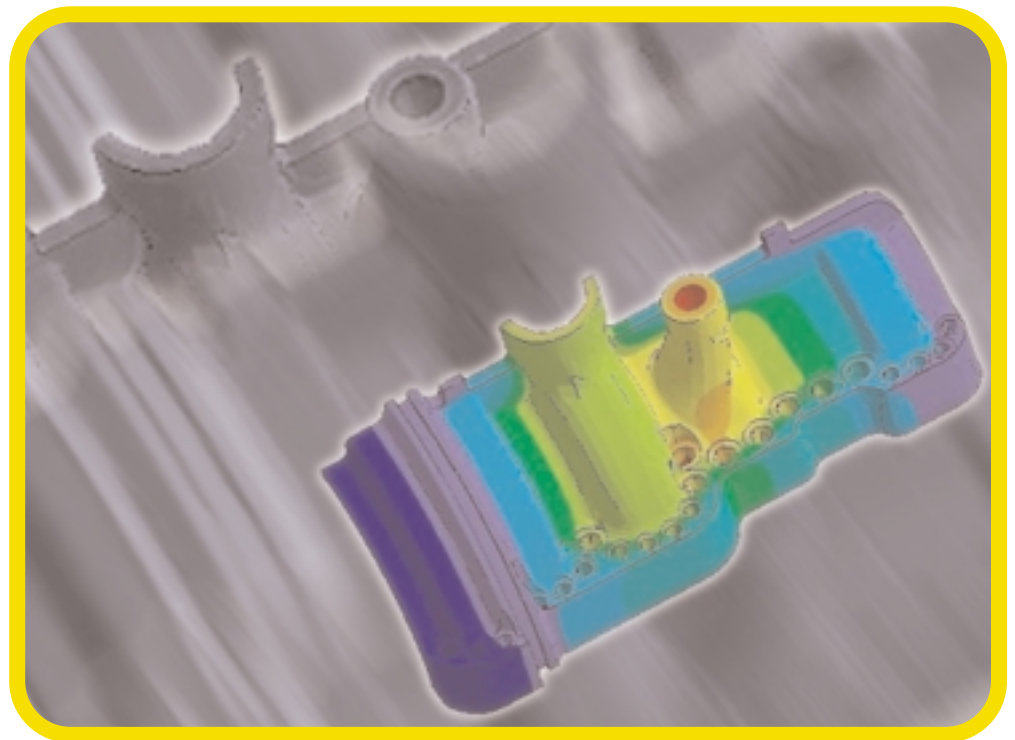


Engineering Applications of ANSYS[®] Inside Siemens AG



Compiled and Edited by
Gerhard Müller
Siemens AG, Erlangen, Germany

Table of Contents

	Profile: Siemensii
A.	Preface3
B.	Historical background3
	Numerical simulations at Siemens AG have a long tradition3
C.	Review of early ANSYS applications4
	Defining a geometry using ANSYS Rev. 2 in the design process5
D.	ANSYS examples7
	Calculation of pipe supports with the ANSYS-Program7
	Structural analysis of complex systems in nuclear power plants8
	Fatigue analysis using ANSYS9
	Eigenvalues of a magnet11
	Transient thermal study of directly buried gas-insulated transmission lines (GIL)12
	Temperature distribution of crossing cable systems14
	Electrothermal analysis of switching elements for a superconducting fault current limiter15
	Stress engineering of stencil masks for ion projection lithography16
	Detuning of an HF oscillator by the mechanical vibration of an acoustic sounder17
	Potential characteristic of the gate of a power MOSFET calculated with the finite element method18
	Calculation of the temperature distribution via a refrigerator seal19
	Computation of the S-parameters of a multipole filter by an equivalent circuit20
	Development of a parametric catenary/pantograph model23
	Finite element calculation of containers as part of railway wagons24
	Calculation of ride comfort of a railway vehicle including flexible car body25
E.	DesignSpace examples29
	Transformer housing29
	Stress distribution of a flange29
	FE analysis of a contact spring with DesignSpace30
	Turbine casing31
	Mode shape analysis of hot saws31
F.	Acknowledgement32
G.	Literature33
H.	Biographical notes34

Profile: Siemens

Siemens - Global network of innovation

Siemens, headquartered in Berlin and Munich, is one of the world's largest electrical engineering and electronics companies, and one of the richest in tradition. In fiscal 2000 (ended September 30, 2000), the company had 447,000 employees and posted sales of €78.4 billion and net income of 3.38 billion before extraordinary items. Extraordinary items, primarily from the successful public listings of Epcos and Infineon, totaled €4.52 billion. The company boasts an impressive international presence, focusing on the core business segments of Information and Communications, Automation and Control, Power, Transportation, Medical, and Lighting. This constellation of businesses is both a uniquely broad and highly focused spectrum of activities. In the course of restructuring its business portfolio, Siemens has spun off its Components segment and divested Siemens Nixdorf Retail and Banking Systems, as well as its power cable business, some of its communication cable activities, and selected activities in the Automation and Control segment. Siemens currently holds 71 percent of Infineon Technologies AG, but intends to completely withdraw over the medium term.

In addition to its 180,000 employees in Germany, Siemens has 267,000 employees working in more than 190 countries around the world, in hardware and software production, engineering, service, sales, and development. The company operates some 500 production facilities in more than 50 countries, underlining its status as an innovative global player.

Siemens' international business is conducted by sixteen operating Groups. The company is also an equal partner with Bosch in a joint venture within the household appliances sector. Siemens' Information and Communications segment has been re-organized to comprise three new Groups: Information and Communication Mobile (mobile networks and end-user devices), Information and Communication Networks (convergence solutions for the Internet and telecommunications networks), and Siemens Business Services (services for electronics and mobile business). The company has strengthened its expertise in Internet technologies by acquiring a number of U.S. start-ups. These will be publicly listed on the American Nasdaq technology exchange under the name "Unisphere." In the computer business, the 50/50 joint venture Fujitsu Siemens Computers was formed effective October 1, 1999. Siemens Building Technologies, headquartered in Zurich, was established following the acquisition of the industrial activities of Switzerland's Electrowatt. Siemens has considerably extended its position in the fossil-fuel power plant business by acquiring the conventional power business of Westinghouse in the U.S. Siemens' nuclear power business is being funneled into a joint venture with France's Framatome. Finally, the company's acquisition of Atecs Mannesmann will enable it to strengthen its automotive engineering and production and logistics systems businesses. Siemens Automotive will be merged with the Atecs company VDO and expanded to create a world market leader in automotive electronics. In the high-growth e-logistics busi-

ness, Siemens Production and Logistics Systems will attain a world-leading position through its merger with Dematic.

All of these portfolio optimization measures aim at placing Siemens' businesses in number-one or number-two global market positions. If the company does not achieve this aim with a business, it has four options: buy, cooperate, sell, or close.

The company's business strategy is focused on the growing world market for electrical engineering and electronics, which currently has a volume of more than €2,000 billion. Thanks to rapid advances in electronics, growth rates in the field of electrical engineering will remain well above the industry average as a whole. International business remains a growth driver for Siemens. In fiscal 2000, international business accounted for 76 percent of the company's total sales. Twenty-seven percent of this business was generated in the Americas, while Asia-Pacific accounted for roughly 13 percent. Germany's share of the total was about 24 percent, and its European neighbors contributed 31 percent. Five years ago, Germany alone accounted for 41 percent of the total.

In fiscal 2000, spending on research and development rose to more than €5.6 billion, or just over seven percent of sales, underscoring the enormous importance of innovation at Siemens. The company's innovative strength is further evidenced by the 8,200 invention disclosure reports it submits each year. In terms of registered patents, Siemens ranks

number one in Germany. The company is also number one in registrations at the World Intellectual Property Organization (WIPO). In the U.S., we are currently ranked ninth in patents approved. Roughly three-fourths of all Siemens' products and services are less than five years old. This is made possible by the efforts of some 50,000 R&D employees at Siemens, almost one-third of whom work outside of Germany. The company has its own R&D departments in more than 30 countries.

Growth and innovation are two pillars of Siemens' top initiative, a program that has been successfully implemented for several years now. At the beginning of fiscal 1999, the company made economic value added (EVA) the obligatory performance measure within the entire Siemens organization. Profitability is measured exclusively in terms of a business's ability to generate returns exceeding the cost of capital. Siemens achieved a positive EVA for the company as a whole for the first time in fiscal 2001—a year ahead of schedule.

With some 600,000 shareholders, Siemens is one of Europe's largest public corporations. Just over 50 percent of the share capital is held outside of Germany, and a considerable proportion of the shareholders are Siemens employees. The Siemens share is traded on the eight German stock exchanges, as well as in Amsterdam, Brussels, London, Paris, Vienna, Basel, Geneva, and Zurich. A public listing of the Siemens share in New York occurred in March 2001.

About Framatome ANP

Founded in January 2001, Framatome ANP (Advanced Nuclear Power) combines the former nuclear activities of Framatome and Siemens to form the world's premier nuclear supplier. In the company, AREVA has a 66% stake and Siemens 34%. (When Framatome S.A. was merged—as of September 5, 2001, into the new company AREVA—the shares originally held by Framatome S.A. were assigned to AREVA). Framatome ANP is headquartered in Paris with regional subsidiaries in the U.S. and Germany, and has a

workforce of approximately 13,000 worldwide.

Framatome ANP's focus includes comprehensive engineering, instrumentation and control, nuclear services, heavy component manufacturing, modernization, fuel assemblies for many reactor designs—including those supplied by other vendors—and the development and turnkey construction of nuclear power plants and research reactors.

A. Preface

In the modern product design process, the finite element method has become an everyday tool used to predict the behavior of components and assemblies. Interpretation of analysis results helps the engineer predict behavior and reduce the number of prototypes, physical tests, and development time scales & costs, all the while increasing innovation. Analysts and designers work together in the design process—using advanced optimization methods—to find the best answers.

The finite element program experts within Siemens mainly use to perform advanced coupled-physics, numerical simulations to solve complex engineering problems is ANSYS.

This paper presents an overview of the use of ANSYS in a number of different engineering fields such as power generation, transportation, medical components, electronic devices, and household appliances.

First a historical background on the use of ANSYS at Siemens is provided.

B. Historical background

Numerical simulations at Siemens AG have a long tradition

In the early seventies the KWU Group—a subsidiary of Siemens—started to design nuclear power plants. Numerical simulation of nuclear components was necessary to ensure a high level of quality and reliability^{1/}.

Several engineering departments began to use the finite element method to solve the complex problems they encountered. To satisfy the growing analysis requirement, data centers were established at Siemens sites in Erlangen and Offenbach.

The computers at those data centers were mainframes located in large air-conditioned rooms, representing a large investment.

After a short time, many computer programs written within KWU and other applications licensed from commercial software vendors were applied by thousands of users within Siemens. One of those programs was ANSYS Rev. 2, shortly followed by Revision 3.

The first ANSYS seminar was held at KWU, Erlangen, in 1976 with attendees from Swanson Analysis Systems, Inc. (the forerunner of ANSYS, Inc.). Approximately 40 to 50 ANSYS users from Siemens attended. In 1979 the first ANSYS seminar was held at KWU in Berlin.

Beginning in the early eighties, a special team of FE experts was formed within KWU to consult with the internal users of finite element programs. This team organized ANSYS user meetings that have subsequently been held every year, mostly in Erlangen. A summary of one meeting was published in FEN (Finite Element News) 1986, Issue 1 (February) ^{2/}, illustrating that many of the items discussed are still applicable to today's users.



Figure 1
A photograph taken at the 1986 ANSYS German User's meeting picturing (from left to right) Gerhard Müller SIEMENS AG, Dr. Günter Müller, President and CEO of CADFEM, the ANSYS support distributor in the German-speaking countries of Europe, Dr. John A. Swanson, President of Swanson Analysis Systems Inc, Mrs. Sue Batt, Swanson Analysis Systems Inc.

A representative of Swanson Analysis Systems, Inc. (later ANSYS Inc.) has visited each annual user meeting since 1976. Additional seminars were scheduled when required. This was especially the case in 1983, when Revision 4.0, with substantial enhanced capabilities, was released. Five-day training seminars were held in Erlangen and Offenbach, covering linear and nonlinear structural mechanics, as well as heat transfer and the associated use of the graphics capabilities.

To help German speaking users, especially students, a book on the use of ANSYS—written in German—was published in 1989 /3/. Concurrently, SASI released an educational version of ANSYS that was a limited solution size version of rev. 4.2. Only linear statics problems could be solved. The hardware requirements for this educational version were a PC with 512 kByte memory and a processor chip extended by a numerical coprocessor such as 8087 or 80287. The minimum disc capacity was 10 Mb. The supported operating system was MS-DOS Version 2.11 or, alternatively, PC-DOS Version 2.1 or later. The element library available consisted of two-dimensional elements such as spar, beam, axisymmetric conical shell, and isoparametric solid elements. Three-dimensional elements consisted only of spring, mass and shell elements.

One year later, the educational version for thermal capabilities was released (Version 4.2b). The usable elements were: the radiation link, heat conducting bar, two-dimensional isoparametric solid, the three-dimensional thermal shell, and the thermal mass.

With the release of Revision 5.0 in 1992, the use of ANSYS increased rapidly within the Siemens Group. The early contract with ANSYS was renewed and extended for all worldwide Siemens sites. The successful long-term cooperation with CADFEM GmbH—the exclusive ANSYS Support Distributor for Germany—continued.

Today, the local consulting team at KWU supports the users, handles the ANSYS contract for users within Siemens' global enterprise, and organizes users meetings and update seminars. The development of computers and the multipur-

pose capabilities of ANSYS are the main reasons why ANSYS is one of the most applicable software tools for numerical simulations. The capability to read CAD data using workstations and PCs makes the use of ANSYS easy and most practical.

