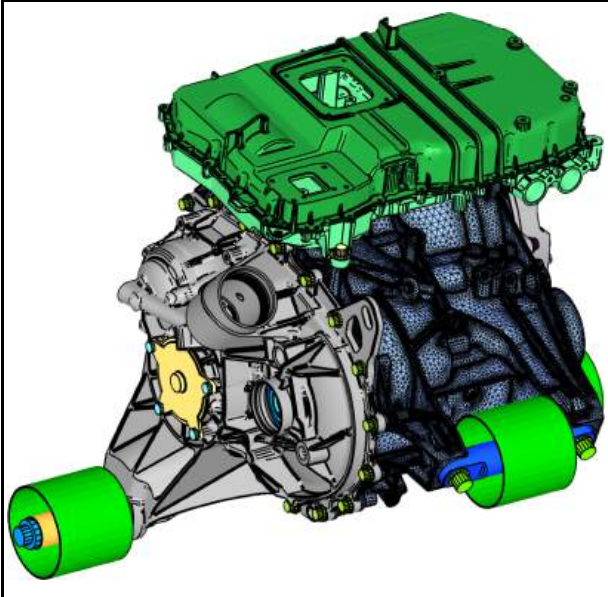




Product Description Version 21

Ingenieurgesellschaft für
technische Software mbH



© **INTES GmbH**, July 2026 (rev. 21.00)

The words in **green** colour are collected in the index at the end of this document.

The solid only finite element model (34 M DOFs, 11.6 M nodes and 90 000 CA-DOFs) of an eDrive unit for quasi-linear structural analysis appears by courtesy of ZF Friedrichshafen AG, Germany.

Address: **INTES GmbH**
Breitwiesenstr. 28
D-70565 Stuttgart

Phone: **+49 711 784 99 - 0**
 Fax: **+49 711 784 99 - 10**

E-mail: **info@intes.de**
 WWW: **<https://www.intes.de>**

Abaqus is a registered trademark of Dassault Systèmes Simulia Corp., Providence, RI, USA.

Adams is a registered trademark of Hexagon AB, Stockholm, Sweden.

ADSTEFAN is a registered trademark of Hitachi Industry Control & Solutions, Ltd., Ibaraki, Japan.

CATIA is a registered trademark of DASSAULT SYSTEMS, and in Canada, IBM is the registered user under No. RU 81167.

COMREL is a registered trademark of RCP GmbH, München, Germany.

AVL Excite is a registered trademark of AVL List GmbH, Graz, Austria.

HyperMesh, HyperView, H3D and MotionSolve are registered trademarks of the Altair Engineering Inc., Big Beaver, USA.

I-DEAS is a registered trademark of SIEMENS PLM Software Inc., Plano, USA.

MEDINA and CAE-Datenschienen are registered trademarks of the T-Systems ITS GmbH, Stuttgart, Germany.

MATLAB is a registered trademark of The Mathworks Inc., Natick, MA, USA.

MpCCI is a registered trademark of FhG SCAI, St. Augustin, Germany.

Nastran is a registered trademark of the National Aeronautics and Space Administration (NASA).

NX is a registered trademark of Siemens PLM Software.

MSC Patran is a registered trademark of Hexagon AB, Stockholm, Sweden.

PERMAS is a registered trademark of **INTES** Ingenieurgesellschaft für technische Software mbH, Stuttgart, Germany.

Simdrive 3D is a registered trademark of CONTECS Engineering Services GmbH, Berlin, Germany

SIMPACT is a registered trademark of Dassault Systèmes Simulia Corp., Providence, RI, USA.

STAR CD is a registered trademark of Computational Dynamics Ltd., London, England.

VAO is a registered trademark of CDH AG, Ingolstadt, Germany.

Virtual.Lab is a registered trademark of LMS International, Leuven, Belgium.

VisPER is a registered trademark of **INTES** Ingenieurgesellschaft für technische Software mbH, Stuttgart, Germany.

The use of registered names or trademarks does not imply, even in the absence of further specific statements, that such names are free for general use.

Contents

	Page		Page
INTES	5	Kinematic Constraints	64
Company Profile	5	Handling of Singularities	65
Services	5	Element Library	66
PERMAS	7	Standard Beam Cross Sections	67
Overview	7	Design Elements for Optimization	67
Introduction to PERMAS	7	SPR Stresses and Absolute Error Indicator	68
Benefits of PERMAS	8	Material Properties	68
What's New in PERMAS Version 21	9	Sets	69
What's New in VisPER Version 21	14	Mathematical Functions	69
Universal Features	16	Loads	70
Available VisPER Modules	16	Model Verification	71
Available PERMAS Modules	16	Interfaces	72
Performance Aspects	17	Matrix Models	73
Parallelization	18	Combination of Results	74
Areas of Application	18	Transformation of Results	74
Reliability	19	Comparison of Results	74
Quality Assurance	19	XY Result Data	75
Applications	21	Cutting Forces	75
Car Body Analysis	21	System of Units	75
Engine Analysis	24	Restarts	76
Part Connections	27	Open Software System	76
Brake Squeal Analysis	29	Direct Coupled Analyses	76
Rotating Systems	31	Coupling with CFD	76
Analysis of Machine Tools	32	PERMAS Analysis Modules	79
Actively Controlled Systems	35	PERMAS-MQA – Model Quality Assurance	79
Robust Optimum Design	37	PERMAS Thermo-Mechanics	81
Experimental Modal Analysis	38	PERMAS-LS – Linear Statics	81
VisPER	41	PERMAS-CA – Contact Analysis	81
The VisPER History	41	PERMAS-CAX – Extended Contact Analysis	85
VisPER – A Short Introduction	41	PERMAS-CAMG – Contact Multigrid Analysis	85
VisPER-BAS – Basic Module	42	PERMAS-CAU – Contact Geometry Update	87
Postprocessing	47	PERMAS-NLS – Nonlinear Statics	88
VisPER-TOP – Topology Optimization	47	PERMAS-NLSMAT – Extended Material Laws	91
VisPER-OPT – Design Optimization	49	PERMAS-NLSA – Nonlinear statics advanced	92
Design Wizard	50	PERMAS-BA – Linear Buckling	93
VisPER-FS – Fluid-Structure Coupling	51	PERMAS-HT – Heat Transfer	94
VisPER-CA – Contact Analysis	54	PERMAS-NLHT – Nonlinear Heat Transfer	95
Substructuring	55	PERMAS Vibro-Acoustics	97
Evaluation of Spotwelds	56	PERMAS-DEV – Dynamic Eigenvalues	97
VisPER-PCGen – PERMAS Component Generator	57	PERMAS-DEVX – Extended Mode Analysis	98
PERMAS Basic Functions	59	PERMAS-MLDR – Eigenmodes with MLDR	100
Substructuring	59	PERMAS-DRA – Dynamic Response	101
Submodeling	59	PERMAS-DRX – Extended Dynamics	104
Variant Analysis	60	PERMAS-FS – Fluid-Structure Acoustics	104
Cyclic Symmetry	61	PERMAS-NLD – Nonlinear Dynamics	106
Surface and Line Description	61	PERMAS-HBM – Harmonic Balance Method	107
Automated Coupling of Parts	62	PERMAS Design-Optimization	109
Automated Spotweld Modeling	63	PERMAS-OPT – Design Optimization	109
Local Coordinate Systems	64	PERMAS-TOPO – Layout Optimization	114
		PERMAS-AOS – Advanced Optim. Solvers	117
		PERMAS-RA – Reliability Analysis	119
		PERMAS Special Modules	121

PERMAS-LIFE – Basic Fatigue Lifetime Analysis	
121	
PERMAS-LA – Laminate Analysis	122
PERMAS-WLDS – Refined Weldspot Model	122
PERMAS-AERO – Generalized Inertia Relief	123
PERMAS-EMS – Electro- and Magneto-Statics	124
PERMAS-EMD – Electrodynamics	124
PERMAS-XPU – GPU accelerator	125
Interfaces	127
Overview	127
PERMAS-MEDI – MEDINA Door	127
PERMAS-PAT – PATRAN Door	127
PERMAS-ID – I-DEAS Door	127
PERMAS-AD – ADAMS Interface	128
PERMAS-EXCI – EXCITE Interface	128
PERMAS-SIM – SIMPACK Interface	129
PERMAS-HMS – MotionSolve Interface	129
PERMAS-H3D – HYPERVIEW Interface	129
PERMAS-VAO – VAO Interface	129
PERMAS-VLAB – Virtual.Lab Interface	129
PERMAS-ADS – ADSTEFAN Interface	129
PERMAS-MAT – MATLAB Interface	130
PERMAS-NAS – NASTRAN Door	130
PERMAS-ABA – ABAQUS-Door	131
PERMAS-CCL – MpCCI Coupling	131
PERMAS-SIM3D – SIMDRIVE 3D Interface	132
More Interfaces	132
PERMAS for Education	135
Installation and beyond	137
Supported Hardware Platforms	137
Licensing	137
Maintenance and Porting	137
User Support	138
Additional Tools	139
Documentation	140
Training	140
Future Developments	140
Additional Information	141
Index	143

INTES



Figure 1: INTES Head Office in Stuttgart

Company Profile

INTES company was founded as an FE technology enterprise in 1984. Its competence in every aspect of Finite Element (FE) technology is provided by INTES to its clients not only through the high-end software system PERMAS. The full range of development know-how of INTES is also made available to its clients by the provision of top-notch services and expert consultancy. INTES activities mainly concentrate on the

- development and distribution of the FE solver PERMAS and the joint pre- and post-processor VisPER,
- development of new and efficient numerical and graphical methods,
- development of software for new hardware architectures (such as parallel computers),
- increasing customer productivity by creation of efficient analysis workflows, e.g. coupling of PERMAS and VisPER with other software systems (such as pre- and postprocessors and MBS systems),
- consultancy and training of users,
- consulting by analysis projects.

The international support of PERMAS clients is supported in France by INTES France and in Japan by INTES Japan. In addition, partners are supporting and distributing the software in other countries.

For all of its customers, INTES wants to be a competent partner in all respects regarding the Finite Element Method. Above all, satisfaction of the customers with all the software and services is of prime

importance to the company.

“Making Realistic Simulations Practical”

Underlines our focus on technology and innovation.

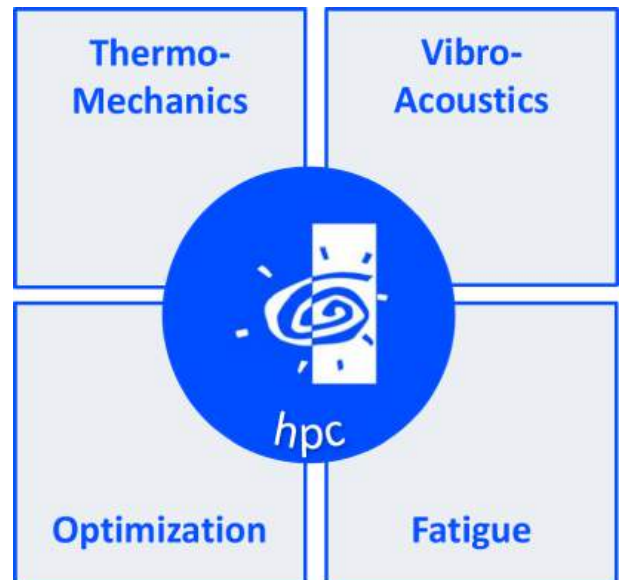


Figure 2: Major application areas with the shared HPC kernel technology

Services

INTES offers a number of services to its customers including:

- Developments for PERMAS and VisPER:
 - Interfaces to other software packages,
 - Efficient and reliable simulation workflows and process automation,
 - New analysis capabilities,
 - New finite elements,
 - Customer specific developments.
- Installation of PERMAS and VisPER on new hardware platforms as well as consultancy concerning the optimum hardware configuration,
- Software maintenance,
- FEM training,
- FEM research and development,
- Configuration and installation of add-on software products,
- Engineering:
 - modeling with VisPER, ANSA, MeshWorks,
 - ...

- simulation with PERMAS,
 - workflow and process development,
 - engineering method development,
 - Know-How transfer, based on clients projects.
- Introduction of FE analysis in enterprises, continuous consultation service (hotline), and support on current projects.

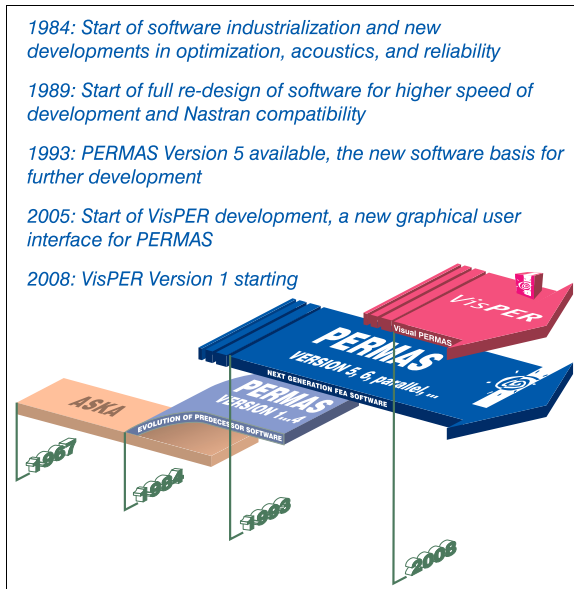


Figure 3: Synopsis of PERMAS history

PERMAS

Overview

This product description provides information on all essential characteristics of PERMAS and its application. Therefore, the description is organized into seven parts set forth below:

- The **introduction** gives some good reasons for the application of the Finite-Element-Method (FEM) and PERMAS. The particular **benefits** of PERMAS are presented on pages 8 to 19.
- **Applications** using several functional modules are illustrated on pages 21 to 38
- The features of **VisPER** are described on pages 41 to 56.
- The **universal features** of PERMAS, which are not related to a single module, are explained on pages 59 to 76.
- The available **functional modules** are described on pages 79 to 124.
- The **interfaces** are collected on pages 127 to 131.
- Additional information about the **installation and further aspects** of PERMAS is given on pages 137 to 141.

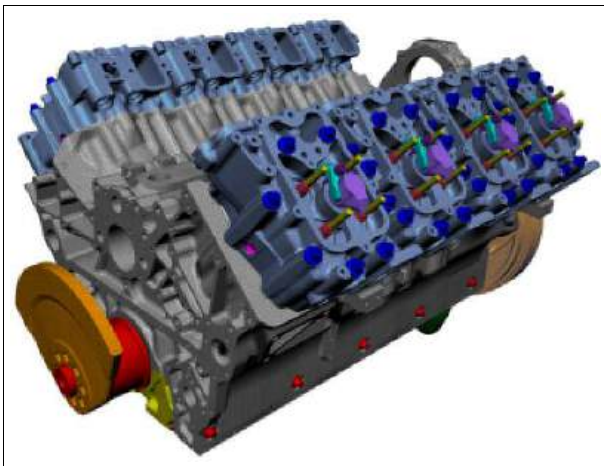


Figure 4: V8 engine by courtesy of FPT Motoren-forschung AG in Arbon, Switzerland.

Introduction to PERMAS

PERMAS is a general purpose software system to perform complex calculations in engineering using the finite element method (FEM), and to optimize the analyzed structures and models. It has been developed by INTES and is available to engineers as an analysis tool worldwide.

PERMAS enables the engineer to perform comprehensive analyses and simulations in many fields of applications like stiffness analysis, stress analysis, determination of natural modes, dynamic simulations in the time and frequency domain, determination of temperature fields, acoustic fields, and electromagnetic fields, analysis of anisotropic material like fibre-reinforced composites.

PERMAS computes a large number of results during the course of these analyses, which may be used in the assessment of the structural behavior like deflections, stresses and strains, natural frequencies and mode shapes, strain energy distribution, sound vibration power density, time history and interaction with other parts of the structure.

Independent of the area of application, these results provide a lot of valuable information for the design and development process. A number of essential **benefits** can be derived from the early use of the FEM:

- Safe accomplishment of customer requirements.
- Reduction of expensive manufacturing and testing of prototypes.
- Simulation of extreme conditions.
- Shorter development and design cycles.
- Significant suggestions for design optimization:
 - topology optimization,
 - sizing optimization,
 - shape optimization,
 - parameter studies by sampling.
- Improvement of structural reliability.
- Analysis in case of malfunction of a structure during operation.
- Long term quality improvements.

In view of today's increasing requirements for short design cycles and high quality products, the finite element analysis becomes an indispensable tool for the daily development work. Moreover, complex products are often developed in distributed struc-

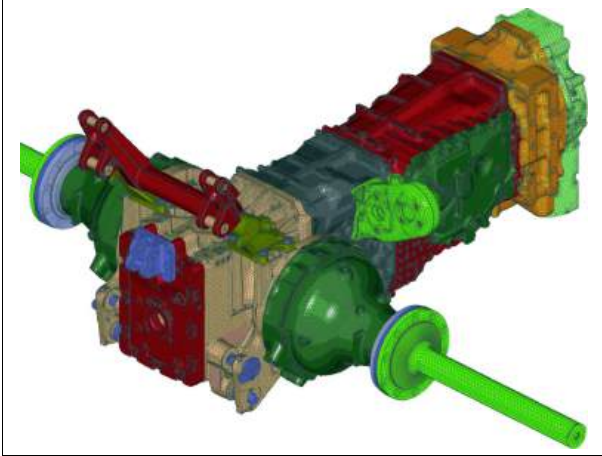


Figure 5: Model of a tractor transmission
ZF Friedrichshafen AG, Friedrichshafen, Germany.

tured companies. This makes interdependencies between different components of the product visible in time only if they are simulated and analyzed on the computer. At the same time, the quality assurance of analysis results is of great importance. Hence, the choice of the right analysis tool is of crucial significance.

Benefits of PERMAS

PERMAS is an internationally established FE analysis system with users in many countries. INTES has developed the system and, additionally, offers **individual consultation and user support** and all training required. The consultation covers all requests regarding the use of the software but also basic questions regarding the idealization and physical modeling.

The benefits arising from the use of PERMAS can be characterized by the following points:

- As a general purpose software package PERMAS provides for **powerful capabilities**, which cover a wide range of applications from mechanics to heat transfer, fluid structure acoustics and electrodynamics.
- Integrated optimization algorithms allow PERMAS not only to analyze models but also to determine optimized parts which fulfil many different conditions. The optimization methods include topology optimization, sizing and shape optimization, and reliability analysis to take into

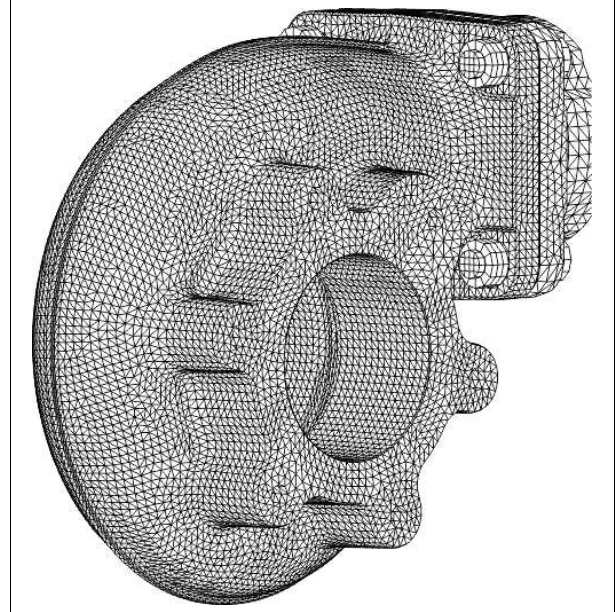


Figure 6: Turbocharger housing
BorgWarner Turbo Systems Engineering GmbH,
Kirchheimbolanden, Germany.

account uncertain model parameters.

- The graphical user interface VisPER supports the user in verifying his models and in evaluation of the analysis results. Moreover, VisPER provides advanced modeling features, e.g. for generation of fluid meshes, and for the set-up of optimization models in particular.
- Efficient equation solvers and optimized data storage schemes provide PERMAS with **ultimate computing power** with low resource consumption. Moreover, the software is continually adapted to the most advanced and powerful computers.
- PERMAS, a well-proven and mature software, has been available for many years and in numerous structural analysis departments. There, the **reliability of the software** is appreciated above all.

On the subsequent pages all these points are specified in more detail.

PERMAS is an advanced software package with up-to-date user conveniences. The PERMAS development aims to implement future-oriented functionalities in close cooperation with the users and to provide currently most advanced algorithms. In this way, PERMAS today faces the requirements of tomorrow.

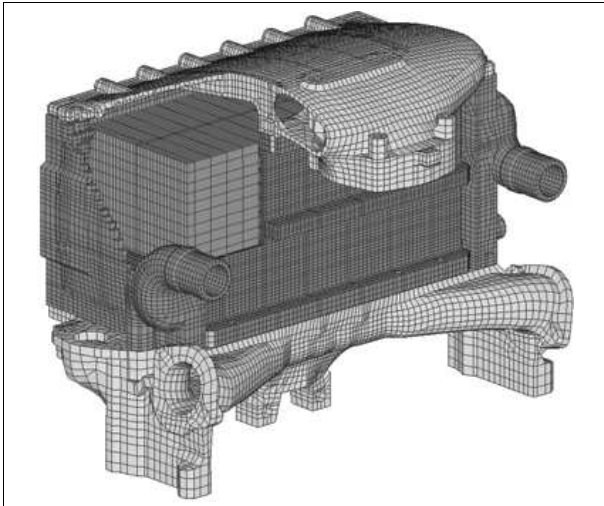


Figure 7: Charge air cooler
Behr GmbH & Co., Stuttgart, Germany.

What's New in PERMAS Version 21

Version 21 of PERMAS is the result of intensive development efforts since the shipment of the last major release, V20. It offers significant improvements in the areas of contact modeling, contact multigrid solvers, dynamic analyses, as well as data handling and optimization.

With **VisPER** (i.e. Visual PERMAS) we provide a dedicated tool to accelerate simulation workflows for PERMAS. The new features of VisPER Version 21 will be introduced in the next section (see page 14).

PERMAS4EDU — Free Education Edition

In addition to the commercial versions of PERMAS and VisPER, Release 21 introduces the new edition **PERMAS4EDU** (PERMAS for Education), available free of charge for learning and training purposes. It enables students and users to fully engage with advanced FE analysis (for non-commercial use). Examples, see also: [github repository](#)

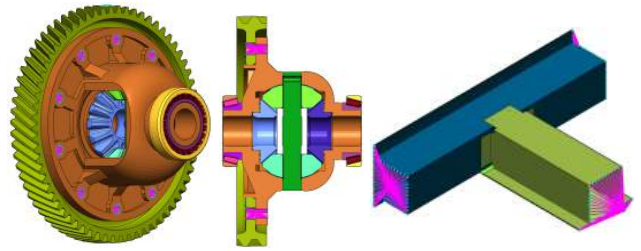


Figure 8: Two models with only one \$CONTACT AUTO[SEARCH] definition

Extensions to contact analysis (modules CA, CAX, and CAU from page 81):

- **automatic contact partner search**

\$CONTACT AUTO[SEARCH] automatically will find potential contact partners, introduces the necessary surfaces and the corresponding contact definitions in the model.

This feature dramatically speeds up contact modeling, makes the use of contact analyses more accessible, and significantly increases overall modeling efficiency and reliability due to the following reasons:

- One contact definition leads to lots of contacts, i.e. less effort is necessary to define contacts.
- An automatic handling of contact normals, kinks and already defined MPCs leads to less tries for having a correct working contact model, i.e. contact modeling is faster and more reliable.
- No manual modeling of surfaces is necessary.
- For contact definitions with shells, the normal vector of the shell elements is automatically corrected if it does not match the contact definition.
- Manual contact definitions take precedence over automatic detection

Figure 8 shows two models where only one **\$CONTACT AUTO[SEARCH]** is necessary for contact modeling. The internally created contact definitions and the generated surfaces can be exported with **ITEM CONTACT AUTO[SEARCH]**. Also refer to examples CA23 and CA24.

- **automatic surface splitting**

at large kinks, see Fig 9.

Compared to PERMAS Version 20, this feature

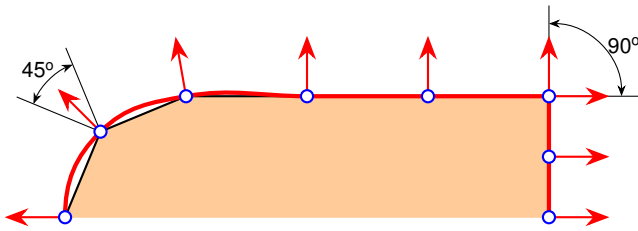


Figure 9: Automatic surface split at large kinks

substantially reduces modeling effort and improves the efficiency and robustness of model preparation. For the example shown in Fig. 10, only one surface per wheel is required. In earlier PERMAS versions, a total of 168 surfaces had to be defined manually, corresponding to four surfaces per tooth for 21 teeth on each wheel.

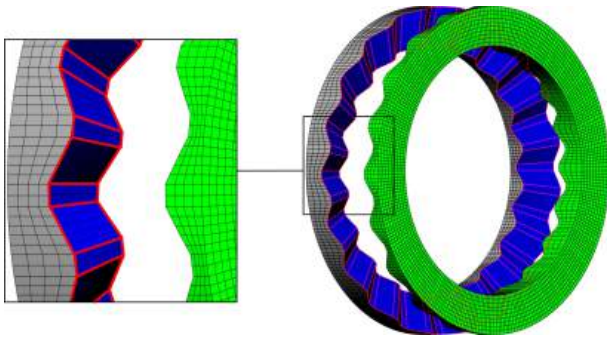


Figure 10: Contact modeling effort: Only 1 surface instead of 84 surfaces per wheel

- In contact analyses involving shell elements, the **shell thickness** is now **automatically considered** in the **contact gap width**.

Friction solver for contact multigrid analysis (module CAMG from page 86):

In PERMAS Version 20 a completely **new stabilized contact algorithm** with great performance improvements was made available for normal contact. In PERMAS Version 21 this algorithm was extended to frictional contact problems. A completely new innovative and extremely robust algorithm with high performance is provided. Still *accurate Coulomb friction* law is solved.

The runtime for this larger OEM Axle with friction and two time steps was improved by factor 3.7 from Version 20 up to Version 21. Please note the difference to solve this without CAMG (V17).

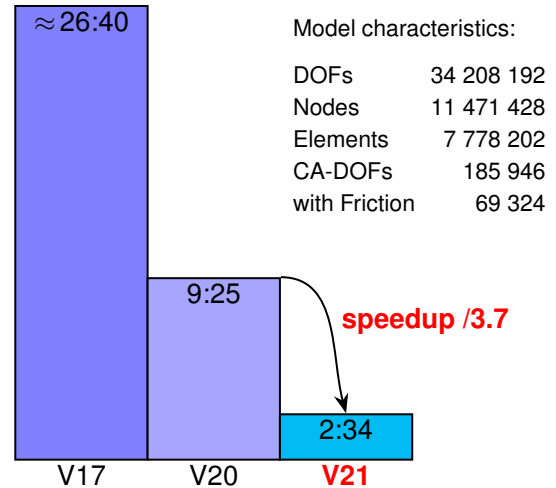


Figure 11: Performance improvement [hh:mm] for CAMG license package (friction)

Extensions to nonlinear static analysis (module NLS, page 88):

- In PERMAS Version 21 a lot of work was done to improve stability, robustness and convergence behavior of nonlinear analyses, see Fig. 12-14.

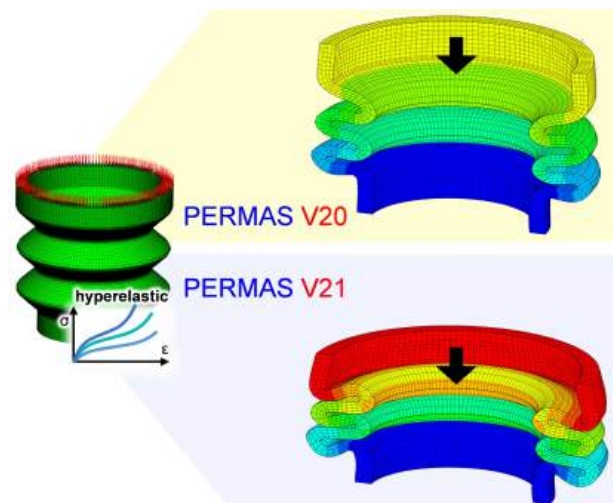


Figure 12: Stability and robustness improvement

Extensions to nonlinear static advanced analysis (module NLSA, page 92):

- A stability check of hyperelastic materials input data **\$HYPERELASTIC** using Hill's criterion for local convexity was introduced. The results are

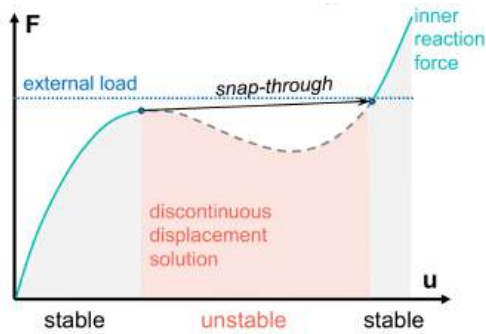


Figure 13: Snap-through solution

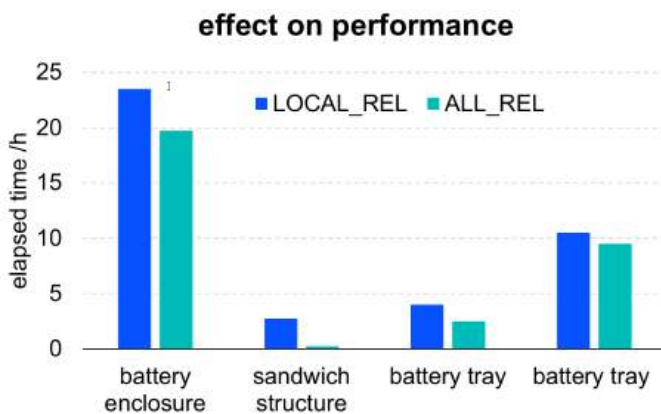


Figure 14: Effect of the convergence criterion on performance

provided as a table in the result file (.res), see also example [MNL16](#):

> Check local convexity for hyperelastic material (Hill criterion)

Stable strain range for MAT_HYEL		
load case	compressive strain	tensile strain
uniaxial	overall stable	overall stable
planar	overall stable	overall stable
equibiaxial	overall stable	overall stable

Extensions to dynamic response analysis (module DRA and HBM, pages [101](#), [107](#)):

V21 offers comprehensive extensions for dynamic response and harmonic balance analysis.

- **Dynamic contact for:**
 - a broad range of analyses
 - * [DIRECT TIMEhistory](#)
 - * [DIRECT FSCoupled TIMEhistory](#)
 - * [MODAL TIMEhistory](#)
 - * [MODAL FSCoupled TIMEhistory](#)
 - * [DIRECT HARMonic BALance](#)
 - * [MODAL HARMonic BALance](#)

- * [MODAL FSCoupled HARMonic BALance](#)
- convenient modeling like in statics
- * [Surface to Surface](#)
- * [Surface to Node](#)
- * [Node to Node](#)
- Very easy to use with rich postprocessing

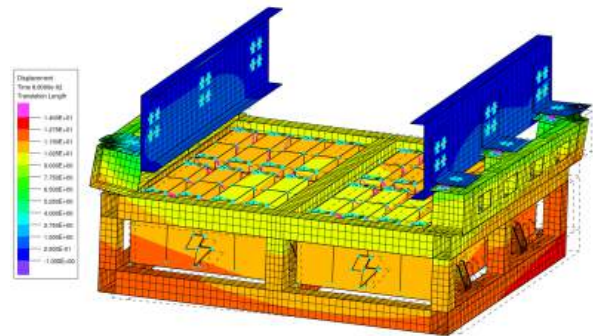


Figure 15: Dynamic contact example: Battery pack with Modal Timehistory

- **Extended loads for transient response analysis (\$TRANSLoad):**
 - Option **PREScribed = VELO/ACCE**: assign prescribed velocities or accelerations.
 - **TYPE = PSD**: assign a power spectral density (PSD) signal. It will be automatically converted to the time domain. See example [DRA25V1](#).
 - **FILTER option**: Assign a **bandpass** filter or use the **DETREND** option to remove unwanted frequency content or drift from VELO/ACCE signal, see fig. [16](#).

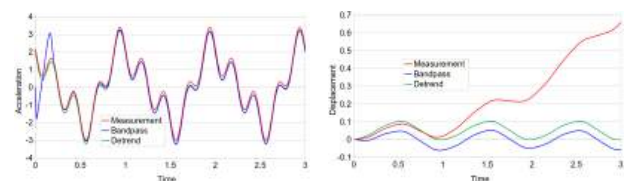


Figure 16: Excitation (acceleration) and response (displacement) of a signal for bandpass filter and DETREND option

- New time integration solvers: **TWOSTEP** and **SEMIIMPLICIT**

Extensions to fluid-structure coupled analysis (module FS, page [104](#)):

- A completely new analysis type **DIRECT FSCoupled TIMEhistory** is now available. This enables direct fully-coupled fluid-structure acoustic time analysis for high frequency problems, see fig. 17.
- Moreover, a new **DIRECT FSCoupled RANDOM** analysis is available (e.g. for diffuse noise), see fig. 18.

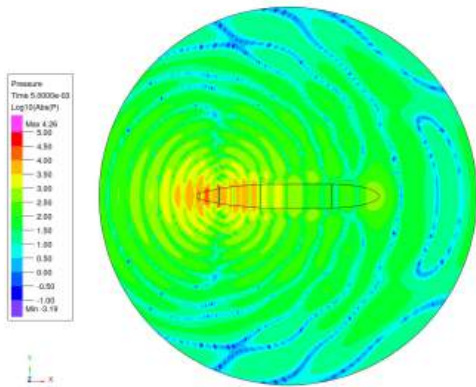


Figure 17: Pressure distribution at $t = 0.005\text{ s}$: maximum amplitude of the shock

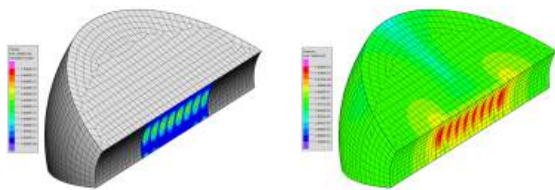


Figure 18: FS random, velocity (left) and pressure distribution (right) at 1800.0 Hz

Extensions to mode analysis (modules DEV, DEVX and MLDR, pages 97, 98 and 100):

- Check for **globality of modes**

Mode	Frequency	Globality	
1	2.69480e+02	36.14 %	
2	2.70244e+02	32.98 %	
3	2.70244e+02	32.98 %	
...	...	...	
24	2.70434e+02	32.75 %	
25	2.70439e+02	19.81 %	
26	2.70439e+02	19.81 %	
...	...	...	
39	2.70471e+02	29.63 %	
40	2.70471e+02	34.68 %	
41	5.39208e+02	72.23 %	
42	5.39208e+02	72.23 %	
43	6.24631e+02	90.54 %	
44	7.52688e+02	66.68 %	
45	7.52688e+02	66.68 %	
46	9.84031e+02	37.78 %	
...	...	...	

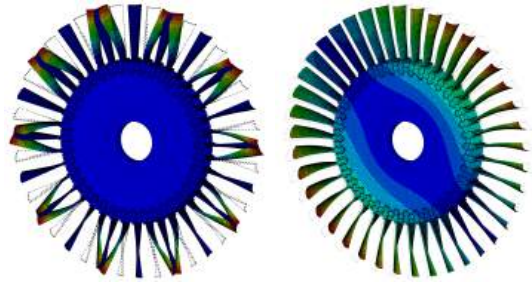


Figure 19: Local mode 25 (left) and global mode 42 (right)

- Enhanced LOG for unstable frequencies for **MODAL ROTATING**
- MLDR can handle many additional static modes (e.g. due to contact)
- MAC for complex modes to correlate with measurements, see Fig 20

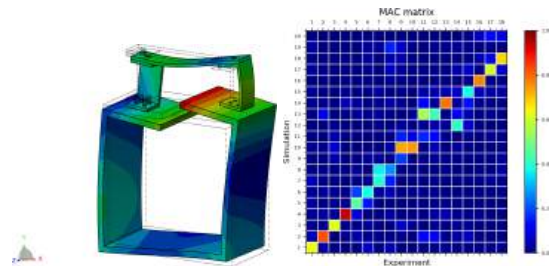


Figure 20: MAC for complex modes, BARC structure

Extensions to design optimization and topology optimization (modules OPT and TOPO, pages 109 and 114):

- Fully integrated **damage-driven optimization**

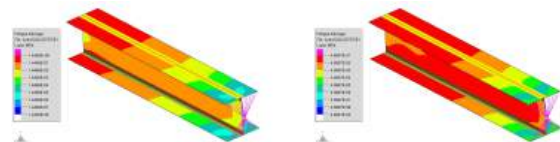


Figure 21: Initial (left) and final damage (right)

- New solver for **bead optimization**, see fig. 22.
- TOPO: Option to use integration point stresses for global stress based optimization.

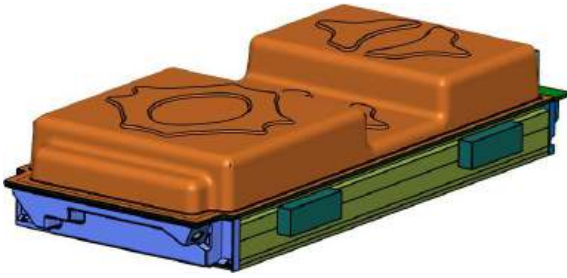


Figure 22: Bead design

PERMAS Binary Model Data (module BMOD)

A new PERMAS **binary model format** based on the HDF file standard was introduced in PERMAS Version 21.

- **Fast Model Import**

Compared to the traditional ASCII DAT format, the new binary format significantly accelerates model import, as shown in Fig. 23.

- **Efficient Storage Usage**

Compared to (compressed) ASCII and other binary formats

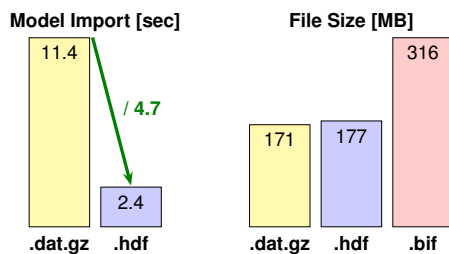


Figure 23: Performance (left) and file size (right) for PERMAS binary model with 4.5 Mio nodes & 2.5 Mio elements

The available workflows for generating and importing PERMAS binary model data are illustrated in Fig. 24.

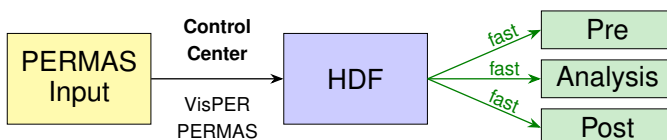


Figure 24: Workflow for PERMAS binary model

Extensions to basic module (module MQA, page 79):

A lot of UCI and DAT file command extensions took place, see software release note for details.

- **Matrix input**

The direct matrix input definition, **\$MATRIX**, has been relocated from **\$SYSTEM** variant to **\$CONSTRAINTS** variant. In addition, several new features have been introduced, such as:

- support for both symmetric and non-symmetric matrices,
- support for mass and damping matrices (besides stiffness),
- automatic generation of nodal degrees of freedom (DOFs) when they are not already defined,
- consideration of matrix entries during re-ordering and MLDR processing.

What's New in VisPER Version 21

VisPER (i.e. Visual PERMAS) is a dedicated tool to provide analysis workflows (see Figs. 83 and 71) and to improve pre- and post-processing for PERMAS. VisPER is released together with PERMAS 21. More information on VisPER can be found from page 41.

We are able to offer with MeshWorks (DEP) a mesher/preprocessor that is integrated as third party product within our licensing system. The PERMAS Control Center (permascc) has been extended accordingly.

- **Bead optimization workflow:**

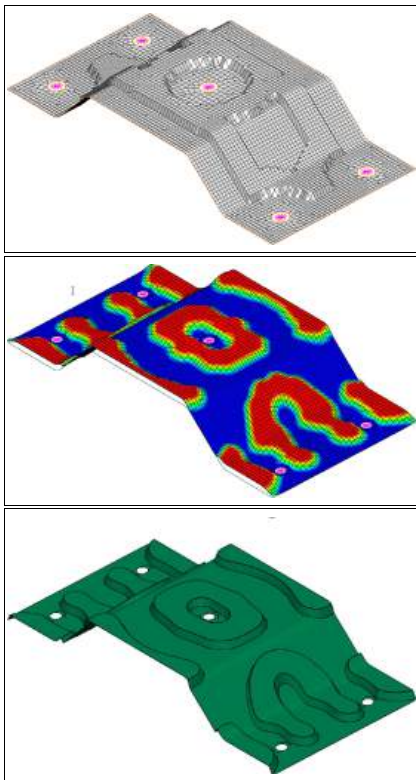


Figure 25: Bead and bead design wizard

- VisPER Supports the setup of a bead optimization (*BeadWizard*) as well as bead post-processing (*BeadDesignWizard*), see figure 25.
- **Model verification:**
 - Quick model insight by exploded view visualization. See figure 26
 - Fully integrated check of mass/inertia data for structural and fluid mass, see fig. 27. With convenient handling of substructure models
 - **Model Information Tree** with complete

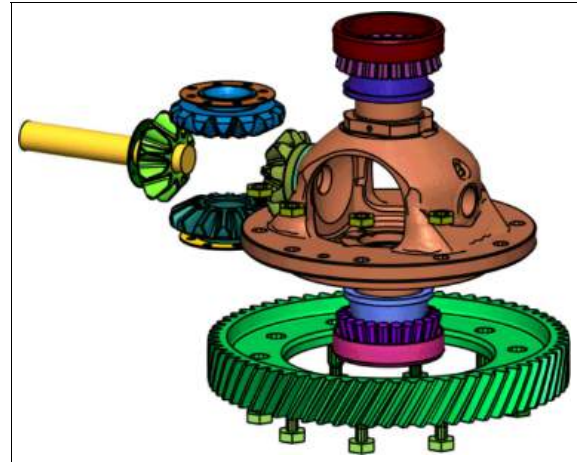


Figure 26: Exploded view

model info on all supported and unsupported items

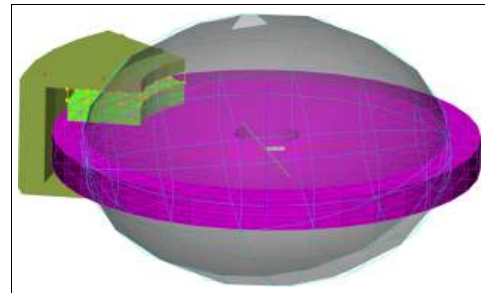


Figure 27: Mass/inertia characteristics

- **Model completion:**
 - Edit of material properties.
 - Improved dialogs, e.g. for MPCs.
- **Model Update:**
 - Automatic model update for sizing and shape optimization, see figure 28. For sizing, new geodat and elprop assignments are performed, while a coordinate update may be performed for shape.
- **Wizards:**
 - CA Wizard
 - * Support of contact parameters for dynamic contact.
- **Graphical User Interface:**
 - Configuration of user defined toolbars possible.
- **Post-processing:**
 - New Views Dialog Bar to handle view settings.
- **Tools:**
 - Using PYTHON scripting enables automated post-processing including image and video

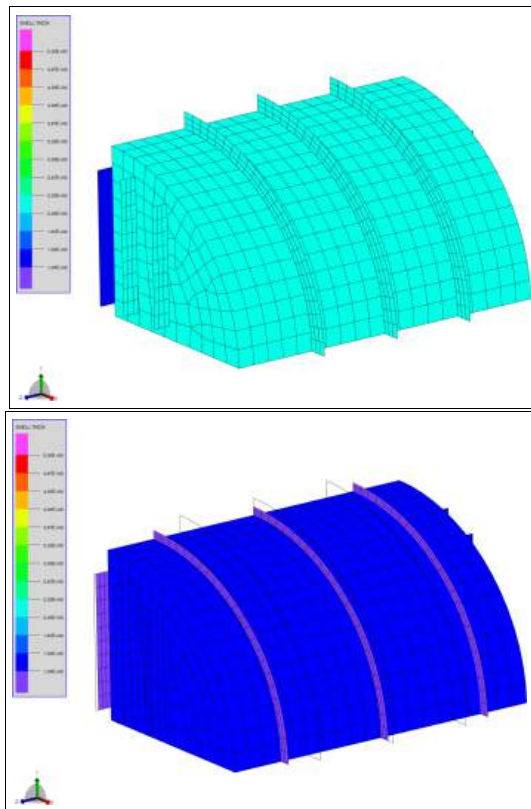
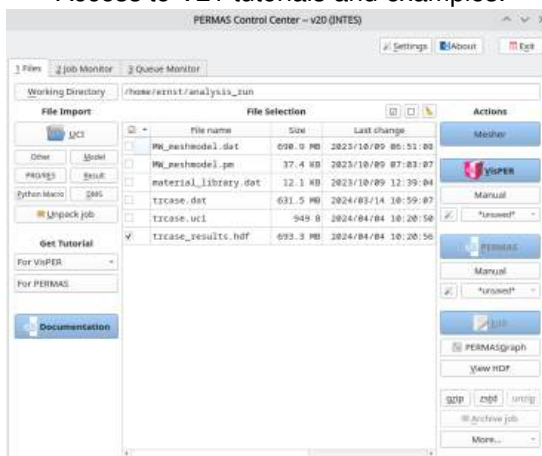


Figure 28: Sizing and Shape Update

generation. Model information, images, and videos may be exported to MS Word, Excel, and PowerPoint.

– PERMAS Control Center

- * Revised and extended workflow tool.
- * Configurable mesher and further function plugins.
- * Access to V21 tutorials and examples.



Universal Features

The outstanding mostly module-independent basic features of PERMAS are as follows (see pages 59 to 76):

- Hierarchical substructuring, with automatic sub-component insertion (see page 59)
- Submodeling (see page 59)
- Variant analysis (see page 60)
- Cyclic symmetry (see page 61)
- Surface and Line Description (see page 61)
- Automated coupling of parts (see page 62)
- Automated spotweld modeling (see page 63)
- Local coordinate systems (see page 64)
- Multiple kinematic constraints (see page 64)
- Automatic detection of singularities (see page 65)
- Same elements for different analysis types (element library, see page 66)
- Standard beam cross sections (Seite 67)
- Design elements for optimization (page 67)
- SPR Stresses and Absolute Error Indicator (page 68)
- General material description (see page 68)
- Node and element sets (see page 69)
- Mathematical functions (see page 69)
- All kinds of loading (see page 70)
- Model verification (see page 71)
- Integrated interfaces to pre- and post-processors (see page 72)
- Input and Output of Data Objects and matrices (see page 73)
- Combination, transformation, and comparison of results (see page 74)
- Output of XY result data (see page 75)
- Calculation of cutting forces (see page 75)
- System of units (see page 75)
- Restart facility (see page 76)
- Open software through Fortran and C interfaces (see page 76)
- Direct coupling of different analysis types (see page 76)
- Coupling with CFD (see page 76)

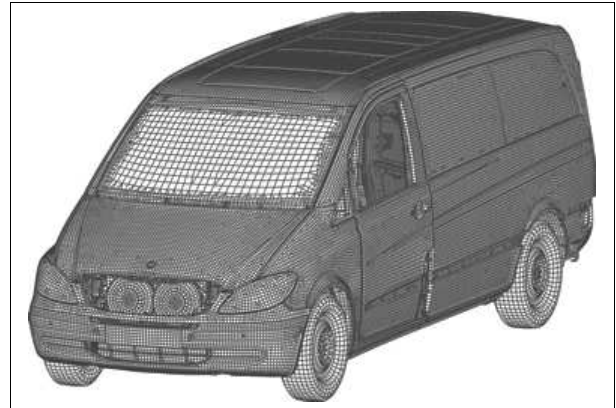


Figure 29: Model of a transport vehicle
courtesy of Daimler AG,
Commercial Vehicle Division in Stuttgart

Available VisPER Modules

The below listed functional modules are explained in more detail on pages 42 to 54:

- | | |
|------------------------------|----------|
| • Basic module | (VBAS) |
| • Topology optimization | (VTOP) |
| • Design optimization | (VOPT) |
| • Fluid-structure coupling | (VFS) |
| • Contact analysis | (VCA) |
| • PERMAS Component generator | (VPCGEN) |

Available PERMAS Modules

The below listed functional modules are explained in more detail on pages 79 to 131:

- | | |
|--|----------|
| • Model Quality Assurance | (MQA) |
| • Linear Statics | (LS) |
| • Contact Analysis | (CA) |
| • Extended Contact Analysis | (CAX) |
| • Contact Geometry Update | (CAU) |
| • Contact Multigrid Solver | (CAMG) |
| • Nonlinear Statics | (NLS) |
| • Extended Nonlinear Material Laws | (NLSMAT) |
| • Advanced Nonlinear Statics | (NLSA) |
| • Buckling Analysis | (BA) |
| • Heat Transfer | (HT) |
| • Nonlinear Heat Transfer | (NLHT) |
| • Dynamic Eigenvalue Analysis | (DEV) |
| • Extended Dynamic Eigenvalue Analysis | (DEVX) |
| • Eigenmodes with MLDR | (MLDR) |
| • Dynamic Response Analysis | (DRA) |

- Extended Dynamic Response Analysis (DRX)
- Harmonic Balance Method (HBM)
- Fluid-Structure Acoustics (FS)
- Nonlinear Dynamics (NLD)
- Design Optimization (OPT)
- Layout Optimization (TOPO)
- Advanced optimization solvers (AOS)
- Reliability Analysis (RA)
- Laminate Analysis (LA)
- Fatigue Analysis (LIFE)
- Refined Weldspot Model (WLDS)
- Aeroelasticity (AERO)
- Steady-state electromagnetics (EMS)
- Electrodynamics (EMD)
- Use of GPU (XPU)
- Interfaces to various pre-/post-processors
 - MEDINA (MEDI)
 - PATRAN (PAT)
 - I-DEAS (ID)
- Interfaces to other analysis packages
 - ADAMS (AD)
 - DADS (DADS)
 - SIMDRIVE3D (SIM3D)
 - SIMPACK (SIM)
 - EXCITE (EXCI)
 - MOTIONSOLVE (HMS)
 - HYPERVIEW (H3D)
 - VAO (VAO)
 - Virtual.Lab (VLAB)
 - ADSTEFAN (ADS)
 - MATLAB (MAT)
 - NASTRAN (NAS)
 - ABAQUS (ABA)
 - MpCCI (CCL)

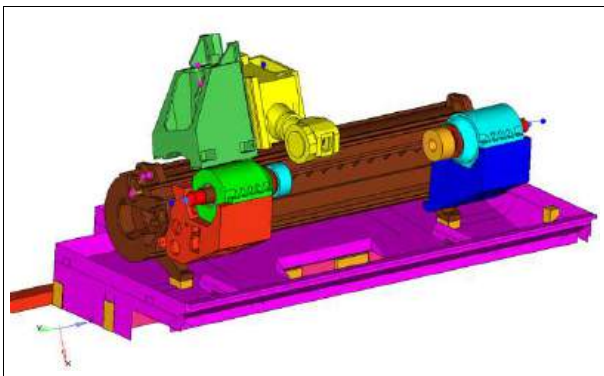


Figure 30: Machine tool
INDEX-Werke GmbH & Co. KG, Esslingen, Germany

Performance Aspects

By ongoing further developments of the equation solvers PERMAS achieves a very high computation speed. Both, direct and iterative solvers, are continuously optimized.

- Very good multitasking behavior due to a high degree of computer utilization and a low demand for central memory.
- The central memory size used can be freely configured – without any limitation on the model size.
- The disk space used can be partitioned on several disks – without any logical partitioning (e.g. optimum disk utilization in a workstation network).
- There are practically no limits on the model size and no explicit limits exist within the software. Even models with many million degrees of freedom can be handled.
- By using well-established libraries like BLAS for matrix and vector operations, PERMAS is adapted to the specific characteristics of hardware platforms and thus provides a very high efficiency.
- Another increase of computing power has been achieved by an overall *parallelization* of the software.
- By simultaneous use of several disks (so-called disk striping) the I/O performance can be raised beyond the characteristics of the single disks.
- PERMAS can be invoked using an option for using direct I/O. If PERMAS DMS-Files are put on SSD systems, the I/O is performed directly to these systems, which can reduce run time for I/O bound jobs significantly (see Fig. 36).
- Disk I/O can be avoided at all, if large memories are used. The memory size may readily exceed 1 TiB.

Parallelization

PERMAS is also fully available for parallel computers. A general parallelization approach allows the parallel processing of all time-critical operations without being limited to equation solvers. There is only **one** software version for both sequential and parallel computers.

PERMAS supports the parallelization on shared memory computers. There, the parallelization is based on POSIX Threads, i.e. PERMAS is executed in several parallel processes, which all use the same memory area. This avoids additional communication between the processors, which fully corresponds with the overall architecture of such systems.

In addition, PERMAS allows asynchronous I/O on this architecture, which realizes better performance by overlapping CPU and I/O times.

Parallelization does not change the sequence of numerical operations in PERMAS, i.e. **the results of a sequential analysis and a parallel analysis of the same model on the same machine are identical** (if all other parameters remain unchanged).

PERMAS is able to work with constant and pre-fixed memory for each analysis. This also holds for a parallel execution of PERMAS. So, several simultaneous sequential jobs as well as several simultaneous parallel jobs or any mix of sequential and parallel jobs are possible.

The parallelization is based on a mathematical approach, which allows the automatic parallelization of sequentially programmed software. So, PERMAS remains generally portable and the main goal has been achieved: *One single* PERMAS version for all platforms.

Parallel PERMAS is available for all platforms, where a sequential version is supported, too.

The parallelization on several cores can be extended by the use of a GPU (Graphical Processing Unit) of Nvidia, where a Tesla K20c or better is supported. For compute bound solution steps, the GPU can essentially accelerate the analysis run (see also module XPU on page 125).

The parallel execution of PERMAS is very simple. Because there are no special commands neces-

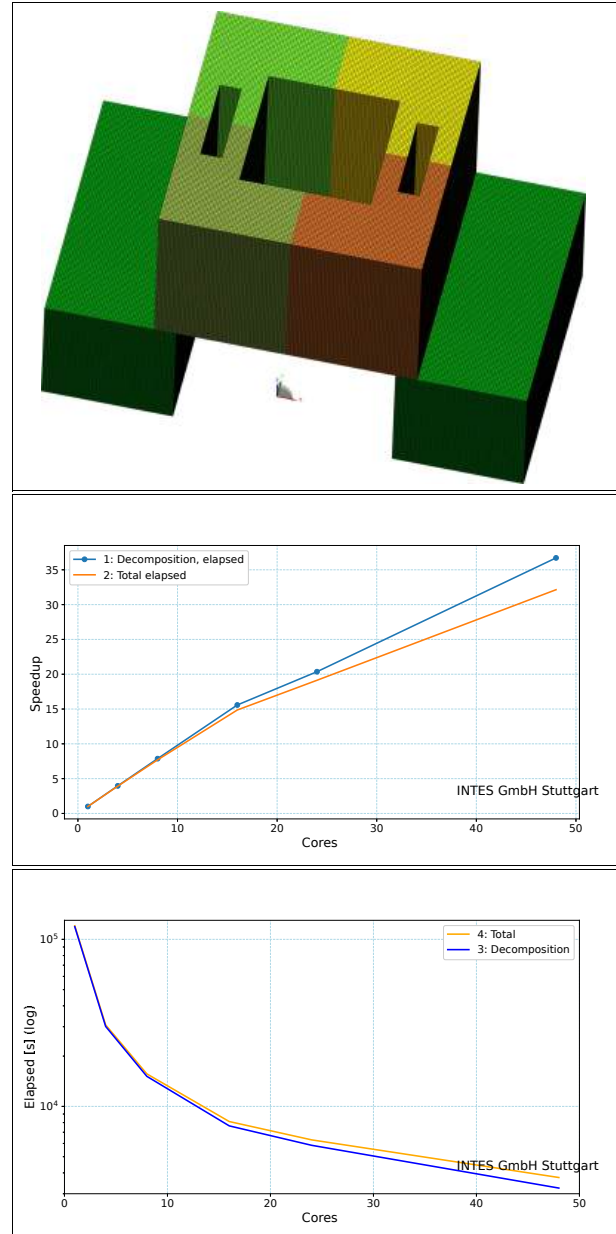


Figure 31: Static analysis with 5 loading cases
11.6M nodes, 1.4M HEXE27, 34.8M Dof
speedup and runtime on 2*24 Intel Xeon 8260L

sary, a sequential run of PERMAS does not differ from a parallel one - except for the shorter run time. Only the number of parallel processes or processors for the PERMAS run has to be defined in advance.

Areas of Application

Presently, PERMAS is used in the following branches of industry:

- Automotive industry
- Aerospace industry
- Ship building industry
- Mechanical engineering
- Offshore- and power plant engineering
- Plant- and equipment engineering

Reliability

Nowadays, not all results of FE analyses can be proven by experiments. They are often directly used in the development process. Moreover, the models become more and more complex and the results have to be produced faster and faster. Early detection of possible modeling errors and their elimination means a great challenge to the analysis software. To this end, PERMAS and VisPER make a substantial contribution.

- **Robustness of the software:** Low system error rate due to advanced software engineering methods and intensive software testing.
- **Model verification:** The basic PERMAS-MQA module provides tools for model quality assurance (see page 79). Beside automatic model testing, many quantities and model properties can be exported for visualization and checking in a postprocessor (see section Model Verification on page 71). In addition, VisPER provides a model verification environment for a growing number of modeling parameters (see page 42).
- **Safe use:** Expensive faulty runs are avoided by the task scanning concept of PERMAS-MQA. Firstly, these give an estimation of the necessary computer resources, which allow for a more reliable planning of large model analyses. In addition, numerous modeling deficiencies can be detected, which directly improves the reliability and quality of the subsequent analysis.
- **Correctness of results:** The quality of results is ensured by comprehensive and continuous verification (using the tests of NAFEMS and SFM).

Above all, the application of well-proven algorithms and esteemed development tools results in the high quality of the software.

A broad traditional PERMAS user base from different branches of industry essentially contributes to the reliability of the software.

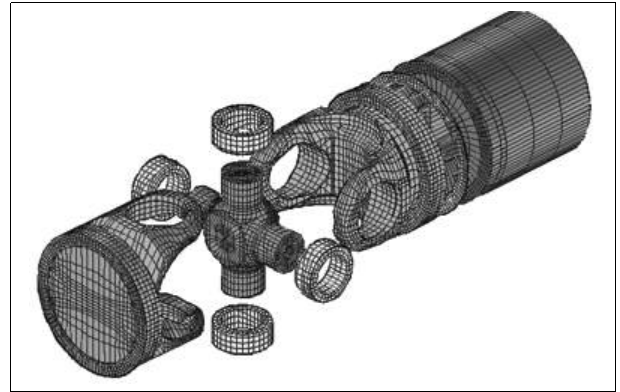


Figure 32: Model of a cardan shaft
Voith Turbo GmbH & Co. KG, Heidenheim, Germany.

Quality Assurance

INTES develops high quality software und offers all related services. All phases of the software development are performed on the basis of established standards and appropriate tools in order to achieve a maximum of product quality.

Some important aspects of quality assurance are:

- Especially developed for the management and administration of the software, a development tool provides for a safe software database, which includes all modifications and new sub-routines and manages them in a unique and apprehensible way.
- A problem report management system gathers all messages regarding software problems and development requests as well as other user requests together with the subsequently elaborated solutions and responses. A 'Technical Newsletter' issued regularly informs the users about all inquiries made and the pertinent solutions.
- An ever growing library of software test runs daily ensures the equally high quality of the software. Problem cases extracted from the problem report management system lead to an extension of the test library in order to preclude the re-occurrence of problems handled in the past.

Applications

Car Body Analysis

Finite Element Analysis of car bodies comprise a broad variety of modeling levels from *BIW (body-in-white)* to trimmed bodies and acoustic models taking into account enclosed and even surrounding air. This variety of structural variants corresponds to different targets from simple stiffness issues up to complex comfort tasks. Therefore, a lot of different methods are applied in car body analysis ranging from linear static analysis up to fluid-structure coupled acoustics.

A typical characteristic of car body models is the use of shell elements. Most frequently, quadrangular linear shell elements are used (together with triangular shell elements). Dependent on the mesh size, up to several million shell elements are used to model car bodies. A car body consists of a larger number of structural parts (typically 50 to 100) which are joined by different techniques like spot welding (see page 63 and module WLDS on page 123), bonding, laser welding. In order to generate the meshes of all parts efficiently, *incompatible meshing* (see page 62) is used for independent meshing.

A special feature in VisPER supports post-processing of spotwelds (see page 56) in very large body structures.

Static analysis

For computations of static stiffness of a car body, linear static analysis is used. For some load cases like towing or light impact calculations of *inertia relief* (see page 81) are applied.

To check the force flow through any structural member, *cutting forces* (e.g. through a column or sill, see page 75) can easily be derived and a summary of the forces and moments is exported (and printed).

Dynamic analysis

It is an important issue in dynamic analysis that all masses are taken into account. The matching of masses between the real structure and the simulation model is very important. Masses and moments of inertia can be calculated by the simulation and compared to the expected values.

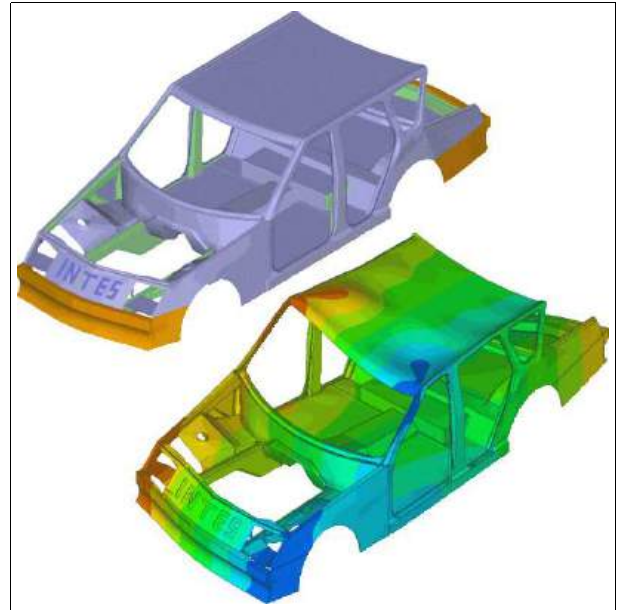


Figure 33: Workshop example INTEScar under torsional loading

An eigenvalue analysis is performed as a basis for subsequent response analysis. Because cars are not supported on ground, a free-free vibration analysis has to be performed. A check on the rigid body modes is highly recommended and supported by corresponding printed information. The frequency range for the eigenvalue analysis depends on the intended frequency range of the subsequent response analysis. A certain factor (2 to 3) on the intended frequency range is frequently applied in order to get good response results over the full frequency range.

Flexible bodies are often incorporated in *MBS (Multi-Body Systems)* models. Usually, this is done on the basis of modal models. PERMAS supports a number of interfaces to export flexible bodies in special formats (see page 128).

Due to the cut of eigenfrequencies beyond the frequency range, response results can be insufficient in the quasi-static range (between zero frequency and first eigenfrequency). This quasi-static response can be improved by taking relevant static mode shapes which are computed automatically from given static load cases (see page 103).

Structural modifications of the car body (BIW) are usually done for only a few parts, e.g. the front of the car. Then, there is no need to repeat the full analysis of the car from scratch but the rear car can be

reduced by dynamic condensation (see page 98). Using dynamic condensation so-called *matrix models* (see page 73) are generated which represent the reduced part of structure. These matrix models are used in each analysis of the remaining structure. In this way, run time for variants (e.g. of the front car) is reduced drastically.

For the subsequent response analysis (see page 101), there are methods for the frequency domain (i.e. frequency response analysis) and for the time domain (i.e. time-history response analysis). These methods are available as modal methods (based on previously determined eigenfrequencies and mode shapes) and as direct methods (based on full system matrices). For realistic models, the direct methods are much more time consuming than modal methods. But the direct methods are very accurate and can be used on a case-by-case basis to check the accuracy of the modal models.

