

Online Ψ Calculator

ISSUE 2/2017

Title Story // RDO in civil engineering

Simulation workflow using ANSYS EKM and optiSLang

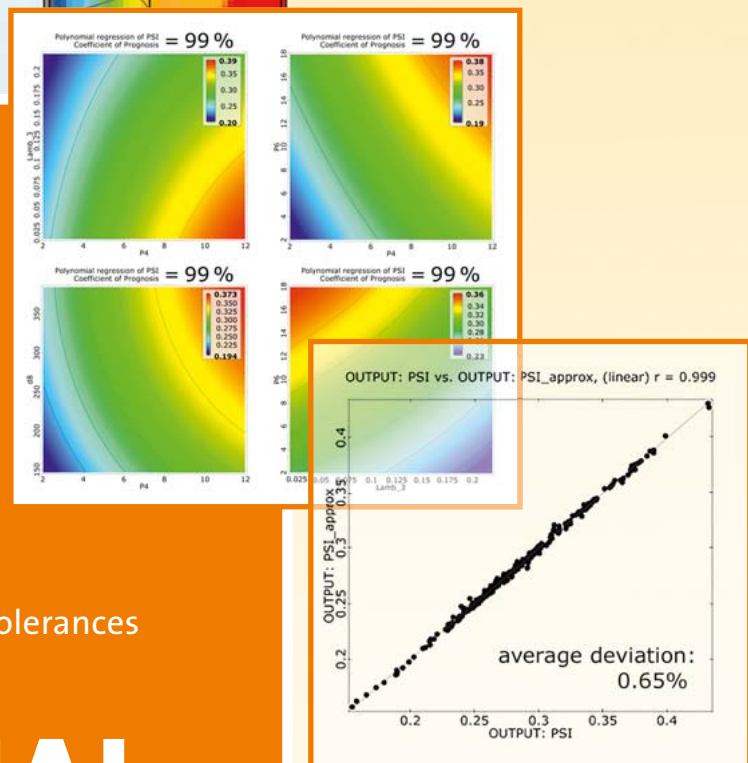
Parameter identification for a material model of an elastomer rubber boot

Metamodels in a cyber-physical system

Fatigue Analysis considering manufacturing tolerances

TMF panel optimization

RDO-JOURNAL



optiSLang

ETK

SoS

CADFEM ANSYS SIMULATION CONFERENCE

**The Simulation Driven
Product Development Conference**

Austria

April 24th - 25th 2018, Linz

www.simulation-conference.at

Germany

October 10th - 12nd 2018, Leipzig

www.simulation-conference.com

Czech Republic & Slovakia

June 6th - 8th 2018, Vysoké Tatry (SVK)

www.svsfem.cz

Russia

October 23rd - 25th 2018, Moscow

www.ansysconference.ru

Switzerland

June 14th 2018

www.simulation-conference.ch

United Kingdom & Ireland

November 1st - 2nd 2018, Dublin

www.cadfemukandireland.com

CADFEM[®]

ANSYS[®]

www.simulation-conference.com

PRACTICAL APPLICATION OF CAE-BASED RDO IN CIVIL ENGINEERING

Today, CAE-based product optimization is state-of-the-art in many industries, e.g., automotive, aero or tooling industry. In civil engineering, construction products often have been developed based on practical experience. The large amount of corresponding tests is still causing high development costs and long product development cycles. One reason CAE-based product development has not yet adapted in civil engineering at a higher level is the requirement of considering complex phenomena during the simulation of construction materials and products. Those include non-linear stress-strain behavior, anisotropic strength behavior, cracking and brittle material failure. Due to the progress in simulation technology in the last decades, modern CAE-tools have been meeting these requirements. For example, ANSYS provides software for very realistic simulations of nonlinear brittle material characteristics and their associated crack evolution. With the automated coupling of simulation tools and by using methods of CAE-based sensitivity analysis, optimization and robustness evaluation, material properties or behavior can be systematically analyzed. This enables the identification of crucial parameters for product enhancement. Furthermore, experimentally obtained results can be included, e.g., in the verification or calibration of simulation models to reduce the amount of hardware testing. Thus, customers can optimize their products and achieve significantly shortened development times.

For a modern CAE-based prototyping of construction products, Dynardo provides consulting service, training as well as software. optiSLang, the toolbox for variation analysis, supports the automated application of simulation-based sensitivity studies, model calibration and product optimization. Additionally, multiPlas offers the required material characteristics as well as their mechanical, thermal or hygric effects. Thus, multi physical simulations of non-linear materials within the ANSYS simulation environment can be efficiently conducted.

The title story of this journal will present some practical examples of virtual product development, optimization and sales support in civil engineering at the HALFEN GmbH. As a leading company in the fields of anchoring, framing and reinforcement technology, HALFEN has successfully implemented CAE-based RDO in the development of various product lines.

Apart from that, we again have selected case studies and customer stories concerning CAE-based Robust Design Optimization (RDO) applied in different industries.

I hope you will enjoy reading our magazine.

Yours sincerely



Johannes Will
Managing Director DYNARDO GmbH

Weimar, October 2017

CONTENT

2 // TITLE STORY // CIVIL ENGINEERING

Practical application of CAE-based RDO in civil engineering

8 // CUSTOMER STORY // PROCESS ENGINEERING

Simulation workflow building using ANSYS EKM and optiSLang

11 // CUSTOMER STORY // AUTOMOTIVE ENGINEERING

Parameter identification for a hyperelastic material model of an elastomer rubber boot

17 // CUSTOMER STORY // ELECTRICAL ENGINEERING

Metamodels in a cyber-physical system

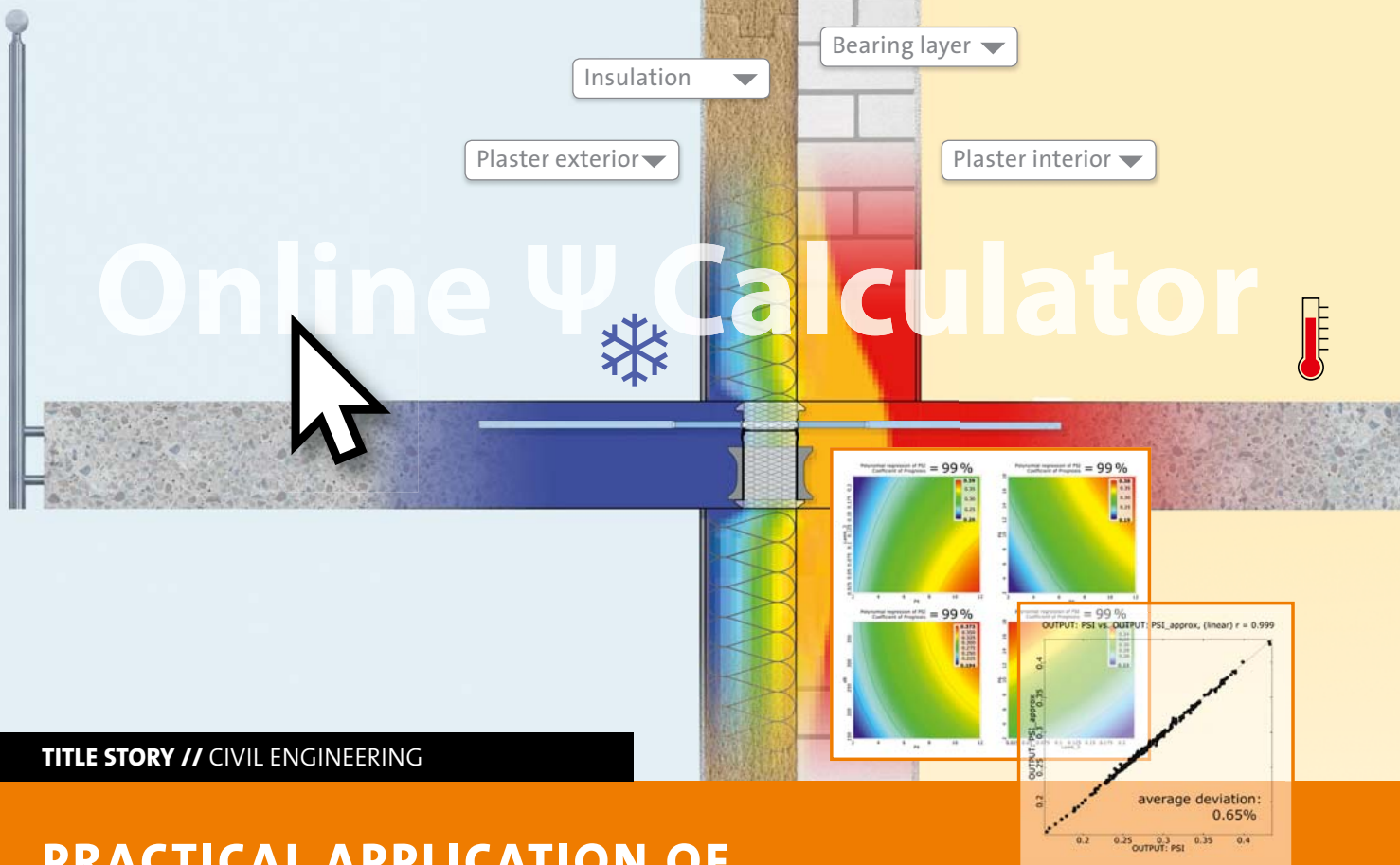
24 // CUSTOMER STORY // TURBO MACHINERY

Analysis of low cycle fatigue considering geometric manufacturing tolerances

30 // CUSTOMER STORY // AEROSPACE INDUSTRY

TMF Panel optimization

Online Ψ Calculator



TITLE STORY // CIVIL ENGINEERING

PRACTICAL APPLICATION OF CAE-BASED RDO IN CIVIL ENGINEERING

At HALFEN, CAE-based virtual product development enables the automated determination of relevant parameters for various product combinations and mounting conditions.

Introduction HALFEN

HALFEN was founded in 1929 and is one of the leading international companies in the fields of anchoring, framing and reinforcement technology in civil engineering. With about 1100 employees and more than 60 branches, HALFEN offers worldwide intelligent technical solutions and customized engineering consulting. The company's portfolio comprises approximately 20,000 products, yielding an enormous variety of available technical solutions. Accordingly, there is a high demand for variation analyses to determine relevant parameters. Using CAE-based RDO at HALFEN ensures that these complex correlations are understood and taken into account in product development. Since 2010, a close cooperation between HALFEN and Dynardo GmbH has been developed in order to efficiently implement CAE-based RDO for the development of various product lines.

Approaches to the development of construction products

In civil engineering, similar to other industries, the global competitive pressure is also demanding more cost-efficient production of particularly reliable and ecologically more

compatible products. In the past, construction products were often developed based on experiments or practical observations. The success of such a product development was usually not very controllable and often trusted to chance. The normally large amount of required tests caused relatively high development costs and time, leading to decreased efficiency.

Today, the combination of modern numerical simulation and CAE-supported optimization methods enables a very efficient virtual product development, as it is already well established in other industries, such as in the automotive sector. By using powerful simulation software, developers can literally "look" into their products and, thus, can directly visualize the effects of changed conditions. The analysis of different variations of input parameters can be conducted by simulation at a fraction of the comparable experimental effort. The automated coupling of simulation tools with methods of sensitivity analysis, mathematical optimization and robustness evaluation enables a targeted, best possible optimization of aimed product properties. Here, also experimentally obtained results can be included, e.g., for verifying or calibrating simulation models.

A special characteristic of many building materials is their non-linear and often brittle properties. With the help of modern simulation methods, e.g., the Finite Element Method (FEM), nowadays, simulations of nonlinear brittle material characteristics and the associated crack evolution can be conducted very realistically. For the non-linear simulation of building materials using FEM, elasto-plastic material models have proven to be very capable over the past few years. Dynardo has been developing the material model library multiPlas for the FE program system ANSYS. It is especially suitable for brittle materials, e.g., concrete, porous concrete, steel-fiber concrete, reinforced concrete, masonry, sand-lime brick or brick. In addition to mechanical characteristics, also thermal and hygric effects can be included in multiphysical simulations. Thus, new and more comprehensive possibilities for the simulation of the component behavior are opening up including all product stages from production up to the mounting.

In the following passages, practical examples will show the application of simulation-based optimization methods and sensitivity analyses in the development of construction products. For this purpose, Dynardo has been developing the optimization platform optiSLang for sensitivity analyses, robustness evaluation, single and multi-target optimization as well as stochastic analysis.

CAE-based RDO in virtual product development

The process of a simulation-based optimization is unitized in the following steps:

1. The generation of a suitable parametric simulation model and process

The simulation model can be composed of several partial models, just as the simulation process can include several different analyses (e.g., thermal, hydraulic or mechanical analysis – see Fig. 1). The entire simulation process, including the simulation model, is set in such a way that it can be run in batch mode. The input values (i.e., geometry parameters, material characteristics, influencing conditions) are defined as variables. Finally, the result values and plots

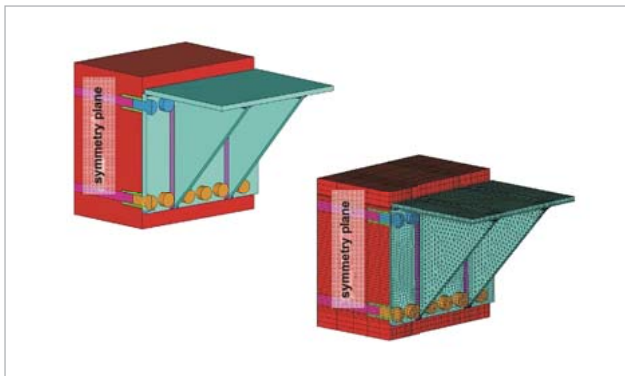


Fig. 1: FEM simulation of a steel anchoring

required for product development are automatically generated after each simulation.

2. The conduction of a sensitivity analysis

The aim of a sensitivity analysis is to investigate the sensitivity of individual output parameters, which are important for product development in regard to the variation of input parameters. For this purpose, different designs are generated and simulated by varying the input parameters within predefined parameter limits. For a meaningful statistical evaluation, it is necessary to scan the entire design space within the upper and lower limits of all parameters as accurately as possible. For accomplishing this task, optiSLang provides powerful sampling techniques (i.e., Latin Hypercube Sampling).

In order to identify the correlations between input parameters and result variables, metamodels representing the best possible prognosis quality are generated and applied with the help of optiSLang. With sufficient accuracy, these metamodels are then capable of determining in real-time the values of the result variables according to the respective input parameters. If there is an adequate prognosis quality of the metamodels, they can be used instead of solver runs. As a result of a sensitivity analysis, it is possible to identify the relevant input parameters for the optimization (i.e., those parameters which substantially influence the result variables). Subsequently, unimportant input parameters are filtered out and the parameter space required for optimization can be reduced. Further important knowledge can be gained from the variability space of the result values and from the possible indication of competing optimization targets.

3. The conduction of an optimization

In this step, target functions and constraints (or restrictions that cannot be violated by optimization) must be defined. optiSLang offers different strategies for the optimization, i.e., gradient methods, response surface methods, particle swarm methods, as well as evolutionary and genetic methods. The suitability of the individual optimization methods depends, i.e., on the number and the type of variables, as well as on the characteristics of the target variables. In addition to the direct simulation results, e.g., stresses, deformations, loadings, heat transfer, even cost or production-relevant parameters can be particularly considered.

4. The conduction of a robustness evaluation

After the optimization procedure, the robustness of the optimal design should be verified. For this purpose, a stochastic analysis is conducted for the optimized product, taking into account the natural scattering (distribution) of the important input parameters. As a result of the robustness evaluation, the statistical measures (mean values, standard deviations and variational coefficients) of the result variables or product properties can be verified. If there is a small scatter of important product properties, the design can be considered robust.

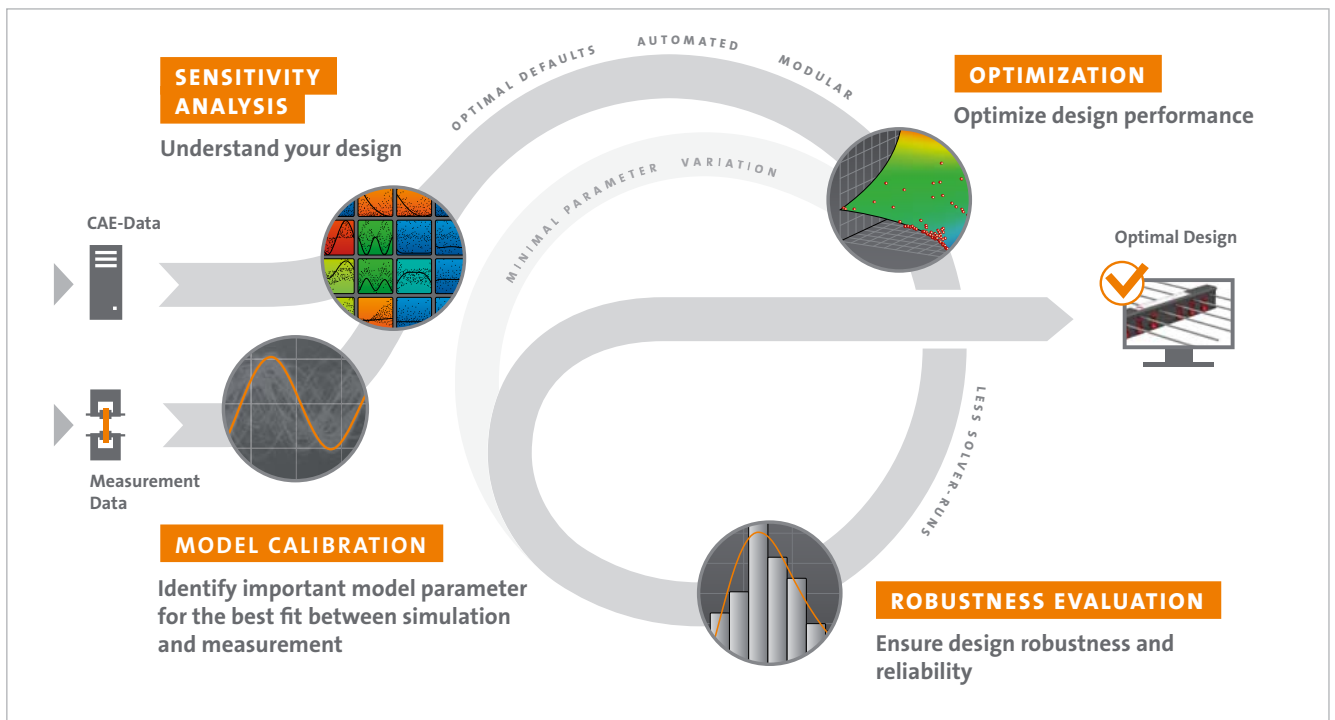


Fig. 2: Schematic chart of Robust Design Optimization based on multiple designs, measurement or observation points using optiSlang

Example of practical RDO application at HALFEN

With the HALFEN HIT ISO-element, the company offers a versatile and innovative product line for thermally insulated balcony connection. In practice, they are used, e.g., to prevent mold growth and to ensure increased structural physical requirements. The elements consist of load-bearing and heat-insulating parts for the thermal separation of reinforced concrete components (Fig. 3). The ETA-13/0546 approval and the corresponding CE-marking as well as the ICC-ES approval (ES-3799) means the elements are certified and can be used in over 30 countries in Europe and USA. The patented modular system considerably reduces transportation and storage costs by up to 65 % as well as the CO₂ emission and increases the flexibility and planning reliability in the precast concrete industry. The HALFEN Insulated Connection already comes with a fire resistance of REI 120 as standard. This avoids incorrect planning or wrong assembly in respect of the fire protection requirements.

For the thermal insulation certificate according to the Energy Saving Regulation (EnEV), simulations of thermal bridges have to be conducted. In order to determine these bridges regarding balcony connections, ψ -values are required for modeling the construction. The HALFEN HIT ISO-elements are very flexible and can be applied for various installations, e.g., for different wall constructions as a thermal insulation composite system, a monolithic wall, a multi-leaf wall or in a sandwich construction. In addition, a variation of floor and balcony mountings, required load capacities or window connections can be considered to select the right heat transfer coefficients (ψ -values) for further calculations. Altogether, more than 100 million individual variations would be basi-



Fig. 3: HALFEN HIT ISO-element HIT-HP/SP MVX for the thermally insulated connection of balconies and projecting cladding parts

cally possible. In the past, production engineers and architects had to commission thermal calculations based on the particular case or had to use catalog values for standardized cases. On behalf of HALFEN, Dynardo generated parametric FE models and metamodels in ANSYS optiSlang for the simulation of the HIT ISO-elements. Subsequently, the ψ -values can be calculated in real time depending on the input parameters. Thus, HALFEN can provide end users with the appropriate characteristic values via an MOP service on their website. In the following, the methodology of this service will be presented by explaining the individual five project steps.

1. Generation of parametric FE simulation models for the HALFEN HIT-MVX ISO-elements

In order to simulate the variations, parametric FE models were generated in ANSYS optiSLang representing the different installation conditions (e.g. with or without a window or balcony door, above and below the respective connection).

These parameters include:

- Two different joint widths
- Two to twelve different compression shear bearings and two to eighteen tension bars (depending on load carrying capacity)
- Four different types of wall mounting
- Up to eight geometry and material parameters for wall mounting (Fig. 4 top left)
- Up to three concrete coverings
- Up to nine geometry and material parameters for floor slab mounting (Fig. 4 top right)

The FE models also take into account the various outside and inside conditions of temperature. For each design, five structural-physical response variables are calculated: ψ -value, U-value, $\Theta_{si,min}$ -value, an equivalent thermal conductivity value, and the f_{Rsi} -value.

2. Validation of the simulation model

Fig. 5 exemplarily shows the calculated temperature profiles for two different conditions with or without windows. These temperature gradations as well as derived and calculated response variables were compared with real measured values. The outcome indicated a high correspondence and accuracy of the calculation results in regard to the measured values.

3. Sensitivity analyses regarding the relevant parameter spaces for various product lines using the sampling strategies of optiSLang

A total of 48 different installation conditions with approx. 20 parameters were calculated using an automated workflow. Up to 700 designs were generated and calculated for each installation condition using the advanced Latin Hypercube sampling available in optiSLang. The FE calculations were processed in parallel with ANSYS Parametric Pack licenses on the Dynardo cluster. Thus, up to 64 designs could be parallelly calculated. The calculation time for one design was approx. 10 minutes (using 4 CPUs). The max and min limits of the parameter variation ranges were considered according to the possible installation conditions.

4. Generation and verification of metamodels

Metamodels with prognosis quality of 99 percent Coefficient of Prognosis (CoP) value were created for all response variables of the examined mounting conditions. The latest optiSLang MOP technology of Kriging or Moving Least Square approximations (Fig. 6/7, see next page) was applied to achieve an high and accurate prognosis quality (CoP-values).