ANSYS is used by many different Siemens businesses involved in the production of power generation and distribution, automotive components, transportation systems, medical equipment, MEMS, and electronic devices. The use of ANSYS now covers a wide range of fields not only in structural mechanics, but also heat transfer, fluid dynamics, magnetism, piezoelectricity, etc. Brief details of some these applications are presented here; further published papers of analyses using ANSYS can be found in the proceedings of the CADFEM Users Meetings (/7/ .../22/).

C. Review of early ANSYS applications

During the late seventies, a large number of simulations in the design process of nuclear power plants were performed using ANSYS. Most of the load cases were transient and temperature-dependent, including varied pressure cycles. A number of geometries were represented by axisymmetric two-dimensional models, e.g. structures such as nozzles /23/. Also, three-dimensional transient calculations were performed with a very large amount of computer resources. Several batch jobs that took a couple of weeks to obtain the final results of the analysis existed. In the case of linear static analysis for complex structural components without transient effects, shell or volume models with an increased number of elements were created.

Before detailed analysis became commonplace, beam element models were used to study the behavior of complex structural components. The main benefit was to receive a very quick answer regarding the global stress distribution. The most tedious and time-consuming work of those models was to calculate the moments of inertia and find the right orientation, due to poor capability of plots showing the element coordinate systems. Furthermore, post-processing of the results was complicated because no graphic algorithms were available for beam elements to show the stresses.

With the continued development of computer hardware, shell elements were used to model thin structures in combination with beam elements to simulate the boundary conditions. To solve models with large numbers of elements, the so-called substructure technique was used. It was possible to subdivide structures into discrete parts represented only by a stiffness matrix generated from a separate finite element model. Using this procedure, the parts of the structures were combined numerically to form the complete component. A number of calculations had to be made before the results could be shown.

Defining a geometry using ANSYS Rev. 2 in the design process

The first case described is the analysis of a square support beam using Rev. 2 of ANSYS [4, 7]. It is interesting for today's users to learn how geometry was defined for Rev. 2 without the help of any geometry modeling capabilities or possibilities to read CAD data. No graphics exist for the original analysis, but the geometry shown in Figure 1 is similar to that which is described here.

The first task was to discretize the structure into nodes and elements. For this discretization process, the experience of the user was needed. First, to devise a finite element model that would yield an acceptable stress distribution (reasonable element shapes and sizes). Second, to optimize the wave front (number of active equations in the solution process), to allow the model to be solved in the available memory of the computer in a reasonable period of time.

To create the finite element model, the user would separate the geometry into four flat panels and then generate a pattern of nodes to produce quadrilateral shell elements on each panel. The number of nodes on each edge could not be too high, in order to minimize the wave front. The frontal method equation solver has to build up the element matrices from the node and element numbers. In order to minimize the wave front, the numerical difference of the node numbers defining an element needed to be kept as small as possible.

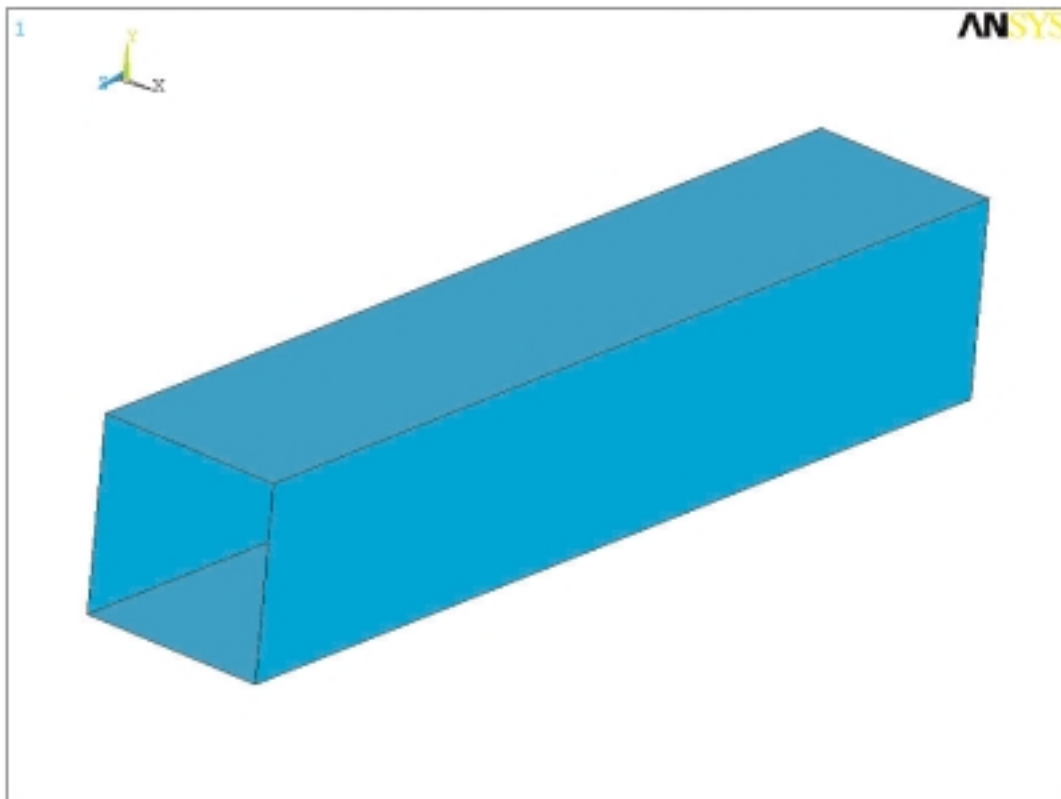


Figure 1
Simple geometry model.

For demonstration purposes, only the simple model was meshed (Fig. 2). It can be seen that the sorting of the elements is important for the wave front. The element connectivity should be started along the shortest edge of the structure in the direction of the arrow. The advantage of the frontal equation solver is that solution times and memory requirements are independent of the largest node number (unlike other methods).

The next problem to overcome was the assembly of the finite element meshes on each side panel. To achieve this, the nodes on each mating edge of the panels needed to have the same numbers and the same spatial definitions to enable assembly. A command to merge coincident nodes like “nummrg,node” was not available in Rev. 2.

A further consideration during the model development was to minimize the amount of data to be entered using the generation capabilities available in the program. Data entry was a laborious task and could take hours of manual typing on a computer key cardpunch or computer terminal. ANSYS allowed nodes to be generated using a geometric increment

or filling node numbers between the existing ones. Using a node number increment, a given pattern of elements could be generated. At the time, a lot of users wrote their own small FORTRAN programs to generate the input data sets for special classes of geometry.

One additional item should be mentioned to review the use of ANSYS in the past, which was the issue of graphical output. It is very tedious to perform a finite element calculation without graphics. After using the drum plotter located in the data center for many years, the first graphical screen was delivered. The time to get a plot was then reduced by a factor of 5 to 10. There were no interactive graphics like today’s programs. A hardware-dependent plot file was written using software libraries provided by the hardware vendor. This was one of the major changes of graphic systems using finite element software.

This short review of the difficulties in the past is intended to show the development of the enhanced capabilities of the ANSYS program. Today, this simple model can be defined and meshed within a few minutes.

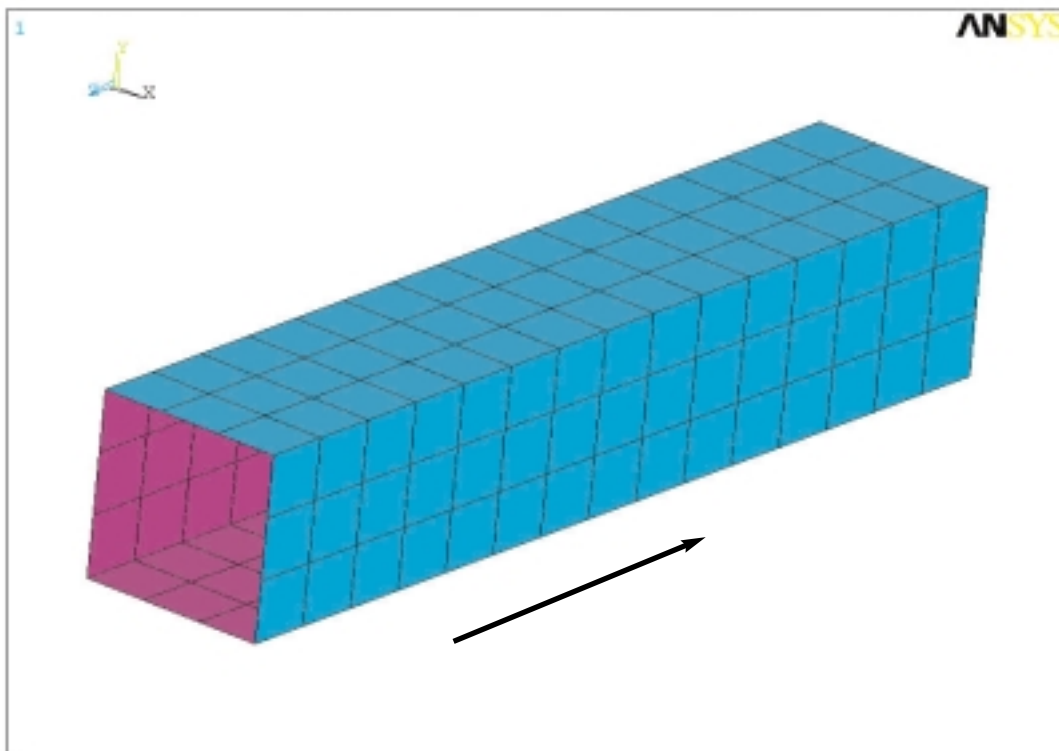


Figure 2
Meshed simple geometry model.

D. ANSYS examples

Calculation of pipe supports with the ANSYS-Program

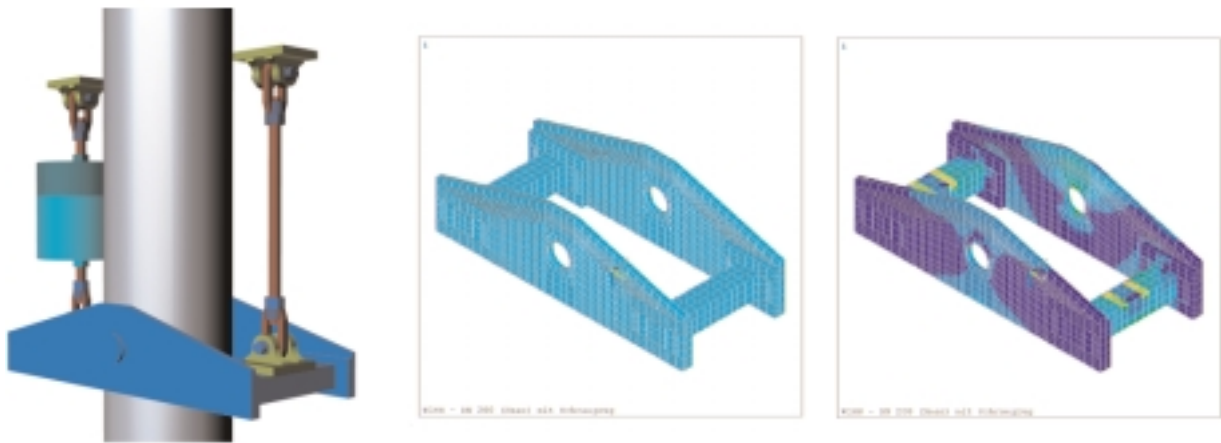
K. Kullig, Framatome ANP,* Erlangen, Germany

In the construction of power stations, the structural integrity and function of pipe systems is proved in the course of the valid regulation for specified loads. These systems consist of pipes and their supports, which transmit the loads from the pipeline to the building structure.

For the separate pieces of the supports—for example, pipe clamps or axial supports—the allowable loads are calculated.

Axial supports:

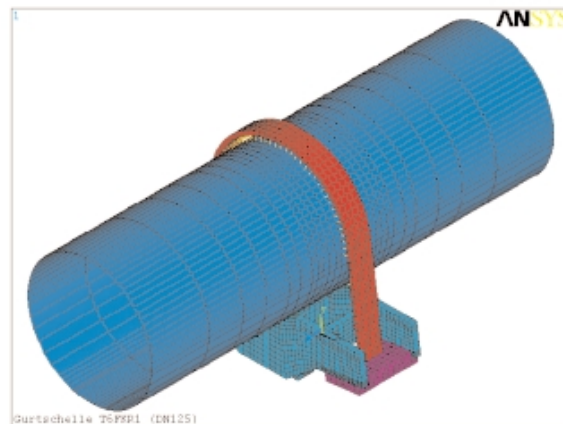
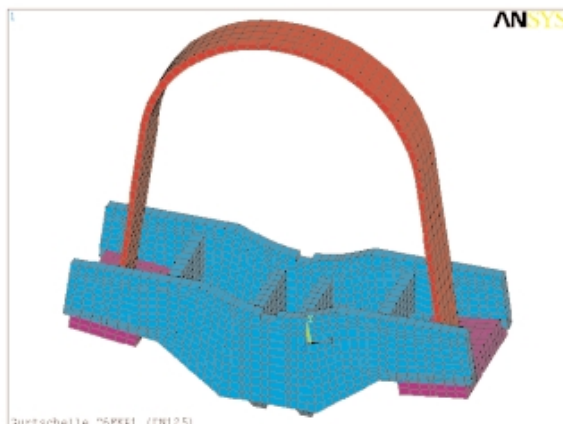
The loads from the piping to the pipe support are transmitted by pipe lugs (round lugs). During the calculation, the inclined position of the support is considered because of the lateral movement of the piping. Because the calculation is performed using the FE method, the allowable loads can be computed for this complex support.



