08	0.00069044	2.00E+01
2.144730E+08	0.000905715	2.00E+01
2.232710E+08	0.00113216	2.00E+01
2.309010E+08	0.00136818	2.00E+01
2.374720E+08	0.00161686	2.00E+01
2.434780E+08	0.00186615	2.00E+01
2.489900E+08	0.00212343	2.00E+01
2.537010E+08	0.00237927	2.00E+01
2.580740E+08	0.00264977	2.00E+01
2.618640E+08	0.00291306	2.00E+01
2.749570E+08	0.987724	2.00E+01
1.197440E+08	**0.000000E+00**	**2.00E+02**
1.372210E+08	0.000150232	2.00E+02
1.526430E+08	0.000318051	2.00E+02
1.661990E+08	0.000501695	2.00E+02
1.779330E+08	0.000705482	2.00E+02
1.877650E+08	0.000916054	2.00E+02
1.964240E+08	0.0011459	2.00E+02
2.042710E+08	0.00137793	2.00E+02
2.109960E+08	0.00162412	2.00E+02
2.168660E+08	0.00187285	2.00E+02
2.224920E+08	0.00212359	2.00E+02
2.273760E+08	0.00238524	2.00E+02
2.315650E+08	0.00264341	2.00E+02
2.357800E+08	0.00291061	2.00E+02
2.394380E+08	0.00317783	2.00E+02
2.427480E+08	0.00345259	2.00E+02
2.458020E+08	0.0037248	2.00E+02
2.483570E+08	0.00400585	2.00E+02
2.607750E+08	0.987776	2.00E+02
9.395560E+07	**0.000000E+00**	**4.00E+02**
1.109670E+08	0.000147702	4.00E+02
1.257960E+08	0.00031501	4.00E+02
1.391810E+08	0.000502729	4.00E+02
1.513970E+08	0.000693089	4.00E+02
1.615460E+08	0.000901722	4.00E+02
1.719130E+08	0.00110838	4.00E+02
1.807870E+08	0.00132821	4.00E+02
1.896110E+08	0.00155609	4.00E+02
1.973380E+08	0.00177833	4.00E+02
2.040020E+08	0.0020176	4.00E+02
2.108630E+08	0.00227036	4.00E+02
2.166270E+08	0.00250988	4.00E+02
2.217540E+08	0.00276262	4.00E+02
2.270110E+08	0.00302181	4.00E+02
2.316240E+08	0.00328666	4.00E+02
2.359570E+08	0.00353864	4.00E+02

2.399900E+08	0.00380089	4.00E+02
2.435650E+08	0.00407479	4.00E+02
2.473230E+08	0.00434701	4.00E+02
2.504700E+08	0.00461697	4.00E+02
2.534860E+08	0.00488039	4.00E+02
2.565580E+08	0.00516621	4.00E+02
2.590850E+08	0.0054415	4.00E+02
2.618460E+08	0.00571469	4.00E+02
2.642040E+08	0.00598377	4.00E+02
2.664980E+08	0.00626864	4.00E+02
2.688010E+08	0.00655337	4.00E+02
2.707800E+08	0.00684093	4.00E+02
2.727910E+08	0.00711289	4.00E+02
2.746250E+08	0.00740163	4.00E+02
2.761730E+08	0.00769286	4.00E+02
2.899820E+08	0.987431	4.00E+02
6.813530E+07	**0.000000E+00**	**6.00E+02**
7.328360E+07	0.000257529	6.00E+02
7.829640E+07	0.000503261	6.00E+02
8.290950E+07	0.000765946	6.00E+02
8.762440E+07	0.00101438	6.00E+02
9.253260E+07	0.00127428	6.00E+02
9.694570E+07	0.00153873	6.00E+02
1.008660E+08	0.00179445	6.00E+02
1.045960E+08	0.00206517	6.00E+02
1.084920E+08	0.00232106	6.00E+02
1.118360E+08	0.00259532	6.00E+02
1.151410E+08	0.00286988	6.00E+02
1.177690E+08	0.00313745	6.00E+02
1.208230E+08	0.00341427	6.00E+02
1.235960E+08	0.00368045	6.00E+02
1.263380E+08	0.00397337	6.00E+02
1.286670E+08	0.0042436	6.00E+02
1.311810E+08	0.00451206	6.00E+02
1.337120E+08	0.00479363	6.00E+02
1.357820E+08	0.0050794	6.00E+02
1.377000E+08	0.00535333	6.00E+02
1.394580E+08	0.00562869	6.00E+02
1.413970E+08	0.00591563	6.00E+02
1.432950E+08	0.00621616	6.00E+02
1.449730E+08	0.00649218	6.00E+02
1.463780E+08	0.0067707	6.00E+02
1.478850E+08	0.00706152	6.00E+02
1.494900E+08	0.00735141	6.00E+02
1.508070E+08	0.00763067	6.00E+02
1.521220E+08	0.00792319	6.00E+02
1.533070E+08	0.00820362	6.00E+02
1.546070E+08	0.00849623	6.00E+02
1.557300E+08	0.00877717	6.00E+02
1.568860E+08	0.00907105	6.00E+02
1.579110E+08	0.00937932	6.00E+02
1.589350E+08	0.00964791	6.00E+02
1.600560E+08	0.00995523	6.00E+02
1.610410E+08	0.0102506	6.00E+02

1.618970E+08	0.0105471	6.00E+02
1.627120E+08	0.0108175	6.00E+02
1.637110E+08	0.0111259	6.00E+02
1.644790E+08	0.0113966	6.00E+02
1.653270E+08	0.0117063	6.00E+02
1.660060E+08	0.0119911	6.00E+02
1.668310E+08	0.0122878	6.00E+02
1.675400E+08	0.0125722	6.00E+02
1.683160E+08	0.0128692	6.00E+02
1.689570E+08	0.0131675	6.00E+02
1.696200E+08	0.0134655	6.00E+02
1.702530E+08	0.0137505	6.00E+02
1.709090E+08	0.0140486	6.00E+02
1.715030E+08	0.0143472	6.00E+02
1.720500E+08	0.0146329	6.00E+02
1.726920E+08	0.0149442	6.00E+02
1.731860E+08	0.0152172	6.00E+02
1.737160E+08	0.0155295	6.00E+02
1.741900E+08	0.015829	6.00E+02
1.745880E+08	0.0161028	6.00E+02
1.833170E+08	0.988303	6.00E+02
1.000000E+06	**0.000000E+00**	**1.12E+03**
1.050000E+06	0.019989	1.12E+03
1.050000E+06	0.99	1.12E+03
1.000000E+06	**0.000000E+00**	**1.16E+03**
1.050000E+06	0.0179	1.16E+03
1.050000E+06	0.99	1.16E+03
1.000000E+06	**0.000000E+00**	**2.00E+03**
1.050000E+06	0.0179	2.00E+03
1.050000E+06	0.99	2.00E+03

Yield Stress

ONLY FOR MAGMA.

