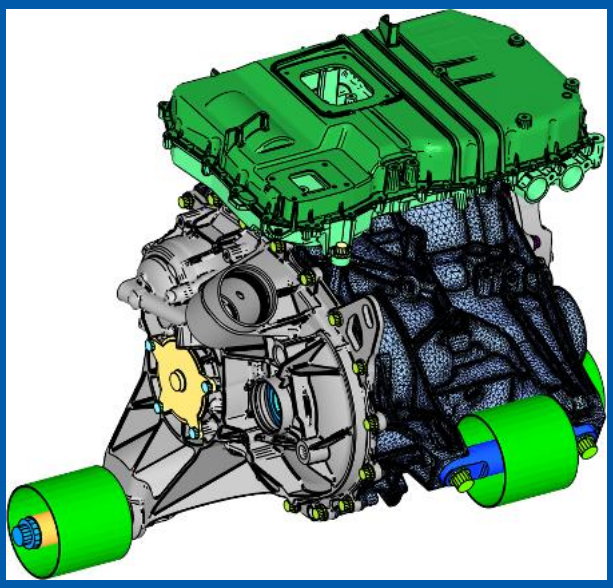


PERMAS

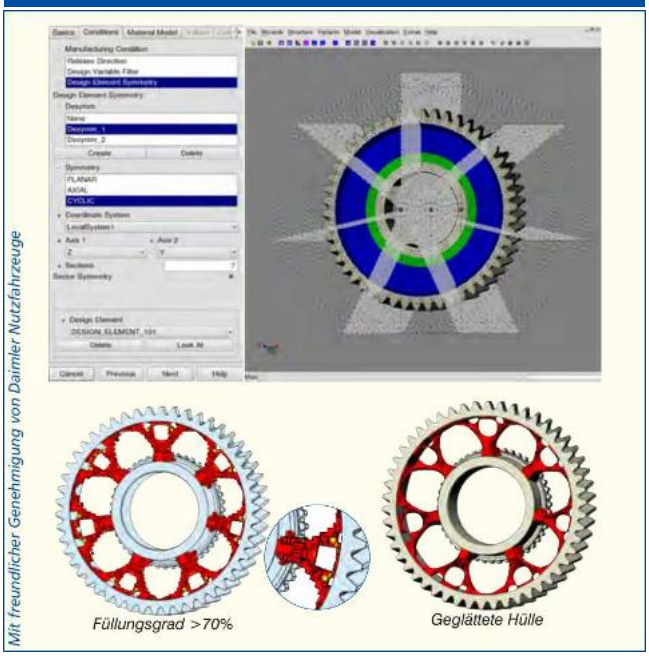
E

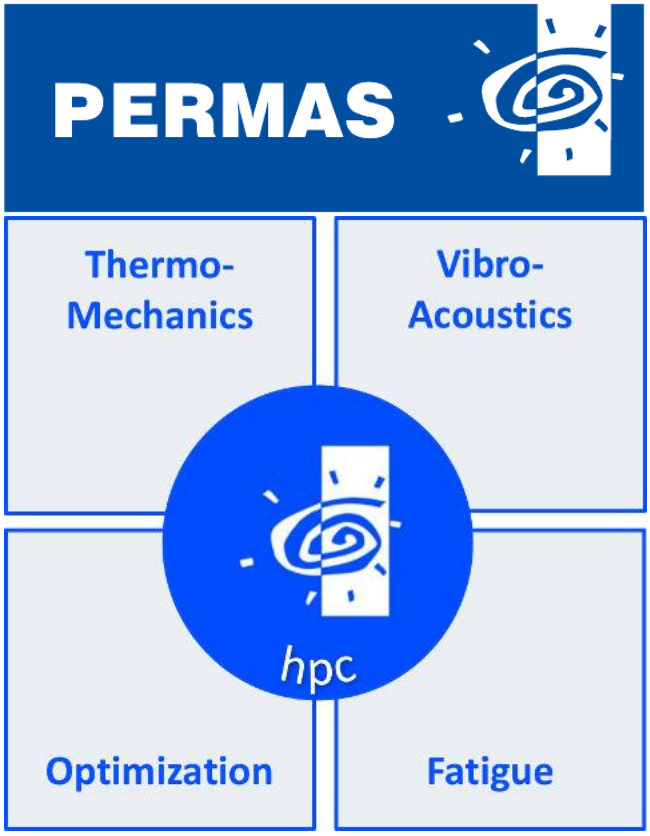
Version 21

INTES is a privately held and independent enterprise for Finite Element Technology (FE Technology) with offices in Stuttgart, Paris and Tokyo.

For all of its customers, INTES is a competent partner in all aspects of FE Technology. Above all, satisfaction of the customers with all the software and services is of prime importance to the company.

INTES offers the FE software PERMAS (with VisPER as graphical user interface), related software developments, training courses, consulting and simulation services.





PERMAS is making realistic simulations practical.

PERMAS provides extremely fast and accurate solutions for realistic simulations of large models and complex situations in time.

PERMAS effectively supports performance based design decisions without the need to sacrifice accuracy.

Customer value with PERMAS:

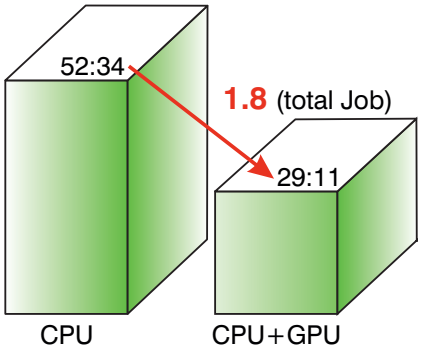
- To reach improved understanding of product performance and better product designs.
- More design iterations for more accurate models for the same simulation costs.
- Better product designs through effective and rapid optimization of complex situations.
- Reduced effort, cost and time to achieve reliable performance driven designs.
- Reduced risk and uncertainty through robust design and reliability analysis.

Profile/ Strengths

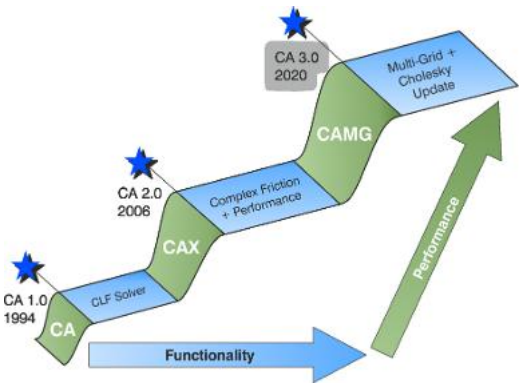
INTES

High performance computing
for large and very large models, with fast solvers, parallel (direct) I/O and parallelization.

In combination with GPU acceleration.



2*Intel 6146(40core)
Nvidia Tesla V100
GPU benefit for contact analysis

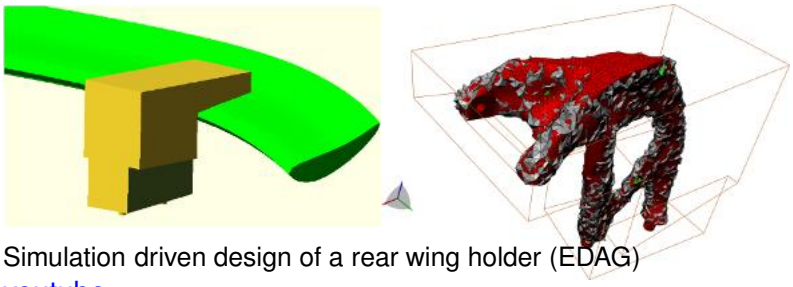


Development of Contact solvers

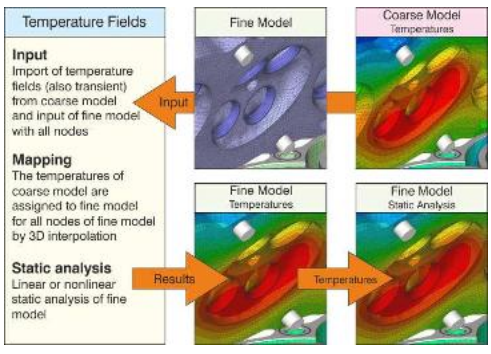
Special high performance algorithms
for contact analysis, for eigenvalues with MLDR method, for fluid-structure coupling, harmonic balance method.

