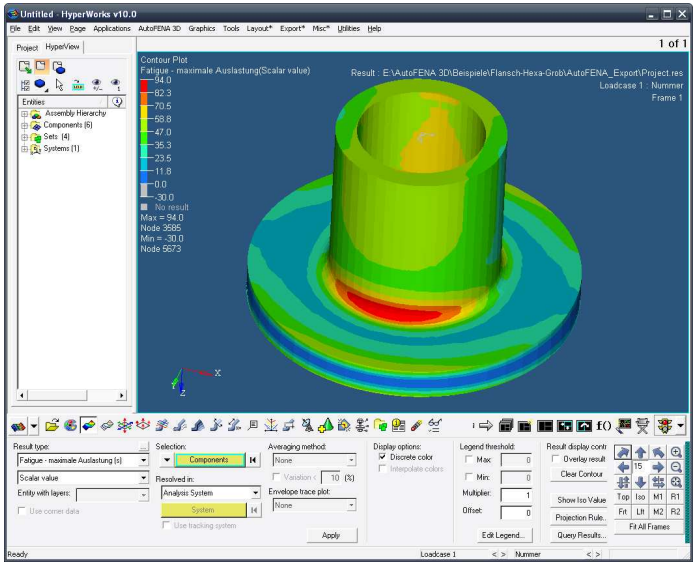
AutoFENA 3D:
Assessment of static strength and fatigue life according to FKM-Guideline within HyperWorks

EHTC 2010, Versailles

Dr.-Ing. Wolfgang Feickert
Dipl.-Ing. (FH) Tim Kirchhoff
Ingenieurbüro Huß & Feickert, Liederbach



www.ihf-ffm.de

Static strength and fatigue life according to FKM-Code in HyperWorks

Simulation explicit and implicit FE-Method

- Linear and nonlinear structural mechanics
- Dynamic
- Optimisation
- Thermal Transport
- Fluid Dynamic
- Crash
- Drop Test
- Containment Test

Software- und Product Development

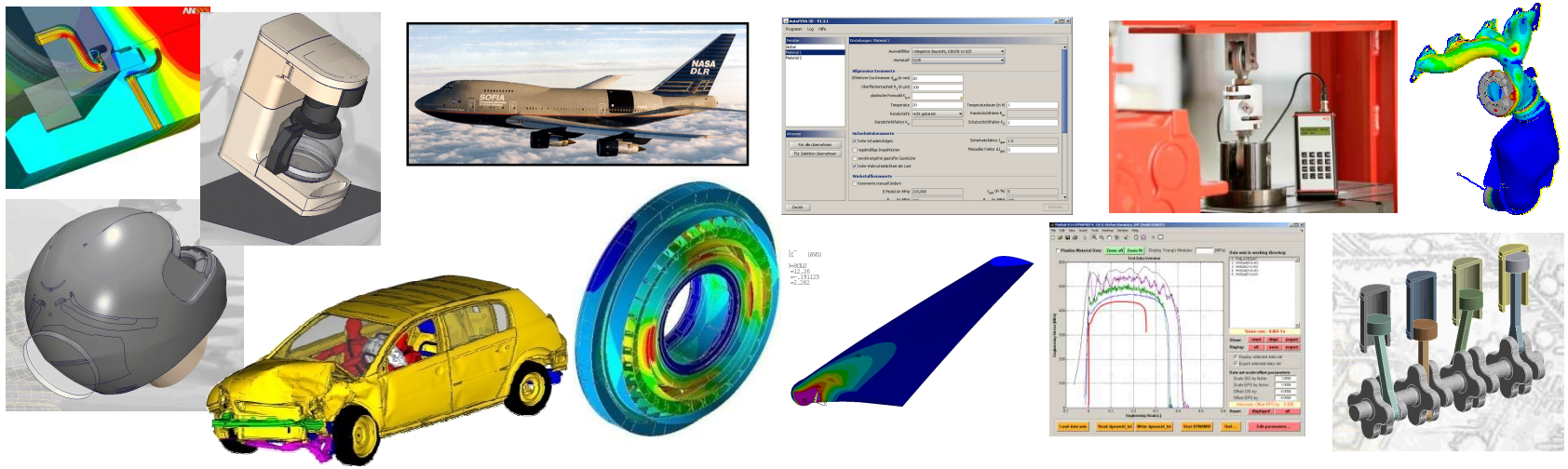
- Software Development
- AutoFENA 3D
- DYNAMID
- Concept Development
- CAD Design
- Assessment
- Preparation Of Drawings

Experimental Services

- Durability Testing
- Acceleration Measurement
- Modal Analysis
- Temperature- und Strain Gauge-Measurement

SoftwareTraining

- Training Course in ANSYS and LS-DYNA
- Sales Support for ANSYS and LS-DYNA
- Reseller for Design-Space

