Finite Element Method (FEM)

Technical Principles

Examples

Description of the Operation of Simrit Seals using the Finite Element Method (FEM)

Numerical simulation procedures are now an essential component of modern development processes. These methods can be used at a very early stage of the development process of components to test their feasibility. The development process is significantly faster and more economical and the quality of the product can be significantly improved. Such simulations cannot of course entirely replaced component testing, because it is impossible to simulate reality exactly. However, the number of tests required can be significantly reduced.

A common simulation procedure is the "Finite Element Method (FEM)". A component with a complex shape is divided into a large finite number of small elements, referred to as finite elements. Depending in the problem and the geometry the elements are triangles and squares or pyramids or cubes. The division into small discrete elements that describe the geometry of the component under consideration make it possible to describe the behaviour of the component numerically and to predict it on the computer. This makes it very easy to test different geometries, to test different materials and to investigate the behaviour of the component under various loads without having to manufacture prototypes at every stage and run tests. The simulation procedure can also be used to "look inside" the component to gain a better understanding of its behaviour.

The prerequisite for rational use of the simulation methods is to imitate real life as closely as possible. This includes accurate knowledge of the loads to be applied to the component (temperatures, pressures, forces, displacements, ...), and particularly the correct description of the behaviour of the material. Behaviour of the material is understood as the interaction between application of an external load (e.g. a force) and the reaction of the material (e.g. a change in length). This interaction is generally determined by tensile trials on a simple test body. The result of such as experiment is described in (→ Fig. 1) with an elastomer as an example, where the mechanical tension is shown as the force and the elongation as the change in length. In the simplest case there is a linear relationship between the force applied and the change in length. This can be observed e.g. with metal materials if the load does not exceed a specific value (yield point). In (→ Fig. 1) it can be see that with elastomers there is not a linear relationship between tension and elongation but this relationship is non-linear. Another difference from metal materials is that elastomers are frequently very strongly deformed, elongations of over 100% are not uncommon.

To be able to represent the material behaviour in a computer simulation, the relationship between tension and elongation must be described in the form of "material models". This is very simple for the linear tension-elongation relationship. However, it is much more complicated for elastomers: There is a whole variety of material models for non-linear material behaviour in the literature. These models are all part of the class of hyperelastic material models.

The material parameters are included in the material models. The parameters can be used to modify the material model for different materials. There is only one material parameter for linear material behaviour: the pitch of straight lines, known as the modulus of elasticity of the material. The number of material parameters under the hyperelastic laws varies depending on the model between one and infinitely many parameters.

A selection of these material models is shown in (→ Fig. 2) in addition to the material testing. It can clearly be seen that these models predict the actual material behaviour with varying degrees of accuracy. In particular not all models are capable of describing the upturn in the tension-elongation relationship. The material models vary even more when predicting a compressive stress (→ Fig. 3), where the same material parameters were used to describe the tensile trials. The difference between two basically different classes of hyperelastic material models can be clearly seen here: on one hand there is the class of phenomenological material models. They describe the tension-elongation curve with various mathematical functions. Some examples are the Mooney-Rivlin model and the Ogden model. These models describe loads to which the material parameters were adapted (e.g. a tensile load) very well. However, problems are frequently encountered if other types of loads (e.g. a compressive load) must be extrapolated. In extreme cases tensile loads may occur with compressive deformation, which physically indicates an unnatural behaviour. As a result many different experiments are required when using this class of material models, which makes determining the material parameters very expensive and complex.

The second class of material models are referred to as physically motivated models. They describe the physics of the material, i.e. they have a background based on physical properties. As a result these models basically predict physically reasonable results, and it is sufficient to determine the material parameters on the basis of only one test (e.g. a tensile trial). A disadvantage of this class of material models is that development is very complex, which is why there are very few physical material models for elastomers. The best-known model is the Neo-Hooke model, which however is not capable of describing the increase in stiffness in the tensile trial. This is why we have developed our own, physically based material model, which correctly describes the complete mechanical behaviour of elastomers. Use of the Freudenberg material model has the following advantages: it can correctly predict the strongly non-linear material behaviour of elastomers, including the upturn with very large deformations, for loads of up to several hundred percent elongation.