Integrated optimization
for topology, bead, (free-) shape, sizing, laminates, robust design, sampling . . .

Automated workflows for generative design

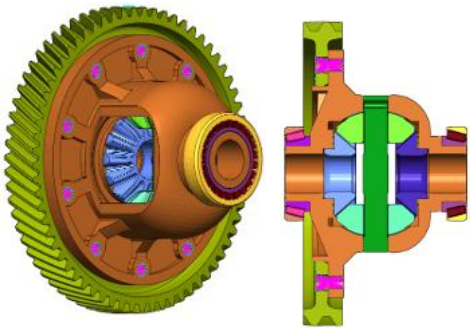


Simulation driven design of a rear wing holder (EDAG)
youtube



Mapping of temperature fields

Productivity tools
for automatic part coupling and incompatible meshes, with substructuring and submodelling.



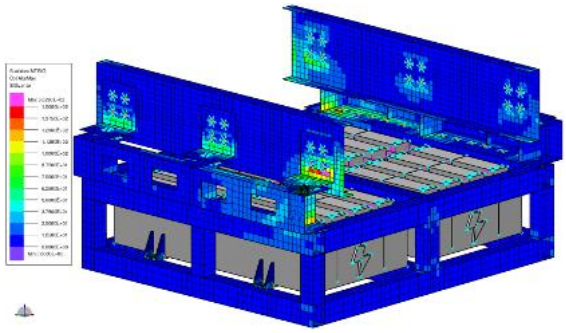
Differential model with multiple contact partners

Contact in Dynamics

- Same contact description as in statics
- For modal/direct timehistory and HBM solutions
- Also for fluid-structure (FS) systems and brake squeal

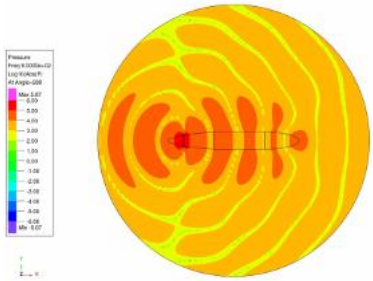
CONTACT AUTOSEARCH

- Automatic detection and definition of contact partners
- Handling of contact normals, kinks and MPCs
- Automatic handling of shell normals for contact and shell thickness
- Manual contact definitions preceede automatic detection

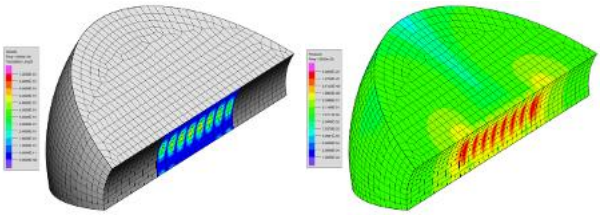


Battery pack with contact in dynamics, stresses

Direct methods for FS acoustics



Direct fscoupled frequency, pressure distribution at $900\,Hz$

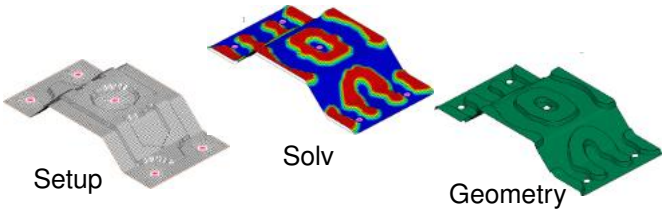


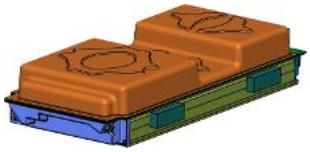
FS random response, velocity (left) and pressure distribution (right) at $1800.0\,Hz$

Bead design in PERMAS/VisPER

- Bead Wizard for bead optimization setup
- Specific optimization Solver
- Bead postprocessing by Bead Design Wizard

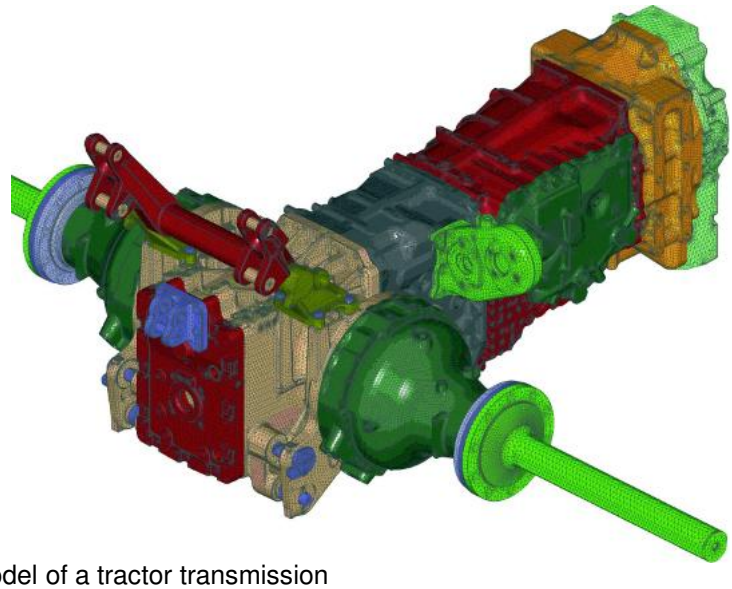
Complete analysis workflow provided





Beads, battery housing

INTES



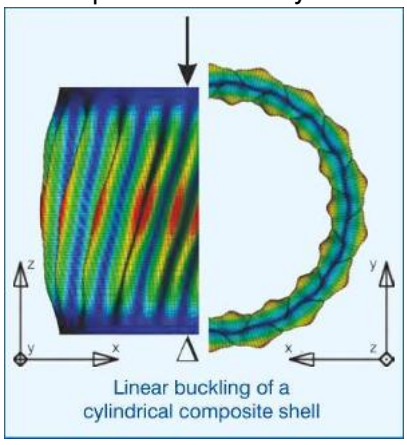
Model of a tractor transmission
ZF Friedrichshafen AG, Friedrichshafen, Germany.

Contact is a nonlinear boundary condition, which can be coupled with other linear conditions as well as with nonlinearities. At the same time, the contact can be calculated between different bodies, between a body and ground, but also when self contact occurs.

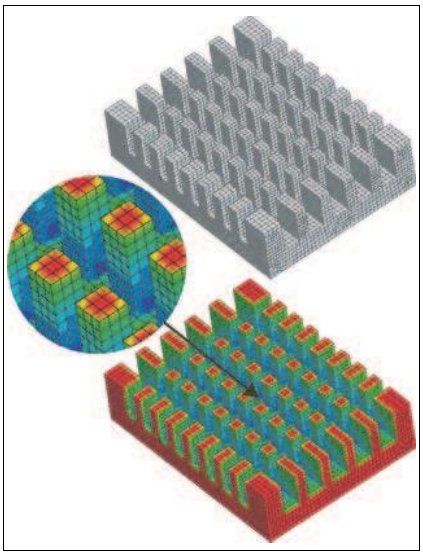
The contact analysis can be performed with or without friction, where isotropic or orthotropic sticking and sliding friction is applied following Coulomb's law of friction.

Thermomechanics comprises functionalities for the determination of temperature fields, contact analysis, linear and nonlinear static analysis as well as buckling.

