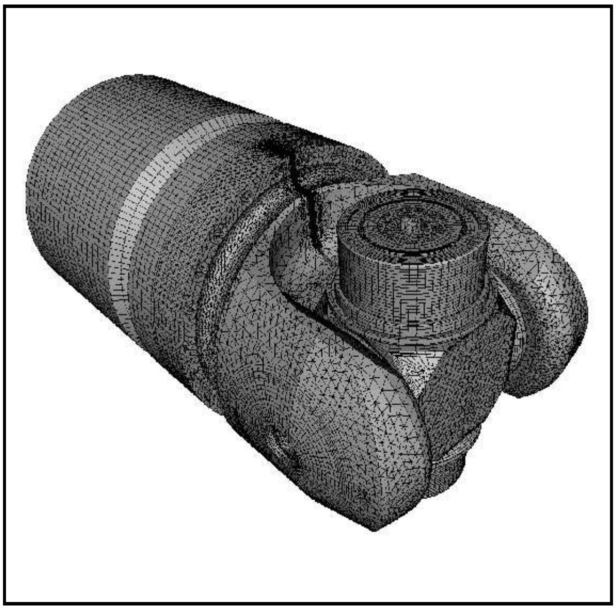


PERMAS

Short Description
Version 9.0

Ingenieurgesellschaft für
technische Software mbH
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 postprocessors),
- consultancy and training of users,
- performance of analysis projects.

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

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.

INTES 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 and analysis services
  - with MEDINA, I-DEAS, PATRAN
- introduction of FE analysis in enterprises, continuous consultation service (hotline), and support on current projects.

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

The finite element model of a cardan shaft on the frontpage appears by courtesy of Voith Turbo GmbH & Co. KG in Heidenheim, Germany. The model has 857,000 elements, 1,140,000 nodes, 25,000 contacts, 3,200,000 degrees of freedom.

Contents

Overview

This short description provides information on all essential characteristics of PERMAS and its application. Therefore, the description is organized into five 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 4 to 9.
- The module-independent **universal features** of PERMAS are explained on pages 10 to 20.
- The available **functional modules** are described on pages 20 to 38.
- Additional information about the **environment** of PERMAS is given on the remaining pages 39 to 40.

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.

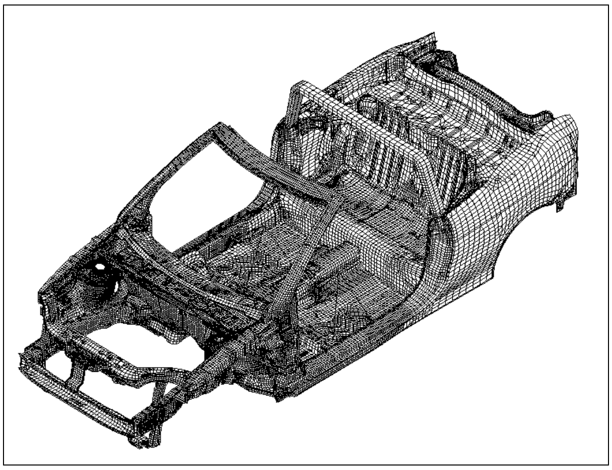
PERMAS enables the engineer to perform comprehensive analyses and simulations in many fields of applications like stiffness analysis, stress analysis, determination of natural modes, dynamic simulations in the time and frequency domain, determination of temperature fields and electromagnetic fields, 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 behaviour 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 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.



Car body model, Wilhelm Karmann GmbH, Osnabrück, Germany.

Benefits of PERMAS

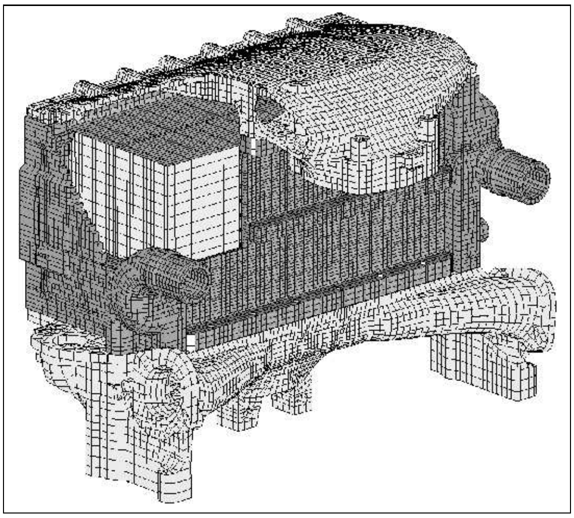
PERMAS is an internationally established FE analysis system with users in several countries. INTES has developed the system and, additionally, offers **individual consultation and user support** and all training required. The consultation covers all requests regarding the use of the software but also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

The benefits arising from the use of PERMAS can be characterized by the following three 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.
- Efficient equation solvers and optimized data storage schemes provide PERMAS with **ultimate computing powers** 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 three 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.



Charge air cooler, Behr GmbH & Co.

What's New in PERMAS Version 9

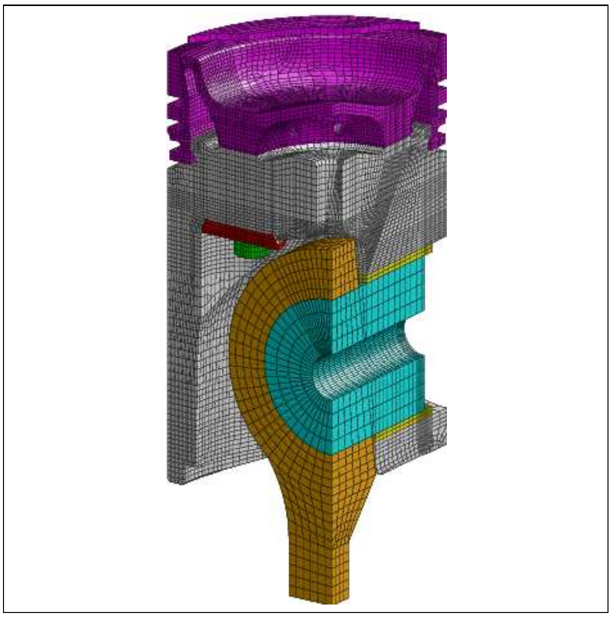
The new Version 9 of PERMAS is the result of about 18 months development work since the shipment of the predecessor version 8. For the regular reader of our Short Description of PERMAS, a rough overview summarizes the main changes in the new version. Of course, a complete and detailed Software Release Note is available with Version 9 in addition.

PERMAS V9 offers again improved computing performance. Better scalability in parallel execution, disk space savings, and improved MPC processing are the main features to enhance performance.

- **New modules:**
  - Dynamic condensation with the module 'Extended Dynamic Eigenvalue Analysis' (DEVX, see page 24).
  - DADS interface (see page 37).
  - SIMPACK interface (see page 37).
- **Major extensions:**
  - Contact with pretension (see page 22).
  - Combined geometric and material nonlinearities (see page 23).
  - Steady state results in time domain derived from frequency response (see page 25).
  - Support of frequency constraints in module TOPO (see page 30)
  - Combination of reliability analysis and optimization for a robust design (see page 32)
  - Incompatible connection of shell elements with volume elements (see page 12).
  - XY data in coordinate directions (see page 18).
  - Distributed element loads as a function of coordinates.
  - Data line generation in PERMAS input.
- **New elements:**
  - New shell family for linear and nonlinear applications with 3 and 4 nodes for linear shape functions and 6, 8, and 9 nodes for quadratic shape functions. These elements are using a 3-dimensional shell formulation and are designed to work for material non-linearities.
  - New HEXE8 element using the EAS (Enhanced Assumed Strain) formulation with 9 or 21 extended strain modes. This element is available in addition to the previous HEXE8 element.
  - New TET10 element improving the use of TET meshes in contact analysis. This ele-

ment is available in addition to the previous TET10 element.
- New general beam elements with and without warping allowing one intermediate cross section between the two nodes of the element.
- New axisymmetric shell element with corresponding fluid element, coupling element, and wave element.
- In addition to the already existing linear plot elements, the corresponding quadratic plot elements have been implemented.

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



Ship engine piston, Mahle GmbH, Stuttgart, Germany.

For all system platforms an update to the current release of the operating system had been performed (see Page 39).

Universal Features

The outstanding module-independent basic features of PERMAS are as follows (see pages 10 to 19):

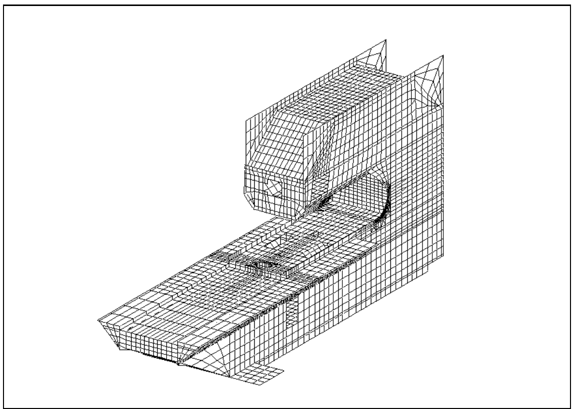
- Variant analysis (see page 10)
- Surface and Line Description (see page 10)
- Automated coupling of parts (see page 11)
- Automated spotweld modelling (see page 12)
- Multiple kinematic constraints (see page 12)
- Automatic detection of singularities (see page 13)
- Hierarchical substructuring, with automatic subcomponent insertion (see page 13)
- Same elements for different analysis types (see page 14)
- General material description (see page 15)
- Node and element sets (see page 16)
- Mathematical functions (see page 16)
- All kinds of loading (see page 16)
- Integrated interfaces to pre- and post-processors (see page 17)
- Input and Output of Data Objects and matrices (see page 18)
- Combination and transformation of results (see page 18)
- Output of XY result data (see page 18)
- Calculation of cutting forces (see page 19)
- Restart facility (see page 19)
- Open software through Fortran and C interfaces (see page 19)
- Direct coupling of different analysis types (see page 19)
- Coupling with CFD (see page 20)

Available PERMAS Modules

The below listed functional modules are explained in more detail on pages 20 to 38:

- Model Quality Assurance (MQA)
- Linear Statics (LS)
- Contact Analysis (CA)
- Nonlinear Statics (NLS)
- Buckling Analysis (BA)

- Dynamic Eigenvalue Analysis (DEV)
- Extended Dynamic Eigenvalue Analysis (DEVX)
- Dynamic Response Analysis (DRA)
- Extended Dynamic Response Analysis (DRX)
- Fluid-Structure Acoustics (FS)
- Heat Transfer (HT)
- Laminate Analysis (LA)
- Design Optimization (OPT)
- Layout Optimization (TOPO)
- Reliability Analysis (RA)
- Steady-state electromagnetics (EMS)
- Electrodynamics (EMD)
- Model editor FELIX
  - Preprocessor (FEPRE)
  - Postprocessor (FEPOST)
- Interfaces to various pre-/post-processors
  - MEDINA (MEDI)
  - PATRAN (PAT)
- Interfaces to CAD systems with Pre- and Post-processor
  - CATIA (CAT)
  - I-DEAS (ID)
- Interfaces to other analysis packages
  - ADAMS (AD)
  - DADS (DADS)
  - SIMPACK (SIM)
  - MATLAB (MAT)
  - NASTRAN (NAS)
  - FIRST/PIMO3D (FIRST)
  - MpCCI (CCL)



Blanking Press,
Trumpf GmbH + Co., Ditzingen

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.