The dynamic loading (or excitation) can be specified by forces (and moments) or prescribed displacements (or rotations) and a frequency or time function which describes the course of the excitation dependent on frequency or time.

- In frequency domain, the discretization of the excitation frequency range is an important accuracy parameter for the resulting response graphs. In particular, the discretization of peaks is important and this is supported by generation of clusters of excitation frequencies around eigenfrequencies.
- If a time function is provided by measurements, beside a time-history response an alternative approach is also available to get a periodic response result. An internal *FFT (Fast Fourier transformation)* is available to detect the main excitation frequencies. For each of these frequencies a frequency response can be performed (with just one excitation frequency). The result of all these harmonic response results can then be superimposed in the time domain to get the periodic response (or *steady state response*). Fig. 126 shows an example.
- In time domain, the sampling rate should be related to the time characteristics of the excitation function.

For response analysis, the specification of damping is very important. There are a lot of ways to specify damping (see page 102). In particular, trimmed

bodies require a detailed and accurate modeling of all additional springs, masses, and dampers.

The results from a frequency response analysis are any complex primary result (displacements, velocities, or accelerations) and secondary result (e.g. stresses, strains, sound radiation power density) for all nodes at any excitation frequency. Frequently, so-called *transfer functions* are more important than the full fields of result quantities. Transfer functions describe the relation between the excitation points and any target point of interest (by a unit excitation) for all excitation frequencies.

In order to reduce computational effort for response analysis the user can specify the requested results in advance. In case of requested transfer functions, the response analysis can be restricted to just a node set.

Fluid-structure dynamics

Coupled simulation of structure and air is seen as natural extension of structural dynamics. This extension is needed, because noise in a car is a combination of structural-borne and air-borne noise. Noise at the driver's ear is important for the comfort and the acoustic quality of a car.

As a first step the interior of the car is modeled by so-called fluid elements which are classical volume elements but with a pressure degree of freedom. In order to model the coupling between structure and air physically, there are additional coupling (or interface) elements which contain both the displacement and pressure degrees of freedom and represent the physical compatibility condition between structure and air.

To facilitate the two modeling steps for fluid and coupling elements of the car interior, VisPER contains an easy-to-use wizard starting from the structural mesh and generating the fluid mesh and the coupling elements step by step in an almost automatic way (see page 51). Typically, the coupling elements are compatible with structural elements of the interior surface, but the fluid elements representing the enclosed air are incompatibly meshed, because the mesh for the air is usually much coarser than for the structure. The wizard derives the appropriate element edge length from the requested frequency range.

The fluid may contribute to the damping by so-called

volumetric drag which represents the absorption in a fluid volume. The coupling elements contribute to the damping by surface absorption which represents a normal impedance of the coupling surface.

After completing the fluid-structure model, the analysis steps are very similar to structural dynamics of cars as described above (see also page 104 for more functional details).

- A coupled eigenvalue analysis is available to derive the coupled eigenfrequencies and mode shapes. The mode shapes consist of two corresponding parts, a displacement mode shape of the structure and a pressure mode shape of the fluid.
- Excitations can now also be specified in the fluid by a pressure signal.
- Based on coupled eigenfrequencies and mode shapes, modal frequency response analysis and modal time-history response analysis can be performed in the same way as for the sole structure.

In addition to modal methods, also a direct frequency response is available for fluid-structure coupled analysis.

From the coupled response results, all results as described for structural response calculations can be obtained. In addition, the pressure field in the air and transfer functions from structural points to pressure points are available (and vice versa). Moreover, sound particle velocities (as vector field or magnitudes) can be derived from the pressure field.

In addition to enclosed air in a car, the surrounding air can also be modeled and coupled to the structure. This feature can be used to calculate noise transition through the structure (from the road or from air flow induced noise to the driver's ear).

High performance

Continuous effort is spent in improving and accelerating the speed of algorithms. In car body analysis emphasis is put on the following achievements:

- For large models (millions of degrees of freedom) and many modes (thousands of modes), eigenvalue analysis is made much faster by *MLDR (Multi-Level Dynamic Reduction)*. Details can be found on page 100. This method is available for both structural dynamics and coupled fluid-structure dynamics.

- In frequency response analysis many different dynamic load cases (several hundreds) are often applied. So-called *assembled situations* (see page 103) are used to solve these load cases simultaneously instead of one after the other.
- In frequency response analysis the equation solving can be made much faster (for a high number of modes and many excitation frequencies) using an iterative solver.

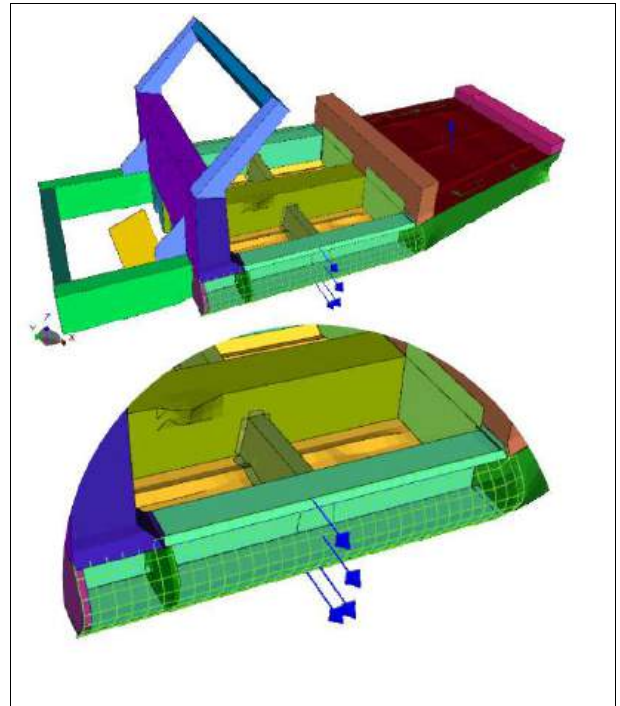


Figure 34: Shape optimization of a sill with transition to neighbored parts

Optimization

Supported by VisPER and PERMAS, optimization tasks for the car body can be solved in an integrated way. So, the optimization model is part of the model description and can easily use all available references to existing model parts like node and element sets. Although all available optimization types (as described on pages 109 to 119) can be used for car bodies, the most important ones are as follows:

- **Sizing:** This is used to optimize element properties like shell thickness, beam cross section, spring stiffness, and damper properties.
- **Shaping:** This is used to optimize geometry of parts by modifying node coordinates (also pos-

sible with incompatible meshes).

- **Bead design:** This is used to position and shape beads in shell structures (see example in Fig. 35).

All these optimization types can be combined in one optimization project. Static and dynamic analysis can be used simultaneously for optimization tasks. The optimization modeling is fully supported by VisPER (see details on page 49). Even post-processing of optimization results can be made with VisPER.

Optimization of transfer functions due to sizing, shaping, and bead design is of major importance in dynamic analysis. This frequency response optimization can be used with an objective transfer function (i.e. a frequency dependent limit of amplitudes).

If the objective transfer function is derived from experimental results, then the optimization process is named *model updating*. By this process selected model parameters are modified in order to fit the simulation transfer function to the experimental one.

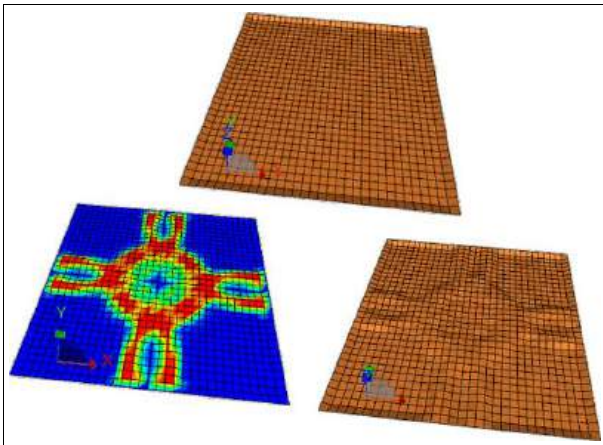


Figure 35: Bead design of a plate with positioning and height of beads

Engine Analysis

Many physical effects play an important role during a mechanical analysis of combustion engines. In static analysis such effects are leak tightness and durability under changing temperature conditions and in dynamic analysis there are sound radiation and frequency responses of complex engine assemblies. At least in static analysis the influence

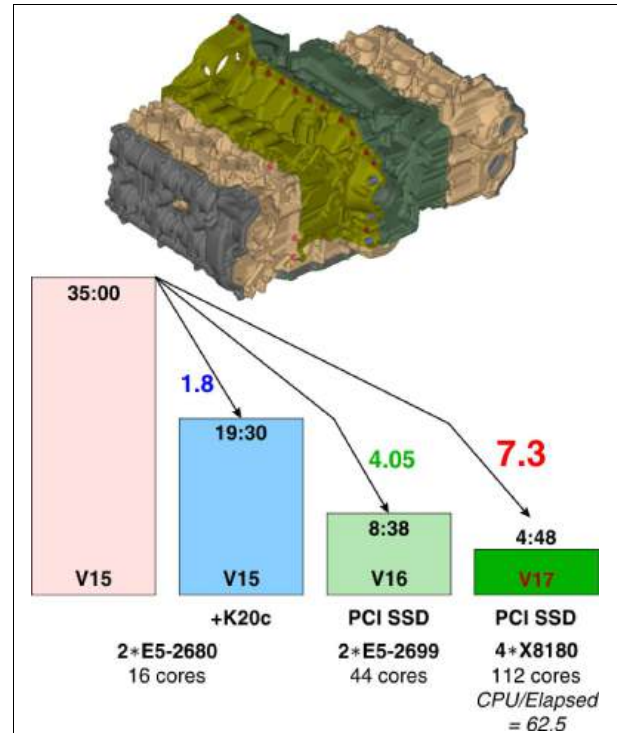


Figure 36: Contact analysis of an engine with 56 Million DOF, 37 time steps, 2 different temperature states, and CAS files, direct I/O with PCI SSD.

of temperature requires a coupled analysis taking heat transfer into account. Modeling the mounting of an engine requires the consideration of bolt loading conditions where the correct sequence of bolt pre-stressing and operating loads is of major importance. In addition, nonlinear material behavior has to be considered. These and other effects are important for engine analysis.

Heat Transfer

Applications are e.g. the analysis of operating temperatures and the aging in an oil bath by simulating the cooling down process. The following features are available:

- Nonlinear material behavior with temperature-dependent conductivity and heat capacity,
- Temperature-dependent heat convection for the modeling of heat exchange with the surrounding,
- **Automatic solution method** for nonlinear heat transfer with **automatic step control** and several convergence criteria, i.e. an automatic load stepping for steady-state analyses and an auto-

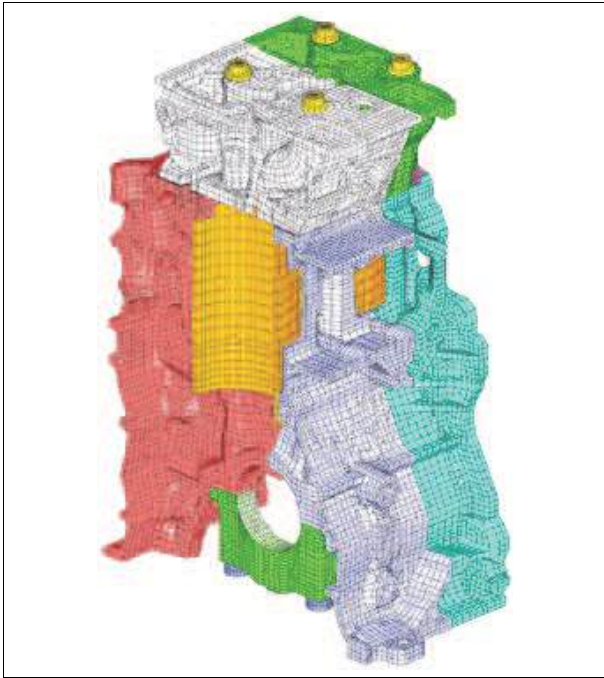


Figure 37: Simple engine model
Daimler AG, Commercial Vehicle Division)

matic time stepping for transient analyses,

- Convenient and very detailed specification possible for loading steps and points in time where results have to be obtained,
- Full coupling to subsequent static analysis (steady-state and transient),
- Heat exchange by radiation can be included, if this makes a relevant effect on the temperature field.
- If temperature fields are available for cylinder head and engine block, then other parts may not yet have temperatures, like gaskets or bolts. Then *temperature mapping* procedures using the submodel technique are available to provide all parts with proper temperatures (see submodeling on page 59).

Statics

Static deformations are calculated under various loads with linear and nonlinear material behavior:

- Nonlinear material models:
 - plastic deformation,
 - nonlinear elastic,
 - creep,
 - cast iron with different material behavior under tension and compression.
- *Gasket elements*:

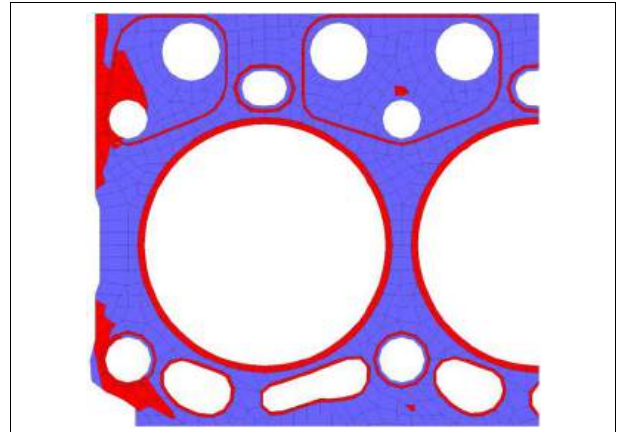


Figure 38: Contact status for gasket elements

- for convenient simulation of sealings,
- the behavior of sealings is described by measured pressure-closure curves,
- input of many unloading curves possible.
- Contact analysis:
 - many contacts possible ($> 100,000$),
 - unrivaled short run times,
 - most advanced solver technology,
 - friction can be taken into account with transitions between sticking and sliding,
 - bolt conditions can be applied in one step,
 - specification of a realistic loading history,
 - If an engine has many parts, which are connected only by contact, the RBM assistant in VisPER helps to avoid rigid body modes by applying compensation springs (see page 45).
 - contact results: contact pressure, contact status, contact forces, saturation, etc..
- *Submodeling*:
 - for subsequent local mesh refinements,
 - automatic interpolation of displacements to get kinematic boundary conditions for a finer mesh,
 - then, a local analysis is performed e.g. to achieve more accurate stresses.

High performance

Due to typically large models in engine analysis all analysis methods are oriented towards highest possible performance. The following points can be highlighted:

- outstanding performance through special algorithms for large models with nonlinear material and contact,

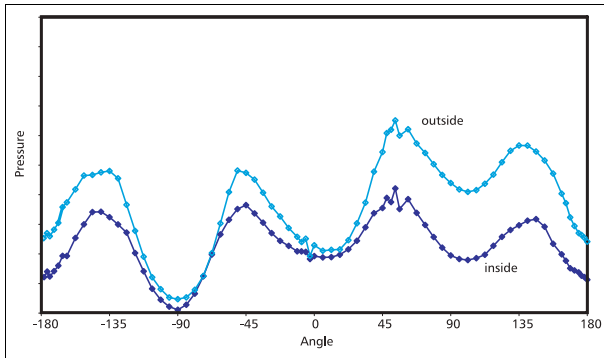


Figure 39: Pressure distribution at stopper over the angle

- contact algorithms have been strictly designed to meet the needs of large models with many contacts,
- unrivaled fast method for linear material and contact.
- Gasket elements can be handled as integral part of the contact iteration instead of a feature in nonlinear material analysis (i.e. **CCNG analysis**, Contact Controlled Nonlinear Gasket analysis). If no other material nonlinearities are present in the model, run time reduction factors can be higher than 10 (e.g. for analysis of combustion engines with pretension, temperature loads, and cylinder pressures). In cases, where other material nonlinearities are present in the model, a run time reduction by a factor of about 2 can still be achieved.
- An additional speed-up can be obtained, if a contact analysis is repeated. The resulting contact status of a contact analysis is stored in so-called **contact status files**. These contact status files may be used as starting point for the subsequent contact analysis. In case of small changes, this will essentially reduce the run time of the new contact analysis.
- If several temperature fields are used several times in an engine analysis, e.g. to calculate several load cycles with different temperatures like in a cold and hot engine, then a special algorithm can be used to accelerate the analysis significantly (see Fig. 36).

Dynamics

By using the same software for dynamic and static simulations only one structural model is necessary. All dynamic methods are available for engine analysis (see pages 97 to 104). Some important points

are:

- Eigenvalues and mode shapes for large solid models can be calculated using MLDR (see page 100).
- Fast **dynamic condensation** methods support the efficient analysis of engines with many attached parts (DEVX, see page 98).
- By using **dry condensation** (page 98) even fluids can be integrated in a dynamic model without taking along pressure degrees of freedom (e.g. in an oil pan).
- Calculation of sound particle velocity is supported for the evaluation of noise emission of engines.

In order to facilitate the transition from static analysis with contact to dynamic analysis, a **contact locking** feature is provided (see page 85). By using this feature, the results of any loading state in static analysis of an engine can be used for a subsequent dynamic analysis. A contact pressure dependent threshold value for the locking of contacts is available to fit dynamic results to experiments, if necessary.

Part Connections

The modeling of part connections essentially determine the quality of simulation results. On the other hand modeling of connection details is sometimes complex and time consuming. Hence, analysts want to have simplified models for various connections giving satisfactory simulation results. Consequently, part connection is a typical modeling feature in the area of tension between modeling effort and result quality.

There are two different classes of part connections which will be subsequently described in more detail:

- Structural connections,
- Connection elements.

Structural Connections

Parts can be coupled at their surfaces in different ways:

- For the connection of e.g. two cylindrical parts, where the inner part has a slightly greater outer diameter than the inner diameter of the outer part, a press fit connection can be used, where the interference is directly modelled without contact. As a consequence, a press fit connection will remain under all loads.
- Tied coupling (i.e. coupling remains under both tension and compression). This is typically achieved by kinematic constraints (see page 64).
- Contact (i.e. connection can open and close during loading). This is the topic of contact analysis (see page 81).

In all cases, the surfaces of coupled parts can be meshed compatibly or incompatibly (see the part coupling on page 62). The latter is an advantageous feature reducing modeling effort, because parts can be meshed independently.

Tied coupling can be used with all analysis types (like static and dynamic analysis). But in case of contact, e.g. a subsequent dynamic analysis needs one additional analysis step. This step includes a contact analysis where the final contact status is locked (i.e. *contact locking*, see page 85). In this way, a linearization of the contact problem is achieved. Fig. 40 describes the process of dynamic analysis for engine structures under pretension load. Before bolt pretension is applied, the parts of an engine can vibrate separately, but af-

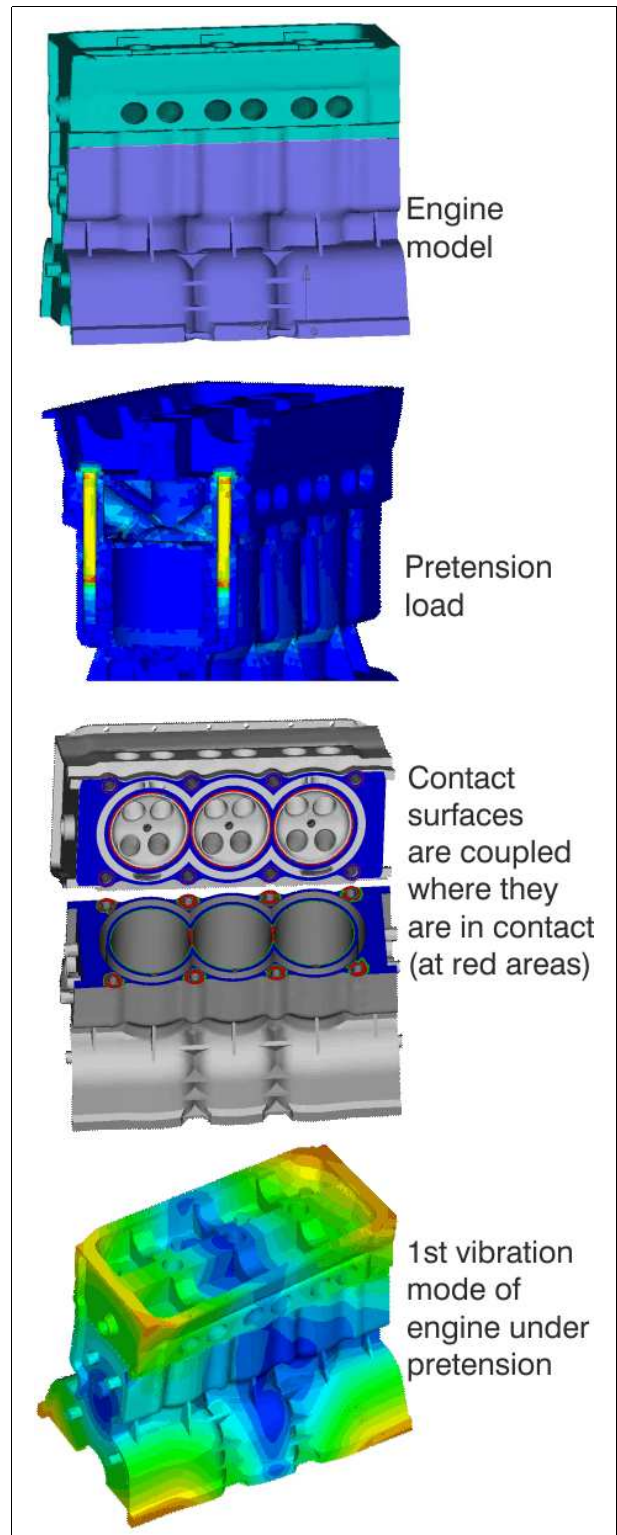


Figure 40: Dynamic analysis of an engine under pretension load

ter bolt pretension is applied, the engine assembly vibrates as one single body. This behaviour is achieved by locking the contact areas where the parts are in contact. Other areas are kept uncou-

pled, where no contact is in place.

Once, contact locking is applied, eigenvalue analysis and frequency response analysis can be used. Even optimization of frequency response functions can be used, e.g. to reduce sound radiation of engine. Fig. 41 shows the effect of moving the ribs on the engine surface and the effect of increasing rib thickness to reduce sound radiation of the engine block. The ribs are meshed incompatibly from the engine block. Hence, the ribs can easily be moved on the surface without re-meshing of engine block and ribs. This is used by a shape optimization.

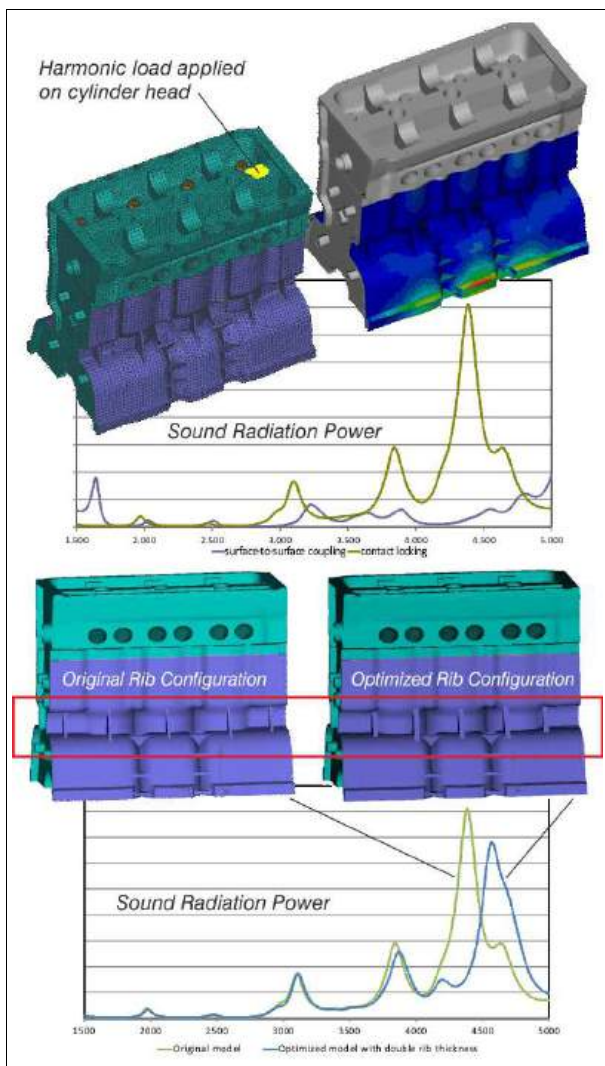


Figure 41: Optimization of rib thickness and rib position to reduce sound radiation from engine block surface

Connection Elements

Following features can be seen as connection elements:

- Bolt connections:**
 They are often used under pretension. So, contact analysis is applied to define prestressed bolts (see page 83 for more details). The thread coupling of bolts is of particular importance for short bolts (like in Fig. 42), because any cut through the shaft of a short bolt will be warped under pretension.
- Weld spot connections:**
 Typical weld spot connections consist of an element at the weld point location, which is used to model the additional spotweld stiffness and an MPC condition which couples the element forces to the connected flanges. These flanges typically have incompatible meshes (see page 63 for more details on automated spotweld modeling). A refined spotweld model is also available which shows improved stiffness representation and reduced sensitivity against different mesh sizes at the connected flanges (see page 123).
- Sealing connections:**
 For convenient modeling of sealings gasket elements are available which define the nonlinear behavior in a preferential direction by force-displacement curves. Contact analysis is used to solve static sealing problems, where the force-displacement curves are handled as an internal contact. In dynamic analysis, the typical frequency-dependent stiffness and damping of a sealing is modeled by spring-damper systems (see next list item and Fig. 181).
- Spring-damper connections:**
 In dynamic analysis, many joints have an influence on stiffness and damping. Such joints are modeled with spring-damper systems which also allows to model frequency dependent stiffness and damping properties (see Fig. 181).

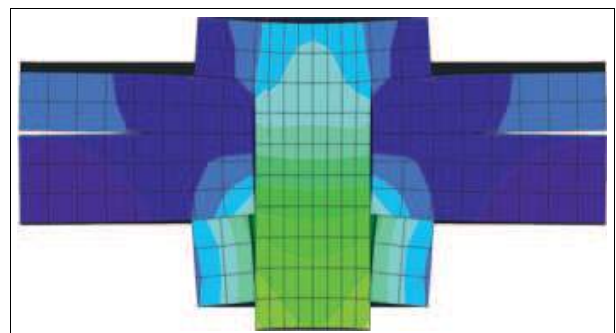


Figure 42: Short bolt under pretension

Brake Squeal Analysis

Brake squeal is a known phenomenon since brakes are used, and despite intensive research for many decades there are still coming new cars to the market which are squealing so heavily that expensive warranty cases arise for the manufacturers. This holds not only for passenger cars but also for commercial vehicles, the same for rail cars or aircraft brakes or even bicycles. Also, not only disk brakes but also drum brakes are affected.

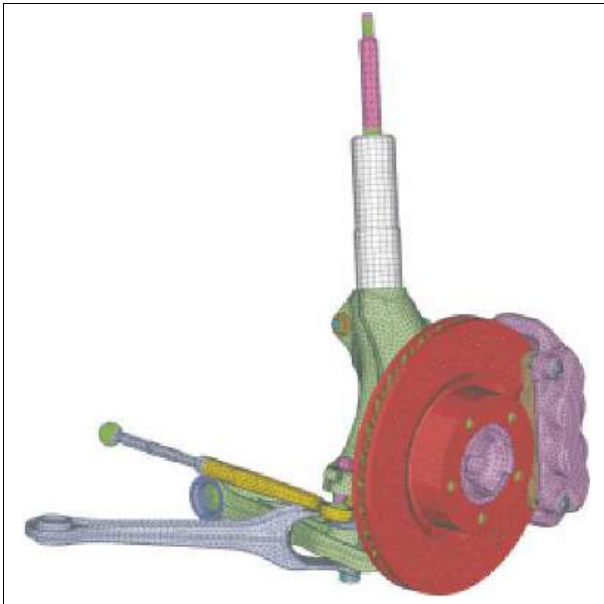


Figure 43: Brake model

by courtesy of Dr. Ing. h.c. F. Porsche AG in Stuttgart, Germany.

There is no lack of numerical approaches to make brake squeal computable but up to now the complexity of the phenomenon has prevented massive computations in this field due to very long computing times. As long as one set of parameters for one brake requires many hours of computing time, it is practically impossible to study geometrical modifications to get a configuration which does not exhibit squealing under all typical operating conditions.

Brake squeal is widely understood as friction induced dynamic instability. Therefore, two principal approaches are available: Transient analysis and complex eigenvalue analysis. Due to high computational effort for transient analyses obviously stability is more effectively studied by a complex mode analysis.

The analysis can be split up in following steps:

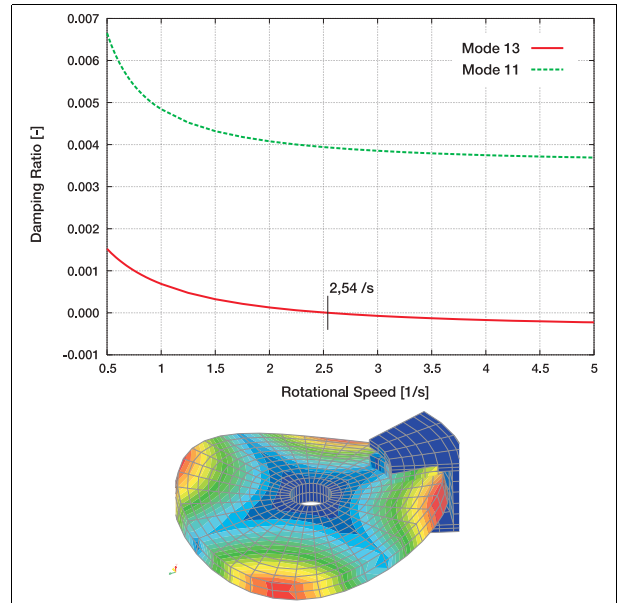


Figure 44: Simple brake model (1)
with an unstable bending mode ($m=2$, $n=1$) at 2,54 rps

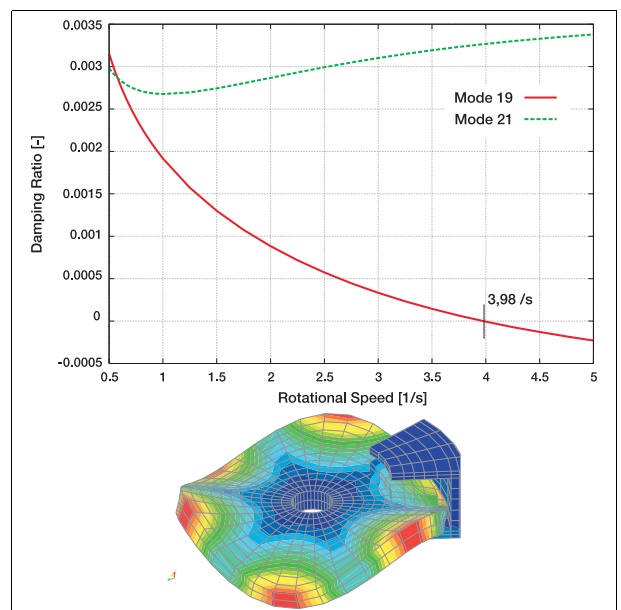


Figure 45: Simple brake model (2)
with an unstable bending mode ($m=3$, $n=1$) at 3,98 rps

- A linear static analysis with contact and friction under brake pressure and rotation. There, sliding between disk and brake pad can be prescribed by a rigid body motion to determine the sliding velocity.
- A real vibration mode analysis using the previously calculated contact status. This requires a linear model for the contact status which is achieved by *contact locking*.
- A complex mode analysis with additional fric-

tional and rotational terms. Gyroscopic and stiffness terms are taken into account which consider the disk as elastic structure in an inertial reference system. Additional stiffness and damping terms are derived from the frictional contact state perviously calculated in the contact analysis.

As usual instabilities are detected by a complex mode analysis if the real part of the complex eigenvalue becomes positive or the effective damping ratio becomes negative.

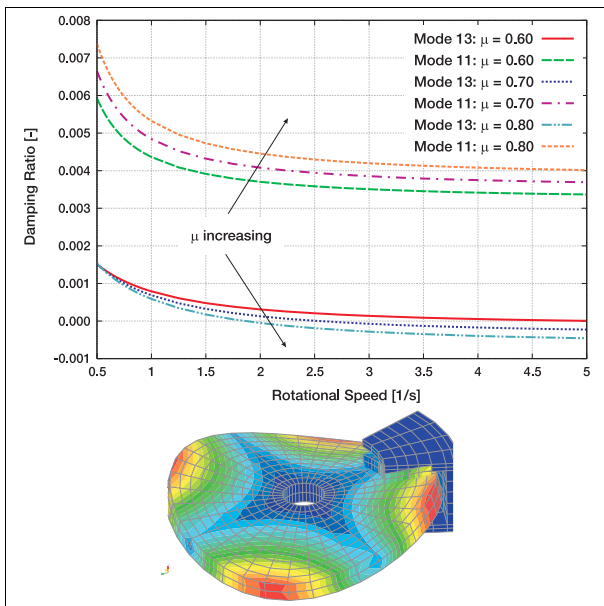


Figure 46: Simple brake model (3) with an unstable bending mode ($m=2, n=1$) which becomes unstable at different rotational speeds dependent on the frictional coefficient.

A complex mode analysis is performed in one computing run for the full range of interesting rotational speeds. By this sweep all relevant points of instability are obtained for one set of brake parameters. Successive computing runs are then used to study parameter modifications in order to establish a stability map for all important influencing effects.

As an example a simple brake is used which exhibits various instabilities at different rotational speeds (see Fig. 44 and Fig. 45). Each of the diagrams shown was generated by one single computing run.

This analysis is repeated several times to get the influence of the frictional coefficient between brake disk and brake pad (see Fig. 46).

A similar study was made to get the influence of

Young's modulus of the brake pad. Figure 47 shows a corresponding example.

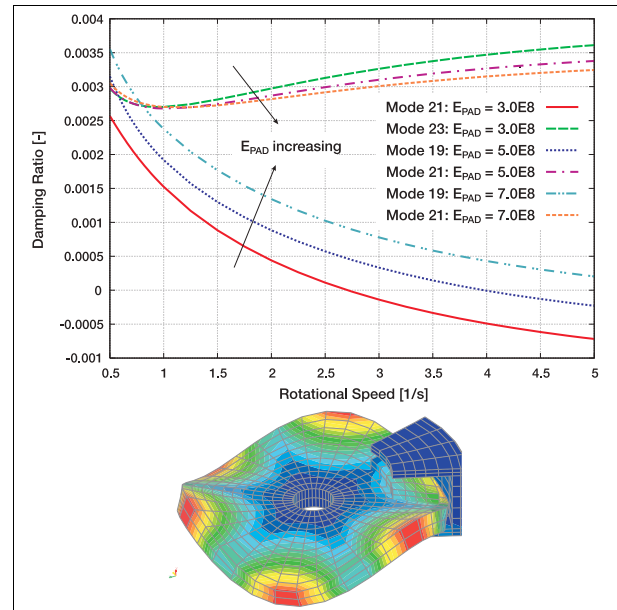


Figure 47: Simple brake model (4) with an unstable bending mode ($m=3, n=1$) which becomes unstable at different rotational speeds dependent on Young's modulus of the brake pad.

To illustrate the required run times for such analyses corresponding computing times are given for a large industrial model with the following characteristics:

480,000	Elements
900,000	Nodes
2.1 Million	Unknowns
218	Real eigenmodes
240	Complex eigenfrequencies
80	Rotational speeds

For this example the full elapsed run time with PERMAS Version 18 is 12 minutes on an Intel E5-2697 v4 processor (18 cores / 2.3 GHz) with 105 GB memory.

By such computing times an extensive parameter study of a brake will be possible in short time.

Such a parameter study can be used to generate a stability map of a brake (see Fig. 48). There, all unstable modes from complex eigenvalue analysis are collected for a large number of different parameter sets. This allows the identification of frequencies where squealing could occur. The information contained can be used to detect those parts of a brake which are candidates for modification to im-

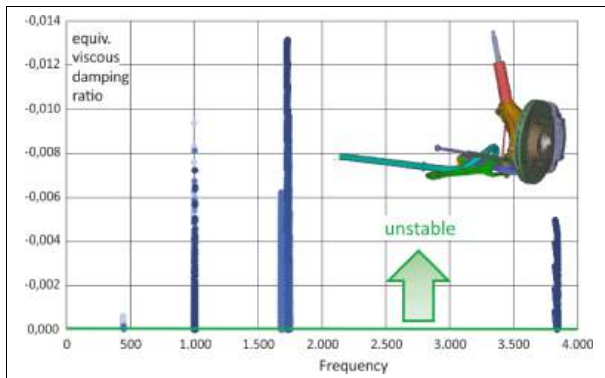


Figure 48: Stability map of a brake for 7110 parameter sets from rotational speeds, Young's modulus of disk, and frictional coefficient between disk and pad. The total run time for this stability map was 5 hours 12 minutes.

prove brake squeal behaviour.

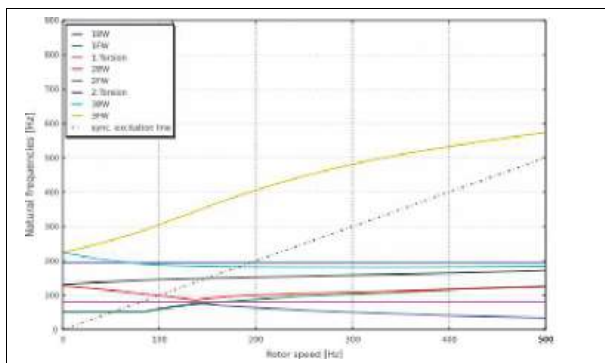


Figure 49: Campbell diagram for the evaluation of rotor dynamics

Rotating Systems

The available static and dynamic analysis capabilities can be used to analyze rotating systems, which imply additional constraints to the solution.

Fig. 50 provides an overview on the analysis capabilities for rotating structures. Both co-rotating and inertial reference systems can be applied.

Static Analysis

In a quasi-static analysis, which may include contact at the hub, the centrifugal forces due to rotation are taken into account. The reference system is co-rotating or inertial (with axisymmetric rotor). The static analysis is possible below critical speed.

	Co-rotating reference system	Inertial reference system
Elastic rotor	arbitrary shape	axisymmetric
Several rotors	one single speed	different speeds
Bearing stiffness	isotropic	arbitrary, speed dep.
Stator	no	arbitrary shape
Static analysis	subcritical	subcritical
Dynamic analysis	sub- and overcritical	sub- and overcritical
Additional matrices	geometric stiffness, centrifugal stiffness, Coriolis matrix	geometric stiffness, convective stiffness, gyroscopic matrix
Modal damping +	material, viscous	speed dep. bearing, material, viscous in stator
Campbell diagram in one analysis	with mode tracking and stability evaluation	with mode tracking and stability evaluation
Modal and direct response	harmonic, periodic (steady-state), in time domain	harmonic, periodic (steady-state), in time domain
Sizing and shape optimization	for rotor	for rotor, stator, and bearing
Active damping	of rotor	of stator and bearing
Model reduction	of rotor	of rotor and stator

Figure 50: Rotor dynamics capabilities

In a linear analysis, the centrifugal stiffness and the geometric stiffness at the given rotational speed are taken into account. In a geometrically nonlinear analysis, an update of the centrifugal forces will take place.

Dynamics

In order to get the relation between eigenfrequencies and rotational speed an automatic procedure is available (see Fig. 49 and page 98) which directly generates all values for a *Campbell* diagram.

For dynamics of rotating systems, the assumption is a linearized equation of motion with constant coefficients. A co-rotating or inertial reference system is taken. If rotating and non-rotating parts are present, the rotating part can be modeled as elastic body. The rotational speed is expected to be constant.

In the case of a coupling of rotating and non-rotating parts in a **co-rotating reference system**, no restrictions have to be observed for the rotating parts, but the non-rotating parts have to provide isotropic support to the rotor.

For such configuration, all direct and modal methods in time and frequency domain can be applied in the subcritical and overcritical frequency range. During response analysis the *Coriolis matrix* is taken into account.

In the case of dynamics in an **inertial reference system**, no additional restrictions have to be observed for the non-rotating parts, but the rotating parts have to be axisymmetric.

Also for such configuration, all direct and modal methods in time and frequency domain can be applied taking into account the *gyroscopic matrix*. Modal methods remain applicable even for the over-critical range of rotational speeds.

To determine the *critical rotation speed* a Campbell diagram can be used. In the co-rotating reference system, the Campbell diagram will show zero eigenvalues at certain rotational speed.

Rotation speed dependent stiffness and viscous damping of rotor supports can be taken into account during complex eigenvalue analysis and for the generation of a Campbell diagram. This feature can be modeled by a special element (i.e. CONTROL6, siehe Abb. 175).

For dynamics modal *steady-state response* is of particular importance. There, the static stresses under centrifugal load are determined first. Then, with geometrical and centrifugal stiffness, the static displacements are derived. On the basis of real eigenvalue analysis, several modal frequency response analyses are performed for each harmonic. After back transformation to physical space, the results for all harmonics and the static case are superposed in the time domain (see page 102).

Analysis of Machine Tools

For the development of machine tools, dynamic behaviour of the complete system is of utmost importance for the efficiency and precision of the machines. The complete system consists of structural parts, drives in different axes, and control. During machining the interaction between workpiece and tool generates cutting forces which may cause vibrations in the system. These vibrations have to be sufficiently damped by all system components. Finally, the machine tool has to provide high speed and high precision.

All dynamic analysis methods can also be used for machine tool analysis, like eigenmode and frequency response analysis, complex mode analysis,

and time-history response analysis. In addition, optimization methods can be used to propose model modifications which improve the characteristics of the machine tool like weight, static response, and dynamic response.

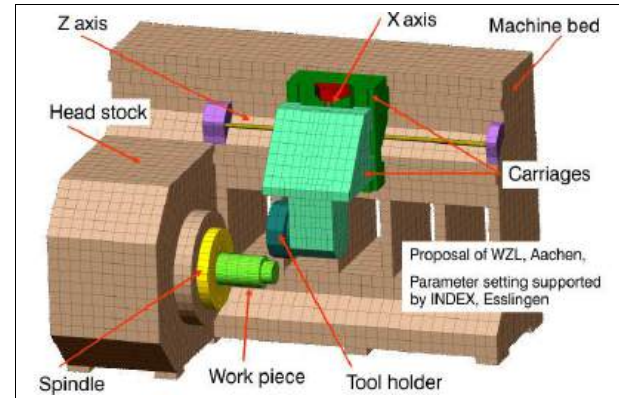


Figure 51: Simplified model of a turning machine as proposed by WZL in Aachen and supported by INDEX, Esslingen for setting suitable parameters

An example of a turning machine has been proposed by WZL Institute in Aachen (see Fig. 51). The parameter setting was supported by INDEX-Werke GmbH & Co. KG in Esslingen.

The following typical machine components were used to set-up the model:

- **Structural components:**
Machine bed, carriages, and headstock are usually modeled by solid elements (see Fig. 51).
- **Guide rails:**
They are part of structural components, but their proper connection is modeled by spring-damper combinations, where the spring and damper forces are connected to the solid structures taking incompatible meshes properly into account.
- **Ball screw drives:**
They are modeled by beam elements. Their function is to transform a rotational motion of the drive to a translational motion of the carriage. This transformation is achieved by a proper MPC condition taking the diameter of the screw and the pitch of its thread into account.
- **Spindle with workpiece:**
Spindle and workpiece are usually modeled with solid elements. The coupling to the headstock is done by a special element combined with a suitable MPC condition. The special element is used to provide rotation speed dependent stiffness and damping values for the spindle support

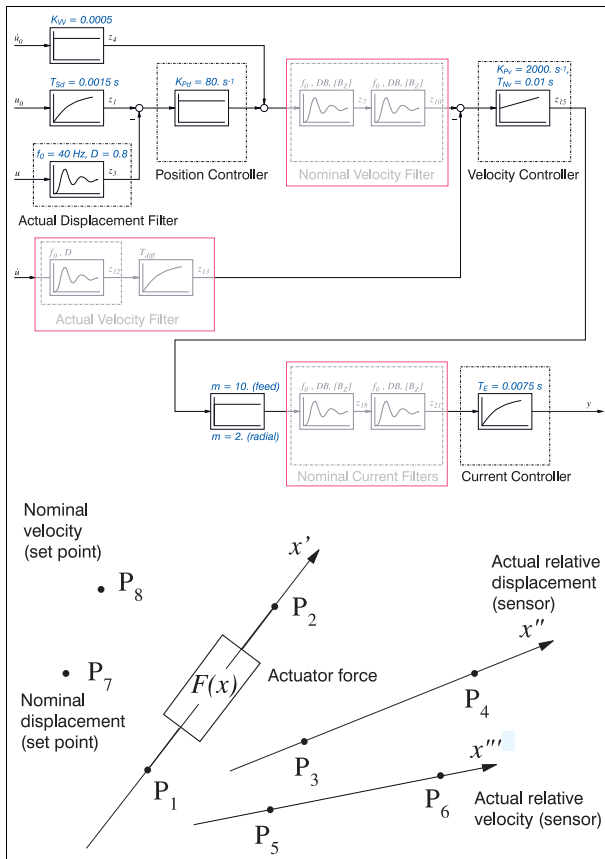


Figure 52: Block diagram of used controller and corresponding finite element representation

(see Fig. 175).

- **Control:**

In order to include a controller in the model a finite element has been developed which represents a so-called *cascade controller* as shown in Fig. 52. This cascade controller is able to represent position and velocity control beside some filters and a current controller. This control element comprises the typical control steps used in machine tools and has been developed based on the experience of machine tool manufacturers.

The actuator force is applied as a moment on the ball screw drives of both feed axes (Z axis and X axis). The sensors for relative displacements are located between machine bed and Z carriage and between Z carriage and X carriage respectively. The sensors for relative velocities (based on rotation speed sensors) are located at the ball screw drives.

The parameter settings have to reflect the actual controller which will be used for the machine tool. So, the parameters have to be provided to

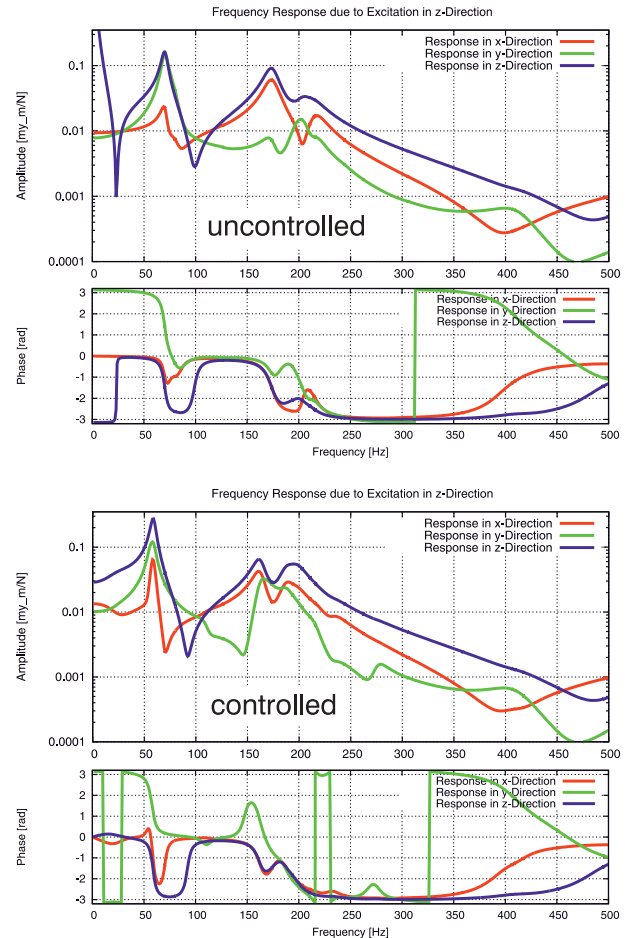


Figure 53: Frequency response between tool and workpiece due to excitation in Z direction for uncontrolled (above) and controlled (below) machine

the analyst by the controller development (like other parameters, e.g. material). Then, the analyst will be able to support the machine design by full system analysis.

In the example model, there are two controllers, one for the translational motion parallel to the spindle axis (Z axis) and one for the motion in radial feed direction (X axis). The parameters of the controllers are almost the same except the parameter m which reflects the moved masses leading to higher value for the translational motion.

Typical methods for dynamic analysis of machine tools include:

- **Eigenvalue analysis:**

Modal methods are often more efficient due to the reduction of the model size. But direct methods are also available.

- **Modal frequency response analysis:**

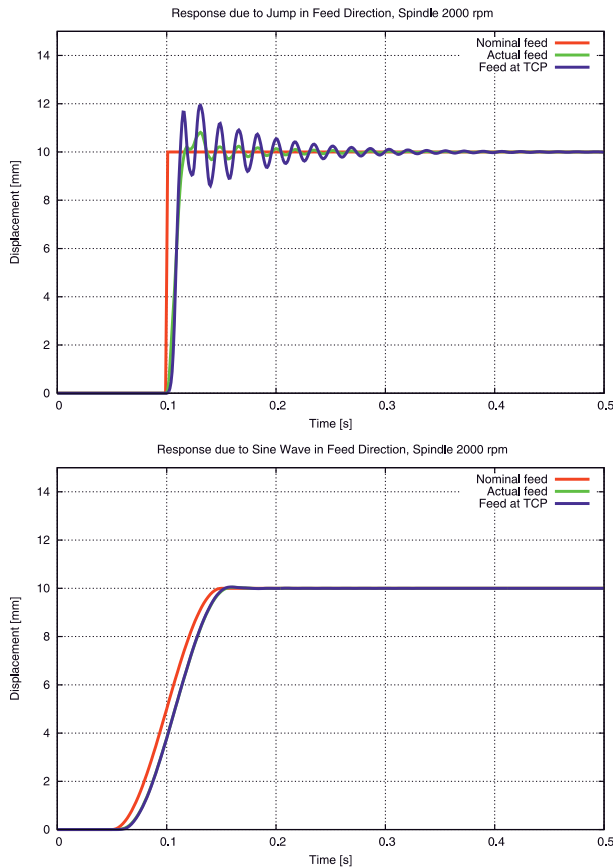


Figure 54: Time-history response due to excitation in Z direction for a sudden position jump (above) and a smooth position change (below) for the controlled machine

Fig. 53 shows the frequency response curves for amplitudes and phases between tool and workpiece for an excitation in Z direction under uncontrolled (above) and controlled (below) conditions.

- **Modal time-history response analysis:**

Fig. 54 shows the response to a sudden jump (above) and a smooth position change (below) of the tool in translational Z direction. It is worth mentioning that this calculation already takes the rotation of the spindle into account (at 2000 rpm).

In addition, it is possible to use nonlinear support conditions (like those for the spindle support) in modal time-history response analysis.

- **Complex mode analysis:**

The damping effect on the eigenfrequencies due to guides, spindle support, and control can be determined using complex eigenvalue analysis (including gyroscopic effects).

- **Stability analysis of process:**

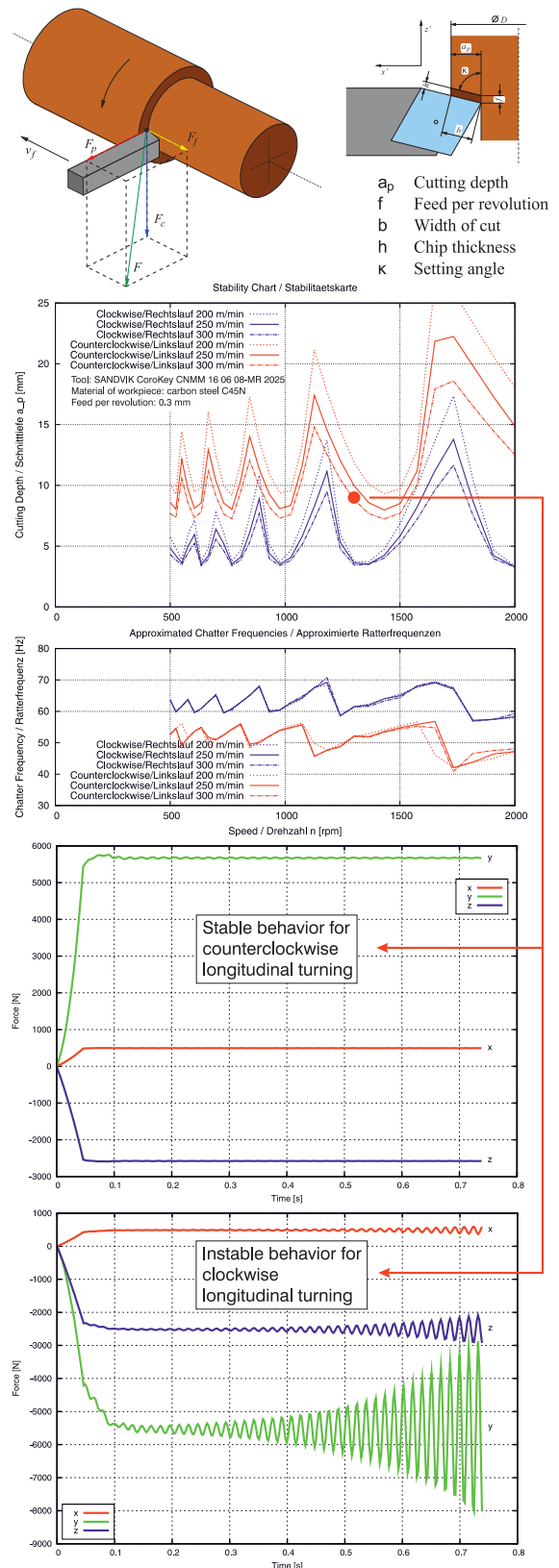


Figure 55: Stability chart for longitudinal turning with stable and instable behaviour under counterclockwise and clockwise turning.

For the design of a machine tool, the prediction of instabilities during operation is of utmost importance to the manufacturer. The goal is to design a machine tool which works stable with high precision at high speed. The source of instability is the interaction between tool and workpiece, where the energy for selfexciting vibrations is coming from the rotating spindle.

In order to make the nonlinear turning process predictable a cutting force model has been developed. The cutting force model depends on the cutting speed, the cutting depth, and the feed per revolution. The cutting force model requires a number of coefficients which reflect the combination of a specific turning tool with the material of the workpiece. These coefficients have to be calibrated based on a few experiments with the same combination.

On this basis a stability chart and the related chatter frequencies can be derived (see. Fig. 55). There, one important effect of spindle rotation can be seen. The stability depends on the rotating direction of the spindle. Here, counterclockwise rotation allows for higher cutting depths than clockwise rotation of the spindle.

Fig. 55 also shows the time-history response for one point of operation (see red dot at 1300 rpm) which leads to stable behaviour for counterclockwise spindle rotation and to instable behaviour for clockwise spindle rotation.

Some examples of additional applications of optimization are:

- Topology optimization of structural parts for lighter machine tools taking control and dynamic conditions (like eigenfrequencies, frequency response) into account.
- Shape optimization for getting optimized positions of supports to ground.
- Sizing optimization of controller parameters to move eigenfrequencies and to improve frequency response.

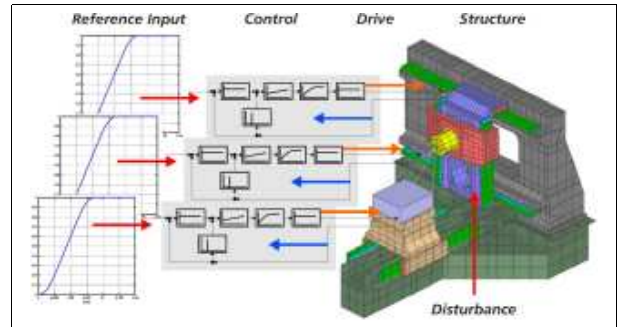


Figure 56: Principles of coupled analysis with control

Actively Controlled Systems

In the past, the design of machine tools structures and their control have been made separate from each other. Today's drives have an essentially higher dynamics and this separation of the design is no longer appropriate due to a strong coupling between machine and control dynamics. So, the coupled simulation of structural dynamics and control becomes a basic requirement for a successful overall design.

Lightweight structures tend to show higher sensibility to vibrations. Sometimes, such vibrations can be damped by active control (see example of a composite box girder in Fig. 57). A dynamic analysis taking control into account provides a lot of information for the design of the structures like the required damping forces and the related power needed.

For the analysis of controlled structures, the following features are supported:

- Linear *control elements*:
 - Three-term (PID) controller,
 - Various cascade controllers.

Those control elements link a dynamic vibrational state (measured by a sensor) with a driving force using classical linear control parameters.

Linear controller elements are handled in the same way as any other element, i.e. they are defined by their topology together with some property values. Of course, there may be any number of controller elements in one model.

An eight node controller is also available as user programmable element where the user has to provide element stiffness and viscous damping matrices via subroutine.

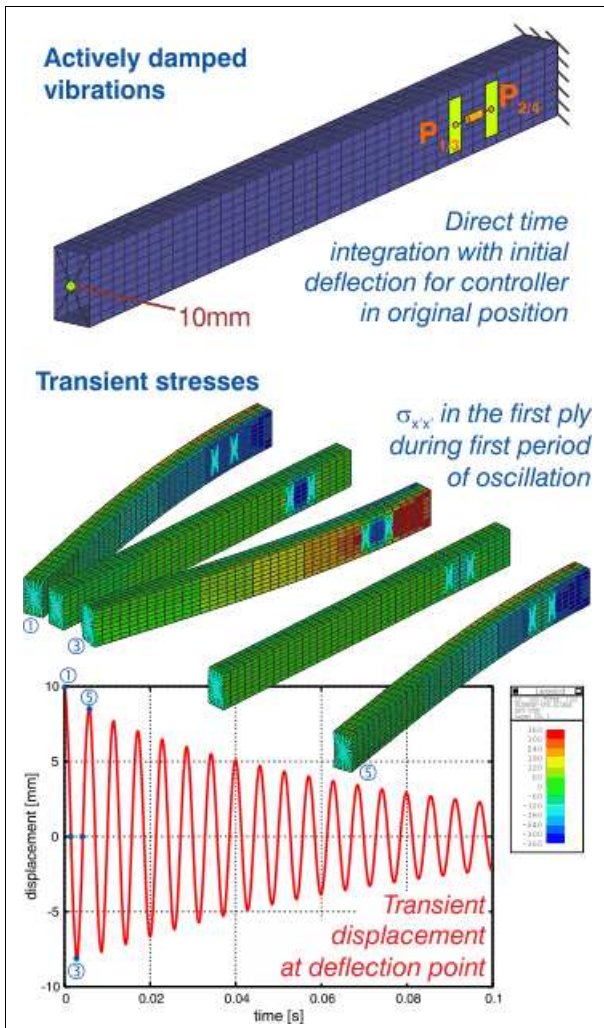


Figure 57: Active damping of a composite box girder

- Solution methods:
Linear controller elements may be used only in dynamic analyses, especially within the following solution methods:
 - Direct response analysis in frequency and time domain.
 - Modal response analysis in frequency and time domain, where the modal basis is enhanced by static mode shapes to represent the internal state variables of the controller elements.
 - Complex eigenvalue analysis to judge the effect of controllers on the dynamic behavior.
- Additional static mode shapes:
Beside the representation of internal state variables in modal space, static mode shapes added to the modal basis may be used to improve the accuracy of results of solution meth-

ods in modal space (see page 103).

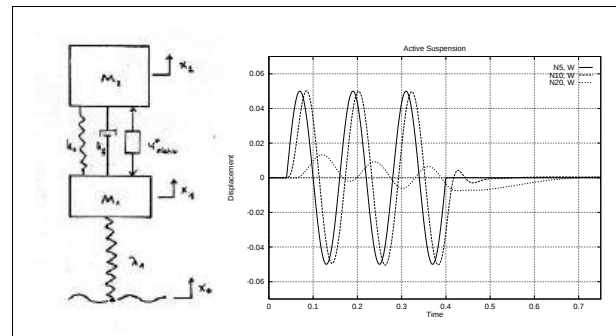


Figure 58: Active suspension unit u_{aktiv}^* between wheel M1 and body M2 as a function $f(x_0, x_1, x_2, \dot{x}_1, \dot{x}_2)$ with a harmonic base excitation. The body does not show any increase of the amplitude.

- Nonlinear control elements:
In addition, for nonlinear control additional control elements are available, where the dependency of the controller force on any result value in the model may be described by a general function, e.g. by a FORTRAN or C subroutine. Due to nonlinearities, the application of these elements is restricted to modal or direct transient response analysis.

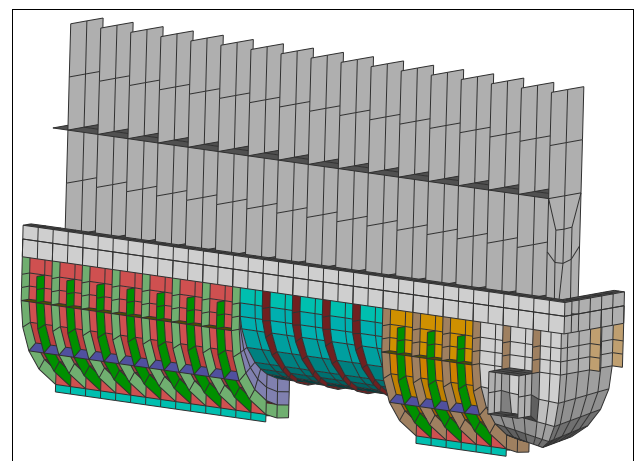


Figure 59: Optimization of a water box with 18 design variables and 19 stochastic basic variables

		Basic	Optimized	One Step
Mass:	M	$4.09 \cdot 10^{-4}$	$3.15 \cdot 10^{-4}$	$3.21 \cdot 10^{-4}$
Failure probability:	P_f	$4.78 \cdot 10^{-6}$	$6.80 \cdot 10^{-3}$	$1.55 \cdot 10^{-6}$
Failure rate (one of):		209205	147	645161

Robust Optimum Design

In order to achieve a *robust design*, it is not sufficient to perform a simple optimization:

- Optimization often leads to reduced safety margins.
- The optimized design may have other critical parameters than the initial design.
- A “reliable” optimum may be different than a deterministic one.

The proposed solution is a combination of optimization and reliability analysis. For this combination two different approaches are available:

- **Two step approach**
 - Basic design
 - * FE-Analysis (e.g. static analysis),
 - * Reliability analysis.
 - Optimization taking into account reliability
 - * Optimization,
 - * Reliability analysis of optimized design,
 - * If not sufficient:
 - Modify the design model according to reliability results,
 - Repeat the optimization and the reliability loop.
- **One step approach**
 - Combined optimization and reliability analysis,
 - Reliability as design constraint in optimization.

For the one-step approach, the interplay of design variables in optimization and basic variables in reliability analysis is as follows:

- Design Variables
 - define the design state of the structure,
 - may be modified by the optimizer,
 - may be assigned to one of the following types:
 - * Deterministic design variable,
 - * Deterministic mean value of a stochastic design variable.
- Basic Variables
 - Basic variables define the stochastic properties of the problem,
 - Following types of basic variables are possible:
 - * Stochastic properties of the structure,
 - * Stochastic design variable with deterministic mean value,

- * Load factors,
- * *Limit state function* parameter,
- * Parameter of another basic variable.

In the one step approach, there two different states belong to each design point:

- Design state,
- Limit state.

For each state one FE-analysis is necessary. Therefore, for each step during optimization at least two FE-analyses must be performed.

The design state is the actual optimization state. It is given by:

- Actual values of design variables,
- Mean values of basic variables.

The objective function is evaluated for the design state. The design constraints are evaluated for the design state. The final design state must fulfil the design constraints.



Figure 60: Optimization of PROTEUS satellite with 28 design variables and 30 stochastic basic variables (Alcatel Space S.A.)

