# SIMULATION OF COMBINED FORMING AND INJECTION MOLDING PROCESSES

Dipl.-Ing. Timo Mueller, Prof. Dr.-Ing. Frank Henning

Karlsruhe Institute of Technology, Lightweight Technology

***Abstract**: Injection and compression molding of fiber reinforced thermoplastic materials are proven and efficient processes to produce lightweight parts for automotive applications. By adding continuous fiber reinforced composites or sheet metal, the possible range of applications can be greatly increased while retaining the advantages of the molding processes, cost efficiency and design flexibility. So called hybrids parts are used for more and more applications, especially in the automotive industry. The reinforcements are formed by the closing motion of the mold and the flow of the molding material. A novel approach is suggested using Abaqus/Explicit coupled eulerian lagrangian simulation to model combined forming and molding processes. All existing material models and material data for forming simulation available in Abaqus/Explicit can be used. The molding material is simulated using VUEOS und VUVISCOSITY, enabling the use of material data available for commercial mold filling simulation tools. All thermoplastic molding processes, including injection molding, injection compression and compression molding can be simulated. Three examples are used to demonstrate the application to simple injection molding, a combination of injection molding and metal forming and a combination of compression molding and organic sheet forming.*

***Keywords**: Injection Molding, Compression Molding, In-Mold Forming, Overmolding, Metal Forming, Preimpregnated Sheet Forming, Organic Sheet Forming, Coupled Eulerian Lagrangian Simulation, ABAQUS CEL, Folgar-Tucker, Fiber Orientation, Tait Equation, pvT model, ABAQUS VUEOS, ABAQUS VUVISCOSITY, Hybrid Parts, Automotive Lightweight Technology, Long Fiber Reinforced Thermoplastics*

## 1. Introduction

Injection and compression molding of fiber reinforced thermoplastic materials are proven and efficient processes to produce lightweight parts for automotive applications. By adding continuous fiber reinforced composites or sheet metal, the possible range of applications can be greatly increased while retaining the advantages of the molding processes, cost efficiency and design flexibility. So called hybrids parts are used for more and more applications, especially in the automotive industry. The reinforcements are formed by the closing motion of the mold and the flow of the molding material.

Figure 1: Overmolded hybrid Part. Source: Engel Austria

Specialized software is used to simulate injection molding, but to date there is no solution to simulate the combination of forming and molding. ABAQUS Explicit is often used for sheet metal and composites forming simulation, but not for injection molding simulation.

For practical purposes, the forming and mold filling stages are simulated separate from each other. While injection molding simulation software allows simulating anisotropic or deformable inserts, the available material models are very limited and not suited for a complete forming simulation.

A Solution which combines the best of both worlds would be a great benefit for the development of hybrid fiber reinforced parts.

## 2. Abaqus CEL

For most applications ABAQUS\Explicit is used with a Lagrangian formulation. In a traditional Lagrangian analysis nodes are fixed within the material, and elements deform as the material deforms. Lagrangian elements are always 100% full of a single material, so the material boundary coincides with an element boundary. Many material models for different applications exist in Abaqus/Explicit, or have been implemented using VUMAT.

But simulating process like injection or compression molding, where very large deformations occur, is not possible, because the elements would deform to a point where they would not be able to function properly. Injection molding software packages use an Eulerian Formulation, where material flows through elements which are fixed in space. This is better suited for injection molding simulation, although problems arise if the mesh needs to change to account for a deforming insert or a changing mold cavity, for example in compression molding. The problem is to combine both kinds of simulation using different software packages.

Abaqus/Explicit offers what is called coupled eulerian lagrangian (CEL) analysis, which has been used for a wide range of specialized applications, but is less well known than the conventional functions of Abaqus/Explicit. CEL offers the capability to combine materials in conventional lagrangian formulation and eulerian formulation in one simulation, interacting trough Abaqus/Explicit contact. The lagrangian part of the simulation is set up in the same way as a normal Abaqus/Explicit simulation.