Pipe clamps:

Apart from the calculation of allowable loads for the clamp, the deformations and stresses of the pipe due to the strain with the clamp are critically checked. There are different deformations of clamp and pipe, dependent upon the pipe wall thickness under the same stretching force. The nominal diameter of the pipe also has an influence on the stretching behavior.

A sufficient number of contact elements are used to model the contact area between belt and pipe wall, and between yoke mounting and pipe wall, so the influence of friction may be considered.



*- See company profile on page iii of this document.

Structural analysis of complex systems in nuclear power plants

H.-J. Lomoth, K. Wirtz, Framatome ANP,* Erlangen, Germany

In the structural analysis of complex piping systems, with built-in components and rather sophisticated loading, ANSYS offers the possibilities of simulating the situation in a realistic way.

The requirements for such kinds of analyses are typical for nuclear power plant pipework. Traditionally, the analyses were performed step-by-step from the global system analysis to disseminate a local component analysis from global behavior and section loads in order to perform the required assessments.

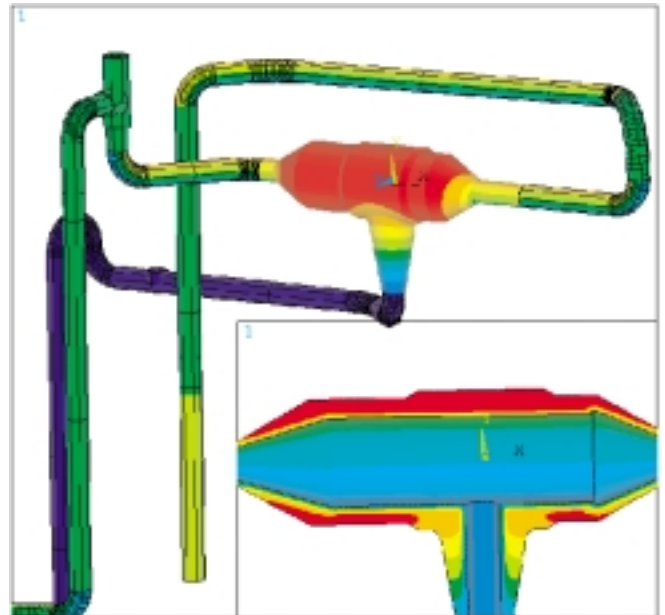
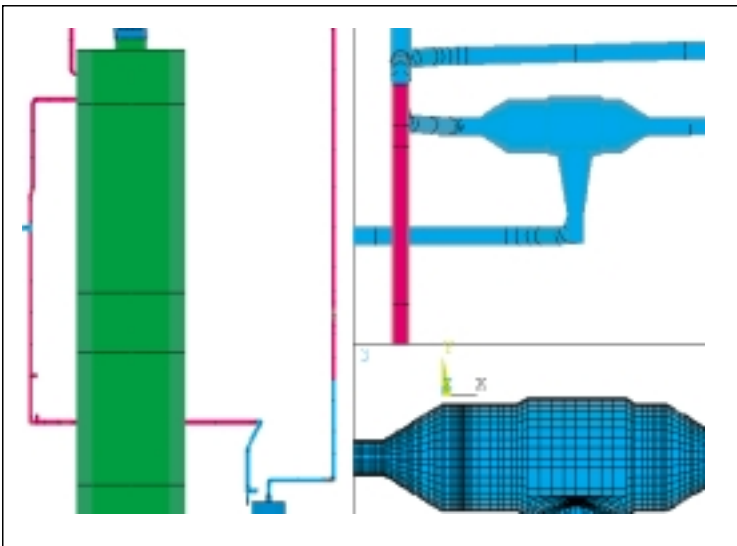
The interface loads had to be considered in a suitable manner. In the case of transient thermal loads, there is a major effort concerned with integration of the detailed component structure to the system model. This has been realized for current structural analyses performed for nuclear power plants. The essential advantages are:

1. There is only one structural model with which to work.

2. The transfer of interface loads, which was very often a compromise, no longer exists.
3. The loads and the results for the global structure, as well as for the detailed local structure, can be displayed simultaneously in a realistic manner.

We favor the ANSYS code for such applications because all the necessary tools are available for this kind of simulation procedure:

- Loads can be introduced by location and with time-dependency.
- Thermal gradients due to stratification can be applied to the available pipe and elbow elements.
- Coupling equations are generated automatically between different element types such as beam and volume elements.
- A useful possibility of programming specific demands is available with the ANSYS Parametric Design Language (APDL).



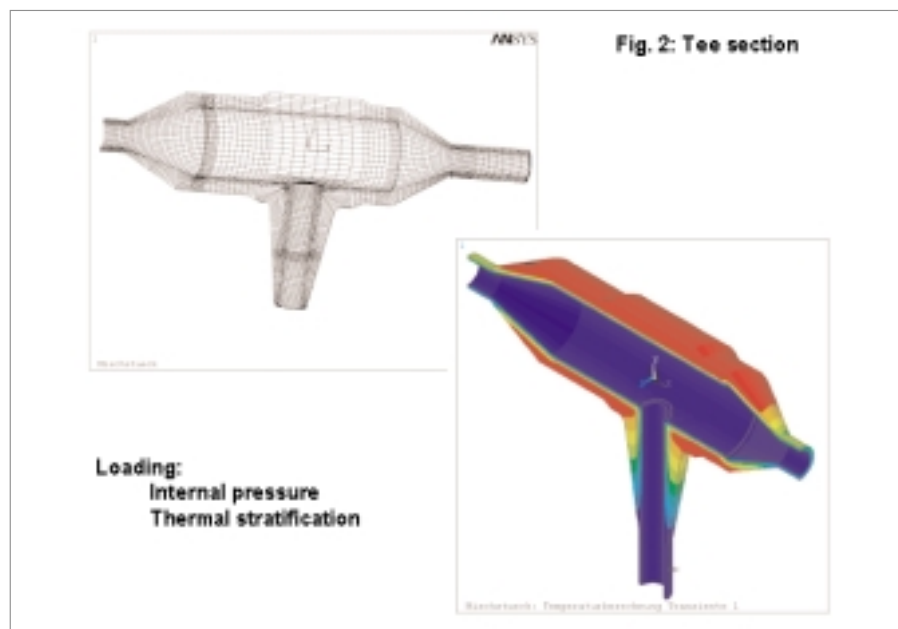
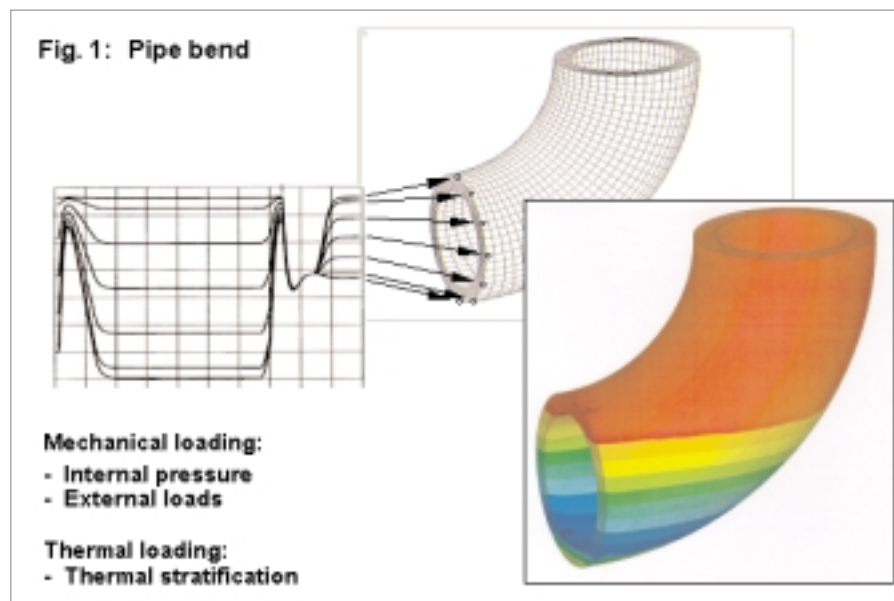
*- See company profile on page iii of this document.

Fatigue analysis using ANSYS

H. Lang, Framatome ANP,* Erlangen, Germany

The structural analysis of mechanical components or other structures is comprised of strength analysis, as well as analyses demonstrating resistance to cyclic-load-induced failure, or fatigue failure. Depending upon the complexity of the geometry and load, loading is determined by analysis, by experiment, or by numerical analysis using the finite element method.

For elastic fatigue analyses, the stress ranges must be derived from the stress history of key loading conditions and their frequency. In cases where large numbers of load events are involved, this constitutes a major undertaking. The use of suitable counting methods is required for determining realistic stress ranges and their frequencies. The determined loads constitute a load regime and are evaluated (i.e. the usage factor is determined) on the basis of the rele-



vant codes and standards. In cases where the use of an elastic analysis method does not qualify the component, the fatigue resistance can be demonstrated by the more detailed elasto-plastic method.

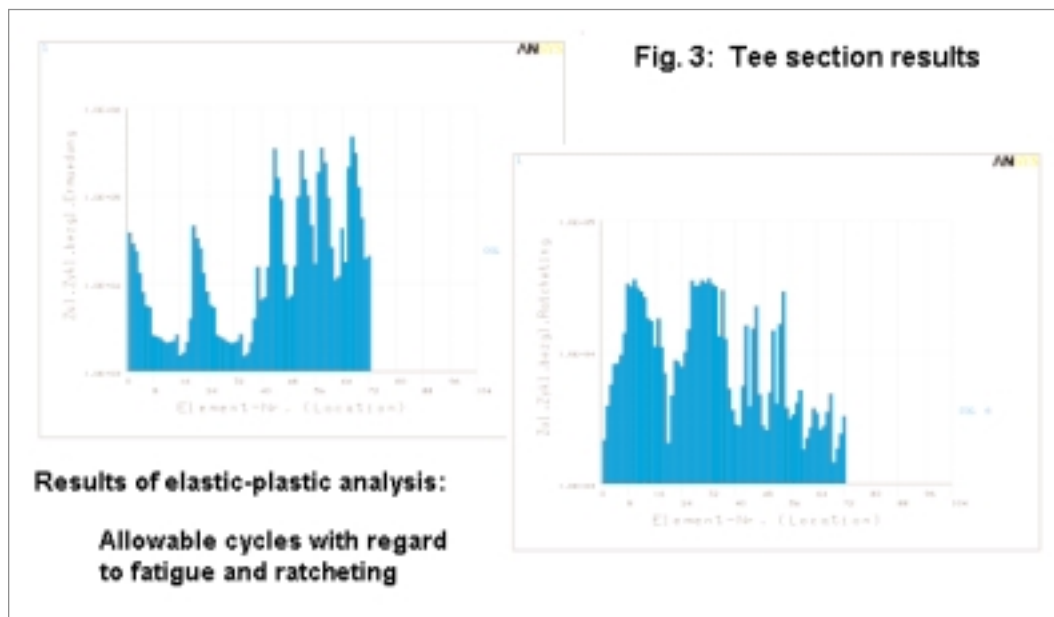
In the case of complex structures or load histories, we favor the ANSYS code for determining the detailed stress and strain histories for our fatigue assessment. ANSYS offers the necessary tools to establish and manage the required data for fatigue analysis.

A pipe bend is shown as an example in Figure 1. The FE model of the bend and the key loading points are indicated. The bend is part of a larger model that uses thermal stratification as thermal loads and internal pressure, and external loading as mechanical loads. The model also displays the history of one of the 29 transients. The seven curves correspond to the temperature history over the circumference.

The results plot shows the temperature distribution for one of the approximately 1,500 calculated load steps that make up the 29 transients.

The elasto-plastic method is applied to a tee section shown in Figure 2. The loading is made up of internal pressure, external loads and thermal stratification. Furthermore, the temperature distribution is plotted for a specific point in time.

By determining the stress or strain intensity ranges from all the calculated load steps, the specific fatigue curve must be introduced to calculate the allowable cycles as shown in Figure 3 (on the left). Additionally, a limitation of the incremental strain gain per load cycle is possible to ensure resistance of the component to ratcheting-induced failure (see bar charts, Figure 3, below).



Eigenvalues of a magnet

P. Dietz, Siemens AG,

Medical Solutions, Erlangen, Germany

One of the uses of ANSYS in the medical department was to understand and influence the expansion of vibrations inside an MR magnet. The source of the loading is inside the cylinder. The result was to place a ring, acting as a stiffener, to reduce the amplitudes of the displacements outside the magnetic field.

The figure shows the displacements of one mode shape as contours on the displaced structure.



Figure 1
Displacement contours of the magnet.

Transient thermal study of directly buried gas-insulated transmission lines (GIL)

Dr. A. Chakir, Siemens AG, Power Transmission and Distribution, Berlin, Germany

To confirm the reliability of the GIL, a directly buried single-phase line was laid in the earth on ground in Berlin at a depth of between 0.7 m and 3 m in a long-term test that lasted over six months. The total length of the pipe was approximately 100 m.

The buried GIL was cyclically subjected to high current rating (4000 A) and high voltage, which correspond to a heating and a cooling phase, in order to simulate a service period of fifty years. Extensive temperature measurements were taken in order to monitor temperatures in the GIL and also to be able to compare them with calculated values.

