

Product Description Version 18

Ingenieurgesellschaft für
technische Software mbH



© **INTES GmbH**, July 2020 (rev. 18.02)

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

The finite element model of a power cutter for dynamic analyses on the frontpage appears by courtesy of ANDREAS STIHL AG & Co. KG in Waiblingen, Germany.

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

Adams is a registered trademark of MSC Software Corporation, Santa Ana, USA.

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-Datenschiene 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 MSC Software Corporation, Santa Ana, USA.

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

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: **<http://www.intes.de>**

Contents

	Page		Page
INTES	5	Standard Beam Cross Sections	61
Company Profile	5	Design Elements for Optimization	62
Services	5	SPR Stresses and Absolute Error Indicator	62
PERMAS	7	Material Properties	62
Overview	7	Sets	63
Introduction to PERMAS	7	Mathematical Functions	63
Benefits of PERMAS	8	Loads	64
What's New in PERMAS Version 18	9	Model Verification	64
What's New in VisPER Version 18	14	Interfaces	65
Universal Features	16	Matrix Models	67
Available VisPER Modules	17	Combination of Results	67
Available PERMAS Modules	17	Transformation of Results	68
Performance Aspects	18	Comparison of Results	68
Parallelization	18	XY Result Data	68
Areas of Application	19	Cutting Forces	69
Reliability	19	System of Units	69
Quality Assurance	19	Restarts	69
Applications	21	Open Software System	69
Car Body Analysis	21	Direct Coupled Analyses	70
Engine Analysis	24	Coupling with CFD	70
Part Connections	26	PERMAS Analysis Modules	71
Brake Squeal Analysis	28	PERMAS-MQA – Model Quality Assurance	71
Rotating Systems	30	PERMAS Package TM /Thermo-Mechanics	73
Analysis of Machine Tools	31	PERMAS-LS – Linear Statics	73
Actively Controlled Systems	35	PERMAS-CA – Contact Analysis	73
Robust Optimum Design	36	PERMAS-CAX – Extended Contact Analysis	77
Experimental Modal Analysis	37	PERMAS-CAMG – Contact Multigrid Analysis	78
VisPER	39	PERMAS-CAU – Contact Geometry Update	79
The VisPER History	39	PERMAS-NLS – Nonlinear Statics	80
VisPER – A Short Introduction	39	PERMAS-NLSMAT – Extended Material Laws	83
VisPER-BAS – Basic Module	40	PERMAS-NLSA – Nonlinear statics advanced	84
VisPER-TOP – Topology Optimization	43	PERMAS-BA – Linear Buckling	84
VisPER-OPT – Design Optimization	44	PERMAS-HT – Heat Transfer	85
VisPER-FS – Fluid-Structure Coupling	46	PERMAS-NLHT – Nonlinear Heat Transfer	86
VisPER-CA – Contact Analysis	49	PERMAS Package VA /Vibro-Acoustics	88
Substructuring	50	PERMAS-DEV – Dynamic Eigenvalues	88
Evaluation of Spotwelds	51	PERMAS-DEVX – Extended Mode Analysis	89
PERMAS Basic Functions	53	PERMAS-MLDR – Eigenmodes with MLDR	90
Substructuring	53	PERMAS-DRA – Dynamic Response	91
Submodeling	53	PERMAS-DRX – Extended Dynamics	94
Variant Analysis	54	PERMAS-FS – Fluid-Structure Acoustics	95
Cyclic Symmetry	55	PERMAS-NLD – Nonlinear Dynamics	97
Surface and Line Description	55	PERMAS Package DO /Design-Optimization	98
Automated Coupling of Parts	56	PERMAS-OPT – Design Optimization	98
Automated Spotweld Modeling	57	PERMAS-TOPO – Layout Optimization	102
Local Coordinate Systems	58	PERMAS-AOS – Advanced Optim. Solvers	106
Kinematic Constraints	58	PERMAS-RA – Reliability Analysis	107
Handling of Singularities	59	PERMAS Special Modules	109
Element Library	60	PERMAS-LA – Laminate Analysis	109
		PERMAS-WLDS – Refined Weldspot Model	110
		PERMAS-GINR – Generalized Inertia Relief	110

PERMAS-EMS – Electro- and Magneto-Statics	111
PERMAS-EMD – Electrodynamics	111
PERMAS-XPU – GPU accelerator	111
Interfaces	113
PERMAS-MEDI – MEDINA Door	113
PERMAS-PAT – PATRAN Door	113
PERMAS-ID – I-DEAS Door	113
PERMAS-AD – ADAMS Interface	114
PERMAS-EXCI – EXCITE Interface	114
PERMAS-SIM – SIMPACK Interface	114
PERMAS-HMS – MotionSolve Interface	114
PERMAS-H3D – HYPERVIEW Interface	114
PERMAS-VAO – VAO Interface	115
PERMAS-VLAB – Virtual.Lab Interface	115
PERMAS-ADS – ADSTEFAN Interface	115
PERMAS-MAT – MATLAB Interface	115
PERMAS-NAS – NASTRAN Door	115
PERMAS-ABA – ABAQUS-Door	116
PERMAS-CCL – MpCCI Coupling	117
More Interfaces	117
PERMAS for Education	119
Installation and beyond	121
Supported Hardware Platforms	121
Licensing	121
Maintenance and Porting	121
User Support	122
Additional Tools	122
Documentation	123
Training	124
Future Developments	124
Additional Information	124
Index	125

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),
- 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.

Services

INTES offers a number of services to its customers including:

- Developments for PERMAS and VisPER:
 - Interfaces to other software packages,
 - New modeling processes,
 - 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, MEDINA, ...
 - simulation with PERMAS,
- Introduction of FE analysis in enterprises, continuous consultation service (hotline), and support on current projects.

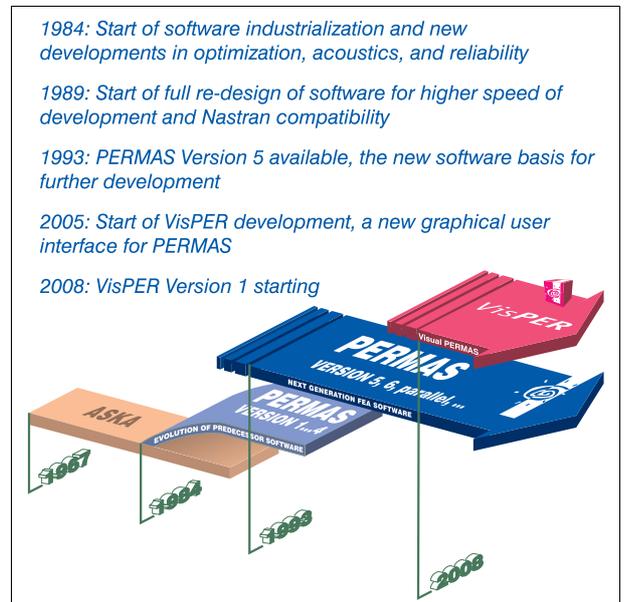


Figure 2: Synopsis of PERMAS history

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 37
- The features of **VisPER** are described on pages 39 to 51.
- The **universal features** of PERMAS, which are not related to a single module, are explained on pages 53 to 70.
- The available **functional modules** are described on pages 71 to 111.
- The **interfaces** are collected on pages 113 to 117.
- Additional information about the **installation and further aspects** of PERMAS is given on pages 121 to 124.

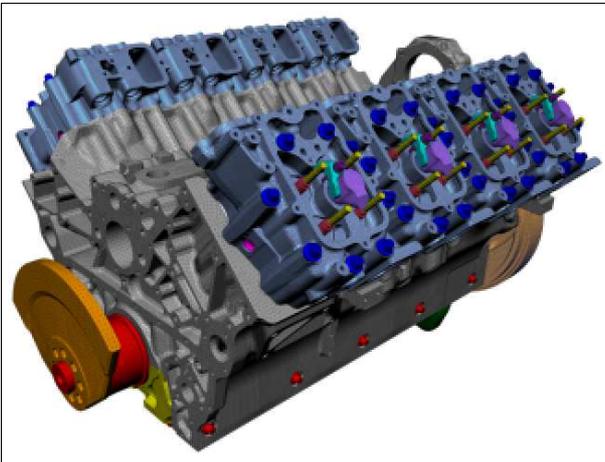


Figure 3: V8 engine by courtesy of FPT Motorenforschung AG in Arbon, Switzerland.

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-

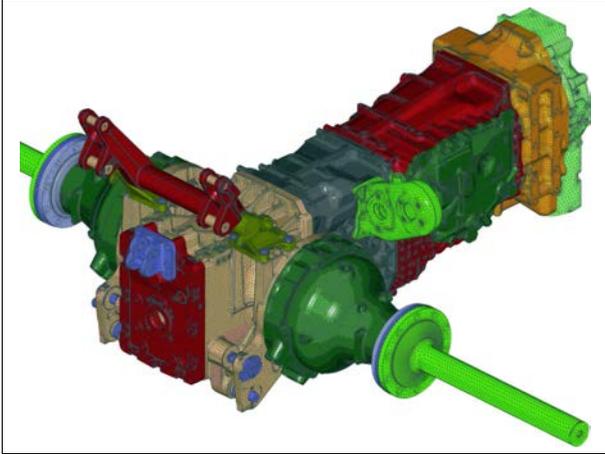


Figure 4: Model of a tractor transmission
ZF Friedrichshafen AG, Friedrichshafen, Germany.

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

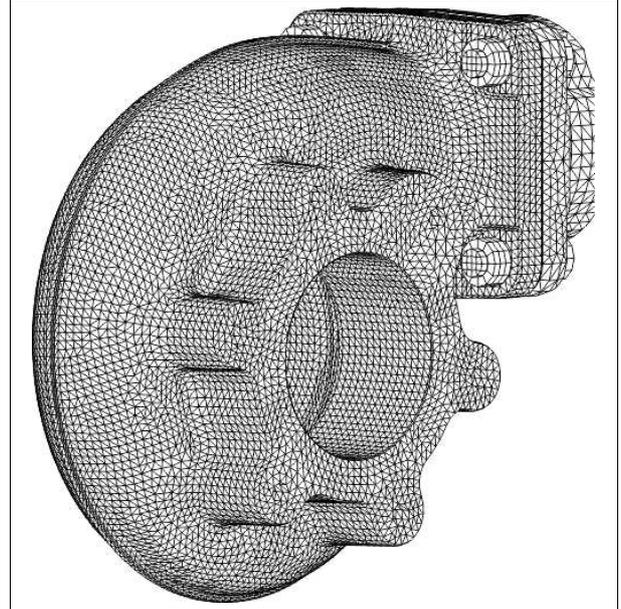


Figure 5: Turbocharger housing
BorgWarner Turbo Systems Engineering GmbH,
Kirchheimbolanden, Germany.

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.

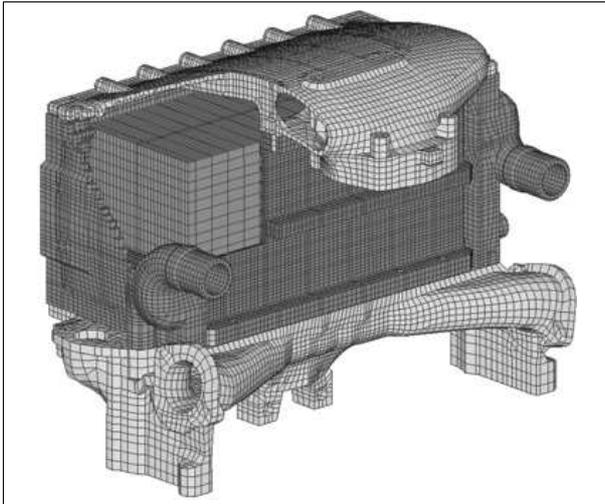


Figure 6: Charge air cooler
Behr GmbH & Co., Stuttgart, Germany.

What's New in PERMAS Version 18

The new Version 18 of PERMAS is the result of about 24 months of development work since the shipment of the predecessor version 17. For the regular reader of our Product Description of PERMAS, a rough overview summarizes the main changes in the new version. Of course, a complete and detailed Software Release Note is available with Version 18 in addition.

With *VisPER* (i.e. Visual PERMAS) we provide a dedicated tool to improve pre- and post-processing for PERMAS. To achieve a consistent version numbering, the new VisPER has the same Version No. 18 as PERMAS. The new features of VisPER Version 18 will be introduced in the next section (see page 14).



Besides the standard commercial PERMAS and VisPER software, now a new PERMAS4EDU (PERMAS for Education) edition of this software package is available for education and further training purposes. It is free-of-charge and does not per-

mit commercial use. There are a number of limits on model size (see page 119), and most of the analysis functions are available.

PYTHON is now the standard scripting tool for PERMAS and VisPER (see page 122). In order to provide a standard tool (i.e. *pyINTES*) for PERMAS and VisPER, the PERMAS installation provides this tool in addition. For advanced usage of PERMAS and VisPER, it is highly recommended to have this standard tool available. Scripts from INTES for special operations may be provided using this standard tool. Now, Python 3 is used as standard version.

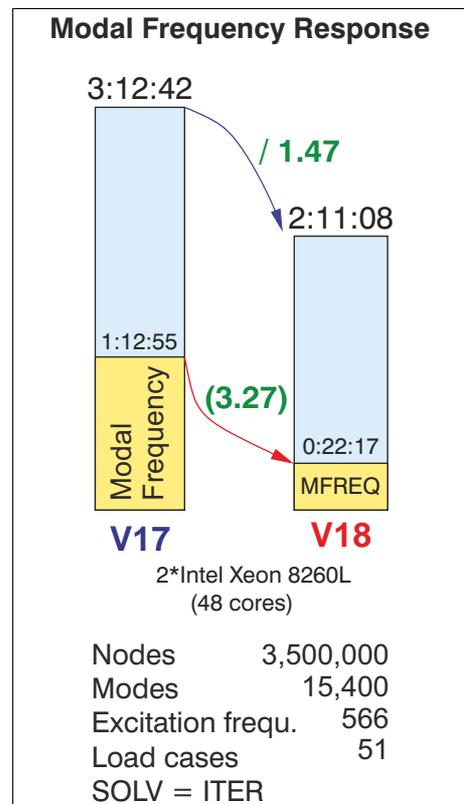


Figure 7: New iterative solver for modal frequency

PERMAS Version 18 offers again improved computing performance:

- Frequency response analysis has been significantly accelerated by automatic selection of solver algorithm and run time reduction for the iterative solver (see Fig. 7).
- The runtime for contact analysis of large models could be reduced again.
- By ongoing parallelization, nonlinear stress calculations now show improved speed-up.
- For heat transfer, conductivity and heat capacity computation is parallelized (see Fig. 8).

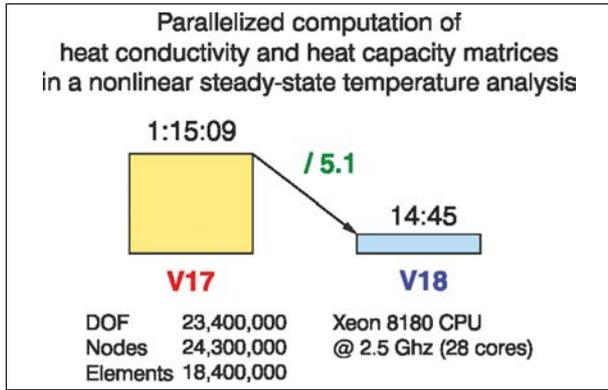


Figure 8: Parallelization of conductivity and capacity matrices.

- The list of supported Nvidia Tesla graphic cards in Module XPU (see page 111) has been extended. Now, beside Kepler also Pascal and Volta graphic cards are supported by module XPU (Cuda 10.1 is recommended).
- The *HDF import* in VisPER has been significantly accelerated. Post-processing may start much faster even with large result data sets (see Fig. 19).

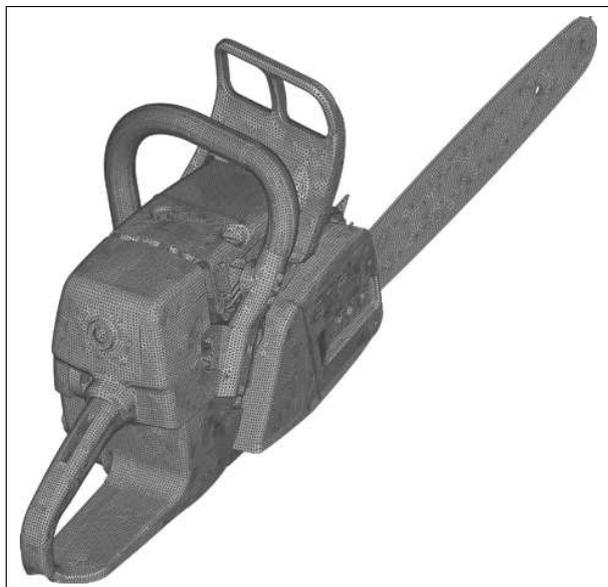


Figure 9: Model of a chain saw
Andreas Stihl AG & Co. KG, Waiblingen, Germany.

The list of major software extensions in PERMAS is as follows:

- **New modules:**
 - A new module *CAMG* establishes the 3rd generation of PERMAS contact solvers. It is an own and unique technology branch of INTES. The module enables significant run-

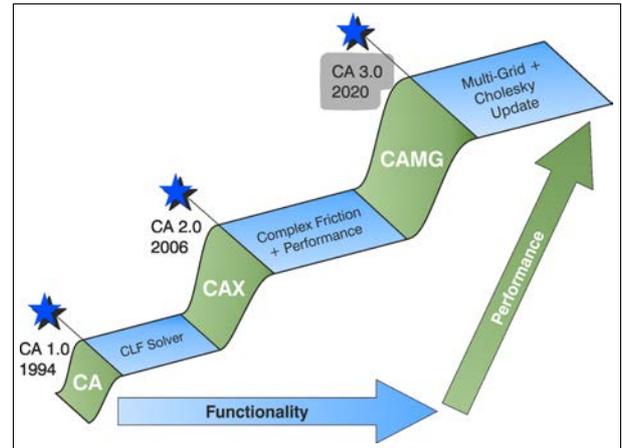


Figure 10: Evolution of contact solvers

time improvements for larger contact models by using an innovative *Contact-Multigrid-Solver*. Fig. 10 shows the major three steps in developing the alternative *CLF* (Condensed Lagrange Flexibility) solver since year 1994. Fig. 11 shows the runtime reduction using CAMG compared to the previous software release V17 for a large engine model. For further information, see from page 78.

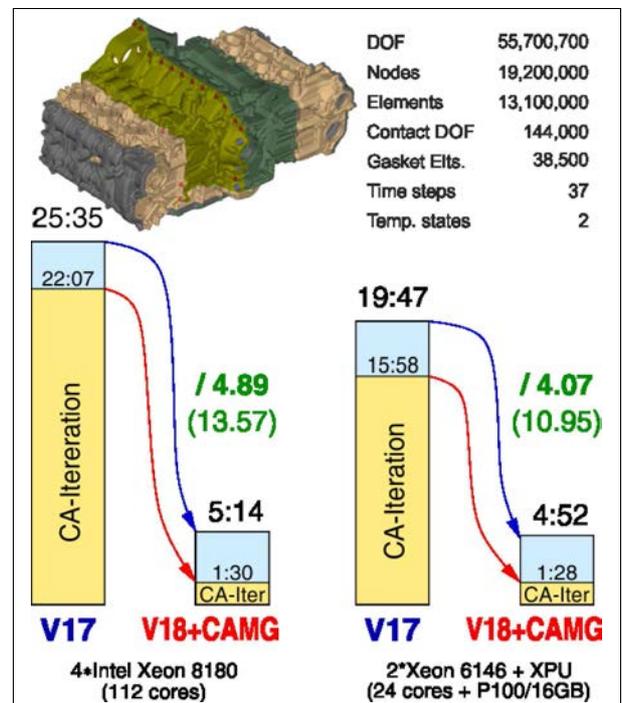


Figure 11: Contact analysis of an engine with 56 Million DOF, 37 time steps, 2 different temperature states

- The new module *NLSA* enables *hyperelastic material behavior* (quasi-incompressible) to-

gether with large-strain elements in nonlinear analyses (see Fig. 12). For further information, see from page 84.

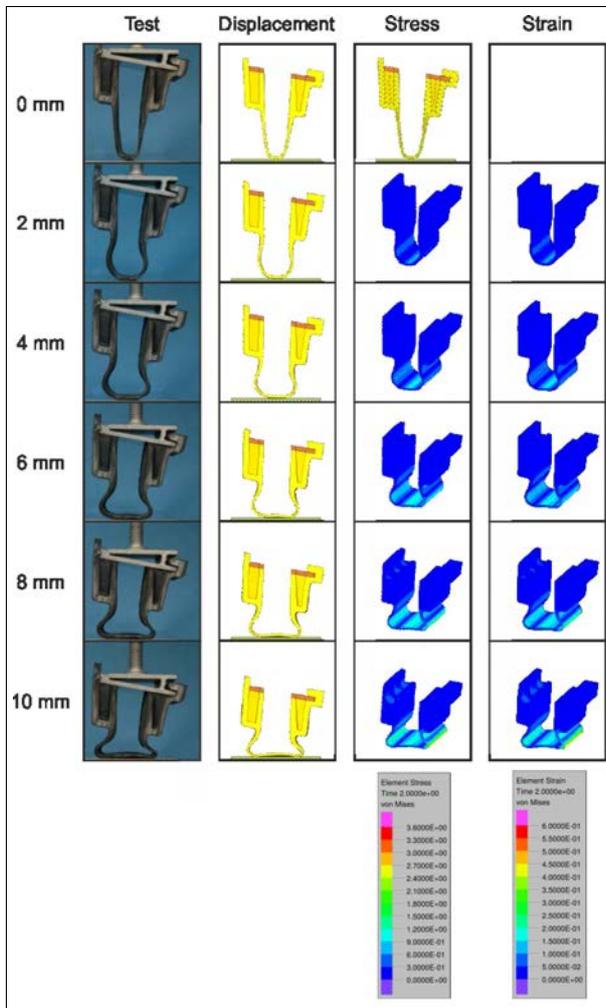


Figure 12: Large strain analysis (by courtesy of Mahle Behr GmbH, Stuttgart, Germany)

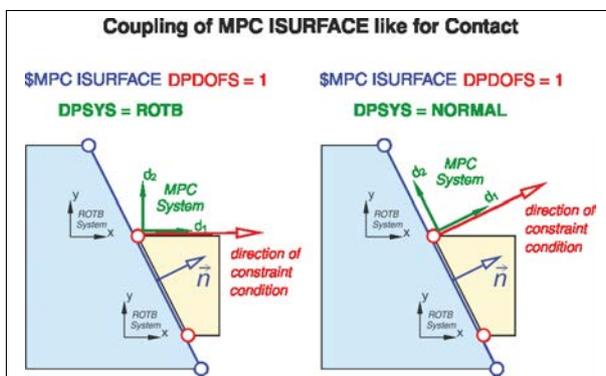


Figure 13: Surface coupling

• **Major extensions:**

- Extensions to basic module (module MQA, page 71):

- * **MPC** conditions have been revised and their application has been extended. Up to now, MPC conditions are part of the invariant model description like elements. Now, MPC conditions are also possible in the constraint variant as a variable model part, allowing to switch easily between a fixed connection (“tied contact”) and a contact definition. Moreover, contact locking for linearization of the model (i.e. switching from contact to linear dynamics) is now also possible in the constraint variant.
- * The coupling of surfaces (“tied contact”) has been extended to easily use the normal and tangential directions of the surface. This allows the surface coupling definition like for contact, which facilitates the switching between surface coupling and contact (see Fig. 13).
- * The coupling of nodes by MPC conditions has been extended to achieve an analogous description as for node-to-node contact including contact to ground.
- * Overlapping regions of MPC conditions usually lead to duplicated definitions of dependent degrees of freedom. Now, an automatic handling of such situations overcomes the resulting over-constraints.
- Extensions to linear static analysis (module LS from page 73):
 - * An additional second geometrical stiffness is available to specify a constant and a variable part of the geometrical stiffness (e.g. for a rotating systems with press fit).
 - * New options for a quasi-static analysis are available (**Inertia Relief**, see page 73) in order to select the center of gravity, to automatically detect rigid body modes and to take added mass as well as load correction matrices into account.
- Extensions to contact analysis (modules CA, CAX, and CAU from page 73):
 - * New contact summary table with contact area data on result file.
 - * Table “Statistics of contact neighborhood search” with additional information such as: minimal geometrical gapwidth, neighborhood information, and number of contact pairs with constraints.

- * Coefficient of sliding friction can be defined with a load-dependent function.
- * More consistent definition of initial gap for gasket elements.
- Extensions to nonlinear static analysis (module NLS, page 80):
 - * Support of *transverse isotropic plasticity* for *short fibre material*.
 - * Improved lever arm correction for geometrically nonlinear MPC conditions and support of rigid bodies with one dependent degree of freedom.
 - * A spring element (2 nodes, 6 degrees of freedom) with co-rotational formulation for geometrically nonlinear analysis.
 - * Parallel computation of nonlinear element stress (see Fig. 14).

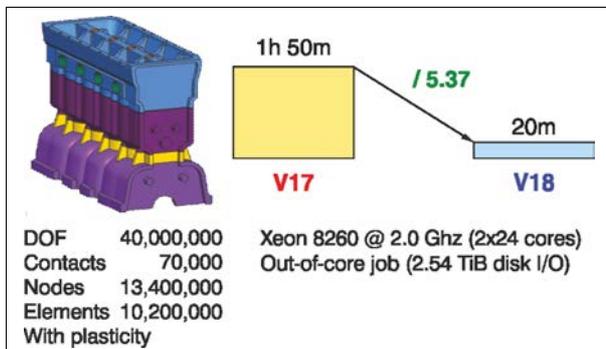


Figure 14: Parallel nonlinear element stress computation of a motor with 10.2 Mio elements

- Extensions to linear buckling analysis (module BA, page 84):
 - * An automatic *shift method* for the stiffness matrix has been added to selectively analyze a certain range of buckling factors. Moreover, the sign of the load factors to be computed, can be chosen.
 - * Buckling may be combined with topology and shape optimization.
- Extensions to mode analysis (modules DEV and DEVX, pages 88 and 89):
 - * Rigid body modes are listed in a table on the result file. This table has been extended by an estimate of the quality.
 - * If the first eigenfrequencies seem to be non-detected rigid body modes, they are moved to the rigid body modes.
 - * Equivalent viscous damping ratios are available for real (overcritically damped) eigenmodes in modal complex eigen-

- value analysis.
- * In break squeal analysis, the dependency of friction coefficients on sliding velocity is taken into account.
- * Gyroscopic matrices are now available for beam elements on the rotor axis in a fixed frame of reference.
- Extensions to dynamic response analysis (module DRA, page 91) and fluid-structure acoustics (module FS, page 95):
 - * Specification of prescribed velocities and prescribed accelerations is possible for direct and modal frequency response analyses.
 - * For the modal frequency response analysis and the modal fluid and modal fluid-structure frequency response analysis, a new auto-solve option has been implemented, which selects the most appropriate solver out of a set of available solvers: direct, iterative, and SMW solvers. The *Shermann-Morisson-Woodbury* (SMW) solver has been upgraded and is now faster with many linear damper elements. In addition, frequency-dependent symmetric control elements are now supported. The new iterative solver is now much faster for big problems with excellent convergence behaviour for most problems.
 - * In modal transient analyses initial displacements and velocities may be specified.
 - * In modal transient analyses and modal general condensation, material or structural damping may be replaced by equivalent viscous damping calculated by referring to the real eigenfrequencies of the corresponding eigenvalue analysis.
- Extensions to design optimization and topology optimization (modules OPT and TOPO, pages 98 and 102):
 - * Sampling, topology optimization, and shape optimization are now possible also for buckling analysis.
 - * Sampling and all optimization methods are now possible for consecutive analyses including contact coupling, such as brake squeal analyses.
 - * The OC method for freeform optimization (see page 99) has been completely

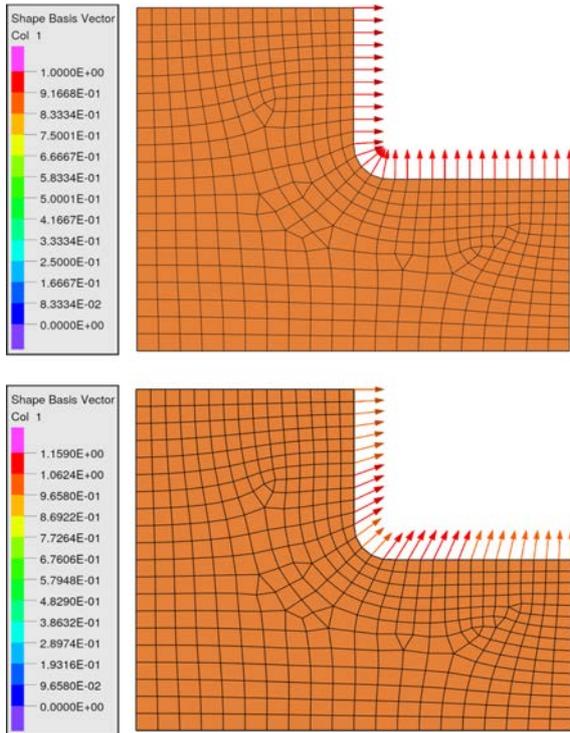


Figure 15: Improved shape changes

revised, i.e. better solver, improved filter and stepsize handling and special treatment of degenerated elements. Midnodes of quadratic elements may now 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. 15 with non-normal thickness changes).

- * Extended manufacturing conditions for freeform and topology optimization (see Fig. 23).
- * If quantities should just be monitored but not used as design constraints, a definition of specific result items can be made to view the course of these items over the iterations.
- * New design constraints have been implemented:
 - Center of gravity,
 - Inertia properties,
 - Load scale factors in buckling analysis,
 - Resulting pressures (sampling only).
- * Improved minimum and maximum membersize handling (see Fig. 16).

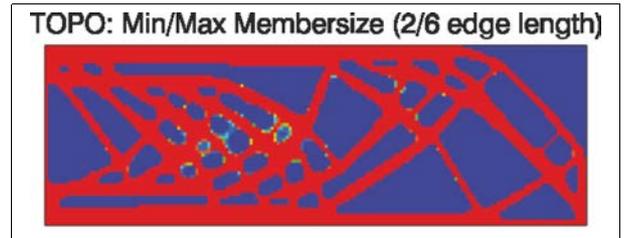


Figure 16: Extended Membersize handling

– **New elements:**

- * Solid large strain elements for hyperelastic materials are available. Existing solid elements (see Fig. 12) are used for this purpose automatically provided that a hyperelastic material is assigned.

Many smaller extensions of almost all functional modules had been performed in addition. Moreover, all interfaces were updated and adapted to the new functionalities. Major interface enhancements are:

– **ABAQUS (ABA)** (see page 116):

- * Support of part instances,
- * Support of hyperelastic material data,
- * Abadoor translation report (pptx/xlsx) in VisPER.

– **MEDINA (MEDI)** (see page 113):

- * Support of zero force elements with Medina 9.0.4.1 ff. and support of physical description label,
- * Export of new PERMAS Version 18 results.

– **I-DEAS (ID)** (see page 113):

- * Support of export in I-DEAS Master Series format (topology and results).

– **EXCITE (EXCI)** (see page 114):

- * Support of Excite binary format *exb* and of inertia invariants,
- * Parallel calculation of inertia invariants (runtime reduction from 27.5h to 7.3h),
- * Recovery matrix to separate file.

– **SIMPACT (SIM)** (see page 114):

- * Support of SIMPACK Version 9 as default.

For all system platforms an update to the current release of the operating system had been performed (see Page 121).

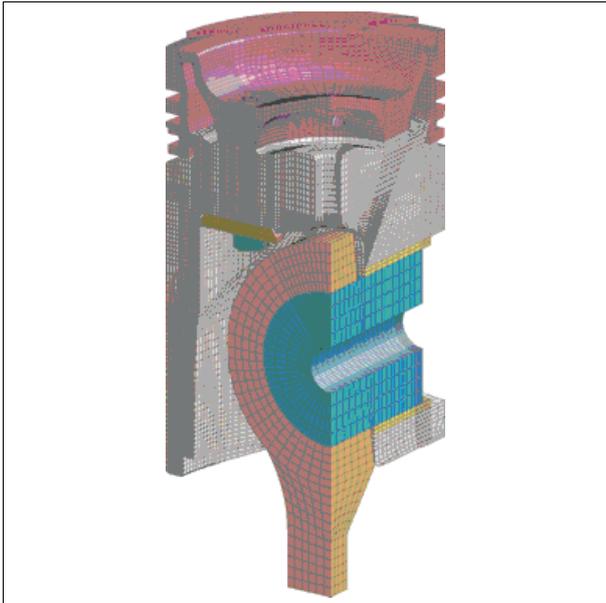


Figure 17: Ship engine piston
Mahle GmbH, Stuttgart, Germany.

What's New in VisPER Version 18

VisPER (i.e. Visual PERMAS) is a dedicated tool to provide analysis workflows (see Figs. 18 and 24) and to improve pre- and post-processing for PERMAS. VisPER Version 18 is released together with PERMAS 18. More information on VisPER can be found from page 39.

- **Some highlights:**
 - The postprocessing-performance has been significantly improved (see Fig. 19).
 - The GUI for postprocessing (i.e. Results&Data dialog) is now based on a table with checkboxes, which is easy to use also for non-experts (see Fig. 20).
 - A new camera model with non-clipping zoom allows for any level of detail (especially for contact models).
 - Multiple synchronized or independent viewers allow flexible insights for model checking and postprocessing (see Fig. 21).

The list of major software extensions in VisPER is divided in four sections as follows:

- **Model completion:**
 - MPC definitions may, as part of the constraint variant, be modified or exchanged by contact conditions.
 - Functions are available to define distributed loads (also at nodes/surfaces). The func-

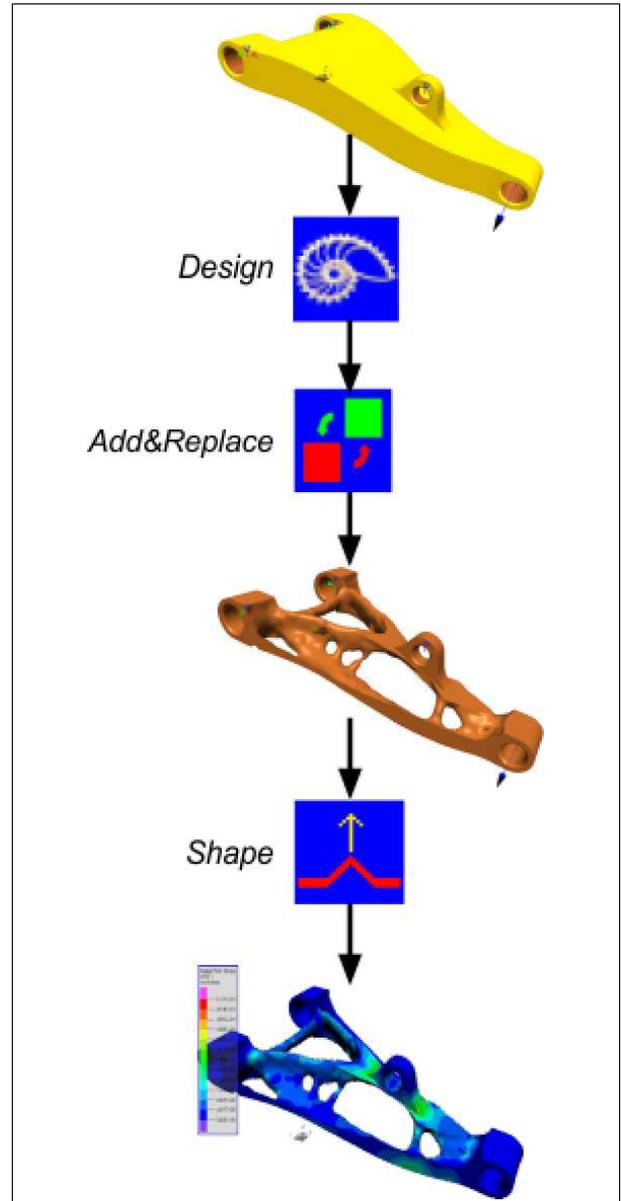


Figure 18: Operations of design wizard

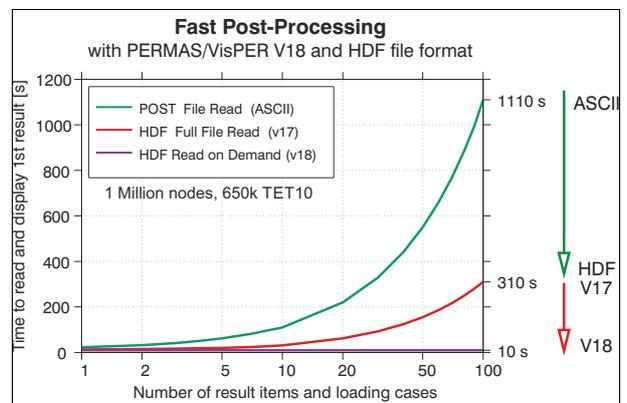


Figure 19: HDF postprocessing performance

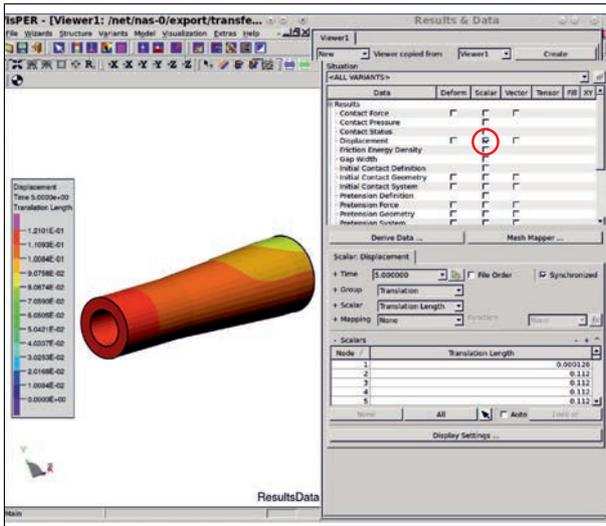


Figure 20: New post-processing

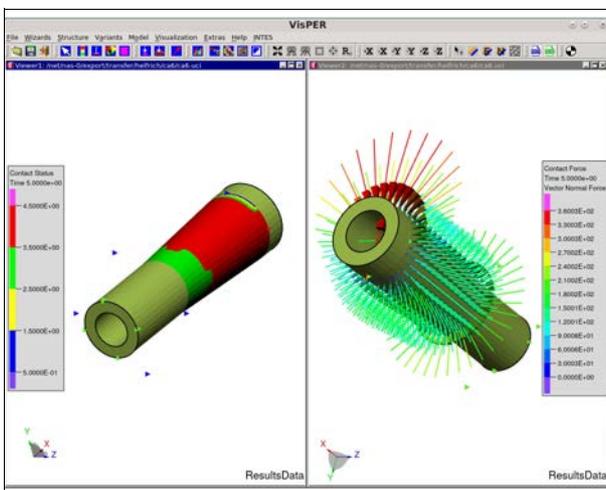


Figure 21: Multiple viewers

tion evaluation may be verified using the Results&Data dialog (see Fig. 22).