		Basic	Optimized	One Step
Mass:	M	324.8	308.9	312.5
Maximum stress:	σ	$9.6 \cdot 10^7$	$1.2 \cdot 10^8$	$5.5 \cdot 10^6$
Failure probability:	P_f	$8.2 \cdot 10^{-7}$	$4.2 \cdot 10^{-5}$	$1.0 \cdot 10^{-6}$
Failure rate (one of)		1.2 Mio	23809	1 Mio.

The limit state describes for a given design state the corresponding failure state. It is given by:

- Actual values of design variables,
- Actual values of basic variables.

The limit state function is evaluated for the limit state. The design constraints are meaningless for

the limit state, e.g. the limit values for the limit state function and the limit values for the design constraints are different.

The combined analysis gives the following results:

- **Final design state**
 - Objective function value,
 - Design variable values,
 - Elasticities of design variables with respect to objective function,
 - Probability of failure,
 - Values of active constraints.
- **Final limit state**
 - Basic variable values,
 - Parameter sensitivities of the limit state function.
- **Always available:**
 - Selected data for each iteration.

Experimental Modal Analysis

For structures under dynamic loads, simulations and experiments are frequently used side by side. This leads to mutual benefits. On the one hand, simulation provides a means to identify preferred points to measure. On the other hand, experimental results can be used to identify differences between experimental and simulation model, which provide a basis for model updating to fit the test results by simulation.

One important comparison between experimental modal analysis (EMA) and Dynamic Eigen-Value analysis (PERMAS module DEV) is between measured and computed natural frequencies and between measured and computed mode shapes. While the natural frequencies can be compared directly, the comparison of mode shapes is usually made using MAC matrices (MAC - Modal Assurance criterion), see also page 74. To this end, each mode shape of the experiment is compared with each computed mode shape and vice versa. The values of a MAC matrix is between Zero and One. Values near One indicate a strong similarity of the mode shapes, while small values indicate different mode shapes.

PERMAS is capable to read model and results (from a Universal File) and to use them subsequently to

generate and export directly the MAC matrix with the computed mode shapes.

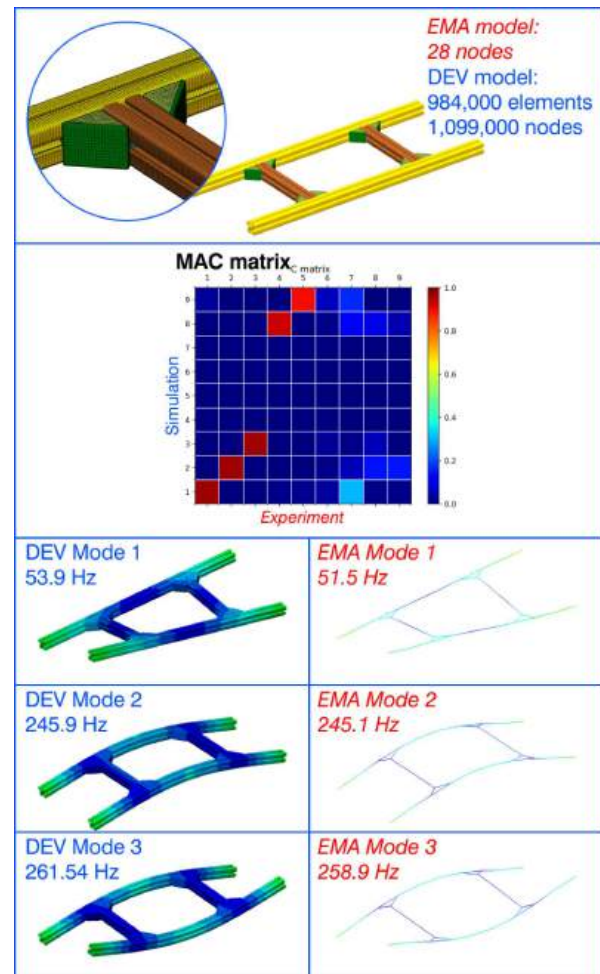


Figure 61: Example of EMA

A ladder frame with measured and computed natural frequencies and mode shapes. The MAC matrix side shows strong correlation for five modes.

Fig. 61 shows an example, which kindly has been made available by Prof. Dr.-Ing. Jörg Bienert of Ingolstadt University of Applied Sciences. He has determined the experimental results for the ladder frame structure, while the simulation and the comparison have been made by INTES. The location of the sensors does not fit to nodes of the FE mesh. So, interpolation regions were used to connect the sensor locations with the neighboured nodes of the FE mesh. By doing so, the computed results are available at the same points as the measured results and the comparison can be performed directly by generating the corresponding MAC matrix.

For the first three modes, the computed and measured natural frequencies and mode shapes are

shown. We see from the MAC matrix that the modes four to seven were not available in the experimental results. The experiment used one-dimensional sensors, which perfectly detect displacements normal to the ladder frame. But the computed modes four to seven show displacements in the plane of the ladder frame, which are not detectable by the used sensors.

More information about eigenvalue analysis can be found on page 97.

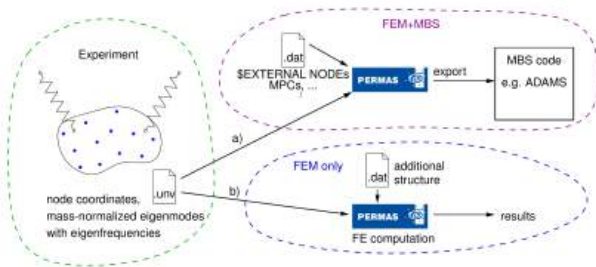


Figure 62: Workflow for Experimental Modal Analysis

PERMAS Version 21 offers a **possibility to reconstruct a matrix model from (measured) eigenmodes**. The workflow is illustrated in Fig. 62.

VisPER



Figure 63: The logo of VisPER

The VisPER History

PERMAS as a software for numerical analysis provides many specific productivity functions which are not sufficiently supported by general purpose pre-/post-processors. In order to accelerate the model completion process, a suitable *GUI* is required.

The initial version of the new graphical user interface VisPER (for Visual PERMAS) has been announced on the PERMAS Users' Conference 2008 in Stuttgart. The current version of VisPER is released with the corresponding PERMAS major release.

VisPER – A Short Introduction

VisPER (Visual PERMAS) is a GUI based model editor. It supports workflows to efficiently complete finite element models for specific applications with PERMAS. To this end VisPER efficiently fills the gap between FE models generated by a mesher and PERMAS models which are ready to run (see Fig. 65).

VisPER can also be used as post-processor for PERMAS.

In order to demonstrate the potential of enhanced model editing a number of special functions are supported by VisPER:

- Brake squeal analysis,
- Part replacement,
- Design by Simulation,
- Topology optimization,
- Sizing optimization,
- Fatigue analysis

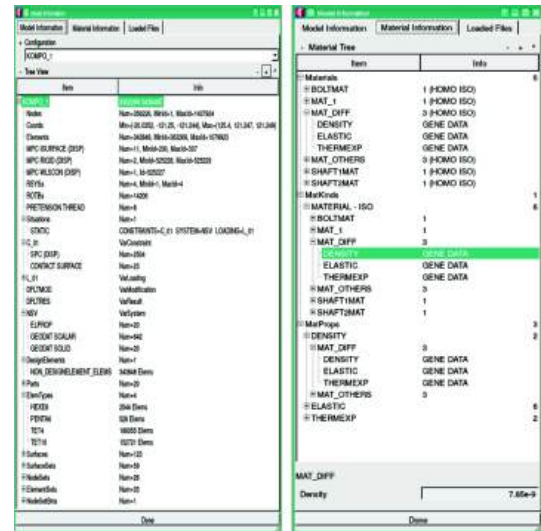


Figure 64: Model information tree with model and material information

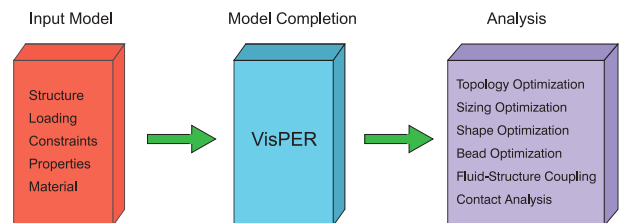


Figure 65: Model editing with VisPER

- Shape optimization (with free-form optimization),
- Bead design,
- Sampling,
- Fluid-structure coupling,
- Contact analysis (with bolt pretension),
- Press fit.

The use of VisPER is justified, if

- a FE analysis of a supported type is to be performed and a mesh is already available,
- a FE model is to be inspected for checking purposes (e.g. loads and boundary conditions),
- a substructure model has to be created and verified,
- results of a FE analysis with PERMAS should be post-processed.

Some of the advantages related to the use of VisPER are:

- VisPER provides an easy and fast way to complete a model,
- The model completion represents the logical structure of a PERMAS model. This is aimed at

a reliable and nearly error-free specification process and shortens the way to a correct model,

- Due to a recording facility and a fully integrated scripting capability (*Python*) VisPER is highly customizable,
- The model data generated in VisPER can be exported separately and used in PERMAS together with the already existing model data,
- VisPER uses the same data structures and input facilities as PERMAS. Hence, models in both programs are identical without translation or interpretation.

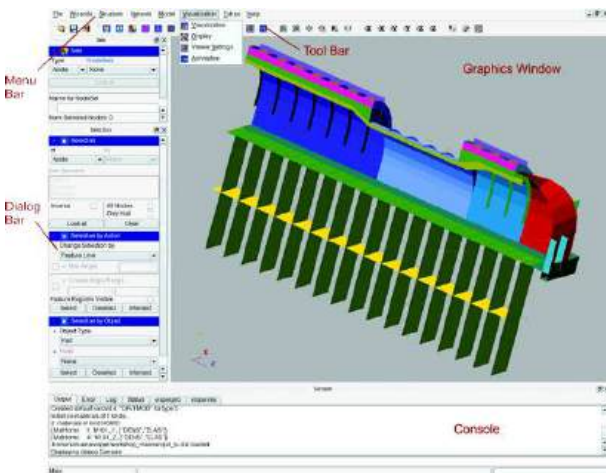


Figure 66: Main Window of VisPER

Fig. 66 shows the main window of VisPER. Each function can be accessed via pull down menus. Shortcuts are predefined for many functions which may be configured or new shortcuts may be added. The tool bar provides a fast access to frequently used menu functions. A dialog bar provides a range of commands corresponding to a selected item. The console provides a feedback of VisPER to the user's actions (including comments and error messages). The console and any dialog bar can be hidden to maximize the graphics window. For frequent use they may be pulled out of the main window to see them permanently.

VisPER-BAS – Basic Module

This module comprises the complete VisPER infrastructure, the graphical user interface, and all basic functions for pre- and post-processing.

The infrastructure includes:

- configuration of VisPER with regard to a site, a user, and the integration of user-defined documentation (like *tooltips*),
- import and export of files (like model data and results),
- geometry handling
 - VisPER reads stl and step formats for geometry. The handling and visualization is performed in a separate dialog bar.
 - Geometry Display/Meshing, see figure 84:
 1. Visualize Geometry
 2. Create discrete geometry
 3. (triangular) Surface meshing
 4. Tet-meshing
- Mesh-check functionality is provided in addition
 - VisPER can export geometry in step or stl format
- generation and use of macros,
- creation and use of PERMAS user control files (i.e. UCI).

The graphical user interface includes:

- the menu elements like menu bar, tool bar, dialog bar, wizards, consol panel, and information panel,
- the GUI elements for color definition, slider animation, and font selection,
- the control of mouse, space mouse, viewer buttons, and camera interaction,
- the interactive manipulations with the mouse,
- the measuring of a distance between nodes or elements,
- the definition and use of shortcuts.

Fig. 68 shows the top-level menus of VisPER, which gives a good overview on the supported PERMAS features.

The following wizards are part of the basic module:

- Add&Replace wizard,
- Brake Squeal wizard,
- Design wizard,
- Pressfit wizard,
- Pretension wizard,
- Sampling wizard.

In the following, these wizards are explained in more detail:

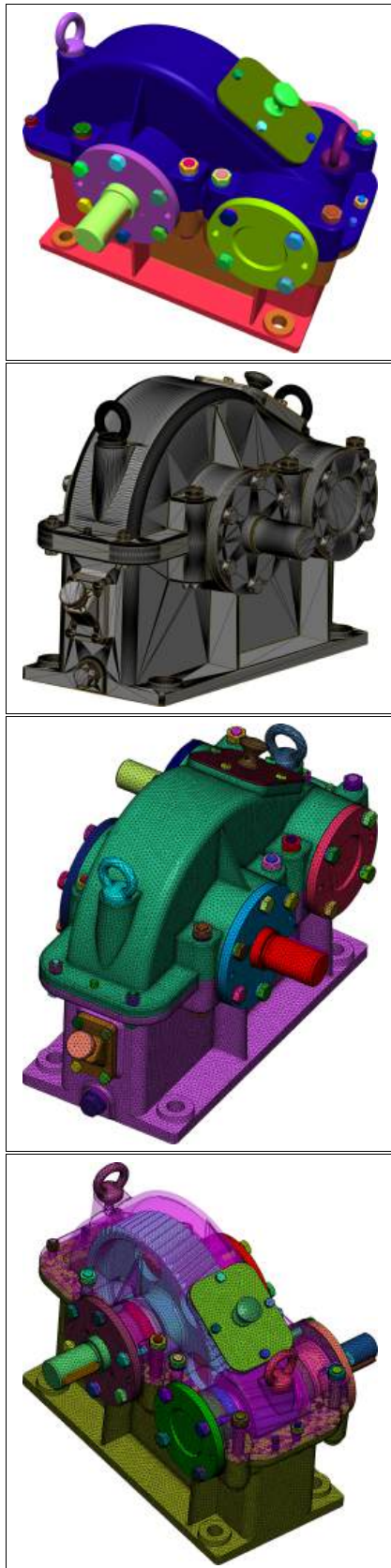


Figure 67: Geometry processing, transmission housing

File	Wizards	Structure	Variants	Model	Geometry	Visualization	Extras	Additional	Help
Open	Add&Replace	Selection	Constraints	Config.	Geom-Display	Results&-Data	Console	Check-Mesh	About
Import	Brake	Contlock	Loading	Functions	Geom-Meshing	Annotations	Snapshots	Fix-Mesh	What's
Open Macro	Design	El. Quality	Modification	Materials	Check-Mesh	Clip-planes	Videos		This
PY									
Export	Contact	Elements	Results	Messages	Tet-Meshing	Display	Edit Macro		Manual
Save	Pressfit	ElSystems	System	Measurements	Patch-Meshing	Face Normals	Recorder		Short-
									cuts
SyncDMS	Pretension	Lines	Create New	Information		Show/Hide	Abaqus El.	Report	
Create	FS	MPCs	Parameters	Relabel		Viewer Settings	Customize		
UCI									
Exit	Free-shape	Nodes		Search For					
	Shape	Parts							
	Sizing	Reference Systems							
	Topo	Rotated Bases							
	Sampling	Sets							
	Sub-structure	Subcomp.							
		Surfaces							

Figure 68: VisPER Menu with all top-level menu icons.

The *Add&Replace Wizard* supports the following two operations:

- To add a new part to an existing assembly,
 - Automatic relabeling of conflicting node and element numbers,
 - Optional positioning to correct misalignment, wrong direction, different origin, etc.,
 - Create connections like MPC, contact pretension, pressfit, etc..
- To replace an existing part in an assembly by a new part (see Fig. 71).
 - Recover existing sets and connections,
 - Fit surfaces to new geometry,
 - Relabeling of nodes and elements,
 - Manage replaced and new part simultaneously,
 - Keep model consistent.

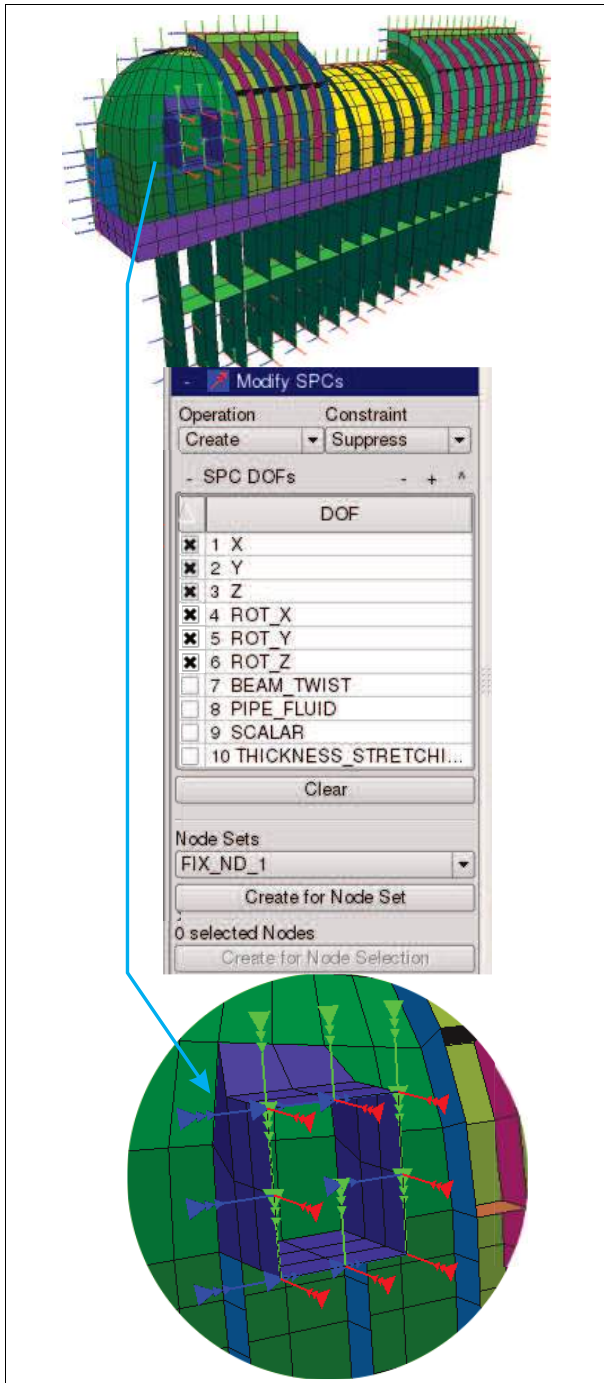


Figure 69: Specifying single point constraints

The *Brake Squeal Wizard* allows an easy description of complex brake models according to PERMAS brake squeal technology (see Fig. 72). The key features of the wizard to provide model standardization and to ensure process stability are:

- Fast setup of brake squeal analysis,
- Guided definition of additional physics for the dynamics of the analysis task,
- Checking of dynamic definitions and their com-

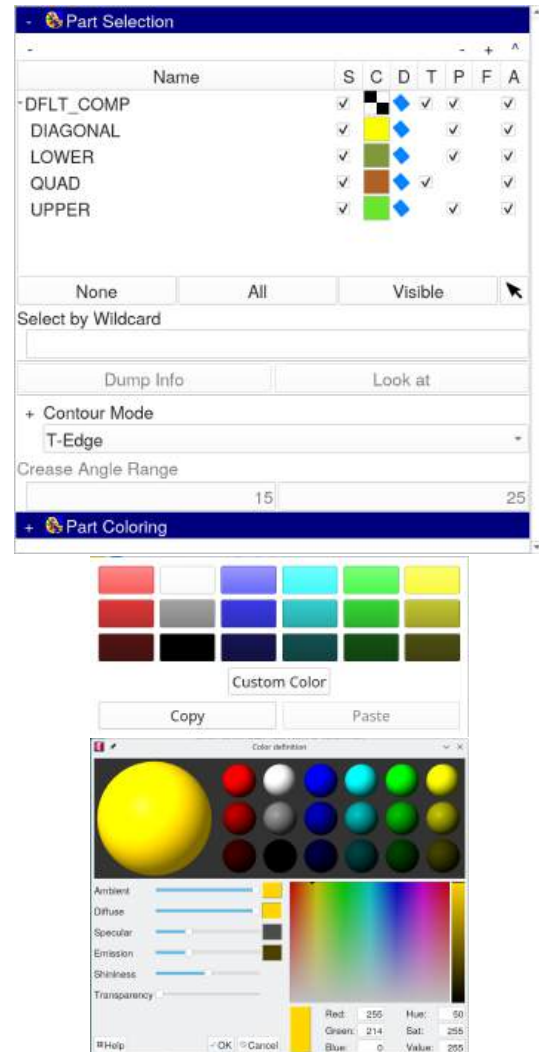


Figure 70: Graphics entities for parts and element groups (select S=Show, C=Colour, D=Display Mode, T=Thickness, P=Profile, F=Free nodes, A=Active).

patibility,

- UCI file generation after wizard completion.

The *Design Wizard* supports a simulation driven design (see Fig. 83) by generating and smoothing a hull for the topology optimization result, repair the hull mesh, add and checking manufacturing constraints, and generating a TET mesh from the hull:

- Topology optimization has great potential to create new design ideas,
- Clear separation of material/void areas essential for automated process to select basic geometry,
- After having derived a new design, further detailed requirements have to be fulfilled (e.g. by

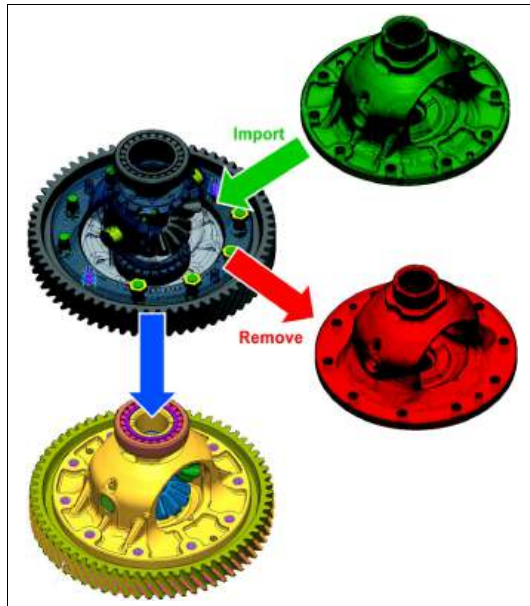


Figure 71: Replacement of model part

shape optimization),

- Hence, a simulation process chain has to be followed that best meets the requirements,
- The simulation chain targets to bridge the gap between topology and shape optimization,
- The goal is a digital prototype enabling performance based design decisions.

The *Sampling Wizard* supports the set-up of a sampling model by preparing the sampling situation, assigning the selected variables, defining the sample value ranges, and specifying the result item for which the sampling should be performed (see also page 79).

The *Pressfit Wizard* supports the set-up of a pressfit connection by selecting the pressfit partners, defining the pressfit parameters and loads, and specifying the load history, if any.

The *Pretension Wizard* identifies possible candidates for a pretension connection. It supports the set-up of a pretension definition by selecting the pretension partners, defining the pretension parameters and loads, and specifying the load history, if any.

A *RBM Assistant* (Rigid Body Mode Assistant) is available which provides an efficient workflow to generate elastic supports for static models. In particular for contact models, compensation springs can be comfortably generated by this assistant (see page 84).

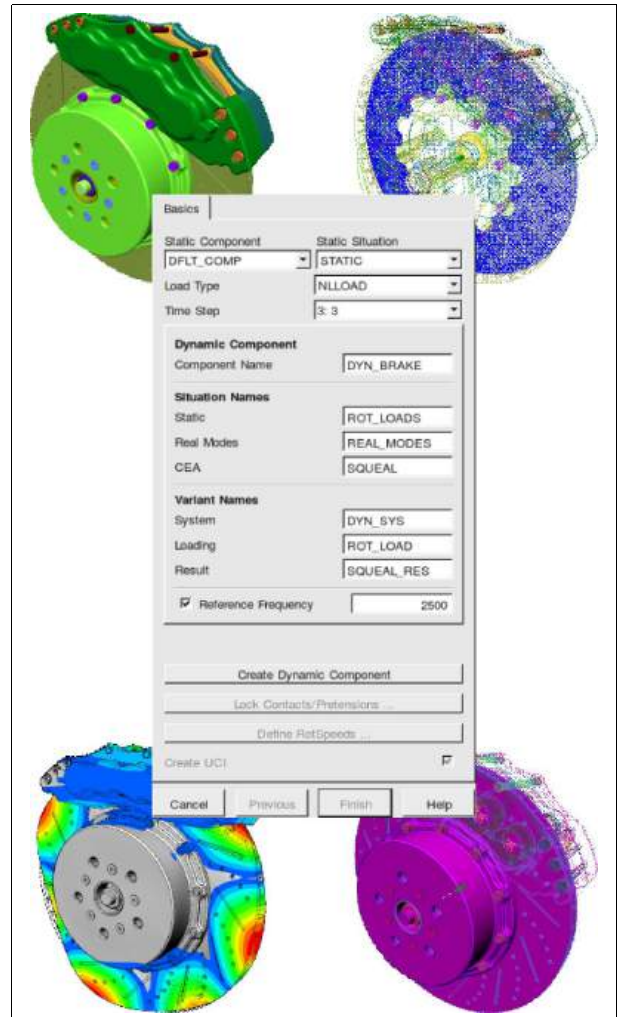


Figure 72: Preprocessing of brake models

VisPER also provides some useful tools to support the cooperation with PERMAS, e.g.:

- A *Model Information Tree* provides access to all model data. With a Context menu the detailed access to these data is enabled. In combination with a *Reference Tree* a powerful tool to inspect complex data relations is available
- A *message tree* is provided to summarize diagnostics and to provide additional information (like tool tips). The dialog provides parsing of the PERMAS protocol file. There is a direct relation to model entities mentioned in the diagnostics.
- VisPER can be configured to submit a PERMAS batch job.
- Imported results and the related post-processing model can be exported to other formats (like MEDINA or HyperView).

VisPER contains a complete set of *post-processing*

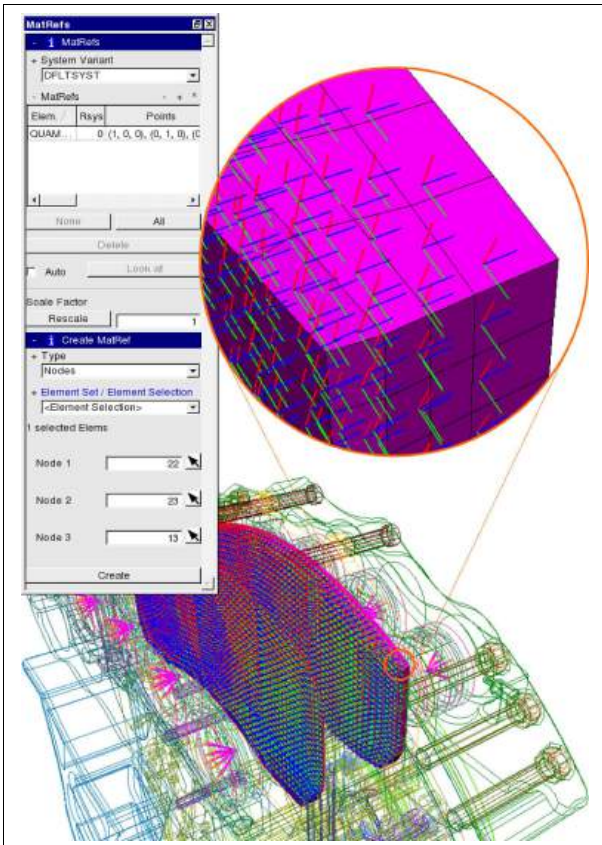


Figure 73: Material reference system for anisotropic material

functions, e.g.:

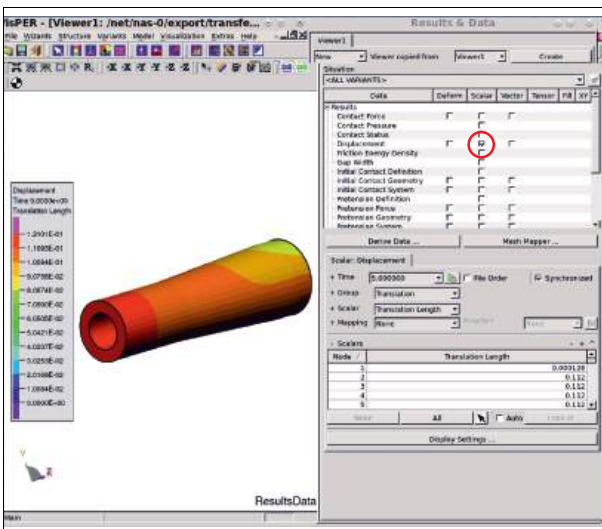


Figure 74: Post-processing

- Complete overview on all available results (see Fig. 74).
- Multiple viewers see fig. 75
- Direct selection of results for visualization from the overview.

- Dependent on the result type, deformations, scalar results (colored on model surface), vector results (arrow length and arrow color), tensor results (like stresses), filling ratios (like from topology optimization) and XY data can be visualized.
- Single result columns, single components or component groups (like translation or rotation) as well as mapping using a defined function (like LOG or ABS) can be selected.
- A great variety of arithmetic operations on result combinations is possible (e.g. difference of kinetic and strain energy density). Each single column may be used as operand. All functions with matching number of input parameters can also be used (e.g. logarithmic function to post-process acoustic levels).
- The visualization of tensors and principal stress components is colored due to values.
- Viewing angles and direction as well as part selection or cutouts can be defined.
- Optionally, the visualization may include a legend, where the relation of color and value can be adapted as well as the size and position of the legend.
- Viewings may be stored as pictures, and animations can be specified and stored as GIF or video.
- Particular result values (e.g. at nodes or elements) may be included in the evaluation by annotations.
- Clipping planes may be defined and allow for a view on the interior of a model.

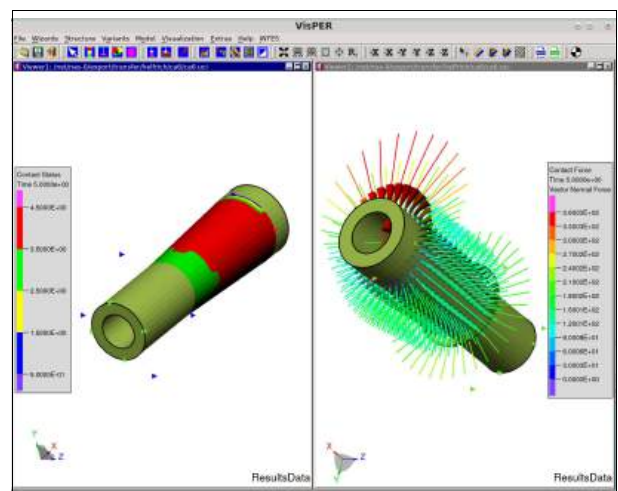
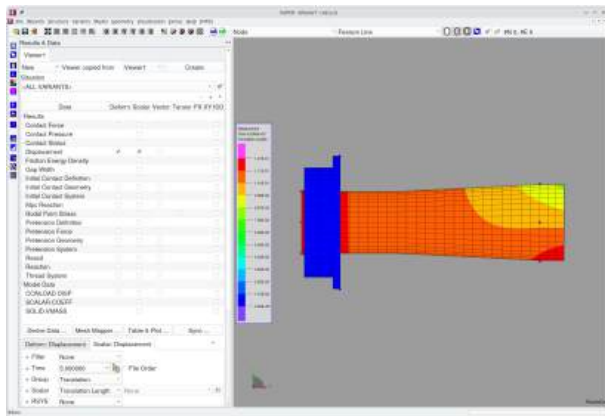


Figure 75: Multiple viewers



Postprocessing

VisPER is a fully featured tool for postprocessing of PERMAS results. It is easy to use by a central sheet summarizing the available FEA result types.



Figure 76: Isosurfaces of fluid pressure distribution

- The iso-surface visualization allows to evaluate scalar field values, like e.g. pressure values. Multiple iso-surfaces may be created to see a distribution of the values. See figure 76.
- Generic table xy-plot function. E.g. contact pressure along a (node) path.. See figure 77.
- Transformation of results in local coordinate systems.
- Support of element stress resultant results.



- Creation of orbit plots for rotordynamics, see figure 78.

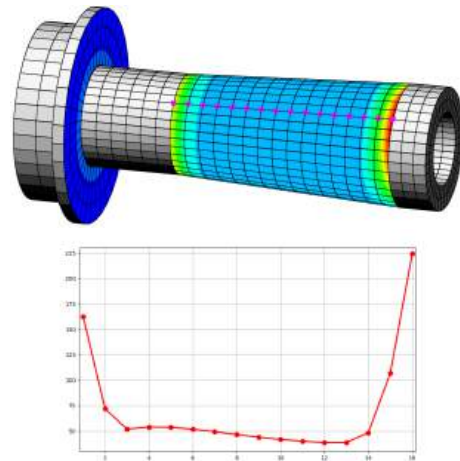


Figure 77: Generic plot function

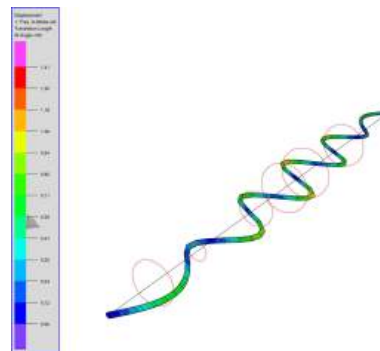
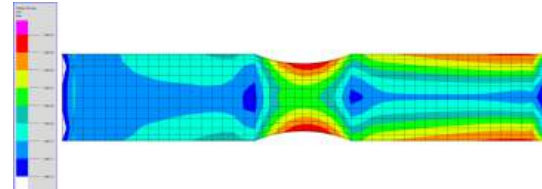
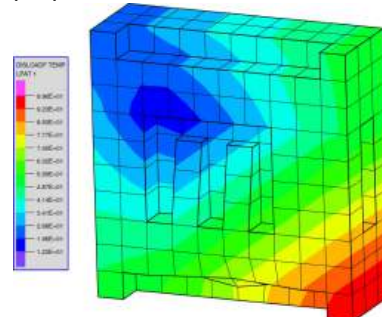


Figure 78: Orbit plot

- Support of fatigue results.



- Verification of model data like temperatures, properties, etc.



VisPER-TOP – Topology Optimization

This module provides a *wizard* supporting the set-

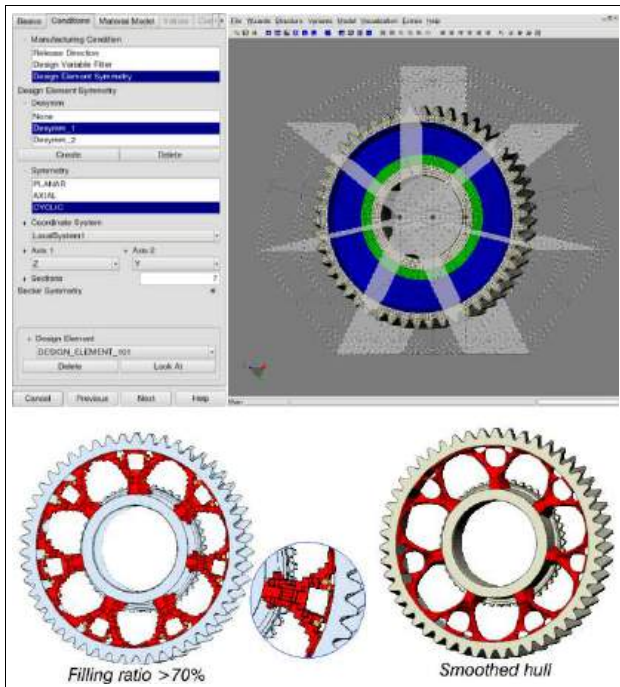


Figure 79: Topology optimization of a gearwheel
Definition of cyclic symmetry and release directions, max. stress at interface between design space and gear rim, max. allowed weight, minimum compliance as objective function (by courtesy of Daimler AG, Stuttgart).

up of an optimization model for topology optimization tasks. The details of topology optimization in PERMAS can be found on page 114.

The wizard provides guidance through the optimization modeling by the following steps:

1. The basics contain the specification of the design space and the related design elements. For the filling ratio a number of conditions can be specified like initial fill, modification limit, and minimum and maximum values.
2. The *manufacturing constraints* contain the definition of release directions, filters for minimum and maximum member size, and symmetry conditions (see Fig. 79 as an example). A special function allows the specification of a *fixed mold parting line* which separates opposing release directions.
3. Design constraints can be specified dependent on certain analysis types for various result quantities (like compliance, weight, displacement, frequency). One of the constraints is selected as design objective.

In addition, post-processing of topology optimization results is supported by VisPER, too. *Hull generation*, *smoothing* and *polygon reduction* can be performed beside the export of smoothed surface (see page 117 for details of these results). Fig. 79 also shows the smoothed hull of the gear wheel body.

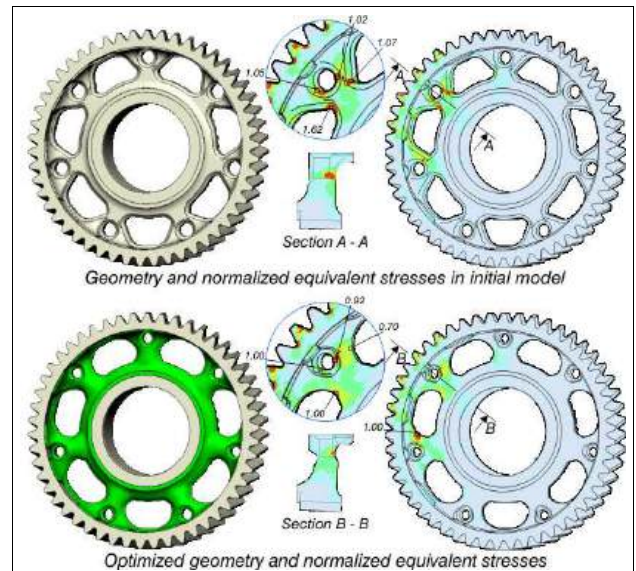


Figure 80: Shape optimization of a gearwheel
Definition of cyclic symmetry, max. stress, minimum weight as objective function (by courtesy of Daimler AG, Stuttgart).

VisPER-OPT – Design Optimization

This module provides three *wizards* supporting the set-up of an optimization model for design optimization tasks, one wizard for parametric shape optimization, one wizard for sizing optimization, and one for non-parametric free-form optimization. The details of design optimization in PERMAS can be found on page 109.

Design optimization comprises the following tasks:

- Sizing optimization which modifies the properties of elements (geometrical properties or material properties).
- Shape optimization which modifies the node coordinates of a model by a parametric approach.
- Bead design which modifies the coordinates of a shell model to achieve certain stiffness targets by bead generation.
- Free-form optimization which modifies the node coordinates of a model by a non-parametric approach.

For *sizing optimization*, the wizard provides guidance through the optimization modeling by the following steps:

1. The global selection of properties and their relation to design elements including design and modification limits. The design elements are specified by element sets.
2. The basics contain the specification of the design space and the related design elements.
3. The detailed properties can be specified for each design variable.
4. Design constraints can be specified dependent on certain analysis types for various result quantities (like compliance, weight, displacement, frequency, temperature, contact pressure). One of the constraints is selected as design objective.

For *shape optimization* and *bead design*, the wizard provides guidance through the optimization modeling by the following steps:

1. The basics contain the specification of the design space and the related design elements.

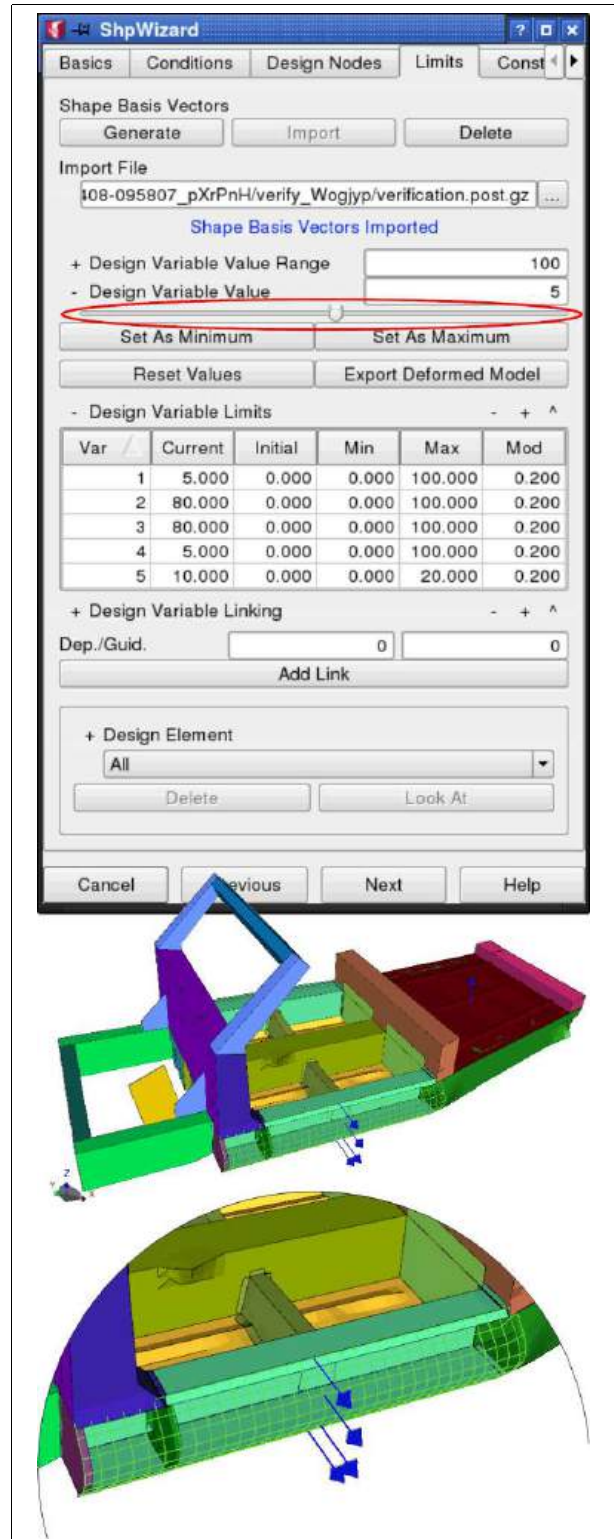


Figure 81: Definition of shape basis vectors of a car sill. The shape changes can be visualized by moving the slider in the menu.

2. The conditions contain the definition of restraints for shape changes, restrictions on bead design, and symmetry conditions.

- Definition of *shape basis vectors (SBV)* as design variables and a number of conditions like initial values, modification limit, and minimum and maximum values. The shape basis vectors are calculated by PERMAS and provided for visualization. See Fig. 81 as example for the specification of shape basis vectors by *morphing*.
- Design constraints can be specified dependent on certain analysis types for various result quantities (like compliance, weight, displacement, frequency, temperature, contact pressure). One of the constraints is selected as design objective.

A wizard for *free-form optimization* is available for a parameter-free shape optimization to reduce stress concentrations (see Fig. 82).

In addition, post-processing of design optimization results is supported by VisPER, too:

- Parameters of the optimization process are available as XY plots for objective function history, constraint history, design variable history, and max. constraint violation history. The visualization of history plots allows a direct link between a touched curve in the plot and a highlighted area in the structure, e.g. a design variable history shows the related design elements.
- Resulting shell thicknesses can be presented.
- Shape changes can be visualized by showing the model with new coordinates.
- A *hull generation* is supported also for the new shape after shape optimization.

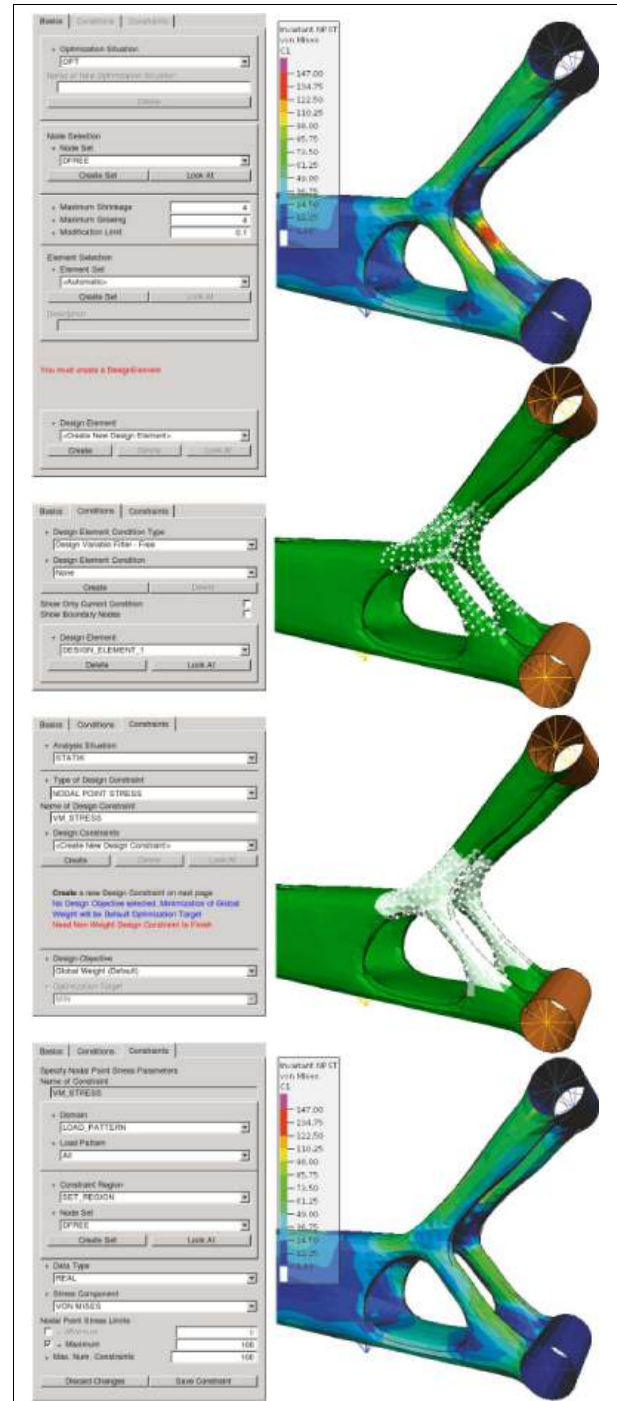


Figure 82: Pre- and post-processing for free-form optimization:

- Initial shape with too high stresses
- Definition of surface node set to specify design area
- Automatic selection of design space in thickness direction
- Modified shape after free-form optimization with reduced stresses

Design Wizard

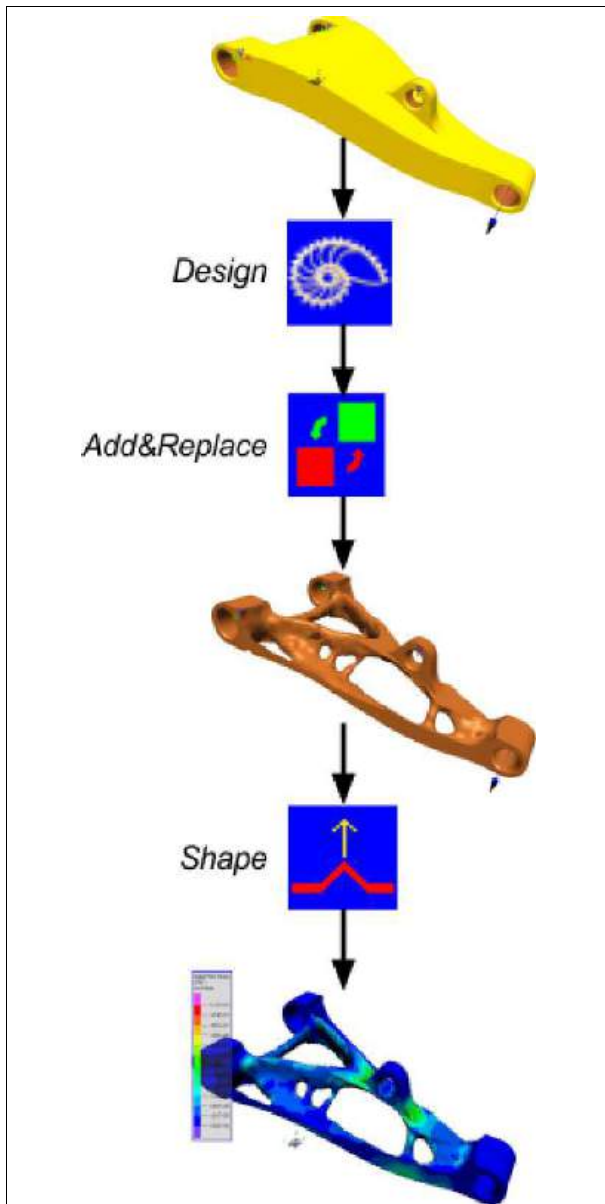


Figure 83: Operations of design wizard

This wizard plays a central role to provide a workflow for “*Design by Simulation*” or for “*Generative Design*”, see fig. 83. It provides functionality for:

- Creation of a regular hull based on a topology or shape optimization
 - Smoothing and remeshing of the hull to prepare a new tessellated geometry representation
 - Checking of manufacturing conditions, see figure 85
 - Geometry re-creation to prepare a new design variant or to export a geometry to CAD.
- This process allows also a re-use of unchanged geometry “*Hybrid Geometry*”

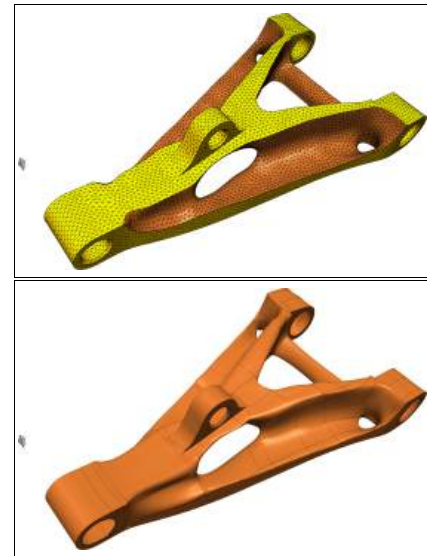


Figure 84: Parametric geometry creation

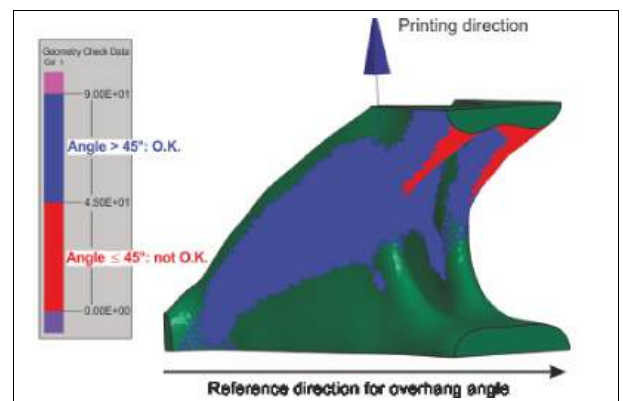


Figure 85: Check of overhang angle
red: critical angle, blue: non-critical, other colors: no value (i.e. >90°).

VisPER-FS – Fluid-Structure Coupling

This module mainly provides a *wizard* supporting the semi-automatic fluid meshing of a cavity, where the surrounding structure is given as input. The wizard supports the following steps, where the numbers correspond to the numbers in Figs. 86 and 87:

1. A recommended mesh size is calculated based on a frequency range for subsequent analysis.
2. *Hole detection* is performed automatically. Only those holes are detected which are larger than a defined minimum element size.
3. Due to difficult topological situations, some holes cannot be detected automatically. Then, undetected holes can be specified in addition.

4. Hole meshing is performed automatically using elements which do not introduce any stiffness or mass (like plot elements). They are just used to specify the topology of the hole and to limit the subsequent cavity meshing.

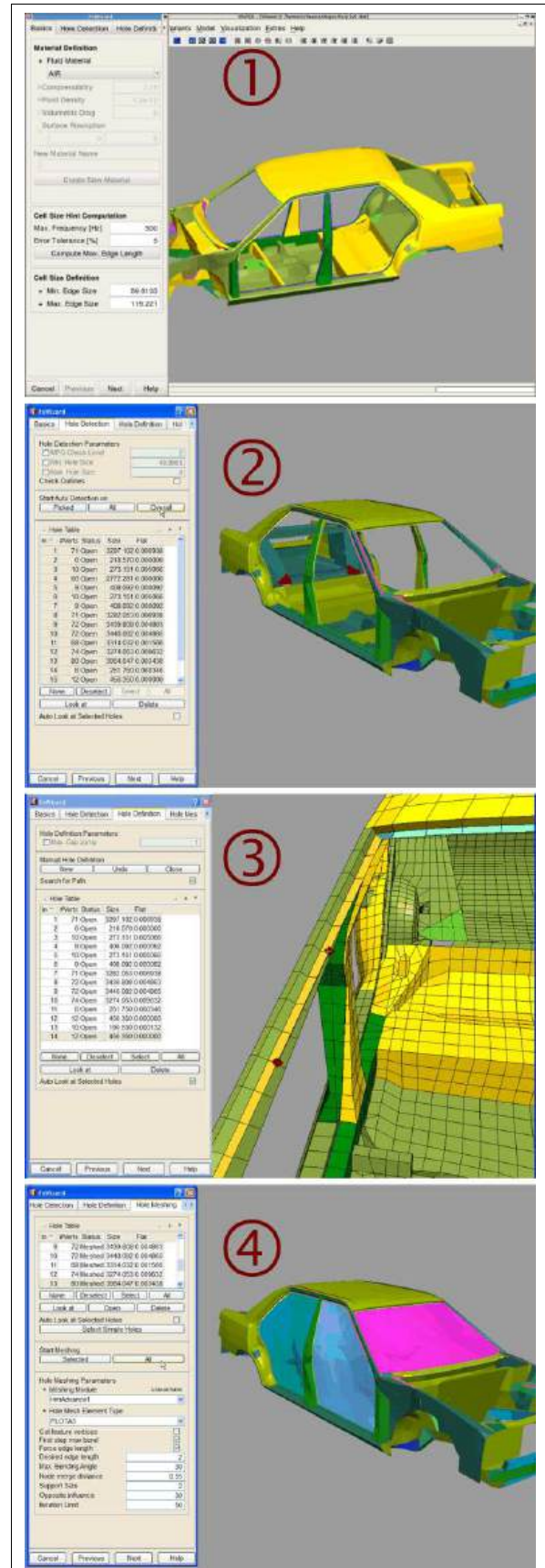


Figure 86: Steps 1 to 4 of the FS wizard

5. Seed point definition is made by the user. This specifies the starting point for the cavity meshing.
6. Cavity meshing is performed automatically. *Voxel* meshing is used and leads to a hexahedra-dominant mesh. The hexahedra-dominant mesh provides a low number of pressure degrees of freedom compared to tetrahedra meshes (where the element orientation influences the result, too).
7. When meshing is finished, there are some elements penetrating the hull of the cavity. These penetrations are resolved by a relaxation process which is performed automatically.
8. The last step automatically generates the coupling elements between fluid and structure. Their nodes have both pressure and displacement degrees of freedom.

Details of the fluid-structure acoustic simulation can be found on page 104.

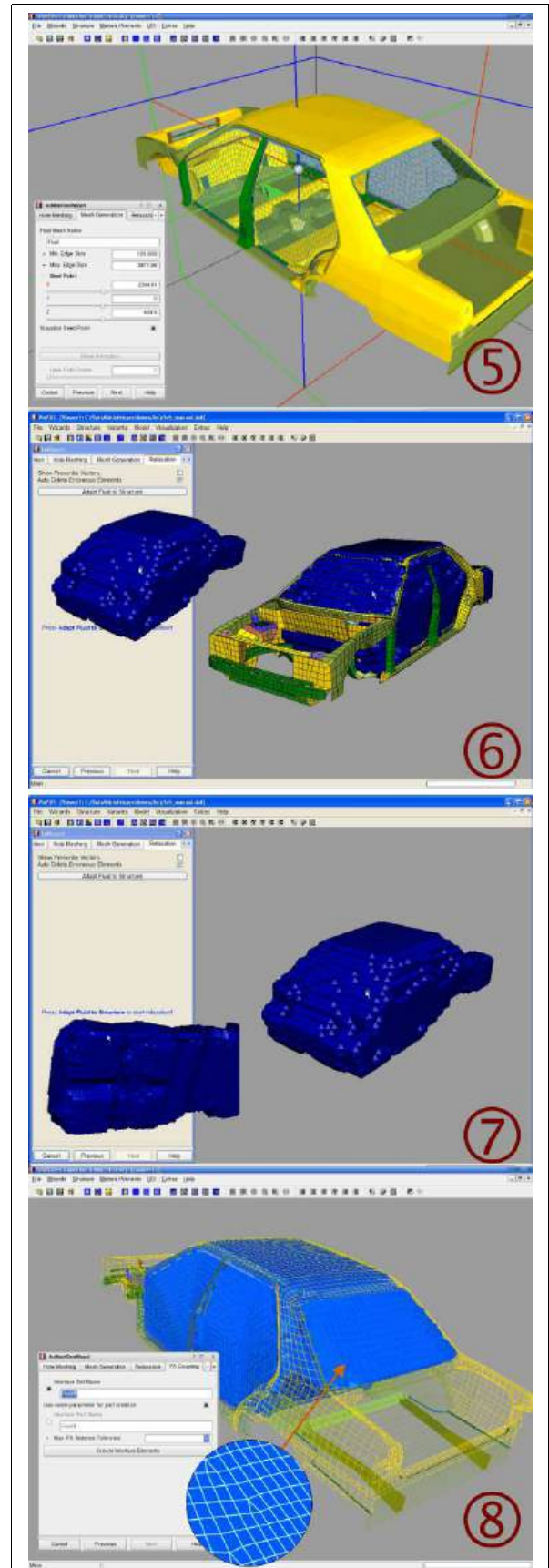


Figure 87: Steps 5 to 8 of the FS wizard

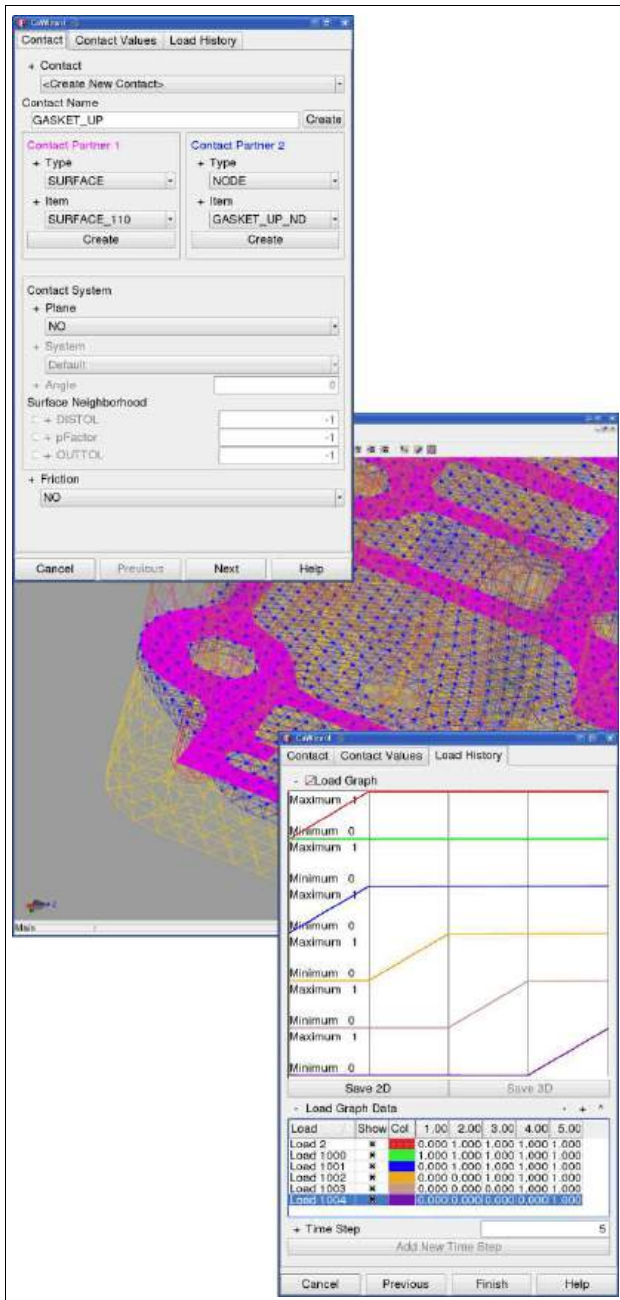


Figure 88: Contact modeling
Contact topology and load history definition

VisPER-CA – Contact Analysis

This module provides two *wizards* supporting the set-up of a contact model and bolt pretension. The details of contact analysis in PERMAS can be found on page 81.

The *contact wizard* provides guidance through the contact modeling by the following steps and provides a graphical check of already existing contact definitions:

- Definition of the contact topology with node-to-node, surface-to-node, or surface-to-surface relationship.
- Definition of the contact details like initial gap width and frictional coefficients. This information is specified as load case dependent data.
- Definition of a load history, which specifies the activation and deactivation for all loadcases in a contact analysis (or in a static analysis in general). See the example in Fig. 88.
- Checking of contact definitions by highlighting while the model is shown by contour lines.

A *pretension wizard* allows an easy description of complex bolt definitions according to PERMAS pretension technology:

- Bolts are detected automatically and grouped with respect to their length and direction.
- The coupling surfaces in the thread area are proposed and can be adjusted or confirmed after visual checking.
- The wizard is capable to detect holes for screw candidates.

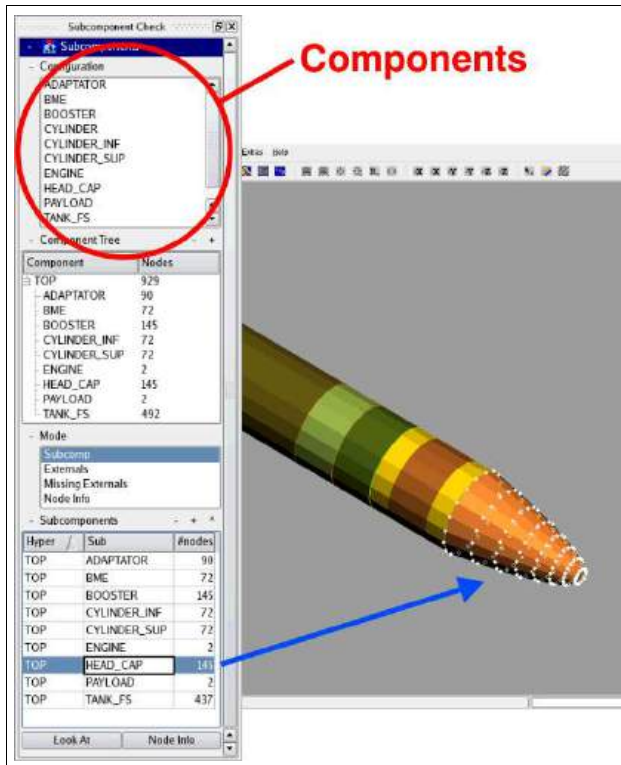


Figure 89: Navigation in the component tree of a substructure model

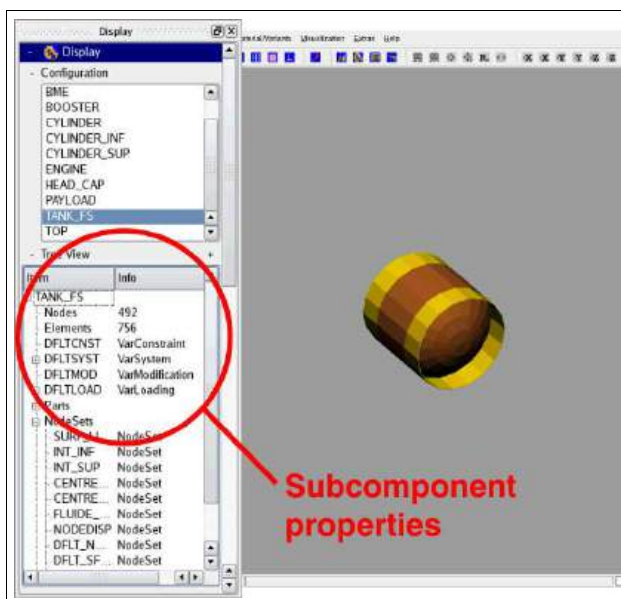


Figure 90: Visualization of components of a substructure model

Substructuring

A *substructure wizard* has been provided to facilitate the set-up of substructure models. This function is part of module VBAS (see page 42).