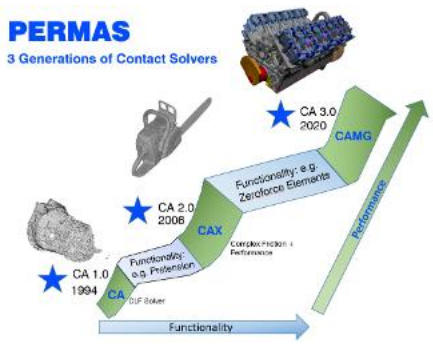
The analysis of linear and nonlinear, steady-state and transient **heat transfer analysis** is supported. Heat conductivity, heat capacity, and heat convection with radiation can be used accordingly. Previously calculated temperature fields can be directly used in subsequent static analysis.



The linear **buckling** analysis is a classical method to study shell and beam structures. It can be used also after a nonlinear static analysis in order to detect additional bifurcation points.

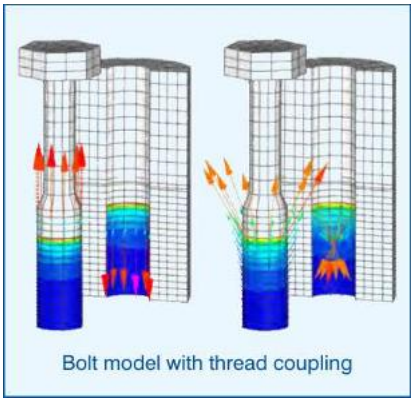


Analysis of cooling element with radiation

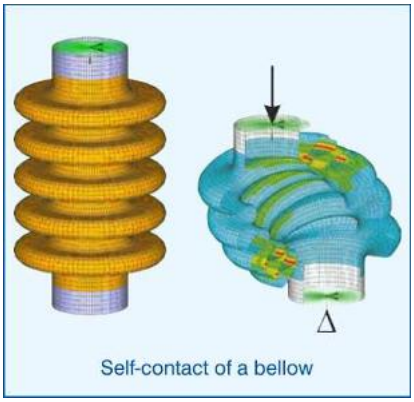


The CLF (Condensed Lagrange Flexibility) method provides high performance contact analysis. The integrated ContactAnalysisMultiGrid (CAMG) solver reduces drastically run times for large contact models (>10,000 contacts).

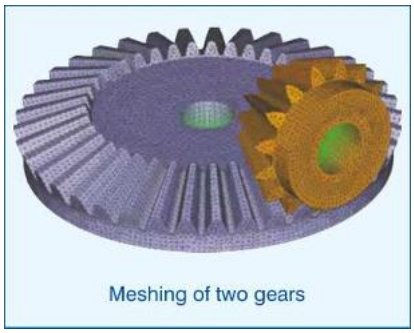
Contact can be calculated between compatibly and incompatibly meshed parts. In both cases, the resulting contact pressure is available.


Bolt model with thread coupling

Bolt pretension is also made by contact. The coupling of screw and mating thread takes place at the thread area, while the bolt and the bolt hole remain perfectly cylindrical. By doing so, the radial widening of the bolt hole and the twist of the bolt can be modeled by a flank angle and the pitch of the thread without modeling the thread itself.


Self-contact of a bellow

By **updating the contact geometry**, large sliding displacements can be calculated. Together with geometrical nonlinear effects, large rotations are carried out.


Meshing of two gears

Nonlinear gaskets are modeled by special gasket elements, which are fully handled by contact analysis.

Purely **force-guided contacts** are also possible with larger initial gap widths.

An **obtained contact status** can be re-used in a subsequent variant analysis. This has the potential to reduce the run time of the variant analysis significantly.

To perform a **subsequent dynamic or heat transfer analysis**, the achieved contact status can be frozen, where a pressure dependent coupling is supported.

Nonlinear material behaviour can use different constitutive models: nonlinear elastic material, elastic-plastic material (von Mises, Tresca, Drucker-Prager, Mohr-Coulomb, cast iron), simple visco-plastic material, creep. At the same time, the material can be temperature-dependent. In case of plasticity isotropic or kinematic hardening can be taken into account (the latter also as nonlinear kinematic hardening). A user-defined material is possible.

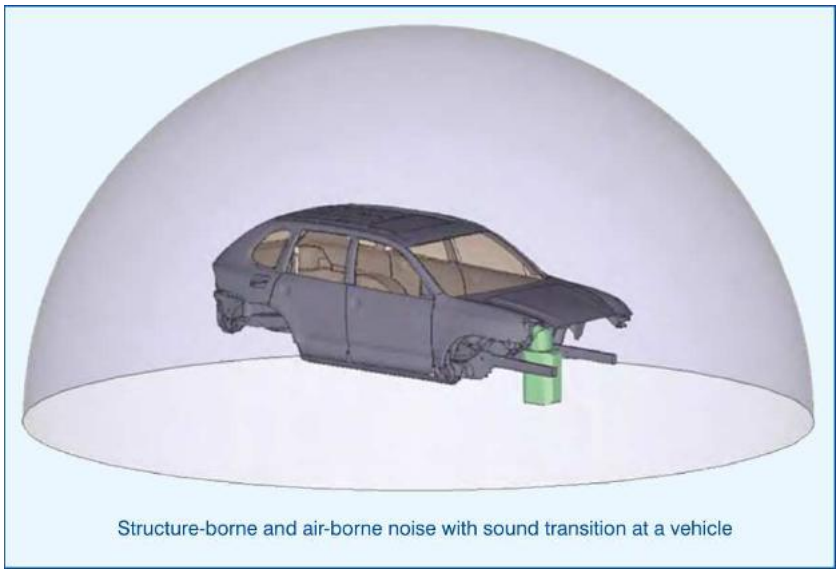
Beside isotropic plasticity models, transverse isotropic plasticity for short fiber reinforced plastics is also supported.

For large models and local nonlinear effects, substructuring and submodeling can be used in nonlinear analysis.

Geometric nonlinearities can be combined with other linear or nonlinear effects. Also nonlinear buckling of shell structures is supported with linear or nonlinear material behaviour. It can be amended by linear buckling in each load steo in order to detect bifurcations.

Large strain analysis for hyperelastic materials is supported, where the same element types are used as for other linear or nonlinear analyses.

INTES

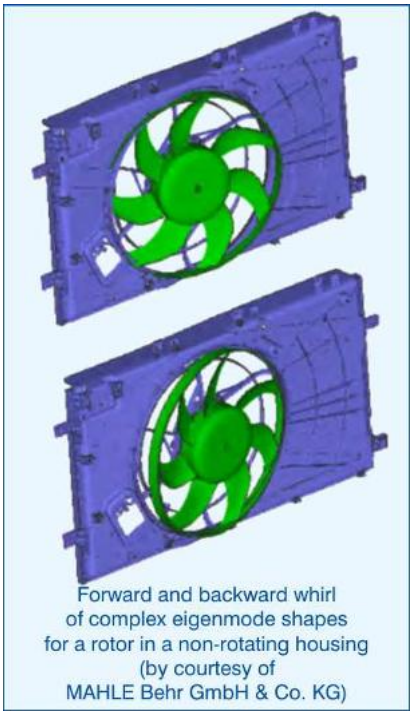


Structure-borne and air-borne noise with sound transition at a vehicle

In a real **eigenvalue analysis** of a structure the elastic stiffness can be extended by geometric stiffness and pressure stiffness, if needed. In rotor dynamics, centrifugal stiffness or convective stiffness can be taken into account as the case may be calculated in a co-rotating or inertial reference system.

For **structural dynamics** functionalities are provided in modal space by real and complex eigenvalue analyses, dynamic condensation, and response analysis in frequency and time domain (also steady-state). Additionally, spectral and random response analysis are available. Direct solutions for response analysis in frequency and time domain are available, too.

For both a **Fluid** only and the coupled **Fluid-Structure-Acoustics**, real eigenvalue analysis, dynamic condensation, and response analysis in frequency and time domain as well as random response are supported in modal space. A direct solution for the response analysis in frequency domain is also available.