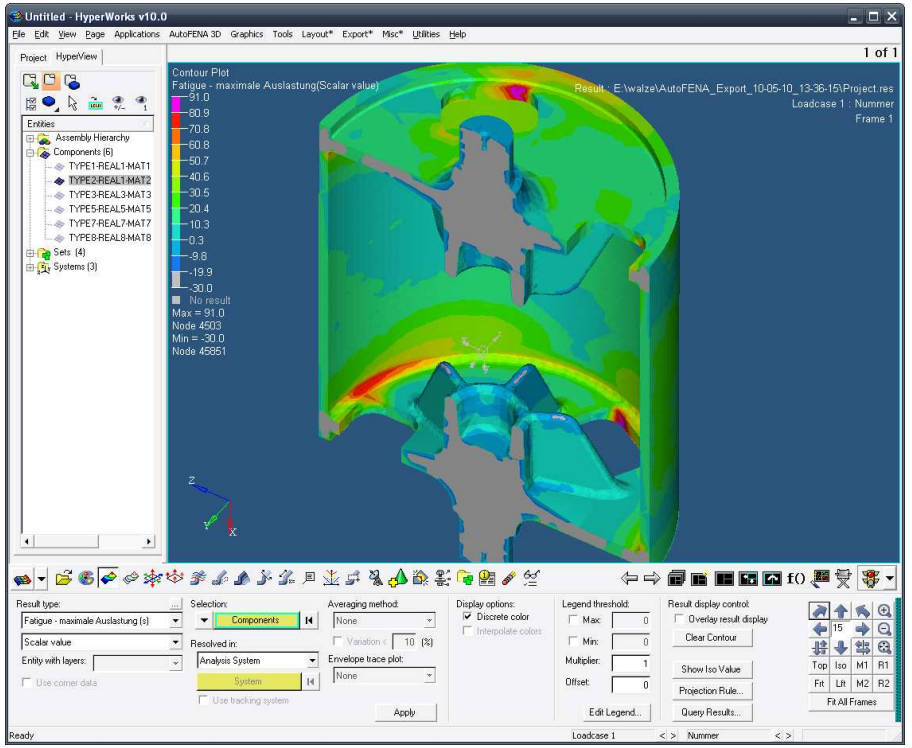


www.ihf-ffm.de

ingenieurbüro huß & feickert

Overview

- German FKM-Guideline
- Properties of AutoFENA 3D
- Software features
- Quality of software
- Further developments
- Conclusions
- Video about AutoFENA 3D



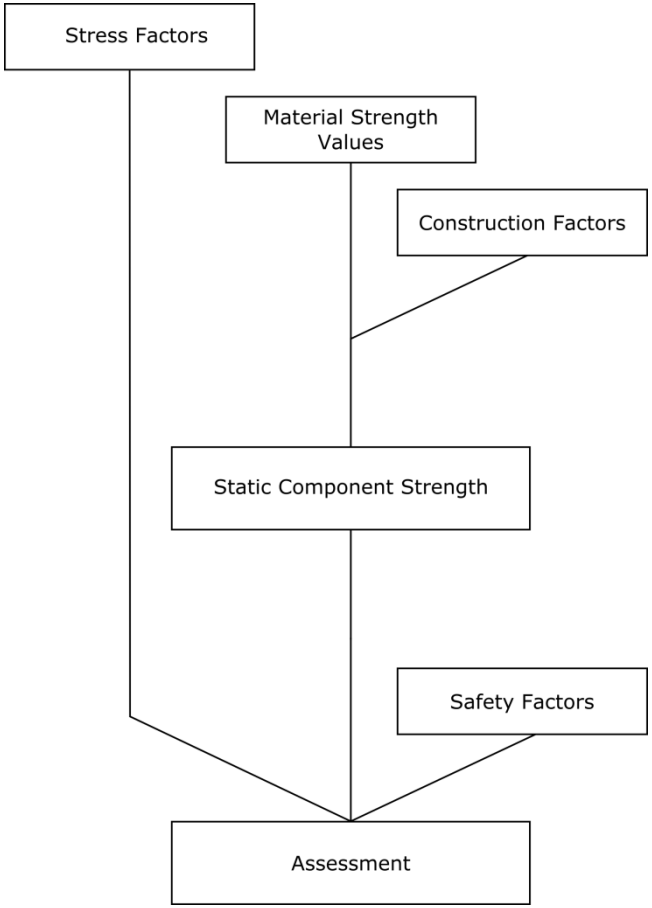
FKM-Guideline

- Well accepted and widely-used verification procedure in German speaking countries for analytical strength assessment for machine components
- Publisher: German Engineering Federation
  Verband Deutscher Maschinen- und Anlagenbau
  ForschungsKuratorium Maschinenbau
- The Guideline is continuously further developed within different research projects
- It covers a static strength and fatigue life assessment for steel, cast irons (both up to 500°C) and aluminium (up to 200°C)
- The Guideline represents the state of the art