Because only simple tensile trials are required to determine material parameters, the parameters of new materials can be determined very quickly and the influence of the material on the behaviour of the component can be simulated. In addition, the correct material behaviour is reproduced for any desired stress types by the Freudenberg material model. Even the behaviour of complex components can be simulated and the behaviour of such components can be simulated under any desired loads. All simulations of elastomer components at Freudenberg are conducted exclusively with this material model for this reason.

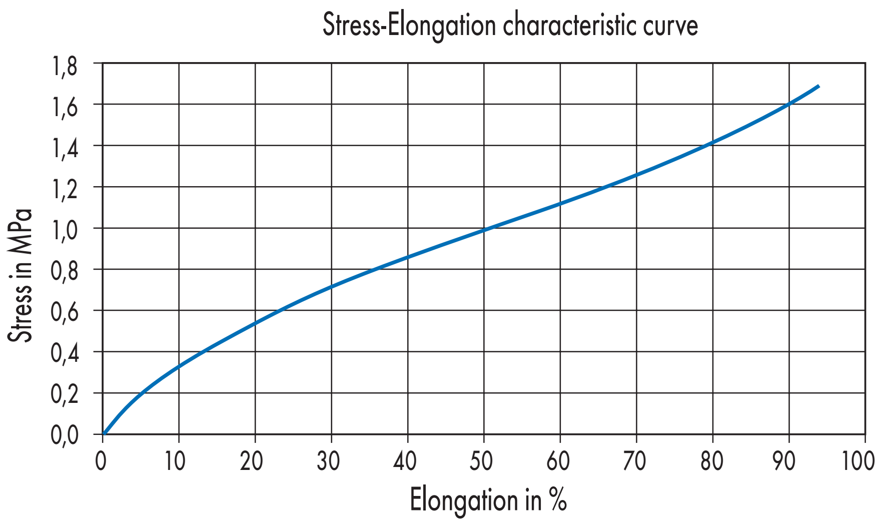

Fig. 1 Tensile trials on an elastomer

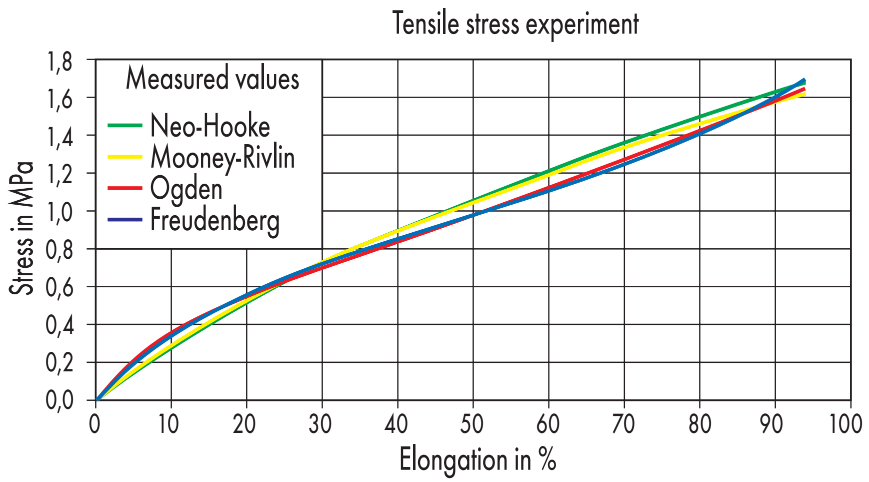

Fig. 2 Material models adapted to a single-axis tensile trial

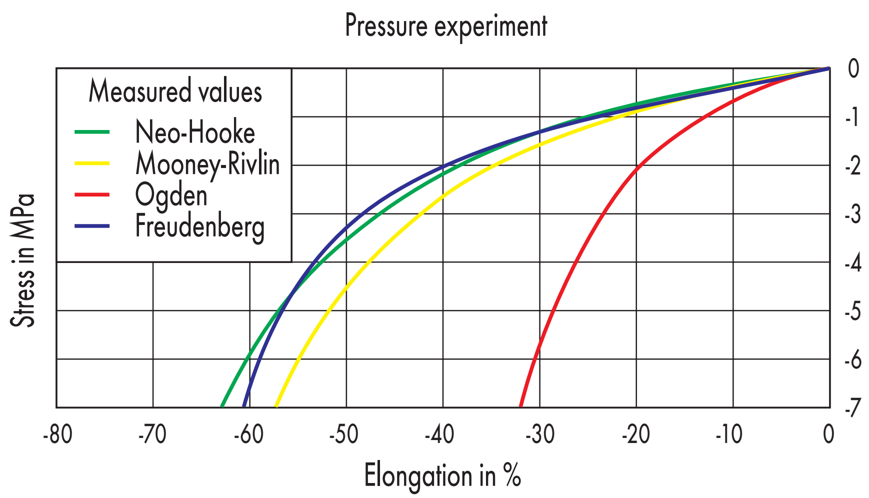

Fig. 3 Material models adapted to a single-axis compressive trial

Examples

Simmerring