- A new measurements dialog is available for verification and documentation purposes.
- Highly extended relabelling functionality to resolve naming conflicts of variants.
- New Load History Assistant for specification of nonlinear loading history and time selection for result export.

• **Wizards:**

- New design constraints (like center of gravity, inertia parameters, load scale factors) are supported in the optimization wizards as well as design constraint linking.
- Support of 3D printing restrictions in Topology and Freeform Wizard (see Fig. 23).
- Improved smoothing and extended check ca-

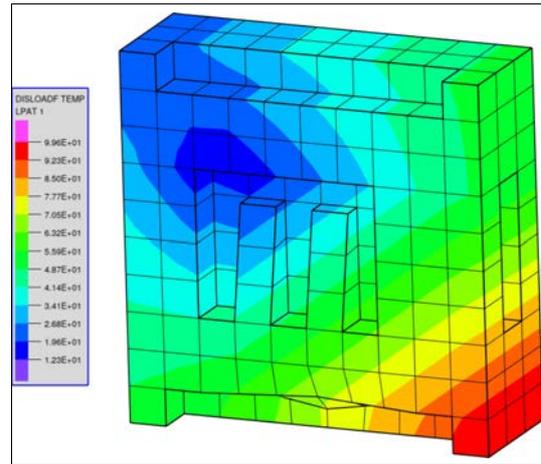


Figure 22: Function evaluation

pabilities in the Design Wizard.

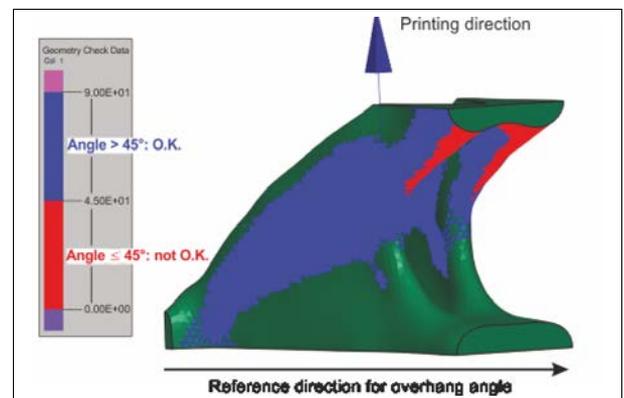


Figure 23: Check of overhang angle
red: critical angle, blue: non-critical, other colors: no value (i.e. >90°).

• **Post-processing:**

- 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.

• **Tools:**

- Using PYTHON scripting enables standardized post-processing including image and video generation. Model information, images, and videos may be exported to MS Word, Excel, and PowerPoint.
- An additional menu to provide functions to check and improve closed 2D (e.g. STL) meshes:
 - * Check and improve triangle meshes: No.

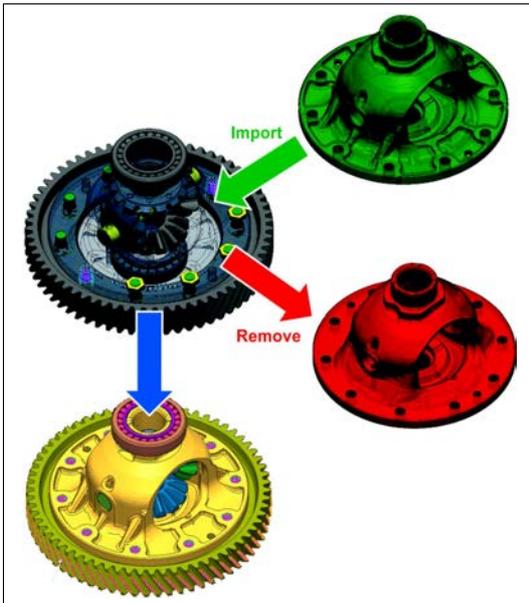


Figure 24: Replacement of model part

of neighbors, angle between neighbors, element orientation, penetration.

- * For fixing meshes, a specific dialog is provided to improve the element quality of selected elements by a minimal modification of their element node coordinates, preserving the mesh topology. This is performed, using mathematical optimization methods.
- * A dialog is available to create a patch mesh with linear triangle elements, based on a surface mesh (i.e. STL). To ensure, that the initial mesh is consistent, a check of the initial triangle mesh is useful.

Universal Features

The outstanding mostly module-independent basic features of PERMAS are as follows (see pages 53 to 70):

- Hierarchical substructuring, with automatic sub-component insertion (see page 53)
- Submodeling (see page 53)
- Variant analysis (see page 54)
- Cyclic symmetry (see page 55)
- Surface and Line Description (see page 55)
- Automated coupling of parts (see page 56)
- Automated spotweld modeling (see page 57)
- Local coordinate systems (see page 58)
- Multiple kinematic constraints (see page 58)
- Automatic detection of singularities (see page 59)
- Same elements for different analysis types (element library, see page 60)
- Standard beam cross sections (Seite 61)
- Design elements for optimization (page 62)
- SPR Stresses and Absolute Error Indicator (page 62)
- General material description (see page 62)
- Node and element sets (see page 63)
- Mathematical functions (see page 63)
- All kinds of loading (see page 64)
- Model verification (see page 64)
- Integrated interfaces to pre- and post-processors (see page 65)
- Input and Output of Data Objects and matrices (see page 67)
- Combination, transformation, and comparison of results (see page 67)
- Output of XY result data (see page 68)
- Calculation of cutting forces (see page 69)
- System of units (see page 69)
- Restart facility (see page 69)
- Open software through Fortran and C interfaces (see page 69)
- Direct coupling of different analysis types (see page 70)
- Coupling with CFD (see page 70)

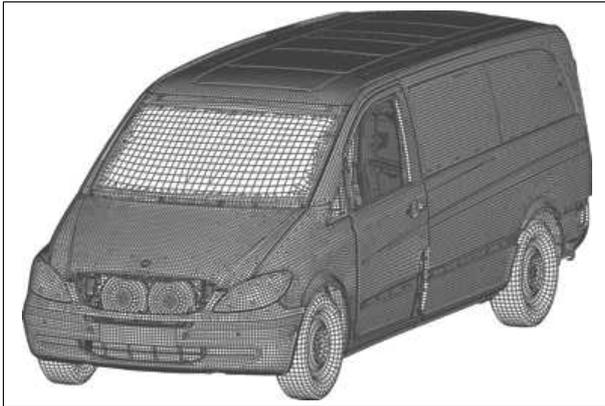


Figure 25: Model of a transport vehicle
courtesy of Daimler AG,
Commercial Vehicle Division in Stuttgart

Available VisPER Modules

The below listed functional modules are explained in more detail on pages 40 to 49:

- Basic module (VBAS)
- Topology optimization (VTOP)
- Design optimization (VOPT)
- Fluid-structure coupling (VFS)
- Contact analysis (VCA)

Available PERMAS Modules

The below listed functional modules are explained in more detail on pages 71 to 117:

- 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)

- Fluid-Structure Acoustics (FS)
- Nonlinear Dynamics (NLD)
- Design Optimization (OPT)
- Layout Optimization (TOPO)
- Advanced optimization solvers (AOS)
- Reliability Analysis (RA)
- Laminate Analysis (LA)
- Refined Weldspot Model (WLDS)
- Generalized Inertia Relief (GINR)
- 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)
 - SIMPACK (SIM)
 - EXCITE (EXCI)
 - MOTIONSOLVE (HMS)
 - HYPERVIEW (H3D)
 - VAO (VAO)
 - Virtual.Lab (VLAB)
 - ADSTEFAN (ADS)
 - MATLAB (MAT)
 - NASTRAN (NAS)
 - ABAQUS (ABA)
 - MpCCI (CCL)

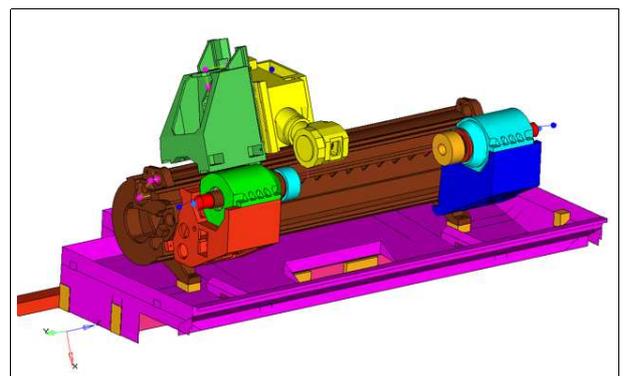


Figure 26: Machine tool
INDEX-Werke GmbH & Co. KG, Esslingen, Germany

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. 121).
- Disk I/O can be avoided at all, if large memories are used. The memory size may readily exceed 256 GB.

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 111).

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

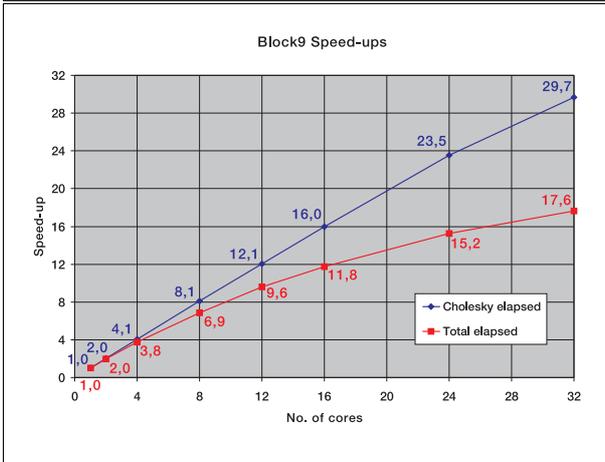
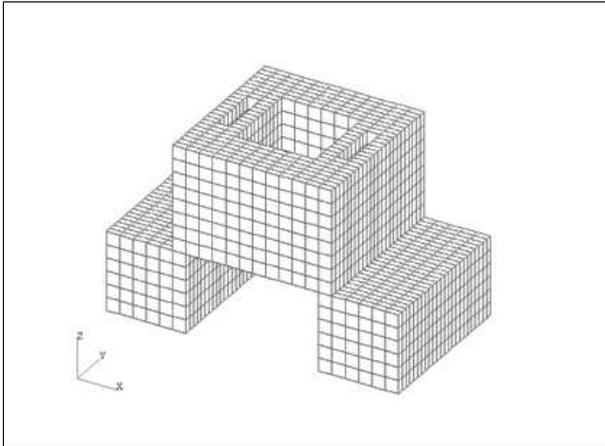


Figure 27: Static analysis with 3 loading cases
1.5M nodes, 176k HEXE27, 4.4M Dof
run time on Intel Boxboro

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 71). 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 64). In addition, VisPER provides a model verification environment for a growing number of modeling parameters (see page 40).
- **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.

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

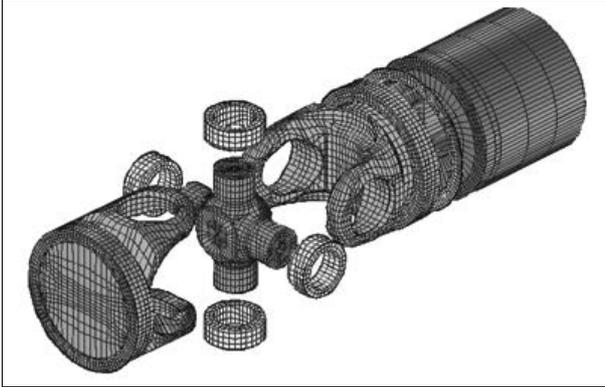


Figure 28: Model of a cardan shaft
Voith Turbo GmbH & Co. KG, Heidenheim, Germany.

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 57 and module WLDS on page 110), bonding, laser welding. In order to generate the meshes of all parts efficiently, *incompatible meshing* (see page 56) is used for independent meshing.

A special feature in VisPER supports post-processing of spotwelds (see page 51) 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 73) are applied.

To check the force flow through any structural member, *cutting forces* (e.g. through a column or sill, see page 69) 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.

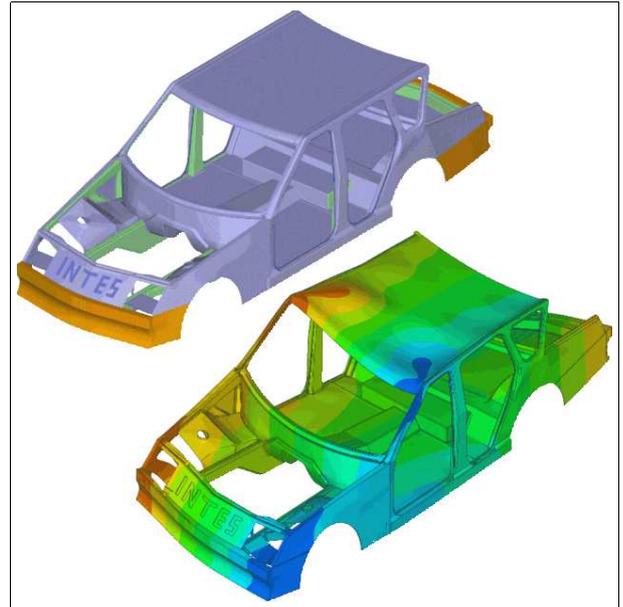


Figure 29: Workshop example INTEScar under torsional loading

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 114).

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 93).

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 89). Using dynamic condensation so-called *matrix models* (see page 67) 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 91), 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. 103 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 93). In particular, trimmed bod-

ies 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 46). 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 95 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 90. 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 93) 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.

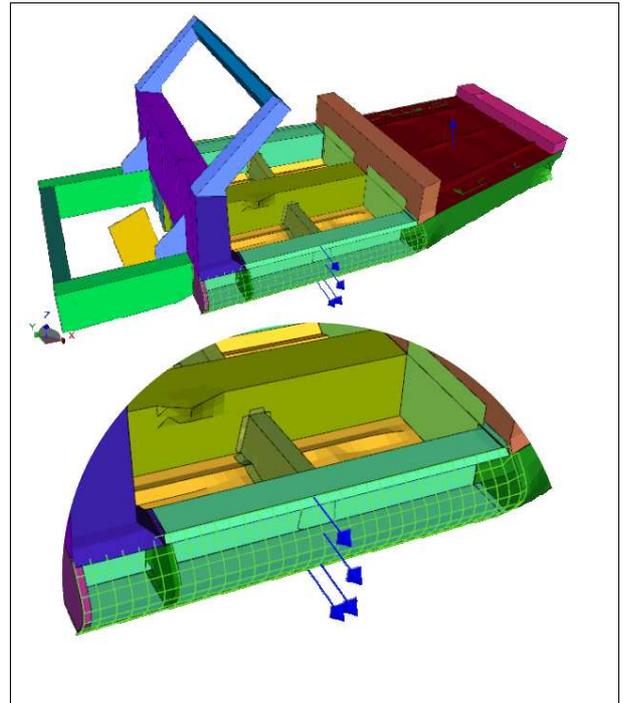


Figure 30: Shape optimization of a sill with transition to neighbored parts

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 98 to 107) 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 possible with incompatible meshes).

- **Bead design:** This is used to position and shape beads in shell structures (see example in Fig. 31).

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 44). 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.

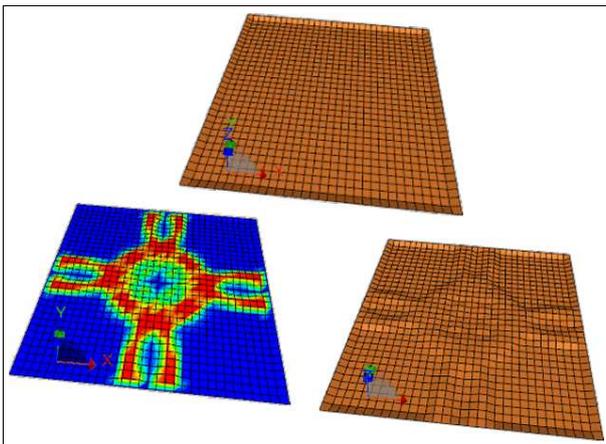


Figure 31: Bead design of a plate with positioning and height of beads

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 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.

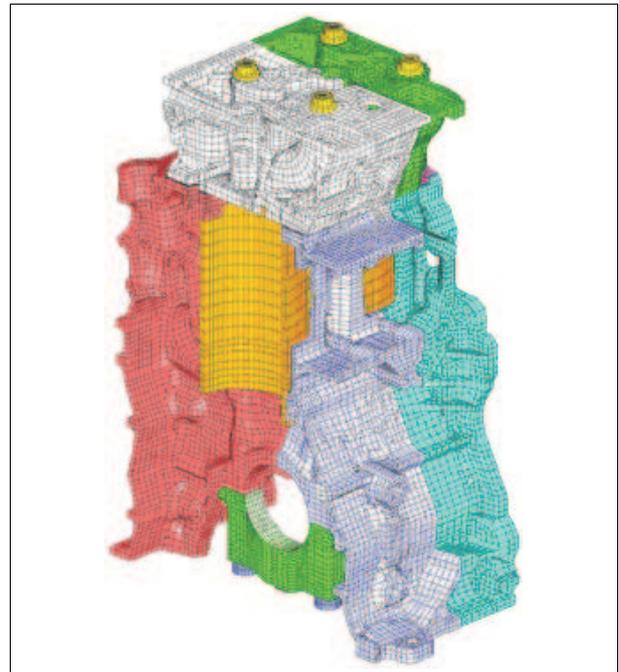


Figure 32: Simple engine model
Daimler AG, Commercial Vehicle Division)

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 automatic 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 53).

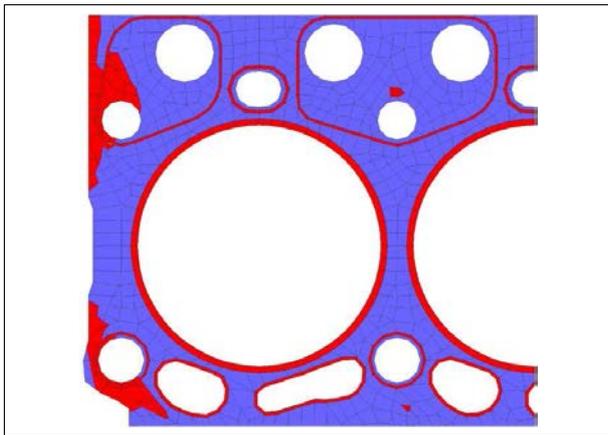


Figure 33: Contact status for gasket elements

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*:
 - 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 con-

nected only by contact, the RBM assistant in VisPER helps to avoid rigid body modes by applying compensation springs (see page 42).

- 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.

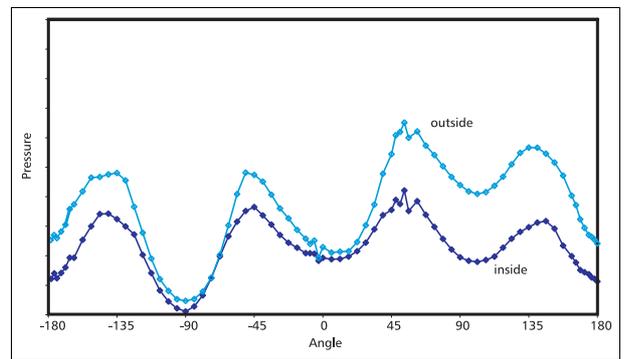


Figure 34: Pressure distribution at stopper over the angle

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,
- 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. 121).

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 88 to 95). Some important points are:

- Eigenvalues and mode shapes for large solid models can be calculated using MLDR (see page 90).
- Fast *dynamic condensation* methods support the efficient analysis of engines with many attached parts (DEVX, see page 89).
- By using *dry condensation* (page 89) 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 77). 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 58).
- Contact (i.e. connection can open and close during loading). This is the topic of contact analysis (see page 73).

In all cases, the surfaces of coupled parts can be meshed compatibly or incompatibly (see the part coupling on page 56). 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 77). In this way, a linearization of the contact problem is achieved. Fig. 35 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-

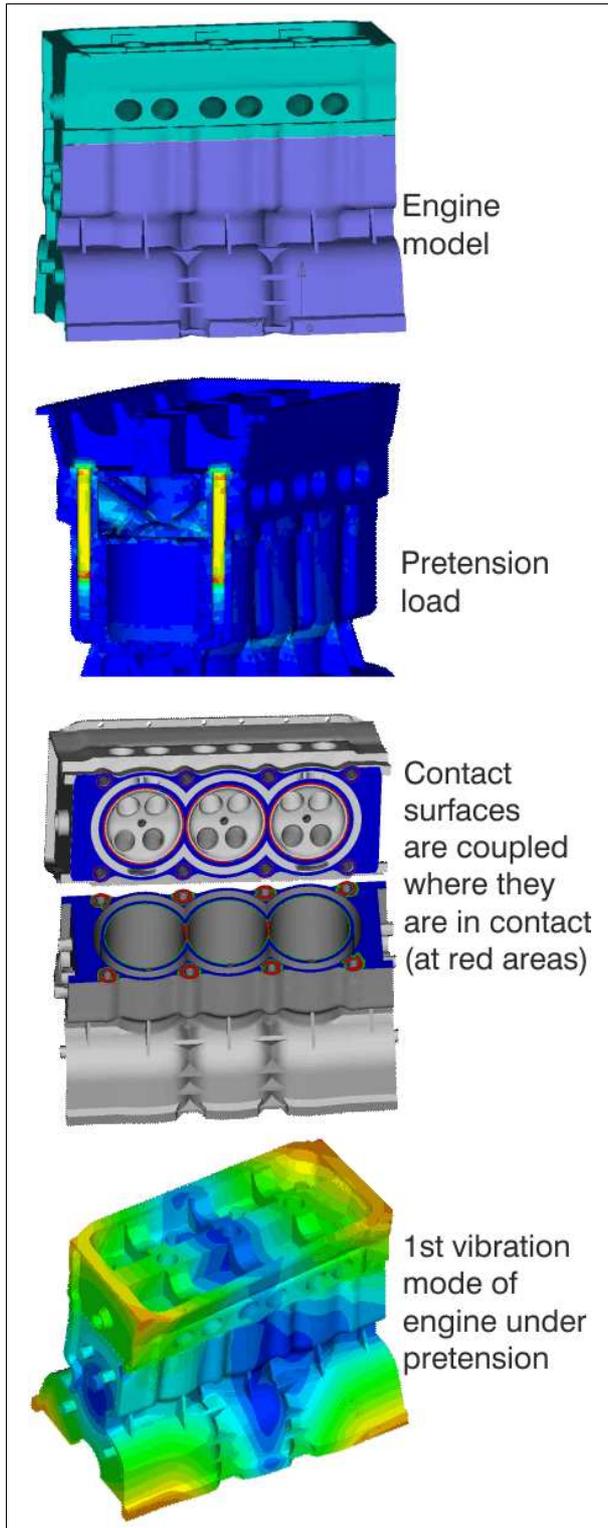


Figure 35: Dynamic analysis of an engine under pretension load

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. 36 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.

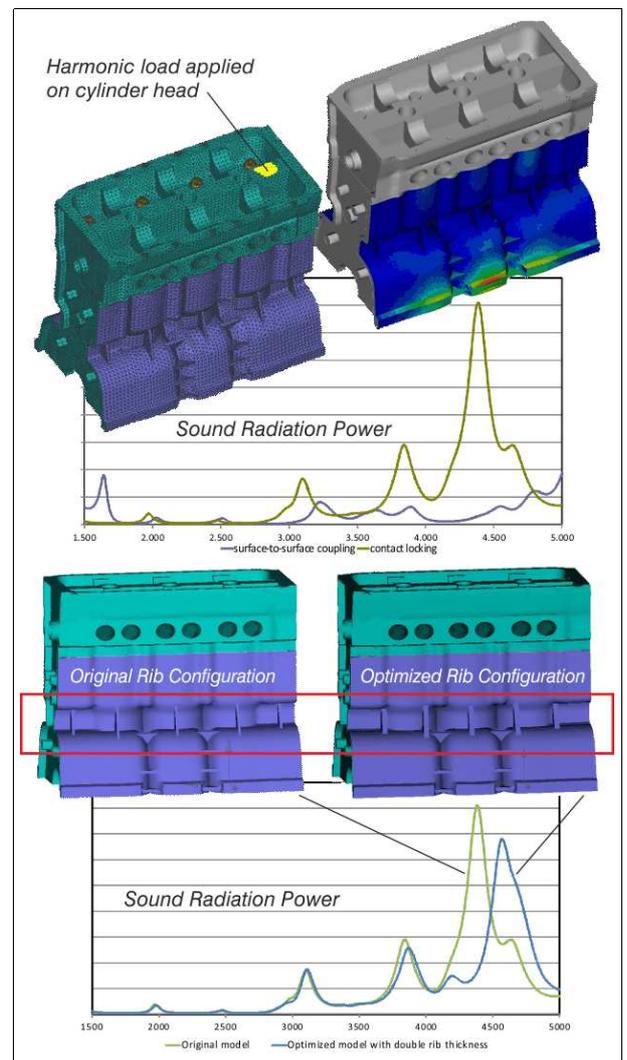


Figure 36: Optimization of rib thickness and rib position to reduce sound radiation from engine block surface

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 75 for more details).

The thread coupling of bolts is of particular importance for short bolts (like in Fig. 37), 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 57 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 110).

- **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. 148).

- **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. 148).

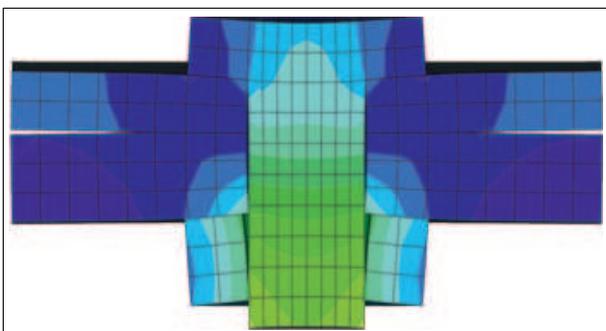


Figure 37: Short bolt under pretension

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.

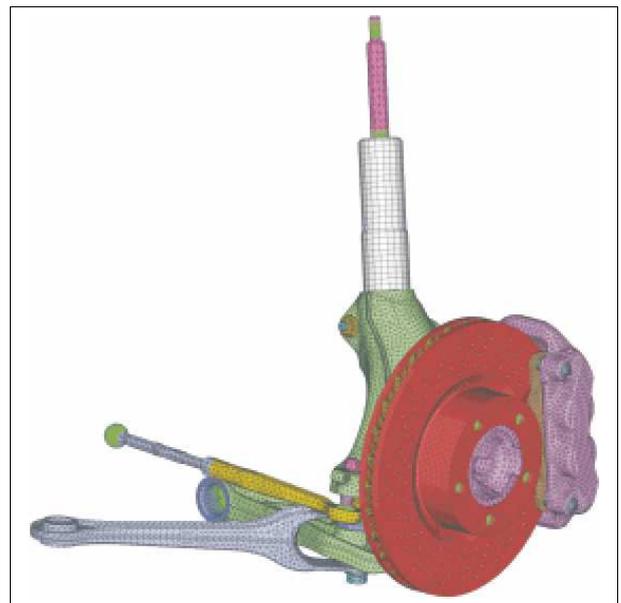


Figure 38: Brake model

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:

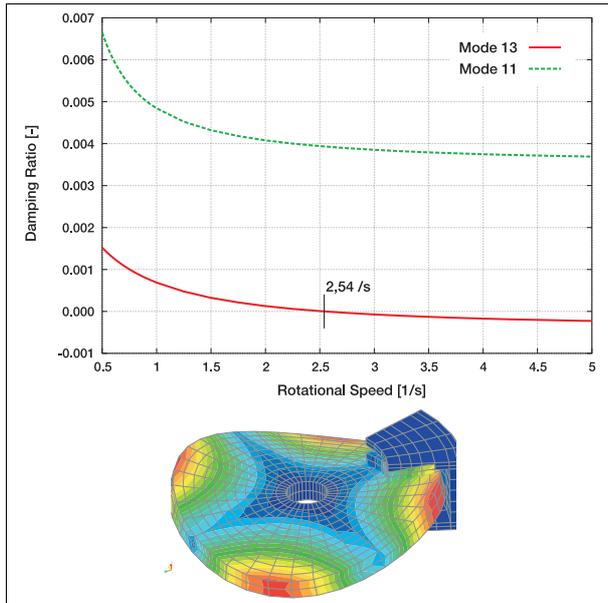


Figure 39: Simple brake model (1) with an unstable bending mode ($m=2, n=1$) at 2,54 rps

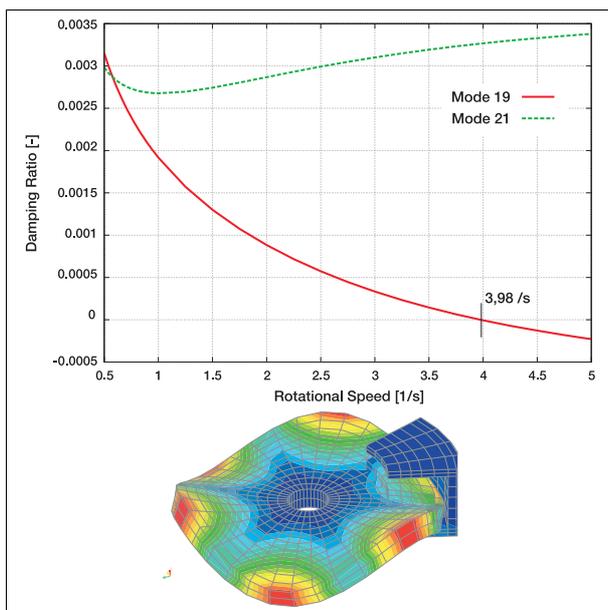


Figure 40: Simple brake model (2) with an unstable bending mode ($m=3, n=1$) at 3,98 rps

- 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.

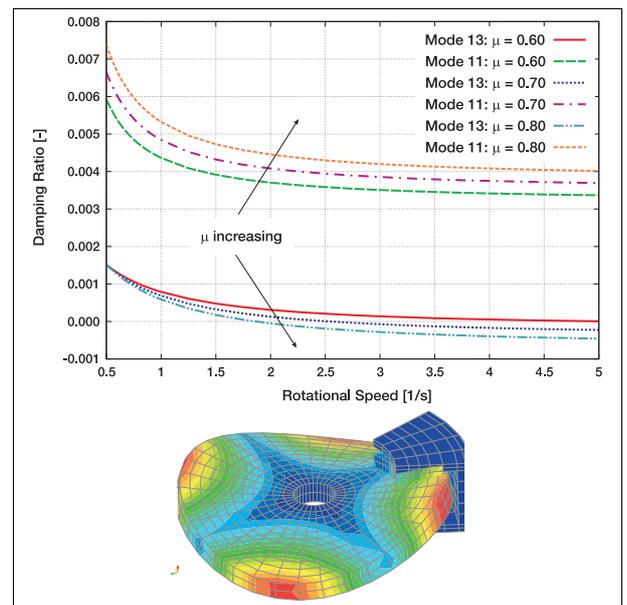


Figure 41: Simple brake model (3) with an unstable bending mode ($m=2, n=1$) which becomes unstable at different rotational speeds dependent on the frictional coefficient.

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. 39 and Fig. 40). 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. 41).

A similar study was made to get the influence of

Young's modulus of the brake pad. Figure 42 shows a corresponding example.

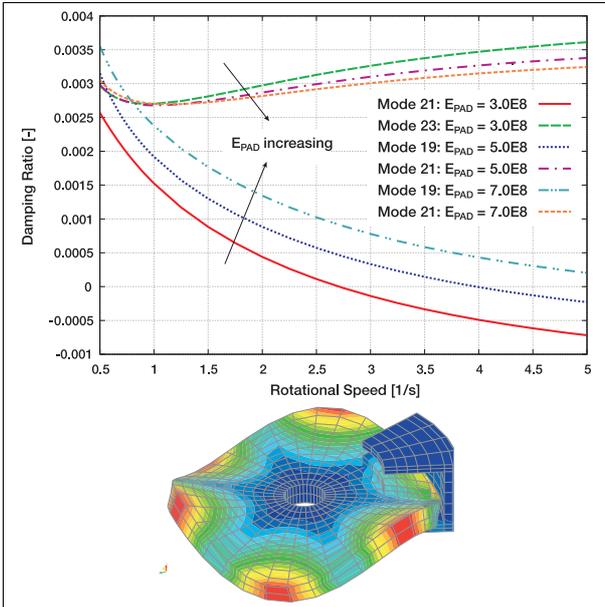


Figure 42: Simple brake model (4) with an unstable bending mode (m=3, n=1) which becomes unstable at different rotational speeds dependent on Young's modulus of the brake pad.

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. 43). 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-

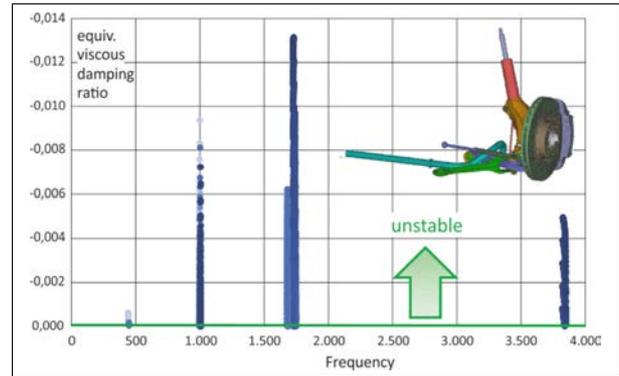


Figure 43: Stability map of a brake for 7110 parameter sets from rotational speeds, Young's modulus of disk, and frictional coefficient between disk and pad. The total run time for this stability map was 5 hours 12 minutes.

prove brake squeal behaviour.

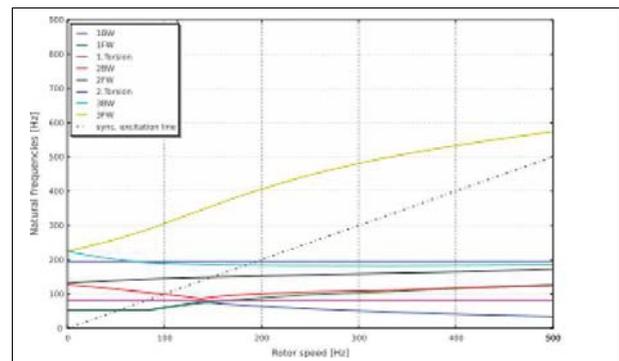


Figure 44: Campbell diagram for the evaluation of rotor dynamics

Rotating Systems

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

Fig. 45 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 45: 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. 44 and page 89) 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 overcritical 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. 144).

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 92).

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.

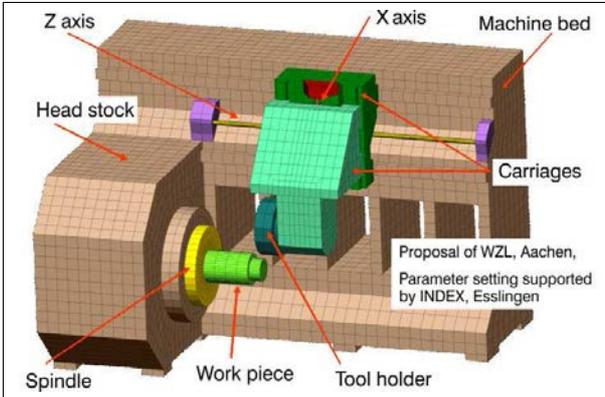


Figure 46: Simplified model of a turning machine as proposed by WZL in Aachen and supported by INDEX, Esslingen for setting suitable parameters

An example of a turning machine has been proposed by WZL Institute in Aachen (see Fig. 46). 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. 46).
- **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

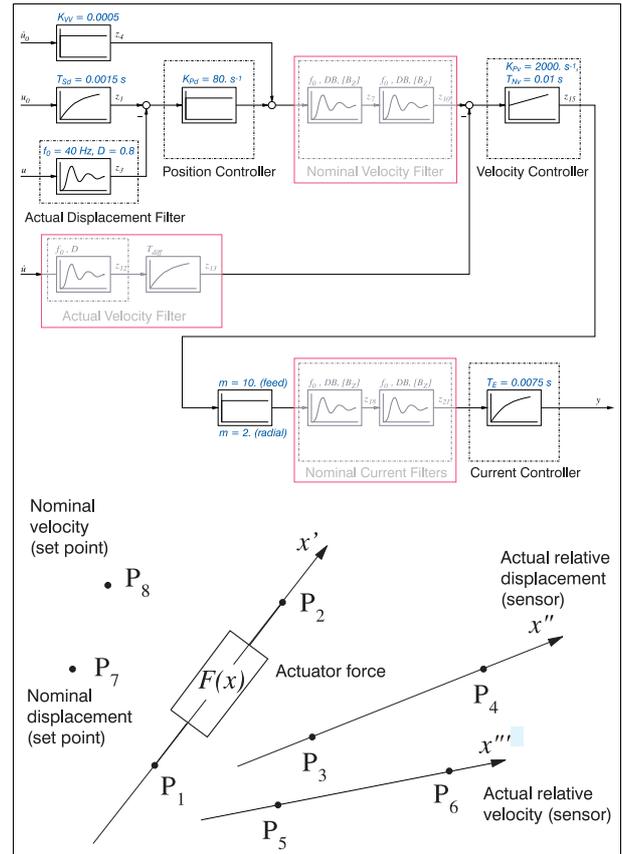


Figure 47: Block diagram of used controller and corresponding finite element representation

(see Fig. 144).