Forward and backward whirl of complex eigenmode shapes for a rotor in a non-rotating housing (by courtesy of MAHLE Behr GmbH & Co. KG)

The modeling of a fluid is using volume elements and the connection to the structure is using coupling elements. For the boundaries of surrounding fluids, radiation boundary condition elements and semi-infinite elements are available.

The real eigenvalue analysis in coupled fluid-structure acoustics results in eigenmodes, which consist of displacements for the structure and a corresponding pressure field for the fluid.

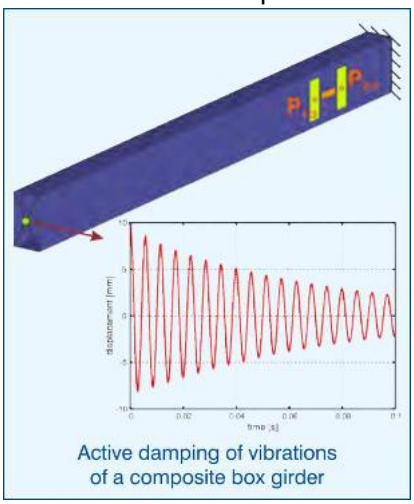
For very large models with a high number of modes to be calculated, a special eigenvalue solver using the **MLDR method** (Multi-Level Dynamic Reduction) provides an extraordinary efficient procedure.

The **complex eigenvalue analysis** is based on real eigenvalues and mode shapes. For rotating structures, a Campbell diagram for an arbitrary number of rotating speeds can be generated in one single run.

Many options for the modeling of **damping** are available like material damping, proportional damping, viscous damping elements, modal viscous damping, modal structural damping, and direct input of damping matrices (also in modal space).

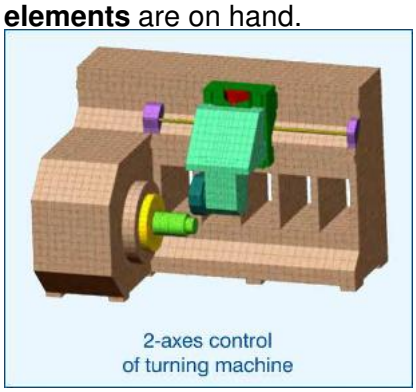
In frequency domain, structural damping can be described as a function of frequency. A frequency-dependent viscous damping is supported by a special element.

The modeling of **active damping** is possible using control elements, which combine dynamic vibrations (detected at a sensor) with a driving force (applied at the actuator) using classical linear or nonlinear control parameters.



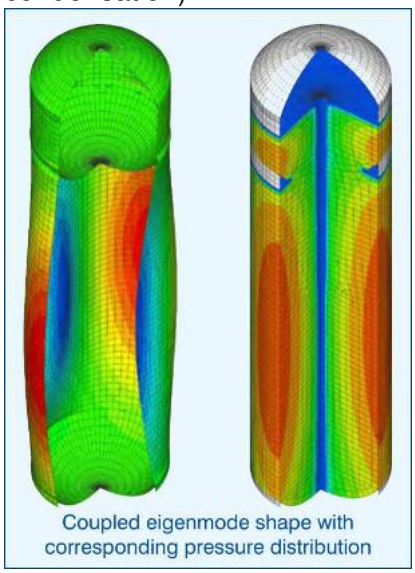
Active damping of vibrations of a composite box girder

For complex position and velocity control (like for machine tools) additional **control elements** are on hand.



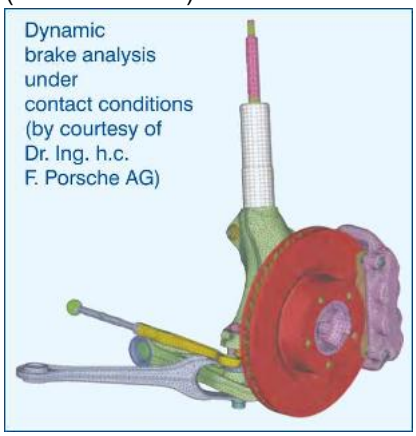
2-axes control of turning machine

In case of coupled **fluid-structure acoustics**, a structure with enclosed fluid can be condensated in a way that no pressure degrees of freedom are present in the condensated system (so-called dry condensation).



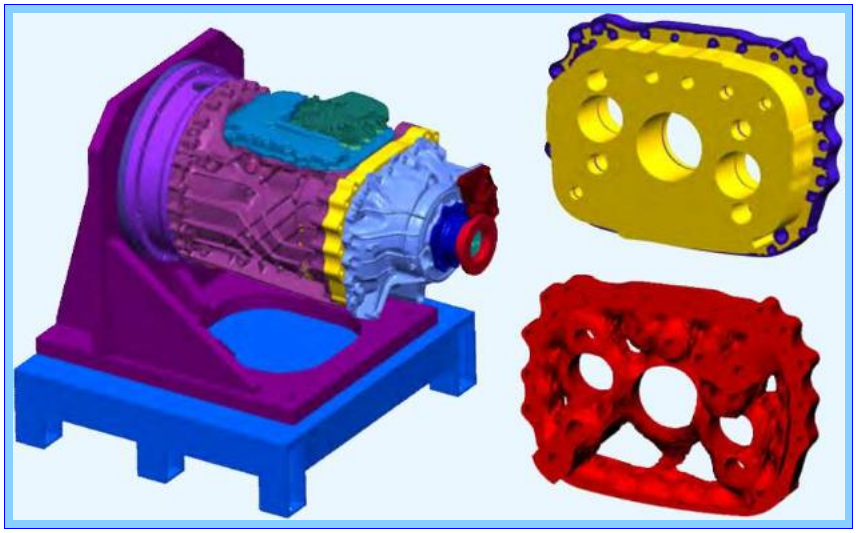
Coupled eigenmode shape with corresponding pressure distribution

For **dynamic condensation**, an extended Craig-Bampton method can be applied (i.e. MBCB Mixed Boundary Craig-Bampton), which allows the use of vibration modes under different boundary conditions (also free-free).



In **dynamic brake analysis** the contact status between brake pad and disc can be frozen for a subsequent real and complex eigenvalue analysis to identify instabilities indicating brake squeal. An additional parameter study using the integrated sampling method will give important hints on the brake's potential for improvement.

Optimal Design

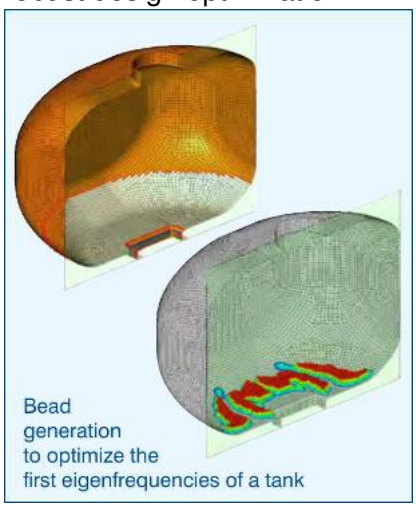


Topology optimization of a bearing support (by courtesy of ZF Friedrichshafen AG, Friedrichshafen)

A parametric **shape optimization** is performed by several methods like position optimization, bead generation, or shape optimization using shape basis vectors.

The **position optimization** can be characterized by a change of the position of two or more parts to each other or by a change of the position of boundary conditions in order to fulfil certain conditions like minimum deformations.

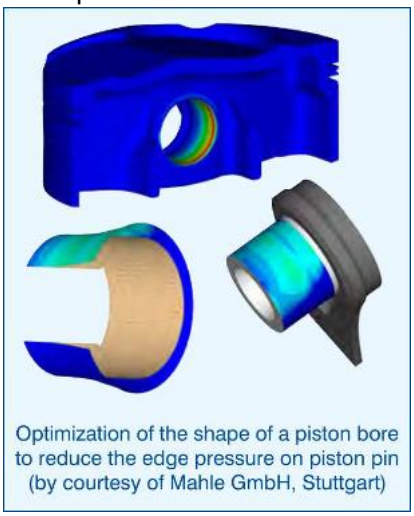
All optimization methods in PERMAS are fully integrated. Besides sizing optimization, topology optimization, and shape optimization are available. In addition, reliability analysis is available to handle uncertain model parameters. An optimization under reliability constraints is supported as robust design optimization.



