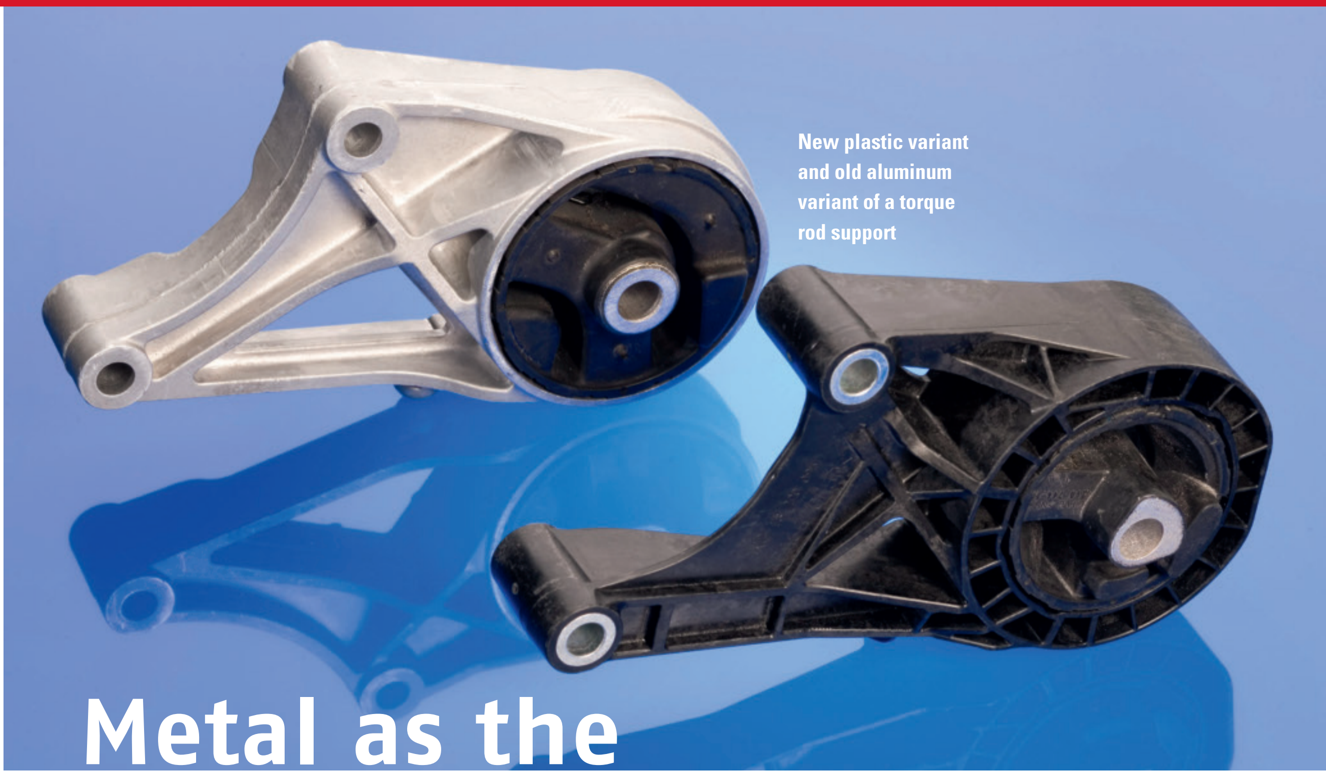

New plastic variant and old aluminum variant of a torque rod support

Metal as the Virtual Benchmark

Integrative Simulation. With Integrative Simulation, part three, which has now been further refined once again, it is possible to predict the strength and fracture behavior of highly-stressed plastic parts, such as engine supports or chassis bearings, in a highly accurate manner. This makes it possible to implement applications in plastics that were formerly confined exclusively to metals.