The GIL and its surrounding soil forms a system of thermally coupled bodies with inner heat generated from the circulating electrical current in both the conductor and the enclosure. Convection and radiation remove the heat losses from the conductor to the enclosure, while the heat transfer in the annulus by conduction is negligible. This heat compounds the losses—via the Joule effect—from the enclosure and dissipates into the soil mainly in the radial direction to the surface of the soil, where it ultimately dissipates in the ambient air by convection. The soil parameters were taken from various references about the Berlin soil properties.

Comparisons of the measured and the calculated temperatures over sixteen days, with two cycles occurring, above (TM1), below (TM3) and to the side (TM3) of the enclosing tube (Location 1) are shown in Fig. 3. The calculations concur with the measured values. The maximum temperatures rose slowly during the short cycles and, after eight days, reached 35°C. During the second period, the cooling phase was extended from four hours to sixteen hours, which was why the temperatures in the GIL system fell (Fig. 2). In this case, the maximum enclosing tube temperatures were less than 33°C.

The non-steady-state temperature distribution of a buried GIL and its surrounding soil, corresponding to a real application case, is presented in Fig.4. A double GIL system (6 phases) buried underground at a depth of 1.2 m (distance from the soil surface to the tube axial axis) and axial distance between phases is 1.3 m. The outer surface of the GIL enclosure is protected by a polypropylene anticorrosion serving layer with a thickness of 3 mm and thermal resistivity of 3.5 km/W. The thermal resistivity of the soil and backfill surrounding the GIL is taken as 1.1 Km/W. The soil temperature is taken as 35°C and the air 40°C. These values are used to calculate the summer ratings of buried systems in particular countries. The insulation gas was taken to be a mixture of nitrogen and sulfur hexafluoride (SF6) at a pressure of 7 bar (absolute) at 20°C.



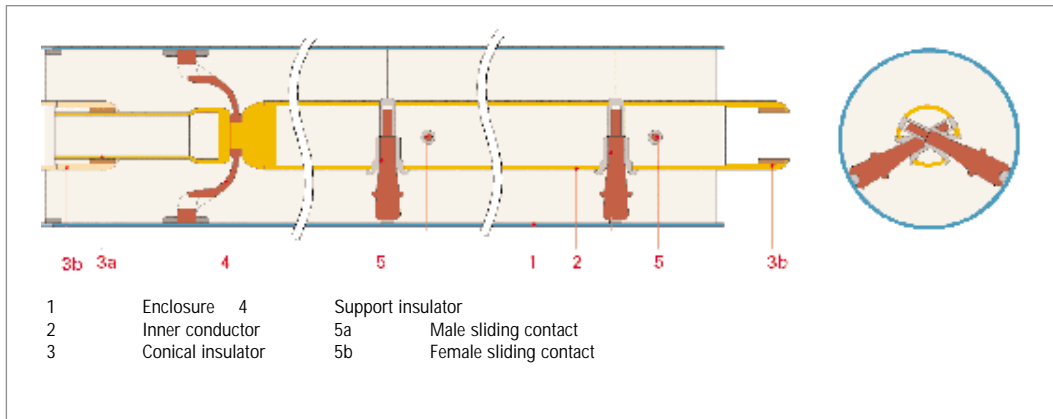


Figure 2
 Long-term test of directly buried GIL and a straight construction unit

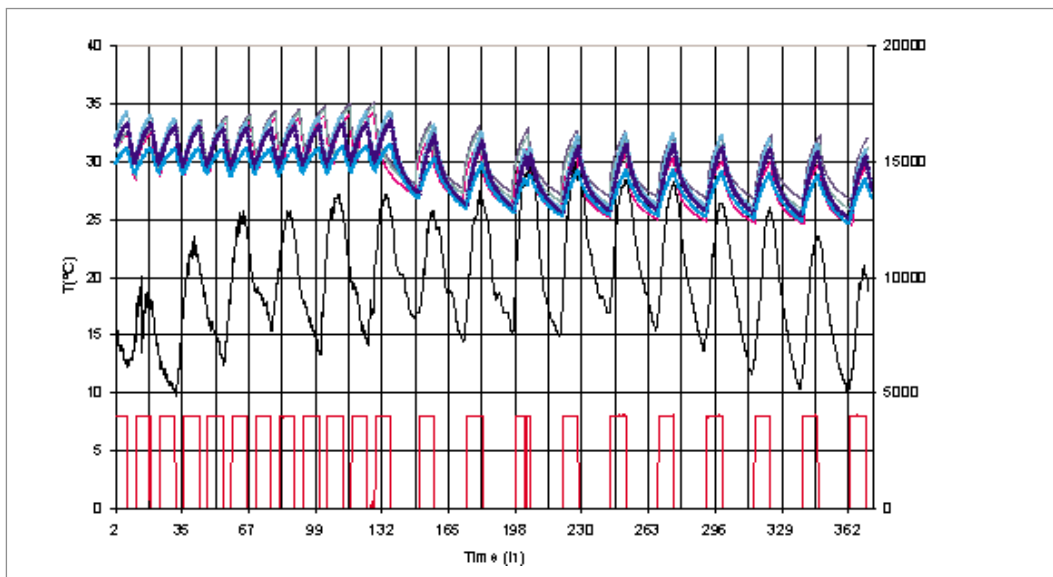


Figure 3
 Comparison of calculation and experimental results of overload current rating of a directly buried GIL long-duration test, short and long cycle.

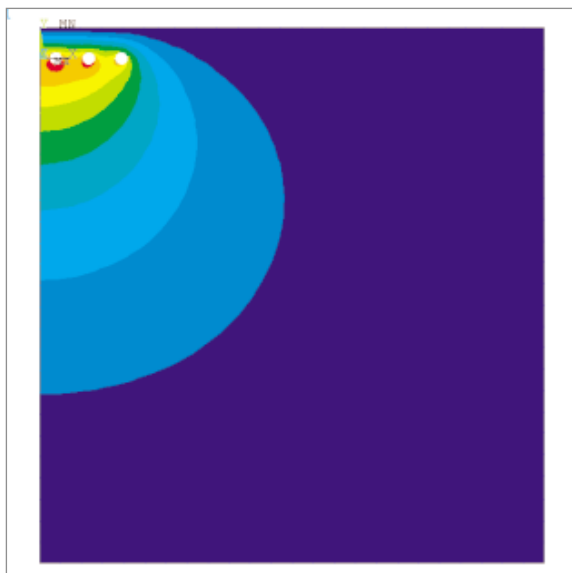


Figure 4
 Steady-state temperature distribution of a directly buried GIL by continuous current rating 1700 A, two systems (6 phases).

Temperature distribution of crossing cable systems

Dr. W. Fricke, Siemens AG, Industrial Solutions and Services, Erlangen, Germany

The current rating of a power cable system depends mainly upon the rate at which the heat produced by the cable transmission losses can be dissipated to the surroundings. In practical installations, the current rating of a power cable circuit and the required conductor cross-sectional area for a given transmission capacity are often determined by only a few thermal critical route sections. Detailed knowledge of the thermal conditions and temperature distributions within these sections can help to avoid oversizing and unnecessary material costs.

In this particular case, a new underground power cable system has to cross one that has already been installed. The

existing system shown in the lower portion of Figure 1 produces heat, due to its transmission losses, which is transferred to the surrounding soil by heat conduction. The crossing part of both cable systems—including the surrounding soil—was modeled with ANSYS using a three-dimensional model. The additional heating of the new cable system was determined to be caused by the losses of the existing one. The simulation shows a temperature rise that was approximately 30% lower than estimated with conventional simplified methods. Therefore, in this particular instance, it can be verified that the thermal critical sections do not require a larger conductor size of the new cable system than that calculated for the route sections away from the crossing.

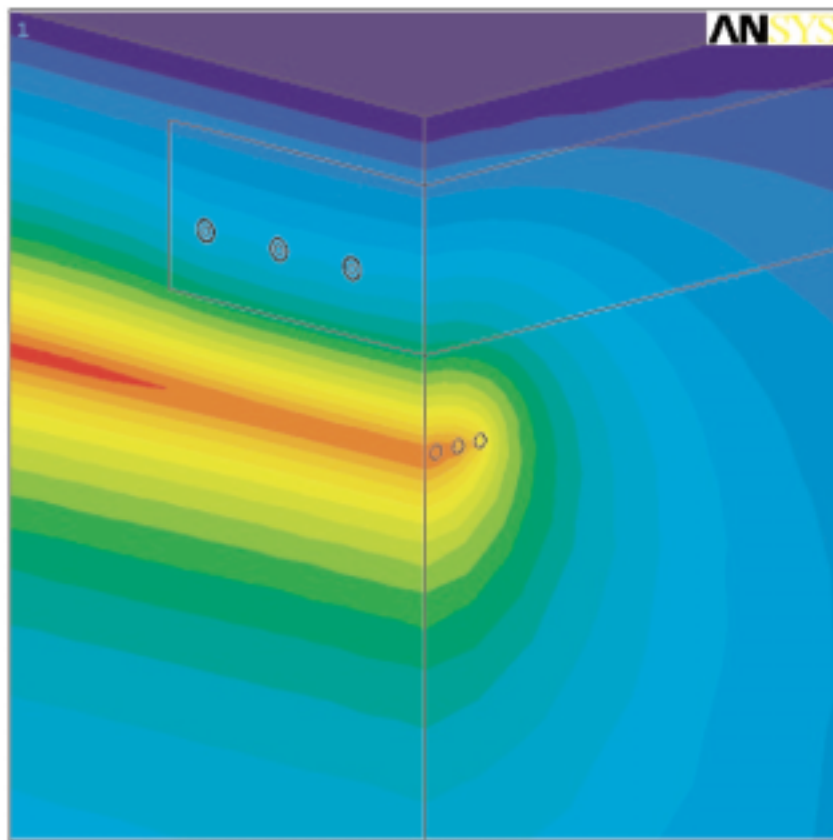


Figure 1
Temperature rise in a cable system caused by the losses of a crossing second system.

Electrothermal analysis of switching elements for a superconducting fault current limiter

R. Röcklein, Siemens AG, Corporate Technology, Erlangen, Germany

High-temperature superconducting fault current limiters work at liquid nitrogen temperature. For normal operation, the limiter is superconducting. With higher current densities (fault) the superconductors become resistive and therefore limit the current in a very short time. For a high resistance in the limiting state, the current path should be as long as

possible on a predefined substrate area. High temperatures induced during the switching operation should not destroy these switching elements. ANSYS was used to perform non-linear, time-dependent, electro-thermal calculations for the optimization of geometric parameters.



Figure 1
Experimental model
layer of gold, thickness 200 μm .

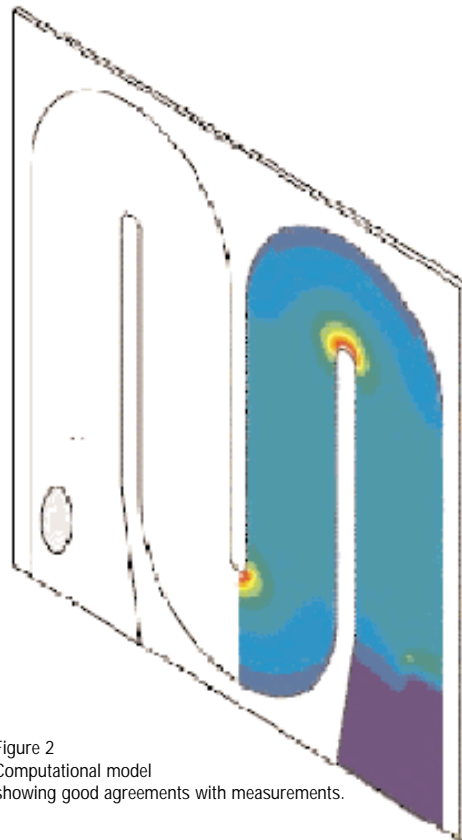


Figure 2
Computational model
showing good agreements with measurements.

Stress engineering of stencil masks for ion projection lithography

Dr. F.M. Kamm, Infineon Technologies AG, Munich, Germany

In recent years considerable progress has been made in the reduction of the minimum pattern size of integrated circuits (ICs). However, since there is a continuing demand to further decrease device dimensions well below 100nm, classical lithography technologies, based on standard optical methods, will soon reach the physical limit of minimum feature size. Therefore, alternative technologies for next-generation lithography (NGL) are currently being developed.

One prospective NGL technology under consideration at Infineon Technologies is ion projection lithography (IPL). This technology requires open stencil masks, which consist of 3 μm thin silicon membranes supported by a 700 μm thin silicon ring (figure 1). Due to the high aspect ratio of the membranes (126 mm diameter), the system is extremely sensitive to mechanical distortions. Since pattern dimensions below 100nm are to be exposed, very high requirements on placement accuracy in the range of a few nanome-

ters are given. Thus, it is necessary to both minimize and predict mechanical distortions due to mounting of the masks, and due to intrinsic layer stress of the membrane layer. For this reason finite element calculations using ANSYS/Mechanical are performed, in order to predict the distortions that appear during mask processing and mounting. The results of these calculations are used to pre-distort the stencil patterns and thus to compensate for mechanical distortions. Additionally, it is necessary to calculate local stiffness variations that result from patterning the open stencils into the membrane. Due to the removal of material during the patterning process, the elastic properties of the silicon membrane are modified and can be characterized by an effective medium with reduced stiffness. Since these stiffness variations are pattern-dependent and thus vary locally, their influence on local distortions has to be considered (Figure 2). The effective stiffness is calculated by finite element modeling using ANSYS/Mechanical.

