# HIGH PERFORMANCE PLASTIC COMPONENTS FOR ENGINE MOUNT APPLICATIONS

*Scott Bykowski, Hans-Jürgen Karkosch, Diethard Schneider*
*ContiTech Vibration Control*
*Holger Klink, Marios Lambi BASF*

## Abstract

In the face of dwindling resources, rising energy prices and increasing environmental pollution, reductions in consumption and emissions are topics of increasing importance in all fields of technology. To achieve these goals specifically in the automotive industry, new engine concepts are needed in connection with thorough-going implementation of lightweight construction. Whereas weight-optimized plastics components are already utilized in many vehicle subsystems and components, steel and/or aluminum structures are usually used for load-bearing structural elements. This statement also applies fundamentally for the engine mounting subsystem. In this area, plastics components have been used previously only for subordinate, moderately loaded semi-components.

Now for the first time, a mechanically highly-loadable torque reaction mount has been conceived as a plastics structural part and implemented in the series production of a vehicle with a transverse-mounted engine. In addition to weight reduction, it also helps create a more advantageous load distribution on the axles. Load reduction on the front axle has positive effects on driving dynamics and safety.

The paper begins by stating fundamental requirements for components of engine mounting systems. The principle procedure in developing load-bearing plastics components includes the topics of integrative simulation, laboratory component tests and in-vehicle testing.

## Introduction

Resources are becoming increasingly scarce, energy prices are rising and environmental pollution is increasing. Against this background, the subject of reducing consumption and emissions is becoming increasingly important in all areas of technology. Meeting the targets in the automotive industry in particular will require new drive concepts combined with the consistent implementation of lightweight construction. Weight-optimized plastic components are already being used in many vehicle subsystems and components, but steel and/or aluminum are normally used for load-bearing elements. In general, this goes for the engine suspension subsystem as well. In this area, plastic components were previously only used for sub-ordinate, moderately-loaded part components.

The article deals with series development of mechanically high-load bearing plastic components for engine suspension of automobiles. The use of lightweight construction components contribute to reducing weight and a more favorable axle load distribution. The reduction of the front axle load positively affects driving dynamics and safety. The present article discusses the following points:

- Fundamental requirements for components of engine suspension systems
- Principal procedures when developing highly resilient plastic components
- Integrative simulation for computer-aided component design
- Component tests in laboratory and vehicle trials

## Engine Suspension Requirements

The development of an engine mount system is a complex process requiring close coordination of a range of departments. An interdisciplinary working group collaborates from the beginning of a development project. This team includes employees from the system simulation, design, analysis, noise & vibration and durability. The fundamental tasks in developing an engine mount system are shown in Figure 1.