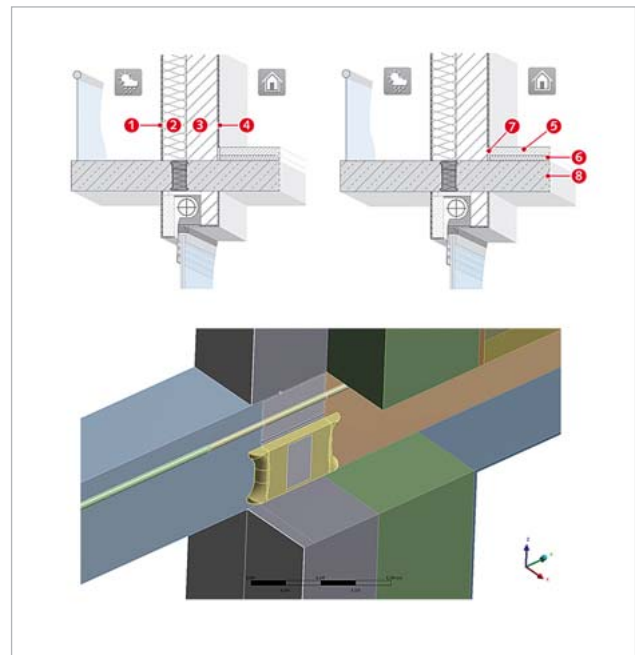


Fig. 4: Parametric wall construction (top left), parametric floor slab mounting (top right), parametric FE model (bottom)

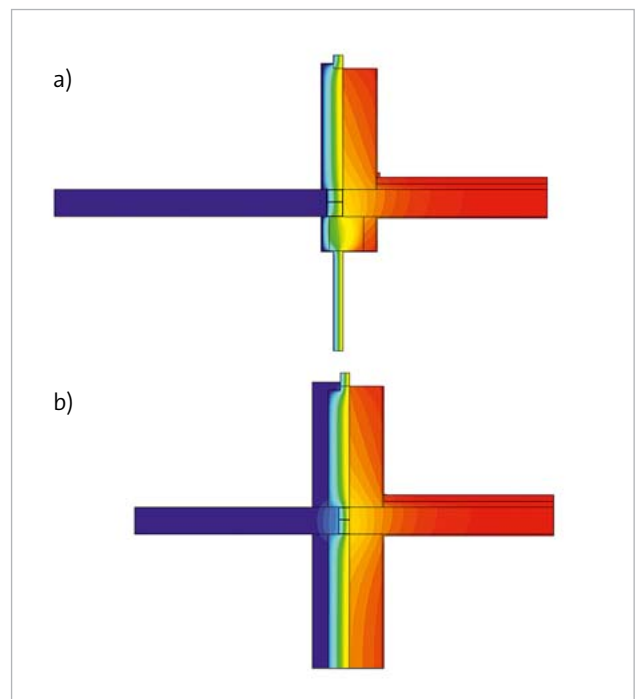


Fig. 5: Temperature distribution from thermal analysis for different conditions, ETICS facade with windows on top and bottom (a), sandwich construction with window on top (b)

5. Implementation of a metamodel-based service as a real-time solver solution at the HALFEN website

For the integration of the metamodels into a web application, Dynardo developed a MOP service providing the metamodels as stand-alone and real-time solver solution. The HALFEN Ψ -Calculator allows end users an efficient and professional calculation of structural-physical properties regarding their particular construction to find the corresponding thermal values.

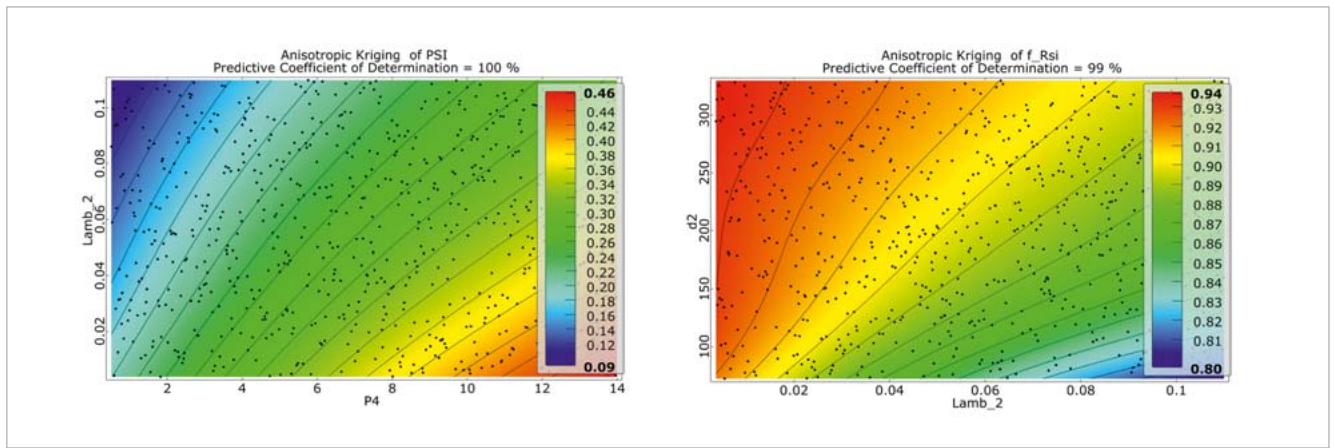


Fig. 6: Metamodels for different response variables (left: Ψ -value, right: $f_{R,SI}$ -value)

The required parameters are entered in five simple steps:

- Wall construction
- Wall mounting
- Floor slab mounting
- Window / door
- HALFEN HIT ISO-element

Regarding the wall construction, the user can choose between thermal insulation composite systems (ETICS), as well as monolithic, double-leaf or sandwich constructions.

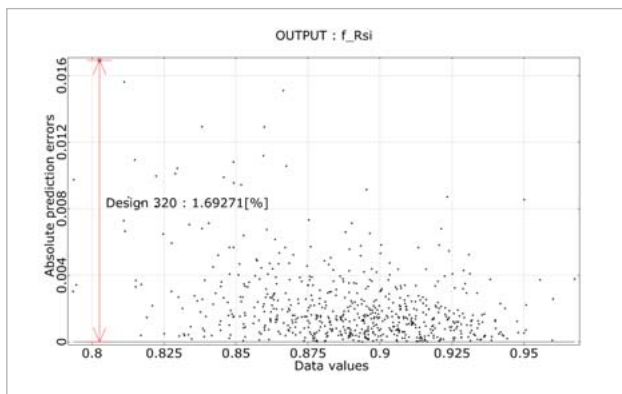


Fig. 7: New output of the maximum local deviation of a metamodel in optiSLang

All wall constructions consist of different layers, e.g., exterior plaster, insulation or bearing layer. The materials, their thermal conductivity and the dimensions of the individual layers can be defined in further steps. Standard materials with their properties will be suggested for a quick choice. The tool also offers menu items regarding required windows or doors above the balcony.

The results of the Ψ -value calculation can be processed to a PDF file containing all relevant parameters as a attachment to, e.g., planning or project documents. The PDF output can be completed with user's individual project data. The results can be saved and reused via link for editing or changing conditions at any time.

Link to the MOP-web service:

<http://psi.halfen.com/de-de/start>

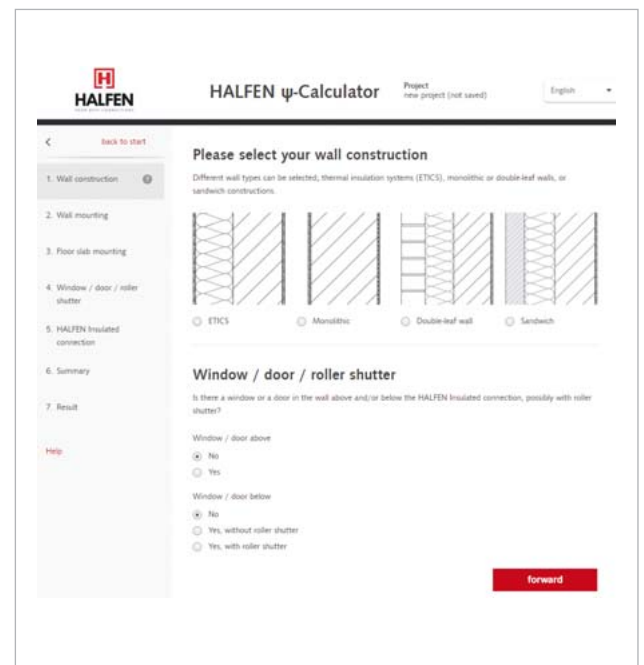


Fig. 8: Screenshot of HALFEN Ψ -Calculator

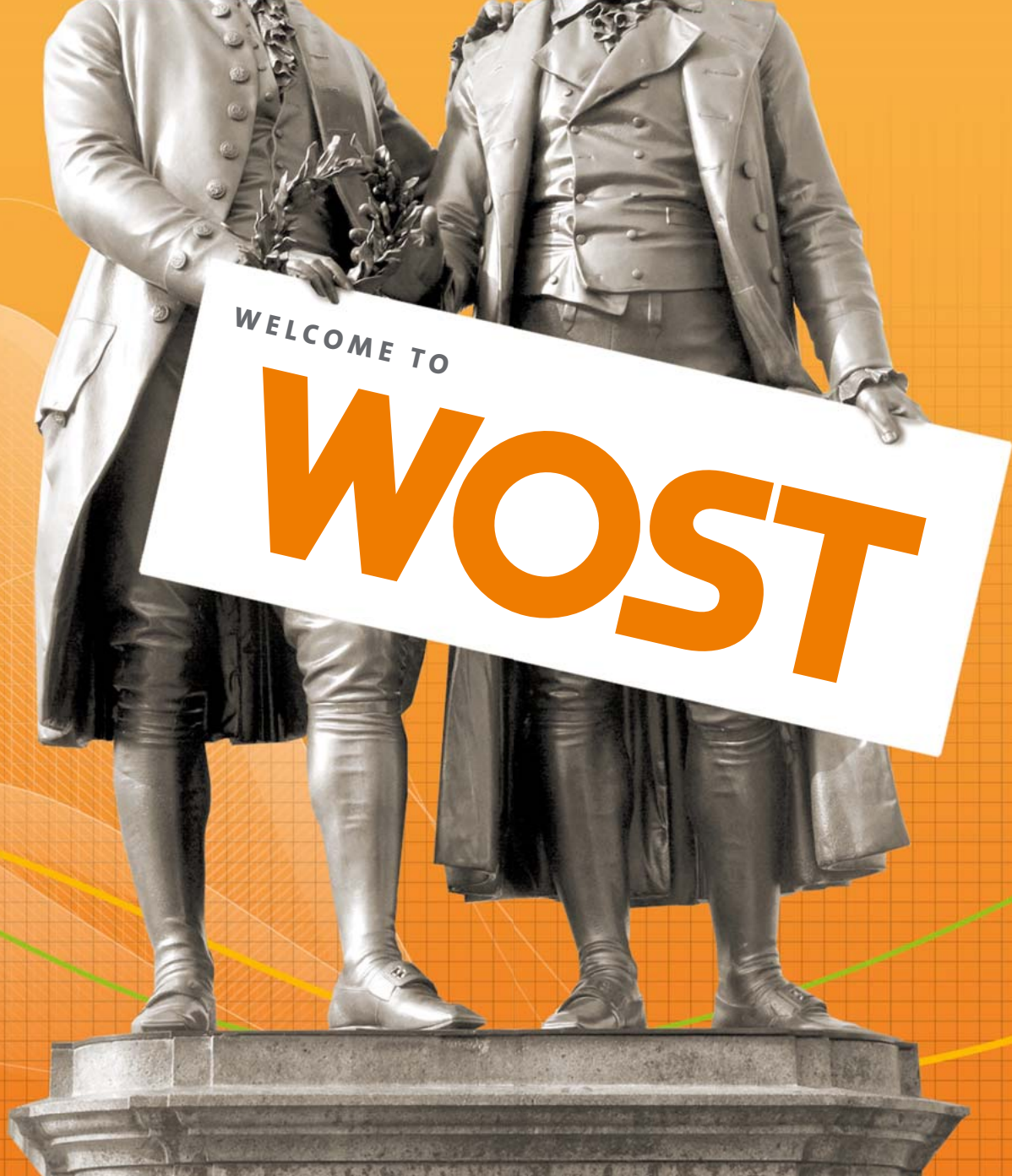
Conclusion

As in many other industries, RDO has been also established in civil engineering. HALFEN has implemented a virtual product development and optimization, which is successfully applied in all production phases. The described MOP web service provides end users, e.g. engineers or architects, with an easy access to individual HALFEN products. It is thus directly supporting the sales and service activities of HALFEN.

Authors //

Dr.-Ing. Roger Schlegel (Dynardo GmbH)

Dr. Raimo Füllsack-Köditz, Dr. Thorsten Heidolf (HALFEN GmbH)



ANNUAL WEIMAR OPTIMIZATION AND STOCHASTIC DAYS

Your conference for CAE-based parametric optimization, stochastic analysis and Robust Design Optimization in virtual product development.

The annual conference aims at promoting successful applications of parametric optimization and CAE-based stochastic analysis in virtual product design. The conference offers focused information and training in practical seminars and interdisciplinary lectures. Users can talk about their experiences in parametric optimization, service providers present their new developments and scientific research institutions inform about state-of-the-art RDO methodology.

Take the opportunity to obtain and exchange knowledge with recognized experts from science and industry.

You will find more information and current dates at:
www.dynardo.de/en/wosd.

We are looking forward to welcoming you to the next Weimar Optimization and Stochastic Days.



CUSTOMER STORY // PROCESS ENGINEERING

SIMULATION WORKFLOW BUILDING USING ANSYS EKM AND OPTISLANG

At MANN+HUMMEL, simulation teams implement optiSlang in ANSYS Engineering Knowledge Manager (EKM) to include CAE-based design optimization in the development of their filtration products.

Introduction

Simulation Management plays an important role in sustaining a high quality standard of simulation results. Apart from people and hardware management, in simulation workflows other challenging areas of software administration has to be considered regarding data, process, quality, as well as knowledge management. Having become aware of these requirements, ANSYS has already come up with a software solution called ANSYS Engineering Knowledge Manager (EKM).

At MANN+HUMMEL, as a leading global specialist, engineers in Filter element team use EKM workflows to improve the team collaboration and the management of projects by implementing engineering workflows formerly limited to CAD related ecosystems.

ANSYS Workbench can manage various levels of interfacing software with user-friendly drag and drop options. In CAE simulations, a lot of data has to be handled regarding cleaned up CAD, meshes, boundary conditions, result files, post processing and reports. Thus, it is also important to access data on a real time basis by synchronizing it across

servers at different locations. The storage of data should also be ensured in a structured way similar to a PLM database to enable a seamless collaboration.

Simulation Management at MANN+HUMMEL

The simulation management at MANN+HUMMEL includes data as well as process, workflow, quality and knowledge management aspects. Here, the fundamental task is to keep track of the project management regarding time, resources and quality requirement. Using EKM workflows, engineers can create GUI driven workflows, control all areas of simulation management and include other supporting software like optiSlang. Quality checklists have been integrated which help in quality audits and implemented EKM sections supports the engineer with knowledge management.

In ANSYS EKM, every step of the simulation process, such as geometry cleanup, meshing, setup of boundary conditions, running simulations, post processing or report preparations can be generated as a node and be used for different projects. EKM displays these steps as check boxes in its user

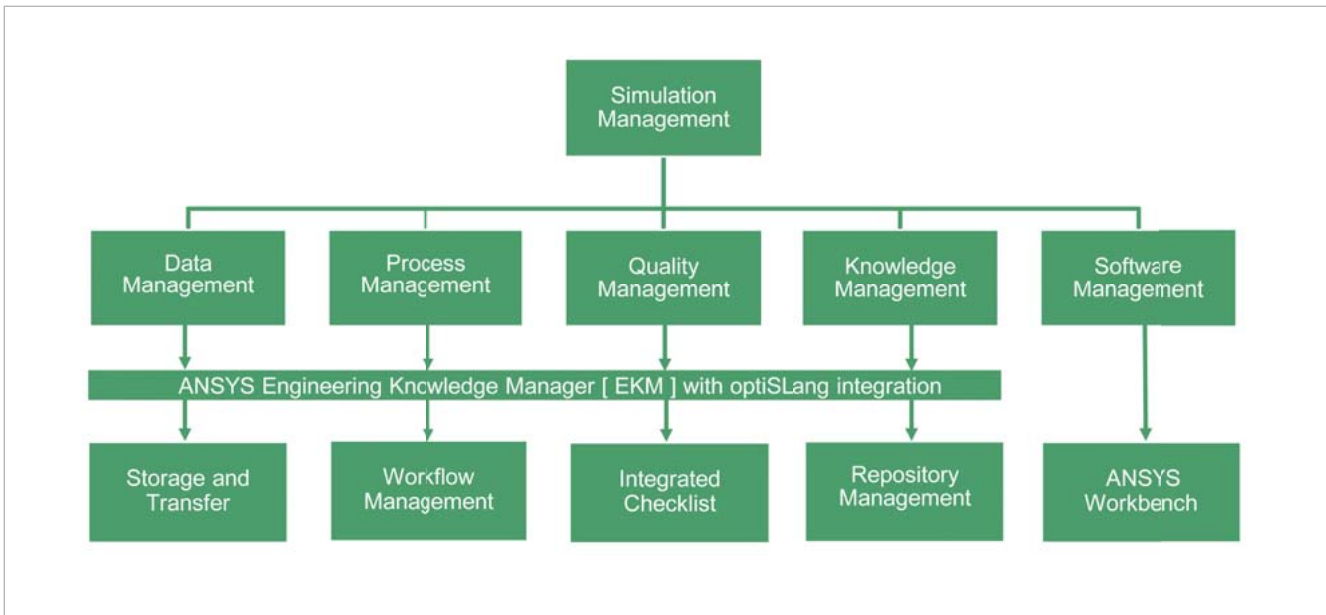


Fig.1: Simulation Management at MANN+HUMMEL

interface. Furthermore, engineers working on simulation projects can keep updating these workflows, which not only serves as a part of quality management for audit trials, but also shows the status of projects to all involved stakeholders.

The software optiSLang enables the integration of procedures of CAE-based sensitivity analysis, optimization and robustness evaluation into EKM workflows. Nowadays, design engineers do not only require simulation plots indicating whether an analyzed component fulfills design criteria or not. To meet the pressure of cost and time efficiency in product development, every engineer wants to know what can improve the design and to what extent a product is sensitive towards scattering conditions requiring more iterative design studies. Here, MANN+HUMMEL simulation teams see an opportunity of helping designers by running required design optimization studies with optiSLang. In recent times, at least 50 % of simulation projects have been subjected to sensitivity analysis to efficiently explore practical design solutions and improvements. This collaboration between simulation engineers and designers opens up further optimization capa-

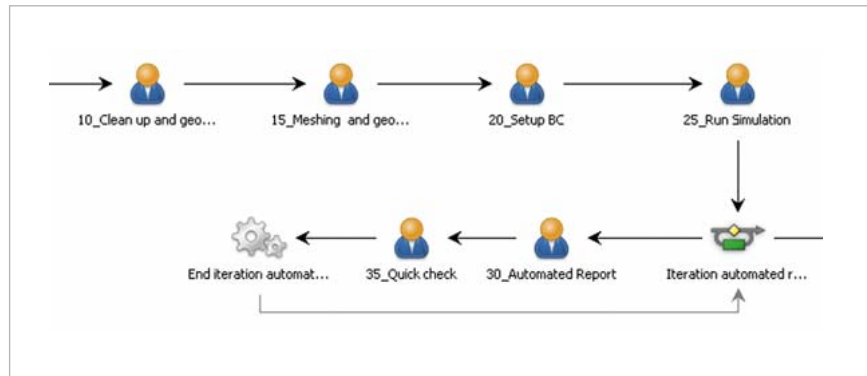


Fig.2: Creation of guided workflows in EKM

Task	Status	Assignee
Project Check	✓ Completed	Employee Name XXX
Materials	✓ Completed	Employee Name XYX
Clean-up and geometry generation	✓ Completed	Employee Name XXX
Meshing	🔄 In Progress	Employee Name XXX
Setup Boundary Conditions	⊖ Not Started	*Meshing

Task: Setup Boundary Conditions

- Workbench-File downloaded from project folder "0_Work"
- (Optional) Symmetry defined
- Support defined
- Loads and/or displacements defined
- Contacts defined
- Materials defined
- Boundary Conditions defined
- Preprocessing strategy document updated
- Workbench-File uploaded as a new version on EKM

Fig.3: Status control of workflows via EKM user interface

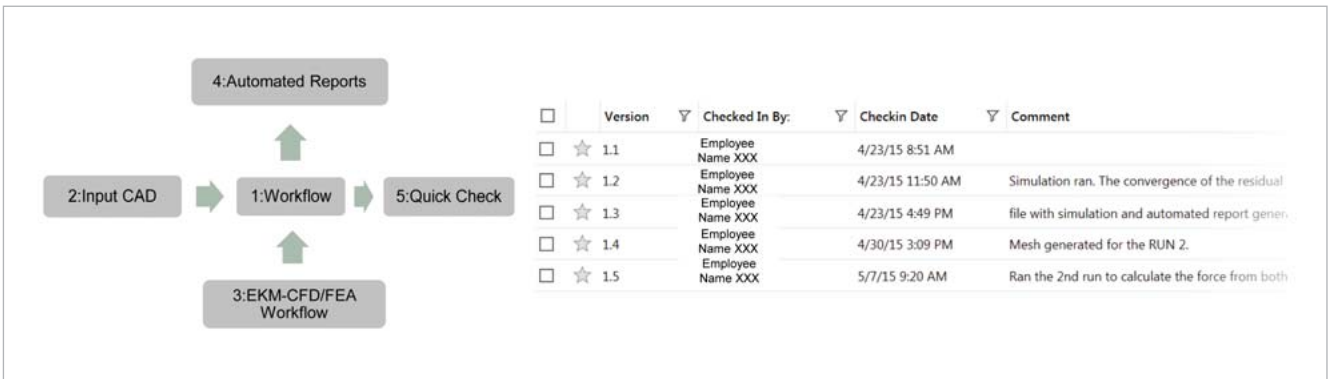


Fig.4: Meta data and version control

bilities by exploring larger design spaces. Furthermore, the resulting technical decisions save time and cost in prototyping.

At Mann+Hummel, simulation teams not only help in virtual product evaluation, they also guide the product development through geometry optimization pointing out what parameters can result in meeting complex design requirements. The output of optimization tools like optiSLang is extremely helpful for making precise design decisions. The sensitivity analysis supports the designers in time-constrained projects to identify efficiently relevant parameters with potential for optimization. The filter element teams at Mann+Hummel handle all simulation projects by using such workflows.