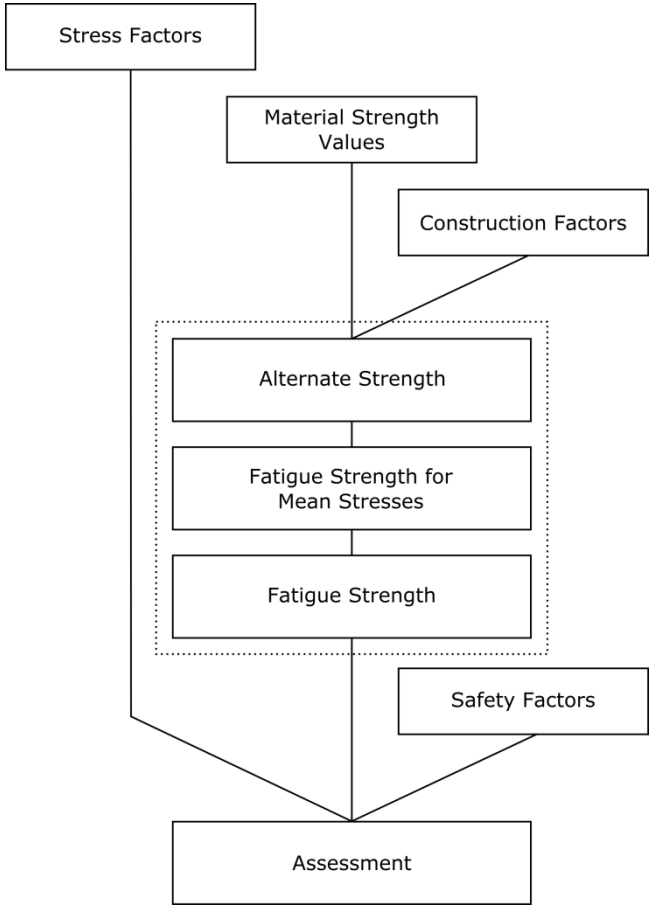
FKM-Guideline

- Assessment for
  - Static strength, limit state
  - Fatigue limit
  - Fatigue strength
- Calculation bases on
  - Nominal stresses in a section
  - Local stresses (effective notch stress)

FKM-Guideline



Static stress assessment



Fatigue stress assessment

FKM-Guideline

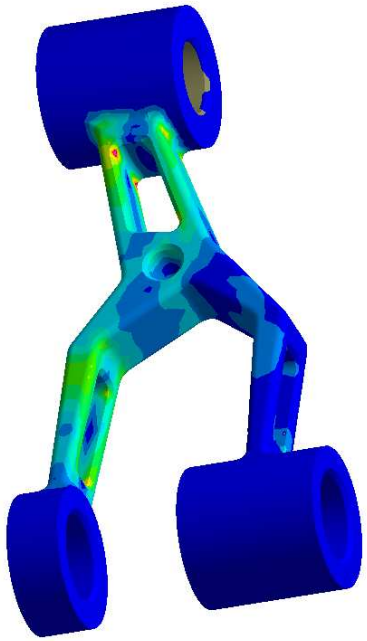
- Assessment procedure using local stresses
  - Fatigue action: determine effective notch stress
  - Fatigue resistance: against effective elastic notch stress in terms of a universal S-N curve
  - Summation of cumulative damage
  - Use adequate safety factors depending on regular / not regular inspection provided, probability of occurrence of load, non destructive testing and consequences of failure (damage), material
- Results of FEM-analysis can be used
- Critical points must be chosen by the user

FKM-Guideline

- Necessary input values
  - Method of calculation
  - Amplitude und mean value of elastic principle stresses at the reference point on the surface
  - Stress gradient normal to the surface or stress amplitude inside the part at a given depth
  - Behaviour of stress components (synchronously)
  - Material properties and surface treatment
  - Operating temperature and exposure time
  - Design: Thickness, average surface roughness
  - Safety requirements: resulting damage, probability of occurrence of load, inspection provided, non destructive testing, overload case

Software tool AutoFENA 3D

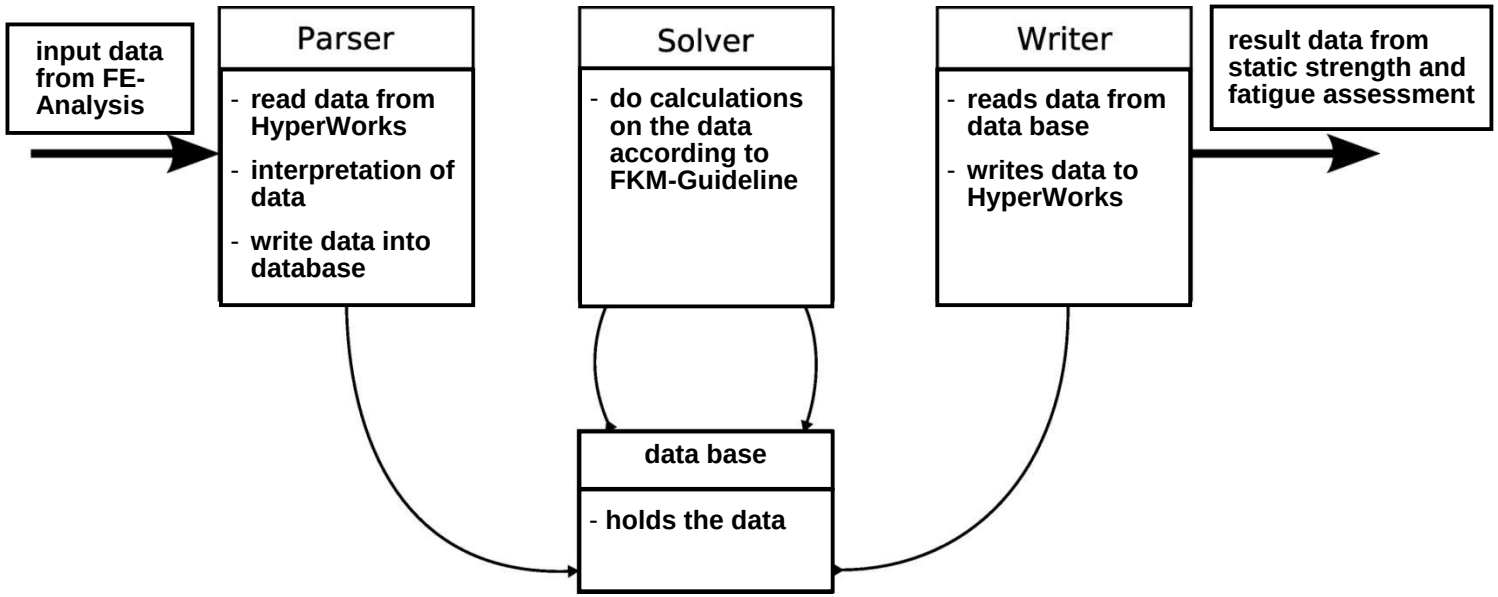
- Assessment strictly according to FKM- Guideline for FE-results
  - Program bases on the actual 5th revised edition 2003
  - Some future development are optional covered
- Assessment for the entire surface of a loaded structure
  - Assessment point = Finite Element Node
- Automatic calculation of the stress gradient normal to the surface at each assessment point leads to notch sensitivity parameter



