

AUTOMOTIVE STABILIZER BARS – STRENGTH CALCULATIONS OF STABILIZER BARS USING FINITE ELEMENT METHOD – DIRECTIONS AND GENERAL INSTRUCTIONS FOR CALCULATION

ADAM MARKUS WITTEK¹, DAMIAN GAŚKA², TOMASZ MATYJA³

ThyssenKrupp Federn & Stabilisatoren GmbH

Silesian University of Technology

Summary

The function of stabilizer bars in motor vehicles is to reduce the body roll during cornering. The body roll is influenced by the occurring wheel load shift and the change of camber angle. Decisive is the steering performance which may be purposefully adjusted towards understeer or oversteer when designing the stabilization. The article contains the further outline of the calculation methods for stabilizer bars. Modern technological and structural solutions in contemporary cars are reflected also in the construction and manufacture of stabilizer bars. A proper construction and the selection of parameters influence the strength properties, the weight, durability and reliability as well as the selection of an appropriate production method. An improper preparation of Finite Element Method calculation models consequently leads to wrong results. It is particularly difficult to interpret the results and to find an error if we do not have a comparative calculation base (such as results of fatigue tests, analytical strength calculations). The article contains practical directions and general instructions for calculation necessary for a correct preparation of calculation models, for a proper performance of calculations and a proper interpretation of results using Finite Element Method.

Keywords: stabilizer bar, FEA calculations

¹ ThyssenKrupp Federn & Stabilisatoren GmbH, Oeger St. 85, 58089 Hagen, Germany,
e-mail: adam.wittek@t-online.de, tel. +49 233 14 07 105

² Silesian University of Technology, Faculty of Transport, ul. Krasińskiego 8, 40-019 Katowice, Poland,
e-mail: damian.gaska@polsl.pl, tel. +48 32 603 43 93

³ Silesian University of Technology, Faculty of Transport, ul. Krasińskiego 8, 40-019 Katowice, Poland,
e-mail: tomasz.matyja@polsl.pl, tel. +48 32-603 41 83

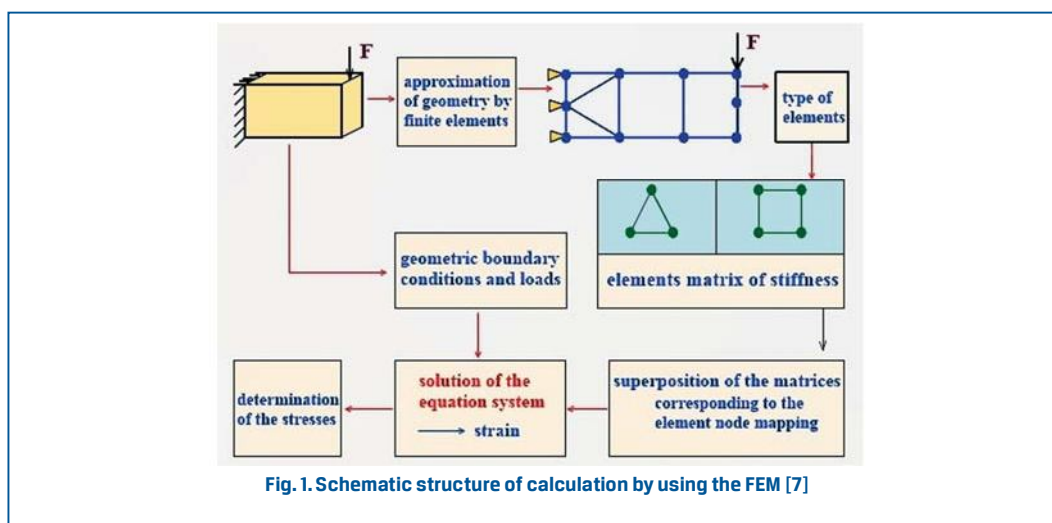
1. Introduction

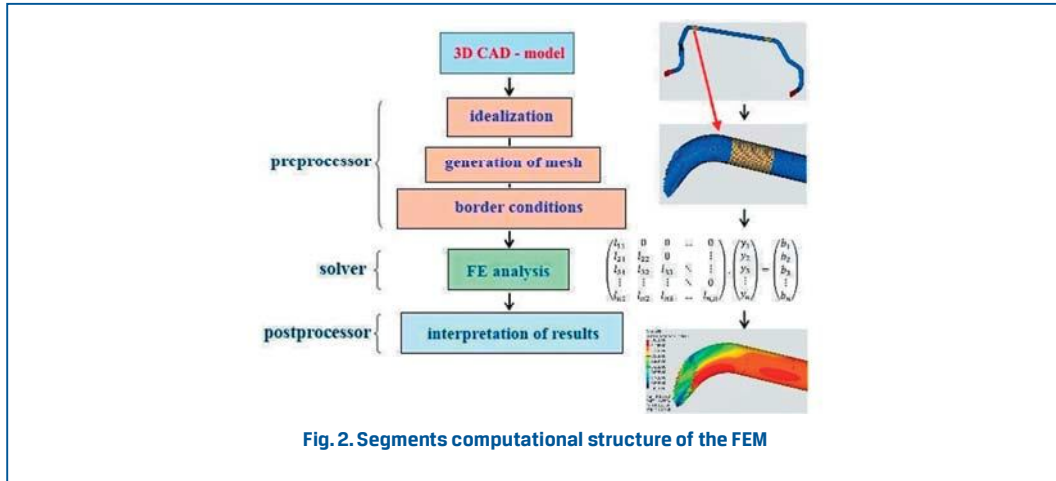
FEM is a method of approximation (by discretisation) of partial differential equations. Three basic elements of methodology for the solution of a given problem by means of FEM include [2, 7, 9]:

- a weak (variational, integral) formulation of a differential problem,
- a decomposition of the calculation domain into elements,
- taking simple basic functions in the domain as a basis of approximation.

