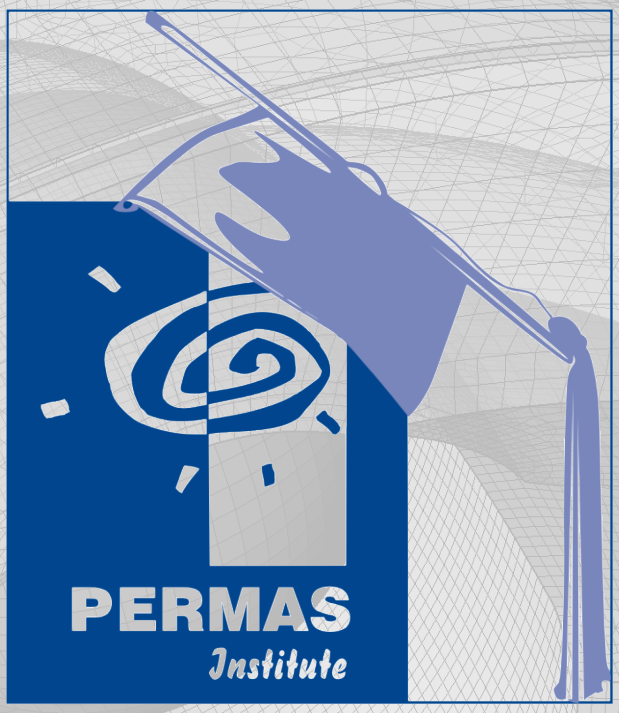


Training

2024

Ingenieurgesellschaft für
technische Software mbH

INTES

PERMAS

Training Courses 2024

PERMAS Basics Training:

Introduction (2 days)
(Basic-1, Basic-2)

PERMAS Special Training:

Contact Analysis
(CA-1) (1 day)
Contact Analysis Advanced
(CA-2) (1 day)
Dynamics I
(DYN-1) (1 day)
Dynamics II
Advanced Applications
(DYN-2) (1 day)
Dynamics – Special Subjects
(DYN-ROT, DYN-RAND, DYN-NL/HBM)
(Each ½ day)
Fluid-Structure Acoustics
(FS) (1 day)
Nonlinear Statics
(NLS-1, NLS-2) (2 days)
VisPER Basics
(VISPER) (1 day)
Design Optimization
(OPT-1, OPT-2) (2 days)
Topology Optimization
(TOPO) (1 day)
Advanced Modeling Features
(AD-1) (1 day)
Heat Transfer
(HT) (1 day)
Substructure Technique
(SUB) (1 day)

All training courses can be book individually with us.

Venue:
Online

Participation fee:
The fee for each online training course amounts to 980,00 EUR plus VAT per day for up to two participants. In case of mere online training, each training day is carried out over two mornings. The training fee includes all training
documents.

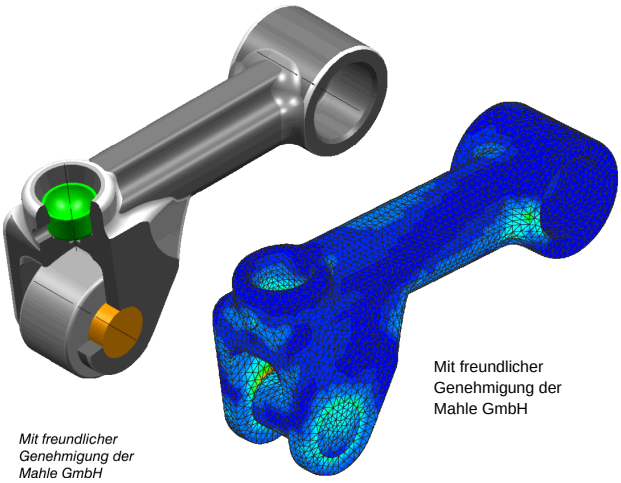
Registration:
Please write to us for all arrangements.

Organization and information:
Claudia Krauss
INTES GmbH
Breitwiesenstr. 28
70565 Stuttgart
Tel.: 0711 78499-0
Fax: 0711 78499-10
E-Mail: Claudia.Krauss@intes.de

PERMAS Basic Training Course (BASIC-1/BASIC-2)

This training course provides the basic knowledge of working with PERMAS. The aim is that at the end of the two training blocks the participants are able to describe and analyze a mechanical part with the help of PERMAS.

The operation of PERMAS and the configurable working environment for a calculation run with PERMAS as well as the data management will be explained. The handling of the PERMAS documentation and the usage of PERMAS tools are also shown.