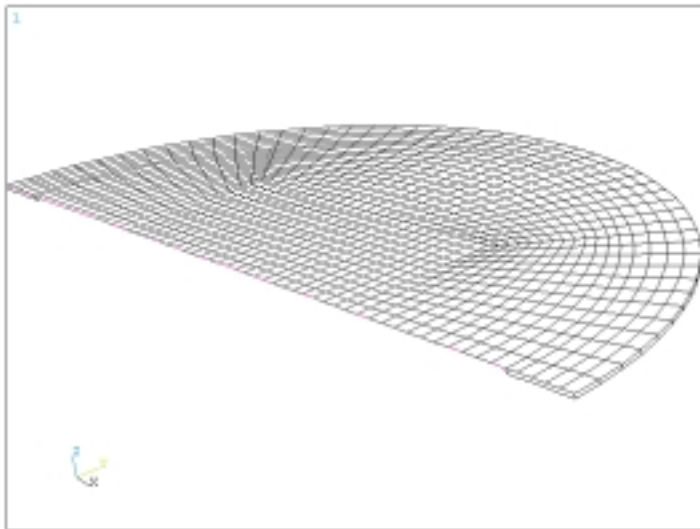


Figure 1
Cross-sectional view of the stencil mask geometry. The membrane (126 mm diameter) is formed by a 3 μm thin Si-layer, supported by a 700 μm thin Si support ring.



Figure 2
Calculation of local distortions (max. 10 nm) of a 6x17 μm mask area under applied stress. The effective stiffness is calculated by averaging over the area.

Detuning of an HF oscillator by the mechanical vibration of an acoustic sounder

R. Romatowski, Siemens AG, Information and Communication Mobile, Kamp-Lintfort, Germany

The circuit board of a modern mobile phone (Figure 1) carries many different function units (e.g. base band chipset, display, audio components, HF chipset etc.). Under unfavorable circumstances, the mechanical coupling of these components may lead to unwanted influences. In this case, the acoustic sounder (driven by a 3 kHz square wave signal) caused a malfunction of an HF oscillator by mechanical vibrations of the circuit board.

To research this problem with ANSYS, a modal analysis of the oscillator housing was calculated and the first natural frequency of the soldered housing was identified at 15.4 kHz (Figure 2), i.e. the first eigenmode of the housing was excited by the 5th harmonics of the sounder signal. By changing the material properties of the oscillator housing, the eigenvalues were slightly shifted to higher frequencies (approx. 22 kHz), so the excitation of the housing and the malfunction of the oscillator circuit was prevented.

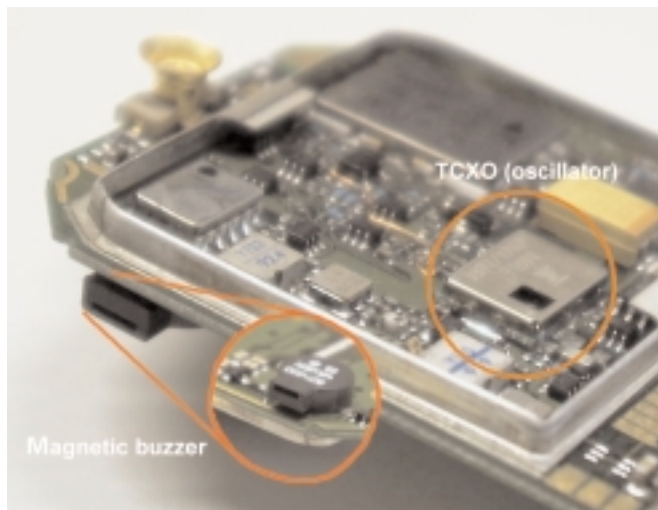


Figure 1
Circuit board of a mobile phone.

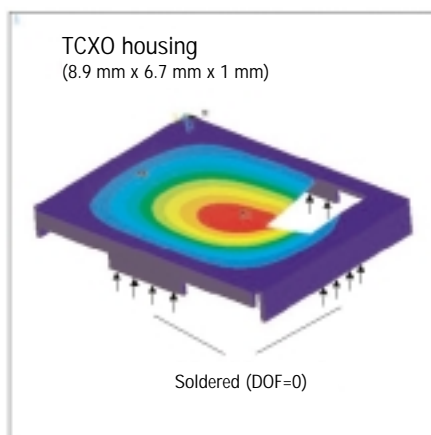


Figure 2
Modal analysis of the housing delivers the natural frequencies ($f_1=15.4$ kHz).

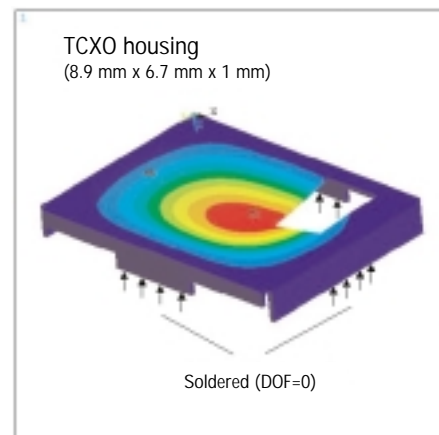


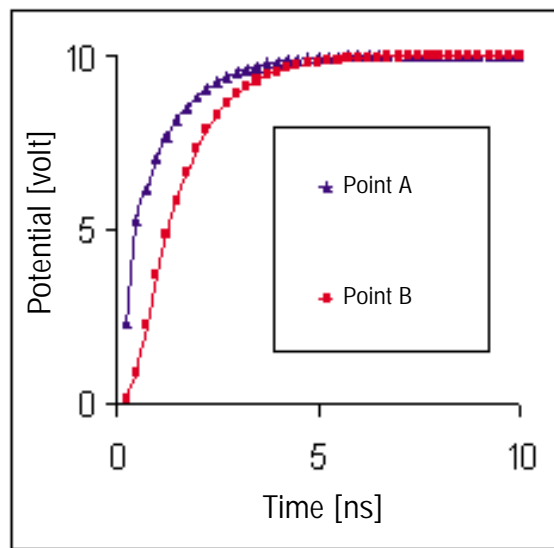
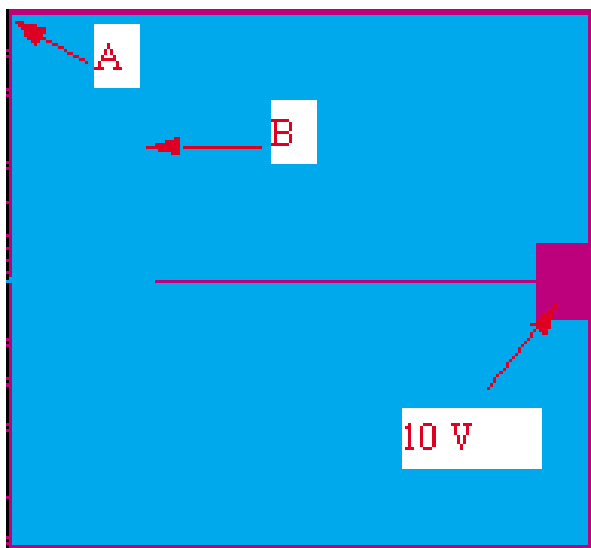
Figure 3
Modal analysis of the housing with changed material properties ($f_1=21.7$ kHz).

Potential characteristics of the gate of a power MOSFET calculated with the finite element method

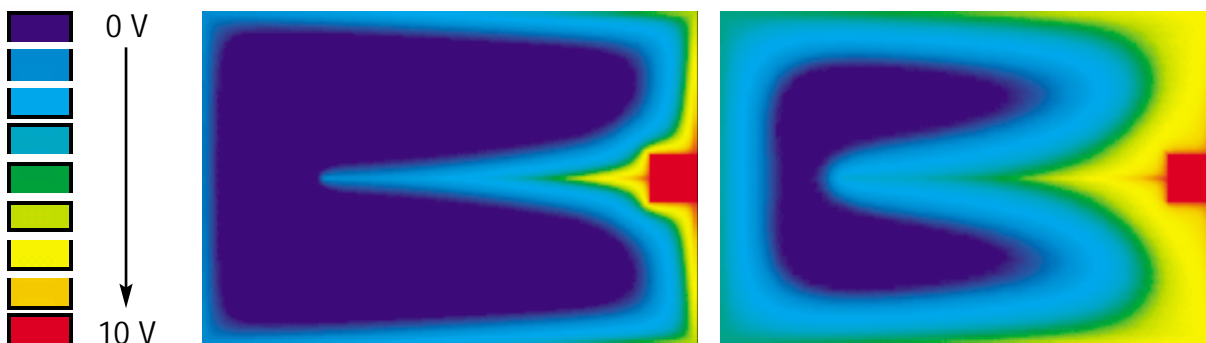
S. Ramminger, Siemens AG, Corporate Technology, Munich, Germany

To ensure the reliable and homogeneous power-on of semi-conductors, it is important to structure the gate region so that the gate voltage simultaneously reaches all the gate zones on the chip. At the first moment of power-on, the voltage (10 V) is applied only to the bond surface of the external contacts and is then raised within a few nanoseconds to increase potential in the entire gate region.

If the chip surface is considered as a 2-D resistance load, Ohm's law and the continuity equation can be used to derive a transport equation that corresponds to the heat conduction equation, with the exception of the coefficients. The ANSYS finite element simulation can then be used to solve an equivalent heat transport problem in order to optimize the gate design.



Transient 2-D finite element calculation of the potential at the gate of a power MOSFET after the turn-on pulse of the gate structure.



Calculation of the temperature distribution via a refrigerator seal

Dr. A. Feinauer, Bosch–Siemens Hausgeräte GmbH, Giengen, Germany

G. Müller, Siemens AG, Power Generation, Erlangen, Germany

The main objective of the calculation of the temperature distribution was to optimize the sealing of a refrigerator. An initial calculation was intended to show the present conditions before the geometry or the material of the sealing is changed.

The boundary conditions consisted of two different constant temperatures with convective heat transfer inside and outside the refrigerator. To keep the calculation as simple as possible, it was assumed that no convection of air is present within the complete model. Only heat conductivity was considered. The assumption that the temperature distribution over the height is constant reduces the problem to a two-dimensional model.

The existing CAD model was used to generate the finite element model. Using only an area resulting from a Boolean operation of the geometry of the half-CAD model reduced the size of the three-dimensional model. This geometry was transferred to ANSYS via IGES. There were a number of different materials that had to be defined and assigned to the different areas, regarding insulation, air inside the sealing, etc.

After defining the boundary conditions, a steady state calculation was performed. The results are shown in Figure 1. With the assumptions mentioned above, the results show a good agreement with measured values.

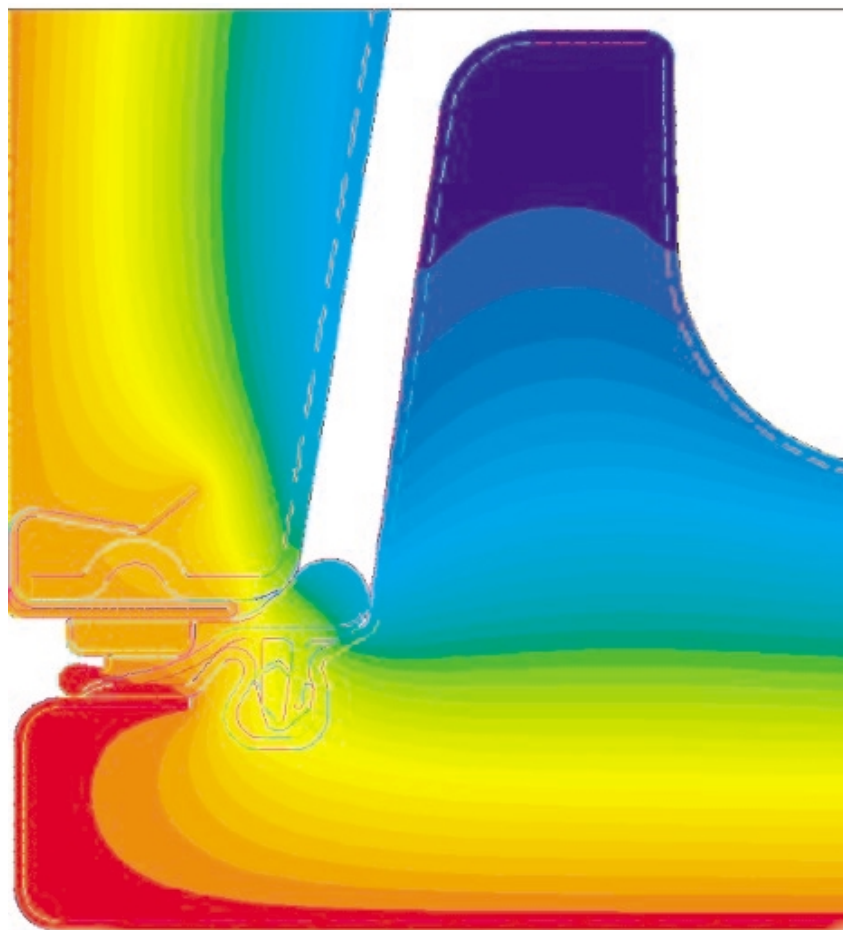


Figure 1
Temperature distribution across the sealing of a refrigerator.