- **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. 47. 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

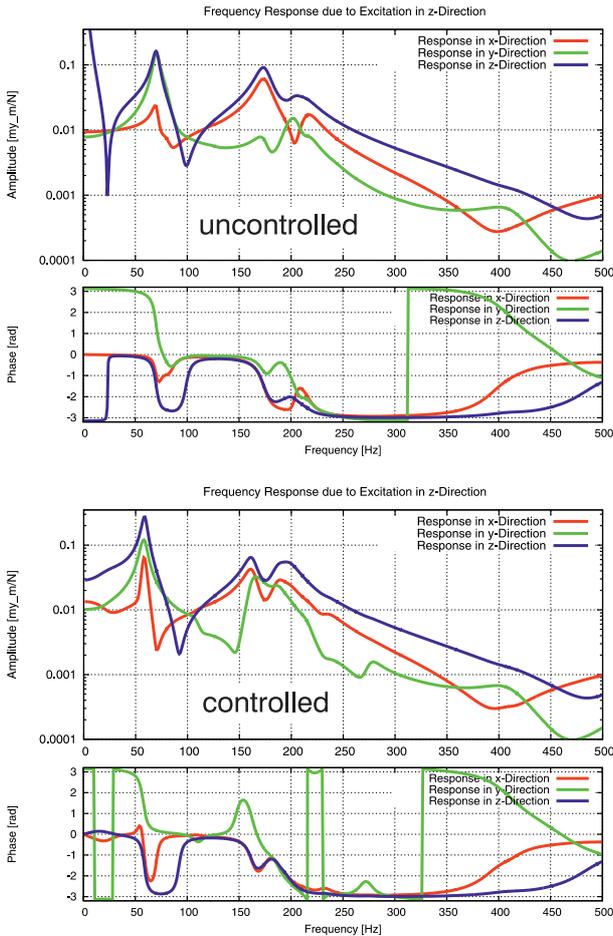


Figure 48: Frequency response between tool and workpiece due to excitation in Z direction for uncontrolled (above) and controlled (below) machine

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:**

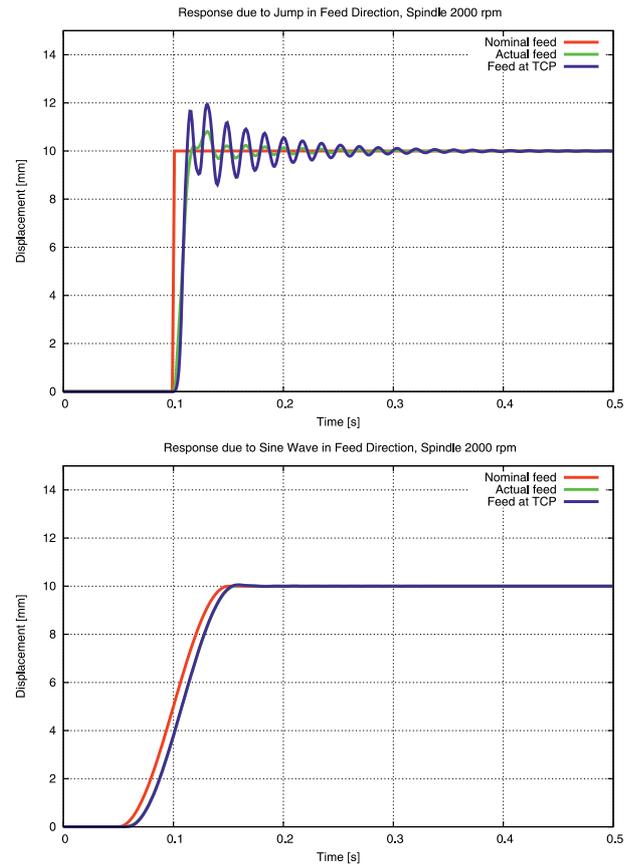


Figure 49: Time-history response due to excitation in Z direction for a sudden position jump (above) and a smooth position change (below) for the controlled machine

Fig. 48 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. 49 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:**

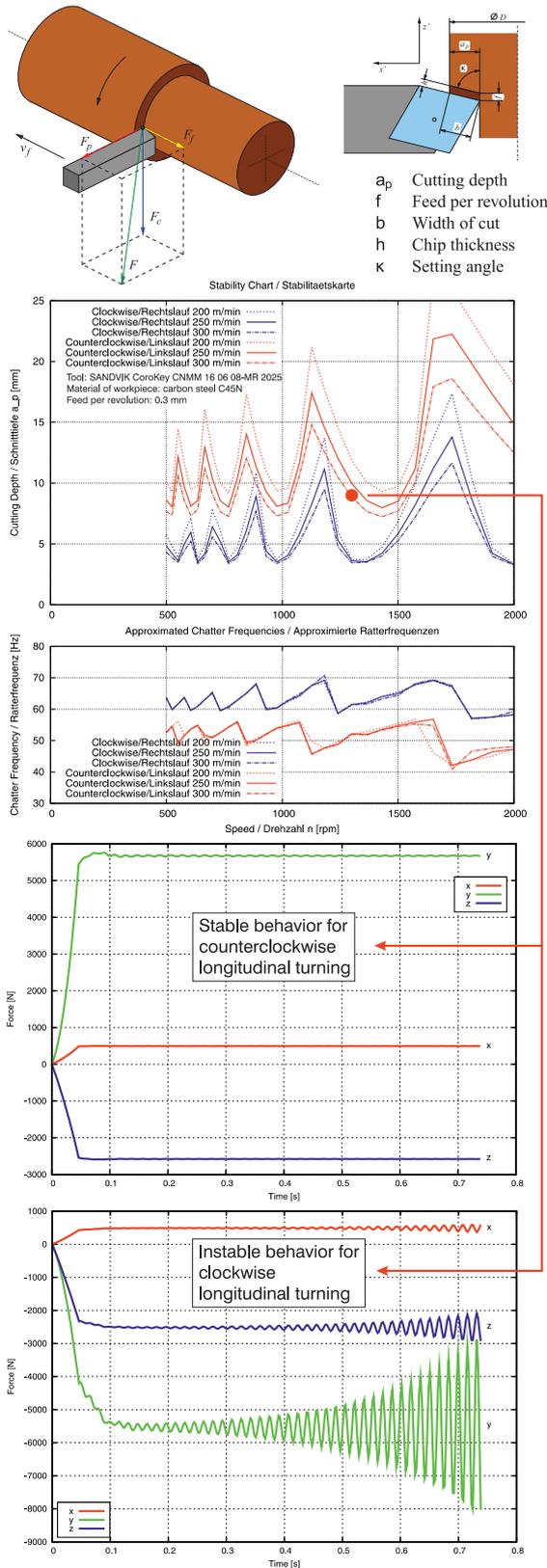


Figure 50: Stability chart for longitudinal turning with stable and unstable behaviour under counterclockwise and clockwise turning.

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 self-exciting 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. 50). 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. 50 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.

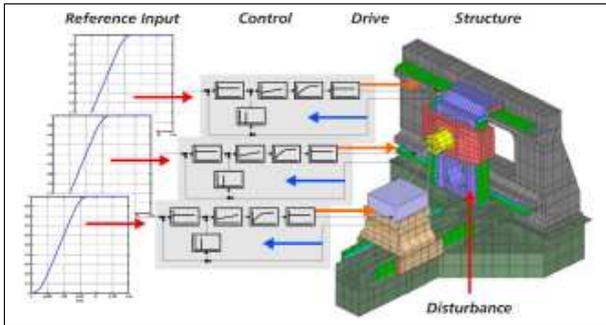


Figure 51: Principles of coupled analysis with control

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. 52). 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.

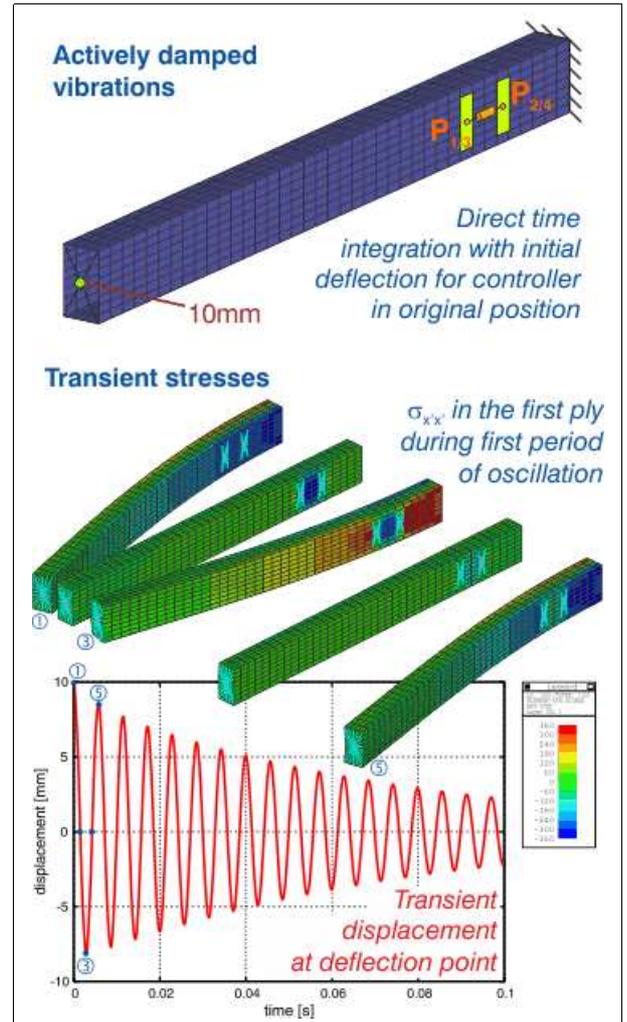


Figure 52: Active damping of a composite box girder

- 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 93).

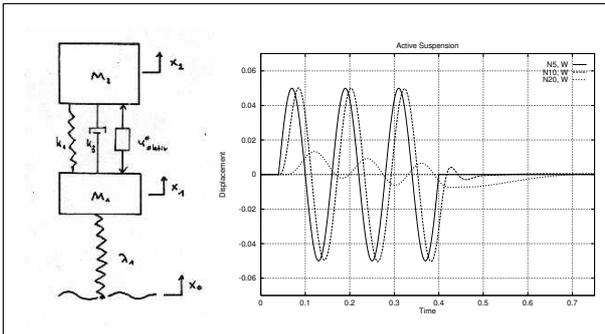


Figure 53: Active suspension unit

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.

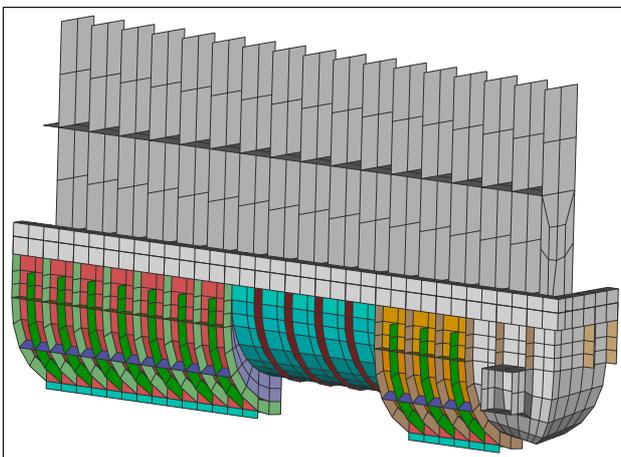


Figure 54: Optimization of a water box with 18 design variables and 19 stochastic basic variables

		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.

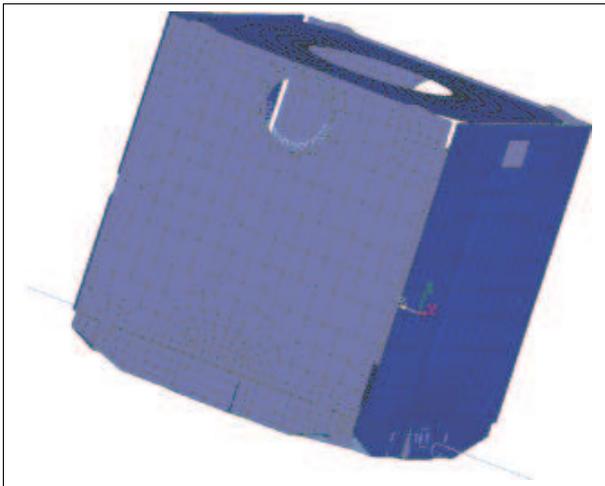


Figure 55: Optimization of PROTEUS satellite with 28 design variables and 30 stochastic basic variables (Alcatel Space S.A.)

		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 68. 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.

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 88.

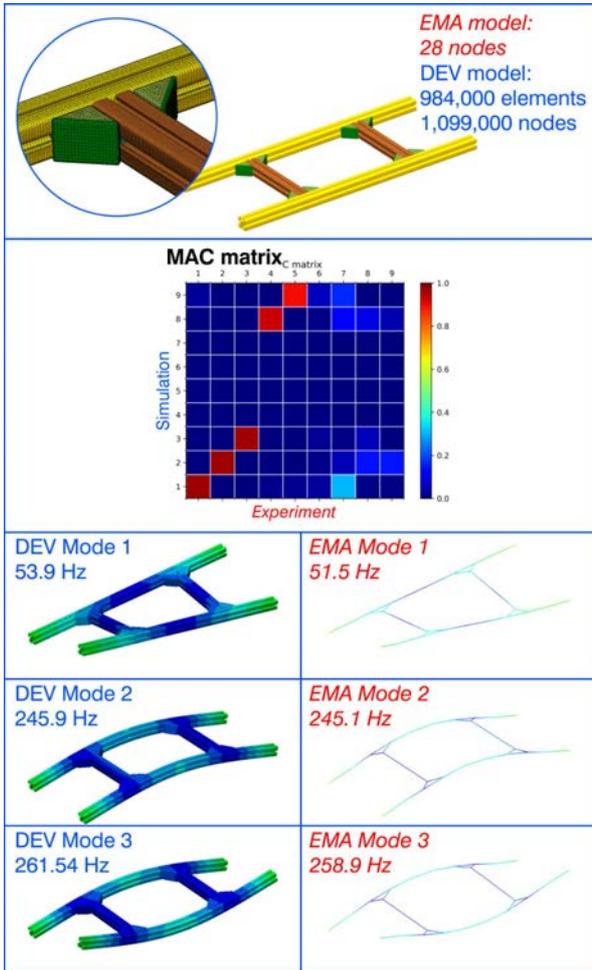


Figure 56: Example of EMA

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

Fig. 56 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 neighbored 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

VisPER



Figure 57: The logo of VisPER

The VisPER History

PERMAS as a software for numerical analysis provides many functions which are not sufficiently supported by general purpose pre-processors. So, for many years the developers of PERMAS have looked for a suitable *GUI* to support PERMAS as good as possible and to provide a more effective model description.

After three years of development the new graphical user interface VisPER (for Visual PERMAS) has been announced on the PERMAS Users' Conference early April 2008 in Stuttgart. The current version is VisPER Version 18 which is available with PERMAS Version 18.

VisPER – A Short Introduction

VisPER (Visual PERMAS) is a GUI based model editor. It is used to 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. 58).

VisPER can also be used as post-processor for PERMAS. In particular, post-processing is provided for those functions, where VisPER is used as model editor.

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,

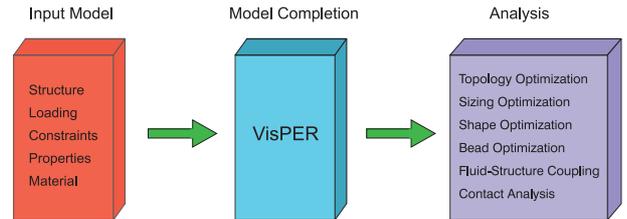


Figure 58: Model editing with VisPER

- Topology optimization,
- Sizing optimization,
- 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.

Fig. 59 shows the main window of VisPER. Each function can be accessed via pull down menus. Shortcuts are predefined for many functions which

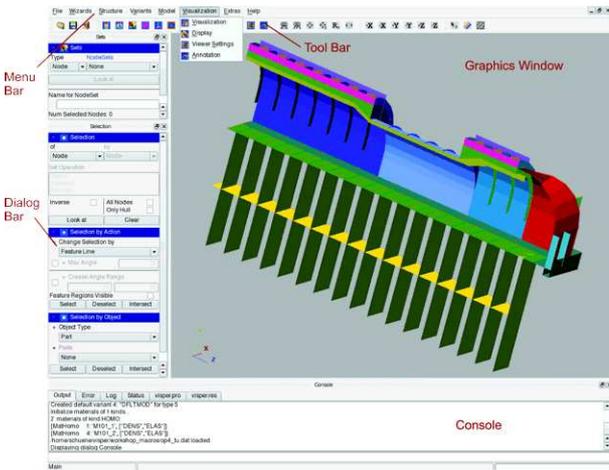


Figure 59: Main Window of VisPER

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),
- 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,

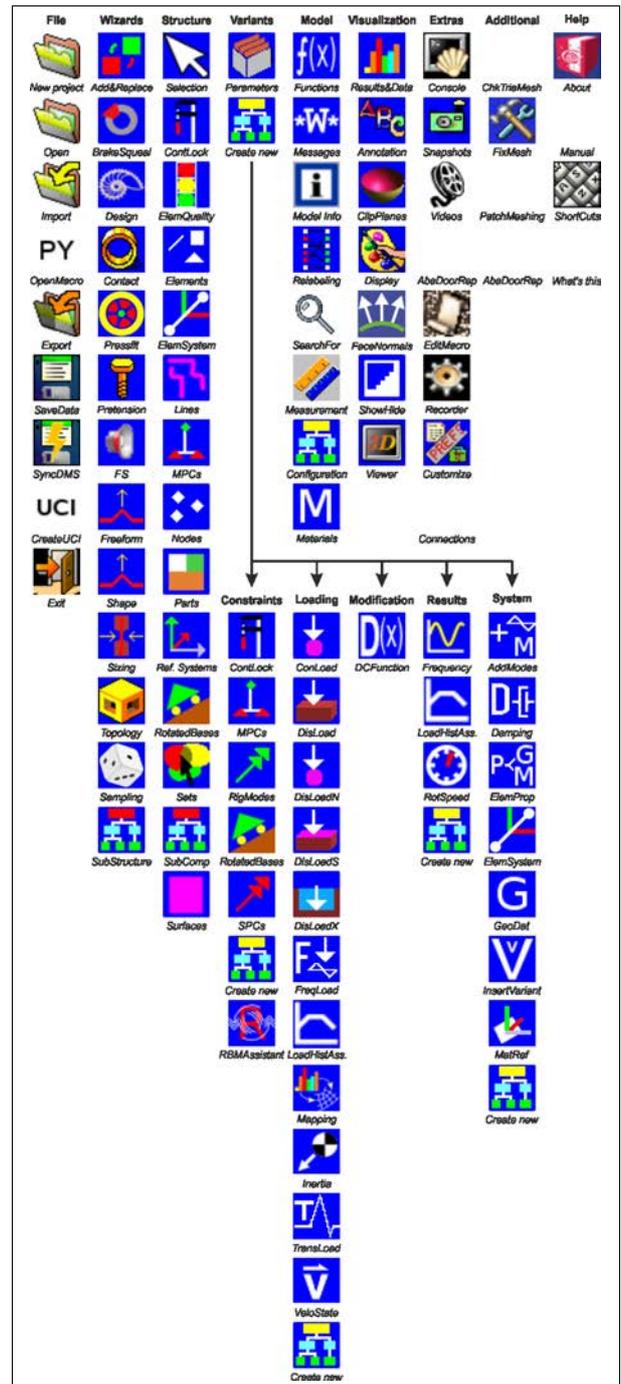


Figure 60: VisPER Menu with all top-level menu icons.

- 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. 60 shows the top-level menus of VisPER, which

gives a good overview on the supported PERMAS features.

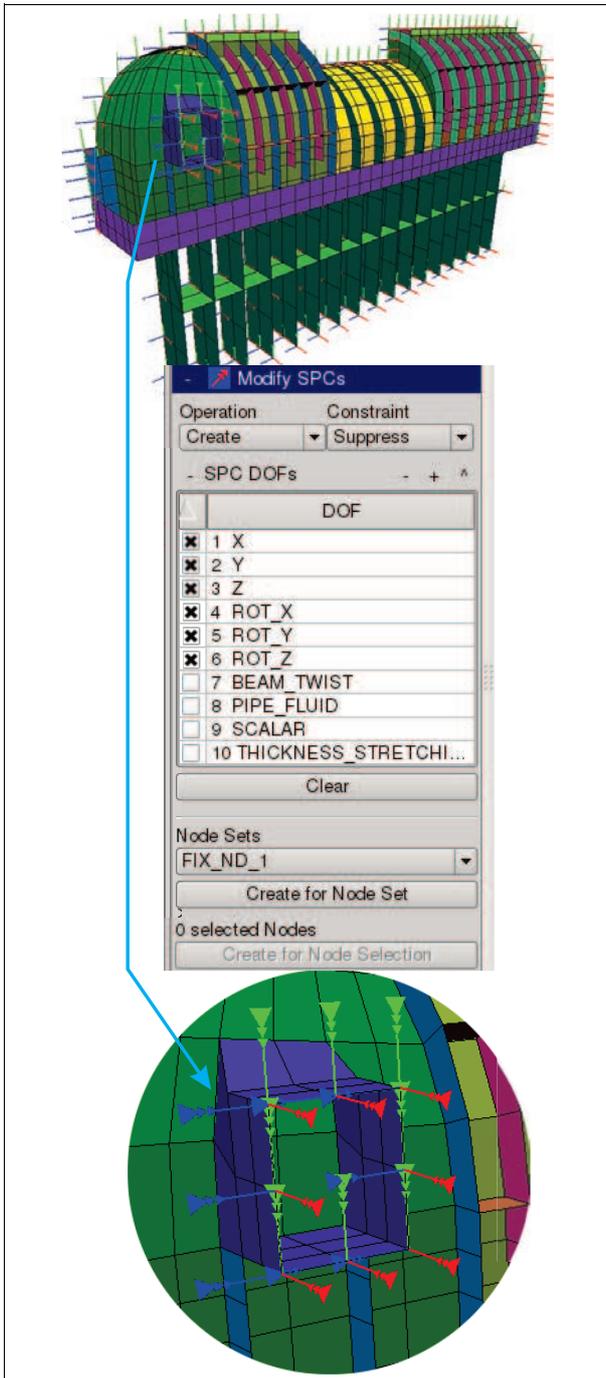


Figure 61: Specifying single point constraints

The following wizards are part of the basic module:

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

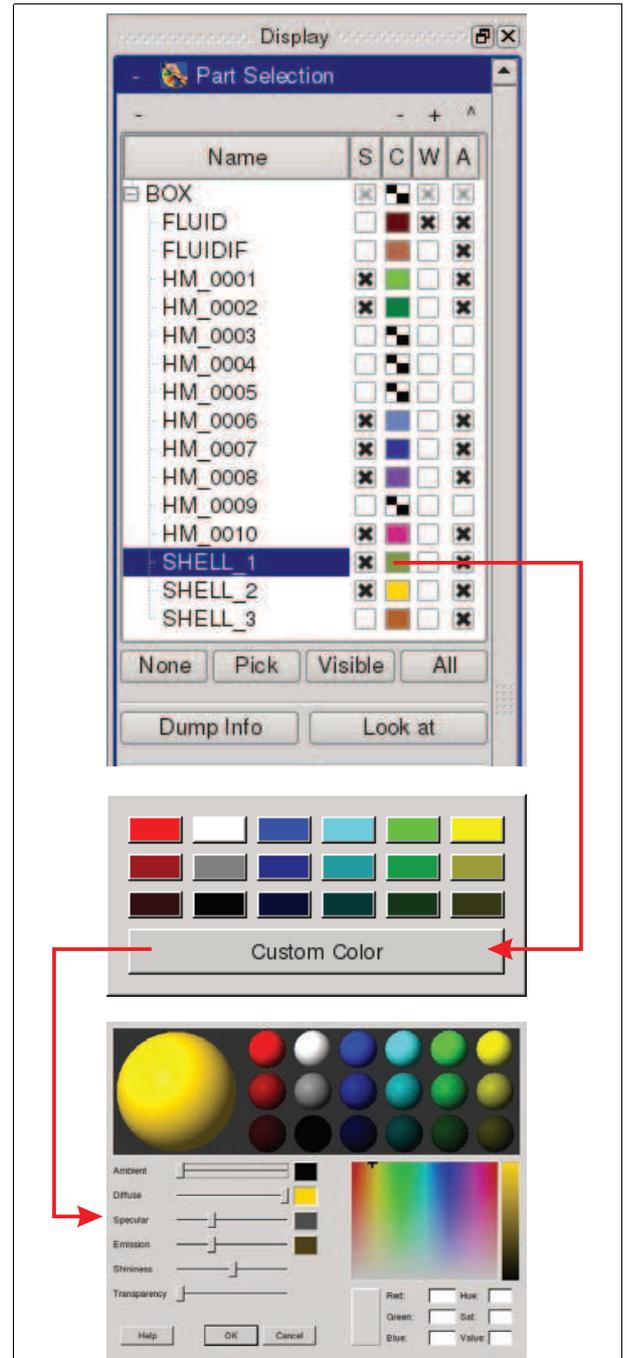


Figure 62: Graphics entities for parts and element groups (select S=Show, C=Colour, W=Wireframe, A=Active).

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

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 pre-tension, pressfit, etc..
- To replace an existing part in an assembly by a new part (see Fig. 24).
 - Recover existing sets and connections,
 - Fit surfaces to new geometry,
 - Relabeling of nodes and elements,
 - Manage replaced and new part simultaneously,
 - Keep model consistent.

The *Brake Squeal Wizard* allows an easy description of complex brake models according to PERMAS brake squeal technology (see Fig. 63). 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 compatibility,
- UCI file generation after wizard completion.

The *Design Wizard* supports a simulation driven design (see Fig. 18) 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 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 71).

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

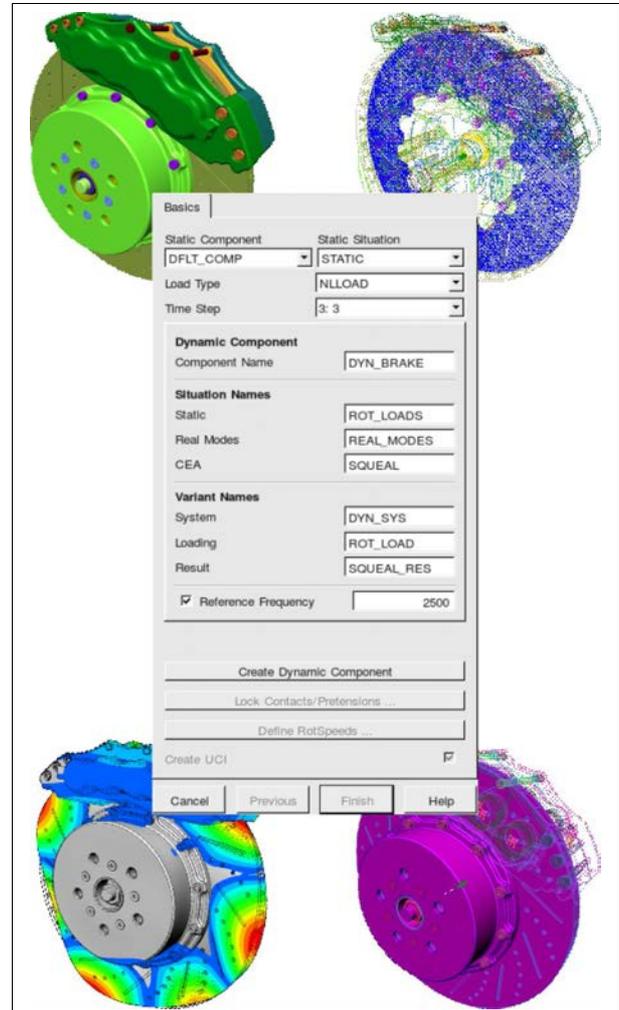


Figure 63: Preprocessing of brake models

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 75).

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

- 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.
- Information about the model is provided by a tree view of all processed input files.

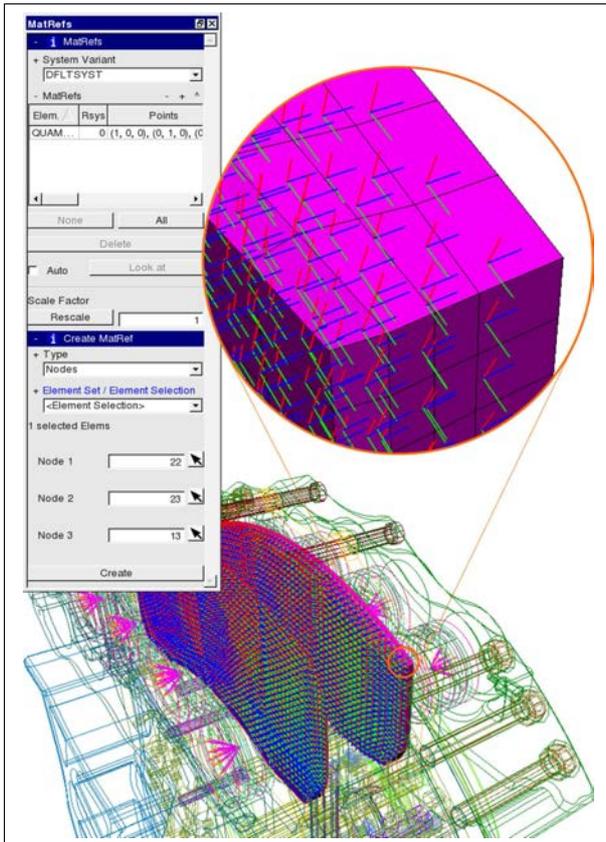


Figure 64: Material reference system for anisotropic material

- 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* functions, e.g.:

- Complete overview on all available results (see Fig. 65).
- 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.
- Viewing angles and direction as well as part se-

lection 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.

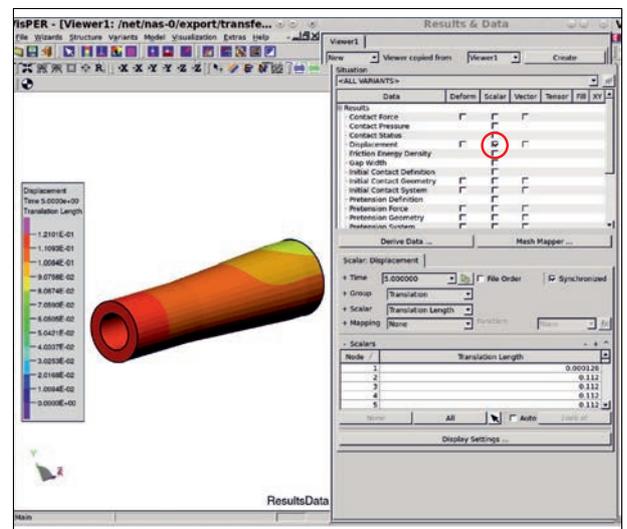


Figure 65: New post-processing

VisPER-TOP – Topology Optimization

This module provides a *wizard* supporting the set-up of an optimization model for topology optimization tasks. The details of topology optimization in PERMAS can be found on page 102.

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

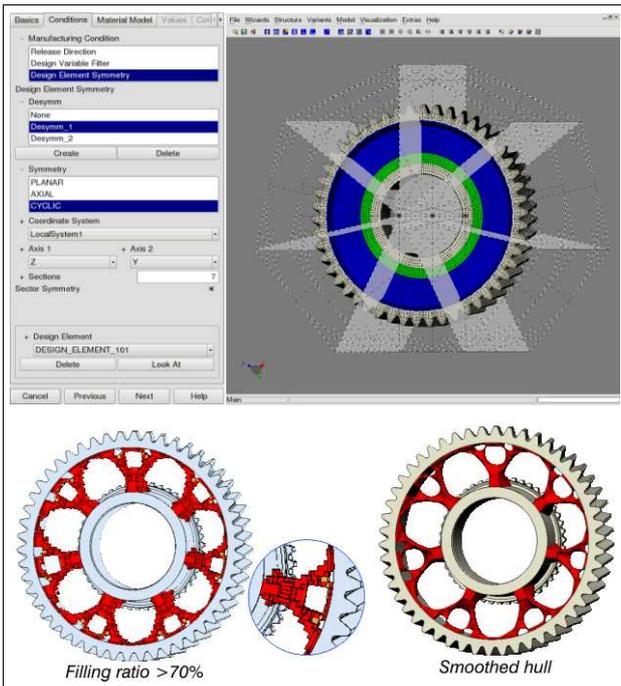


Figure 66: Topology optimization of a gearwheel
 Definition of cyclic symmetry and release directions, max. stress at interface between design space and gear rim, max. allowed weight, minimum compliance as objective function (by courtesy of Daimler AG, Stuttgart).

and maximum member size, and symmetry conditions (see Fig. 66 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 105 for details of these results). Fig. 66 also shows the smoothed hull of the gear wheel body.

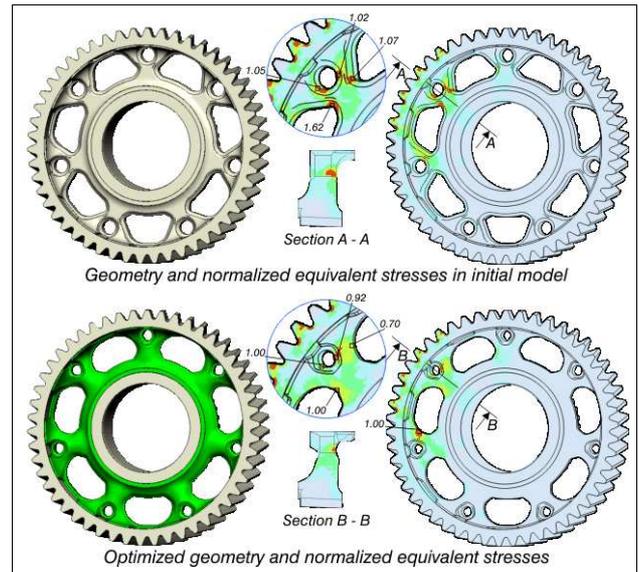


Figure 67: Shape optimization of a gearwheel
 Definition of cyclic symmetry, max. stress, minimum weight as objective function (by courtesy of Daimler AG, Stuttgart).

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 98.

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.
2. The conditions contain the definition of restraints for shape changes, restrictions on bead design, and symmetry conditions.
3. 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. 68 as example for the specification of shape basis vectors by *morphing*.
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.

A wizard for *free-form optimization* is available for a parameter-free shape optimization to reduce stress concentrations (see Fig. 69).

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

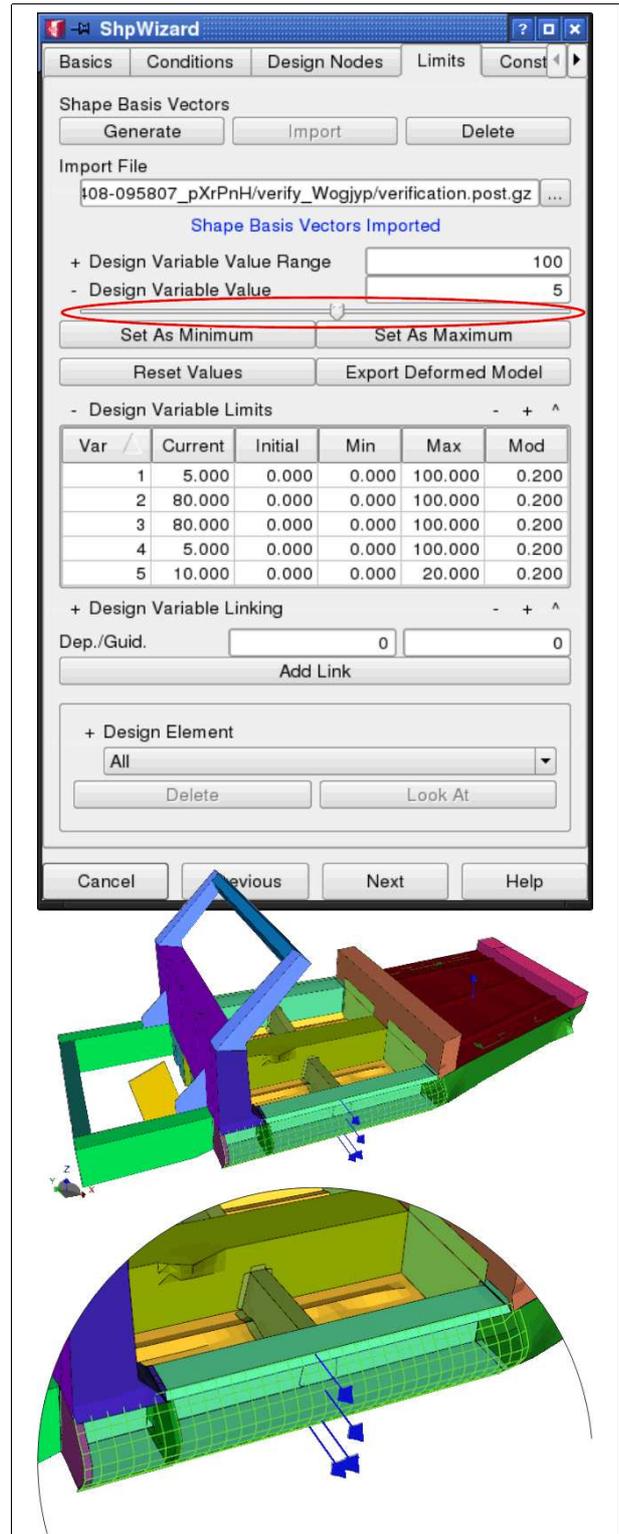


Figure 68: Definition of shape basis vectors of a car sill. The shape changes can be visualized by moving the slider in the menu.

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.

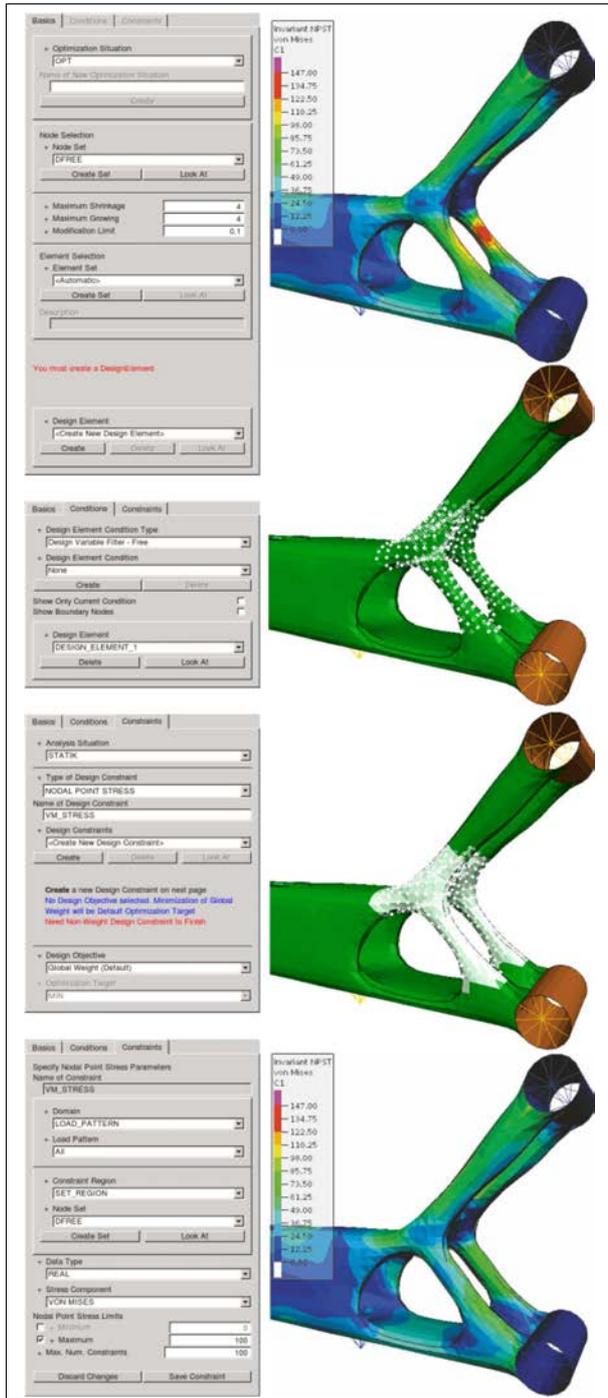


Figure 69: Pre- and post-processing for free-form optimization:

- 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
- 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.