Future prospects

As a conclusion, CAE-based optimization is becoming a mandatory value adding activity in simulation studies and workflow modification. At MANN+HUMMEL, over the years,

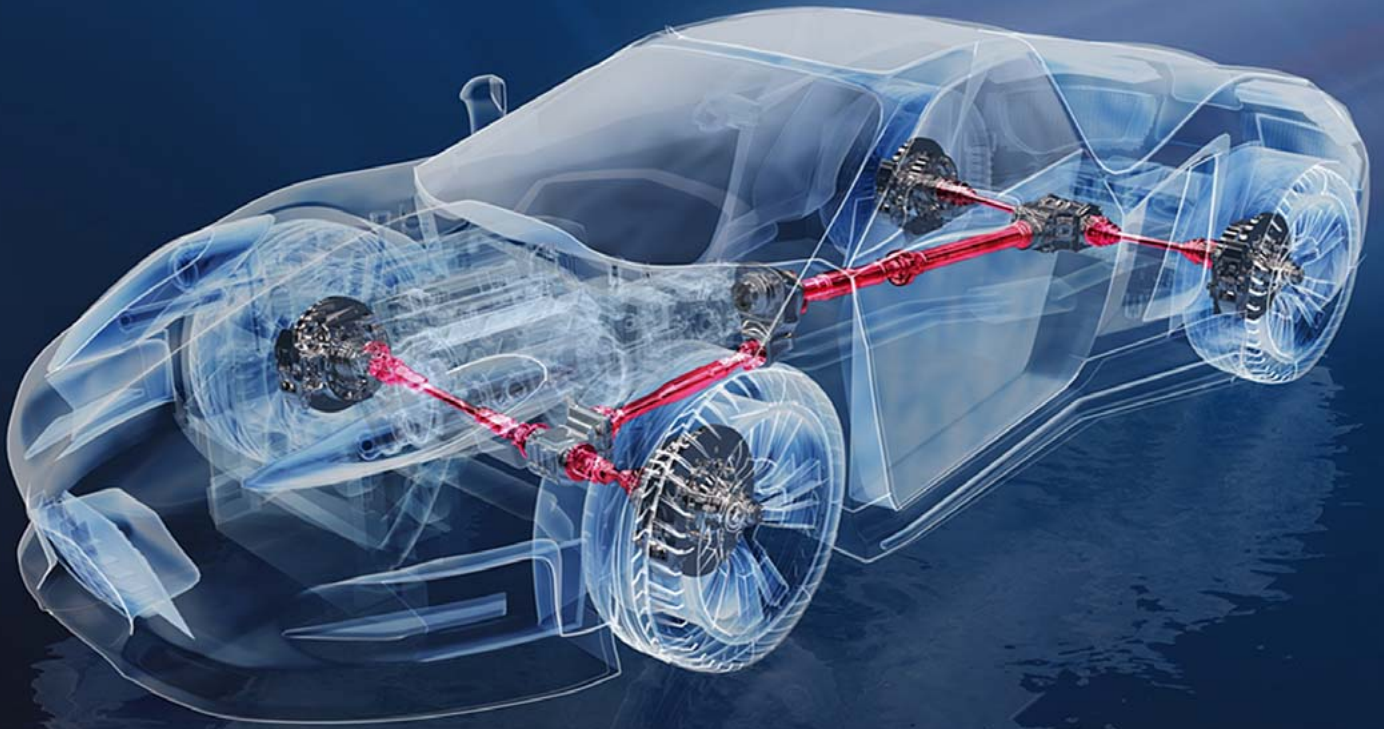
the integration of parametric capabilities of ANSYS Design modeler and ANSYS Mesher has formed a strong base for optimization studies. Here, optiSLang can be used inside ANSYS Workbench to enable strong roadmaps for optimization areas of any kind of CAE simulations. In the future, this will bring about an intensive collaboration between CAD/CAX, simulation, optimization, DOE environments, as well as respective databases. Thus, an iterative design process will be ensured which finalizes the design geometry before it is released back to the CAD database. Completed manufacturing drawings will then be created with emphasis on modified workflows which can be easily implemented by optiSLang.

Author //
Nagendra Prasad (Mann+Hummel)

DYNARDO LIBRARY

Our internet library is an extensive source for your research on CAE topics and CAE-based Robust Design Optimization (RDO).

www.dynardo.de/en/library.html



CUSTOMER STORY // AUTOMOTIVE ENGINEERING

PARAMETER IDENTIFICATION FOR A HYPERELASTIC MATERIAL MODEL OF AN ELASTOMER RUBBER BOOT

In order to develop a reliable model for a rubber boot of a constant velocity (CV) joint, optiSLang was used for parameter identification and direct optimization to fit the simulated stress strain curves to the experimental data.

Introduction

IFA develops and manufactures side and longitudinal drive shafts as well as their corresponding components, such as cross and constant velocity joints. Depending on the field of application, there are different CV joints.

The purpose of these joints is to transmit torque, to bend the shaft and to ensure a length compensation due to the kinematics of the drive train. The advantage of them is that the rotary movement can be uniformly transmitted at all angular positions. CV joints essentially consist of the following steel components: hub, outer ring, bearing cage with balls, the sealing cap as well as the connecting part pin and the sealing component elastomer rubber boot (Fig. 1). The rubber boot is clipped on the pin as well as in the crimp of the sealing cap where it flexibly seals the CV joint. This prevents a contamination from the outside even in bended position or during length compensation. Furthermore, the lubrication of the joint will be ensured. In the case of a positive length compensation, the rubber boot will be exclusively stretched. During a negative length compensation of the joint, the rubber boot rolls off without creasing on the inside of the sealing cap as well as on the pin and the hub. The reduction of the

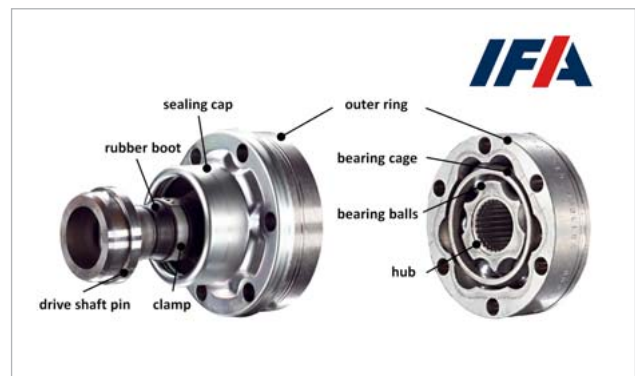


Fig.: 1: CV joint manufactured by IFA

working space simultaneously leads to a pressure increase inside the joint. This may cause an “s-punch shape” deformation of the rubber boot (Fig. 2 see next page). The resulting contact with other components could damage the surface of the boot, eventually leading to its failure and causing even more damage to other joint components. The aim of the simulation is to obtain a reliable prognosis of the deformation of the rubber boot as well as to conduct a leakage test for predefined joint positions.



Fig. 2: "S-punch shaped" deformation (left) and optimal rolling off behavior (right) of the rubber boot

Methodical approach

For the generation of a hyperelastic material model, measurement data from two material tests were used and the corresponding material parameters were determined by curve fitting.

Measurement data and material model

For the curve fitting, tensile as well as shear test data (simple shear tests) were available in the form of multi-hysteresis plots. Thus, on the one hand, the material behavior could be described in a one-dimensional stress state and, on the other hand, in a plane distortion state (Fig. 3). Four amplitudes were tested, each with five cycles (Fig. 4). The arithmetical mean value of the loading and unloading curve of the last cycle with maximum amplitude was chosen as the respective reference curve. It could be assumed that non-elastic effects were minimized with increasing cycle numbers and a quasi-static material state was established.

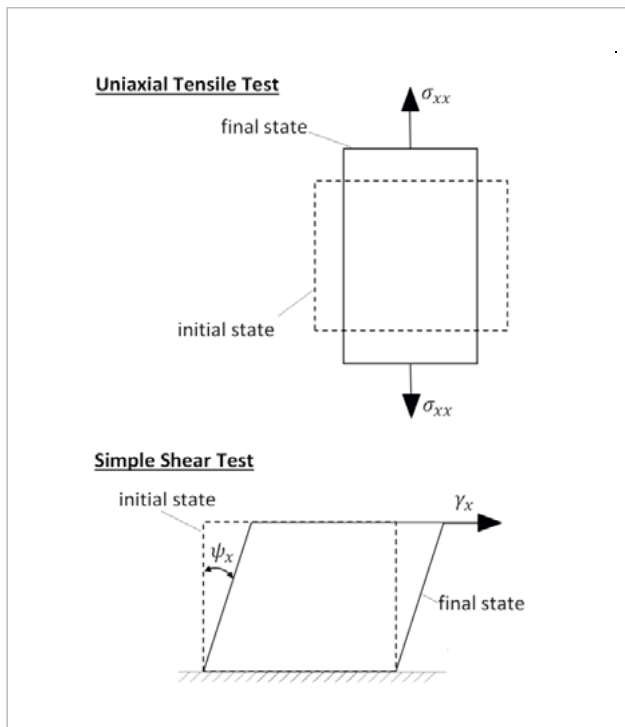


Fig. 3: Schematic diagram of the single-axis tensile test (top) and the simple shear test (bottom)

This project focused on the determination of the material constants μ_i and α_i for the third-order Ogden model. This model describes the distortion energy \widehat{W} as a function of the spatial strains λ_i in the form of equation (1).

$$\widehat{W} = \sum_{i=1}^{\infty} \frac{\mu_i}{\alpha_i} (\lambda_1^{\alpha_i} + \lambda_2^{\alpha_i} + \lambda_3^{\alpha_i} - 3) \quad (1)$$

The assumption of the incompressibility according to equation (2) applies for the volume expansion J (see also [1]).

$$J = \lambda_1 \lambda_2 \lambda_3 = 1 \quad (2)$$

For loading and unloading, an average curve had to be found that passes through the origin point of the coordinate system. In the area of plastic expansion, the curve should have a slope as small as possible. Using one curve, the different material effects could thus be described in a corresponding manner (Fig. 5) because the third-order Ogden material model does not allow to consider inelastic effects directly, such as hysteresis loops and plastic strains.

Parameter identification

The correlation between material parameters and the material behavior occurring during loading was determined inversely using the simulation of the material tests (Fig. 6).

Unfortunately, in ANSYS Workbench version 17.0, there is no option to use experimental test data from a tensile test and a simple shear test in the curve fitting simultaneously.

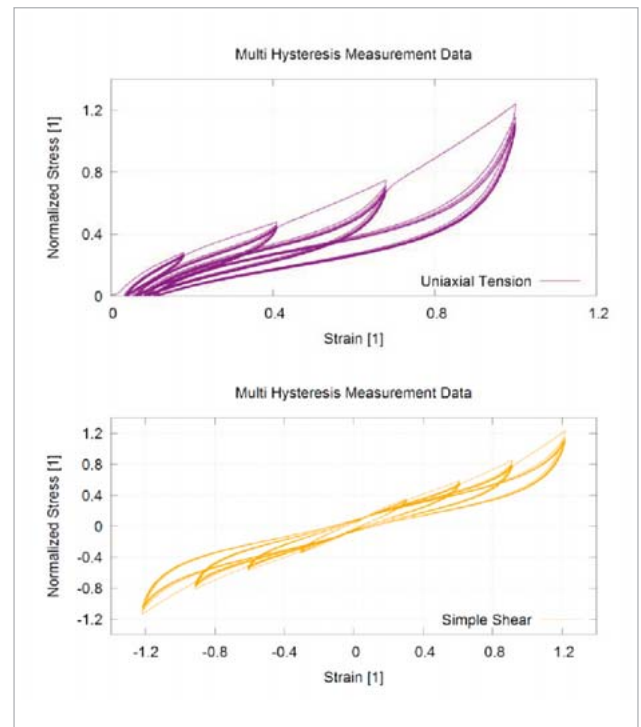


Fig. 4: Measurement data of the tensile test (top) and the simple shear test (bottom) [2]

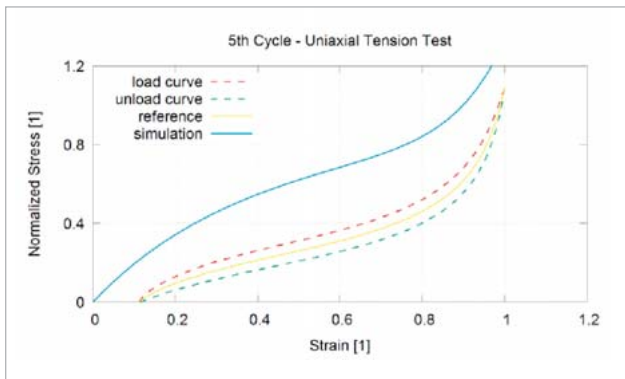


Fig. 5: Hysteresis of the normalized stress-strain curve for the tensile test

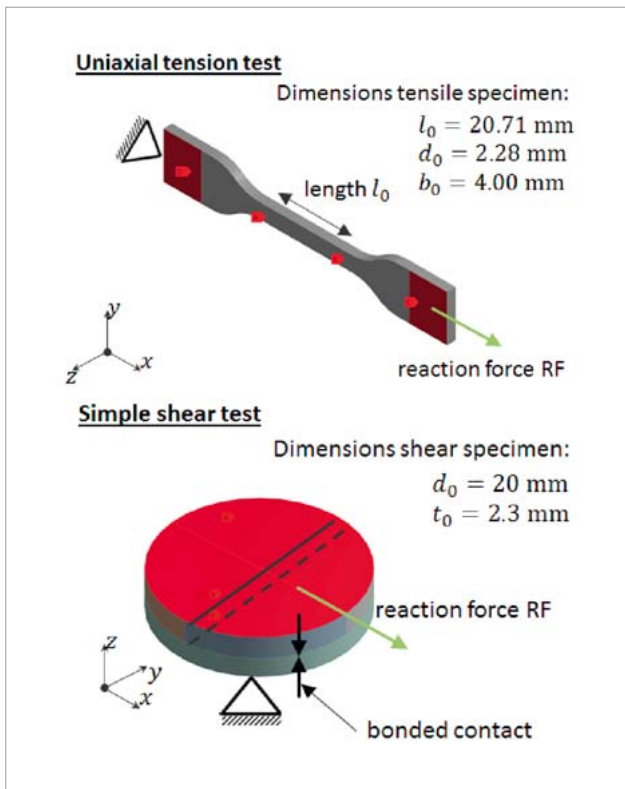


Fig. 6: Simulation models of the material tests for tensile test (top) and simple shear test (bottom)

The sensitivity study and the optimization (curve fitting) were carried out in optiSLang for parameter variation as well as for matching the reference curves with the simulation results.

Sensitivity analysis

The sensitivity study analyzes the influence of the individual parameters on the system behavior and limits their range of values for the subsequent curve fitting (Table 1).

In the parameter setting, the values α_2 and μ_3 were defined as dependent parameters according to (3.1) and (3.2). Thus, an equal distribution of parameter variation over more than one magnitude of the exponent was reached by using an Advanced Latin Hypercube Sampling. As a result, only the value ranges of the exponents were varied.

i	α_i	μ_i	a_i	b_i
1	2 ... 4,5	0 ... 0,3	1	1
2	1	50 ... 100	-2 ... -1,5	1
3	16,5 ... 20,5	1	1	-6 ... -4

Table 1: Parameter range of the sensitivity study

$$\alpha_i = 10^{a_i} \tag{3.1}$$

$$\mu_i = 10^{b_i} \tag{3.2}$$

For a robust physical material behavior, parameters with the same indices should have the same sign in accordance to condition (4). [1]

$$\alpha_i \mu_i > 0 \tag{4}$$

The equidistant discretized curves could be compared by using the Euclidean norm (NRMS) according to equation (5).

$$NRMS = \frac{\sqrt{\frac{1}{n} \sum_{i=1}^m (\sigma_{ref} - \sigma_{sim})^2}}{(\sigma_{max} - \sigma_{min})} \tag{5}$$

The prognosis capability of the simulation models proved to be very sufficient, which was indicated by a high Coefficient of Prognosis (CoP) for the overall model (see Fig. 7, red marking).

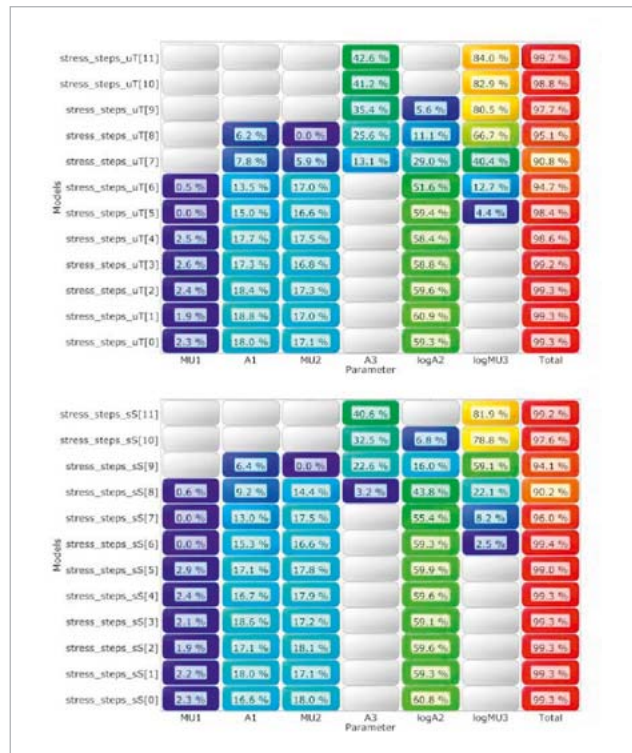


Fig. 7: CoP matrix of discretized signals for tensile test (top) and shear test (bottom)

A coupling of the significant parameters among each other was indicated by the sum of the single CoPs, which showed the influence of the input parameters. The parameters α_2 in the form of $\log A2$ and α_1 , as well as μ_1 had a significant influence of up to 60 percent of the total strain. The remaining 40 percent showed an increasing influence of the parameter μ_3 , in the form of the exponents $\log MU3$ and α_3 . For a subsequent optimization, the parameter range was considered to be sufficiently accurate since stress-strain curves of the calculated parameter variants covered the reference curves (Fig. 8).

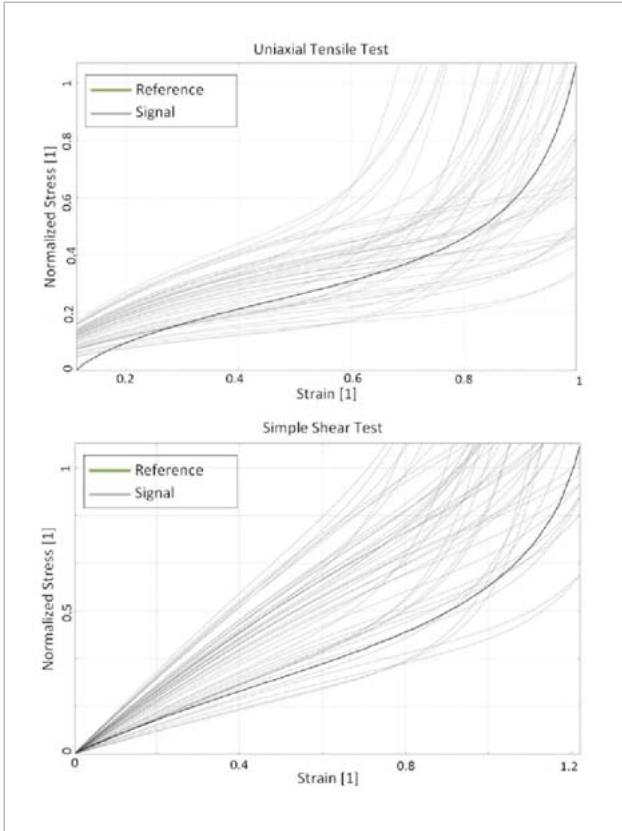


Fig. 8: Stress-strain curves for tensile test (top) and shear test (bottom)

Optimization

The fitting of the stress-strain curves from the material test simulations to the reference curves was defined as an optimization objective. The sum of the individual Euclidean norms per experiment defined the target function and had to be minimized according to equation (7). In contrast to simple shear tests, the measured values of the tensile test can be determined relatively easily with high accuracy and were therefore more weighted in the simulation.

$$\sum_{i=1}^n NRMS = \alpha \cdot NRMS_1 + \beta \cdot NRMS_2 \quad (7.1)$$

$$\sum_{i=1}^n NRMS \rightarrow 0 \quad (7.2)$$

The highly non-linear behavior of the Ogden material model required a direct optimization. Here, the ARSM algorithm (Adaptive Response Surface Method) was used. A comparison of the reference curves with the stress-strain curves of the best design parameter set showed a very accurate correspondence to the tensile test over the entire strain range (Fig. 9). A sufficient coverage of the curves up to a strain of 1 was achieved for the simple shear test.

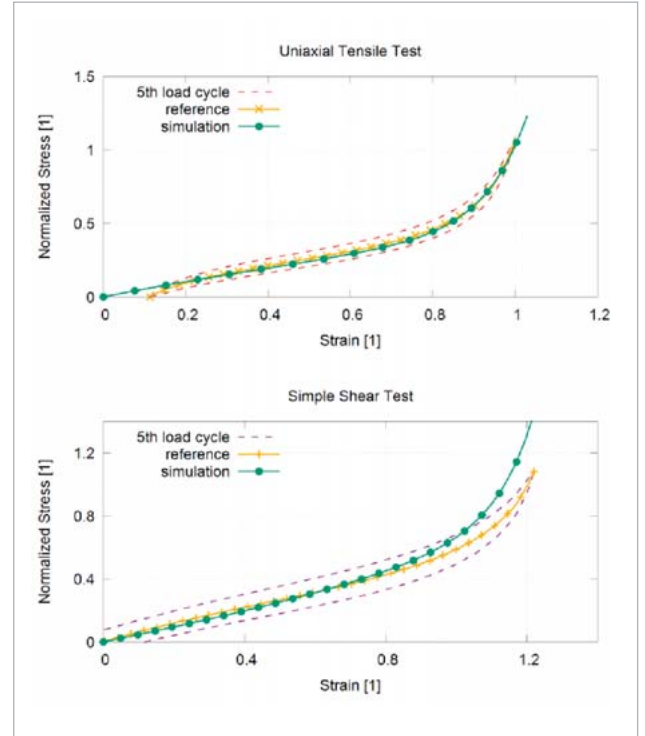


Fig.9: Comparison of the material test simulation with measurement data for tensile test (top) and shear test (bottom)

Axial-symmetrical 2D-simulation of an elastomer rubber boot

The third-order Ogden material model (Fig. 10) could then be used to analyze the component deformation. The aim was

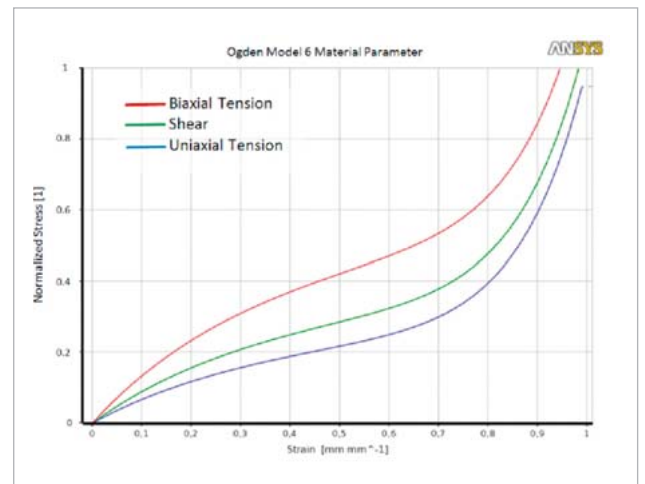


Fig. 10: Normalized stress-strain curves of the tensile (biaxial, uniaxial) and shear components

to obtain a reliable prognosis of potential contact points between the boot and other joint components in order to eliminate damage to the boot surface in operational load cases.