Integration in HyperWorks

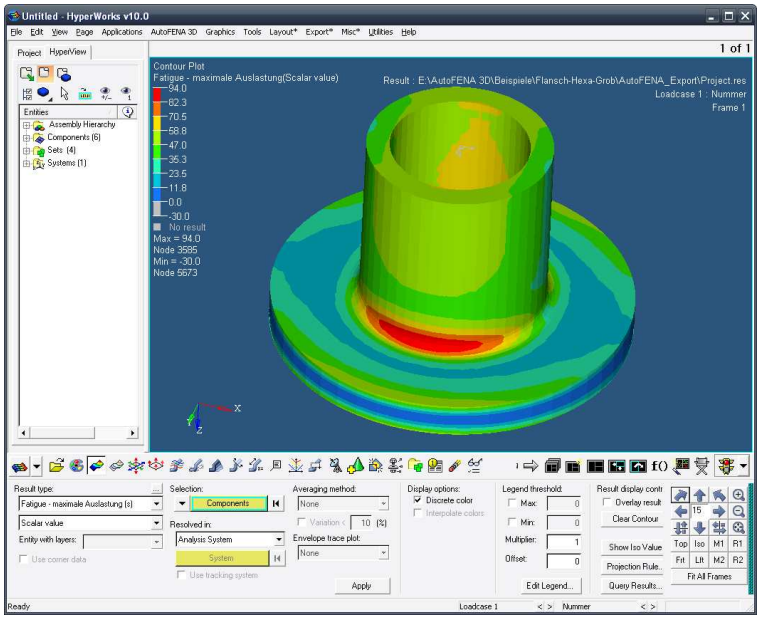
- Necessary stress data will be provided by the HyperWorks suite
- Integration as entry in the menu
- Stresses, node- and element-data at all surface nodes will be read out from the HyperWorks database
- Results from Radioss, NASTRAN, ANSYS and Abaqus are supported
- The results of the static and fatigue calculation using the FKM-Guideline are exported as HyperMesh result file
- The results of the static and fatigue assessment can be visualized in 3D within HyperView

Program structure



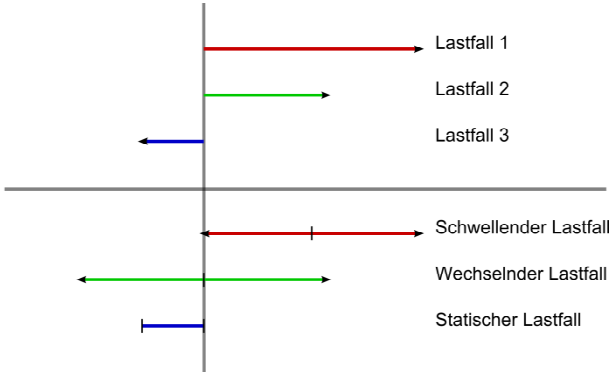
Assessment procedure

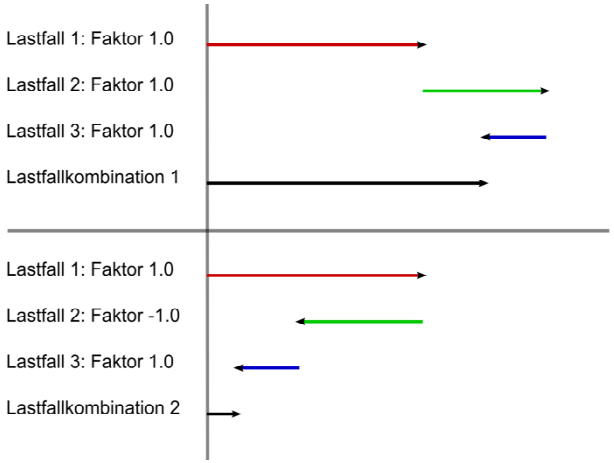
- Read in the results of the Finite Element calculation into HyperView
- Export the data to AutoFENA 3D
- Automatic generation of load case combinations
- Assessment for the entire surface
  - Static strength
  - Fatigue limit
  - Fatigue strength
- Examination
  - Post-Processing in HyperView
  - Report in html-Format