While in the classical direct method of variational calculus, such as the Ritz method or the Galerkin method, the trial functions cover the entire domain (e.g. a structure), in case of the FEM such trial functions are selected which are not equal to zero respectively only in a subdomain, a so-called finite element. Fig. 1 clearly shows the individual steps when using the FEM [7, 9, 13].

After modelling, at first the geometry of the examined structure is decomposed into individual subdomains (finite elements). The cantilever beam shown in Fig. 1 is examined here for example as a plane elastic problem (modelling discussion). The middle plane of the cantilever beam is decomposed into finite elements. This leads to the so-called FE network. Various types of finite elements are available for the integration of the problem. In the present case, triangular and quadrangular elements were used for this purpose. The stiffness properties of the overall structure are described by the sum of the stiffness matrices of the individual finite elements. They are superimposed in addition to the overall stiffness matrix. Then, in conjunction with the geometric boundary conditions and the loads, a system of linear equations arises from the overall stiffness matrix. The unknown items to be calculated in this system of equations are the deformations of the so-called nodal points. Subsequently, stresses in each individual finite element can be calculated from the deformations using the trial functions [7, 9, 13].





The problem solution by means of FEM allows obtaining an approximate solution. The error of a FEM solution can be pre-estimated based on the form of the problem to be solved, the geometric characteristics of the elements and the properties of the approximation space. Unfortunately, the estimation is expressed by means of an unknown exact solution (nevertheless it is significant to determine the convergence of the method, its optimality, and to compare it with other approximations of the same problem) [2, 10, 11].

Computer programs in which the finite element method is used consist of 3 parts (Fig. 2) [2, 7, 9]:

1. **a preprocessor** in which the problem to be solved is created,
2. **a processor**, i.e. the calculating part,
3. **a postprocessor** for the graphical presentation of obtained results.

For the users of such programs, the most labour- and time-consuming stage of the problem solution is the decomposition into finite elements in the preprocessor. An improper decomposition into finite elements causes that wrong results are obtained.

A finite element is a simple (plane or three-dimensional) geometric figure for which discriminated points called nodes and certain interpolation (nodal, shape) functions used for the description of the distribution of the analysed item inside it and on its sides were determined. The nodes are situated at the vertices of a finite element, but they can also be located on its sides and inside it. If the nodes are situated at the vertices only, the finite element is called a linear element (because the interpolation functions are linear then). Otherwise they are higher order elements [2, 7].

The order of an element is always equal to the order of interpolation functions (shape functions). The number of shape functions in an individual finite element is equal to the number of its nodes. The shape functions are always structured in such a way that at the nodes to which they refer their values amount to one and at the other nodes they have the value of zero.

2. FEA calculations

2.1 Areas of the application of the FEM method

Advantages of using FEM:

- reduction of time of the design process,
- reduction of cost of the design process,
- reduction of the production costs,
- material savings,
- possibility of an early identification of weak points of the design,
- improvement in quality of the design process, thus of the manufactured element,
- optimisation of design,
- reduction of the number of experimental tests,
- flexible adaptation of the design for further implementations (so-called supervision of model variants).

Requirements and assumptions necessary to achieve the above - mentioned benefits [6, 9, 12]:

- Powerful software (**ABAQUS**, ADINA, ALGOR, **ANSYS**, ANTRAS, COSAR, **COSMOS**, ISAFEM, LUSAS, **MARC**, MECHANICA, NASTRAN, NISA, OPTISTRUCT, PERMAS, **RADIOSS**, ...),
- Powerful Hardware (PC, workstation, large - capacity computer),
- FEM - theory (knowledge of the basics),
- Engineering knowledge to critically assess the results,
- Modeling understanding (real construction ⇒ calculation model).

Table 1. Examples of FEM applications [12]

linear elastostatics	<ul style="list-style-type: none"> • Boolean reversible material behavior
nonlinear elastostatics	<ul style="list-style-type: none"> • nonlinear material behavior (elastoplasticity) • geometric nonlinear problems (instability problems, large displacements at low strains)
linear Elastodynamics	<ul style="list-style-type: none"> • natural oscillations • free oscillations • forced oscillations • randomly excited vibration
nonlinear Elastodynamics	<ul style="list-style-type: none"> • reply / time - behavior • stability and resonances
static and dynamic aeroelasticity	<ul style="list-style-type: none"> • structural behavior under inflow
linear and nonlinear thermoelasticity	<ul style="list-style-type: none"> • mechanical stress under high temperatures
heat transfer problems	<ul style="list-style-type: none"> • steady and unsteady heat conduction
liquid flows	<ul style="list-style-type: none"> • seepage flow, velocity pressure and temperature fields viscous flows
electrical engineering	<ul style="list-style-type: none"> • electromagnetic fields
acoustics	<ul style="list-style-type: none"> • sound pressure distribution

Principal procedure for the calculation of a component (e.g. a stabilizer bar) using the FEM [6, 9, 12]:

- replacement of the real design by a calculation model (preprocessor, e.g. HyperMesh),
- calculation of the deformations, stresses, internal forces, bearing reactions etc. under predefined loads and bearing conditions (FE analysis program, main processor, e.g. OptiStruct),
- evaluation of the results (postprocessor, e.g. HyperMesh),
- targeted optimisation of the design based on the calculated deformations, stresses, etc.,
- possibly a recalculation with a modified design or a modified model or manufacture of the component.

