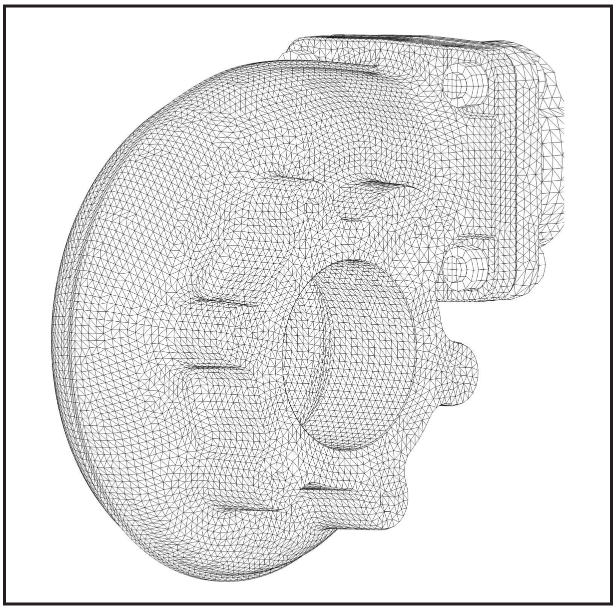

PERMAS

Short Description
Version 13

Ingenieurgesellschaft für
technische Software mbH
INTES

Contents

The finite element model of a turbocharger housing on the frontpage appears by courtesy of BorgWarner Turbo Systems Engineering GmbH in Kirchheimbolanden, Germany.

Address: **INTES GmbH**
Schulze-Delitzsch-Str. 16
D-70565 Stuttgart

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

E-mail: **info@intes.de**
WWW: **http://www.intes.de**

INTES

Company Profile

INTES company was founded as an FE technology enterprise in 1984. Its competence in every aspect of Finite Element technology is provided by INTES to its clients not only thru 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 PERMAS,
- development of new and efficient numerical methods,
- development of software for new hardware architectures (such as parallel computers),
- coupling of PERMAS with other software systems (such as CAD systems and pre- and post-processors),
- consultancy and training of users,
- performance of analysis projects.

The international support of PERMAS clients is supported in France by INTES France and in Japan by INTES Japan.

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.

Services

INTES offers a number of services to its customers including:

- Developments for PERMAS:
  - Interfaces to other software packages,
  - New analysis capabilities,
  - New finite elements,
  - Customer specific developments,
- Installation of PERMAS 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, MEDINA,
  - simulation with PERMAS,
- Introduction of FE analysis in enterprises, continuous consultation service (hotline), and support on current projects.

PERMAS

Overview

This short 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 6 to 13.
- **Applications** using several functional modules are illustrated on pages 15 to 25
- The features of **VisPER** are described on pages 27 to 34.
- The **universal features** of PERMAS, which are not related to a single module, are explained on pages 38 to 51.
- The available **functional modules** are described on pages 53 to 74.
- The **interfaces** are collected on pages 76 to 79.
- Additional information about the **installation and further aspects** of PERMAS is given on pages 81 to 84.

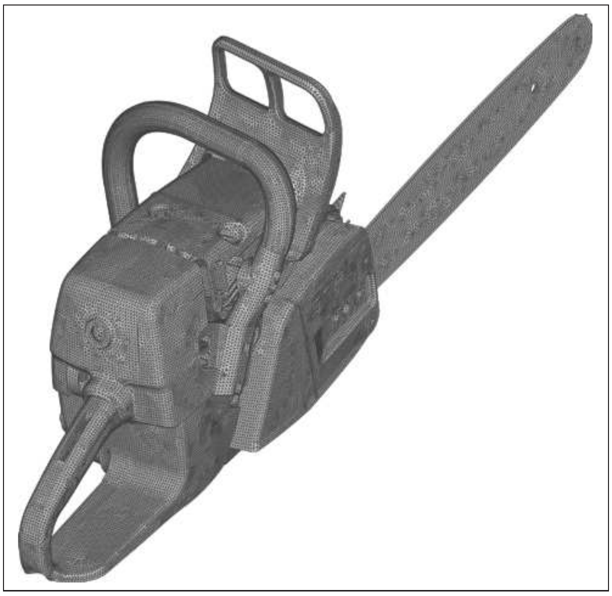

Figure 1: Model of a chain saw
Andreas Stihl AG & Co. KG, Waiblingen,Germany.

Introduction to PERMAS

PERMAS is a general purpose software system to perform complex calculations in engineering using the finite element method (FEM). 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 and electromagnetic fields, analysis of anisotropic material like fibre-reinforced composites.

PERMAS determines 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:
  - check of design variants,
  - insight to correlated structural factors,
  - detection of structural performance reserves,
  - hints for saving material.
- 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 indispensible tool for the daily development work. Moreover, complex products are often developed in distributed structured companies. This makes interdependencies

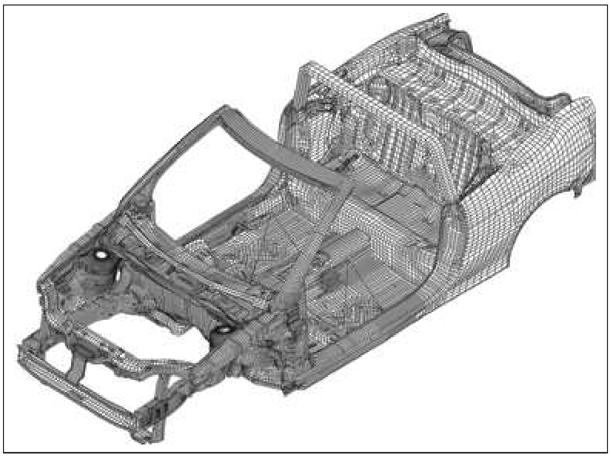

Figure 2: Car body model
Wilhelm Karmann GmbH, Osnabrück, Germany.

between different components of the product visible in time only if they are simulated and analysed 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 account uncertain model parameters.
- The graphical user interface VisPER supports the user in verifying his modells and in evaluation of the analysis results. Moreover, VisPER provides advanced modelling 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.

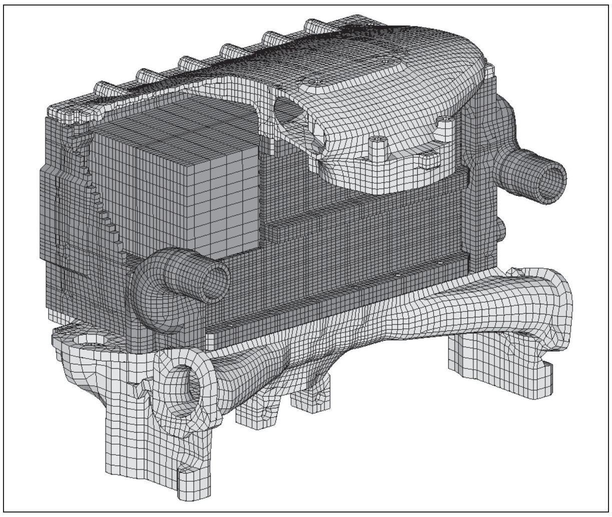

Figure 3: Charge air cooler
Behr GmbH & Co., Stuttgart, Germany.

What's New in PERMAS Version 13

The new Version 13 of PERMAS is the result of about 24 months of development work since the shipment of the predecessor version 12. For the regular reader of our Short 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 13 in addition.

Great effort has been spent in the past years to provide with *VisPER* (i.e. Visual PERMAS) a dedicated tool to improve pre- and post-processing for special PERMAS functions. The second regular VisPER Version 2 is released at about the same time as PERMAS Version 13. More information on VisPER can be found on page 27.

PERMAS Version 13 offers again improved computing performance:

- Gasket elements are now handled by contact analysis instead of nonlinear material algorithm (leading to run time reduction by a factor of larger than 10, if no other material nonlinearities are present; see module NLS on page 57),
- Contact analysis with normal contacts only or with normal and frictional contacts shows run time reductions of up to 40 percent for some models (see module CA on page 54),
- A new eigenvalue solver kernel shows significantly improved performance for a larger number of eigenmodes (see module DEV on page 59),
- Faster calculation of modal participation factors for a modal timehistory response analysis (see module DRA on page 61),
- Iterative solver option for modal frequency response analysis (recommended for a large modal basis; see module DRA on page 61),
- Improved performance and disk space reduction for direct transient temperature analysis (see module HT on page 66),
- Iterative solver option for steady-state temperature analysis (see module HT on page 66),

Version 13 has started to parallelize element operations. A number of element routines are parallelized and will result in a corresponding speed-up. Mainly, linear and non-linear static analysis as well as linear dynamic analysis will benfit from this additional parallelization. However, there are still element subroutines which are running in sequential mode only. It is the intention to complete this parallelization of element operations in the next releases.

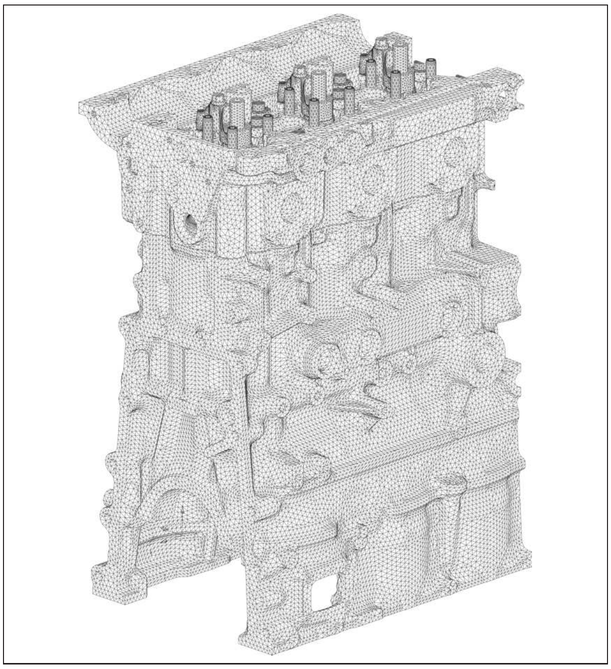

Figure 4: Half model of a six-cylinder Diesel engine
Daimler Trucks North America in Detroit, Michigan, USA.

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

- **New modules:**
  - In order to provide more optimization methods a new module AOS (Advanced Optimization Solvers) has been created. It contains additional derivative based methods but also derivative free methods, as well as globalized and global methods. With these new methods it is now possible to optimize nonlinear analysis tasks like contact problems or nonlinear material analysis. Global methods can be used to optimize highly nonlinear optimization problems, where derivative based methods fail (see module AOS on page 71).
  - A new interface to AVL EXCITE is available (see module EXCI on page 77).
  - A new interface to Altair's MotionSolve is available (see module HMS on page 77).
- **Major extensions:**
  - Extensions to basic module (module MQA, page 53):
    - Most of the names defined by the user in the input deck can now have long names (up to 40 characters). These include names for materials, components, variants, situations, geometrical data, sets, contacts, parts, pretensions. Some of

these names can be further described using a free text, like contacts, parts, and pretensions.

  * A *user stop file* is provided to explicitly break iterative procedures without losing already computed results, like contact analysis, nonlinear static analysis, nonlinear heat transfer analysis, or optimization.
  * A special comment feature supports improved communication with *SDM* (*Simulation Data Management*) systems. These comments provide a means to describe many entities 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.

– Extensions to contact analysis (modules CA and CAX, page 54):
  * Gasket elements can now be handled as integral part of the contact iteration instead of a feature in nonlinear material 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.
  * Information on gap widths is now available as scalar result and as vector result.
  * Contact locking was extended to couple frictional degrees of freedom even in cases where the direction of these degrees of freedom does not match the local system at the node.

– Extensions to nonlinear static analysis (module NLS, page 57):
  * *Initial conditions* (e.g. from *casting*) can be described as initial strains (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.
  * Extrapolation of stresses for linear and nonlinear elements is now fully provided. For linear elements, stresses are calculated at the stress points and either assigned or extrapolated to the element corner nodes. For nonlinear elements, stresses are calculated at the integration points and either assigned to the element nodes or extrapolated to the element corner nodes. If stresses are not assigned to the element midside nodes, these stresses will be calculated through linear interpolation from the element corner nodes. In plasticity analysis, stresses at the element nodes are showing stresses at the integration points or they are showing extrapolated stresses which do not lie on the yield surface any more. In case of subsequent generation of nodal point stresses in post-processing, the original element stress results can be seriously modified.

– Extensions to dynamic response analysis (module DRA, page 61):
  * Modeling of damping was extended to any component including subcomponents by *Rayleigh damping* and *component structural damping*. In subcomponents, damping is applied to Guyan as well as Craig-Bampton reduced parts. This type of damping can also be used with fluid-structure coupled analysis.
  * In addition, modal damping can be specified by a modal damping matrix. The input of matrix elements can be as viscous damping ratio, damping value, or material damping coefficient. The modal damping matrix may be diagonal, symmetric, or non-symmetric. The modal damping matrix is applicable to systems with modal degrees of freedom including Craig-Bampton modes.
  * For rotating structures, any number of rotational speeds is now 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 dy-

namic response analysis. This procedure makes the generation of Campbell diagrams very efficient and the response analysis of rotating structures is facilitated.

- Extensions to design optimization (module OPT, page 67):
  * New *design elements* for shape optimization, bead design, and free coordinate modifications are available. Shape optimization is supported for any arbitrarily shaped design space, with an arbitrary number and location of design nodes, and with a smooth interpolation between design spaces (and boundary). Bead design supports automatic generation of bead design variables according to a user-defined reference.
  * Shape optimization and bead design can always be combined with sizing optimization.
  * Contact analysis results are available as new design constraints: contact pressure, contact reaction forces, and contact gap widths.
  * Heat flux results from heat transfer analysis is available as new design constraint.
  * Material parameters are available as design variables in optimization.
- In topology optimization (module TOPO, page 69) a *fixed mold parting line* can be defined for opposite release directions.
- In reliability analysis (module RA on page 73) material parameters can now be used in the same way as for optimization.
- 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 55), material curves for plasticity, and transient loads in timehistory response analysis.

- **New elements:**
  - A new nonlinear control element with 3 nodes and 6 natural forces. This element has been developed in order to model e.g. nonlinear interaction of two parts like between tool and workpiece, or between hub and shaft in a journal bearing.

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:

- **MEDINA (MEDI)** (see page 76):
  - Support of MEDINA 8.2,
  - Support of new contact menus,
  - Enable material and property labels as comment,
  - Transfer of connector label to corresponding property label,
  - Support of contact names,
  - Long names for sets, components, variants, situations, contact, geometrical data,
  - Long material names as default for MEDINA door,
  - Export of new PERMAS Version 13 results.
- **NASTRAN (NAS)** (see page 78):
  - Long names for sets, components, situations, geometrical data,
  - PBARL support (rod, tube, and bar).
- **HyperView (H3D)** (see page 78):
  - Support of HyperView 8.0 Libraries,
  - Post-processing on HyperView 11.0 libraries possible.
- **ADAMS (AD)** (see page 77):
  - Support of MNF library ADAMS 2007 r1,
  - Extensions for invariants (mass, inertia),
  - Support of modal general condensation.

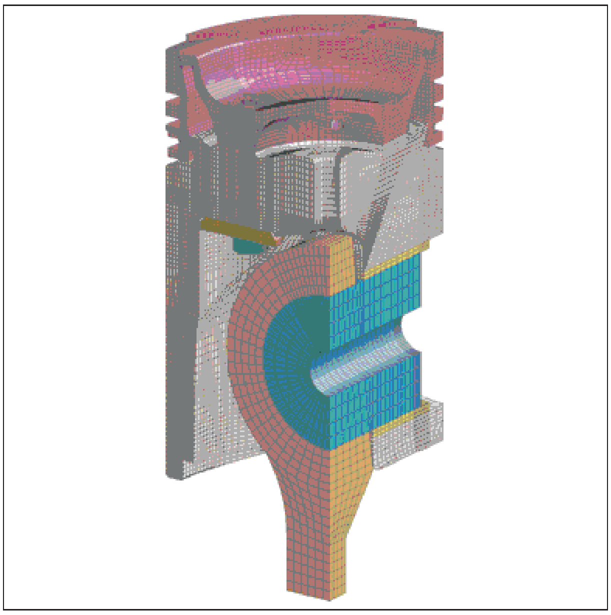

Figure 5: Ship engine piston
Mahle GmbH, Stuttgart, Germany.

The PERMAS Tools (see page 82 have been extended to use additional formats (MEDINA, PATRAN) for XY data plotting.

For all system platforms an update to the current release of the operating system had been performed (see Page 81).

Universal Features

The outstanding mostly module-independent basic features of PERMAS are as follows (see pages 37 to 51):

- Hierarchical *substructuring*, with automatic sub-component insertion (see page 37)
- Submodeling (see page 37)
- Variant analysis (see page 38)
- Cyclic symmetry (see page 39)
- Surface and Line Description (see page 39)
- Automated coupling of parts (see page 39)
- Automated spotweld modeling (see page 41)
- Multiple kinematic constraints (see page 41)
- Automatic detection of singularities (see page 42)
- Same elements for different analysis types (Element library, see page 42)
- Standard beam cross sections (Seite 44)
- Design elements for optimization (page 44)
- Error estimation and refinement indicator (page 44)
- General material description (see page 45)
- Node and element sets (see page 45)
- Mathematical functions (see page 45)
- All kinds of loading (see page 46)
- Model verification (see page 46)
- Integrated interfaces to pre- and post-processors (see page 47)
- Input and Output of Data Objects and matrices (see page 49)
- Combination, transformation, and comparison of results (see page 49)
- Output of XY result data (see page 50)
- Calculation of cutting forces (see page 50)
- Restart facility (see page 50)
- Open software through Fortran and C interfaces (see page 51)
- Direct coupling of different analysis types (see page 51)
- Coupling with CFD (see page 51)

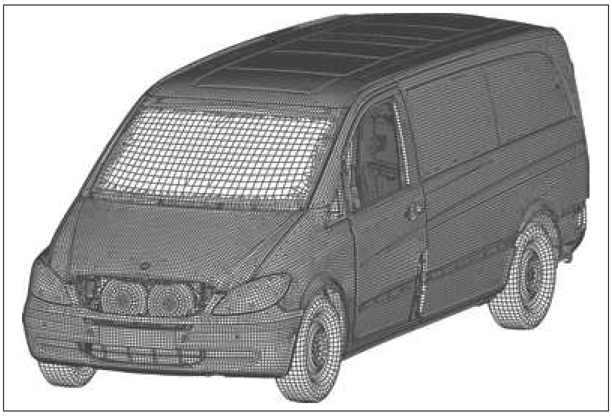

Figure 6: Model of a transport vehicle
courtesy of Daimler AG,
Commercial Vehicle Division in Stuttgart

Available VisPER Modules

The below listed functional modules are explained in more detail on pages 28 to 33:

- 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 53 to 79:

- Model Quality Assurance (MQA)
- Linear Statics (LS)
- Refined Weldspot Model (WLDS)
- Contact Analysis (CA)
- Extended Contact Analysis (CAX)
- Nonlinear Statics (NLS)
- Extended Nonlinear Material Laws (NLSMAT)
- Buckling Analysis (BA)
- 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)
- Heat Transfer (HT)
- Nonlinear Heat Transfer (NLHT)
- Laminate Analysis (LA)
- Design Optimization (OPT)
- Layout Optimization (TOPO)
- Advanced optimization solvers (AOS)
- Reliability Analysis (RA)
- Steady-state electromagnetics (EMS)
- Electrodynamics (EMD)
- Interfaces to various pre-/post-processors
  - MEDINA (MEDI)
  - PATRAN (PAT)
- Interfaces to CAD systems with Pre- and Post-processor
  - 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)
  - MATLAB (MAT)
  - NASTRAN (NAS)
  - MpCCI (CCL)

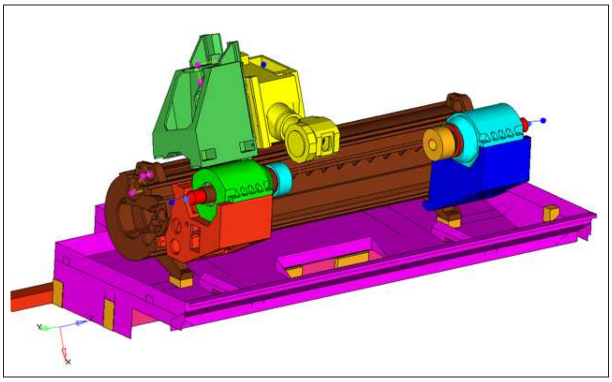

Figure 7: Machine tool
INDEX-Werke GmbH & Co. KG, Esslingen, Germany

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.

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.

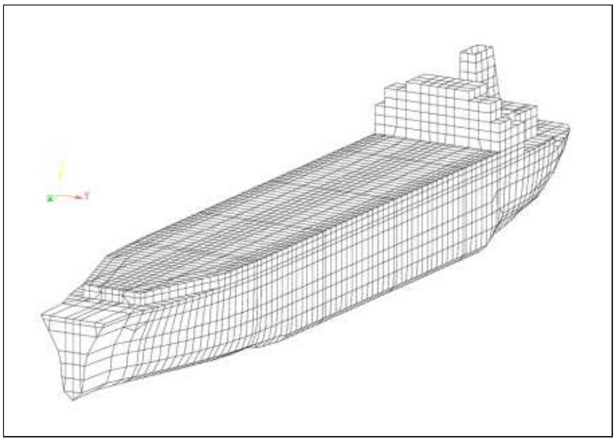

Figure 8: Methane Carrier

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 parallel execution of PERMAS is very simple. Because there are no special commands necessary, 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.