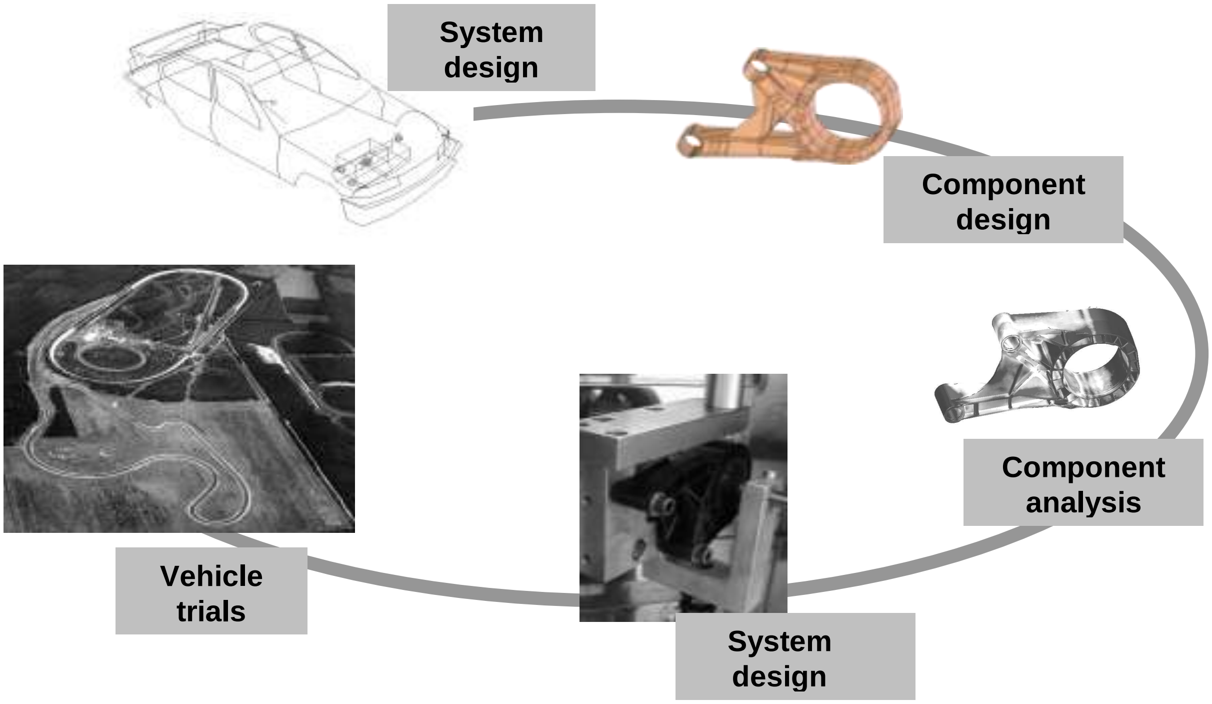


*Fig. 1: Development process for engine mounting system*

The engine mounts are the primary connecting link between the engine/drive unit and the chassis or body. Thus, it is subject to a variety of influences and interactions of a range of sub-systems. The main tasks which must be performed by the mounting system include the following [1]:

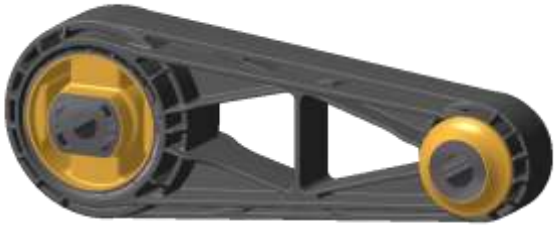
- Positioning of the engine/gearbox assembly
- Support function (absorption of the weight forces)
- Structure-borne noise insulation
- Support of the forces and moments which occur when driving (e.g. drive momentum, mass forces due to acceleration, deceleration or cornering)
- Limiting movements of the assembly (with the aim of guaranteed avoidance of contact with the body)
- Damping of shocks and vibrations caused by the road surface (e.g. shaking)
- Holding the engine in the event of a crash

In addition to the requirements mentioned here, compliance with the component properties at the specified ambient conditions must be guaranteed. In the engine mount area, the standard temperatures are between -30°C to 120°C. Also, heavy soiling can impair the function of the mount.

## Selection of Suitable Plastic Components

The described installation location and the requirements of the engine mount result in a wide range of options for the use of plastic as reinforcement in the engine mount. As these components are generally made of steel or aluminum today, a high level of design stiffness must be achieved for implementation in plastic. This results in new geometries. Figure 2 shows examples of components which are particularly suitable for implementation in plastic based on their requirement profile. The development of such components is explained below using a torque reaction mount as an example.

**Torque reaction mount** **Engine mount**



**Roll restrictor**

*Fig. 2: Examples of the use of plastic in engine mounts*

## Development Methods and Component Design

Figure 3 shows the development sequence of a plastic component using the torque reaction mount as an example. Here, the installation space for the implementation of bearing components as a connecting element between the engine and the body is critical. The connections to the powertrain and the body are prescribed. Topology optimization is a proven method for an initial inspection of the engine mount which must be implemented in the existing installation space. Initial computer models were drawn up on this basis. At this early stage, the subsequent component weight can already be estimated.