The Abaqus data presented next (left) was extracted from the Abaqus plasticity data.

E [Pa]	Temp. [C]
142959000	20
119744000	200
93955600	400
68135300	600
1000000	1120
1000000	1160
1000000	2000

E [MPa]	Temp. [C]
1.42959E+02	20
1.19744E+02	200
9.39556E+01	400
6.81353E+01	600
1.00000E+00	1120
1.00000E+00	1160
1.00000E+00	2000

The graphics of some of the stress material data can be found in:

Young's Modulus - *Figure 10.34*; Poisson's Ratio - *Figure 10.35*; Expansion Coefficient - *Figure 10.36*; Plasticity - *Figure 10.37*

10.5 Keywords of the Abaqus input files

Next, as a reference, we present the keywords of our Abaqus input (.inp) files corresponding to the Cylinder simulations.

Before Shakeout

```
**
** PARTS
**
*Part, name=Cast
*Element, type=DC3D4
** Section: Section-1-_PICKEDSET2
*Solid Section, elset=_PickedSet2, material=CAST-MAT
1.,
*End Part
**
*Part, name=Mold
*Element, type=DC3D4
** Section: Section-2-_PICKEDSET2
*Solid Section, elset=_PickedSet2, material=MOLD-MAT
1.,
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Cast-1, part=Cast
*End Instance
**
```

```
*Instance, name=Mold-1, part=Mold
*End Instance
**
*Surface, type=ELEMENT, name=CAST-EXT-SURF
*Surface, type=ELEMENT, name=MOLD-INT-SURF
*Surface, type=ELEMENT, name=MOLD-EXT-SURF
*End Assembly
**
** MATERIALS
**
*Material, name=CAST-MAT
*Conductivity
*Density
*Latent Heat
*Specific Heat
*Material, name=MOLD-MAT
*Conductivity
*Density
*Specific Heat
**
** INTERACTION PROPERTIES
**
*Surface Interaction, name=CAST-MOLD-CONTACT-INTERACTION-
PROPERTY
1.,
*Gap Conductance
1000.,   0.
  0.,1000.
**
** PHYSICAL CONSTANTS
```

```
**
*Physical Constants, absolute zero=-273.15, stefan
boltzmann=5.67e-08
**
** PREDEFINED FIELDS
**
** Name: Field-1   Type: Temperature
*Initial Conditions, type=TEMPERATURE
_PickedSet48, 1400.
** Name: Field-2   Type: Temperature
*Initial Conditions, type=TEMPERATURE
_PickedSet49, 20.
**
** INTERACTIONS
**
** Interaction: CAST-MOLD-CONTACT-INTERACTION-PROPERTY-1
*Contact Pair, interaction=CAST-MOLD-CONTACT-INTERACTION-
PROPERTY
MOLD-INT-SURF, CAST-EXT-SURF
** ----------------------------------------------------------------
**
** STEP: Before Shake-Out
**
*Step, name="Before Shake-Out", extrapolation=PARABOLIC,
inc=28800
*Heat Transfer, end=PERIOD, deltmx=10.
10., 28800., 1e-12, 28800.,
**
** INTERACTIONS
**
** Interaction: SURFFILM-1
```

```
*Sfilm
MOLD-EXT-SURF, F, 20., 20.
** Interaction: SURFRADIATE-1
*Sradiate
MOLD-EXT-SURF, R, 20., 0.76
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
NT,
*Output, history, frequency=0
*End Step
```

After Shake out

```
**
** PARTS
**
*Part, name=Cast
*Element, type=DC3D4
** Section: Section-1-_PICKEDSET2
*Solid Section, elset=_PickedSet2, material=CAST-MAT
1.,
*End Part
**
**
```

```
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=Cast-1, part=Cast
*End Instance
**
*Surface, type=ELEMENT, name=CAST-EXT-SURF
*End Assembly
**
** MATERIALS
**
*Material, name=CAST-MAT
*Conductivity
*Density
*Latent Heat
*Latent Heat
**
** INTERACTION PROPERTIES
**
*Film Property, name=Conv-HTC
**
** PHYSICAL CONSTANTS
**
*Physical Constants, absolute zero=-273.15, stefan
boltzmann=5.67e-08
**
** PREDEFINED FIELDS
**
** Name: Cast-Initial   Type: Temperature
```

Bestpfe.com

```
*Initial Conditions, type=TEMPERATURE, file=g:/dokument/5-ace-R3.odb, step=1, inc=506

** ----------------------------------------------------------------

**

** STEP: Before Shake-Out

**

*Step, name="Before Shake-Out", extrapolation=PARABOLIC, inc=43200

*Heat Transfer, end=PERIOD, deltmx=1.

10., 43200., 1e-12, 43200.,

**

** INTERACTIONS

**

** Interaction: SURFFILM-1

*Sfilm

CAST-EXT-SURF, F, 20., Conv-HTC

** Interaction: SURFRADIATE-1

*Sradiate

CAST-EXT-SURF, R, 20., 0.76

**

** OUTPUT REQUESTS

**

*Restart, write, frequency=0

**

** FIELD OUTPUT: F-Output-1

**

*Output, field

*Node Output

NT,

*Output, history, frequency=0

*End Step
```

Stress Analysis

```
**
** PARTS
**
*Part, name=CAST
*Element, type=C3D4
** Section: Cast-Sect
*Solid Section, elset=_PickedSet2, material=CAST-MAT
1.,
*End Part
**
**
** ASSEMBLY
**
*Assembly, name=Assembly
**
*Instance, name=CAST-1, part=CAST
*End Instance
**
*End Assembly
**
** MATERIALS
**
*Material, name=CAST-MAT
*Density
*Elastic
*Expansion
*Plastic
**
```