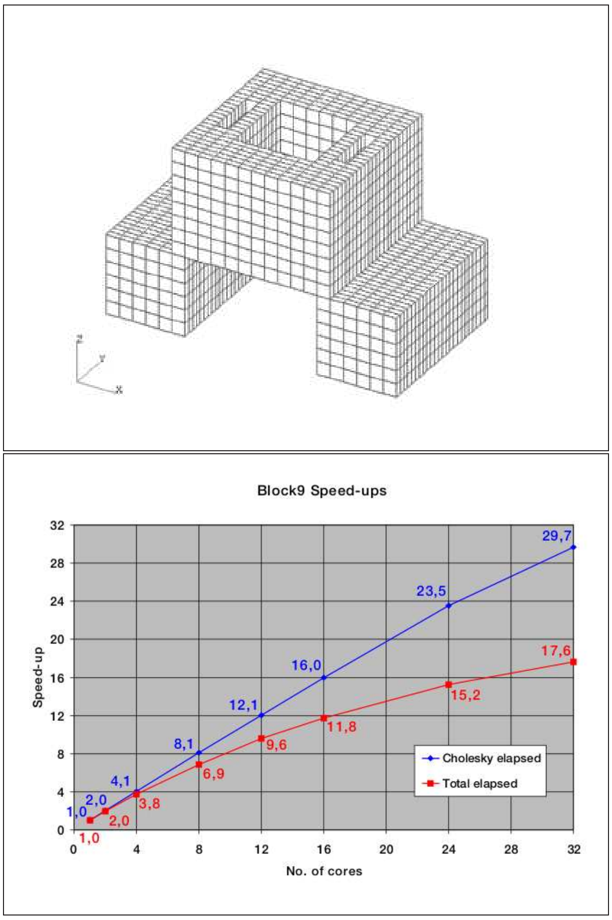

Figure 9: Static analysis with 3 loading cases
1.5M nodes, 176k HEXE27, 4.4M Dof
run time on Intel Boxboro

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 makes 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 53). 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 46). In addition, VisPER provides a model verification environment for a growing number of modeling parameters (see page 28).
- **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.

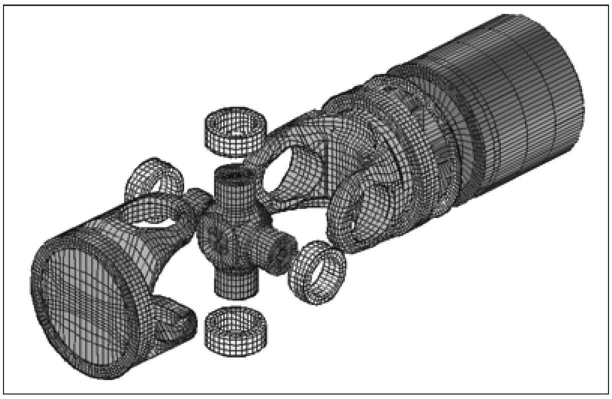

Figure 10: Model of a cardan shaft
Voith Turbo GmbH & Co. KG, Heidenheim, Germany.

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 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 subroutines 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 tests run 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-occurance 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 41 and module WLDS on page 54), bonding, laser welding. In order to generate the meshes of all parts efficiently, *incompatible meshing* (see page 39) is used for independent meshing.

A special feature in VisPER supports post-processing of spotwelds (see page 34) 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 impacts calculations of *inertia relief* (see page 54) are applied.

To check the force flow through any structural member, *cutting forces* (e.g. through a column or sill, see page 50) 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.

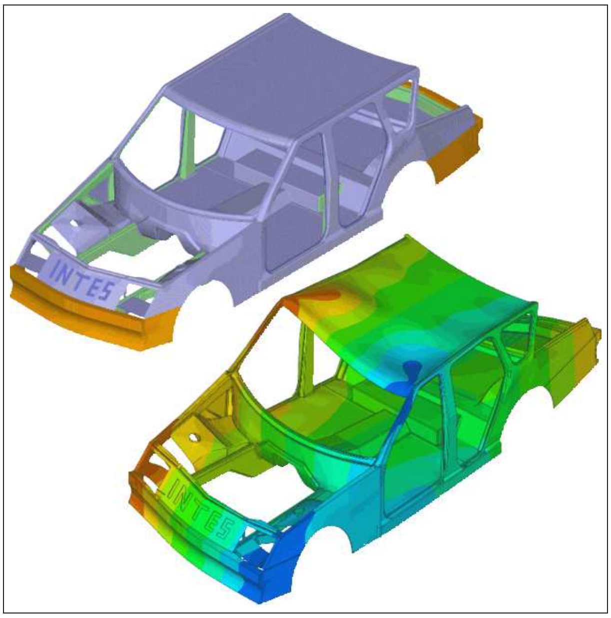

Figure 11: Workshop example INTEScar
under torsional loading

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 77).

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 63).

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 60). Using dynamic condensation so-called *matrix models* (see page 49) 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 61), 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. 62 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 62). In particular, trimmed bodies require a detailed and accurate modeling of all additional springs, masses, and dampers.

The results from a frequency response analysis are any complex primary result (displacements, veloctities, or accelarations) 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 repsonse 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 31). 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 64 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 61. 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 63) 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.

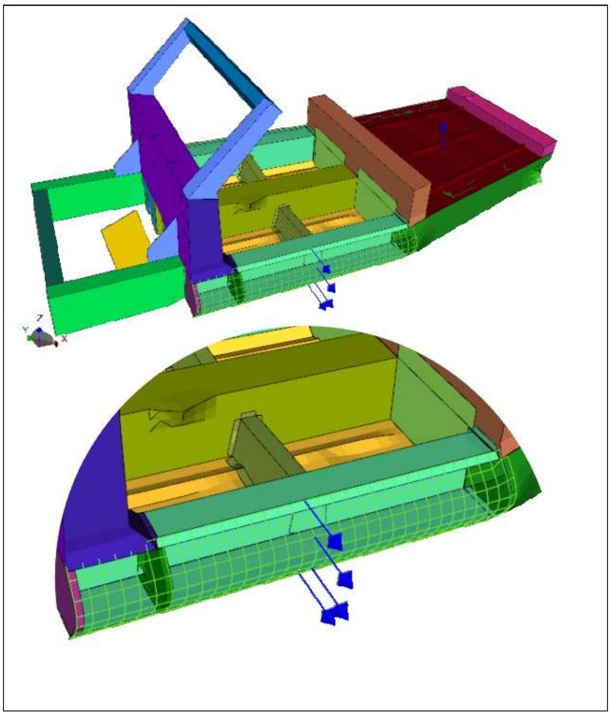

Figure 12: Shape optimization of a sill
with transition to neighboured parts

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 67 to 73) 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.

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 29). 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 experimantal 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.

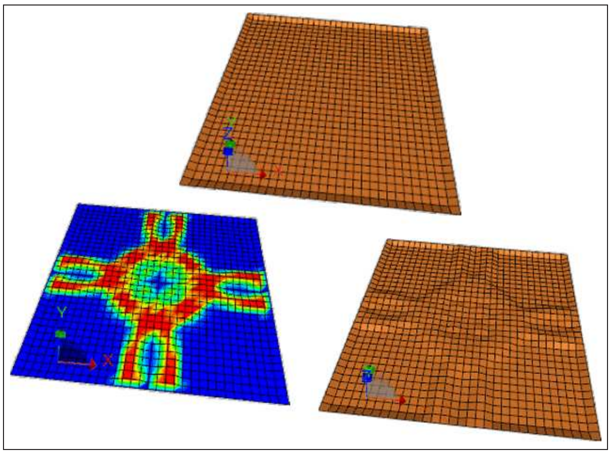

Figure 13: Bead design of a plate
with positioning and height of beads

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.

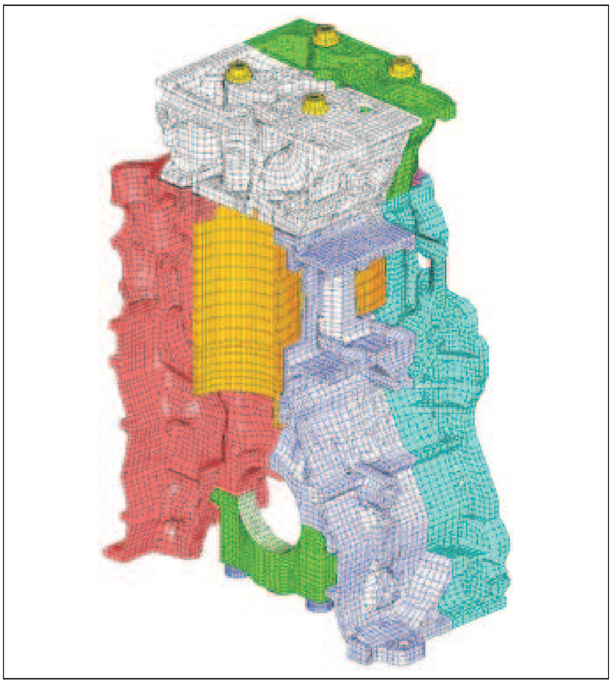

Figure 14: Simple engine model
Daimler AG, Commercial Vehicle Division)

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.

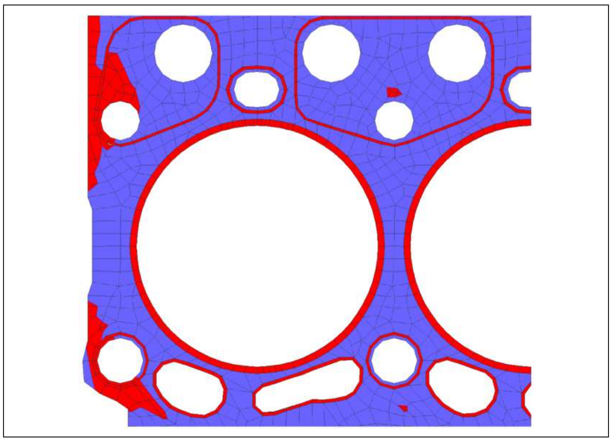

Figure 15: Contact status for gasket elements

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 ($>$ 30,000),
  - 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,
  - 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.

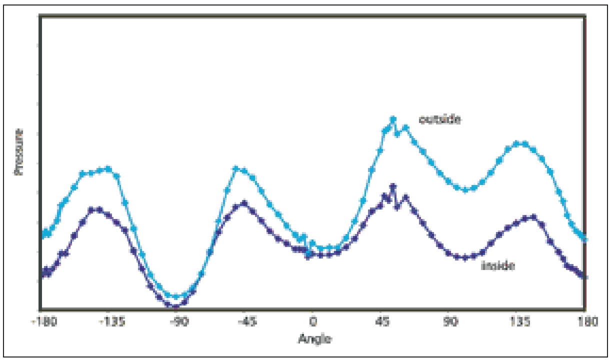

Figure 16: Pressure distribution at stopper over the angle

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.
- An additional speed-up can be obtained, if block and cylinder head behave linear. Then, they can be taken as substructure and after condensation only the nonlinear parts remain in the (much smaller) top component.
- Gasket elements can now be handled as integral part of the contact iteration instead of a feature in nonlinear material 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.

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 59 to 64). Some important points are:

- Eigenvalues and mode shapes for large solid models can be calculated using MLDR (see page 61).
- Fast *dynamic condensation* methods support the efficient analysis of engines with many attached parts (DEVX, see page 60).
- By using *dry condensation* (page 60) 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.

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.

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:

- 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 frictional 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

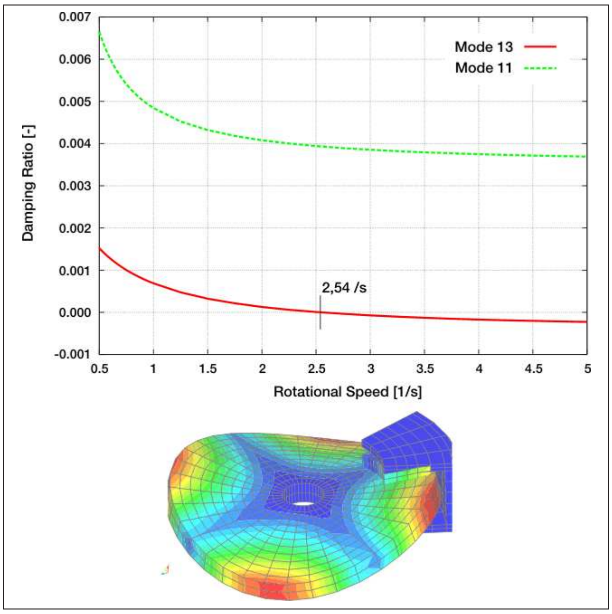

Figure 17: Simple brake model (1)
with an unstable bending mode (m=2, n=1) at 2,54 rps

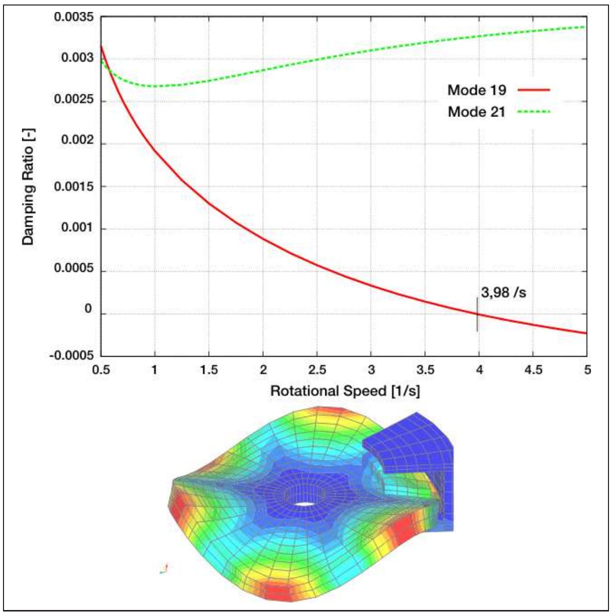

Figure 18: Simple brake model (2)
with an unstable bending mode (m=3, n=1) at 3,98 rps

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.

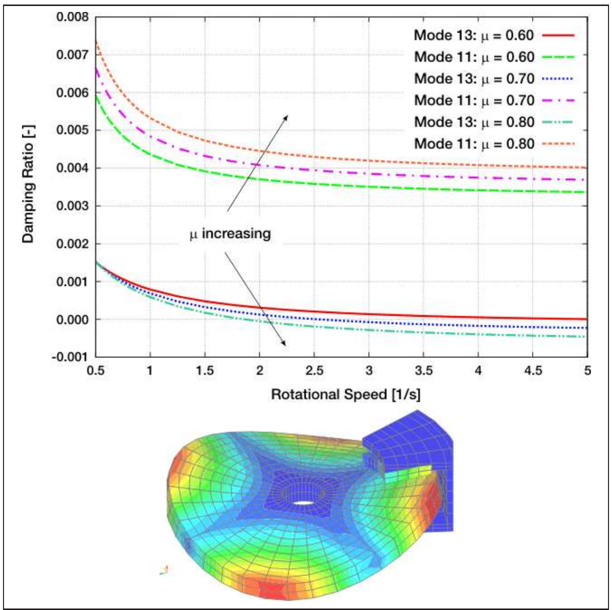

Figure 19: Simple brake model (3)
with an unstable bending mode (m=2, n=1) which becomes unstable at different rotational speeds dependent on the frictional coefficient.

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. 17 and Fig. 18). 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. 19).

A similar study was made to get the influence of Young's modulus of the brake pad. Figure 20 shows a corresponding example.

To illustrate the required run times for such analyses

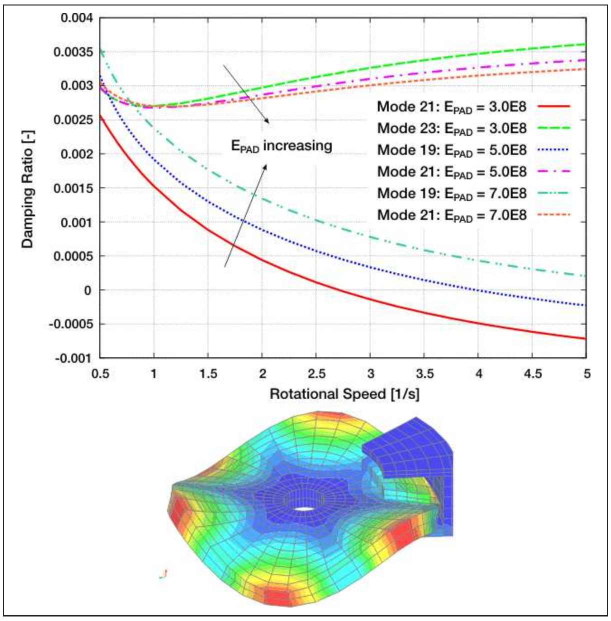

Figure 20: Simple brake model (4)
with an unstable bending mode (m=3, n=1) which becomes unstable at different rotational speeds dependent on Young's modulus of the brake pad.

corresponding computing times are given for a large industrial model with the following characteristics:

300 000	Elements
500 000	Nodes
1.5 Million	Unknowns
158	Real eigenmodes
316	Complex eigenfrequencies
80	Rotational speeds

For this example the full elapsed run time with PERMAS Version 12 was 1 hour 10 minutes on a 4-CPU Itanium machine with 8 GB memory. The required disk space is about 90 GB.

On a 4-CPU quad core machine the run time was reduced to 19 minutes.

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

Rotating Systems

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

Fig. 22 provides an overview on the analysis capa-

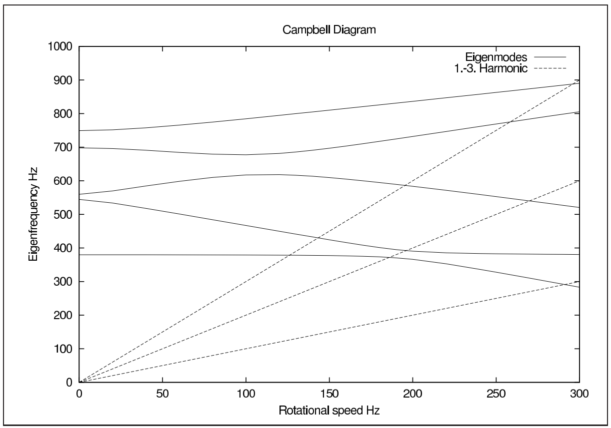

Figure 21: Campbell diagram
for the evaluation of rotor dynamics

bilities for rotating structures. Both co-rotating and inertial reference systems can be applied.

	Co-rotating reference system		Inertial reference system	
	subcrit.	overcrit.	subcrit.	overcrit.
Elastic rotor	✔	✔	✔	✔
Static analysis	✔	–	–	–
Supports	symmetric		arbitrary	
Campbell diagram	✔	✔	✔	✔
Modal response				
- in frequency domain	✔	✔	✔	✔
- steady-state	✔	✔	✔	✔
- in time domain	✔	✔	✔	✔
Direct response				
- in frequency domain	✔	✔	✔	✔
- steady state	✔	✔	✔	✔
- in time domain	✔	✔	✔	✔

Figure 22: Rotor dynamics capabilities

Statics

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. The static analysis is possible below critical speed.

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 page 60) 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 modelled 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 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.

For dynamics modal steady-state response is of particular importance. There, the static stresses unter 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 62).

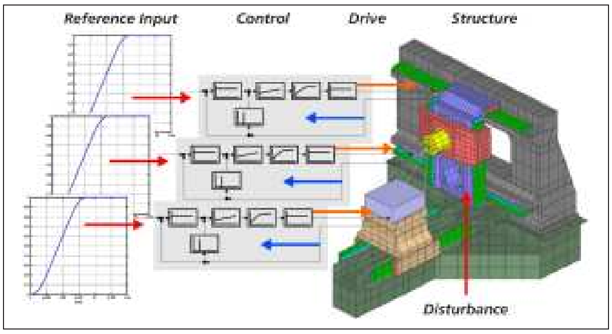

Figure 23: Principles of coupled analysis
with control

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.

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.
- 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 methods in modal space (see page 63).

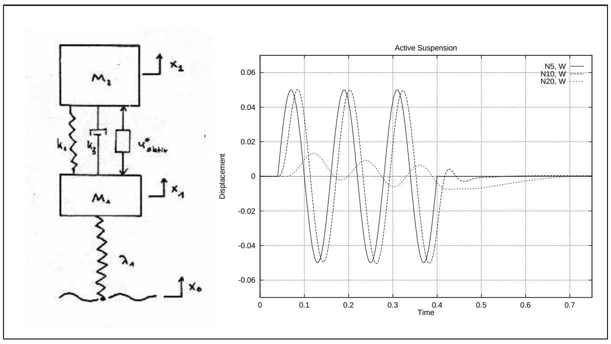

Figure 24: Active suspension unit
u^*_{aktiv} between wheel M1 and body M2 as a function $f(x_0, x_1, x_2, \dot{x}_1, \dot{x}_2)$ with a harmonic base excitation. The body does not show any increase of the amplitude.

- Nonlinear controller element:
  In addition, for nonlinear controls another control element is 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 that element is restricted to modal or direct transient response analysis.

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 inital design.
- A "reliable" optimum may be different than a deterministic one.

The proposed solution is a combination of optimiza-

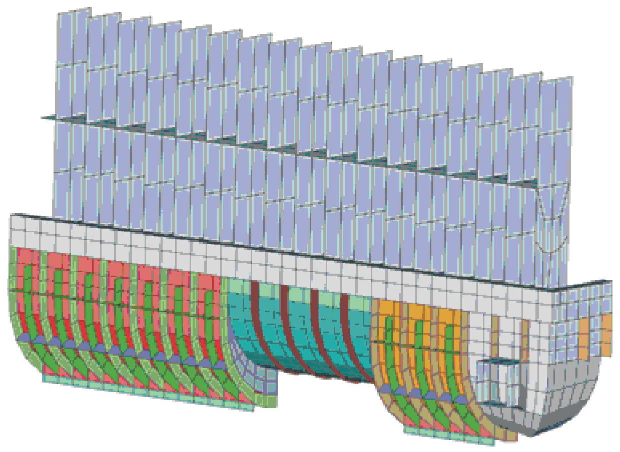

Figure 25: Optimization of a water box
with 18 design variables and 19 stoachstic basic variables

		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

tion 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 fulfill the design constraints.