Load case combination

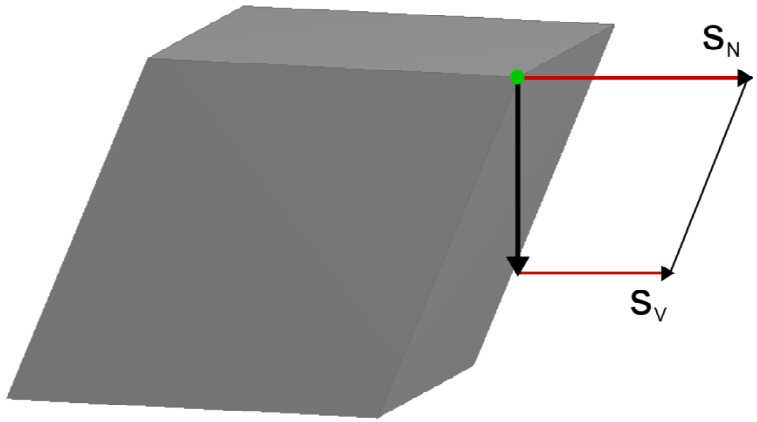
- Two load case combination methods
  - Independent load cases (e.g. pressure, wind)
    Definition of a arbitrary stress ratio R
    All linear combinations of the load cases will be calculated automatically
  - Dependent load cases (e.g. dead load, bolt force, pressure), which are already calculated within the FEM-Code (appropriate for non linear loads)
    The stresses to calculate the amplitude vary between these load cases
- Simplified procedure for non-proportional stresses according to FKM-Guideline





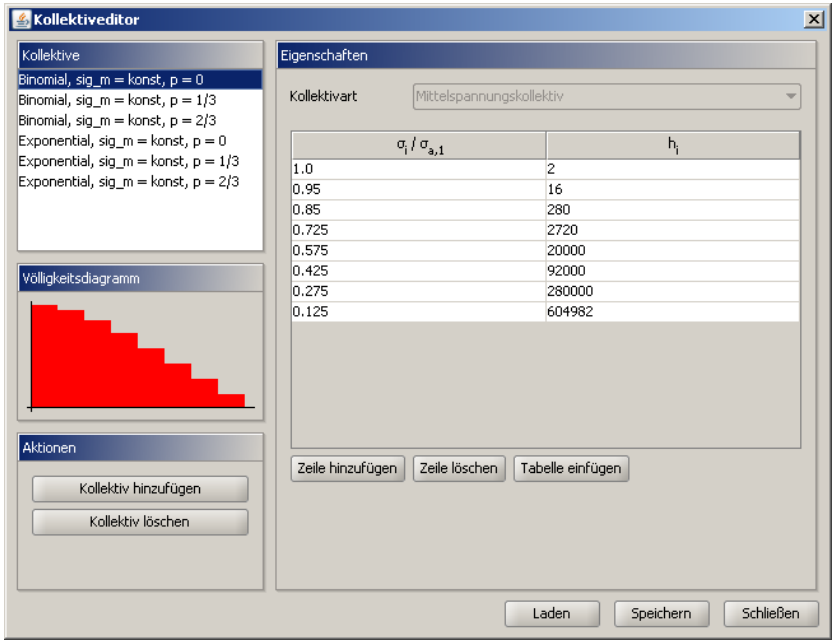
Calculation of stress gradient

- The stress gradient is necessary to calculate the plasticity factor (additional support)
- The stress gradient is calculated by the form functions of a 4-Node-Tetrahedron and a 8-Node-Hexahedron
- The stress gradient is calculated for every node on the surface