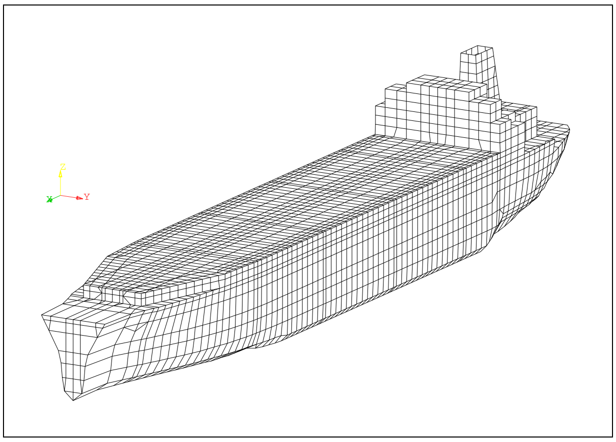
Parallelization

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.

Because the principal architectures of available parallel computers are either shared memory or distributed memory, PERMAS offers two different parallelization strategies, too:

- On shared memory computers 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.

- On distributed memory computers the parallelization is based on MPI (Message Passing Interface), a standard for the control of communication between different processors. This tool is required for parallel use of different processors to solve a single ananlysis task.



Methane Carrier

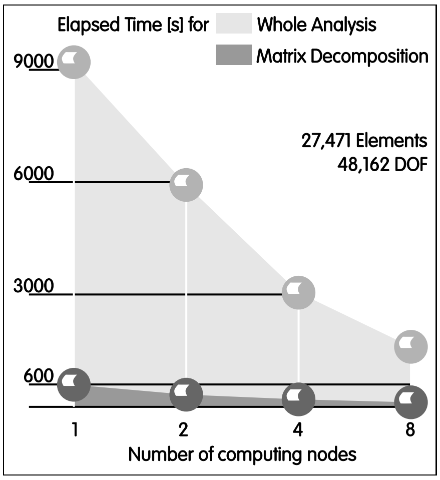
In addition, PERMAS allows asynchronous I/O on both different architectures, which realizes better performance by overlapping CPU and I/O times.

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 UNIX platforms, where a sequential version is supported, too.



Eigenvalue Analysis, Methane Carrier, run time on IBM SP2

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.

Areas of Application

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

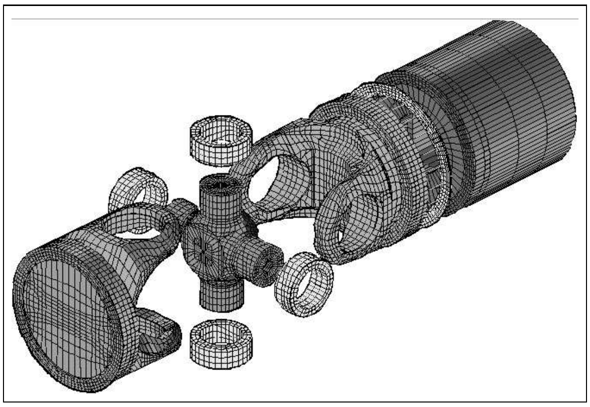
- Automotive industry
- Aerospace industry
- Ship building industry
- Mechanical engineering
- Offshore- and power plant engineering
- Plant- and equipment engineering

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 20).
- **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.



Model of a cardan shaft, Voith Turbo GmbH & Co. KG

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:

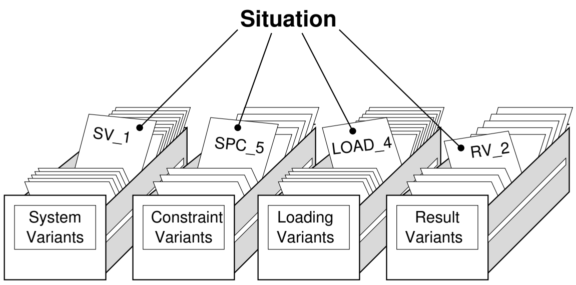
- Especially developed for the management and administration of the software, a development tool provides for a safe software database, which includes all modifications and new subroutines and manages them in a unique and apprehensible way.
- A problem report management system gathers all messages regarding 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.

Basic Functions:

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.

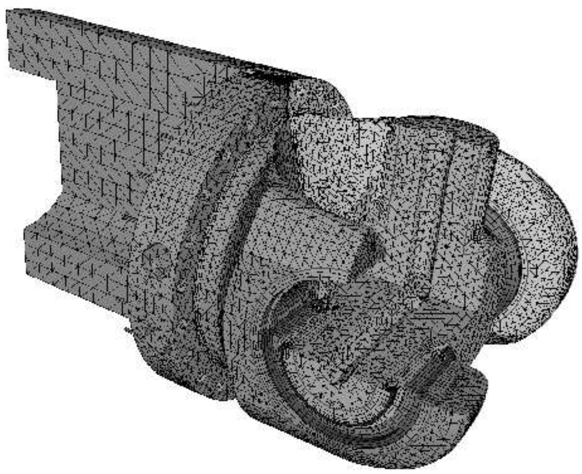


Variants in PERMAS

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

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

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



Cardan shaft model with incompatible meshes in sectional view (Voith Turbo GmbH & Co. KG)

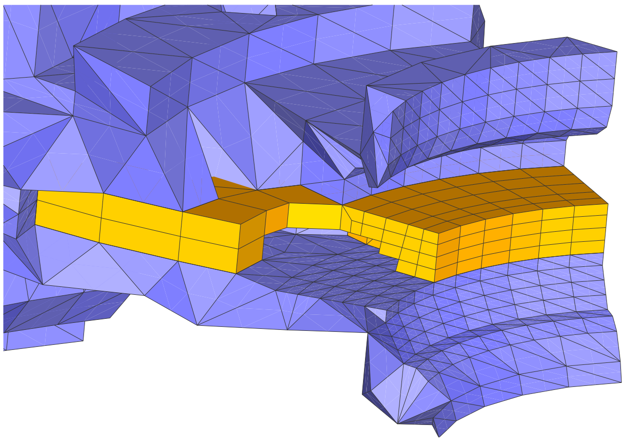
Surface and Line Description

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 16),
- by specifying geometry elements (see element library page 14).

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 16),
- by specifying geometry elements (see element library page 14).

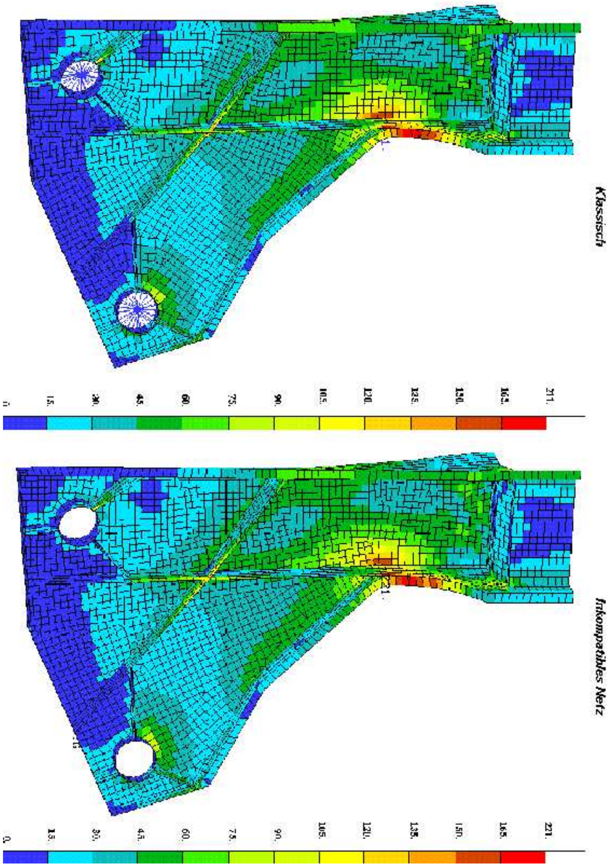


Element transition (HEXE8/TET10) with incompatible meshes

Automated Coupling of Parts

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

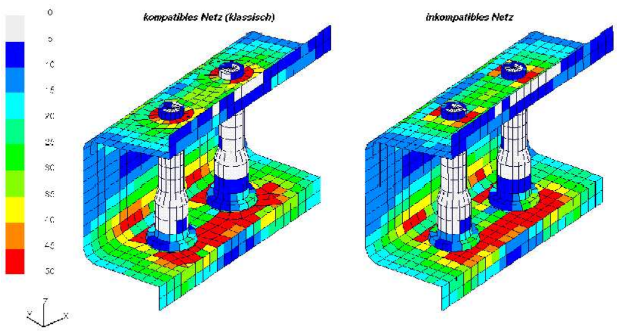
- It allows for a much more flexible organization of the model generation where single parts are administrated and exchanged.
- It facilitates a fast modification cycle for virtual prototypes.
- Accurate mesh transitions without bad elements are possible.
- Welding spots and surface contact become much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



