Link of injection molding simulation with FEM structure mechanical analysis

C. Hinse, R. Haag

SimpaTec GmbH, Aachen, Germany

Abstract: One major benefit of using a 3D injection molding simulation is the capability to simulate the fiber orientation and the anisotropic warpage, with the option, to transfer the material properties to structural mechanical analysis. With this option, it is possible to reduce the over dimensioning of fiber reinforced parts and thus resulting in substantial savings of mass, material and costs .This work will demonstrate the benefit of FEM combined with injection molding simulation but also the risks that can occur.

Keywords: injection molding simulation, fiber orientation, anisotropic warpage simulation

1. Introduction

As a way of reducing weight and fuel consumption, plastics are continuing to substitute metals in the automotive industry. Just as important as the lower density of plastics is the possibility of being able to modify mechanical and thermal properties to suit the loads by incorporating different components, such as glass fibres and fillers. Since thermoplastics exhibit a sharp drop in elastic modulus at elevated temperatures, but an automotive component also has to offer sufficient rigidity even in extreme conditions, the designer has to choose an adequate elastic modulus and appropriate glass fibre reinforcement.

When glass fibres and other fillers are used, the isotropic material approach employed in standard FEA calculations loses its validity. Instead, components consisting of such reinforced materials behave extremely anisotropic on account of their complexity. This anisotropy causes a warpage that is related to the fibre orientation and leads to different stress states that need to be evaluated differently in the structural mechanical calculation. In extreme cases, a part will fail on account of unfavourable fibre orientations in an area that was not critical in an isotropic FEA calculation. An approach that takes account of these effects is described below. As an example the investigation of a runner system of a bumper will be shown. As the fiber orientation is determined by the filling behavior the mechanical properties of the part are directly linked with the injection molding simulation. The example describes two different runner designs and shows the influence of these designs on the mechanical behavior in a dynamic load case. The material is a PA66 from DuPont with 45% fibers. The injection molding simulation is done with Moldex3D.

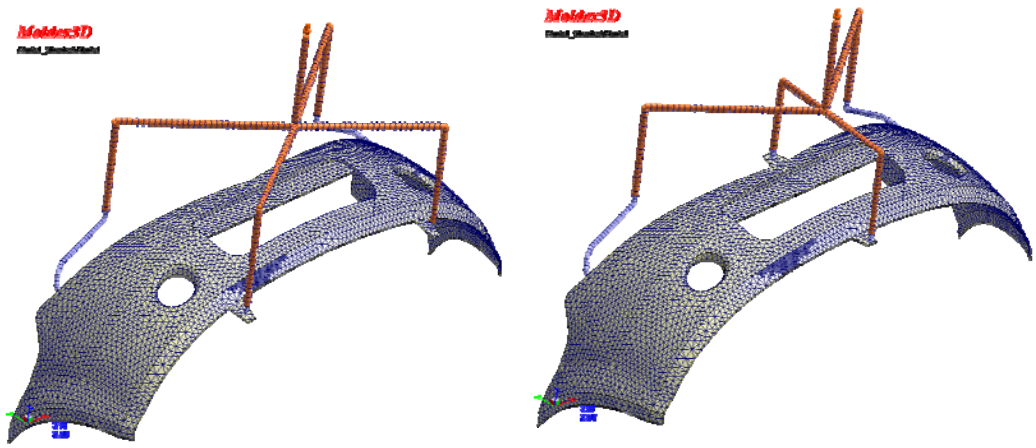


Fig. 1: Two different runner designs

2. Anisotropic warpage analysis

To perform anisotropic FEA, it is necessary to calculate the fiber orientation tensor during the filling process. The Folgar-Tucker equations describe the change of fiber orientation caused by shear effects and vorticity. The 3x3-tensor is symmetric, so 6 components have to be calculated.

$$\frac{\partial A_{ij}}{\partial t} + u_k \frac{\partial A_{ij}}{\partial x_k} = A_{ik} W_{kj} - W_{ik} A_{kj} + \lambda \left(A_{ik} D_{kj} + D_{ik} A_{kj} - 2 A_{ijkl} D_{kl} \right) + 2 C_I \dot{\gamma} \left(\delta_{ij} - 3 A_{ij} \right)$$

W_{ij} and D_{ij} : Vorticity, deformation tensor
A_{ijkl} : 4^{th} order fiber orientation tensor
C_I : Interaction coefficient of value from 10^{-2}~10^{-3}

Fig. 2 shows the fiber orientation of the two runner systems. Shown is the maximal eigenvalue of the orientation tensor. The red color means high orientation, the blue color random orientation of the fibers. Clearly can be seen the welding lines split over the part. Welding lines occur where two melt fronts come together. They result in a weak stiffness because two reasons. First of all there is no fiber connection over the welding line and second the plastic does not fit very well together because of lower temperature at the melt front. Welding lines always show a random like orientation of the fibers. In design 1 there is one long welding line throughout the whole part. In design 2 there are two welding lines.