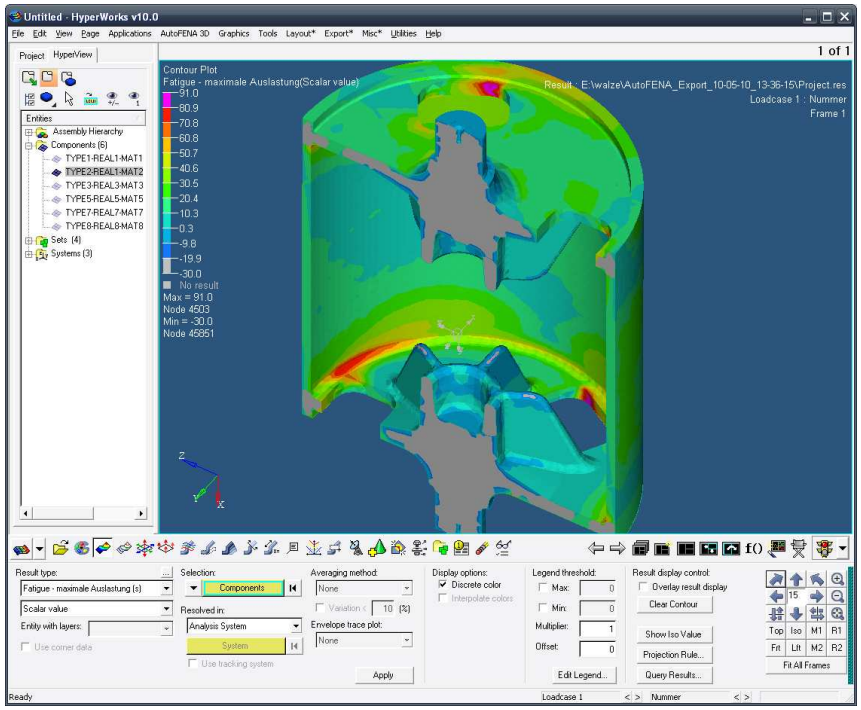
Strength Assessment

- Static strength assessment
- Fatigue limit
- Fatigue strength
  - Definition of stress collective
  - Procedure according to Miner rule consequent
  - Procedure according to Miner rule elementary



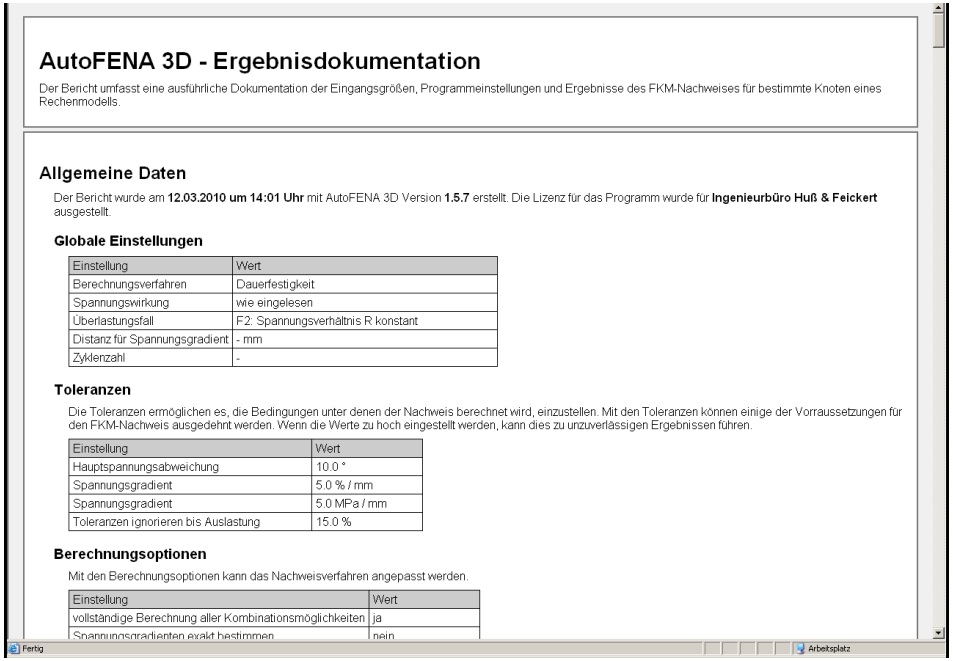
Examination

- Visualisation of results in HyperView
- Full field display of the static and fatigue utilization
- All post processing features of Hyper View can be used
- Further output to evaluate the reliance of the results
  - Non-averaged and averaged strength capacity consumed
  - positive stress gradient
  - misaligned stress tensor



Documentation

- Documentation in html-Format
- Detailed Documentation of critical or user defined Nodes



AutoFENA 3D - Ergebnisdokumentation

Der Bericht umfasst eine ausführliche Dokumentation der Eingangsgrößen, Programmeinstellungen und Ergebnisse des FKM-Nachweises für bestimmte Knoten eines Rechenmodells.

Allgemeine Daten

Der Bericht wurde am **12.03.2010 um 14:01 Uhr** mit AutoFENA 3D Version **1.5.7** erstellt. Die Lizenz für das Programm wurde für **Ingenieurbüro Huß & Feickert** ausgestellt.

Globale Einstellungen

Einstellung	Wert
Berechnungsverfahren	Dauerfestigkeit
Spannungswirkung	wie eingelesen
Überlastungsfall	F2: Spannungsverhältnis R konstant
Distanz für Spannungsgradient	- mm
Zyklenzahl	-

Toleranzen

Die Toleranzen ermöglichen es, die Bedingungen unter denen der Nachweis berechnet wird, einzustellen. Mit den Toleranzen können einige der Vorraussetzungen für den FKM-Nachweis ausgedehnt werden. Wenn die Werte zu hoch eingestellt werden, kann dies zu unzuverlässigen Ergebnissen führen.