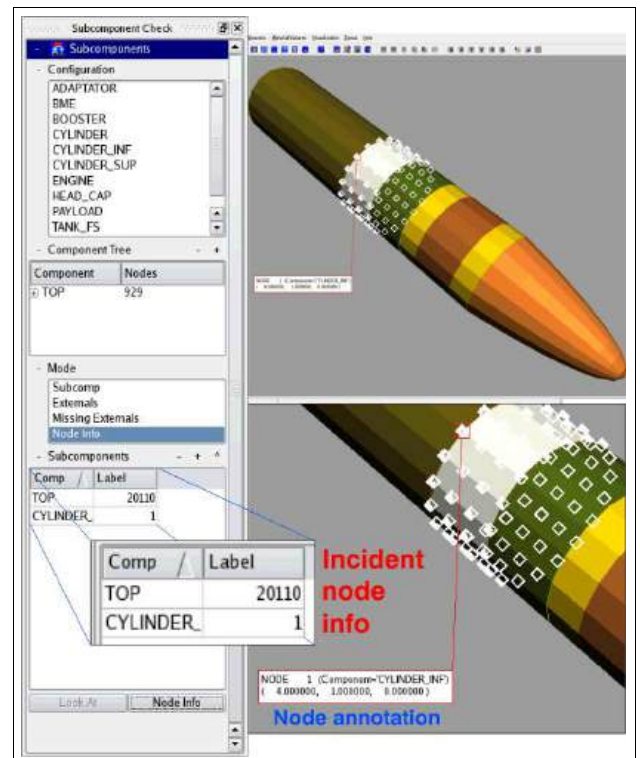


Figure 91: Visualization of coupling nodes of a substructure model

Because VisPER uses the data structures of PERMAS, substructure models can be directly imported. Substructuring is explained in more detail on page 59.

The verification of substructure models includes the following points:

- Navigation through the component tree (see Fig. 89).
- Visualization of single components (see Fig. 90).
- Visualization of coupling nodes (see Fig. 91).
- Check of element properties (see Fig. 92).
- Display of mode shapes on full structure (see Fig. 93).

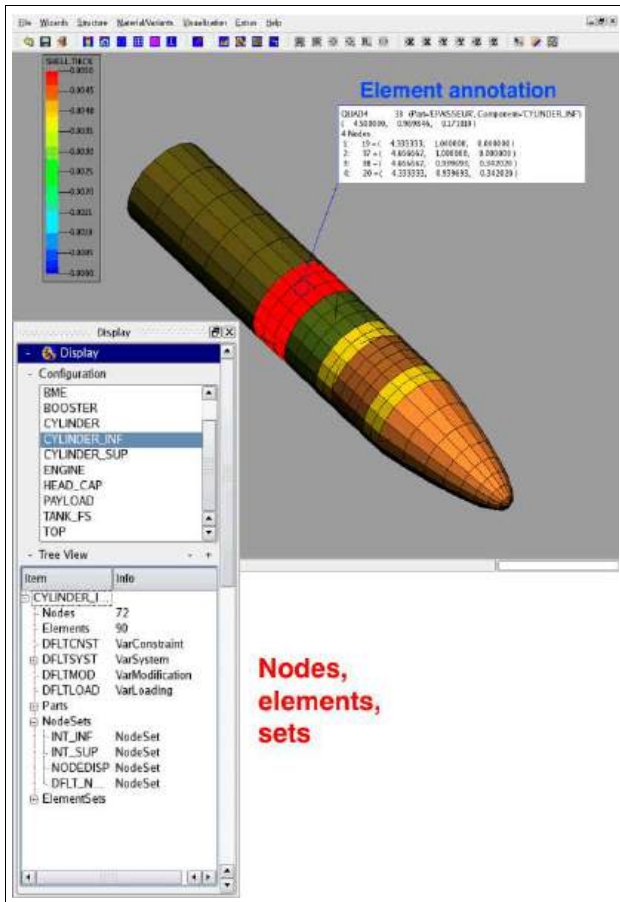


Figure 92: Checking element properties in a substructure model

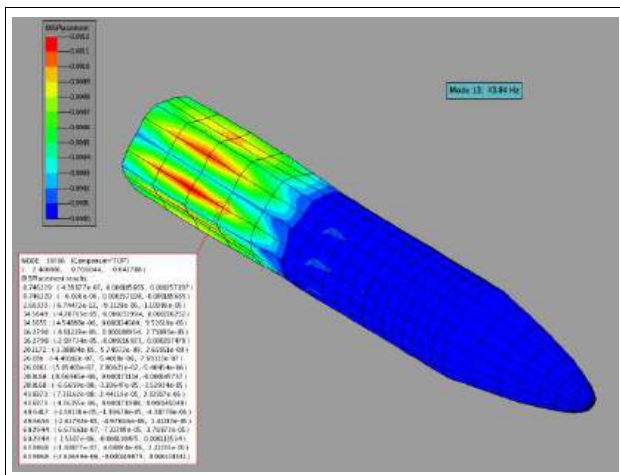


Figure 93: Display and animation of mode shapes on full substructure model

Evaluation of Spotwelds

A special post-processing feature has been included in VisPER providing the evaluation of spotweld forces. This function is part of module VBAS (see page 42).

Certain threshold values can be specified to show green, yellow, and red colour for certain levels of spotweld forces. This can be used as a light signal to mark uncritical, problematic, and critical force values. A full evaluation of spotwelds is possible together with the stresses in the flanges. This is combined with a spotweld representation as half spheres (see Fig. 94). In this way a fast overview on a high number of spotwelds in a large structure is provided (e.g. of a car body). An automatic detection of critical model areas enables a fast navigation in the model.

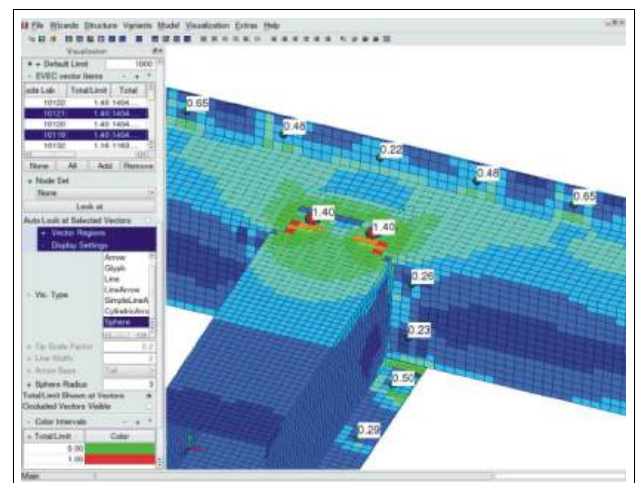


Figure 94: Evaluation of spotweld forces together with element stresses



Figure 95: PCGen logo

VisPER-PCGen – PERMAS Component Generator

PCGen is a modeler add-on for VisPER, specialized in quasi axisymmetric structures such as fluid tanks, see figure 98.

PCGen is meant as a (pre-)design tool. It allows the creation of complete PERMAS models, ready for simulation and even optimization, from scratch and with little knowledge of the PERMAS solver. It allows fast design to simulation loops.

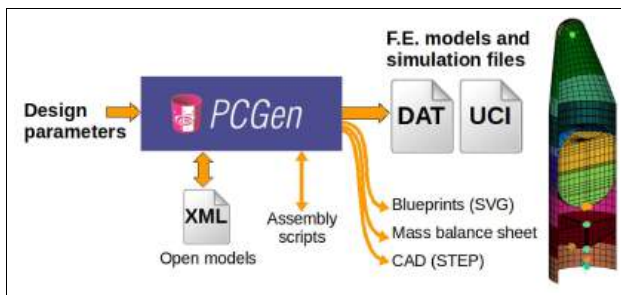


Figure 96: PCGen files

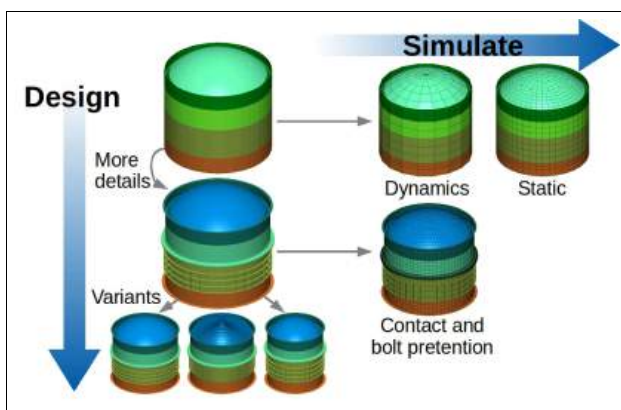


Figure 97: PCGen Workflow

PCGen relies on a library of parametrized objects to generate complete FE models, and requires no

CAD data to generate a mesh. This makes it possible to create complete finite element simulation files (**DAT** and **UCI**) at early design stages, when **CAD** data is not yet available and when design evolves fast. PCGen also generates quickly mesh variants for various analysis purposes (e.g. fine mesh for static buckling and coarser mesh for dynamics). Optionally, the geometry can be exported as a **CAD STEP** format.

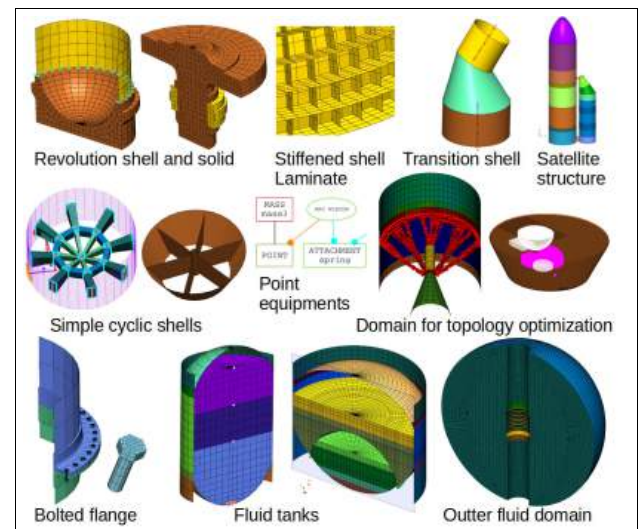


Figure 98: PCGen featured products

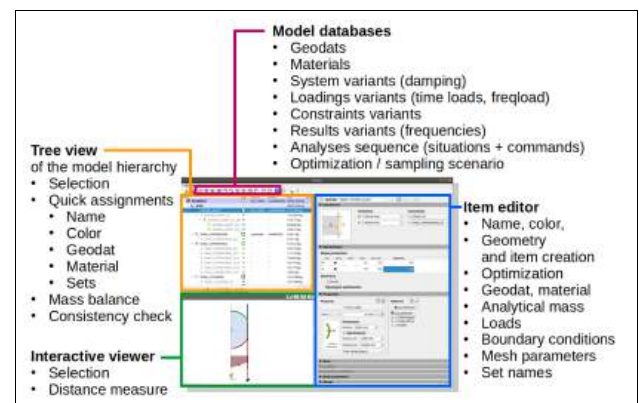


Figure 99: PCGen GUI

The user interface is organized along the main modeling tasks. The model has a hierarchical organization, which allows properties inheritance for a faster modification. PCGen provides various feedbacks to the designer, such as model consistency and completion, and analytical masses. Documentation and tutorials are available.

PERMAS Basic Functions

Substructuring

PERMAS allows to decompose a model into substructures – the so-called **Components**. Like single elements in a FEA-model, these Components may be inserted into a superior structure – the **Configuration**.

- The number of Components is not limited and **each Component** may be **arbitrarily large and complex**.
- Components and Configurations are identified by user defined names.
- Each Component has its own name index for element-, node-IDs, etc.
- Each Configuration may consist of an arbitrary number of Component levels and each level may contain elements, loads and constraints. The specification of the coupling degrees of freedom in each component (the so-called 'external' degrees of freedom) allows the **automatic assembly** of the complete Configuration. The *condensation* of the components is performed using *Guyan's* reduction.
- In addition to the *static condensation* using *Guyan's* reduction a *dynamic condensation* using the *Craig-Bampton* method or the *Mixed-Boundary Craig-Bampton* method is available.
- Specific reordering concepts in conjunction with explicit and iterative condensation schemes and automatic selection of algorithms provide **highest solution efficiency**.

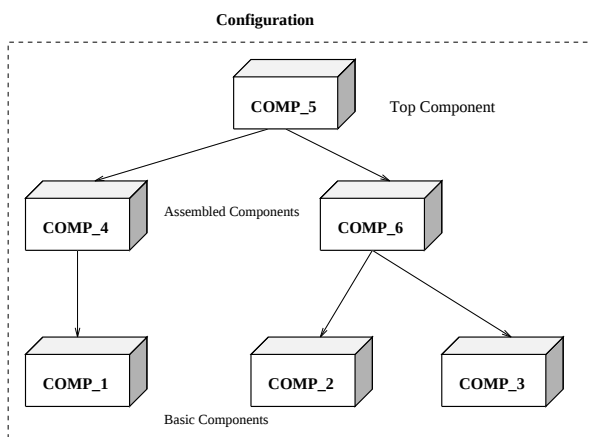


Figure 100: Substructuring in PERMAS

- The data base can hold an arbitrary number of Configurations.

- Multiple Configurations may share the same Components.
- Components may be extracted and saved for future substructuring either by their model description or by the corresponding condensed matrix models (see section Matrix Models, page 73).
- Such matrix models may be forwarded to customers and suppliers in place of the real geometric models.
- Substructuring permits the separated modeling and verification for all parts of the structure, prior to the final assembly.
- Single FEA models from distinct modeling sources, can be easily combined.
- Areas with design variations or nonlinear properties may be assembled into separate Components, thus concentrating modifications and iterations on this Component only. This will measurably cut computation time and resources.

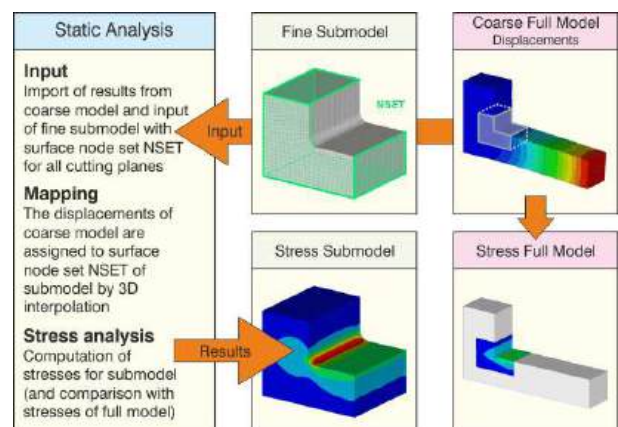


Figure 101: Submodeling in static analysis

Submodeling

This feature supports the use of previously calculated results from a coarse (global) model as boundary conditions for a refined mesh of a part of the model. This enables e.g. a subsequent more precise analysis of stresses (see Fig. 101).

In a static analysis the displacements at the boundary of the refined part are taken from the full model as prescribed values.

For temperature fields, the interpolation on the side of the fine mesh is usually done in 3D (see Fig. 102).

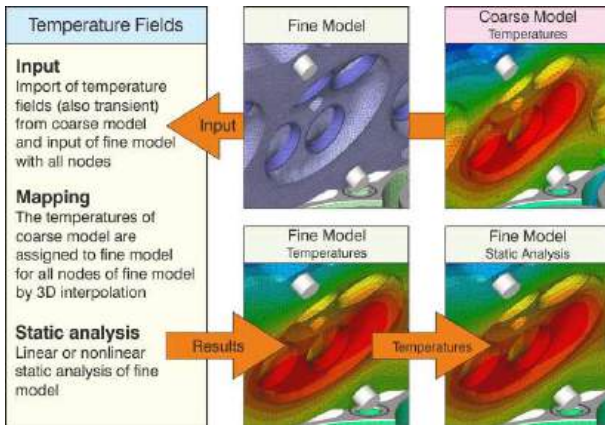


Figure 102: Mapping of temperature fields

Sometimes temperature fields have also to be extrapolated to nodes which do not yet have temperatures (e.g. additional parts like gaskets or bolts in engine analysis). This extrapolation can be performed by the mapping or by an additional temperature field analysis (or both as explained by the example in Fig. 103).

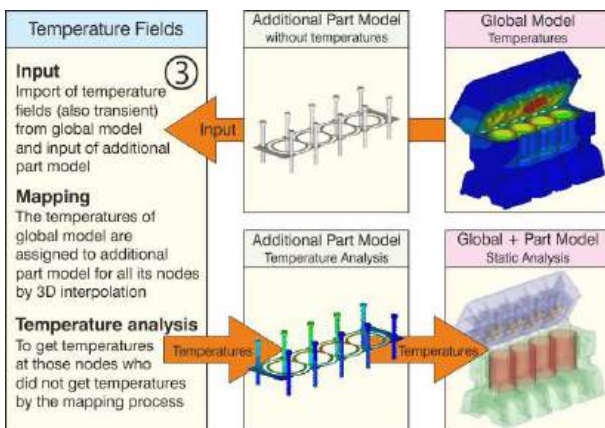


Figure 103: Mapping with extrapolation of temperature fields

Variant Analysis

PERMAS offers an easy way for examining different variants of one FEA model. Variable model properties are held within the so-called **Variants**:

- **System Variant:**
 - material assignment of elements,
 - element properties (thickness, cross section etc.),
 - element local coordinate systems.
- **Constraint Variant:**
 - suppressed degrees of freedom,
 - prescribed degrees of freedom,
 - contact definitions,
 - coefficients of general kinematic constraints,
 - local coordinate systems for the degrees of freedom at each node.
- **Loading Variant:**
 - Dynamic loads and an arbitrary number of static load cases or combinations hereof.
- **Result Variant:**
 - For more detailed specification of required results, like
 - * load pattern combination rules,
 - * list of excitation frequencies for frequency response,
 - * load steps where results are requested.
- **Modification Variant:**
 - specification of design model for sensitivity analysis and optimization.

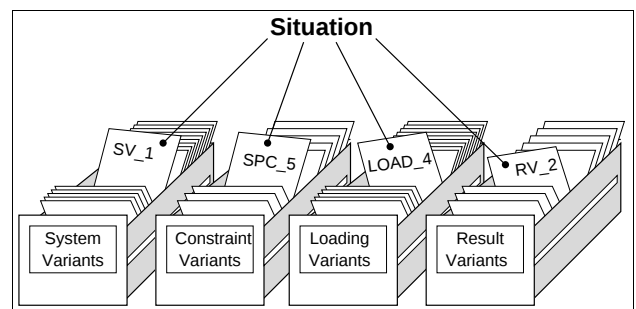


Figure 104: Variants in PERMAS

Basic properties like nodal point coordinates, element topology and global coordinate systems are invariant.

Single Variants may be selected and examined together as a so-called **Situation**.

- Variants and Situations are identified by user defined names.
- The number of Variants and Situations is not limited.
- There is no restriction in combining the variant definitions with any **substructuring** feature.
- In each substructure the selection of system, constraint, loading and result Variants is independent of the selections made for other substructures (e.g. useful for models with symmetry/antisymmetry).
- PERMAS keeps a record of all calculation steps already completed. In this manner, multiple cal-

culations are avoided when using a Variant repeatedly.

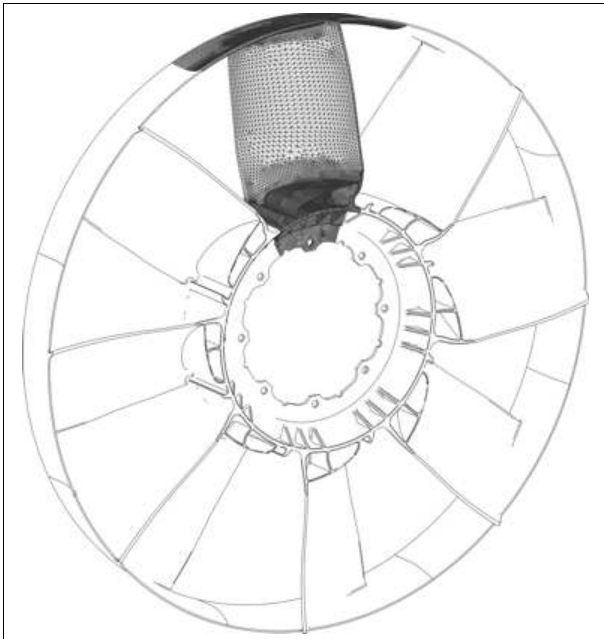


Figure 105: Cyclic symmetry of a fan
Behr GmbH & Co., Stuttgart, Germany.

Cyclic Symmetry

Cyclic symmetric structures like Fig. 105 are characterized by a number of identical sectors each rotated by a multiple of the sector angle around the axis of symmetry.

Cyclic symmetric structures can be handled directly for static analysis with matching cyclic symmetric loads and real eigenvalue analysis (see modules LS and DEV on pages 81 and 97). There, the analysis of the whole structure is replaced by a series of analyses for one sector with different boundary conditions. In addition, the number of nodal diameters has to be specified, i.e. number of waves along the perimeter.

Substructuring can be used to model a sector and the analysis of cyclic symmetry is performed in the top component.

In topology optimization (see module TOPO on page 114) the result can be forced to become cyclic symmetric.

A *Coleman transformation* or *Multi-Blade Coordi-*

nate transformation (MBC transformation) is available for rotating cyclic symmetric structures. A real eigenvalue analysis based on a previous static analysis is used to start a complex eigenvalue analysis to evaluate the stability of a rotor with anisotropic elastic support.

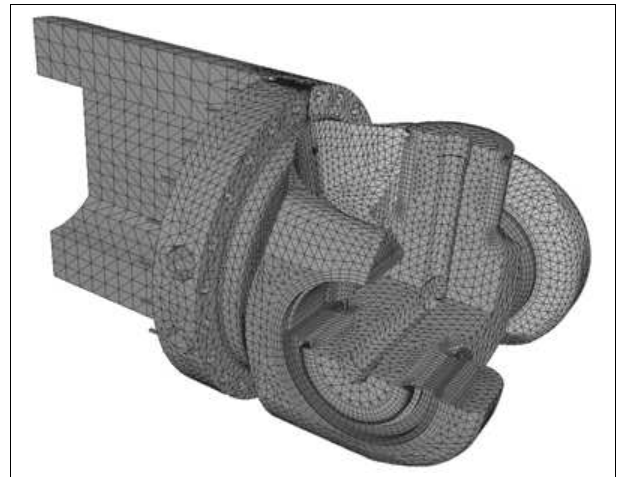


Figure 106: Incompatible meshes
Cardan shaft model in sectional view
(Voith Turbo GmbH & Co. KG)

Surface and Line Description

The description of surfaces in PERMAS is used for the specification of structural parts which have to be coupled automatically (see next section). This description is made using one of the following methods:

- by specifying element surfaces, e.g. for a set of volume elements (for Sets see page 69),
- by specifying geometry elements (see element library page 66).

For surfaces very accurate coordinates are frequently required (as e.g. for contact with incompatible meshes), which are not available through the pre-processor used. Such surfaces can be smoothed internally by correction of runaways, which leads to very precise surfaces for more accurate analysis results.

For surfaces with quadratic elements, a linearization is available, which makes the midside nodes linearly dependent on the corner nodes. If these surfaces are used in contact definitions, then the surfaces will

also provide contact pressure results. A special export item allows to visualize the surface definitions. Definition and visualization of surfaces are fully supported by VisPER.

In the same way does the description of lines serve as specification of structural parts which have to be coupled automatically along lines (see next section). This description is made using one of the following methods:

- by specifying element edges, e.g. for a set of face elements (for Sets see page 69),
- by specifying geometry elements (see element library page 66).

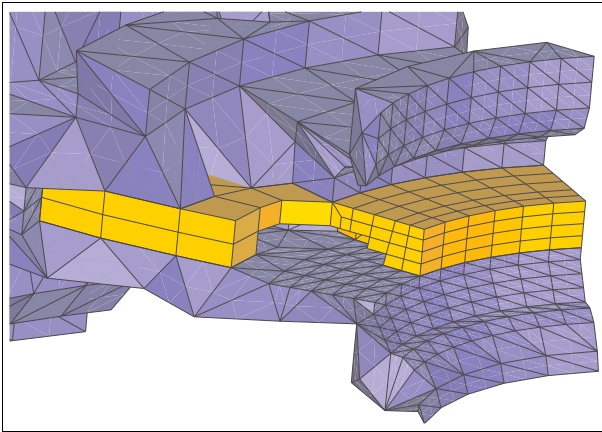


Figure 107: Incompatible meshes
Element transition (HEXE8/TET10)

Automated Coupling of Parts

The automated coupling of incompatible meshed parts brings a number of benefits for the user:

- It allows for a much more flexible organization of the model generation where single parts are administrated and exchanged.
- It facilitates a fast modification cycle for virtual prototypes.
- Accurate mesh transitions without bad elements are possible.
- Welding spots and surface contact become much easier with automated part coupling.

The coupling definition consists of the following steps:

- Definition of a guiding surface and the degrees of freedom to be coupled

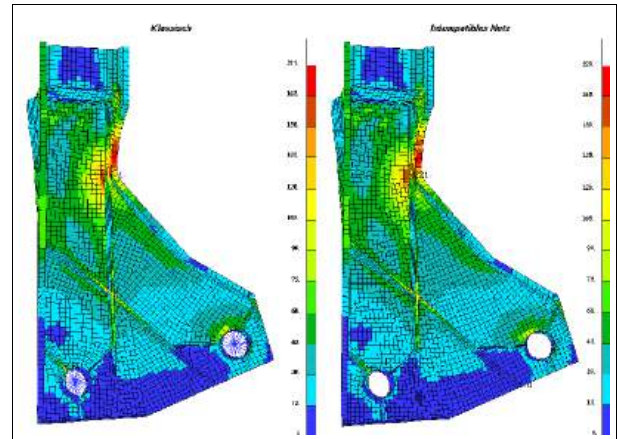


Figure 108: Ribbed shell model
with compatible meshes (left) and
incompatible meshes (right)

- Definition of a dependent surface or node set

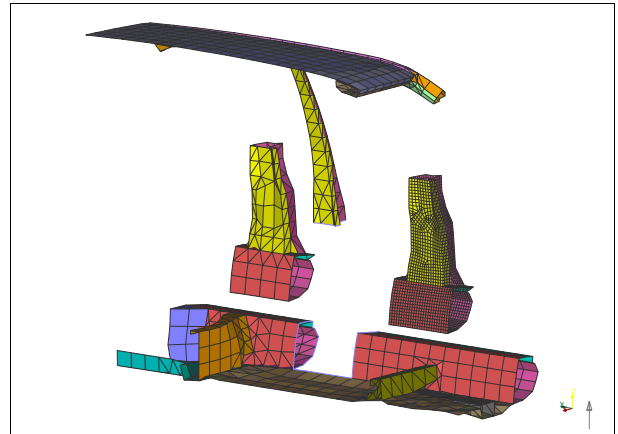


Figure 109: Application of local mesh refinement

Subsequently the neighborhood computation takes place and the parts are connected by **MPC**-conditions automatically. The result of the neighborhood computation is available for post-processing and verification purposes.

The coupling is a general feature that may also be used for coupled analyses, where different mesh densities occur due to the modeled physics. One example is a coupled fluid-structure acoustic computation, where the acoustic mesh may be coarser than the mechanical part.

One valuable advantage of incompatible meshes is their use in shape optimization. The position of bolts or ribs can be optimized in a shape optimization without any remeshing of the structure. Fig. 111 shows an example, where the initial position and the

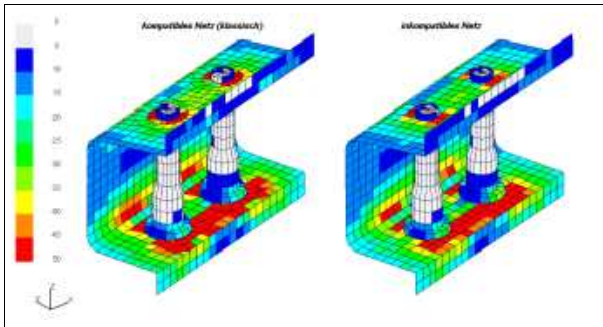


Figure 110: Results for part assembly with compatible (left) and incompatible (right) mesh

final position of the bolts are very different. Nevertheless, after a number of iterations the final position is reached which is still symmetric and shows identical bolt forces for all the bolts.

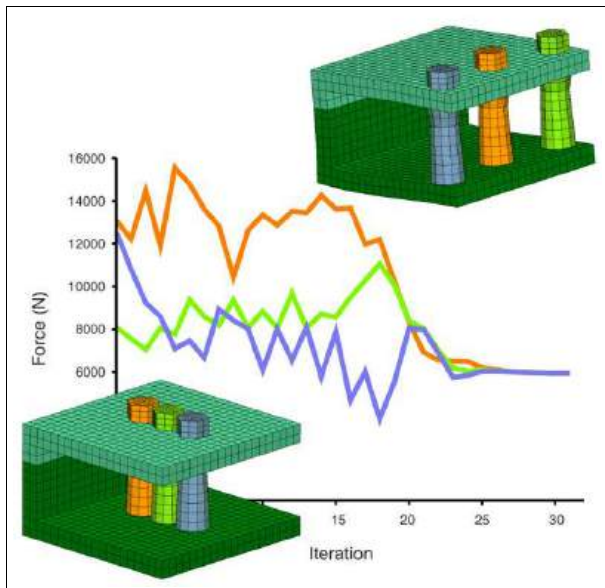


Figure 111: Shape optimization with incompatible meshes to achieve smallest relative displacements and identical bolt forces (top: initial design, bottom: final design; in between: bolt forces for each iteration)

Automated Spotweld Modeling

For the automated modeling of spotwelds neighbouring surfaces are connected using predefined points:

- Specification of spotweld positions
- Selection of (incompatible) faces

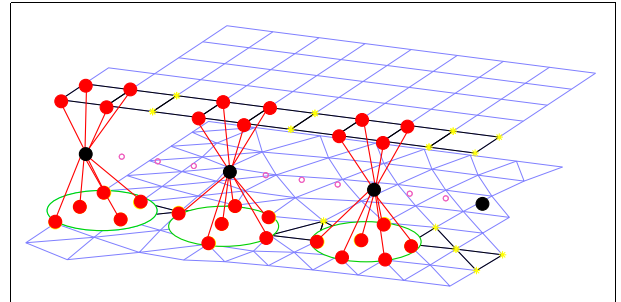


Figure 112: Connection of spotweld forces

- Specification of spotweld stiffness and (optionally) the spotweld diameter



Figure 113: Spring element at spotweld position

The spotweld stiffness is modeled by a spring element, which is coupled to the neighboured parts by automatically generated *MPC* conditions (see Fig. 113). Available results are the spring forces and the reaction forces at the coupling nodes of the joint parts.

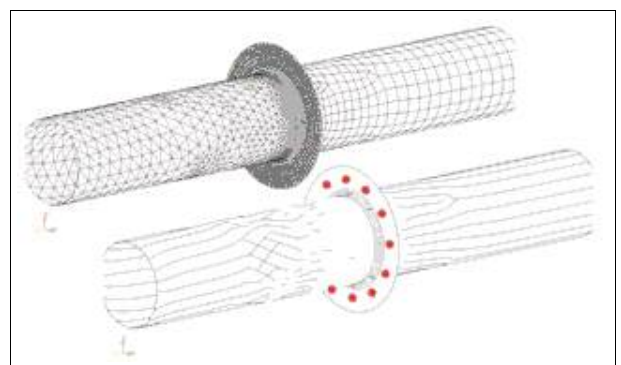


Figure 114: Spotwelds and incompatible meshes

For verification purposes the topology and the connecting vectors of the generated spotweld connections can be issued for graphical post-processing (see Fig. 115).

A more sophisticated spotweld model has been de-

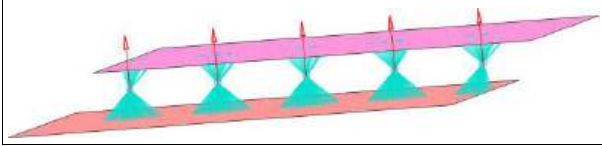


Figure 115: Verification of spotweld connections

veloped with module WLDS (see page 123). Using very simple modeling it achieves best possible results.

Local Coordinate Systems

Local coordinate systems are used as reference for node coordinates or other coordinate systems to facilitate the model description. Also, rotated node bases, loads, and anisotropic materials may make reference to local coordinate systems. The types of local coordinate systems can be seen from Fig. 116. Local coordinate systems may be defined in a visual manner by VisPER.

Kinematic Constraints

For the specification of single point kinematic constraints (absolute constraints) suppressed and prescribed degrees of freedom are available for zero or non-zero displacements, respectively.

Multilinear kinematic constraints (relative constraints) between several degrees of freedom are described by the use of so-called Multi Point Constraints (MPCs).

PERMAS offers a great number of various MPCs – all of them comfortable tools for modeling:

- Multiple degrees of freedom may be forced to have identical freedom values by simple **Assignment** (for modeling swivels, hinges or sliding surfaces and for boundary conditions in cyclic symmetry).
- **Rigid Bodies** allow the modeling of rigid parts within an elastic structure. There can be one or several guiding degrees of freedom and one or several dependent degrees of freedom, too.
- **Interpolation Regions** may be used for mesh refinements, coupling of incompatible meshes,

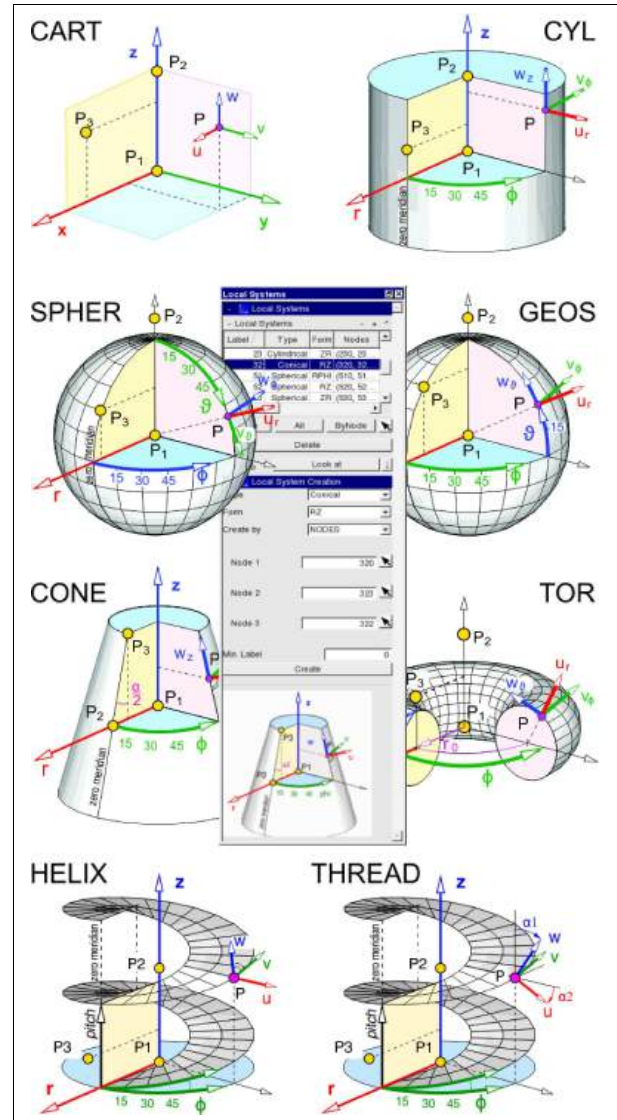


Figure 116: Local coordinate systems in VisPER

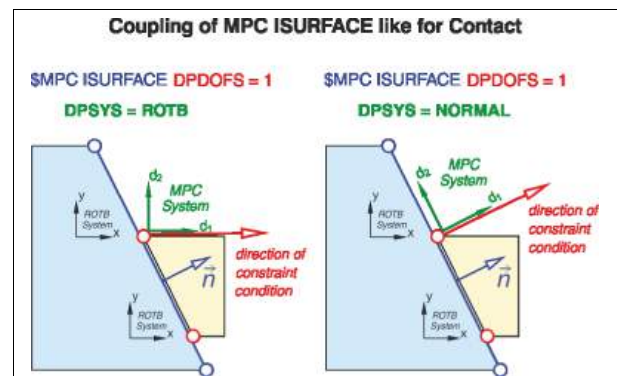


Figure 117: Surface coupling

distributing loads, or transfer of results between different meshes:

- lines with 2 or 3 guiding nodal points,

- triangular and quadrilateral areas with 3 or 6 and 4, 8 or 9 guiding nodes, respectively,
- Volume areas as hexahedra (with 8, 20, 27 nodes), pentahedra (with 6, 15, 18 nodes), tetrahedra (with 4, 10 nodes), pyramids (with 5 nodes).
- **Coupling with surfaces** may be used for coupling of incompatible meshes. All degrees of freedom at all nodes belonging to dependent node set, dependent surface or dependent surface set are forced to have displacements (or temperatures etc.) which are determined by interpolation of the corresponding values at the nodes belonging to guiding surface or guiding surface set. This coupling may be used also for a **Volume-Shell Transition**, i.e. the connection of plates and shells with solid element structures (see Fig. 118). If the dependent nodes do not lie in the guiding surface, then either the node is projected onto the surface or a rigid lever arm is generated in order to represent the kinematics of the coupling correctly.

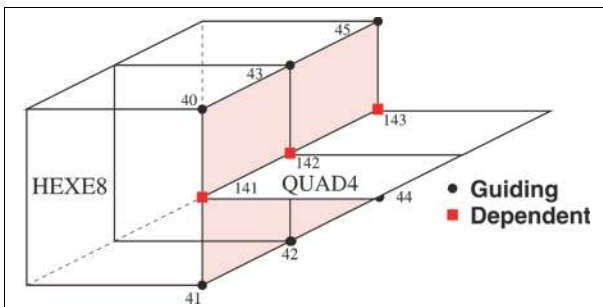


Figure 118: MPC example: Volume Shell Transition

- **Coupling with volumes** may be used for coupling of incompatible meshes. All degrees of freedom at all nodes belonging to dependent node set are forced to have displacements (or temperatures etc.) which are determined by interpolation of the corresponding values at the nodes belonging to guiding volume, which is defined by referenced elements.
- **Press fit** to conveniently model e.g. shaft-hub connections where the shaft has a slightly larger outer diameter than the inner diameter of the hub. Typically, such configurations are modeled using contact. If any change in the contact is not expected, it would be more efficient to use a general MPC to model the interference in order to keep a problem in the linear range instead of a nonlinear contact analysis. It was proven that contact analysis and general MPC give identi-

cal results for corresponding models (see Fig. 119). A press fit command supports the automatic generation of the MPC conditions.

- **General MPCs** allow any linear combination of the involved degrees of freedom.

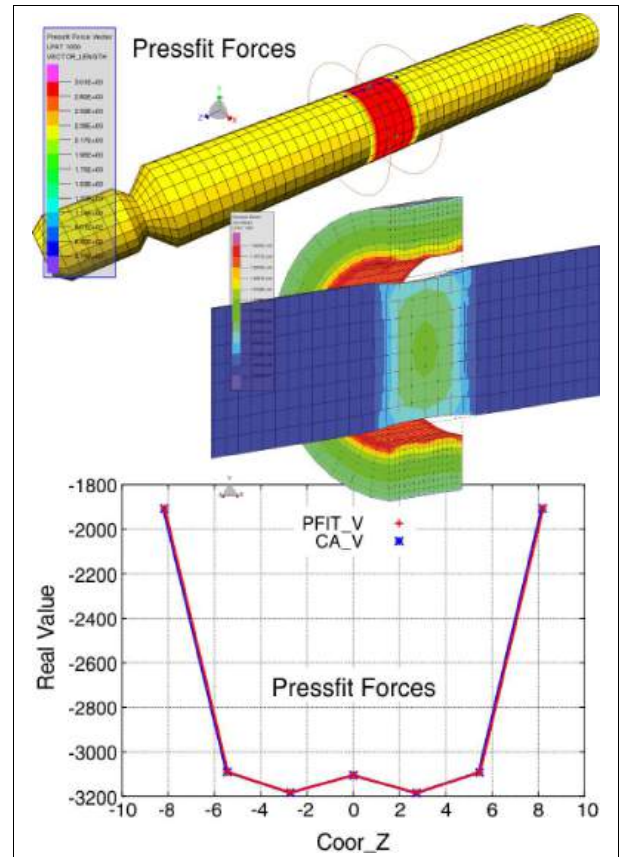


Figure 119: Press fit between shaft and hub
Identical stress results of MPC and contact solution.

The number of dependent degrees of freedom for each MPC is not limited. Also multi-level conditions (hierarchical MPCs) may be used as long as there is no recursive interdependence.

The coupling of single components using the *sub-structure technique* is defined by 'external degrees of freedom'. On the higher component's level these external degrees of freedom can be part of single or multipoint constraints.

Handling of Singularities

In static analysis there are two different kinds of singularities, which are detected automatically and

communicated to the user in a suitable way:

- For redundant degrees of freedom, which have no stiffness (e.g. perpendicular to rods or membranes), the user gets a list on the result file.
- For rigid body degrees of freedom the related displaced shapes are issued on a post-processor file. They can easily be inspected in order to detect the missing supports or other modeling errors.

VisPER provides an RBM Assistant (see page 45) which allows for the elimination of rigid body modes prior to analysis.

In dynamic mode analysis the rigid body modes are detected and decoupled automatically.

In dynamic response analyses in the time or frequency domain the absolute response results from a superposition of the elastic and the rigid body response.

Element Library

The PERMAS elements are generally usable for different types of physical degrees of freedom (like displacements, temperature, electromagnetic potential, etc.), beside some specific, application-dependent elements. Currently, the following elements are provided (see also Fig. ELTS):

- **Solid Elements:**
 - tetrahedra with 4 or 10 nodal points and straight or curved edges,
 - pyramid element with 5 nodal points,
 - pentahedra with 6, 15, or 18 nodal points and straight or curved edges,
 - hexahedra with 8, 20, or 27 nodal points and straight or curved edges.
 - hexahedra with 8, 20, or 27 nodes, pentahedra with 6, 15, or 18 nodes, tetrahedra with 4 or 10 nodes, and pyramid element with 5 nodes as *fluid elements*.
 - Two different special hexahedra with 8 nodes for dynamic visco-elastic calculations (see page 103).
- **Flange (Rod) and Membrane Elements :**
 - flange elements with 2 or 3 nodal points,
 - triangular elements with 3 or 6 nodal points and straight or curved edges,
 - quadrilateral elements with 4, 8 or 9 nodal points and straight or curved edges,
 - quadrilateral shear panel with 4 nodal points.
- **Beam Elements** with 2 nodal points, optionally with or without rigid lever arms (offset nodes):
 - beams with arbitrary shaped solid cross section,
 - thin-walled profiles and tubes with open or closed cross sections,
 - thin-walled open or closed tubes with cross sections tapered along the beams' length axis,
 - fluid-filled or fluid-surrounded straight or curved pipe elements.
- **Plate and Shell Elements :**
 - triangular or quadrilateral elements for thin and moderately thick plates and shells.
 - triangular or quadrilateral elements *sandwich shells*.
 - a thin-walled triangular plate element following *Kirchhoff's* Theory.
 - triangular (with 3 or 6 nodes) and quadrilateral elements (with 4, 8, or 9 nodes) with 3-dimensional shell formulation for linear and non-linear material.
 - triangular and quadrilateral elements for layered cross-sections (*composites*).
- **Discrete Elements:**
 - linear and non-linear spring elements,
 - various mass elements,
 - linear and non-linear damper elements,
 - control elements,
 - elements with direct matrix input,
 - hysteresis elements.
- **Elements for contact analysis:**
 - Gasket elements defined by a nonlinear pressure-closure curve. This curve can also reflect an additional gap to the neighbored part in the unloaded initial configuration.
 - Compensation springs to ground or between 2 nodes with 3 translational DOF, or between two nodes with 1 translational DOF.
- **Scalar Elements:**
 - spring elements between 2 degrees of freedom or for a support of 1 degree of freedom to ground,
 - damper elements between 2 degrees of freedom or for a connection of 1 degree of freedom to ground,
 - scalar masses for 1 or 2 nodal points,

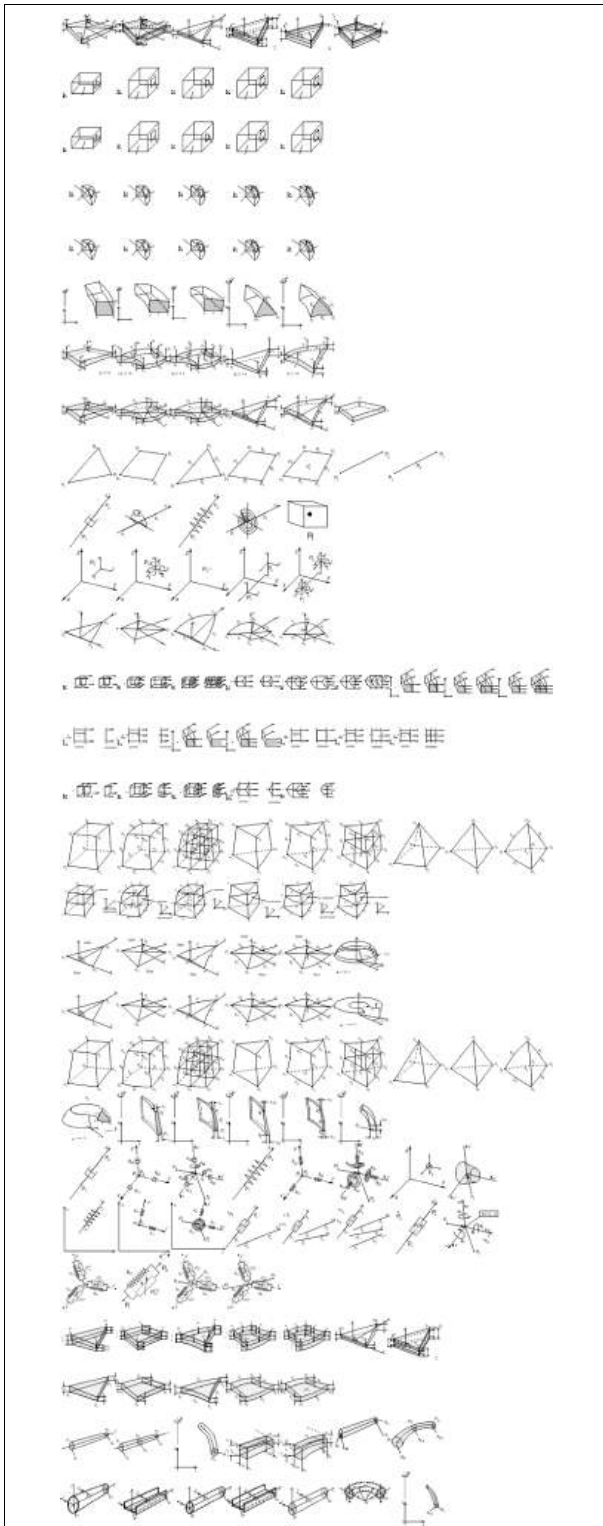


Figure 120: Overview on element types

- scalar mobility element for fluid meshes.
- **Load carrying membranes** in form of triangular respectively quadrilateral areas for load application and stress evaluation.
- **Plot Elements** in form of points, lines and trian-

gular respectively quadrilateral areas for result evaluation.

- **Geometry Elements** in form of lines and triangular respectively quadrilateral areas for line and surface definition.
- **Convectivity Elements** to model the convectivity behavior and radiation on free surfaces in thermal analysis.
- **Fluid-Structure Coupling Elements** for coupled acoustics and surface absorption.
- **Surface Wave Elements** for acoustic analyses.
- **Semi-Infinite Elements** for acoustic and electromagnetic analyses.
- **Radiation Boundary Condition (RBC) Elements** for the modeling of acoustic radiation effects.
- **Axisymmetric Elements** for structures, heat transfer, acoustics, and electromagnetics.

Standard Beam Cross Sections

The following standard geometries are supported for beam cross sections (see also Fig. 121):

- General thin-walled open section,
- Thin-walled circular cross section,
- Thin-walled box cross section.
- Solid circular cross section,
- Solid box cross section.

Using those cross sections allows to get stress results at certain points of the cross sections.

The parameters of the cross sections are directly available as design parameters in an optimization run (see page 109).

Design Elements for Optimization

For design optimization purpose (see page 109), all parts of the structure which may be modified are assigned to so-called Design Elements. They are used to define possible modifications for elements and nodes belonging to the design element.

To facilitate shape optimization a number of additional design elements are available to collect structural elements belonging to one design element. All

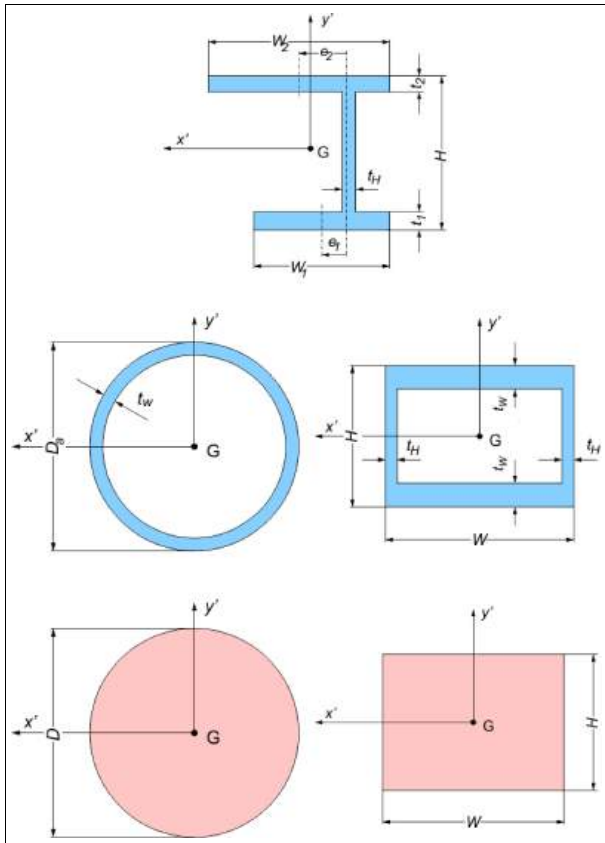


Figure 121: Standard beam cross sections

types of finite elements are allowed to be part of these design elements:

- design element for smooth shape optimization,
- design element for bead design,
- design element for free-form optimization,
- design element for topology optimization.

SPR Stresses and Absolute Error Indicator

The element size in FE meshes influences the accuracy of the results (in particular stress results). But the 'right' element size is a local characteristic and depends on the force flow and its gradients. Particularly, notch stresses are highly dependent on element size.

Beside the standard classical stress calculation another stress calculation of *SPR stresses* is provided to improve this alternative stress calculation. The standard classical stress calculation is not changed. The *Smooth Patch Recovery Method* has been pub-

lished by *Zienkiewicz*. This method aims to get more accurate stresses by using a patch of neighbored elements instead of just one element (see Fig. 122). SPR stresses are available at element nodes (for volume, flange, membrane, and shell elements) and nodal points. This include also principal stresses.

In addition to the SPR stresses, reliable *Stress Gradients* are available for the normal of surfaces. These gradients could be used for fatigue life calculations.

In addition to the SPR stresses, an *Absolute Error Indicator (AEI)* is now available, which provides the difference between the classical stresses and the SPR stresses. The value is an absolute indicator, because it is not a relative deviation of the stresses but the direct difference and, therefore, has the same unit of measurement as the stresses.

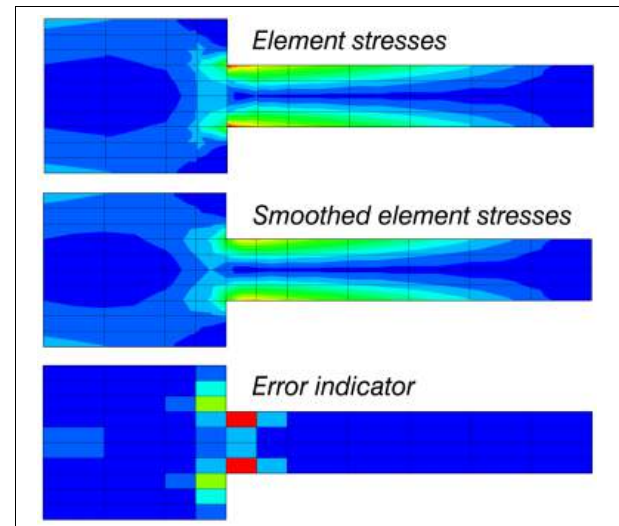


Figure 122: Element stresses, smoothed element stresses, and absolute error indicator

Material Properties

The description of material is made independently of the other model data. In doing so, the needed material parameters can easily be taken from already existing data sets building a certain kind of material data base.

Beside isotropic material all kinds of *anisotropy* can be applied in a simple and suitable way: transverse isotropic, orthotropic, monoclinic, and general (triclinic) material.

Because of the application of PERMAS in different fields, different material properties can be defined like elasticity, density, compressibility, damping, thermal expansion, heat conductivity, heat capacity, absorption, volume drag, electric conductivity, dielectricity, magnetic permeability. There, only the required data are really used, all additional material properties don't have any effect.

For nonlinear material, additional input facilities are provided like stress-strain curve, yield load, creep behavior.

Almost all material properties can be specified as temperature-dependent. Following a given temperature field, the actual material properties are determined by an interpolation from the given distribution function.

The definition of damping can be frequency-dependent.

Layered composites (laminates) can be defined by different materials (like fibre-reinforced material) for each layer (see page 122).

Um bei einem Datenaustausch mit Kunden und Lieferanten die Materialdaten in den Modelldaten nicht offenlegen zu müssen, können diese verschlüsselt werden. PERMAS kann die Daten lesen und verarbeiten, legt sie aber garantiert nicht offen.

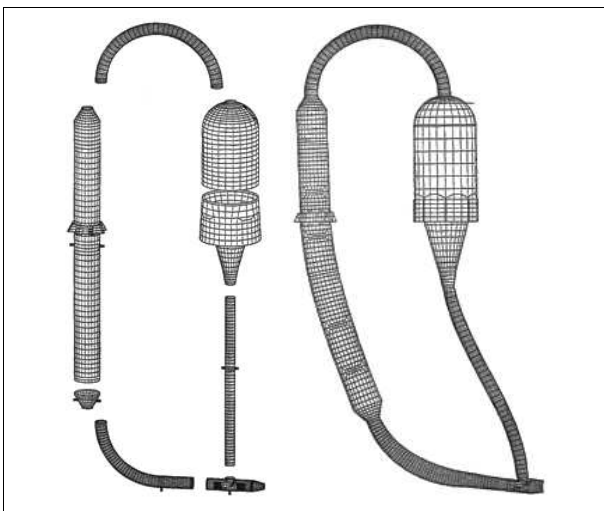


Figure 123: Mode shape of a reactor

Sets

For a lucid model description node and element sets may be used. Such sets may be generated from other sets using Boolean operations. In addition, several sets can be collected in a set bin, which denotes a set of sets.

Beside the model description such sets may be used also for the specification of result output. For the print output of results and the export to connected post-processors the amount of issued results can be restricted to the relevant data by sets. In case of analyses in the time or frequency domain, the required run time and disk space can be drastically reduced by the specification of sets for which the results are to be determined.

All interfaces preserve the sets and their identifiers from the pre-processing via the solver to the post-processor. An accompanying text marks a set more precisely.

Mathematical Functions

For the description of complex data relations a library of mathematical functions is provided for: polynomes, trigonometric functions, exponential functions, and discrete functions. E.g. they allow for the definition of time-dependent loading and temperature-dependent loads.

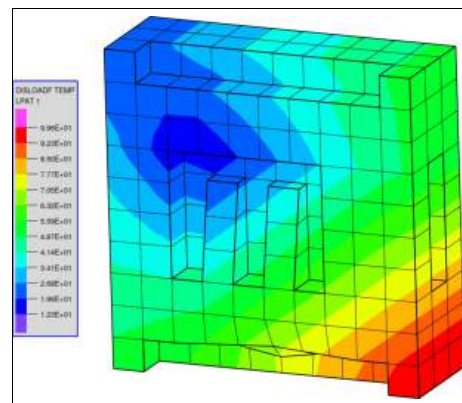


Figure 124: Function evaluation

The most convenient way of using functions is to directly formulate the function via a math notation with option *FORMULA*.

Beside the library functions also tabular functions or user functions (defined by Fortran or C subroutines) may be defined and used.

On all of these functions sums, products, or chains may be defined in order to specify complex compound functions.

In VisPER, a special functions dialogue is available to prepare, manipulate, and visualize functions (see Fig. 125).

In addition, a special function to derive the *Fourier* coefficients of a time signal is provided. Time-dependent excitations are analyzed by a Fourier transformation and the resulting frequency-dependent coefficients are used in a frequency response analysis (see Fig. 126).

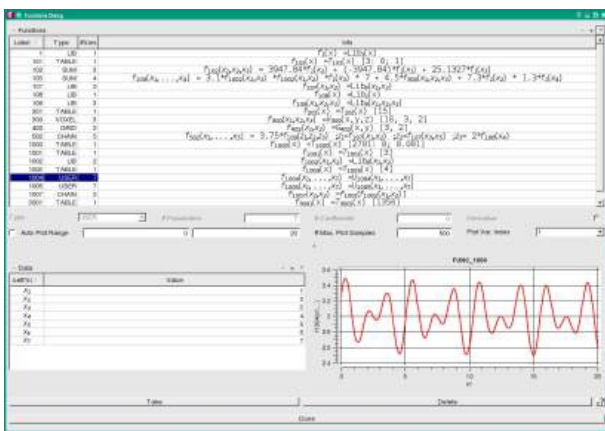


Figure 125: Functions dialogue in VisPER

Loads

In static analyses the following mechanical loads may be used:

- Global loads:
 - Inertia loads,
 - *Inertia relief* (quasi-static acceleration).
- Nodal loads:
 - Concentrated loads,
 - Distributed loads,
 - Prescribed displacements,
 - Temperature loads,
 - State of contact.
- Element loads:
 - Distributed loads,
 - Initial strains.
- Surface loads:
 - Distributed loads,
 - Hydrostatic pressure.

For distributed surface loads, e.g. not specified at the nodes of a mesh but a coarser grid, special load elements (see page 67) can be used which are coupled to the main structure by interpolation (see page 64).

In heat transfer analyses the following thermal loads may be used:

- Nodal loads:
 - Concentrated heat flows,
 - Distributed heat flow,
 - Prescribed temperatures.
- Element loads:
 - Distributed heat flow.

Temperature fields may be taken directly from a previous heat transfer analysis. One temperature field may be used to define temperature dependent material properties, and another temperature field may be used for initial strain calculation or as initial condition for a transient thermal analysis.

All distributed element loads can be defined as coordinate-dependent.

Load case combinations may be applied to create new loads from existing patterns.

Time-dependent loads are defined as a product of a static or thermal load and a time-dependent function. In addition, initial conditions for time-dependent mechanical or thermal analysis can be

specified. These can simply be taken from a previous analysis.

For frequency-dependent dynamic loads it always holds that they are defined as a product of a static load and a spectrum of amplitudes and phase angles.

Time dependent loads of a periodic process can be transformed to frequency dependent loads by an automatic *Fourier* analysis (see Fig. 126). Then, one or more frequency response analyses can be performed. Finally, by superposition of the results in the time domain a periodic response can be achieved.

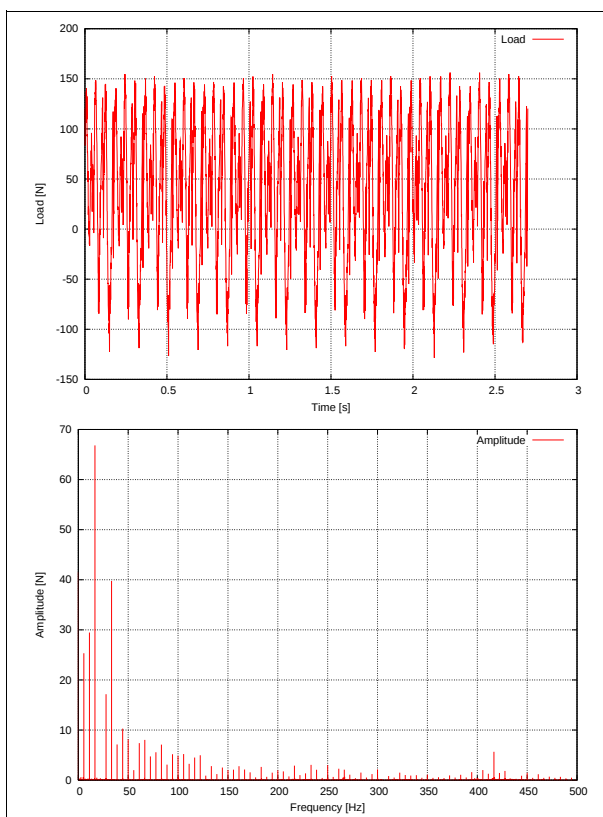


Figure 126: Example for the transformation of a periodic load (above in time domain, below in frequency domain)

Model Verification

With increasing complexity of models the need to verify the correctness of model descriptions increases. PERMAS pays attention to this point by providing model properties in a form that they can be post-processed like results.

It is an important principle of model description that all quantities which can be specified can also be exported for visualization and checking purposes. In particular, this holds for all automatically generated quantities which facilitate model description.

Since VisPER is available, verification of PERMAS models became much easier. It does not only provide visual means to check the model, but it provides also capabilities to visualize the verification data generated by PERMAS (see examples below). For PERMAS models, VisPER is the recommendable verification tool.

The following list gives a few examples of verification aids:

- Element test results are exported for identification of erroneous elements.
- Mean element thickness at nodes can be exported to check the thickness distribution in a shell structure.
- Rigid body modes detected during the analysis can be exported for viewing in order to verify support conditions.
- For MPC interpolation regions (see MPC conditions on page 64) the offset vectors are exported to verify the result of the automatic interpolation.
- For spotweld connections the generated weld base vectors are issued (see Fig. 115).
- VisPER can be used to check distributed loads (even those with a function dependent on geometric position).
- Element temperature loads and distributed nodal temperature loads can be exported as temperature result for visualization.
- Initial strains are available for postprocessing.
- For contact modeling a number of entities are available for verification like contact definition, contact geometry, contact coordinate system, initial contact status, initial gap width (see Fig. 127).
- For the pretension of bolts an additional number of description entities can be postprocessed like pretension definition, pretension coordinate system, pretension thread vectors (flank normal, downhill and pitch direction).
- For heat transfer analyses with heat exchange by radiation the radiating surface can be coarsened automatically. The element mesh of the coarsened surface can be exported for checking.

- In fluid-structure coupled analysis the interface elements and their orientation can be checked.
- In optimization applications the use of design elements can be checked by VisPER (see pages 47 and 49). Besides, the assignment of finite elements to design elements can also be checked.

For a number of input data, PERMAS can generate postscript files using *gnuplot* (if *gnuplot* is available on the currently used machine). It will generate *gnuplot* command files, and a table with the xy-data of the curves will be exported to a file. Such input data include load history (see page 83), material curves for plasticity, and transient loads in timehistory response analysis.

Interfaces

The integration of PERMAS in the pre- and post-processor is of top priority for the interface development. Therefore, all interfaces are directly integrated without any separate software tools. These interfaces are denominated as 'Doors', which allow for a very direct access to the original model data. Above the model description, some interfaces allow for standard solutions, which make the working environment more comfortable.