Mit freundlicher Genehmigung der Mahle GmbH

Particular attention is paid to the clear separation of the analysis control (UCI file, User Control Interface) from the input of the model data in PERMAS format (DAT file). The syntax and functional division of the PERMAS commands in the individual sections in the UCI are discussed in detail, as well as the structural setting of FE models in data blocks in the DAT that are grouped according to their physical meaning.

The first training block shows which UCI commands are necessary to carry out a static analysis with PERMAS and to interpret them using PERMAS log files. Based on the basic concepts for the analysis and model structure, simple examples are used to illustrate how special results can be generated. The participants can apply the acquired knowledge directly on corresponding exercises.

In the second training block, the focus is on creating and describing models in PERMAS format. The participants learn to fully describe an FE model for PERMAS with the help of VisPER and to control the division of the model data in the different data blocks. The flexible variant principle of PERMAS is explained in detail and it is shown how a large number of constructive variants can be set up and analyzed in a single calculation run. The participants will directly use the acquired knowledge in suitable exercises.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

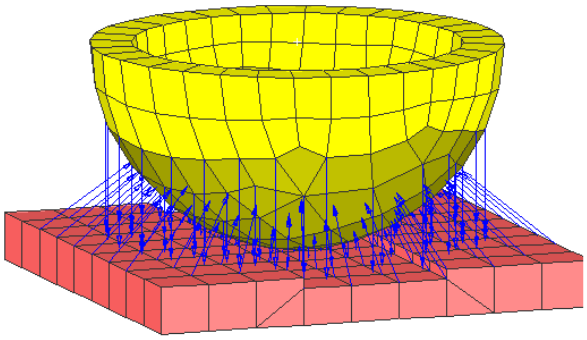
Duration 8 hours; German or English; at INTES, at your place or online

Topics: PERMAS product overview, introduction and basic definitions, operation and environment, interfaces, command language (UCI), error messages, short introduction on VisPER, integration in pre-/post processors, merging of input data, linear statics, data input, data structure, basics of variant analysis, exercises.

Prerequisite: Basic knowledge of structural mechanics and FEM

Contact Analysis Basics (CA-1)

After a short theoretical introduction to contact analysis, the participants learn how to contact definitions in PERMAS are implemented and how they can be used. The various modeling options for contacts (node or surface contact) are discussed in detail. The participants receive tips and hints on how to choose the best way to model the contact based on their problem.



The numerous contact results made available by PERMAS are presented and discussed. In addition, it will be explained how these results are to be interpreted and what possibilities they offer to check the model for plausibility.

The topic of the storage of rigid body modes, which are held by contact forces during the analysis, are treated. In particular, the CAZERF elements (zero force springs) are presented and discussed.

Another focus of the training is the contact geometry update. The definition in PERMAS is presented. Practical examples are shown, possible pitfalls are discussed and practical solutions are shown.

The definition of non-linear load histories for PERMAS is discussed and the screw preload will be introduced briefly.

The exercises offer the participants room for discussions and the possibility to put into practice what they have learned. In addition, they receive practical tips and thus learn how to apply the newly gained knowledge to own models.

It is recommended to attend this training course together with the PERMAS basic training.

Dates: Individual appointment by arrangement

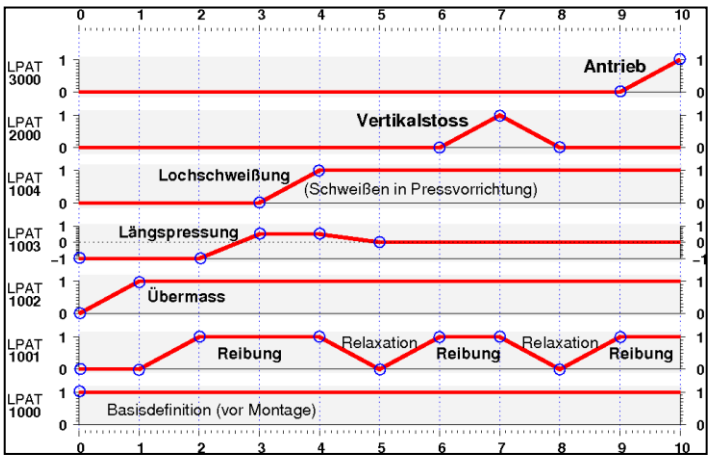
Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Topics: Normal contact, theory, node contact, surface contact, modeling, practice, screw preload, contact geometry update, load history, exercises

Prerequisite: Basic knowledge of structural mechanics and FEM as well as PERMAS basic knowledge and basic training.

Contact Analysis Advanced (CA-2)



This training course aims at the experienced users of PERMAS contact calculation because it provides very in-depth and detailed information about the efficient use of PERMAS in this area.