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. 70 and 71:

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.

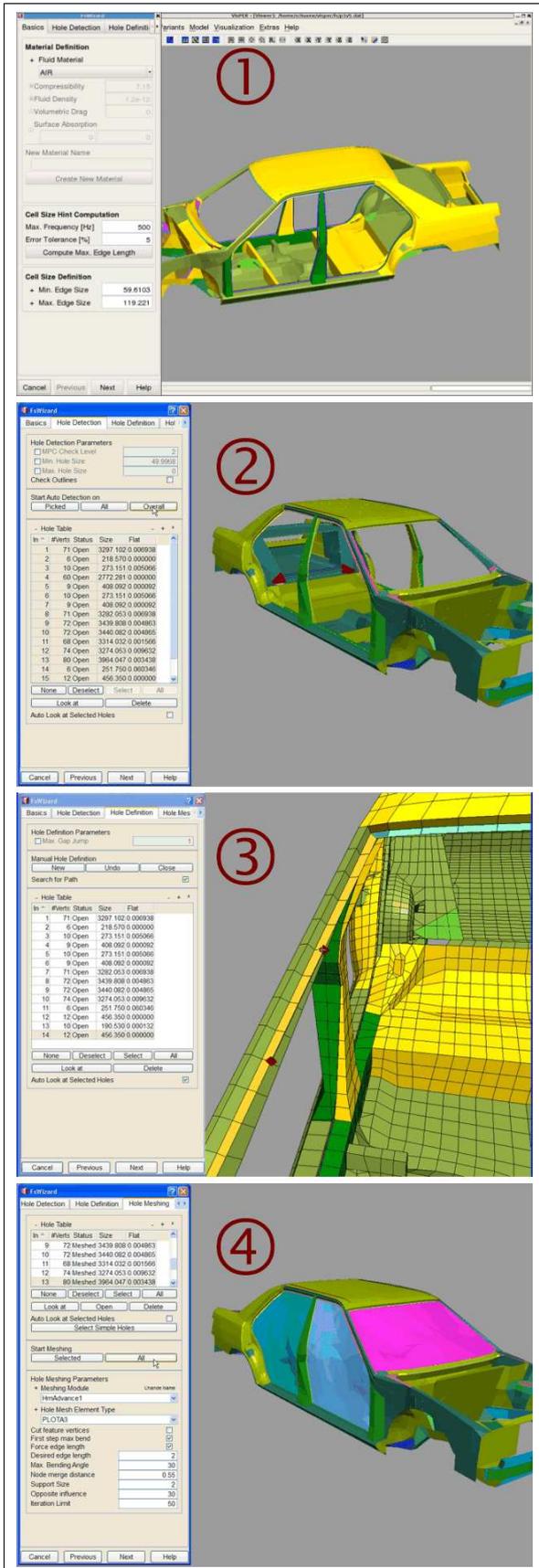


Figure 70: Steps 1 to 4 of the FS wizard

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 95.

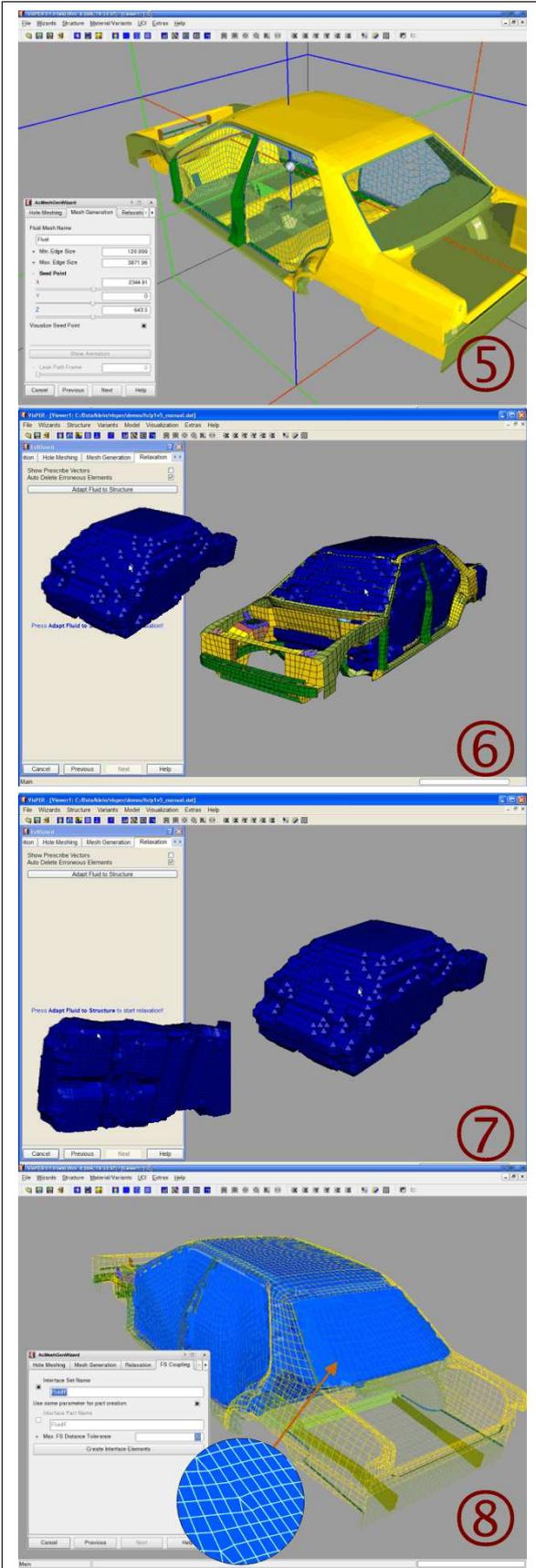
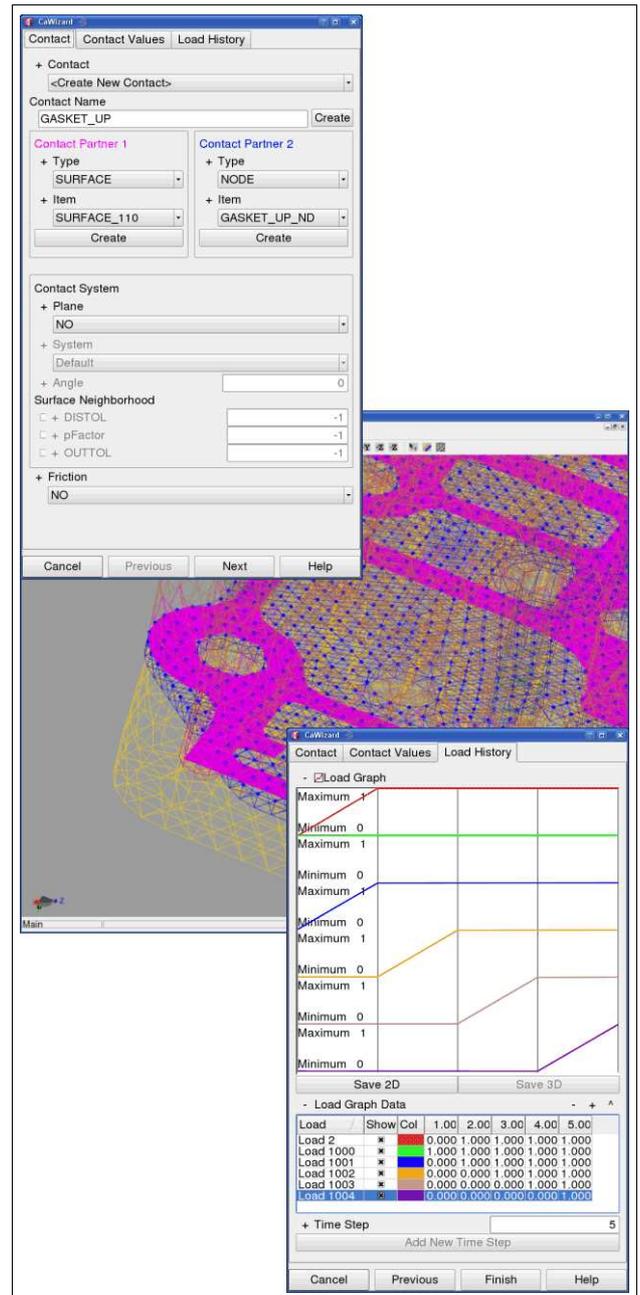


Figure 71: Steps 5 to 8 of the FS wizard



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 73.

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. 72.
- 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.

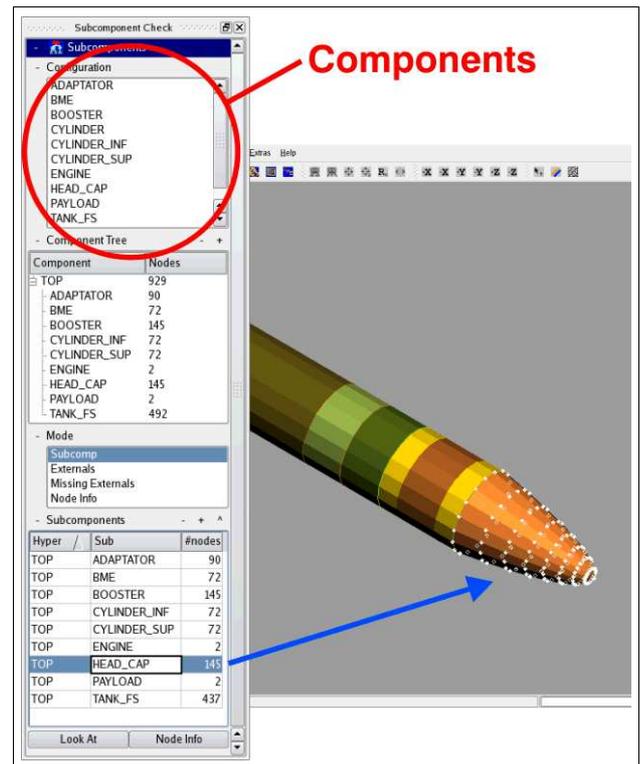


Figure 73: Navigation in the component tree of a substructure model

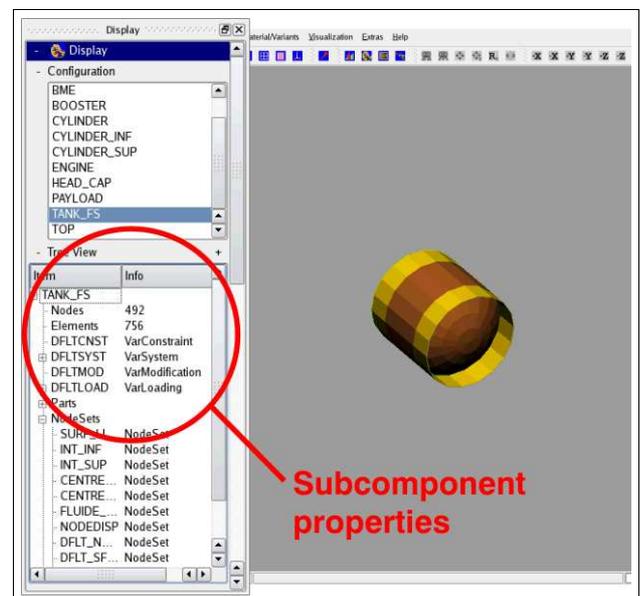


Figure 74: Visualization of components of a substructure model

Substructuring

A *substructure wizard* has been provided to facilitate the set-up of substructure models. This function is part of module VBAS (see page 40).

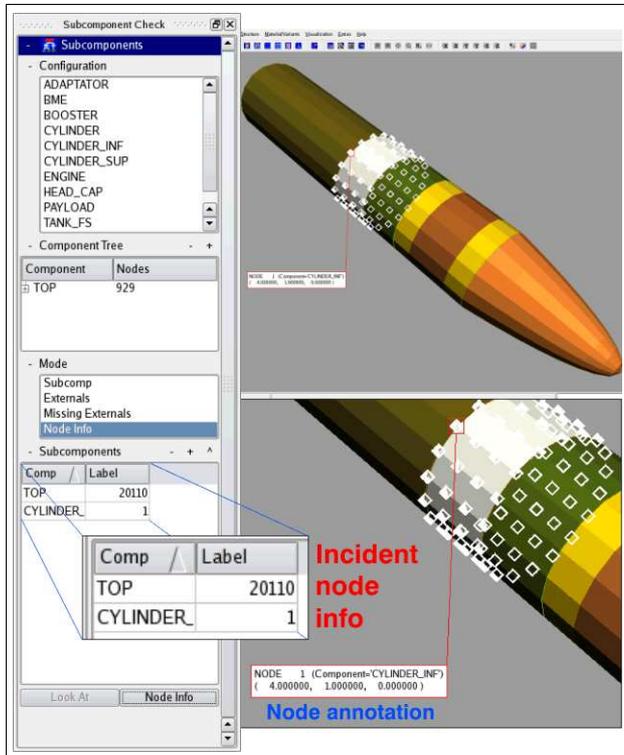


Figure 75: Visualization of coupling nodes of a substructure model

Because VisPER uses the data structures of PERMAS, substructure models can be directly imported. Substructuring is explained in more detail on page 53.

The verification of substructure models includes the following points:

- Navigation through the component tree (see Fig. 73).
- Visualization of single components (see Fig. 74).
- Visualization of coupling nodes (see Fig. 75).
- Check of element properties (see Fig. 76).
- Display of mode shapes on full structure (see Fig. 77).

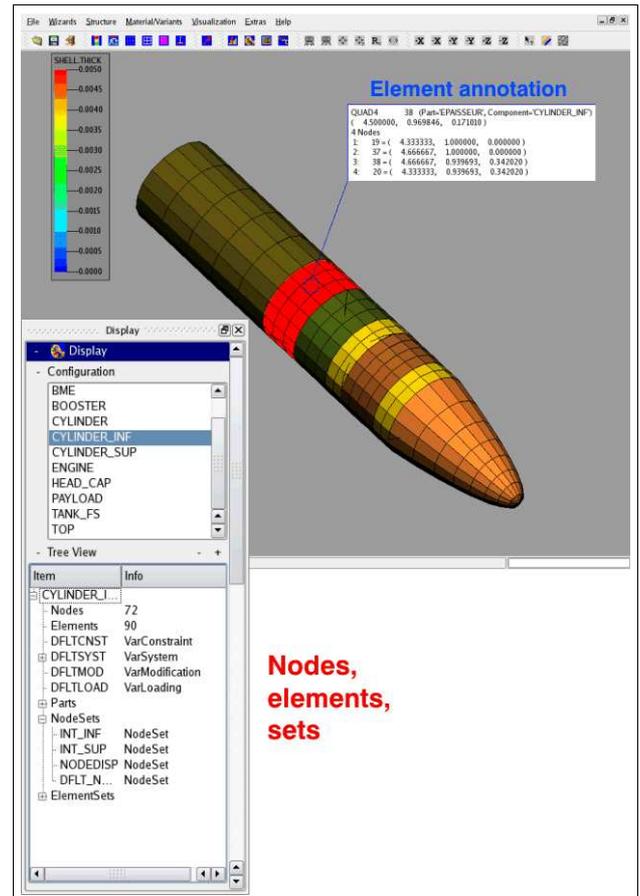


Figure 76: Checking element properties in a substructure model

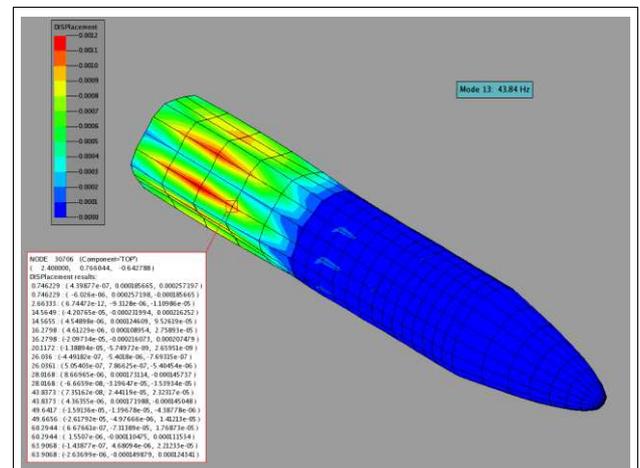


Figure 77: Display and animation of mode shapes on full substructure model

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 40).

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. 78). 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.

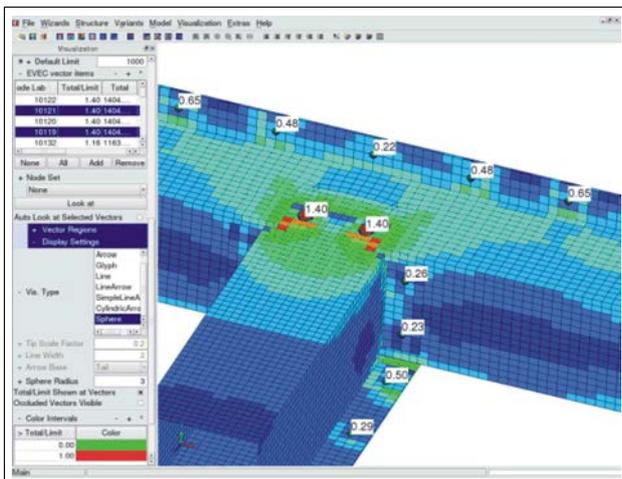


Figure 78: Evaluation of spotweld forces together with element stresses

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**.

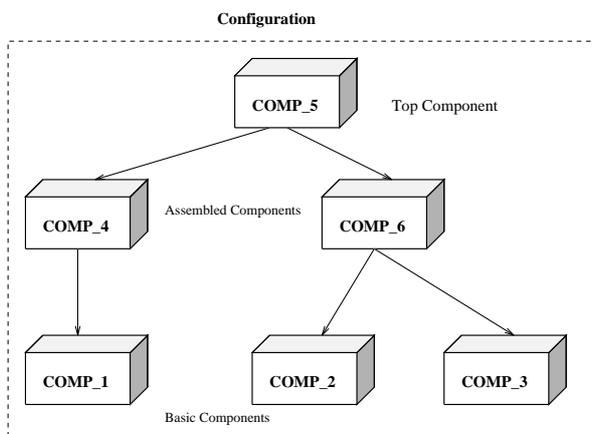


Figure 79: Substructuring in PERMAS

- 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 67).
- 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.

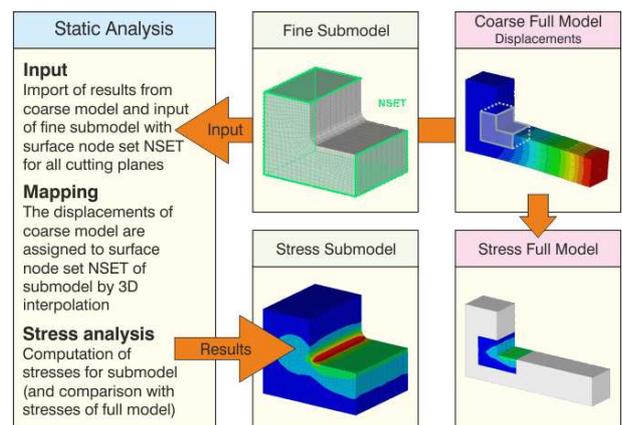


Figure 80: Submodeling in static analysis

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. 80).

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. 81).

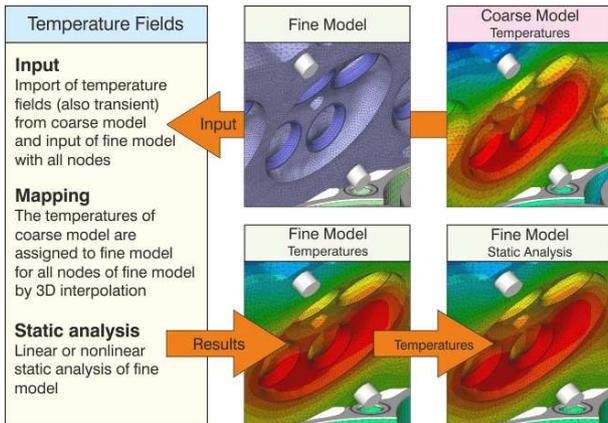


Figure 81: Mapping of temperature fields

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. 82).

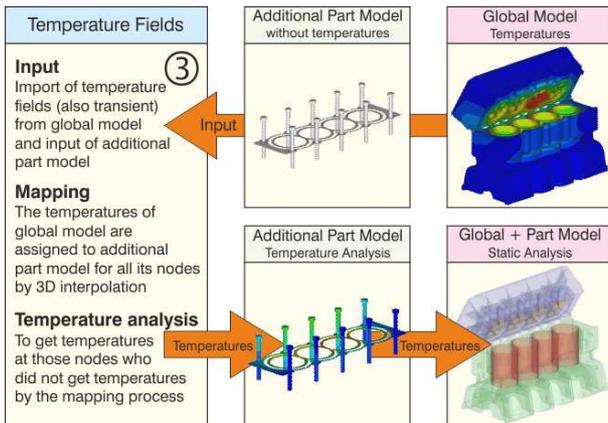


Figure 82: Mapping with extrapolation of temperature fields

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.

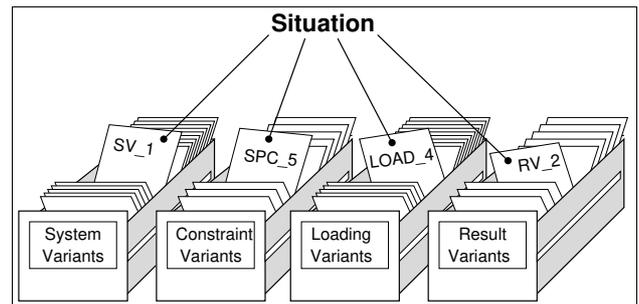


Figure 83: Variants in PERMAS

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.

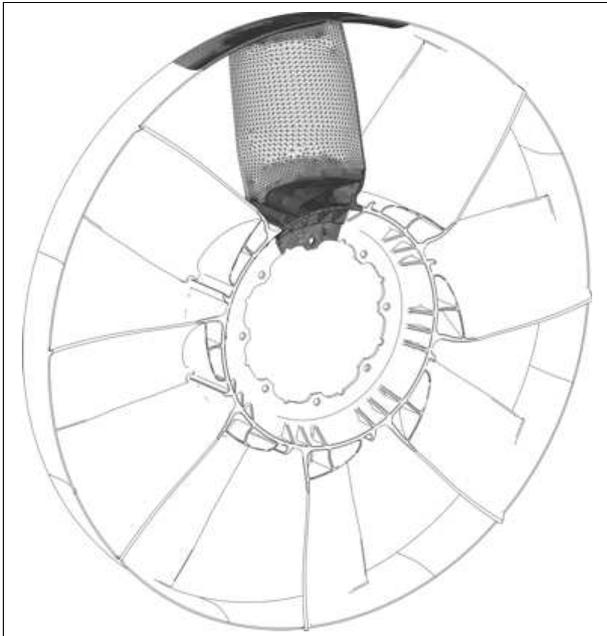


Figure 84: Cyclic symmetry of a fan
Behr GmbH & Co., Stuttgart, Germany.

Cyclic Symmetry

Cyclic symmetric structures like Fig. 84 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 73 and 88). 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 102) 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.

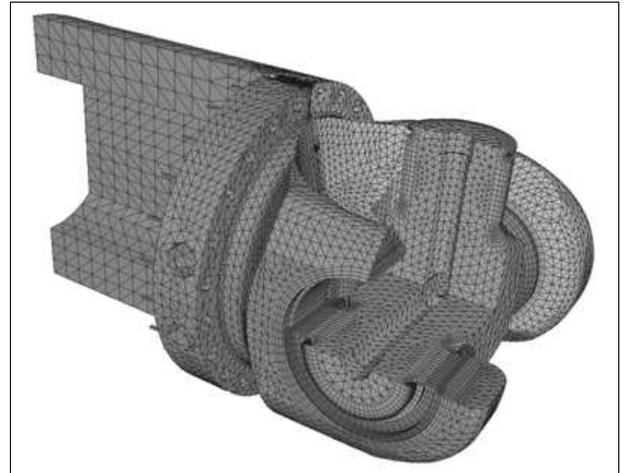


Figure 85: Incompatible meshes
Cardan shaft model in sectional view
(Voith Turbo GmbH & Co. KG)

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 63),
- by specifying geometry elements (see element library page 60).

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 63),
- by specifying geometry elements (see element library page 60).

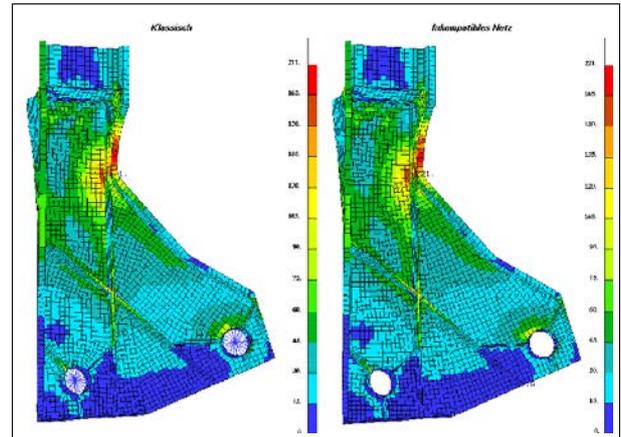


Figure 87: Ribbed shell model with compatible meshes (left) and incompatible meshes (right)

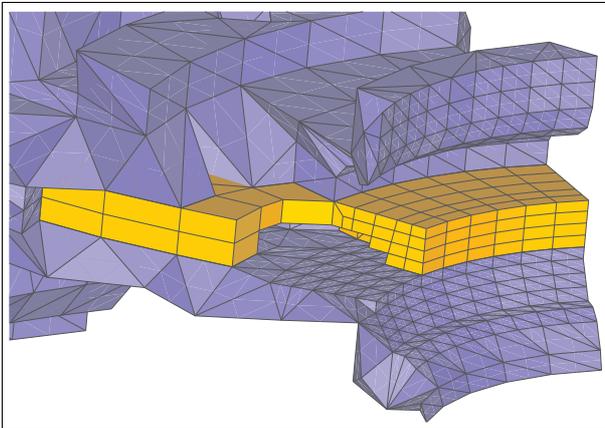


Figure 86: Incompatible meshes
Element transition (HEXE8/TET10)

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

- Definition of a dependent surface or node set

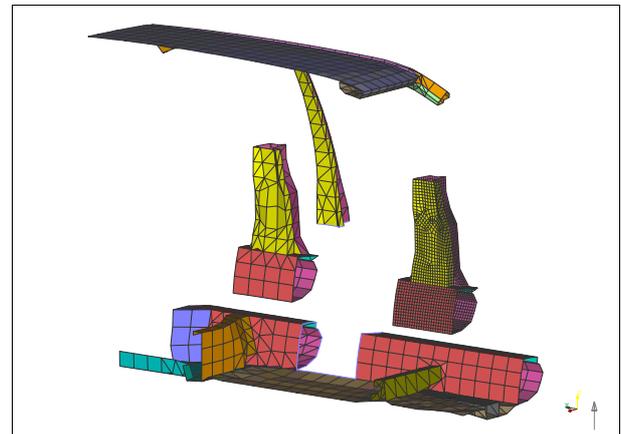


Figure 88: Application of local mesh refinement

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. 90 shows an example, where the initial position and the

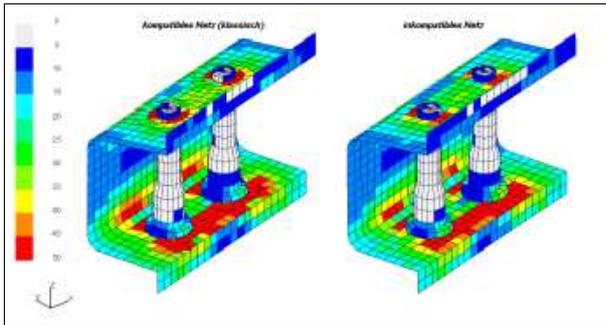


Figure 89: Results for part assembly with compatible (left) and incompatible (right) mesh

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.

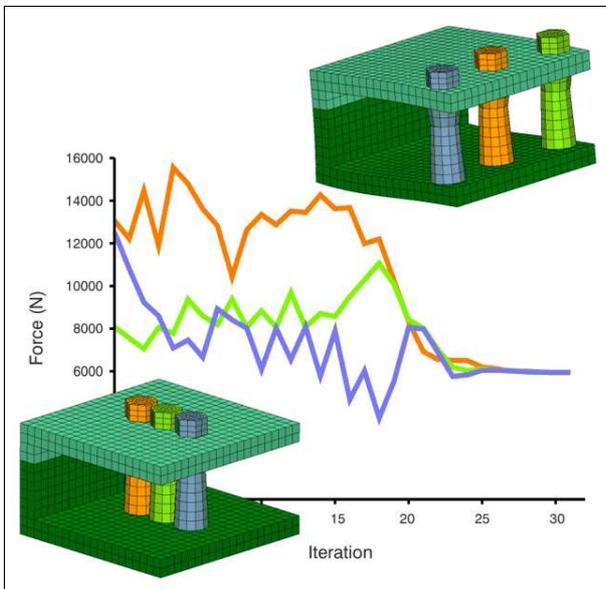


Figure 90: Shape optimization with incompatible meshes to achieve smallest relative displacements and identical bolt forces (top: initial design, bottom: final design; in between: bolt forces for each iteration)

Automated Spotweld Modeling

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

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

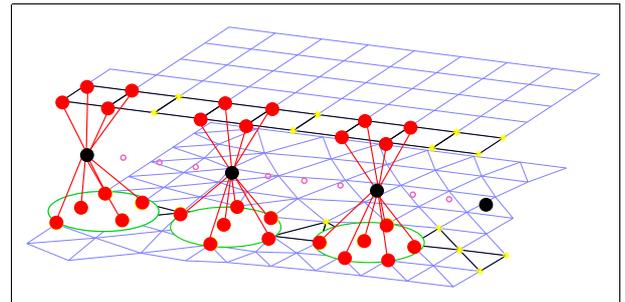


Figure 91: Connection of spotweld forces

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

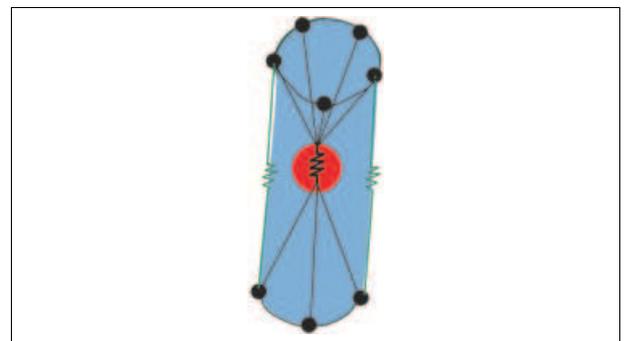


Figure 92: Spring element at spotweld position

The spotweld stiffness is modeled by a spring element, which is coupled to the neighboured parts by automatically generated *MPC* conditions (see Fig. 92). Available results are the spring forces and the reaction forces at the coupling nodes of the joint parts.

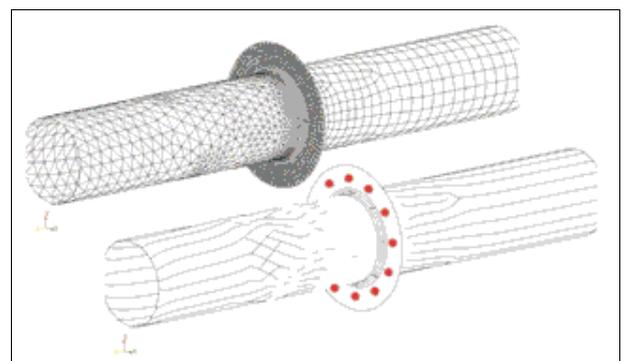


Figure 93: Spotwelds and incompatible meshes

For verification purposes the topology and the connecting vectors of the generated spotweld connections can be issued for graphical post-processing (see Fig. 94).

A more sophisticated spotweld model has been de-

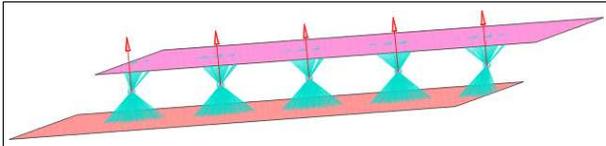


Figure 94: Verification of spotweld connections

veloped with module WLDS (see page 110). 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. 95. 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,

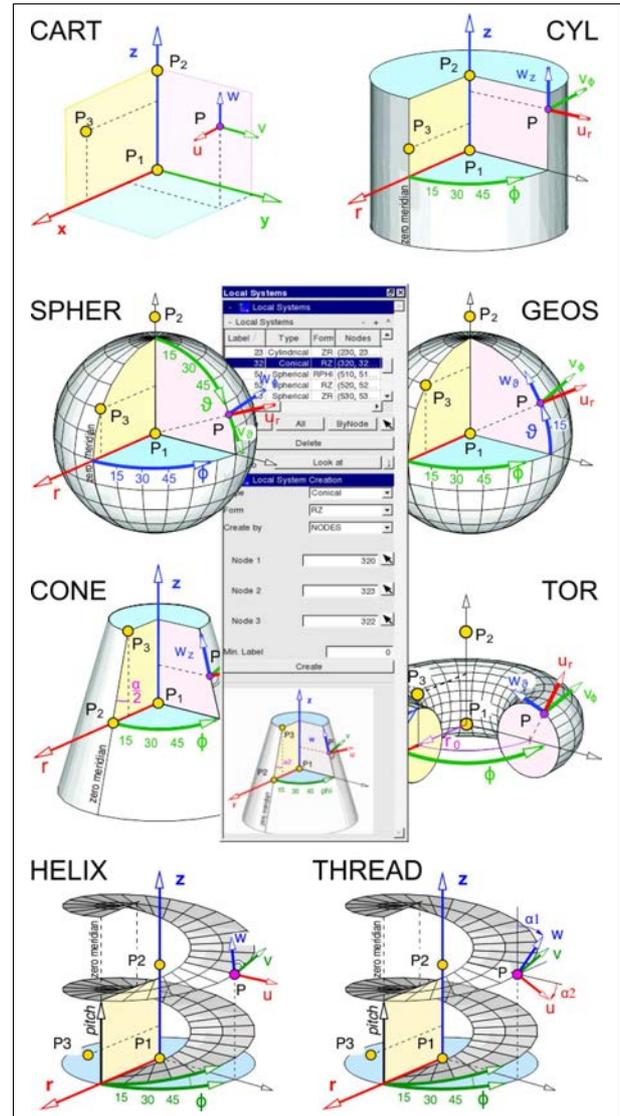


Figure 95: Local coordinate systems in VisPER

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. 96). 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.

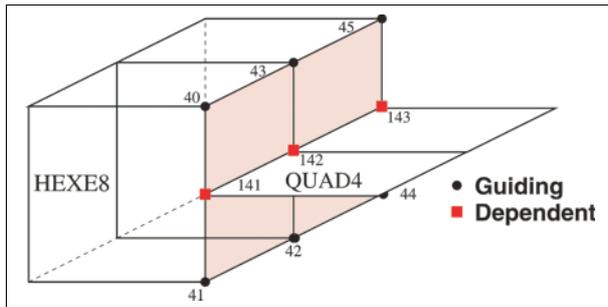


Figure 96: MPC example: Volume Shell Transition

- **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 are 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 identical results for corresponding models (see Fig. 97). A press fit command supports the automatic generation of the MPC conditions.
- **General MPCs** allow any linear combination of the involved degrees of freedom.

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

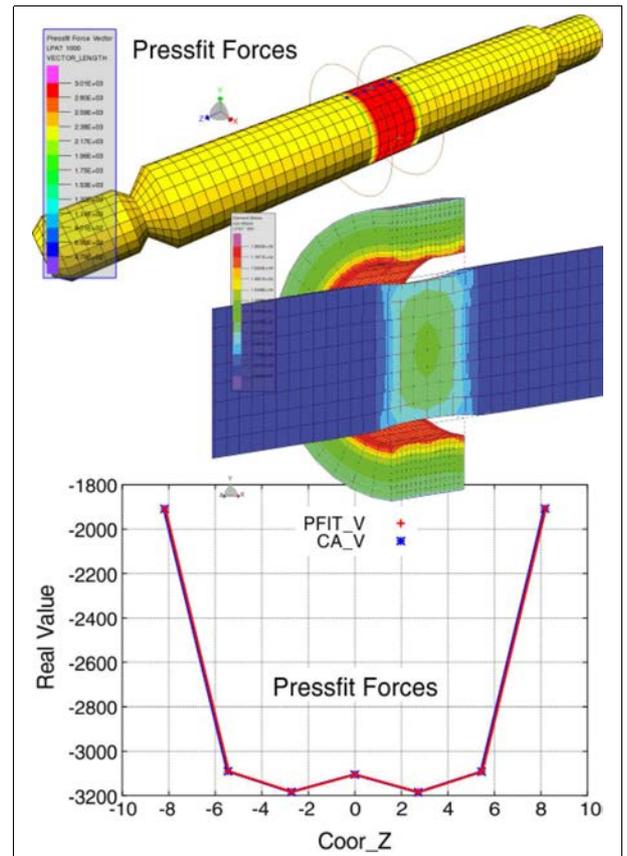


Figure 97: Press fit between shaft and hub
Identical stress results of MPC and contact solution.

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 42) 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 heaxaedra with 8 nodes for dynamic visco-elastic calculations (see page 94).
- **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.