2.2 3D CAD – models

The component or the subassembly to be examined is usually in the form of a 3D CAD model, thus it accurately describes the geometric relationships.



3D model - SolidWorks, Catia, Creo, HiCAD, Inventor, SolidEdge / alternative: step, iges

Fig. 3. Sample models of stabilizers made with using of SolidWorks

However, most stabilizer bar models developed using such CAD programs such as: SolidWorks, CATIA are not suitable for automatic mesh generation. Such models require a further processing, e.g. using the HyperMesh program. The advantages thereof, in addition to considerably shorter computation times, include a smaller model size and a better handling as well as a reduced data base. Based on this data, such simplifications as the removal or blanketing of insignificant details such as holes or chamfers are made where necessary. Such steps are referred to as idealisation. Many components show symmetry and make it possible to reduce the calculation model. Due to the use of symmetry and antisymmetry boundary conditions the same statements as for a full model arise as a result.

2.3 Preprocessing – Creation of the mesh, the loads and boundary conditions

In the next step, starting from the geometry, the FE network is generated (Figs. 4, 5d). It consists of finite elements and nodes. Various types of elements for integration such as beams, shells, solids, springs, gaps, rigid bodies etc. are at the user's disposal [3, 9, 16].

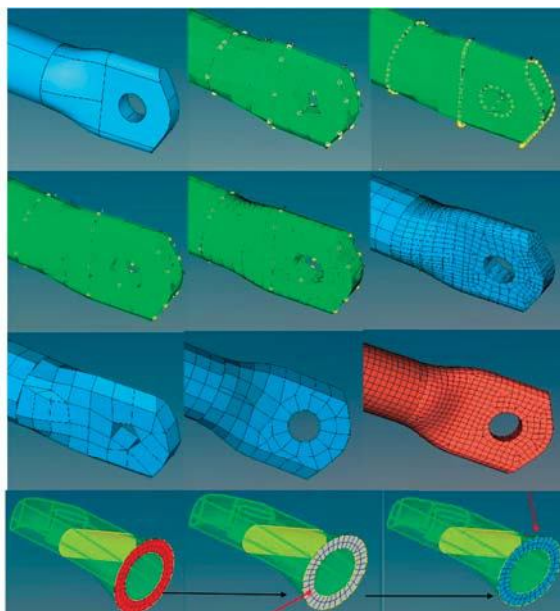


Fig. 4. Examples of mesh generation and optimization with the use of HyperMesh [10]

This step is referred to as discretisation and depending on the complexity of a structure it can take from a few seconds to several hours.

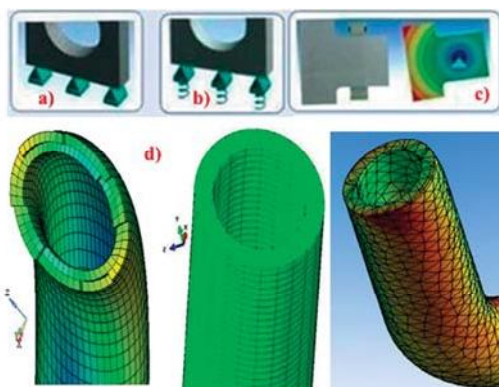


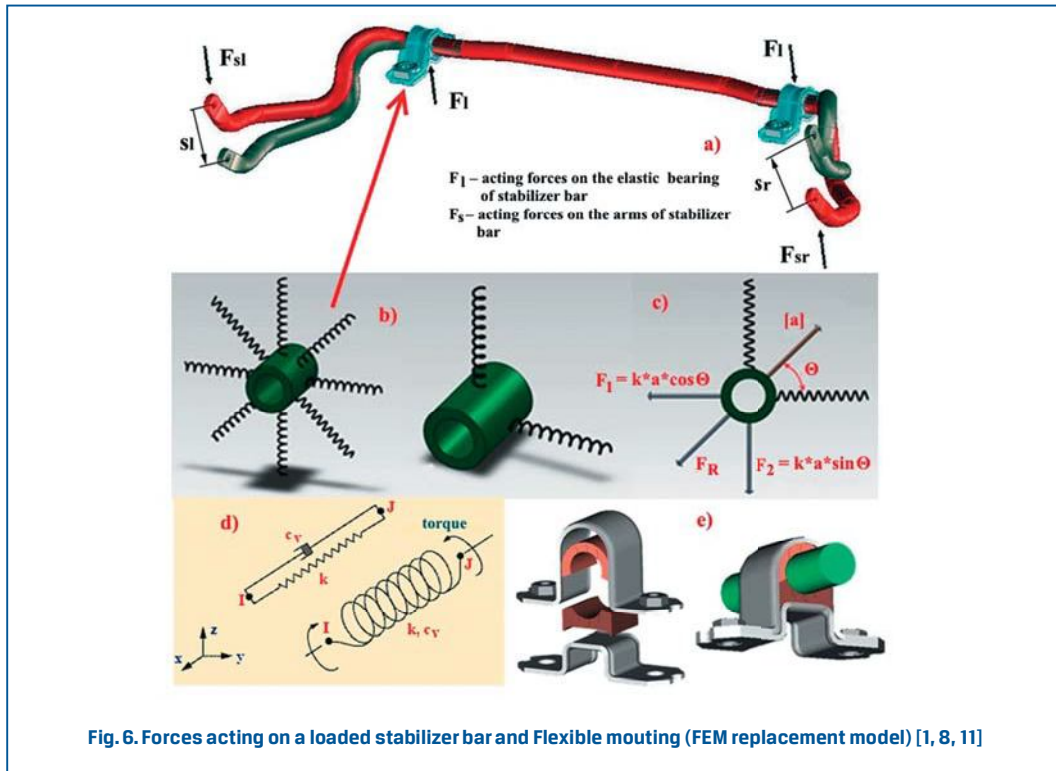
Fig. 5. Scope of the bearings and mesh for FEM friction free bearings a) [8], elastic mounting b) [8], external displacement c) [8], mesh varieties: HyperMesh, Abaqus, Ansys d)