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

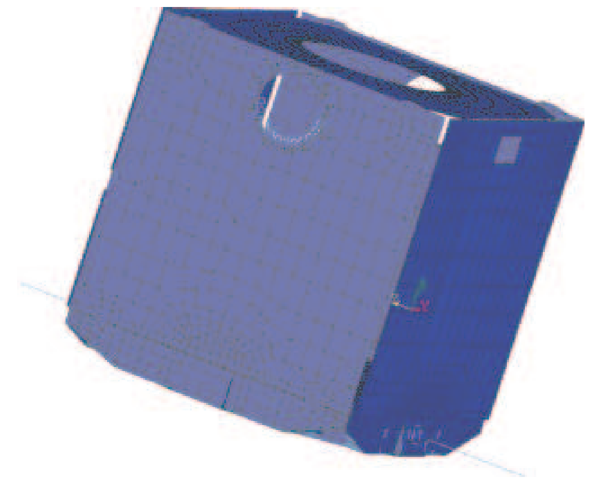

Figure 26: Optimization of PROTEUS satellite with 28 design variables and 30 stoachstic basic variables (Alcatel Space S.A.)

		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.

- **Final limit state**
  - Basic variable values
  - Parameter sensitivities of the limit state function
- **Always available:**
  - Selected data for each iteration

VisPER

Figure 27: The logo of VisPER

The VisPER History

PERMAS as a software for numerical analysis provides many functions which are not sufficiently supported by available 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 a new graphical user interface for PERMAS has been announced on the PERMAS Users' Conference early April 2008 in Stuttgart. This product is named VisPER (for Visual PERMAS). The current version is VisPER Version 2 which is available with PERMAS Version 13.

VisPER was developed in co-operation by science+computing ag (s+c), Tuebingen (www.science-computing.de) and INTES. The graphical functionality was developed by s+c whereas INTES supplied the data base functions, the interfaces and the computational functions.

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 fills efficiently the gap between FE models generated by a standard pre-processor and PERMAS models which are ready to run (see Fig. 28).

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.

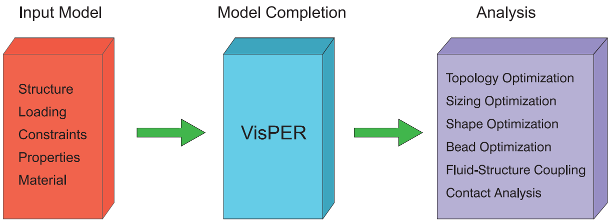

Figure 28: Model editing with VisPER

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

- Topology optimization,
- Sizing optimization,
- Shape optimization,
- Bead optimization,
- Fluid-structure coupling,
- Contact analysis.

The use of VisPER is justified, if

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

To achieve this, loads, kinematic boundary conditions (including MPC conditions) can be visualized, sets and surfaces can be created.

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 is guided using a *wizard* concept which 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 than PERMAS. Hence, models in both programs are identical without translation

or interpretation.

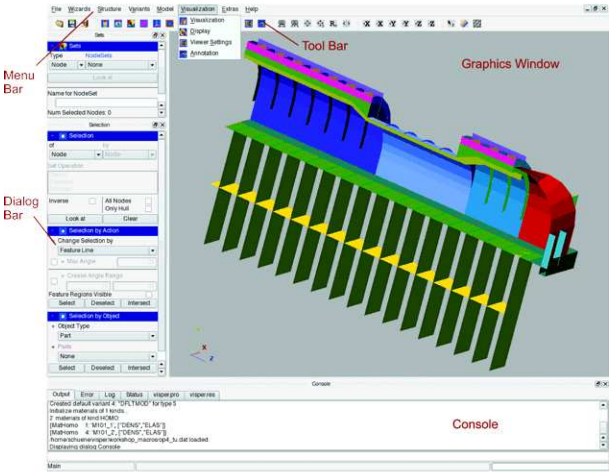

Figure 29: Main Window of VisPER

Fig. 29 shows the main window of VisPER. Each function can be accessed via pull down menus. Shortcuts are predefined for many functions which may be changed 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,
- 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.

The basic functions for pre- and post-processing include:

- functions on model categories like element quality, subcomponents, sets, surfaces, MPC, local systems, nodes, elements, and grouping.
- functions on variants like single point constraints, geometrical element data, and element properties,
- functions on modal data like search and presentation of model information, model configuration with definition of situations, and material definition,
- visualization functions on scalar results, vector results, displacements, and XY data. In addition, a mesh mapper supports the 3D-visualization of points (by a cube) and lines (by a pipe with polygonal cross sections). Generation of snapshots and videos is also supported,
- functions for clipping planes, show/hide elements, and annotations.

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 69.

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 *manufacturing conditions* contain the definition of release directions, filters for minimum and maximum member size, and symmetry conditions (see Fig. 30 as an example). A special

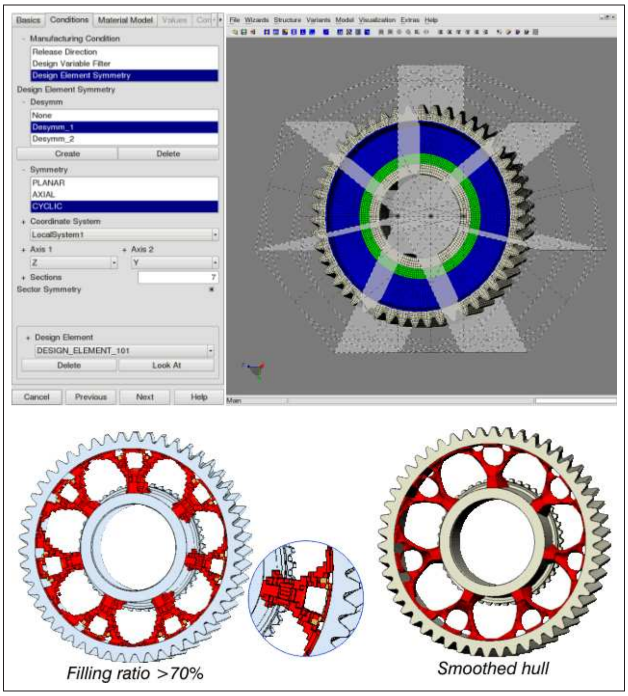

Figure 30: Topology optimization of a gearwheel
Definition of cyclic symmetry and release directions, max. stress at interface between design space and gear rim, max. allowed weight, minimum compliance as objective function (by courtesy of Daimler AG, Stuttgart).

function allows the specification of a *fixed mold parting line* which separates opposing release directions.

3. For the filling ratio a number of conditions can be specified like initial fill, modification limit, and minimum and maximum values.

4. 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 71 for details of these results). Fig. 30 also shows the smoothed hull of the gear wheel body.

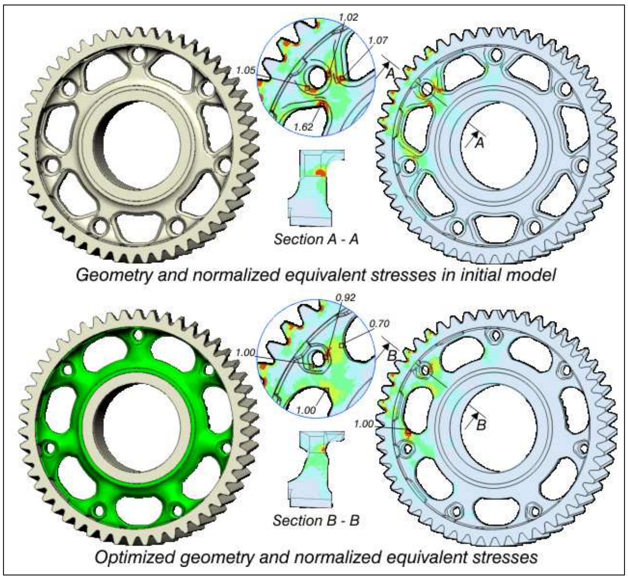

Figure 31: Shape optimization of a gearwheel
Definition of cyclic symmetry, max. stress, minimum weight as objective function (by courtesy of Daimler AG, Stuttgart).

VisPER-OPT – Design Optimization

This module provides a wizard supporting the setup of an optimization model for design optimization tasks. The details of design optimization in PERMAS can be found on page 67.

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.
- Bead design which modifies the coordinates of a shell model to achieve certain stiffness targets by bead generation.

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 modifcation 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, 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. 32 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.

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

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

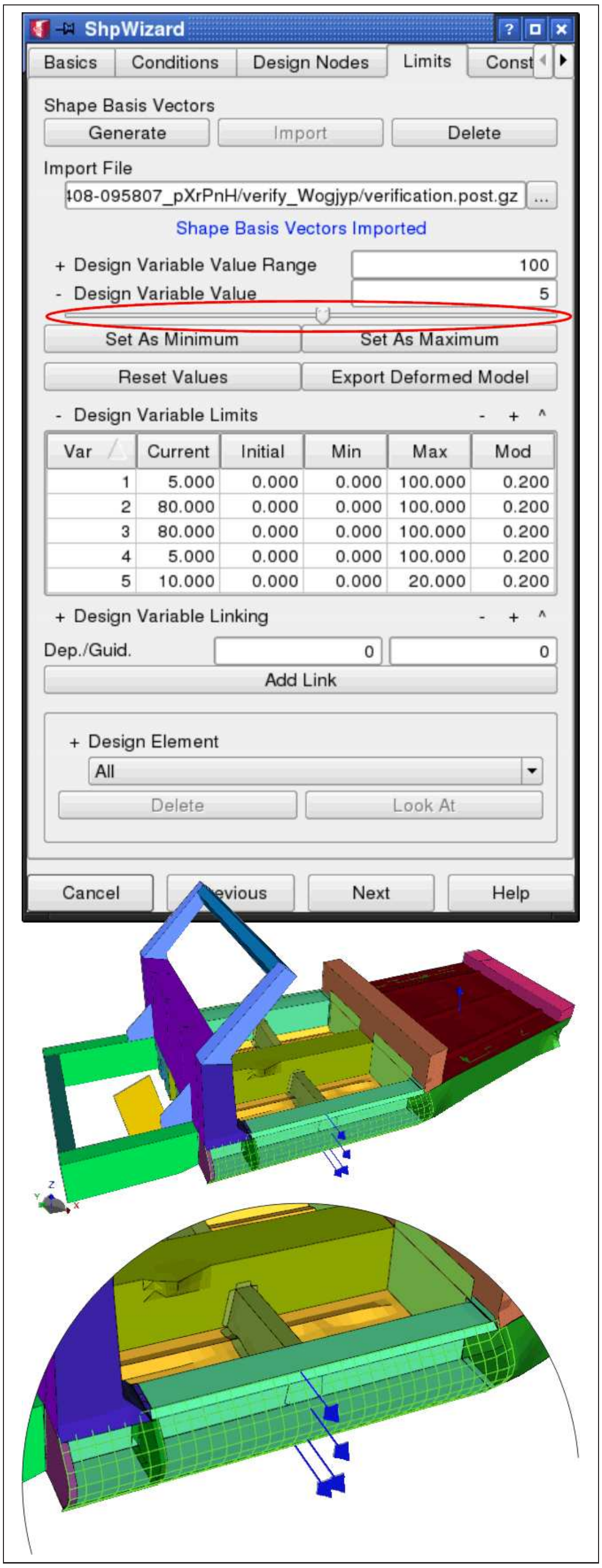

Figure 32: Definition of shape basis vectors of a car sill. The shape changes can be visualized by moving the slider in the menu.

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. 33 and 34:

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 the previously specified mesh 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. There, elements are used which do not introduce any stiffness or mass. They are just used to specify the topology of the hole and to limit the subsequent cavity meshing.

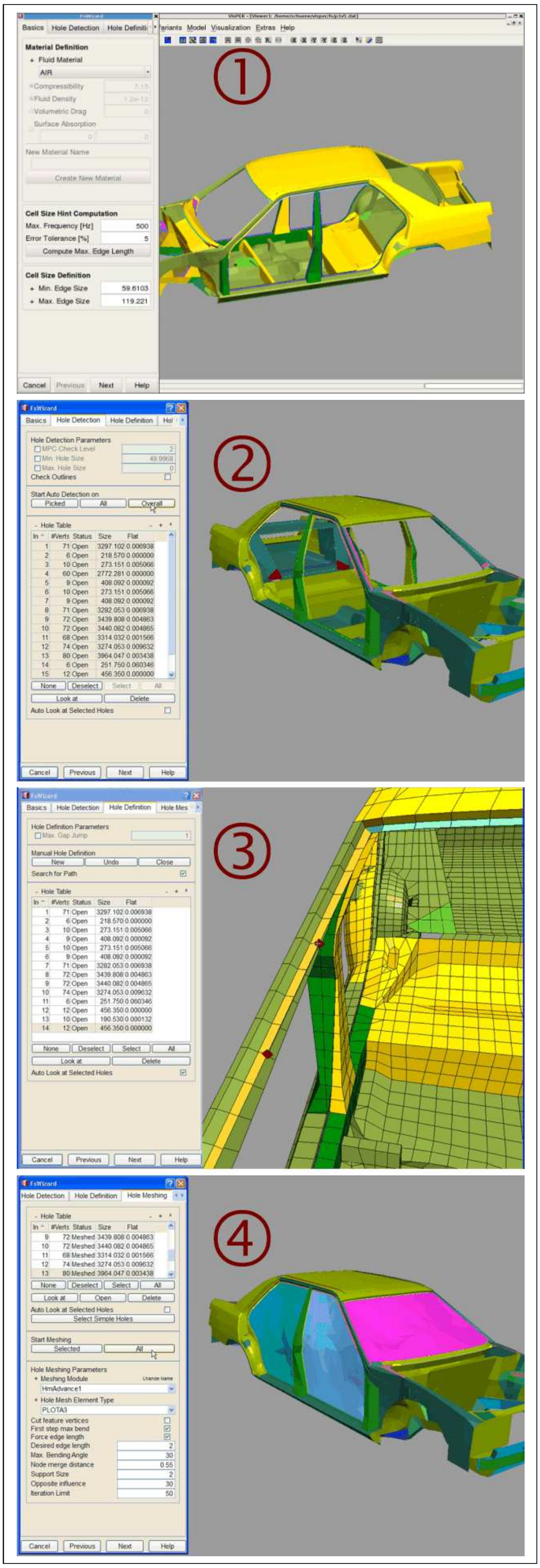

Figure 33: Steps 1 to 4 of the FS wizard

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 mesh. The hexahedra 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 64.

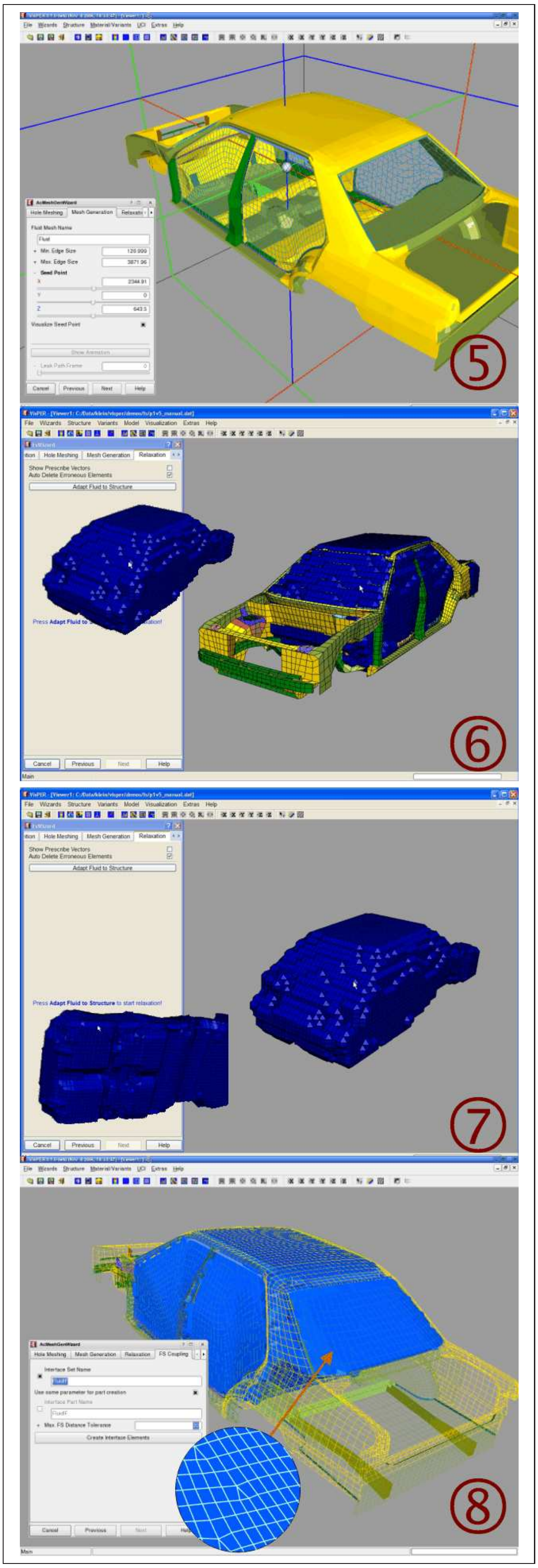

Figure 34: Steps 5 to 8 of the FS wizard

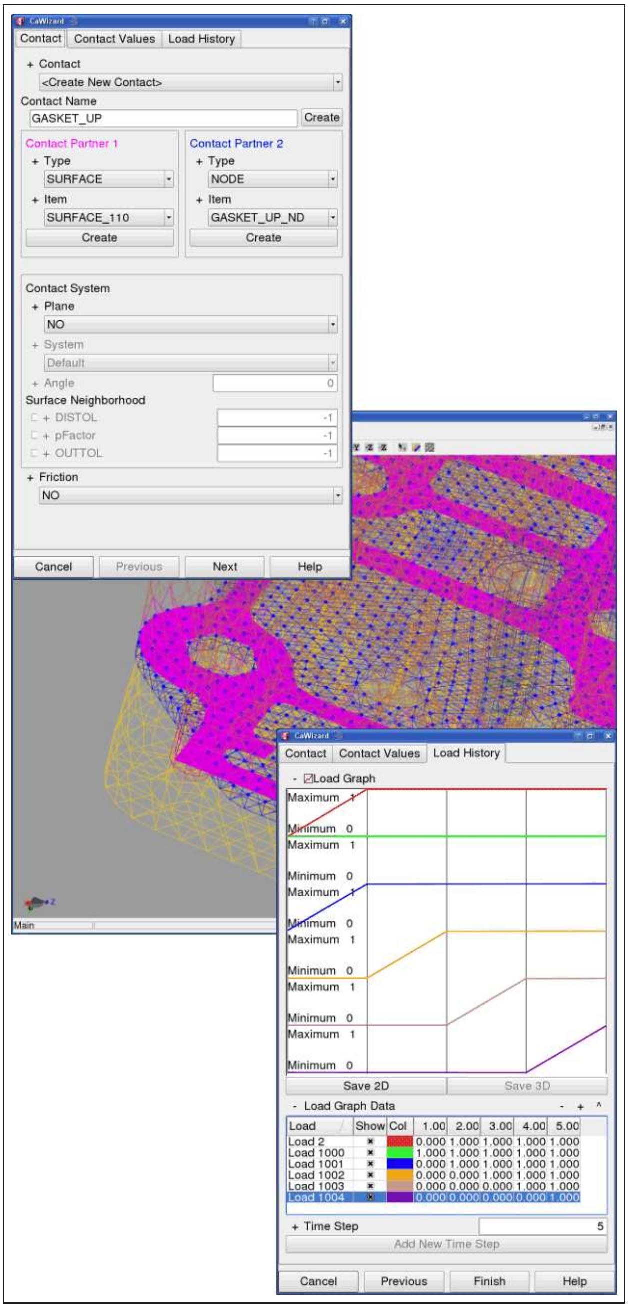

Figure 35: Contact modeling
Contact topology and load history definition

VisPER-CA – Contact Analysis

This module provides a wizard supporting the set-up of a contact model. The details of contact analysis in PERMAS can be found on page 54.

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

- Definition of the contact topology with node-to-node, surface-to-node, or surface-to-surface relationship.
- Definition of the contact details like intial 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. 35.

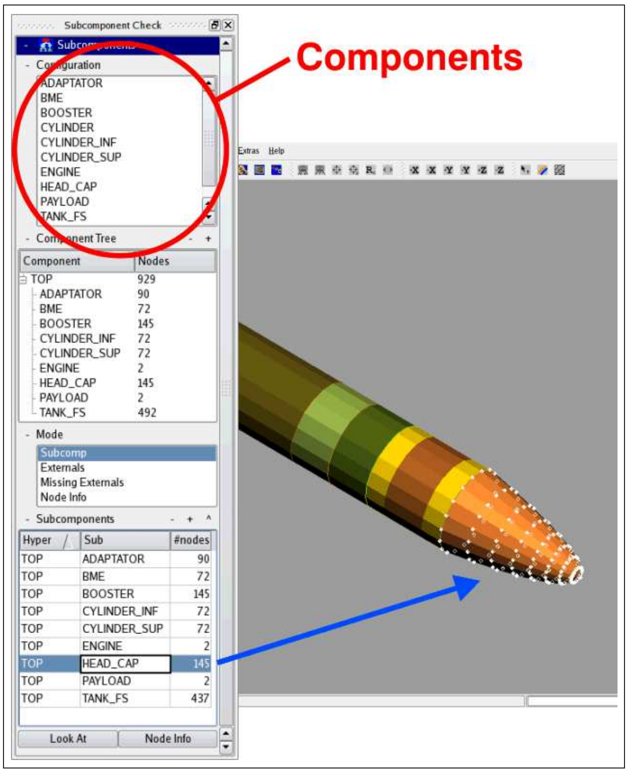

Figure 36: Navigation in the component tree
of a substructure model

Substructuring

Because VisPER uses the data structures of PERMAS, substructure models can be directly imported. Substructuring is explained in more detail on page 37.