Ribbed shell model with compatible meshes (top) and incompatible meshes (bottom)

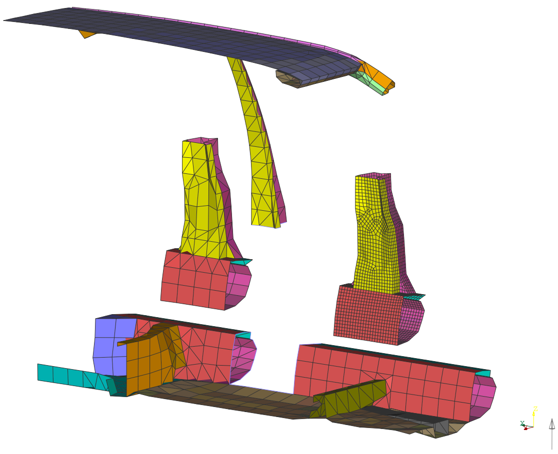
The coupling definition consists of the following steps:

- Definition of a guiding surface and the degrees of freedom to be coupled
- Definition of a dependent surface or node set



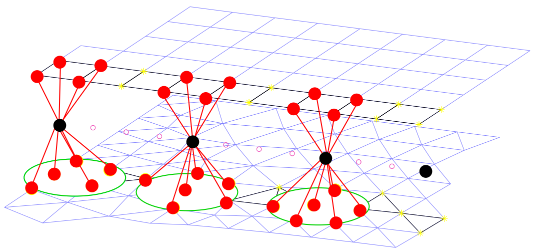
Results of compatible (left) and incompatible (right) part assembly

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



Application of local mesh refinement

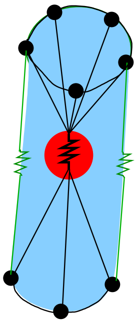
The coupling is a general feature that may also be used for coupled analyses, where different mesh densities occur due to the modelled physics. One example is a coupled fluid-structure acoustic computation, where the acoustic mesh may be coarser than the mechanical part.



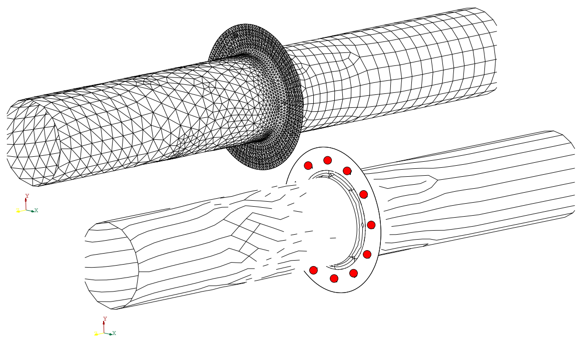
Automated Spotweld Modeling

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

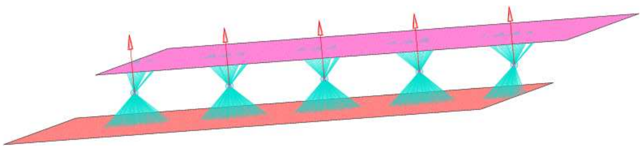
- Specification of spotweld positions
- Selection of (incompatible) faces
- Specification of spotweld stiffness and (optionally) the spotweld diameter



The spotweld stiffness is modeled by a spring element, which is coupled to the neighboured parts by automatically generated MPC conditions. 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.



Verification of generated spotweld connections

Kinematic Constraints

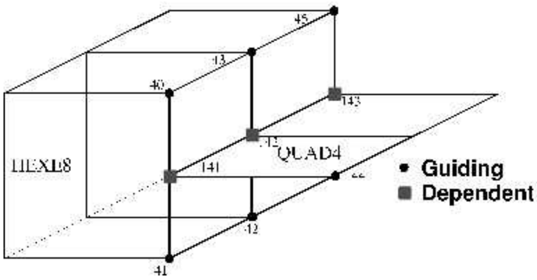
For the specification of single point kinematic constraints (absolute constraints) suppressed and prescribed degrees of freedom are available for zero or non-zero displacements, respectively.

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

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

- Multiple degrees of freedom may be forced to have identical freedom values by simple **Assignment** (for modelling swivels, hinges or sliding surfaces and for boundary conditions in cyclic symmetry).
- **Rigid Bodies** allow the modelling 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 and distributing loads:
  - lines with 2 or 3 guiding nodal points,
  - triangular and quadrilateral areas with 3 or 6 and 4, 8 or 9 guiding nodes, respectively.

  The interpolation regions may be used also for a **Volume-Shell Transition**, i.e. the connection of plates and shells with solid element structures. 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 10 and the Automated Coupling of Parts on page 11).
- **General MPCs** allow any linear combination of the involved degrees of freedom.



MPC example: Volume Shell Transition

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.

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 postprocessor file. They can easily be inspected in order to detect the missing supports or other modelling errors.

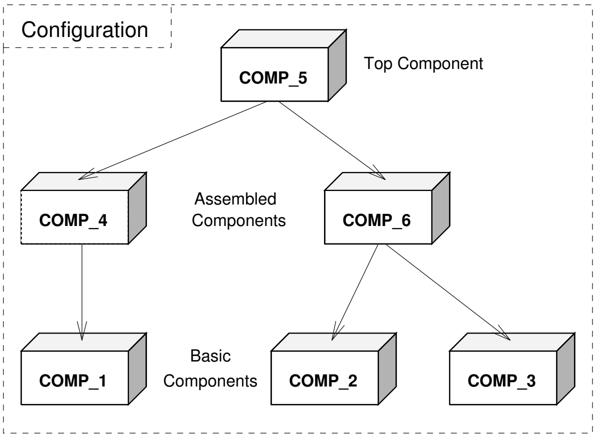
In dynamic mode analysis the rigid body modes are detected and decoupled automatically.

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.

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.



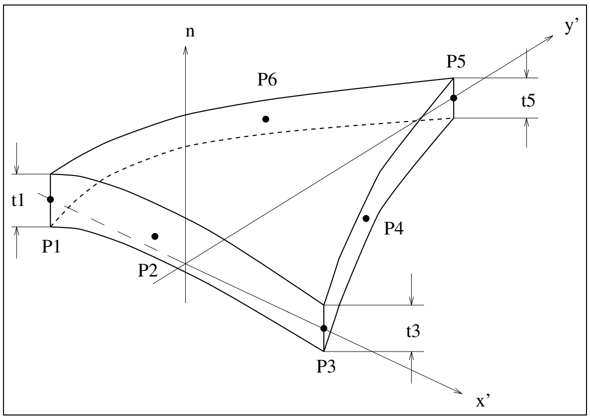
Substructuring in PERMAS

- 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**.
- 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.
- Such matrix models may be forwarded to customers and suppliers in place of the real geometric models.
- Substructuring permits the separated modelling and verification for all parts of the structure, prior to the final assembly.
- Single FEA models from distinct modelling 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.

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**:
  - tetrahedron elements with 4 or 10 nodal points and straight or curved edges,
  - pyramid element with 5 nodal points,
  - pentahedrons with 6, 15 and 18 nodal points and straight or curved edges,
  - hexahedrons with 8, 20 and 27 nodal points and straight or curved edges.
- **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.



Triangular element with curved edges

- **Beam Elements** with 2 nodal points, optionally with or without rigid lever arms (offset nodes):
  - beams with arbitrary shaped solid cross section,
  - thin-walled profiles and tubes with open or closed cross sections,
  - thin-walled open or closed tubes with cross sections tapered along the beams' length axis,
  - fluid-filled or fluid-surrounded straight or curved pipe elements.
- **Plate and Shell Elements**:
  - triangular or quadrilateral elements for thin and moderately thick plates and shells.

- triangular or quadrilateral elements sandwich shells.
- a thin-walled triangular plate element following Kirchhoff's Theory.
- triangular (with 3 or 6 nodes) and quadrilateral elements (with 4, 8, or 9 nodes) with 3-dimensional shell formulation for linear and non-linear material.
- Triangular and quadrilateral elements for layered cross-sections (composites).

- **Discrete Elements**:
  - linear and non-linear spring elements,
  - various mass elements,
  - linear and non-linear damper elements,
  - control elements with user defined control mechanism.
  - 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.
- **Plot Elements** in form of lines and triangular respectively quadrilateral areas for result evaluation.
- **Convectivity Elements** to model the convectivity behaviour 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.

Material Description

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.

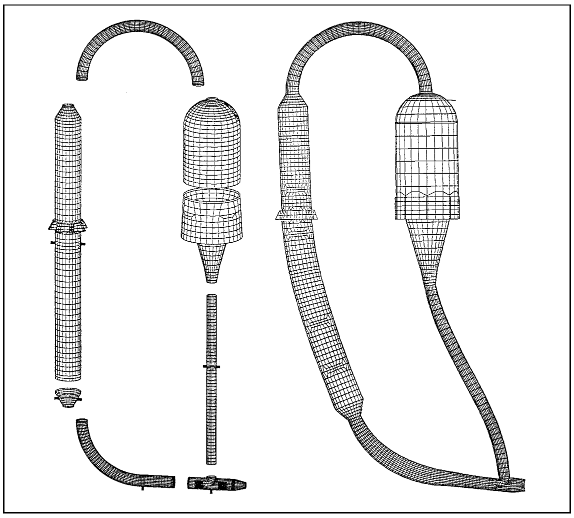
Because of the application of PERMAS in different fields, different material properties can be defined like elasticity, density, compressibility, damping, thermal expansion, heat conductivity, heat capacity, absorption, volume drag, electric conductivity, dielectricity, magnetic permeability. There,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

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

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 34).



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 (quasistatic acceleration).
- Nodal loads:
  - Concentrated loads,
  - Distributed loads,
  - Prescribed displacements,
  - Temperature loads,
  - State of contact.
- Element loads:
  - Distributed loads,
  - Initial strains.

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.