The aim is to generate a FE network where the elements have as little deviations from the ideal form as possible and at the relevant points have a sufficiently small edge length. In addition, the selection of the correct type of element is of major importance.

Apart from the FE network, loads and boundary conditions are also required for most calculations. Such factors as forces, moments, displacements, pressures, temperatures, speeds, accelerations, etc. are used as well. The boundary conditions describe the bearing of the model and possible symmetries. In case of (both solid and tubular) automotive stabilizer bars the best calculation results are achieved by defining the following boundary conditions (Fig. 6):

- the selection of mounting (Fig. 6e) and of the method of stabilizer bar bearing in the vehicle (Figs. 5a and b, 6a, b, c and d),
- the definition of forces or displacements of the ends of a loaded stabilizer bar (Figs. 5c, 6a).

This data has been made available by the manufacturers of cars and flexible bearings used in motor vehicles.



2.4 FEA calculation

After finishing the FE model including the loads and boundary conditions, the actual calculation can be carried out. For this purpose the model is transferred to the solver that carries out various plausibility checks in the first instance and then checks the elements for the compliance with the predefined criteria. If e.g. material properties are missing or elements are too much distorted, then the calculation is terminated yet before the actual

analysis. If this step has been successful, then the conversion of the physical model into a mathematical model takes place. For this purpose, at first the stiffness matrix which represents the properties of the elements is generated. Since the matrices are positively defined and symmetrical the computing time can be considerably reduced. For this purpose, the matrix is reduced by a decomposition based on the Gaussian elimination to the extent that a triangular coefficient matrix is formed. This step is indeed highly computationally intensive; however, a significant time advantage arises as an end result. Together with the loads a system of equations arises which is to be solved by the solver and gives the searched items (displacements, stresses, etc. – Fig. 7) as a result.

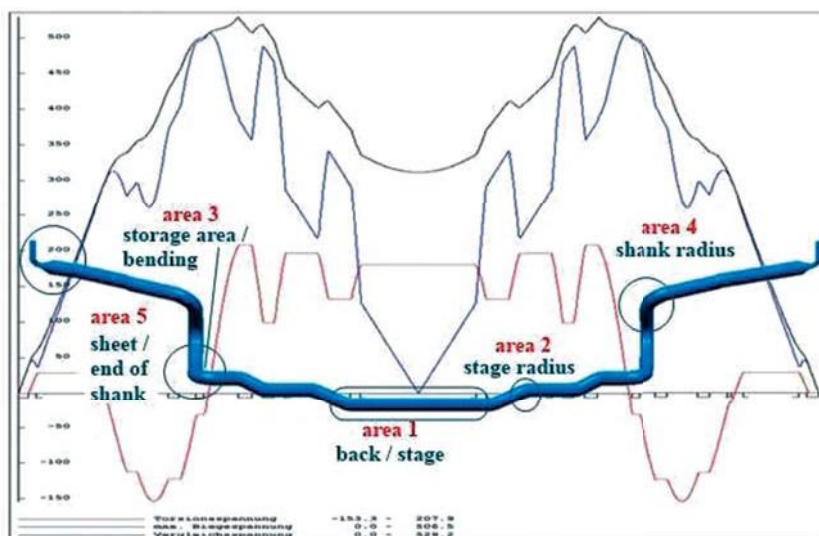


Fig. 7. Example size of stress (zone of calculation) in the loaded stabilizer bar
 Area 1 and 5 – small strength
 Area 2 and 4 – average strength
 Area 3 – area of high strength

An example illustrating the theoretical assumptions of the FEM in the computational practice of stabilizer bars can be a prismatic tension/compression member (Fig. 8).