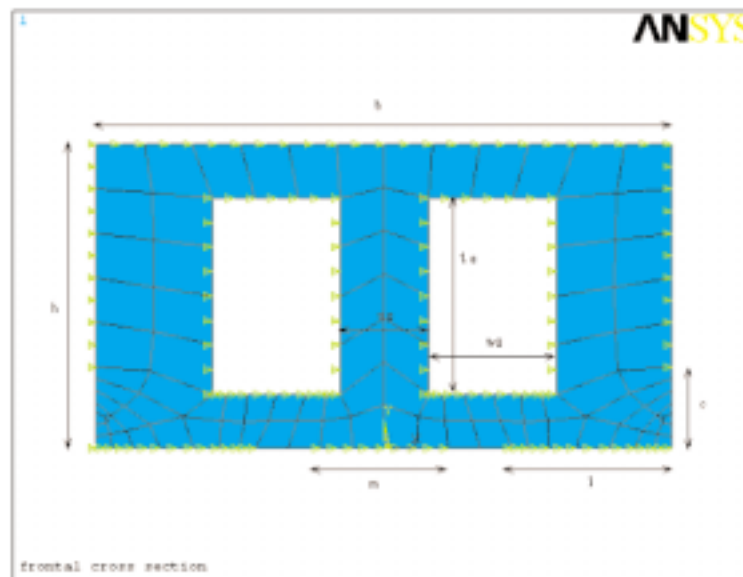
Computation of the S-parameters of a multipole filter by an equivalent circuit

I. Gavela Perez, J.M. Riedler, and M. Stadler, EPCOS AG, Deutschlandsberg, Austria

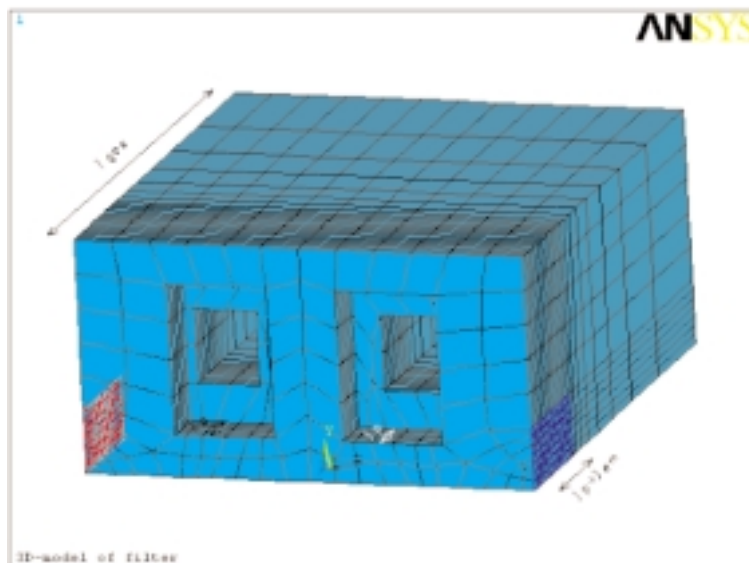
In this example, the S-parameters of a multipole filter (2-5 poles) are computed as a function of frequency by means of an equivalent circuit model. A static 2-D and/or 3-D finite element model is used to calculate the capacitance and inductance. As an approximation, only TEM-wave propagation is taken into account. Furthermore, ohmic resistances are omitted. The mutual capacitances of all metal layers are computed. Afterwards, the capacitance matrix is inverted to compute the inductance matrix. An equivalent circuit is derived. Results are compared with other programs and experiments.

2-D and 3-D FEM models:

In the case of 2-D, the three-dimensional filter is sliced into a finite amount of segments whose cross sections are plain symmetric in the z-direction. Here the frontal cross section of a two-pole filter (the green triangles designate the metal parts) is shown.



The 3-D extrusion has the following form:



Except for the frontal surface and the bottom around the IO pads, the ceramic block shown in this figure is covered by a metal surface that is on ground potential. The two holes are also covered by metal and can be considered as two coupled lines (see the equivalent circuit shown in the next plot).

Computation of capacitances and inductances:

In the computation of the capacitance matrices of a system of transmission lines (1, 2, 3, ..., n) is considered, and a ground conductor 0 that attains potential 0. Let the line i be on potential v_i .

Then, the capacitive coupling is described by the following set of electrostatic equations (the dash denotes the quantity per unit length):

$$\begin{aligned} q'1 &= c'11 v1 + c'12 v2 + \dots + c'1n vn , \\ q'2 &= c'21 v1 + c'22 v2 + \dots + c'2n vn , \\ &\dots \\ q'n &= c'n1 v1 + c'n2 v2 + \dots + c'nn vn . \end{aligned}$$

To compute the elements of the capacitance matrix, the boundary condition $v1 = -1$ on conductor 1 is applied and for all other conductors ($i \neq 1$) $v_i = 0$. After solving the finite element model we extract the charges of each conductor i. Thus, we get the matrix elements $c'11$, $c'21$, etc. Repeating

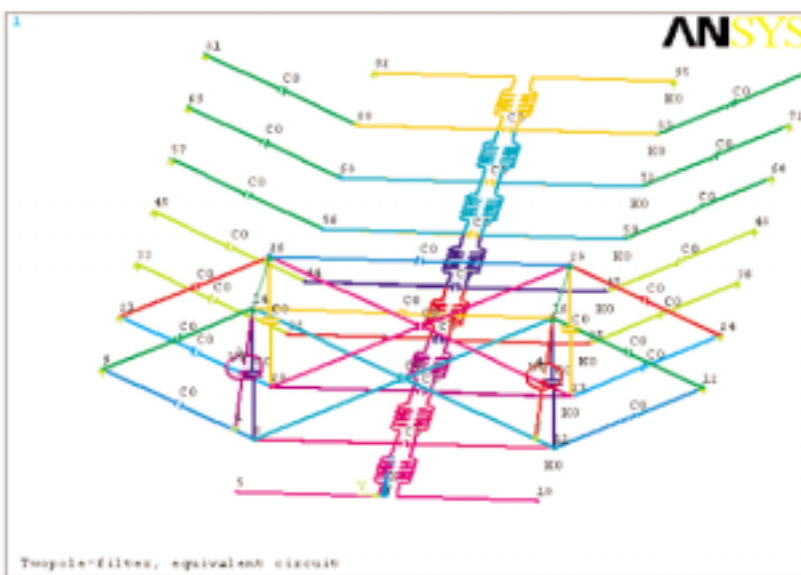
this procedure for any conductor, we get the rest of the elements. The inductance matrix can be computed by inversion of the conductance matrix and multiplication by the factor $\epsilon_r/c2$.

In the case of a 2-D FEM, the procedure is repeated for each cross section. Thus capacitances and inductances per unit length are obtained. To convert them to ordinary capacitances and inductances, a multiplication by the lengths of the corresponding segments (l_p , l_{em} , l_{eh} , ...) is required.

Equivalent circuit:

The equivalent circuit of the two-pole filter is presented in the figure shown below.

The filter is modeled by two coupled lines that are realized by a sequence of inductances and capacitances shown in the above plot. The right sequence of nodes (10,11, ... 82,95) corresponds to line 1, the left sequence (5,8,20, ... 80,92) to line 2. A voltage of 1 Volt is applied to the input port, and 0 Volts to the output port. The currents flowing through the voltage sources can be extracted from the postprocessor. They are interpreted as Y-parameters ($Y11$, $Y21$; $Y22=Y11$ due to symmetry) and converted to S-parameters by well-known formulas.



Results:

The following plot shows the S21-parameter (logplot) and the voltage standing wave ratio (linear plot) of a two-pole filter.

Some highlights:

Series circuit of inductances versus parallel: In connecting the nodes either impedance or admittance was used in such a way that the schematics show maximum simplicity.

Boundary conditions for vector potential, 2-D: It was discovered that the constraint to a constant value on a conductor is the corresponding boundary condition that yields the correct magnetic field for frequencies in the limit of small depths of the skin effect.

Boundary conditions for vector potential, 3-D: For conductors with varying cross sections, a vector potential derived as the gradient of a scalar field is the reliable boundary condition that pertains for small depths of currents according to skin effect.

De-coupled components of vector potential: For the boundary conditions mentioned above, the three Laplacian equations of the vector potential are de-coupled. Therefore, results can be obtained by also solving the scalar Laplacian equation for each component.

Parameter extraction from 3-D models: The correct procedure would be to extract capacitances from the scalar

Laplacian equation and extract inductances from vector Laplacian equations. However, this requires four times the computation time compared to a single solution of the scalar Laplacian equation. For special cases, there is no need to solve the vector potential equations.

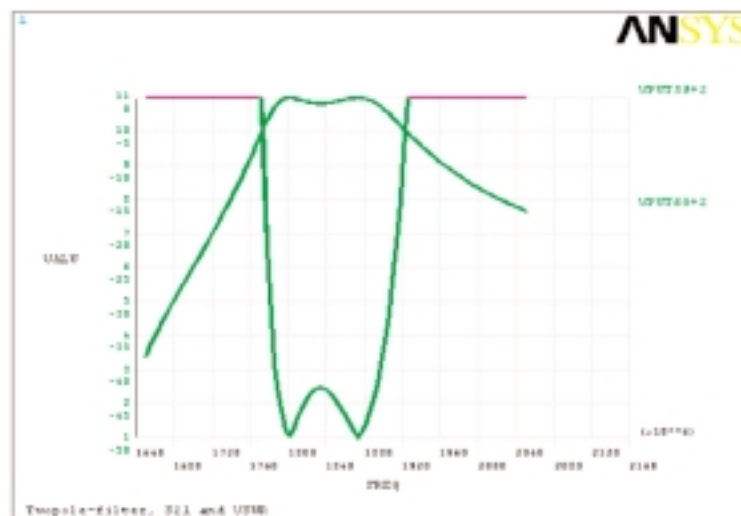
There is a simpler method, but only applicable for lines with constant cross section. Extract capacitances from finite the element model, divide by length, compute inductance per unit length, and finally get the inductances by multiplication with length.

Extend analysis to higher order modes beyond TEM: Assume a box with conducting walls, except front and back wall, and low height compared to the other dimensions.

Discretize the system by a net of inductances in x- and y-direction, where a capacitance is connected to ground in any crossing point of four inductances. Such a system gives the same results as are known from the analysis of a rectangular cavity.

Conclusion:

This method is well suited for interactive development where one wishes to get an overview of the S-parameters as a function of geometry. The computation is really fast (1 minute on a 200 MHz processor, for the two pole example). The ideas attached in this example could give rise to a new concept for simulating high frequency circuits.



Development of a parametric catenary/pantograph model

Dr. T. Reichmann, Siemens AG, Transportation Systems, Erlangen, Germany

At Transportation Systems, a parameterized FEA model of the catenary/pantograph system has been developed. It is used for the static design—as well as for the simulation—of dynamic behavior. This allows the effects of various geometric and mechanical parameters of stability and contact

quality of the construction to be studied. Design criteria are maximum bending stresses and contact force and displacements as a function of time. In this way, an optimum configuration can be achieved in an iterative procedure.

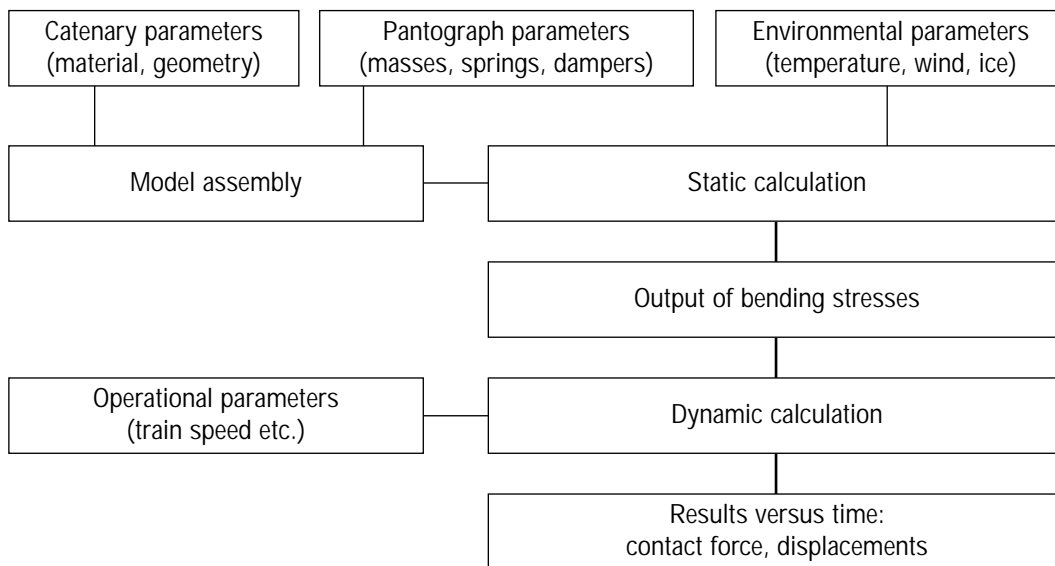


Figure 1
Flow diagram of the parametric model.