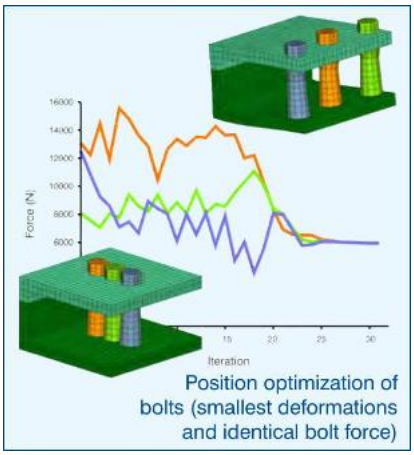
To start optimization a design space exploration using **sampling** is a good first step before starting optimization. In this way different parameter settings can be used to get more information on the effect of certain parameters and the sensitivity of result quantities due to parameter changes.

Dependent on the optimization method, different analysis types can be used in optimization loops, like static analysis, contact analysis, linear buckling analysis, nonlinear material behavior, real and complex eigenvalue analysis, modal frequency response analysis, steady-state heat transfer analysis. Limits of design variables or numerous result quantities can be used as objective or side constraint.

The **bead generation** is used with shell structures in order to achieve certain static or dynamic properties of the optimized parts by a suitable bead pattern.

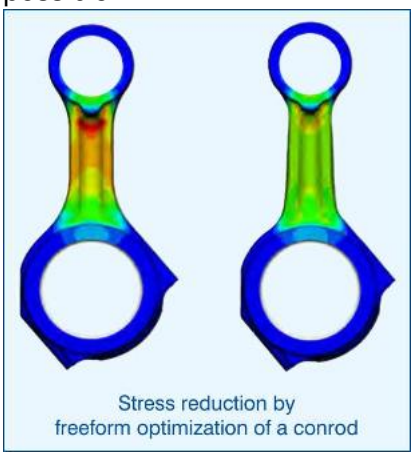


Optimization of the shape of a piston bore to reduce the edge pressure on piston pin (by courtesy of Mahle GmbH, Stuttgart)



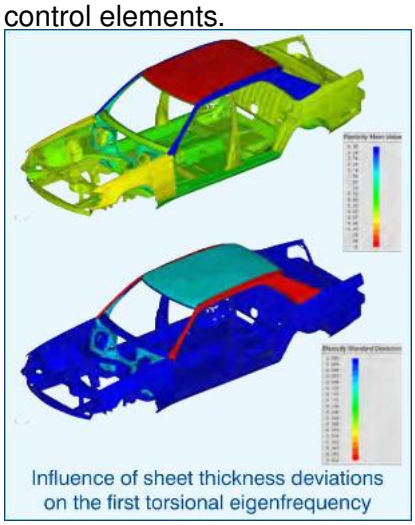
Position optimization of bolts (smallest deformations and identical bolt force)

The non-parametric **freeform optimization** is mainly used to homogenize stress fields or to optimize the weight of parts under stress limits. To achieve that, material can be added or released at complex surfaces in normal direction. Additional constraints like displacmenets or release directions are also possible.

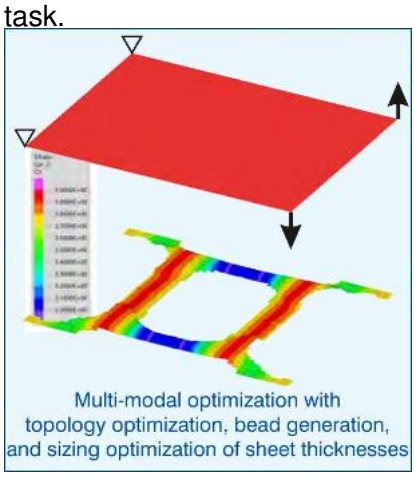

Stress reduction by freeform optimization of a conrod

The **reliability analysis** handles uncertain model parameters and their influence on the structural behavior. For a given failure mode, it calculates the probability of failure and its sensitivities with regard to the uncertain variables.

A **sizing optimization** uses other model parameters than node coordinates as design variables like shell thicknesses, beam cross sections, material parameters, property values of spring, mass, and damper elements or even parameters of control elements.


Influence of sheet thickness deviations on the first torsional eigenfrequency

Due to the unification of optimization methods like parametric shape optimization, sizing and topology optimization, all these methods can be used simultaneously in a **Multi-Modal Optimization** to jointly solve an optimization task.

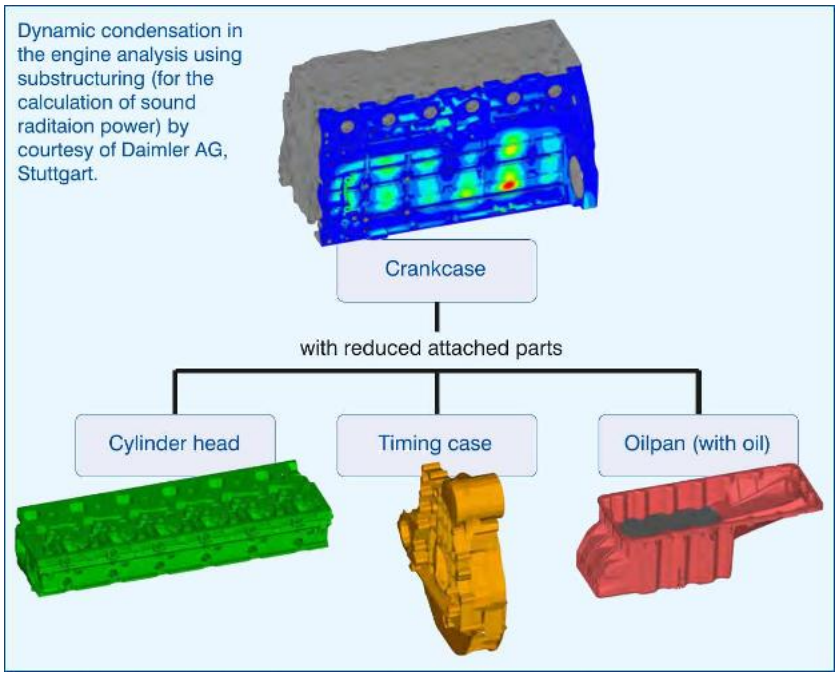

Multi-modal optimization with topology optimization, bead generation, and sizing optimization of sheet thicknesses

The **topology optimization** starts from a design space in order to find the optimal material distribution in this design space for a given design objective. Objective and side constraints can be defined in the design space or outside in the other parts of the structure. In addition, manufacturing constraints can be specified for symmetries, release directions, minimum and maximum member sizes, and overhang angles as 3D printing condition to influence the design. The element filling ratio is used as design variable, which directly influences stiffness and mass of each element. After convergence, the elements in the design space are either those with filling ratio near one, which represent the desired structural behaviour, or those with filling ratio near zero, which are not needed to achieve this behaviour. After an automatic surface smoothing of the remaining structure, it can be exported as mesh or geometry (STL).

Topology optimization may also be used with other quantities like sheet thicknesses (so-called **Free Sizing**).

Laminates can be optimized by free sizing for new design concepts and by sizing optimization of ply thickness and angle also with ply failure constraints.

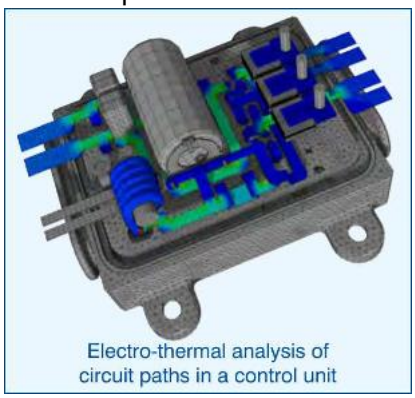
Other Functions



By **substructuring**, an FE model can be split into an arbitrary number of substructures (so-called components). The components can be assembled like single elements to get a complete structure (so-called configuration).