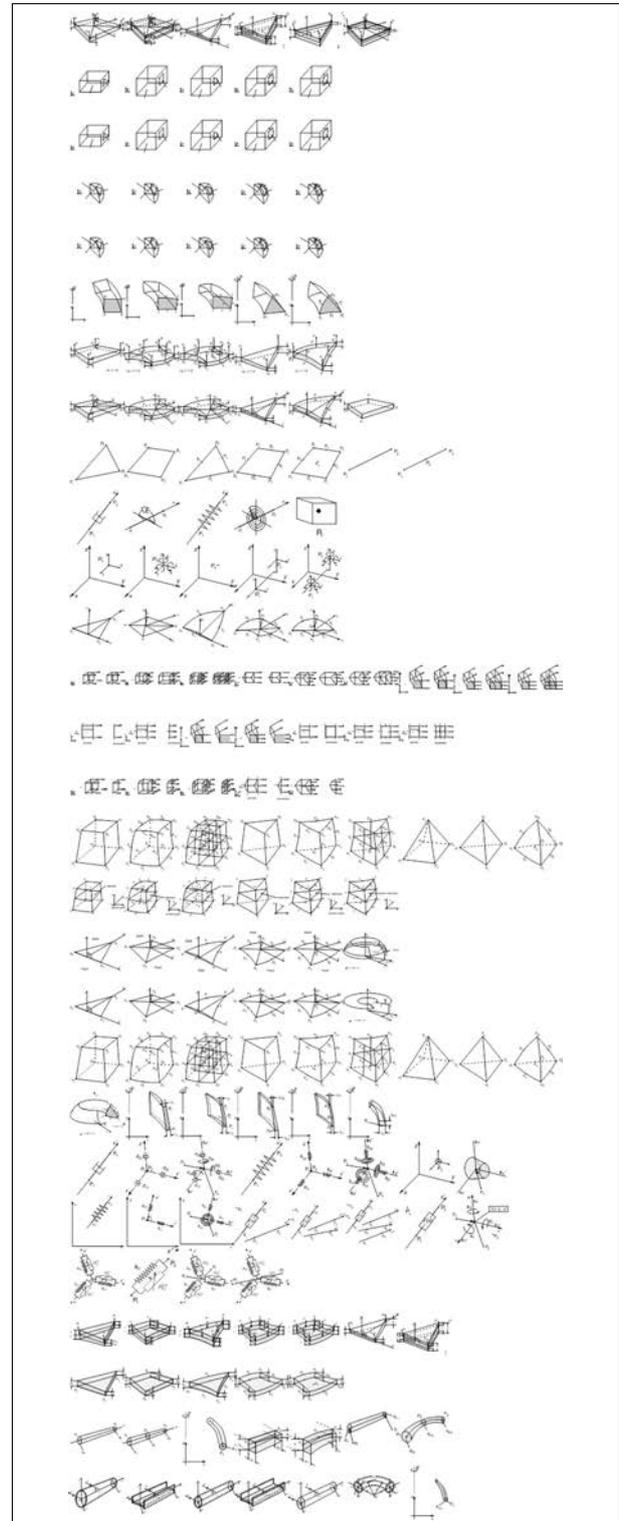


Figure 98: Overview on element types

- **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.
- **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,
 - 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 triangular 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) Ele-**

ments 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. 99):

- 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 98).

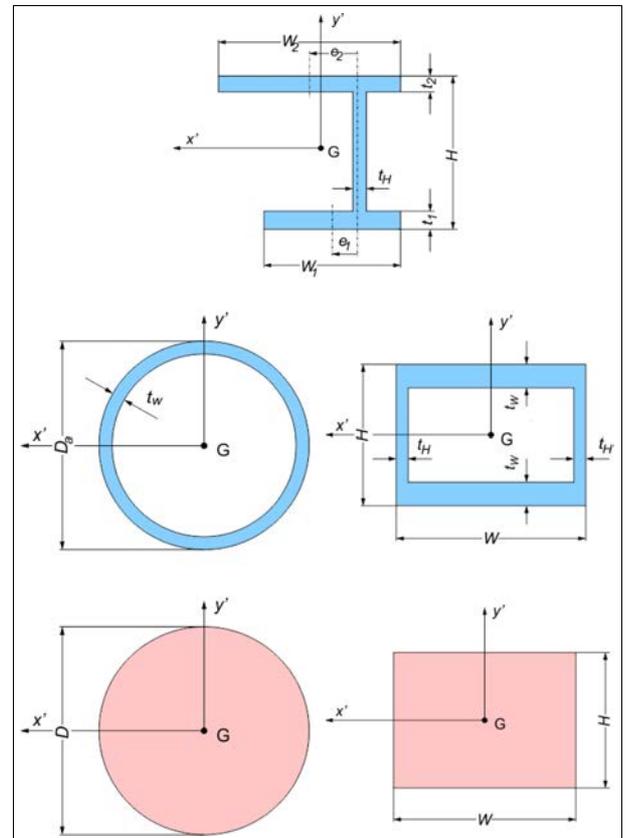


Figure 99: Standard beam cross sections

Design Elements for Optimization

For design optimization purpose (see page 98), 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 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 published by *Zienkiewicz*. This method aims to get more accurate stresses by using a patch of neighboured elements instead of just one element (see Fig. 100). 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.

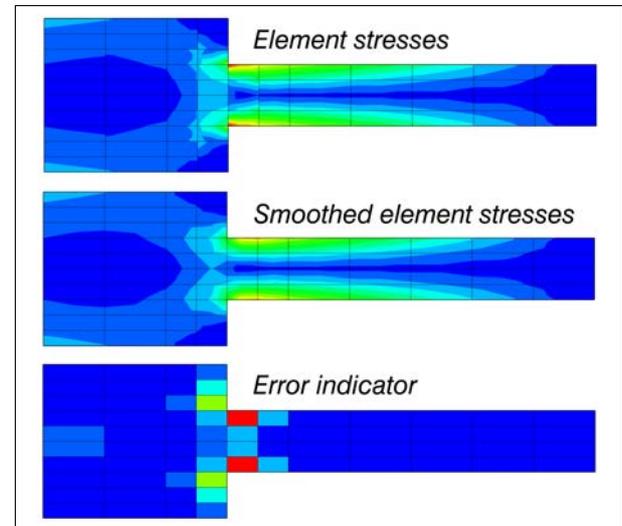


Figure 100: Element stresses, smoothed element stresses, and absolute error indicator

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 (trigonal) 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 deter-

mined 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 109).

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

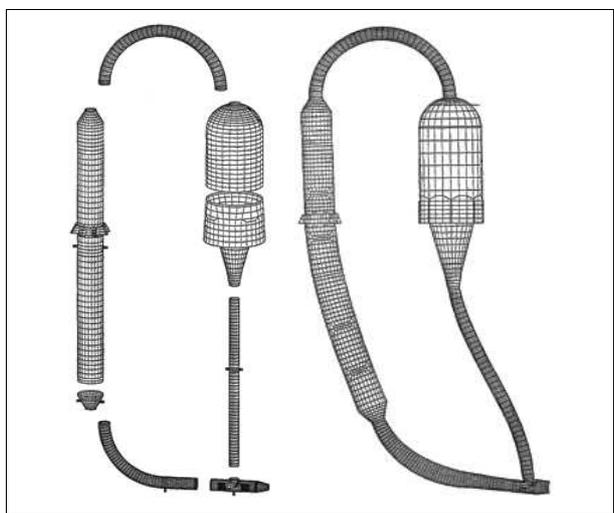


Figure 101: Mode shape of a reactor

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.

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. 102).

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. 103).

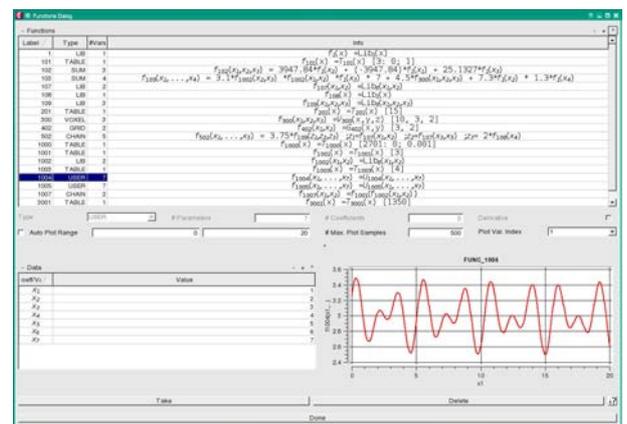


Figure 102: Functions dialogue in VisPER

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 61) can be used which are coupled to the main structure by interpolation (see page 58).

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. 103). 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.

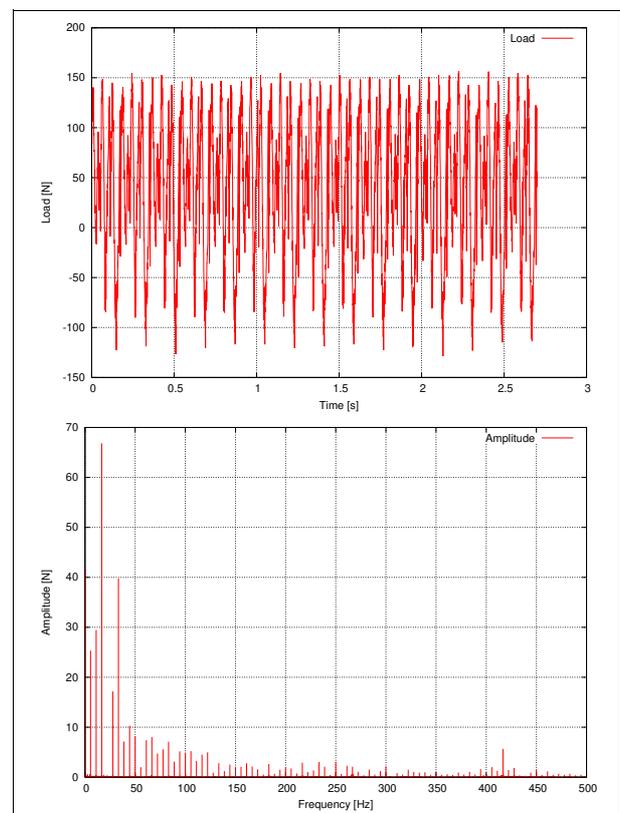


Figure 103: Example for the transformation of a periodic load (above in time domain, below in frequency domain)

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 58) 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. 94).
- 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. 104).
- 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 43 and 44). 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 75), 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 39)
- MEDINA (page 113)
- PATRAN (page 113)
- I-DEAS (page 113)
- ADAMS (page 114)
- SIMPACK (page 114)
- EXCITE (page 114)
- MOTIONSOLVE (page 114)
- HYPERVIEW (page 114)
- VAO (page 115)
- Virtual.Lab (page 115)
- ADSTEFAN (page 115)
- MATLAB (page 115)
- NASTRAN (page 115)
- ABAQUS (page 116)

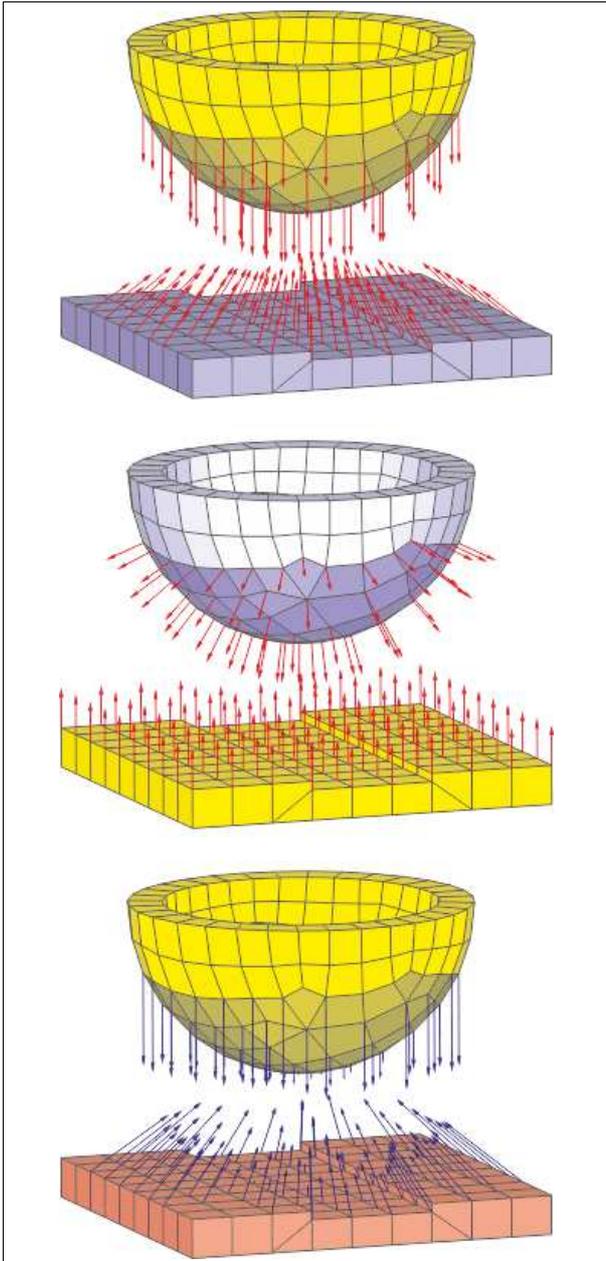


Figure 104: Verification of a contact model

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

- MpCCI (page 117)

Moreover, a growing number of extra interfaces to PERMAS are available from partner companies or INTES (see page 117).

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:

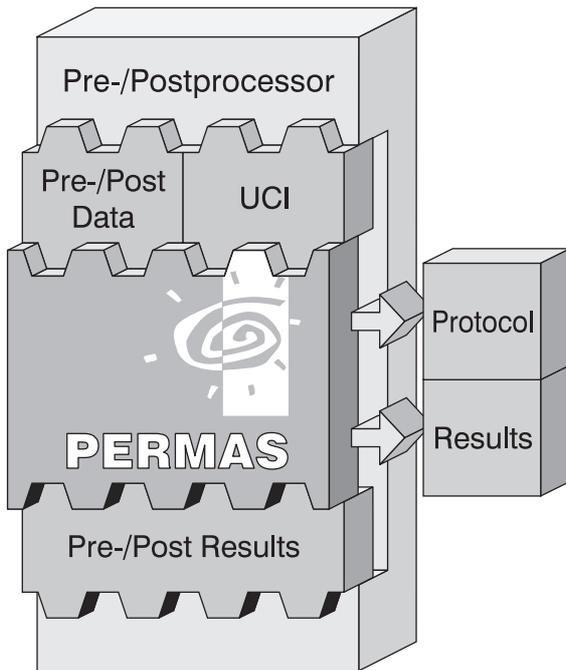


Figure 105: Integration of PERMAS
in pre- and post-processor

- 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 89). 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 63) 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, cylindrical, 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:

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

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 37).

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 75), material curves for plasticity, and transient loads in timehistory response analysis.

Within *PERMAS Tools* plots can be created using *PERMASgraph* (see page 122 for details).

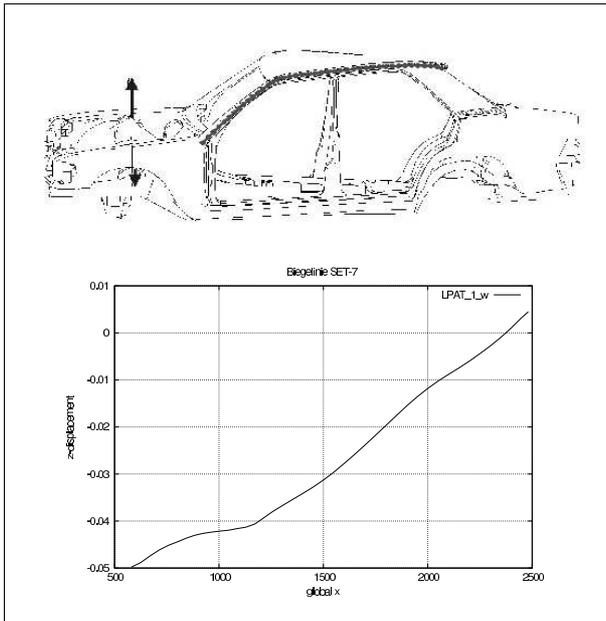


Figure 106: Determination of elastic line for torsional load case (displacement for the marked nodes)

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. 107).

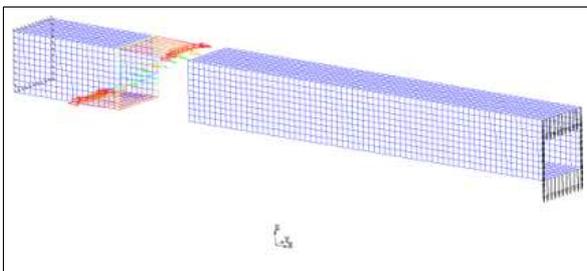


Figure 107: Determination of cutting forces

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 63).

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.

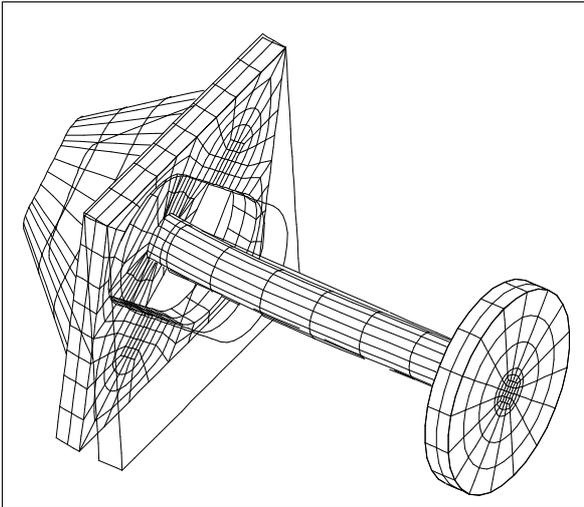


Figure 108: Clutch Element, Static Displacements

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.

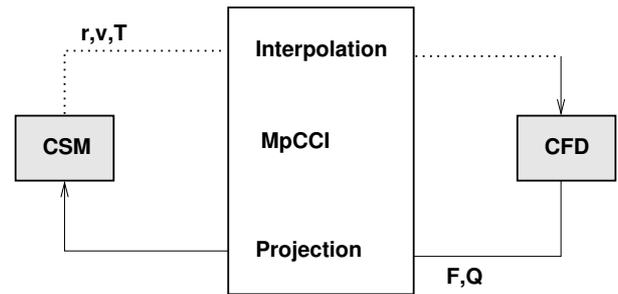


Figure 109: Coupling of CFD and CSM

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 \mathbf{F} or pressure resp., heat flux \mathbf{Q} , displacement \mathbf{r} , velocity \mathbf{v} , tem- perature \mathbf{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.

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. 43. So, all possible design variables (see page 98) or basic variables (see page 107) can be used for sampling and all analysis steps supported by optimization (see page 98) and reliability analysis (see page 107) 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 59).
- **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 65).

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 Package TM / Thermo-Mechanics

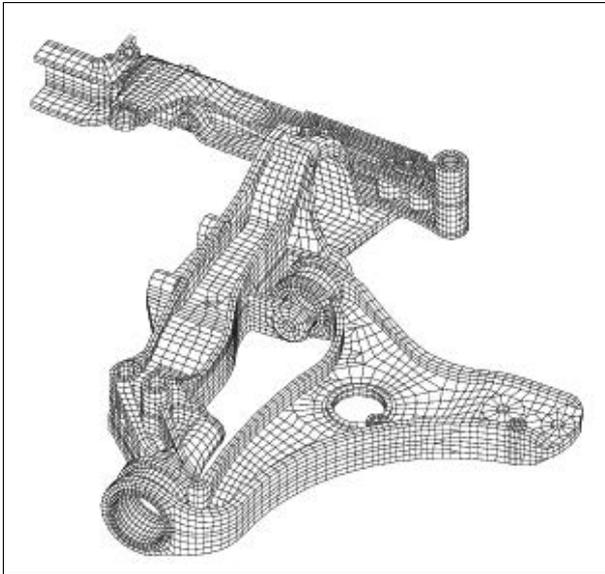


Figure 110: Model of a front axle
Porsche AG, Weissach

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

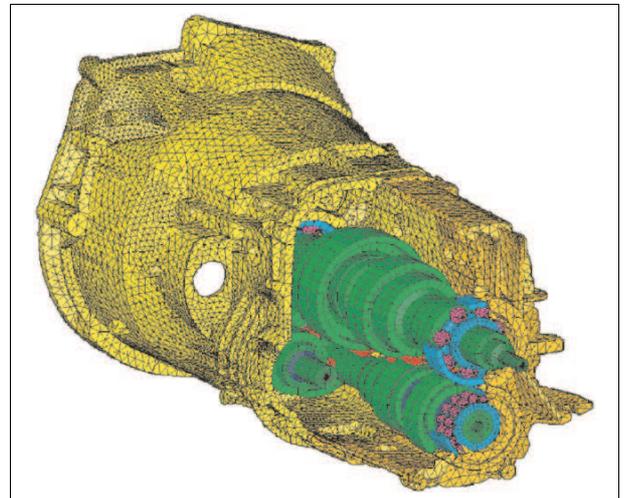


Figure 111: Bearing housing ZF AG

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 58).
- 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 64).
- The following primary results are calculated:
 - Displacements,
 - Mass and moments of inertia.
- From that additional results are derived:
 - Reaction forces,
 - Stresses and stress resultants,

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.

In addition *self contact* is supported by contact analysis (see Fig. 123), 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. 112).

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 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 59).

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-CAX provides a suitable extension of contact analysis (see next section). Both modules together cover a wide application area for contact analysis. This combination is complemented by module PERMAS-CAU (see section after next section) which takes into account large relative displacements of contacting bodies.

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. 114).

The load history can be amended by pretensioning (e.g. of bolts), where the contact analysis is used to

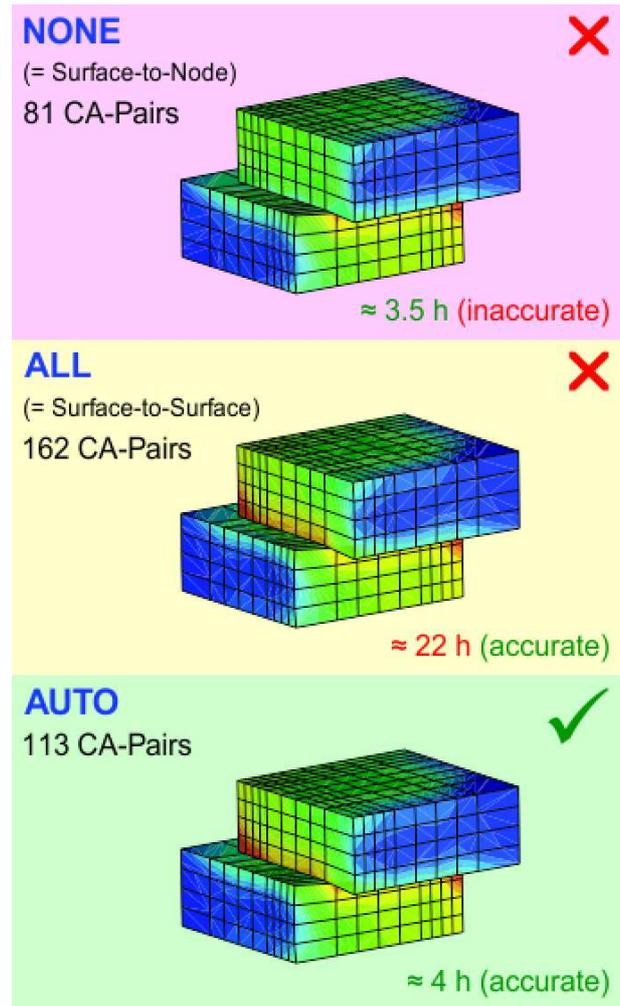


Figure 112: Surface-to-surface contact with COMPLEMENT option: Best compromise between accuracy and computation time (von Mises nodal stresses are shown). The numbers are taken from an industrial application.

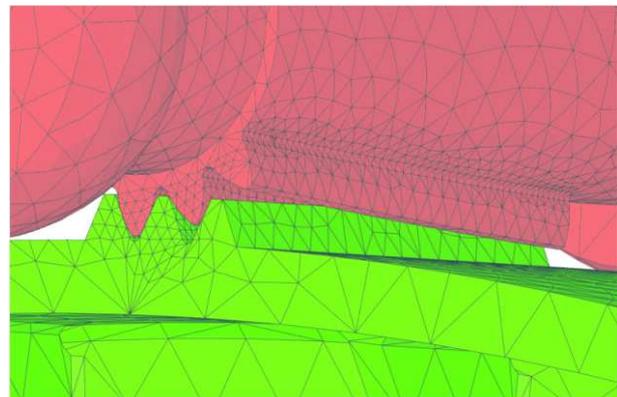


Figure 113: Contact with incompatible meshes

describe the pretension. In this way, the screw tightening torque is modeled by a known contact force in

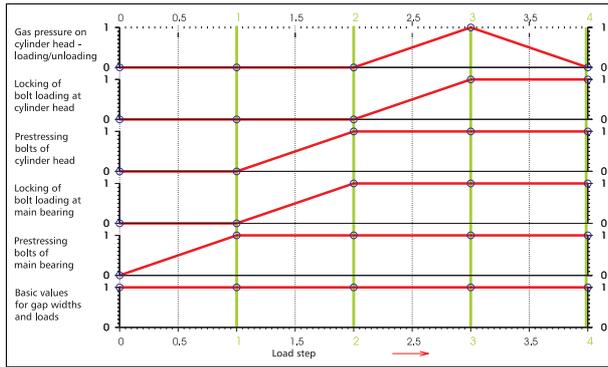


Figure 114: Representation of a load history

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. 115). In addition, Fig. 116 shows an application on 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 contact force (see Fig. 117). 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 42).

Comprehensive checks allow the verification of contact models like type of contact, its geometry (gap-width and normal vector, see also Fig 104), 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 degrees of freedom. This procedure has the following

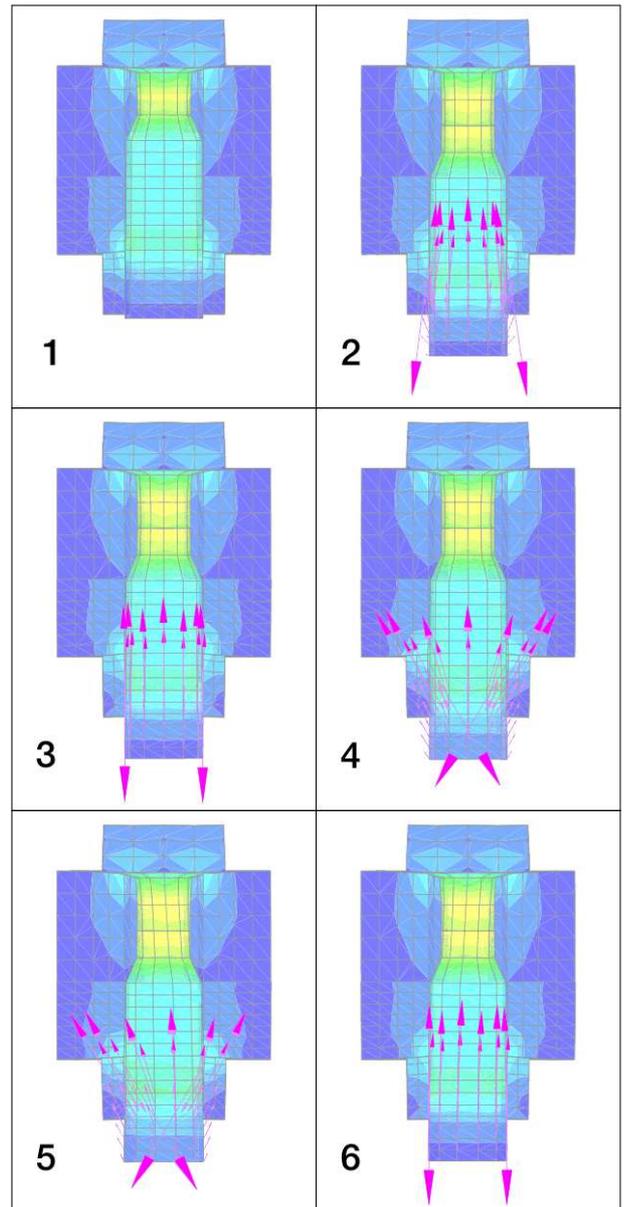


Figure 115: Bolt pretension Nodal point stresses and pretension forces for deformed mid-size screws:

- (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

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 intro-

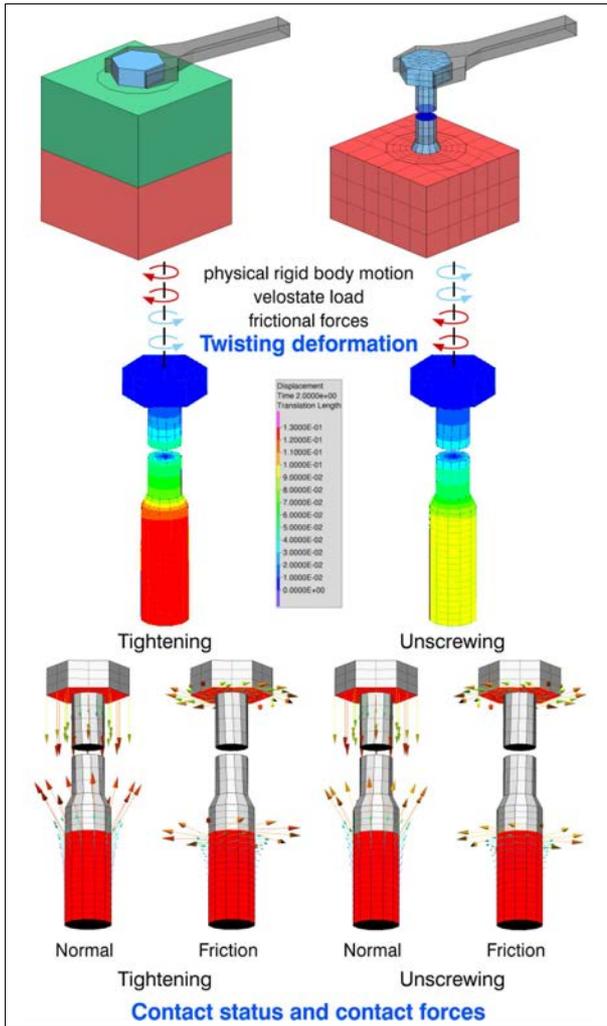


Figure 116: Thread definition on bolt surface.

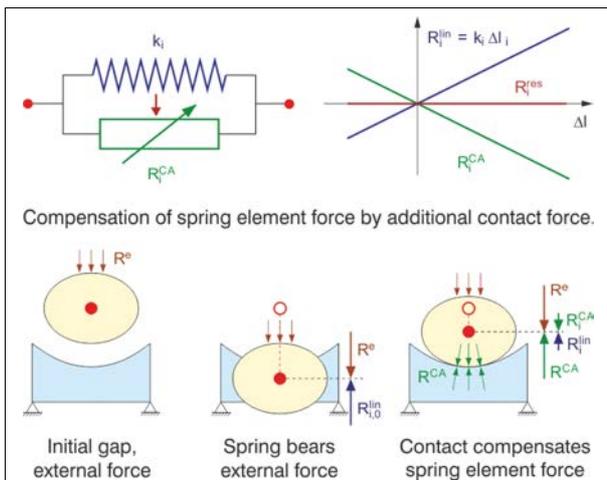


Figure 117: Force-guided contact with the aid of compensation spring

duced by the modeling of contacts.

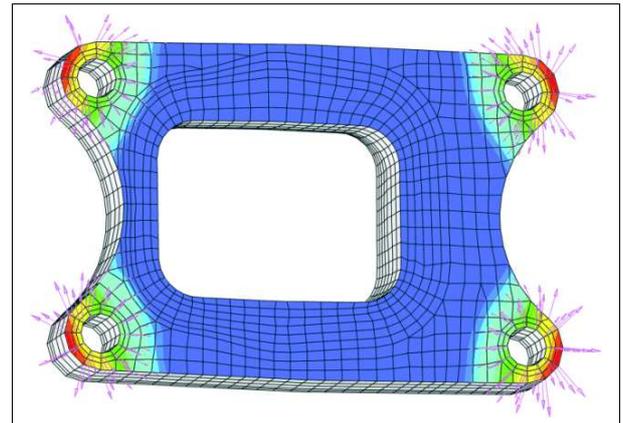


Figure 118: Contact pressure and shear vectors

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. 118), 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. 119, where the stress result is almost the same with and without midside nodes.

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.

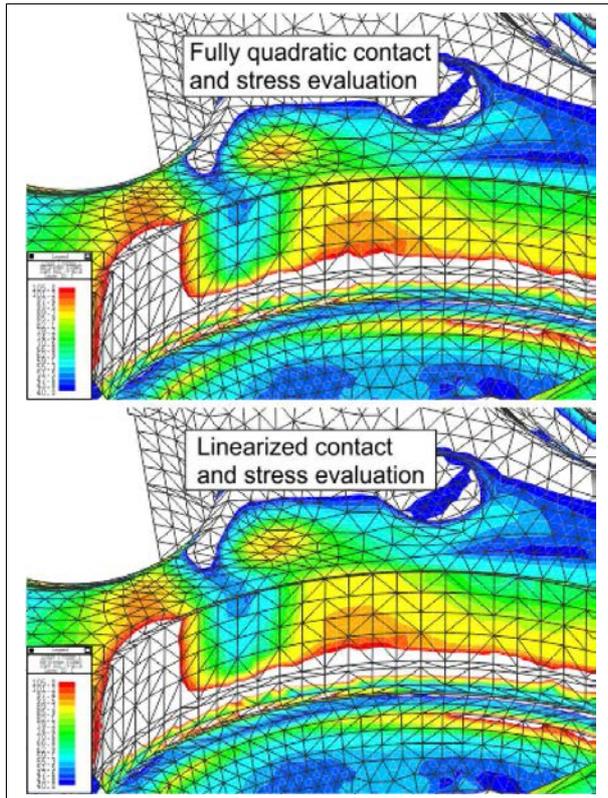


Figure 119: Nodal point stresses on surface with linearization (bottom), and without linearization as reference (top).

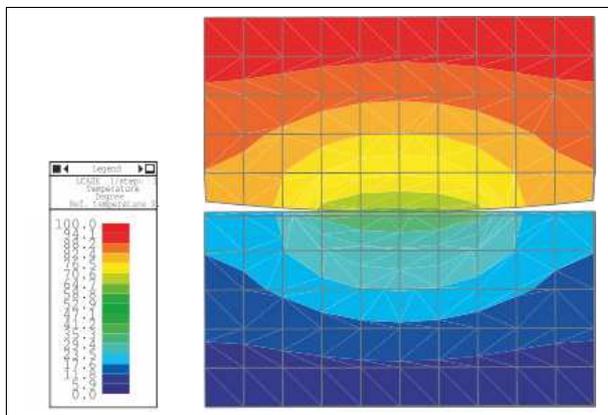


Figure 120: Temperature distribution in two contacting bodies

For subsequent analyses, contact states can be 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. 120), or *submodeling*.

PERMAS-CAX – Extended Contact Analysis

This module has been designed to provide new contact solution algorithms for ambitious slip-stick problems and for large contact models (more than 10,000 contact node pairs) in order to essentially accelerate contact analysis runs again (by a factor of 3 and beyond). The module is used as an add-on to module PERMAS-CA (see preceding section).

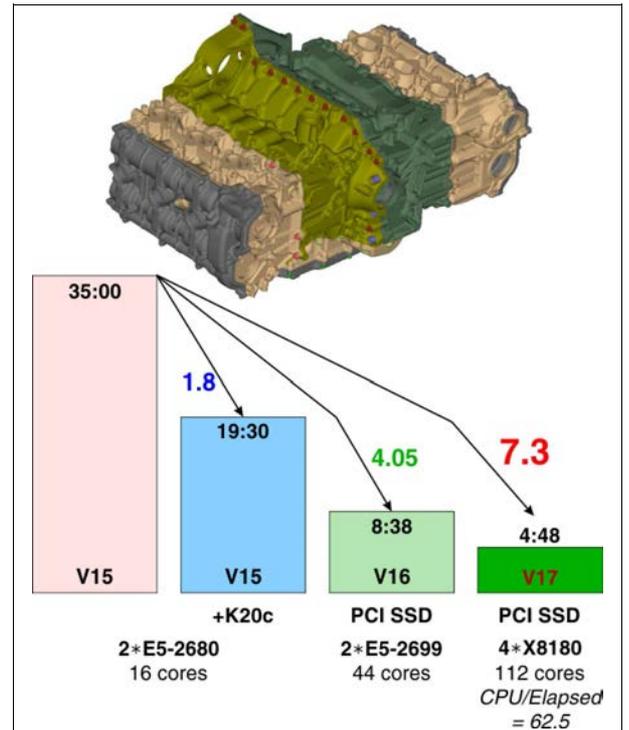


Figure 121: Contact analysis of an engine with 56 Million DOF, 37 time steps, 2 different temperature states, and CAS files, direct I/O with PCI SSD.

The functionality comprises the following features:

- High performance iterative solution algorithms to accelerate standard contact analysis with normal and frictional contact.
- An additional very stable iteration method for critical slip-stick problems (e.g. if all contact pairs get into sliding state, see Fig. 122).

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.

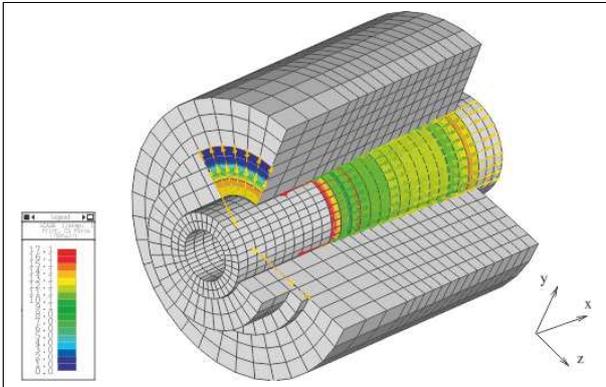


Figure 122: Conical *press fit*
(slipping forces after assembly)

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 28).

PERMAS-CAMG – Contact Multigrid Analysis

This is a new module in PERMAS V18 and allows great runtime improvements for larger contact models by using an innovative **Contact-Multigrid-Solver**. This solver consists of two orthogonal approaches. Both approaches significantly reduce the order of floating point operations and thus the runtime benefit increases with the size of the contact problem.

- **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 build by an efficient update using the Cholesky factor of the previous 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

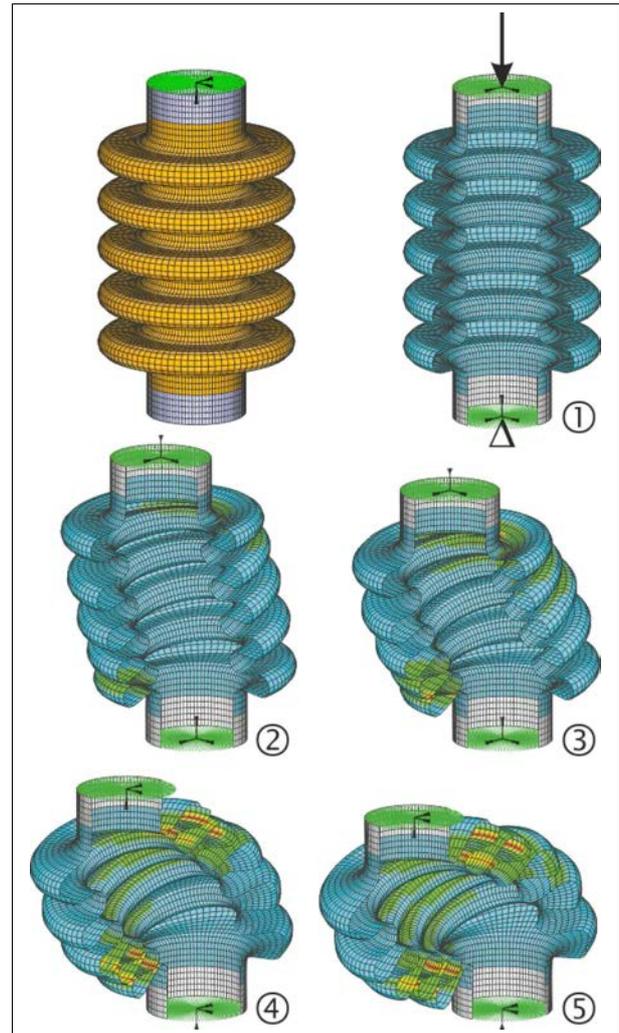


Figure 123: Self contact of a bellows

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 124 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.