Beside its own input and output formats PERMAS offers Doors not only to various pre- and post-processors but also to model files of other FEA systems:

- VisPER (page 41)
- MEDINA (page 127)
- PATRAN (page 127)
- I-DEAS (page 127)
- ADAMS (page 128)
- SIMPACK (page 129)
- EXCITE (page 128)
- MOTIONSOLVE (page 129)
- HYPERVIEW (page 129)
- VAO (page 129)
- Virtual.Lab (page 129)
- ADSTEFAN (page 129)
- MATLAB (page 130)
- NASTRAN (page 130)
- ABAQUS (page 131)

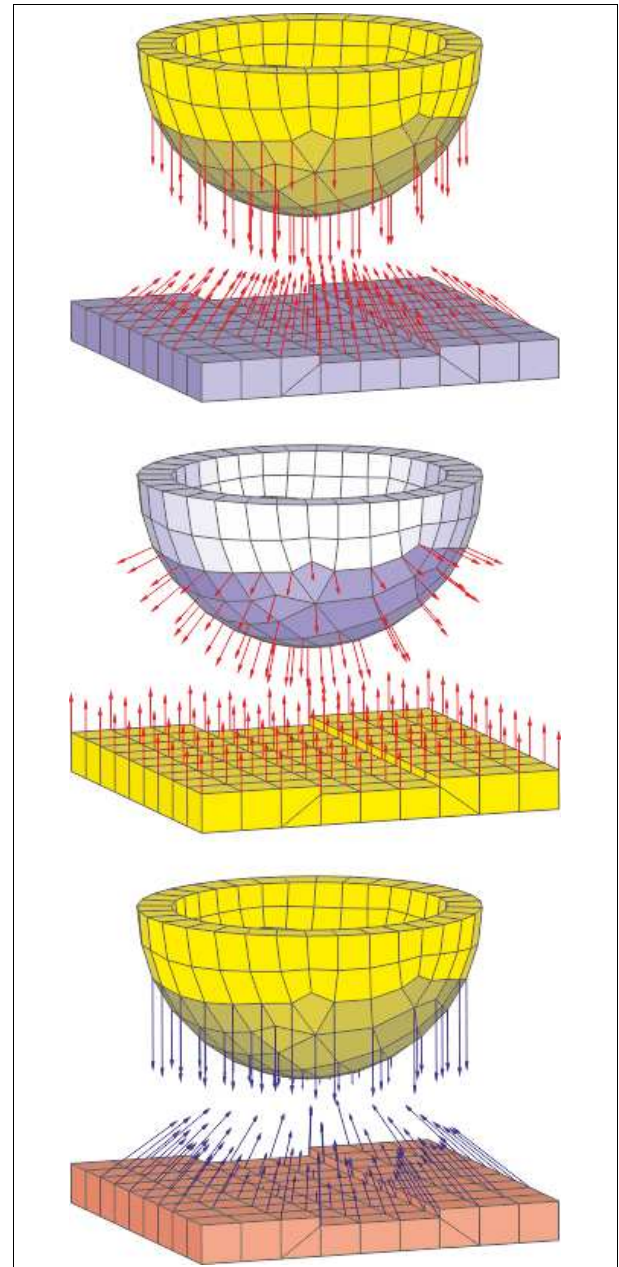


Figure 127: Verification of a contact model

Top: contact normal vector
Middle: surface normal vector
Bottom: nadir vector

- MpCCI (page 131)

Moreover, a growing number of extra interfaces to PERMAS are available from partner companies or INTES (see page 132).

All users with a heterogeneous environment of pre- and post-processors benefit from the excellent interfaces to other CAE products.

During **input** complete FE-models are read and

translated into autonomous PERMAS data structures. If requested, any part may be combined with or completed by additional PERMAS data files.

During **output** the calculated results may be written in various forms (listing, xy-plot files, several post-processor formats). In addition, the complete FE model may be output as PERMAS model file or as post-processing model in diverse pre- and post-processor formats – independent of the kind of input.

For the direct PERMAS model and results export, an ASCII and a binary format is available. The exported model and results can be read by VisPER for post-processing, but also PERMAS is capable to read at model and at least primary results in these formats.

The ASCII format is written in compressed format (using *gzip*) by default. The binary results export format is based on the widely used *HDF5* library (see <https://www.hdfgroup.org/HDF5/>). The main effect of this binary format is the reduction of time needed to export the results. In addition, a reduction of file size is achieved compared to the *gzip*-ed ASCII export file format.

The result evaluation even for very large models is supported by comprehensive selection facilities for the exported result data. Beside the selection of relevant loading cases, time steps, and frequencies the specification of node and element sets is used to reduce the amount of exported results.

This high-level Door concept endows a number of **advantages**:

- High comfort:
 - no external interface necessary,
 - similar command syntax for all Doors,
 - no intermediate files necessary.
- The translation process is very fast.
- Simple **input mixing**:
 - Several input models may be concatenated to one big model, no matter from which Door the model files were read.
 - The model description may be spread on several files, utilizing the possibilities of all Door input formats involved.
- With the direct translation, it is even possible to support features not quite compatible with any DAT file input.
- Quality assurance for modeling is improved:

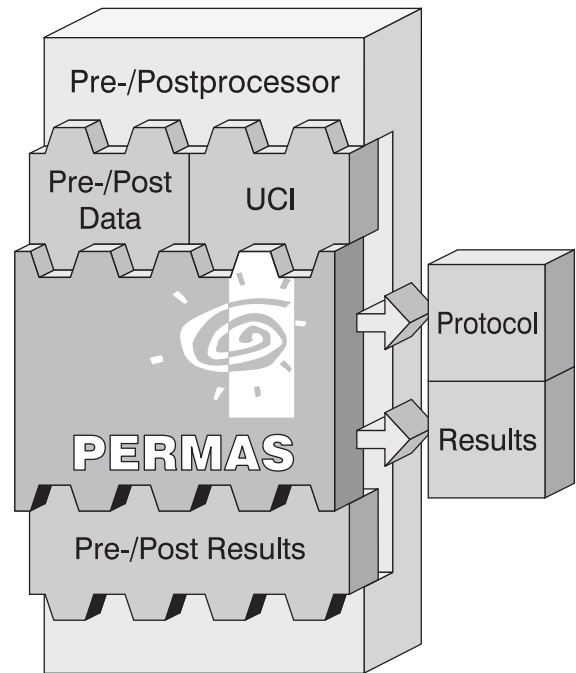


Figure 128: Integration of PERMAS in pre- and post-processor

- Common consolidation of all input data (extensive model testing).
- External identifiers are preserved and are used throughout the PERMAS run.
- Additional data may be given by a separate input file – the original input file remains untouched.
- The mixing feature enables the user to define difficult parts with the most convenient input format. Naturally, this will be the least error susceptible definition, too.
- Because the export format is independent of the input format, PERMAS allows for the translation of any pre-processor format to any post-processor format.

Matrix Models

PERMAS data objects for results or matrices can be output to or input from external files in different ways and in binary or ASCII format.

This tool can be used to organize the data exchange with third party software, where no direct interface is available. In addition, it can also be used to store the data for later use in another PERMAS run to save

computer time.

A typical application is the intermediate storage of condensed models. There beside the condensed matrices also a condensed model can be generated and exported in order to use them in another run. For a detailed post-processing the results can then be transformed back to this model. To reduce size of matrices and results this transformation can be restricted to sets in advance.

Condensation of substructures is used beside static analyses mainly for vibration analysis and response analysis in time and frequency domain. There, beside stiffness and mass matrices eigenvalues, mode shapes, loads, and damping matrices are provided. Additional static mode shapes can also be taken into account.

Another application is the generation of modal models (e.g. by *generalized modal condensation*, see page 98). These modal models are intended for modal response analyses outside of PERMAS.

All standard ASCII output files can be generated as compressed files (gzip) saving disk space and time (for large files). Because the input is possible in compressed format, too, there is no explicit need to store model and result data in full ASCII format.

Special formats available for matrix export are MATLAB format and *Rutherford-Boeing* format.

Combination of Results

After the computation, different primary and secondary results may be combined to new results. For this purpose different summation rules and mathematical functions (see page 69) are applicable. In this way, even results from different variants may be combined.

The generated results may overwrite existing results or generate new ones. The output of the combined results is exactly like the output of the original results.

In VisPER a great variety of arithmetic operations on result combinations is possible (e.g. difference of kinetic and strain energy density). Each single column may be used as operand. All PERMAS functions

with matching number of input parameters can also be used (e.g. logarithmic function to post-process acoustic levels).

Transformation of Results

All nodal results are calculated in the global coordinate system of the actual Component. From these results the following transformations may be performed:

- into the displacement coordinate system of the respective node.
- into a specific coordinate system for all nodes (cartesian, cylindric, spheric).
- using a special transformation for each node.

All transformations may be performed later in backward direction to the Component system.

Beside the transformation of real results, complex results of a frequency response analysis can be transformed to another coordinate system, too.

Comparison of Results

The following facilities are available for the comparison of dynamic analysis results between two variants:

- **MAC** (Modal Assurance Criterion):
Compares two sets of eigenvectors (with same number of nodes) from different situations and gets their degree of correspondence. MAC factors are computed as:

$$MAC = \frac{(\mathbf{X}_1^t \mathbf{X}_2)^2}{diag(\mathbf{X}_1^t \mathbf{X}_1) diag(\mathbf{X}_2^t \mathbf{X}_2)}$$

- **CoMAC** (Coordinate Modal Assurance Criterion):
Compares two sets of eigenvectors (with same number of nodes) from different situations and gets their degree of correspondence. CoMAC factors are computed as:

$$COMAC = \frac{diag(\sum_{m=1}^M |\mathbf{X}_1^m \mathbf{X}_2^m|)^2}{diag(\sum_{m=1}^M \mathbf{X}_1^m)^2 \cdot diag(\sum_{m=1}^M \mathbf{X}_2^m)^2}$$

The result gives a measure of the correlation of the vector sets in each degree of freedom direction.

- **COF** (Cross Orthogonality Factors):
Compares two sets of eigenvectors (with same number of degrees of freedom) from different situations and gets their degree of correspondence. Cross orthogonality matrix and factors are computed as :

$$COFM = \mathbf{X}_1^t \mathbf{K} \mathbf{X}_2; \quad COF = COFM^t COFM.$$

All comparisons may be performed for two variants of a model and the compared model parts can be restricted to specified node sets.

One possible application of MAC factors is the comparison of results from *Experimental Modal Analysis* and from FE analysis (see page 38).

XY Result Data

On the basis of element and node sets, xy data can be generated directly and issued on all connected output files for graphical processing.

Usually, xy data are directly extracted from the corresponding result item such, that the abscissa value is given by the column (i.e. loading case, iteration) and the ordinate values are the result data. If the columns are associated with certain values (like frequencies, time, load factors), these values will be used for the abscissa.

As a special case in order to study e.g. the stiffness behavior of a part, it is possible to create xy data for nodal point results with the coordinate direction as abscissa. Then, the abscissa values are prepared in ascending coordinate direction for the selected nodes. A local coordinate system may be used to select the coordinate directions.

For a number of input data, PERMAS can generate postscript files using *gnuplot* (if gnuplot is available on the currently used machine). It will generate gnuplot command files, and a table with the xy-data of the curves will be exported to a file. Such input data include load history (see page 83), material curves for plasticity, and transient loads in timehistory response analysis.

Within *PERMAS Tools* plots can be created using *PERMASgraph* (see page 139 for details).

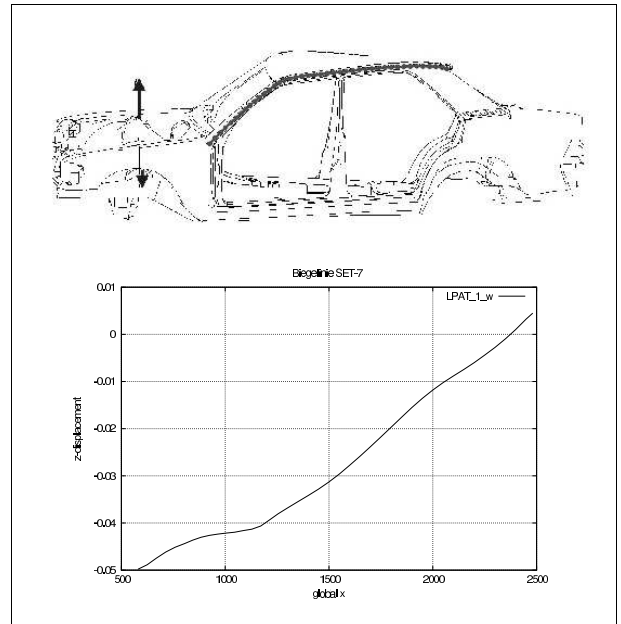


Figure 129: Determination of elastic line for torsional load case (displacement for the marked nodes)

Cutting Forces

On the basis of element and node sets the cutting forces can be determined and exported for post-processing for almost all points of a structure. In addition, the sum of all forces and moments over a cut is calculated and printed for a previously specified point in space (see Fig. 130).

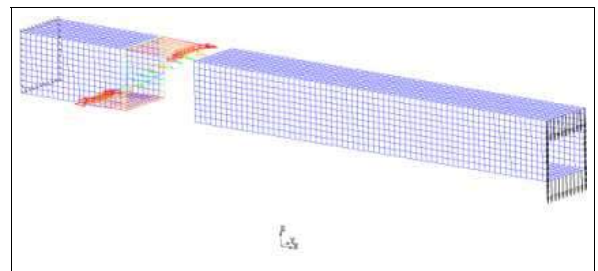


Figure 130: Determination of cutting forces

System of Units

A consistent choice for the system of units for all input data is under user's responsibility.

For the processing of the input data, tolerances of linear measures (like coordinate differences, gap widths) are very important. The standard is the use

of millimeter based input. All these length tolerances can be changed to e.g. meter dimensions by specifying the relevant conversion factor (e.g. 1.E-3) to keep the absolute tolerance values.

Restarts

Each PERMAS run opens a data base file, which may be used for subsequent runs. At every restart, the latest status of the data base is always available from the previous run. This includes all intermediate results possibly obtained only by using considerable computation time.

In doing so, e.g. different load variants in several runs may be processed without assembling and decomposing the stiffness matrix every time.

Open Software System

PERMAS is an open software system with respect to its capability to include user-written routines in every program execution, which may be invoked during run-time.

On the one hand, this feature is used to offer maximum flexibility in defining data dependencies, e.g. with user-defined mathematical functions (see page 69).

On the other hand, PERMAS may be used as subroutine library in order to write own solutions or result evaluations. These can be invoked at the right place during program execution.

The programming language is Fortran 77/90 in any case, but on some platforms C is also available. The PERMAS library routines are available as Fortran programs.

To become independent of a compiler, scripts based on *python* can also be used to perform certain operations during a PERMAS analysis.

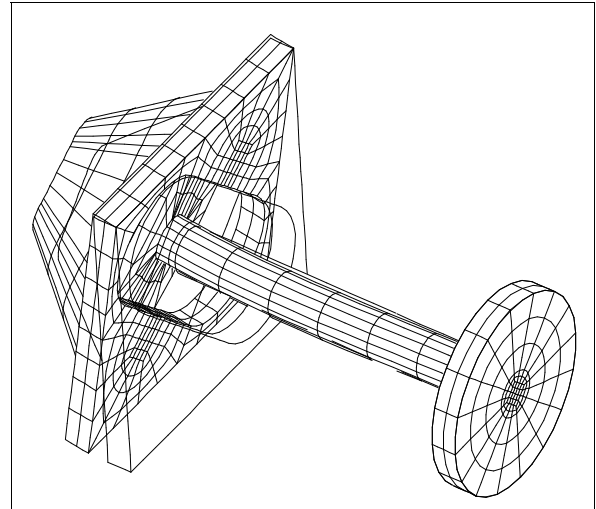


Figure 131: Clutch Element, Static Displacements

Direct Coupled Analyses

PERMAS enables different coupled analyses in one run, e.g.

- initial values for time-dependent dynamic analysis may be taken from a previously performed static analysis.
- initial values for a transient thermal analysis may be taken from another transient thermal analysis.
- a thermal stress analysis may be performed on the basis of a previous thermal analysis.
- a thermal or mechanical analysis on the basis of a previous electromagnetic analysis.
- a (fully) coupled fluid structure acoustics analysis.
- all optimizations, if different analysis types are used simultaneously (like static and eigenvalue analysis).

For all kinds of coupled analysis the same types of elements may be used for the different partial analyses. As far as appropriate, most element types are available for all different analysis types.

Coupling with CFD

A coupling of structural mechanics and computational fluid dynamics has been realized within the scope of the CIPAR ESPRIT project by a loose coupling approach.

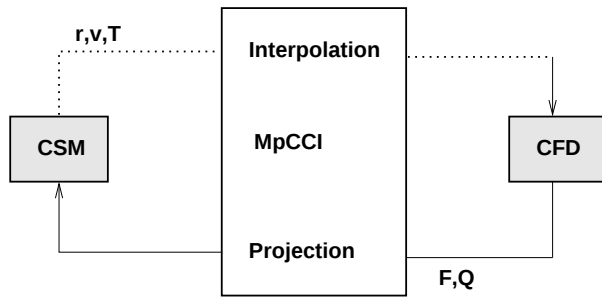


Figure 132: Coupling of CFD and CSM

Within the project a general COupled COmmuni-
cation LIBrary (COCOLIB) has been developed by
FhG/SCAI, which allows for a weak coupling of a
structural mechanics software (CSD) and a fluid dy-
namics software (CFD). Both packages are running
simultaneously and the data exchange is during run
time following previously defined criteria and cou-
pling algorithms (Gauss-Seidel, Jacobi). This soft-
ware has been further developed and is now called
MpCCI (more information at [www. mpcci.org](http://www.mpcci.org)).

The transferred boundary conditions comprise ther-
mal and mechanical quantities (force **F** or pressure
resp., heat flux **Q**, displacement **r**, velocity **v**, tem-
perature **T**). There the coupling library does any in-
terpolation and projection necessary due to incom-
patible meshes between structure and fluid.

The calls to *MpCCI* have been integrated into
PERMAS. So, PERMAS and any CFD code where
MpCCI is adopted, too, can be used to solve
applications in mechanical, thermal, or thermo-
mechanical coupling. Module PERMAS-CCL pro-
vides all necessary functions from the PERMAS
side to support the coupling.

PERMAS Analysis Modules

PERMAS-MQA – Model Quality Assurance

The PERMAS-MQA basic module builds the system kernel for all other software modules.

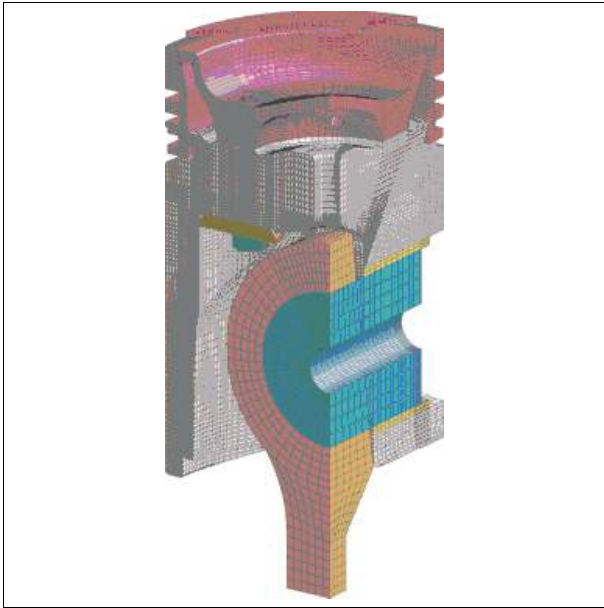


Figure 133: Ship engine piston
Mahle GmbH, Stuttgart, Germany.

Among others this *system kernel* contains the data management system, the UCI command language, the standard input of PERMAS models, the standard output of results, the model consolidation, the *substructure technique*, numerous element types, and *sampling* for parameter studies.

The characteristic features of PERMAS-MQA are the concepts and tools for the **quality assurance** of the analysis process and for the investigation of different model parameters by **sampling**.

The **quality assurance** of finite element models becomes more and more important.

- Today, Finite Element calculations are used in the product development to early assess different design variants and to accelerate the development process.
- The skill of FEA systems users changes from an expert level to a more general background.
- The FE applications become more and more complex.

Design exploration by **Sampling** is the repeated execution of analyses where the values of design or basic variables are specified explicitly. Sampling works analogous to optimization and reliability analysis. An example is shown in Fig. 48. So, all possible design variables (see page 109) or basic variables (see page 119) can be used for sampling and all analysis steps supported by optimization (see page 109) and reliability analysis (see page 119) can be used for sampling as well. For all desired results, sampling collects this data and prepares special XY data for convenient post-processing. Some examples for suitable variables are geometric parameters, Young's modulus for stiffness considerations, or different load parameters. In order to reduce the number of samples without losing information about parameter influence a special sampling method (i.e. *LHC Latin Hypercube Sampling*) is available.

Beside the software quality (see page 19) the reliability of FEA results depends on the following points:

- **Model testing by VisPER:**
PERMAS-MQA is also the basic module of VisPER. So, all model tests available in this module are also part of VisPER. But VisPER adds additional visualization functionality to these tests, which facilitates the understanding and the repair of detected problems.
- **Comprehensive model testing:**
PERMAS performs very intensive tests of the input data. There are several thousands of different plain text system messages to react to complex inconsistencies in an appropriate way. In particular, the automatic detection of singularities can save much time for the user (see page 65).
- **Avoidance of erroneous analysis runs:**
In PERMAS a **task scanner** has been introduced in order to avoid faulty runs:
 - The analysis steps are checked in a group-wise manner to verify the feasibility of the complete analysis.
 - The resources in terms of CPU time and disk space are estimated in advance.
 - The input model data are checked for completeness and compatibility with respect to the analysis steps requested.
 - Even the control of those PERMAS modules can be checked which are not licensed on the active platform.

- The model tests can be used in addition to similar tests of the applied pre-processor.

- **Relief of the user:**

Cumbersome routine work can be reduced and the overview of all model related information can be improved by comfortable input facilities, direct interfaces, and all of the test tools listed above. So, the user can concentrate on the objectives of the analysis and the evaluation of the results.

In particular, comfortable interfaces allow for a smooth transfer of model data from the pre-processor (see page 72).

A comment feature supports improved communication with *SDM (Simulation Data Management)* systems. These comments provide a means to describe any entity in the model description in any desired level of detail. To this end, the comments can be included in the model input file or the comments are linked to an additional file which also can include *XML* documents.

PERMAS Thermo-Mechanics

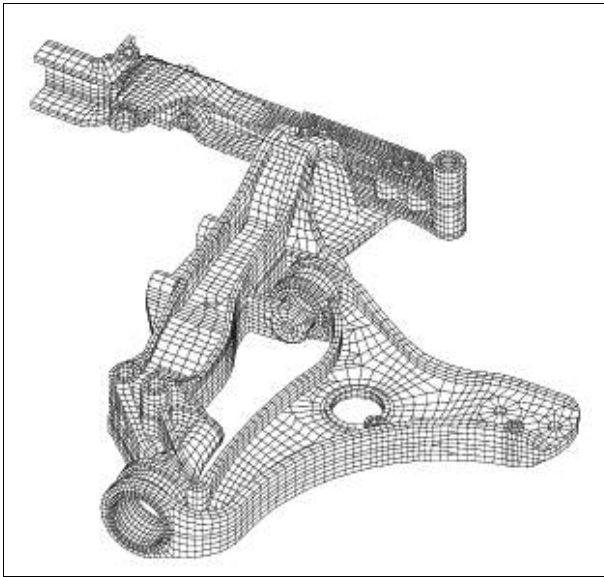


Figure 134: Model of a front axle
Porsche AG, Weissach

- Residual forces,
- Strain energy density, i.e. mass- or volume-specific strain energy.

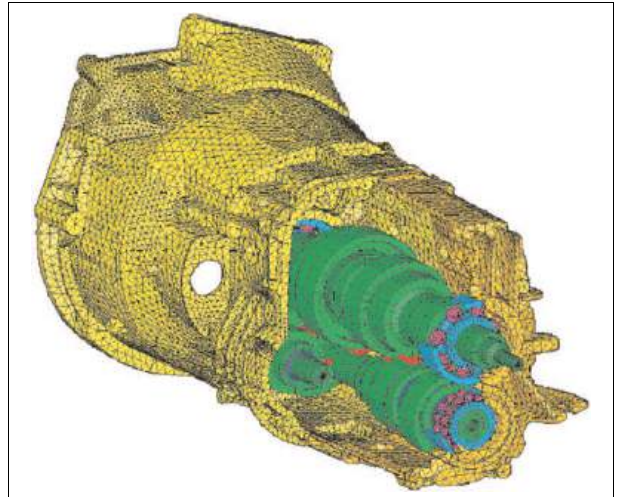


Figure 135: Bearing housing ZF AG

PERMAS-LS – Linear Statics

This module allows for linear elastic calculations, based upon the assumptions of small displacements, small strains, and linear material behavior (isotropic or anisotropic). Therefore, this module often builds the first step in Finite Element analysis.

- The following definitions of kinematic boundary conditions are available:
 - Suppressed degrees of freedom,
 - Prescribed degrees of freedom,
 - Linear constraints (*MPCs*) (see page 64).
- For free or partially free structures a quasi-static analysis can be performed (*Inertia Relief*). There, on the basis of a rigid body decoupling the inertia forces are computed which are in equilibrium with the applied forces. Subsequently, a static analysis is performed under the applied loads and these inertia forces.
- Different kinds of static loading are available (see page 70).
- The following primary results are calculated:
 - Displacements,
 - Mass and moments of inertia.
- From that additional results are derived:
 - Reaction forces,
 - Stresses and stress resultants,
 - Strains,

PERMAS-CA – Contact Analysis

Static analyses with non-linear boundary conditions (contact problems) can be analyzed using the PERMAS-CA module.

Contact boundary conditions may be present between elastic bodies or between elastic bodies and a rigid counterpart. The bodies may behave also non-linearly.

\$CONTACT AUTO[SEARCH] is intended to boost the problem setup for a contact analysis. It detects possible contact candidates by distance criteria and defines corresponding contact definitions, see Fig. 136

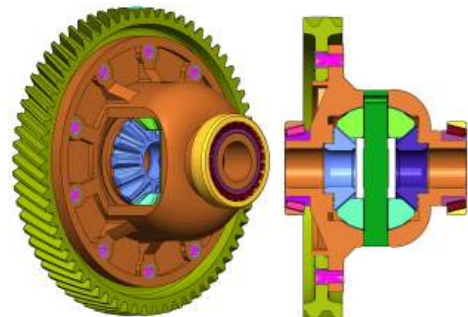


Figure 136: Differential model with only one **\$CONTACT AUTO[SEARCH]** definition

An automatic internal surface split at large kinks is a basic feature that is used by autosearch, but can be used in general for all contact definitions, see Fig. 137.

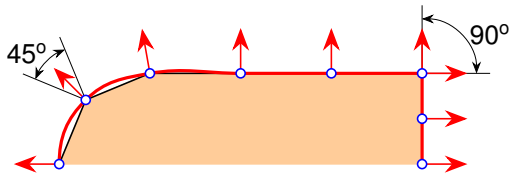


Figure 137: Automatic surface split at large kinks

In addition *self contact* is supported by contact analysis (see Fig. 148), i.e. contact models where a surface may get in touch with itself, e.g. due to warping or other forms of large deformations, and where contacting regions are not known in advance.

Several methods to describe contacts are available:

- specification of contact node pairs,
- specification of nodesets for each contact zone (the node pairs are detected automatically),
- assignment of nodes/nodesets to surfaces (incompatible meshes),
- definition of general surface-to-surface contact (incompatible meshes).

The feature to define contact with incompatible meshes allows the independent meshing of the contacting bodies. This simplifies the modeling of complex contact surfaces (like tooth contact between gearwheels) essentially.

In surface-to-surface contacts, more contacts are taken into account than in surface-to-node contacts. More contacts typically give more accurate results but also longer computation times. Less contacts does not mean that the results become inaccurate, but it is possible e.g. for coarse meshes of contact surfaces. In order to get the best out of a contact analysis by spending only moderately more computation time, an optional automatism has been developed (*option COMPLEMENT*), which reduces the number of contacts in surface-to-surface contact areas in a way that the accuracy is preserved (see Fig. 138).

The direction of contact and the initial gap width may be specified explicitly or determined automatically from the geometry. Any *press fit* is easily modeled by the specification of a negative gap width. A given

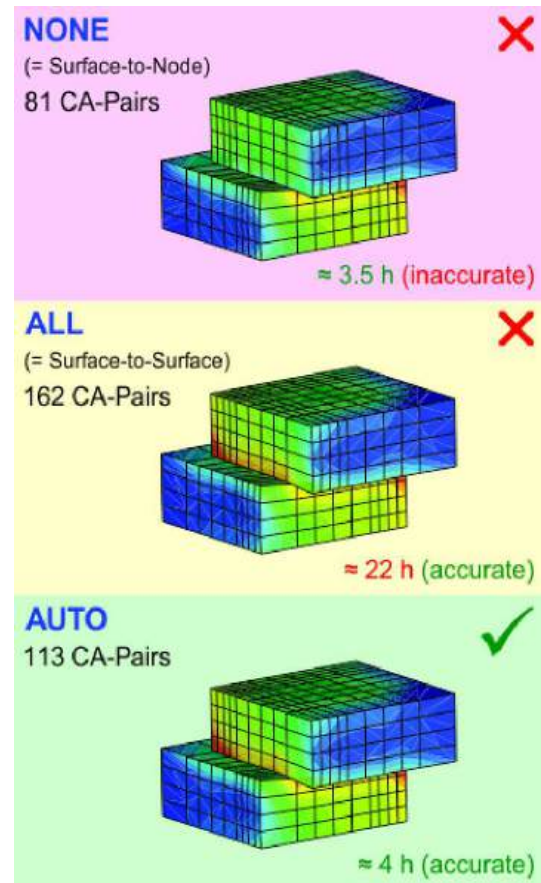


Figure 138: Surface-to-surface contact with COMPLEMENT option: Best compromise between accuracy and computation time (von Mises nodal stresses are shown). The numbers are taken from an industrial application.

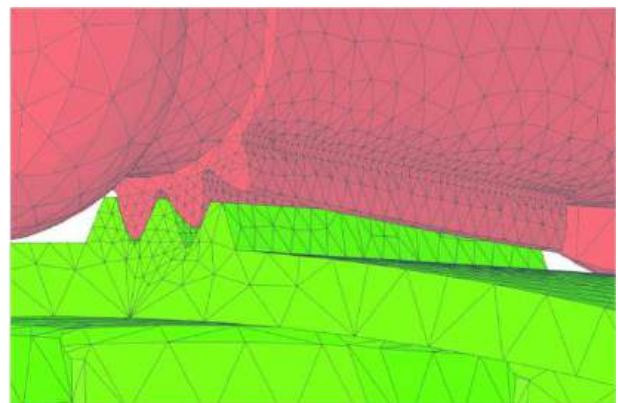


Figure 139: Contact with incompatible meshes

gap may be scaled by a user-defined function, which may depend on the position in space and topological information, i.e. a *gap function*.

Modeling of *press fit* can be made with MPC conditions, alternatively, if the press fit will not open a gap

during loading (see page 65).

The contact analysis can handle isotropic or anisotropic frictional contact with sticking or slipping according to *Coulomb's law*.

For large contact problems and for complex frictional tasks module PERMAS-CAMG provides a suitable extension of contact analysis (see page 86 ff.). Both modules together cover a wide application area for contact analysis. This combination is complemented by module PERMAS-CAU (see page 87 ff.) which takes into account large relative displacements of contacting bodies.

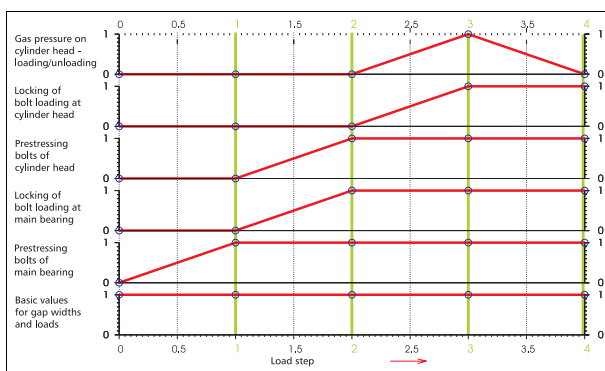


Figure 140: Representation of a load history

The specification of a load history allows the correct simulation of assembly and working loads and any contact situation with slipping and sticking friction. This facilitates the convenient simulation of such situations in a quasi-static analysis. A postscript plot file of the load history can be exported to view its graphical representation (see Fig. 140).

The load history can be amended by pretensioning (e.g. of bolts), where the contact analysis is used to describe the pretension. In this way, the screw tightening torque is modeled by a known contact force in the barrel of the bolt.

A generalized concept for **bolt pretension** is provided. Beside the classical approach using a cutting plane with pretension in normal direction, a new approach using cylindrical thread coupling with pretension in axial direction is available. This highly innovative feature offers a convenient definition which can take into account the detailed effects of radial spreading and axial torque caused by the thread's flank and pitch geometry without the need of modeling the flank shape or thread line explicitly (see Fig. 141). In addition, Fig. 142 shows an application on

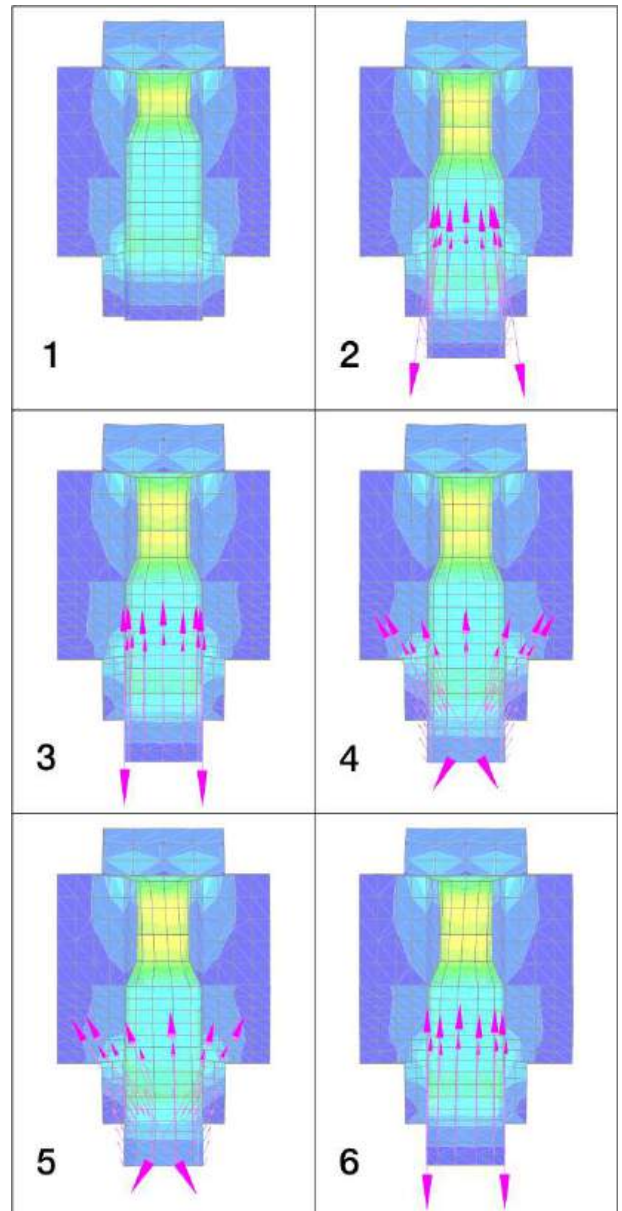


Figure 141: Bolt pretension
Nodal point stresses and pretension forces
for deformed mid-size screws:

- (1) with cutting plane in bolt
- (2) with radially joined thread
- (3) thread without radial coupling
- (4) thread with flank but no pitch angle
- (5) thread with flank and pitch (standard M10)
- (6) left-handed thread with pitch but no flank angle

tightening and unscrewing of a bolt.

Contacting bodies are often only supported by contact forces. To provide an elastic foundation to such bodies, a family of *compensation springs* (also called *zero force springs*) can be used, where the spring force is compensated by an additional con-

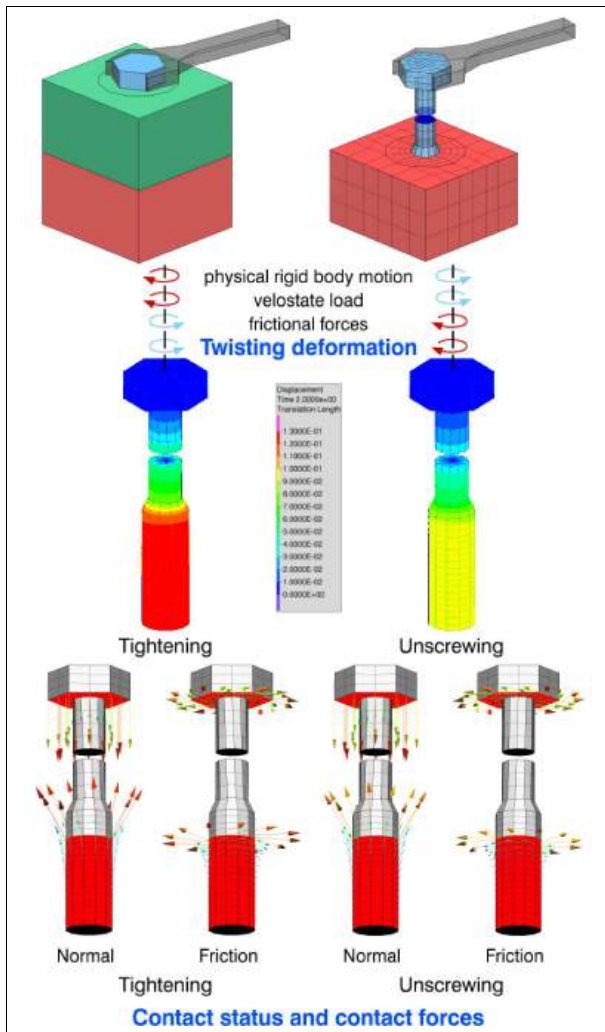


Figure 142: Thread definition on bolt surface.

tact force (see Fig. 143). By this means, a *force-guided contact* is facilitated. In addition, the *RBM Assistant* in VisPER helps to place and properly define such elements (see page 45).

Comprehensive checks allow the verification of contact models like type of contact, its geometry (gap-width and normal vector, see also Fig 127), and the contact coordinate system (for normal and frictional force directions). In addition, the contact status is available in all iteration steps for checking purposes.

For frictional contact the quality of surfaces is of utmost importance. Therefore, PERMAS can smooth contact surfaces in order to essentially improve frictional behavior.

The analysis procedure uses a reduced flexibility model which is derived from the set of contact de-

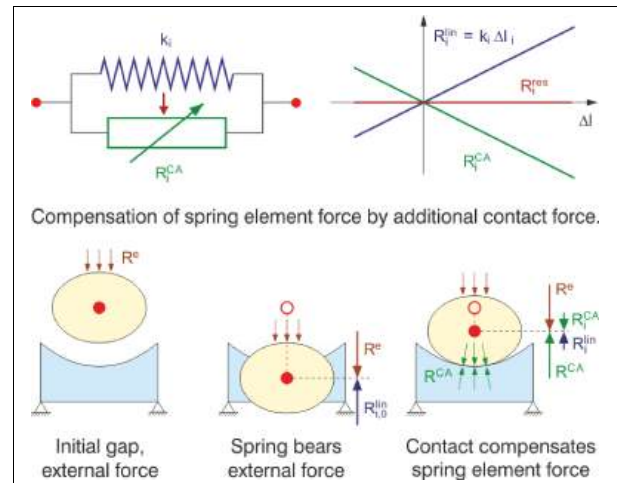


Figure 143: Force-guided contact with the aid of compensation spring

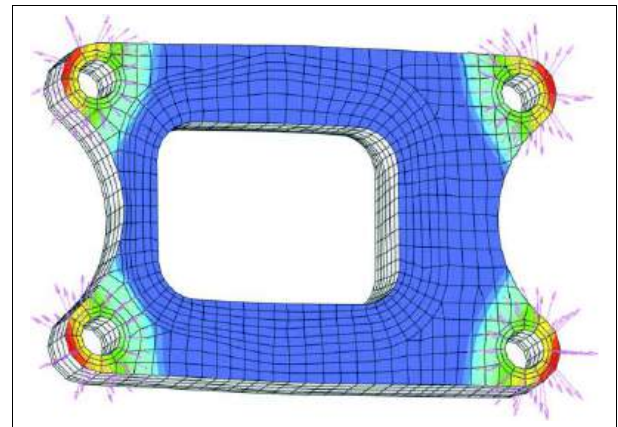


Figure 144: Contact pressure and shear vectors

grees of freedom. This procedure has the following advantages:

- The iteration is very efficient making it best suited for extremely large models with an arbitrary number of contact nodes.
- The accuracy of the results is fully preserved, because no additional stiffnesses are introduced by the modeling of contacts.

The simultaneous analysis of an **arbitrary number of loading cases** is possible. The contact parameters, i.e. gap width and coefficients of friction, may be different for each loading case. The contact boundary conditions are taken into account automatically by the static analysis procedure. No additional user request is required for a contact analysis.

For efficient calculation of successive contact variants *contact status files* are available for easy job recovery and considerable run time reductions.

In addition to all results usually derived from a static analysis the contact analysis provides for the contact status, the contact forces, the contact pressure (see Fig. 144), the gap widths, and the relative gap displacements.

For contact surfaces derived from TET10 elements, an option is available to interpolate the midside nodes, which accelerates contact analysis. To improve the stress result quality, an additional result interpolation for the midside nodes is provided. The achieved improvement is shown in Fig. 145, where the stress result is almost the same with and without midside nodes.

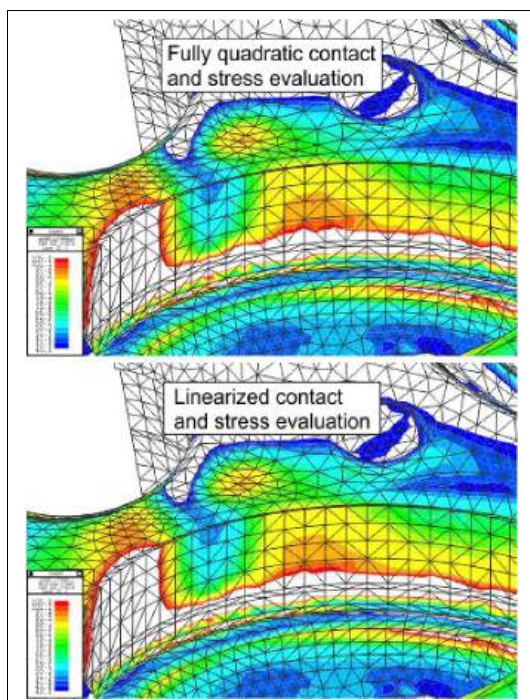


Figure 145: Nodal point stresses on surface with linearization (bottom), and without linearization as reference (top).

For each contact definition with a non-zero sum of contact forces, there exists a *line of action* with a minimal resultant torque due to the corresponding normal and frictional contact forces. The point on that line of action, which lies closest to a given centroid (e.g. the center of contact node coordinates) is defined as the contact's *center of pressure (COP)*. The coordinates of the COP and the sum of forces and moments is written to the result file by request. This result is available for all load steps performed in a linear or nonlinear contact analysis.

For subsequent analyses, contact states can be

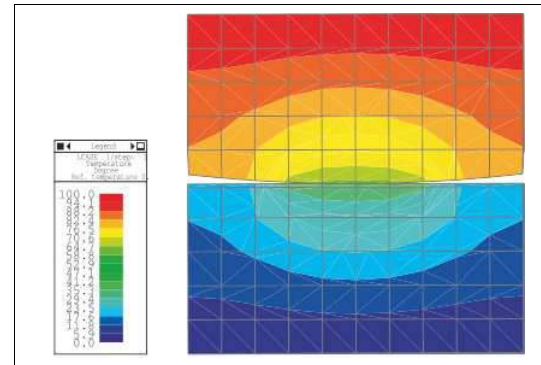


Figure 146: Temperature distribution in two contacting bodies

locked. This *contact locking* leads to linear constraints according to the current contact state. To achieve this, the active contacts are automatically transformed into kinematic constraints. With this new model various kinds of subsequent analyses are possible like eigenvalue analysis, heat transfer analysis (see Fig. 146), or *submodeling*.

PERMAS-CAX – Extended Contact Analysis

The module provides basic contact solution algorithms for ambitious *slip-stick* problems, see Fig. 147 and for larger contact models. It is used as a standard add-on to module PERMAS-CA (see preceding section).

Gasket elements are handled as integral part of the contact iteration through contact controlled non-linear gasket analysis (*CCNG analysis*) by default. This leads to a remarkable reduction of run time compared to the classical solution method with non-linear material behaviour. In particular, this run time reduction will be much higher, when the nonlinear features are contact and gasket elements only. The run time reduction is still significant, if additional nonlinearities are applied.

For sliding friction between dynamically moving parts a velocity field can be prescribed to take it into account in a quasi-static contact analysis (see section Brake Squeal Analysis on page 29).

PERMAS-CAMG – Contact Multigrid Analysis

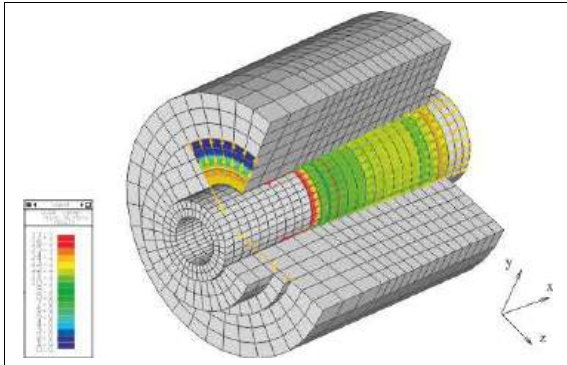


Figure 147: Conical *press fit*
(slipping forces after assembly)

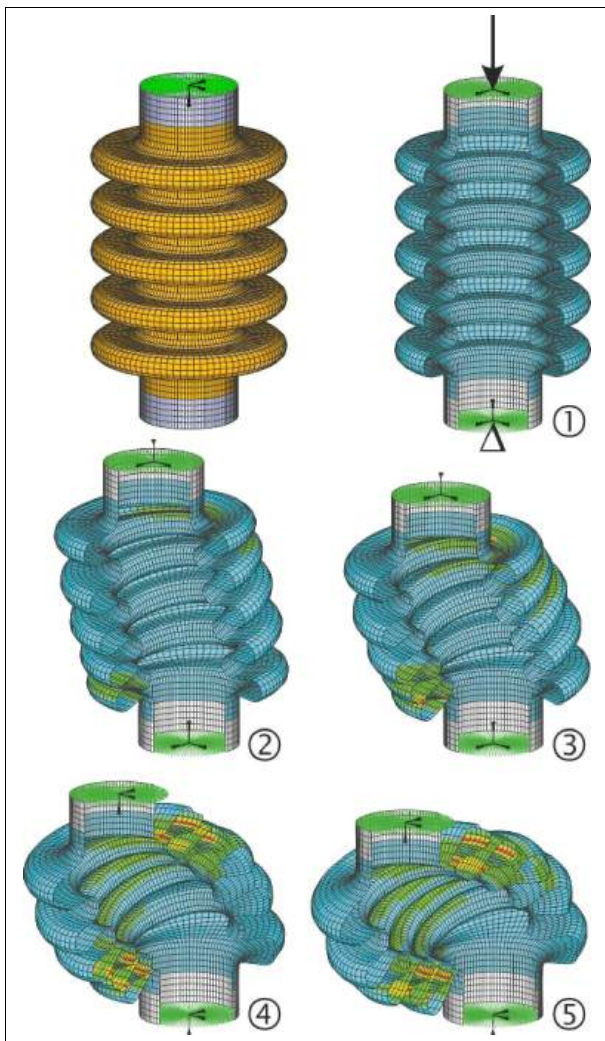


Figure 148: Self contact of a bellow

This represents the third generation of contact solvers (see fig. 149) and allows great runtime improvements for larger contact models by using an innovative **Contact-Multigrid-Solver**. This solver consists of two orthogonal approaches. Both ap-

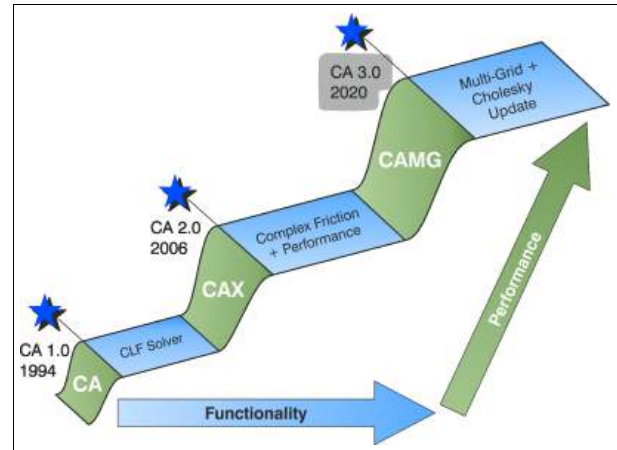


Figure 149: Evolution of contact solvers

proaches significantly reduce the order of floating point operations and thus the runtime benefit increases with the size of the contact problem.

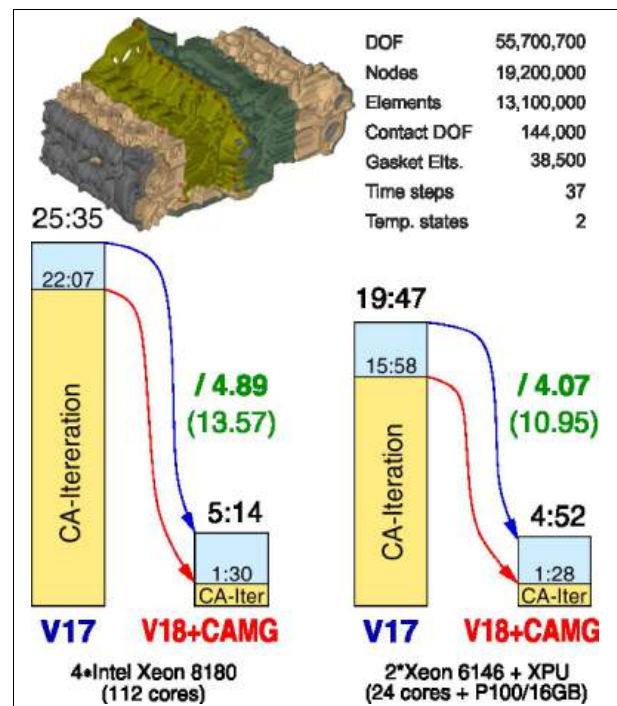


Figure 150: Contact analysis of an engine with 56 Million DOF, 37 time steps, 2 different temperature states

- Cholesky update:**
 First an innovative solver strategy based on an update of the Cholesky factor of the flexibility matrix instead of a decomposition in each iteration was implemented. The Cholesky factor for the current active set of contact degree of freedoms is built by an efficient update using the Cholesky factor of the previ-

ous active set. Thus a cost-intensive complete (re-)decomposition in each iteration step is no longer necessary. Since only the solving step of the linear system in each iteration for the current active set is accelerated, the contact algorithm is not changed itself (mathematical identical iteration, same iteration, same result, only change in numerics) but only the strategy to solve the linear system for current active flexibility is highly accelerated. This new approach is highly adapted for efficient parallelization and is automatically detected if the **CAMG license** is available.

- **Multigrid coarsening:**

This approach is based on a hierarchical coarsening of the contact degrees of freedom. In a first step only a part of these contact degree of freedoms are considered to solve a reduced contact problem, which solution can be used (after a suitable mapping) as initial (caso-file like) solution for the contact iteration on the original (fine) level. The contact problem on the reduced coarse level is also solved efficiently by using the Cholesky update approach. In contrast to the Cholesky update this approach changes the contact algorithm. It may happen, that more (but cheaper) iterations are necessary.

Figure 151 shows the computation time and the speedup (total and only contact iteration) for a large contact model with friction with 119 000 contact DOFs (total 226 900 00 DOFs), 9 000 000 nodes and 5 800 000 elements. the calculation was done on a machine with 48 cores (2*Intel Xeon 8260L). For this model we get a speedup of 7.43 (8.37 for the contact iteration only) for the whole computation time with the new **CAMG** license module, using only the CAX license module we get a speed up of 1.58 compared to PERMAS version 17.

With PERMAS V21, especially for complex friction problems, a big step towards stability and performance has been realized, as shown in Fig. 152.

PERMAS-CAU – Contact Geometry Update

This module has been designed to provide new

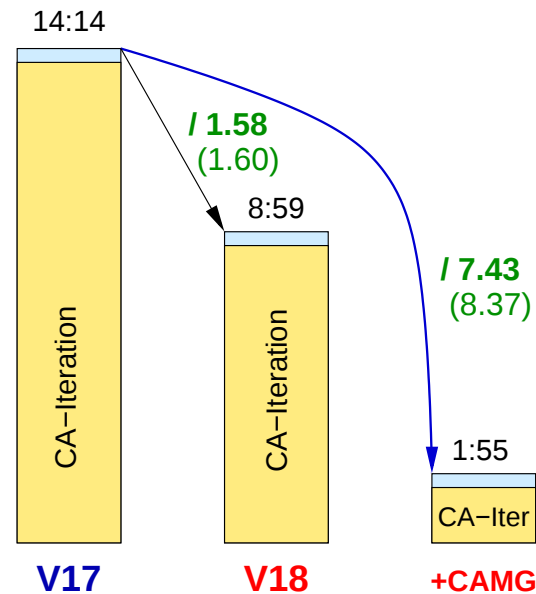


Figure 151: Computation time ([hh:mm]) and speedup for large contact model

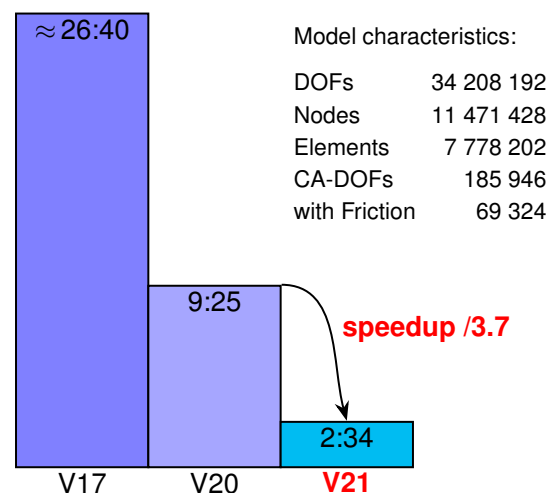


Figure 152: Performance improvement [hh:mm] for CAMG (friction)

contact solution algorithm for significant changes of contact geometry during loading. The contact geometry is characterized by the nadir points (i.e. the projections of the contact points of one body to the contact surface of the other body) and the contact direction (i.e. the normal to the surface at each contact node). The following application cases require the use of contact updates:

- In case of large relative displacements of two bodies in contact, the update of the contact properly reflects the change of the relative position of both bodies (see Fig. 153 and Fig. 154).
- If very accurate contact results are required,

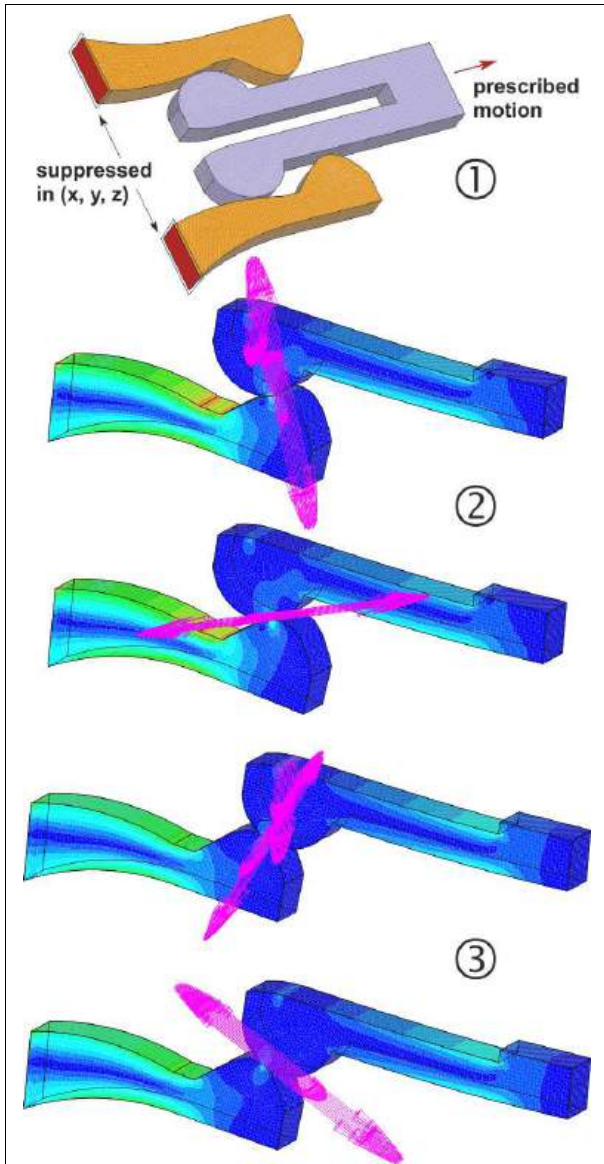


Figure 153: Clip example for contact update
Large relative motion with friction. Picture 1 shows the model; pictures 2 show the contact pressure (above) and the frictional shear (below) with nodal point stresses before the bottleneck is reached; pictures 3 show the contact pressure (above) and the frictional shear (below) with nodal point stresses after the bottleneck is passed.

contact updating even reflects very small changes of the contact geometry which gives exact contact forces, displacements, stresses, etc..

The following features are supported by this module:

- Special modeling is not required. Because the update process changes the position of the contacting surfaces, the contact geometry must be specified by surfaces (e.g. surface-to-surface or

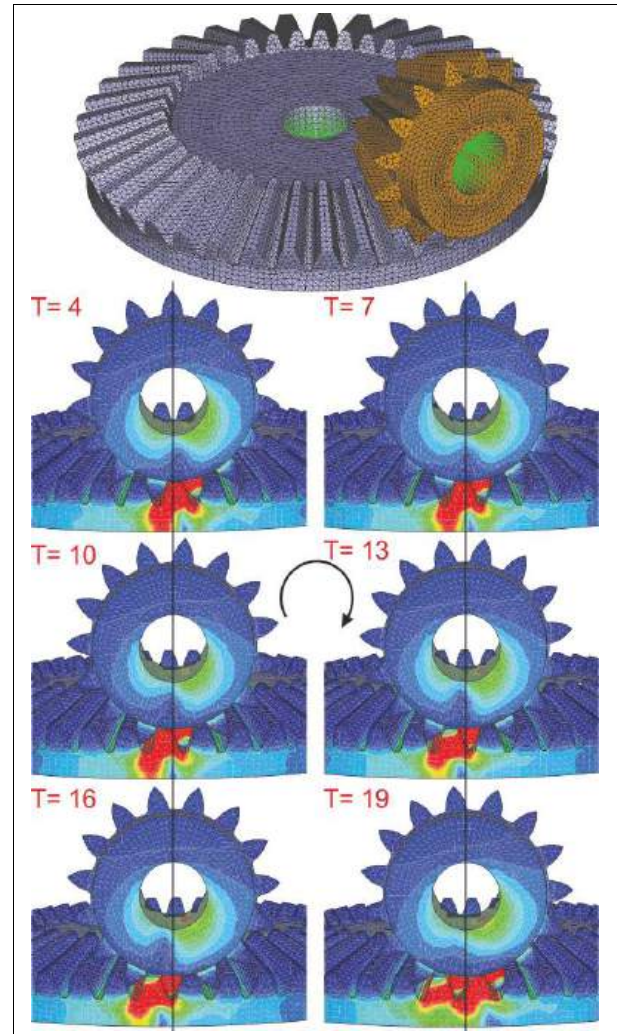


Figure 154: Driven gear set

surface-to-node). Hence, node-to-node contact and contact to ground are not updated.

- Contact updating works with and without friction.
- Contact updating works with linear and nonlinear static analysis.

The geometry update is principally explained in Fig. 155. Steps 1 and 2 show the typical contact procedure and steps 3 and 4 show the first iteration of the updating process which will be repeated until a given number of iterations is performed or until a given accuracy is achieved.

PERMAS-NLS – Nonlinear Statics

Geometrically nonlinear behavior

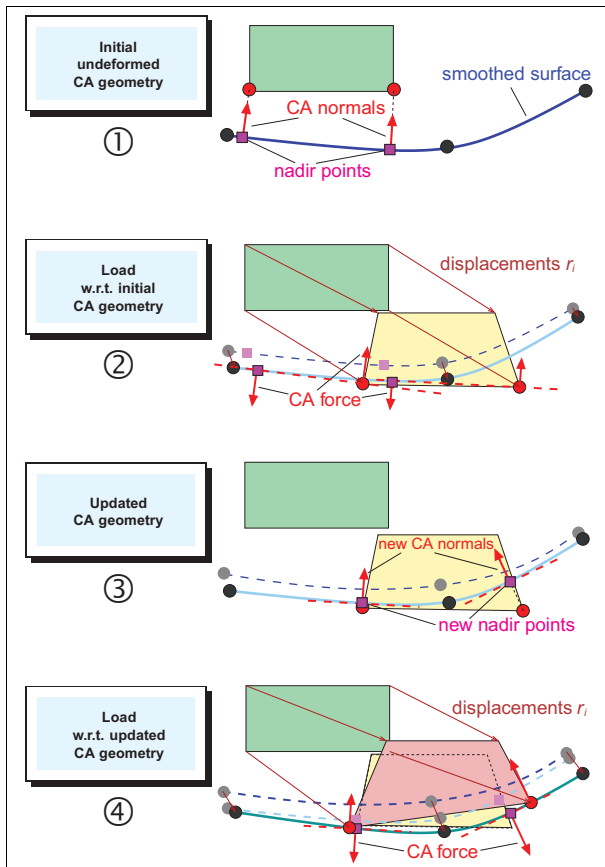


Figure 155: First steps in contact update

This module part enables the geometric nonlinear analysis of models via the *Co-Rotational* method. Large displacements and rotations are taken into account under the assumption of small strains (linear strain tensor). The resulting system of nonlinear equations is solved efficiently by the *Newton-Raphson* method combined with a *Line Search*. The load increments may be controlled manually or automatically. The nonlinear characteristic of the solution path can be shown by load vs. displacement XY plots.

Nonlinear buckling problems can be analyzed with an *arclength* method which enables the determination of the solution path beyond bifurcation and limit points. Again PERMAS offers an increment control, which automatically identifies critical regions and enables an efficient solution. Examples for elastic buckling are shown in Fig. 157 and Fig. 217, while elastic-plastic buckling is shown in Fig. 158. In order to determine critical points and buckling modes a linear buckling analysis may be performed after each converged increment based on the current stiffness.