The focus of this training course is on all the terms that the user already knows from his daily work with contact analysis. What was previously perceived only peripherally will now be deepened and it will be explained how the information and results during the contact analysis can be interpreted, evaluated, and employed.

It will be discussed when contact with friction makes sense and how the friction affects that result. The convergence behavior will also be discussed in this context. It will be shown which parameters have an impact on the convergence and how these parameters might be improved. This main focus area also includes the contact status files in which the contact status is saved. They can be used, for example, as a starting point for subsequent analysis or model variants and thus significantly accelerate convergence.

Another focus is the combination of contact and other non-linearities. These include in particular the contact geometry update in connection with geometry non-linearity and the gasket elements (sealing elements) which play a decisive role in the engine calculation.

In addition, the possibilities for screw pretension are explained in detail. The parameters of preload force, flank angle and thread pitch are discussed, as well as the implementation of the preload process.

Based on interesting examples, the training content is presented, discussed, and deepened, so that the participants will be able to apply the acquired knowledge to their own models.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

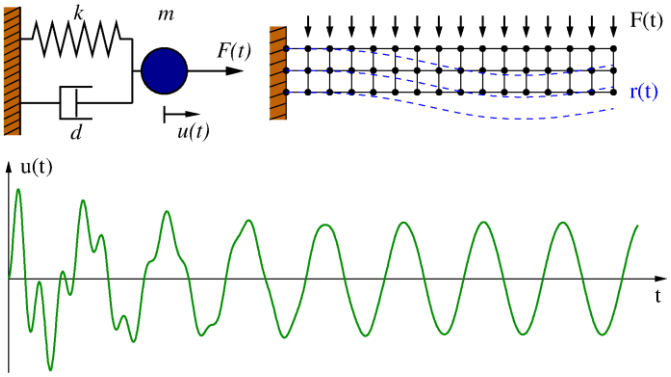
Topics: Contact results in detail, nonlinear load history, intermediate processing, bolt preload, contact geometry update, contact with shells, static friction, sliding friction, slip-stick, relaxation, saturation, limit load, iteration, convergence, performance, preload, tips & tricks, subsequent analysis (e.g. eigenvalues), case studies, discussion.

Prerequisite: Practical experience with contact analysis and CA-1.

Dynamics - Basics (DYN-1)

The training course dynamics I offers an overview of the basic capabilities of PERMAS in dynamics.

A simple single degree of freedom system is used for a short introduction into dynamics, which is then extended to finite element models. After this, the course discusses the definition of density and mass, which is compulsory for analysis in dynamics.



A major part of the training course is modal analysis, i. e. the computation of eigenfrequencies or natural frequencies and the matching mode shapes. The influence and the treatment of rigid body modes is discussed in detail – this allows for the computation of unsupported systems.

The second part of the training course deals with the analysis of vibrating systems which are excited by external loads. Procedures for the response computation in time and frequency domain offered by PERMAS are introduced. The definition of loads and damping, choice of important parameters for the computations and the interpretation of results are discussed.

The training course is completed by an introduction to modal transformation and the subsequent response computation in modal space.

Finally, a short overview of matching examples and further training courses in dynamics is given.

All topics of the training course are complemented by short exercises using simple but realistic examples, which can be executed at the computers in our lecture room or at home if you participate online.

Dates: Individual appointment by arrangement

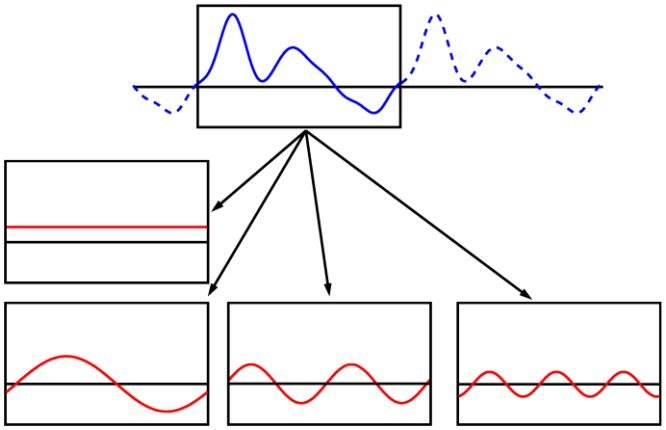
Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Topics: Definition of mass, eigenfrequencies and mode shapes, rigid body modes, analysis in time domain, analysis in frequency domain, damping, modal transformation, analysis in modal space

Prerequisites: Basic knowledge of structural dynamics and FEM as well as PERMAS basic training BASIC-1 and BASIC-2.

Dynamics II – Advanced Applications (DYN-2)



In the training course Dynamics II, the solution methods of the basic course Dynamics I are discussed in more detail and augmented by further techniques.