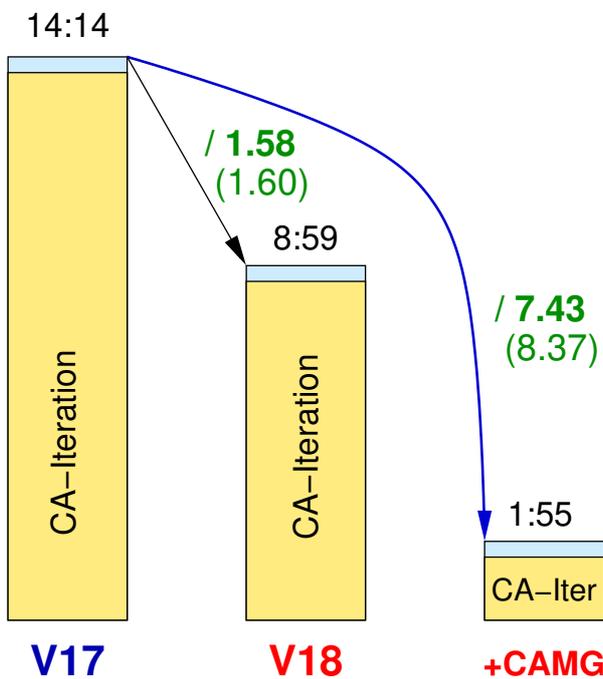


Figure 124: Computation time (hours) and speedup for large contact model for V18 and V18+CAMG compared to V17

PERMAS-CAU – Contact Geometry Update

This module has been designed to provide new 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 the contact surface of the other body) and the contact direction (i.e. the normal to the surface at each

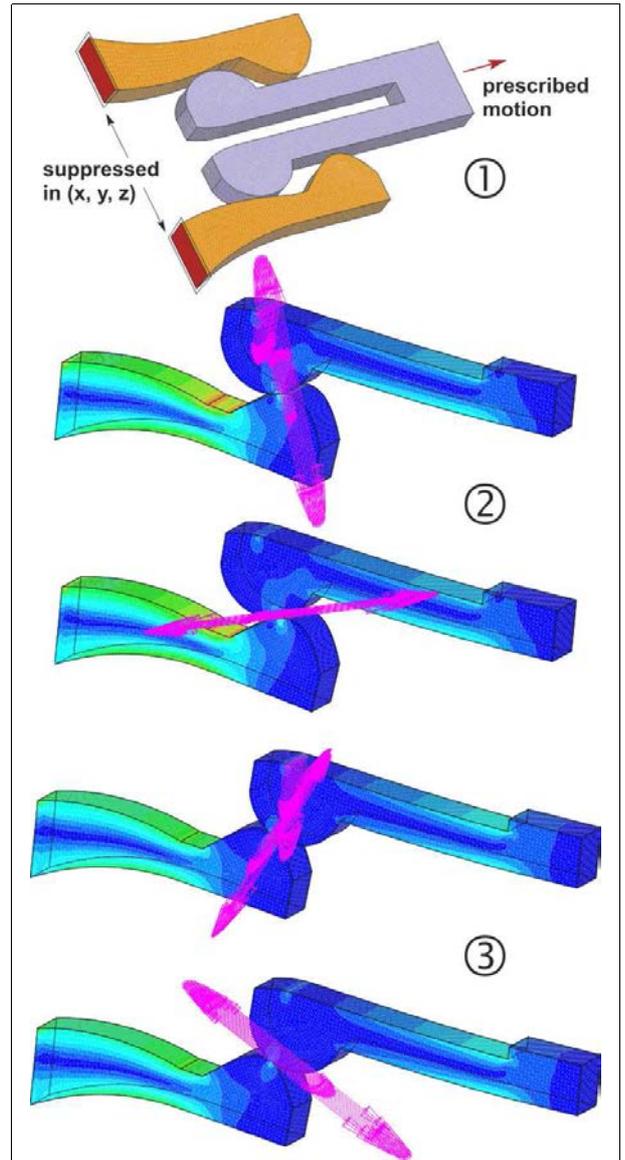


Figure 125: Clip example for contact update Large relative motion with friction. Picture 1 shows the model; pictures 2 show the contact pressure (above) and the frictional shear (below) with nodal point stresses before the bottleneck is reached; pictures 3 show the contact pressure (above) and the frictional shear (below) with nodal point stresses after the bottleneck is passed.

contact 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. 125 and Fig. 126).
- If very accurate contact results are required, contact updating even reflects very small changes of the contact geometry which gives

exact contact forces, displacements, stresses, etc..

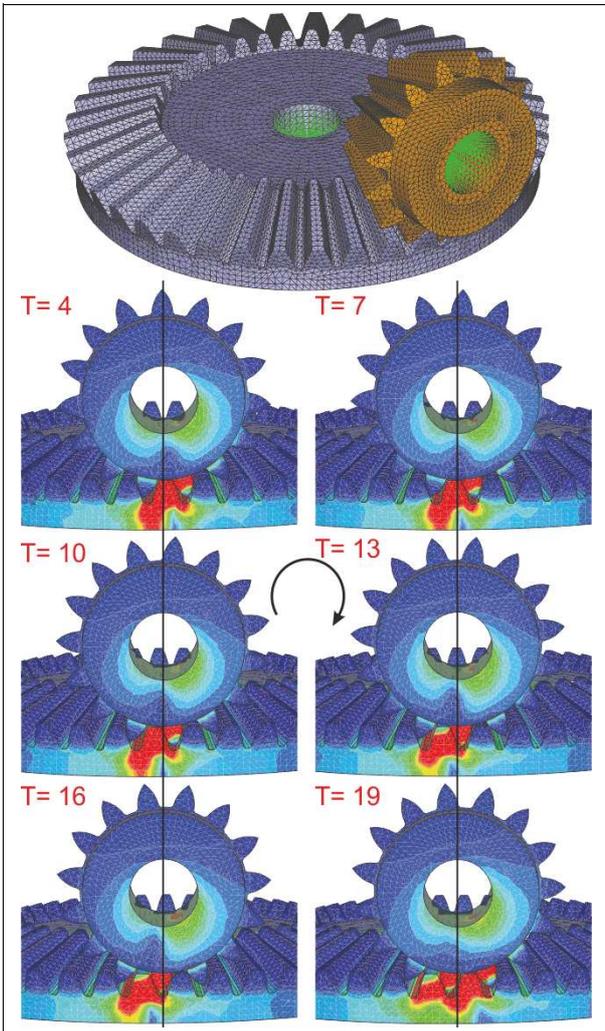


Figure 126: Driven gear set

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 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. 127. 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.

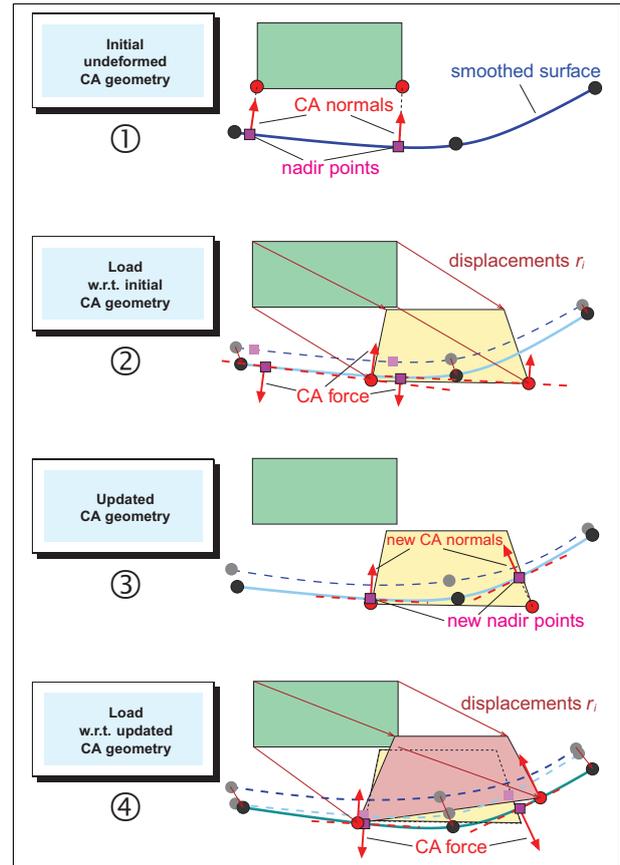


Figure 127: First steps in contact update

PERMAS-NLS – Nonlinear Statics

Geometrically nonlinear behavior

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. 128 and Fig. 176, while elastic-plastic buckling is shown in Fig. 129. 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.

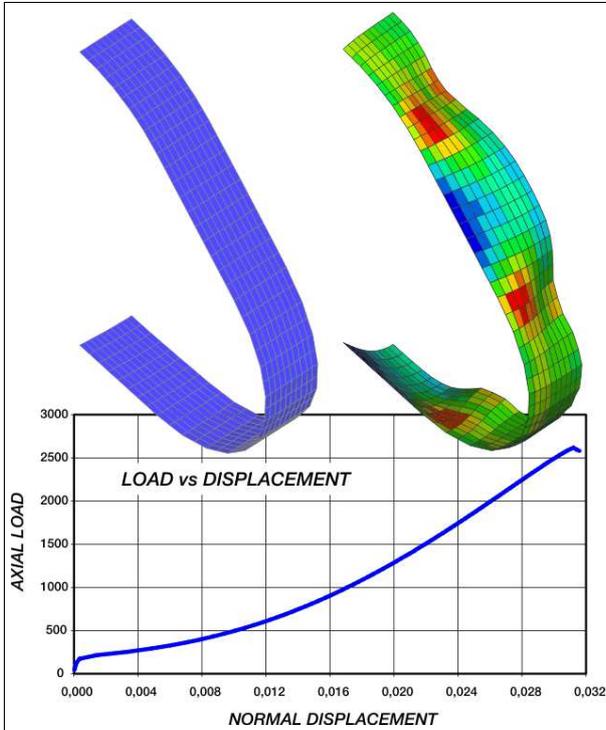


Figure 128: Nonlinear NAFEMS Test

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 78) in particular for efficient analysis of combustion engines (see engine example in Fig. 121).

Material nonlinearities

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

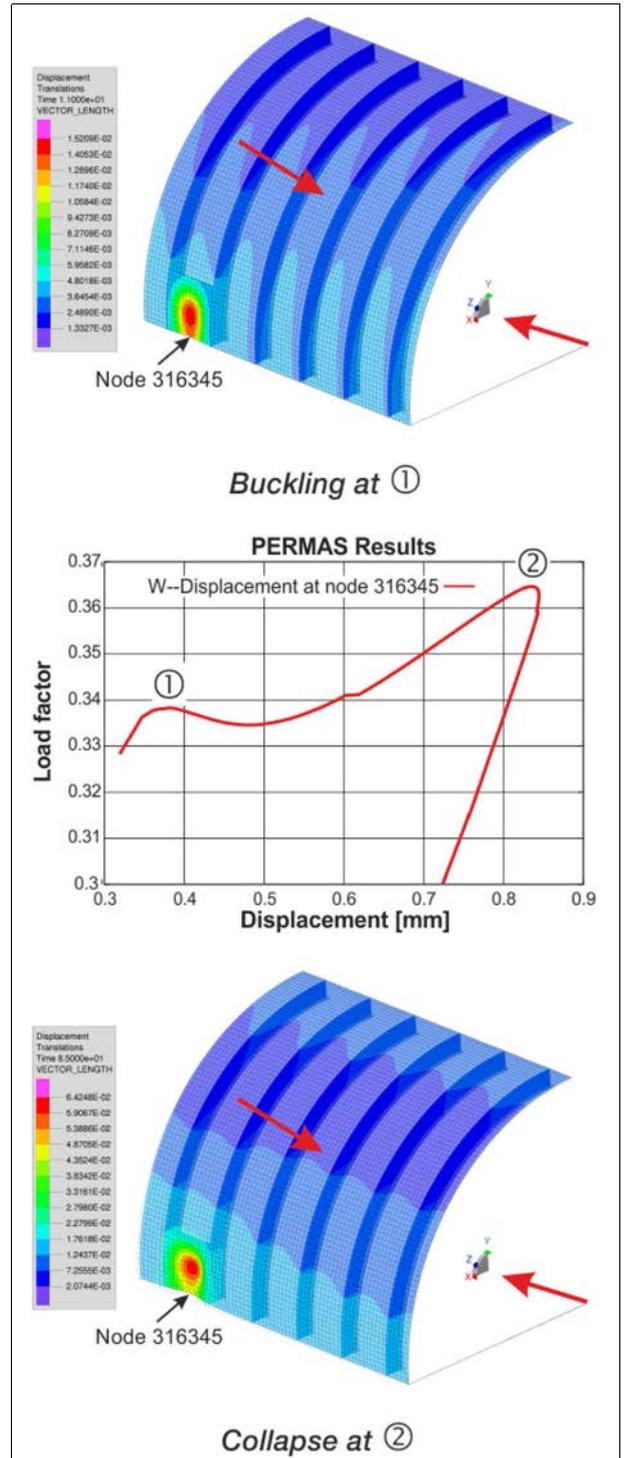


Figure 129: Elastic-plastic buckling with post-buckling behavior

- 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 73) 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. 130 compares two simple beam models with identical material and loading but different fibre orientation.

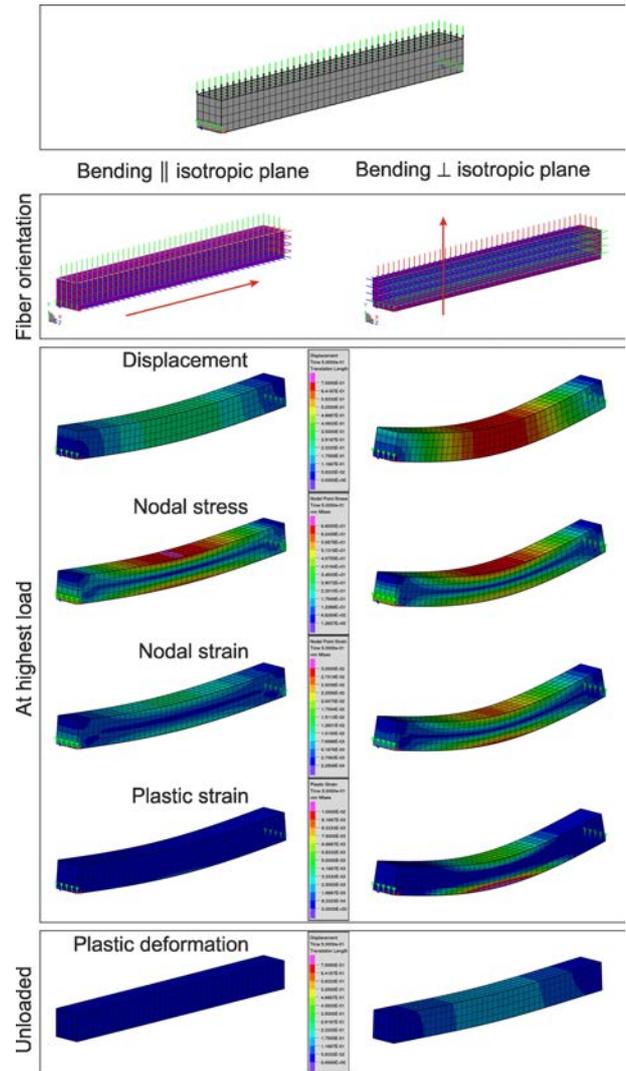


Figure 130: Transverse isotropic plasticity

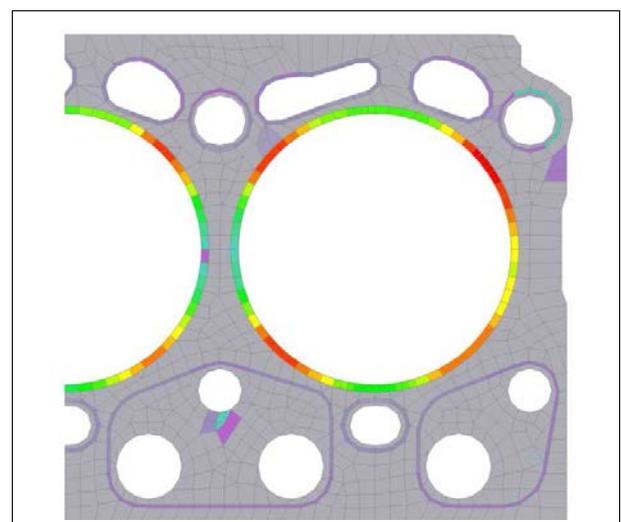


Figure 131: Pressure of cylinder head gasket

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. 129). 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 84).

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 53), 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 53). 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:

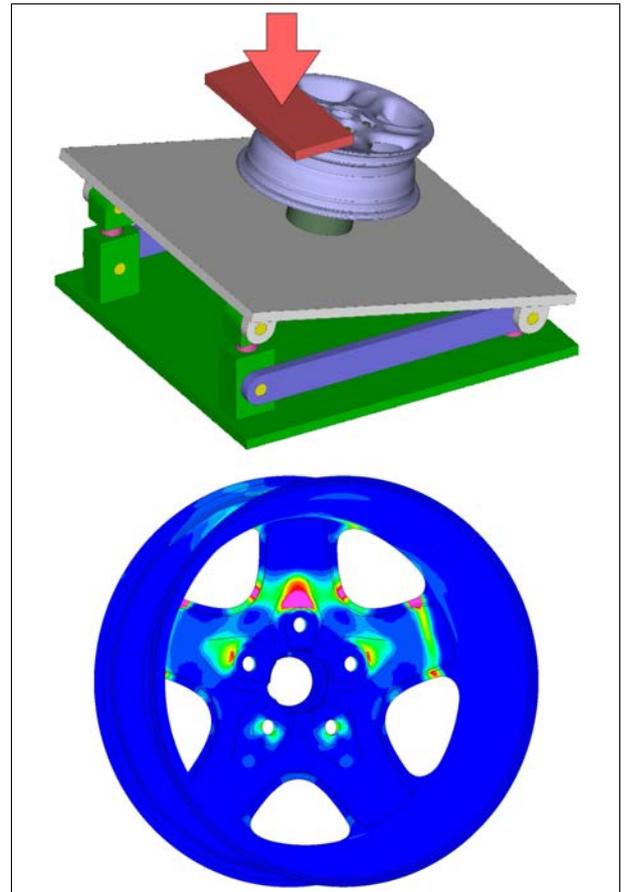


Figure 132: Impact test of a wheel with effective plastic strain.

- A material law for *cast-iron* is available taking into account the different behavior under tension and compression (see Fig. 133).
- Furthermore, a *nonlinear kinematic hardening* model (following *Armstrong-Frederick*) is available as a model for cyclic loading.

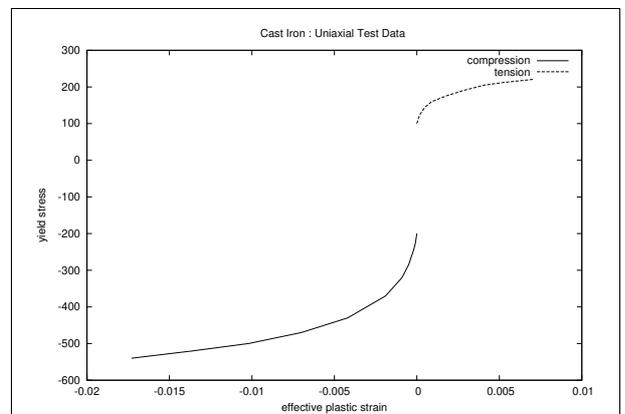


Figure 133: Uniaxial test data of a cast iron material

PERMAS-NLSA – Nonlinear static advanced

This is a new module in PERMAS V18 and 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. 134). 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.

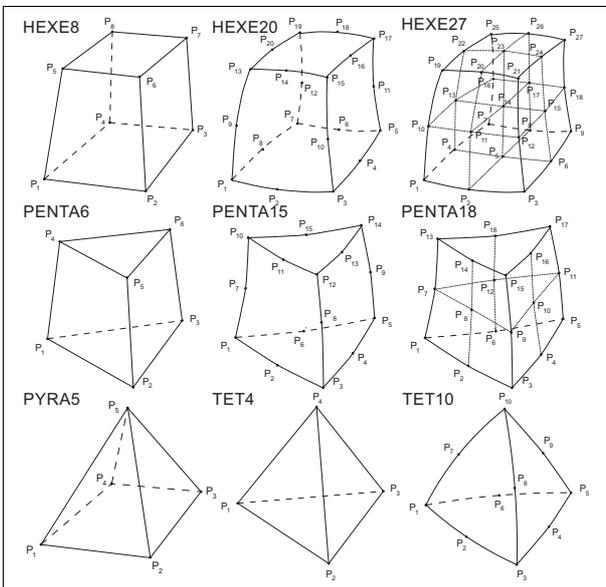


Figure 134: Large strain elements

- 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.:

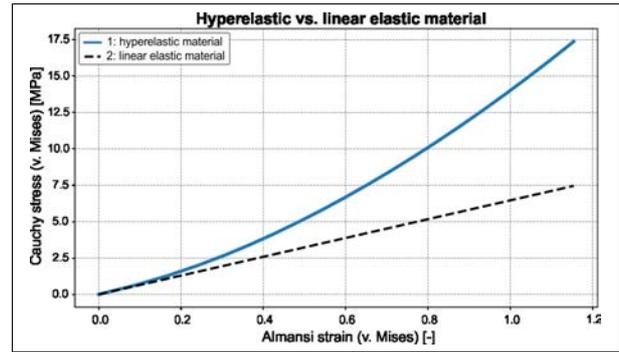


Figure 135: Hyperelastic material with non-linear relation between stress and strain (blue curve) and elastic material with linear relation between stress and strain (dashed curve)