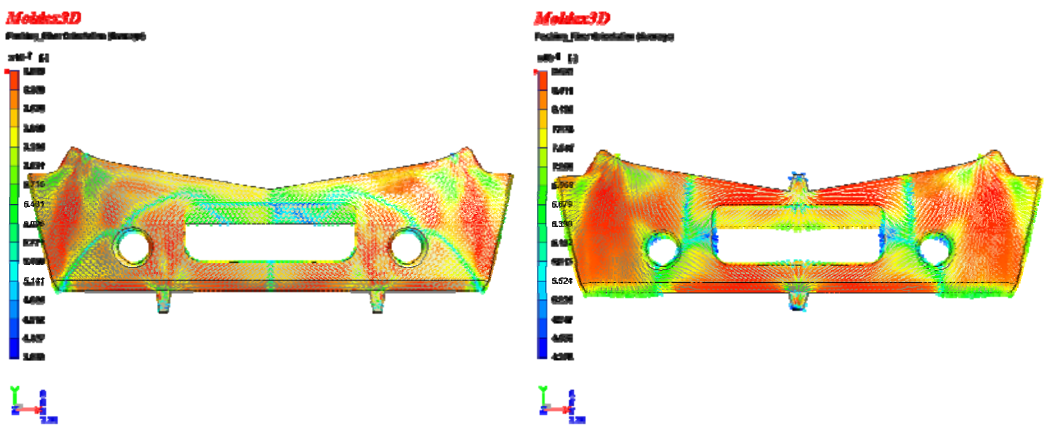


Fig. 2: Comparison of the fiber orientation

After the calculation of the fiber orientation the anisotropic mechanical properties has to be calculated. This is done by a so called micromechanical model like Halpin-Tsai or Tandong-Weng. These models provide the mechanical properties in the fiber system. The ingredients are the mechanical properties of the pure polymer und the glass fibers, the volume fraction and the length/diameter ratio of the fibers. Besides a direction depend Young Modulus one receives a direction depend Poisson ratio and shear modulus. Obviously the model only distinguishes between the fiber direction and the two orthogonal directions. To get the mechanical properties in the x-,y-,z- system, the local anisotropic properties have to be rotated by the fiber orientation tensor. This has to be done for each element

Through an interface between Moldex3D and Abaqus the anisotropic mechanical properties can be transferred for an anisotropic load case simulation. This transfer can also be done onto an independent Abaqus mesh. This interface is

Halpin-Tsai-Model

$$E_{11} = E_m \frac{1+\xi\eta\Phi_f}{1-\eta\Phi_f} \qquad \xi = 2\frac{l}{d} \qquad \eta = \frac{M_R - 1}{M_R + \xi}$$

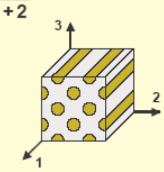
$$E_{22,33} = E_m \frac{1+\xi\eta\Phi_f}{1-\eta\Phi_f} \qquad \xi = 2 \qquad M_R = \frac{E_f}{E_m}$$

$$G_{12,13} = G_m \frac{1+\xi\eta\Phi_f}{1-\eta\Phi_f} \qquad \xi = 1 \qquad M_R = \frac{G_f}{G_m}$$

$$G_{23} = G_m \frac{1+\xi\eta\Phi_f}{1-\eta\Phi_f} \qquad \xi = \frac{K_m/G_m}{(K_m/G_m)+2}$$

$$v_{12,13} = v_f\Phi_f + v_m(1-\Phi_f)$$

$$v_{21,31} = v_{12}\frac{E_{22}}{E_{11}} \qquad v_{23,32} = \frac{1}{2}\frac{E_{22}}{G_{23}} - 1$$