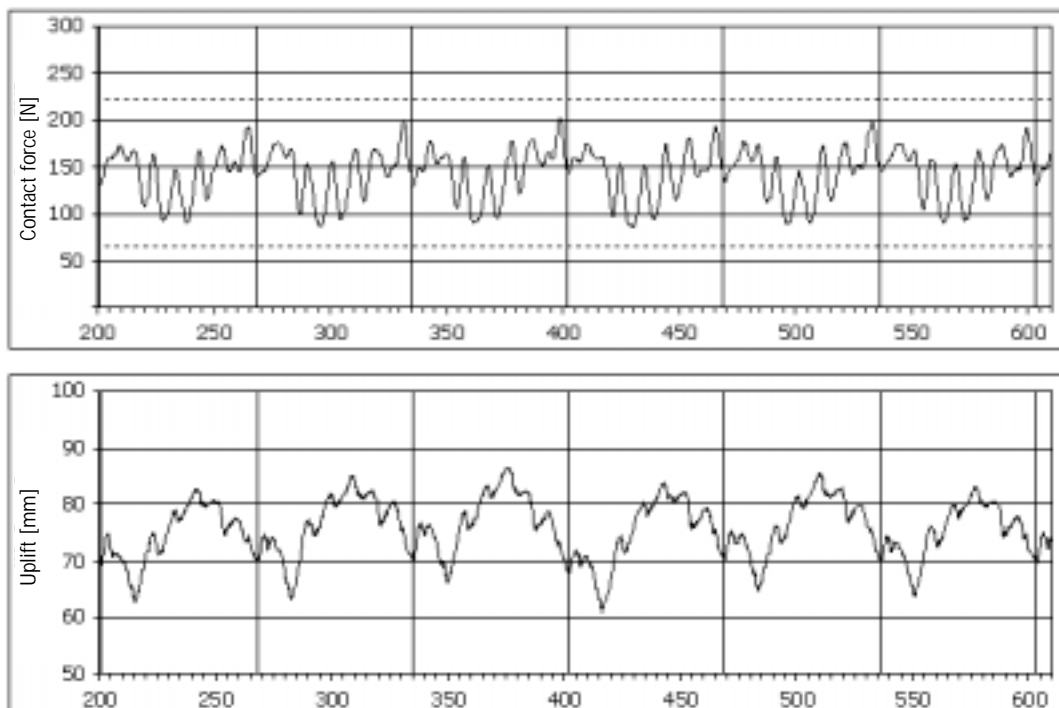


Figure 2
Results obtained from the transient simulation.

Finite element calculation of containers as part of railway wagons

Dr. H. Spanier, Siemens AG, Transportation Systems, Erlangen, Germany

The electrical components of a railway wagon are often placed inside containers located beyond the cabin. These containers should be designed to meet the loading requirements in operation.

One of the designs used by Siemens is a plug-connected system. This system consists of outside metal sheets stiffened by extrusion-pressed profiles and bolted together by some special connectors at the edges. The material used for all parts is aluminum. This complete system is joined with brackets to structure the bottom of the wagon.

In the finite element calculation, the profiles are modeled with beam elements. Shell elements represent the metal sheets at the sides. The sheets on the top and bottom are not modeled because of handling reasons. The electrical components placed inside are modeled with mass elements. The loading consists of accelerations. Mostly, the force placed upon the bolts is reviewed. Stress is also considered.

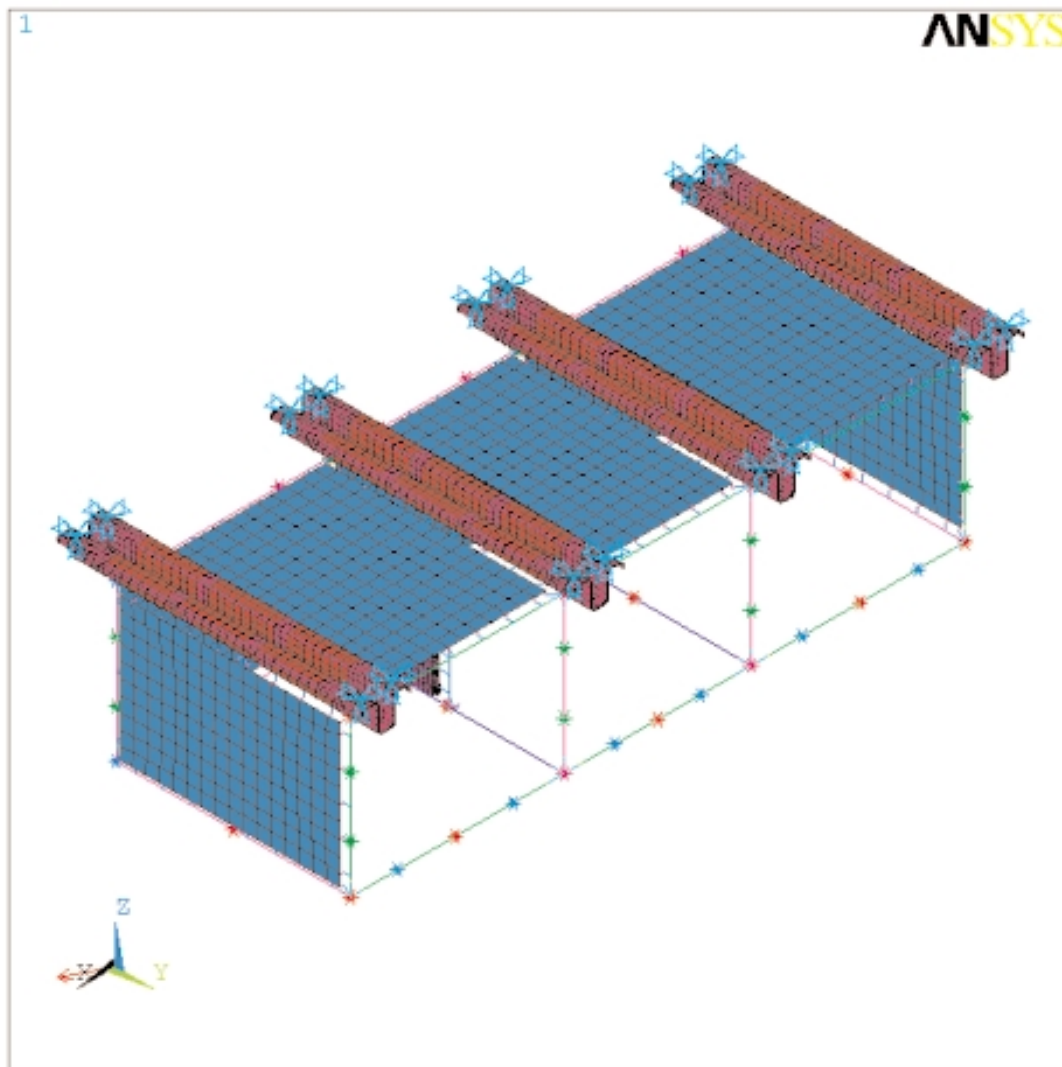


Figure 1
Finite element model of a container for the electrical components of a railway wagon.

Calculation of ride comfort of a railway vehicle including flexible car body

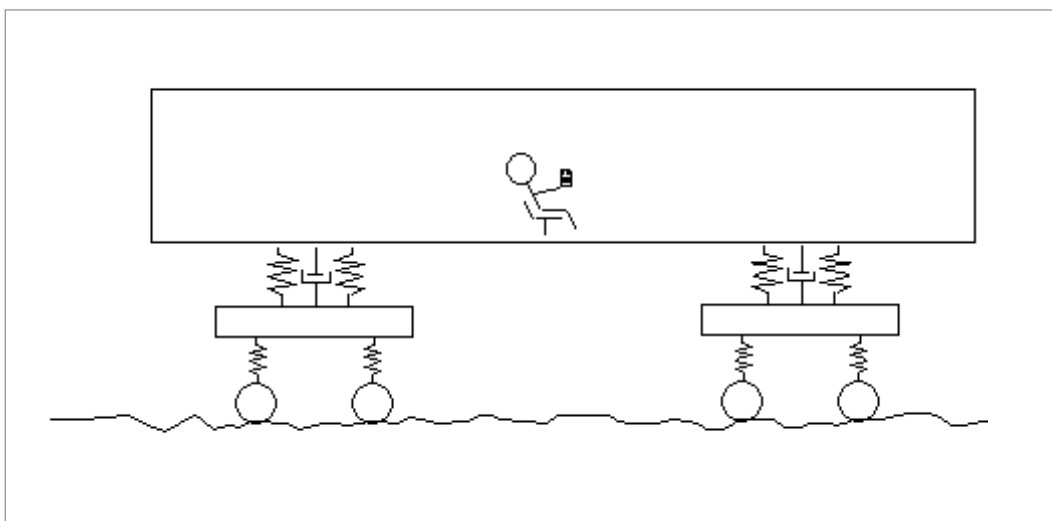
Ride comfort optimization for DESIRO-UK at Siemens Transportation Systems with ANSYS & SIMPACK
R. Gansekow, Siemens AG, Transportation Systems, Krefeld, Germany



The railway vehicle as a vibration system

A railway vehicle is designed to follow the track as accurately and safely as possible—while minimizing the transmission of track irregularities into the car body—in order to make passengers feel comfortable and prevent damage from loads. Therefore, the standard railway vehicle has two

stages of suspension, including springs and dampers, and an intermediate mass, the bogie frame. This system can reduce shocks in the car body and will be able to vibrate in its eigenfrequencies, possibly excited by the track irregularities.

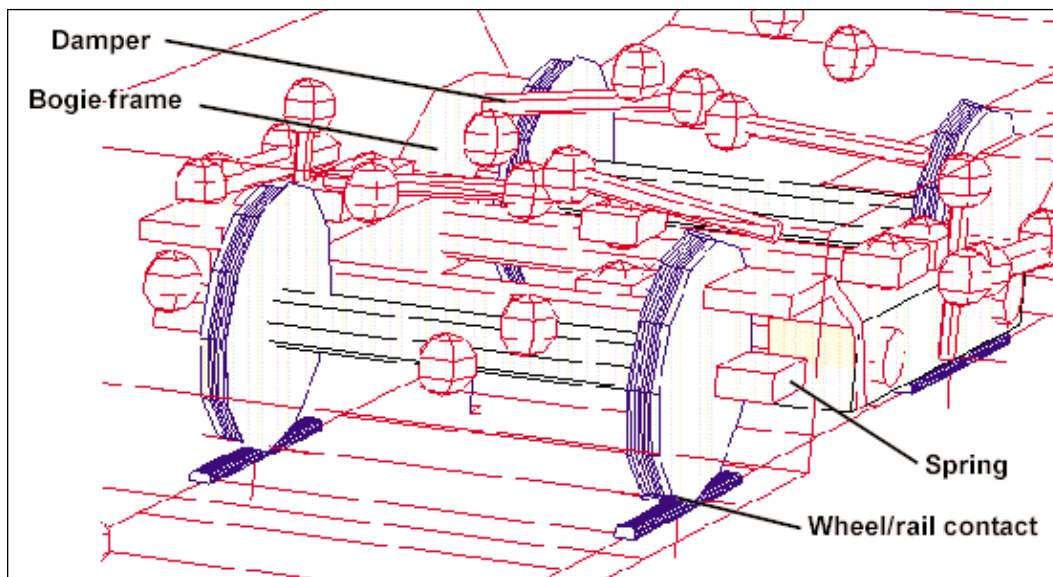


Model of a railway vehicle.

Calculation of Ride Comfort

The railway vehicle is represented by a multibody system calculation model including mass, stiffness and damping. DLR Munich has developed the multibody code SIMPACK. Special enhancements for the wheel/rail contact

have been added in cooperation with Siemens. The vibrations excited from measured or theoretical track irregularities are calculated in the frequency or time domain.



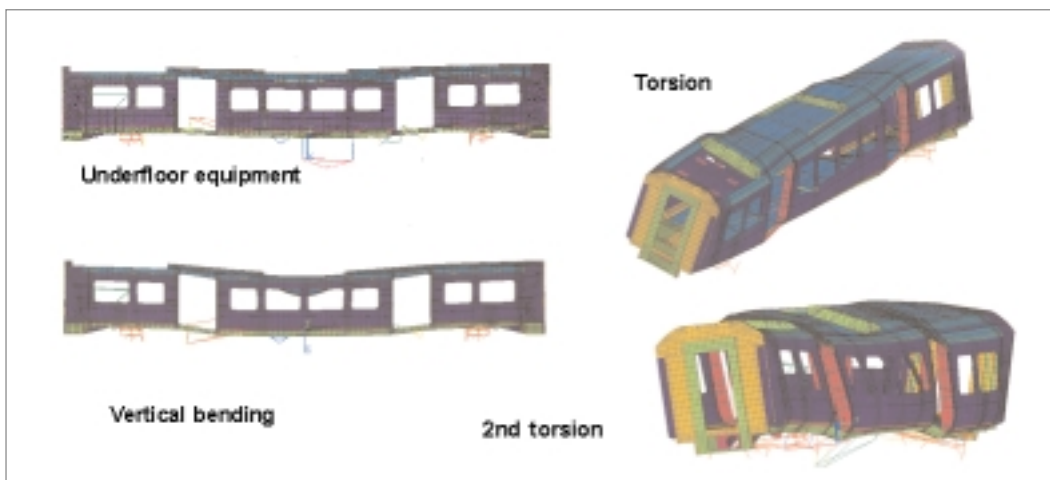
Detailed model of a running gear.

Flexible vibration modes of the car body

In predicting ride comfort, the car body can not be assumed to be rigid. The first vibration modes of the car body lie within the frequency range that affects human perception of ride comfort.

The flexible car body is modeled with the FEA method in ANSYS. A typical car body model has 900,000 degrees of freedom. The large alloy extrusions are modeled

using shells on the middle planes. The stiffness of the glass windows is included. Under-floor equipment has to be represented by the mass and mounting stiffness. Seats, carpets and interior trim are modeled by additional mass elements. Typical vibration modes are vertical and lateral bending, torsion, and floor and side-wall deflection.