STEFAN GLASER
ANDREAS WÜST
BERNHARD AUMER

PE104283

Alongside the aspects of safety and comfort, the development of state-of-the-art vehicles is currently being driven ahead by the need to save costs and, above all, to bring about weight savings and hence also achieve a reduction in consumption and emissions. Although a large number of steel and aluminum parts in the engine compartment have given way to lighter-weight plastic parts with a higher level of functional integration, load-bearing structural elements have so far been excluded from this. Power train bearings, such as engine bearings, spring supports and chassis components subject to high mechanical stresses have also been made predominantly of metals to date. Plastics were scarcely used in this field, since little was known about the behavior of plastics under a high level of stressing, and it was not possible to design plastic parts with a sufficient degree of accuracy.

For plastics to make their way into power train bearings, they need to have strength properties that come close to ▶

Stress-strain Curves

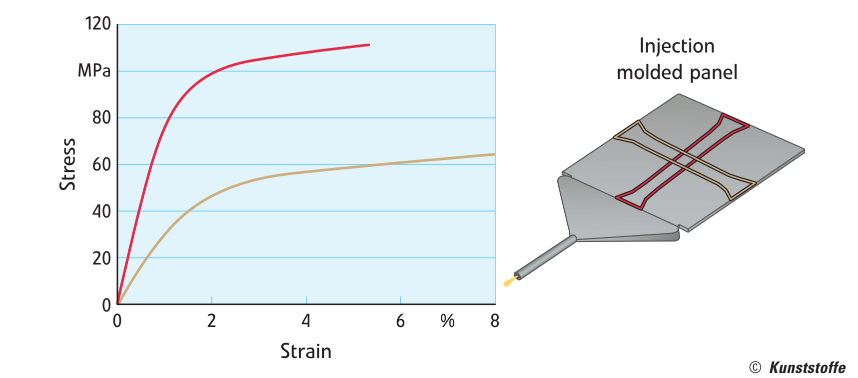

Fig. 1. Material behavior as a function of the fiber orientation

Translated from Kunststoffe 7/2008, pp. 88–92

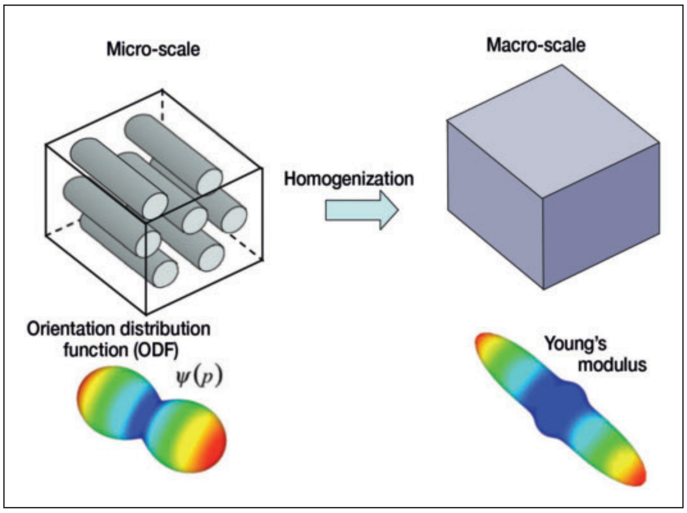

Fig. 2. Virtual modeling of compounds

those of metals. Only engineering plastics are suitable here – in the form of compounds incorporating reinforcing fibers. This reinforcement makes the local description of the complex material behavior of plastics even more complex: on the one hand, the reinforcing fibers introduce a directional dependency into the material, which varies at each point of the part and is highly dependent on the production process, and, on the other hand, complex phase interactions exist between the plastic and the reinforcing fibers, which have a key influence on the compound strength.