The eulerian part consists of a mesh defining the control volume in which the material can move and deform. Only 8-node hexahedra are available for eulerain meshes, one normal and one thermally coupled element. The eulerian mesh and lagrangian mesh can overlap, they will not interact with each other. Each eulerian element can either be empty, completely filled with material or anyway in between. This is tracked by its eulerian volume fraction (EVF). If a material is completely filled by one or more materials, its EVF is 1.0, if it is completely empty, its EVF is 0.0. Empty space is filled with what is called "void" material, which has neither mass not strength. An element with EVF 0.0 is completely filled with void material. To define the position of material inside the eulerian mesh at the beginning of the simulation, the EVF of the elements containing material is set to the appropriate value. Empty elements do not need to be defined, as they are by default filled with void material. Abaqus/CAE offers the volume fraction tool to compute the EVF of the eulerian mesh based on a geometric model of the material distribution.

Based on the EVF of each element, Abaqus computes the boundary of the fluid in each time increment. The surface reconstruction algorithm approximates the material boundaries within an element as simple planar faces, taking the EVF of neighboring elements into account. The surface may be discontinuous, and a fine mesh is required to get an accurate description of the materials position. Also, displaying the results in Abaqus/CAE can be a challenge.

Boundary Conditions for eulerian meshes are the same as in a conventional analysis, but some differences exist. Displacement boundary conditions cannot be used on eulerian meshes. Material that flows over the mesh boundary is lost. To stop material from flowing out of the mesh, the velocity boundary condition normal to the mesh surface is set to zero. This type of boundary definition can also be used to define a forced in- or outflow of material by setting a non-zero value. If material flows into the control volume, it will have the same state and material content as the material that exist within the element. If the velocity boundary conditions in-plane to the mesh boundary are set to zero, the fluid will stick to the boundary. There are some specialized eulerian boundary conditions to control in- and outflow of void and material.

Multiple eulerian materials interact with each other with a sticky behavior. It occurs by default. Much more possibilities exist for the contact between an eulerian and a lagrangian material. By default, the contact between these two is frictionless and cannot transmit tensile stresses. The eulerian surface can be used to define specific interactions with lagrangian surfaces.

Because the contact is based on the eulerian surface, the eulerian mesh does not need to conform to the shape of the lagrangian surface. It is always possible to use a simple, rectangular mesh for the eulerian domain. Thermal interactions are also possible, although gap radiation and gap conductance as a function of clearance cannot be used.

All Abaqus/Explicit material models can be used for the eulerian domain, but for mold filling simulation, the equation of state materials in combination with a viscosity model are the best suited ones. Abaqus also offers the possibility for user defined VUEOS and VUVISCOSITY.

## 3. Injection Molding Simulation

Injection molding simulation for thermoplastics has been used for long time, well proven models for injection molding materials exist, and a large amount of test data for numerous materials is available. For thermoplastic materials, the cross viscosity model is often used in a combination with a time-temperature shift function based on the work of Williams, Landel and Ferry (Williams, 1955).

$$\eta = \eta_{inf} + \left(\eta_0 - \eta_{inf}\right) \frac{1}{1+\left(\frac{\eta_0 * \dot{\gamma}}{\tau^*}\right)^{1-n}} \quad (1)$$

The melt viscosity $\eta$ is defined as a function of the infinite shear viscosity $\eta_{inf}$ which defines a lower limit for the viscosity and the zero shear viscosity $\eta_0$, which defines the upper limit the viscosity approaches for low shear rates. The stress level at which the transition between Newtonian and shear thinning behavior occurs is $\tau^*$. The behavior in the shear thinning area is defined by the shear rate $\dot{\gamma}$ and the power law index $n$. The Williams-Landel-Ferry equation changes the value for the zero shear viscosity depending on the temperature of the material. The formula uses three fitting coefficients $A_1$, $A_2$ and $D_1$, the melt temperature $T$ and a reference temperature $T^*$, which is typically set to the glass transition temperature of the material.

$$\eta_0 = D_1 * \exp\left(-\frac{A_1 * (T - T^*)}{A_2 + (T - T^*)}\right) \quad (2)$$

$A_2$ and $T^*$ can be further described as being dependent on pressure $p$ and additional fitting coefficients $A_3$, $D_2$ and $D_3$.

$$A_2 = A_3 + D_3 * p \quad (3)$$
$$T^* = D_2 + D_3 * p \quad (4)$$

The Cross-WLF viscosity model is available in Abaqus/Explicit. To incorporate the pressure influence on the glass transition temperature, it was implemented using VUVISCOSITY. The model and parameter sets typically available only describe the viscosity above the glass transition temperature of the model. The viscosity at glass transition temperature was therefore set as a limit. In conventional mold filling simulations using finite volume approaches, the infinite shear viscosity is often set to zero, but setting it to low, non-zero values often helps in improving simulation performance. Figure 2 shows the viscosity of a typical thermoplastic molding material.