Interfaces

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

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

- MEDINA (page 35)
- CATIA (page 36)
- I-DEAS (page 37)
- PATRAN (page 36)
- ADAMS (page 37)
- DADS (page 37)
- SIMPACK (page 37)
- NASTRAN (page 38)
- MATLAB (page 37)

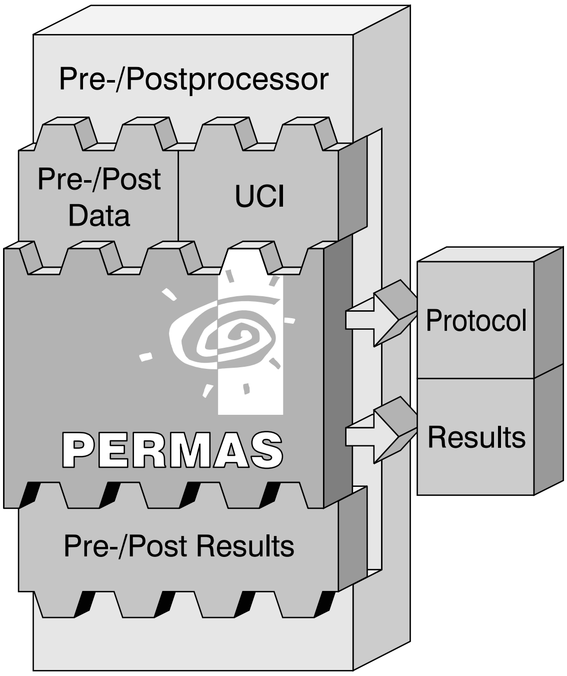
Moreover, a growing number of extra interfaces to PERMAS are available from partner companies or INTES.

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

During **input** complete FE-models are read and translated into autonomous PERMAS data structures. If requested, any part may be combined with or completed by additional PERMAS data files.

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

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.



Integration of PERMAS in pre- and post-processor

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

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

Input and Output of Data Objects

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.

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 16) 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.

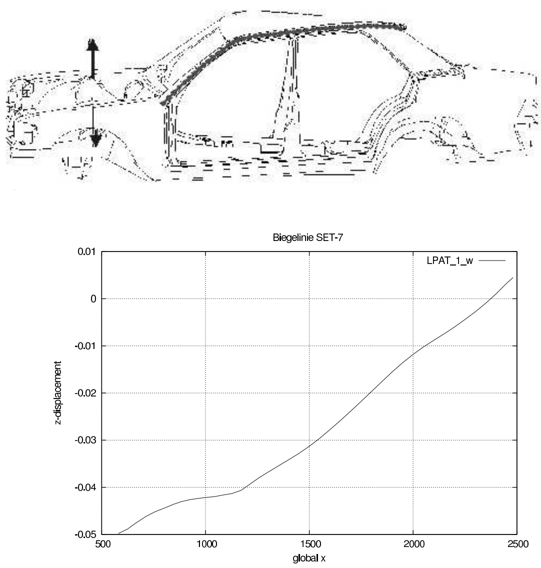
Beside the transformation of real results, complex results of a frequency response analysis can be transformed to another coordinate system, too.

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.

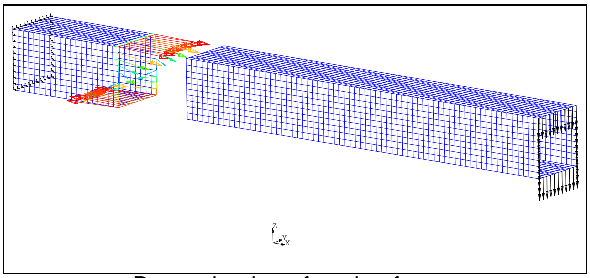
As a special case in order to study e.g. the stiffness behaviour of a part, it is possible to create xy data for nodal point results with the coordinate direction as abscissa. Then, the abscissa values are prepared in ascending coordinate direction for the selected nodes. A local coordinate system may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



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.



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.

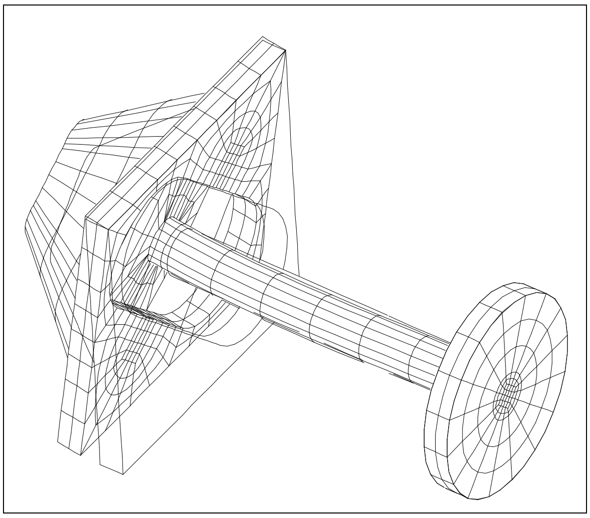
Open Software System

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 16).

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.



Clutch Element, Static Displacements

Direct Coupled Analyses

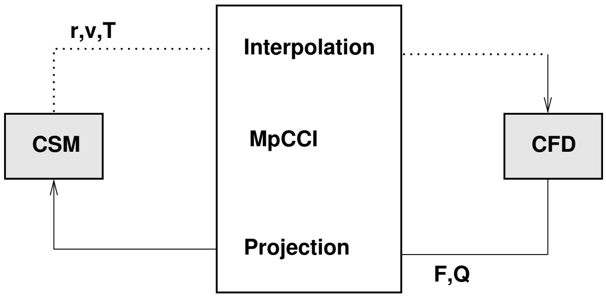
PERMAS enables different coupled analyses in one run, e.g.

- 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.

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.



Within the project a general COupled COmmunication LIBrary (COCOLIB) has been developed by GMD/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 (available from PALLAS GmbH 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 and STAR-CD. So, both coupled packages 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.

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.

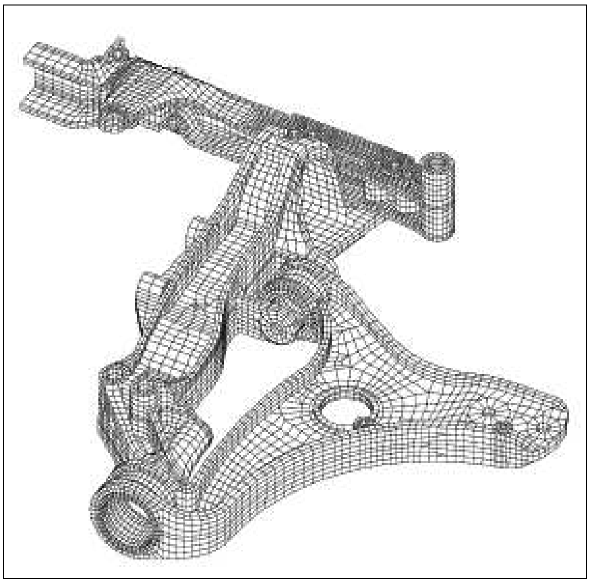
- Today Finite Element calculations are used in the product development to early assess different design variants and to accelerate the development process.
- The skill of FEA systems users changes from an expert level to a more general background.
- The FE applications become more and more complex.

Beside the software quality (see page 9) the reliability of FEA results depends on the following points:

- **Comprehensive model testing:**
  PERMAS performs very intensive tests of the input data. There are about 4000 different plain text system messages even 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 13).
- **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 17).

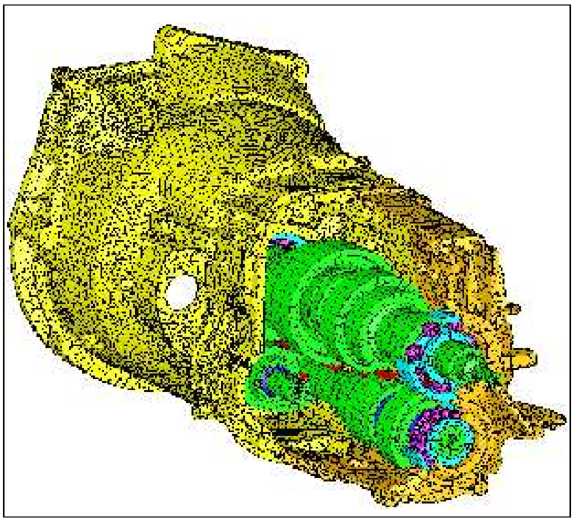


Model of a front axle, Porsche AG, Weissach

PERMAS-LS – Linear Statics

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

- The following definitions of kinematic boundary conditions are available:
  – Suppressed degrees of freedom,
  – Prescribed degrees of freedom,
  – Linear constraints (MPCs) (see page 12).
- Different kinds of static loading are available (see page 16).
- 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.



Bearing housing ZF AG

PERMAS-CA – Contact Analysis

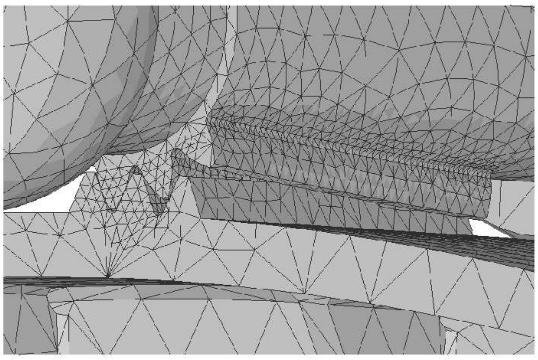
Static analyses with non-linear boundary conditions (contact problems) can be analyzed using the PERMAS-CA module.

Contact boundary conditions may be present between elastic bodies or between elastic bodies and a rigid counterpart.

There are several methods to describe contacts:

- by specification of the contact nodes in pairs,
- by specification of nodesets for each contact zone (the node pairs are detected automatically),
- assigment of nodes/nodesets to surfaces (incompatible meshes),
- 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.



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.

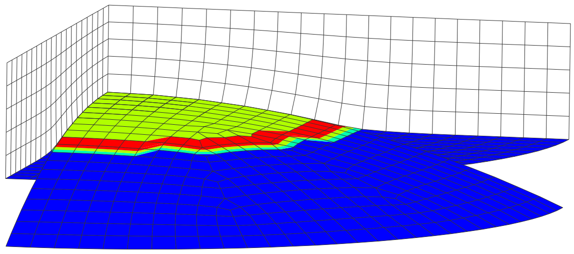
The contact analysis can include frictional contact using Coulomb friction (for slip and stick), which may be either isotropic or anisotropic.