With the Integrative Simulation method developed by BASF, it is now possible to design highly-stressed parts in plastic in a highly accurate manner in terms of their mechanical strength. This is achieved on the basis of a special material model which – in the same way as the crash simulation [1 to 3] presented four years ago – describes key aspects of the material behavior for parts made of fiber-reinforced plastics [4, 5]. While, in the case of the crash simulation, the part behavior is studied during short-time, high-speed impact, it is now a matter of observing mechanical strength under static loading. The pedals, engine bearings and chassis bearings, or steering rods, that have so far been made of cast aluminum, for example, are now also of interest for plastics injection molding. This is because they can now be designed on a virtual basis at the computer, using the Integrative Simulation tool that has been further refined. As in a large number of other cases, this not only leads to a considerable weight reduction but also makes it possible to design more complex parts, integrate additional functions and hence reduce the costs by comparison with conventional parts.

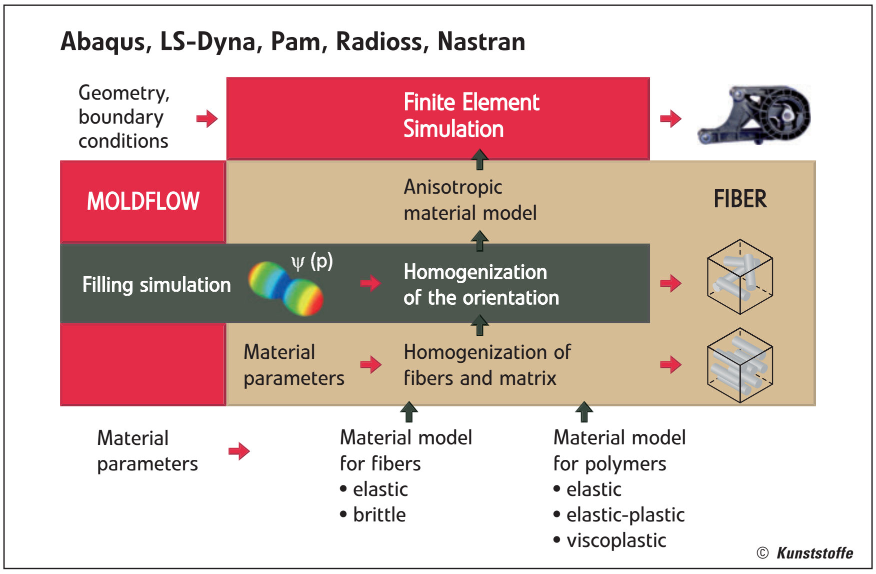

Fig. 3. Integrative Simulation: implementation

Describing the Behavior of Plastics Numerically

One of the biggest challenges in the strength calculation for thermoplastic components is achieving a numerical description of the typical behavior of plastics. The nonlinear viscoplastic behavior of the thermoplastic matrix plays a key role here. Hence a large number of plastics have a yield stress in the compression range that is considerably higher than their yield stress in the tension range. In addition to this, in the case of high-level strains, inelastic components remain, which no longer fully relax again when the load is removed. Plastics thus display highly complex, nonlinear behavior. Further difficulties are encountered in the numerical treatment of the components studied here due to the short glass fiber filling in the matrix. The orientation of the fibers during the filling process gives rise to anisotropic, or direction-dependent, mechanical properties [6]. This not only means anisotropic stiffnesses but also anisotropic values for the yield stress and elongation at break. The numerical material description used for the fiber-reinforced plastic must, therefore, be in a position to describe the anisotropy through the orientation of the glass fibers, the nonlinear stress-strain relationship, the strain rate dependence, the anisotropic failure and the tensile/compressive asymmetry. In addition, it must be possible to readily link the program system to the process simulation. These requirements on the material model are far more demanding than what it was possible to achieve in a CAE design process just a few years ago.