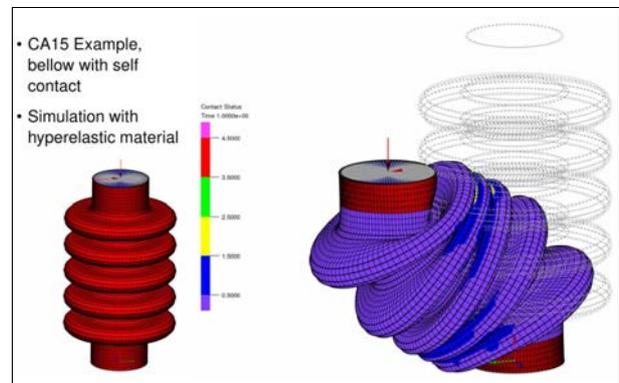


Figure 136: Bellow with self contact

```

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
!
$EXIT 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.

The calculation of *modal participation factors* allows

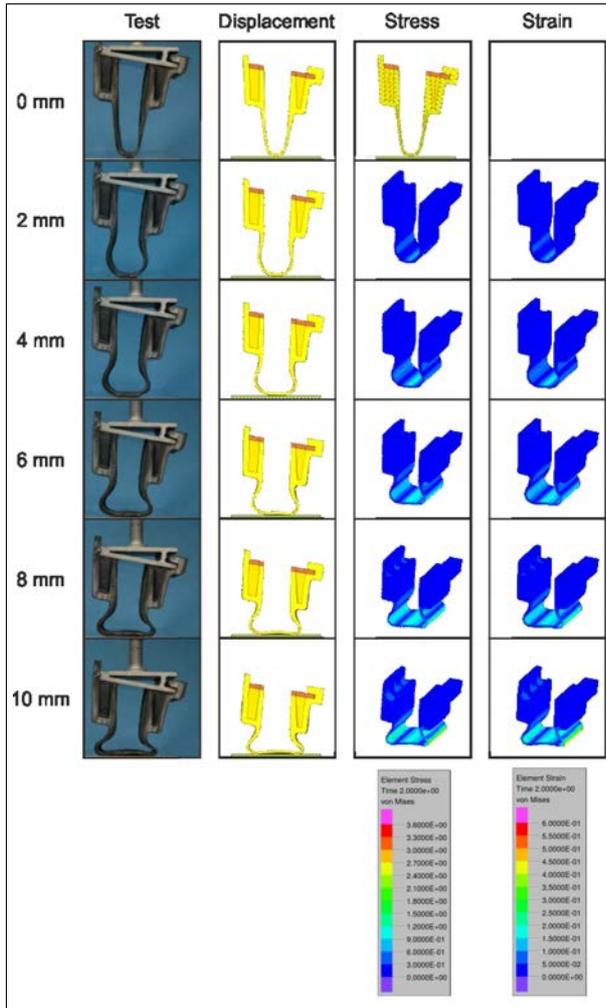


Figure 137: Large strain analysis (by courtesy of Mahle Behr GmbH, Stuttgart, Germany)

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*.

Sampling, topology optimization, and shape optimization are now possible also for buckling analysis (see Fig. 138).

PERMAS-HT – Heat Transfer

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

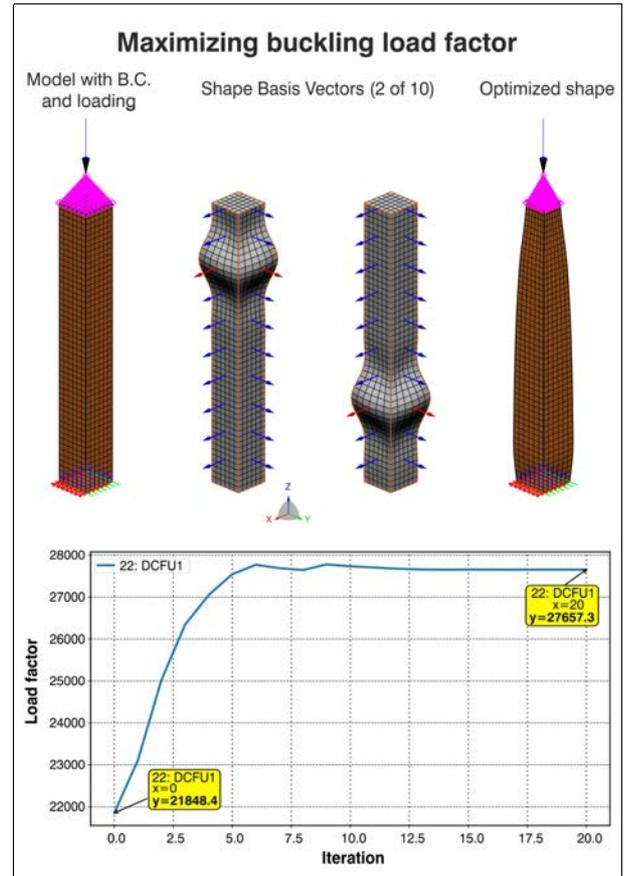


Figure 138: Maximizing buckling load factors by parametric shape optimization.

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 63).

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.

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 70).

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.

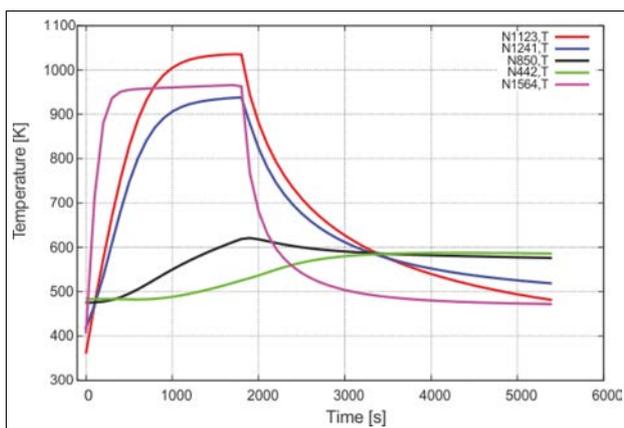


Figure 139: Transient temperatures at nodes over time

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.
- Transient 'loads' are built conveniently by combination of a stationary 'load pattern' with time-dependent functions (see page 63).
- 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.

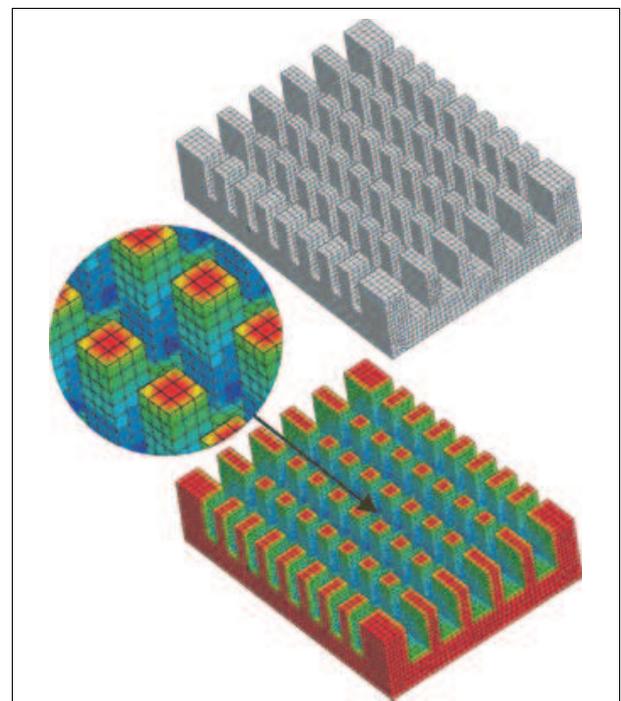


Figure 140: Analysis of cooling element with radiation

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).

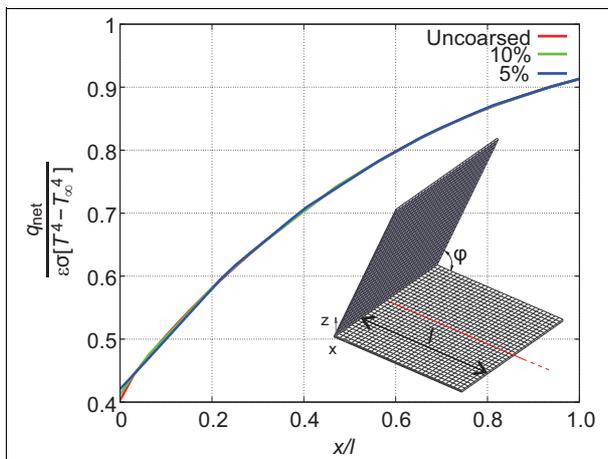


Figure 141: Heat radiation between adjoint plates

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 Package VA / Vibro-Acoustics

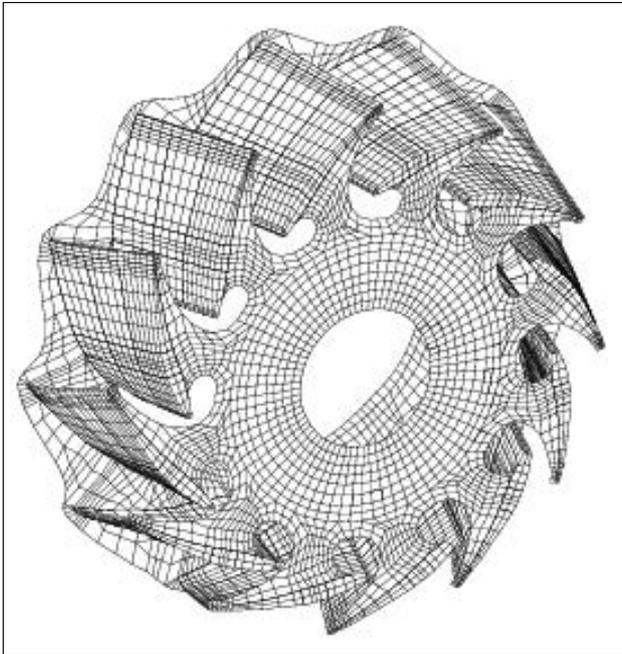


Figure 142: Half Model of a Turbine
Half Model of a Turbine, 8th Mode Shape with Antisymmetric Boundary Conditions

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.

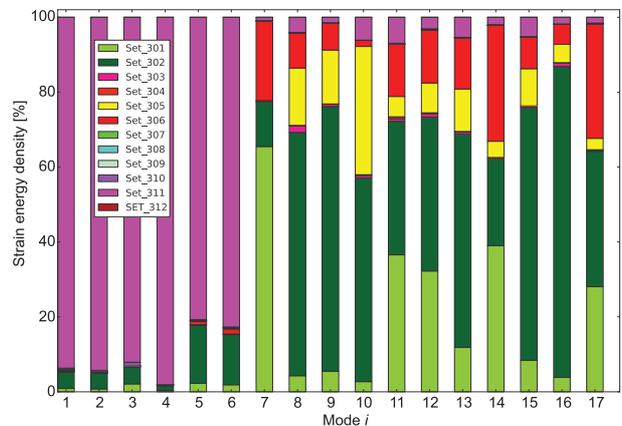


Figure 143: Analysis of strain energy density in element sets and for various modes

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. 143).
- 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 68).
- 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 53).

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 95):

- **“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

For rotational systems (see also page 30) 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

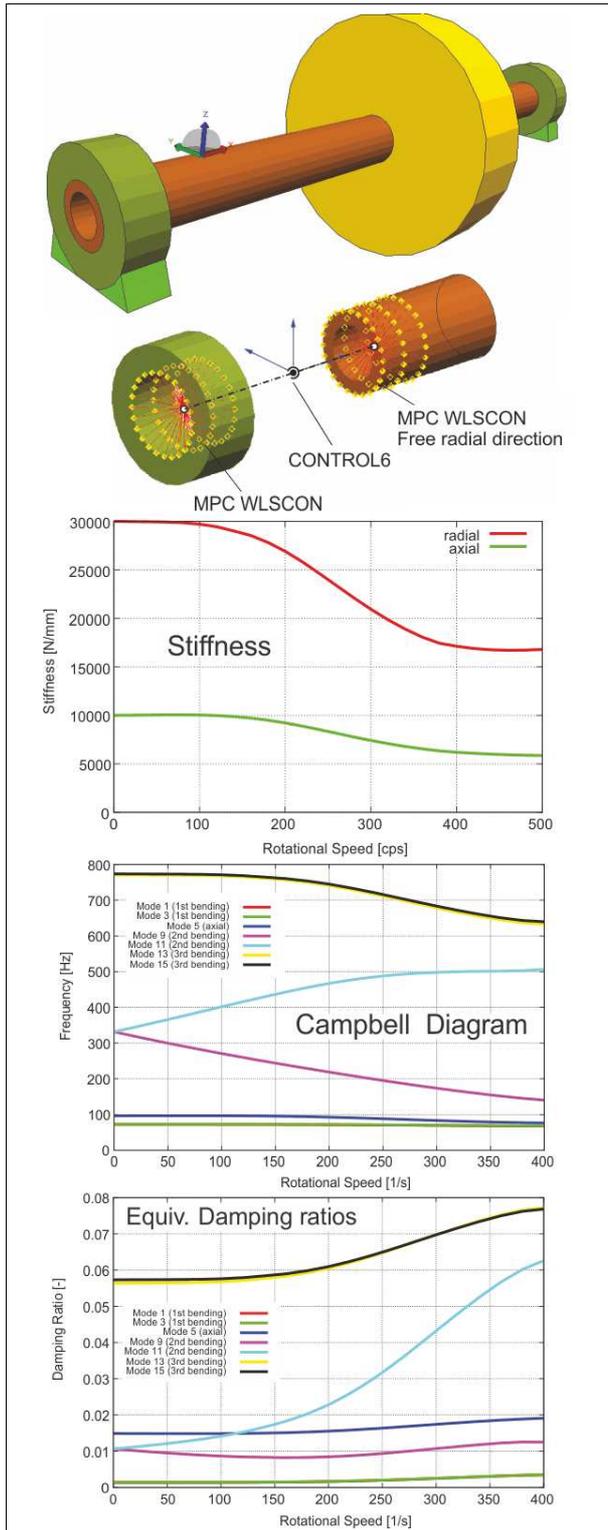


Figure 144: Rotor with rotation speed dependent supports

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.

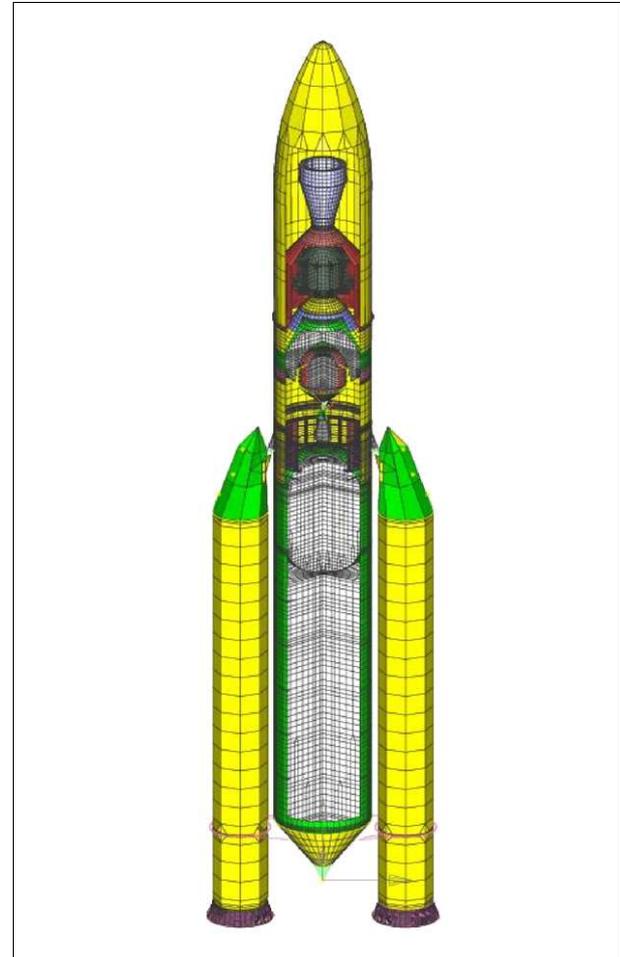


Figure 145: Ariane 5 launcher model by courtesy of ASTRIUM Space Transportation, Les Mureaux

PERMAS-MLDR – Eigenmodes with MLDR

The calculation of eigenvalues with modules DEV (page 88) and DEVX (page 89) is complemented by another method. This method can also be used for the calculation of coupled fluid-structure modes.

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 53 and module DEVX on page 89). 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 93) and static mode shapes (see page 93) are considered. Also static modes from inertia relief analysis (see page 73) 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 nonlinearities 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.

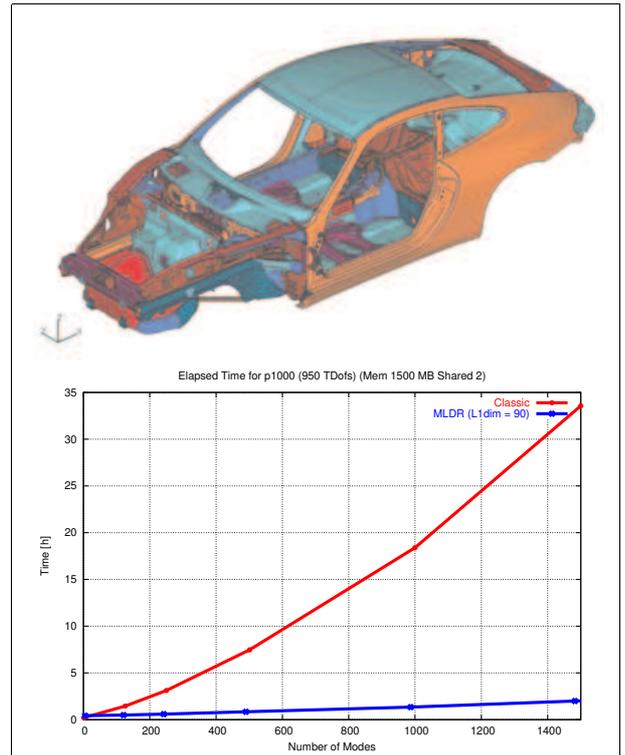


Figure 146: Comparison of elapsed run times for Subspace Iteration (upper curve) and MLDR (lower curve) with increasing number of modes (157412 Nodes, 164301 Elements (QUAD4), 944472 Unknowns)

PERMAS-DRA – Dynamic Response

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

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

- 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* and *HHT (Hilber-Hughes-Taylor)*.
- Local nonlinear effects are taken into account by
- nonlinear spring elements,

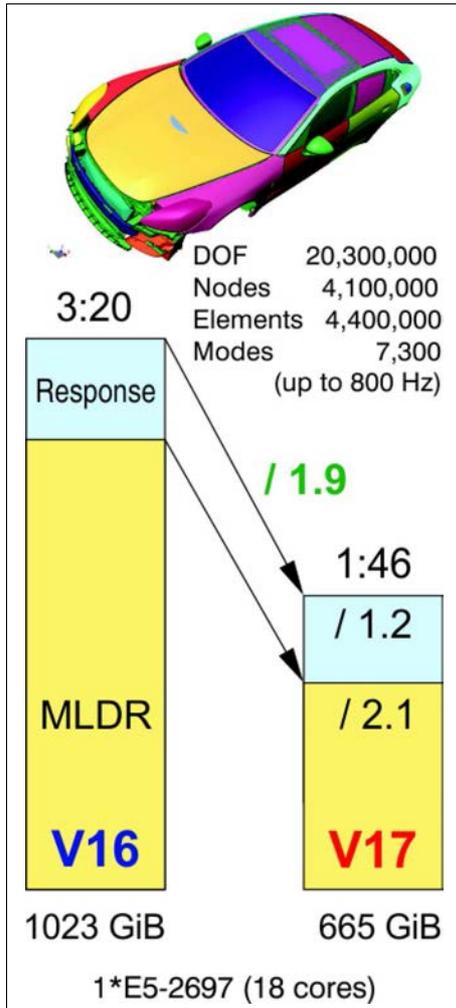


Figure 147: NVH analysis of a car body

- nonlinear damper elements, and
- nonlinear control elements.
- 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. 148 and Fig. 144).
- 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.

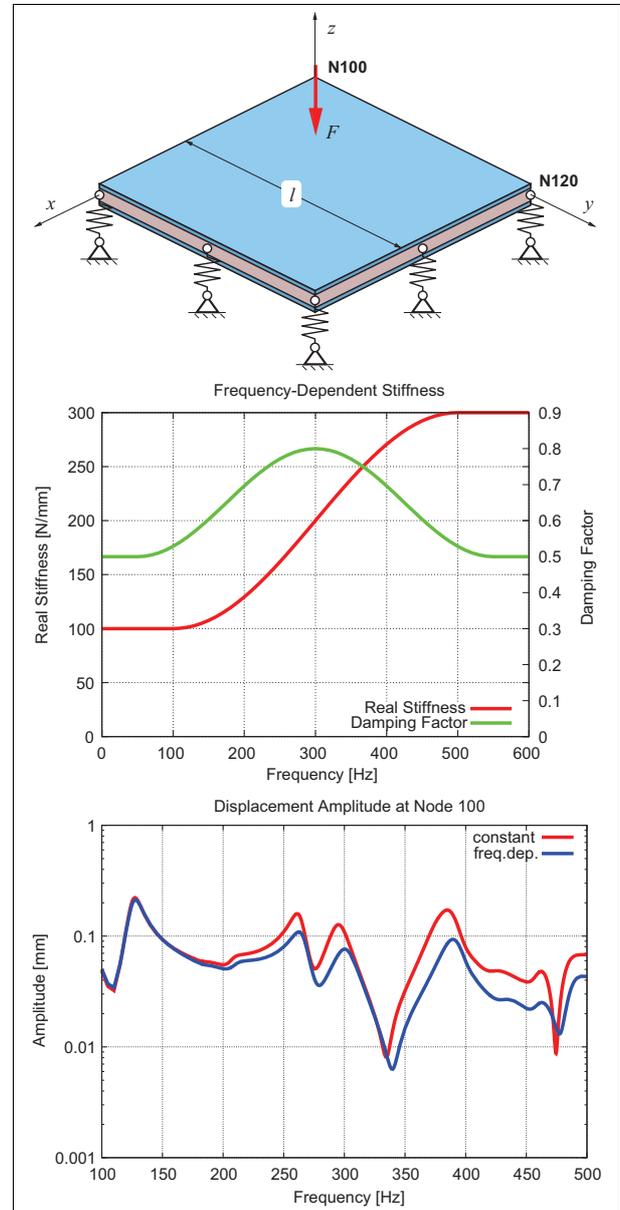


Figure 148: Frequency-dependent stiffness and frequency dependent viscous damping in frequency response analysis

By specification of a node set (see page 63) 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:

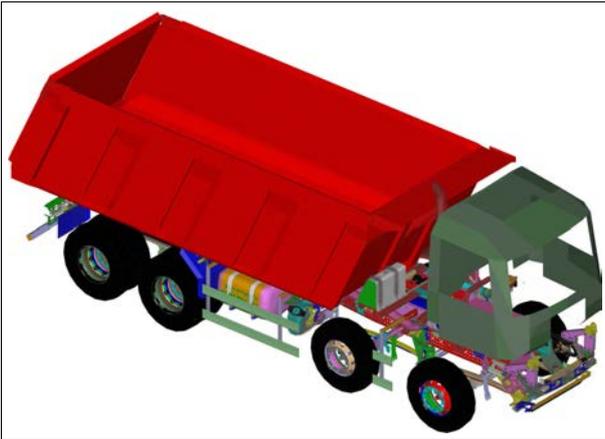


Figure 149: The model of a tipper truck (by courtesy of Daimler AG, Stuttgart, Germany).

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

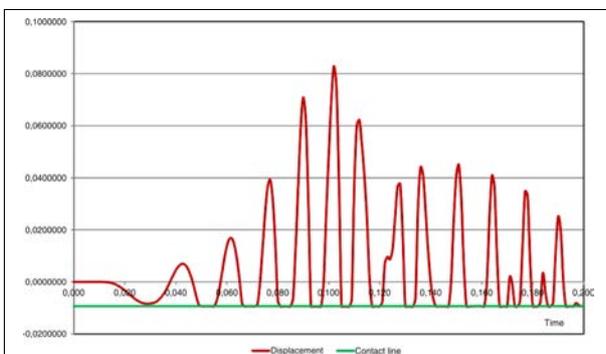


Figure 150: Transient response under contact conditions

- The excitation is defined by static loading cases modulated by functions of time or frequency re-

spectively (see page 63). 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 analysis with many different load cases. Instead of solving all dynamic load cases separately, a combined response analysis can be performed.

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 method is implemented as a 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.

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. Fig. 152 shows an overview on the analysis capabilities for fluid only and for coupled fluid and structure.

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

Acoustics	Fluid only	Fluid coupled with structure
Degrees of freedom	pressure variation	+ displacements
Element types	fluid, radiation bound. cond., surface waves, semi-infinite elements for added masses	+ coupling, structure
Material values	density, compressibility (also incompressible)	+ density, Young's modulus, Poisson's ratio
Real eigenvalues and eigenmodes	yes	coupled, with energy distribution
Additional static modes	yes	yes
Dynamic Condensation	Craig-Bampton	dry (only for displacement dof) or wet (also pressure dof)
Damping	absorption at surfaces and in fluid volume	+ any damping of the structure
Modal dynamic response	in frequency and time domain	in frequency and time domain
Direct dynamic response	in frequency and time domain	in frequency domain
Primary results	pressures	+ displacements, velocities, accelerations
Secondary results	sound particle velocity	+ stresses, strains, etc.

Figure 152: Overview of acoustic capabilities for fluid only and for fluid coupled to a structure.

- 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.

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:

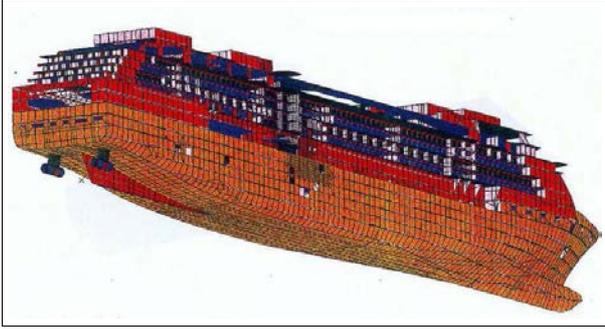


Figure 153: Ship model

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

- 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 88), 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 90).

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*).
- 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 random response* analysis for FS coupled analyses is available.

By specification of a node and/or element set (see page 63) 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.

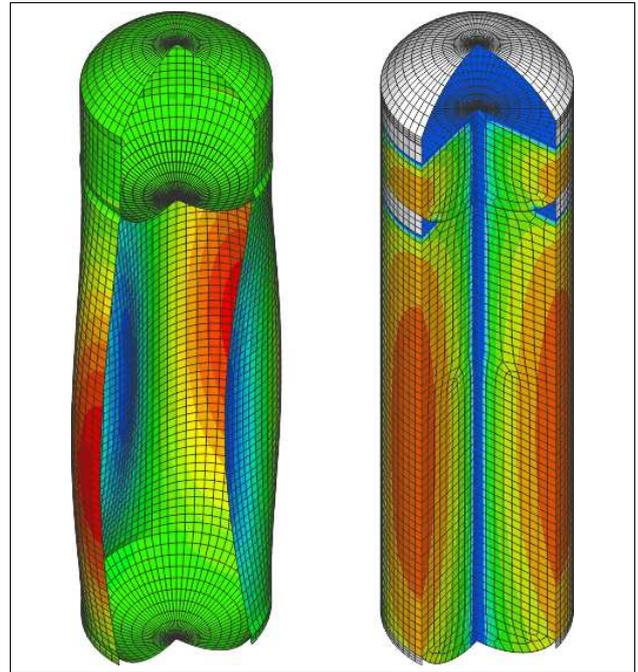


Figure 154: A coupled vibration mode shape of a fluid-filled space tank with corresponding pressure mode

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 63). 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 Package DO / Design-Optimization

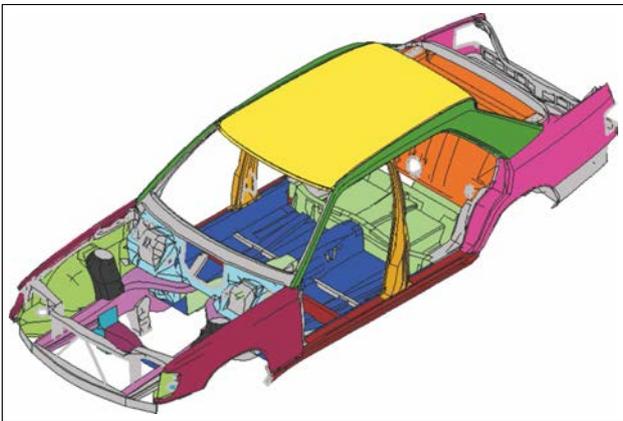


Figure 155: Frequency response optimization of a body-in-white with shape and sizing parameters (see also Fig. 158)

PERMAS-OPT – Design Optimization

Beside the pure FE modeling, PERMAS also allows the definition of a design model and its automatic optimization.

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 61),
 - 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 62),
 - use of *Shape Basis Vectors* (see page 45),
 - use of incompatible meshes (for *positioning*) without remeshing (see Fig. 90),
 - *bead design* (see Fig. 157).
 - Nonparametric *free-form optimization* (see on page 100).

- **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,
- stresses,
- compliance,
- weight,
- contact gap widths,
- contact pressure,
- contact 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. 156).

The optimization itself is performed using one of the

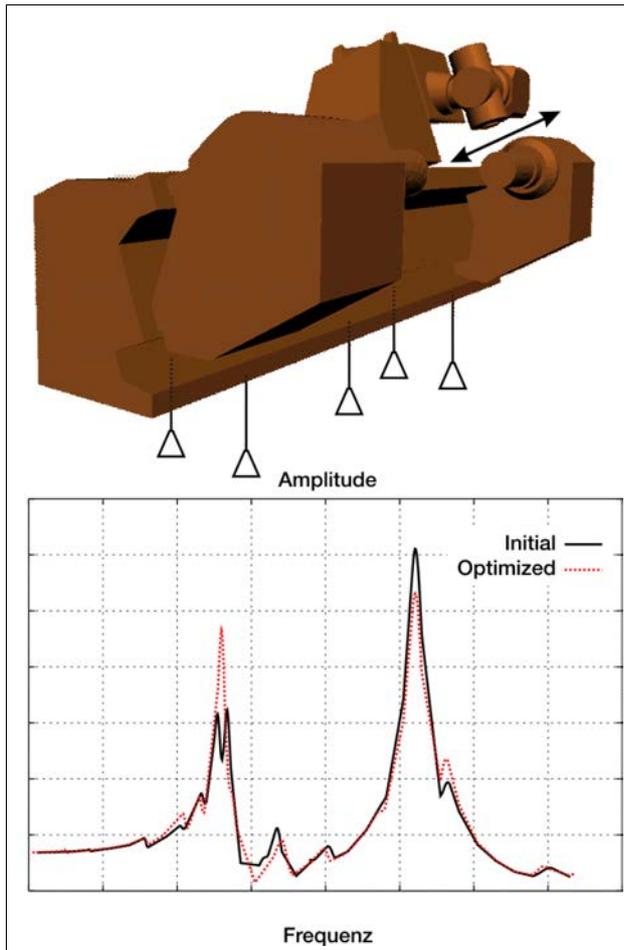


Figure 156: Frequency response optimization for a machine foundation where the position of the base spring elements is optimized which are coupled to the machine structure by interpolating surfaces. Design objective is the lowest displacement of the tool center point.

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 100).

More algorithms are available with module AOS (see below on page 106).

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 73),
- Eigenvalue analysis,
- Modal frequency response analysis,
- Steady-state heat transfer analysis.

With the aid of module AOS (see page 106) additional solvers are available for Optimization:

- Contact analysis,
- Nonlinear material behavior.
- Complex eigenvalue analysis (including rotating structures)

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 loading cases as well as different boundary conditions using variant analysis (see page 54.) 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 ex-

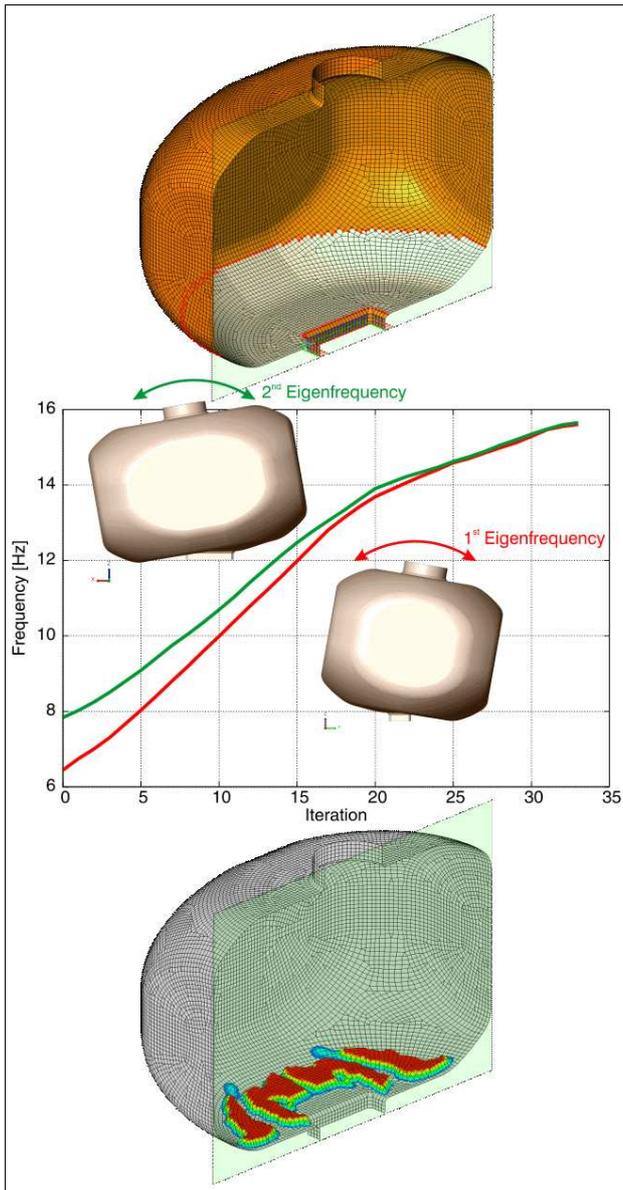


Figure 157: Bead generation for plastic tank

ported 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.

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. 160). This opens the most easy way to define shape optimization of free-form geometries. The set-up of this optimization is supported by a

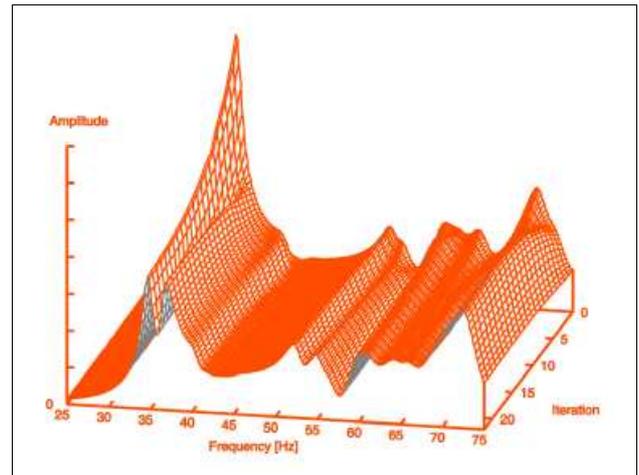


Figure 158: Iteration history of a frequency response optimization

freeshape wizard in VisPER (see page 45).

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. 159 with non-normal thickness changes).
- Release directions are available for free-form optimization as manufacturing constraint.

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. 161 and Fig. 162).

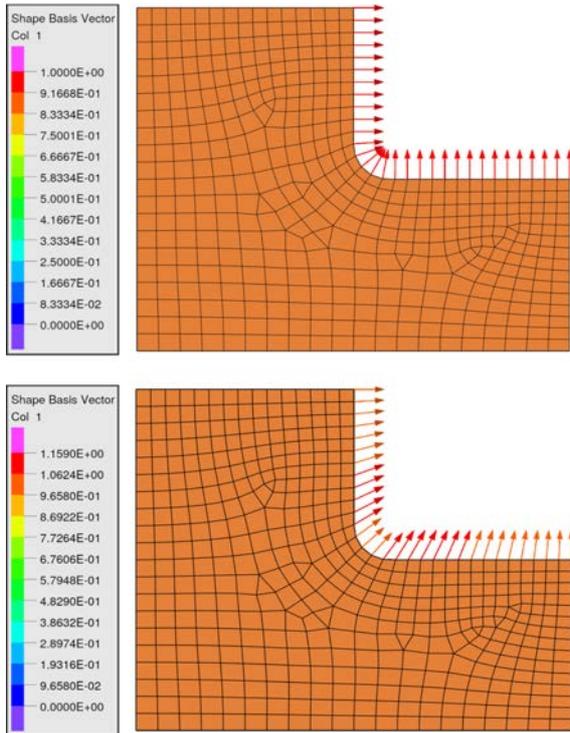


Figure 159: Improved shape changes

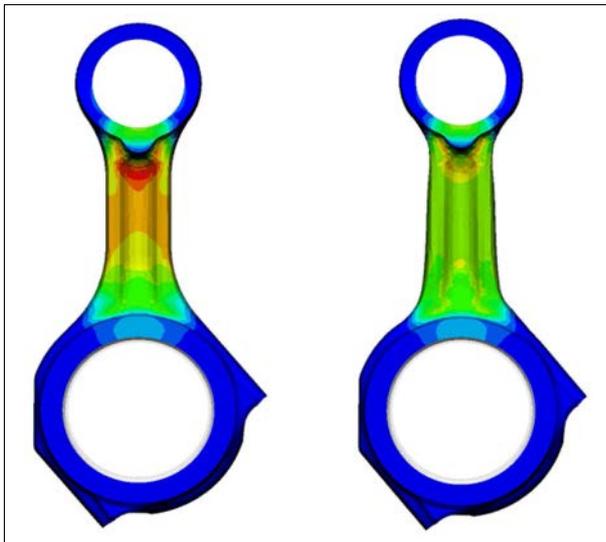


Figure 160: Stress homogenization of conrod

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. for the calculation of safety factors). To achieve that a script has to be written, which invokes the external tool and provides nodal results in a specific PERMAS compliant format.

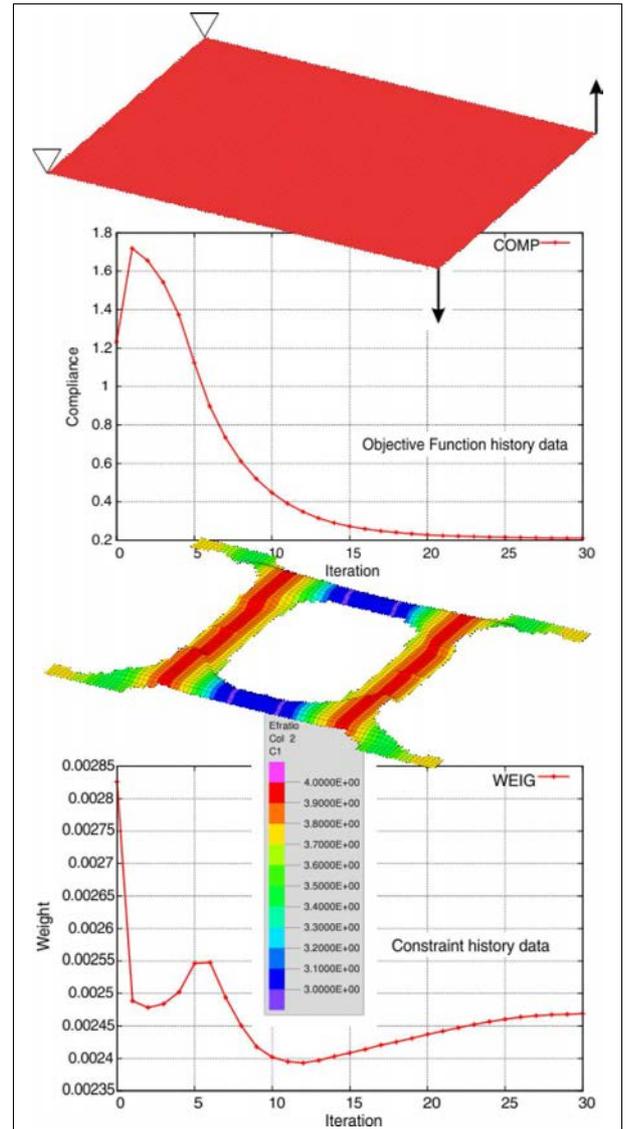


Figure 161: Multi-modal optimization with topology optimization, bead optimization, and free sizing for the sheet thicknesses.

Laminate Optimization

Topology optimization is used to get ply shapes from the optimized thickness distributions (see page 105 and fig. 164).

Sizing of laminates is then supported, where ply thicknesses and angles can be optimized. *Ply failure criteria* (see page 109) may be used as constraints for the laminate 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

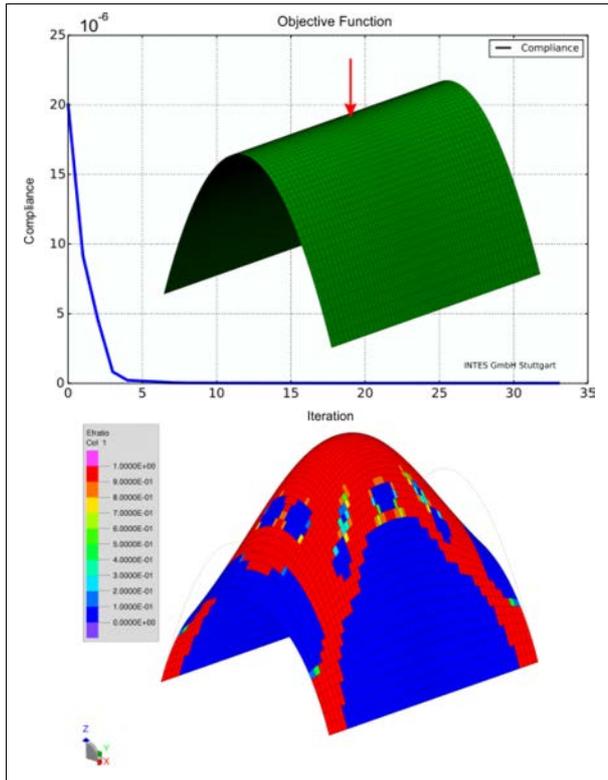


Figure 162: Topology and shape optimization simultaneous solution as multi-modal optimization

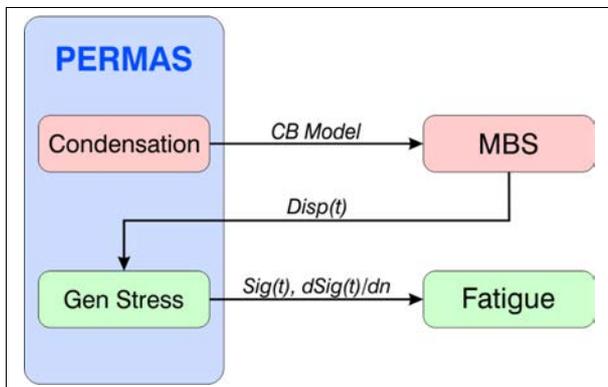


Figure 163: Use of MBS and Fatigue coupled to PERMAS.

reliable regarding uncertain model parameters (see page 36 for more details).

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

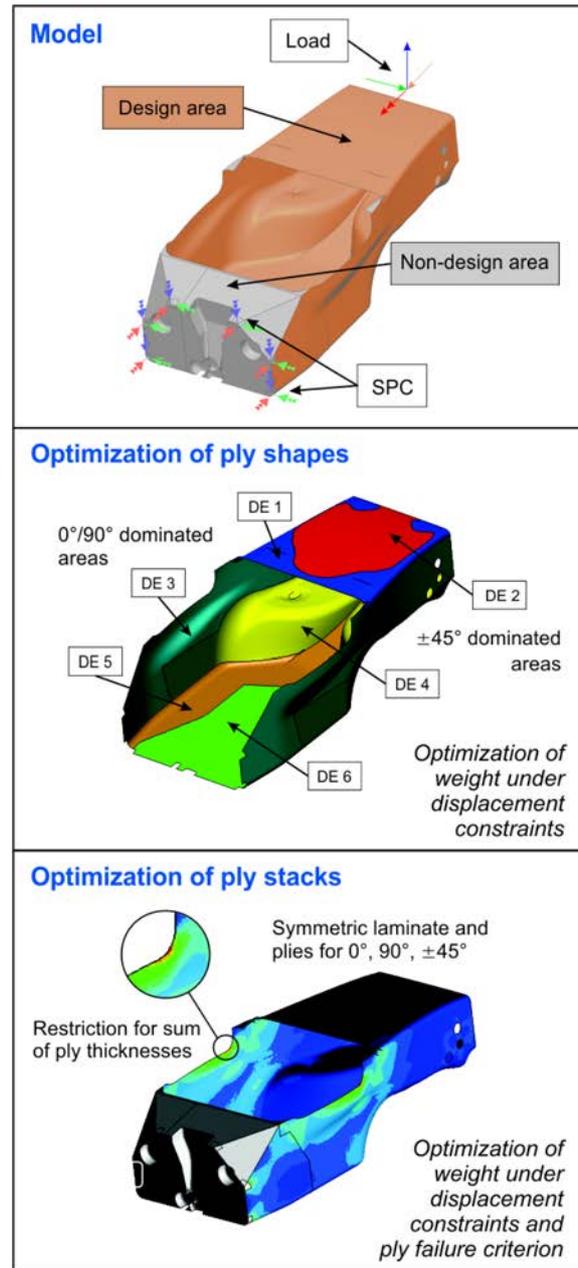


Figure 164: Optimizing laminates for a racing car monocoque with free sizing (see 105) to get a thickness distribution of the layers and with sizing (see page 98) to optimize ply thickness and angle.

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.

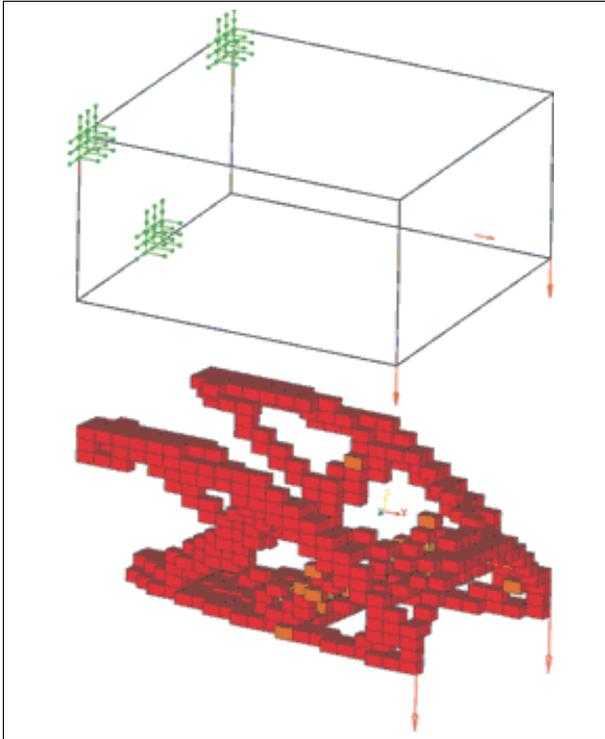


Figure 165: Layout optimization
Design space with boundary conditions and loading (above) and the optimal material distribution (below)

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.

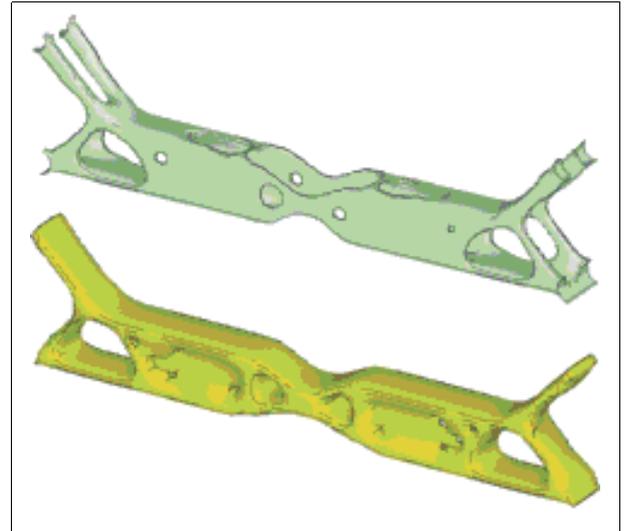


Figure 166: Different results for topology without (above) and with (below) release directions

- **Manufacturing constraints:**

- *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. 166). 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 checkerboard filter).
- *Frozen regions:* Such regions of the design space may not be changed during the optimization process.

- The overhang angle can be designed, which is important for 3D printing designs (see Fig. 167).

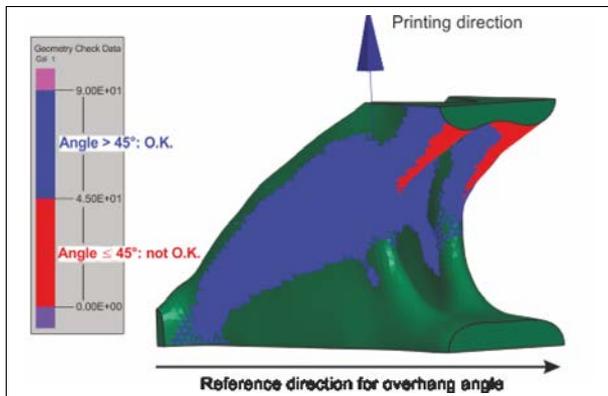


Figure 167: Check of overhang angle
red: critical angle, blue: non-critical, other colors: no value (i.e. $>90^\circ$).

- **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 106) maximum member size in topology optimization is exactly formulated.

A number of analysis options are available for the optimization like

- Linear statics,

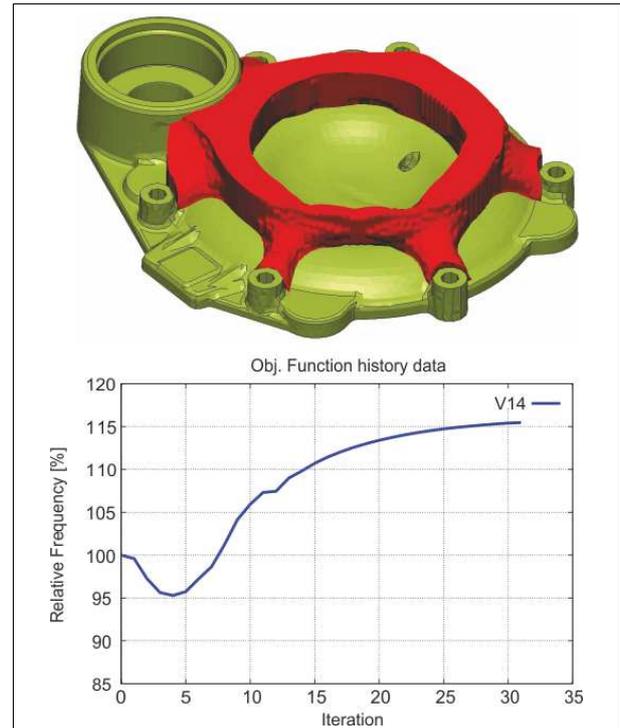


Figure 168: Topology optimization of a gearbox cover to maximize the second eigenfrequency with release direction (by courtesy of ZF Friedrichshafen AG)

- 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)*

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

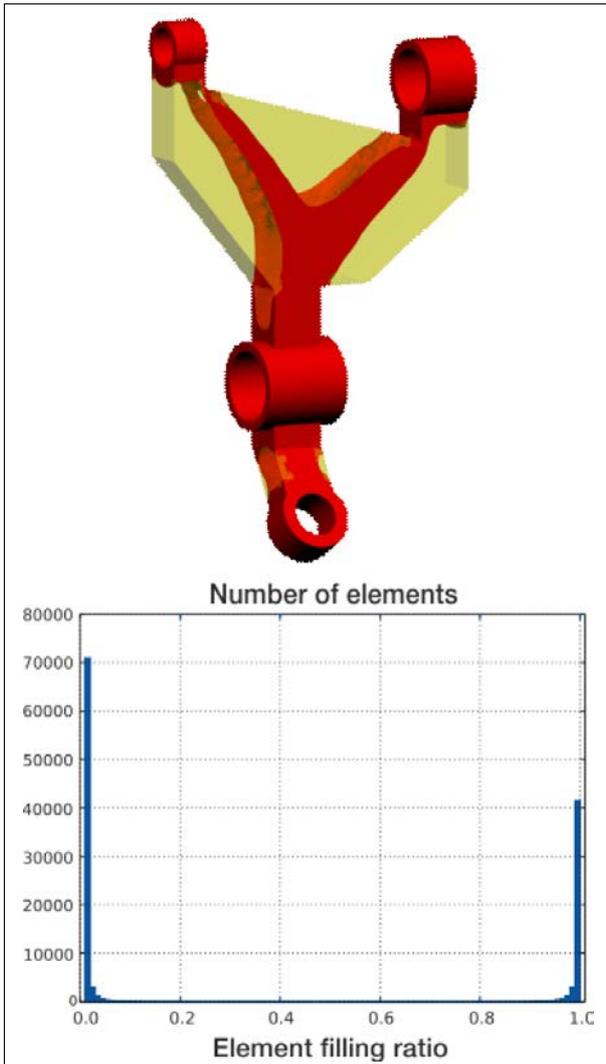


Figure 169: Fully converged topology

directly for further analysis and design steps (see Fig. 169).

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 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 con-

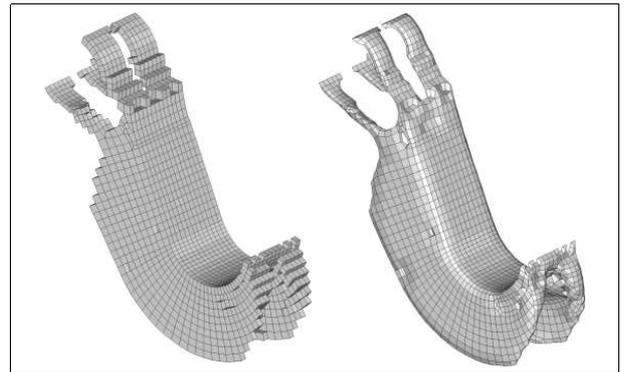


Figure 170: Layout-Optimization of a crane hook (on the left side the primary result, on the right side the smoothed surface)

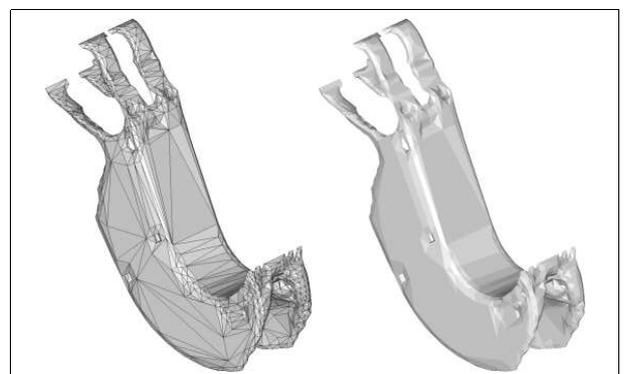


Figure 171: Polygon reduction (on the left side the triangle mesh, on the right side graphical representation of the surface)

straints.

- 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 101).

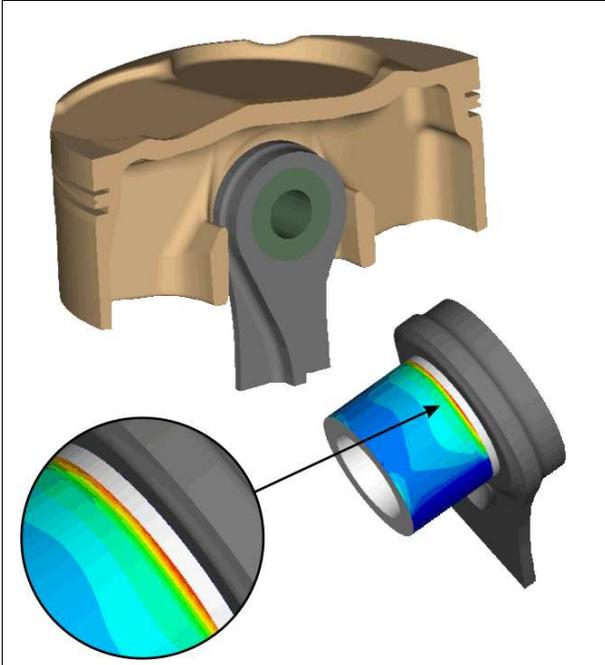


Figure 172: Optimization of contact pressure between piston and piston pin (Mahle GmbH, Stuttgart). The figure shows the edge pressure between both parts (for the optimized result see also next Fig. 173).

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 chosen 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.

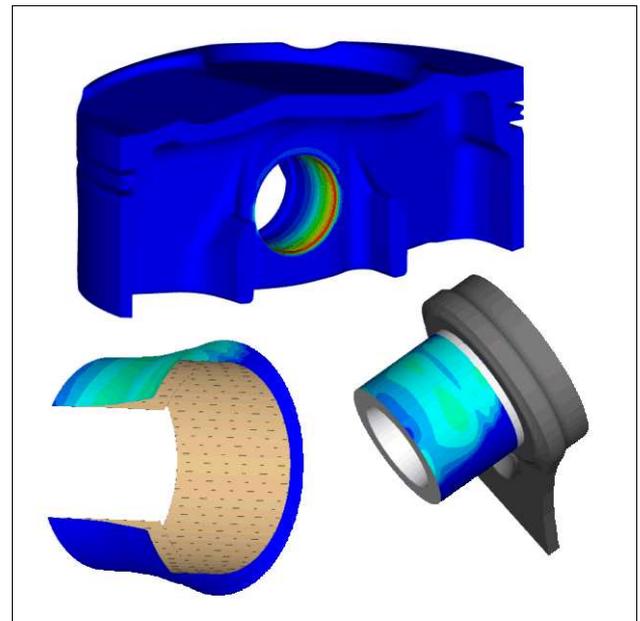


Figure 173: Optimization of contact pressure between piston and piston pin (Mahle GmbH, Stuttgart). The figure shows the optimized geometry of the piston and the reduced edge pressure.

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 com-

prise 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 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. 174 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 53).

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.

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 results of such calculations are compared to typical limiting constraints such as a maximum stress or de-

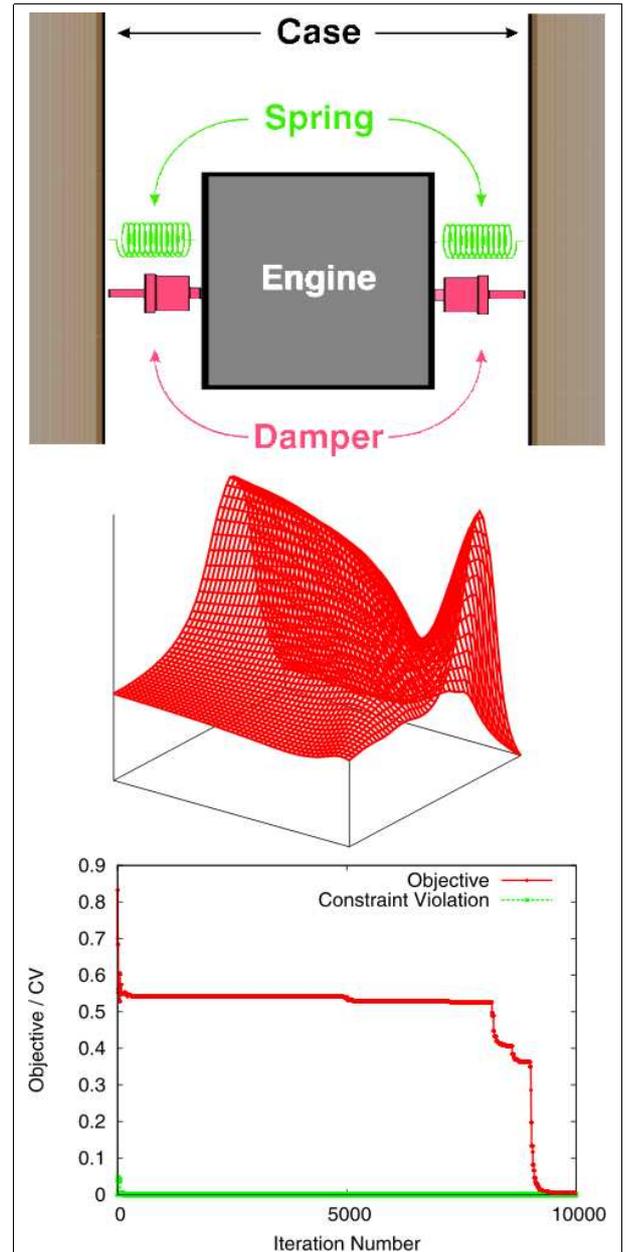


Figure 174: Global optimization of a spring-damper system with the LDR method. Top: model with 4 design variables, Middle: 2 design variables fixed at the optimum, objective function dependent on the 2 free variables, Bottom: course of objective function and constraint violation during the iterations.

flection 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 charac-

teristics 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-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 homogenization then takes place in PERMAS following the *Extended Laminat Theory (ELT)*.

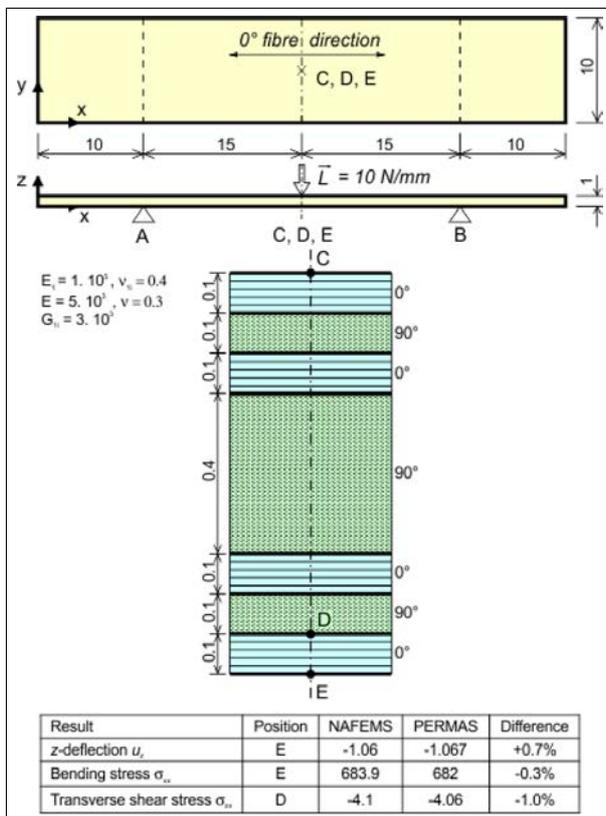


Figure 175: Laminate analysis of a *NAFEMS* composite benchmark.

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. 175), also nonlinear analysis (like in Fig. 176) is supported.

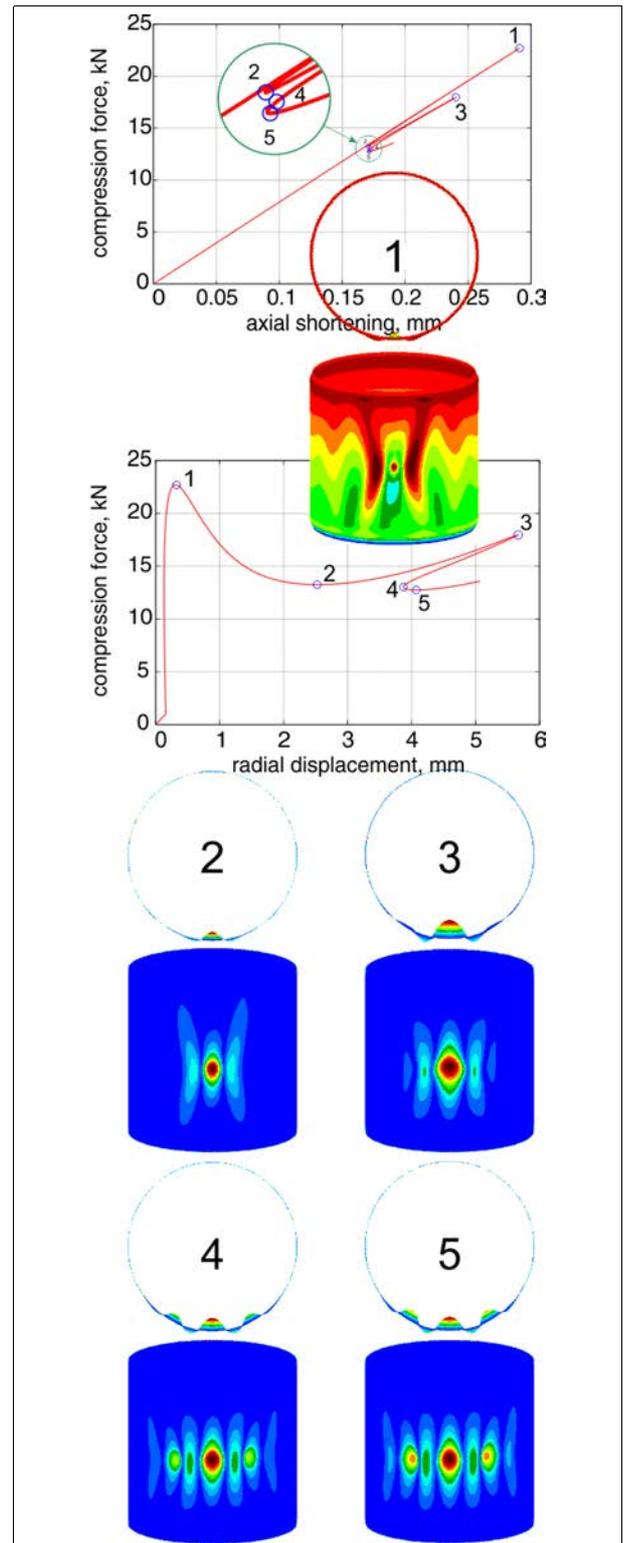


Figure 176: Nonlinear buckling with post-buckling behavior for a laminated cylinder shell

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.

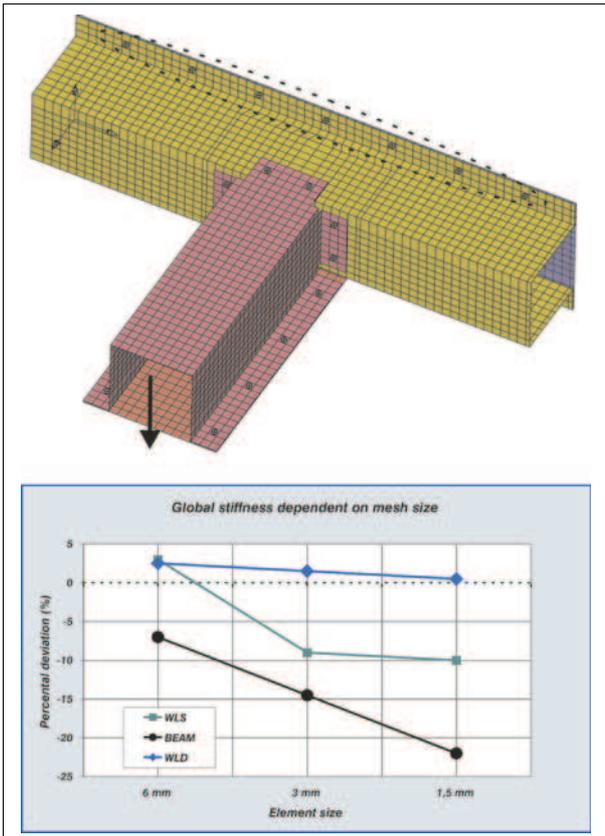