At first, further options for the computation of eigenfrequencies and mode shapes are explained. Especially the MLDR method is very useful, as it allows much faster computations in many cases.

The mode shapes used for transformation to modal space can be complemented by additional basis vectors, so called addmodes. The different kinds of addmodes and their influence on the results are discussed.

The definition of damping already addressed in the course Dynamics I is covered in more detail, such that the participants get an overview of the different ways to define damping in PERMAS.

Assembly situations, which are often used in customers' applications, as well as nonstructural mass is introduced. A large chapter deals with the primary and secondary results of the analysis procedures, including their interpretation.

Moreover, the special analysis methods of Steady State Response and Spectral Response are discussed. The use of files to save intermediate results is shown.

The training finishes with a short overview of further analysis method in dynamics, which are presented in special courses in detail.

All topics of the training course are complemented by short exercises using simple but realistic examples, which can be executed at the computers in our lecture room or at home if you participate online.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Topics: Vibration Analysis, MLDR method, addmodes, damping II, Assembly Situations, nonstructural mass, results, Steady State Analysis, Spectral Response, using files, special topics

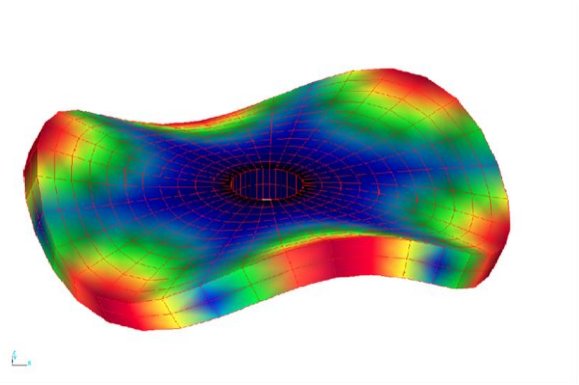
Prerequisites: Training Dynamcis I

Special Training – Dynamics

Rotating Systems (DYN-ROT)

The training course gives an overview of the possibilities of computing rotating systems in PERMAS, either in a co-rotating or a fixed reference frame. This includes evaluation of the stability.

The course starts with simple bodies, continues with rotating axes and also covers break squeal analysis.

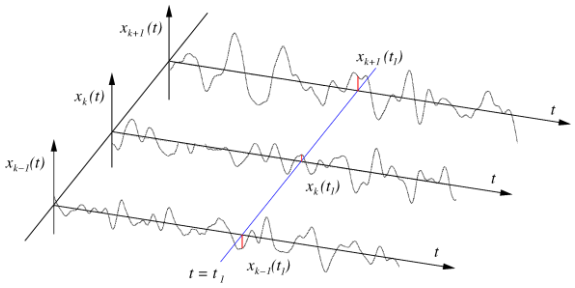


Topics: Complex eigenvalues, rotating systems, break squeal analysis, cyclic symmetry

Random Vibrations (DYN-RAND)

This course is about the computation of structures under a random load in PERMAS.

It starts with a short introduction into the theory of random processes and the prerequisites for a computation using PERMAS. This is followed by the definition of loads and the computational procedure. Finally some examples are shown.



Topics: Random processes, random loads, solution, results, load by matrix, dimensions, examples

Prerequisites: DYN-1 and DYN-2 for all special topics

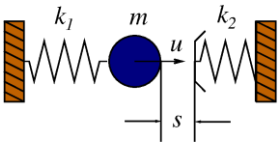
Dates: Individual appointment by arrangement

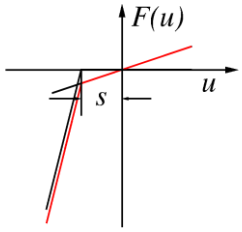
Please contact claudia.krauss@intes.de

Duration 4 hours each; German or English; at INTES, at your place or online

Nonlinear Dynamics / Harmonic Balance Method

(DYN-NL/HBM)



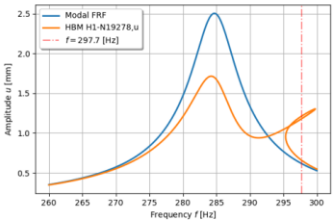


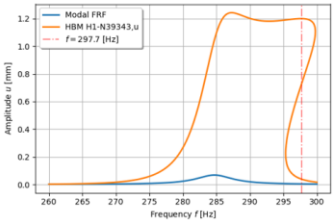
Several nonlinear effects can be handled by dynamic analyses in PERMAS. The nonlinearities are modeled by special discrete non-linear elements.