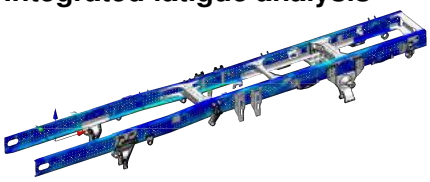
A configuration can be assembled by an arbitrary number of levels up to the top component (i.e. the root of the substructure tree). Each level may introduce new elements, loads, and boundary conditions.

Further analysis capabilities for electrodynamics, some special functions like laminate analysis, and an innovative new spot weld concept as well as the interfaces, which are directly supported by PERMAS to other software products.



Linear static and dynamic electrodynamic tasks can be solved with the corresponding modules. Generated heat according to the Joule effect or induced forces can be directly used in subsequent structural analysis.

Integrated fatigue analysis



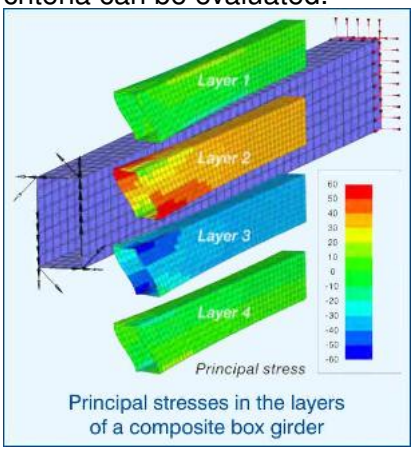
Courtesy of Daimler Truck, Stuttgart

PERMAS provides an easy-to-use and fully integrated high cycle fatigue lifetime analysis capability. By this, huge external data transfer can be avoided and HPC technology allows for very low elapsed times. The fatigue analysis may follow a (nonlinear) static analysis as well as a timehistory analysis (modal or direct) Multiple fatigue analyses in one run allow for parameter Studies.

For the reduction of components, **static and dynamic reduction** is available. By this way, matrix models can be generated, which represent the FE models of the reduced components and are suitable for model exchange between cooperation partners without exchanging the model details.

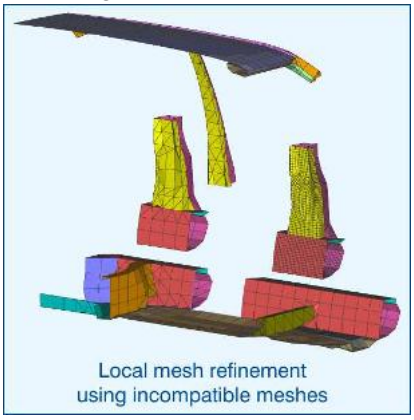
Other Functions

For modeling and analysis of laminated **composites**, triangular and quadrangular shell elements can be used. Their properties are described by the layer set-up or directly by ABD matrices. Ply failure criteria can be evaluated.



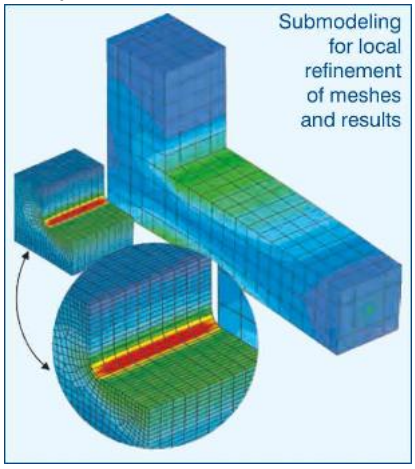
Principal stresses in the layers of a composite box girder

A special **spot weld element** is available, which essentially reduces the sensitivity of stress results due to the mesh size of the flanges.

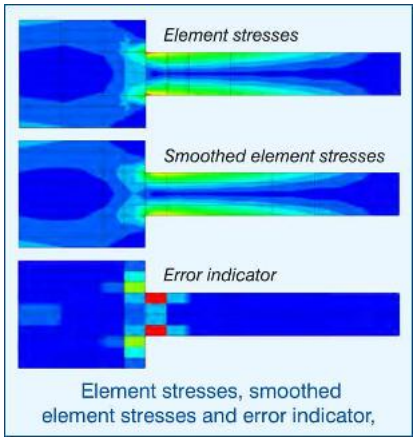


Local mesh refinement using incompatible meshes

Results, which are calculated with a coarse mesh, can be applied as boundary conditions for a finer meshed part of the structure (e.g. in order to calculate more precise stresses). This **submodeling** technique also enables nonlinear analysis of parts based on linear analysis of the complete structure.



By **automated part coupling** through incompatible meshing of parts, more flexible and faster modeling is enabled, because meshes need not to be made compatible by expensive mesh adaptations and replacing parts by other meshes become much easier.



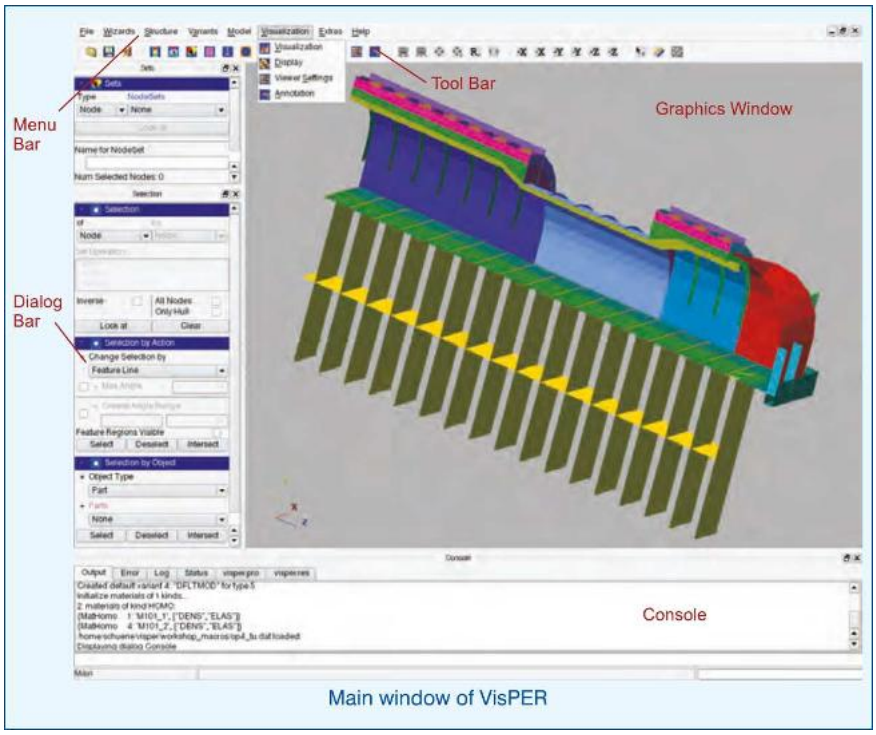
Element stresses, smoothed element stresses and error indicator,

A Smooth Patch Recovery (SPR) method is applied to get **SPR stresses** beside classical stress calculation. The difference between both stresses defines an (Absolute) **Error Indicator** (AEI), which can be used for mesh validation and further mesh refinement, if required. In addition, the stress gradient normal to the surface is calculated at the surface nodes.