The specification of a load history allows the correct simulation of any contact situation with slipping and sticking friction. This facilitates the convenient simulation of such situations in a quasi-static analysis.

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

Comprehensive checks allow the verification of contact models like type of contact, its geometry (gapwidth and normal vector), 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.

To support the use of automatic TET mesh generation for contact models a modified TET10 element is available, which gives improved results compared with the classical TET10 element.



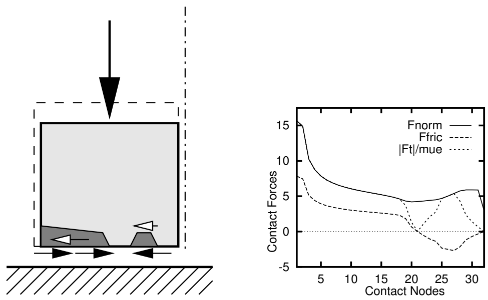
Contact with friction between cylinder and sphere
(with sticking and slipping regions)

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 modelling of contacts.

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

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

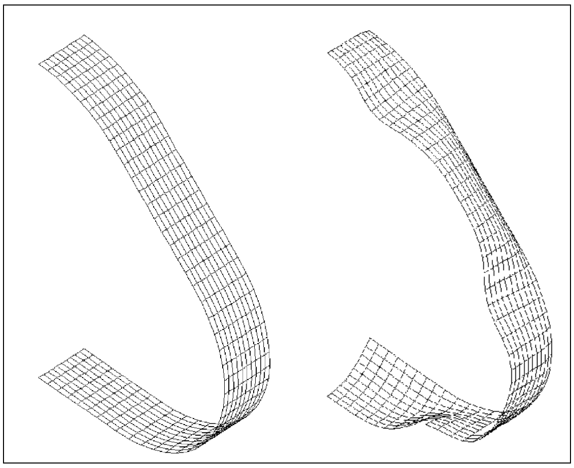


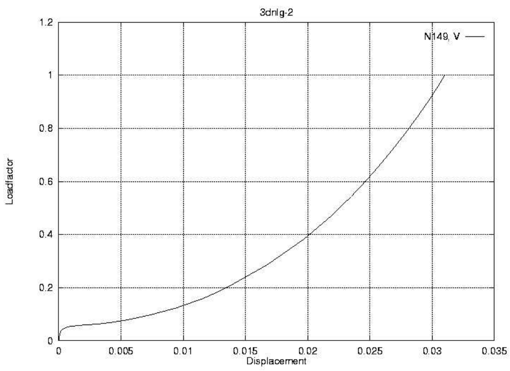
Contact with friction

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, Quasi Newton BFGS, 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.





Nonlinear NAFEMS Test

Material nonlinearities

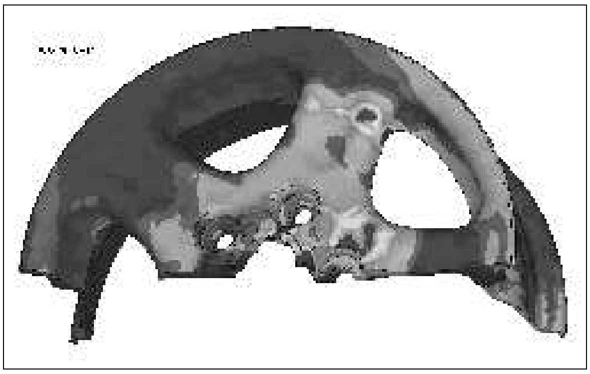
This part of the module allows the analysis of nonlinear material behaviour of models with small strains:

- Nonlinear elsticity (of Cauchy type)
- Plasticity (von Mises, Tresca, Drucker-Prager, Mohr)
- 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.

An incremental and iterative solver strategy is based on Newton-Raphson, Modified Newton-Raphson, and Quasi-Newton 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).



Impact test of a wheel

Combination of material and geometrical nonlinearities

Analyses with material nonlinearities can take into account the geometrical nonlinear effects, too. Such situations are often observed with flexible shell structures.

General

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

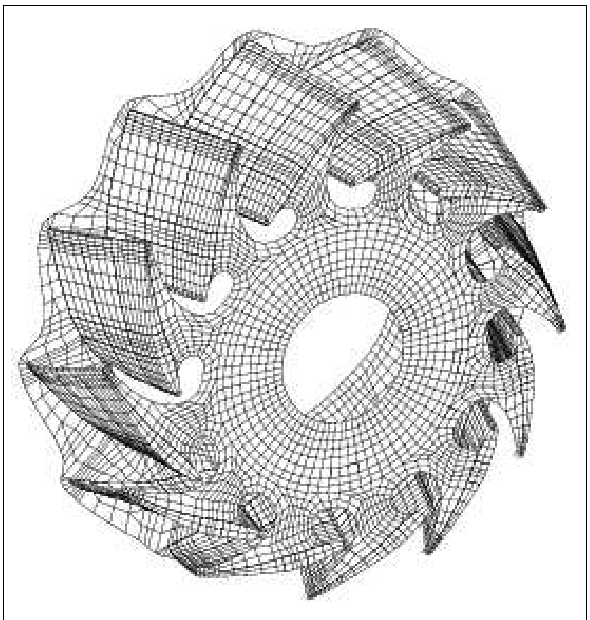
The results of a nonlinear analysis may be used for subsequent analysis like a dynamic mode analysis.

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 pre-buckling behavior.

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



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

PERMAS-DEV – Dynamic Eigenvalues

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

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 or Cross Orthogonality factors (COF) can be generated to compare modes between two different modal analyses.
- As a measure for the completeness of the modal model, effective masses are generated and printed on the result file.

PERMAS-DEVX – Dynamic (Condensation)

This module provides additional methods for dynamic eigenvalue analysis.

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 14).

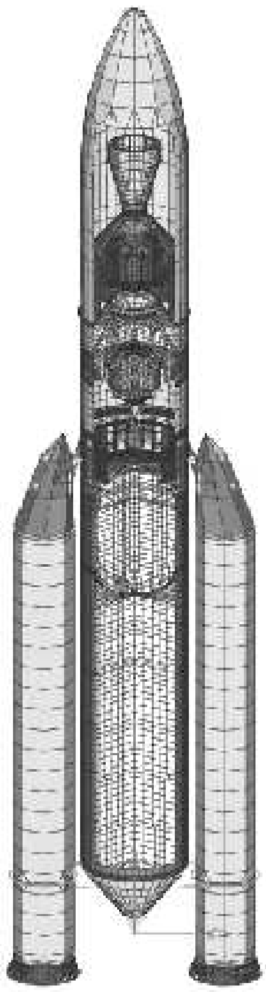
The functionality may be summarized as following:

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

Two condensation options are available for coupled fluid-structure acoustics (see also page 27):

- **„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.
  - Condensation points are not in touch with pressure dofs.
- **„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.



Ariane 5 model by courtesy of EADS Launch Vehicles, Les Mureaux

PERMAS-DRA – Dynamic Response

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

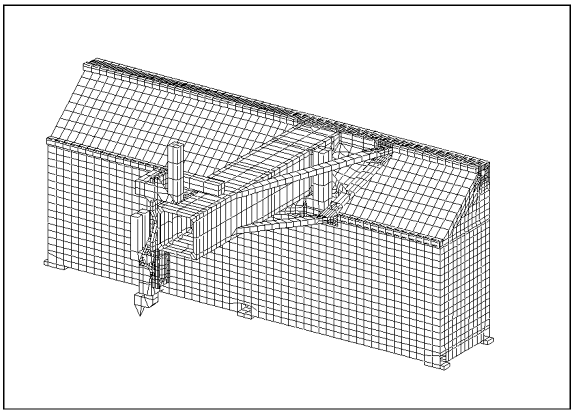
The solution of the dynamic equation is performed either directly using physical coordinates or in modal coordinates after a transformation into the modal space.

- The response in the **time domain** (transient response) is determined by an integration of the equation of motion:
  - Absolute transient response with or without rigid body response.
  - Direct integration of the equation of motion or integration after a transformation to the modal space.

  Local nonlinear effects are taken into account by
  - nonlinear spring elements,
  - nonlinear damper elements, and
  - general 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 adddition, a static load case can be taken into account. This is facilitated for all periodic excitations with known harmonic composition.



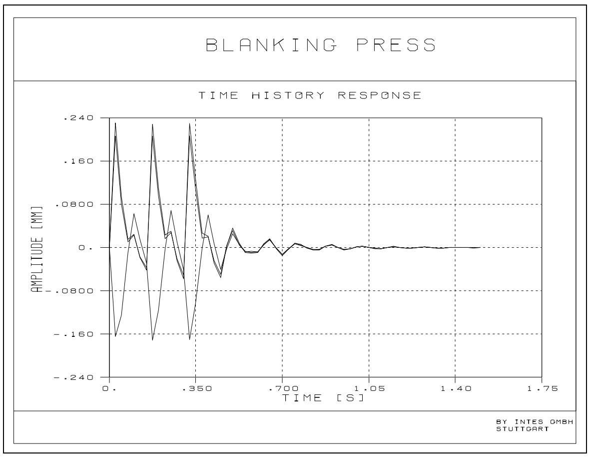
3D laser cutting and welding machine, Trumpf GmbH + Co., Ditzingen

By specification of a node set (see page 16) 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 modelled 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,
  – direct input of damping matrix,
  – skew-symmetric Coriolis matrix due to gyroskopic effects.

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



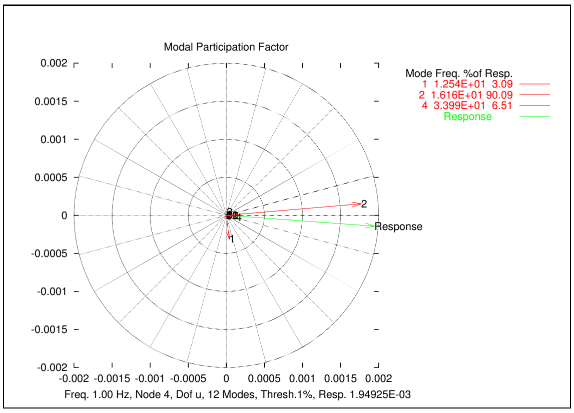
Transient response of some degrees of freedom

- The excitation is defined by static loading cases modulated by functions of time respectively frequency (see page 16). 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.