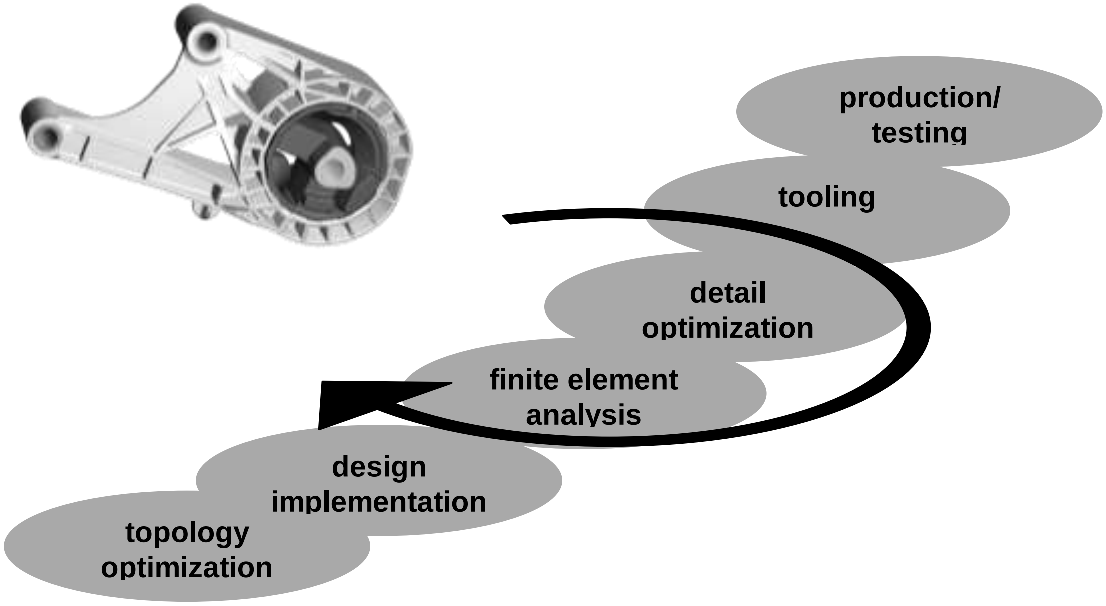
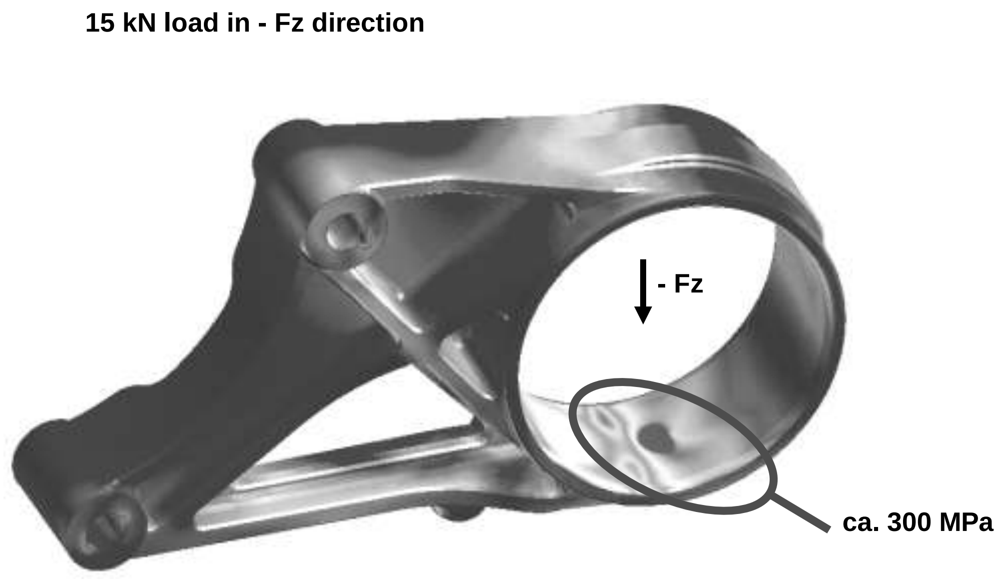


*Fig. 3: Development process*

Comprehensive calculations and simulations are performed as the key part in the development sequence. Based on the results, stress analyses, tool concepts and manufacturing parameters are specified. Only then is the injection molding tool commissioned. This procedure has now proven itself in a series of components. Consistency right up to series production is guaranteed by in-house manufacturing of both the elastomer and plastic components.

## Computer Structure Design – Integrative Simulation

Before the topology could be optimized and design implemented, an initial assessment of the feasibility of the torque support described here was made. For this, the existing aluminum design based on a standard PA 66 GF50 was calculated with conventional simulation methods. If the calculated stresses in such a procedure are significantly greater than the permitted values for the respective plastic, there is generally no point in developing the component further with conventional simulation methods. Even if the stresses can be controlled with extensive use of materials, the larger component volume and resulting increased use of plastic renders its economic feasibility questionable.



**15 kN load in - Fz direction**

*Fig. 4: Feasibility study of initial design (aluminum)*

Figure 4 shows the results of the estimation of the loaded condition in the Fz direction. The maximum stress is approx. 300 MPa. If one takes into account that standard PA 66 GF 50 units are designed for an average stress of 90 MPa at 120°C, the component would appear to be unfeasible with standard development methods. For this reason, the project was continued using integrative simulation, which is explained in more detail below [2].