Figure 177: Quality of the refined weldspot model

PERMAS-WLDS – Refined Weldspot Model

The modeling of weldspot connections is described on page 57. This modeling gives a good representation of the global stiffness.

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 rep-

resentation (see Fig. 177). Among others, this is achieved by an internal calibration of the stiffness using a volume model.

PERMAS-GINR – Generalized Inertia Relief

Inertia relief performs a quasi-static analysis of a free body, i.e. of an unconstrained structure (see page 73). 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. 178).

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.

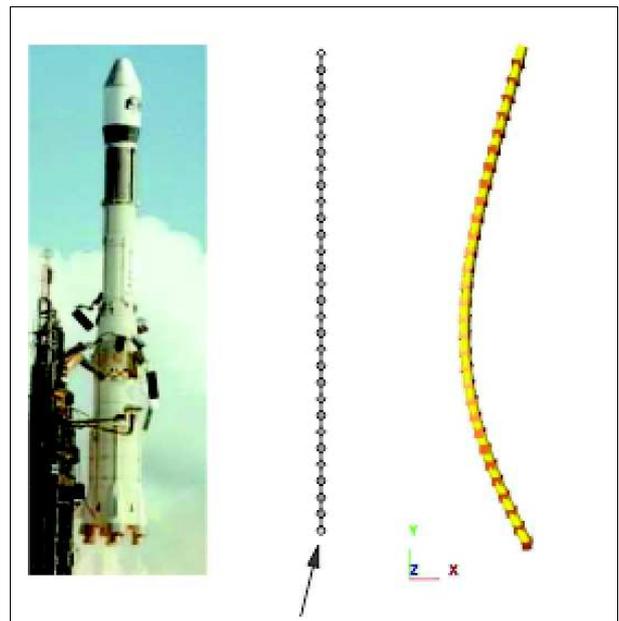


Figure 178: Displacements of flying rocket under thrust and dead weight.

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 59).

Heat induced by an electrical field can be used for a subsequent thermal analysis (see page 85). From that thermal stresses can be derived performing a subsequent static analysis (see page 73).

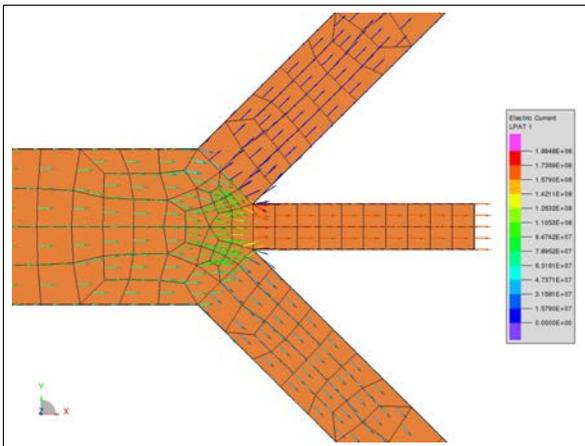


Figure 179: Electric current in an electric junction

Forces induced by a magnetic field can be used in a subsequent static analysis (see page 73).

PERMAS-EMD – Electrodynamics

A solution of *Maxwell's* equations is available for different problem cases:

- eddy currents
- induction problems (see Fig. 180)
- 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 91).

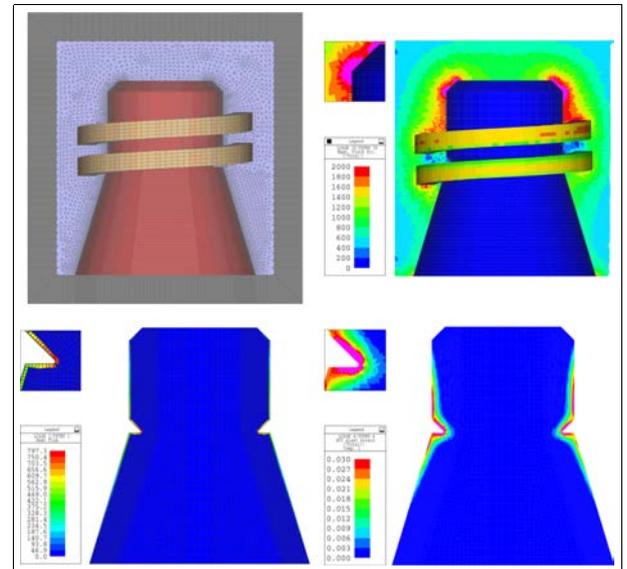


Figure 180: Example for electrodynamic induction, model and magnetic field strength (top), induced heat flux and plastic strains (below).

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

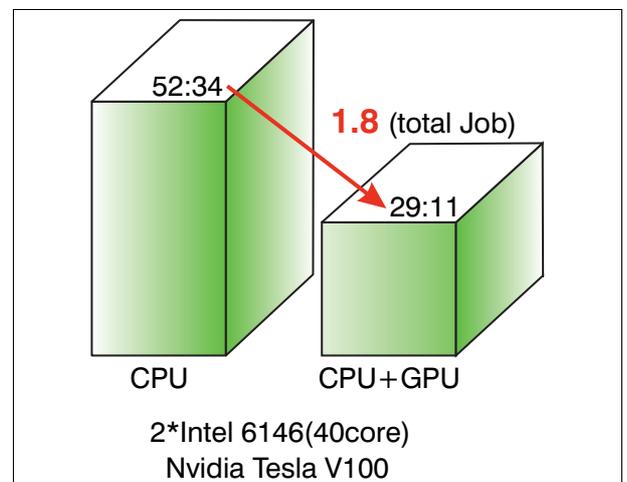


Figure 181: XPU benefit for a large contact analysis

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 121), 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. 181

Interfaces

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 58). Beside Components, different Situations with constraint and load variants may be specified within MEDINA (see page 54).

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.

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 37).

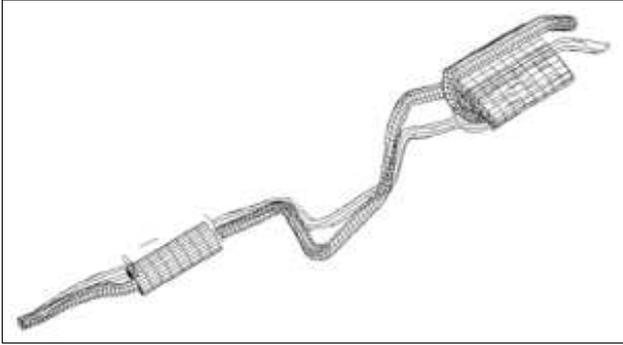


Figure 182: Mode Shape of an Exhaust System

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 88).

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 90).

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. 163).

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 90).
- 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 88).

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:
 - 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.

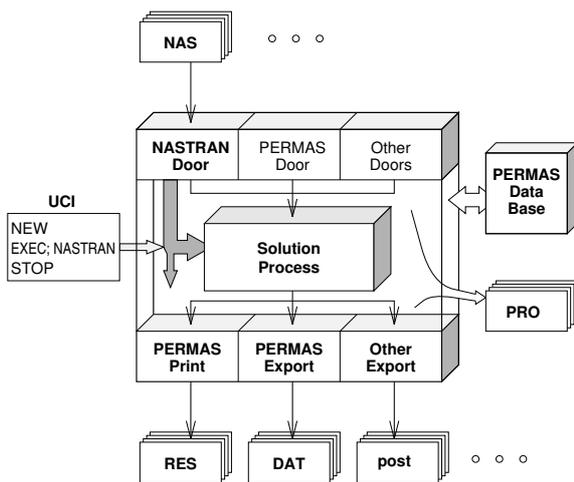


Figure 183: UCI trigger for NASTRAN-Task

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 executed individually, leaving all remaining subcases 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

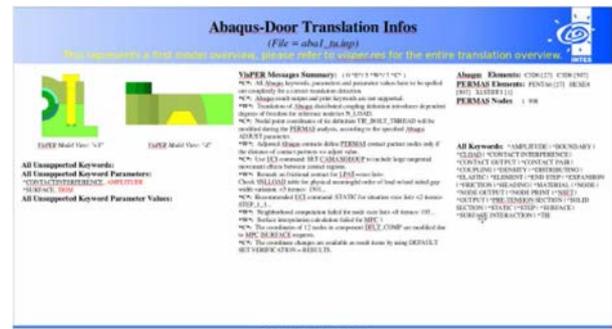


Figure 184: Abadoor translation report

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. 185).

Step History Overview					
Permas			Abaqus		
Situation [UCI command]	LFAT	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 2 3	4	General	STATIC	CLOAD DLOAD TEMPERATURE

Figure 185: 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 70.

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 MEDINA format (.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.
- **FEGraph (von Mach Engineering, www.vonmach.de):** This software works as a comprehensive post-processor to PERMAS and processes the PERMAS 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 114) which exports the results for post-processing with HyperView.
- **MAGMALink (MAGMA, www.magma-soft.de):** This software is a module for the transfer of casting simulation results out of MAGMASOFT to FE meshes for stress and durability analysis.
- **Material data base MARLIS (M-Base, [© INTES GmbH Stuttgart](http://www.m-

</div>
<div data-bbox=)**

base.de): This material data base contains material data of steel sheets and is capable to issue the material properties as PERMAS material description.

- **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.

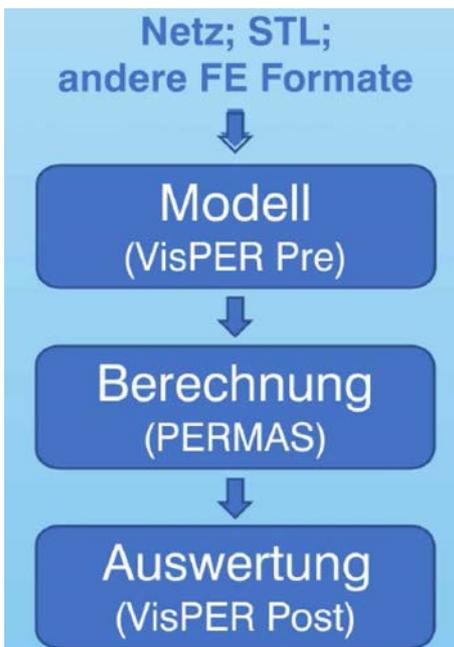


Figure 186: Analysis Workflow

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

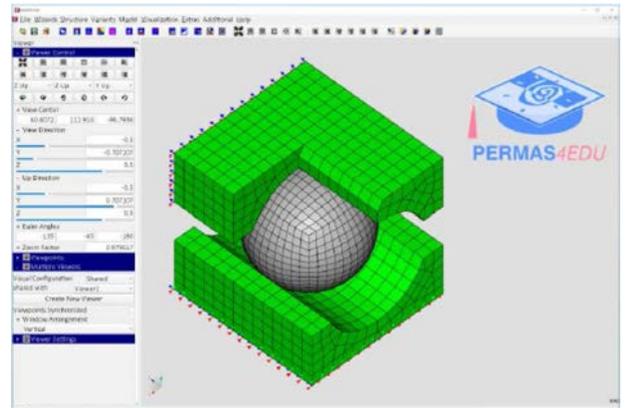


Figure 187: Contact model

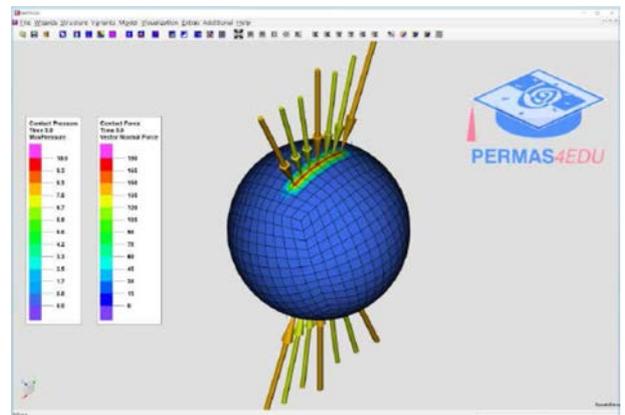


Figure 188: Contact pressures and forces

Installation and beyond

Supported Hardware Platforms

For PERMAS, the following platforms are supported:

Architecture	Operating system
PC x86(_64)	LINUX Debian 9/10, LINUX Ubuntu 16.04/18.04/20.04, LINUX RedHat EL6/EL7/EL8, LINUX SuSE 12/15 and compatibles Nvidia GPU (see page 111)
PC x86_64	Windows 10 64 bit

For VisPER, the following platforms are supported:

Architecture	Operating system
PC x86(_64)	LINUX glibc \geq 2.11, NVIDIA graphics
PC x86_64	Windows 10 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.

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 (GNATS) 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 sub-

scribed 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.

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.
 - 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.
-

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 submit a PERMAS or VisPER job and to set the most common job parameters. It also support data handling and job control (see Fig. 189), 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.

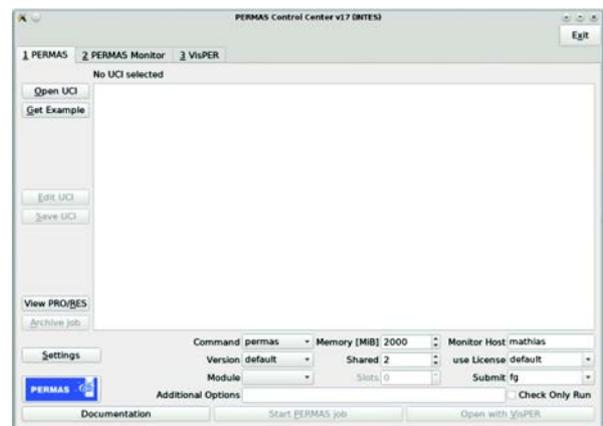


Figure 189: PERMAS Control Center

- **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 (.post or .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. 143).

- 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 user's site as well as training.

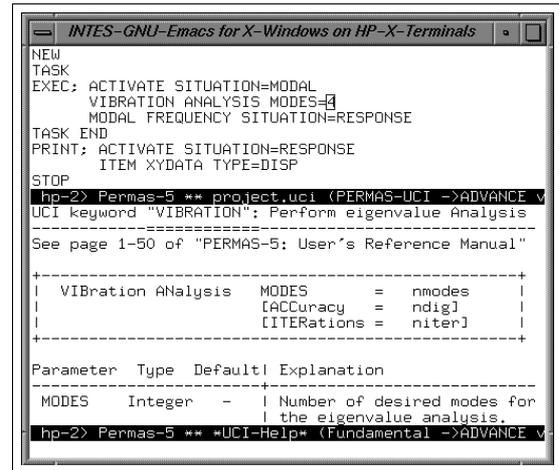


Figure 190: Context-Help for UCI-Files inside Editor

Documentation

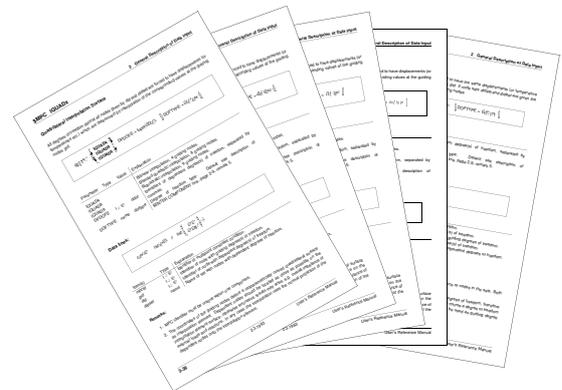


Figure 191: PERMAS documentation

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. Beside 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.

- **Simulation driven design**
to provide solutions ready to produce, e.g. for 3D-printing.

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:	Rolf Fischer
Phone:	+49 (0)711 784 99 - 13
Fax:	+49 711 784 99 - 10
E-mail:	info@intes.de
WWW:	http://www.intes.de
Address:	INTES GmbH Breitwiesenstr. 28 D-70565 Stuttgart

Index

- α -method, 97
- 10% rule, 95
- ABDG matrices, 109
- Absolute Error Indicator, 62
- ACP, 99
- active control, 35
- active set strategy, 106
- Adapted Convex Programming, 99
- Add&Replace wizard, 41
- AEI, 62
- anisotropy, 62
- architecture
 - 64 bit, 121
- arclength, 80
- Armstrong-Frederick, 83
- assembled situations, 23, 93
- basic variable, 108
- Bayliss-Turkel, 95
- bead design, 24, 45, 98
- beam elements
 - standard cross sections, 61
- BFGS, 106
- BIW, 21
- body-in-white, 21
- bolt pretension, 75
- brake squeal analysis, 28
- brake squeal wizard, 42
- buckling
 - linear, 84
 - nonlinear, 81
- CAMG, 10
- Campbell, 31, 89
- car body analysis, 21
- cascade controller, 32
- casting, 82
- Cauchy, 81
- CB condensation, 89
- CCNG analysis, 25, 77, 81
- center of pressure, 76
- CLF, 10
- Co-Rotational, 80
- COF, 68
- Coleman transformation, 55
- CoMAC, 68
- compensation springs, 75
- complex eigenvalues, 89
- component, 53
- composites, 61, 109
- condensation, 53, 89
 - Craig-Bampton, 89
 - dry, 26
 - dynamic, 26, 53, 89, 91, 114
 - generalized modal, 67, 88
 - Mixed-Boundary Craig-Bampton, 89
 - static, 53, 114
- configuration, 53
- CONLIN, 99
- constraint
 - kinematic, 58
- contact
 - analysis, 73
 - locking, 26, 29, 77
 - multigrid, 78
 - status files, 26
 - wizard, 49
- contact locking, 26, 29, 77
- contact status files, 26, 76
- contact wizard, 49
- control elements, 35
- convectivity elements, 85
- coordinate system
 - local, 58
- COP, 76
- Coriolis matrix, 31
- Coulomb, 74
- Coulomb's law, 74
- coupling
 - analyses, 70
 - CFD, 70
 - surfaces, 58
 - volumes, 59
- CQC, 95
- Craig-Bampton, 53, 89
 - mixed-boundary, 53, 89
- critical rotation speed, 31
- cutting forces, 21, 69
- cyclic symmetry, 55
- damping, 93
 - frequency-dependent, 92
 - material, 93
 - matrix, 93
 - modal, 93
 - proportional, 93
 - Rayleigh, 93, 96
 - rotation speed dependent, 31, 92

- structural, 93
 - viscous, 93
- design constraint function, 104
- design constraint linking, 98
- Design wizard, 42
- DiRect, 107
- Dividing Rectangles, 107
- Drucker-Prager, 81
- dynamic condensation, 97

- earthquake spectral response, 94
- effective masses, 88
- eigenvalues
 - complex, 89
 - real, 88
- electro-statics, 111
- element library, 60
- elements
 - axisymmetric, 61
 - beam, 60
 - contact analysis, 61
 - convectivity, 61, 85
 - discrete, 61
 - flange, 60
 - fluid, 60
 - fluid-structure coupling, 61
 - gasket, 25
 - geometry, 61
 - load, 61, 64
 - membrane, 60
 - plate, 60
 - plot, 61
 - radiation boundary condition (RBC), 61
 - rod, 60
 - scalar, 61
 - semi-infinite, 61
 - shell, 60
 - solid, 60
 - surface waves, 61
- ELT, 109
- EMA, 113
- engine analysis, 24
- Engquist-Majda, 95
- exb, 13, 114
- Experimental Modal Analysis, 68, 113
- experimental modal analysis, 37
- Extended Laminat Theory, 109
- external solver, 101

- failure function, 108
- Fast Fourier transformation, 22
- FFT, 22

- fibre-reinforced composites, 109
- fixed mold parting line, 44, 103
- floating license, 121
- fluid elements, 60
- force-guided contact, 75
- FORM/SORM, 108
- Fourier, 63, 64
- free sizing, 105
- free-form optimization, 45
- frequency response, 92
- friction
 - Coulomb, 74
- frozen regions, 103
- function
 - design constraint, 104
 - failure, 108
 - gap, 74
 - limit state, 108
 - mathematical, 63

- gap function, 74
- gasket elements, 25, 77, 82
- GCA, 104
- GDC, 107
- Global Convex Approximation, 104
- Global Design Centering, 107
- gnuplot, 65, 68
- green, 2
- GUI, 39
- Guyan, 53, 89, 114
- gyroscopic matrix, 31
- gzip, 114

- hardening, 82
 - isotropic, 82
 - kinematic, 82
 - mixed, 82
 - nonlinear kinematic, 83
- HDF import, 10
- HDF5, 66
- heat transfer, 85
- Hilber-Hughes-Taylor, 91, 96
- Hoffman, 110
- hole detection, 46
- hull generation, 44, 46, 105
- hyperelastic material, 84
- hyperelastic material behavior, 10

- incompatible meshing, 21
- inertia relief, 11, 21, 64, 73, 82, 89, 93, 99
- interfaces, 65
- interpolation region, 58

- Kirchhoff, 61
- laminate optimization, 101, 105
- large strain, 84
- Latin Hypercube Sampling, 71
- layout optimization, 102
- LDR, 107
- LHC, 71
- license manager, 121
- limit state function, 37, 108
- line description, 55
- line of action, 76
- Line Search, 80
- linear buckling, 84
- load correction matrix, 110
- MAC, 68, 88
- machine tool, 31
- magneto-statics, 111
- manufacturing constraints, 43, 103
- material
 - cast-iron, 83
 - Drucker-Prager, 81
 - Mohr-Coulomb, 81
 - properties, 62
 - Tresca, 81
 - viscoplastic, 81
 - von Mises, 81
- matrix
 - Coriolis, 31
 - gyroscopic, 31
- matrix models, 22, 67
- maximum member size, 103
- Maxwell, 111
- MBC transformation, 55
- MBCB condensation, 89
- MBS, 21
- message tree, 42
- minimum member size, 103
- MLDR, 23, 90, 91
- MMO, 100
- modal analysis, 86, 88
- modal participation factors, 84, 94
- modal random response, 96
- model updating, 24
- model verification, 64
- Mohr-Coulomb, 81
- Monte Carlo, 108
- morphing, 45
- MPC, 11, 56 to 58, 73, 113
 - as constraint, 11
 - coupling of surfaces, 58
 - coupling of volumes, 59
 - general, 59
 - interpolation region, 58
 - press fit, 59
 - rigid body, 58
- MpCCI, 17, 70, 117
- MS, 107
- Multi-Blade Coordinate transformation, 55
- Multi-Body Systems, 21
- Multi-Level Dynamic Reduction, 23, 91
- multi-modal optimization, 100
- multi-objective design optimization, 98
- multi-start method, 107
- NAFEMS, 109
- Newmark, 91, 96, 97
 - β , 91, 96
- Newton, 106
- Newton-Raphson, 80, 82, 97
- NLSA, 10
- nodelock license, 121
- non-parametric optimization, 100
- OC, 99
- optimality criteria method, 99
- optimization
 - bead, 98
 - contact pressure, 106
 - derivative-free, 106
 - design elements, 62
 - free-form, 98, 100
 - frequency response, 98
 - global, 106
 - layout, 102
 - non-parametric, 100
 - positioning, 98
 - shape, 98
 - size, 98
 - topology, 102
 - trust region, 106
- optimization
 - laminate, 101, 105
- optimization for a robust design, 101
- option COMPLEMENT, 74
- parallelization, 18, 87, 121
- Pareto, 98
- part connections, 26
- part coupling, 56
- PD, 104
- PERMAS Tools, 68
- PERMASgraph, 68, 122

- plasticity
 - isotropic, 81
 - transverse isotropic, 12
- ply failure criteria, 101, 110
- polygon reduction, 44, 105
- post-processing, 43
- power spectral density, 95
- pre-buckling behavior, 85
- press fit, 74, 78
- Pressfit wizard, 42
- pretension wizard, 49
- Primal-Dual Solution, 104
- probability of failure, 108
- Prony series, 94
- pyINTES, 9, 122
- Python, 39
- python, 9, 69, 122

- quality assurance, 71

- radiation, 87
- random response analysis, 95
- Rayleigh, 93, 96
- RBM Assistant, 42, 75
- real eigenvalues, 88
- reduction
 - Guyan, 53, 89, 114
- reference system
 - co-rotating, 31
 - inertial, 31
- release directions, 103
- reliability analysis, 107
- response
 - in frequency domain, 92
 - in time domain, 91
 - random, 95, 96
 - spectral, 94
 - steady-state, 92
 - transient, 93
- response spectrum, 94
- response surface methods, 108
- restart, 69
- results
 - combination, 67
 - comparison, 68
 - transformation, 68
 - xy data, 68
- robust design, 36, 101
- rolling, 82
- rotating systems, 30
- rotor dynamics, 31
- Rutherford-Boeing, 67

- sampling, 71
- Sampling wizard, 42
- sandwich shells, 61
- SBV, 45
- SCP, 106
- SDM, 71
- self contact, 73
- sets
 - element sets, 63
 - node sets, 63
- Shape Basis Vectors, 98
- shape basis vectors, 45
- shape optimization, 45, 98
- Shermann-Morisson-Woodbury, 12, 94
- shift
 - buckling, 12
 - dyn. eigenvalues, 88
- shift method, 12, 85, 88
- short fibre material, 12, 82
- Simulation Data Management, 71
- situation, 54
- sizing, 98
- sizing optimization, 44
- SLP, 106
- Smooth Patch Recovery Method, 62
- smoothing, 44, 105
- SMW, 12, 94
- solver
 - CLF, 10
- spectral response analysis, 94
- spotweld, 57
- SPR stresses, 62
- SQP, 106
- static mode shapes, 93, 96
- steady state response, 22
 - rotating systems, 31
- steady-state response, 31, 92
- stiffness
 - centrifugal, 88
 - convective, 88
 - frequency-dependent, 92
 - geometric, 88
 - pressure, 88
 - rotation speed dependent, 31, 92
 - spotweld, 57
- STL, 105
- stochastic analysis, 107
- strain
 - large, 11
- Stress Gradients, 62
- submodeling, 25, 53, 77, 83

subspace iteration, 88
substructure technique, 59, 71, 83, 97
substructure wizard, 50
substructuring, 53, 54, 99, 103, 114
summation rules, 94
surface description, 55
system kernel, 71
system of units, 69

task scanner, 71
temperature field, 85
temperature mapping, 25
Thomas, 82
tied contact, 11
tooltips, 40
topology optimization, 102
transfer function, 22
transient response, 91
transverse isotropic plasticity, 12
Tresca, 81
trust region, 106
Tsai-Wu, 110

user defined material, 82

variant, 54
variant analysis, 54
view factors, 87

visco-elastic material, 94
visco-plasticity, 81
VisPER, 5, 9, 14, 39
volume-shell transition, 59
von Mises, 81
voxel, 47

wizard, 43, 44, 46, 49
 add&replace, 41
 bead design, 45
 brake squeal, 42
 contact, 49
 design, 42
 fluid-structure coupling, 46
 free-form optimization, 45
 pressfit, 42
 pretension, 49
 sampling, 42
 shape optimization, 45
 sizing optimization, 44
 substructure, 50
 topology optimization, 43

WLIN, 107

XML, 72

zero force springs, 75
Zienkiewicz, 62

Contact:

Phone: +49 (0)711 784 99 - 0

Fax: +49 711 784 99 - 10

E-mail: info@intes.deWWW: <http://www.intes.de>

Address: INTES GmbH

Breitwiesenstr. 28

D-70565 Stuttgart