Additional tools are available for the further processing of results:

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



Modal grid participation factors

PERMAS-DRX – Extended Dynamics

This module comprises additional methods for structural response analysis:

- Spectral Response Analysis (or Earthquake Spectral Response Analysis)

Spectral Response Analysis

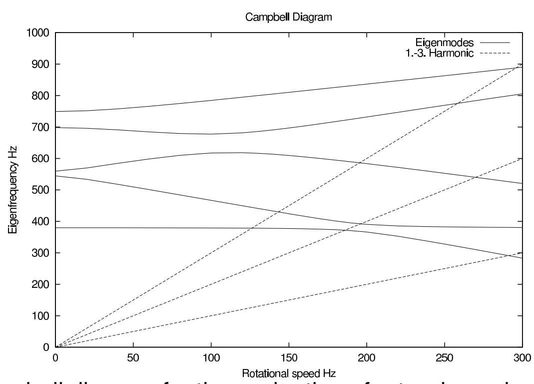
In case of a prescribed ground motion, like in earthquake analysis, the dynamic response behavior is determined by a special method, which results in maximum response values. There, the following requirements have to be fulfilled:

- 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.



Campbell diagram for the evaluation of rotor dynamics

Analysis of Rotating Systems

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

Statics

In a quasistatic analysis, which may include contact at the hub, the centrifugal forces due to rotation are taken into account. The reference system is co-rotating. 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

For dynamics of rotating systems, the assumption is a linearized equation of motion with constant coefficients. Usually, a co-rotating reference system is taken. If rotating and non-rotating parts are present, the rotating part is taken as rigid and the reference system is the inertial system.

In the case of a coupling of rotating and non-rotating parts, no restrictions have to be observed for the non-rotating parts, but the rotating parts have to be taken as rigid and at a constant rotational speed. In addition, the rotating parts have to be symmetric.

For such configuration, all direct and modal methods in time and frequency domain can be applied including steady-state response, where for a periodic excitation several frequency response analysis results are superposed (see also page 25).

In the case of dynamics in a co-rotating reference system, no additional restrictions have to be observed for the non-rotating parts, but the support has to be symmetric and the rotational speed of the reference system has to be constant.

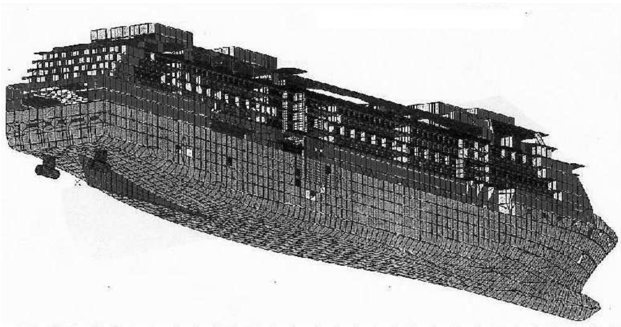
Also for such configuration, all direct and modal methods in time and frequency domain can be applied including steady-state response, where for a periodic excitation several frequency response analysis results are superposed (see also page 25).

For dynamics in the co-rotating reference system, 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.

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 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.



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 modelled 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 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 24), where the mass of the fluid is taken into account to calculate the structural modes.

For the calculation of the dynamic response behavior, modal superposition and direct integration methods are available:

- The response in the **frequency domain** (frequency response) is determined by the solution of the linear complex system of equations for each excitation frequency requested. In general, a fully populated equation system has to be solved.

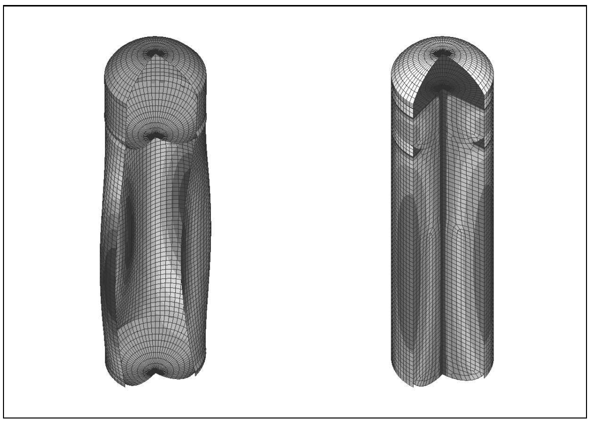
By specification of a node and/or element set (see page 16) the run time for modal superposition methods can be drastically reduced when the results are only determined for the set members. The **reduced** response results in enormous disk space savings.

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 16). The load definitions may consist of:

- Structural loads as described for the DRA module
- prescribed pressures.



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

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-HT – Heat Transfer

Temperature fields will be analyzed using the PERMAS-HT module.

- 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 16).

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.

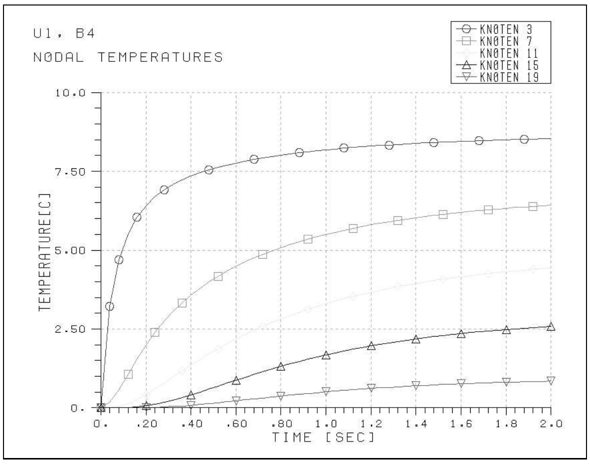
All finite elements applicable in static analysis may be used in heat transfer analysis. For shell elements a temperature gradient between top and bottom surface is allowed.

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 20).

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.



Transient Temperatures at selected 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 16).
- 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, for transient analyses the temperature or the heat flux may be issued for any point in order to generate xy-plots.

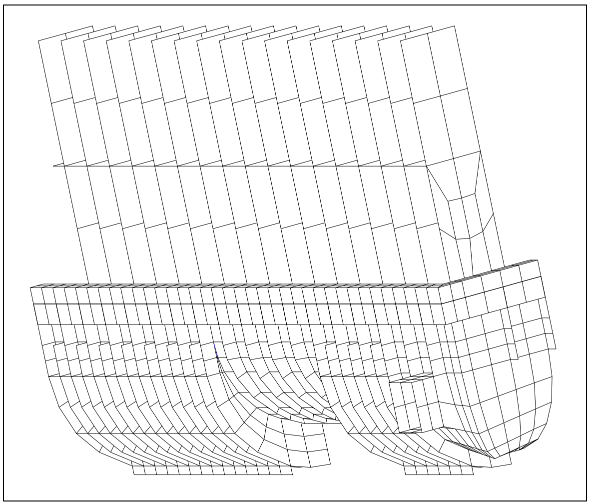
Arbitrarily composed element sets allow for the output of the heat flux through a part of the surface in absolute or area specific values. For transient analyses these heat balance results may be issued to produce xy-plots.

PERMAS-OPT – Design Optimization

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

The following design variables are provided:

- areas of cross section, inertia moments and general functions between these properties for beam elements
- thicknesses/offsets/nonstructural mass of shell elements
- stiffness and mass of spring elements
- node coordinates for shape optimization



Optimization of a water box

In each optimization constraints may limit the value range for design variables as well as for the response quantities like displacements, forces, stresses and compliance. Such constraints can be defined for eigenfrequencies, too.

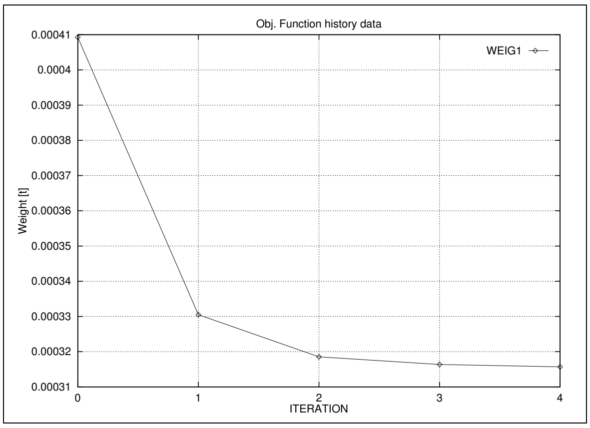
Optimization for a robust design is achieved by additional reliability constraints. Then, the design fulfills all of the above mentioned constraints and it is also reliable regarding uncertain model parameters (see page 32 for more details).

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

The optimization allows taking into account several loading cases as well as different boundary conditions using the variant analysis (see page 10.) In addition, dynamic mode frequencies can also be optimized, where a mode tracking during the structural changes is performed automatically.

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.