VisPER	Adams MNF
MEDINA	AVL Excite
Patran	Altair MotionSolve
I-deas	VLAB Motion, Durability
Hypermesh, Hyperview	Simpack
Ansa	Simdrive 3D
Nastran BDF+OP2	MeshWorks
Simlab	VAO
NX	Femfat
Abaqus INP	Ensight
ADSTEFAN	modeFrontier
Magmasoft	Novus Nexus

Interfaces: blue: in PERMAS, black: external

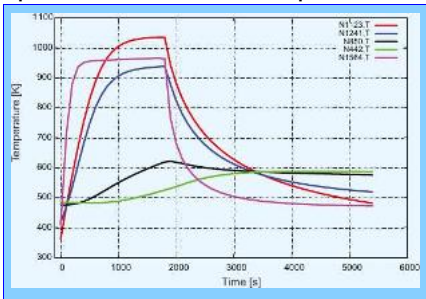
PERMAS is an open system and maintains numerous **interfaces** to other software products.



Main window of VisPER

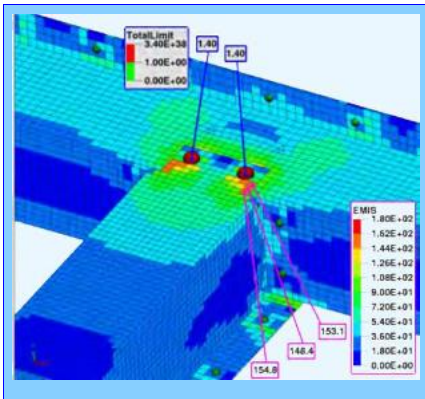
VisPER is the graphical pre- and post-processor of PERMAS. It comprises the pre-processing of PERMAS models (mainly on the basis of already existing meshes) and the post- processing of PERMAS results.

In post-processing, XY plots can be generated using VisPER or “PERMASgraph”, a specialized tool for XY plots.



Temperature distribution over time with PERMASgraph

For the evaluation of spot weld forces, a special method is available, which uses traffic light labeling to visualize critical and non-critical values of the normal and shear forces in spot welds. The stresses can be shown in addition for a full assessment of the stresses in a flange.



Evaluation of spot weld forces

Beside the specification of element properties, MPC conditions, boundary conditions, and loads, VisPER comprises specialized **wizards**, which guide the user through the respective modeling steps:

General:

- Assembly and part exchange
- Brake squeal analysis
- Design by simulation
- Pressfit
- Sampling

Contact:

- Contact modeling
- Bolt pretension

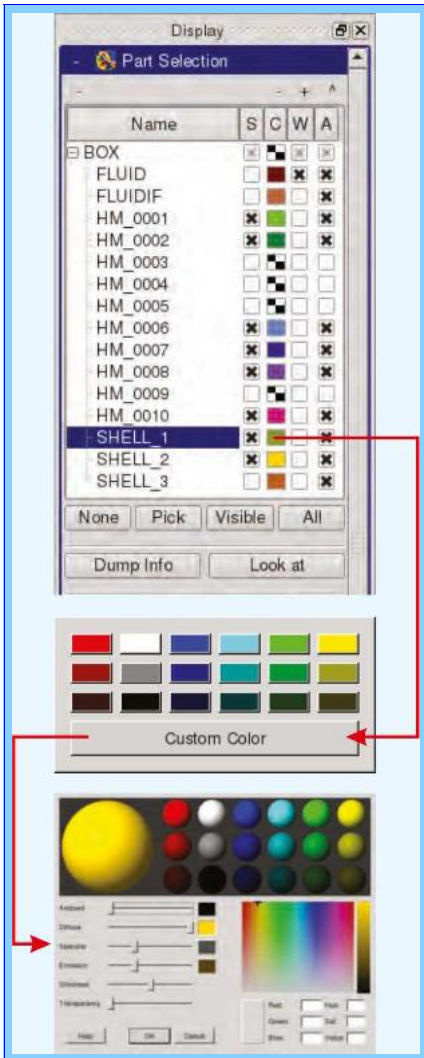
Optimization:

- Topology optimization
- Sizing optimization
- Shape optimization (with bead generation and position optimization)
- Freeform optimization

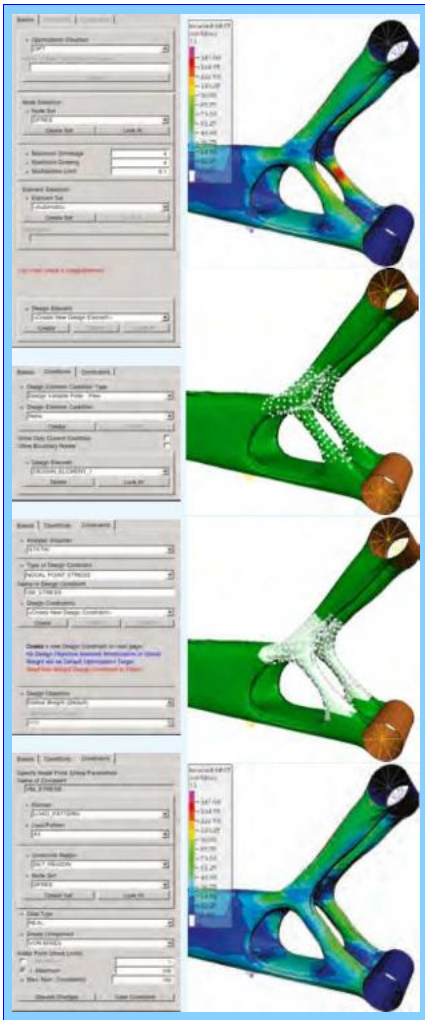
Others:

- Fluid-structure coupling
- Substructuring
- Rigid body mode decoupling (RBM assistant)

VisPER also performs the task of model verification by visualization and evaluation of numerous PERMAS verification results, like projection vectors and unconnected nodes in surface to surface MPC connections. In addition, a message dialog evaluates the PERMAS diagnostic messages and shows the affected model parts directly.



Graphical properties of parts and element sets (S=shown,C=colored,W=wireframe,A=active)

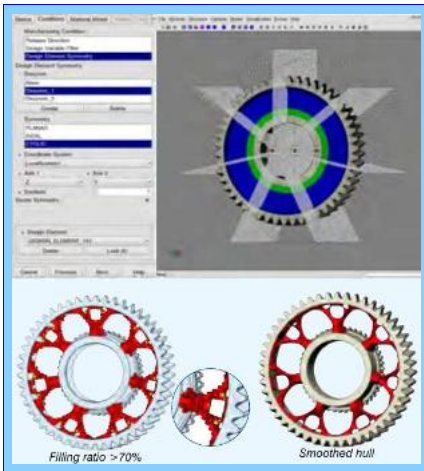


Pre-and post-processing for a freeform optimization

VisPER is based on the infrastructure of PERMAS, which provides most of the interfaces in both PERMAS and VisPER in the same way, where VisPER additionally gives the direct visual feedback about the read model to the user. In addition, results of PERMAS can be exported in other formats.

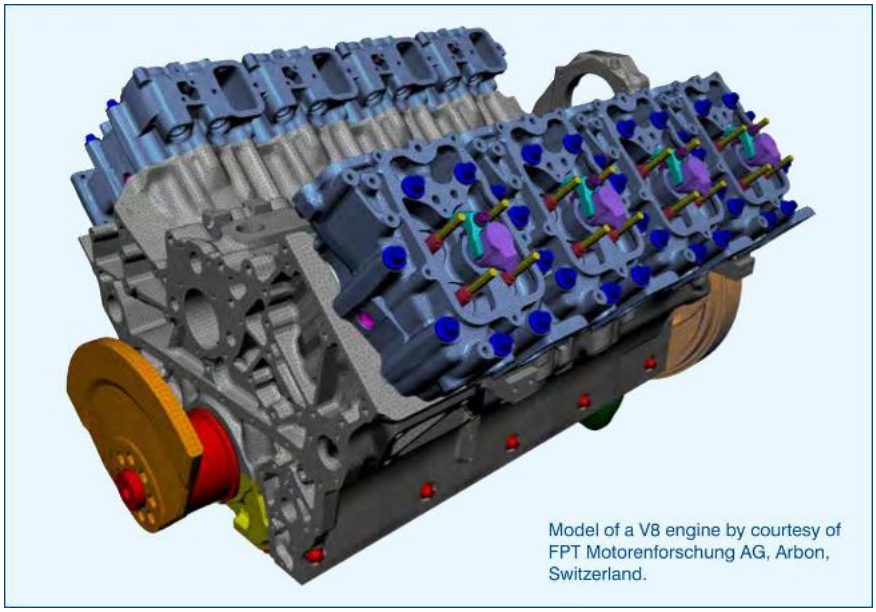
VisPER has a modern interactive graphical user interface for easy and intuitive application. A simple selection of the shown model parts, working with transparency and cutting planes, the simple generation of animations, and many other useful interactions are supporting the effective and efficient working with this tool.