BESTPFE.COM
List of research project topics and materials

```
** PHYSICAL CONSTANTS
**
*Physical Constants, absolute zero=-273.15, stefan
boltzmann=5.67e-08
**
** BOUNDARY CONDITIONS
**
** Name: BC-6 Type: Displacement/Rotation
*Boundary
_PickedSet74, 2, 2
_PickedSet74, 3, 3
** Name: BC-7 Type: Displacement/Rotation
*Boundary
_PickedSet76, 1, 1
_PickedSet76, 2, 2
** Name: BC-9 Type: Displacement/Rotation
*Boundary
_PickedSet81, 1, 1
_PickedSet81, 2, 2
_PickedSet81, 3, 3
**
** PREDEFINED FIELDS
**
** Name: Cast-Initial Temp   Type: Temperature
*Initial Conditions, type=TEMPERATURE
_PickedSet49, 1400.
** ----------------------------------------------------------------
**
** STEP: Stress-1
**
```

```
*Step, name=Stress-1, inc=28800
*Static
0.01, 28800., 1e-11, 28800.
**
** PREDEFINED FIELDS
**
** Name: BSO   Type: Temperature
*Temperature, file=C:/ProgHJ/ABAQUS/5-ace-R3.odb, bstep=1,
binc=1, estep=1, einc=506
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
NT, U
*Element Output, directions=YES
EE, PE, PEEQ, S, THE
*Output, history, frequency=0
*End Step
** ----------------------------------------------------------------
**
** STEP: Stress-2
**
*Step, name=Stress-2, inc=43200
*Static
0.01, 43200., 1e-11, 43200.
```

```
**
** PREDEFINED FIELDS
**
** Name: ASO   Type: Temperature
*Temperature, file=C:/ProgHJ/ABAQUS/9-fg-R6.odb, bstep=1,
binc=1, estep=1, einc=999
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
NT, U
*Element Output, directions=YES
EE, PE, PEEQ, S, THE
*Output, history, frequency=0
*End Step
```

10.6 Thermal Expansion Coefficient calculation Magmasoft - Abaqus

Magmasoft uses a local definition of the thermal expansion coefficient while Abaqus uses a global definition. Taking this into account, if the thermal expansion coefficient material data from the Magma database will be used in Abaqus, a conversion of the data must be performed.

We carry the conversion in Matlab as follow:

```
%%Adapting the Magmasoft Thermal Expansion Coefficient data to Abaqus


%Thermal Expansion Coefficient from Magmasoft, AlphaM:

AlphaM=[1.07E-05
1.07E-05
1.37E-05
1.37E-05
1.52E-05
1.52E-05
1.33E-05
1.33E-05
1.00E-10
1.00E-10
];


%Global Temperature from Magmasoft, T:

T=[20
199
201
399
401
599
601
1159
1161
2000
];


% Relative Temperature change (used in Magmasoft), DTM (Delta T
Magmasoft):

A(1)=T(1);

for i=2:length(T)

    A(i)=T(i)-T(i-1);
end

DTM=A';
```

```
%Relative Thermal Strain (used in Magmasoft), SM (Strain Magmasoft):

for i=1:length(T)

    B(i) = AlphaM(i) * DTM(i);

end

SM=B';


%Absolute Thermal Strain (to be used in Abaqus), SA (Strain Abaqus):

C(1)=SM(1);

for i=2:length(T)

    C(i)=SM(i)+C(i-1);

end

SA=C';


%Absolute Reference Temperature (to be used in Abaqus), RT (Reference
Temperature):

RT=0;


%Global Temperature for Abaqus considering the Reference Temperature,
TA (Temperature Abaqus):

D(1)=0;

for i=2:length(T)

    D(i)=T(i)-RT;

end

TA=D';


%Thermal Expansion Coefficient for Abaqus, AlphaA:

E(1)=0;

for i=2:length(T)

    E(i)=SA(i)/TA(i);

end

AlphaA=E'
```

10.7 Conduction Interaction Vs Tie Constraint

As mentioned in the step 10 of *Section 10.1.1.3.1,* we could have simulated the conductive heat transfer between the casting and the mold in Abaqus by means of a Tie Constrain, which represents perfect conduction, or by a Contact Interaction property where a value for the heat transfer coefficient must be defined. We compare simulations of the Cylinder model before shake-out using a HTC of 1000 in Magmasoft, a Contact Interaction with an equivalent HTC of 1000 in Abaqus and using a Tie Constraint in Abaqus. Finally we decide that the Contact Interaction approach will produce better comparisons with the HTC=1000 simulation in Magmasoft. In *Figure 8.1* the comparison of these simulations is presented.

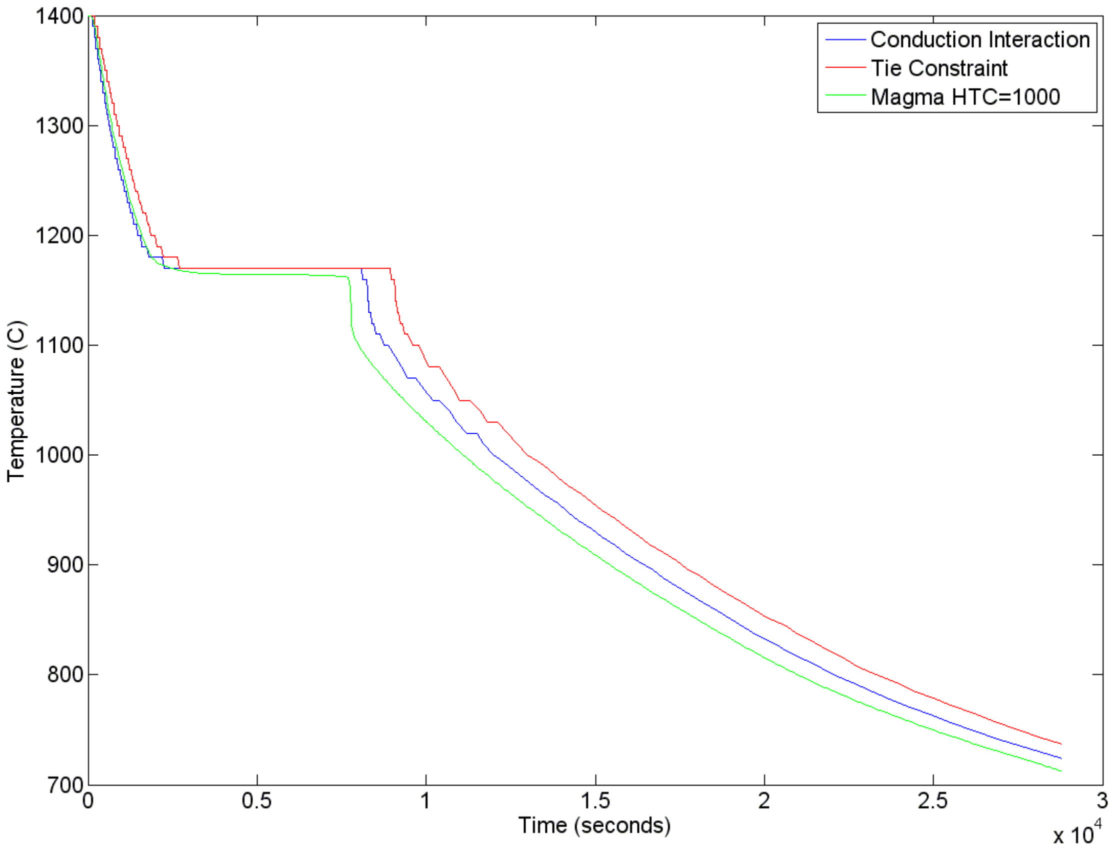


Figure 10.89. Comparison between Conduction Interaction with HTC=1000, Tie Constraint and Magmasoft HTC=1000