In the component design method commonly used in the industry today, a stress analysis is performed separately to the process simulation. Finite element analyses are used to evaluate the mechanical load capacity. Key material values are used which are only average values in many respects. Thus, these results only allow the mechanical load capacity to be evaluated to a restricted degree. Subsequent process simulation, in which the tool filling behavior and the fiber orientation are determined, serves merely to optimally design the tool.

In order to make better use of the performance of reinforced fiber polyamides, other influencing factors must be taken into consideration. This is where integrative simulation (figure 5) comes into play.

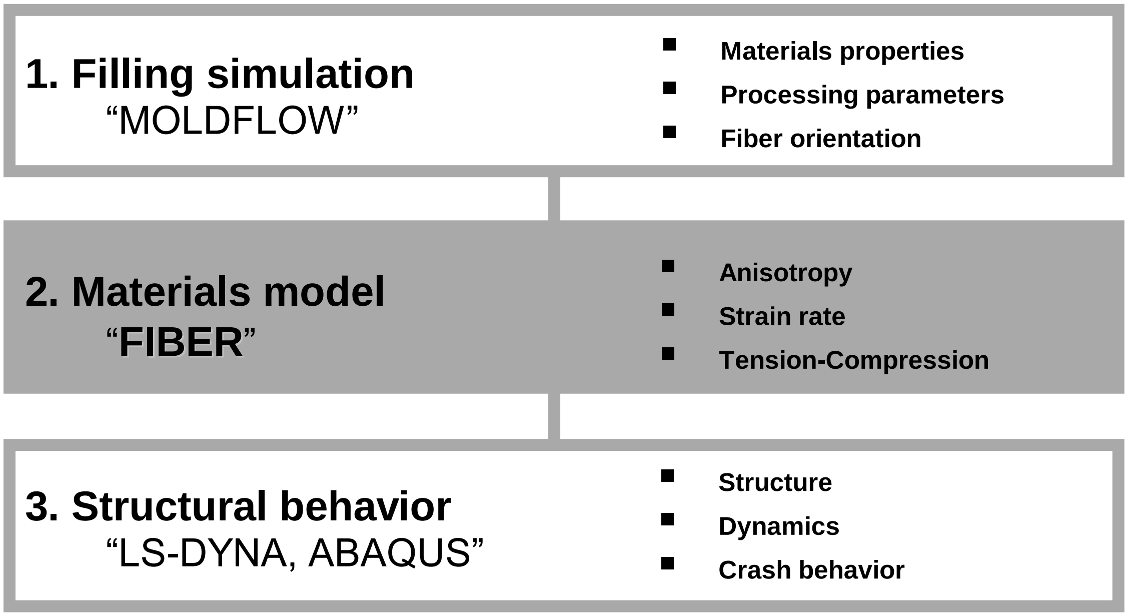
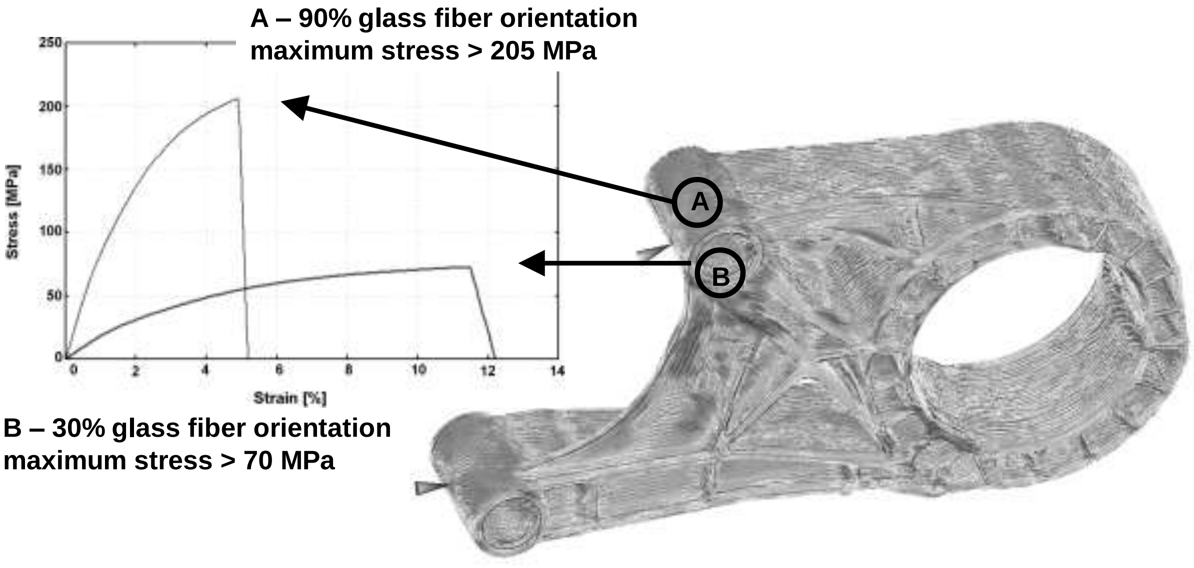