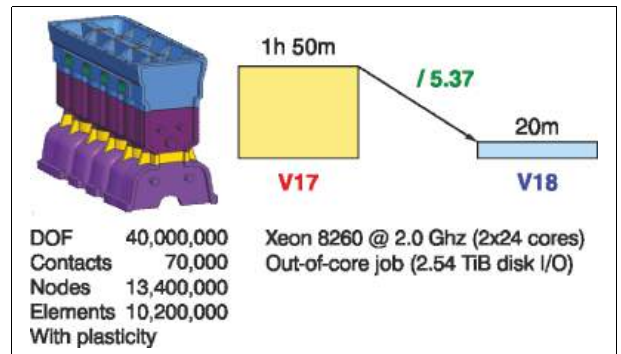


Figure 156: Parallel nonlinear element stress computation of a motor with 10.2 Mio elements

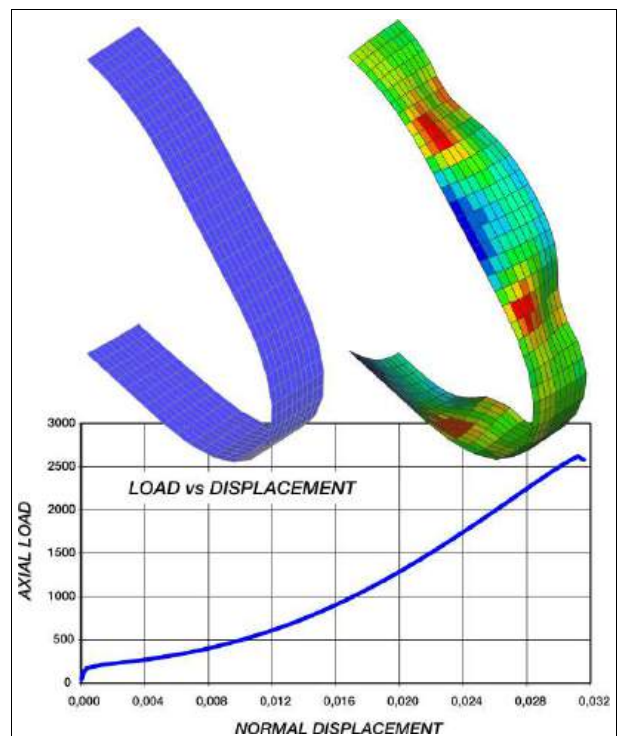


Figure 157: Nonlinear NAFEMS Test

Linear but temperature-dependent material

When temperature is changing during a nonlinear load history, then the update of the stiffness and load matrices requires a nonlinear solution approach. If material and geometric nonlinearities can be neglected, a special algorithm is available which has a high potential for saving run time compared to a fully nonlinear solution. This approach can be combined with nonlinear gaskets using *CCNG analysis* (see page 85) in particular for efficient analysis of combustion engines (see engine example in Fig. 36).

Material nonlinearities

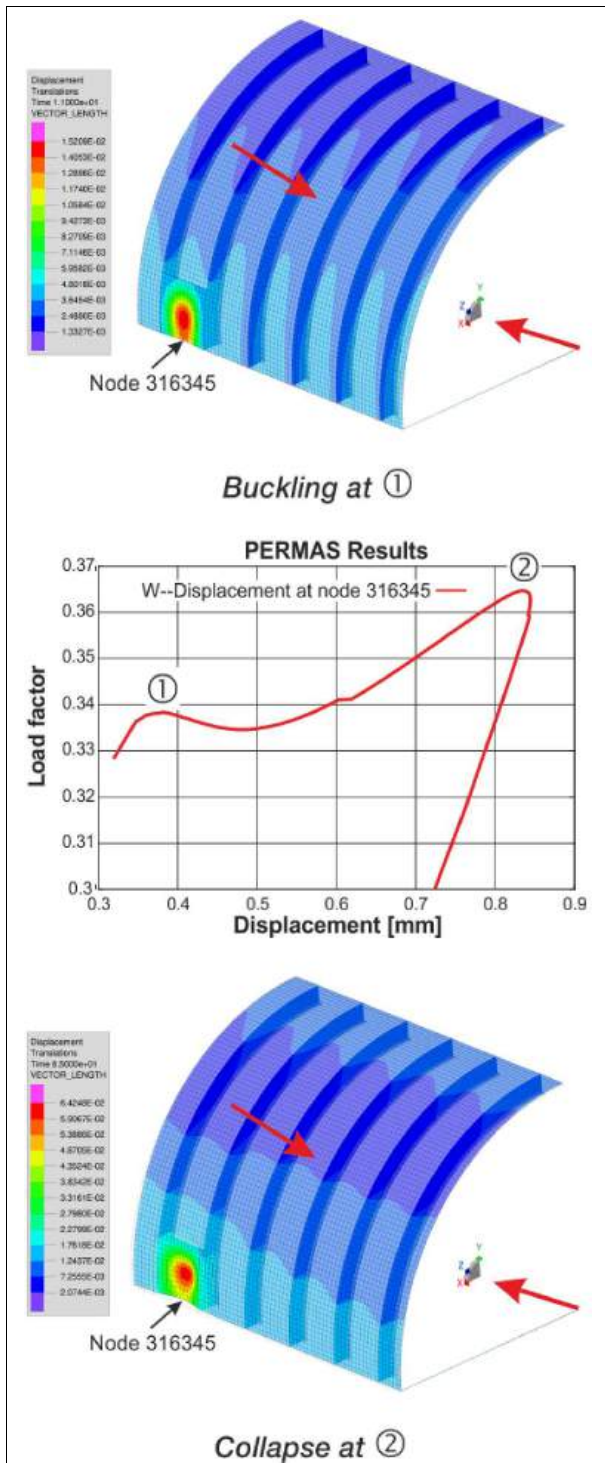


Figure 158: Elastic-plastic buckling with post-buckling behavior

This part of the module allows the analysis of nonlinear material behavior of models with small strains under isotropic conditions:

- Nonlinear elasticity (of *Cauchy* type)
- Plasticity (*von Mises*, *Tresca*, *Drucker-Prager*, *Mohr-Coulomb*)

- *Visco-plasticity* of power-law type for von Mises yield criterion.
- Creep with
 - nonlinear elasticity or
 - plasticity

The material can be defined temperature-dependent for Young's modulus, yield stress, and the stress-strain curves. A time-dependent characteristic is present for creep calculations in addition. *Hardening* in plasticity can be defined as *isotropic* or *kinematic* or *mixed*.

For the use of shell elements with material nonlinearities, an element family of elements with a 3-dimensional shell formulation is available, which is applicable for linear analyses, too. This element family (triangles and quadrangles with linear and quadratic shape functions) has been designed for nonlinear analysis with already existing shell models.

For the modeling of gaskets a family of *gasket elements* is available. These elements are used to define the nonlinear behavior in a preferential direction by a measured force-displacement curve of the real gasket.

An incremental and iterative solver strategy is based on *Newton-Raphson*, Modified Newton-Raphson, and *Thomas* method. An automatic load step control allows for an optional specification of initial load step and total applied load (or time). The material laws may be defined either in tabular form or as user-written subroutine (Fortran or C).

Applications using *inertia relief* (see page 81) can also take nonlinear material behavior into account.

Structural behavior can be influenced by any pre-treatment (like *casting*, *rolling*). The resulting internal strains can be used as initial conditions (without displacements).

User defined material laws can be used. User subroutines allow the incorporation of own material laws. The subroutine does the necessary calculation of stresses and strains together with the tangent matrix associated with the material law.

In addition to isotropic plasticity, there is also transversely isotropic plasticity supported. Typical applications are for *short fibre material*. Fig. 159 compares two simple beam models with identical mate-

rial and loading but different fibre orientation.

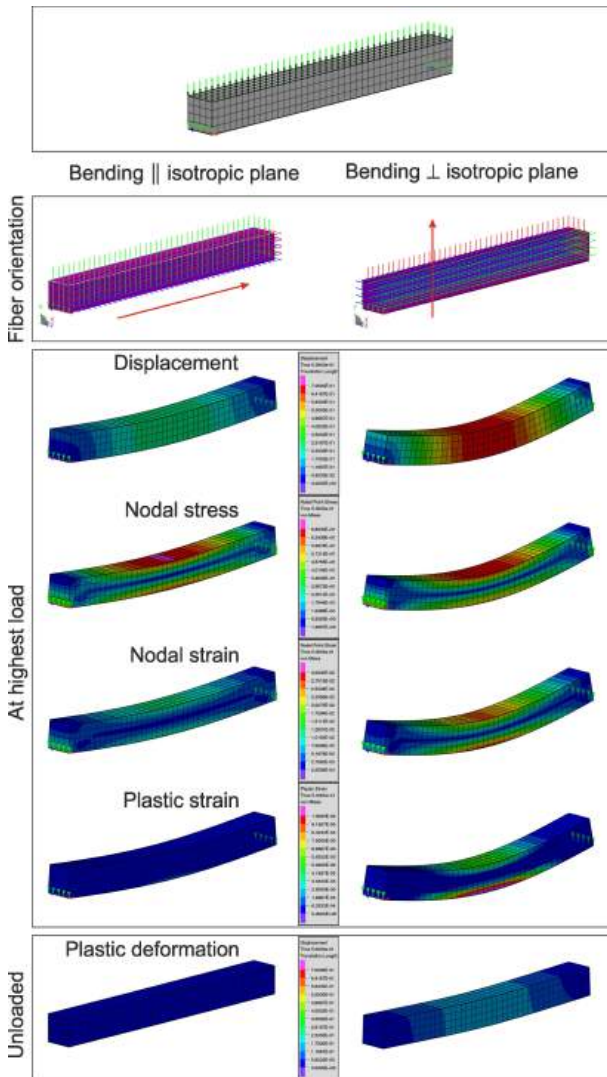


Figure 159: Transverse isotropic plasticity

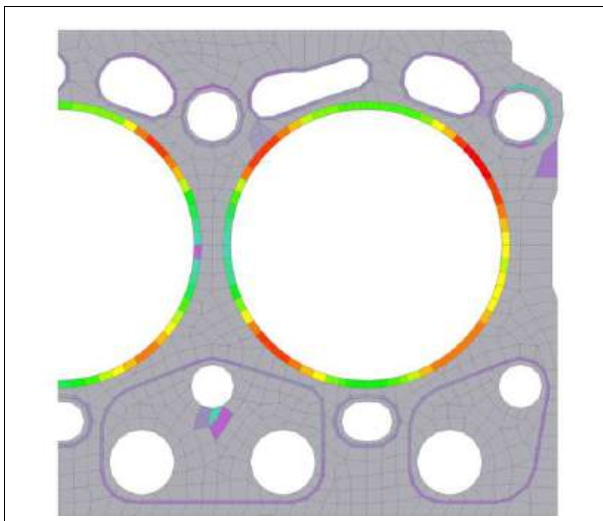


Figure 160: Pressure of cylinder head gasket

Combination of material and geometrical nonlinearities

Analyses with material nonlinearities can take into account the geometrical nonlinear effects, too. There, also follower loads like pressure loads, temperature loads, and inertia loads can be taken into account.

An application example is post-buckling of curved shells (see Fig. 158). This allows the static analysis beyond plastic buckling. In addition, after nonlinear analysis a linear buckling can be performed to study all possible buckling modes at this point of loading (see Module BA on page 93).

General

In case of contact definitions the nonlinear analysis takes them into account automatically performing a nonlinear contact analysis.

Initial states like for rotating structures can be taken into account in nonlinear analyses.

The results of a nonlinear analysis may be used for subsequent analysis like a dynamic mode analysis.

In many cases, the major part of the model is linear. This is an ideal prerequisite to apply *substructure technique* (see page 59), where the linear parts are put in subcomponents and all nonlinear parts are put in the top component. This procedure will lead to a significant reduction in run time.

If many loading cycles have to be calculated for realistic models (e.g. with creep in engine models), then a recommendable strategy is to use *submodeling* to reduce the the model to the area of interest. The results of the global analysis are mapped to a sub-model (see section on submodeling on page 59). Then, the nonlinear analysis will be performed for the submodel. Run times are reduced dependent on the size of the submodel and may allow for time consuming nonlinear analysis.

PERMAS-NLSMAT – Extended Material Laws

This module comprises a number of additional material laws to complement the standard material laws in module NLS:

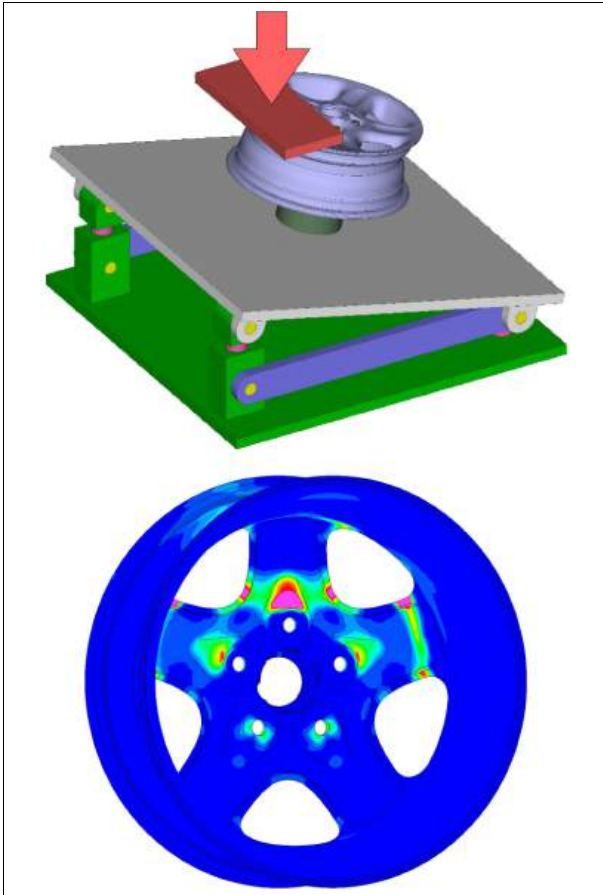


Figure 161: Impact test of a wheel with effective plastic strain.

- A material law for **cast-iron** is available taking into account the different behavior under tension and compression (see Fig. 162).
- Furthermore, a **nonlinear kinematic hardening** model (following **Armstrong-Frederick**) is available as a model for cyclic loading.

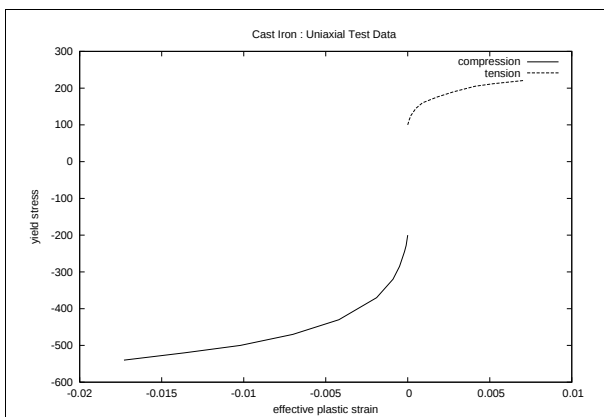


Figure 162: Uniaxial test data of a cast iron material

PERMAS-NLSA – Nonlinear statics advanced

This module enables **hyperelastic material** (quasi-incompressible) together with **large strain** elements in nonlinear analyses.

- The complete volume type element family has been extended to support large strains and hyperelastic materials (see Fig. 163). If the element properties of the elements are assigned to a hyperelastic material definition, then the nonlinear static solvers will use a large-strain formulation. For all other analyses, or if no hyperelastic material is assigned, the elements are used in their classical small-strain formulation.

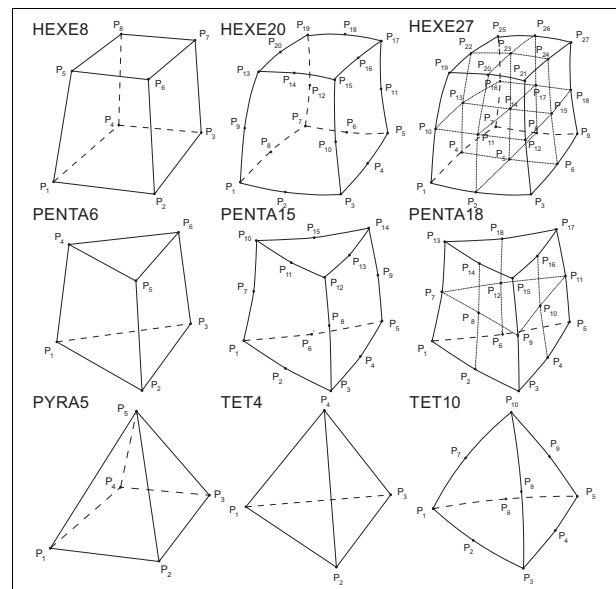


Figure 163: Large strain elements

- New hyperelastic material description for taking into account non-linear stress-strain relations (also nearly incompressible material behaviour is included). With the selection of the strain potential type the following material models are selectable:
 - Polynomial model
 - Reduced polynomial model
 - Neo-Hooke model
 - Mooney-Rivlin model
 - Yeoh model
 - Ogden model
- In order to control the linearised small-strain behaviour for every hyperelastic material definition the linearised Young's Modulus and Poisson's Ratio is printed on the result file, e.g.:

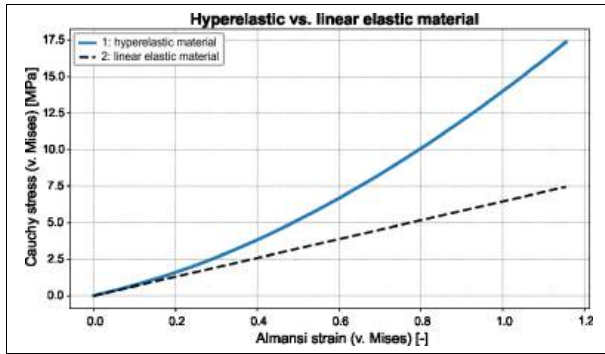


Figure 164: Hyperelastic material with non-linear relation between stress and strain (blue curve) and elastic material with linear relation between stress and strain (dashed curve)

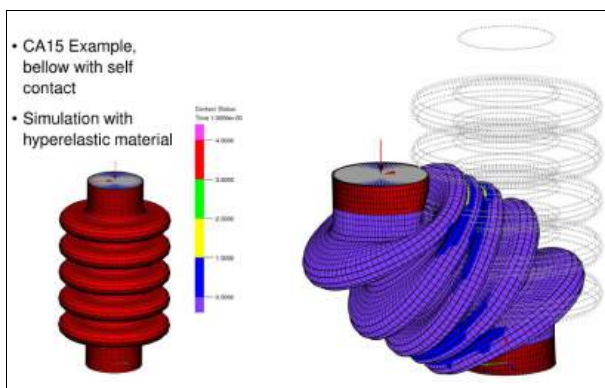


Figure 165: Bellow with self contact

```

SENTER MATERIAL
|
$MATERIAL NAME=MAT_NEOHOOKE TYPE=ISO
$HYPERELASTIC SPTYPE = NEO_HOOKE
1.e4 3.e-5 ! C_10 D_1
SEND MATERIAL
|
$MATERIAL NAME=MAT_MOONEY
$HYPERELASTIC SPTYPE = MOONEYRIVLIN
1.5e4 0.5e4 3.e-5 ! C_10 C_01 D_1
SEND MATERIAL
|
SEXIT MATERIAL

```

> Linearization of hyperelastic material for zero strains

Linearized Hyperelastic Material Data (Reftemp = 0.0E+00)		
Material	E	nu
MAT_NEOHOOKE	54.545E+03	363.636E-03
MAT_MOONEY	100.000E+03	250.000E-03

PERMAS-BA – Linear Buckling

Based on a linear static analysis the related buckling modes with load factors and mode shapes can be determined.

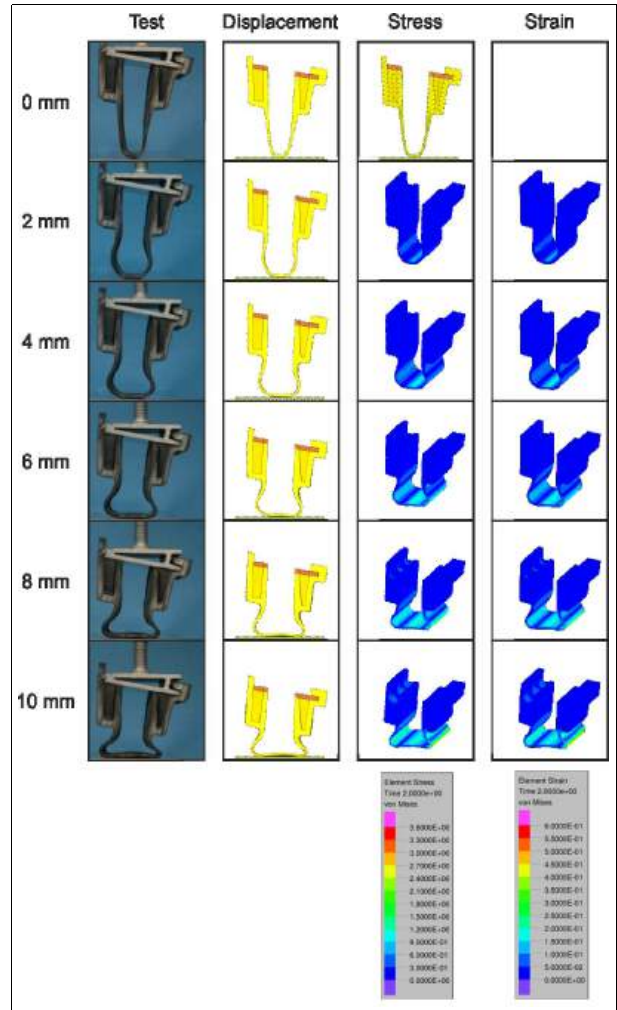


Figure 166: Large strain analysis
(by courtesy of Mahle Behr GmbH, Stuttgart, Germany)

The calculation of *modal participation factors* allows for the assessment of the nonlinearity of the *pre-buckling behavior*.

Load factors and mode shapes are available for any kind of post-processing. For the load factors, a limit or an interval may be specified. The eigenvalue analysis is performed using an advanced *shift method*.

The analysis may also be performed at a singular step of a nonlinear analysis to understand the failure modes at a *bifurcation point*.

Sampling, topology optimization, and shape optimization are now possible also for buckling analysis (see Fig. 167).

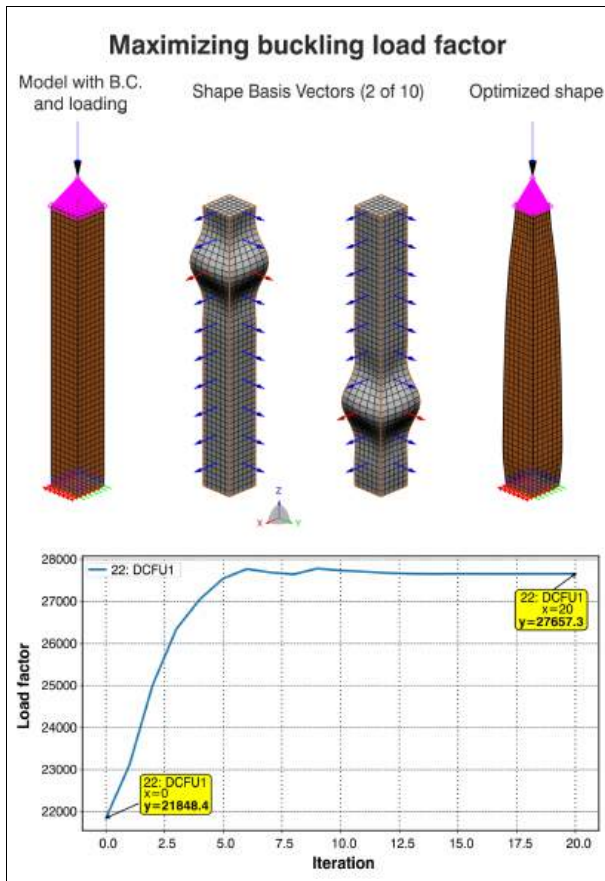


Figure 167: Maximizing buckling load factors by parametric shape optimization.

PERMAS-HT – Heat Transfer

Temperature fields will be analyzed using the modules PERMAS-HT and PERMAS-NLHT (see next section).

- The temperature field may be steady-state or transient.
- Nonlinear material data for conductivity and heat capacity may be specified by tabular input.
- Temperature and space-dependent convectivity coefficients may be specified in a very general way by functions (see page 69).

PERMAS-HT provides a complete set of *convectivity elements* to model surface convection. In addition, they allow for the determination of the surface area of a set of elements in order to get the drained or injected heat through the related surface. Moreover, an optional film thickness may be specified for the convectivity elements, which allows to model the heat capacity of boundary layers in transient analysis.

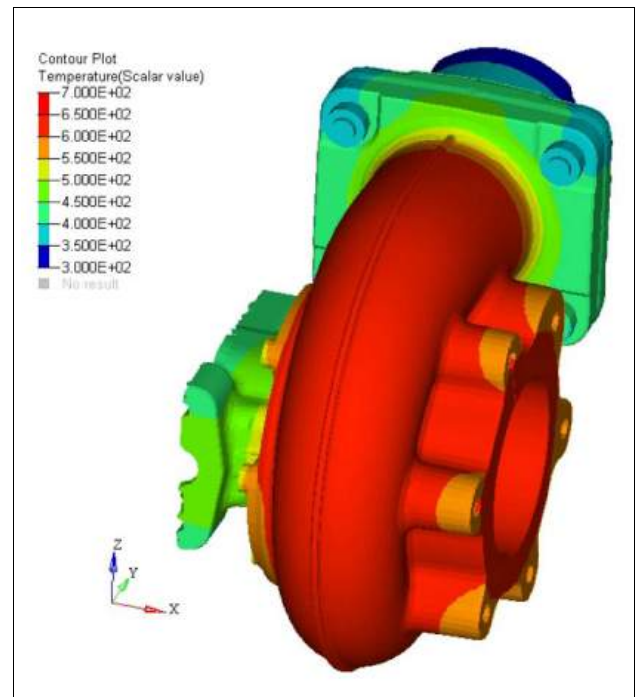


Figure 168: Temperature field of a turbocharger

All finite elements applicable in static analysis may be used in heat transfer analysis. For shell elements a temperature gradient between top and bottom surface is allowed.

Coupled analysis of thermo-mechanical problems is fully automatic, i.e. the resulting temperature field is directly used to derive the related displacements, strains and stresses. The material data for the static analysis (elasticity and thermal expansion data) may be temperature-dependent.

In addition, using PERMAS-CCL convective boundary conditions can be imported from a CFD analysis or fully coupled analyses can be performed, respectively (see page 76).

A transient analysis may be continued by reference to the results of the previous run. Among others, during the simulation of complete cycles of thermal loads this feature easily allows for sudden changes of the surrounding conditions.

A *modal analysis* allows for the computation of eigenvalues and eigenmodes for heat transfer problems.

Available loads and boundary conditions:

- Stationary 'loads' may be defined as point heat fluxes or distributed heat fluxes along lines, on surfaces and in volumina.

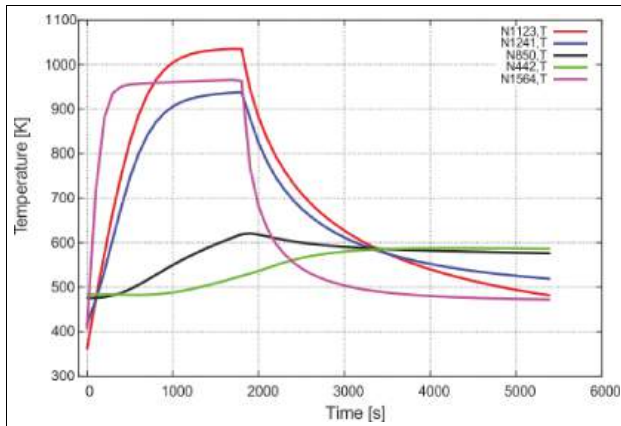


Figure 169: Transient temperatures at nodes over time

- Transient 'loads' are built conveniently by combination of a stationary 'load pattern' with time-dependent functions (see page 69).
- Additional boundary conditions are prescribed temperatures and a surrounding temperature for convectivity elements.

Primary results of a heat transfer analysis are the temperature field and the heat fluxes. In addition, the following derived results are available:

- the gradient of the temperature field,
- the heat flux through any internal face,
- arbitrarily composed element sets allow for the output of the heat flux through a part of the surface in absolute or area specific values.

In addition, for transient analyses primary and derived results may be issued for any point in order to generate XY plots.

PERMAS-NLHT – Nonlinear Heat Transfer

The methods available for nonlinear analyses in PERMAS-HT are complemented by a more advanced set of algorithms to solve higher nonlinear computations for steady-state and transient problems.

In contrary to the methods described in the previous section an automated stepping algorithm is used for both steady-state and transient analysis, which may be complemented by manual selections of explicit points in time or load levels.

For the selection of results, load steps and particular times can be explicitly defined.

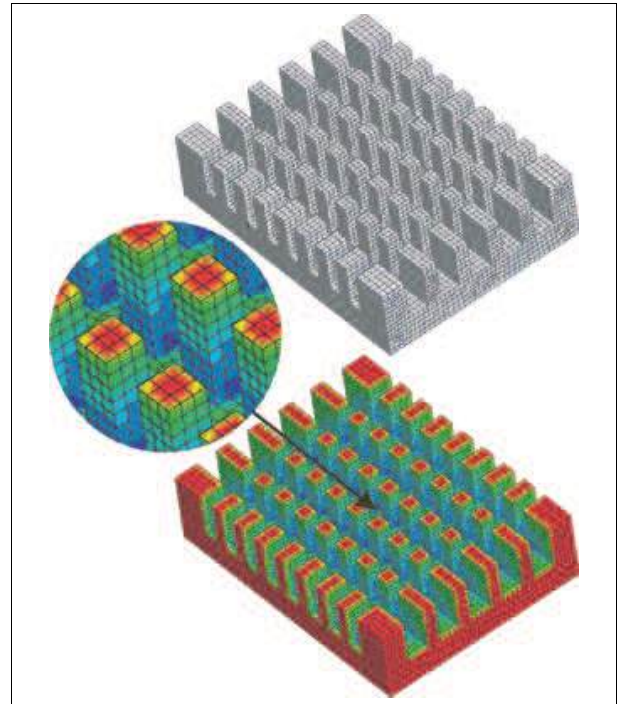


Figure 170: Analysis of cooling element with radiation

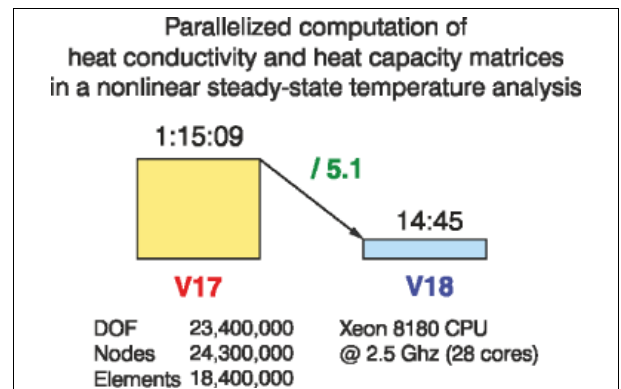


Figure 171: Parallelization of conductivity and capacity matrices.

Radiation with heat exchange

Heat transfer by radiation is increasingly important with higher temperatures and for parts with cavities and self-shadowing effects like brakes, combustion engines, and cooling elements. This allows heat transfer analyses with convection and radiation coupled with heat conduction.

The assumptions for this function are heat exchange between surfaces (no radiation from within bodies), radiation of grey bodies (radiation not dependent on

wave length), and diffuse emission (radiation not dependent on radiating direction).

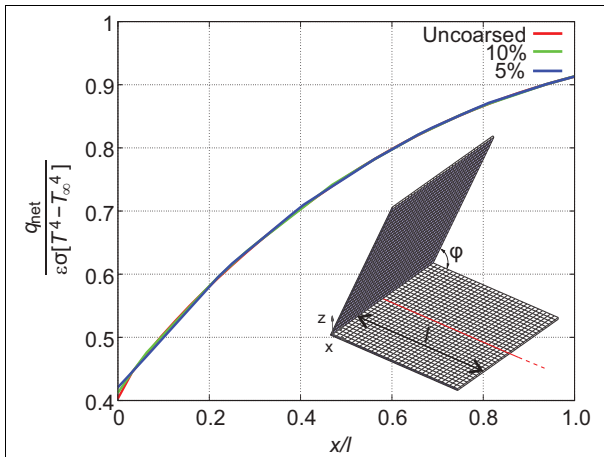


Figure 172: Heat radiation between adjoint plates

The calculation of radiation has the following characteristics:

- The radiation is integrated in the heat transfer analysis process.
- The convection elements are extended to model also radiating surfaces, i.e. all surface elements where radiation has to be taken into account have to be modeled with convection elements.
- There is a direct integration of the *view factors* over the surface elements instead of averaged view factors.
- In order to accelerate the calculation of viewing factors with a very high number of surface elements an automatic (selective) coarsening procedure is provided to reduce the number of surface elements.
- The computational efficiency is obtained by using *parallelization*.
- The coupled solution of the nonlinear heat transfer equation with radiation boundary conditions is performed in a few iteration steps either for steady-state or transient calculations.

PERMAS Vibro-Acoustics

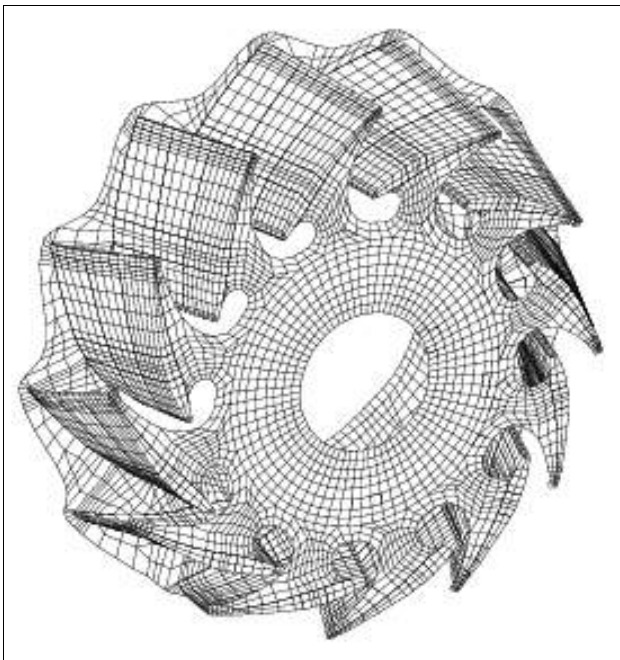


Figure 173: Half Model of a Turbine
Half Model of a Turbine, 8th Mode Shape with
Antisymmetric Boundary Conditions

PERMAS-DEV – Dynamic Eigenvalues

The PERMAS-DEV (Dynamics/Eigenvalues) module provides for the calculation of *real eigenvalues* and mode shapes of the structure (*modal analysis*). The specification of a number of modes and an upper frequency limit is supported. The very efficient *subspace iteration* algorithm used is capable of solving very large eigenvalue problems. Rigid body modes are detected automatically or may be explicitly defined and are decoupled prior to the subspace iteration.

If the number of modes is increasing, one can observe a disproportional increase of solution time, because the last modes take more time than the first modes. Therefore, a *shift method* has been introduced which are used to split the frequency domain in several parts and to solve each part separately. Then, even the last modes can be solved as fast as the first modes resulting in an overall run time reduction for the complete analysis. The shift method is used automatically by PERMAS in cases, where a large number of modes is required (> 5000 modes) or where the standard method takes a long time

to converge. In addition, a higher accuracy of the mode shapes can be expected from this method.

The stiffness matrix can be modified taking into account additional stiffness effects:

- *Geometric stiffness* for any load,
- *Centrifugal stiffness* for rotating parts under constant rotational speed referring to co-rotating reference system,
- *Convective stiffness* for rotating parts under constant rotational speed referring to inertial reference system,
- *Pressure stiffness* for shell elements and fluid-filled pipe elements under pressure.

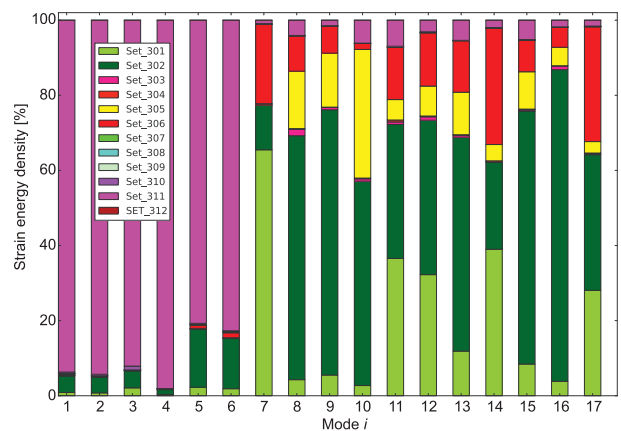


Figure 174: Analysis of strain energy density
in element sets and for various modes

Additional tools are available for the further processing of modes:

- Modal stresses can be derived from modal displacements.
- In addition, modal potential and kinematic energies can be calculated and exported (see Fig. 174).
- For the evaluation of modes, e.g. with respect to local or global mode shapes, energy balances can be determined and exported for all sets in a structure.
- *MAC* (Modal Assurance Criterion) factors and other factors are available to compare modes between two different modal analyses (see page 74).
- As a measure for the completeness of the modal model, *effective masses* are generated and printed on the result file.

A *generalized modal condensation* is available to establish system matrices in modal space for external applications. Export of modal models is either

supported by interfaces (e.g. to MBS systems) or by direct specification of the matrix items.

PERMAS-DEVX – Extended Mode Analysis

This module provides additional methods for dynamic eigenvalue analysis:

- *Dynamic condensation*
- Complex mode analysis
- Eigenfrequencies over rotational speed for rotating structures

Dynamic Condensation

Two methods are available for dynamic *condensation*:

- Craig-Bampton method (CB condensation),
- Mixed-Boundary Craig-Bampton method (MBCB condensation).

Craig-Bampton method:

The *Craig-Bampton* method uses fixed-interface vibration modes and the static deflections due to unit displacements of the interface degrees of freedom for the dynamic reduction of substructures (*CB condensation*). Like for the *Guyan's* reduction, an explicit and an iterative scheme is available in order to achieve good performance (see also page 59).

The functionality may be summarized as following:

- **Supported solutions**
 - Structural dynamics
 - Acoustics
 - Coupled fluid-structure acoustics

Two *condensation* options are available for coupled fluid-structure acoustics (see also page 104):

- **“Dry” Interface**
 - Solution of a coupled eigenvalue problem on subcomponent level, i.e. **isolation** of the acoustic component. External modes are coupled modes.
 - Global solution may be a mechanical vibration analysis.
- **“Wet” Interface**
 - Separate computation of mechanical and acoustic modes on subcomponent level.
 - Global solution is a coupled vibration analysis.

- Condensation of the fluid-structure interface can also be made.

Mixed-Boundary Craig-Bampton method:

The method allows for free (or mixed) boundary conditions to derive the vibration modes (*MBCB condensation*). The corresponding static deflections are derived using *inertia relief*. This method is advantageous when the condensation is used for structures with free boundaries, because the condensation method can represent the boundary conditions. In such cases, less number of required modes and a higher accuracy can be expected from the Mixed-Boundary Craig-Bampton method compared to the classical Craig-Bampton method.

The above mentioned “dry” condensation of parts with enclosed fluids also works with the Mixed-Boundary Craig-Bampton method.

Complex Mode Analysis

This includes the calculation of complex eigenvalues and eigenvectors in modal coordinates. This method is based on a previous solution of the real eigenvalue task.

The results of this analysis are as follows:

- Frequencies
- *Complex eigenvalues*
- Complex eigenfrequencies (each with damping coefficient and circular frequency)
- Equivalent viscous damping ratios
- Complex mode shapes with physical and modal representation. The modal displacements of the complex modes represent the modal participation of the underlying real modes.

A suitable post-processor (like VisPER) can be used to visualize and animate complex mode shapes

Eigenfrequencies of Rotating Systems

The **MODAL ROTATING** command issues a table, that shows whether a rotational speed is unstable and also what the first unstable frequency is:

Iteration progress of MODAL ROTATING stability calculation in situation SQUEAL of component DISC				
	Rotspeed	CEV formulation	1st unstable freq	Result
145630	0.500000	DIRECT	-	STABLE
145630	0.600000	DIRECT	-	STABLE
145630	0.700000	DIRECT	-	STABLE
145630	0.800000	DIRECT	-	STABLE
145630	0.900000	DIRECT	-	STABLE
145630	1.000000	DIRECT	-	STABLE

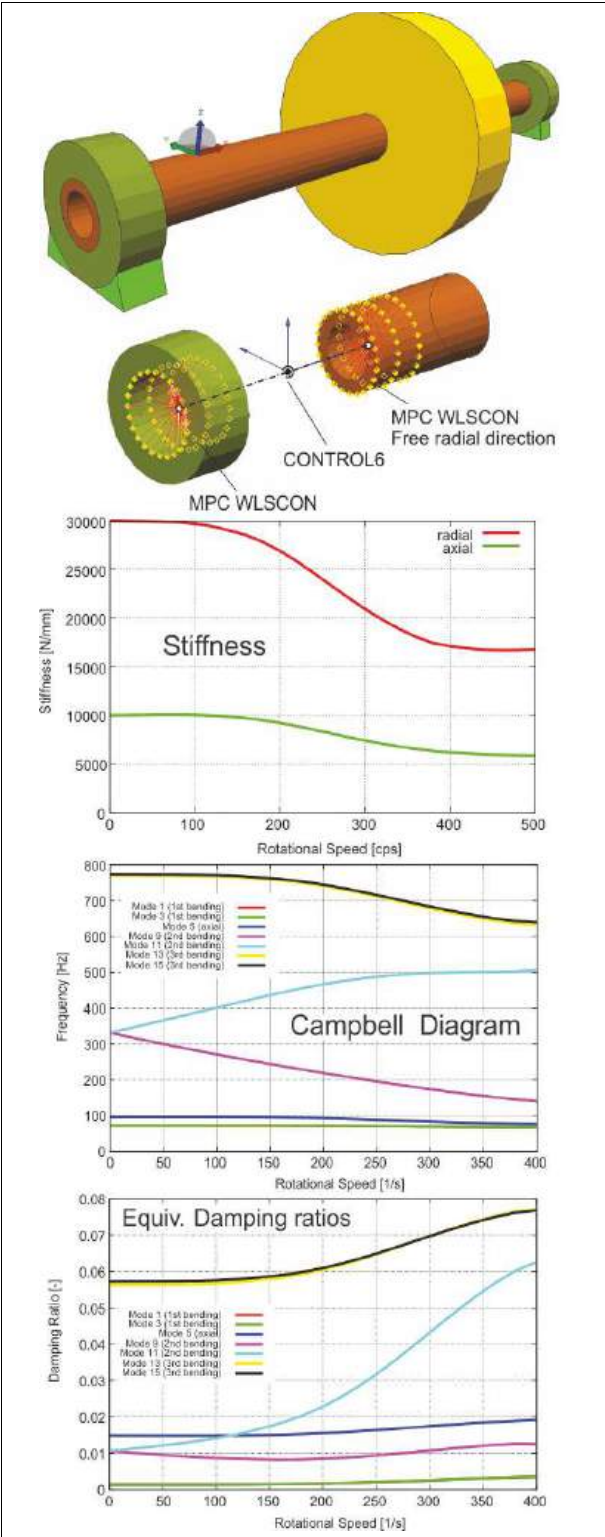


Figure 175: Rotor with rotation speed dependent supports

145630	1.500000	DIRECT	-	STABLE
145630	2.000000	DIRECT	-	STABLE
145630	2.500000	DIRECT	-	STABLE
145630	3.000000	DIRECT	-	STABLE
145630	3.500000	DIRECT	2.760E+03	UNSTABLE
145630	4.000000	DIRECT	2.759E+03	UNSTABLE

145630	4.500000	DIRECT	2.757E+03	UNSTABLE
145630	5.000000	DIRECT	2.755E+03	UNSTABLE

For rotational systems (see also page 31) it is often required to generate a so-called *Campbell* diagram, which relates the eigenfrequencies to the rotational speed. The values of such a diagram can be generated automatically in one single run. From these values all frequencies of interest can be selected for a subsequent frequency response analysis.

For rotating structures, any number of rotational speeds is defined in a separate input. A reference rotational velocity is used in the static pre-run. From this pre-run, additional matrices are built for the reference rotational velocity. The specified rotational velocities are used to scale the additional matrices during dynamic response analysis. This procedure makes the generation of Campbell diagrams very efficient and the response analysis of rotating structures is facilitated.

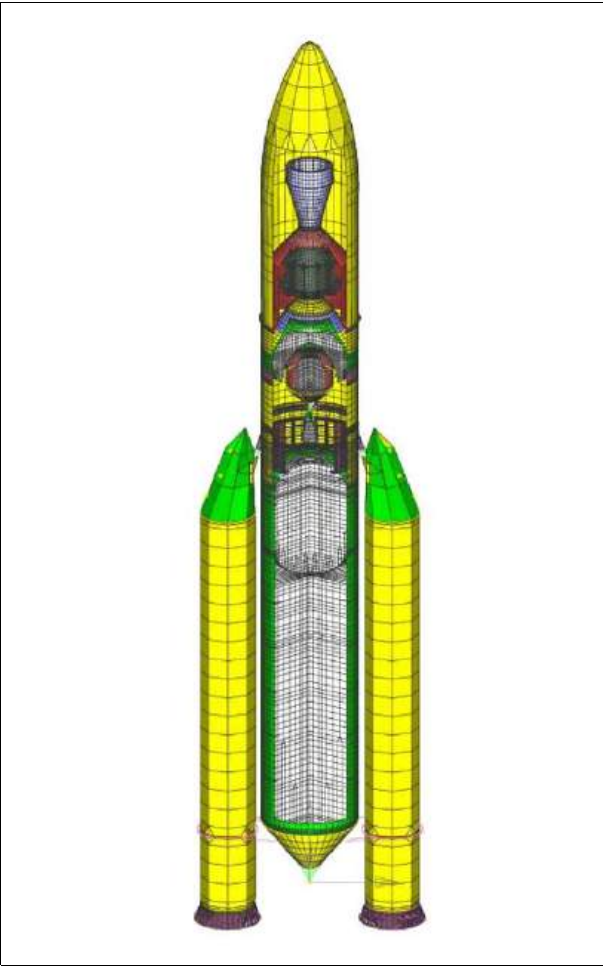


Figure 176: Ariane 5 launcher model by courtesy of ASTRIUM Space Transportation, Les Mureaux

PERMAS-MLDR – Eigenmodes with MLDR

The calculation of eigenvalues with modules DEV (page 97) and DEVX (page 98) is complemented by another method. This method can also be used for the calculation of coupled fluid-structure modes.

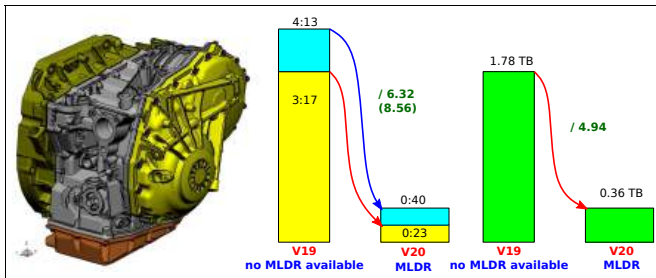


Figure 177: MLDR, 4.5M nodes with CONTLOCK
Elapsed time [h:mm] and disk space

The application of this method is advantageous in those cases where the elapsed run times are mainly determined by I/O like for large models with a high number of modes to be calculated. The larger the models and the larger the number of modes, the higher is the benefit in elapsed run time through the application of the MLDR method.

This benefit in elapsed run time can be essentially raised more, if there is a subsequent dynamic response analysis calculating the response behavior just at a small number of nodes. Then, the generation of the global mode shapes can be saved resulting in considerable computing time savings.

The MLDR method is based on an automatic partitioning of the model where each part does not exceed a preset quantity. In addition, the coupling between the parts has to be as low as possible. These parts are then groupwise combined as substructures using *dynamic condensation* (see substructuring on page 59 and module DEVX on page 98). This procedure is hierarchically carried on until the complete model is represented in one component. In this component only a small number of nodes and elements remain and the dynamic behavior is mainly determined by the modes and frequencies taken over from the substructures and combined following the rules of dynamic condensation. Due to

this procedure the method's name is *Multi-Level Dynamic Reduction (MLDR)*

In order to support a subsequent response analysis, assembled situations (see page 103) and static mode shapes (see page 103) are considered. Also static modes from inertia relief analysis (see page 81) can be used. Also temperature-dependent stiffness is taken into account.

If certain nodes and elements should be present in the main component, the user can specify them explicitly. So, selected model parts can be pushed to the main component and any subsequent processing of the modes is rather beneficial due to the small size of the remaining matrix system. In this way, dynamic simulation, coupling to MBS, optimization of the remaining system, or the consideration of non-linearities can be performed with very low computing times.

Additional reductions of computing time are possible using multi-processor systems, because the method has been fully parallelized. Altogether, the use of MLDR is a big step forward to more productivity and allows, for example, dynamic simulation in a higher frequency range as in the past together with a possible increase in model size for more accurate results.

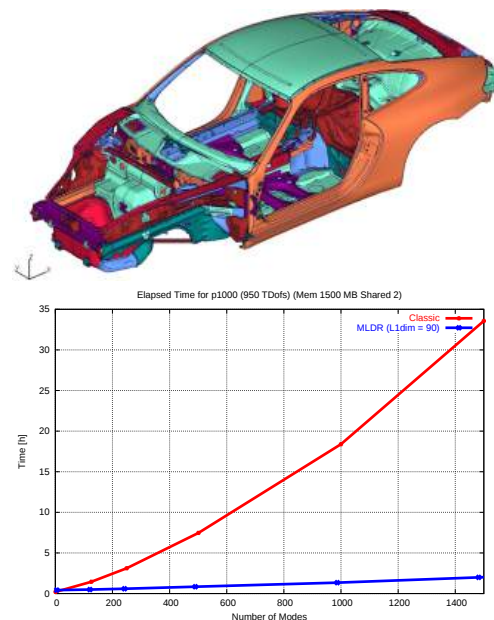


Figure 178: Comparison of elapsed run times for Subspace Iteration (upper curve) and MLDR (lower curve) with increasing number of modes (157412 Nodes, 164301 Elements (QUAD4), 944472 Unknowns)

PERMAS-DRA – Dynamic Response

The PERMAS-DRA (Dynamic Response Analysis) module allows for the determination of structural responses in the time or frequency domain.

Feature	Options
Frequency response	modal and direct
Timehistory response	modal and direct
Material properties	constant, frequency dependent, visco-elastic
Damping	via material, modal, elements, direct input
Loads	concentrated, distributed, inertia, PSD
Real and complex eigenvalues	yes
Additional static modes	yes
Assembly Situations (multiple RHS)	for frequency and timehistory
Nonlinearities	contact, nonlinear springs, dampers, control elements, geometric, pressure and centrifugal stiffness
Primary results	displacements, velocities, accelerations
Secondary results	reaction forces, stresses, strains, sound rad. power etc.

Table 1: Overview of dynamic response capabilities

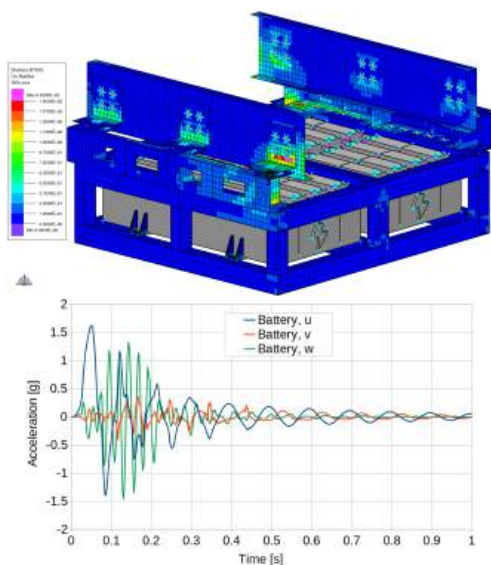


Figure 179: Battery pack, contact with dynamic shock

The solution of the dynamic equation is performed either directly using physical coordinates or in modal coordinates after a transformation into the modal space.

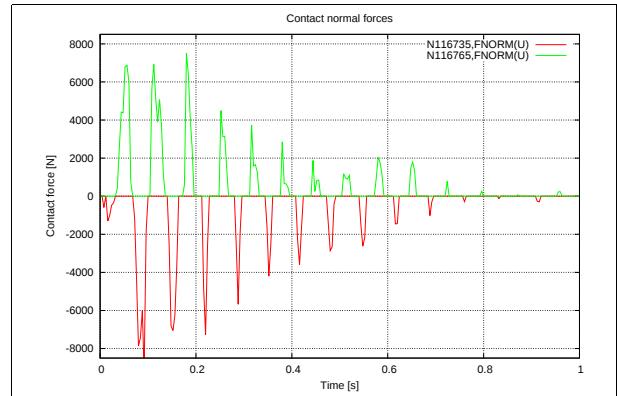


Figure 180: Contact forces, dynamics

- The **response in the time domain** (*transient response*) is determined by an integration of the equation of motion:
 - Absolute transient response with or without rigid body response.
 - Direct integration of the equation of motion or integration after a transformation to the modal space. Available solvers are *Newmark β* , *HHT (Hilber-Hughes-Taylor)*, *TWOSTEP* and *SEMIIMPLICIT*.

Local nonlinear effects are taken into account by

- nonlinear spring elements,
- nonlinear damper elements,
- nonlinear control elements,
- and contact conditions.

- The **response in the frequency domain** (*frequency response*) is determined by the solution of the linear complex equation system for each excitation frequency requested:
 - Absolute frequency response with or without rigid body response.
 - Direct solution of the equation system or solution after a transformation to the modal space.

Frequency and rotation speed dependent stiffness and damping can be taken into account with **CONTROL6** element (see examples in Fig. 181 and Fig. 175).

- Without running through the transient phenomenon a calculation of the **steady-state response** can be performed. To achieve that, a number of frequency response analyses are superimposed in the time domain. In addition, a static load case can be taken into account. This is facilitated for all periodic excitations with

known harmonic composition.

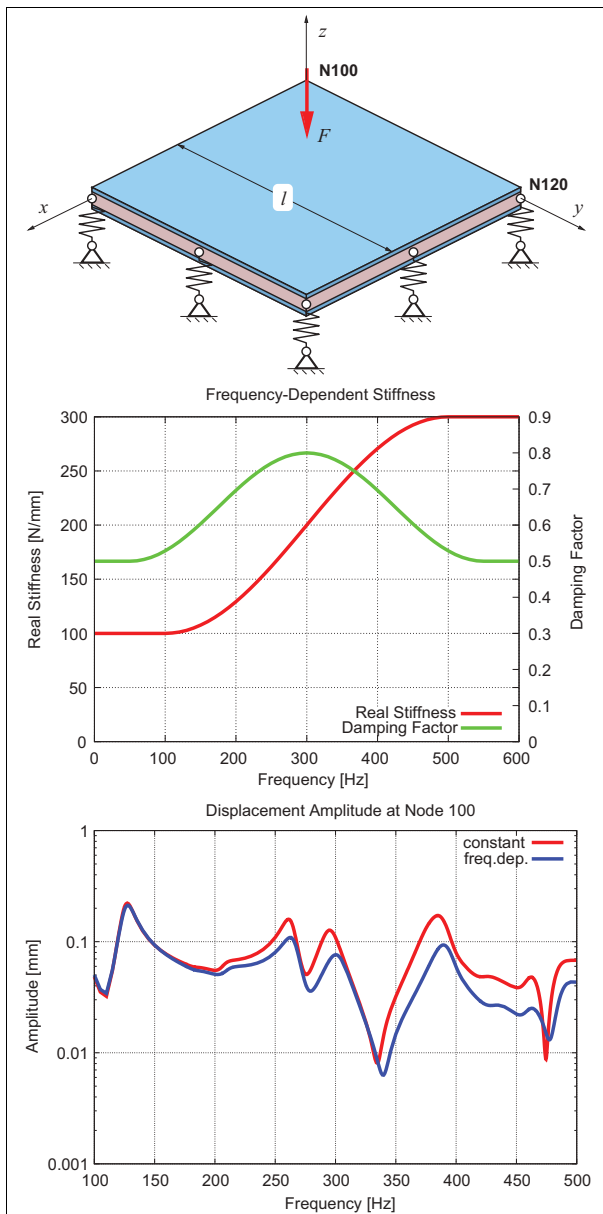


Figure 181: Frequency-dependent stiffness and frequency dependent viscous damping in frequency response analysis

By specification of a node set (see page 69) the run time and disk space for modal superposition methods can be drastically reduced when the results are determined for the set members only.

The features below hold for both time-history and frequency response methods:

- The **damping** properties may be modeled by one of the following methods:
 - material or structural damping for elements,
 - global structural damping for components,

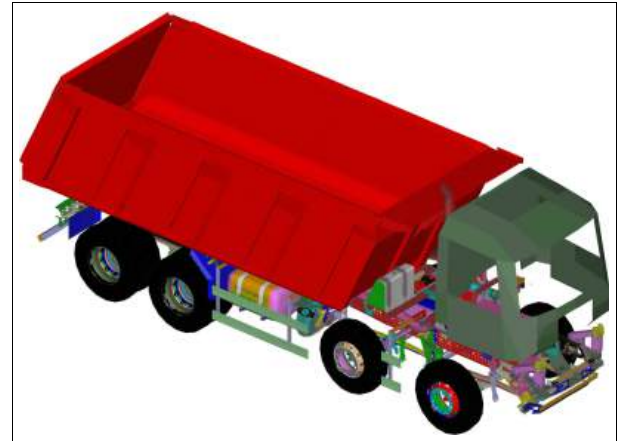


Figure 182: The model of a tipper truck (by courtesy of Daimler AG, Stuttgart, Germany).

- proportional damping (Rayleigh damping),
- viscous damper elements,
- modal viscous damping,
- modal structural damping,
- modal structural and proportional damping for subcomponents,
- direct input of modal damping matrix ,
- direct input of damping matrix.

For calculations in frequency domain, the structural damping may be defined as a function of frequency, alternatively.

Frequency-dependent viscous damping parameters can be specified using a special element (i.e. CONTROL6 element).

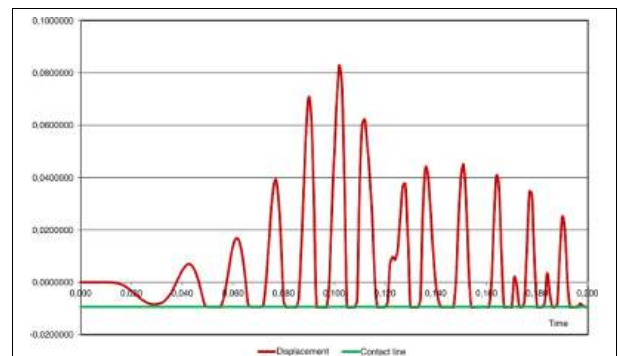


Figure 183: Transient response under contact conditions

- The excitation is defined by static loading cases modulated by functions of time or frequency respectively (see page 69). The load definitions may consist of:
 - concentrated forces or moments,
 - distributed loads (loads applied to lines, surfaces or volumes),

- inertia loads,
- prescribed displacements.
- The primary results are:
 - displacements,
 - velocities, and
 - accelerations.
 For the drawing of XY plots, these data may be output as a function of time or frequency respectively.
- Moreover, the following results may be derived:
 - reaction forces,
 - stresses and stress resultants,
 - strain energy,
 - kinetic energy,
 - specific sound radiation power density.

When modal methods are applied additional functions are available:

Static Mode Shapes can be generated in order to enhance the modal basis. This has the following two advantages:

- In the low frequency range the results will become essentially more accurate.
- They allow to consider non-structural degrees of freedom like the internal state variables of control elements.

The static mode shapes can be specified using one of the following cases:

- directly by nodal displacements,
- by external loads,
- by results from another analysis,
- by natural loads of specified elements (like spring forces),
- implicitly by internal degrees of freedom of controller elements,
- by inertia loads as generated by *inertia relief* analysis.

Assembled situations can be used to highly accelerate frequency response and timehistory analyses with many different load cases/time steps. Instead of solving all dynamic load cases separately, a combined response analysis can be performed.

For models with many eigenvalues, many excitation frequencies, and only some dampers the computation of the frequency response is expensive. As long as the damping is not frequency dependent (only a few discrete dampers), a diagonalisation of

the system is possible via a single value decomposition (complex eigenvalues). An explicit inversion of the resulting system matrix may be performed, applying the *Shermann-Morisson-Woodbury SMW* formulation. The subsequent computation of each frequency point is cheap. This **SMW solver** is much faster than the general solver for modal frequency response analysis.

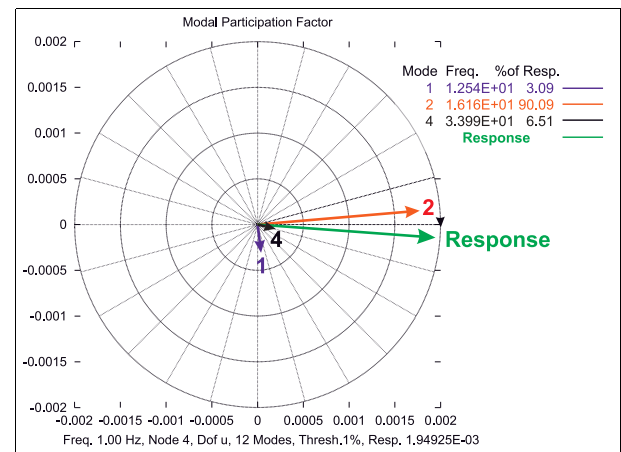


Figure 184: Modal grid participation factors

Additional tools are available for the further processing of modal results:

- For the evaluation of a structural response *modal participation factors* of the primary results can be derived and exported.
- For the evaluation of the contribution of single degrees of freedom to a structural response node participation factors can be derived and exported.
- For a transient response a statistical evaluation of stresses or element forces over the time domain can be performed, which gives the maximal and effective values. These statistical values can be used in durability considerations, for example.

Visco-elastic Material

For *visco-elastic material* in dynamics, a method based on *Prony series* has been introduced. This is for materials with frequency dependent data in direct timehistory or direct frequency response analysis (like rubber, booster propellant, etc.). In order to take the different terms of the Prony series into account, two element types have been developed: One hexahedron element with 8 nodes, where one element with different nodes is used for one term of the Prony series, and another hexahedron ele-

ment with 8 nodes, where one element with internal nodes for all the different terms of the Prony series is used. The latter element uses an internal condensation method. Therefore, the results differ slightly between both element types. The user may decide to use one of the approaches.

Skew Symmetric Pressure Stiffness

The symmetric part of the pressure stiffness can be used in eigenvalue analysis. In addition, the skew symmetric part of the pressure stiffness may be used in dynamic response analyses and complex eigenvalue analyses.

PERMAS-DRX – Extended Dynamics

This module comprises additional methods for structural response analysis:

- Spectral Response Analysis (or *earthquake spectral response* Analysis),
- Random Response Analysis.

Spectral Response Analysis

In case of a prescribed ground motion, like in earthquake analysis, the dynamic response behavior is determined by a special method, which results in maximum response values. There, the following requirements have to be fulfilled:

- uni-directional and translational motion of the ground,
- no other loads,
- analysis can be performed in modal space,
- only modal viscous damping.

After the specification of the direction of the ground motion the load is defined by a spectrum of the transient excitation (*response spectrum*). Then the analysis is performed as follows:

- Calculation of all dynamic modes up to the highest interesting frequency.
- Calculation of the maximum mode contributions.
- Summation of the maximum contribution factors using one of 7 available *summation rules* (like *CQC* or *10% rule*).
- Export or print of peak values.

Random Response Analysis

Frequently, vibrational loads are not predictable like

for cars on a bumpy road, for a house under wind loads, or for a ship on rough sea.

Conveniently, such stochastic loads can be described by random processes. Correlations between such processes and their transformation into frequency domain are leading to the central concept of *power spectral density*.

One special phenomenon is white noise which describes a constant power spectral density over the full range of frequencies.

The loads are specified as power spectral densities and the results are derived as RMS quantities and power spectral densities as well.

This analysis possible either as modal method (i.e. an eigenvalue analysis is performed first followed by the response analysis in modal space and a subsequent back transformation into physical space where the results are made available for export and post-processing), or as a direct method (also for fluid-structure coupled systems).

A LOG file table issues the computation progress for random response analyses:

Progress of DIRECT FSCOUPLD RANDOM calculation in situation RAND of component CYLINDER_IN_WATER			
	Step	Frequency	Rank loading matrix
181731	1/21	1000.000000	59
181731	2/21	1050.000000	63
181731			..
181732	20/21	1950.000000	117
181732	21/21	2000.000000	120

PERMAS-FS – Fluid-Structure Acoustics

The PERMAS-FS (Fluid-Structure Acoustics) module allows for the calculation of acoustic modes, as well as eigenvalues of coupled fluid-structure systems and the computation of coupled or uncoupled response in the frequency or time domain. This computation is provided either directly or in modal coordinates. Table 2 shows an overview on the analysis capabilities for fluid only and for coupled fluid-structure systems.