The verification of substructure models includes the following points:

- Navigation through the component tree (see Fig. 36).
- Visualization of single components (see Fig. 37).
- Visualization of coupling nodes (see Fig. 38).
- Check of element properties (see Fig. 39).

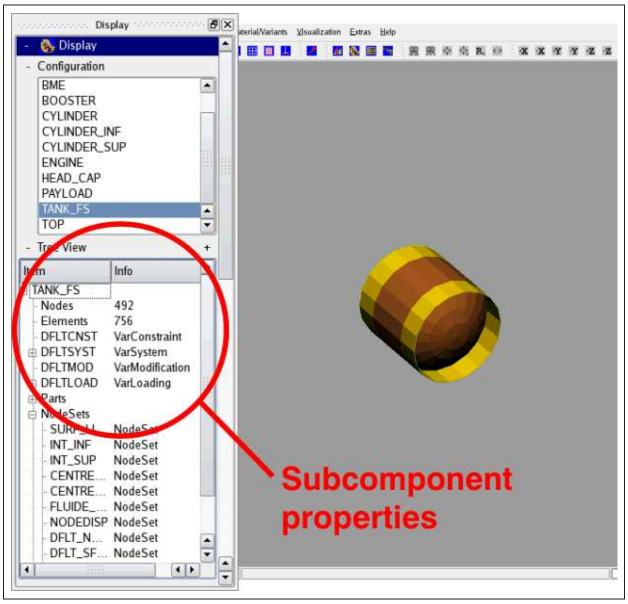

Figure 37: Visualization of components of a substructure model

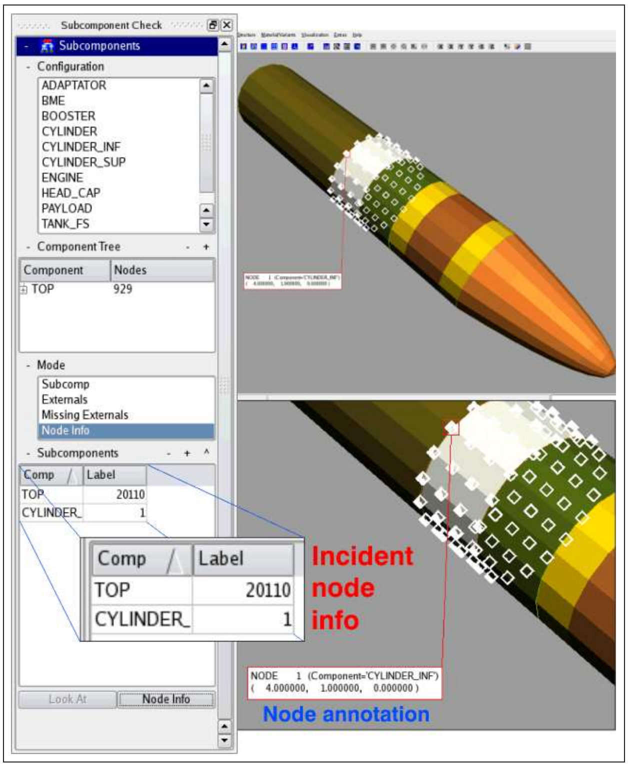

Figure 38: Visualization of coupling nodes of a substructure model

- Display of mode shapes on full structure (see Fig. 40).

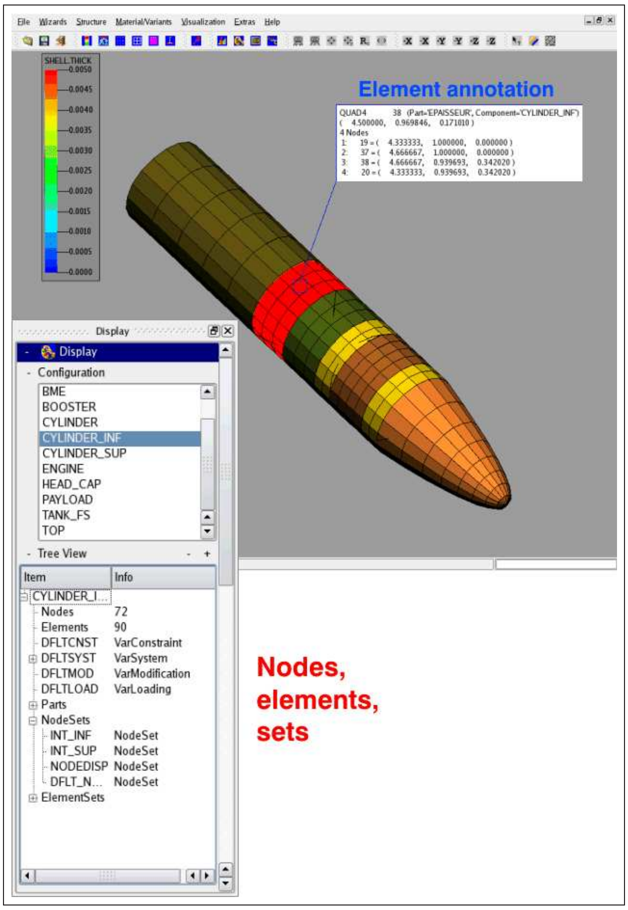

Figure 39: Checking element properties in a substructure model

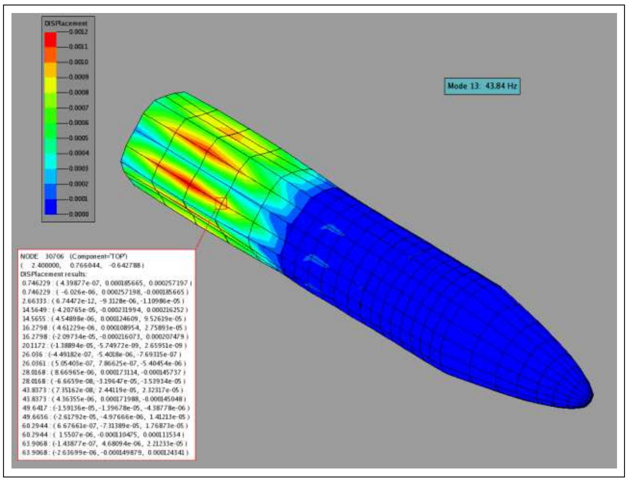

Figure 40: Display of mode shapes on full substructure model

Evaluation of Spotwelds

A special post-processing feature has been included in VisPER providing the evaluation of spotweld forces. 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. 41). 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.

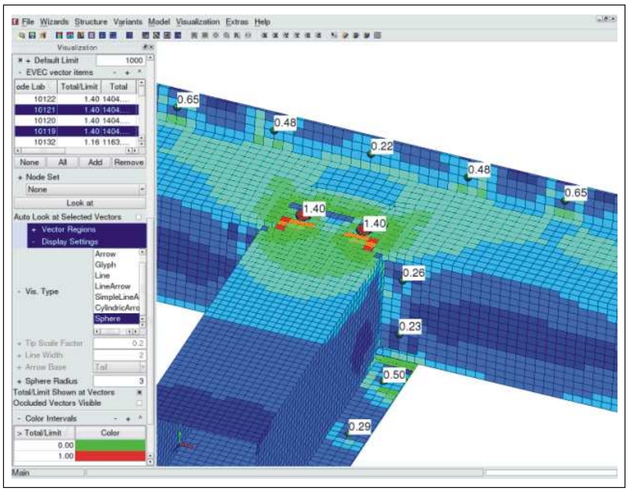

Figure 41: Evaluation of spotweld forces
together with element stresses

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 is available.
- Specific reordering concepts in conjunction with explicit and iterative condensation schemes and automatic selection of algorithms provide **highest solution efficiency**.

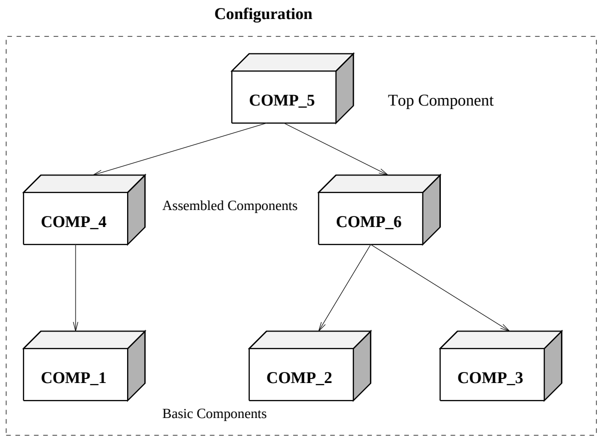

Figure 42: Substructuring in PERMAS

- 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 49).
- 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.

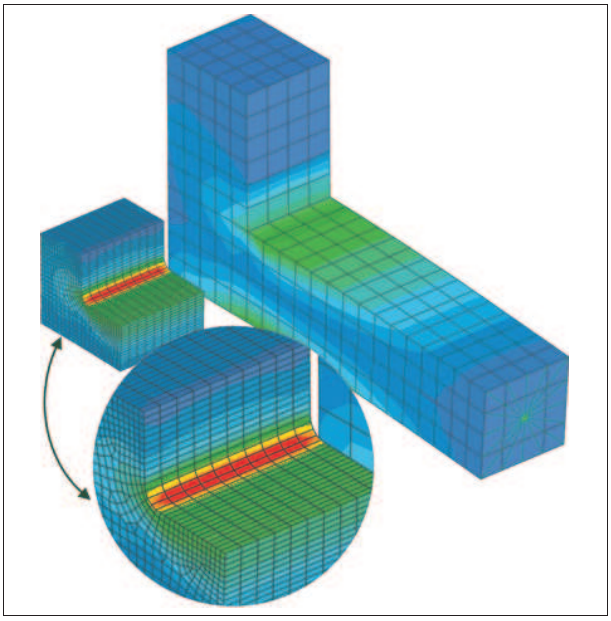

Figure 43: Submodeling
Example for coarse model with a subsequent fine partial model of the notch

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. 43).

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

The same holds for the use of temperature fields in a heat transfer analysis.

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.

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.

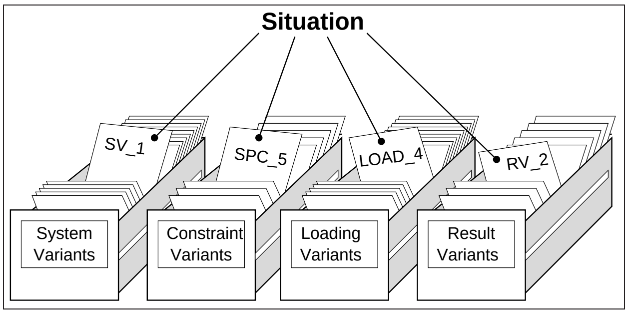

Figure 44: Variants in PERMAS

- 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 calculations are avoided when using a Variant repeatedly.

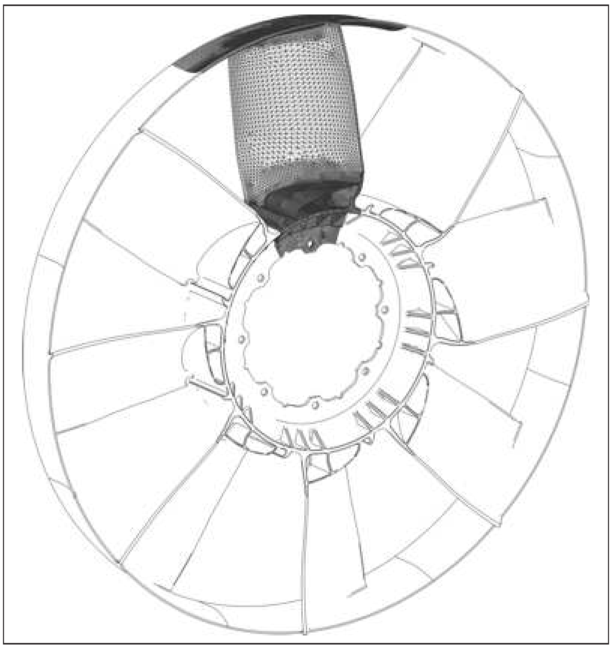

Figure 45: Cyclic symmetry of a fan
Behr GmbH & Co., Stuttgart, Germany.

Cyclic Symmetry

Cyclic symmetric structures like Fig. 45 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 53 and 59). 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 69) the result can be forced to become cyclic symmetric.

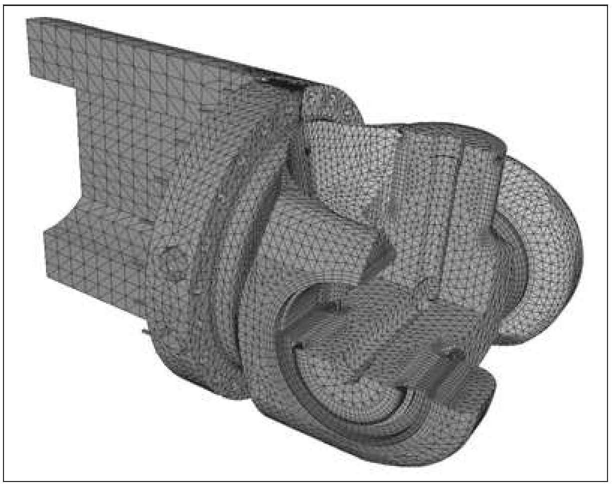

Figure 46: Incompatible meshes
Cardan shaft model in sectional view
(Voith Turbo GmbH & Co. KG)

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 45),
- by specifying geometry elements (see element library page 42).

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.

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 45),
- by specifying geometry elements (see element library page 42).

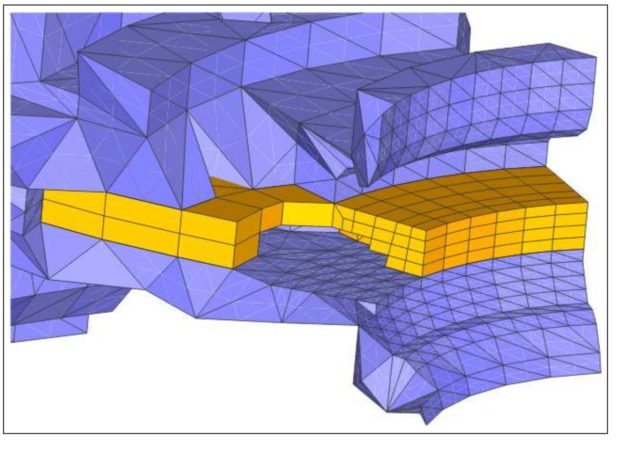

Figure 47: Incompatible meshes
Element transition (HEXE8/TET10)

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.

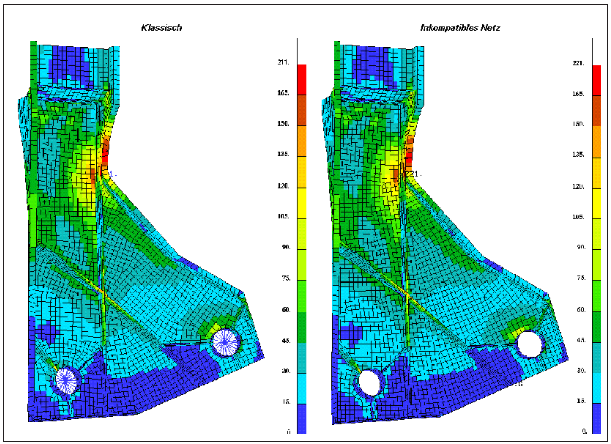

Figure 48: Ribbed shell model
with compatible meshes (left) and
incompatible meshes (right)

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

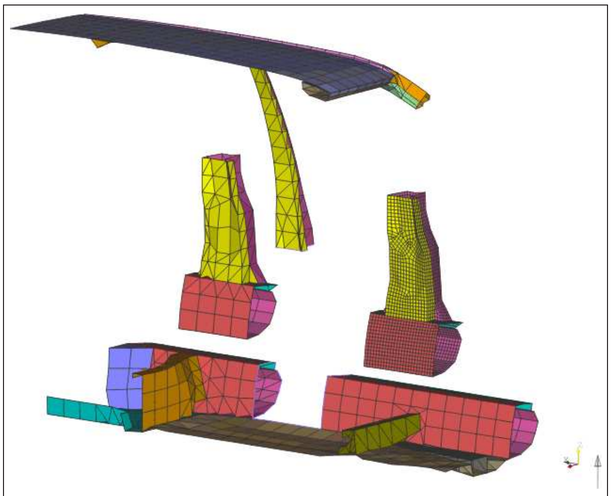

Figure 49: Application of local mesh refinement

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.

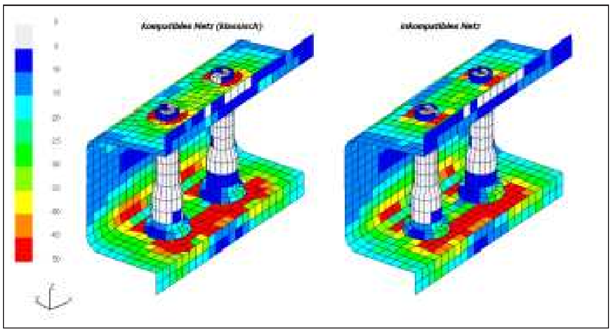

Figure 50: Results for part assembly
with compatible (left) and incompatible (right) mesh

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

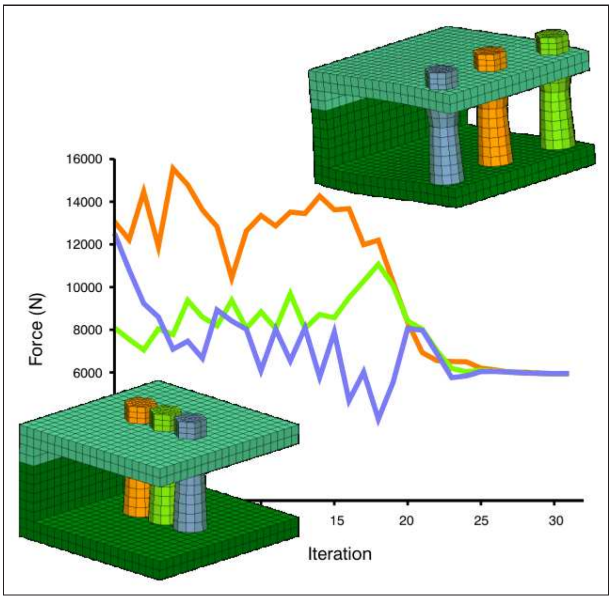

Figure 51: Shape optimization
with incompatible meshes to achieve smallest relative displacements and identical bolt forces (top: initial design, bottom: final design; in between: bolt forces for each iteration)

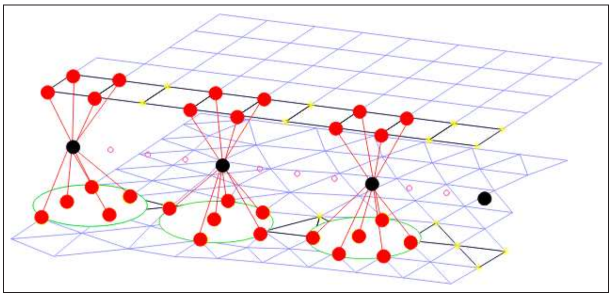

Figure 52: Connection of spotweld forces

Automated Spotweld Modeling

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

- Specification of spotweld positions
- Selection of (incompatible) faces
- Specification of spotweld stiffness and (optionally) the spotweld diameter

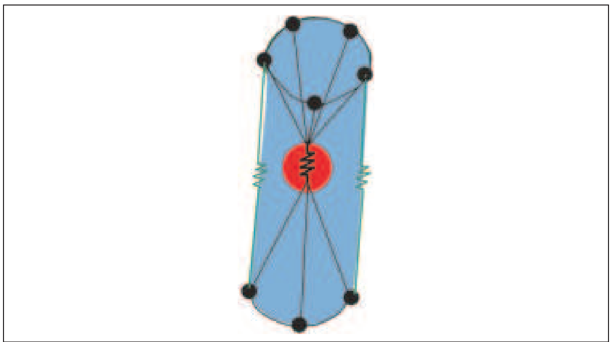

Figure 53: Spring element at spotweld position

The spotweld stiffness is modeled by a spring element, which is coupled to the neighboured parts by automatically generated *MPC* conditions (see Fig. 53). Available results are the spring forces and the reaction forces at the coupling nodes of the joint parts.

For verification purposes the topology and the connecting vectors of the generated spotweld connections can be issued for graphical post-processing (see Fig. 55).

A more sophisticated spotweld model has been developed with module WLDS (see page 54). Using very simple modeling it achieves best possible results.