Anisotropic Material Properties

Figure 1 sets out a number of the properties of the polymer that have been mentioned [6]. The stress-strain curves (tension) shown differ on the basis of their orientation (longitudinal and transversal). At right angles to the fiber orienta-

i Manufacturer

BASF SE
Fachpressestelle Kunststoffe
KT/KC - E100
D-67056 Ludwigshafen
Germany
Tel. +49/6 21/ 60-43348
Fax +49/6 21/60-49497
www.basf.com

tion, a considerably higher elongation at break is achieved than longitudinally to this. Conversely, the tensile stress at break at right angles to the fiber orientation is lower than in the longitudinal direction. These effects can be illustrated through the stress flux in the compound. Stress acting along the length of the fibers flows primarily into the fibers, while, with transverse stressing, the stress is eliminated via the matrix. The difference can certainly attain an order of magnitude of two to three. It is thus incorrect to assume that use can be made of a universally valid, global scaling factor which scales down the longitudinally measured curves in order to make allowance for the orientation. A scaling factor of this type can only be found for a special response by the structure (e. g. a shift) in a special load case. If this factor is then used for the same component in a different load case, this will generally lead to incorrect statements. This can be due to the fact that, in the first case, for example, a transversally-oriented area of the component has the biggest influence on the response behavior, while, in the second load case, a longitudinally-oriented area in a completely different section of the component is responsible.

Numerical Material Model and Establishment of the Data

The failure behavior of a specific material is generally highly dependent on the micromechanical data of the fiber-matrix composite under observation. A numerical material model that is capable of describing the effects mentioned has been developed at BASF and integrated in FE analysis software. The numerical description is performed in accordance with a material law which is based on a viscoplastic formulation for the polymer material and an elastic model for the fibers. This is combined with a micromechanical model for describing the composite material, i. e. the fiber-reinforced polymer material. The micromechanical model of the composite material is based on a homogenization process according to Mori and Tanaka and also Eshelby [1, 7 to 9]. Here, the contributions to the material behavior of the two phases, i. e. the polymer and the fibers, are weighted in respect of each other in numerical terms. The parameters employed are the fiber content and geometry and also the orientation distribution density of the fibers.

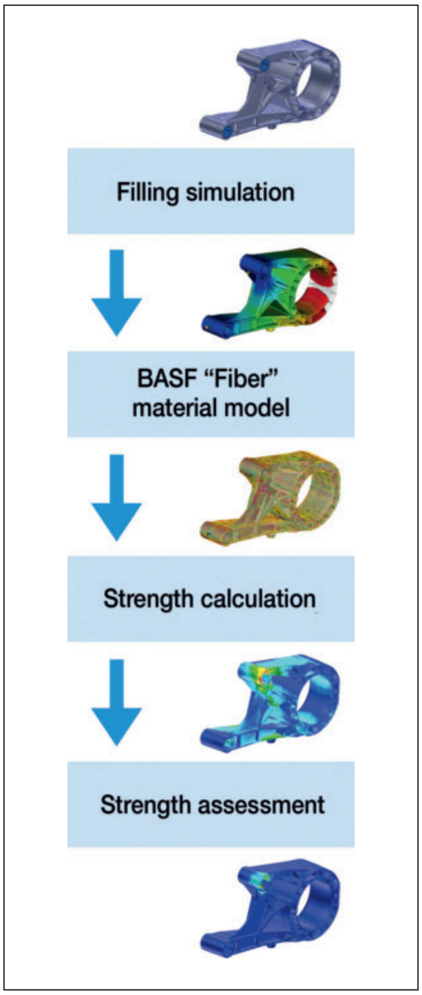

Fig. 4. Integrative strength simulation: dataflow

The material law can be used to describe the anisotropy due to the fibers contained in the polymer, the non-linearity and the dependence on the strain rate resulting from the polymer material, and also the failure behavior. Failure occurs if the polymer matrix fractures, the fibers break, or the matrix becomes detached from the fibers. The material law can additionally be readily linked to a process simulation.