E_{11} : E-Modulus longitudinal
$E_{22,33}$: E-Modulus transverse
E_m : E-Modulus matrix
E_f : E-Modulus fibre
G_m : Shear modulus matrix
G_f : Shear modulus fibre
v_m: Poisson ratio matrix
v_f: Poisson ratio fibre
l : Fibre length
d : Fibre diameter
Φ_f: Fibre volume fraction

developed by Moldex3D and is commercially available. The transfer can only be done from shell to shell or solid to solid. A reduction from solid to shell is not yet available. The interface is transferring the fiber orientation, the local stiffness matrix and the residual stresses for each integration point. Because of the transfer of the stiffness matrix the method is limited to linear calculation. For plastics this limitation is important as plastics reacts very strongly nonlinear. The residual stresses are overestimated and have to be corrected by some correction factor. To get ride of this problem a nonlinear stress calculation has to be implemented in Moldex3D.

In the future the interface to Abaqus will be expended so that nonlinear calculations are possible. Also simulation in the field of crash or fatigue will be possible. Also information about the location of welding lines or the pressure distribution during filling can be used.

3. Results

In Abaqus/Explicit a dynamic simulation is done. With a speed of 8 km/h and a weight of 4 tons the bumper hits a barrier. The simulation does not consider crash or break. It only considers elastic behavior.

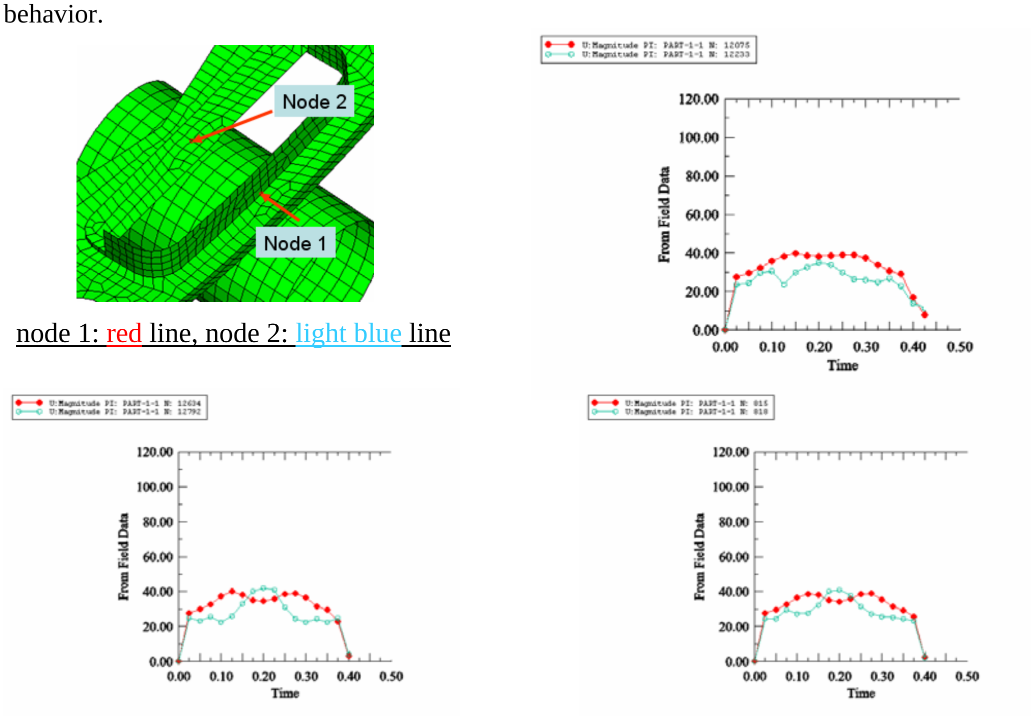


Fig. 3: Deformation history of the bumper, different designs

On the part there are two detection points to describe the deformation history. Fig. 3 shows the deformation history of the bumper. On the above right side the figure shows the deformation under the assumption of an isotropic material. It shows that the node 1 always experiences a higher deformation than node 2. As soon as we consider anisotropy the deformation shows a different picture. Under the fiber influence the node 2 experiences a higher deformation than node 1, independent of the runner design. The difference between the two runner designs is very small compared to the difference between isotropic and anisotropic calculation. Therefore it is very important to consider the fiber effects in the FEA. The calculation out of the isotropic simulation would be to better control the mechanical behavior around the area of point 1. But in reality – under consideration of anisotropy – the point 2 shows the more severe deformation. This is the risk of an isotropic calculation to make the wrong conclusions.