10.8 Results comparison with and without symmetry

If the problem in hand is symmetric, is very convenient from a time-saving point of view, to perform the analysis in the smaller part that represent the whole geometry when appropriate boundary conditions are used.

We carry out our simulation procedure in the using symmetry conditions for the Cylinder, the Original Hub and the Optimized Hub.

Next, we present a comparison between the stress results obtained from the whole geometry and from the symmetric geometries. The comparison includes a CPU simulation time comparison.

10.8.1 Cylinder

10.8.1.1 Geometry

A 1/8th of the cylinder has been used for the symmetric analysis as shown in *Figure 10.90*. Accordingly, a 1/8th of the mold has also been used.

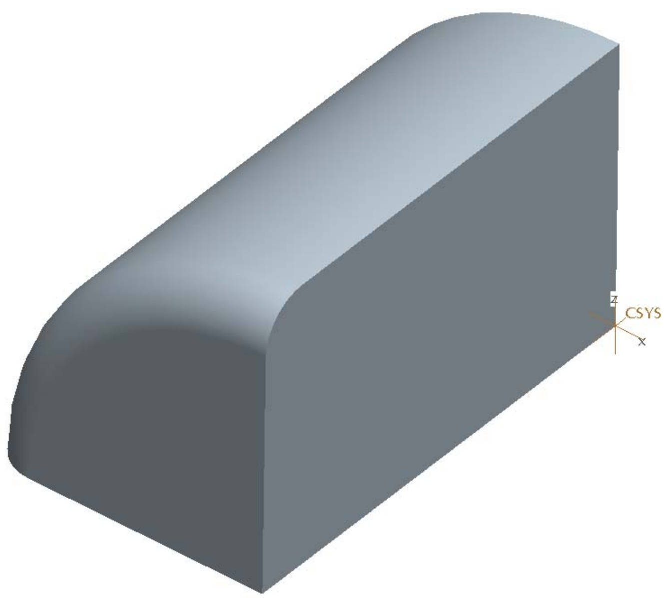


Figure 10.90. 1/8th of the Cylinder geometry as used in the symmetry analysis

10.8.1.2 Simulation time

		Thermal	Stress	Total
Abaqus	Symmetry	2hrs. 4min.	26min.	2hrs. 30min.
	No Symmetry	7hrs. 12min.	24min.	7hrs. 36 min.
Magma	Symmetry			09hrs. 11min.
	No Symmetry			03hrs. 16min.

10.8.1.3 Thermal results

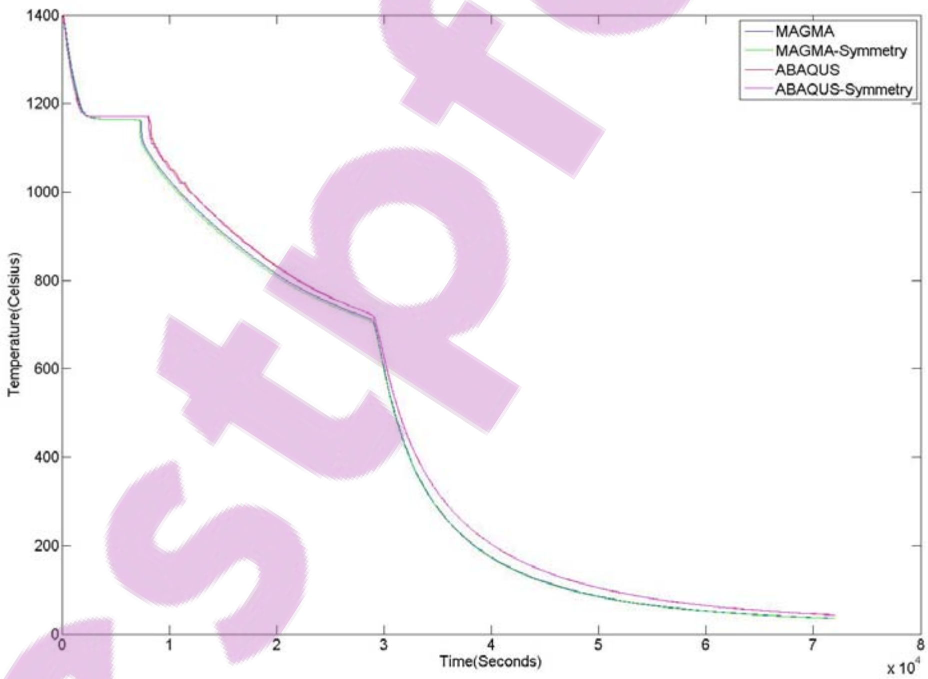


Figure 10.91. Thermal results comparison of the Cylinder model with and without symmetry