Einstellung	Wert
Hauptspannungsabweichung	10.0 °
Spannungsgradient	5.0 % / mm
Spannungsgradient	5.0 MPa / mm
Toleranzen ignorieren bis Auslastung	15.0 %

Berechnungsoptionen

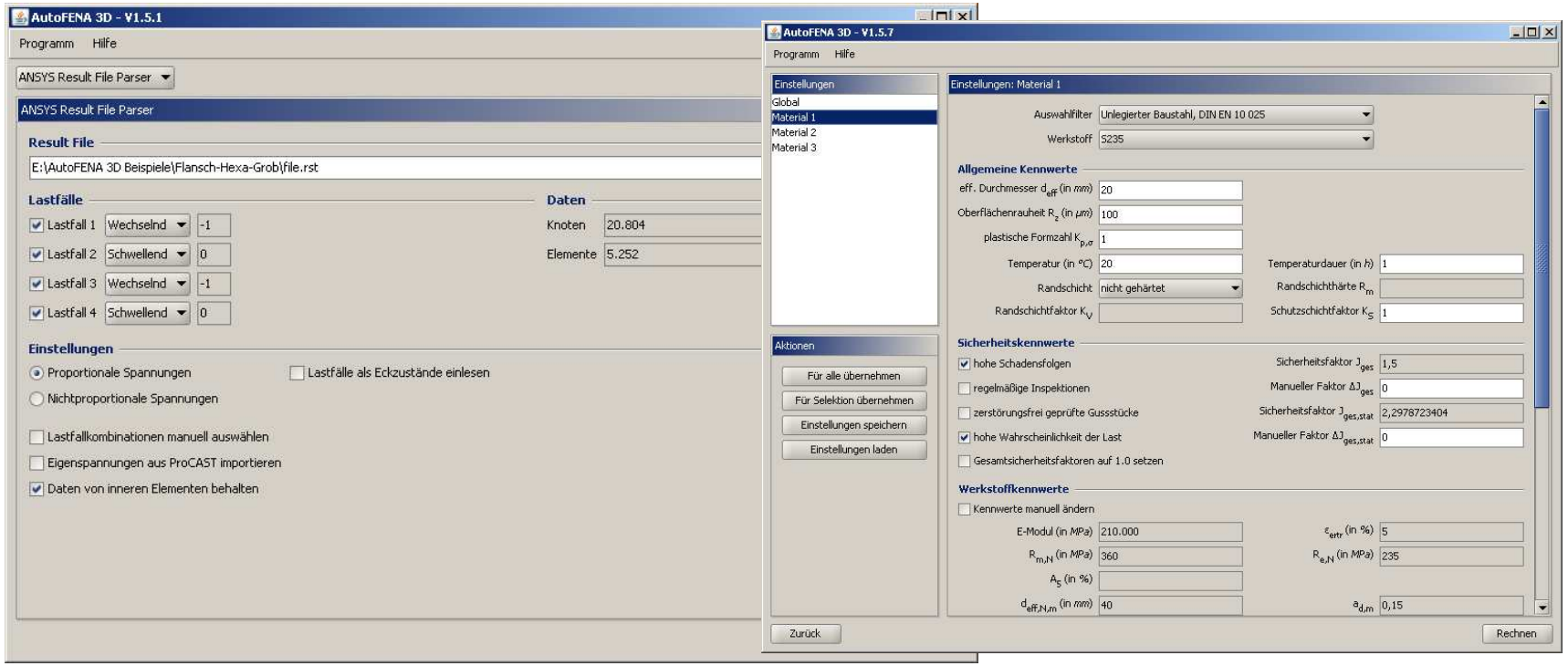
Mit den Berechnungsoptionen kann das Nachweisverfahren angepasst werden.

Einstellung	Wert
vollständige Berechnung aller Kombinationsmöglichkeiten	ja
Spannungsgradienten exakt bestimmen	nein

Fertig | Arbeitsplatz

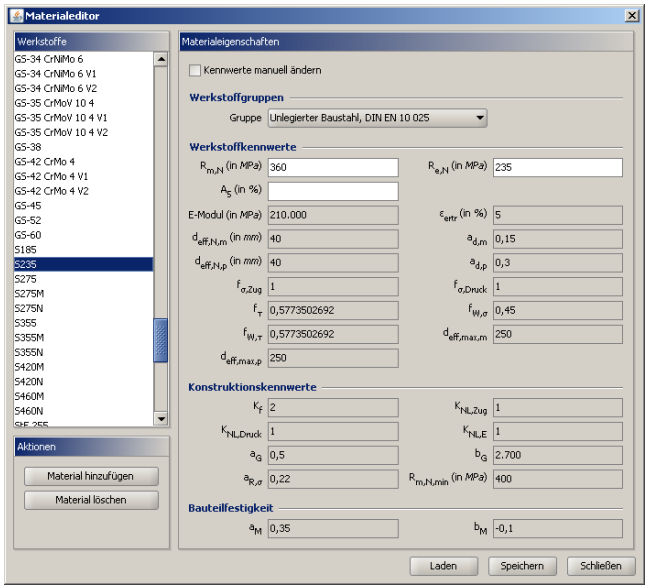
Using

- Easy to use GUI
- Storage of assessment parameters in a "Parameter-File", new calculations can be carried out with an unchanged parameter set