The fluid may be compressible or incompressible. All fluid absorption or damping properties are also

Acoustics	Fluid only	FS coupled
Degrees of freedom	pressure	+displacements
Elements	fluid, radiation bound. cond., surface waves, semi-infinite elem. (added masses)	+coupling, structure
Material values (also frequency dependent)	density, compressibility (or incompressible)	+density, Young's modulus, Poisson's ratio
Real eigenvalues, eigenmodes	yes	fully coupled, with energy distribution
Additional static modes	yes	yes
Dynamic condensation	Craig-Bampton	dry (for displacement dofs only), wet (also for pressure dofs)
Damping	surface absorption, fluid volume	+any structural damping
Modal response	frequency, time domain, random response	frequency, time domain, random response
Direct response	frequency, time domain, random response	frequency, time domain, random response
Assembly Situations	yes	yes
Primary results	pressures	+displacements, velocities, accelerations
Secondary results	sound particle velocity	+stresses, strains, etc.

Table 2: Overview of acoustic capabilities for fluid only and for fluid coupled to a structure.

available for the uncoupled computation. The fluid damping may be frequency dependent.

A number of special features is provided to model boundary conditions:

- Surface waves are modeled by specific elements.
- Special coupling elements are provided at the boundary of the fluid to the structural model. These elements are also used to model surface absorption. In addition, another acoustic damping facility is available through volumetric dampers (like seats in a car).
- Semi-infinite elements are provided to handle an infinite surrounding space.
- Radiating boundary conditions (RBC) can be modeled using special element families, one following the theory of *Bayliss-Turkel* and another

the theory of *Engquist-Majda*.

For the coupling elements mentioned above the face normal has to be oriented from the fluid to the structure. This condition is checked automatically in order to avoid conflicts and sources of mistakes in the coupling of fluid and structure.

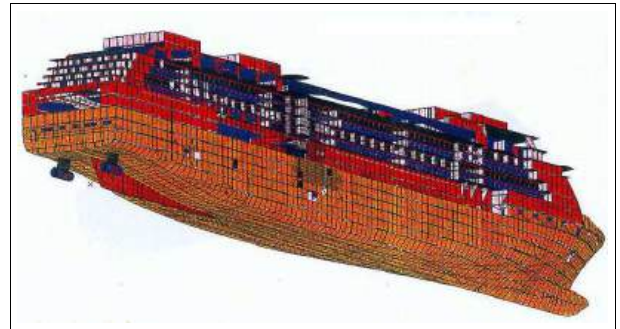


Figure 185: Ship model

by courtesy of Chantiers de l'Atlantique, Saint-Nazaire

For the calculation of dynamic mode frequencies, a difference is made between the fully coupled modes and the structural modes modified by the additional fluid mass:

- The calculation of real eigenvalues and mode shapes of the coupled structure is realized by a simultaneous vector iteration. The specification of a number of modes and an upper frequency limit is supported. The special formulation of the algorithm used is capable of solving very large eigenvalue problems in an efficient way.
- The added mass problem can be solved with the standard structural eigenvalue solver (see page 97), where the mass of the fluid is taken into account to calculate the structural modes.

The runtime for a coupled eigenvalue analysis in case of large models with a high number of modes can be drastically reduced by the MLDR method (see page 100).

For the calculation of the dynamic response behavior, the following methods are available:

- The response in the **time domain** (transient response) is determined by a modal solution of the equation of motion. Available solvers are *Newmark β* and HHT (*Hilber-Hughes-Taylor*). Moreover, two other solvers *TWOSTEP*, *SEMI-IMPLICIT* are available
- The response in the **frequency domain** (frequency response) is determined by the modal or direct solution of the linear complex system

of equations for each excitation frequency requested. In general, a fully populated equation system has to be solved. In modal space an iterative solver makes the analysis much faster.

- Additional *static mode shapes* can be taken into account.
- The *modal/direct random response* analysis for FS coupled analyses is also available.

By specification of a node and/or element set (see page 69) the run time for modal superposition methods can be drastically reduced when the results are only determined for the set members. The **reduced** response results in enormous disk space savings.

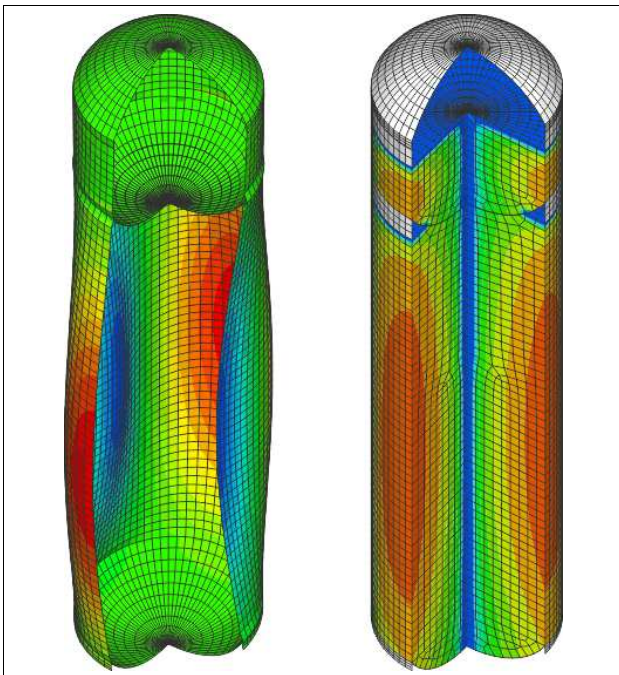


Figure 186: A coupled vibration mode shape of a fluid-filled space tank with corresponding pressure mode

The following damping features are available:

- For the structure:
 - viscous damping elements,
 - material or structural damping for elements, also frequency dependent, if required,
 - global structural damping,
 - Rayleigh damping.
- For the fluid:
 - boundary absorption,
 - volumetric absorption (also frequency-dependent).
- For the coupled system:
 - modal viscous damping (coupled system).

The excitation is defined by static loading cases modulated by functions of frequency (see page 69). The load definitions may consist of:

- Structural loads as described for the DRA module
- prescribed pressures.

The primary results are:

- displacements,
- pressures,
- velocities, and
- accelerations.

For the drawing of XY plots, these data may be output as a function of time or frequency respectively. Moreover, the following results may be derived:

- reaction forces,
- stresses and stress resultants,
- strain energy,
- kinetic energy,
- specific sound radiation power density,
- sound particle velocity.

PERMAS-NLD – Nonlinear Dynamics

This module provides time integration in structural analysis including nonlinearities:

- Material nonlinearities like creep, nonlinear elasticity, plasticity, and visco-plasticity.
- Nonlinear elements like nonlinear springs or gasket elements as well as control elements.
- Large translational motions of elastic bodies handled by updating MPC conditions (with incompatible meshes).

Geometric nonlinear effects and contact are not yet included.

Time integration is done using the *Newmark* method or the generalized *α -method*. The latter includes numerical damping to stabilize the integration scheme.

Different solution methods like *Newton-Raphson* or modified Newton-Raphson are available. An automatic time stepping technique supports the use of appropriate time steps.

Substructure technique and *dynamic condensation* can be used to reduce the purely elastic parts before the nonlinear dynamic analysis starts.

PERMAS-HBM – Harmonic Balance Method

The *Harmonic Balance Method* (HBM) is a steady state method to calculate solutions to nonlinear differential equations. In PERMAS it is used in the context of a nonlinear frequency response.

A new class of applications is opened with this functionality, such as:

- *Rattle noise*
- *Squeak noise*
- Bolted joints and friction between parts
- Vibro-Impact problems
- Blade-Tip/casing contact problems
- Lumped multi-degree-of-freedom (MDOF) and FE models with local nonlinearities

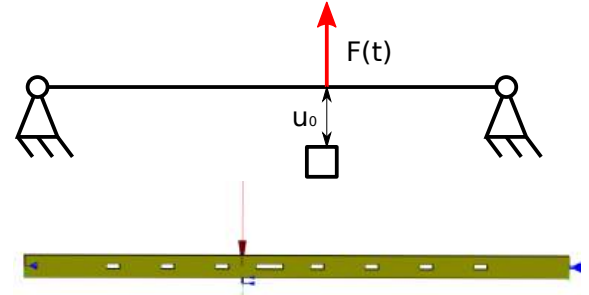
HBM, features

- Many features of direct/modal frequency solutions available.
- Applicable nonlinear elements: nonlinear spring and damper elements (NLSTIFF, NLDAMP), control elements (CONTROL3/6) and a new hysteresis element (HYST3).
- Dynamic contact supported.
- Alternating Frequency-Time (AFT) scheme for efficient calculation of arbitrary nonlinear force for nonlinear elements.
- Optional integrated *Craig-Bampton reduction* of linear part or manual substructuring.
- Arbitrary number of harmonics and sampling size for nonlinear elements.
- *Hysteresis* effects.
- *Path following* capability. See figure 188 for further informations. Please note, that for a certain frequency more than one solution may exist.
- Arbitrary periodic loads.
- Solution may be transformed to the time domain.
- Assessment of stability of HBM solutions (e.g. *Koopmann-Hill* method)
- Secondary results in frequency or time domain (e.g. sound radiation).

- Excellent parallelization.

Example HBM, contact and fatigue

- Beam with contact. The beam is fixed on the left and right side with a barrier at load initiation. It has some weak spots.
- Harmonic loading $F(t) = F_0 \cos(2\pi ft)$



- Elements: 10224 HEXE8, 1 NLSTIFF, 40800 DOF
- Contact modelling with NLSTIFF, initial gap $u_0 = 1$ mm.
- Contact modelled with smoothed penalty contact function, see fig. 187.

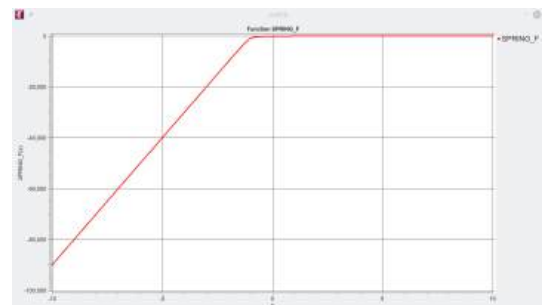


Figure 187: Smooth contact function

Response curve of tip displacement

Comparison between a modal frequency and harmonic balance calculation

k: Fourier series expansion (k=0 constant part; k=1 linear solution). k>1 higher harmonics,
 $\Rightarrow$ Fatigue damage checked for 38 and 46 Hz

Comparison of fatigue damage

The new HBM and LIFE module functionalities may be used together.

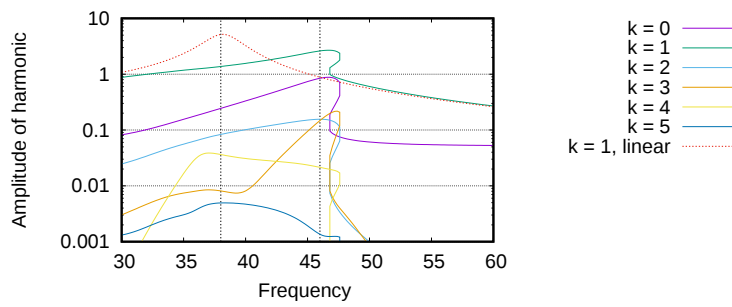
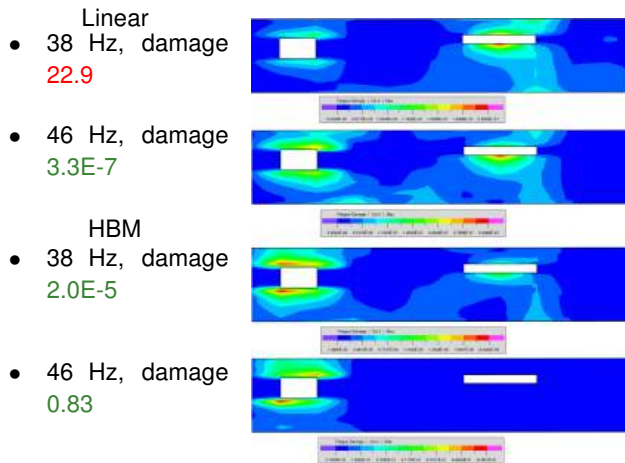


Figure 188: HBM solution



So the linear solution leads to misleading fatigue results.

PERMAS Design-Optimization

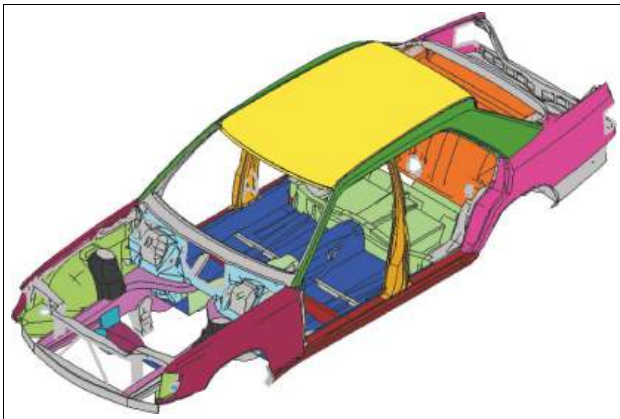


Figure 189: Frequency response optimization of a body-in-white with shape and sizing parameters (see also Fig. 193)

PERMAS-OPT – Design Optimization

Beside the pure FE modeling, PERMAS also allows the definition of a design model and its automatic optimization.

Feature	Options
Opt. Domains	Sizing, shape (parametric and non-parametric), beads, material/laminates
Objective/Constraints	Single objective or pareto opt., various constraints and constraint functions
Optimization solvers	Sensitivity based, finite differences, optimality criteria, global
Solutions	Multiple analyses can be handled
Loads/Steps	Handling of all relevant data
Primary results	XY-data of design variable/constraint histories

Table 3: Overview of optimization capabilities

The following design variables are provided:

- **Sizing:**
 - areas of cross section, inertia moments and general functions between these properties for beam elements,
 - all parameters of standard beam cross sections (see page 67),
 - thicknesses/offsets/nonstructural mass of membrane and shell elements,
 - stiffness and mass of spring elements,
 - mass of mass elements,
 - damping parameter of damping elements,
 - parameters of control elements,
 - convection film coefficients,
 - material parameters.
- **Shape optimization:**
 - node coordinates for shape optimization,
 - use of design elements (see page 67),
 - use of *Shape Basis Vectors* (see page 50),
 - use of incompatible meshes (for *positioning*) without remeshing (see Fig. 111),
 - *bead design* (see Fig. 192). With V21, a complete workflow is provided by PERMAS and Visper, see Fig. 191.
 - Nonparametric *free-form optimization* (see on page 111).
- **Design variable linking**

In each optimization constraints shall limit the value range for design variables as well as for the response quantities like:

- displacements, velocities, accelerations,
- element forces, reaction forces,
- damage,
- stresses, (plastic) strains,
- compliance,
- weight,
- contact gap widths/pressure/forces,
- eigenfrequencies,
- sound radiation power density,
- temperatures,
- heat fluxes,
- Center of gravity,
- Inertia properties,
- Load scale factors in buckling analysis,
- Resulting pressures (sampling only).
- general constraints as combination or arbitrary function of the above mentioned quantities. Such functions include global criteria like max/min, absmax/absmin, or RMS.
- element quality, where the PERMAS element test is mapped to a continuous variable with values between 0. (i.e. perfect element) and 1. (i.e. erroneous element). This design constraint will help to avoid the failure during optimization due to collapsing elements.
- *Design constraint linking* is available as powerful tool to build constraint equations for multiple result values.

The objective function of an optimization may be the

weight or any other specified constraint. An arbitrary number of design constraints may be declared as design objective. The maximum value will be minimized, whereas all others become constraints. For *multi-objective design optimization* a *Pareto* optimization may be performed using a suitable sampling capability.

Dependent nodes are also allowed for shape modifications. This allows the use of incompatible meshes to realize larger modifications without the need to remesh a structure (see example in Fig. 190).

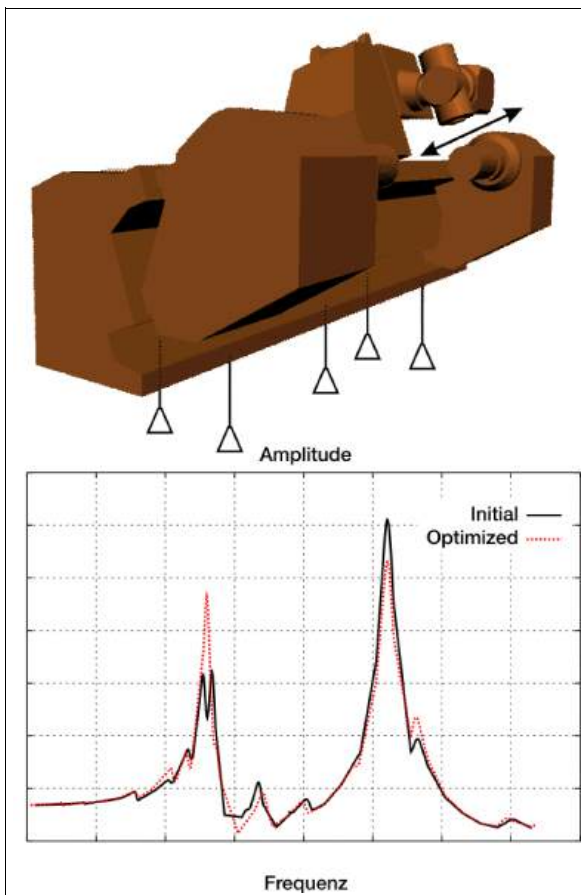


Figure 190: Frequency response optimization for a machine foundation where the position of the base spring elements is optimized which are coupled to the machine structure by interpolating surfaces. Design objective is the lowest displacement of the tool center point.

The optimization itself is performed using one of the following algorithms:

- **CONLIN** (Linear Convex Programming): A simple and robust method using analytical derivatives, only useful with linear analyses.
- **ACP** (*Adapted Convex Programming*): This out-

of-core and parallelized solver is recommended for large optimization tasks, nonlinear behavior, and complex manufacturing conditions.

- **OC** (*Optimality Criteria Method*): Used for free-form optimization tasks (see below page 111).

More algorithms are available with module AOS (see below on page 117).

Optimization is equipped with a general break/restart facility. To this end, a running optimization can be stopped and restart files are prepared. So, the restart can be made at any already performed optimization loop. Before restart, optimization parameters can be modified to influence the convergence behavior of the optimization. The restart uses the restart file to continue the optimization from the already reached status.

The following solvers are available for optimization:

- Linear statics,
- Linear buckling analysis,
- Linear temperature analysis,
- **Inertia relief** (see page 81),
- Eigenvalue analysis,
- Modal frequency response analysis,
- Steady-state heat transfer analysis.

With the aid of module AOS (see page 117) additional solvers are available for Optimization:

- Contact analysis,
- Nonlinear material behavior.
- Complex eigenvalue analysis (including rotating structures)
- Nonlinear temperature analysis

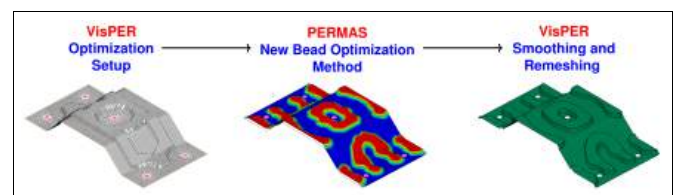


Figure 191: Bead optimization workflow in PERMAS and VisPER V21

For frequency response optimization amplitudes, phases, real, and imaginary values of the above listed results are available for constraint or objective definition. The limits for the constraints can be made dependent on frequency.

The optimization allows taking into account several

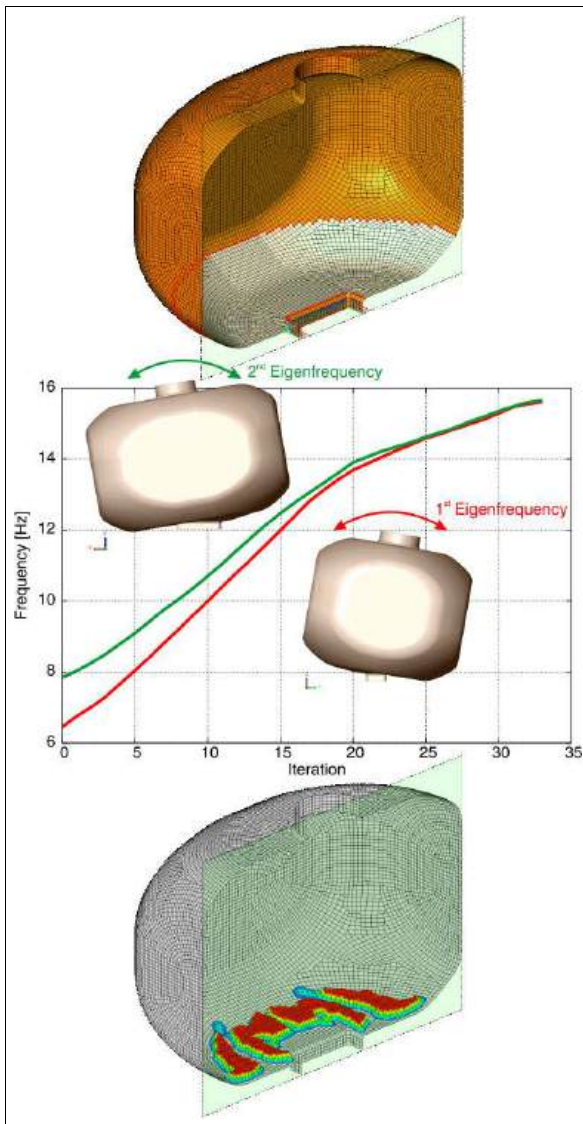


Figure 192: Bead generation for plastic tank

loading cases as well as different boundary conditions using variant analysis (see page 60.) In addition, dynamic mode frequencies can also be optimized, where a mode tracking during the structural changes is performed automatically.

If a small part of a structure is optimized, **substructuring** can be used to reduce run time by separating the design space in the top component. So, the reduction of the unmodified parts has to be done only once.

The results of an optimization are the history of the objective function and an overview on the validity of the design after each iteration. In addition, the values of the design variables and the constraints are available as a function of the iterations performed.

These functions may easily be viewed as XY plots. The export of sensitivities is also possible.

Moreover, element properties may be prepared for result processing (i.e. thickness distribution) and exported for post-processing.

The results of a shape optimization can be exported as displacements for post-processing with the original model or as new model with identical topology and modified coordinates.

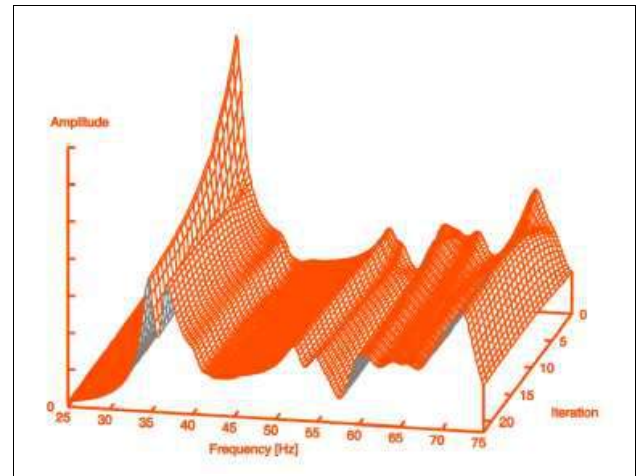


Figure 193: Iteration history of a frequency response optimization

Free-Form Optimization

A *non-parametric optimization* feature has been integrated which allows a *free-form shape optimization* of structures for minimizing stresses by homogenization or for limiting stresses at minimum weight (see Fig. 195). This opens the most easy way to define shape optimization of free-form geometries. The set-up of this optimization is supported by a freeshape wizard in VisPER (see page 50).

Free-form optimization has the following features:

- Possible objectives or constraints are weight, stress (von Mises stress, principal stress), effective plastic strain, and nodal values generated by external tools, if a local change of the part thickness influences the local value of the objective (e.g. safety factors).
- Additional constraints could be stresses outside the design area, displacements or compliance as stiffness constraints or any other constraint as long as (semi-)analytic sensitivities are available.
- Degenerated elements get a special handling in

order not to interrupt the optimization process.

- Midnodes of quadratic elements may be used to define design nodes and design variables.
- Moreover, a surface smoothing functional is integrated in the computation of improved coordinate modifications (see Fig. 194 with non-normal thickness changes).
- Release directions are available for free-form optimization as manufacturing constraint.

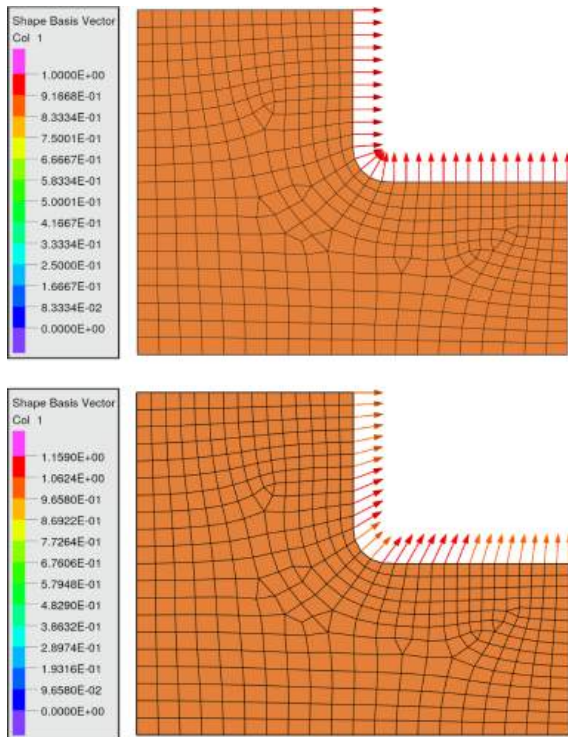


Figure 194: Improved shape changes

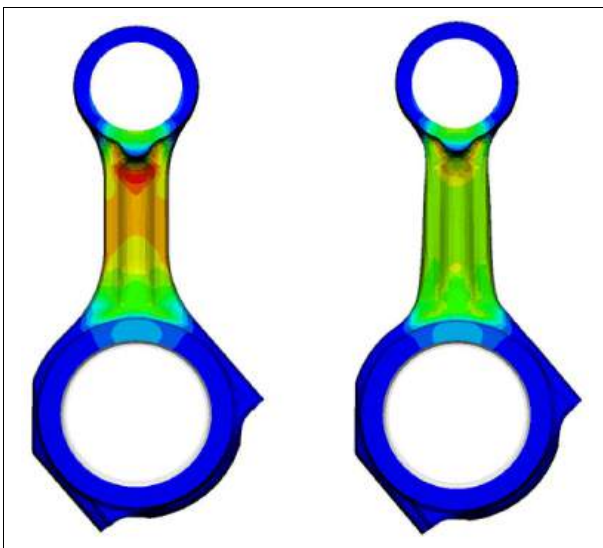


Figure 195: Stress homogenization of conrod

Multi-Modal Optimization

Due to a harmonization of the optimizers now any combination of topology, sizing, and shape optimization can be used simultaneously in one single *multi-modal optimization (MMO)*. To get to this point, the mentioned optimization types are using the same solver. This opens the door to a wider field of optimization tasks (see Fig. 196 and Fig. 197).

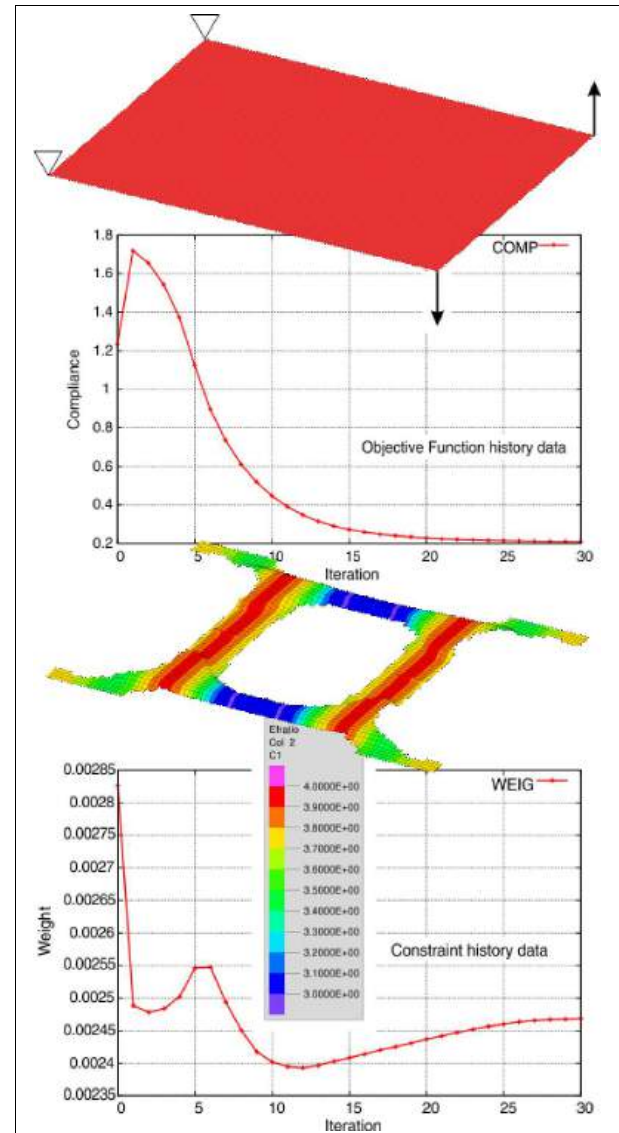


Figure 196: Multi-modal optimization with topology optimization, bead optimization, and free sizing for the sheet thicknesses.

Laminate Optimization

Topology optimization is used to get ply shapes from the optimized thickness distributions (see page 117 and fig. 198).

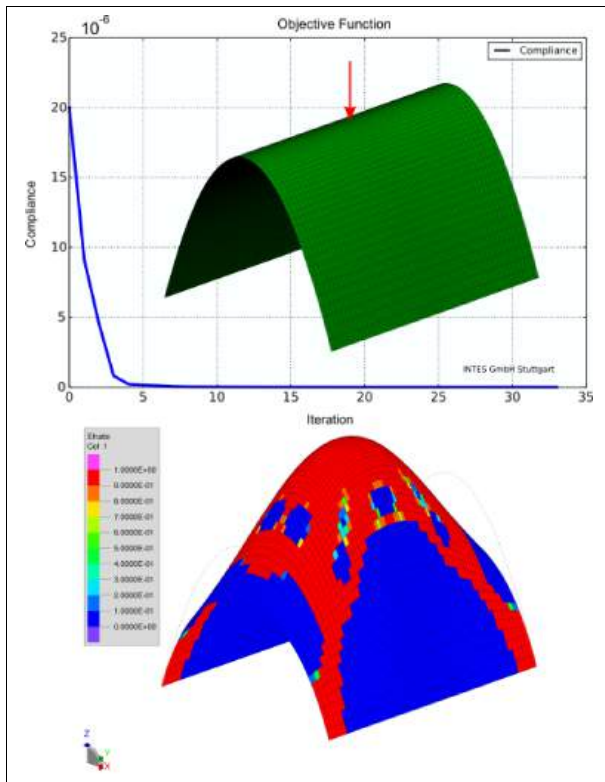


Figure 197: Topology and shape optimization simultaneous solution as multi-modal optimization

Sizing of laminates is then supported, where ply thicknesses and angles can be optimized. *Ply failure criteria* (see page 122) may be used as constraints for the laminate sizing optimization.

Fatigue Optimization

A major and unique new feature in PERMAS Version 21 is **damage-driven optimization** based on results from a PERMAS fatigue analysis. Damage may be defined either as an optimization objective or as a constraint, both locally and globally. The complete workflow is fully integrated into PERMAS (see Fig. 199), providing an efficient (sensitivity based) and user-friendly process that eliminates the need for data transfer between different software tools. At present, damage-based optimization is available for shape and sizing optimization.

Robust Design Optimization

Optimization for a robust design is achieved by additional reliability constraints. Then, the design fulfills all of the above mentioned constraints and it is also reliable regarding uncertain model parameters (see page 37 for more details).

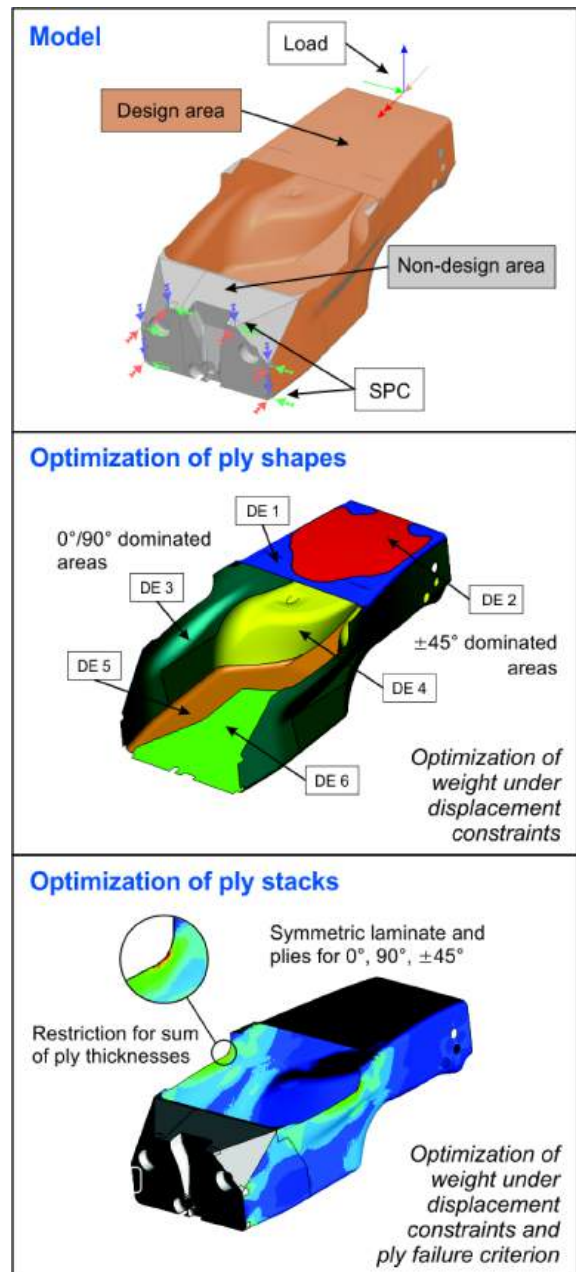


Figure 198: Optimizing laminates for a racing car monocoque with free sizing (see 117) to get a thickness distribution of the layers and with sizing (see page 109) to optimize ply thickness and angle.

Use of External Solvers

In optimization, the use of additional results is required, which are not yet computed in PERMAS directly. So, an *external solver* can be invoked from within the optimization loop (e.g. an MBS solver, see Fig. 200). To achieve that a script has to be written, which invokes the external tool and provides nodal results in a specific PERMAS compliant format.

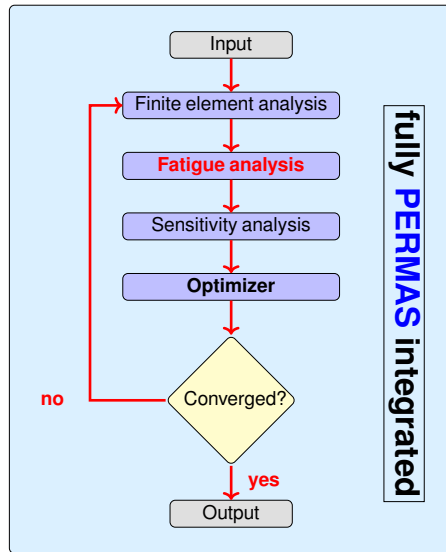


Figure 199: Integrated fatigue optimization workflow

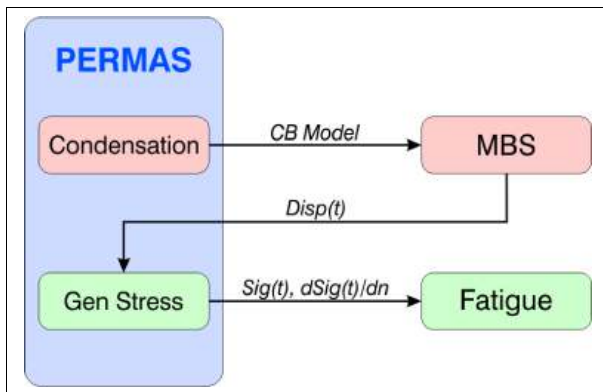


Figure 200: Use of MBS and Fatigue coupled to PERMAS.

PERMAS-TOPO – Layout Optimization

Topology optimization is a method to find in a given limited part of space a finite element structure being optimal relative to a user defined criterion and fulfilling a set of given conditions. For this task a part of the model, the design space, has to be filled with finite elements. Each finite element gets its own design variable, the filling ratio (with values between 0 and 1). It is used for the calculation of a scale factor for the elemental stiffness. If the filling ratio is near zero, so is the stiffness. Then the resp. finite element does not contribute to the mechanical behaviour of the structure and can be neglected. The elements with high filling ratios are the necessary

ones to fulfil the given criteria. It is clear that the layout found by this process consists of a subset of the elements of the design space. The finer you discretize the design space the more detailed the layout result can be, but the higher is the computation effort.

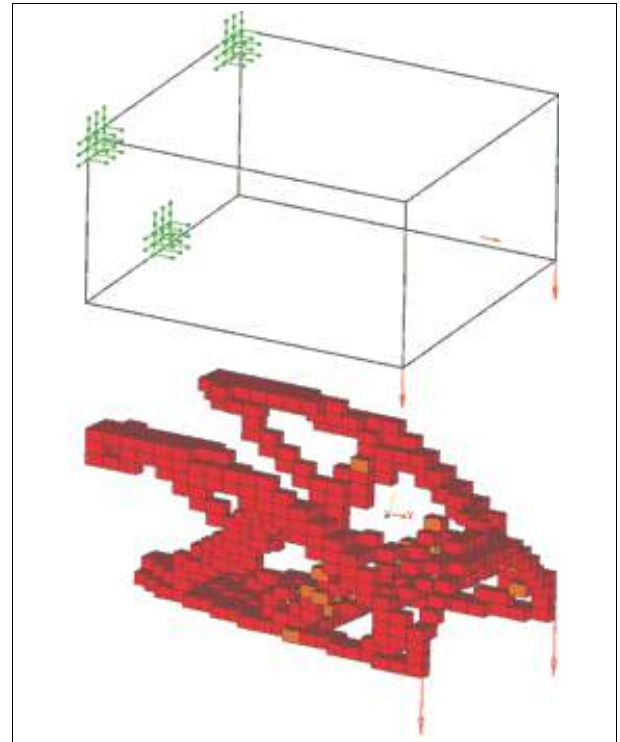


Figure 201: Layout optimization

Design space with boundary conditions and loading (above) and the optimal material distribution (below)

This module supports the concept development stage by providing fully integrated topology optimization strategies:

- Design space specification with variable/fixed parts,
- Provide boundary conditions,
- Provide loads,
- Target definition with remaining volume,
- Additional constraints for the optimization, if any.

For the modeling continua elements like membranes, shells, and solids are used and *substructuring* is supported. Additional modeling parameters are:

- **Fixed/free design elements:**
 - filling ratio per design element,
 - design variable limits,
 - design variable modification limits.
- **Manufacturing constraints:**

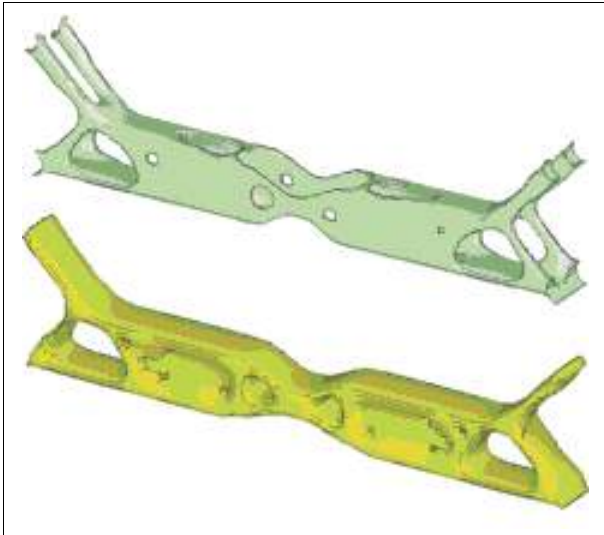


Figure 202: Different results for topology without (above) and with (below) release directions

- **Release Directions:**
For casting special constraints have to be applied in order to get producible parts out of a topology optimization. So, release directions can be specified to control this kind of manufacturing constraint (see Fig. 202). In addition, a minimum residual wall thickness can be specified to obtain a closed structure (like housings). A **fixed mold parting line** can be specified in case of opposing release directions.
- **Symmetry conditions:**
Planar, axial, and cyclic symmetry conditions can be specified to determine the final properties of the layout result.
- **Repetitive structures:** Design element linking is provided to get the same layout for different parts of a structure.
- **Maximum member size:** In combination with release directions a maximum member size option is available to restrict the thickness of remaining structures.
- **Minimum member size:** Minimum member sizes in the remaining structure (i.e. widths and thicknesses) can be controlled by corresponding parameters (so-called checker-board filter).
- **Frozen regions:** Such regions of the design space may not be changed during the optimization process.
- The overhang angle can be designed, which

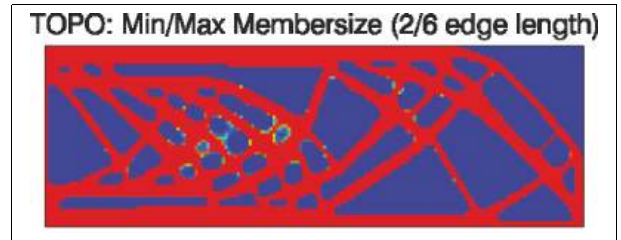


Figure 203: Extended Membersize handling

is important for 3D printing designs (see Fig. 204).

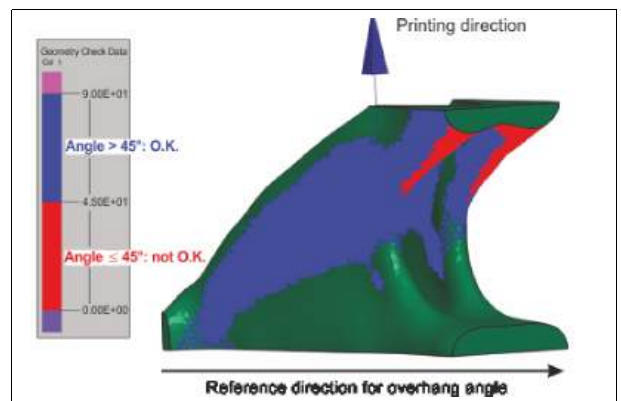


Figure 204: Check of overhang angle
red: critical angle, blue: non-critical, other colors: no value (i.e. >90°).

- **Design constraints and design objective:**

- compliance,
- weight,
- reaction forces,
- eigenfrequency (mode range),
- displacements,
- accelerations, velocities,
- stress (in the non-design space),
- element forces (in the non-design space),
- sound radiation power density (in the non-design space).

Each design constraint can also be used as objective function.

Design constraint functions can be used to build more complex constraints out of the above listed basic constraints.

A general objective function facility can be used to set up an objective function dependent on multiple constraint values (like max/min, abs-max/absmin, RMS).

- **Multi-Modeling**

- several load cases simultaneously with different superposition options,

- different design variants.

With solution options in module AOS (see page 117) maximum member size in topology optimization is exactly formulated.

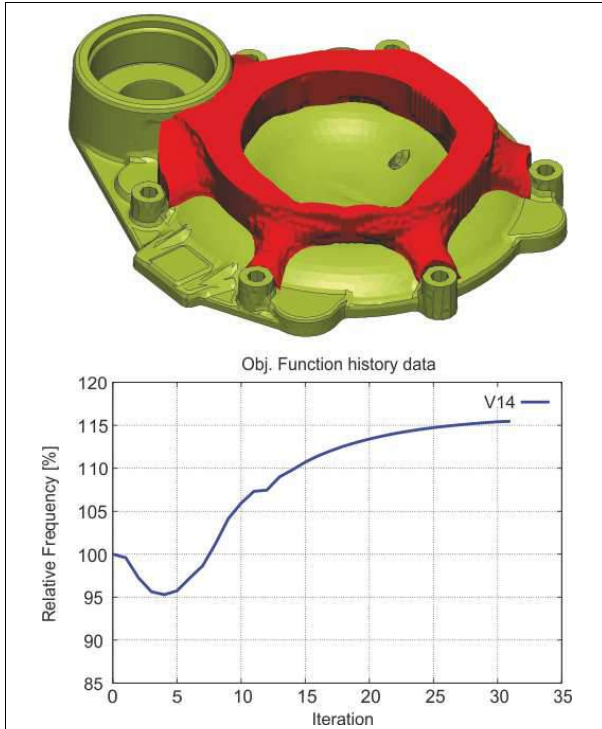


Figure 205: Topology optimization of a gearbox cover to maximize the second eigenfrequency with release direction (by courtesy of ZF Friedrichshafen AG)

A number of analysis options are available for the optimization like

- Linear statics,
- Linear buckling analysis,
- Contact analysis,
- Dynamic mode analysis,
- Modal frequency response analysis.

Because eigenfrequencies and mode shapes change a lot during a topology optimization, strategies for the suppression of local modes are available and of particular importance.

The optimization itself is performed using one of the following algorithms:

- **GCA (Global Convex Approximation)** for eigenfrequencies and a combination of static and dynamic constraints.
- **PD (Primal-Dual Solution)**

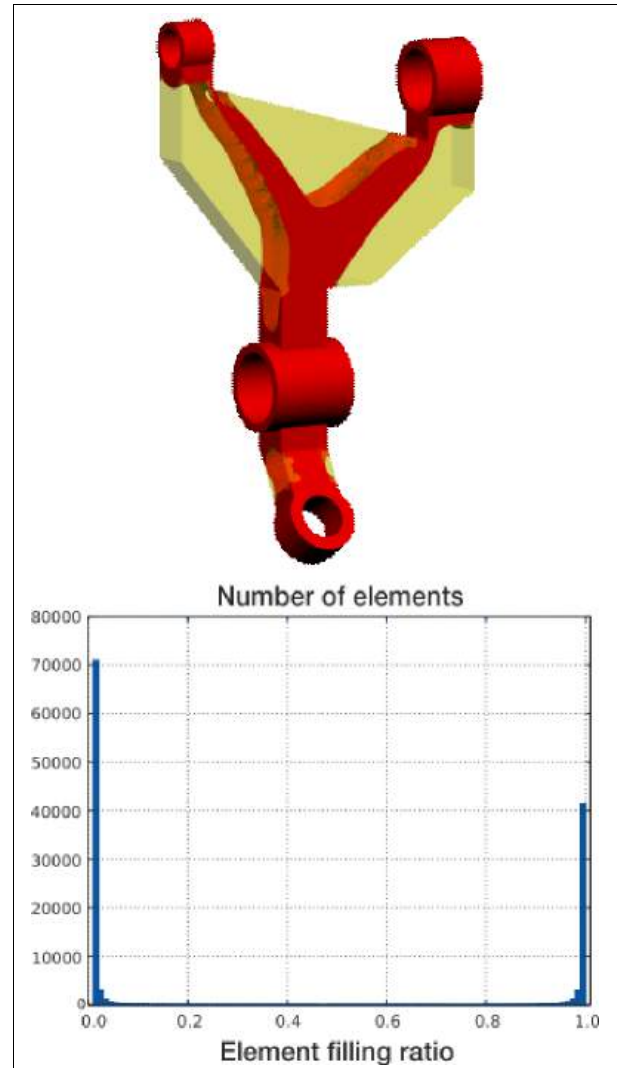


Figure 206: Fully converged topology

The iterations of an optimization can be controlled either by the convergence of the objective function or by a maximum number of iterations.

Topology optimization is able to provide a solution with **clear separation** of filled and void elements. So, elements will show filling ratios near 0 or near 1. This feature avoids misinterpretation of topology optimization results and facilitates the use of the result directly for further analysis and design steps (see Fig. 206).

Beside the history of the objective function, the result of a layout optimization is the element filling ratio. On the basis of the filling ratio, the remaining structure can be visualized in the post-processor easily.

The remaining structure can be further processed

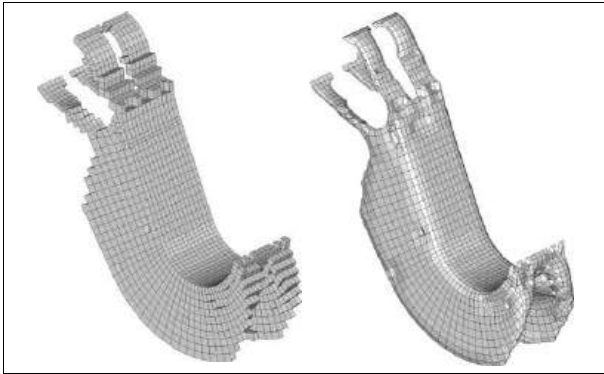


Figure 207: Layout-Optimization of a crane hook
(on the left side the primary result,
on the right side the smoothed surface)

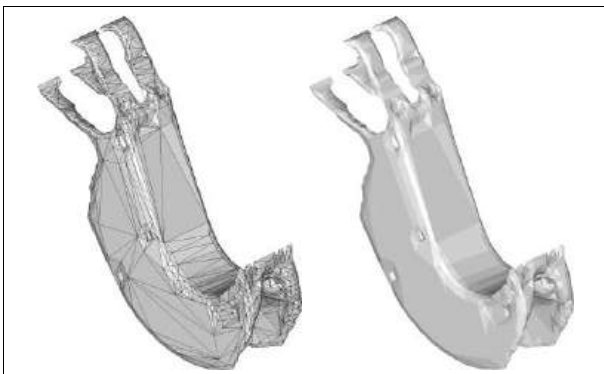


Figure 208: Polygon reduction
(on the left side the triangle mesh,
on the right side graphical representation of the surface)

by one of the following means:

- **Hull generation:**
For an automatically determined or prescribed value of the filling ratio the corresponding surface in space is extracted as mesh out of quadrangles and triangles.
- **Smoothing:**
The hull is smoothed taking into consideration the boundary of parts, loads, and kinematic constraints.
- **Polygon Reduction:**
The mesh is purged of too small triangles and quadrangles in order to achieve a surface description as compact as possible.
- **Export:**
The remaining triangle mesh can be exported as FE mesh for post-processing or as geometry using *STL* format.

Beside the filling ratio as design variable, topology optimization can also handle geometrical properties

of elements (like shell thicknesses). This application is also named *free sizing*.

Laminate Optimization

One application of free sizing is for laminate structures in order to get ply shapes from the optimized thickness distributions. This reflects the fact that for a ply stack under given fiber angles not all plies are needed over the entire structure to bear the loads. The result will specify the element sets which need to have a certain ply of the ply stack. Further optimization steps of laminates are performed by classical sizing (see page 112).

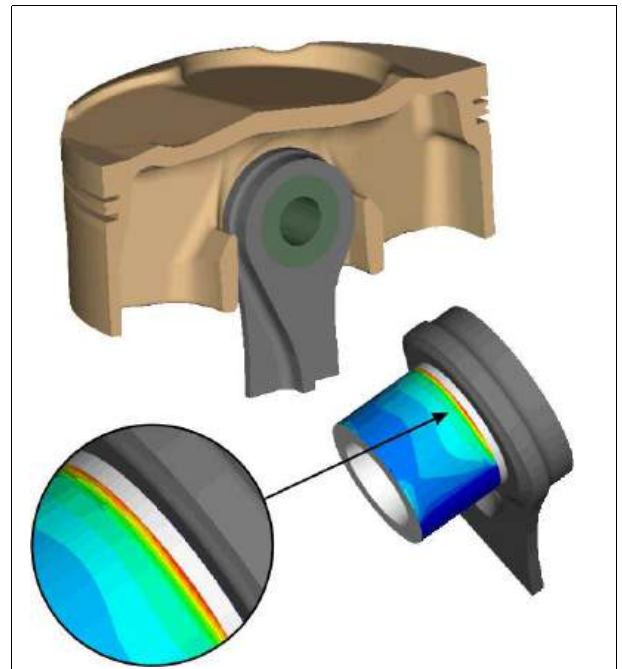


Figure 209: Optimization of contact pressure
between piston and piston pin (Mahle GmbH, Stuttgart).
The figure shows the edge pressure between both parts
(for the optimized result see also next Fig. 210).

PERMAS-AOS – Advanced Optim. Solvers

This module provides additional optimization solvers which essentially extend the range of applications for the integrated optimization in PERMAS. The extensions are as follows:

- By *Trust Region* method based local methods adaptive stepsize control is facilitated. This extends the previous static modification limit cho-

sen by the user.

- Trust region methods keep track of the best point. They reject points, where no improvement is achieved. This extends the previous methods, where any new point is accepted.
- Methods for derivative-free optimization and global optimization are available.

Using these methods, new application fields for optimization are opened, like contact analysis and non-linear material analysis.

The local methods include the following derivative-based methods:

- **SQP** (Sequential Quadratic Programming): This is a damped *Newton* method combined with an *active set strategy* for the optimality equations. It is the best general purpose method (but not necessarily in structural mechanics). Second order information is available by *BFGS* update.
- **SLP** (Sequential Linear Programming): This method uses only linear approximation. Usually, it is slower than SQP due to missing 2nd order information. It is sometimes more robust than other gradient based methods (e.g. in the case of steep gradients).
- **SCP** (Sequential Convex Programming): Usually, best-of-class method for classical optimization problems arising in structural mechanics. Module OPT uses a method which belongs to SCP class of optimization methods.

When derivatives are not available, e.g. in contact problems or nonlinear material behavior, or when the accuracy of computed derivatives is not sufficiently high (like sometimes in frequency response analysis), then derivative-free methods can be applied. The new derivative-free (local) methods comprise the following approaches:

- Derivative-based methods using finite differences (with SQP, SLP, SCP). Functions should be smooth enough and the choice of the finite difference parameter for the interval should not be a problem.
- Derivative-free method **WLIN** (Wedge constraint, LINear approximation). There is no need to choose a finite difference parameter. This method can be used for noisy problems.

When global minima have to be found, local methods are not appropriate any more. For such global

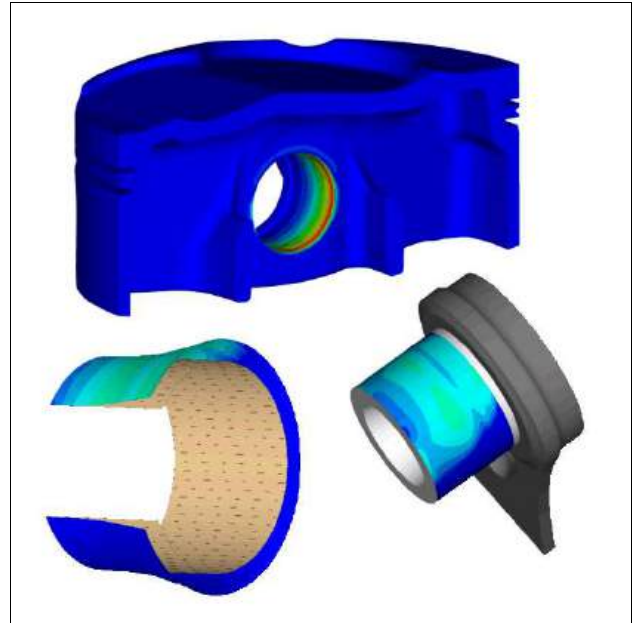


Figure 210: Optimization of contact pressure between piston and piston pin (Mahle GmbH, Stuttgart). The figure shows the optimized geometry of the piston and the reduced edge pressure.

optimization tasks, the following approaches are available:

- By applying the *Multi-Start method* (**MS**) and using random points derivative-based methods can be used to localize minima. This is combined with keeping track of the best point. This approach can be seen as an automatic trial method. A maximum number of loops is used to terminate the analysis.
- Another method is **LDR** (Locally improved variant of the *Dividing Rectangles* (*DiRect*) algorithm). Fig. 211 shows an example for this approach. This method has been generalized to work with constrained problems. It could be improved by solving local subproblems. It generates a sequence of points that is dense in the design space and hence guarantees to approximate the global solution. Because this method is slow and only useful for small models, a suitable model reduction is highly recommended (see substructuring on page 59).

A *Global Design Centering* (**GDC**) option for optimization is available. This option has been developed to search for a domain with maximum stability. A typical example application is brake squeal analysis with stochastic parameters for brake pad material.

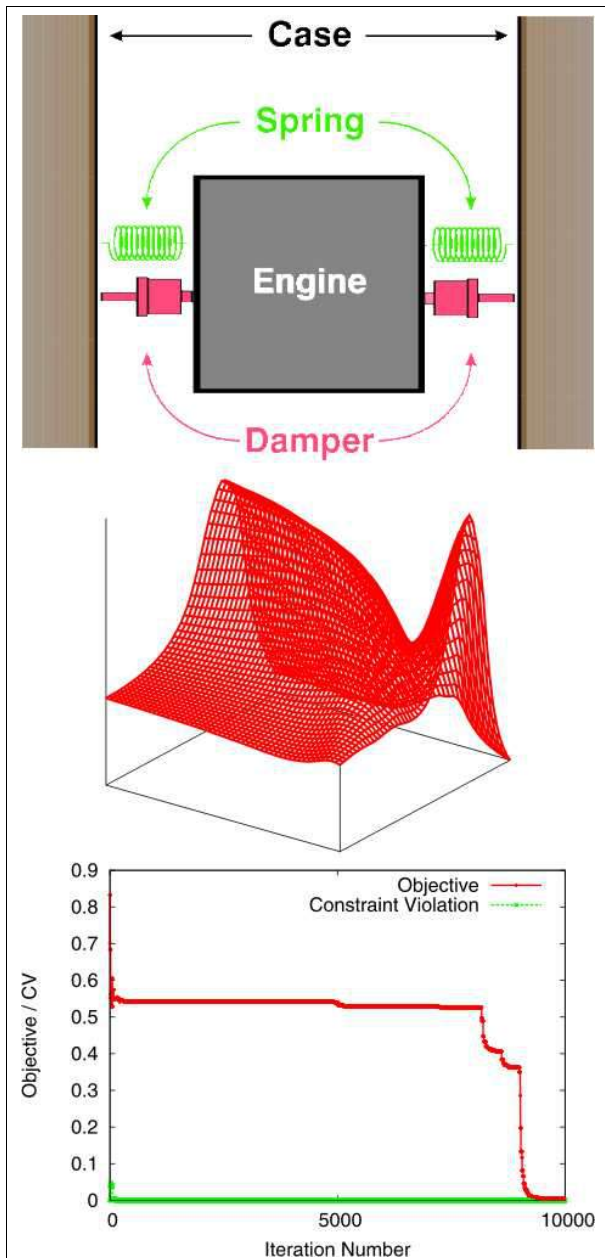


Figure 211: Global optimization

of a spring-damper system with the LDR method. Top: model with 4 design variables, Middle: 2 design variables fixed at the optimum, objective function dependent on the 2 free variables, Bottom: course of objective function and constraint violation during the iterations.

PERMAS-RA – Reliability Analysis

In the classical approach to structural analysis a deterministic model is used to predict the behavior of the design under various loading conditions. The

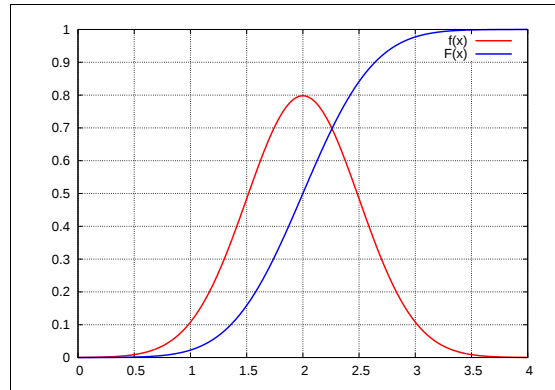


Figure 212: Gaussian Distribution

$f(x)$: Probability density function, $F(X)$: Probability distribution function

results of such calculations are compared to typical limiting constraints such as a maximum stress or deflection under the consideration of safety margins. This is called deterministic approach to the problem of structural safety and the Finite Element Method has become a widespread tool in such procedures.

In contrast to this method, the *stochastic analysis* of a design assumes some properties of a structure or the loads to be uncertain knowing only the characteristics of their probability distributions. The limiting constraints on the design will usually be of the same kind as in the deterministic approach. However, the results from the probabilistic analysis will yield the probability of failure with respect to these constraints and the sensitivity of this probability with respect to the uncertain properties of the model.

This module combines the Finite Element Analysis with the well established COMREL program developed by RCP GmbH, Munich. So, the experience comprised in both software systems could be merged in a single application simplifying the approach to the Stochastic Finite Element Method.

The procedure in reliability analysis comprises the following three steps:

- Definition of uncertain quantities in structural analysis (like geometrical or load parameters) by *basic variables* with an assigned distribution function.
- Definition of *limit state functions* (or *failure functions*) related to result quantities of a structural analysis.
- Calculation of the *probability of failure* for each limit state function.

The following quantities can be used as **basic variables**:

- Design parameters (like geometrical data or coordinates),
- Load factors,
- Material parameters,
- Parameters of the limit state functions,
- Parameters of other basic variables,

More than 20 different types of distribution functions are available to describe the basic uncertain variables.

The stochastic analysis performs an assessment of the failure parameters for the following analysis types:

- Linear static analysis,
- Contact analysis,
- Dynamic eigenvalue analysis,
- Frequency response analysis,
- Complex eigenvalue analysis (including rotating structures).

For this purpose, a number of methods are available:

- Efficient sensitivity based methods as First/Second Order Reliability Methods (*FORM/SORM*),
- *Response surface methods*,
- *Monte Carlo* simulation using adaptive sampling,
- Crude Monte Carlo simulation.

The reliability analysis allows to take into account several loading cases as well as different boundary conditions using different failure functions.

- The definition of **Failure functions** is made using
 - General functions,
 - Dependent on
 - * results (displacements, stresses, etc.),
 - * basic variables,
 - * constant values.
- The primary **Results** of such an analysis are
 - Probability of failure for each limit state function,
 - Parameter sensitivities of the limit state functions,
 - Result sensitivities for basic variables (elasticities),
 - Selected data of each iteration for Monte

Carlo simulations.

PERMAS Special Modules

PERMAS-LIFE – Basic Fatigue Lifetime Analysis

The LIFE module provides functionalities to compute high-cycle *fatigue*. It is the result of a cooperation between “German Steinbeis Transfer Center Traffic Engineering (STZ)” and INTES .

A fully integrated fatigue analysis in PERMAS is provided. See figure 214. Instead of an external tool for fatigue, that typically requests a huge amount of stress data, the integrated solution is able to reduce the output data to the final fatigue results.

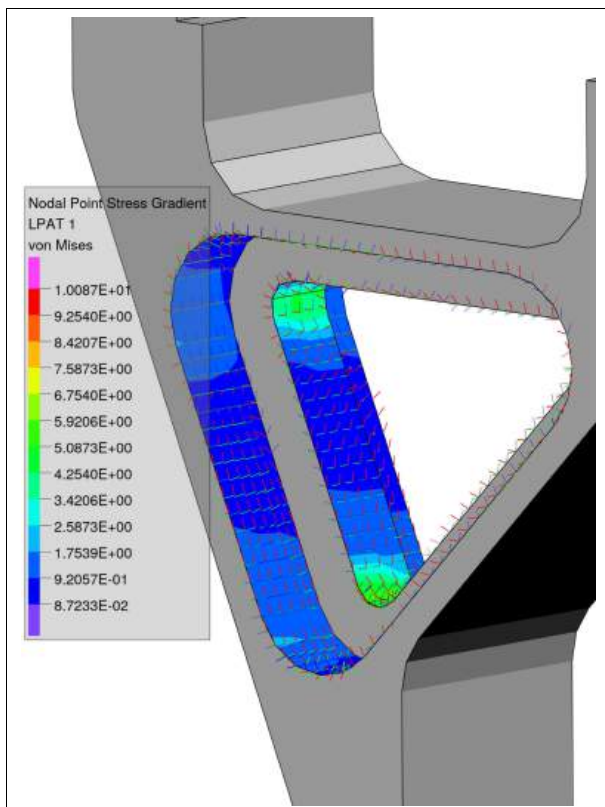


Figure 213: Nodal point stress gradients

Some basic characteristics are:

- Stress based high-cycle fatigue (long term loading, typically >1000 cycles)
- *Multi-axial stress* state on structural surfaces (due to FKM guideline and more) is the basis
- Support of volume elements
- A fatigue analysis may follow a (nonlinear) static analysis as well as transient (timehistory) analysis (modal and direct methods). Load collec-

tives, or time functions for load cycles may be used.