Bestpfe.com

10.8.1.4 Stress results

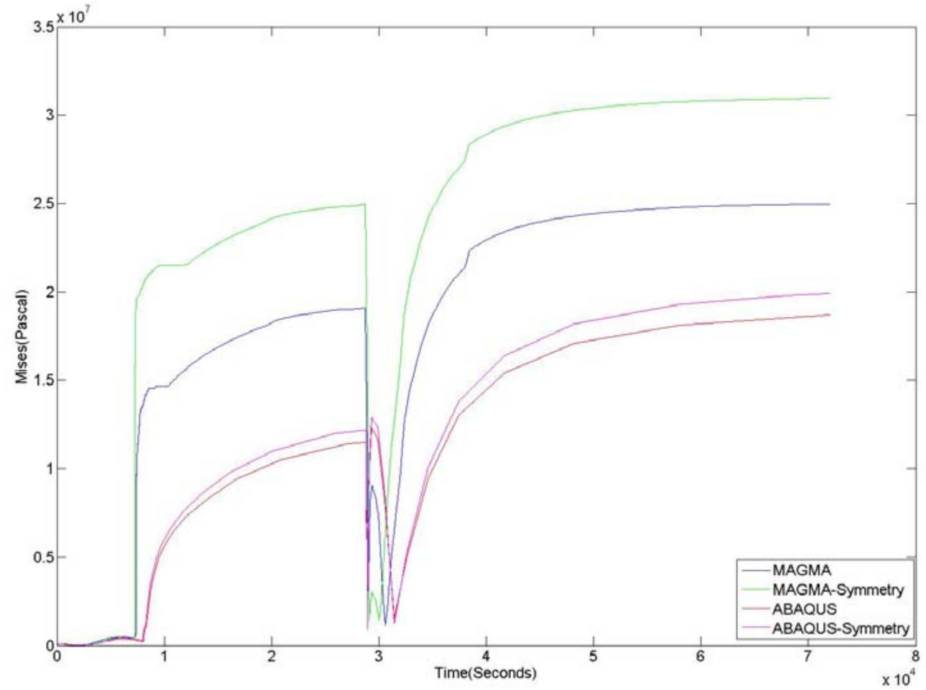
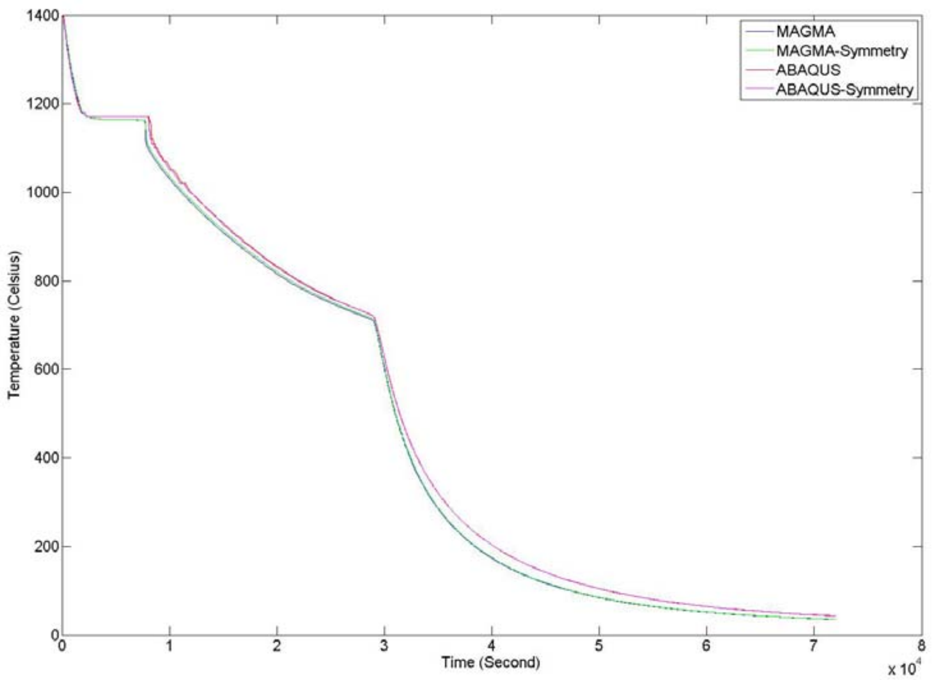


Figure 10.92. Mises comparison of the Cylinder model with and without symmetry



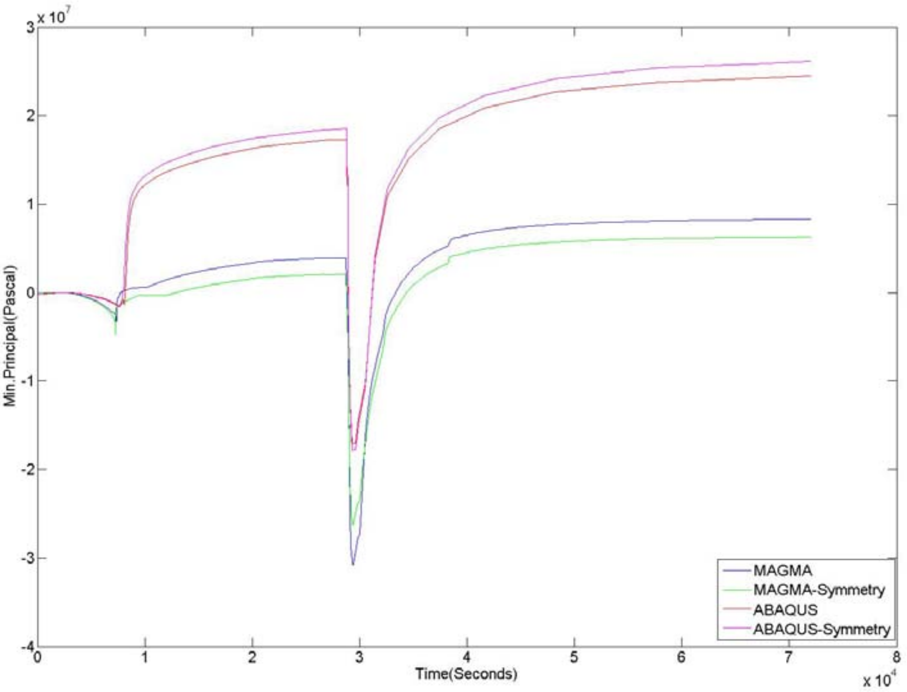


Figure 10.94. Minimum Principal Stress comparison of the Cylinder model with and without symmetry

BESTPFE.COM
List of research project topics and materials

10.8.2 Original Hub

10.8.2.1 Geometry

A half of the Original Hub has been used for the symmetric analysis as shown in *Figure 10.95*. Accordingly, half of the mold has also been used.