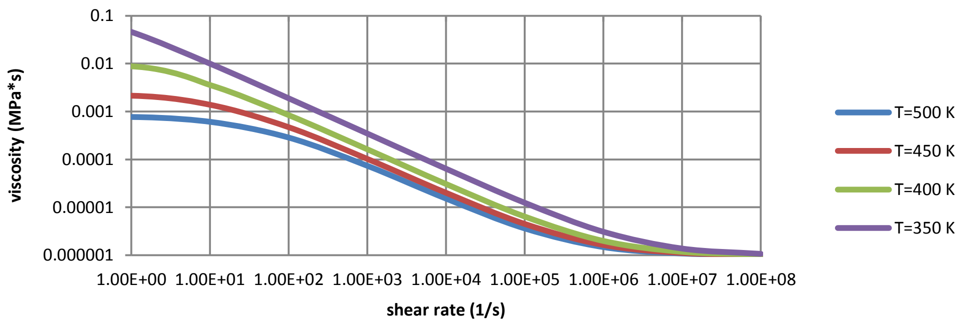


Figure 2: Viscosity for different temperatures at reference pressure

Different Equations of state have been proposed for thermoplastic materials. For semi-crystalline polymers, a two domain formulation of the Tait-Equation is often used. The model is based on the works of Tait (Tait, 1888), although not in the form originally proposed (Hayward, 1967). The basic equation calculates the specific volume of the material for a given temperature and pressure using the specific volume at reference pressure, $v_0$, pressure $p$, the constant $C$ with a value of 0.0894 and the function $B(T)$ describing the pressure sensitivity of the material.

$$v(T,p) = v_0(T)\left(1 - Cln\left(1 + \frac{p}{B(T)}\right)\right) \quad (5)$$

For injection molding, two different sets of parameters are used above and below the crystallization temperature.. An additional term $v_1(T,p)$ is added to adapt the formula to volume changes due to crystallization, some authors attribute this term to Schmidt (Schmidt, 1986).

$$v(T,p) = v_0(T)\left(1 - Cln\left(1 + \frac{p}{B(T)}\right)\right) + v_1(T,p) \quad (6)$$

$$T^* = b_5 + b_6 * p \quad (7)$$

The glass transition temperature $T^*$ is defined by a linear equation similar to one used for the WLF equation, depending on the pressure $p$ and two fitting parameter $b_5$ and $b_6$. For temperatures below the glass transition temperature the subscript s for solid is used, above the subscript m for melt is used.

$$T < T^*$$

$$v_0 = b_{1s} + b_{2s} * (T - b_5) \quad (8)$$

$$B(T) = b_{3s} * \exp(-b_{4s} * (T - b_5)) \quad (9)$$

$$V_{1s} = b_7 * \exp(b_8 * (T - b_5) - b_9 * p) \quad (10)$$

$$T \geq T^*$$
$$v_0 = b_{1m} + b_{2m} * (T - b_5) \quad (11)$$
$$B(T) = b_{3m} * \exp(-b_{4m} * (T - b_5)) \quad (12)$$
$$v_{1m} = 0 \quad (13)$$

The coefficients $b_1$, $b_2$, $b_3$, $b_4$, $b_5$, $b_6$, $b_7$, $b_8$ and $b_9$ are used to fit the model to the material behavior.
ABAQUS/Explicit does not offer this model, although for many applications a tabular form can be used. A different approach is to use the built-in functionality for user defined equations of state, VUEOS. VUEOS calculates the pressure and internal energy based on volumetric strain or density and temperature, making a rearrangement of formula 5 necessary. The equation with the crystallization term cannot be inverted, so the added term will be neglected.

$$p(\rho, T) = b_3 * \exp(b_4 * (T - b_5)) * \left(\exp(\frac{\rho}{C}(b_1 + b_2 * (T - b_5)) - 1\right) \quad (14)$$

$$\frac{dp}{d\rho} = b_3 * \exp(b_4 * (T - b_5)) \left(\frac{(b_1 + b_2 * (T - b_5))}{C}\right) \quad (15)$$

$$\frac{dp}{dE} = \frac{-b_4}{c_w} * b_3 * \exp\big(b_4 * (T - b_5)\big) \left(\exp(\frac{\rho}{C}\big(b_1 + b_2 * (T - b_5)\big) - 1\right) \quad (16)$$

Specific heat and thermal conductivity of thermoplastics are usually temperature dependent. This can be simulated using available ABAQUS/Explicit functionality.