Furthermore, a fluid penetration test was carried out to determine whether the contact pressure on the shaft extension and in the sealing cap could ensure the leak tightness of the joint. For this purpose, a static, two-dimensional and axially symmetrical model of the CV joint was created. For simplification, the rolling elements were dispensed and contact situations between other joint components were replaced by boundary conditions. The non-deformed elastomeric boot (Fig. 11, top) first had to be placed in its installation position (Fig. 11, bottom). The subsequent application of negative length compensation and additional fluid pressure load showed that the boot fixation in the sealing cap and on the pin ensured the tightness of the joint (Fig. 12). Consequently, the sealing cap crimp and the clamp were sufficiently dimensioned to generate the required contact pressure. However, an increasing "s-punch shaped" deformation of the boot could be observed with increasing fluid pressure and length compensation. The deformation was eventually limited by the self-contact of the boot surface (Fig. 13 bottom, see next page).

The maximum major strains (Hencky strains) reached a value of $\varepsilon_{ux}^H = 0.52$. The relation between engineering strains and true strains is given by equation (8), according to [3]. With a strain $\varepsilon = 0.68$, the deformations were within the area of the fitted curves showing a sufficient match with the measured values.

$$\varepsilon_{ux}^H = \ln(\varepsilon + 1) \quad (8)$$

To improve the convergence behavior, a non-linear stabilization method was used to control the dissipation of the element energy [4]. The simulation results of the boot deformations could be sufficiently compared with component deformations in pneumatic experiments, which were used for validation purposes [2].

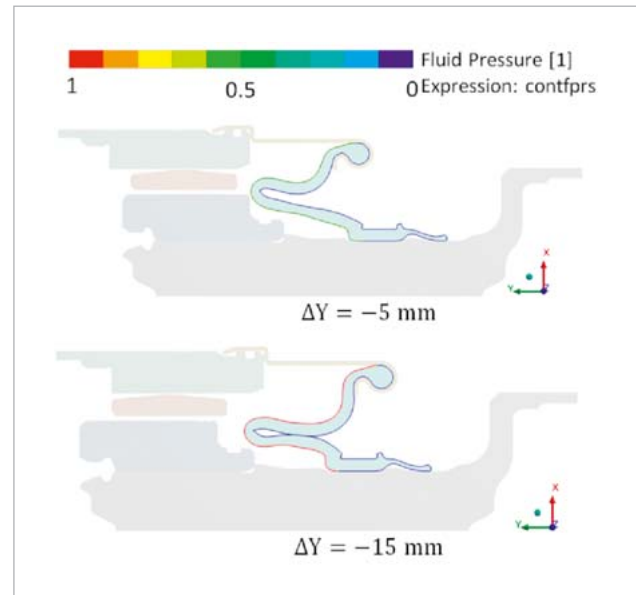


Fig. 12: Fluid pressure as a function of axial length compensation with distinctive s-punch shape (bottom)

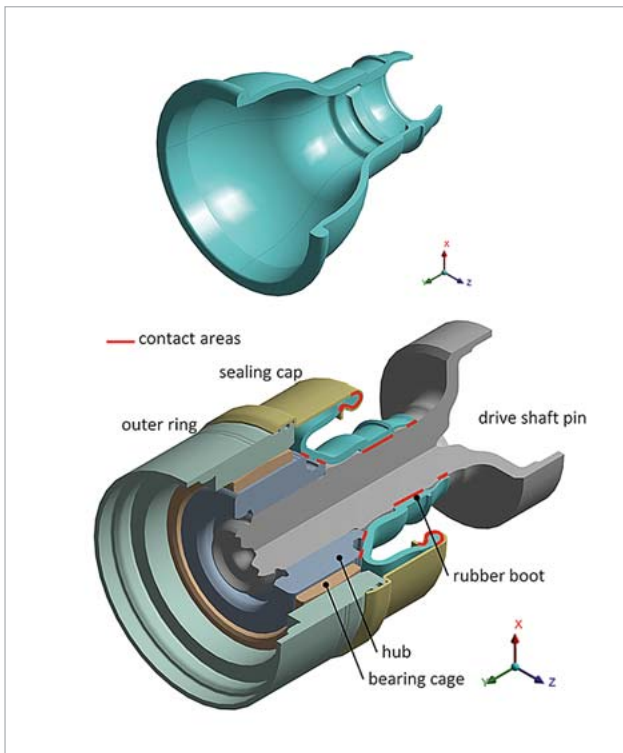


Fig. 11: Boot CAD part (top), 2D-axial-symmetrical model with built-in boot (bottom)

Conclusion

The results of a fluid penetration analysis presented in this article were obtained by using a two dimensional axis symmetric model. The results showed an increased s-punch shaped deformation of the rubber boot caused by increasing a length change compensation of the CV joint and an additional fluid pressure load. Finally, the deformation stopped with self-contact at the boot surface. This result correlated well with pneumatic experiments.

Using optiSLang, tensile test data as well as simple shear test data for the third order Ogden material model could be taken into account. The found parameter set showed a very accurate match for tensile and simple shear test up to a stress of 1. The maximum major strains of this magnitude also corresponded to the analysis of the boot deformation in a two-dimensional, axially symmetrical model. The function and the deformations of the boot could be proven and reliably predicted using a fluid penetration analysis. In a next step following the simulation, the approach shall be transferred to a real three-dimensional joint geometry in order to examine the deformation behavior of the boot during gimbal joint movements.

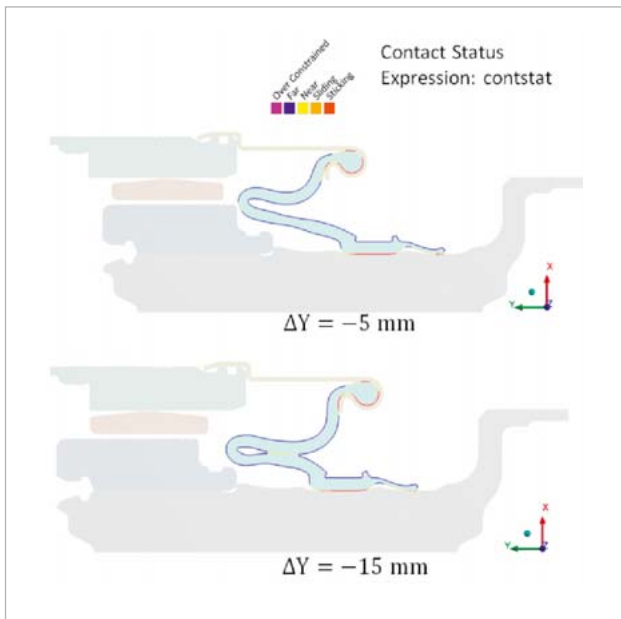


Fig. 13: Contact situation as a function of the axial length compensation and fluid pressure

Authors //

Ingo Grunow, Jean Lefèvre (IFA Composite GmbH)

Sources //

- [1] R. W.Ogden, G. Saccomandi, I. Sagura: Fitting hyperelastic models to experimental data, Computational Mechanics, Springer-Verlag, 2004
- [2] IFA: Interner Prüfbericht, 2015
- [3] H. Mang, G. Hofstetter: Festigkeitslehre, Springer Verlag, 2000 ISBN: 978-3-7091,3762-8
- [4] R. Cisloiu, J. Wang: Understanding Nonlinear Stabilization Features, ANSYS Solutions Magazine Volume 7, Issue , 2006

Getting Started

Sensitivity analysis, multiobjective and multidisciplinary optimization, robustness evaluation, reliability analysis, model calibration and Robust Design Optimization

optiSLang SoS



Info-Events

Webinar Series | In short introductory webinars, we will explain the added value of our software products in practical application examples from various areas of engineering.

Info Day | During an information day, you will get a comprehensive overview and you will have the opportunity to get in direct contact with our consultants in order to discuss specific tasks.

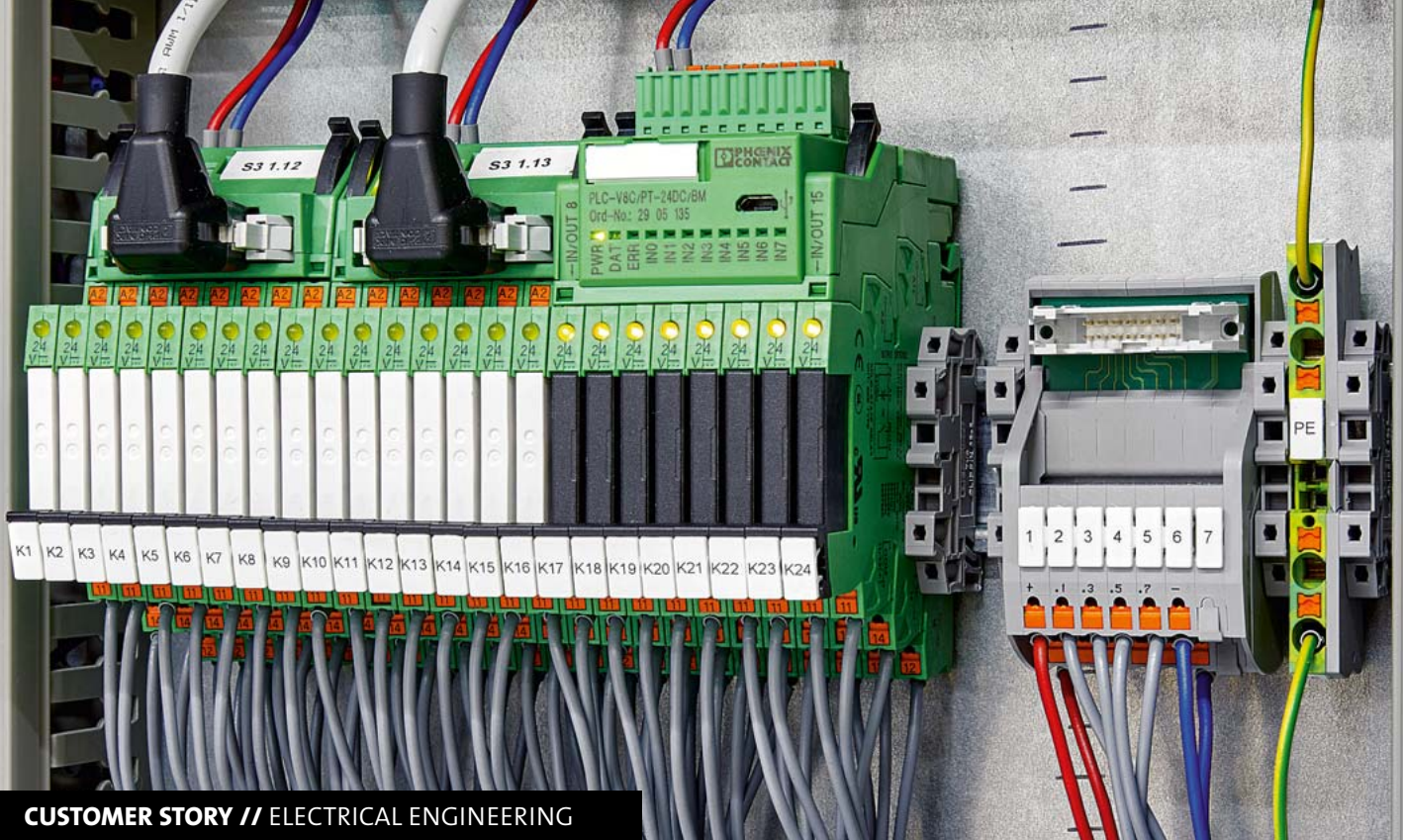
Introductory Training

During **one-day introductory courses** or various **E-learning units** we will acquaint you with the application of our software products by means of illustrative examples.

More ways for getting started are our offers for

Pilot Projects or Advanced Training

Please visit www.dynardo.de for detailed Information.



CUSTOMER STORY // ELECTRICAL ENGINEERING

METAMODELS IN A CYBER-PHYSICAL SYSTEM

In order to determine the remaining lifetime of safety relays, simulation models of the whole system are combined with the real data of an operating elementary relay in a cyber-physical system. Phoenix Contact Electronics GmbH applies optiSlang for parameter identification by means of metamodels used as Functional Mockup Units (FMUs).

Introduction

Safety relays are applied in many fields and industrial environments, including:

- Two-hand control
- Emergency stop
- Control of safety doors

Essential components of many safety relays are forcibly guided contacts according to EN 61810. The operational characteristics of these components enable the analysis of failure evolution. For example, the relay failed due to abrasion of the load contacts caused by its operating function. Such a malfunction would not basically lead to immediate hazardous situations because of redundant arrangements and other internal safety functions. However, the system is switched off and no longer available. If the failure occurs unexpectedly, subsequent disconnections can lead to further malfunctions. The failure of a single component, e.g., in a chained automated process, could interrupt the entire production line. Furthermore, a failure might result in compensation costs for contractually guaranteed services, e.g., in the operation of wind turbines. It is therefore advisable to detect and track

the possibility of an impending failure as accurately as possible for on time replacement of worn components within planned maintenance and operating cycles.

The characteristics of a relay in a switchgear can be described by means of system simulation. The involved physical domains of mechanics, electromagnetics and temperature are considered by using the Finite Element Method (FEM). The derived models of reduced order are transferred into the system simulation in order to be coupled on the system level.

The behavior of the firmware (software) of the switchgear is considered on the system level using a Functional Mockup Unit (FMU). The empirical models, e.g., contact resistance or arc voltage, are displayed in function blocks using the programming language VHDL-AMS. Until now, it has been problematic to integrate the wear behavior of the operating electrical contacts of the relay because an accurate knowledge about load and environmental conditions is often not available. Thus, the prognosis of the system's durability has to be based on statistical test data considering the most critical load and environmental scenarios. This results in a very conservative lifetime estimation. Another problem is

the measurement of the mechanical characteristic values of the operating relay inside the device. These are usually measured on open and freely accessible relays, although, in operation, the relay is encapsulated. These two issues of existing system simulation models need to be considered for an accurate tracking of the lifetime during operation.

In a cyber-physical system, the relay is coupled with a simulation model for data exchange. Based on this data, the device is then analyzed using a simulation model. The results of this analysis are used in the cyber-physical system to predict and track product conditions.

Terms and Definitions of a cyber-physical system

A cyber-physical system is a network of informatic and mechatronic parts connected via data infrastructure (VDI 2013). In the reference architecture industry 4.0 (RAMI 4.0), cyber-physical systems are defined as follows (BMW 2016, IEC 62890): implementation of a (standardized) communication and system infrastructure with required management and productive services as well as with defined QoS (Quality of Service) properties as the basis for an efficient setup and integration of I4.0 systems in an application domain. Therefore, cyber-physical systems are applicable in all areas of the value chain, e.g., marketing, product development, testing, production, sales, customer support, operation, predictive maintenance or return management. For example, a typical application is within a “smart grid”, i.e., systems of power generation are coupled via IT networks for functional optimization. Another example is the communication via “Car-to-X”, i.e., between a network of vehicles as well as central systems, such as manufacturers or transport infrastructure (VDI 2013). The cyber-physical system in this presentation deals with the coupling of measurements of product operation conditions and lifetime estimation for predictive maintenance and optimal operation.

Objectives, Methodology

The aim of the present study is to determine the contact lifetime of the relay NSR01 from Phoenix Contact Electronics GmbH. The analysis is conducted by using a simulation model of a safety relay. The resulting information is provided to the customer and allows the prognosis of malfunctions.

The implementation is achieved by linking a simulation model of the switchgear with the real operating relay by means of a data interface. For this purpose, a simulation model of the entire connector is created. A data interface serves as a connection for transmitting the measured characteristic values to the simulation model. The model is typically derived from simulation models generated during the development of the relay. The software Simplorer (Electronics) from ANSYS Inc. is used for generating the system simulation model. Since the transient coupling of FEM models is

very complex causing a high amount of computation time, such real-time system models coupled to a device are not practical. Therefore, the simulation model considers various physical domains represented by individually derived reduced order models, which are coupled on the system level. In addition, characteristics, e.g., contact resistance or arc voltage, are included by experimentally determined empirical correlations.

The electrical lifetime of the relays is determined under laboratory conditions for different load ranges as a function of the relay parameters and other environmental conditions, e.g., temperature, operating voltage or installation. Based on the measurements, Metamodels of Optimal Prognosis (MOP) are generated using Dynardo's optiSLang. They represent the contact life in the simulation model regarding the respective load areas. The mechanical parameters of the relay in the switchgear are determined by measuring the magnetic field and the coil current. Again, MOPs are generated to calculate the mechanical parameter based on magnetic field measurements. The results, together with other measured parameters, are used to predict the remaining lifetime of the relay (Fig. 1).

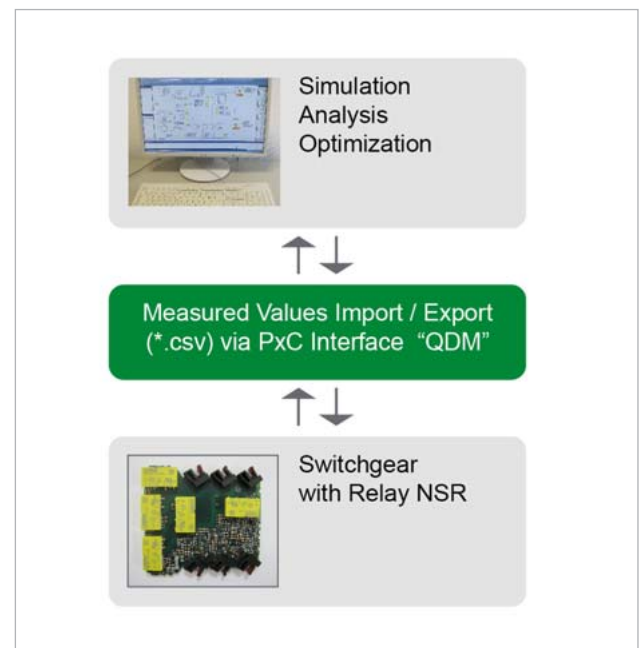


Fig. 1: Linking a simulation model to an operating relay NSR

Relay corresponding physical domains and simulation methods

Mechanics

The mechanical parameters (Fig. 2) are simulated using FEM (ANSYS). The motion ratios as well as force and stress conditions can thus be determined for the various mechanical components of a relay, e.g., contact springs, anchor bearing springs, holding springs, return springs, or transmission ele-

ments. This is achieved by using static and transient (time-dependent) simulation. The transfer of the motion ratios (displacement-time) into a reduced order model is implemented by transforming the eigenfrequency modes of the points (nodes) in a matrix (space state matrices) and the following export via spm file. Here, only linear correlations can be transformed and transferred. For modeling the complex mechanical system of a relay, it is divided into individual linear parts, which become connected with corresponding coupling elements on the system level.

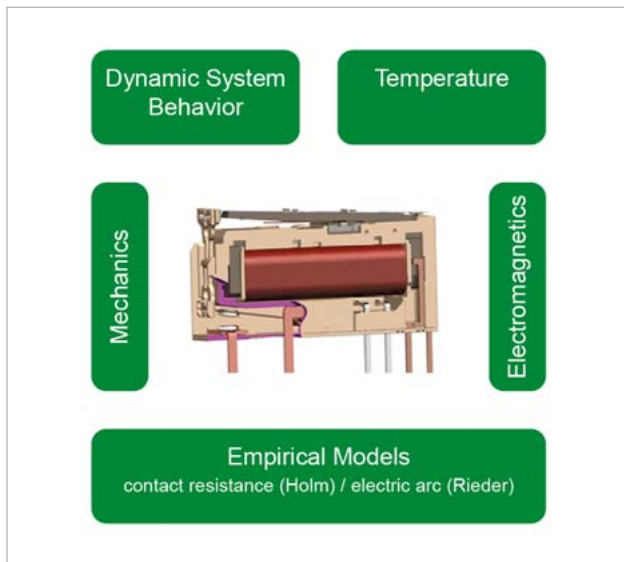


Fig. 2: Physical domains of a relay

Electromagnetics

The simulation of the electromagnetics (Fig. 2) is also conducted by using FEM. The coupling of the complex FEM models can be dispensed if parasitic effects, e.g., eddy currents or iron losses, play only a subordinate role. This is the case if small electromechanical relays are regarded. Instead of coupling the FEM models by co-simulation, the generalized forces from the coupling of the magnetic co-energy and the interlinked flows are simulated using the FEM simulation of the magnetic systems. In this case, the torques and loads of the system are considered in a characteristic diagram including various working points, e.g., rotation angle, stroke or electrical excitation using the dissipation of the magnetic co-energy via angle and stroke. In this characteristic diagram (ECE model), the transmission behavior of the magnetic system is interpolated between the operating points. By linking the magnetic co-energy with the mechanical part by using linked flux, the mechanical feedback effects in regard to the magnetic-electric system are also taken into account.

Temperature

The thermal correlations in the relay can be represented by analytical formulas in combination with measured values. At the system level, thermal correlations are modeled by using numerical blocks.

Dynamics / System Behavior

The dynamics of the relay are considered in the simulation model by transient coupling of the individual subsystems, e.g., mechanical, magnetic or thermal subsystems.

Contact Resistance, Electric Arc

For some characteristic values of relays, e.g., contact resistance of the operating contact or electric arc of the opening contact, empirically determined formulas (Holm, Rieder) are used (Fig. 2). This allows the derivation of electrical characteristics, e.g., current, voltage, of the load circuit as a function of current mechanical values.