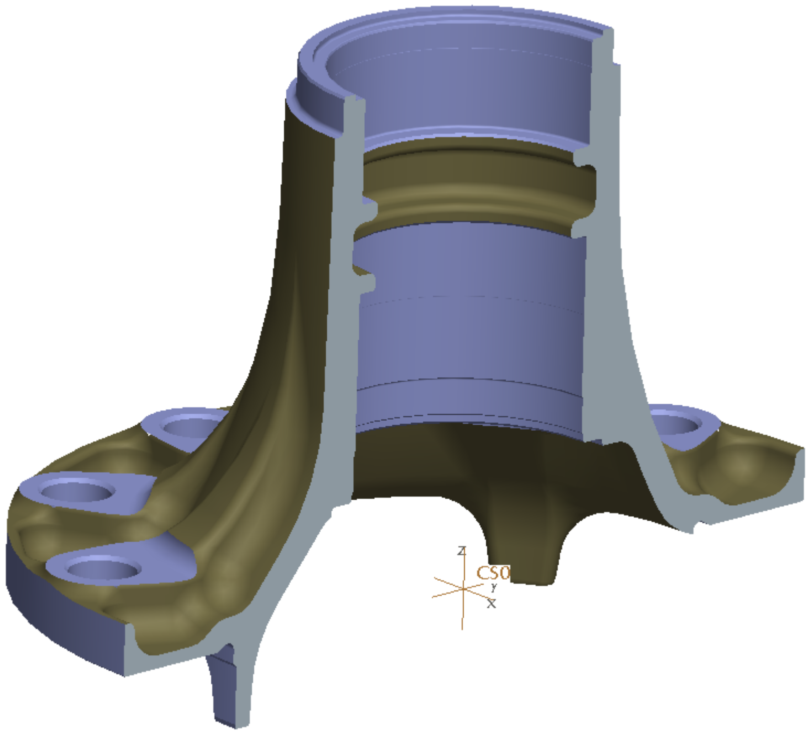


Figure 10.95. Half of the Original Hub geometry as used in the symmetry analysis

10.8.2.2 Simulation time

		Thermal	Stress	Total
Abaqus	Symmetry	41hrs. 18min.	7hrs. 38min.	48hrs. 56min.
	No Symmetry	52hrs. 6min.	10hrs. 50min.	62hrs. 56min.
Magma	Symmetry			09hrs. 20min.
	No Symmetry			08hr. 05min.

10.8.2.3 Stress results

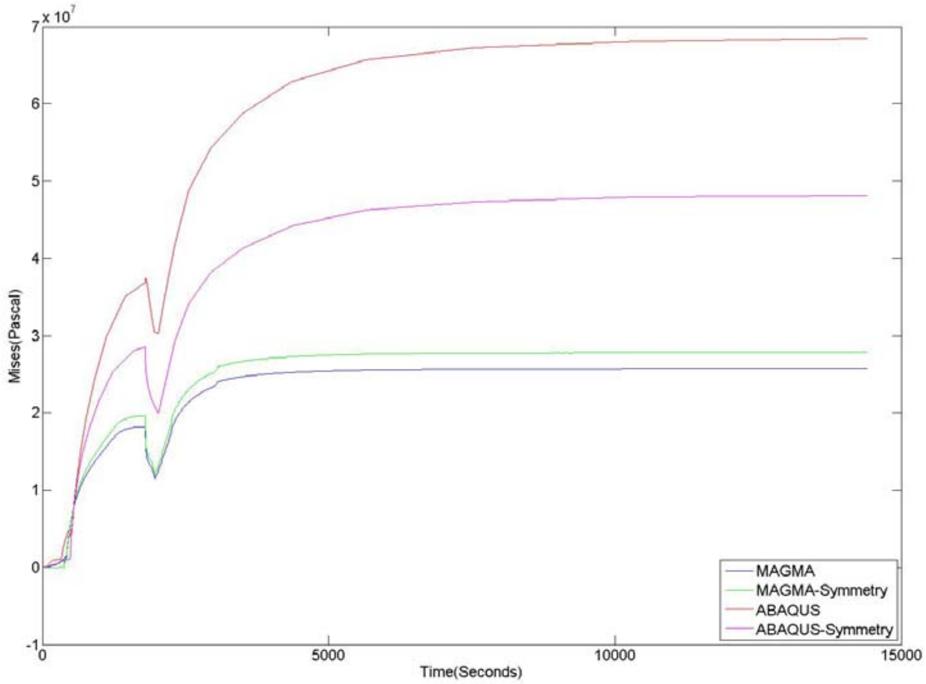


Figure 10.96. Mises comparison of the Original Hub model with and without symmetry

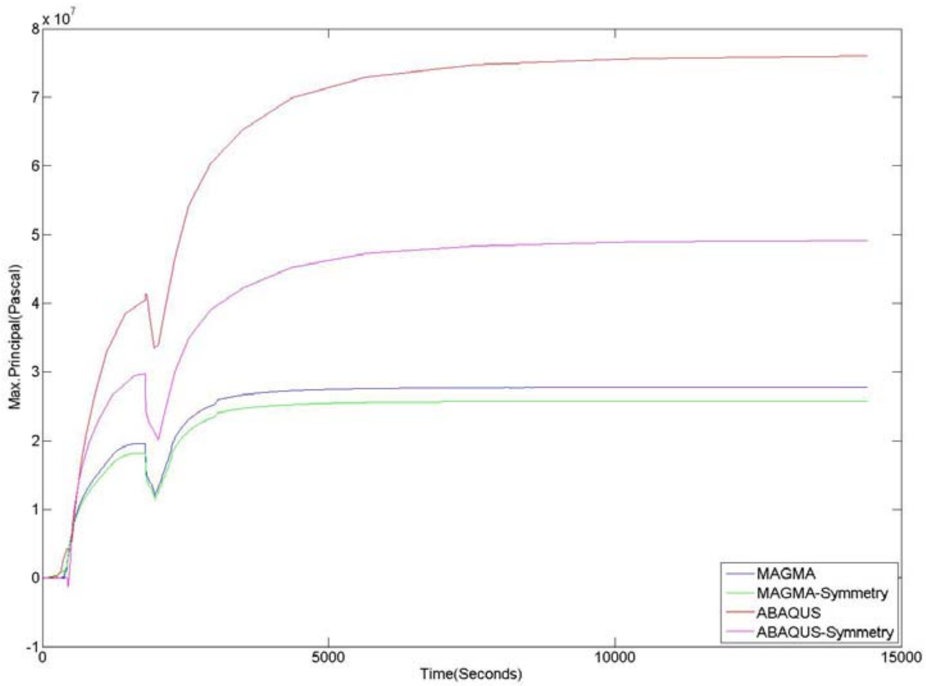


Figure 10.97. Maximum Principal Stress comparison of the Original Hub model with and without symmetry