## 4. Injection molding example

A simple model based on DIN EN ISO 294 will serve as a test case for mold filling simulation.

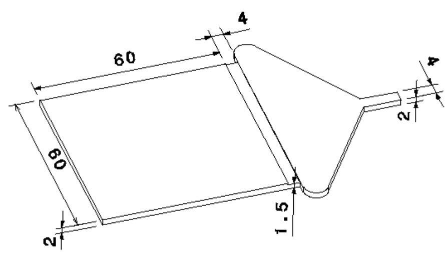


Figure 3: Mold geometry based on DIN EN ISO 294

The part was meshed using C3D8RT elements. Only 8-node hexahedra are available for the eulerian mesh, imposing restrictions on the geometries that can be meshed.

This is generally not a problem, because it is advantageous to model the mold as lagrangian or rigid body and use a primitive geometry overlapping the mold boundaries for the eulerian mesh. This approach offers more options for mechanical contact and heat transfer. For this model though, just the eulerian mesh was created, and boundary conditions were used to prevent material from leaving the mold, and to simulate the sticking of the molten material to the cavity walls. Heat transfer from the hot polymer melt to the colder mold walls is not simulated, because of the short duration of the actual filling process.

A constant flow rate was prescribed as boundary condition for the process, to achieve a filling time of 0.5 s. A velocity boundary condition using a smooth step function was defined on the gate cross section, leading to a constant inflow of material. If material flows into the eulerian mesh crossing the mesh boundary, ABAQUS/Explicit assumes that the material flowing into the elements will have the same state and the volume content as the material that is already present in the elements, so an initial charge was defined at the gate. A temperature boundary condition was also specified on the gate cross section, so all material enters with melt temperature.

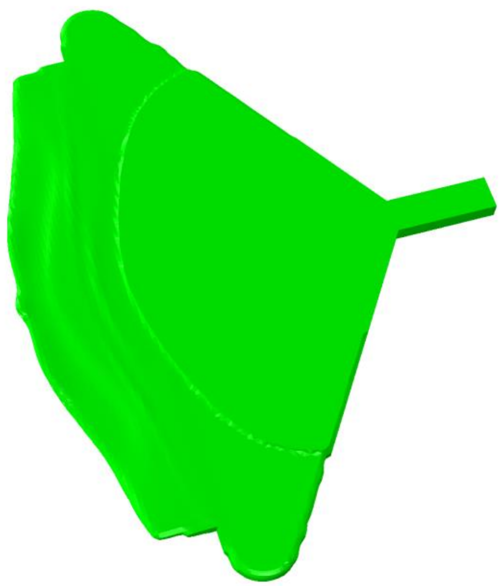

Figure 4: Flow front during filling.

## 5. Metal Forming

Metallic inserts in injection molded parts are widely used for different applications. In most cases, no large deformations of the metal occur, and the process can be simulated using conventional injection molding software. Using the injection molding pressure to form metallic inserts offers advantages for connecting the metallic and plastic parts (Al-Sheyyab, 2008), or to create geometries that cannot be produced using conventional metal stamping tools.
The forming of a cup from stainless steel using a deep drawing process is used as an example. A circular steel blank is formed into a cup similar to experiments used to characterize the deformation behavior of steel. Instead of a deep drawing press, an injection molding machine is used to clamp the steel blank, and instead of a solid punch material is injected onto the blank to deform it.

Figure 5: Stainless steel cup formed using injection molding

The material data for the stainless steel, type 316, were taken from Hussaini (Hussaini, 2014). Temperature dependent plasticity up to 300° C was modeled using available ABAQUS/Explicit models. The model consists of four parts, the eulerian mesh, the lower part of the mold called the die, the blank holder and the steel blank.

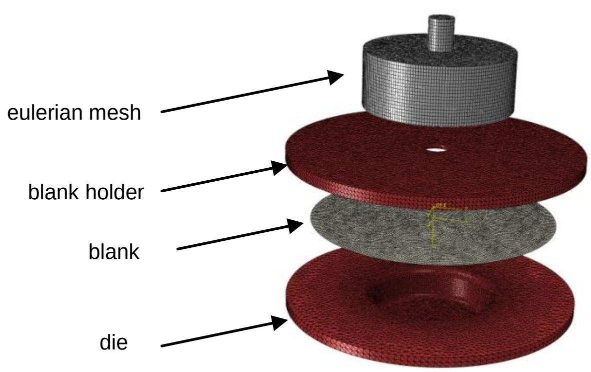


Figure 6: Components of the model.