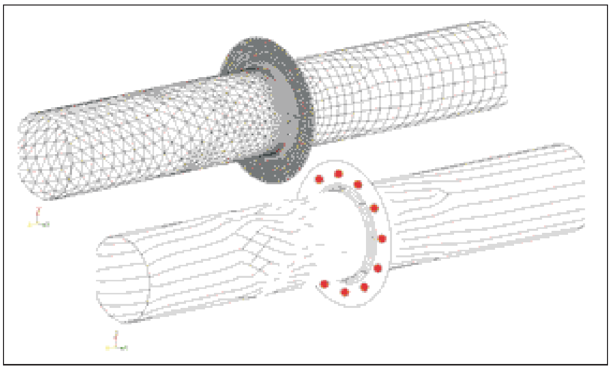

Figure 54: Spotwelds and incompatible meshes

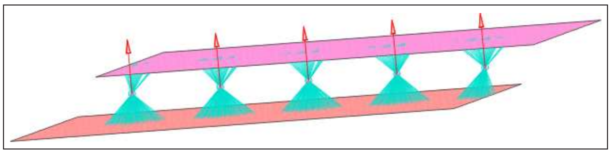

Figure 55: Verification of spotweld connections

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 (*MPC*s).

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, 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).

The interpolation regions may be used also for a **Volume-Shell Transition**, i.e. the connection of plates and shells with solid element structures (see Fig. 56). On the basis of a guiding and a dependent surface, the corresponding constraints for the dependent nodes are generated automatically (see also the Surface Description on page 39 and the Automated Coupling of Parts on page 39).

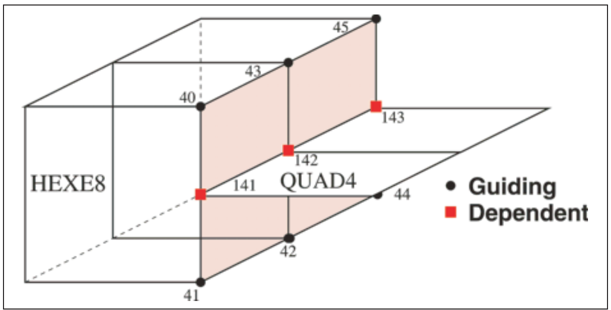

Figure 56: MPC example: Volume Shell Transition

- **General MPCs** allow any linear combination of the involved degrees of freedom.
  General MPC conditions can be used to model **Press Fits**, e.g. shaft-hub connections where the shaft has a slightly larger outer diameter than the inner diamter 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. 57).

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 *substructure technique* is defined by 'external degrees of freedom'. On the higher component's level these external degrees of freedom can be part of single or multipoint constraints.

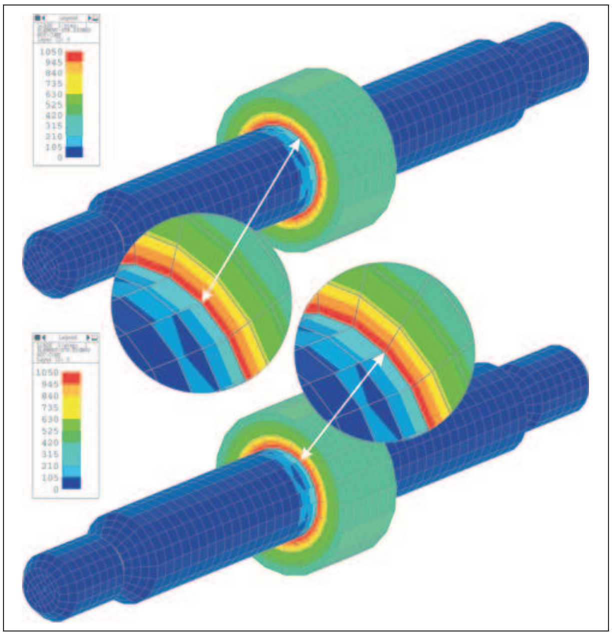

Figure 57: Press fit between shaft and hub
Identical stress results of MPC (above)
and contact solution (below).

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.

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 form 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:

- **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 and pentahedra with 6, 15, or 18 nodes as *gasket elements*.
- **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.

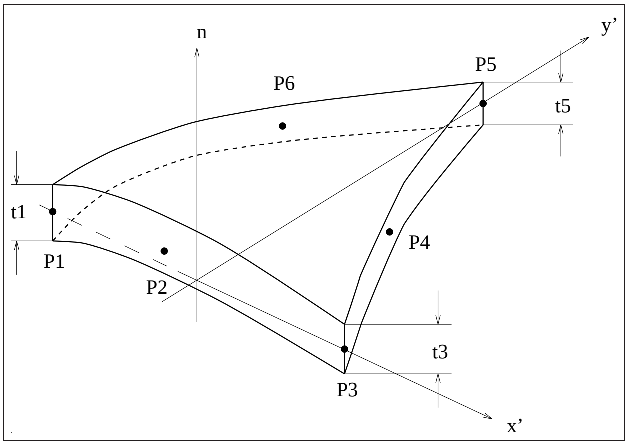

Figure 58: Triangular element with curved edges

- **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.
- **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) Elements** for the modeling of acoustic radiation effects.
- **Axisymmetric Elements** for structures, heat transfer, acoustics, and electromagnetics.

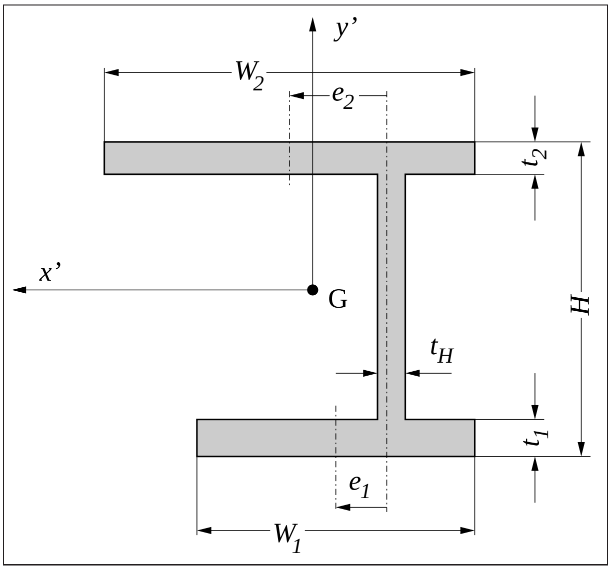

Figure 59: General thin-walled open section

Standard Beam Cross Sections

The following standard geometries are supported for thin-walled beam cross sections:

- General thin-walled open section (see Fig. 59),
- Circular cross section,
- 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 67).

Design Elements for Optimization

For design optimization purpose (see page 67), 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.

Design elements are used according to following rules:

- Each design element contains one or more finite elements and their nodes.
- Each design element may have any number of design variables, at least one. All possible design modifications within the design element are defined dependent on those design variables.
- Each design element has a fixed number of design nodes, a defined geometry and corresponding interpolation functions. Design nodes are needed for shape optimization.
- For optimal sizing problems property dependencies may be defined for the whole design element. They are valid for all corresponding finite elements, independent on its location within the design element.
- For shape optimization the coordinates of the design nodes are made dependent on the design variables, and the coordinates of all nodal points associated to the corresponding design element are modified according to the design element interpolation functions.
- The design elements correspond with the finite elements (like solids, shells, etc.) and their topology.

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 pattern optimization,
- design element for free shape optimization.

Error estimator

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.

With the *refinement indicator* pre-processors like MEDINA may be able to perform an adaptive mesh refinement (see Fig. 60).

But this indicator can also be used to identify those mesh regions where stresses are possibly not very reliable.

In addition, the error estimation can also be used to improve the stresses by a smoothing process without a new mesh or a repeated analysis run.

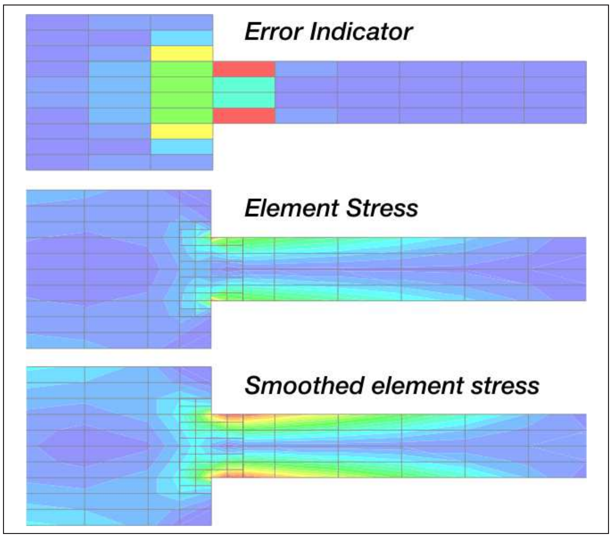

Figure 60: Example for *error indicator*, *element stress*, and smoothed element stress

Material Properties

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

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

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

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

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

The definition of damping can be frequency-dependent.

Layered composites (laminates) can be defined by different materials (like fibre-reinforced material) for each layer (see page 74).

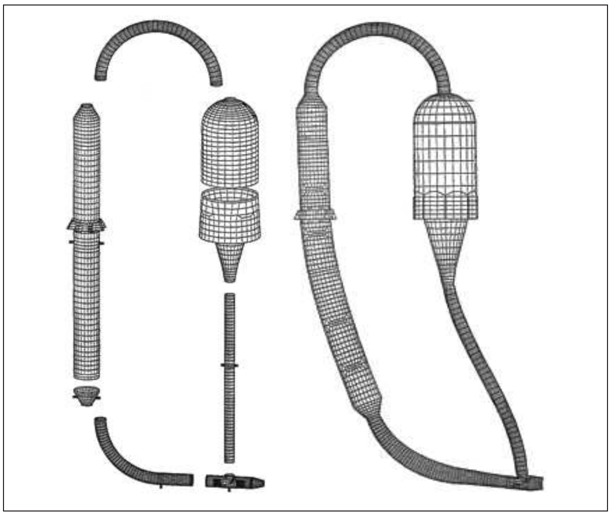

Figure 61: Mode shape of a reactor

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 acompanying 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.

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.

For distributed surface loads, e.g. not specified at the nodes of a mesh but a coarser grid, special load elements (see page 43) can be used which are coupled to the main structure by interpolation (see page 41).

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 like hydrostatic pressure.

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. 62). 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.

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.

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 verifiy

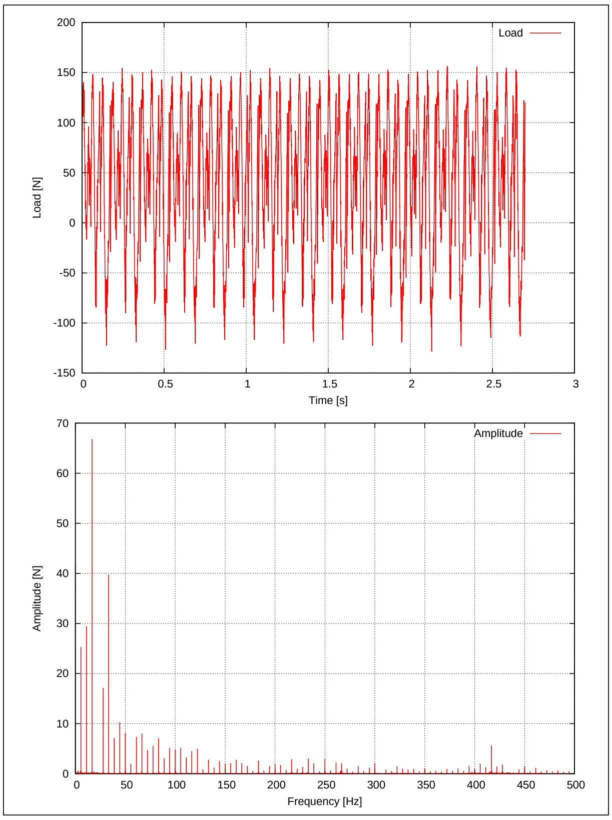

Figure 62: Example for the transformation of a periodic load (above in time domain, below in frequency domain)

support conditions.

- For MPC interpolation regions (see MPC conditions on page 41) 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. 55).
- 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. 63).
- 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 28 and 29. Besides, the assignment of finite elements to design elements can also be checked.

Interfaces

The integration of PERMAS in the pre- and postprocessor 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 postprocessors but also to model files of other FEA systems:

- MEDINA (page 76)
- PATRAN (page 76)
- I-DEAS (page 76)
- ADAMS (page 77)
- DADS (page 77)
- SIMPACK (page 77)
- EXCITE (page 77)
- MOTIONSOLVE (page 77)
- HYPERVIEW (page 78)
- VAO (page 78)
- Virtual.Lab (page 78)
- MATLAB (page 78)
- NASTRAN (page 78)

Moreover, a growing number of extra interfaces to PERMAS are available from partner companies or INTES (see page 79).

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

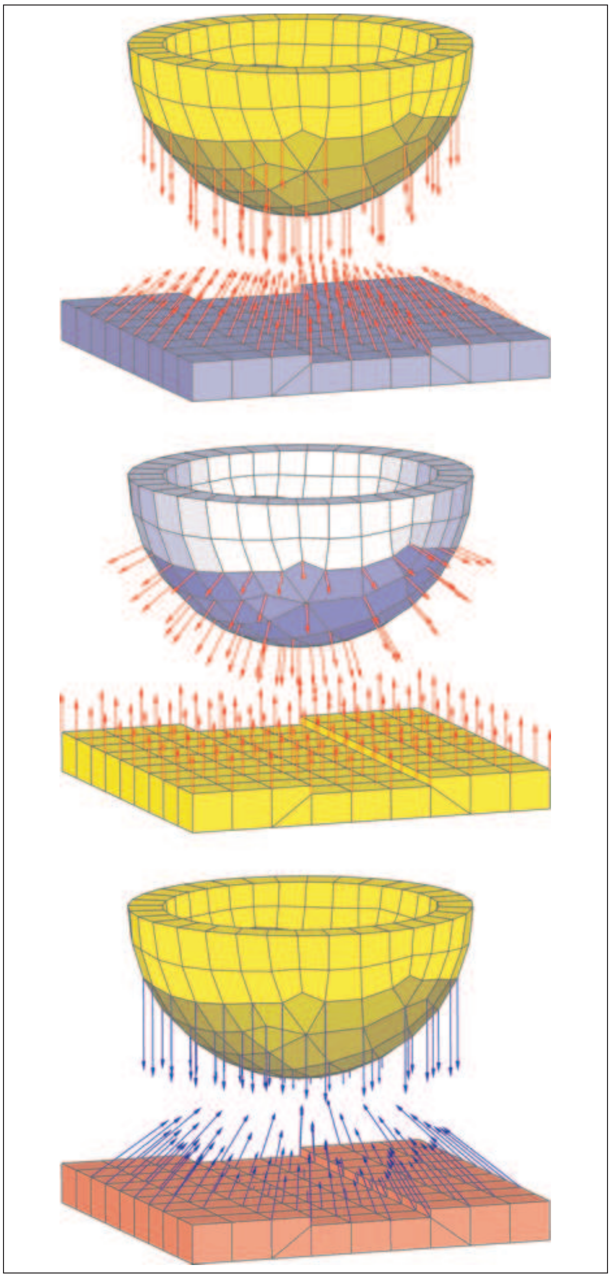

Figure 63: Verification of a contact model
Top: contact normal vector
Middle: surface normal vector
Bottom: nadir vector

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.

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.

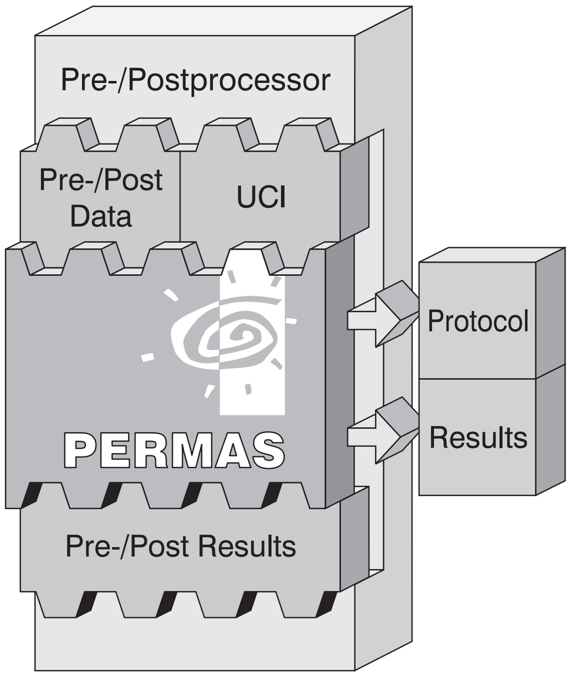

Figure 64: Integration of PERMAS
in pre- and post-processor

- Quality assurance for modeling is improved:

  - 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 of his/her input 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, and damping matrices are provided.

Another application is the generation of modal models (e.g. by *generalized modal condensation* , see page 60). 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 45) 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.

Transformation of Results

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

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

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

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

Comparison of Results

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

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

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

- *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:

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

  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 :

$$COFM = \boldsymbol{X}_1^t \boldsymbol{K} \boldsymbol{X}_2; \qquad COF = COFM^t COFM.$$

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

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.

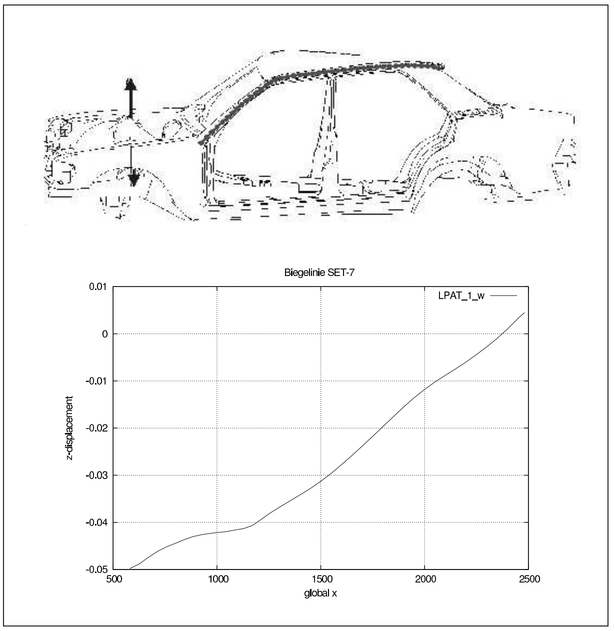

Figure 65: Determination of elastic line for torsional load case (displacement for the marked nodes)

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. 66).

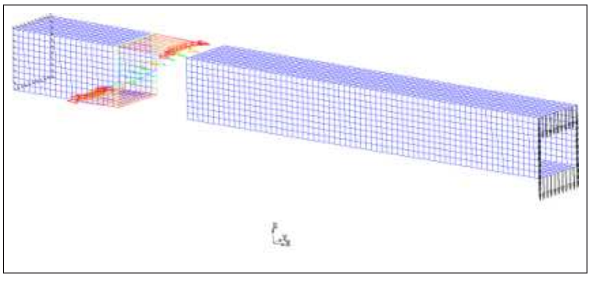

Figure 66: Determination of cutting forces

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 45).

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.

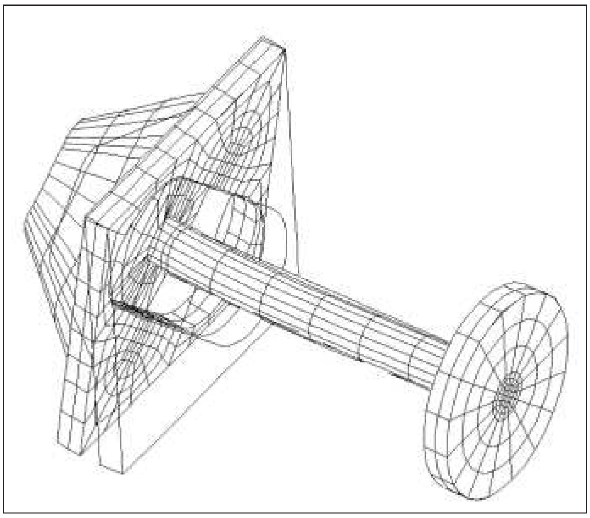

Figure 67: Clutch Element, Static Displacements

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 CISPAR ESPRIT project by a loose coupling approach.

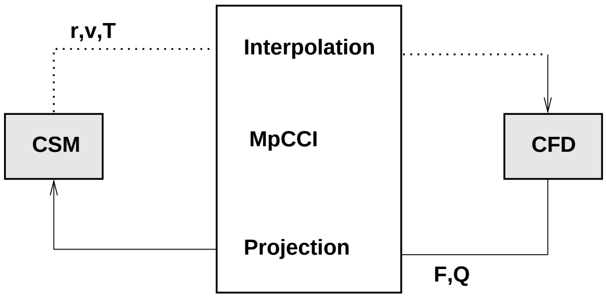

Figure 68: Coupling of CFD and CSM

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

The transferred boundary conditions comprise thermal and mechanical quantities (force **F** or pressure resp., heat flux **Q**, displacement **r**, velocity **v**, temperature **T**). There the coupling library does any interpolation and projection necessary due to incompatible 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 provides 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 software tools like fast vector routines.
The characteristic features of PERMAS-MQA are the concepts and tools for the **quality assurance of the analysis process**.

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.

Beside the software quality (see page 13) the reliability of FEA results depends on the following points:

- **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 42).
- **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 groupwise 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 47).

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.