The deformed geometry of a standard Simmerring in installation position under a 3 bar pressure load is shown in (→ Fig. 4). (→ Fig. 5) shows the optimised profile under the same operating conditions with reference to minimising the contact area of the sealing lip on the shaft.
The animation shows the deformation of a special profile that compensates for large tilts of the shaft. It is clear that in spite of tilting the sealing lip on the unloaded side is still in contact with the shaft. (→ animation on DVD).

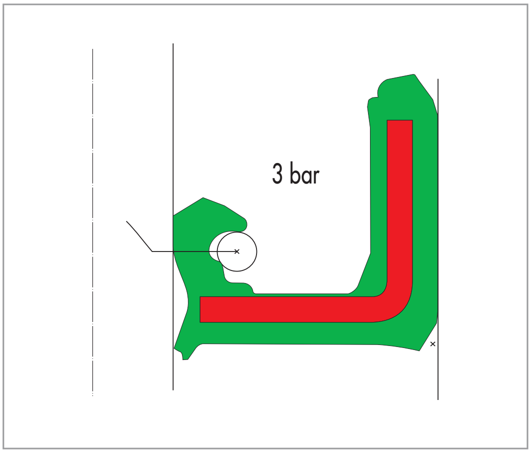

Fig. 4 Simmerring with normal profile

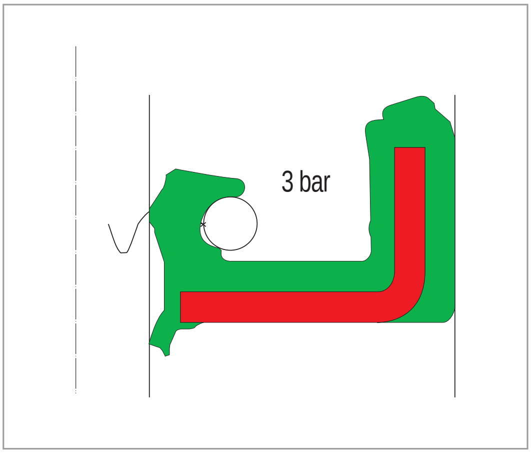

Fig. 5 Simmerring with optimised profile

Hydraulic Seal

The undeformed geometry of the Merkel U-Ring LF 300 is shown in (→ Fig. 6). The U-ring is installed in a groove, and is under pressure in operation. The deformation at a pressure of 100 bar can be seen in (→ Fig. 7).
The animation shows the deformation of the U-ring as the pressure increases from 0 bar to 100 bar. The load involved on the material is shown in addition to the deformation. (→ animation on DVD).