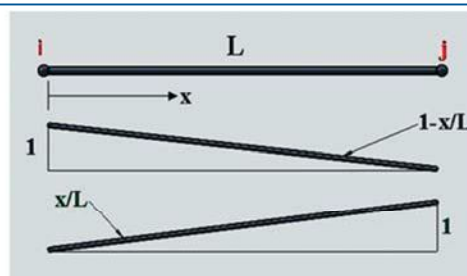


Fig. 8. Prismatic bar compressed and stretched [4]

It is common knowledge that inside a prismatic tension/compression member (i.e. such with a constant cross-section) a constant elongation independent of the local position coordinate x prevails:

$$u(x) = a_1 + a_2 x \quad \text{where } a_1 \text{ and } a_2 \text{ are constants} \quad (1)$$

Thus the longitudinal displacement u changes as a linear function of x [3, 4, 5].

$$\begin{aligned} u(x) &= \left(1 - \frac{x}{L}\right) u_i + \frac{x}{L} u_j \quad \text{respectively } u(x) = \begin{bmatrix} 1 - \frac{x}{L} & \frac{x}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \\ &= [N_i \quad N_j] \begin{bmatrix} u_i \\ u_j \end{bmatrix} \Rightarrow \underline{u} = \underline{N} \underline{a} \end{aligned} \quad (2)$$

where: \underline{N} matrix of interpolation functions or „shape functions“

$$\underline{\varepsilon} = \underline{u}_{,x} = \underline{N}_{,x} \underline{a} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \underline{B} \underline{a} \quad (3)$$

which means nothing else than:

$$\varepsilon = \frac{u_j - u_i}{L} \quad \text{and } \underline{\sigma} = \underline{D} \underline{\varepsilon} = \underline{D} \underline{N}_{,x} \underline{a} = \underline{D} \underline{B} \underline{a} \quad (4)$$

where: D – elasticity matrix, in this case consisting of a single item, namely the E module

Based on these elongation and stress expressions, the form change energy of the bar can be established:

$$U = \frac{1}{2} \int_V \sigma * \varepsilon \, dv = \frac{1}{2} A L \underline{a}^T \underline{B}^T \underline{D} \underline{B} \underline{a} = \frac{1}{2} \frac{E * A}{L} [u_i \ u_j] \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \frac{1}{2} \frac{E * A}{L} (u_i - u_j)^2 \quad (5)$$

According to the Castigliano's theorem applies:

$$\frac{\partial U}{\partial u_i} = F_i \quad \text{or} \quad \frac{\partial U}{\partial u_j} = F_j \Rightarrow \begin{bmatrix} F_i \\ F_j \end{bmatrix} = \frac{E * A}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} \quad (6)$$

We can identify here the structure $\underline{F} = \underline{K} \underline{a}$ with the stiffness matrix of the bar element:

$$\underline{K} = \frac{E * A}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad (7)$$

Which on closer examination proves to be the volume integral $K = \int_V B^T * D * B \, dv$. If the geometry and the material properties of the bar are known, then – based on the specific trial functions – the respective stiffness matrix can be determined by a pre-definable integral.

$$K = \int_V B^T * D * B \, dv \quad (8)$$

Such integrals are numerically implemented in the FE programs. Depending on the type of

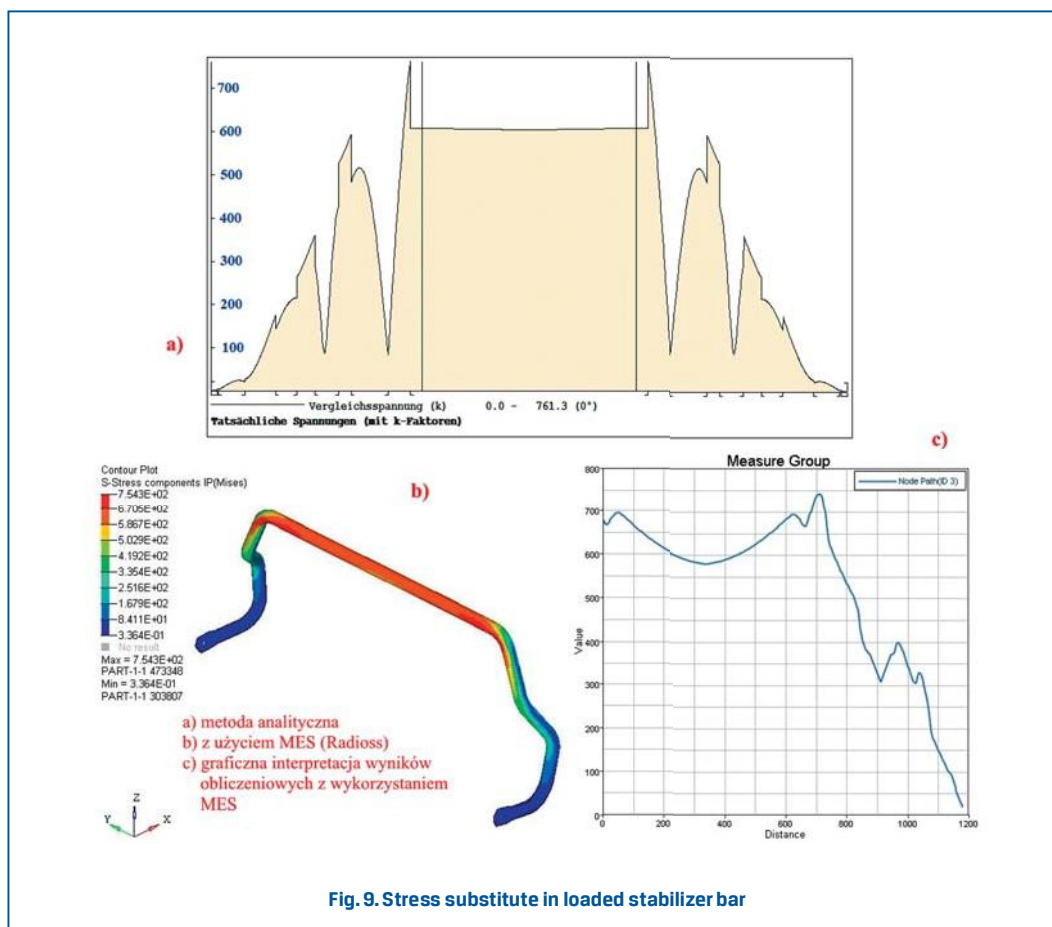
element, a different number of Gaussian integration points which are selected internally in the program, i.e. without the influence on the part of the user, close to the element nodes is brought for this purpose.