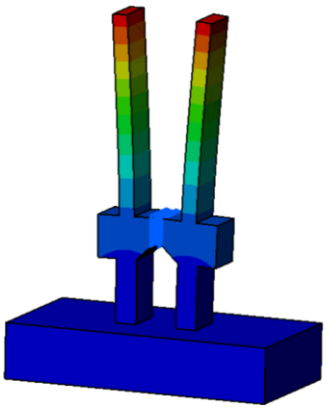
The training shows the calculation methods for non-linear dynamics in the time and frequency domain (Harmonic Balance Method).

The course begins with an introduction to nonlinear dynamics using a single-mass oscillator with a nonlinear spring (Duffing oscillator). Various non-linear effects are demonstrated using this example. This is followed by a definition of the extended periodic approach with a finite Fourier series compared to linear dynamics. The coupling of the harmonics through the non-linear effects is discussed here.

After the theoretical introduction, the PERMAS application begins.







What possibilities does PERMAS offer in harmonic balance? What are the most important parameters and how can I interpret my results? Which element types are supported? How can I incorporate modal reduction into my Harmonic Balance? How are results transformed from the time domain to the frequency domain? Which variables influence the computing time? What are my options for determining whether my solution has converged?

All the training topics are supplemented by short exercises based on simple, practical examples, which participants can carry out on the computer in the seminar room or at the workplace during face-to-face and online training.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

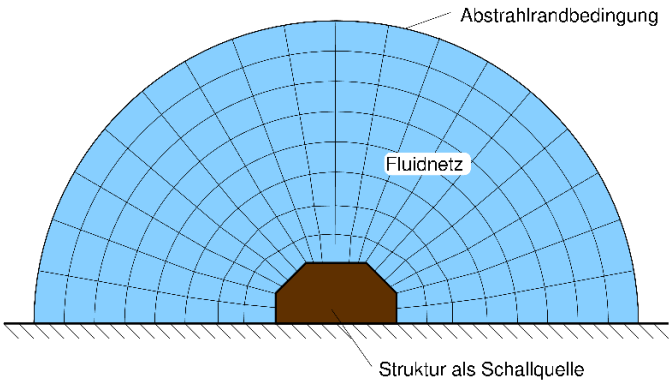
Duration 8 hours; German or English; at INTES, at your place or online

Topics: Introduction to non-linear dynamics, nonlinear elements, controller elements, prony series, cutting process stability, theory and application of harmonic balance, analysis of the results in the frequency/time domain, damping options for harmonic balance

Prerequisites: DYN-1 and DYN-2, as for all special topics

Fluid-Structure Acoustics (FS)

PERMAS offers the possibility to calculate both purely acoustic vibrations (uncoupled) and coupled fluid structure vibrations. The aim of the training is to show the participants the implementation in PERMAS.



In addition, the participants learn how to determine the response behavior in the time and frequency domain (coupled and uncoupled).

The focus of the training is on the coupled fluid-structure tasks and how to solve them in PERMAS. Particular importance is attached to explain to the participants how the modeling is to be carried out. First of all, it is made clear to the user how the discretization of the fluid depends on the wavelength. All necessary modeling steps and definitions are explained using various examples. In addition, there are not always compatible meshes between the structure and the fluid. PERMAS also offers a comprehensive solution for this transition.

Another important issue is sound propagation to infinity. It is shown how PERMAS solves this task with the help of emission boundary condition elements, since finite elements can only describe a limited space. This is also illustrated using an example.

Exercises help to deepen the material and provide space for discussion.

Topics: basics, special elements, boundary conditions, loads, fluid-structure-acoustics (inherent values, response), practical modelling, exercises

Prerequisites: Basic knowledge of structural dynamics and the DYN-1 training

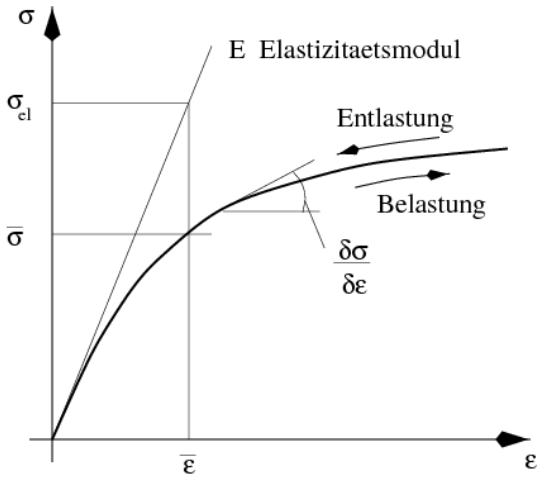
Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Nonlinear Statics (NLS-1/NLS-2)

In this two-day training, the participants will get an overview of the application of nonlinear methods in PERMAS. There will be a special emphasis on the close connection of the theoretical facts with the corresponding formulation in PERMAS.