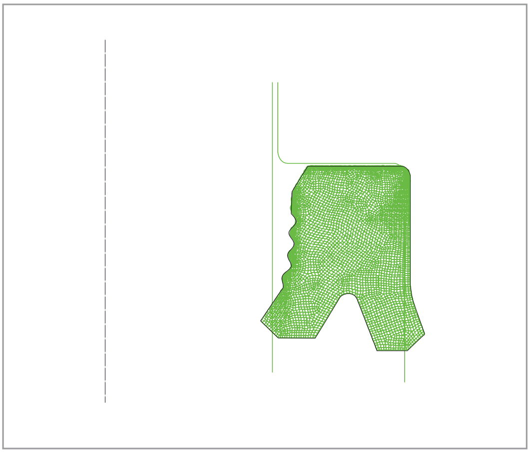

Fig. 6 Merkel U-Ring LF 300 undeformed

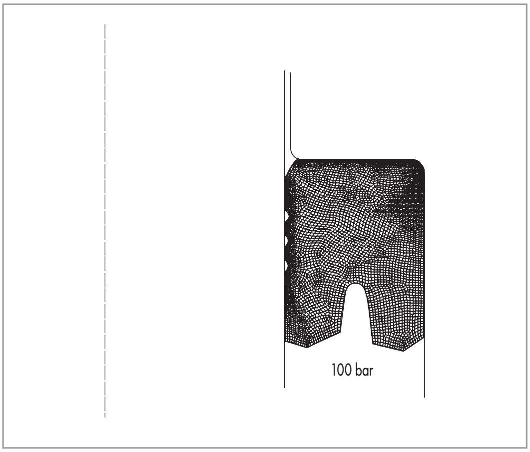

Fig. 7 Merkel U-ring deformed 100 bar

Pneumatic Seal

The undeformed geometry of the pneumatic seal of the Merkel U-Ring NAPN 125 is shown in (→ Fig. 8). The seal is tied into a groove on the piston and installed in the cylinder bore with the piston rod. The deformation of the seal (→ Fig. 9) in operation is obvious at a load pressure of 10 bar.
The animation shows the deformation of the installed pneumatic seal as the pressure increases from 0 to 10 bar. The load involved on the material is also shown in addition to the deformation (→ animation on DVD).

ISC O-Ring

The geometry of a correctly installed standard ISC O-Ring 40-4 at 100 bar is shown in (→ Fig. 10).
(→ Fig. 11) shows the same ring with an excessive gap width in connection with an excessively soft material, resulting in a gap extrusion.
Both the deformation and the load involved on the material can be seen in the two pictures.
The deformation of the ISC O-Ring under pressure can be seen in the animation. The load involved on the material is also shown in addition to the deformation (→ animation on DVD).