Figures 2 and 3 show the main ideas behind the Integrative Simulation method. The statistical orientation distribution function (ODF) describes the number of fibers that can be found at each material point of the plastic part in each given direction. The ODF must be evaluated at each element integration point in an FE analysis. Working on the basis of the ODF, the fiber and matrix material are also homogenized to give a homogenous ▶

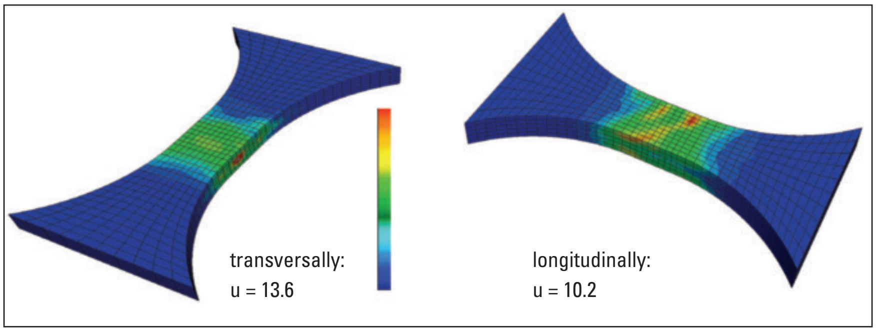

Fig. 5. The degree of stressing on a tension bar (strain given in mm; the color indicates the degree of stressing, e.g. red signifies a high level of stress)

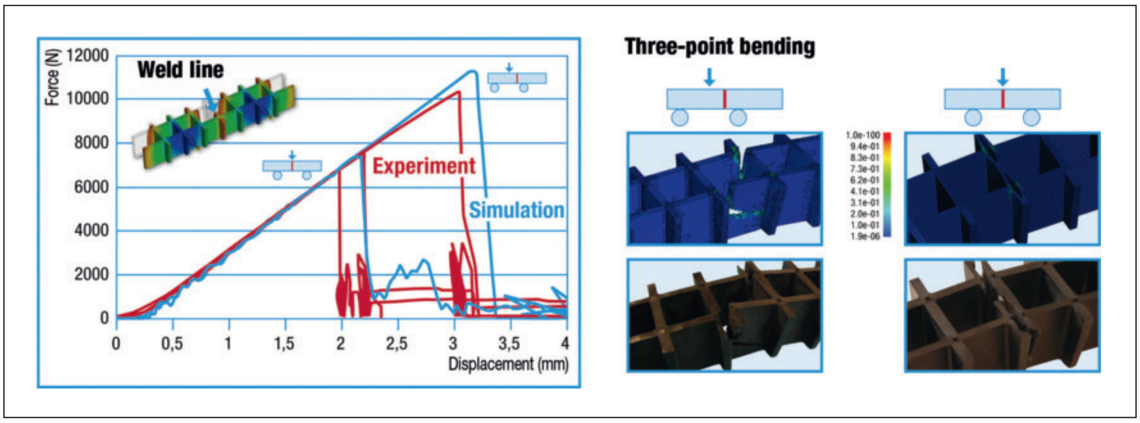

Fig. 6. Results of the three-point bending test on a ribbed structural inlay (right: the two test setups, left: a plot of the force in N versus the displacement in mm for both tests, showing the simulation and experiment in each case)