The first day offers a summary of the theoretical basics of nonlinear statics. These include the material non-linearities of elasticity and plasticity as well as temperature dependent material behavior.

Visco-plasticity and creep will be covered as needed.

An additional chapter deals with the geometric nonlinearity. Here it will be shown which criteria apply to a geometrically nonlinear task and how these are dealt with. On the theoretical and PERMAS-related training topics practical tasks are given.

The second day will give an introduction in the different material models, such as von Mises, Tresca and cast iron. Here the theory will be discussed in detail in order to explain the correct use of the calculation parameters. The available nonlinear strategies and iteration methods will be explained and assigned to the application areas.

Special emphasis will be placed on exercises with different constellations of nonlinear effects. Especially on the second day, more complex examples will be dealt with in order to get to know the different combinations of nonlinearities and iteration methods. Questions of convergence and correct interpretation of nonlinear phenomena will also be deepened.

Topics:

1. Day: Basics in material non-linearities, temperature dependence, creep, visco-plasticity and geometric non-linearity, modeling in PERMAS, exercises

2. Day: Deepening of nonlinear theory, iteration methods, material models, cast iron, combination of material non-linearity with geometric non-linearity, special cases, modeling in PERMAS, exercises

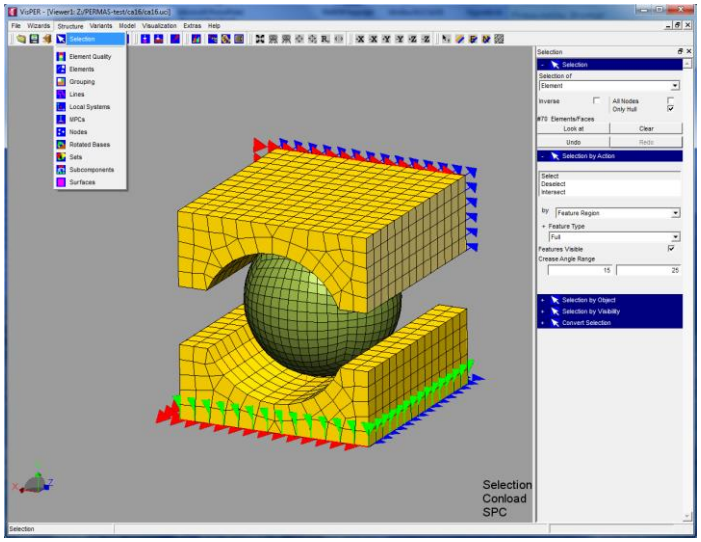
Prerequisite: Basic knowledge of PERMAS

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours per day; German or English; at INTES, at your place or online

VisPER Basics (VisPER)



VisPER is a model editor for PERMAS-specific completion and the graphically supported verification of models. What has to be highlighted above all are an intuitive, self-configurable operation, a model representation consistent with PERMAS and a logical user guidance.

This leads to an efficient handling of even complex models (including substructure) and time-saving processes in the model definition with a low error rate.

VisPER is integrated into the process between networking and calculation. Models in all formats that are supported by PERMAS can be read. VisPER provides functions to carry out all essential definitions up to the computable model. This application profile is rounded off by general and special post-processing functionalities.

The VisPER training is fully interactive. All the functional elements, dialogues and processes that are presented will be immediately reproduced on the computer. In this way, the participants get to know the functionality and working method of VisPER in a quick and easy way. Using examples, they will become familiar with efficient methods in the model definition for PERMAS. Exercises will be worked on in order to consolidate the methods presented.

Topics: Introduction in VisPER, handling and operation, dialog bars, wizard technology, element quality control, model completion, model checking, post-processing, process embedding, referencing, customization, reporting, shortcuts, file export, exercises

Prerequisite: Basic knowledge of PERMAS

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Design Optimization (OPT-1/OPT-2)

This two-day training offers a simple introduction to the optimization functionalities of PERMAS. The first day is dedicated to shape optimization, at the second day the focus is on dimension optimization.

Shape optimization involves modifying existing FE meshes in such a way that an optimum is achieved with regard to a criterion selected by the user. Methods specially tailored to the optimization of volume models (parametric and non-parametric shape optimization) and shell models (bead detection) are discussed. All definitions required for this are shown and executed interactively using examples with the help of the VisPER model editor.

On the second day of training, a procedure for the optimal dimensioning of components will be taught. This is used, for example, to design flanges, to optimize spring stiffness or to find optimal shell thicknesses. The necessary definitions are carried out with the help of VisPER. It is shown how the calculation can be controlled and which results can be generated. Finally, the combination of dimension optimization with shape optimization is presented.

On both days, exercises on optimization tasks from the field of statics and dynamics will be carried out independently by the participants with the support of VisPER.