The target values derived, such as stresses and elongations, are evaluated at the same points and extrapolated to the respective nodes.

2.5 Postprocessing – Analysis of the calculation results

The last step of an FEM analysis is the evaluation of the calculation results. For this purpose the calculated displacements, forces, stresses, etc. are fed into the pre-processor and can be visualised there [3, 9, 16].

An analysis of the calculation results is a difficult and responsible task. Such analysis should include:



1. an evaluation of the calculation model – a definition of areas characterised by the highest, maximum stresses (Figs. 9c, 10a, b and c),
2. a determination of maximum stress values (Fig. 9b and c),
3. a comparison of the curve obtained from analytical calculations, characterising the distribution of stresses in a loaded stabilizer bar with a curve obtained from the FEM calculations (Fig. 9a and c),
4. in case of the occurrence of great differences between the results of analytical and FEM calculations the FEM calculation model, and in particular the boundary conditions, should be checked.

In case of a correctly conducted FEM analysis the deviations of results should not be greater than $\pm 5\%$. Correctly prepared models, a correctly conducted FEM calculation analysis lead to comparable results independent of the solver applied (Fig. 10a, b and c).

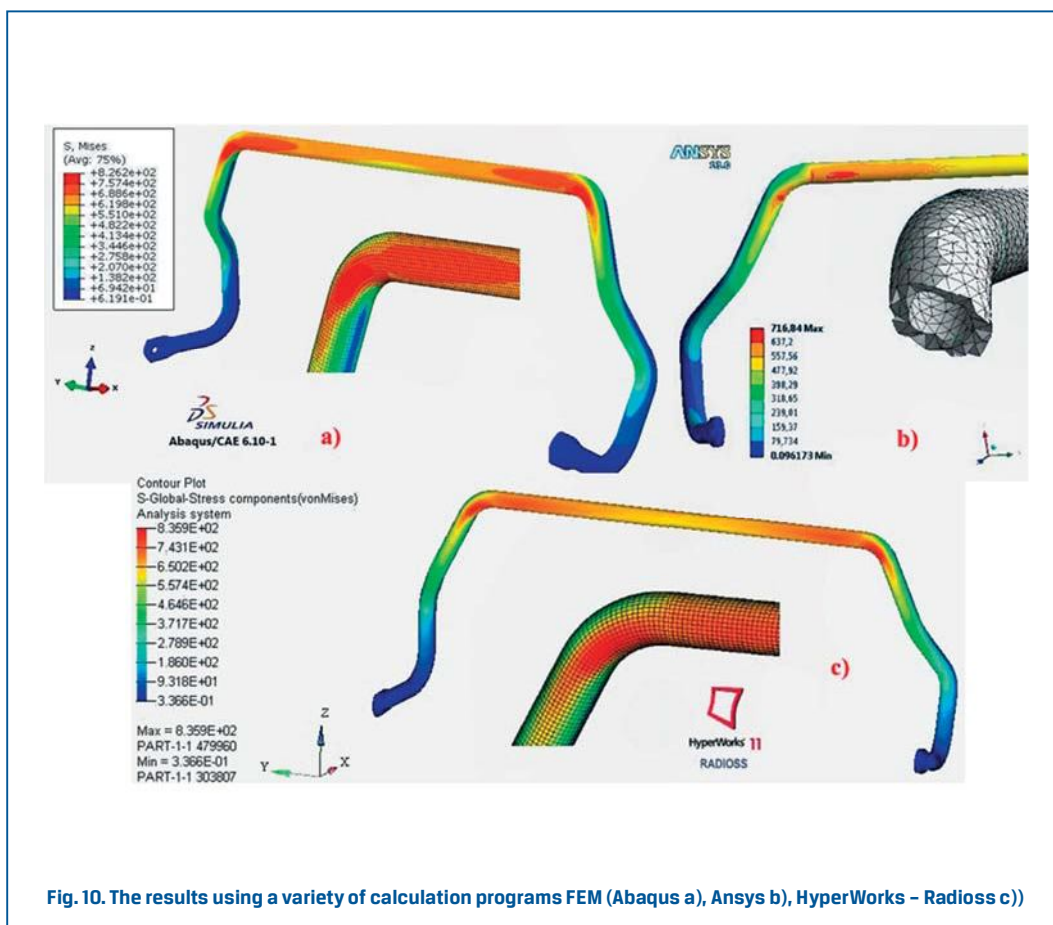
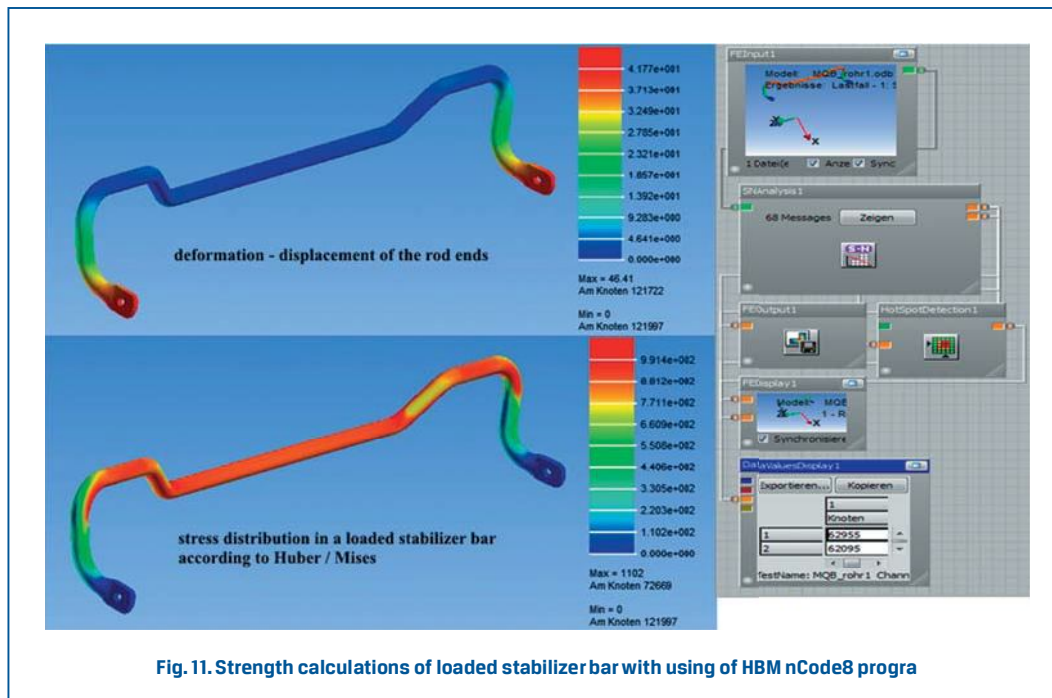