The blank holder clamps the sheet metal to prevent it from buckling while allowing it to move in plane during the drawing process. The clamping force is applied prior to injection. Different interaction properties are used for the contact pairs. A velocity boundary condition is specified on the gate surface to make material flow into the mold cavity. As soon as the free volume inside the mold is filled with material, pressure builds up and the material starts to deform the steel blank.

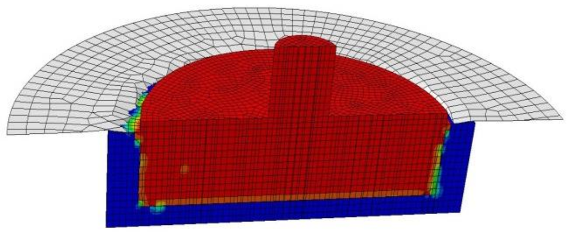

Figure 7: Eulerian material and deformed steel blank.

After the injection, the steel blank is formed into a cup similar to the experiments. Mass scaling can be used to speed up the simulation, because the high stiffness of the steel tends to define the minimal time increment.

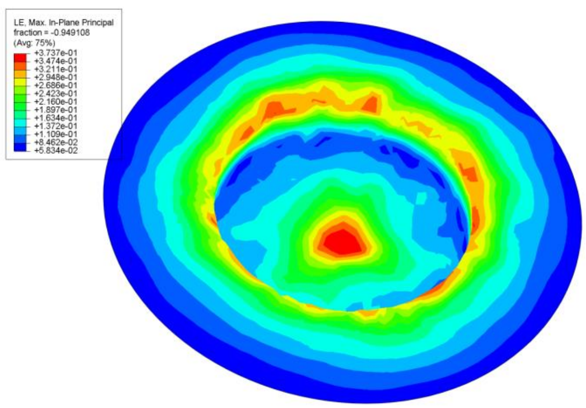


Figure 8: Deformed steel blank after injection molding.

## 6. Preimpregnated Sheet Forming and Compression Molding

Preimpregnated sheets are made of glass or carbon fiber fabrics impregnated with a thermoplastic matrix. Forming and overmolding organic sheets in allows to produce light yet economic parts. The continuous fiber reinforcements provide the load carrying ability of a such a part, while overmolded features like ribs, bosses or gussets add additional functionality. By using Abaqus CEL in combination with the Abaqus fabric material model, forming and overmolding of organic sheets can be simulated. The modeling of overmolding processes is not limited to injection molding, all kinds of processes using thermoplastic materials can be simulated, including injection compression molding and compression molding. As an example, the model of a beam is simulated. The beam consists of a single layer of organic sheet, 1 mm thick, made of a balanced plain weave fabric and a polypropylene matrix, and a layer of polypropylene with long glass fibers. A rib reinforces the beam in the middle. Compression molding will be used for overmolding. Only half the beam is simulated to reduce the computational cost.
The model consists of five parts. Upper and lower mold surfaces are modeled as rigid bodies, the lower one is fixed, and the upper one is moving. The organic sheet is modeled using 3D-shell elements and the Abaqus material model for fabrics. The modeling details of the organic sheet are not part of this work, but the main characteristics, large shear deformation with locking angle, rotation of warp and weft during shear, strong temperature dependence, and different behavior for tension, compression and bending are simulated. Multiple sheets would also be possible. The fourth part defines the initial charge, where the overmolding material is positioned at the beginning of the compression molding process. The fifth part is defining the eulerian domain.