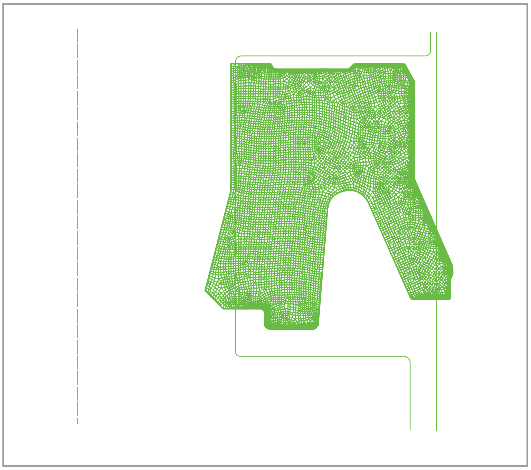

Fig. 8 Merkel U-Ring NAPN 125 undeformed

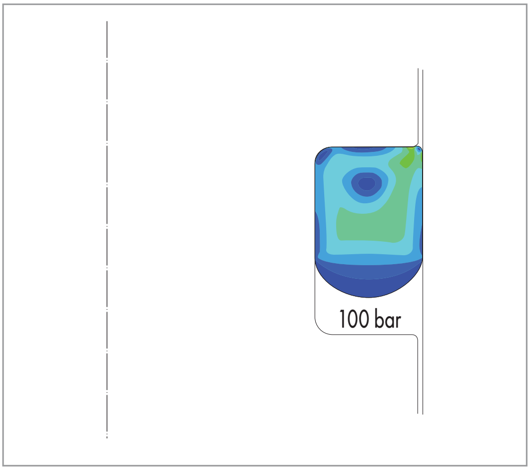

Fig. 10 ISC O-Ring 40-4 100 bar

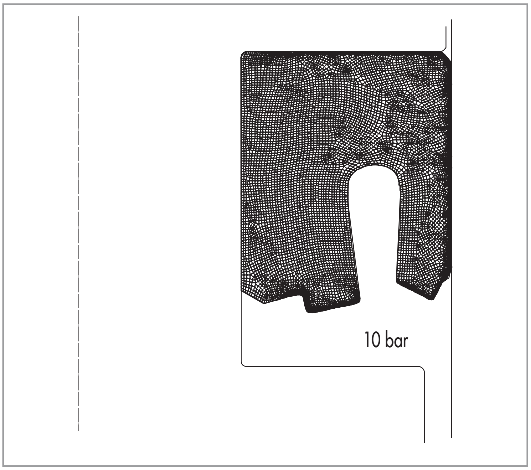

Fig. 9 Merkel U-Ring NAPN 125 deformed 10 bar

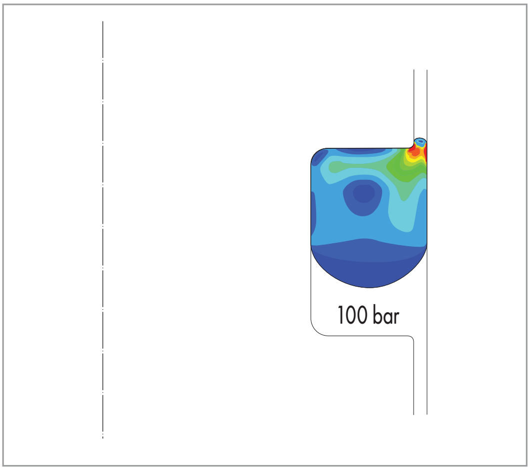

Fig. 11 ISC O-Ring 40-4 100 bar gap extrusion

Elastomer CompositeComponent

The undeformed geometry of a Plug & Seal plug connection is shown in (→ Fig. 12). The deformation of the installed component at a pressure of 10 bar can be seen in (→ Fig. 13).
The animation shows the deformation of the deformed component at a pressure increase of up to 10 bar. The load involved on the material is also shown in addition to the deformation (→ animation on DVD).

Bellows

The installation of a fixed-joint bellows is shown in (→ Fig. 14). The axially sprung and tilted structure of the bellows can be seen in (→ Fig. 15).
The animation shows the axial deflection with the associated tilting of the bellows (→ animation on DVD).