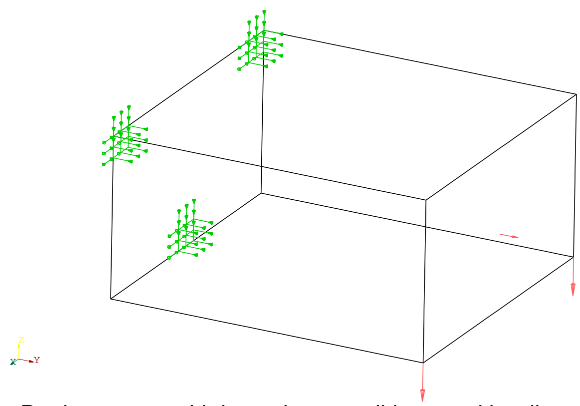


Reduction of weight during the optimization of a water box

PERMAS-TOPO – Layout Optimization

Supports the concept development stage by providing fully integrated topology optimization strategies:

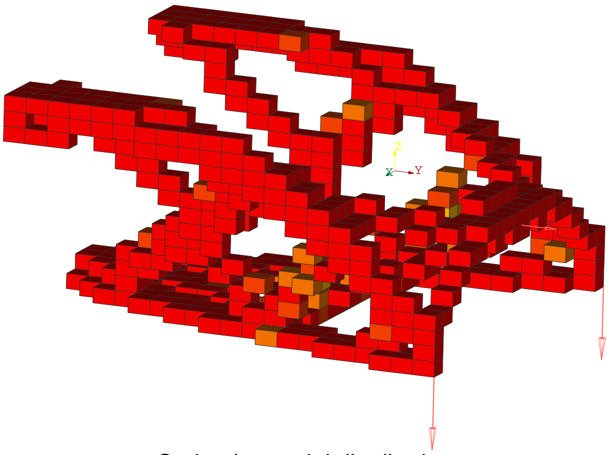
- Design space specification with variable/fixed parts
- Provide boundary conditions
- Provide loads
- Target definition with remaining volume



Design space with boundary conditions and loading

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

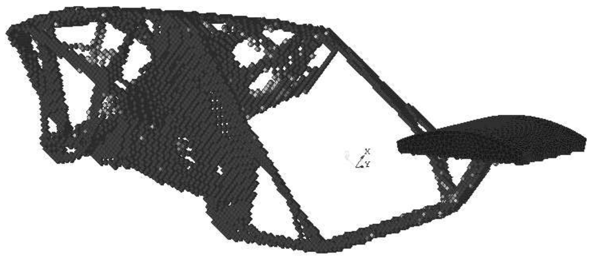
- **Fixed/free Design elements**
  - filling ratio per design element
  - design variable limits
  - design variable modification limits
- **Design objective**
  - compliance, eigenfrequency (mode range)
- **Multimodeling**
  - several load cases simultaneously with different superposition options
  - different design variants



Optimal material distribution

A number of analysis options are available for the optimization like linear statics, **contact analysis**, and dynamic mode analysis. The optimization itself is performed using one of the following algorithms:

- Method of Moving Asymptotes (MMA) with dual or optimality criteria
- Polynomial approach with dual or optimality criteria
- local or global approximation
- Global Convex Approximation (GCA) for eigenfrequencies and a combination of static and dynamic constraints.



Layout-Optimization of a gas pedal (iViP project study by courtesy of Robert Bosch GmbH)

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.

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 variables with an assigned distribution function.
- Definition of limit state functions (or failure functions) related to result quantities of a structural analysis.
- Calculation of the probability of failure for each limit state function.

The following quantities can be used as **basic variables**:

- Design parameters (like geometrical data or coordinates)
- Load factors
- Material parameters
- Parameters of the limit state functions
- Parameters of other basic variables

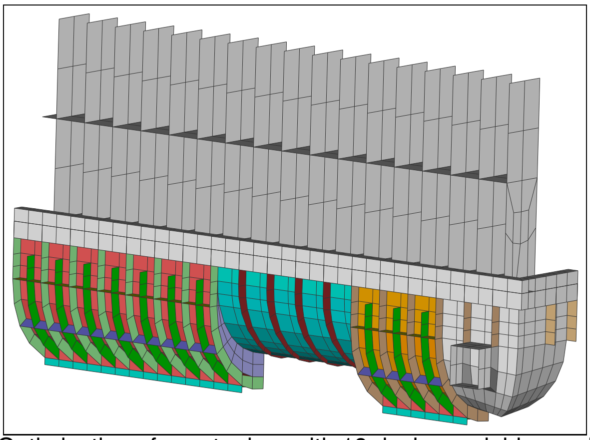
More than 20 different types of distribution functions are available to describe the basic uncertain variables.

The stochastic analysis performs an assessment of the failure parameters of a statically loaded structure. 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



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

Determination of a 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 optimization and reliability analysis. For this combination two different approaches are available:

- **Two step approach**
  - Basic design
    - FE-Analysis (e.g. static analysis)
    - Reliability analysis
  - Optimization taking into account reliability

  - * Optimization
  - * Reliability analysis of optimized design
  - * If not sufficient:
    - · Modify the design model according to reliability results
    - · Repeat the optimization and the reliability loop
- **One step approach**
  - – Combined optimization and reliability analysis
  - – Reliability as design constraint in optimization

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

- Design Variables
  - – define the design state of the structure
  - – may be modified by the optimizer
  - – may be assigned to one of the following types:
    - * Deterministic design variable
    - * Deterministic mean value of a stochastic design variable
- Basic Variables
  - – Basic variables define the stochastic properties of the problem
  - – Following types of basic variables are possible:
    - * Stochastic properties of the structure
    - * Stochastic design variable with deterministic mean value
    - * Load factors
    - * Limit state function parameter
    - * Parameter of another basic variable

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

- Design state
- Limit state

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

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

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

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

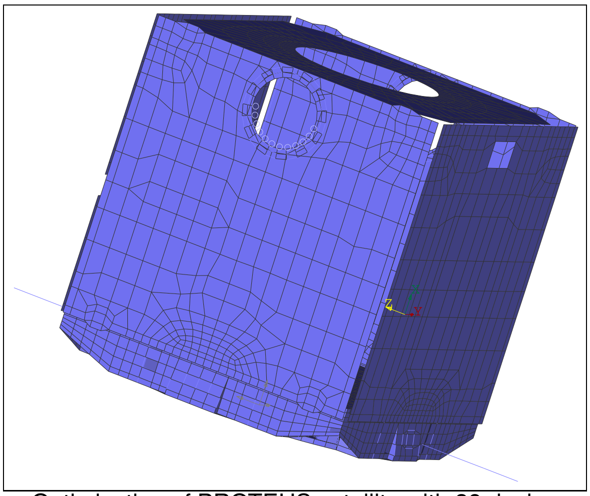
The limit state describes for a given design state the corresponding failure state. It is given by:

- Actual values of design variables
- Actual values of basic variables

The limit state function is evaluated for the limit state. The design constraints are meaningless for the limit state, e.g. the limit values for the limit state function and the limit values for the design constraints are different.

The combined analysis gives the following results:

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



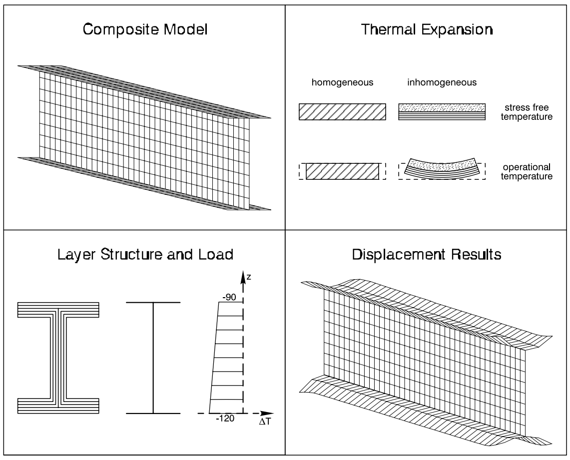
Optimization of PROTEUS satellite with 28 design variables and 30 stoachstic basic variables

		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.

PERMAS-LA – Laminate Analysis

The laminate analysis serves for the modelling 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.



Laminate analysis of a girder

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.

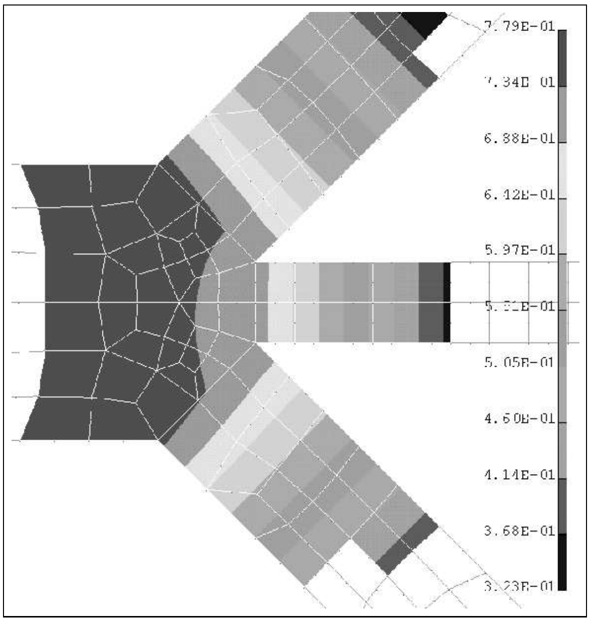
PERMAS-EMS – Electro- and Magneto-Statics

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 13).

Heat induced by an electrical field can be used for a subsequent thermal analysis (see page 29). From that thermal stresses can be derived performing a subsequent static analysis (see page 21).



Scalar potential in an electric junction

Forces induced by a magnetic field can be used in a subsequent static analysis (see page 21).

PERMAS-EMD – Electrodynamics

A solution of Maxwell's equations is available for different 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 25).

FELIX – The PERMAS Model Editor

FELIX is the model editor of PERMAS and comprises the PERMAS-Modules FEPRE and FEPOST. FELIX provides graphical support for specific

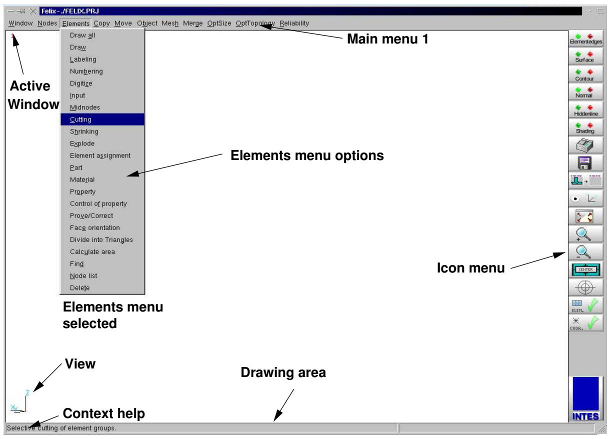
PERMAS input. In addition, standard UCI files can be generated.

FELIX uses the standard PERMAS interfaces for data input and output, which are compatible with all other PERMAS installations. So, FELIX may be used as graphical complement of an existing PERMAS environment.

PERMAS-FEPRE – FELIX Preprocessor

This module is the preprocessor part of the PERMAS model editor FELIX. The PERMAS interfaces to MEDINA, I-DEAS, NASTRAN, and PATRAN are optionally available and allow the direct input of such models.
Special emphasis has been put on the following modeling features to support some of the PERMAS modules:

- **Design Optmization**:
  Supports the preparation of an optimization model for sizing problems.
- **Layout Optmization**:
  Supports the preparation of a topology optimization model.
- **Reliability**:
  Supports the preparation of a model for a subsequent reliability analysis.
- **Dynamics**:
  Supports the preparation of excitation functions in time and frequency domain.