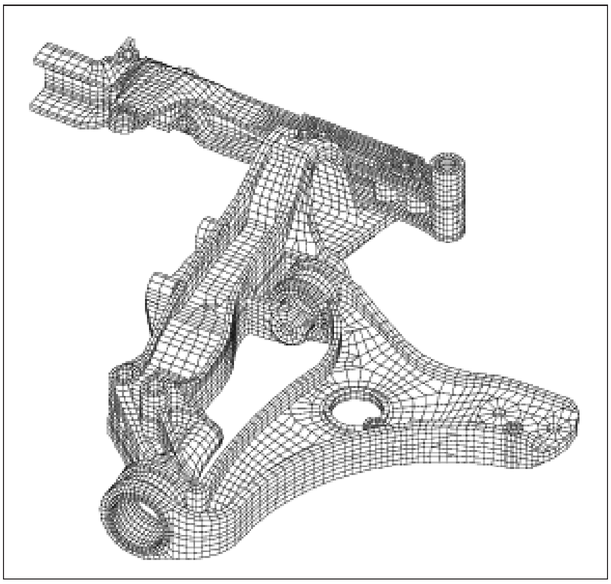

Figure 69: Model of a front axle
Porsche AG, Weissach

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 (*MPC*s) (see page 41).
- 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 unter the applied loads and these inertia forces.
- Different kinds of static loading are available (see page 46).
- The following primary results are calculated:
  - Displacements,
  - Mass and moments of inertia.
- From that additional results are derived:
  - Reaction forces,
  - Stresses and stress resultants,
  - Strains,
  - Residual forces,
  - Strain energy density, i.e. mass- or volume-specific strain energy.

PERMAS-WLDS – Refined Weldspot Model

The modeling of weldspot connections is described on page 41. 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 representation. Among others, this is achieved by an internal calibration of the stiffness using a volume model.

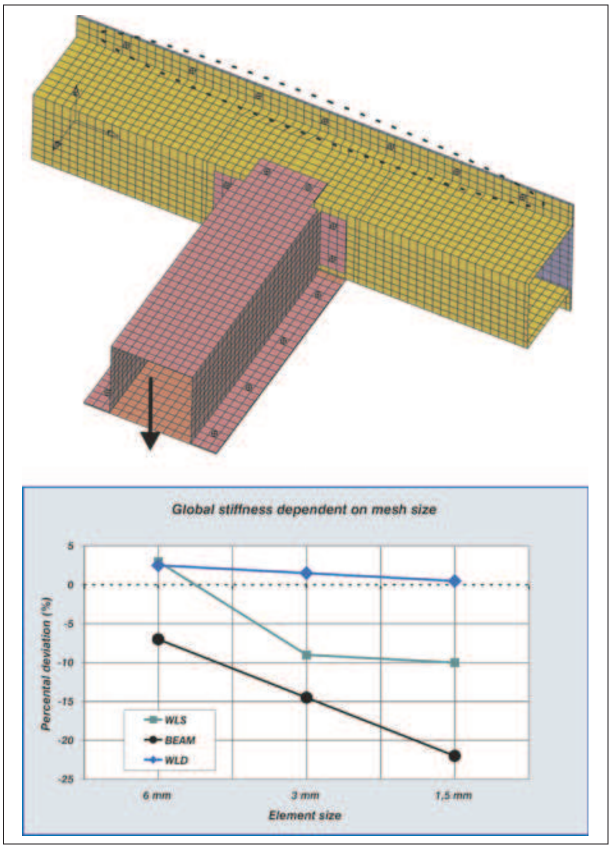

Figure 70: Quality
of the refined weldspot model

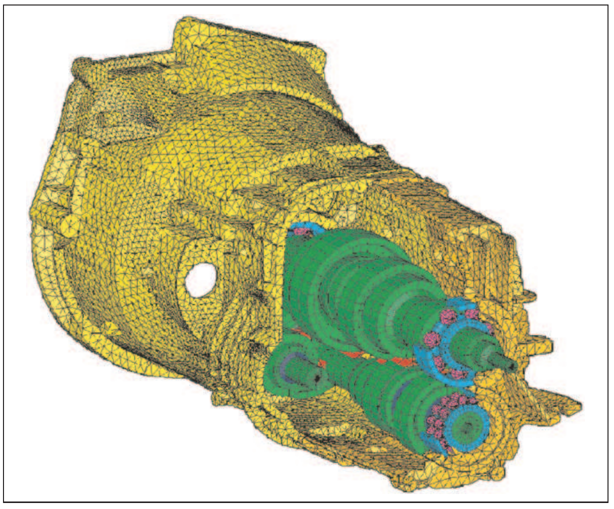

Figure 71: Bearing housing ZF AG

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.

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),
- assigment 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.

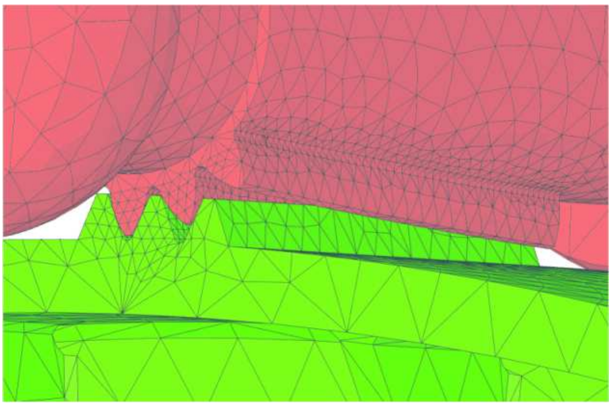

Figure 72: Contact with incompatible meshes

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.

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

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. 73).

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

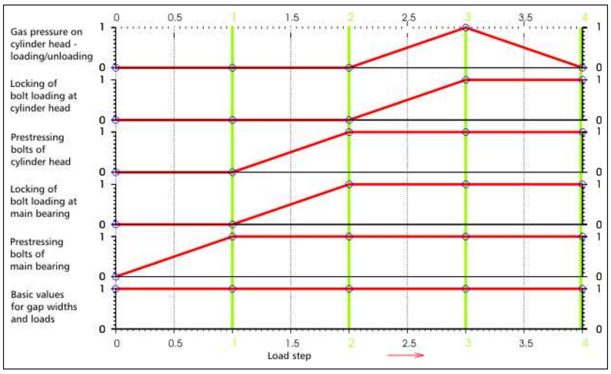

Figure 73: Representation of a load history

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

A generalized concept for **bolt pretension** is provided. Beside the classical approach using a cutting plane with pretension in normal direction, a new approach using cylindrical thread coupling with pretension in axial direction is available. This highly innovative feature offers a convenient definition which can take into account the detailed effects of radial spreading and axial torque caused by the thread's flank and pitch geometry without the need of modeling the flank shape or thread line explicitly (see Fig. 74).

Comprehensive checks allow the verification of contact models like type of contact, its geometry (gapwidth and normal vector, see also Fig 63), and the contact coordinate system (for normal and frictional force directions). In addition, the contact status is available for 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 improve frictional behavior essentially.

The analysis procedure uses a reduced flexibility model which is derived from the set of contact degrees of freedom. This procedure has the following advantages:

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

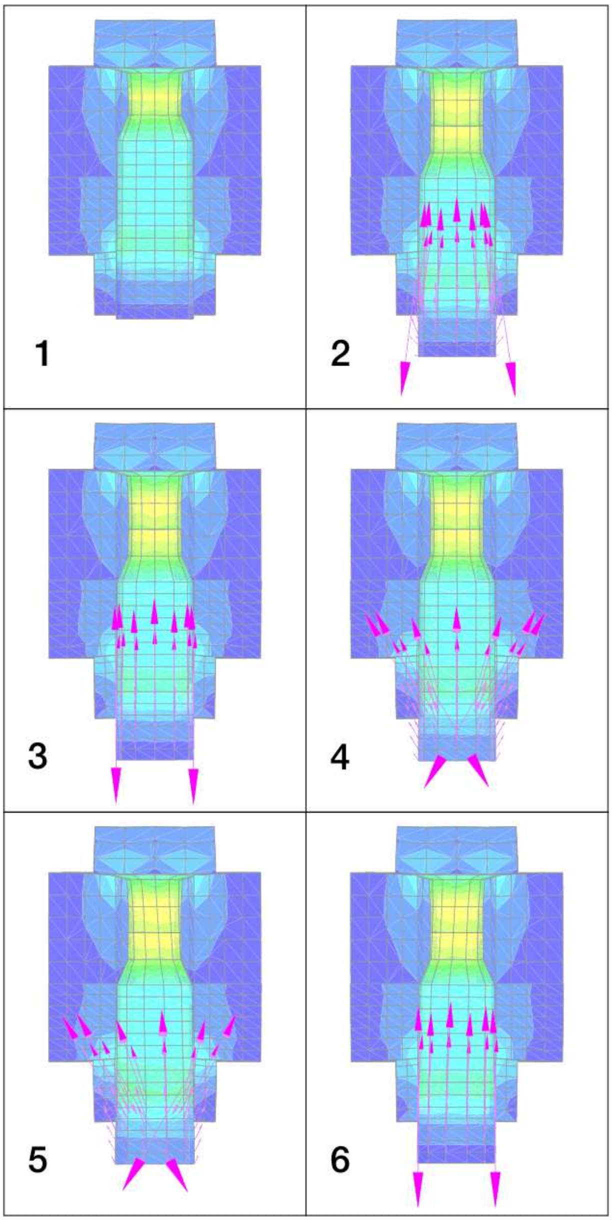

Figure 74: Bolt pretension
Nodal point stresses and pretension forces
for deformed mid-size screws:
(1) with cutting plane in bolt
(2) with radially joined thread
(3) thread without radial coupling
(4) thread with flank but no pitch angle
(5) thread with flank and pitch (standard M10)
(6) left-handed thread with pitch but no flank angle

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.

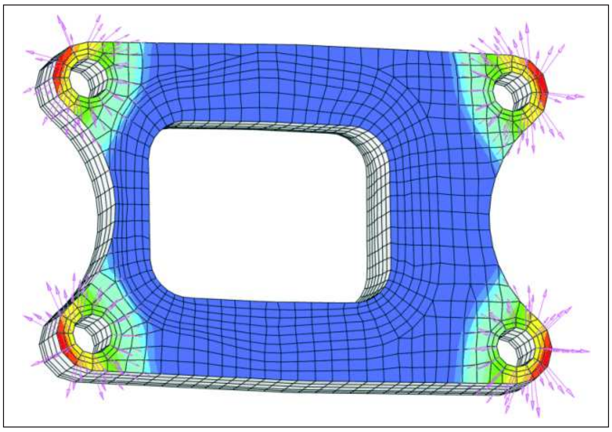

Figure 75: Contact pressure and shear vectors

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. 75), the gap widths, and the relative gap displacements.

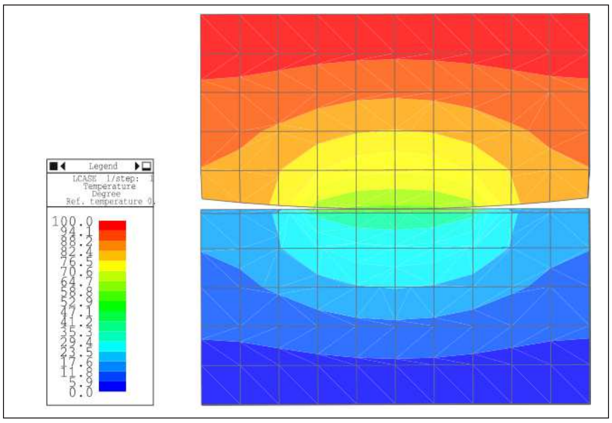

Figure 76: Temperature distribution
in two contacting bodies

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. 76), 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).

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. 77).

Gasket elements can be alternatively handled as integral part of the contact iteration instead of a feature in nonlinear material analysis (see module NLS on page 57). This leads to a remarkable reduction of run time compared to the classical solution method. 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.

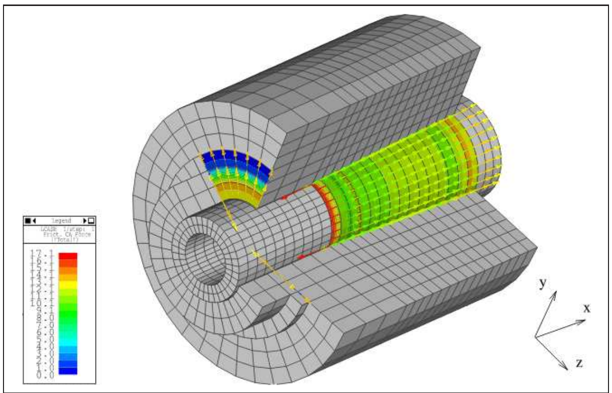

Figure 77: Conical *press fit*
(slipping forces after assembly)

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 20).

PERMAS-NLS – Nonlinear Statics

Geometrically nonlinear behavior

This module part allows for the geometric nonlinear analysis of models. There, large displacements with small strains (i.e. linear elastic material behavior) are assumed. Beside an automatic load step control different nonlinear solvers are available (like *Newton-Raphson*, Modified Newton-Raphson, Secant Newton, Line search, Arc length method). The nonlinear characteristic may be represented by xy-plots, where the load increments can be chosen automatically or manually.

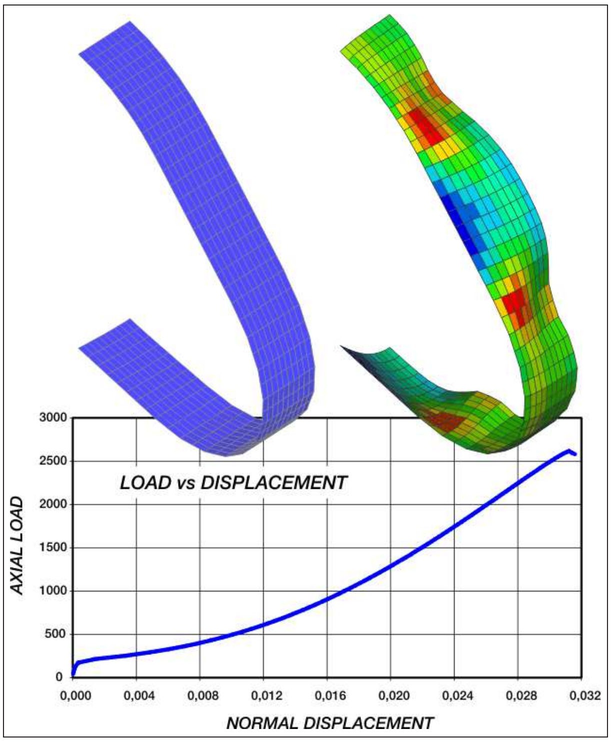

Figure 78: Nonlinear NAFEMS Test

Material nonlinearities

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

- Nonlinear elasticity (of *Cauchy* type)
- Plasticity (*von Mises*, *Tresca*, *Drucker-Prager*, *Mohr-Coulomb*)
- *Visco-plasticity* of power-law type for von Mises yield criterion.
- Creep with
  – nonlinear elasticity or
  – plasticity

The material can be defined temperature-dependent for Young's modulus, yield stress, and the stress-strain curves. A time-dependent characteristic is present for creep calculations in addition. *Hardening* in plasticity can be defined *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 54) can also take nonlinear material behavior into account.

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

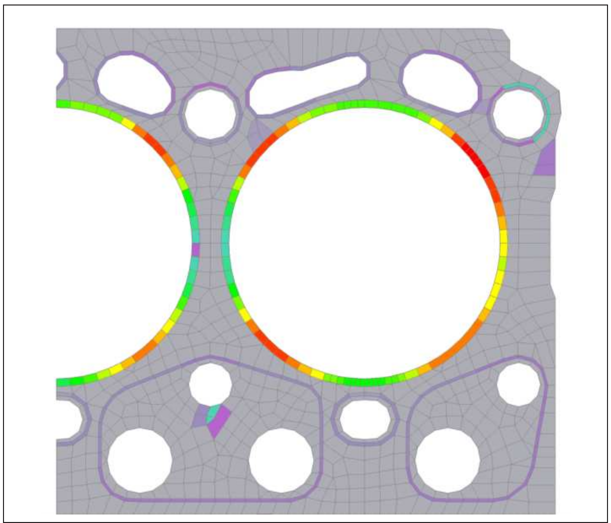

Figure 79: Pressure of cylinder head gasket

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.

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 37), 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.

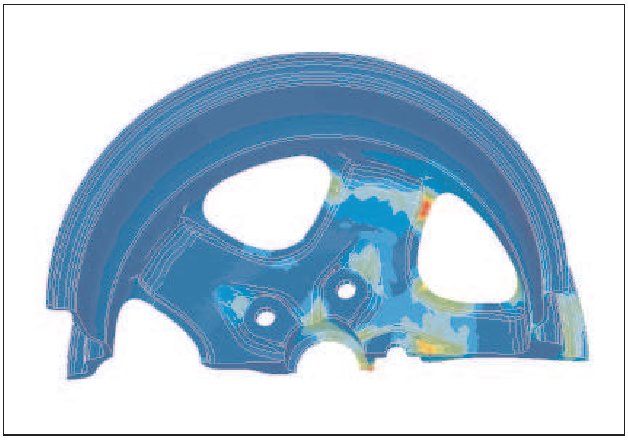

Figure 80: Impact test of a wheel

PERMAS-NLSMAT – Extended Material Laws

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

- A material law for *cast-iron* is available taking into account the different behavior under tension and compression.

- Furthermore, a *nonlinear kinematic hardening* model (following *Armstrong-Frederik*) is available as a model for cyclic loading.

In addition to the PERMAS material laws, a user defined material law can be used. To this end, the user provides special subroutines which do the calculation of stresses and strains together with the tangent matrix associated with the material law.

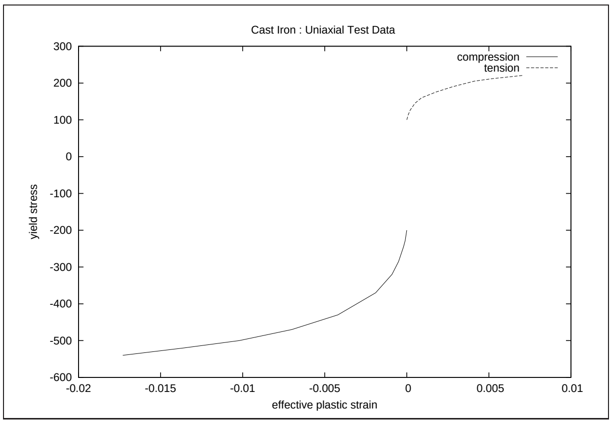

Figure 81: Biaxial test data of a cast iron material

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 for the assessment of the nonlinearity of the *prebuckling behavior*.

Load factors and mode shapes are available for any kind of post-processing.

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

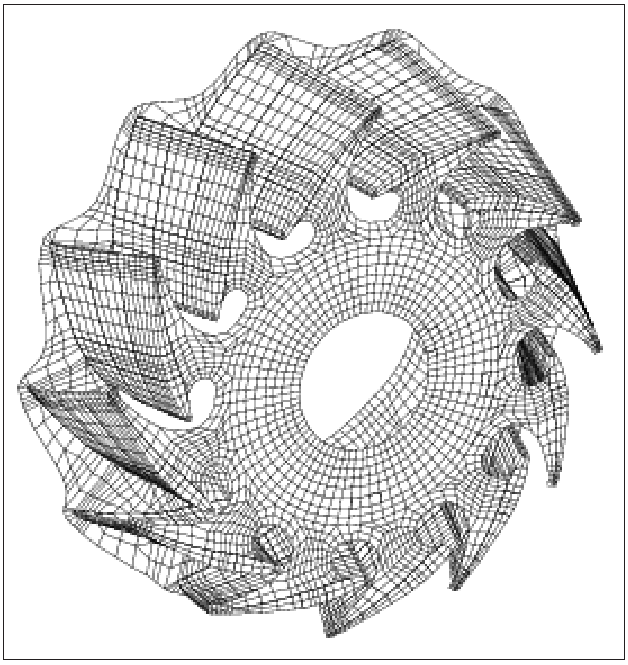

Figure 82: Half Model of a Turbine
Half Model of a Turbine, 8th Mode Shape with Antisymmetric Boundary Conditions

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.

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.

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.
- 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 49).

- As a measure for the completeness of the modal model, *effective masses* are generated and printed on the result file.

For multi-body simulation programs, the mixture between static mode shapes and free vibration modes is a very frequent application, when flexible FE models are integrated. In case of single component models, two sets of vectors are produced: Static mode shapes and dynamic eigenvectors. These mode sets have to be orthogonalized, e.g. in the MBS system. For large models however, this step is very time consuming. Hence, this orthogonalization may be done within PERMAS. A special procedure is available to facilitate this orthogonalization.

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 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

It includes dynamic *condensation* due to the *Craig-Bampton* method. The 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 the substructures. Like for the *Guyan*'s reduction, an explicit and an iterative scheme is available in order to achieve good performance (see also page 37).

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 64):

- **“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.

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 MEDINA) can be used to visualize and animate complex mode shapes

Eigenfrequencies of Rotating Systems

For rotational systems (see also page 21) 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.