Contact resistance according to Holm

$$Rk = 280r\sqrt{E/(Fk - r)} \quad (1)$$

Arc voltage according to Rieder

$$U = s^{1/1.57} * 0.00385^{-1/1.57} * I^{-0.49/1.57} + U_M \quad (2)$$

Contact Lifetime

Normally, for the determination of contact lifetime, characteristic values measured during continuous tests at proper facilities are considered. Here, the failure behavior is determined from a number of tested relays with scattering characteristics and corresponding to the expected basic population of the relays. Usually, a characteristic value of the service life is chosen which represents ten percent failure among the tested relays under these particular conditions (B10-value), see Fig. 3.

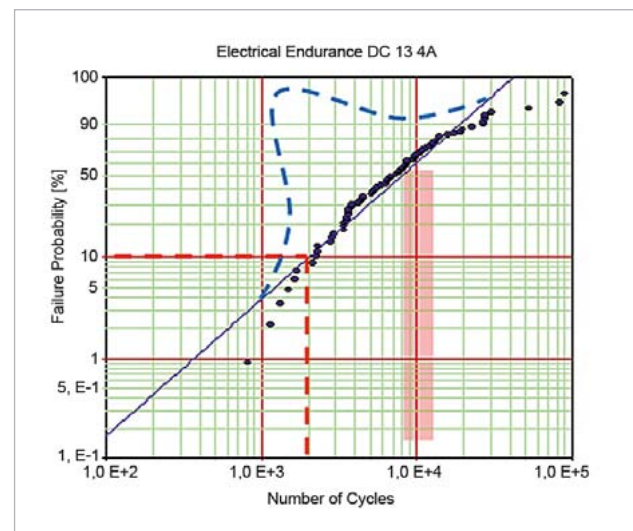


Fig. 3: Contact life of operating contacts – Weibull Diagram

However, the statistical uncertainty in estimating lifetime of a certain relay is a disadvantage of this method. In addition, replacing the relay at 10 percent failure probability is very conservative. Therefore, a more accurate estimation of contact lifetime can be achieved by using metamodels (MOPs)

based on test results. This includes the measurements of the most important electrical and mechanical relay values under laboratory conditions depending on the relay parameters as well as on the load and environmental conditions (Fig.4).

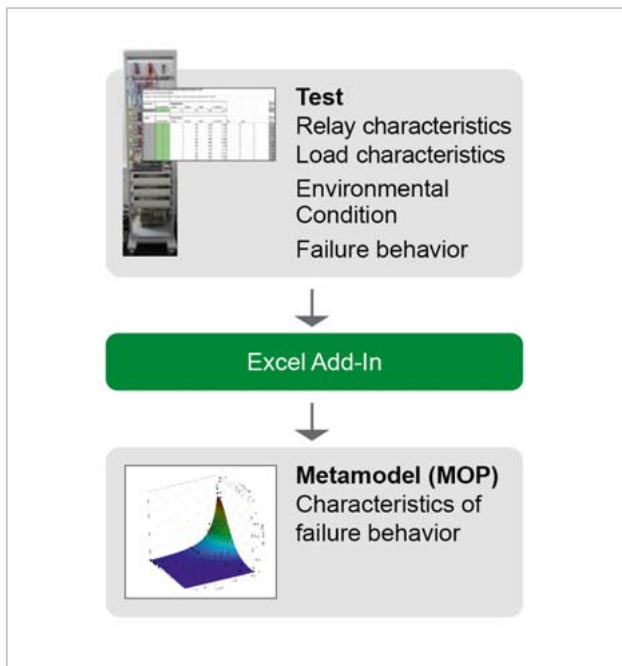


Fig 4: Consideration of experimentally measured contact lifetime parameter using metamodels (characteristics)

These metamodels are used as an FMU in the system simulation model. Since the contact lifetime varies in behavior and dependencies regarding different load types, e.g., ohmic, inductive, capacitive, direct current and alternating voltage, as well as for different load ranges, e.g., small/large current/voltage, the currently valid load range and load type has to be identified and scanned by experiments. This identification is required during the switching on and off operation. The relevant characteristics can thus be selected and activated based on the load profile.

The load identification (Fig. 5) is carried out by transforming the measured signals. Here, on the one hand, capacitive loads, e.g., current peak when activated, are modeled, e.g., by fourier transformations. On the other hand, inductive loads, e.g., slow current rise while activating, are identified, e.g., by Laplace transformations. As an advantage of this method, no signals but transformation coefficients have to be used for the classification of characteristics. This leads to a considerable reduction of calculation effort.

Mechanical Characteristics

The mechanical characteristic values of relays are typically measured under laboratory conditions on open devices (Fig. 6). Because of the harsh operational conditions, relays are usually encapsulated. This has to be considered when calculating the mechanical in regard to the magnetic field.

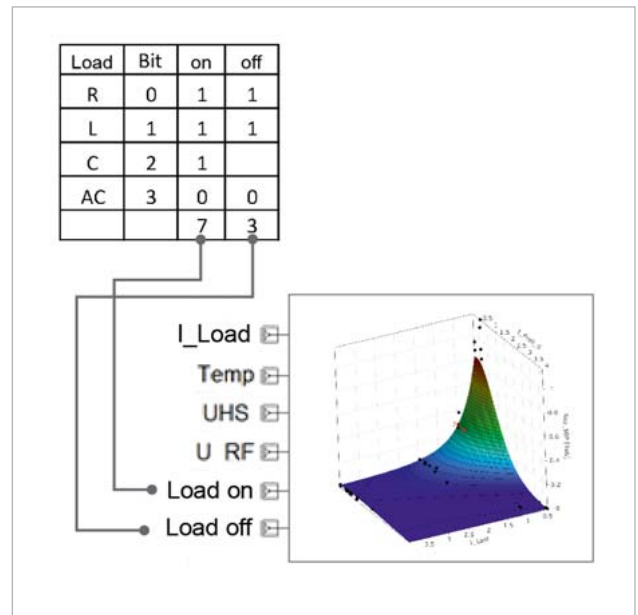


Fig 5: Load types and load area classification



Fig 6: Measurement of mechanical characteristics under laboratory conditions on an open relay

The solution approach has to consider whether the magnetic field, generated by the magnetic circuit, and the anchor position show a correlation. Also a reversed dependence on the measured magnetic field regarding the determination of the position and the rotation angle of the anchor should be observed. The evaluation of the simulation results regarding the magnetic system indicates positions near the relay with a strongly changed magnetic field depending on the anchor position (Fig. 7).

The simulation results of the magnetic flux density B (x, y, z-direction) as a function of different anchor positions (angle of rotation) and coil excitations are converted via Excel interface in optiSLang's binary database. Strong input correlations were found between the excitation current of the coil and the flux densities Bx1 and Bz1(Fig. 8).

The reduction of the strongly correlated input parameters finally results in an MOP with very high prognosis quality regarding the variation of flux densities (Coefficient of Prognosis - CoP of 99 percent). In order to solve the task,

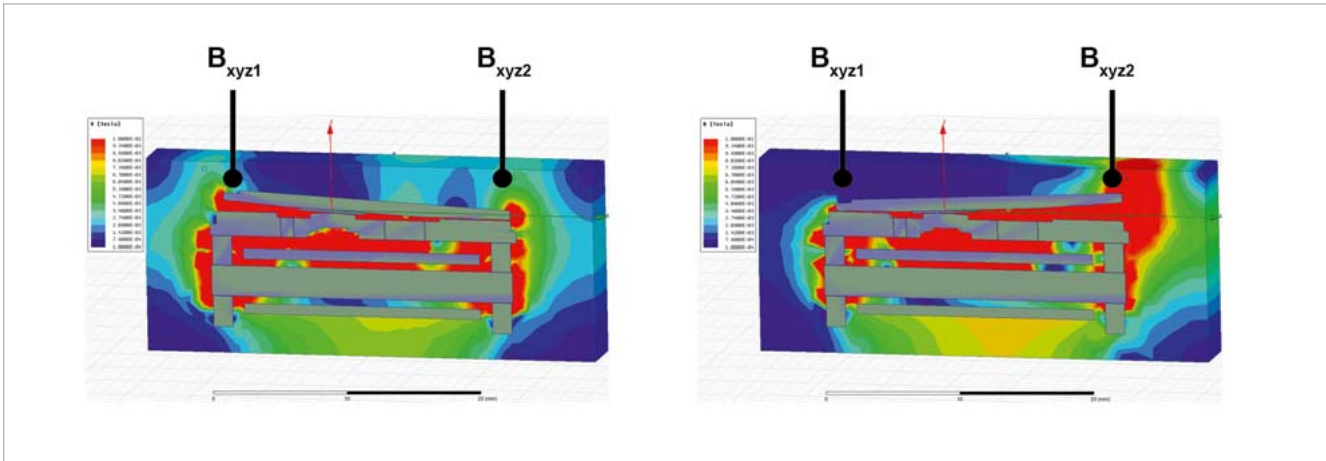


Fig. 7: Magnetic field at different armature positions

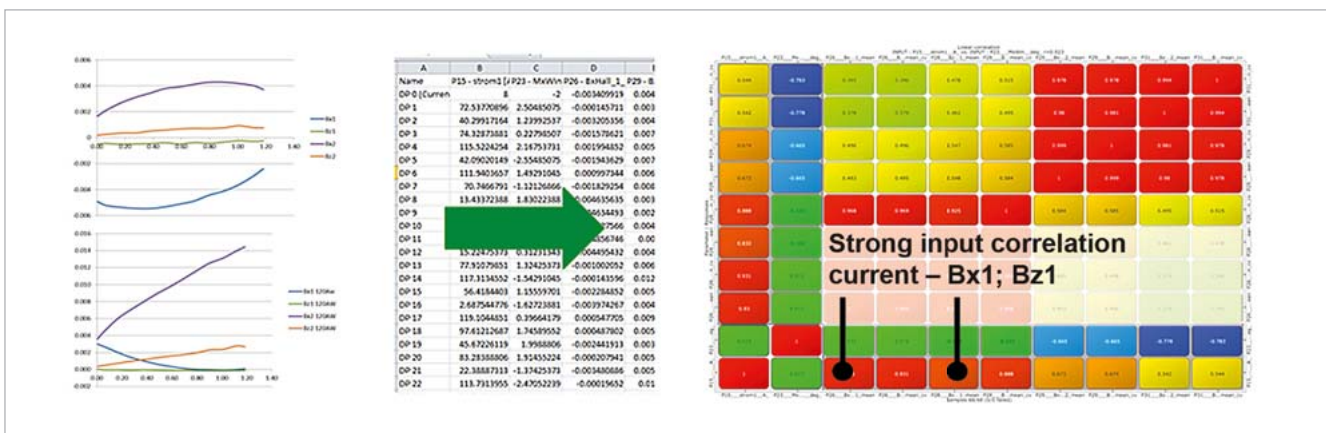


Fig. 8: Simulation and MOP_{Bxyz(1,2)}} = f(Mxwin; current)

the inversion to mechanical parameters needs to produce unique results. Fig. 9 shows the input MOP and Fig. 10 (see next page) the inverted MOP.

The inversion shown in Fig. 10 (see next page) produces unique results. However, the extrapolation within the MOP beyond technically meaningful limits of the rotation angle is a problem. This was solved by conducting a sensitivity analysis with constraints. The resulting MOP is fitted and, thus, capable of representing the anchor position as a function of the magnetic flux density and the excitation current of the coil within meaningful physical limits (Fig. 11, see next page).

Fig.12 (see next page) shows the derivation of different MOPs with varied measurement values of the flux density, i.e., with different effort regarding the number of sensors. Accordingly, under the given conditions and with only one magnetic sensor, it is possible to measure a flux density value and a value of the excitation current of the coil showing a CoP of 93 percent.

Electronics

The electronics simulation is conducted in the environment of the system simulation within the electrical domain. Extensive libraries and algorithms are available for this analysis.

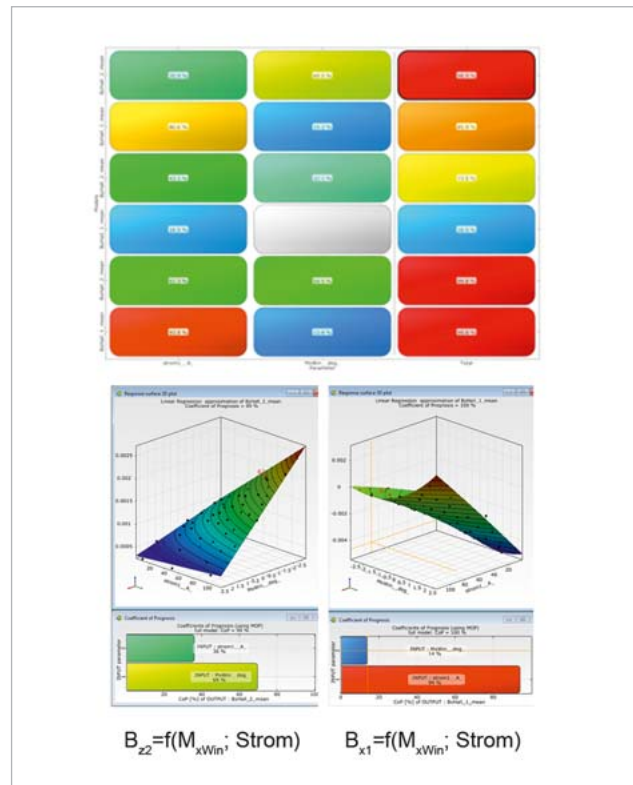


Fig 9: CoP Matrix and MOP without strongly correlated input parameters

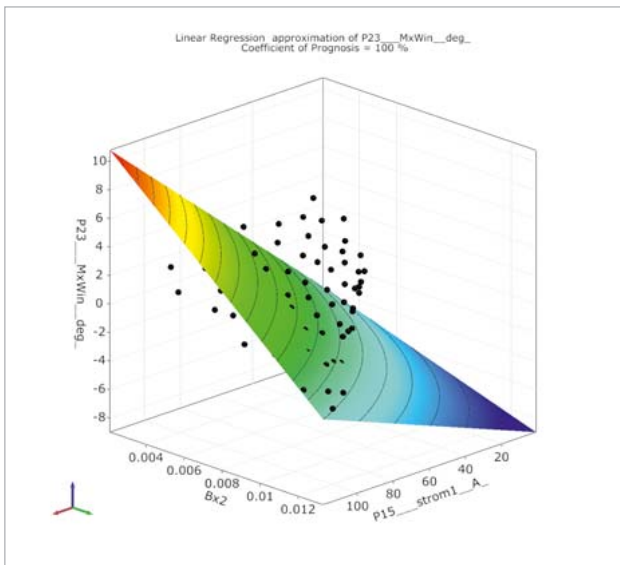


Fig 10: Inversion $M_{xwin}=f(B_{x1222})$; current

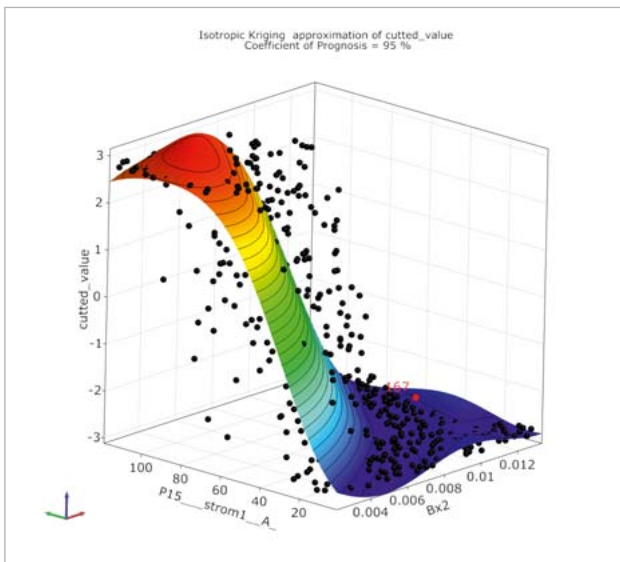


Fig 11: MOP with fitted range of values, MOP: $M_{xwin}=f(B_{x1222})$; current, CoP=96 percent

Firmware

The firmware used in the operational relay is generated with the help of an FMU and is represented in the simulation model using the program module Scade from Ansys Inc.

Simulation model of the relay including meta-models for contact lifetime and mechanical parameters (anchor position)

Fig. 13 schematically shows the block diagram of the simulation model with the objects of various physical domains. The parameters of the relay are transmitted via Simplorer blocks, which provide the measured values as internal parameters. The simulation model of the relay includes all necessary physical domains. Elements of ECE (electromagnetics), spm (mechanics) and Simplorer coupling elements

Nr.	Signal Parameter	CoP	Extrapol.	Signals	Sensors
1	$B_{x1x2z1z2}$ Strom	100	2	5	2
2	B_{x1x2z2} Strom	100	2	4	2
3	B_{x2-x1} Strom	69	1	3	2
4	B_{x2z2} Strom	98	1	3	1
5	B_{z2-x1} Strom	97	1	3	2
6	B_{x2} Strom	96	1	2	1
7	B_{x1x2} Strom	99	(2)	3	2

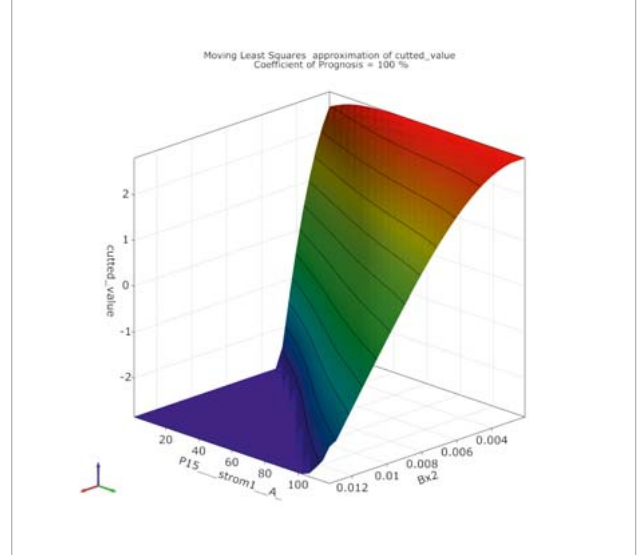


Fig 12: Overview of signal parameters and MOP with a flux density value

were used for the modeling of the relay NSR01. The Simplorer model is connected to the relay via a QDM interface. The communication is carried out by means of a PC interface. On demand, it transmits measured values of the relay to the model for simulating the relay behavior.

Special relay characteristic values and properties, e.g., arc behavior, contact resistance or coil heating, are simulated via VHDL function blocks. The corresponding lifetime characteristic and the anchor position is calculated with the help of MOPs depending on the determined load range. Based on the measured and simulated parameters, the remaining contact life is calculated and displayed via a PC interface. For the different references, please refer to the indicated formatting.

Demonstrator of the safety relay with firmware, electronic components, electromechanical components and load

Demonstrator

The demonstrator of the safety relay is integrated on a test board with an ARM Cortex M0 CPU. In terms of the hardware architecture, this represents real operational conditions. Thus, parts of the firmware could be adopted. The functionality corresponds to “emergency off”, where the load is immediately switched off when receiving a certain

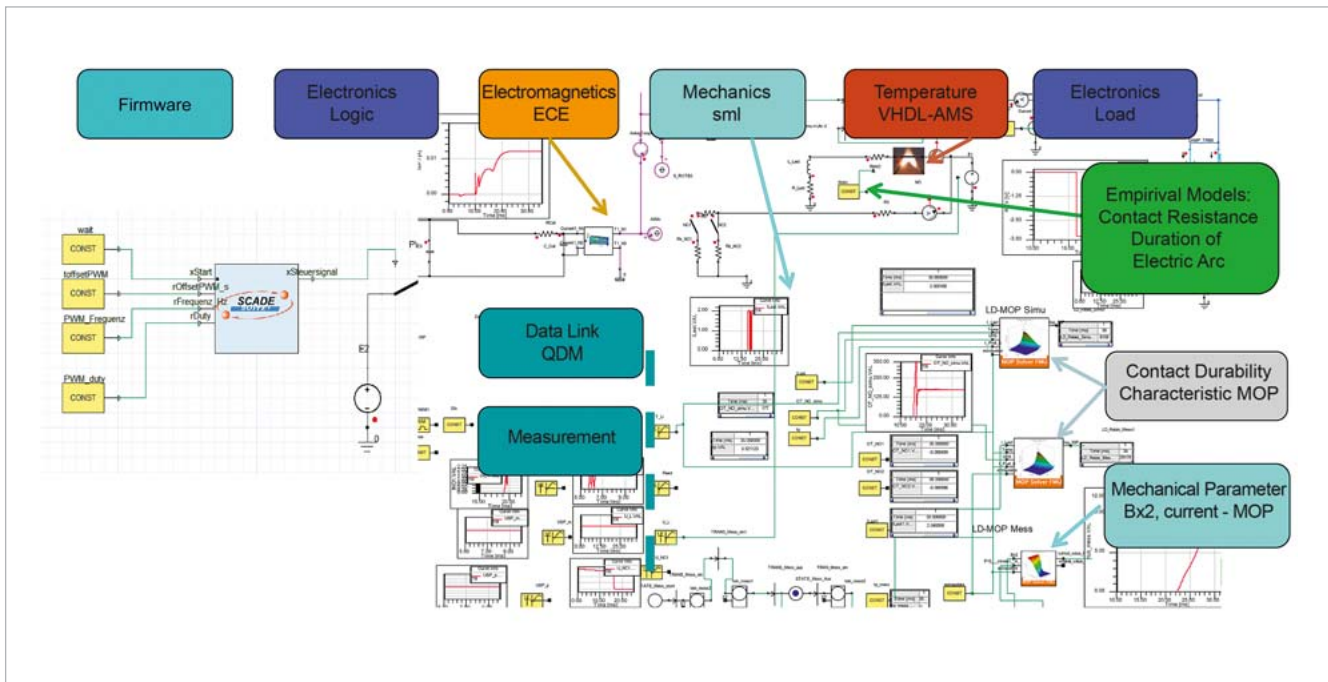


Fig. 13: Simulation model of the relay with metamodel of contact life and anchor position

sensor signal. The reactivation is achieved by means of separate sequences in order to prevent an unintentional restart. The data is transferred via a serial interface to the PC, which also runs the simulation model of the relay. The analog values of the relay and the load are measured by using the AD converter of the board.

Operational case of switch-on relay chatter: danger of contact welding

For the load case of switch-on contact chatter, which has a strong influence on the service life (welding of the contact), the switch-on load current is measured on the demonstrator and the signal is transferred to the simulation model. The analysis of the load current and the calculation of the switch-on contact chatter duration (Prelldauer t_p) is conducted with the simulation model. Using the value of the chatter time, further characteristics of the load (current, voltage) and the characteristic values of the relay, the expected service life up to failure mode is calculated with the load corresponding lifetime characteristic using the MOPs.