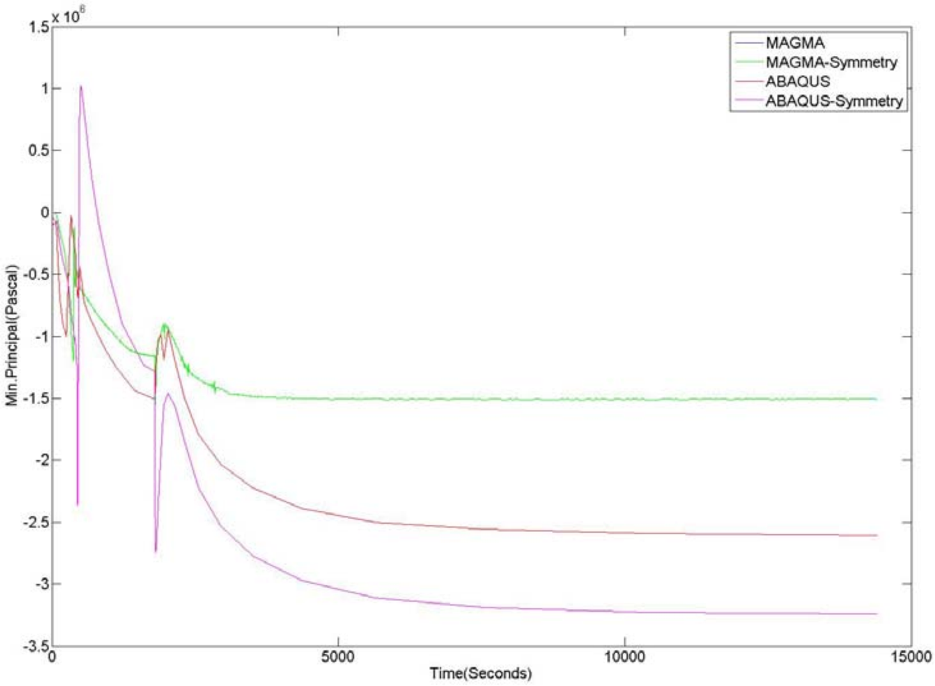


Figure 10.98. Minimum Principal Stress comparison of the Original Hub model with and without symmetry

10.8.3 Optimized Hub

10.8.3.1 Geometry

A half of the Hub has been used for the symmetric analysis as shown in *Figure 10.100*. Accordingly, half of the mold has also been used.

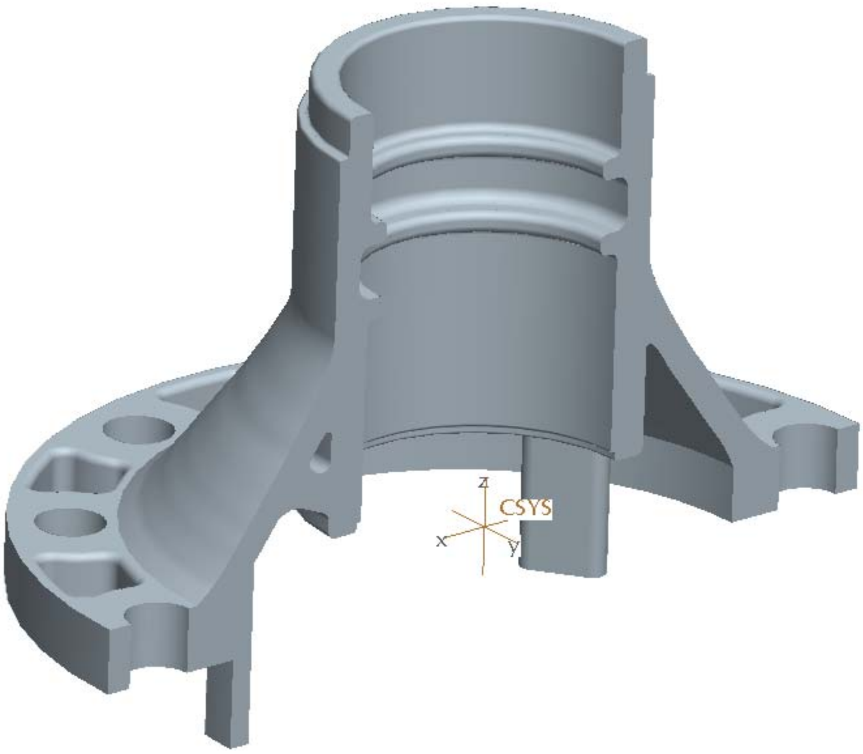


Figure 10.99. Half of the Optimized Hub geometry as used in the symmetry analysis

10.8.3.2 Simulation time

		Thermal	Stress	Total
Abaqus	Symmetry	67hrs. 31min.	7hrs. 54min.	75hrs. 25min.
	No Symmetry	99hrs. 32min.	20hrs. 25min.	119hrs. 57min.
Magma	Symmetry			18hrs. 28min.
	No Symmetry			16hr. 00min.

10.8.3.3 Thermal results

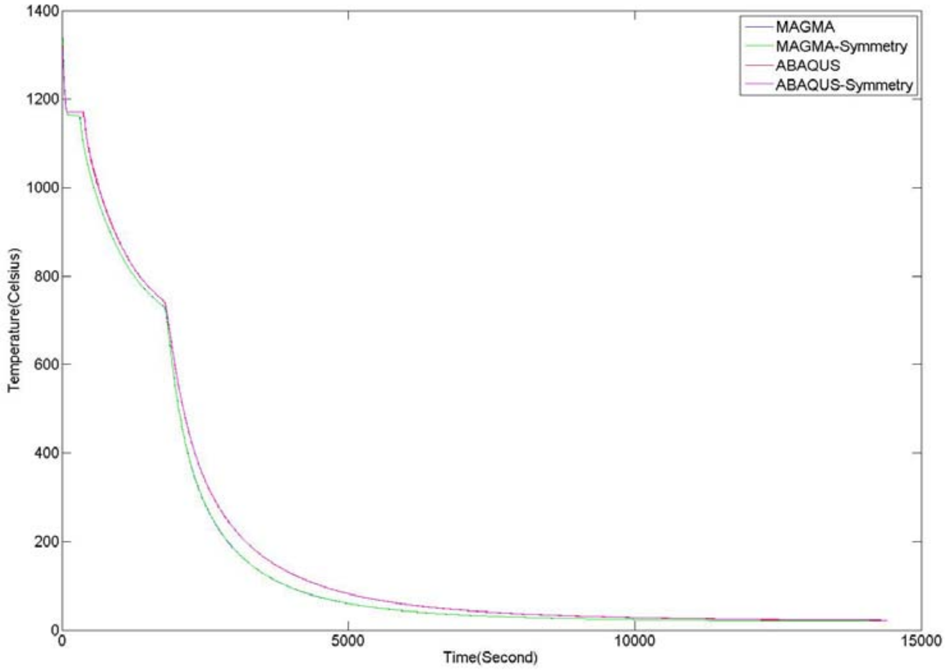


Figure 10.100. Thermal results comparison of the Optimized Hub model with and without symmetry