- Classical nodal point or SPR stresses (see page 68) and precise gradients may be used
- **High performance** implementation (time, memory and I/O optimal, with excellent parallel scaling)
- **No stress export/import**, no external program necessary
- GUI based definition and result evaluation in VisPER
- Potential for future enhanced extra analyses: e.g. combining optimization and fatigue (as objective function and/or constraint)

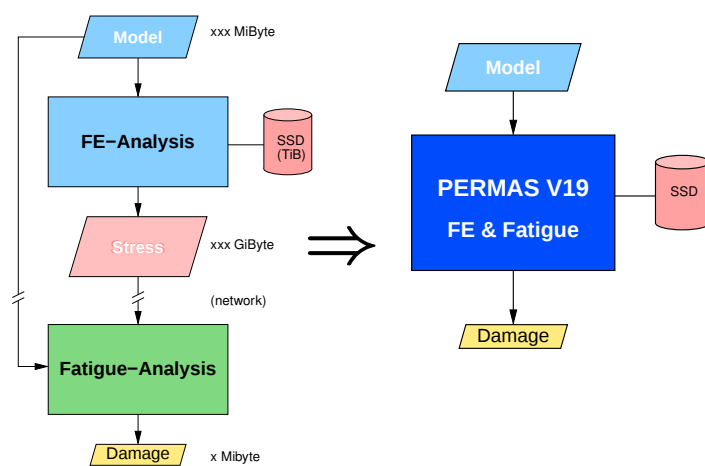


Figure 214: External and integrated fatigue workflow

Example Truck Spring Bracket

The life prediction simulation of a truck spring bracket for 16 torture tracks gives an impression on the handling advantages as well as on runtime gains. Instead of separate fatigue runs for each of the 16 tracks and a final fatigue run on the load collective, everything may be done within a single PERMAS simulation.

7 900 000	Nodes
5 700 000	Elements
82 000	Fatigue nodes
515	Modes
174 000	Time steps
in 16	Torture tracks

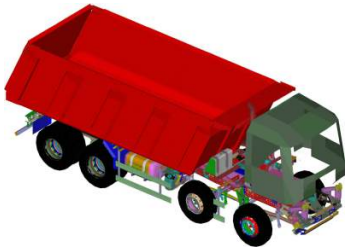


Figure 215: Similar Truck Model

The resources are as follows: **Total job** (Eigenmodes, modal timehistory response+ fatigue calculation):

Elapsed time	[h:mm:ss]	2:58:53
Parallel scaling	[pEff]	11.05
DMS-Disk	[TiB]	1.29

Share due to fatigue calculation (174 000 time steps):

Elapsed time	[mm:ss]	14:09
Parallel scaling	[pEff]	20.19

System: 2*Intel Xeon 8260L, 28 cores, 564 GiB memory

PERMAS V18 needs **5.5 h** with stress export and **4.25 TiB** database disk space plus **≈1.5-12h** for the external fatigue analysis. With PERMAS-LIFE a speedup in overall process time of approximately factor 2-6 is gained for this example. Disk space is scaled down to 30% during the overall process.

PERMAS-LA – Laminate Analysis

The laminate analysis serves for the modeling and analysis of multi-layered *fibre-reinforced composites*. Therefore, PERMAS provides for 3-node and 4-node shell elements.

Usually the laminate is defined ply by ply. Each ply refers to a homogeneous and anisotropic material (frequently defined as orthotropic material) with a given thickness and a ply direction denoting the deviation angle from the material reference system. This definition is supported by VisPER. The homog-

enization then takes place in PERMAS following the *Extended Laminat Theory (ELT)*.

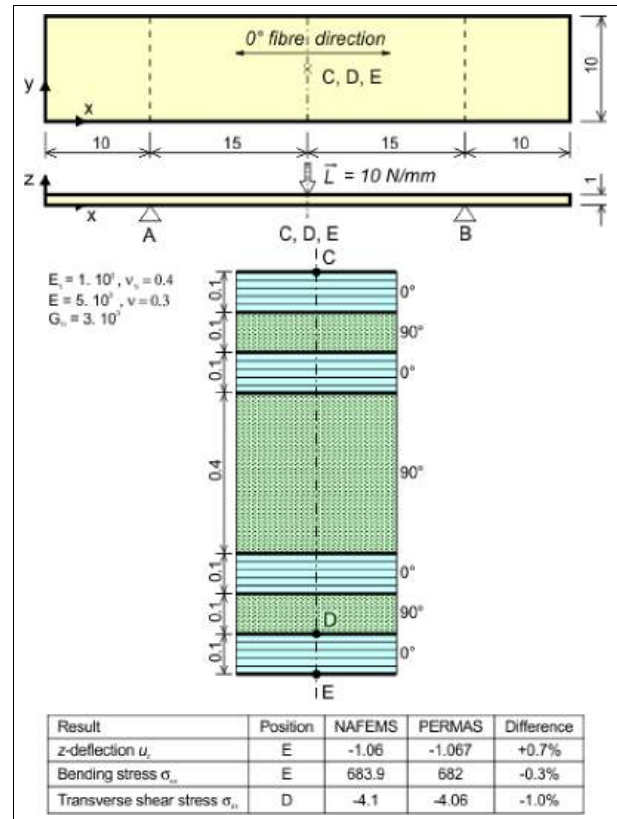


Figure 216: Laminate analysis of a *NAFEMS* composite benchmark.

In rare cases the homogenization may have been done outside of PERMAS. In this case the A, B, D and G matrices may be input directly.

Beside linear analysis (like in Fig. 216), also nonlinear analysis (like in Fig. 217) is supported.

Primary analysis results are element forces, from which the layer stresses and strains are derived. These values are available in the middle of each layer. If one wants to get the stress values between two layers, then a layer with zero thickness may be introduced.

For the further processing of the stresses, a number of *ply failure criteria* are available. Beside *Hoffman* and *Tsai-Wu* criteria, such criteria can also be specified by user subroutines.

PERMAS-WLDS – Refined Weldspot Model

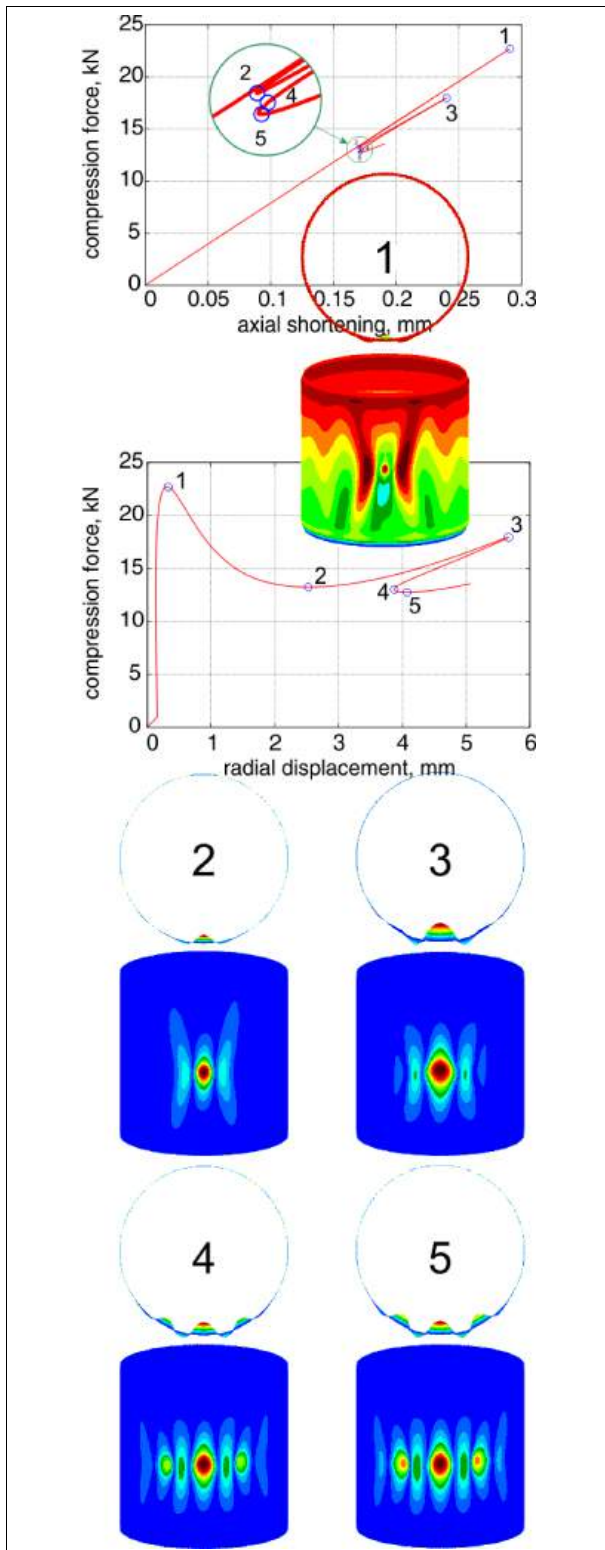


Figure 217: Nonlinear buckling with post-buckling behavior for a laminated cylinder shell

The modeling of weldspot connections is described on page 63. This modeling gives a good representation of the global stiffness.

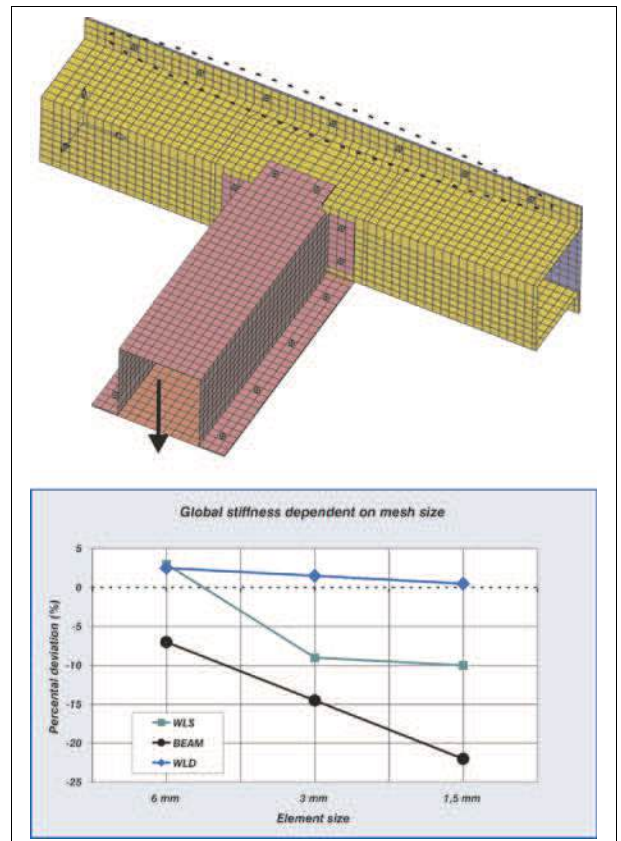


Figure 218: Quality of the refined weldspot model

But along a weld line the weldspot forces can vary very much due to e.g. discretization effects between the incompatibly meshed flanges.

This module offers a refined weldspot model which is characterized by very low variations of the weldspot forces and by an improved stiffness representation (see Fig. 218). Among others, this is achieved by an internal calibration of the stiffness using a volume model.

PERMAS-AERO – Generalized Inertia Relief

Inertia relief performs a quasi-static analysis of a free body, i.e. of an unconstrained structure (see page 81). The structure is assumed to undergo a rigid body acceleration, resulting in a static equilibrium between inertia forces and applied loading. The displacements are calculated as absolute displacement (see Fig. 219).

Like in case of aerodynamic forces, a so-called *load correction matrix* can be specified in addition, which will be taken into account during generalized inertia relief analysis.

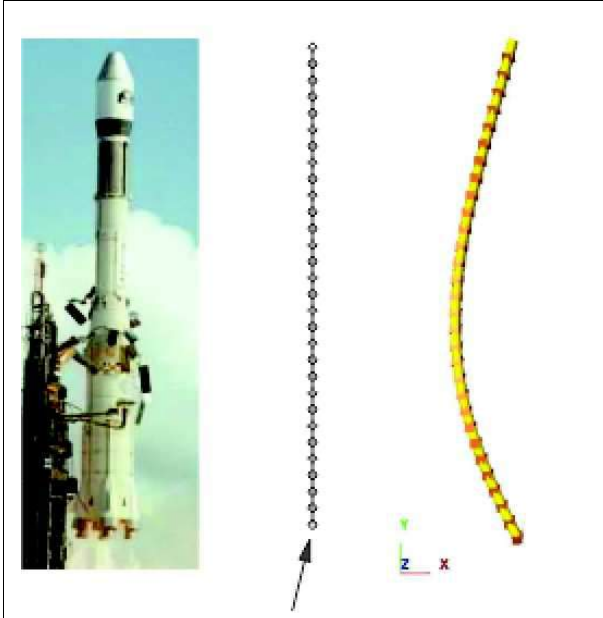


Figure 219: Displacements of flying rocket under thrust and dead weight.

PERMAS-EMS – Electro- and Magneto-Statics

This module allows for steady-state electromagnetic analysis. Magnetic analyses may be based on a previously performed calculation of the steady-state current distribution. The analysis uses a scalar potential for the electric field and a vector potential for the magnetic field.

Various load types are supported.

The determination and handling of singularities is analogous to a static analysis (see page 65).

Heat induced by an electrical field can be used for a subsequent thermal analysis (see page 94). From that thermal stresses can be derived performing a subsequent static analysis (see page 81).

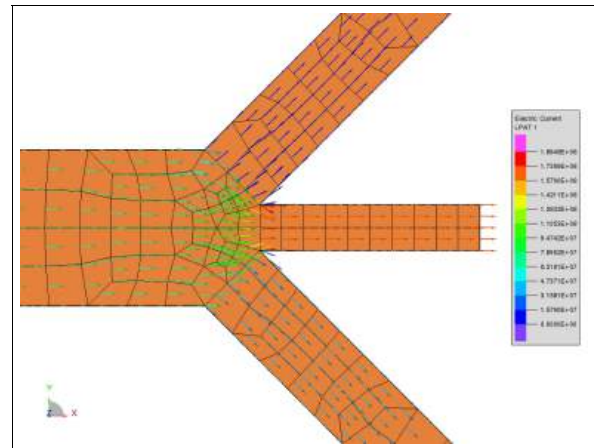


Figure 220: Electric current in an electric junction

Forces induced by a magnetic field can be used in a subsequent static analysis (see page 81).

PERMAS-EMD – Electrodynamics

A solution of *Maxwell's* equations is available for different problem cases:

- eddy currents
- induction problems (see Fig. 221)
- resonant cavities
- wave propagation
- general electrodynamics

All kinds of loading may be applied in dynamics (like

prescribed potential). The specification is made by static loading cases and appropriate time functions like in structural dynamics (see page 101).

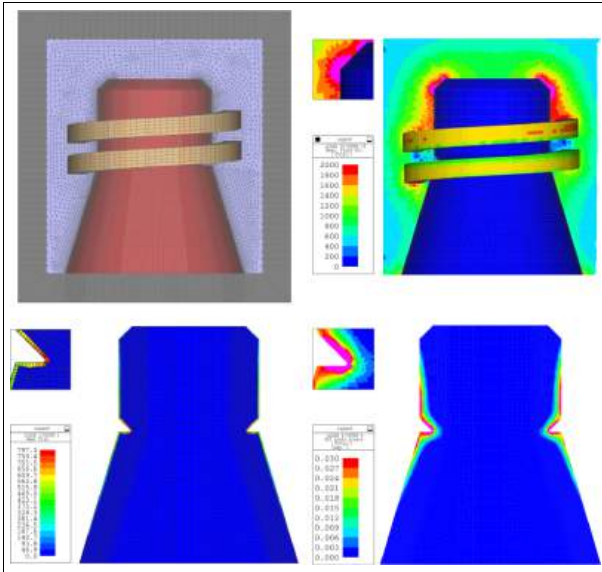


Figure 221: Example for electrodynamic induction, model and magnetic field strength (top), induced heat flux and plastic strains (below).

The coupling between electrodynamics, heat transfer, and static analysis with material nonlinearities is achieved by using only one model, which contains the same solid elements for the different analysis domains. In this way, no interfacing and no mapping is needed between the different analyses.

PERMAS-XPU – GPU accelerator

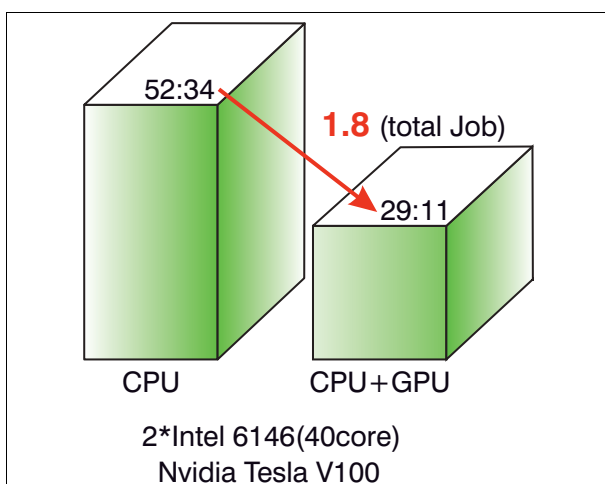


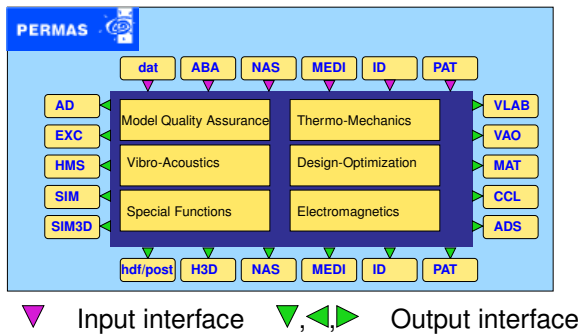
Figure 222: XPU benefit for a large contact analysis

The module XPU supports Nvidia GPU (Tesla K20/K40/K80 [c/m/x] and P100/V100 [c/m/x] with CUDA Version 7/8/9, where P100/V100 needs CUDA 8/9 for full performance) for acceleration of compute-bound analyses by a seamless integration in PERMAS parallelization concept. The functionality is offered together with the S64 versions of PERMAS (see the list of PERMAS operating modes on page 137), since an efficient use of the GPU requires a sufficient core memory of the host machine (≥ 32 GB). The acceleration will be most beneficial for compute-intensive analyses, such as eigenvalue analysis with a high number of modes, a direct fluid-structure coupled response analysis, or large normal contact problems. See fig. 222

Interfaces

Overview

PERMAS is designed as an open software system to integrate other tools/processes.



PERMAS-MEDI – MEDINA Door

This interface directly reads the model from the MEDINA data bus (.bif) and writes the results back to the data bus (.bof).

All MEDINA elements and almost all PERMAS *MPCs* are translated (see page 64). Beside Components, different Situations with constraint and load variants may be specified within MEDINA (see page 60).

It is a very special feature in MEDINA that PERMAS element tests have been integrated for element validation during pre-processing. So, if a model is checked in MEDINA, it will likely pass the PERMAS tests.

The part handling by incompatible line/surface coupling is supported.

The interface supports the following analysis types:

- linear and nonlinear statics,
- contact analysis,
- dynamic mode analysis,
- thermal analysis,
- fluid-structure acoustics (basic modeling and post-processing),
- electromagnetics (basic modeling and post-processing).

The interface is continuously adapted to new versions of MEDINA.

PERMAS-PAT – PATRAN Door

This interface reads the PATRAN database with the model and writes the results onto PATRAN result files.

The modeling is made using custom-made PERMAS preferences. Own solver menus support standard PERMAS solutions from within PATRAN, which may be adapted by the user himself. The following model parts are translated:

- all common elements,
- all kinds of loads incl. inertia loads,
- all PERMAS kinematic boundary conditions,
- sets,
- specifications of substructures,
- variant definitions.

The interface supports the following analysis types:

- linear and nonlinear statics
- contact analysis
- dynamic mode analysis
- thermal analysis
- fluid-structure acoustics

Even for other types of degrees of freedom like electric potential the model may be prepared within PATRAN.

PERMAS-ID – I-DEAS Door

This interface reads the model from an I-DEAS Universal File:

- all common element types
- axisymmetric models
- all kind of loading incl. inertia loads
- many linear kinematic constraints like 'rigid element' and 'coupled dofs'
- laminate material
- sets
- specifications of substructures
- variant definitions

The interface supports the following analysis types:

- linear and nonlinear statics
- contact analysis
- dynamic mode analysis
- thermal analysis

Even for acoustic and electromagnetic analyses the models can be prepared in I-DEAS.

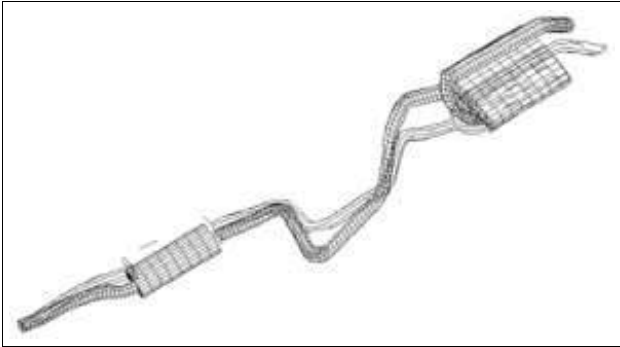


Figure 223: Mode Shape of an Exhaust System

This interface does also have the possibility to read dynamic results from the Universal File. By this feature, the interface became a gateway to *Experimental Modal Analysis (EMA)* (see also page 38).

PERMAS-AD – ADAMS Interface

A post-processing interface for the export of model topology and mass-normalized vibration mode shapes. In addition, generalized stiffnesses and masses are exported and, if needed, also static mode shapes.

The joint orthogonalization of static and dynamic modes can also be very efficiently performed within PERMAS (see module DEV, page 97).

In addition, the export of statically or dynamically condensed models to ADAMS is possible. There, the automatic *substructuring* with MLDR may be used, too (see page 100).

The export is made in the format of ADAMS Version 2007 r1 and up.

PERMAS-EXCI – EXCITE Interface

Post-processing interface for export of model topology, mass-normalized vibration mode shapes and condensed matrices. This interface was developed for EXCITE Version 2013-2 with ExbAPI_V2.0.

A special function is available, which after a vibration analysis of a reduced flexible part may be used to modify mass and stiffness matrices to shift eigenfrequencies according to user specification. The modified data objects are intended to be exported to EXCITE. By this feature, what-if studies are supported to find the optimal properties of the reduced flexible part without modifying the model beforehand.

A filtered backtransformation matrix can be exported for further use after EXCITE execution.

The export can be made in zipped format (by *gzip*) in order to reduce the size of exchanged data in ASCII format.

The Excite binary format *exb* and inertia invariants is also supported.

Response results from EXCITE can be imported in PERMAS to perform further calculations (see Fig. 200).

PERMAS-SIM – SIMPACK Interface

Post-processing interface for export of model topology, mass-normalized vibration mode shapes and condensed matrices, supporting SIMPACK Version 9.

- Based on substructure model.
- *Guyan's* reduction (*static condensation*) by PERMAS.
- Optional also with *dynamic condensation*.
- Or - very interesting - with automatic substructuring using MLDR (see page 100).
- Output of stiffness/mass/etc. on top component level.
- Also export of geometrical stiffness matrices.
- For visualization in SIMPACK the complete model of the uncondensed structure can be exported.

PERMAS-HMS – MotionSolve Interface

Post-processing interface for export of model topology, mass-normalized vibration mode shapes and condensed matrices. This interface was developed for HyperWorks Version 11.

PERMAS-H3D – HYPERVIEW Interface

Post-processing interface for export of model topology and results to HYPERVIEW (from Version 8 onwards). There are formats for HYPERVIEW Version 8 and 11 available.

Supported results are from statics, contact analysis, nonlinear statics, dynamics, acoustics, and heat transfer. The format does not support xy data.

PERMAS-VAO – VAO Interface

Post-processing interface for export of model topology, mass-normalized vibration mode shapes and damping matrices for displacement and pressure degrees of freedom to VAO.

PERMAS-VLAB – Virtual.Lab Interface

Post-processing interface for export of model topology and results to Virtual.Lab:

- Displacements, velocities, accelerations,
- Rigid body mode shapes,
- Assembled loads,
- Reaction forces,
- Stresses,
- Strain energy,
- Kinetic energy,
- Sound radiation power.

The joint orthogonalization of dynamic eigenvectors and static mode shapes can be performed very efficiently in PERMAS (see module DEV, page 97).

PERMAS-ADS – ADSTEFAN Interface

Pre-processing interface for import of temperature fields from a cast analysis using ADSTEFAN. The interface maps the temperature fields to the current FE mesh. A subsequent linear or nonlinear static analysis will be performed to get the stresses during the cooling process. This interface was developed for ADSTEFAN Version 2015.

PERMAS-MAT – MATLAB Interface

A post-processing interface for the export of model topology and matrices.

Often MATLAB is used for the design of controllers. In case of linear controllers the relevant parameters can directly be used in a PERMAS model when control elements are applied.

Moreover, all controllers (in particular nonlinear controllers) can be used in a dynamic transient analysis by providing an appropriate PERMAS user function, which is linked to the software as C or FORTRAN subroutine.

PERMAS-NAS – NASTRAN Door

The NASTRAN Door reads and checks model files compatible to NASTRAN and translates these input files directly into internal PERMAS data structures. The **main capabilities** of the NASTRAN-Door are:

- All Bulk Data formats are supported: Small Field, Large Field and Free Field (with all possibilities for card generation).
- Executive and Case Control sections are translated as well as all global usable statements.
- PERMAS Components and Situations are built according to the Bulk Data model and the selections made by physical Case Control requests.
- Take-over of NASTRAN identifiers, e.g. element-, node- and set-IDs survive the interfacing process and will be taken as PERMAS identifiers.
- Automatic label generation with labels compatible to those generated by NASTRAN.
- Fast Bulk Data sorting with machine independent sort sequence (i.e. equivalent sorting on ASCII and EBCDIC computers).
- NASTRAN-like echo of input statements.
- Extensive error tests are performed:
 - All public NASTRAN statements are recognized and analyzed lexically.
 - All supported statements are completely checked for wrong, missing or contradictory arguments.
- Additional features:
 - Import and export of matrix models in *OP4* format.

- Support of incompatible mesh coupling via: *BGSET*, *BGADD*, *BSURFS*, *BSURF*.
- Calculated results may be referenced under Subcase- or Load-ID.
- Free Field comments within Fix Field statements.
- The INCLUDE statement supports multiple file levels.

A remarkable specialty of the NASTRAN-Door is the ability to utilize the given **control data** as well.

- Control input such as solution-ID, output requests and method selections are converted into task control structures, equivalent to those made by explicit UCI input (User Control Interface).

This so-called **NASTRAN Task** may be started by one special UCI statement.

To execute a NASTRAN-compatible run, only 4 PERMAS UCI-commands are necessary (see figure). Using this simple UCI input, NASTRAN decks can be executed without the necessity of any additional input.

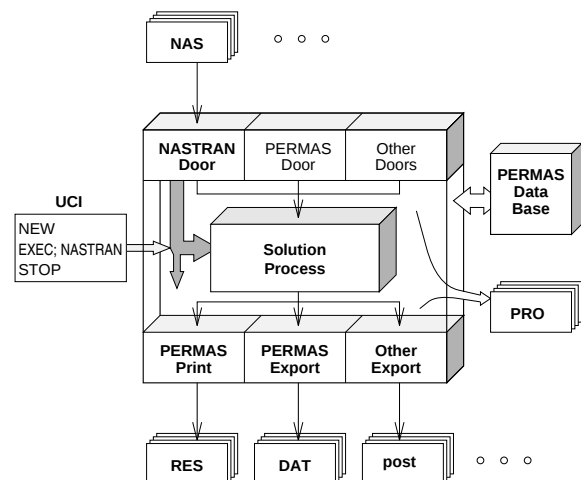


Figure 224: UCI trigger for NASTRAN-Task

Additional features of this NASTRAN Task are:

- Automatic re-translation of its internal task definitions into plain UCI commands echoed to the user.
- The user can choose between the automatic solution control (read from the given NASTRAN model file) and an explicit UCI control.
- The user can extend the NASTRAN Task by additional UCI statements.
- NASTRAN subcases may be selected and ex-

executed individually, leaving all remaining sub-cases untouched.

- Actually the following **solutions** are supported:
 - Linear Static,
 - Normal Modes Analysis,
 - Direct Frequency Response,
 - Direct Transient Response,
 - Modal Frequency Response and
 - Modal Transient Response.
- Supported parameters, which affect the control flow, are also evaluated.

Finally the application of the NASTRAN Task is not restricted to pure NASTRAN-Door input. Even if model additions were made by other input Doors (e.g. DAT file input) – the user is free to execute the NASTRAN control data, automatically.

The export of limited results to certain OUTPUT2 formats. The model data comprise elements, nodes, and local coordinate systems. Supported results are displacements, mode shapes, rigid body mode shapes, and reaction forces.

PERMAS-ABA – ABAQUS-Door



Figure 225: Abadour translation report

The ABAQUS-Door reads and checks model files compatible to ABAQUS and translates these input files directly into internal PERMAS data structures.

This direct interface between two solvers has the advantage that physical meaning of the Abaqus commands can be interpreted during translation to take the adequate equivalent in PERMAS. The user can customize the translation process by his individual door configurations.

The door supports a large number of model and

loading data and provides important information and thematic tables to assure the quality of the translation and analysis process, e.g. an overview on the step history (see Fig. 226).

Step History Overview					
Permas		Abaqus			
Situation [UCI command]	LPAT	Step	Type	Analysis	Supported Step Key
STEP 1_4 [STATIC]	1002	1	General	STATIC	CONTACT INTERFERENCE
	5001,5002	2	General	STATIC	BOUNDARY/CLOAD
	5001,5002	3	General	STATIC	BOUNDARY/CLOAD
	1				CLOAD
	2				DLOAD
	3	4	General	STATIC	TEMPERATURE

Figure 226: Overview on step history

The following analysis procedures are supported:

- linear and non-linear static analysis,
- Steady-state heat transfer analysis,
- Eigenvalue analysis,
- Modal frequency response analysis.

Following modeling features are supported:

- Mass, beam, truss, spring, damper, 2D, 3D, gasket and connector elements,
- Local systems and transformations,
- Various coupling conditions,
- Pretension conditions,
- Anisotropic and temperature-dependent material data,
- Gasket properties and material data,
- Many specifications for contact modeling,
- Heat convection and radiation,
- Static loads and boundary conditions, also centrifugal and inertia loads,
- Complete load history description of supported analysis procedures,
- Dynamic load definitions with real and imaginary parts,
- Support of substructure generation procedure,
- Support of part instances.

For immediate visualization and inspection, VisPER can be used to read ABAQUS input file. In case of missing or not translated modeling features, VisPER may be also be used to complete the PERMAS model.

PERMAS-CCL – MpCCI Coupling

The integration of the coupling library *MpCCI* allows

the coupling of PERMAS with CFD software systems as described in MpCCI section on page 76.

PERMAS-SIM3D – SIMDRIVE 3D Interface

Post-processing interface for export of model topology, mass-normalized vibration mode shapes and condensed matrices, supporting SIMDRIVE 3D and import of MBS results for further processing.

- FEM, MBS, NVH analysis with PERMAS and SIMDRIVE 3D
- Data exchange via single PERMAS HDF file
- Extra data for NVH evaluation:
Mesh and Nodal points (subset of all nodes),
Mode displacements, Modal nodal point stress

More Interfaces

Beside the above described interfaces a number of additional interfaces for PERMAS are available, which have been developed by other software companies to couple their software to PERMAS. The following list only contains those interfaces which became known to the editor before this Product Description was published. So, the list may be not complete or even not correct. In any case, these interfaces are not part of the PERMAS product and their developers have to be contacted to get more information about the contained functionality.

- **Animator4 (GNS, www.gns-mbh.com)**: Post-processing of PERMAS results is done via MEDINA format.
- **ANSA (BETA CAE Systems, www.beta-cae.gr)**: ANSA supports the PERMAS input format for the model and the PERMAS binary (hdf) and ASCII (post) results. Also MEDINA format may be used (.bif, .bof) for the result evaluation with the μ ETA post-processor.
- **Digmat (e-Xstream, www.e-xstream.com)**: This interface generates PERMAS material data input for injection molded parts.
- **EnSight (CEI, www.ensight.com)**: This post-processor takes the results directly from PERMAS files.
- **Evaluator (GNS, www.gns-mbh.com)**: This report generator takes the results directly from PERMAS files.
- **FE-Fatigue (nCode, www.ncode.com)**: The data transfer is possible using MEDINA formats.
- **FEMFAT (MAGNA POWERTRAIN, www.femfat.com)**: The connection between PERMAS and FEMFAT is possible on the basis of the MEDINA export in PERMAS, because FEMFAT also supports this format.
- **HyperMesh (Altair, www.altair.com)**: This interface supports the PERMAS input format. It complements the PERMAS module H3D (see page 129) which exports the results for post-processing with HyperView.
- **Jupiter (Technostar, www.e-technostar.com)**: Jupiter supports the PERMAS input format for the model.
- **MAGMALink (MAGMA, www.magmasoft.de)**: This software is a module for the transfer of casting simulation results out of MAGMASOFT to FE meshes for stress and durability analysis.
- **MAMBA (MAGNA POWERTRAIN)** is a specialized tool to assess fatigue of jointed structures.
- **Material data base MARLIS (M-Base, www.m-base.de)**: This material data base contains material data of steel sheets and is capable to issue the material properties as PERMAS material description.
- **Meshparts** is a pre-/postprocessor with special focus on tooling machines.
- **MeshWorks (DEP, www.depusa.com)** For FEA mesh generation, available within the PERMAS flexible license system. MeshWorks supports PERMAS data input and process automation via VisPER python macros.
- **modeFRONTIER (ESTECO, www.esteco.com)** For simulation process automation and optimization.
- **Novus Nexus (www.novusnexus.com)** For FEA mesh generation. Novus Nexus supports PERMAS data input, upon request.
- **Siemens NX (Siemens PLM Software, www.siemens.com/NX)**: FE models generated with NX can be exported to PERMAS using an interface developed by Dr. Binde Ingenieure (see www.drbinde.de/index.php/en/leistungen-menu/produkte/178-permas). PERMAS results can be post-processed in NX after using an interface tool provided by Siemens PLM Software and developed by VKI

(see

www.vki.com/2013/Products/VdmTools.html).

- **SimLab (Altair, www.altair.com)**: This pre-processor provides an interface in PERMAS input format.

All mentioned names of products and companies belong to their holder. The use does not imply that such names are free for general use.

PERMAS for Education



PERMAS4EDU is the ideal tool to become acquainted with the software or to be used in academic coursework on the Finite Element method. You can get to know the outstanding properties of PERMAS and the wide range of applications with PERMAS4EDU. The model editor VisPER is also included. PERMAS4EDU thus covers the entire process chain from the FE mesh to evaluation.

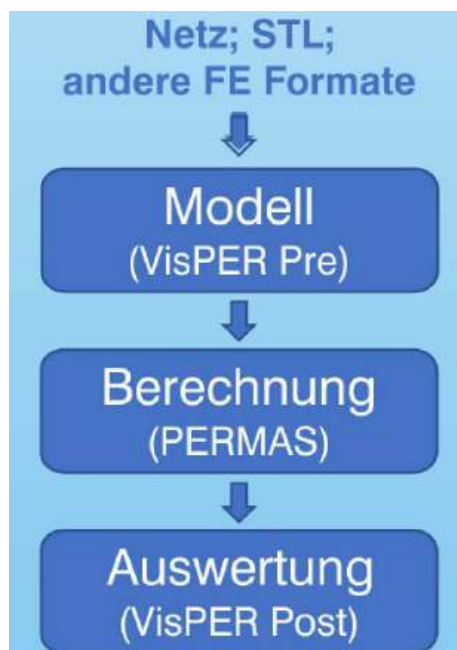


Figure 227: Analysis Workflow

PERMAS4EDU can be obtained through registration at: www.intes.de/edu

- It includes PERMAS, VisPER, and PERMAS Tools
- FEM Process from mesh to evaluation
- Free-of-charge for education and further training, no commercial use permitted

- Limited model sizes e.g. 16,000 nodes, 32,000 elements, and 65,000 degrees of freedom
- Interfaces: VisPER, ANSA, ABAQUS, NASTRAN, MEDINA, and HyperWorks
- Manuals included with >200 application examples and tutorials
- User Forum at INTES for exchange with other users and with a collection of application examples and tips
- Available on Windows and Linux
- A comprehensive collection of examples may be found at: [github repository](#)

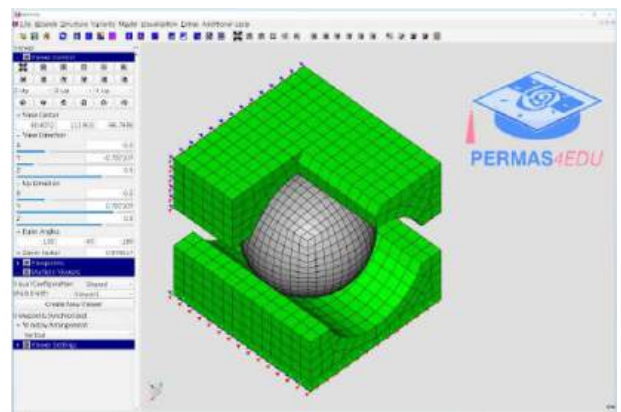


Figure 228: Contact model

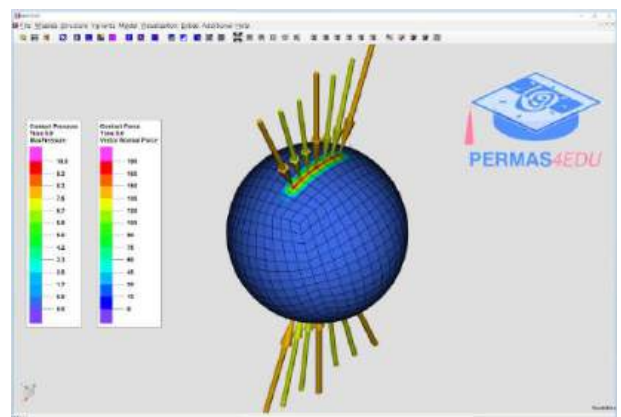


Figure 229: Contact pressures and forces

Installation and beyond

Supported Hardware Platforms

For PERMAS, the following platforms are supported:

Architecture	Operating system
PC x86(_64)	LINUX Debian 11/12, LINUX Ubuntu 20.04/22.04/24.04, LINUX RedHat EL8/EL9, LINUX SuSE 15SP5/15SP6 and compatibles Nvidia GPU (see page 125)
PC x86_64	Windows 10/11 64 bit

For VisPER, the following platforms are supported:

Architecture	Operating system
PC x86(_64)	LINUX glibc ≥ 2.28 , NVIDIA graphics
PC x86_64	Windows 10/11 64 bit

The supported platforms and the related operating systems are always subject to change due to ongoing development activities and new computers on the market. Therefore, any specific case has to be inquired using the contact address on the last page of this document.

The functions described in this document are usually available on all platforms. Nevertheless, some exceptions are certainly possible (i.e. with *parallelization*, CFD coupling, interfaces to third-party products using program libraries (like H3D, VLAB), etc.). Therefore, it is recommended for any specific case to make an inquiry using the contact address on the last page of this document.

PERMAS fully supports *64 Bit architecture* of modern processors. So, the following execution modes are available:

- 64 bit operating system:
 - D64: Double precision floating point operations on 32 Bit machine words with a memory usage of about 7 to 8 GB.
 - S64: Single precision floating point operations on 64 Bit machine words with a practically unlimited memory usage (recommended minimum is 16 GB).

Licensing

The following license types are available:

- **Nodelock license:** The execution of the software is provided for one single computing node. This license type is only available as special function of a floating license.
- **Floating license:** The execution of the software is provided for a computer network, where a fixed number of potential executions can be started on different computing nodes.
- **Flexible license** provides a capability to use floating license tokens.

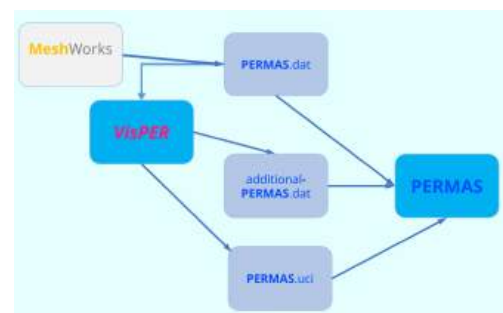


Figure 230: Simulation workflow with MeshWorks

For both nodelock and floating licenses, the *license manager* is responsible for all bookkeeping of license information. The following server architectures are supported:

- **Single server:** The server is acting independent from other license servers in the network. Each server has its own license equipment.
- **Multi server:** Three servers are needed, and two servers have to be available at least. Each server has the total license equipment.

For the management of the license server a WEB interface is available.

Maintenance and Porting

PERMAS is regularly maintained and improved. Within a continual improvement process the actual software version is the best one. On a daily basis numerous software tests are performed and their results are verified.

All incoming problem reports are administrated by a special management system and forwarded to

the responsible engineer. Each resulting correction leads to a unique version number of the software. If a problem is already solved at INTES and there is no workaround for the user, the actual and improved version of the software will be delivered.

Every month a Technical Newsletter is issued, which reports on software corrections, their reasons, and possible workarounds. There, also frequently asked questions are listed with important problem solutions. The Technical Newsletter can be accessed in a reserved section of the INTES homepage (see next Section). The Technical Newsletter can be subscribed by the users, who will receive it automatically via e-mail.

Usually in a 2 years period a larger development step leads to a new major version of the software, which is shipped to all customers having a rental or maintenance contract. Then, also a new version of the user manuals is delivered. After registration, customers may download the new software versions directly.

When new versions of the pertinent pre- and post-processor become available, an adapted version of PERMAS is available in most cases without delay. As soon as changes of the compatibility become known, the users will be notified.

- PERMAS Documentation Templates can be used to organize the workflow and standard PERMAS analysis procedures.
 - Administrative information regarding PERMAS licenses, i.e. on how to change the hardware platform for PERMAS.
-

User Support

PERMAS users have access to the following information sources:

- documentation,
- training,
- support platform on INTES internet homepage,
- hotline services via phone and e-mail.

In particular, the support platform on INTES homepage contains up-to-date information on PERMAS and is an increasing source for useful details:

- The Technical Newsletter contains all available information on software problems, known workarounds, and corrections.
- There is a PERMAS User and Support Forum for the exchange of information between INTES and PERMAS users as well as between all PERMAS users.
- Downloads.

Additional Tools

To support the application of PERMAS, INTES offers some additional tools within the PERMAS Tools which are only available under Linux and Windows operating systems. They comprise:

- **PERMAS control center:** This is a graphical user interface to handle simulation workflows. E.g. to submit a PERMAS or VisPER job and to set the most common job parameters, or to include further tools to the chain. It also supports data handling and job control (see Fig. 231), in detail:
 - Opens and edit an existing PERMAS control file (.uci) or creates a new one.
 - View PERMAS protocol and result files (.pro/.res).
 - Archive Job activates the pack_permas_job script to build an archive containing the complete control and input files of the current control file (.uci).
 - Extracts complete example jobs from PERMAS Examples Manual for trial and inspection.

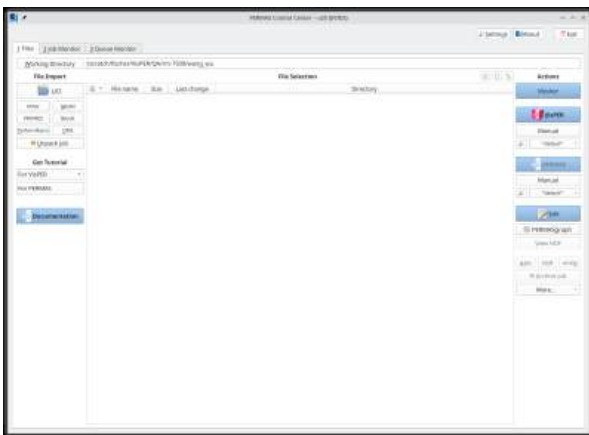


Figure 231: PERMAS Control Center

- **pyINTES:** The INTES tools are completely written in *python*. With pyINTES a standalone python system which contains all required modules to run the INTES scripts is provided. Currently it is using Miniconda2 from Continuum Analytics.
- **PERMASgraph:** A graphical user interface to plot XY data from PERMAS, MEDINA, and PATRAN formats. PERMASgraph supports the following file formats:

- PERMAS native format (.hdf, .post(.gz)),
- in MEDINA native format (.bif, .bof),
- in Patran native format,
- in CSV format (as exported to PERMAS to spreadsheet format; .csv).

The diagrams can be saved in different file formats, such as:

- Portable Network Graphics (.png),
- Encapsulated PostScript (.eps),
- PostScript (.ps),
- Portable Documents Format (.pdf),
- Microsoft Excel File Format (.xls).

The diagrams can also be printed on a local printer.

Additional tools are available for data manipulation, such as:

- Addition/Subtraction of curves,
- Fourier transformation.

A further submenu provides special evaluation tools, such as:

- Analyzer for surface-to-surface coupling,
- Analyzer of potential and kinetic energy of fluid-structure coupled eigenmodes,
- Analyzer of modal participation factors,
- Analyzer for strain and kinetic energy (see example in Fig. 174).

- the integration of external management tools (like Open Grid Engine) in PERMAS for optimum throughput of all available computers with a minimum of effort for administration. This facilitates job distribution and management, particularly in case of parallel applications with PERMAS.
- the fast and comfortable INTES-EMACS text editor incl. online help, documentation for different file formats, and email service. Among others, this tool comprises the following features:
 - Identical edit environment on almost any of the hardware platforms supported by PERMAS.
 - Utilization of function keys and numeric keypad instead of control- and escape-sequences.
 - Context-sensitive help functions and online access to all PERMAS documents.
- the hotline message system OTRS for the reporting of application problems by email.

The PERMAS Tools are available for all PERMAS users. In addition, INTES offers a configuration service and an adaptation to the environment at the

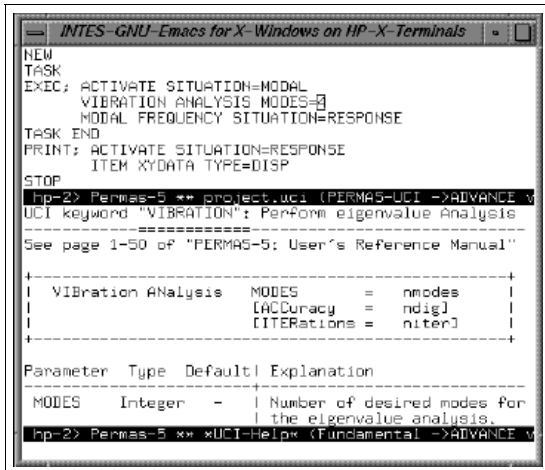


Figure 232: Context-Help for UCI-Files inside Editor

user's site as well as training.

Documentation

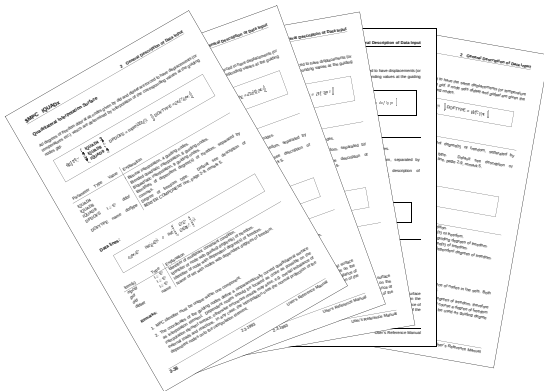


Figure 233: PERMAS documentation

In parallel to the development of PERMAS the documentation is currently updated:

- Apart from the basic documentation
 - *PERMAS Users Reference Manual*,
 - *PERMAS Examples Manual*,
 - *VisPER Users Manual*,
 - *PERMAS Programmers Manual*,
 - *PERMAS Installation Manual*,
 - *PERMAS Tools User Manual*
- the following documents are provided:
- *ABAQUS Door Manual*,
 - *I-DEAS Door Manual*,
 - *MEDINA Door Manual*.
 - *NASTRAN Door Manual*,

– *PATRAN Door Manual*.

- Clearly arranged layout, complete index and cross references using page numbers (even among distinct documents) are a matter-of-course.

For online documentation and help, all manuals are available as PDF files with cross references.

Training

INTES provides training courses for all application fields of PERMAS. Based on a three-day introductory course on linear statics with contact several one- or two-day courses are offered on other topics like contact, heat transfer, dynamics, optimization, etc.

The latest workshop program can be downloaded from the INTES homepage.

Future Developments

In order to provide the FEA user community with a powerful analysis tool continuously, the PERMAS and VisPER development team is intensively working on the extension of already existing functional modules and on the development of new modules and functionalities.

The main lines of future software development are as follows:

- **Integration with CAD/CAE**
This includes e.g. improved and simpler model handling or automatic tools for modeling and result evaluation.
- **Workflows in CAE**
This comprises the analysis of workflows, their automation and improvement. The goal is to make the use of complex workflows as simple as possible.
- **More complex simulations**
This mainly includes functional extensions and coupled analysis features.
- **Higher performance**
Increasing model sizes requires a continuous improvement of the software efficiency. Be-

side algorithmic improvements, this includes the adaptation of the software to new hardware developments.

- **Advanced visualization**

For models and results advanced visualization techniques and tools are developed.

- **Generative design**

to provide solutions ready to produce, e.g. for 3D-printing or to provide simulation based parametric geometry.

Upon your request we are ready to inform you about current development projects and the current planning status for the next major release of PERMAS and VisPER.

Additional Information

For requesting more information and in case of additional questions please contact:

Marketing:	Michael Klein
Phone:	+49 (0)711 784 99 - 13
Fax:	+49 711 784 99 - 10
E-mail:	info@intes.de
WWW:	https://www.intes.de

Address:	INTES GmbH Breitwiesenstr. 28 D-70565 Stuttgart
----------	--

Index

- α -method, 106
- 10% rule, 104
- ABAQUS, 131
- ABDG matrices, 122
- Absolute Error Indicator, 68
- accurate Coulomb friction, 10
- ACP, 110
- active control, 35
- active set strategy, 118
- Adapted Convex Programming, 110
- Add&Replace wizard, 43
- AEI, 68
- anisotropy, 68
- architecture
 - 64 bit, 137
- arclength, 89
- Armstrong-Frederick, 92
- assembled situations, 23, 103
- automatic contact partner search, 9
- basic variable, 119
- Bayliss-Turkel, 105
- bead design, 24, 49, 109
- BeadDesignWizard, 14
- BeadWizard, 14
- beam elements
 - standard cross sections, 67
- BFGS, 118
- BGADD, 130
- BGSET, 130
- bifurcation point, 93
- BIW, 21
- body-in-white, 21
- bolt pretension, 83
- brake squeal analysis, 29
- brake squeal wizard, 44
- BSURF, 130
- BSURFS, 130
- buckling
 - linear, 93
 - nonlinear, 89
- CAD, 57
- Campbell, 31, 99
- car body analysis, 21
- cascade controller, 33
- casting, 90
- Cauchy, 90
- CB condensation, 98
- CCNG analysis, 26, 85, 89
- center of pressure, 85
- Co-Rotational, 89
- COF, 75
- Coleman transformation, 61
- CoMAC, 74
- compensation springs, 83
- complex eigenvalues, 98
- component, 59
- composites, 66, 122
- condensation, 59, 98
 - Craig-Bampton, 98
 - dry, 26
 - dynamic, 26, 59, 98, 100, 129
 - generalized modal, 74, 97
 - Mixed-Boundary Craig-Bampton, 98
 - static, 59, 129
- configuration, 59
- CONLIN, 110
- constraint
 - kinematic, 64
- \$CONSTRAINTS, 13
- contact
 - analysis, 81
 - locking, 26, 27, 29, 85
 - multigrid, 85
 - status files, 26
 - wizard, 54
- contact locking, 26, 27, 29, 85
- contact status files, 26, 84
- contact wizard, 54
- Context menu, 45
- control elements, 35
- convectivity elements, 94
- coordinate system
 - local, 64
- COP, 85
- Coriolis matrix, 31
- Coulomb, 83
- Coulomb's law, 83
- coupling
 - analyses, 76
 - CFD, 76
 - surfaces, 65
 - volumes, 65
- CQC, 104
- Craig-Bampton, 59, 98
 - mixed-boundary, 59, 98
- Craig-Bampton reduction, 107

- critical rotation speed, 32
- cutting forces, 21, 75
- cyclic symmetry, 61
- damping, 102
 - frequency-dependent, 101
 - material, 102
 - matrix, 102
 - modal, 102
 - proportional, 102
 - Rayleigh, 102, 106
 - rotation speed dependent, 32, 101
 - structural, 102
 - viscous, 102
- DAT, 57
- Design by Simulation, 51
- design constraint function, 115
- design constraint linking, 109
- Design wizard, 44
- diffuse noise, 12
- DIRECT
 - FSCOUPLLED, 11, 12
 - HARMONIC, 11
 - TIMEHISTORY, 11
- DiRect, 118
- Dividing Rectangles, 118
- door
 - ABAQUS, 131
 - NASTRAN, 130
- Drucker-Prager, 90
- dynamic condensation, 107
- Dynamic contact for, 11
- earthquake spectral response, 104
- effective masses, 97
- eigenvalues
 - complex, 98
 - real, 97
- electro-statics, 124
- element library, 66
- elements
 - axisymmetric, 67
 - beam, 66
 - contact analysis, 66
 - convectivity, 67, 94
 - discrete, 66
 - flange, 66
 - fluid, 66
 - fluid-structure coupling, 67
 - gasket, 25
 - geometry, 67
 - load, 67, 70
 - membrane, 66
 - plate, 66
 - plot, 67
 - radiation boundary condition (RBC), 67
 - rod, 66
 - scalar, 66
 - semi-infinite, 67
 - shell, 66
 - solid, 66
 - surface waves, 67
- ELT, 122
- EMA, 128
- engine analysis, 24
- Engquist-Majda, 105
- exb, 128
- Experimental Modal Analysis, 75, 128
- experimental modal analysis, 38
- Extended Laminat Theory, 122
- external solver, 113
- failure function, 119
- Fast Fourier transformation, 22
- fatigue, 121
 - analysis, 121
 - high cycle, 121
- Fatigue Optimization, 113
- FFT, 22
- fibre-reinforced composites, 122
- fixed mold parting line, 48, 115
- flexible license, 137
- floating license, 137
- fluid elements, 66
- force-guided contact, 84
- FORM/SORM, 120
- FORMULA, 69
- Fourier, 70, 71
- free sizing, 117
- free-form optimization, 50
- frequency response, 101
- friction
 - Coulomb, 83
- frozen regions, 115
- FSCOUPLLED
 - DIRECT, 11, 12
 - MODAL, 11
- function
 - design constraint, 115
 - failure, 119
 - gap, 82
 - limit state, 119
 - mathematical, 69

- gap function, 82
- gasket elements, 25, 85, 90
- GCA, 116
- GDC, 118
- Generative Design, 51
- Global Convex Approximation, 116
- Global Design Centering, 118
- gnuplot, 72, 75
- GPU, 18
- green, 2
- GUI, 41
- Guyan, 59, 98, 129
- gyroscopic matrix, 32
- gzip, 128
- hardening, 90
 - isotropic, 90
 - kinematic, 90
 - mixed, 90
 - nonlinear kinematic, 92
- HARMONIC
 - DIRECT, 11
 - MODAL, 11
- Harmonic Balance Method, 107
- HDF5, 73
- heat transfer, 94
- Hilber-Hughes-Taylor, 101, 105
- Hoffman, 122
- hole detection, 51
- hull generation, 48, 50, 117
- Hybrid Geometry, 51
- \$HYPERELASTIC, 10
- hyperelastic material, 92
- Hysteresis, 107
- incompatible meshing, 21
- inertia relief, 21, 70, 81, 90, 98, 103, 110
- interfaces, 72
- interpolation region, 64
- Kirchhoff, 66
- Koopmann-Hill, 107
- laminate optimization, 112, 117
- large strain, 92
- Latin Hypercube Sampling, 79
- layout optimization, 114
- LDR, 118
- LHC, 79
- license manager, 137
- limit state function, 37, 119
- line description, 61
- line of action, 85
- Line Search, 89
- linear buckling, 93
- load correction matrix, 124
- MAC, 74, 97
- machine tool, 32
- magneto-statics, 124
- manufacturing constraints, 48, 114
- material
 - cast-iron, 92
 - Drucker-Prager, 90
 - Mohr-Coulomb, 90
 - properties, 68
 - Tresca, 90
 - viscoplastic, 90
 - von Mises, 90
- \$MATRIX, 13
- matrix
 - Coriolis, 31
 - gyroscopic, 32
- matrix models, 22, 73
- maximum member size, 115
- Maxwell, 124
- MBC transformation, 61
- MBCB condensation, 98
- MBS, 21
- MeshWorks, 14
- message tree, 45
- minimum member size, 115
- MLDR, 23, 100
- MMO, 112
- MODAL
 - FSCOUPLLED, 11
 - HARMONIC, 11
 - ROTATING, 12, 98
 - TIMEHISTORY, 11
- modal analysis, 94, 97
- modal participation factors, 93, 103
- modal/direct random response, 106
- Model Information Tree, 45
- model updating, 24
- model verification, 71
- Mohr-Coulomb, 90
- Monte Carlo, 120
- morphing, 50
- MPC, 62 to 64, 81, 127
 - coupling of surfaces, 65
 - coupling of volumes, 65
 - general, 65
 - interpolation region, 64
 - press fit, 65
 - rigid body, 64

- MpCCI, 17, 77, 131
- MS, 118
- Multi-axial stress, 121
- Multi-Blade Coordinate transformation, 61
- Multi-Body Systems, 21
- Multi-Level Dynamic Reduction, 23, 100
- multi-modal optimization, 112
- multi-objective design optimization, 110
- multi-start method, 118

- NAFEMS, 122
- NASTRAN, 130
- Newmark, 101, 105, 106
 - β , 101, 105
- Newton, 118
- Newton-Raphson, 89, 90, 106
- nodelock license, 137
- non-parametric optimization, 111
- Nvidia, 18

- OC, 110
- OP4, 130
- optimality criteria method, 110
- optimization
 - bead, 109
 - contact pressure, 117, 118
 - derivative-free, 118
 - design elements, 67
 - free-form, 109, 111
 - frequency response, 109
 - global, 118
 - layout, 114
 - non-parametric, 111
 - positioning, 109
 - shape, 109
 - size, 109
 - topology, 114
 - trust region, 117
- optimization
 - laminar, 112, 117
- optimization for a robust design, 113
- option COMPLEMENT, 82
- overview
 - dynamic response, 101
 - fluid structure, 105
 - optimization, 109

- parallelization, 17, 18, 96, 137
- Pareto, 110
- part connections, 27
- part coupling, 62
- Path following, 107

- PCGen, 57
- PD, 116
- PERMAS Tools, 75
- PERMAS4EDU, 9
- permascc, 15
- PERMASgraph, 75, 139
- plasticity
 - isotropic, 90
- ply failure criteria, 113, 122
- polygon reduction, 48, 117
- post-processing, 45
- power spectral density, 104
- pre-buckling behavior, 93
- press fit, 82, 86
- Pressfit wizard, 45
- Pretension wizard, 45
- pretension wizard, 54
- Primal-Dual Solution, 116
- probability of failure, 119
- Prony series, 103
- pyINTES, 139
- Python, 42
- python, 76, 139

- quality assurance, 79

- radiation, 95
- random response analysis, 104
- Rattle noise, 107
- Rayleigh, 102, 106
- RBM Assistant, 45, 84
- real eigenvalues, 97
- reduction
 - Guyan, 59, 98, 129
- reference system
 - co-rotating, 31
 - inertial, 32
- Reference Tree, 45
- release directions, 115
- reliability analysis, 119
- response
 - in frequency domain, 101
 - in time domain, 101
 - random, 104, 106
 - spectral, 104
 - steady-state, 101
 - transient, 102
- response spectrum, 104
- response surface methods, 120
- restart, 76

results

- combination, 74
- comparison, 74
- transformation, 74
- xy data, 75

robust design, 37, 113

rolling, 90

ROTATING

- MODAL, 12, 98

rotating systems, 31

rotor dynamics, 31

Rutherford-Boeing, 74

sampling, 79

Sampling wizard, 45

sandwich shells, 66

SBV, 50

SCP, 118

SDM, 80

self contact, 82

SEMIIMPLICIT, 101

SEMIPLICIT, 105

sets

- element sets, 69
- node sets, 69

Shape Basis Vectors, 109

shape basis vectors, 50

shape optimization, 49, 109

Shermann-Morisson-Woodbury, 103

shift

- dyn. eigenvalues, 97

shift method, 93, 97

short fibre material, 90

Simulation Data Management, 80

situation, 60

sizing, 109

sizing optimization, 49

slip-stick, 85

SLP, 118

Smooth Patch Recovery Method, 68

smoothing, 48, 117

SMW, 103

spectral response analysis, 104

spotweld, 63

SPR stresses, 68

SQP, 118

Squeak noise, 107

static mode shapes, 103, 106

steady state response, 22

- rotating systems, 32

steady-state response, 32, 101

STEP, 57

stiffness

- centrifugal, 97
- convective, 97
- frequency-dependent, 101
- geometric, 97
- pressure, 97
- rotation speed dependent, 32, 101
- spotweld, 63

STL, 117

stochastic analysis, 119

Stress Gradients, 68

submodeling, 25, 59, 85, 91

subspace iteration, 97

substructure technique, 65, 79, 91, 107

substructure wizard, 55

substructuring, 59, 60, 111, 114, 128

summation rules, 104

surface description, 61

\$SYSTEM, 13

system kernel, 79

system of units, 75

task scanner, 79

temperature field, 94

temperature mapping, 25

Thomas, 90

TIMEHISTORY

- DIRECT, 11

- MODAL, 11

tooltips, 42

topology optimization, 114

transfer function, 22

transient response, 101

\$TRANSLOAD, 11

Tresca, 90

trust region, 117

Tsai-Wu, 122

TWOSTEP, 101, 105

UCI, 57

user defined material, 90

variant, 60

variant analysis, 60

view factors, 96

visco-elastic material, 103

visco-plasticity, 90

VisPER, 5, 9, 14, 41

volume-shell transition, 65

von Mises, 90

voxel, 53

wizard, [47](#), [49](#), [51](#), [54](#)
 add&replace, [43](#)
 bead design, [49](#)
 brake squeal, [44](#)
 contact, [54](#)
 design, [44](#)
 fluid-structure coupling, [51](#)
 free-form optimization, [50](#)
 pressfit, [45](#)
 pretension, [45](#), [54](#)
 sampling, [45](#)
 shape optimization, [49](#)
 sizing optimization, [49](#)
 substructure, [55](#)
 topology optimization, [47](#)
 bead, [14](#)
 bead design, [14](#)
WLIN, [118](#)

XML, [80](#)

zero force springs, [83](#)
Zienkiewicz, [68](#)

Contact:

Phone: +49 (0)711 784 99 - 0

Fax: +49 711 784 99 - 10

E-mail: info@intes.deWWW: <https://www.intes.de>

Address: INTES GmbH

Breitwiesenstr. 28

D-70565 Stuttgart