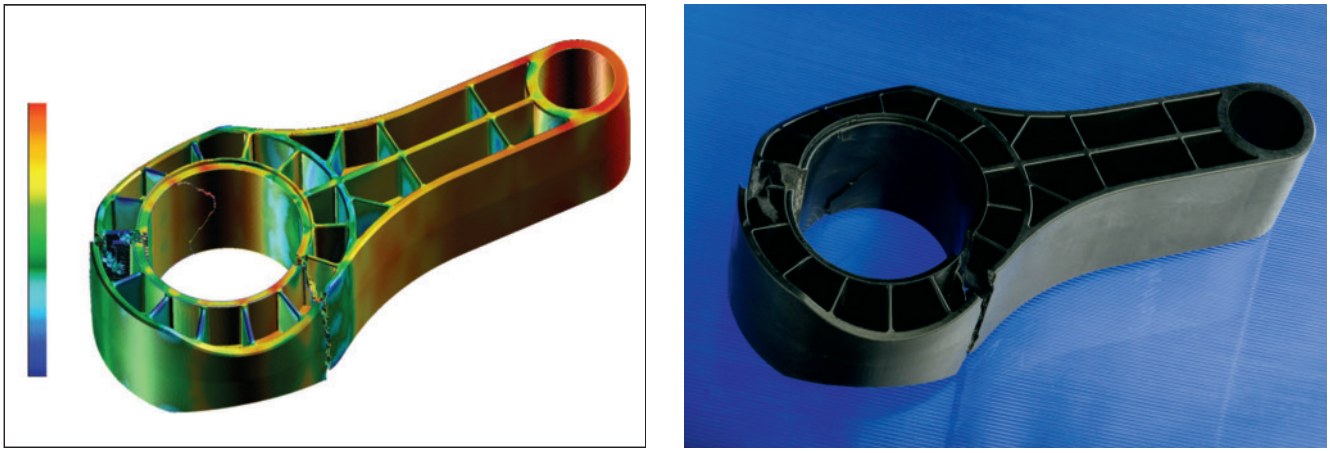

Fig. 7. The prototype of a chassis support in the high-performance polyamide, Ultramid CR, breaks at the predicted point

model for the composite material. Figure 4 shows the data flow of the Integrative Simulation using as an example the torque rod support that is meanwhile ready for production.

It is clear that a material model can only be as good as the measured data on which it is based. In order to establish the anisotropic and strain-rate dependent characteristics of the polymer, an optical high-speed measuring system has been developed at BASF, which includes the control and evaluation software [6]. These material data are already available for a series of high-performance polyamides from the BASF Ultramid CR range. These polymers are precisely tailored to the stringent requirements placed on the components and are subject to a particularly tight material specification and intensive quality controls.

Case Studies for Applications

Tension bar: The simplest part that can be used for verifying the method is a tension bar that is cut out of plastic panels (150 mm × 150 mm). Working on the basis of the computed fiber orientation, the tensile test is simulated on a specimen cut in the injection direction and another specimen taken at right angles to this (Fig. 5). The colors on the specimens indicate the level of loading. This constitutes a measure of the damage to the material and serves as an indicator for material failure. The force/displacement profile is similar to that set out in Figure 1. The simulation reflects the experiment with a good level of accuracy, in line with the requirements.

Rib structure: The mechanical strength of a more complex ribbed part is established in the three-point bending test. The top left of Figure 6 shows mold filling via two lateral gating points and the calculated fiber orientation. This gate position leads to a weld line in the center of the part. In one loading case, the part is bent through the middle and fails in the weld line, which runs longitudinally, in the central rib. The component and the simulation are shown on the extreme right of Figure 6. With a homogeneous material, the component would be expected to tear in the corners of the ribs on account of the notch effect. The fiber orientation, however, means that the weakest point is located directly in the central rib, which is where the component actually fails. In a second load case, the component was bent off-center, one rib field further along. The observed and calculated failure pattern is shown in the center of Figure 6. It is very clear to see that the simulation is capable of depicting inhomogeneous material properties and predicting the failure behavior. The left section of Figure 6 shows a comparison of the force/displacement diagram for the two variants, both as measured and as calculated in each case [10, 11]. In order to investigate the influence of fiber orientation still further and achieve the optimum fiber orientation for the flexural loading case, the mold was converted so that the part was injected in the longitudinal direction. The load-bearing capacity of the part then increases by some 30 % to approximately 14,000 N.