For this case, the simulation shows the following values:

Chatter time t_p (ms)	switches up to failure mode
1.592	297
0.826	29343

Conclusion and Prospects

The basic functional verification of a cyber-physical relay model by means of a demonstrator could be conducted. The simulation model was successfully coupled with a safety relay device for transmission of measured values from the

device to the model. The analysis of status and behavior of the safety relay was also implemented by using the simulation model.

In further development steps, the coupling of the hardware with the simulation model should be improved in order to implement a real-time analysis. In addition, a more precise contact model and the consideration of the load circuit feedback can make a contribution to further enhancement of the relay model.

Author //

Ralf Hoffmann (Phoenix Contact Electronics GmbH)

Sources //

- VDI/VDE Gesellschaft: Thesen und Handlungsfelder; Cyber-Physical Systems; Chancen und Nutzen aus Sicht der Automation, April 2013
- CADFEM: Seminar Systemsimulation , <http://www.cadferm.de/wissen/seminare/ansys-systemsimulation/S/13655.html>
- CADFEM: Seminar Feldberechnung Advanced
- Holm, Ragnar: Die Technische Physik der elektrischen Kontakte, Springer-Verlag Berlin Heidelberg GmbH, 1941
- Riede, W.: Plasma und Lichtbogen, 1967 Vieweg, Braunschweig
- Brauer, J., R.: Magnetic Actuators and Sensors, IEEE Press Wiley 2014
- Vinaricky, E.: Elektrische Kontakte, Werkstoffe und Anwendungen, Springer Berlin-Heidelberg GmbH 2002
- Thomas Most, Johannes Will (Dynardo GmbH): News optislang Algorithms (MOP, Cop, Sensitivity Analysis, Optimization, Reliability Evaluation) WOST 2012



CUSTOMER STORY // TURBO MACHINERY

ANALYSIS OF LOW CYCLE FATIGUE CONSIDERING GEOMETRIC MANUFACTURING TOLERANCES

SIEMENS AG applies ANSYS, Statistics on Structures and optiSLang for probabilistic analyses of geometric variations and their influence on the fatigue behavior of a gas turbine housing.

Motivation

SIEMENS AG LGT manufactures large scale gas turbines for power generation, e.g., the gas turbine SGT5-8000H with 400 MW or CCPP up to 600 MW. The title image shows such a turbine during its assembly. These gas turbines are known for their high efficiency (GT with 40% and GUD even more than 60%). Customers look for power generation units, which are efficient, reliable, available, flexible and cost-effective. The competition with renewable energies imposes strong goals onto power plants to improve the cost effectiveness of their turbines. Technically, this leads to the questions if one can re-define the magnitude of safety factors and lifetime, change maintenance instructions, increase check intervals, or simply find new technical solutions. In any way, the optimization of designs and maintenance cycles will continue towards the limits of product performance. Safety factors are typically applied to material properties (e.g., scatter of properties, distribution in space, scatter in fatigue curve), boundary conditions (e.g., loading, environment parameters, operational parameters) and geometric variations (e.g., manufacturing tolerances). The interaction of these parameters, however, can only be considered by a probabilistic approach.

This article presents a strategy for the probabilistic analysis of geometric variations and their influence on the fatigue behavior of a gas turbine housing. The task is to quantify the influence of the geometric scatter onto stresses and lifetime. This can be done by estimating statistical properties and translating them into failure probabilities. The knowledge can help to adjust safety factors and the duration between maintenance actions or can improve the quality control for manufacturing tolerances.

The workflow of the analyses starts with obtaining knowledge on the real geometric deviations with respect to the target CAD geometry after production. An accurate measurement of the true surface can be done through laser scans. Scans of multiple designs are used to create a statistical model for the geometric variations. Subsequently, the model is capable of generating new virtual random geometries. These serve as input in a CAE analysis. A Monte Carlo-like sampling can be finally used to predict the statistical properties of response quantities, such as stress or durability factors.

Workflow steps

1. Measurements

The generation of the surface laser scans is the first challenge because of the housing size of 4.9 m in diameter, 13.1 m in length and 390 t in weight. Current scanning technology creates large data volumes (more than 1 GB data to store the triangulation of one 180° scan). Several scanner positions are required due to the size of the object. The photogrammetric system GOM Tripod in combination with GOM ATOS Triple Scan was used to support these measurements (Fig. 1).

The possible time to scan the turbine is limited between production and preparation of delivery. Therefore, for some turbine housings only parts of the surface could be scanned. Due to the complex shape of the geometry, also some parts are not accessible by scanning devices. Thus, the triangulation must be edited afterwards, i.e., repair meshes, de-feature, fill holes, remove outliers, reduce number of triangles (see post processing in Fig. 2). The long delivery lead times also cause that only a few turbine housings could be measured within an acceptable project time. Therefore, a very small number of samples is typically available. By taking the symmetry of the geometry into account, a larger number of samples can be obtained virtually.



Fig. 1: Measurement of true geometry using laser scans

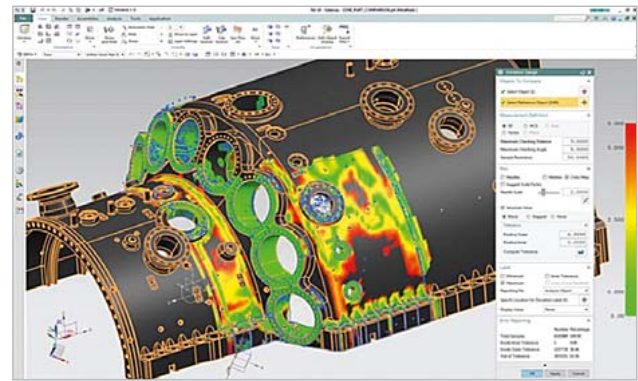


Fig. 2: Post processing of laser scan data in SIEMENS NX

2. Statistical model of geometric tolerances

New random virtual geometries are created in the robustness analysis. Using optiSLang, statistical properties (e.g. distribution type, mean value, standard deviation, correlations) are typically assigned to a small set of parameters. The software uses this information to vary the input parameters according to the pre-defined scheme. The statistical properties must be obtained from the analysis of the laser scans. In this project, a random field model (“statistical shape model”) was chosen to represent the geometric variations (Fig. 3). Here, statistical properties are associated to each point on the housing boundary. This is very different from approaches that try to find a statistical description of CAD parameters. The random field model allows a greater accuracy in the spatial distribution of the variation patterns, because it is tied to the FEM nodes and not to just a few parameters. The parameterization can be automatically obtained by an analysis of the measurements.

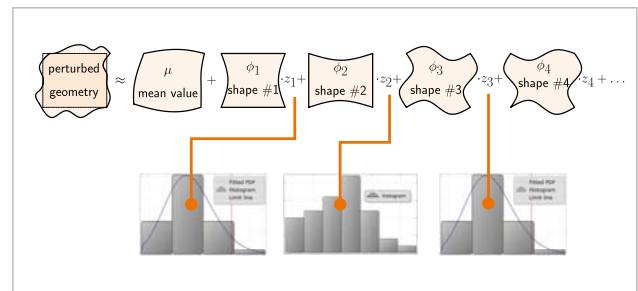


Fig. 3: Statistical shape model (Random field) to include geometric scatter in SoS

First, the measurements are imported to SoS by mapping the measurements (given through STL files defining the boundary) onto the surface of the undeformed FEM mesh. SoS automatically determines the geometric deviation (measured perpendicular to the surface) for each FEM node between each measurement and the reference geometry. A subsequent step is the conduction of a statistical analysis. For each FEM node, the mean deviation and the magnitude of the variation around the mean (= standard deviation) can be determined. A first indication can be derived from this analysis at which locations a large deviation from the CAD

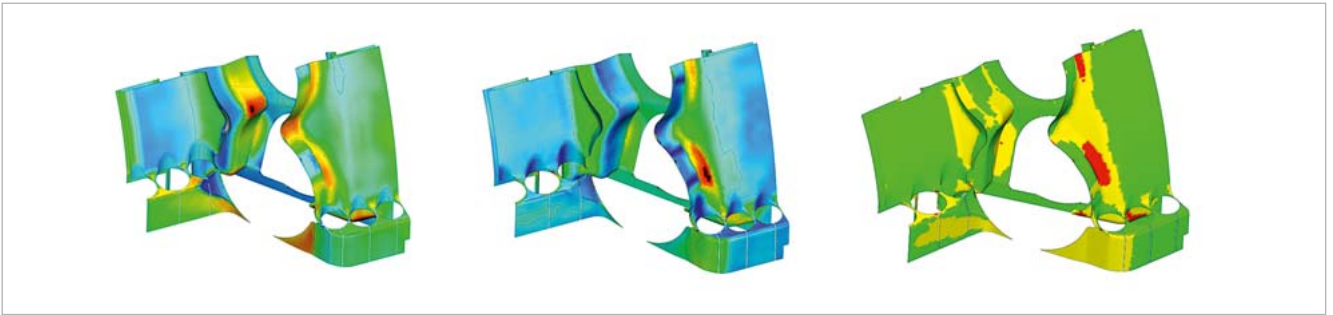


Fig. 4: Statistical analysis of the measured geometric tolerances. Left: mean deviation, Center: standard deviation of geometric scatter, Right: Tolerance analysis based on 3-sigma level (yellow and red are critical locations)

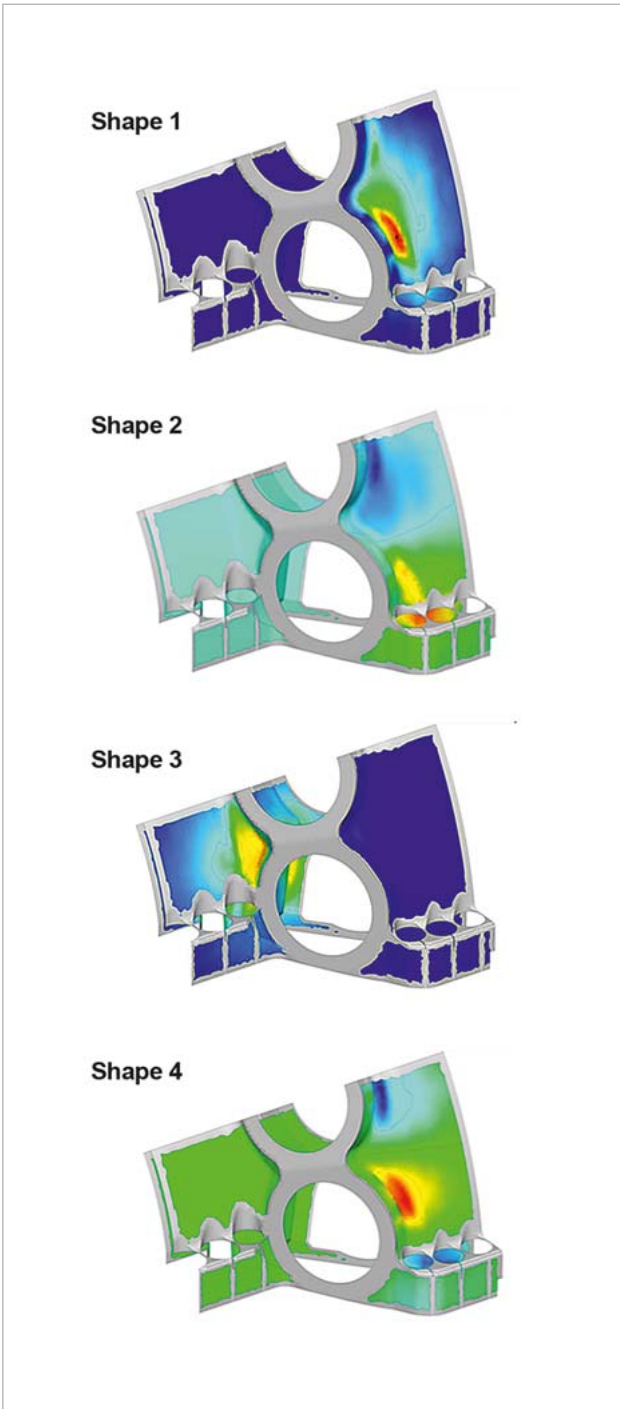


Fig.5: Statistical shape model (random field) of important variation patterns

geometry can be expected (from checking the mean value) and at which locations a large variation is obtained due to the natural variations in the production process (standard deviation), as shown in Fig. 4. Interestingly, these locations are different in our project. If more samples had been available, we would have extended the SoS analysis to search critical locations through quantile values and exceedance probabilities. Anyhow, the analysis can help to investigate the statistics more deeply in the context of quality control.

The next step is the creation of a statistical model for the geometric deviation. The random field model basically consists of “variation patterns” or “scatter shapes”. With a pre-defined sufficiently large set of samples (e.g. 80), SoS first analyzes the data for correlated variation patterns (Fig. 5). Once being identified, SoS can represent each measurement by a series expansion, where each variation shape is scaled by some coefficient and added to the mean value. Typically, only a few variation patterns (5-10) are sufficient to represent the original measurements with high accuracy.

In this project, only a very small number of measurements was available. Therefore, the variation patterns were created using analytical functions based on certain engineering assumptions (e.g. correlation length parameters). Nevertheless, the obtained model is accurate enough to represent the mean value and standard deviation for each FEM node as seen in the measurements. In SoS, this is called a “synthetic random field model”.

The statistical shape model is not created for the whole boundary. Although non-zero deviations were found for all boundary locations in the measurements, the generation of

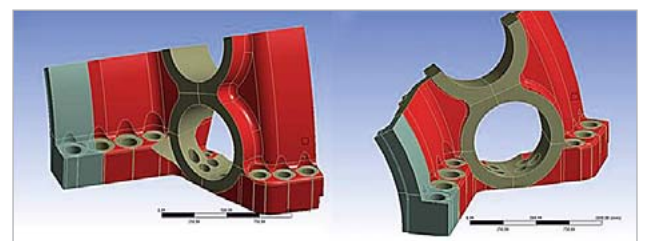


Fig. 6: Red: Surface parts to be varied, Gray: machined surface parts (fixed)

geometric deviations must be restricted to surface patches. Some surface parts are machined after molding, for example, holes are drilled and grinded to attach other structures, such as screws. Further, some surface parts must not be varied to ensure numerical stability of the CAE model, e.g., contact boundaries. The geometric parts subject to variation are shown in Fig. 6.

3. Numerical Model

The numerical model was mainly generated in ANSYS Workbench. Here, a semi analytic thermal model was implemented for the gas flow to be coupled with a transient thermo-mechanical model for the prediction of the spatial distribution of temperature and stresses. Subsequently, a low cycle fatigue (LCF) analysis was carried out with SIEMENS GT internal software. The 3D FEM model includes bolting contacts and a thermal transient defining the time-dependent loading conditions.

The original mechanical model for the 90° housing part consist of 840k nodes and 380k elements. The total computing time for a single design requires more than 30 GB disk space and approximately three days on a medium HPC hardware. Therefore, the run of e.g. 100 samples in a simple Design of Experiments of a robustness analysis is not feasible within project time.

To improve the time and storage demands, only one sub-model was considered with remaining 240k nodes and 270k finite elements (Fig. 7). The hardware demands could be reduced to 7 GB disk space and 2 hours per sample.



Fig. 7: Submodel of turbine housing

The numerical model was created using the CAD geometry model. Once there is an FEM mesh available, the statistical shape model can be built. The geometric changes are not applied to the CAD model, but to the coordinates of the FEM nodes. SoS does not change or re-mesh the FEM mesh, instead, it “morphs” the FEM nodes to their desired positions.

Before the ANSYS Workbench model will be evaluated by optiSlang, SoS prepares APDL macros in the ANSYS Workbench model folder for instructing ANSYS Mechanical how to change the geometry. SoS further uses advanced stabili-

zation and smoothing algorithms, which ensure the stability and computability of the changed FEM mesh. Before doing the robustness analysis, a test run of the solver chain is done using the mean value geometry.

4. Robustness analysis and results

The goal of the robustness analysis is to answer the following questions:

- Is there a difference in stress distribution or endurance factors between a “true” geometry and the reference geometry?
- Is there a significant influence onto stress and endurance factor if the geometric scatter is considered? How large is the influence?

The first question can be answered by transferring each measurement into the CAE model or, simply, by computing the mean geometry. The second question requires a stochastic tolerance analysis. Here, a Design of Experiments (DOE) is virtually created and evaluated. The workflow is illustrated in Fig. 8. This involves a variation of the random field parameters according to their statistical distributions. SoS simplifies the choice by assuming uncorrelated standard-normal variables.

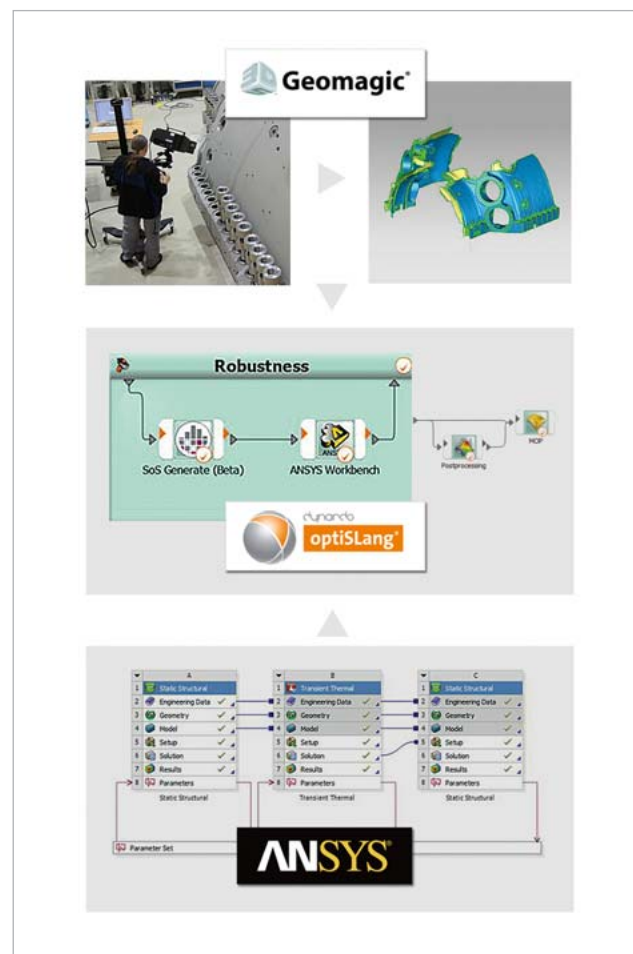


Fig. 8: Workflow of the analysis

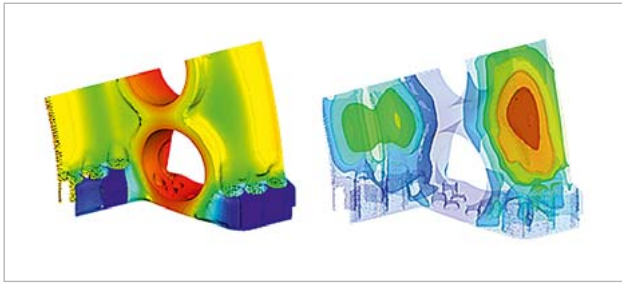


Fig. 9: Statistical analysis of resulting temperature field – mean value (left), standard deviation (right)

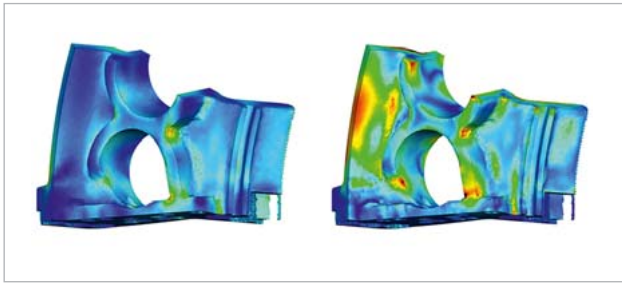


Fig. 10: Statistical analysis of resulting von Mises stress field – mean value (left), standard deviation (right)

The results of the CAE analysis (temperature, stresses, endurance factor, etc.) can either be analyzed in optiSLang (e.g. by their maximum values) or in SoS. For this, the ANSYS RST files are reevaluated by SoS. A statistical analysis can now be conducted for the result quantities (Fig. 9/10). The stresses can be compared, for example, with critical limits for different safety levels (exceedance probabilities). Further, the location of possible critical stresses can be easily identified.

A failure probability was not analysed in this project, because the accuracy of the statistical model is not sufficient due to the small number of measurements. Further, a sensitivity analysis can be conducted with the Field Metamodel of Optimal Prognosis (FMOP). The FMOP associates the sensitivity of the stresses or temperatures to each FEM node for the respective input parameters (Fig. 11). In this analysis, each input parameter represents a certain variation pattern. This indicates which geometric variation shape is relevant for the temperature or stress at critical locations. The result further justifies the quality criteria to geometric tolerances in production to the respective variation shapes.