*Fig. 5: Integrative simulation - flow chart*

It links the results of the filling simulation with a subsequent finite element analysis taking fiber orientation into account. The stresses which can be resisted differ significantly depending on the direction of the fibers. The same applies for the type of load. Thus, the strength in the thrust direction is generally greater than that in traction direction. The influence of temperature and load speed is also taken into account. The local orientation is incorporated for every element based on the filling analysis. Figure 5 shows what this means for the specific component.


**A – 90% glass fiber orientation maximum stress > 205 MPa**

**B – 30% glass fiber orientation maximum stress > 70 MPa**

*Fig. 6: Dependence of properties on glass-fiber orientation*

Two adjacent points of an assembly lug are used to show how the fiber orientation can vary and what that means for the component strength at 120°C. In area A, the fibers are 90% oriented, which results in a maximum permitted stress of approx. 205 MPa. In area B, the orientation is just 30%, which corresponds to a permitted stress of approx. 70 MPa. This behavior is specifically taken into account in component and tool design using integrative simulation. In contrast, conventional simulation methods calculate the entire component with an average material characteristic value. The consequences for the component differ according to the local fiber orientation in the real component. In the best case, the material characteristic value is below the real performance of the material at this point. However, then only an additional, unintentional safety factor is introduced which leads to increased use of materials. The opposite case is significantly more dramatic: The component fails where the simulation did not predict failure.

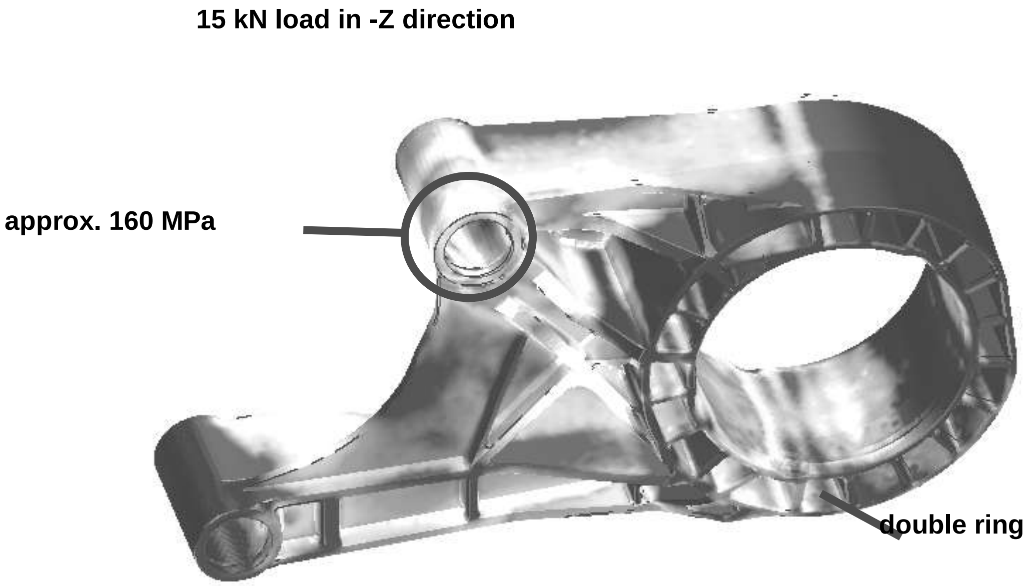


*Fig. 7: Result of integrative simulation*

The result of component optimization via integrative simulation is shown in Figure 7. The major design elements are the double ring on the supporting body and the rib structures along the main load paths to the thread points which are reinforced with metal inserts. The point subject to most stress, at 160 MPa, is the area of the highlighted assembly lug. Using integrative simulation, we were able to show that this corresponds to the performance of the material in this area. The plastic component is designed such that the deformation of the entire system corresponds to the aluminum version under operating loads. Deformation is only slightly greater for extreme loads. However, this does not affect the ride comfort. Suitable design and sufficient clearance ensures that no components approach or touch one another in an unintended manner.