Fig. 10. The results using a variety of calculation programs FEM (Abaqus a), Ansys b), HyperWorks - RADIOSS c))

Further calculations – fatigue strength (HBM nCode8) (Fig. 11 and 12)



3. Conclusions

Each FEM calculation and simulation (according to the matrix – Fig. 1, 2) has to complete the validation and verification process.

Verification & validation plan (V&V plan, Fig. 13):

- General,
- Comment on experiments,
- Uncertainties,
- Verifications,
- Reference values for validation,
- Validation:**

Simulation with experiment:

- Validation of the calculation with a laboratory test,
- Answer: compliance of calculation and experiment acceptable.

Simulation with reality:

- Validation of the model with reality,

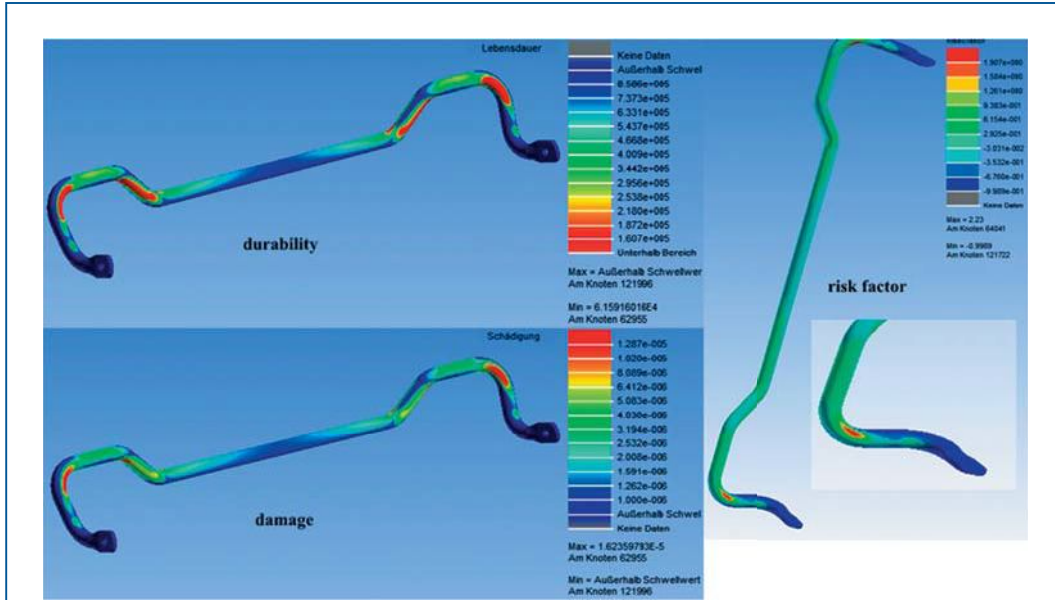


Fig. 12. Fatigue stress calculation of loaded stabilizer with using of HBM nCode8 program