Software and methodology

List of applied software and their particular functions:

optiSLang

- Management and automation of the workflow
- Generation of random parameter values and DOE
- Analyses of scalar responses

ANSYS

- Generation of the thermo-mechanical model

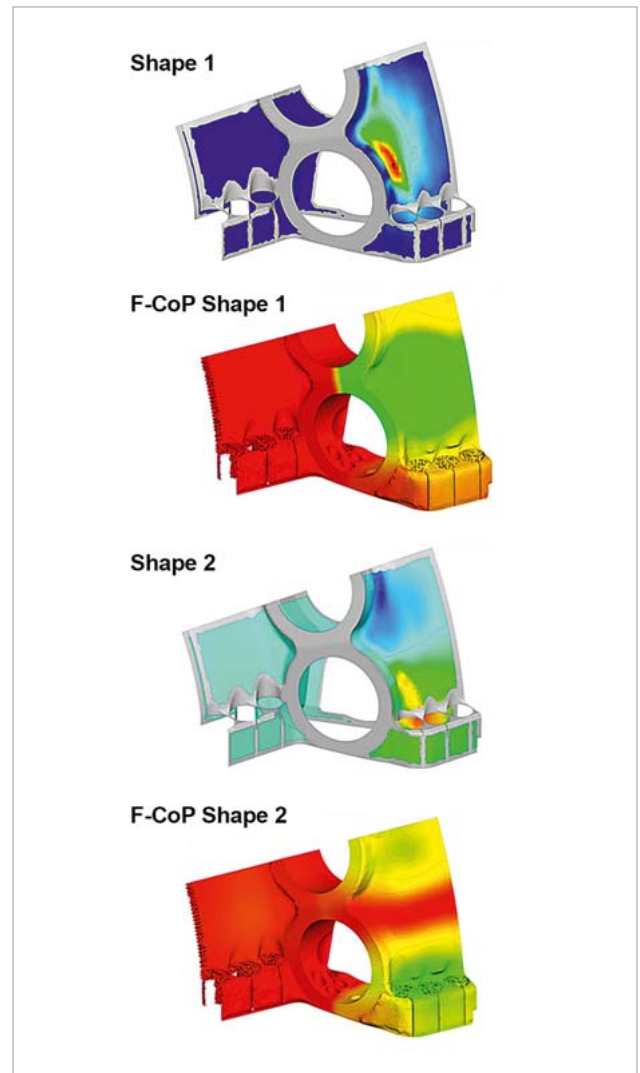


Fig. 11: Sensitivity analysis of temperature field with FMOP – Most dominant geometric variation patterns being responsible for changes in temperature with the corresponding sensitivity (F-CoP) of temperature with respect to the geometric variation patterns

SIEMENS GT

- Low cycle fatigue solver

Statistics on Structures

- Statistical analyses of the measurements
- Generation of new random geometries and transfer into the CAE process
- Statistical analyses of the FEM results in 3D
- Sensitivity analysis of FEM results with FMOP in 3D

Authors //

Uwe Lohse (SIEMENS AG)
 Holger Schulze Spüntrup (ITB Dortmund GmbH)
 Sebastian Wolff (DYNARDO Austria GmbH)

Sources // www.dynardo.de

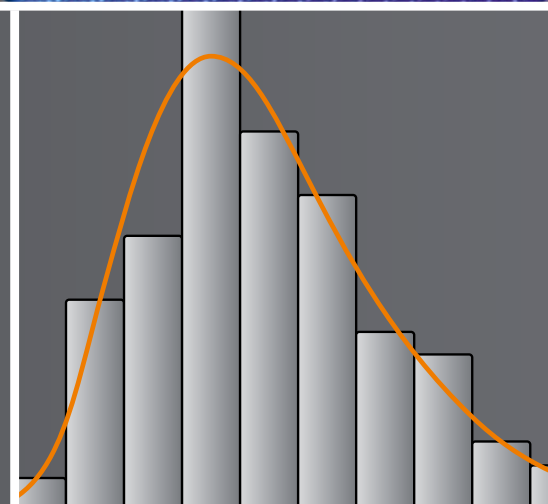
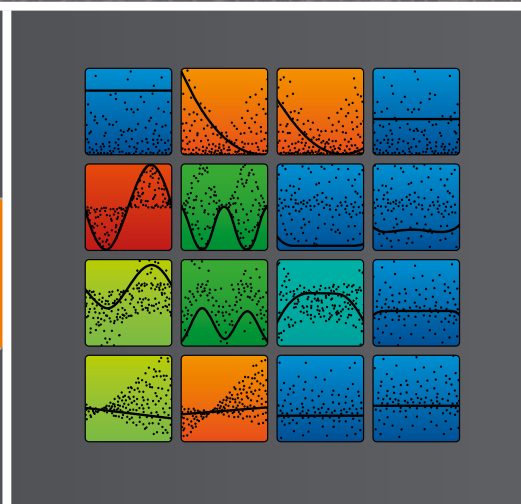
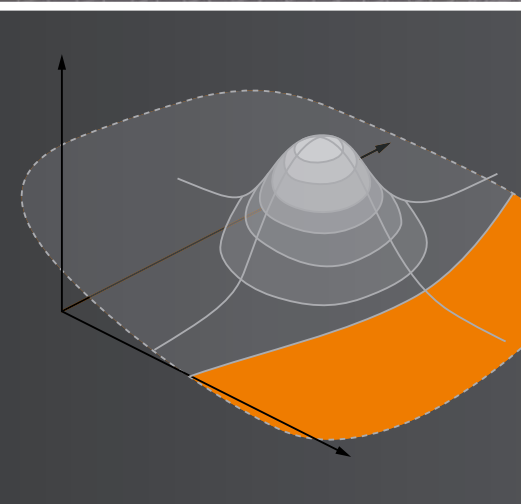
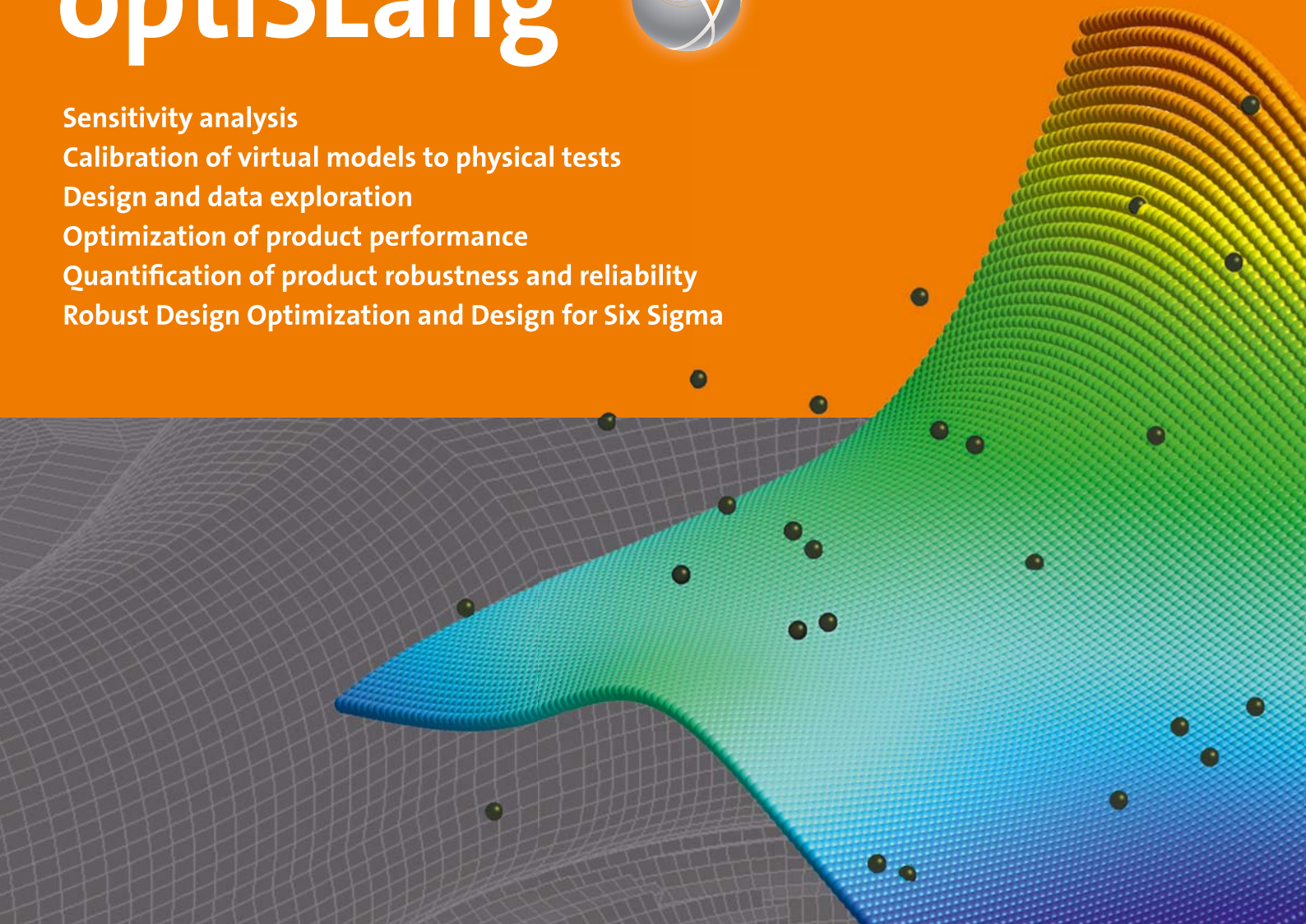
“Investigation of low cycle fatigue on large gas turbine casings under consideration of geometric tolerances”, U. Lohse, B. Voss, Th. Lowitz (Siemens AG Large Gas Turbines), H. Schulze Spüntrup (ITB Dortmund), S. Wolff DYNARDO Austria; presented at the Weimar Optimization and Stochastic Days, June 1-2, 2017

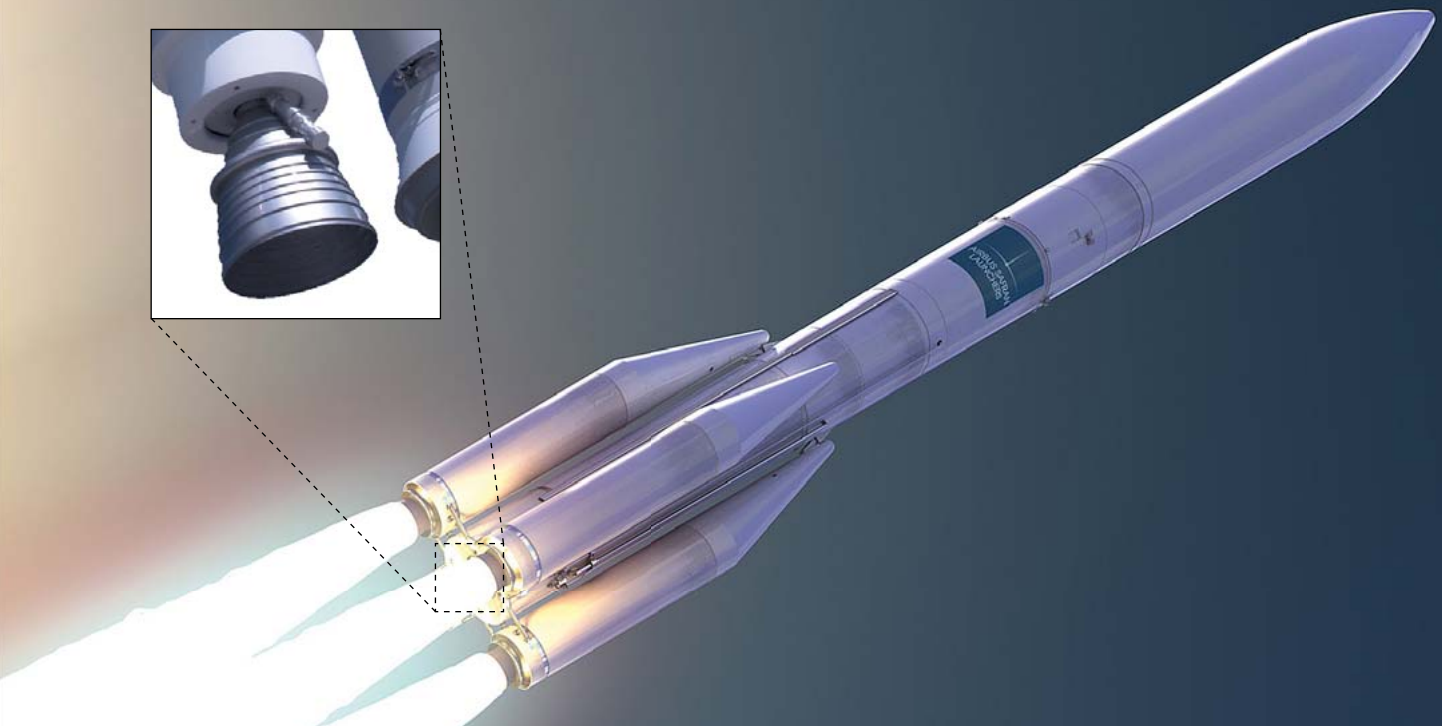
CAE Software & Consulting

optiSLang[®]



- Sensitivity analysis
- Calibration of virtual models to physical tests
- Design and data exploration
- Optimization of product performance
- Quantification of product robustness and reliability
- Robust Design Optimization and Design for Six Sigma





CUSTOMER STORY // AEROSPACE INDUSTRY

TMF PANEL OPTIMIZATION

Ariane Group GmbH developed a simulation procedure in order to reduce the effort on full scale hardware testing. optiSLang was used for parameter identification and optimization of Thermo-Mechanical Fatigue (TMF) panels representing in design and size one part of the combustion chamber of the Ariane 6 European launch vehicle.

Introduction

The propulsion system of a launch vehicle produces thrust in order to lift off and accelerate a carrier rocket into orbit. According to the principle of action and reaction between the combusted reaction gases and the launch vehicle, the acceleration depends on mass and velocity of the emitted matter. To keep the required fuel mass consumption low, a high exhaust velocity is desirable, which in turn requires high pressure levels and hot reaction temperatures inside the combustion chamber. Different concepts are available for the combustion chamber to maintain structural integrity. Here a regenerative cooled combustion chamber is considered, where a cryogenic fluid is fed through cooling channels in the combustion chamber hot gas wall.

Fig.1 shows a combustion chamber with its typical sandwich-like cooling channel structure. It is composed of an inner liner, typically made of a copper alloy, and the outer high-strength jacket, responsible to carry external loads. The hot gas wall as the innermost part of the liner represents the most loaded part of a combustion chamber. It is exposed to large temperature gradients between the combusting medium with up to 3000 K and the cryogenic coolant with around

40 K. The damage behavior of the hot gas wall is intended to be reproduced by the TMF panel tests by using specimen of equal cooling channel geometry and by applying loading conditions similar to those inside the combustion chamber.

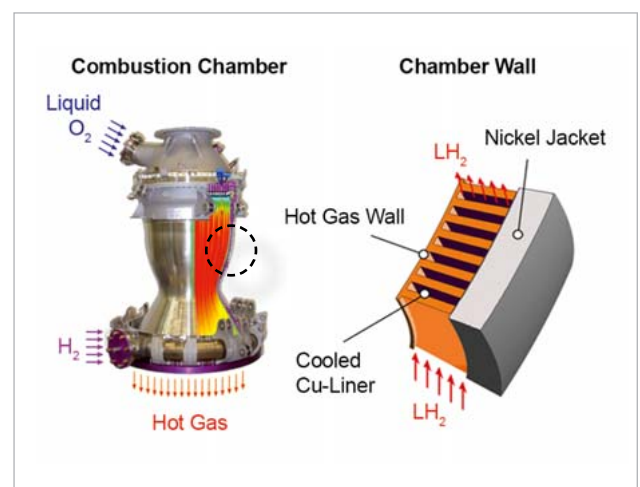


Fig. 1: Functionality of the launcher's main engine with enlarged part of the cooling channel structure of the engine's chamber wall

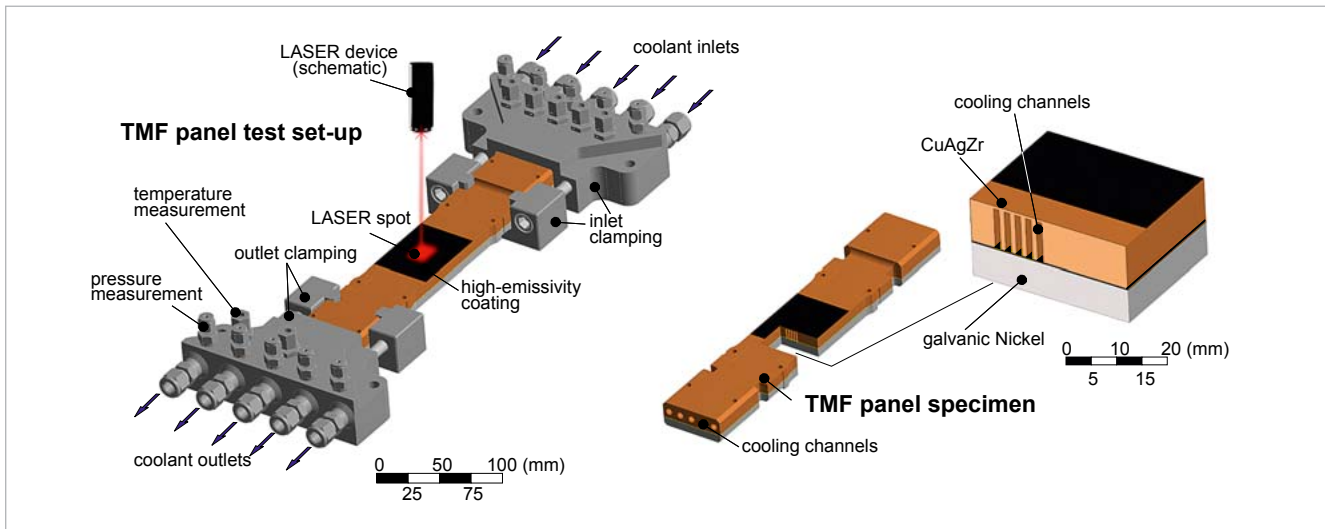


Fig. 2: TMF panel design and test concept [5]

TMF panel test

The TMF panel was created considering two goals. First, for the validation of the damage model, which was created for lifetime predictions on the hot gas wall. A detailed description of the damage model formulation, which accounts for the viscoplastic material behavior, ageing and damage effects under TMF loading conditions, can be found in [5]. With the TMF test, the model is applied to a more complex structure than for the specimen of tensile, fatigue and creep tests in order to justify its applicability on flight hardware. Based on the validated material damage model, a justification capability of today's combustion chambers is provided. Second, panel based TMF testing has the potential to be used in the development process of new combustion chambers as a cost efficient alternative to full-scale tests to investigate the capabilities of new materials or designs. With this intention, this article focuses on the representativeness of the panel's damage behavior compared to the combustion chamber hardware.

TMF panel design

As depicted in Fig. 2, the TMF panel is manufactured out of the liner material CuAgZr that includes five cooling channels in the dimensions of the combustion chamber. On its backside, a Nickel layer is applied by the galvanic deposition process.

In order to generate the high heat input into the panel's hot wall, a 10kW laser device is used to produce a heat flux of approx. 20 MW/m² on the panel surface. Due to the low absorption capacity of the panel material, the laser loaded surface is covered with a high emissivity coating. Further, a nitrogen flow is established that represents the regenerative cooling of the original combustion chamber structure. The pressurized liquid nitrogen circulates through the five cooling channels of the copper alloy part and cools the structure down to keep the hot gas wall temperature at a certain level. A more detailed description of the test stand can be found in [4].

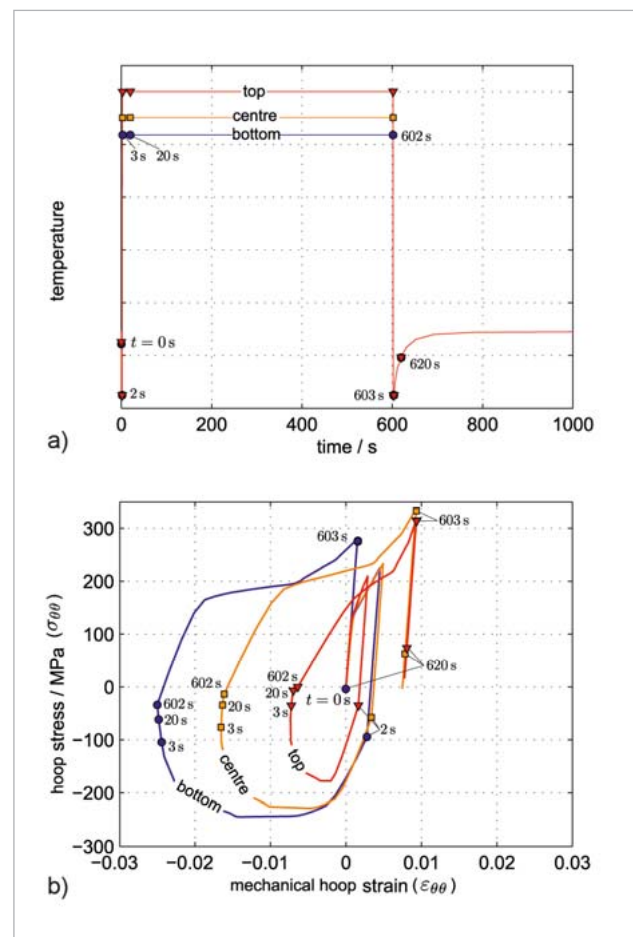


Fig. 3: Stress strain hysteresis in the hot wall ligament during the first load cycle: a) Transient temperatures; b) Stress-strain response in different locations through the liner ligament [5]

Comparison of combustion chamber and panel damage behavior

During a typical load cycle, the combustion chamber is first pre-cooled, which leads to circumferential contraction. Since the liner material usually has a higher coefficient of thermal expansion than the jacket material, a tensile stress

is induced within the hot gas wall during the first two seconds as depicted in Fig. 3b. After ignition, the liner material heats up while the cooled jacket prevents the liner to expand. Hence, a compressive stress state occurs in the hot gas wall that leads to inelastic deformations of the copper material throughout the hot run time of 600s. Once the engine is shut down, a post-cooling phase starts leading back to a tensile stress state. Finally, the temperature returns to an ambient level.

Multiple load cycles of precooling, hot run, post-cooling and return to ambient levels stresses the structure in a domain that is known as thermo-mechanical-fatigue (TMF). These load conditions lead to thinning of the hot gas wall which tends to a roof shaped configuration known as the dog house effect as depicted in Fig. 4b. Microstructural investigations

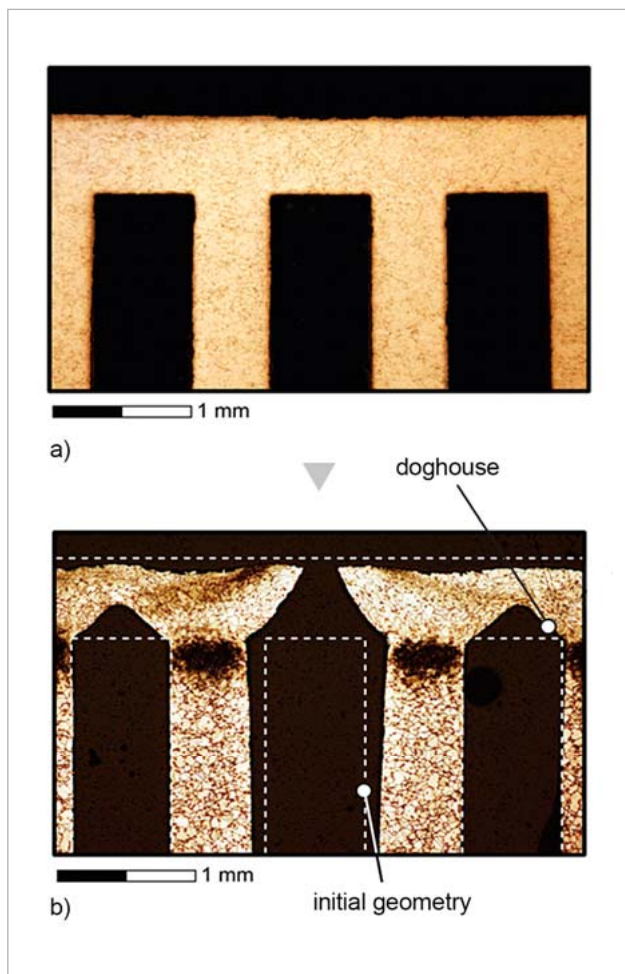


Fig. 4: Shape of the cooling channel structure inside the combustion chamber: a) initial state; b) after hot firing campaign [5]

confirm that ductile damage mechanisms lead to microdefects. During further load cycles, they extend towards macroscale defects. Hot gas wall failure occurs with the creation of macro cracks at the tip of the doghouse as seen in the middle cooling channel of Fig. 4b. Such a crack is not critical for the integrity of the entire engine but still shall be avoided.