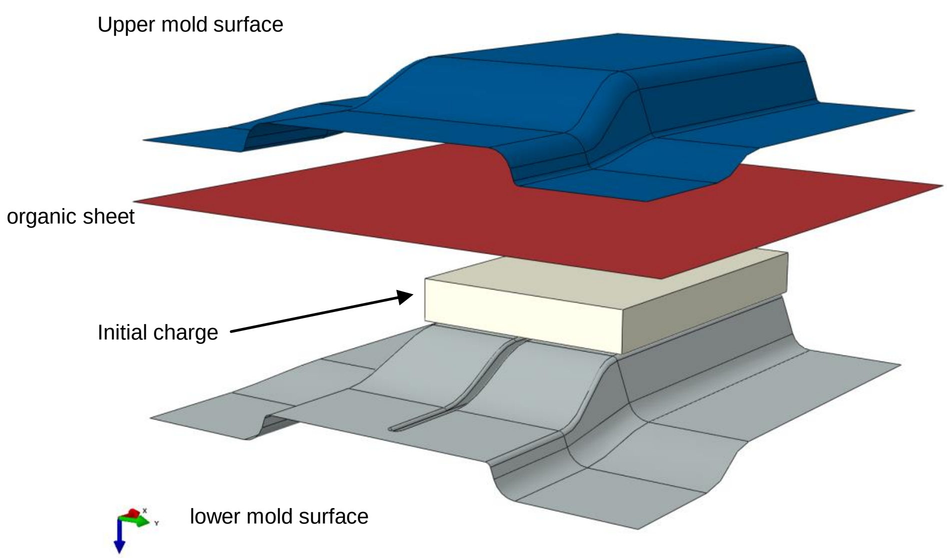


Figure 9: Constituents of the model. The eulerian domain is not depicted.

The organic sheet is pretensioned in the weft direction to keep it from wrinkling during the process. The volume fraction tool is used to define the initial volume fraction of the eulerain elements in the area of the initial charge. The initial charge is at melt temperature at the beginning of the process, and the organic sheet is also heated to forming temperature. Both mold surfaces are set to a constant temperature. A boundary condition is applied to the sides of the eulerian mesh to prevent material from flowing out of the tool. The speed of the upper mold is slowing down the closer it comes to the final part thickness. The contact between organic sheet and eulerian material is established before the upper mold surface comes into contact with the initial charge.

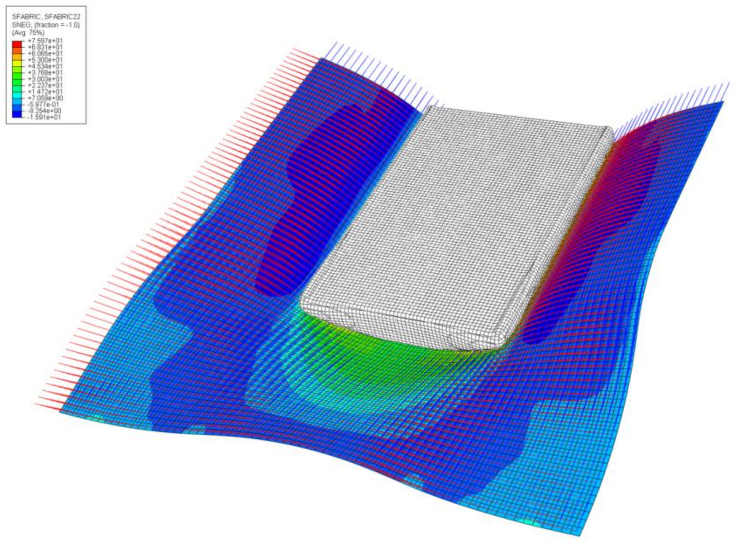


Figure 10: Organic sheet with material orientations and Isosurface representation of the eulerian material in the earlier stages of the compression.

The eulerian material sticks to the organic sheet. As soon as the upper mold surface starts to compress the eulerian material directly, the rate of flow increases because more material is displaced. The material flows along the cavity faster than into the rib, as can be seen in Figure 11. This behavior is often observed in experiments. Conventional mold filling software tends to predict a much faster filling of ribs.

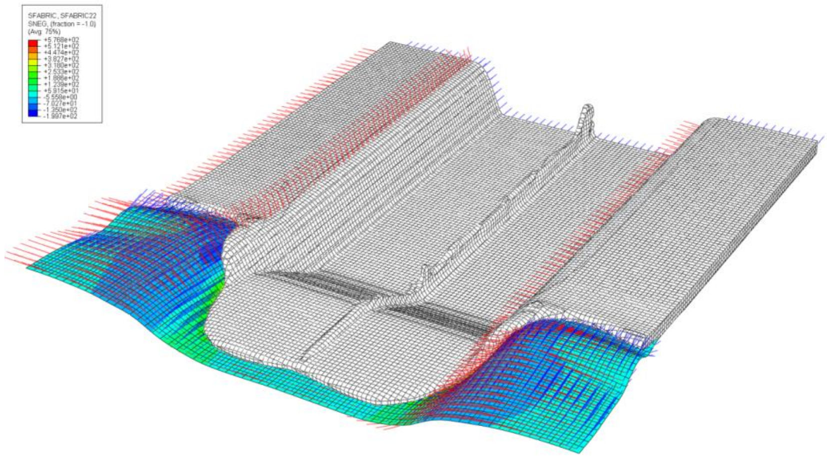


Figure 11: Organic sheet with material orientations and Isosurface representation of the eulerian material near the end of the compression.