The requirements for precise design of components subject to great loads are specially developed materials, like Ultramid® A3WG10CR in this case, in addition to the availability of corresponding material models. These materials are characterized in great detail in terms of the dependence of their properties on fiber orientation and expansion rate. Also, the performance level of the Ultramid® CR materials is greater and the degree of fluctuation is reduced compared with standard materials. This is made possible by a strict selection of materials used and close control in the material manufacturing process. This guarantees highly consistent quality of the components and excellent conformity of calculations with trials.

## Component Tests

When developing plastic-rubber compound components, the individual development steps intermesh directly with regard to component testing. Each individual development step is tested using a suitable method. Accordingly, new components which are developed are not released for vehicle testing until they have met all operational stability requirements in computer simulations and test bench trials in particular.

The component was evaluated in terms of functionality and load capacity with a range of laboratory test methods. The main functional tests include the following:

- Tests for creeping and settling properties
- Determination of static curves
- Dynamic characterization in a broad frequency band

In order to guarantee operational stability, comprehensive service life tests were performed, some of which were highly complex:

- Fracture tests
- Single-stage constant vibration tests
- Abuse tests
- Multi-axle operating load cycle tests

Fracture load tests serve to determine the maximal load capacity of the components and evaluate the computer component design quantitatively. It was shown that integrative simulation allows highly reliable component dimensioning (Figure 8).

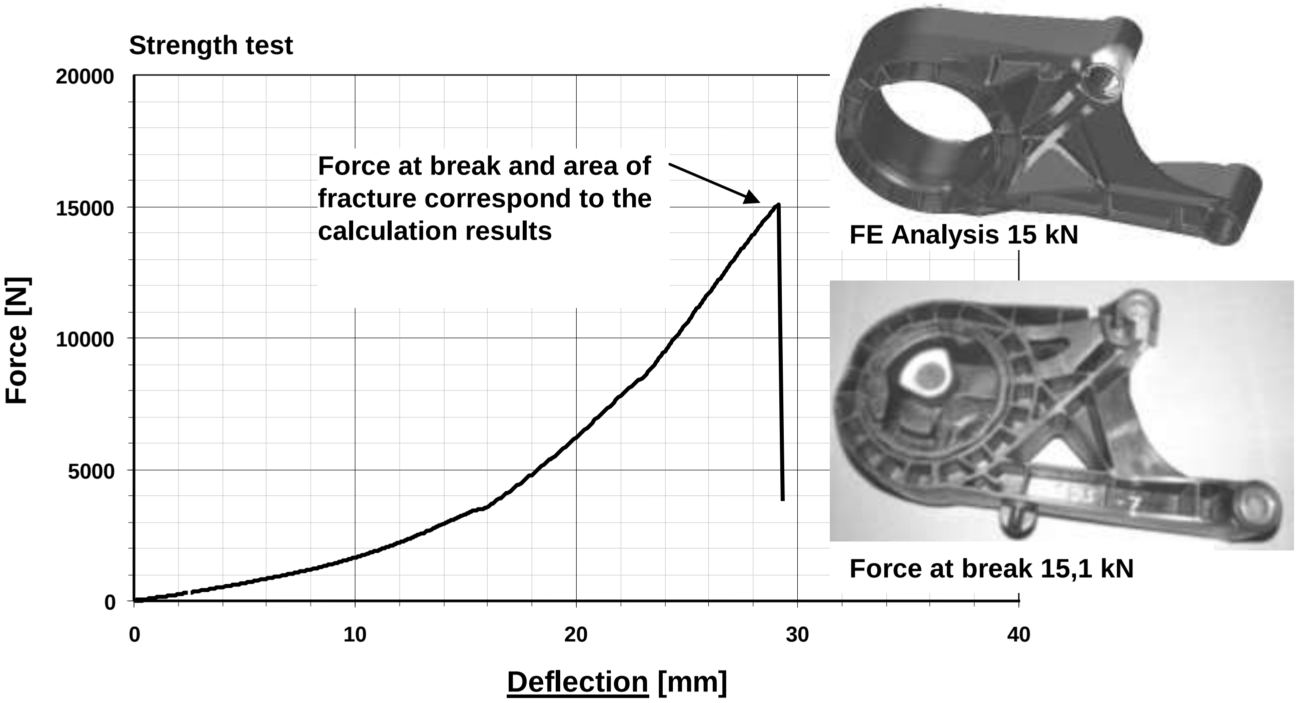


*Fig. 8: Simulation - test comparison*

In order to experimentally simulate abuse loads which seldom occur, load collectives were initially determined in vehicle trials. All tested components demonstrated extreme load bearing capacity in the selected low-cycle fatigue trial (several hundred load cycles not far below the nominal fracture strength). It was significant that no pre-damage occurred in spite of these considerable loads. A final evaluation of the component load capacity was made under realistic conditions based on 3D operating load test cycle trials (Figure 9). The load collectives were measured on representative target vehicles which were equipped with suitable measuring holders.