On the local stress-strain level, the thinning of the hot gas wall corresponds to a circumferential tensile strain, which accumulates from cycle to cycle. In this case, the stress-strain hysteresis is open in the direction of positive hoop strain, see Fig. 5a.

Looking at the behavior of the TMF panel, it occurs that the stress strain hysteresis differs from what is seen in the combustion chamber, see Fig. 5b. During post cooling, the stress

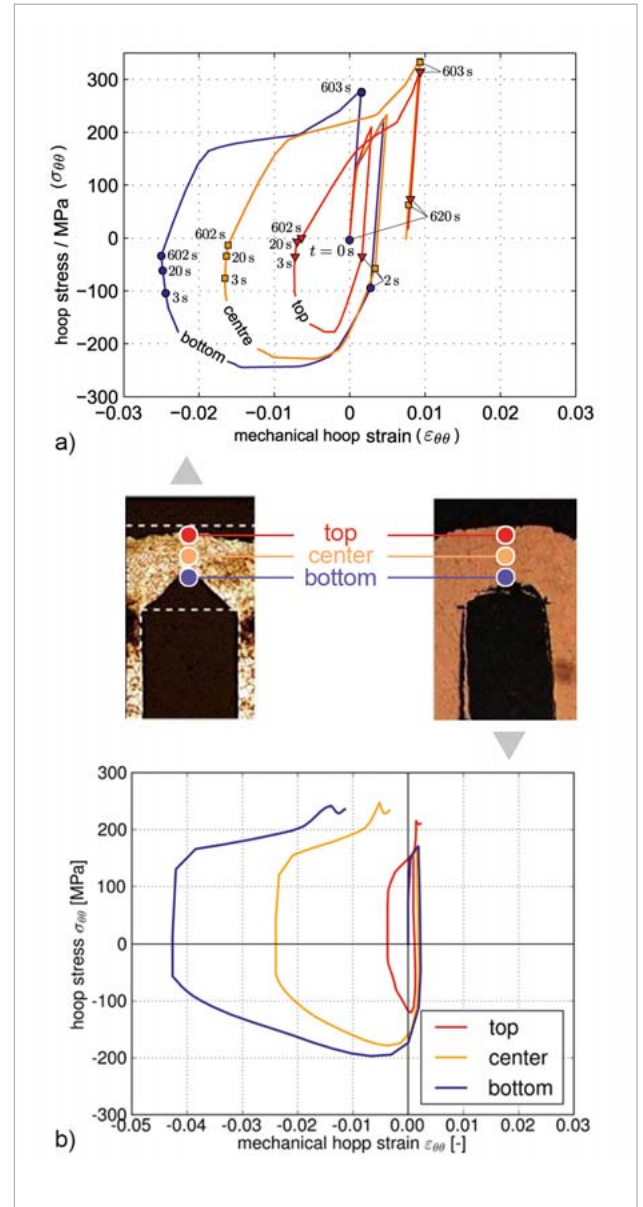


Fig. 5: Stress strain hysteresis in comparison between a) combustion chamber and b) TMF Panel

state is similarly in tension, but the strains get stuck in the compressive domain. As a result, the laser loaded wall of the panel is thickening in contrast to the thinning of the hot gas wall in the combustion chamber. This changes the damage conditions and reduces the representativeness of the panel tests.

In order to achieve a damage behavior in panel tests that is similar to the one found in a combustion chamber hot firing cycle, it is necessary to move the mechanical strain state after post cooling to the tensile domain. Former investigations revealed potential improvements by the adaption of test process parameters, like laser heat input or cooling mass flow, but their implementation on the test were limited. Therefore, further potential for improvement is now investigated on the geometrical level.

Optimization approach

The optimization with optiSlang is based on an ANSYS simulation that delivers the strain response for the designs under investigation. Therefore, a parametrized APDL script (ANSYS Parametric Design Language) is used to create the geometry, build the model, apply the boundary conditions, launch the job and extract all necessary result data. Finally, an error value is returned to optiSlang for each design point. Its minimization corresponds to the evolution towards the best design.

Geometry parameters

Fig. 6 shows a cut through the panel being under investigation. The illustration gives an overview of the design parameters that are modified by the optimizer. Similar to the original design as shown in Fig. 2 (see page 31), the panel consists of a copper liner, a nickel jacket and includes five cooling channels. As a conceptual novelty to the former flat panel design, the current study includes the assessment of curved panels.

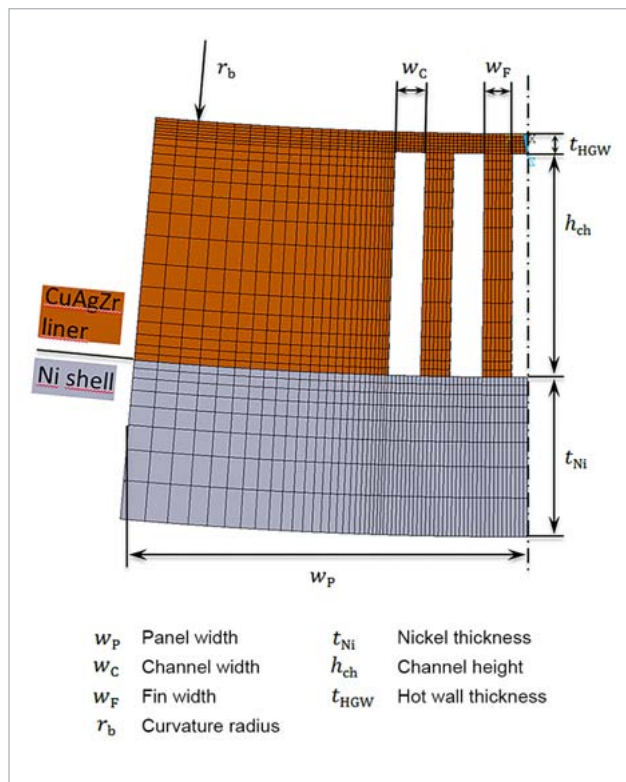


Fig. 6: Modified geometry parameters defining the panel design

The design generation is based on the Latin Hyper Cube sampling method. Thereby, each parameter is uniformly distributed over a band width of $\pm 20\%$ in relation to the initial values of the original design. For the curvature radius, the initial value is correlated to the combustion chamber curvature.

At first, the underlying Finite Element (FE) simulation calculates the temperature field based on the lasers heating input and the convective nitrogen cooling through the cooling channels as schematically displayed in Fig. 2 (see page 31). The subsequent mechanical FE simulation considers the temperature field, internal channel pressure and symmetry conditions along the symmetric plane. With the constitutive material formulation of the copper alloy, the panel deformations are calculated and the stress, strain and damage fields are analyzed. Reference [2] can be consulted for a detailed description of the FE simulation and the post processing used for characterizing the damage behavior inside the laser loaded wall, especially in the hot wall of the mid cooling channel.

The temperature distribution inside the panel as well as the overall stiffness of the panel is influenced due to the variation of the displayed design parameters. Subsequently, the loading of the laser loaded walls changes and leads to the variations of the stress-strain-hysteresis, which is considered during the optimization process.

Optimization criteria

In order to formulate a minimization problem, an error value 'Err' is defined. 'Err' is quantifying the strain deviation of the current design from the goal behavior of the combustion chamber. As exemplarily depicted in Fig. 7 (see next page), for the hot wall center point position of the mid channel, the difference between the mechanical hoop strain after the first load cycle is measured for all three wall positions: top, center and bottom. The geometric mean value of the difference value then defines the error value that is to be minimized:

$$Err = \sqrt{\frac{1}{3} \sum_{bottom}^{top} (\Delta \varepsilon_{\theta\theta})^2} \quad \text{with } \Delta \varepsilon_{\theta\theta} = \varepsilon^{panel} - \varepsilon^{target}$$

The error value is calculated within the APDL script after the FE simulation for each single design and afterward transmitted back to optiSlang.

Sensitivities and best design

For the sensitivity analysis, 100 designs are created by varying the 7 design parameters. 98 designs are calculated successfully and allow the investigation of the model sensitivities according to the described error definition. The same simulation results are used for the generation of a Metamodel of Optimal Prognosis (MOP), which is applied for optimization purposes.

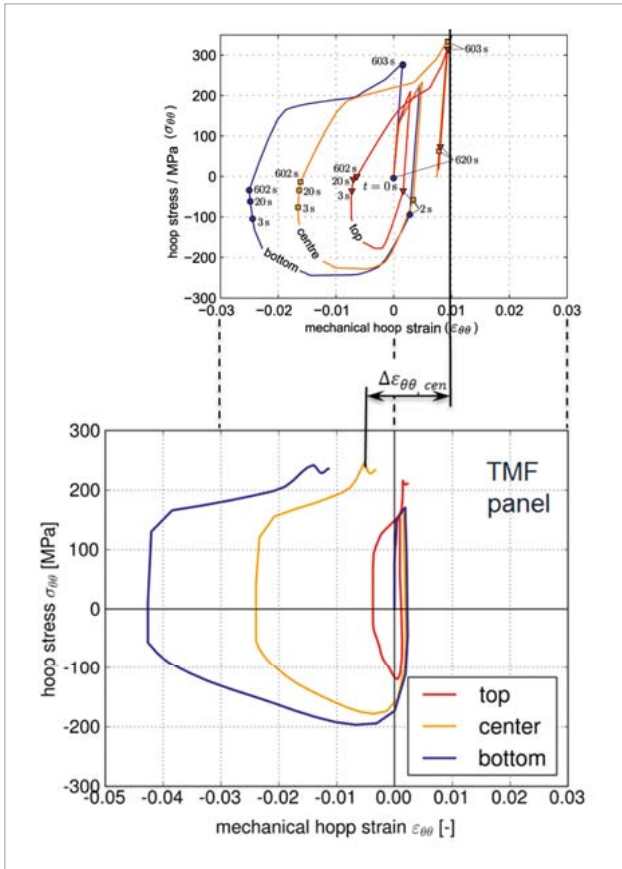


Fig. 7: Lateral mechanical strain deviation after the first load cycle between the goal level of the combustion chamber and the current level of the treated design, here, exemplarily for the center wall position

Parameter sensitivity

The results of the sensitivity analysis provided by optiSlang are presented in Fig. 8. It turns out that the curvature radius shows the highest influence and a large radius reduces the

error value. Although this leads to the notion that maximizing the curvature radius towards a flat shape is beneficial for the damage behavior, it has to be mentioned that the error value minimizes in the upper level of the design space of parameter r_b . Hence, the curvature is needed to be considered for the optimization. Regarding the overall panel width as the second most influential parameter, a more intuitive result could be seen. Having less material in the bulky side volume, the overall cooling behavior is improved, which also stiffens the structure during the cooling phase. While the compressive deformation during the hot phase leads to compressive plastification in lateral direction, the post cooling moves the investigated wall into a tensile stress state increasing its influence by a colder and stiffer side structure. The strain level is then pulled towards the tensile domain in post cooling phase. Further, the residual strain turns out to be sensitive to the liner thickness $t_{liner} = h_{ch} + t_{HGW}$. Here, a thicker liner leads to a reduction of the error value.

It also can be seen that the error values are less sensitive towards the design parameters defining the actual channel structure and hot wall dimension. Therefore, hot wall thickness, channel width and distance can be modified with minor influence on the actual damage behavior. This fact is important regarding the application of the TMF panel test for future combustion chamber validation efforts.

Best design - geometry evolution

From the results of the sensitivity analysis, a Metamodel of Optimal Prognosis (MOP) was created. The actual optimization task was performed on the basis of the MOP, which showed a Coefficient of Prognosis (CoP) = 96%. The result of the optimization incorporates the findings of the sensitivity analysis of a reduced panel width, larger liner thickness

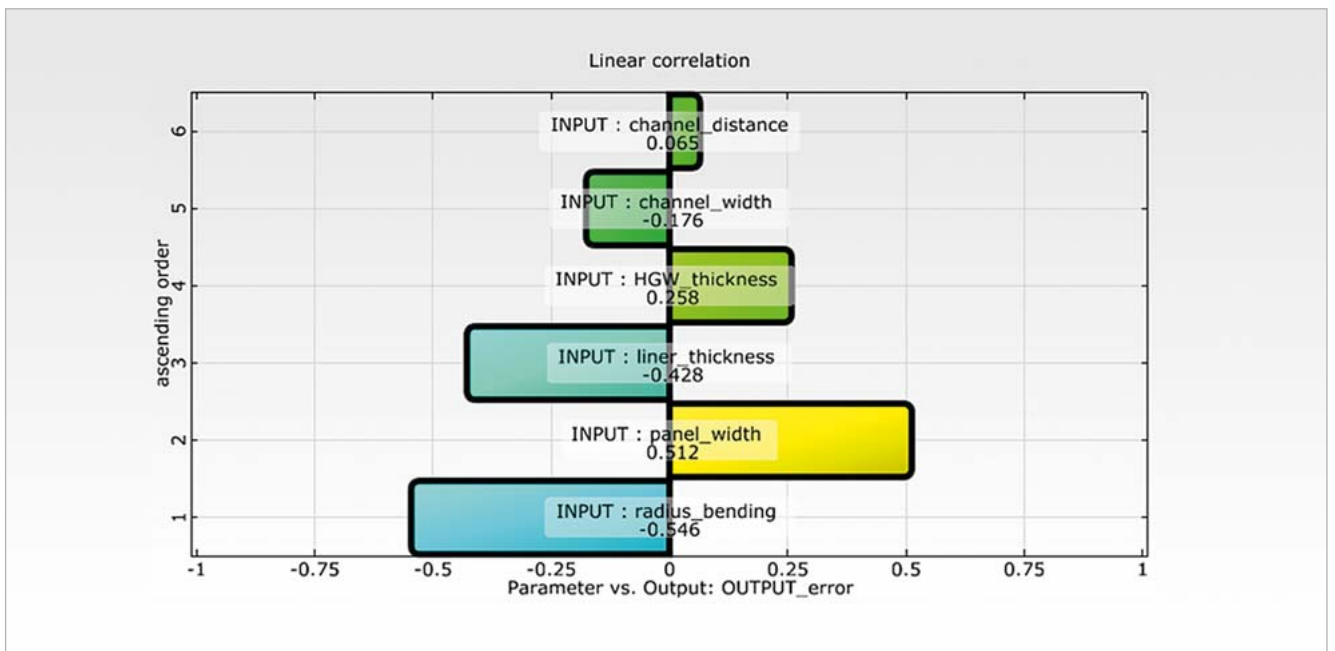


Fig. 8: Sensitivities towards design parameters

and the optimal bending radius as depicted in the geometry drawings in Fig. 9 with the initial geometry on the left and the optimized geometry on the right.

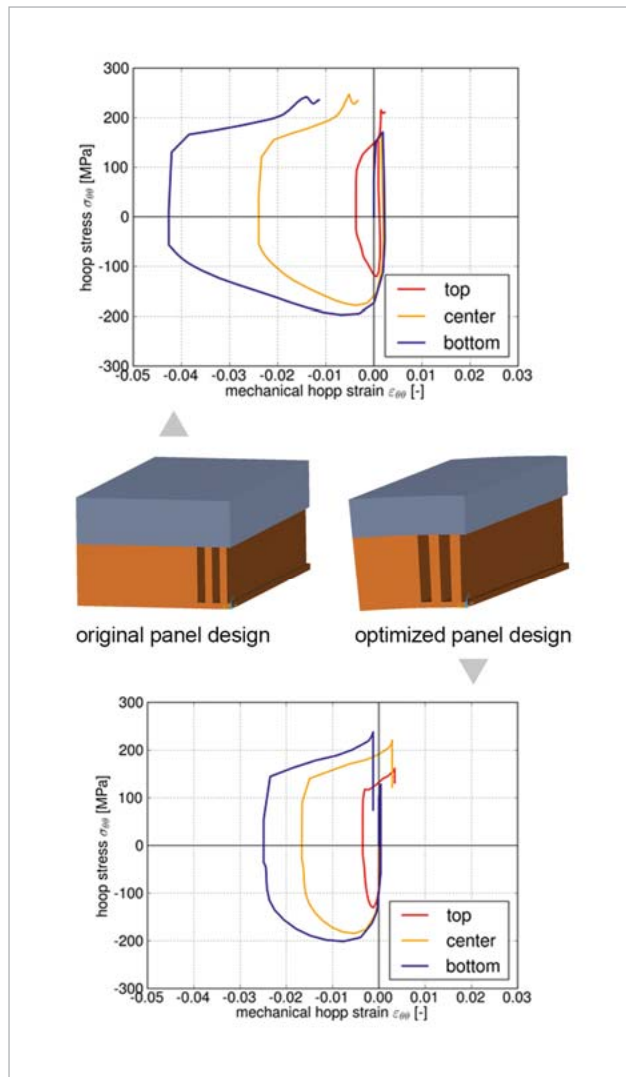


Fig. 9: Original panel design (top) compared to the optimized panel design (bottom)

While observing the stress strain hysteresis of the first cycle of both designs in Fig. 9, it can be seen that the optimized design clearly tends to the tensile strain domain during the cooling phase. Hence, it is shown that modifications of the panel design can be used to change the stress and strain behavior of the hot wall towards the desired direction. Due to the mentioned modifications, the residual strain accumulated in the tensile domain as well as it was observed in the combustion chamber. The optimized panel design increased the representativeness on the hot gas wall damage behavior.

Conclusion

In the present study, the potential enhancement of the currently used TMF panel design was investigated in order to find a panel shape that shows a damage behavior similar to

the combustion chamber. Therefore, an automated TMF panel test simulation was created including the generation of a FE model and running the thermal and mechanical analysis based on prior defined design parameters. In addition, the results of the automatically performed comparison between the behavior of the current design and the one of the combustion chamber were reprocessed back as output variables. With the help of the analyzing capabilities of optiSlang, the sensitivities of the parameter variations regarding the panel's damage behavior could be recognized and verified. It was shown that a curvature of the panel has a high influence on the hot wall behavior as well as on the thickness and width of the panel. On the other hand, the hot wall thickness, channel width and the fin width had a lower influence, which allowed their modification without violating the representativeness of the panel to the combustion chamber. This result is especially important regarding a future application of the TMF panel tests towards combustion chamber qualification. With the results of the sensitivity analysis, a MOP based optimization procedure was launched resulting in a best design capable of fulfilling the objectives of this investigation. Hence, it was found that the right parameter adjustments on the panel design lead to a combustion chamber like damage behavior. This allowed a TMF panel testing of the combustion chamber representatives.

Author //

Marcus Lehmann (Ariane Group GmbH)

Sources //

- [1] Gurson A.L., Continuum theory of ductile rupture by void nucleation and growth: Part I – Yield criteria and flow rules for porous ductile media, *Journal of engineering materials and technology*, 1977.
- [2] Lehmann, M., Schwarz, W., Life time prediction of rocket combustion chambers - current justification approaches and test, 14th European conference on spacecraft structures, materials and environmental testing, Toulouse, ECSS-MET, 2016
- [3] Lemaitre J., Chaboche J.-L., *Mechanics of solid materials*, Cambridge University Press, 1990.
- [4] Riccius J. R., Zamataev E. B., Gernoth A., Schwarz W., Keppeler J., A laser plateau size optimization strategy for TMF tests, 3rd European Conference for Aero-Space Science, EUCASS, Versailles-Paris, 2009
- [5] Schwarz, W., *Modelling of Viscoplasticity, Ageing and Damage for Life Prediction of Rocket Combustion Chambers*, Erlangen, 2013.
- [6] Schwub S., *Mikrostruktur, mechanische Eigenschaften und Schädigungsmechanismen von Kupfer-Silber-Zirkonium Legierungen für Hochtemperaturanwendungen*, Erlangen, 2012.
- [7] Tvergaard V., Needleman A., Analysis of the cup-cone fracture in a round tensile bar, *Acta Metallurgica*, 1984.

Contact & Distributors

Worldwide

Dynardo GmbH
Steubenstraße 25
99423 Weimar
Phone: +49 (0)3643 9008-30
Fax.: +49 (0)3643 9008-39
www.dynardo.de
contact@dynardo.de

Dynardo Austria GmbH
Office Vienna
Wagenseilgasse 14
1120 Vienna
www.dynardo.at
contact@dynardo.at

Worldwide distribution of ANSYS optiSLang

ANSYS, Inc.
Canonsburg
www.ansys.com

Worldwide distribution of optiSLang

LightTrans GmbH
Jena
www.lighttrans.de

Germany

CADFEM GmbH
Grafing b. München
www.cadfem.de

Austria

CADFEM (Austria) GmbH
Vienna
www.cadfem.at

Switzerland

CADFEM (Suisse) AG
Aadorf
www.cadfem.ch

Czech Republic, Slovakia, Hungary

SVS FEM s.r.o.
Brno-Židenice
www.svsfem.cz

Sweden, Denmark, Finland, Norway

EDR & Medeso AB
Västerås
www.edrmedeso.com

United Kingdom of Great Britain and Northern Ireland

CADFEM UK CAE Ltd
Croydon, Surrey
www.cadfemukandireland.com

Ireland

CADFEM Ireland Ltd
Dublin
www.cadfemukandireland.com

Turkey

FIGES A.S.
Istanbul
www.figes.com.tr

North Africa

CADFEM Afrique du Nord s.a.r.l.
Sousse
www.cadfem-an.com

Russia

CADFEM CIS
Moscow
www.cadfem-cis.ru

India

CADFEM Engineering Services India
Hyderabad
www.cadfem.in

USA

CADFEM Americas, Inc.
Farmington Hills, MI
www.cadfem-americas.com

Ozen Engineering Inc.
Sunnyvale, CA
www.ozeninc.com

USA/Canada

SimuTech Group Inc.
Rochester, NY
www.simutechgroup.com

Japan

TECOSIM Japan Limited
Saitama
www.tecosim.co.jp

Korea

TaeSung S&E Inc.
Seoul
www.tsne.co.kr

China

PERA-CADFEM Consulting Inc.
Beijing
www.peraglobal.com

Publication details

Publisher

Dynardo GmbH
Steubenstraße 25
99423 Weimar
www.dynardo.de
contact@dynardo.de

Executive Editor & Layout

Henning Schwarz
henning.schwarz@dynardo.de

Registration

Local court Jena: HRB 111784

VAT Registration Number

DE 214626029

Publication

worldwide

© Images

Fotolia: PRILL Mediendesign, p. 30

Copyright

© Dynardo GmbH. All rights reserved
The Dynardo GmbH does not guarantee or warrant accuracy or completeness of the material contained in this publication.