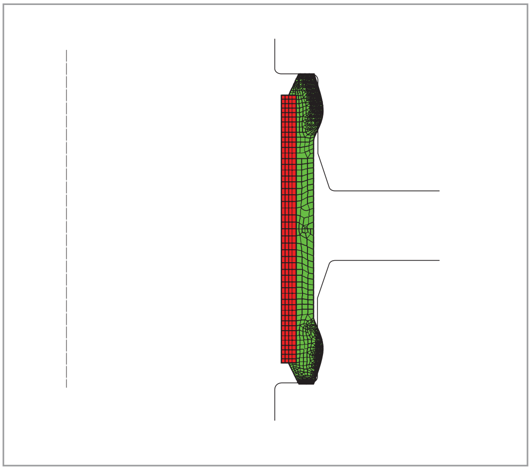

Fig. 12 Plug & Seal plug connection undeformed

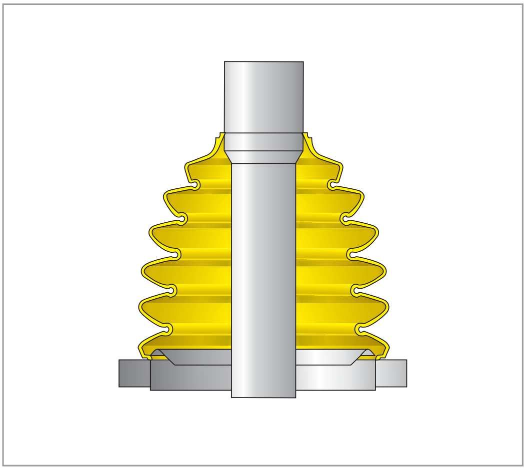

Fig. 14 Bellows installed

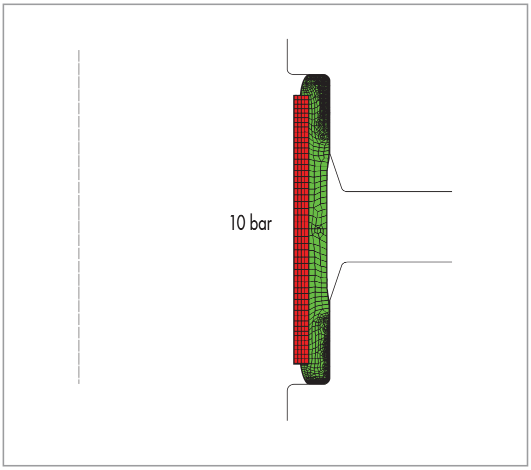

Fig. 13 Plug & Seal plug connection deformed 10 bar

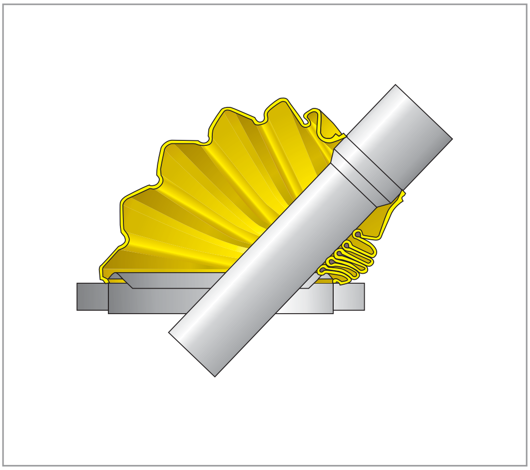

Fig. 15 Bellows deflected and tilted

Diaphragm

The undeformed geometry of a diaphragm in unmounted position is shown in (→ Fig. 16). The axially deflected structure with 0,5 bar pressure on the bottom is shown in (→ Fig. 17). The red nodes represent the area to which a piston is attached.

The animation shows the progress of the piston in the axial direction (first up, then down). Then a pressure of 0,5 bar is applied to the bottom of the diaphragm. The load involved on the material is also shown in addition to the deformation (→ animation on DVD).

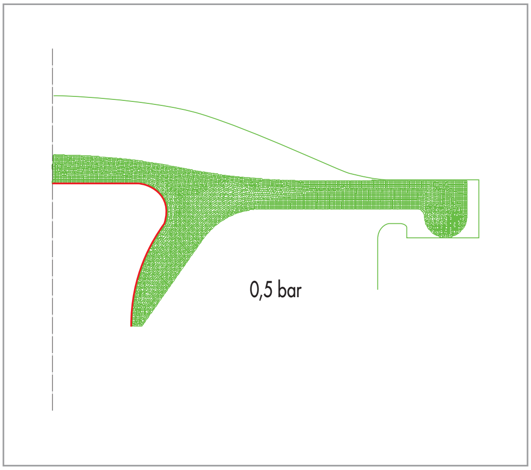

Fig. 16 Diaphragm undeformed

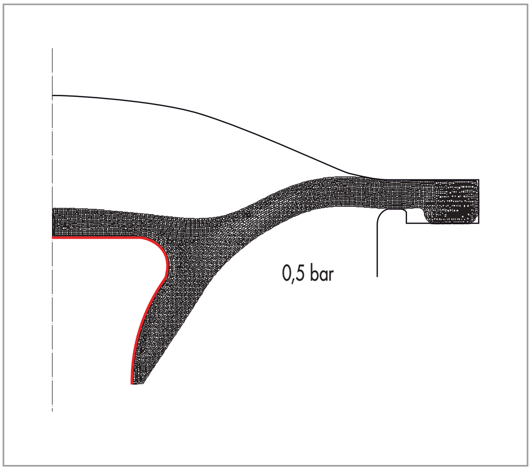

Fig. 17 Diaphragm axially deformed