FELIX Bildschirmeinteilung

The user is guided by a sequence of dialogs in order to have an intuitive guidance to complete all necessary definitions.

There are many verification functions to check the extended models graphically.

PERMAS-FEPOST – FELIX Postprocessor

This module is the postprocessor part of the PERMAS model editor FELIX. It comprises a number of postprocessing features like:

- Visualization of deformations as deformed shape or as animated deformation.
- All other results can be viewed, if they are of scalar or vector type.
- For all xy data generated by PERMAS analysis tools, a diagram can be generated.

A special feature is the postprocessing of welding forces and stresses, where the results at spotwelds can be visualized over whole flanges. An exploded model is used for the representation of welding forces and stress curves along the flanges. In addition, a list of critical is issued (also in Excel format).

All pictures on screen can be issued on PNG or Postscript files for further use in reports or slide shows.

PERMAS-MEDI – MEDINA Door

This interface directly reads the model from the MEDINA data bus (.bif) and writes the results back to the data bus (.bof).

All MEDINA elements and almost all PERMAS MPCs are translated (see page 12). Beside Components, different Situations with constraint and load variants may be specified within MEDINA (see page 10).

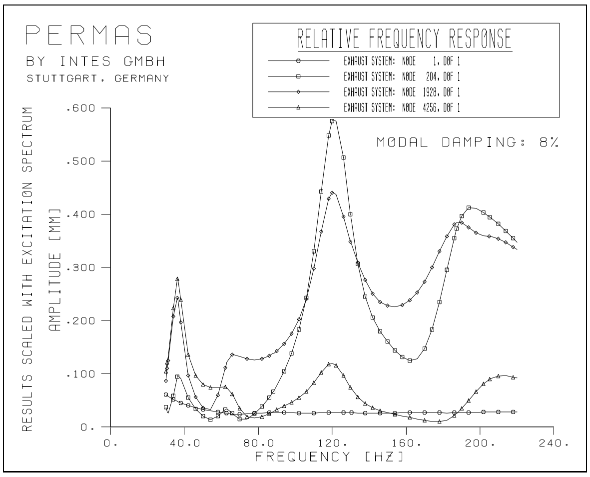
The part handling by incompatible line/surface coupling is supported.

The interface supports the following analysis types:

- linear and nonlinear statics
- contact analysis
- dynamic mode analysis
- thermal analysis

- fluid-structure acoustics (basic modelling and postprocessing)
- electromagnetics (basic modelling and postprocessing)

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



Exhaust System, Frequency Analysis

PERMAS-PAT – PATRAN Door

This interface reads the PATRAN database with the model and writes the results onto PATRAN result files.

The modelling is made using custom-made PERMAS preferences. Own solver menus support standard PERMAS solutions from within PATRAN, which may be adapted by the user himself. The following model parts are translated:

- all common elements
- all kinds of loads incl. inertia loads
- all PERMAS kinematic boundary conditions
- sets
- specifications of substructures
- variant definitions

The interface supports the following analysis types:

- linear and nonlinear statics
- contact analysis
- dynamic mode analysis
- thermal analysis
- fluid-structure acoustics

Even for other types of degrees of freedom like electric potential the model may be prepared within PATRAN.

PERMAS-CAT – CATIA Door

The CATIA integration creates a PERMAS input file directly from within the CAD system:

- Fully integrated interface, the analysis can be done without leaving CATIA
- A PERMAS solver table defines elements, properties, loads and restraints
- specific functions for contact, automated spotweld connection, and variant analysis are provided
- Predefined images facilitate the postprocessing tasks
- sets
- specifications of substructures

The interface supports the following analysis types:

- linear and nonlinear statics
- contact analysis
- dynamic mode analysis
- thermal analysis
- weight analysis

Even for other types of degrees of freedom like pressure or electric potential the models can be prepared in CATIA.

The PERMAS command control can be made within CATIA. All necessary menus are available and mays be adapted by the user to his needs. PERMAS jobs may be sumbitted either as interactive runs or as local or remote batch runs. So, no exit of CATIA is necessary to perform a PERMAS run.

PERMAS-ID – I-DEAS Door

This interface reads the model from an I-DEAS Universal File:

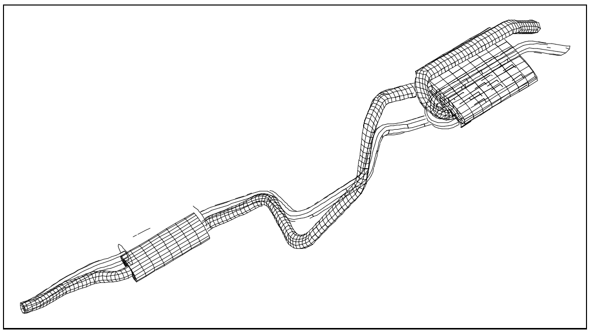
- all common element types
- all kind of loading incl. inertia loads
- many linear kinematic constraints like 'rigid element' and 'coupled dofs'
- sets
- specifications of substructures
- variant definitions

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.



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 correction modes.

In addition, the export of statically or dynamically condensed models to ADAMS is possible.

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 correction modes.

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

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.
- Output of stiffness/mass/etc. on top component level.
- For visualization in SIMPACK the complete model of the uncondensed structure can be exported.

PERMAS-MAT – MATLAB Interface

A post-processing interface for the export of model topology and matrices.

An import of control mechanisms from MATLAB in a dynamic calculation may be performed by using the PERMAS user functions (in C or Fortran).

PERMAS-NAS – NASTRAN Door

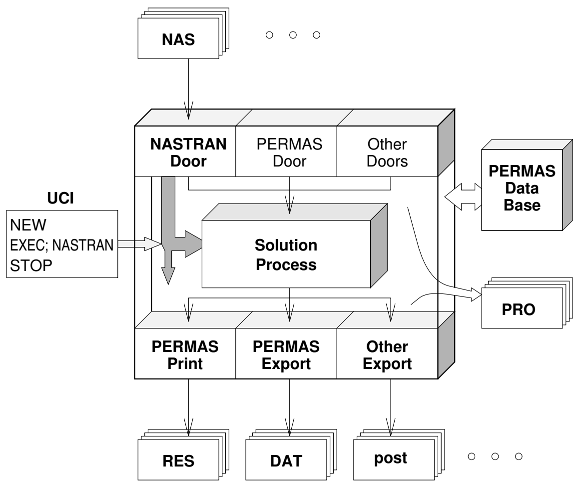
The NASTRAN Door reads and checks model files compatible to NASTRAN and translates these input files directly into internal PERMAS data structures. The **main capabilities** of the NASTRAN-Door are:

- 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.



UCI trigger for NASTRAN-Task

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.

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.

Supported Hardware Platforms

architecture	operating system
COMPAQ Alpha	TRU64
HP PA-RISC	HP-UX
IBM RS6000, SP	AIX
SGI	IRIX
SUN	Solaris
PC	Windows NT/XP
	LINUX

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 four months 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.

Usually once a year 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.

Additional Tools

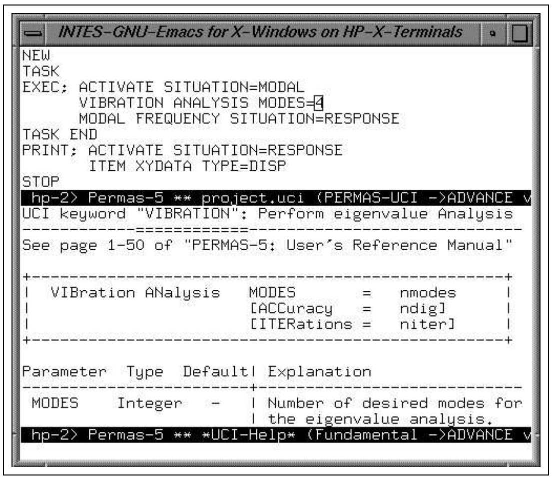
To support the application of PERMAS, INTES offers some additional tools within the INTES Application Environment (IAE).

- 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 INTES-GNATS for the reporting of application problems by email.

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:

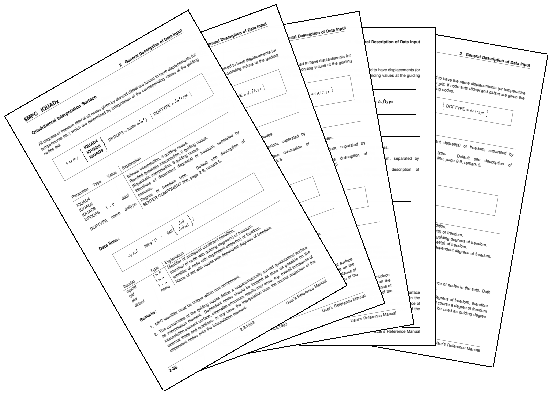
- Identical edit environment on almost any of the hardware platforms supported by PERMAS.
- Utilization of function keys and numeric keypad instead of control- and escape-sequences.
- Context-sensitive help functions and on-line access to all PERMAS documents.



Context-Help for UCI-Files inside Editor

The IAE is available for all PERMAS users under very affordable conditions. In addition, INTES offers a configuration service and an adaptation to the environment at the user's site as well as training.

Documentation



In parallel to the development of PERMAS the documentation is currently updated:

- Apart from the basic documentation
  - *PERMAS Users Reference Manual*
  - *PERMAS-FELIX Users Manual*
  - *PERMAS Examples Manual*
  - *PERMAS Programmers Manual*

  the following documents are provided:
  - *CATIA Door Manual*,
  - *I-DEAS Door Manual*,
  - *MEDINA Door Manual*.
  - *NASTRAN Door Manual*,
  - *PATRAN Door Manual*.
- Clearly arranged layout, complete index and cross references using page numbers (even among distinct documents) are a matter-of-course.

For online documentation and help, all manuals are available as PDF files with cross references.

Training

INTES provides training courses for all application fields of PERMAS. Based on a three-day introductory course on linear statics several one- or two-day courses are offered on other topics like contact, heat transfer, dynamics, optimization, etc.

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**