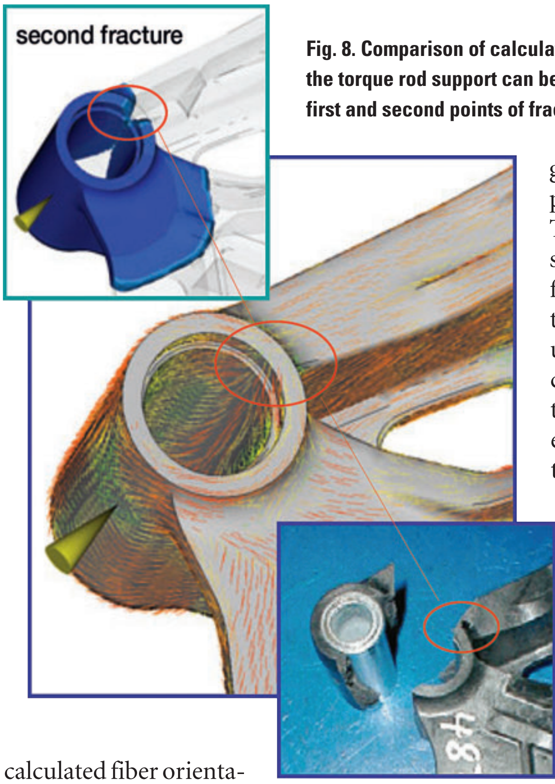

Fig. 8. Comparison of calculation and experiment; the failure of the torque rod support can be predicted highly accurately; the first and second points of fracture are shown

Chassis support: The strength layout of a support shown in Figure 7, a prototype in Ultramid CR, shows the critical points of the part. As predicted in the simulation, the part breaks in the central region. Other critical points are to be found in the end zones, at the lug on the inside, and also opposite the weld line.

Torque Rod Support from ContiTech Vibration Control for Opel

A further example of a mechanical strength layout using the Integrative Simulation tool developed by BASF is the engine bearing for the Opel Vectra that has been in serial production since 2006. In this case, it has proved possible to replace an aluminum part in the engine compartment by a plastic part and thus save a considerable amount of weight, espe-

Fig. 9. Predicted fracture load: 14.4 kN; tested fracture load: 14.4 to 15.0 kN

cially in the front axle zone. The part breaks at two points, both of which have a different fiber orientation (Fig. 8). Figure 9 shows a comparison of the experiment and the prediction of the maximum possible loading. The fracture occurs at the predicted point under the predicted conditions [12].

Conclusion

With the aid of the Integrative Simulation method that has been further refined once again by BASF, it is now possible to predict, with a high degree of accuracy, the mechanical strength and fracture behavior of highly-stressed plastic parts like engine or chassis bearings giving consideration to the production process. By projecting the local material properties (fiber orientation), which are dependent on the production process, onto the mechanical FE model of the part, it is possible to increase the information value of the results to a previously unattainable level. It is thus now possible to implement applications in plastics that were previously reserved exclusively for metals. Integrative Simulation permits the precise determination of local material behavior. Together with numerical optimization processes, which permit an optimum layout of the molded part, this makes it possible to design parts to withstand specific loads in a rapid and targeted manner. A large number of validations (comparisons with tests on real-life parts) have confirmed the simulation results in full. The torque rod support developed by ContiTech and BASF for Opel and produced by ContiTech, which recently entered serial production, provides particularly impressive confirmation of this simulation instrument. ■

REFERENCES

Interested readers can request the extensive literature list from the following e-mail address: sabine.philipp@basf.com

THE AUTHORS

DR. STEFAN GLASER, born in 1961, is head of the unit for the development of CAE methods and their application to engineering plastics at BASF SE, Ludwigshafen, Gemany.

DIPL.-ING. ANDREAS WÜST, born in 1965, works on the development of CAE methods and their application to engineering plastics at BASF.

DIPL.-ING. BERNHARD AUMER, born in 1957, works in application development for engineering plastics at BASF, in the specialist field of the automotive power train.

Contact: sabine.philipp@basf.com