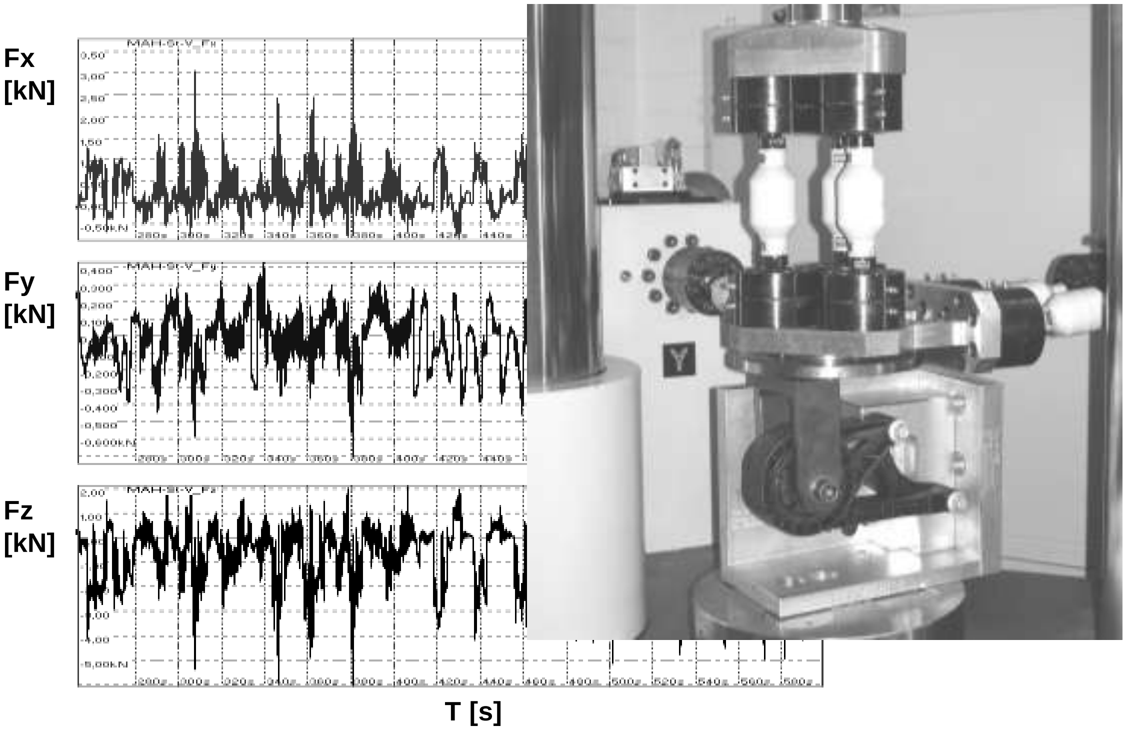


*Fig. 9: Component testing on a 3D test bench*

For specific evaluation of the specific plastic characteristics, the trial plans had to be coordinated accordingly. Of particular note in this regard were the extreme directional dependence of properties (effect of fiber orientation), effects of load direction (traction or thrust), effects of climatic conditions (spray dry or defined humidity) and temperatures from -30°C to 120°C.

A system and component evaluation in vehicle trials with regard to comfort and service life requirements is essential and determines the success or failure of a development. Therefore, the final system validation was performed via vehicle testing. In order to determine the effects on the comfort properties, subjective evaluations were made in accordance with specified plans. In particular, comparison runs of vehicles with aluminum components and plastic components were run. Parallel to subjective evaluation, the transfer properties of the aluminum and plastic components were measured using acceleration sensors and interior microphones. Note that the criteria for the natural component frequency applicable to the metallic components cannot be applied to plastic components. Due to the far higher material damping, plastic components are low in noise even at critical stimulation frequencies. Material damping D of aluminum is approx. 0.02%, while the material damping D of plastic is approx. 2%. The material damping results of the aluminum and plastic supports shown in Figure 10 support this statement.