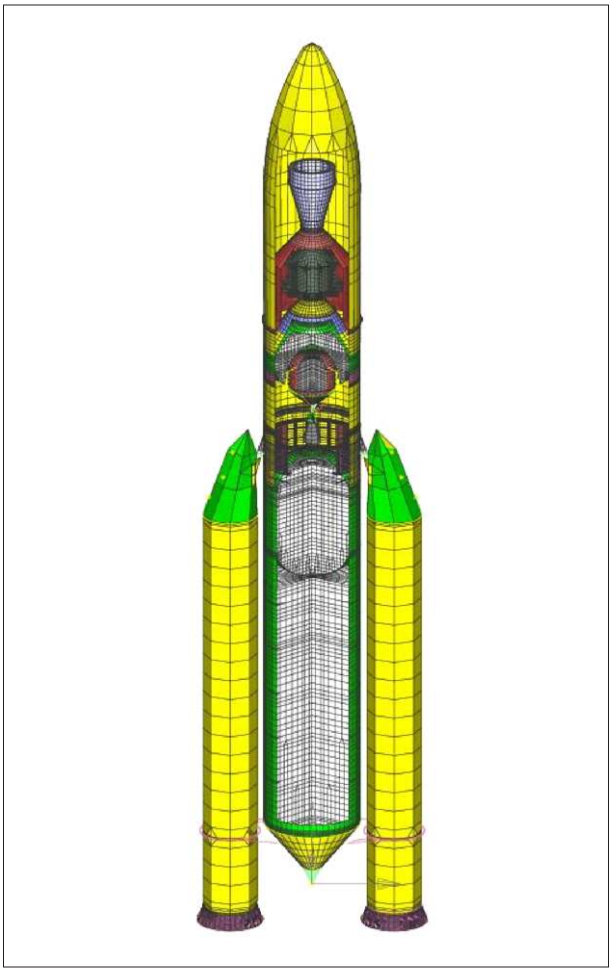

Figure 83: Ariane 5 launcher model
by courtesy of ASTRIUM Space Transportation, Les Mureaux

PERMAS-MLDR – Eigenmodes with MLDR

The calculation of eigenvalues with modules DEV (page 59) and DEVX (page 60) 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 37 and module DEVX on page 60). 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 **M**ulti-**L**evel **D**ynamic **R**eduction (MLDR)

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 frequeny range as in the past together with a possible increase in model size for more accurate results.

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 re-*

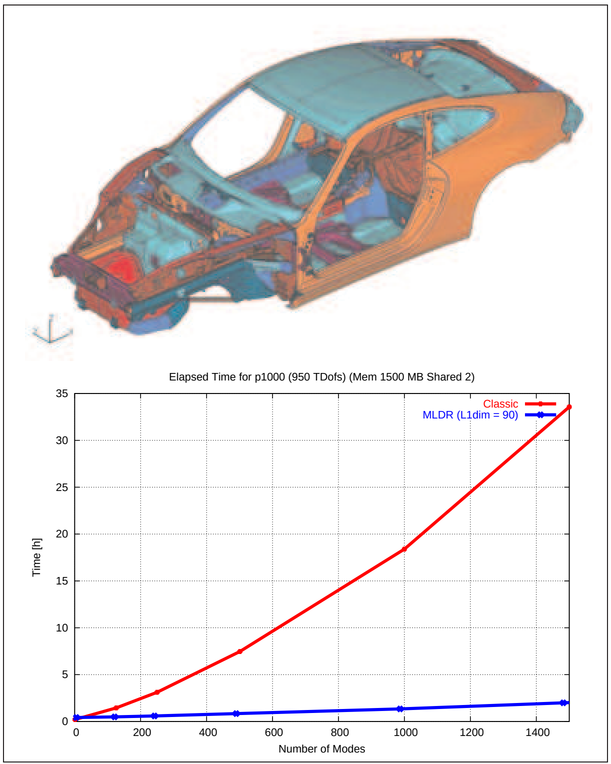

Figure 84: Comparison of elapsed run times for Subspace Iteration (upper curve) and MLDR (lower curve) with increasing number of modes (157412 Nodes, 164301 Elements (QUAD4), 944472 Unknowns)

sponse) 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,
- 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.

- 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 superposed 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.

By specification of a node set (see page 45) the run time and disk space for modal superposition methods can be drastically reduced when the results are determined for the set members only.

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

- The **damping** properties may be modeled by one of the following methods:
  - material or structural damping for elements,
  - global structural damping for components,
  - proportional damping (Rayleigh damping),
  - viscous damper elements,
  - modal viscous damping,
  - modal structural damping,
  - modal structural and proportional damping (also 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.

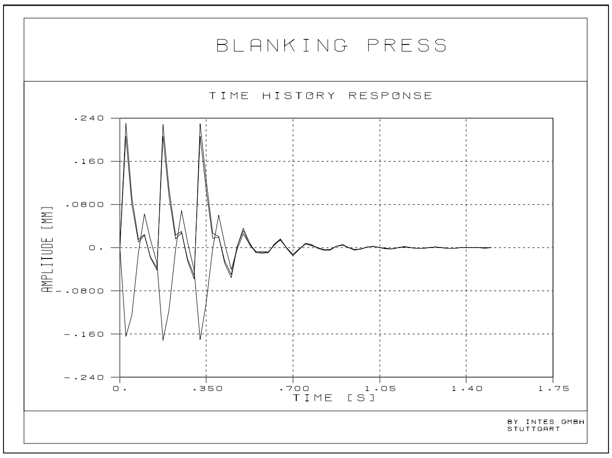

Figure 85: Transient response

- The excitation is defined by static loading cases modulated by functions of time respectively frequency (see page 45). 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 respectively frequency.

- 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 of dynamic modes. 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.

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.

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

- For the evaluation of a structural response *modal participation factors* of the primary results

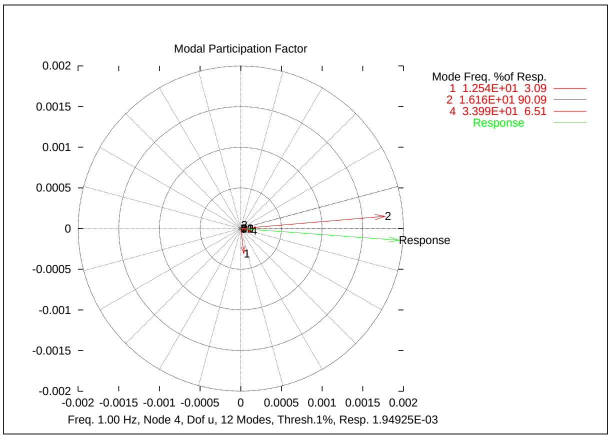

Figure 86: Modal grid participation factors

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

PERMAS-DRX – Extended Dynamics

This module comprises additional methods for structural response analysis:

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

Spectral Response Analysis

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

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

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

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

Random Response Analysis

Frequently, vibrational loads are not predictable like for cars on a bumpy road, for a house under wind loads, or for a ship on rough sea.

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

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

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

This 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.

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.

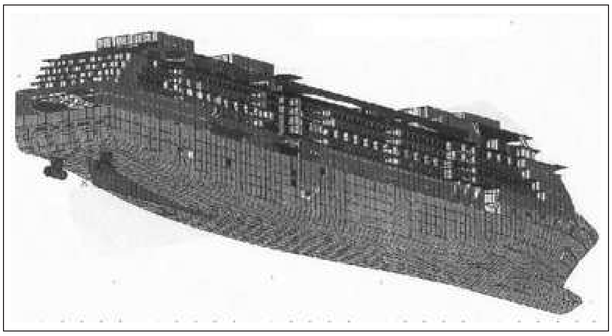

Figure 87: Ship model
by courtesy of Chantiers de l'Atlantique, Saint-Nazaire

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

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

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:

- 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

59), 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 61).

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.

By specification of a node and/or element set (see page 45) 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.

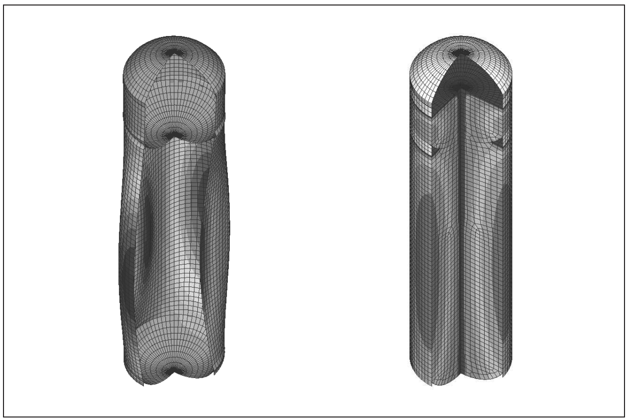

Figure 88: A coupled vibration mode shape of a fluid-filled space tank with corresponding pressure mode

The following damping features are available:

- For the structure:
  - material or structural damping for elements, also frequency dependent, if required.
- 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 45). 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 respectively frequency. 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-HT – Heat Transfer

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

- The temperature field may be steady-state or transient.
- Nonlinear material data for conductivity and heat capacity may be specified by tabular input.
- Temperature and space-dependent convectivity coefficients may be specified in a very general way by functions (see page 45).

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 51).

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.

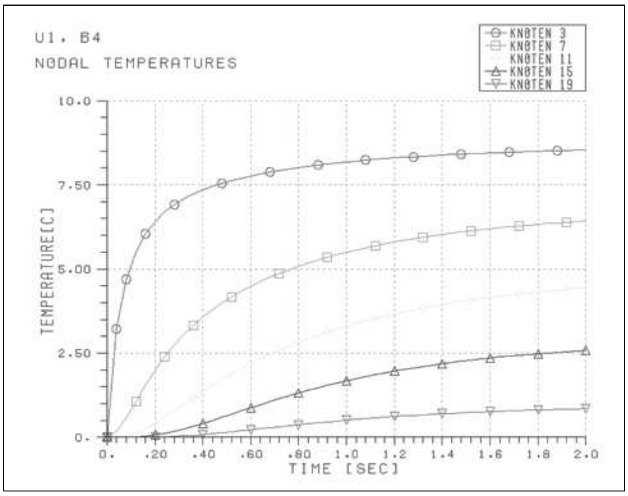

Figure 89: Transient Temperatures at nodal points

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 45).
- 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.

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).

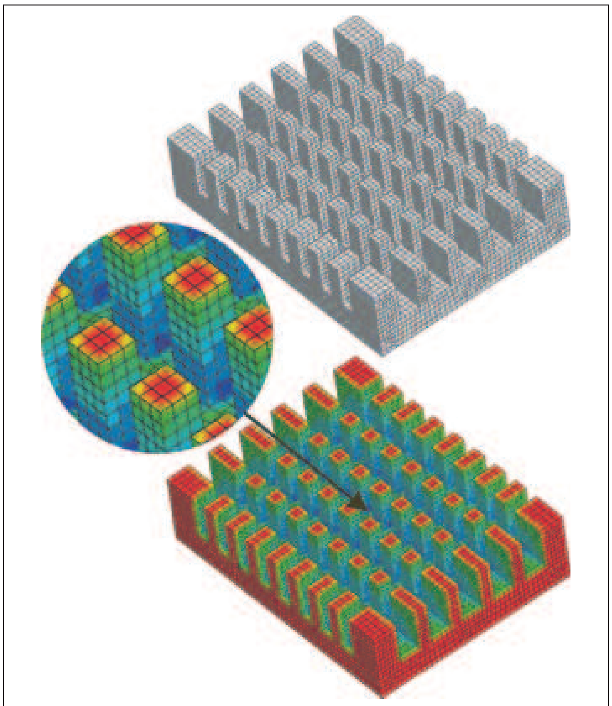

Figure 90: Analysis of cooling element with radiation

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.

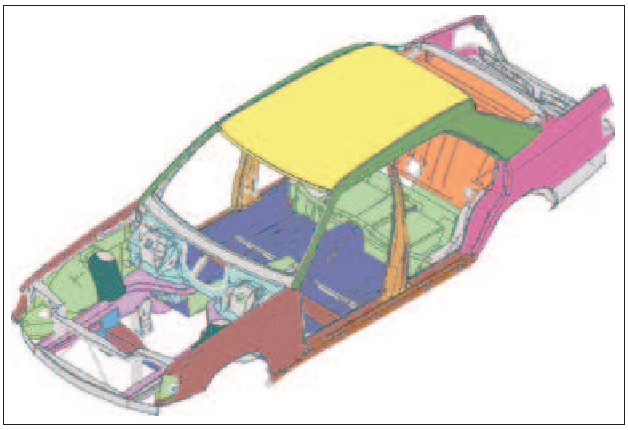

Figure 91: Frequency response optimization of a body-in-white with shape and sizing parameters (see also Fig. 93)

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 44),

- 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 44),
  - *bead design*.
- **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,
- general constraints as combination or arbitrary function of the above mentioned quantities. Such functions include global criteria like max/min, absmax/absmin, or RMS.

The objective function of an optimization may be the weight or any other specified constraint.

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. 92).

The following solvers are available for optimization:

- Linear statics,
- Contact analysis,
- *Inertia relief* (see page 54),
- Nonlinear material behavior,
- Eigenvalue analysis,
- Modal frequency response analysis,
- Steady-state heat transfer analysis.

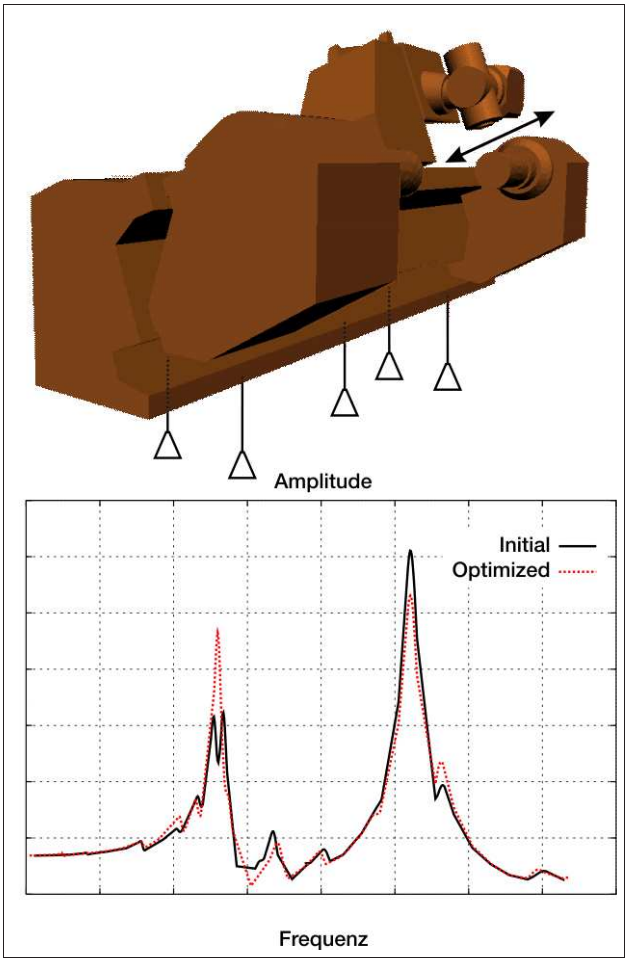

Figure 92: Frequency response optimization
for a machine foundation where the position of the base spring elements is optimized which are coupled to the machine structure by interpolating surfaces. Design optjective is the maximum flexibility of the tool center point.

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.

Different solvers can be combined in one optimization task as well as sizing and shape parameters.

The optimization allows taking into account several loading cases as well as different boundary conditions using the variant analysis (see page 38.) 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, **substruc-**

turing can be used to reduce run time by separating the design space in the top component. So, the reduction of the unmodified parts has to be done only once.

The results of an optimization are the history of the objective function and an overview on the validity of the design after each iteration. In addition, the values of the design variables and the constraints are available as a function of the iterations performed. These functions may easily be viewed as xy-plots. The export of sensitivities is also possible.

Moreover, element properties may be prepared for result processing (i.e. thickness distribution) and exported for post-processing.

The results of a shape optimization can be exported as displacements for post-processing with the original model or as new model with identical topology and modified coordinates.

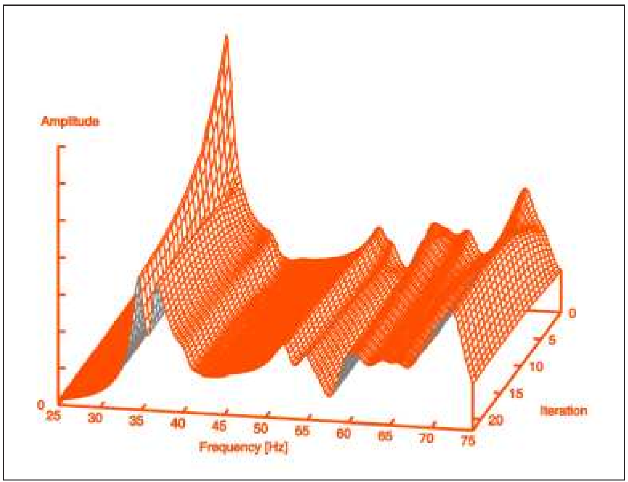

Figure 93: Iteration history
of a frequency response optimization

Optimization for a robust design is achieved by additional reliability constraints. Then, the design fulfills all of the above mentioned constraints and it is also reliable regarding uncertain model parameters (see page 23 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 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.

This module supports the concept development stage by providing fully integrated topology optimization strategies:

- Design space specification with variable/fixed parts,
- Provide boundary conditions,
- Provide loads,
- Target definition with remaining volume,
- Additional constraints for the optimization, if any.

For the modeling continua elements like membranes, shells, and solids are used and *substructuring* is supported. Additional modeling parameters are:

- **Fixed/free design elements:**
  - filling ratio per design element,
  - design variable limits,
  - design variable modification limits.
- **Manufacturing constraints:**
  - *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. 95). 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 direc-

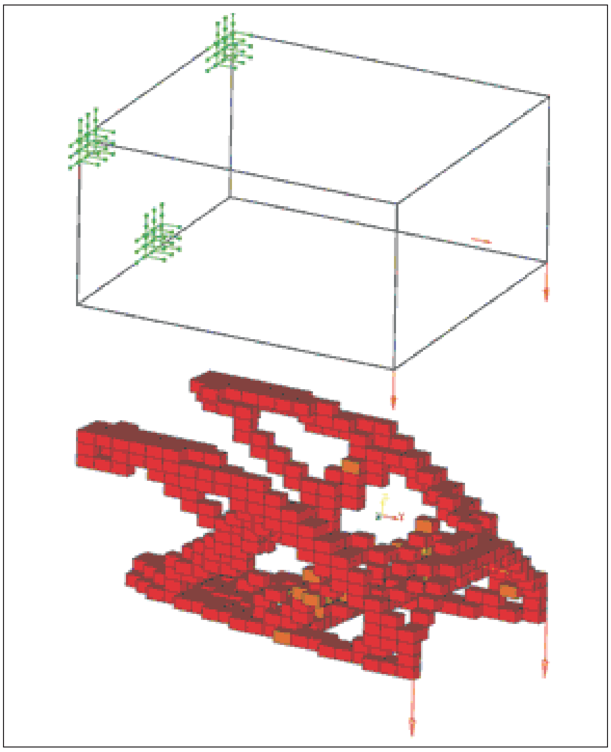

Figure 94: Layout optimization
Design space with boundary conditions and loading (above) and the optimal material distribution (below)

tions.

- 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).

- **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, absmax/absmin, RMS).
- **Multi-Modeling**
  - several load cases simultaneously with different superposition options,
  - different design variants.

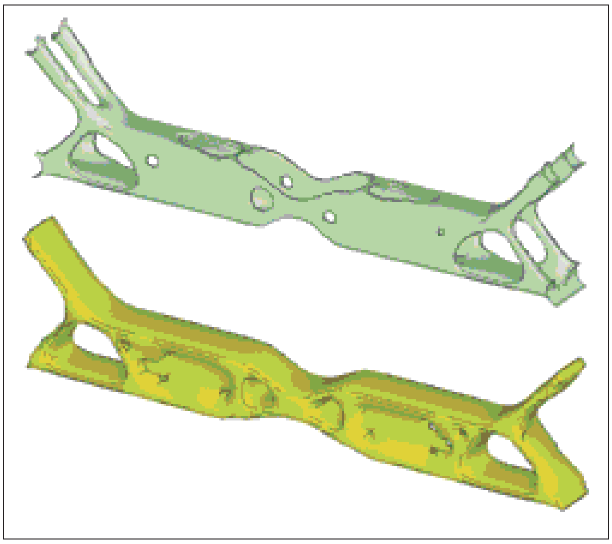

Figure 95: Different results for topology
without (above) and with (below) release directions

A number of analysis options are available for the optimization like

- Linear statics,
- Contact analysis,
- Dynamic mode analysis,
- Modal frequency response analysis.

Because eigenfrequencies and mode shapes change a lot during a topology optimization, strategies for the suppression of local modes are available and of particular importance.

The optimization itself is performed using one of the following algorithms:

- GCA (Global Convex Approximation)
  for eigenfrequencies and a combination of static

and dynamic constraints.

- PD (Primal-Dual Solution)

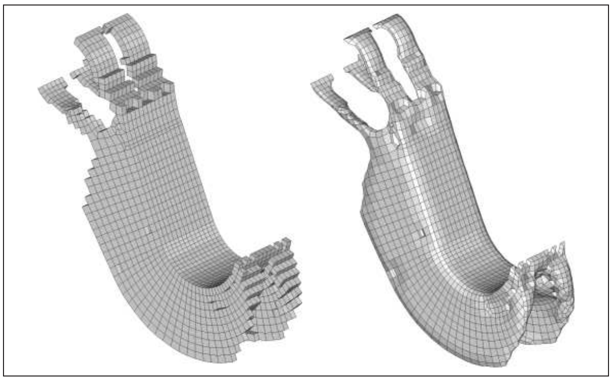

Figure 96: Layout-Optimization of a crane hook
(on the left side the primary result,
on the right side the smoothed surface)

The iterations of an optimization can be controlled either by the convergence of the objective function or by a maximum number of iterations.

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.

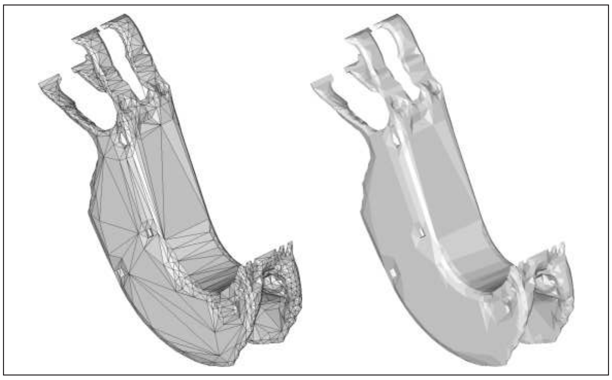

Figure 97: Polygon reduction
(on the left side the triangle mesh,
on the right side graphical representation of the surface)

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 constraints.
- **Polygon Reduction:**
  The mesh is purged of too small triangles and quadrangles in order to achieve a surface description as compact as possible.
- **Export:**
  The remaining triangle mesh can be exported as FE mesh for post-processing or as geometry using *STL* format.