Topics: Introduction to optimization, GUI support by VisPER, simple examples of dimension and shape optimization, different types of analysis (static / dynamic) in one optimization run, combination of dimension and shape optimization, exercises

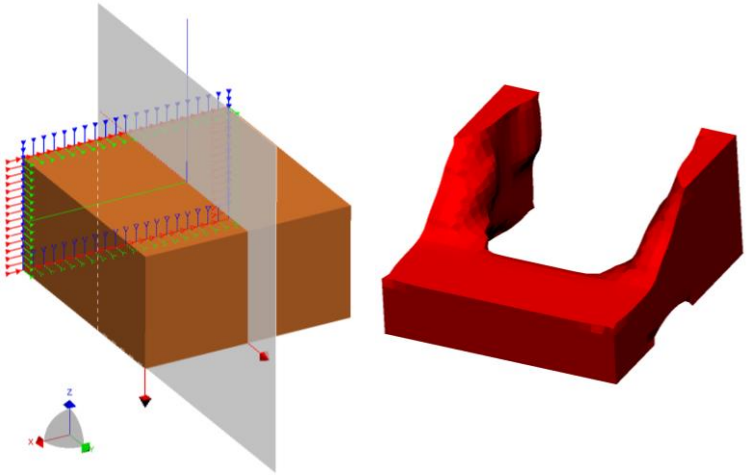
Prerequisite: PERMAS basics. For the second day (OPT-2) the knowledge of the first day (OPT-1) is required.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours each day; German or English; at INTES, at your place or online

Topology Optimization (TOPO)



Topology optimization is a design-finding method. This results in an optimized structure from the automatic selection of finite elements in a given construction space, which achieves a goal specified by the user and also fulfills a number of other side constraints.

The aim of the training is to give the participants an introduction to solutions for typical optimization tasks. The required model definitions and control commands are first explained for simple applications using examples, and then more complex tasks, in particular the consideration of manufacturing boundary conditions such as e.g. demolding directions, component symmetries and wall thicknesses, expanded.

An important part of the training is the processing of exercise examples with the model editor VisPER. Here it is shown, with the help of a wizard, how the model additions necessary for the optimization can be carried out efficiently, completely and error-free. In addition, the diverse possibilities of post-processing are shown, including the efficient converting of the results of a topology optimization in a new design as a basis for a new analysis model or for output to CAD systems.

Topics: Topology optimization, UCI, VisPER, data input, examples, postprocessing, hull generation, exercises

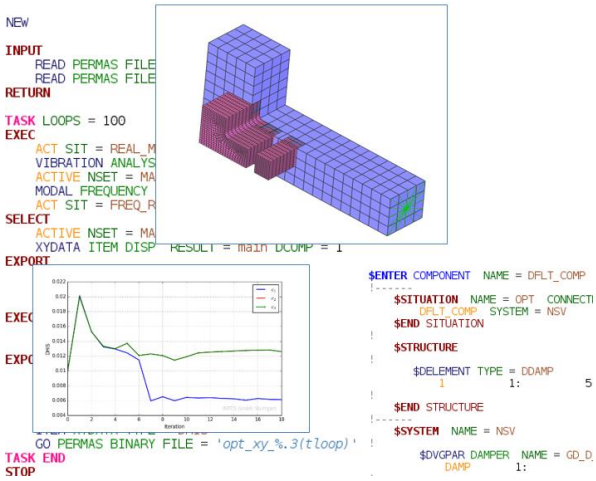
Prerequisite: PERMAS basics.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Advanced Modeling Features (AD)



The aim of the training is to use PERMAS to full capacity and to develop a better understanding of the interaction between UCI and data entry.

One focus is on the MPC conditions available in PERMAS, their peculiarities, restrictions and areas of application.

In addition, the training deals with press fits and spotwelds which are based on internally generated MPC conditions.

Another topic is the parametrization of models using variables in data entry and UCI control.

Furthermore, advanced methods such as sampling, inertia relief and sub-modeling techniques are discussed and deepened using application examples. In another application area auxiliary UCI commands are dealt with in detail.

Exercises on the respective subject areas serve to deepen the learning content.