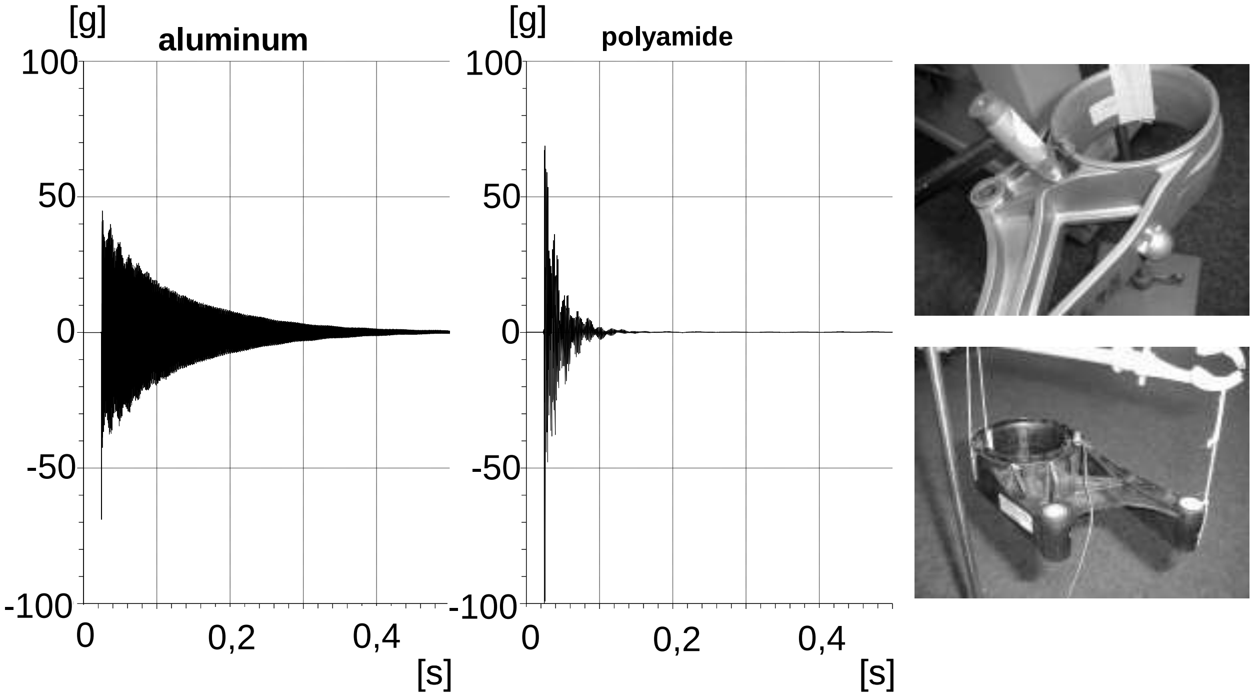


*Fig. 10: Material damping behavior aluminum vs. polyamide*

After the comprehensive laboratory trials, the service life requirements were tested in vehicle trials which involved endurance programs specially designed for global stresses. In this, the vehicles run through an entire vehicle service life taking the global sales network into account, whereby the loading due to poor streets or abuse is examined.

## Summary and Outlook

Based on integrative simulation, high-performance materials, qualified component manufacturing and systematic testing, high load capacity components can be created for engine/transmission mounting. For example, Figure 11 shows a torque support made of plastic compared with the original aluminum component. The first components were approved in 2006. The weight saving compared to the previous aluminum component is approx. 35%.

In view of the increasing scarcity of resources, rising energy prices and the spread of pollution, light construction of automotive parts will grow in importance. Plastic mounts can make a significant contribution to this. Against this background, ContiTech Vibration Control has a range of other plastic components which are almost ready for series production. he weight reduction will be approx. 50% compared with similar aluminum components. This is made possible by developments in component design and using additional potential in process engineering.

*Fig. 11: Torque reaction mounts in plastic and aluminum*

## References

1. Molitor, W.; Berlinger, M.; Karkosch, H.; Eberhard, G.: Innovative Motorlagersysteme und –komponenten; VDI Reifen Fahrbahn Fahrwerk. Hannover: VDI Verlag 2003
2. Frik, S.; Erzgräber, M.; Wüst, A.; Glaser, S.: Entwicklung eines thermoplastischen Lower Bumper Stiffeners für den Fußgängerschutz; Kunststoffe im Automobilbau. Mannheim: VDI Verlag 2007