10.8.3.4 Stress results

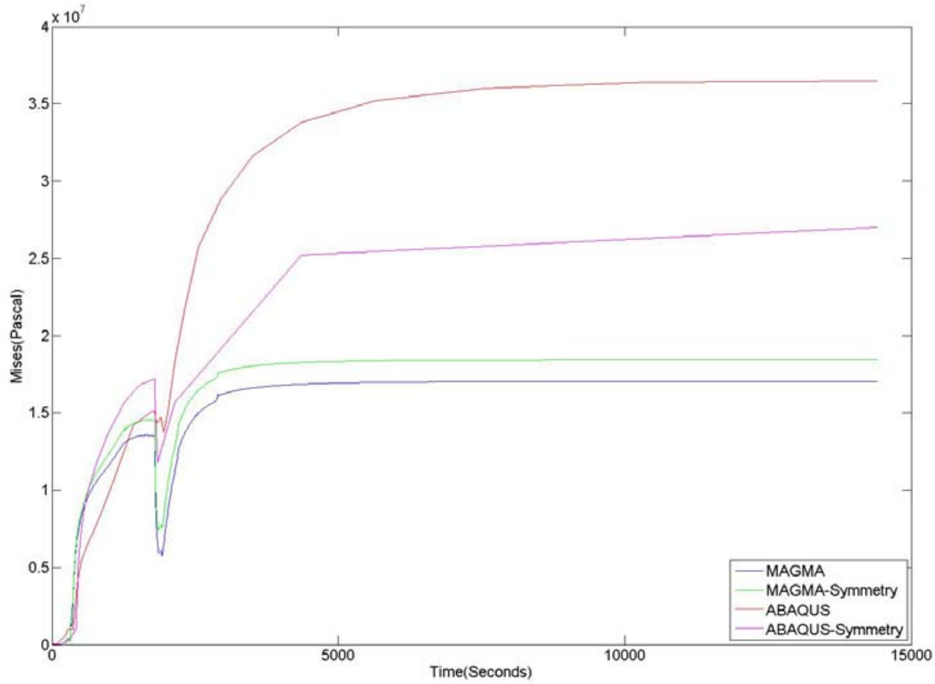


Figure 10.101. Mises comparison of the Optimized Hub model with and without symmetry

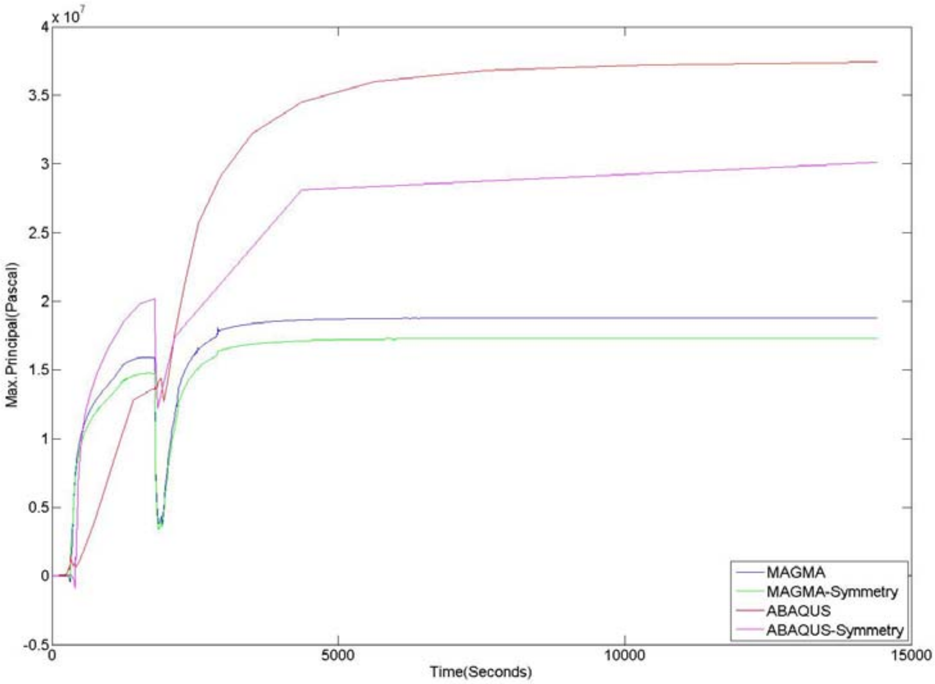


Figure 10.102. Maximum Principal Stress comparison of the Optimized Hub model with and without symmetry

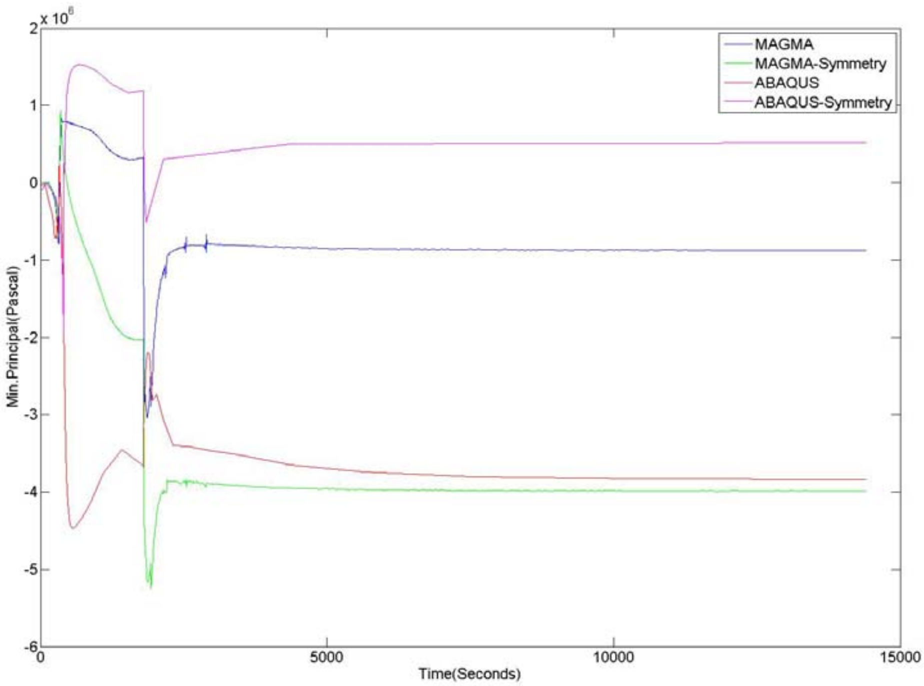


Figure 10.103. Minimum Principal Stress comparison of the Optimized Hub model with and without symmetry