Topics: Efficient usage of PERMAS, UCI definition, tips and tricks, advanced methods for experienced users, MPC conditions, press fits, spotwelds, parametrization, inertia relief, sub-modeling and mapping, sampling, auxiliary UCI commands, exercises

Prerequisite: Basic knowledge in data input and UCI, experience with PERMAS

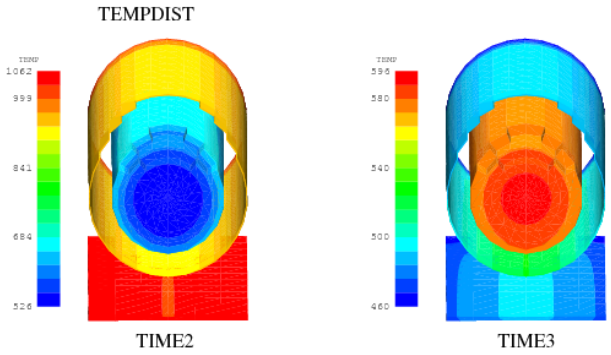
Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

Heat Transfer (HT)

In PERMAS there are several solution methods for the analysis of temperature fields available. These allow the solution of stationary and transient problems in the linear and non-linear area.



The training gives the participants an overview of the available procedures and also some aids in order to make a decision when which procedure should be used.

The model structure, the data input in PERMAS as well as the UCI commands for the controlling of the solution methods are in the foreground of this training. In addition, the participants receive important information and hints from practice which make it much easier to apply them to their own models.

The training also addresses coupled analyses. It will be shown how a thermomechanical analysis is performed with PERMAS. With this type of analysis, the calculated temperature field is applied as a load in a subsequent static analysis. This can be done either within a single PERMAS calculation or in several successive calculations.

Another focus of the training is on the topic of heat exchange through radiation. This is important for components with cavities that are exposed to very high temperatures, such as brake systems, combustion engines or cooler. In this case, PERMAS first determines the visibility factors of the surfaces involved and then, in a non-linear analysis, the exchanged heat flows and the temperature field in the components.

The aim of the training is that the participants learn how to calculate the heat transfer through radiation and convection on the component surface, coupled with heat transfer in the structure.

Application examples and own exercises deepen the training subject.

Topics: Temperature field analysis, linear and nonlinear solution methods, heat transfer through convection and radiation, thermal stresses, material properties, loads and boundary conditions, application examples, exercises

Prerequisite: Basic knowledge of PERMAS

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online

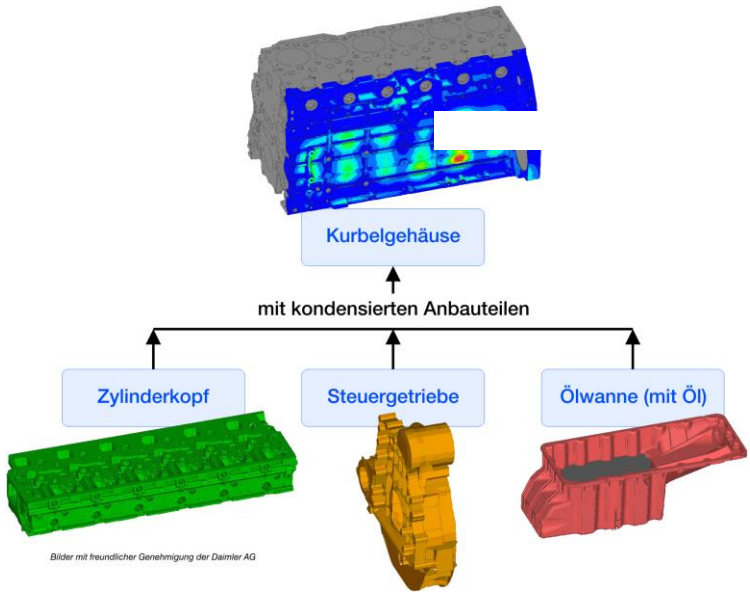
Substructure Technique (SUB)

In many cases, the substructure technique permits a drastic reduction of computing times.

In this training, the participants will get an introduction to the methods of static and dynamic condensation and their application with PERMAS.

After a general overview, the theoretical basics of the methods that are available in PERMAS will be presented.

In the substructure technique, the model is divided into a top component and one or more subcomponents. The necessary definitions in the PERMAS input file, also with regard to the variant concept, will be presented. The UCI commands required in connection with the substructure technique will be shown and discussed using examples. The training also covers the MLDR algorithm which is used for automatic reduction of the model in dynamic analyses.



Mit freundlicher Genehmigung der Daimler AG

Another focus is on the structure and handling of matrix models. This way, condensed models can be saved and used again. This also allows to pass on stiffness and mass properties to third parties without having to disclose the geometry of the model.

Examples and exercises will deepen the training content.

Topics: Theoretical background, Guyan reduction, Craig-Bampton method, mixed boundary dynamic reduction, MLDR method, matrix models, application notes, exercises

Prerequisite: Basic knowledge of data input and UCI, experience with static and dynamic analyses.

Dates: Individual appointment by arrangement

Please contact claudia.krauss@intes.de

Duration 8 hours; German or English; at INTES, at your place or online