## 7. Conclusion & Outlook

Abaqus/Explicit offers a unique possibility for the process simulation of hybrid parts made from thermoplastic molding material and steel or continuous fiber reinforcements. The main advantages are that available material models for forming simulations can be used, and the flexibility to model many different processes like injection or compression molding. A disadvantage is the high computational cost of the simulation.

To improve the performance of mold filling simulation, two features available in Abaqus could be used. Depending on the process, large areas of the eulerian mesh might not be used the whole time. Especially for combined compression molding and forming processes, the eulerian mesh needs to be large enough to incorporate the initial charge and one or more organic sheets, while the elements must small enough to allow for a correct simulation when the tool is close to its final position. A much more efficient solution would be to use eulerian mesh motion, where the mesh moves and deforms to follow the motion of the material inside.

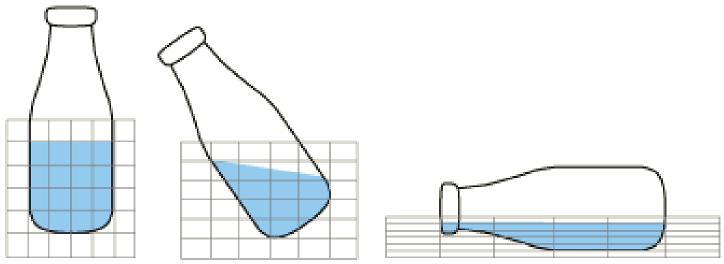

Figure 12: Eulerian mesh motion. The mesh is deformed to enclose the eulerian material (blue).

The other feature is adaptive mesh refinement for the eulerian domain. Abaqus refines elements with sharp gradients of density or pressure, or elements in contact with lagrangian bodys, also leading to a more efficient solution, because the initial element size can be defined coarser without losing accuracy.
The most important area for improvement is the implementation of a solution so simulate the orientation of fibers in the thermoplastic material, for example the Folgar-Tucker model or the ARD-RSC model for long fiber reinforced materials.

## 8. References


Abaqus Users Manual, Version 6.14-2, Dassault Systémes Simulia Corp., Providence, RI.

Al-Sheyyab: "Light-Weight Hybrid Structures - Process Integration and Optimized Performance", Institute of Polymer Technology (LKT), 2008

Hayward, A. T. J., "Compressibility equations for liquids: a comparative study ", British Journal of Applied Physics, Volume 18, Number 7, 1967

Hussaini, S. M., Singh, S. K., Gupta, A. K., "Experimental and numerical Investigation of Formability for Austenitic Stainless Steel 316 at elevated Temperatures", Journal of Materials Research and Technology, Volume 3, Issue 1, January–March 2014

Schmidt, T.W., Menges, G., "Calculation of the Packing Phase in Injection Mmolding with a two-layer segment Model", Proceedings of the 44th ANTEC, SPE, 1986

Tait, P. G., "The  Voyage of H.M.S. Challenger", Collected Scientific Papers, Vol. 2 (Cambridge: Cambridge University Press), 1888

Williams, M. L., Landel, R. F., Ferry, J. D., "The Temperature Dependence of Relaxation Mechanisms in Amorphous Polymers and Other Glass-forming Liquids". Journal of the American Chemical Society 77, 1955

## 9. Acknowledgement

These investigations are carried out through the R&D activities of the project "KITe hyLITE Plus: Technologies and network development for hybrid lightweight construction solutions in the automotive industry". This project is funded by the European Union through the program "European Funds for Regional Development" as well as state government of Baden-Wuerttemberg in Germany. Administrative agency of this program is the Ministry of Rural Development, Food and Consumer Protection. For more information visit www.rwb-efre.baden-wuerttemberg.de

Europäische Union
Europäischer Fonds für regionale Entwicklung

investition in Ihre Zukunft!

Baden-Württemberg
Ministerium für Wissenschaft, Forschung und Kunst