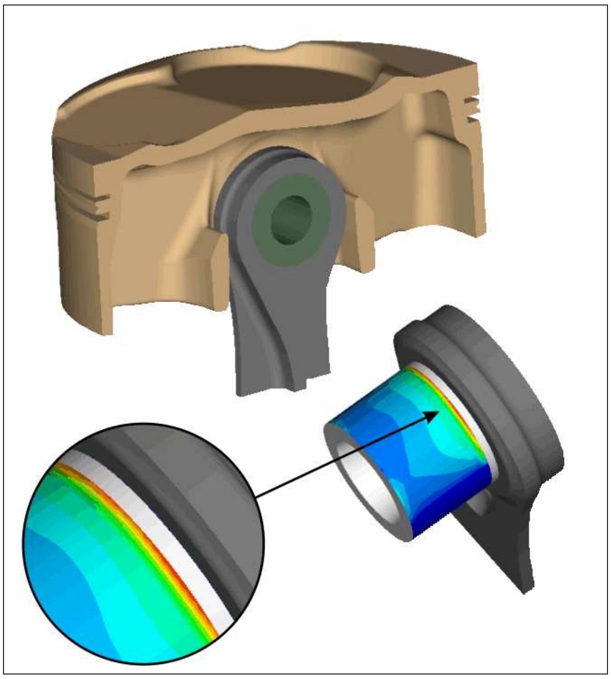

Figure 98: Optimization of contact pressure
between piston and piston pin (Mahle GmbH, Stuttgart).
The figure shows the edge pressure between both parts
(for the optimized result see also next Fig. 99).

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 new *Trust Region* 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.

- New methods for derivative-free optimization and global optimization are available.

Using these new methods, new application fields for optimization are opened, like contact analysis and nonlinear material analysis.

The new 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 structrual mechanics. Module OPT uses a method which belongs to SCP class of optimization methods.

When derivatives are not available, e.g in contact problems or nonlinear material behavior, or when the accuracy of computed derivatives is not sufficiently high (like sometimes in frequency response analysis), then derivative-free methods can be applied. The new derivative-free (local) methods comprise the following approaches:

- Derivative-based methods using finite differences (with SQP, SLP, SCP). Functions should be smooth enough and the choice of the finite difference parameter for the intervall 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. 100 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 37).

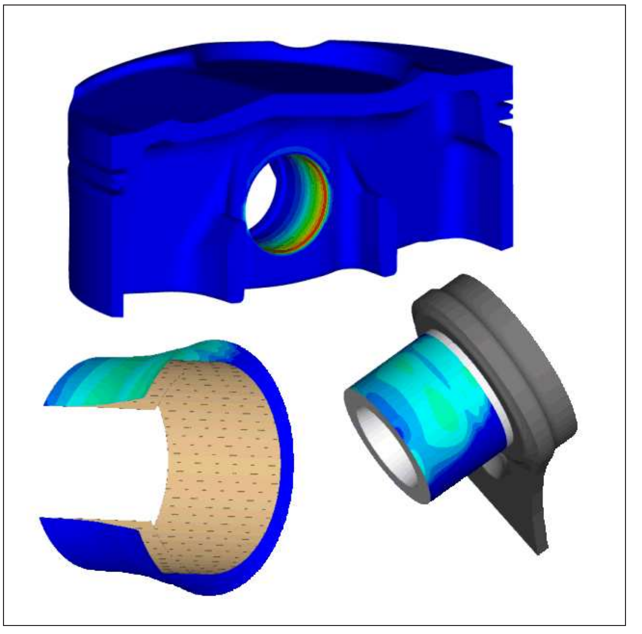

Figure 99: Optimization of contact pressure between piston and piston pin (Mahle GmbH, Stuttgart). The figure shows the optimized geometry of the piston and the reduced edge pressure.

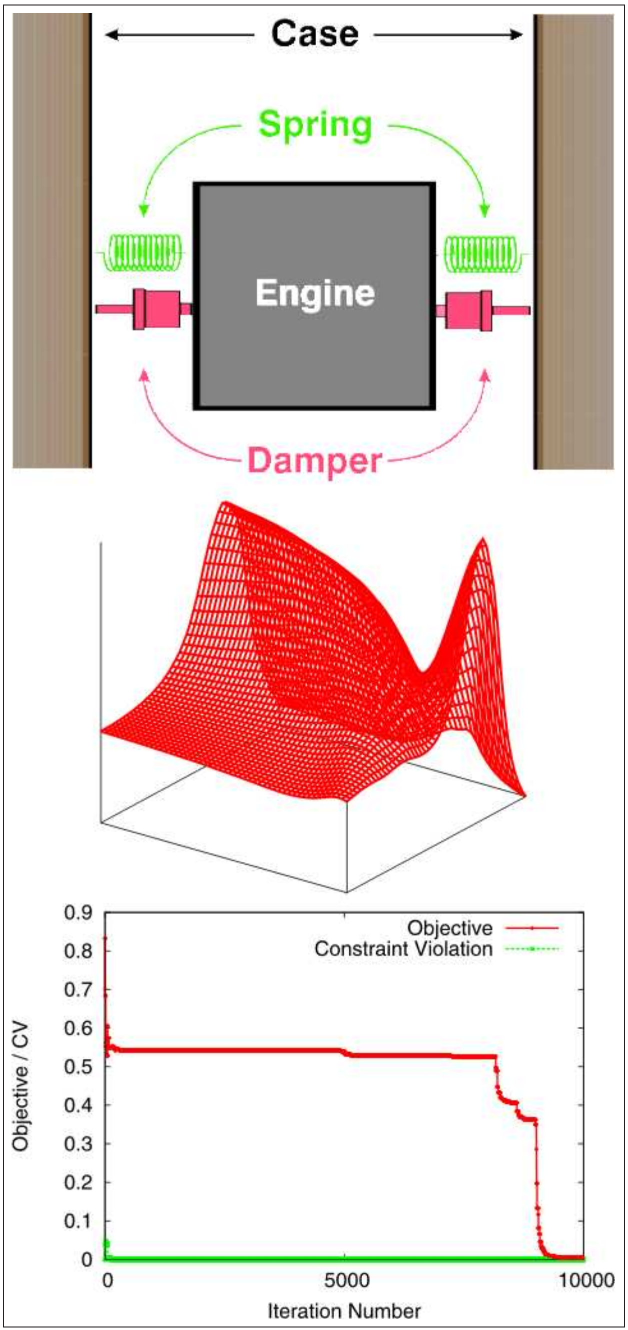

Figure 100: Global optimization of a spring-damper system with the LDR method. Top: model with 4 design variables, Middle: 2 design variables fixed at the optimum, objective function dependent on the 2 free variables, Bottom: course of objective function and constraint violation during the iterations.

PERMAS-RA – Reliability Analysis

In the classical approach to structural analysis a deterministic model is used to predict the behavior of the design under various loading conditions. The results of such calculations are compared to typical limiting constraints such as a maximal stress or deflection under the consideration of safety margins. This is called deterministic approach to the problem of structural safety and the Finite Element Method has become a widespread tool in such procedures.

In contrast to this method, the *stochastic analysis* of a design assumes some properties of a structure or the loads to be uncertain knowing only the characteristics of their probability distributions. The limiting constraints on the design will usually be of the same kind as in the deterministic approach. However, the results from the probabilistic analysis will yield the probability of failure with respect to these constraints and the sensitivity of this probability with respect to the uncertain properties of the model.

This module combines the Finite Element Analysis with the well established COMREL program developed by RCP GmbH, Munich. So, the experience comprised in both software systems could be merged in a single application simplifying the approach to the Stochastic Finite Element Method.

The procedure in reliability analysis comprises the following three steps:

- Definition of uncertain quantities in structural analysis (like geometrical or load parameters) by *basic variable*s 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

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 sensitivites of the limit state functions
  - Result sensitivities for basic variables (elasticities)
  - Selected data of each iteration for Monte Carlo simulations

---

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.

Out of the pre-processing with MEDINA or I-DEAS the geometry, the boundary conditions and loads as well as the material set-up of the laminate is used. From the number and sequence of layers and their thickness, fibre orientation, and material properties the usual ABD matrices are determined.

The analysis results in element forces, from which the layer stresses and strains are derived. Using the MEDINA post-processor, failure criteria, failure indices, and safety factors may be evaluated. As usual, all other post-processing features may be used for composite structures in addition.

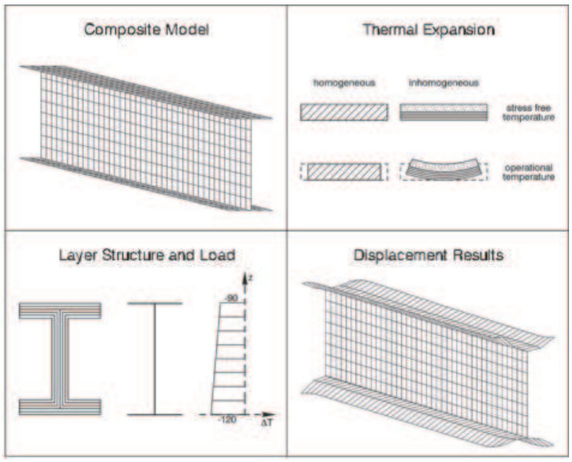

Figure 101: Laminate analysis of a girder

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 analoguous to a static analysis (see page 42).

Heat induced by an electrical field can be used for a subsequent thermal analysis (see page 66). From that thermal stresses can be derived performing a subsequent static analysis (see page 53).

Forces induced by a magnetic field can be used in a subsequent static analysis (see page 53).

PERMAS-EMD – Electrodynamics

A solution of *Maxwell*'s equations is available for dif-

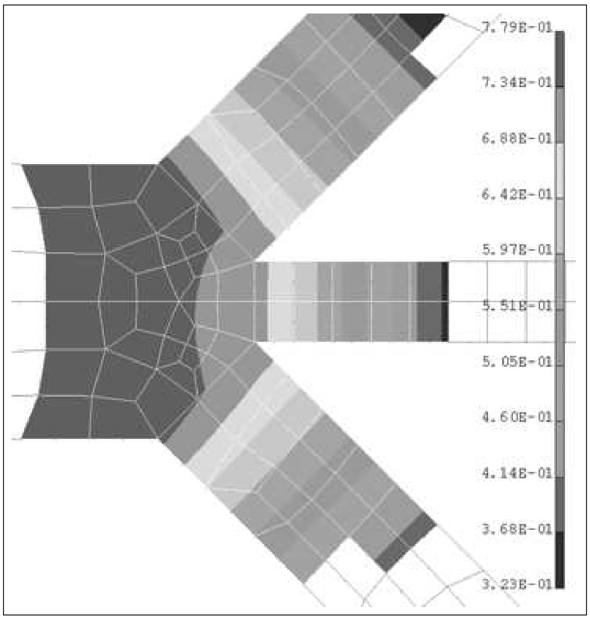

Figure 102: Scalar potential in an electric junction

ferent problem cases:

- eddy currents
- induction problems
- 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 61).

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 *MPC*s are translated (see page 41). Beside Components, different Situations with constraint and load variants may be specified within MEDINA (see page 38).

It is a very special feature in MEDINA that all PERMAS element tests have been integrated for element validation during pre-processing. So, if a model is checked in MEDINA, it will pass the PERMAS tests without any serious problems.

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 continually adapted to new versions of MEDINA and extended to cover new features of PERMAS.

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.

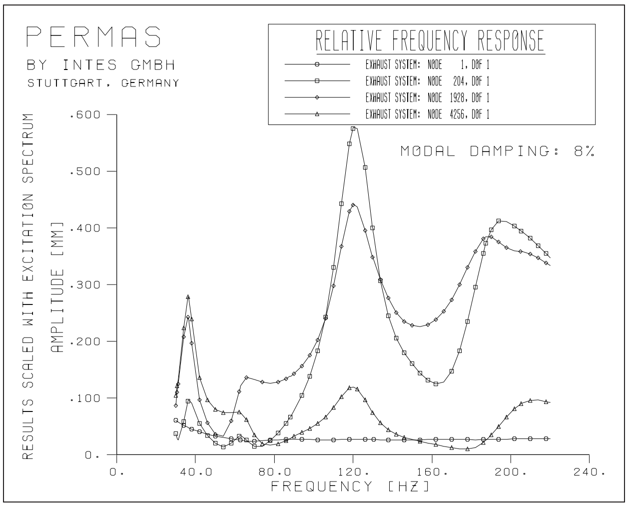

Figure 103: Exhaust System
Frequency response analysis

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.

The PERMAS command control can be made within I-DEAS. All necessary menus are available and may easily be adapted by the user to his needs. So, no exit of I-DEAS is necessary to perform a PERMAS run.

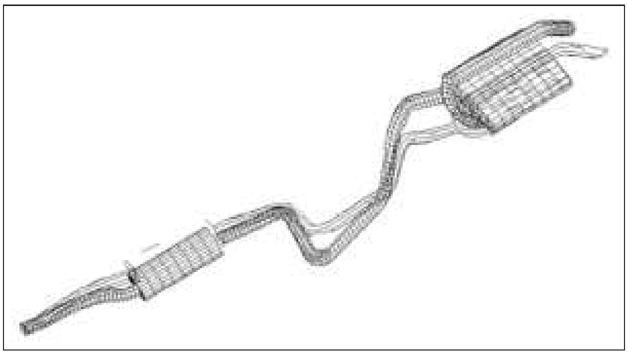

Figure 104: Mode Shape of an Exhaust System

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 59).

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 61).

The export is made in the format of ADAMS Version 9 and up.

PERMAS-DADS – DADS 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.

- Based on single component model.
- Static mode shapes defined by prescribed degrees of freedom.
- Static analysis and dynamic eigenvalue analysis.
- Output based on PERMAS post file format.

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 7.0

PERMAS-SIM – SIMPACK Interface

Post-processing interface for export of model topology, mass-normalized vibration mode shapes and condensed matrices.

- 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 61).
- 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 6 onwards). There are formats for HYPERVIEW Version 6, 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 59).

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.

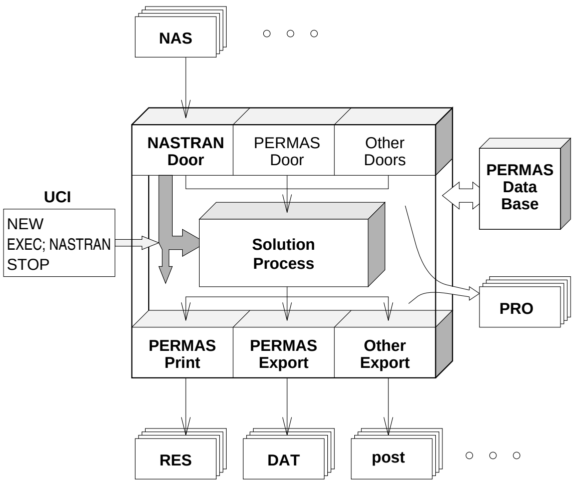

Figure 105: UCI trigger for NASTRAN-Task

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 his/her NASTRAN control data, automatically.

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 51.

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 Short 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.

- **Animator3 (GNS, www.gns-mbh.com)**: Postprocessing of PERMAS results is done via MEDINA format.
- **ANSA (BETA CAE Systems, www.beta-cae.gr)**: ANSA supports the MEDINA-Format which is used by PERMAS, too.
- **CATOPO (CES, www.ces-eckard.de)**: Pre- and post-processing for topology optimization and other applications is provided via PERMAS formats.
- **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 (vMach 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 to the pre-processor complements the PERMAS module H3D (see page 78) which exports the results for post-processing with HyperView.
- **iSIGHT (Engineous Software, www.engineous.com)**
- **MAGMAlink (MAGMA, www.magmasoft.de)**: This software is a module for the transfer of casting simulation results out of MAGMASOFT to FE meshes for stress and durability analysis.
- **Material data base MARLIS (M-Base, www.m-base.de)**: This material data base contains material data of steel sheets and is capable to issue the material properties as PERMAS material description.
- **modeFRONTIER (ESTECO, www.esteco.com)**
- **OptiY (OptiY e.K., www.optiy.de)**
- **pro-fe (CD-adapco, www.cd-adapco.com)**: This pre- and post-processor supports PERMAS formats.
- **SFE CONCEPT (SFE, www.sfe-berlin.de)**: In the concept phase of body-in-white design even automatic optimization with PERMAS can be applied. SFE CONCEPT generates a PERMAS model and reads the relevant results after the analysis run.
- **SimLab (SimLab Corporation, www.simlabcorporation.com)**: This pre- and post-processor provides a bi-directional interface to PERMAS.
- **TOSCA (FE-Design, www.fe-design.de)**: TOSCA is capable to work with PERMAS input ans output directly.

All mentioned names of products and companies belong to their holder. The use does not imply that such names are free for general use.

Installation and beyond

Supported Hardware Platforms

Architecture	Operating system
Itanium II	HP-UX V11 LINUX REDHat AS5
PA-RISC	HP-UX V11
POWER-n	AIX 5.3 LINUX SuSE SLES 10/11
PC x86(_64)	LINUX Debian 5/6, LINUX Ubuntu 10.4, LINUX RedHat AS5, LINUX SuSE 10/11
PC x86	Windows XP/Vista/7
PC x86_64	Windows XP64/Vista64/7

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:

- 32 bit operating system:
  - D32: Double precision floating point operations on 32 Bit machine words with a memory usage of about 2 GB.
- 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.
- **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.

In case of a floating license, the *license server* 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:** At least three servers are needed, and two servers have to be availbale at least. Each server has the total license equipment.

For the management of the license server a WEB interface and a a program interface (e.g. via Python) 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 subscribed by the users, who will receive it automatically via e-mail.

Usually in an 1 to 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. All shipments are made within a short time span.

When new versions of the pertinent pre- and postprocessor 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.

The actual list of supported hardware platforms is subject to continuous changes mainly on the side of the operating systems. Often, different versions of the operating system are supported on one platform. Platforms or operating systems which are hardly or no more used will be regularly discarded.

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.

- the integration of external management tools (like Sun 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.
- the hotline message system OTRS for the reporting of application problems by email.
- The PERMAS Tools comprise a number of tools:
  - **PERMASgui**: This is a small graphical user interface to submit a PERMAS job and to set the most common job parameters,
  - **PERMASgraph**: A graphical user interface to plot XY data from PERMAS, MEDINA, and PATRAN formats.
  - **PERMASmonitor**: This can be used to view the progress of running PERMAS jobs.

In addition to the PERMAS environment INTES offers the installation of a very powerful, universally applicable editor, too. Essentially, this is an extended version of the standard GNU-Emacs – with great enhancements in comfort and functionality.

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 on-line access to all PERMAS documents.

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.

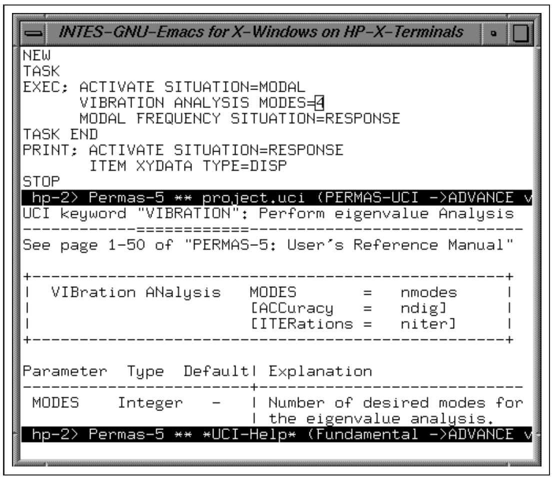

Figure 106: Context-Help for UCI-Files inside Editor

Documentation

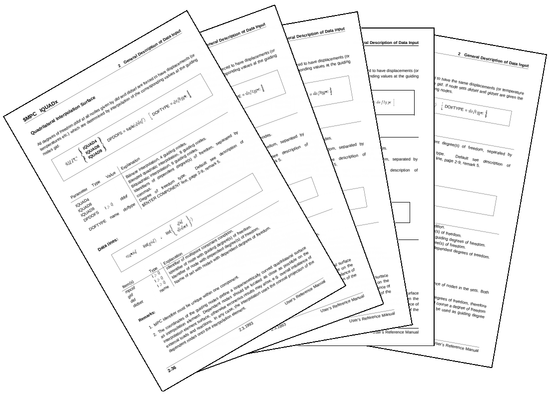

Figure 107: PERMAS documentation

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*

  the following documents are provided:
  - *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 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 development team is intensively working on the extension of already existing functional modules and on the development of new PERMAS modules.

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.
- **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.

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.

Additional Information

For requesting more information and in case of additional questions please contact:

Marketing:	**Reinhard Helfrich**
Phone:	**+49 (0)711 784 99 - 11**
Fax:	**+49 (0)711 784 99 - 10**
E-mail:	**info@intes.de**
WWW:	**http://www.intes.de**
Address:	**INTES GmbH** **Schulze-Delitzsch-Str. 16** **D-70565 Stuttgart**

Index

Contact:

Phone:	+49 (0)711 784 99 - 0	Address:	INTES GmbH
Fax:	+49 (0)711 784 99 - 10		
E-mail:	info@intes.de		Schulze-Delitzsch-Str. 16
WWW:	http://www.intes.de		D-70565 Stuttgart