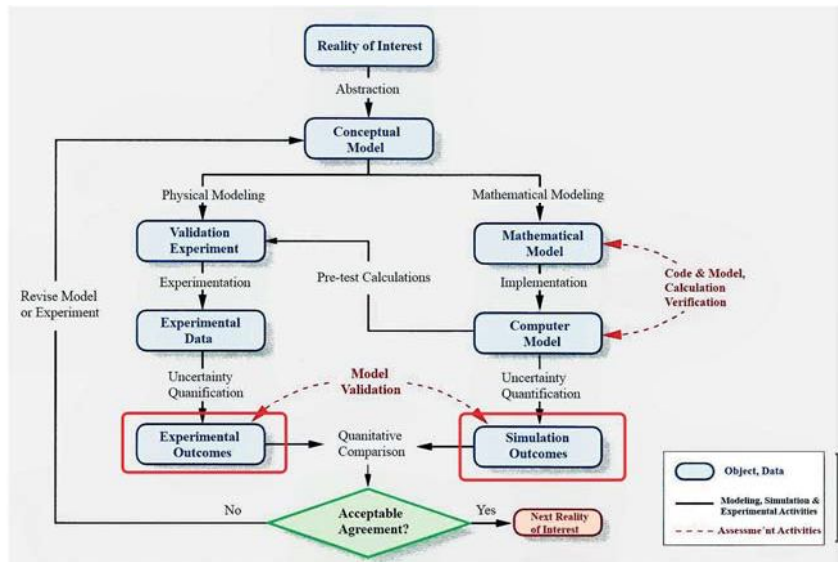


Fig. 13. Detailed model development, verification, & validation process (V&V Plan Thacker et.al., Los Alamos) [15]

- Please note: not the same as validation of the calculation with a test,
- Possibly from the summary of sub-experiments,
- Answer: compliance of calculation with reality acceptable.

Summary - verification of results

Objective of the verification of results:

- Verification of the FEM results: Validity of the FEM solution,
- Validation model: compliance of experiments / calculations with reality,
- Validation of calculation: compliance of experiments with calculation,
- Objective: sufficiently accurate description of reality or experiment!

The computational models of stabilizer bars Land Rover L405 and VW MQB created by the authors were used to present an optimal scheme of the calculation of stabilizers car using FEA. These models in industrial conditions are subject to continuous of verification and validation.

References

- [1] Abaqus Version 6.10.: Volume I: Static and Dynamic Analyses. Dessault Systems 2010.
- [2] BANAŚ K.: *Metoda Elementów Skończonych*, Seminarium BIT CM UJ, Instytut Modelowania Komputerowego, Politechnika Krakowska, 17 maja 2006, 1-25.
- [3] BIELSKI J.: *Wprowadzenie do inżynierskich zastosowań MES*. Politechnika Krakowska, Wydanie 1, Kraków 2010.
- [4] DEGER Y.: *Die Methode der Finiten Elemente*, Kontakt und Studium, Band 551.5. Auflage, Expert Verlag, Renningen 2010.
- [5] FROELICH P.: *FEM – Anwendungspraxis. Einstieg in die Finite Elemente Analys*, 1. Auflage, Vieweg + Teubner, Wiesbaden 2005.
- [6] FISCHER W.: *Finite – Element – Methode*, Skript zur Vorlesung, Fachhochschule Dortmund, 4. Auflage, Wintersemester 2011/12, 1-27.
- [7] GABBERT U.: *Finite – Element – Methode*, Teil 1. Lehrstuhl für Numerische Mechanik, Vorlesung, Otto von Guericke Universität Magdeburg, 2012, 1-48.
- [8] GEBCHARDT CH.: *Praxisbuch FEM mit ANSYS Workbench. Einführung in die lineare und nichtlineare Mechanik*. 1. Auflage Carl Hanser Verlag, München 2011.
- [9] KLEIN B.: *FEM – Grundlagen und Anwendungen der Finite-Element-Methode im Maschinen- und Fahrzeugbau*. 7. Auflage, Vieweg Studium Technik, Wiesbaden 2007.
- [10] Schulungsunterlagen HyperWorks11, Radioss. Fa. ALTAIR Deutschland GmbH, Hannover 2012.
- [11] SHIMOSEKI M., HAMANO T., IMAIZUMI T.: *FEM for springs*. 1. Auflage, Springer Verlag. Berlin – Heidelberg 2003.
- [12] REINERT U.: *Finite – Element – Methode*, Vorlesung. Fachhochschule Bremen, 2012, 1-24.
- [13] STEINBUCH R.: *Finite Elemente - Ein Einstieg*, 1. Auflage, Springer Verlag. Berlin – Heidelberg 1998.
- [14] THACKER B. H.: *Concepts of Model Verification and Validation*, Los Alamos, October 2004.
- [15] WITTEK A. M.: *Wpływ czynników konstrukcyjnych i technologicznych na trwałość stabilizatorów w pojazdach samochodowych*. Ph.D dissertation, Katowice 2013, 117-186.
- [16] FEM kurz und bündig, <http://www.smart-fem.de/fem.html>, Smart Engineering GmbH, 2013.