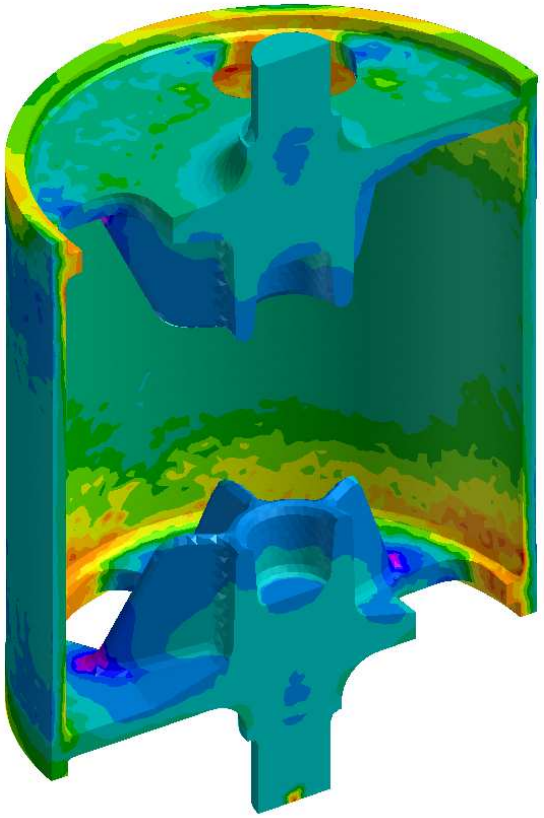
Material database

- Material database with all non-aluminium materials according to the FKM-Guideline (approximate 200 materials)
- User can define his own materials
- Filtering of materials according to material groups
- Material definition will be mapped to parts according to their element component name



Connections

- HyperWorks
  - Actual version 10.0
  - Integration in HyperView
  - Support of solid elements
- Connection to ProCAST cast simulation
  - Import Data from ProCAST

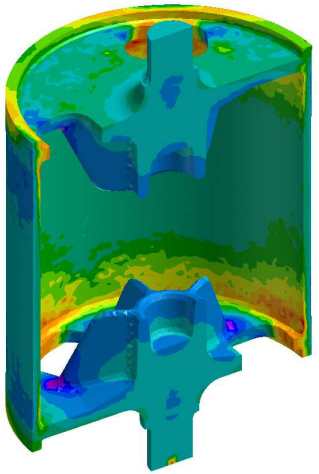


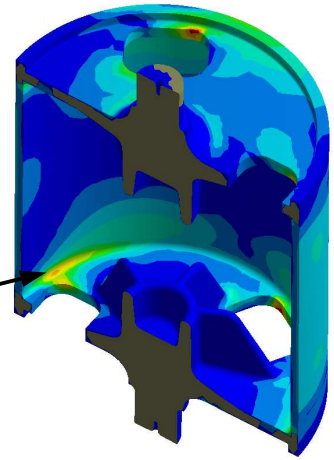
Reliability of the results

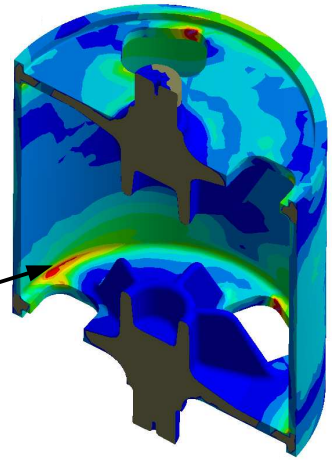
- Assessment of results at each Finite Element Node
  - Comparison of averaged and non-averaged results (analogue element stresses and nodal stresses)
- Misalignment of the stress tensor from the surface normal
  - Indicator for an insufficient discretisation in the interior direction
- Graphical output of error codes
  - e.g. positive stress gradient in the interior direction

Residual Stresses from CASTING

- Consideration of residual stresses due to different cooling rates during the casting process (ProCAST)
- Residual compressive stress at the surface increase the fatigue strength, residual tension stress decrease the fatigue strength
- Under cyclic loading the mean stress is changed by residual stress. Mean stress sensitivity of alloys and grey cast are significantly high
- Result mapping from casting simulation on FE-Model, including transformation of different coordinate systems and use of symmetry







www.ihf-ffm.de

Quality

- Tested with approximately 150 test cases
  - Automatic testing for each new release
- Benchmarks against other software of competitors
- Used by customers like: Liebherr, Heidelberger Druckmaschinen, Simpelkamp and Voith Paper
- Collaboration in Forschungskuratorium Maschinenbau: Fachkreis Bauteilfestigkeit; the research agency of the German Engineering Federation
- Collaboration in project accompanying work shops for the next edition of the FKM-Guideline

Further development

- Collaboration in the development process for the next edition guarantees a continuous progress of the software
- Future development of the FKM-guideline can be offered as optional features before the finishing of the revision process
- Customer request will be implemented in new software releases
- Continuous customisation of all connections
- Next steps:
  - consideration of shell elements
  - consideration of welded seams

Conclusions

- The manual selection of the critical point is not necessary
- No manual data input of stress values into the code procedure
- Thanks to the graphical visualization of the utilization interpreting the results is much easier
- Secondary critical areas can be detected and material in non critical regions can be saved
- The quality of the FE-Model in respect to the assessment can be checked
- The calculation is more reliable
- Using HyperWorks to provide the input data and visualize the results opens a broad range of applications