Comprehensive options for adjustment provide an easy customization to the user's preferred working style. Self-defined abbreviations for standard functions accelerate many operations. Macros using the programming language Python can easily be generated by recording and used later.



Topology optimization of a gear wheel with cyclic symmetry and release directions

INTES



Model of a V8 engine by courtesy of FPT Motorenforschung AG, Arbon, Switzerland.

PERMAS-Modules:

Thermo-Mechanics:
MQA Basic module
LS Linear static analysis
CA+CAX Contact analysis
CAU Contact geometry update
CAMG Contact multigrid solver
NLS Nonlinear static analysis
NLSMAT Extended mat. laws
NLSA Hyperelastic material
BA Linear buckling analysis
HT Heat transfer
NLHT Nonlinear heat transfer
LIFE Basic fatigue analysis

Vibro-Acoustics:
DEV Dynamics (eigenvalues)
DEVXExtended mode analysis
MLDR Eigenmodes with MLDR
DRA Dynamics (response)
DRX Extended dynamics
FS Fluid-structure acoustics
NLD Nonlinear dynamics
HBM Harmonic balance method

Optimal Design:
OPT Design optimization
TOPO Layout optimization
AOS Advanced opt. solvers
RA Reliability analysis

Other Functions:
EMS Electro-/magneto-statics
EMD Electrodynamics
LA Laminate analysis
WLDS Weldspot model
AERO Aeroelasticity/Generalized inertia relief
XPU GPU-accelerator

Interfaces:
MEDI MEDINA-Door
PAT PATRAN-Door
ID I-DEAS-Door
AD ADAMS-Interface
EXCI EXCITE-Interface
SIM SimPack-Interface
SIM3D Simdrive 3D-Interface
HMS MotionSolve-Interface
H3D HYPERVIEW-Interface
VLAB Virtual.Lab-Interface
ADS ADSTEFAN-Interface
MAT MATLAB-Interface
NAS NASTRAN-Door
ABA ABAQUS-Door

VisPER-Modules:

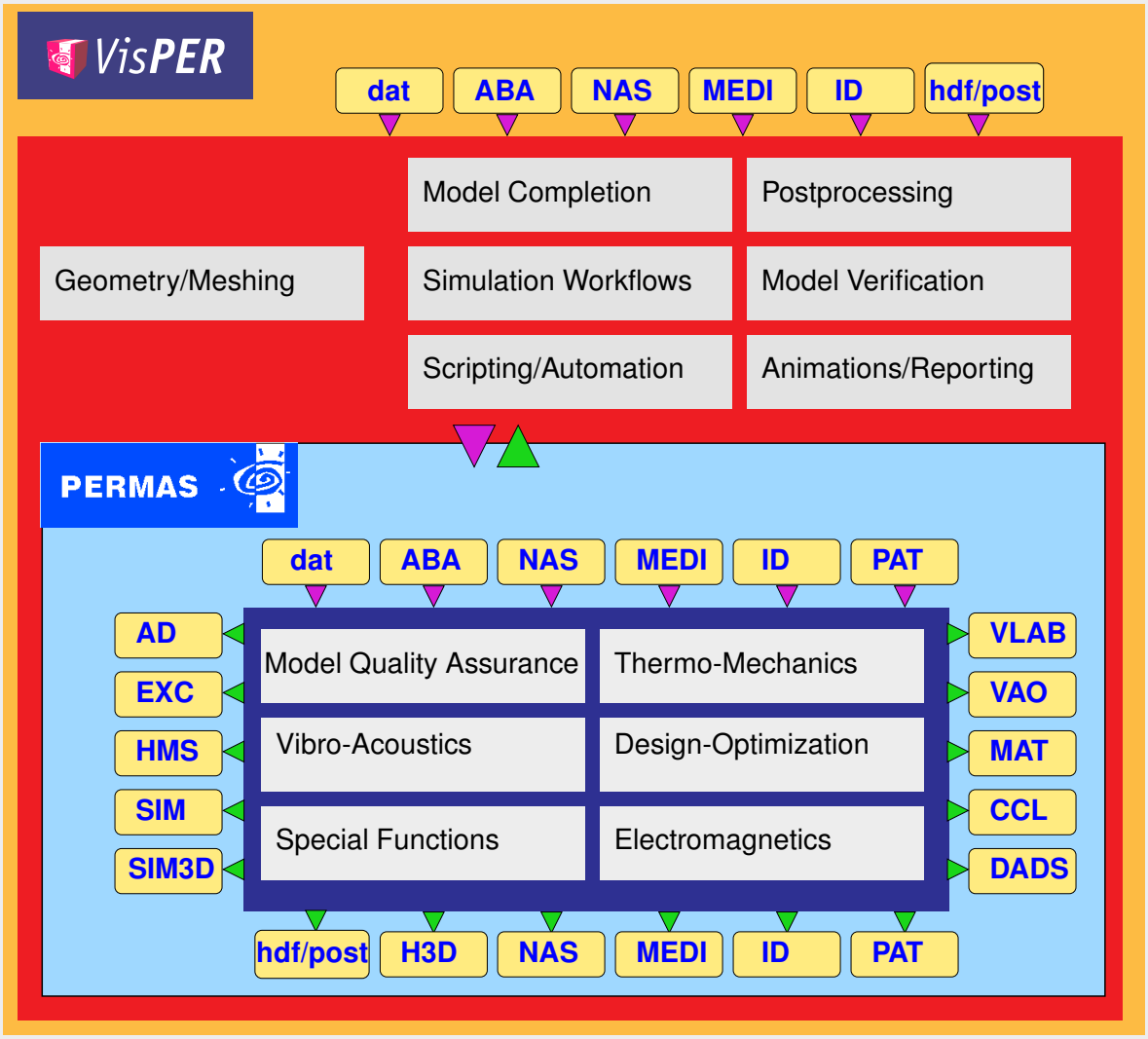
VBAS Basic module
VCA Contact modeling
VOPT Optimization models
VTOP Topology optimization
VFS Fluid modeling

INTES

3rd Party

ANSA Digimat HyperMesh Jupiter Magmalink MARLIS MeshWorks MESHPARTS

Novus Nexus NX SimLab



Animator4 EnSight Evaluator FE-Fatigue FEMFAT MAMBA modeFRONTIER

VisPER comprises PERMAS and uses the same data basis. So, a perfect data compatibility exists between pre-processing, analysis, and post-processing.

HPC (High Performance Computing) through parallelization (multi-threading), the additonal use of a GPU (like Nvidia Tesla Kepler GPU) and special algorithms (like contact, MLDR, fluid-structure-coupling).

Detailed information about the different modules can be found in the PERMAS Product Description on www.intes.de → Company → Publications.

INTES GmbH
Breitwiesenstr. 28
D-70565 Stuttgart
Tel.: +49-711-784990
Fax: +49-711-7849910
E-Mail: info@intes.de
Web: www.intes.de
Copyright 2026 INTES GmbH, Stuttgart, all rights reserved.

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

If not specified differently, all registered names and trademarks are legally protected brands of INTES or other organizations. The registered trademarks of INTES are PERMAS and VisPER. The use of other product names does not imply that these names are not protected. They are used here only to inform the reader.