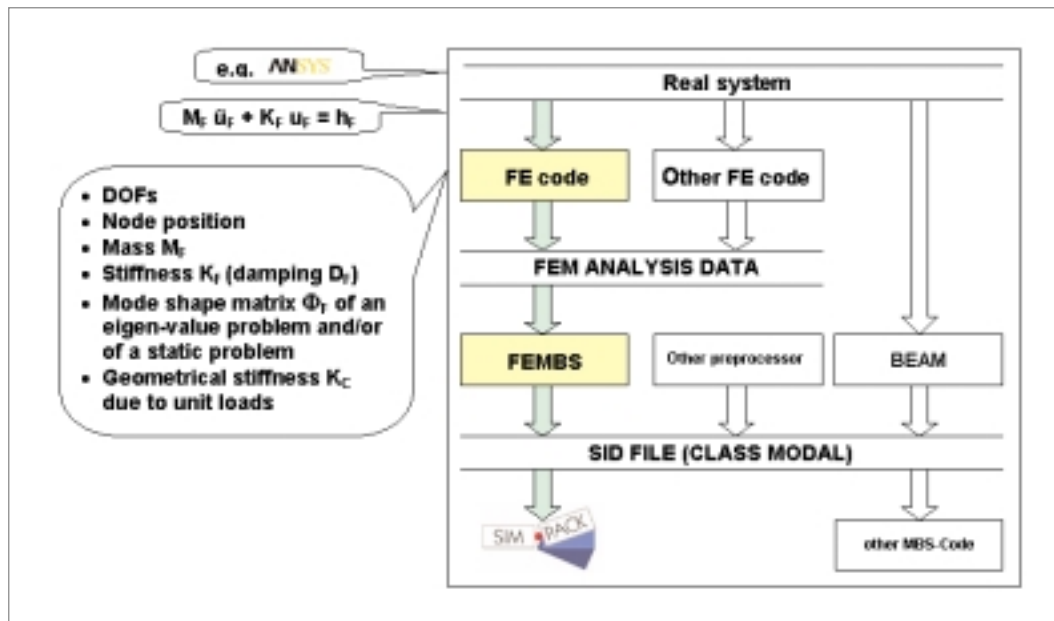


Flexible modes of car body caculated with ANSYS.

Exporting the flexible modes from ANSYS to SIMPACK

Master degrees of freedom (approximately 300-2000) are defined. The model is condensed with the static GUYAN method, resulting in a substructure having much fewer degrees of freedom. The eigenfrequencies and modes are calculated for the substructure.

Care has to be taken to guarantee the accuracy of the reduced model by carefully selecting the master degrees of freedom. The SIMPACK interface, FEMBS, reads the geometry of the substructure, and the results of the eigenmode calculation, and converts them into the so-called “SIDfile” which can be read by SIMPACK.

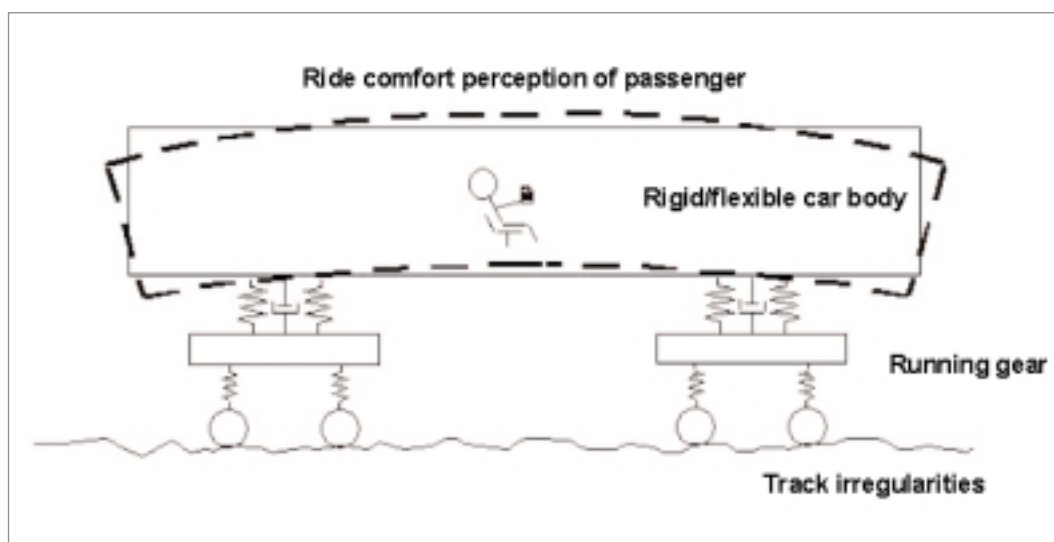


Fembs: Interface between ANSYS and SIMPACK.

SIMPACK model with flexible modes

The car body vibration modes are associated with the “rigid” car body of the multibody system.

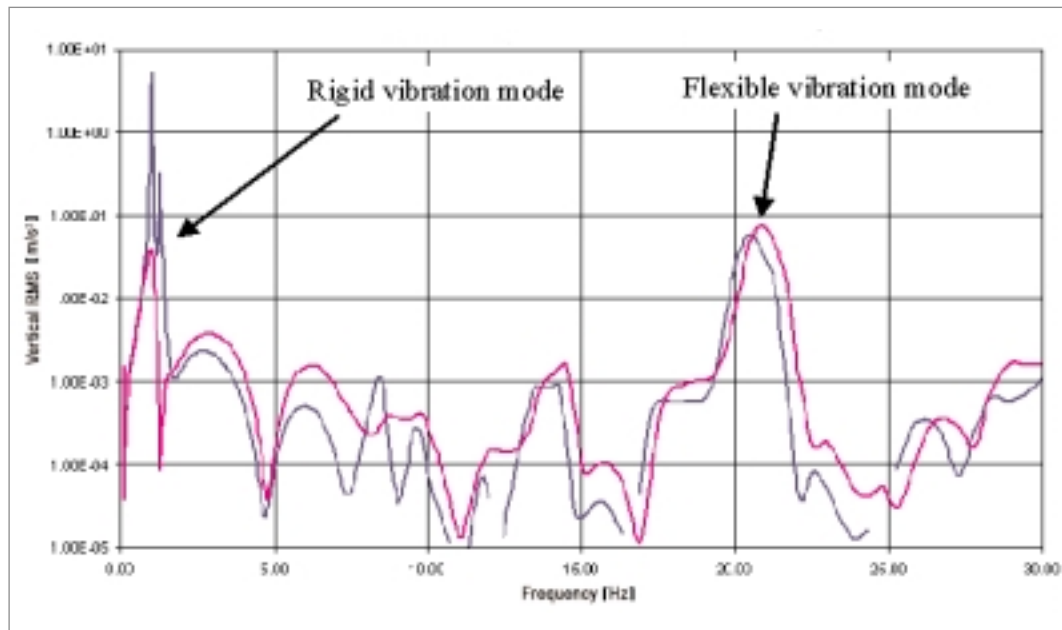
In addition to the six rigid degrees of freedom, the body has modal degrees of freedom linked to the pre-calculated eigenforms.



Multi-body system having flexible modes.

With this model the following analyses can be performed:

- Detection of the coupled vibration modes of the car body and running gear
- Ride comfort evaluation
- Running gear optimization
- Structural optimization of the car body



Vibration analysis by power spectral density.

Summary

Vibration and ride comfort of railway vehicles are evaluated using a combined SIMPACK/ANSYS model, including rigid and modal degrees of freedom. This method has been found to be very accurate, providing the model parameters have been guaranteed by the engineering process or ver-

ified by testing. Testing and risks for future projects can be reduced considerably. This is very important in railway vehicle projects because delivery schedules are short and prototype testing is often impossible.

E. DesignSpace examples

Transformer housing

R. Scheday, Siemens AG, Power Transmission and Distribution, Nuremberg, Germany

The first example is a transformer housing. The geometry was used to locate the eigenfrequencies. The material consists of steel. The boundary conditions are the fixed areas at the bottom. In Figure 1 the structure was rotated to show the fixed areas.

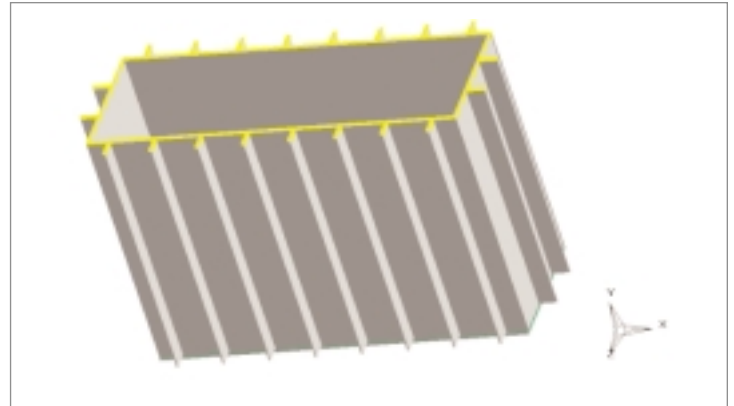


Figure 1
Geometry and support boundaries.

The same view was used to plot one of the mode shapes.

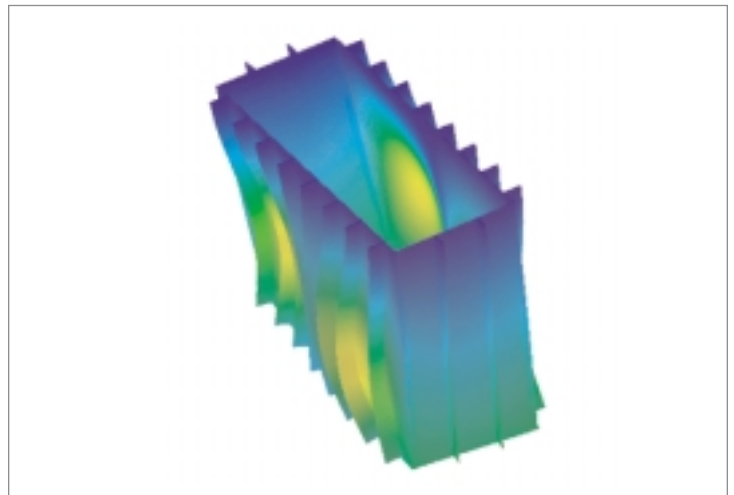


Figure 2
One of the higher mode shapes.

Stress distribution of a flange

D.W. Röder, Siemens AG, Power Transmission and Distribution, Erlangen, Germany

The stress distribution of a flange had to be analyzed because of a number of different forces with different boundary conditions. The geometry was built up with Pro/Engineer and transferred to DesignSpace via the direct interface. A number of calculations had to be performed to study the differences resulting from the various boundary conditions. One of those results is shown in Figure 1.

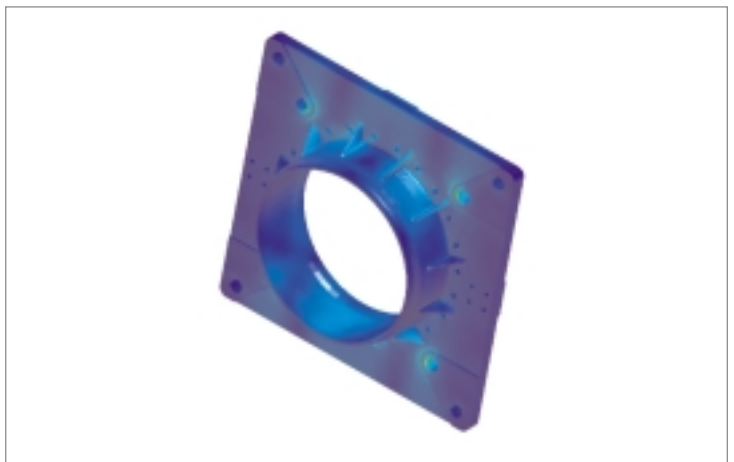


Figure 1
Stress distribution of a flange resulting from one load case in combination with one specific boundary condition.

FE analysis of a contact spring with DesignSpace

P. Janssen, Siemens Metering AG, Zug, Switzerland

This example involves designing a contact spring to be placed in a very small space inside the housing of a device. The spring-back force has to be 3N and the material must be neutral for magnetism. Because spring steel is not magnetic-neutral, the material bronze was chosen. The problem with bronze is that its yield stress value is only 0.25 of that of steel. This is the reason for concentrating on the dis-

placements. Furthermore, it should be kept in mind that the available space within the housing is very small. A first design showed the practical possibilities. Using DesignSpace, the calculation of the stress distribution results in an optimization process, with some iteration, for the designer to optimize the material behavior.

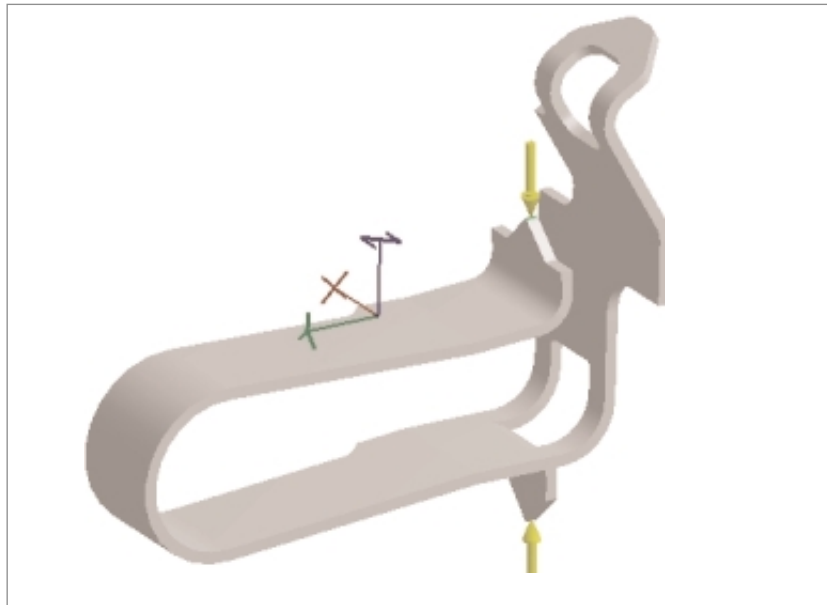


Figure 1
Contact spring with defined loads, the support is at the front side (not seen here).

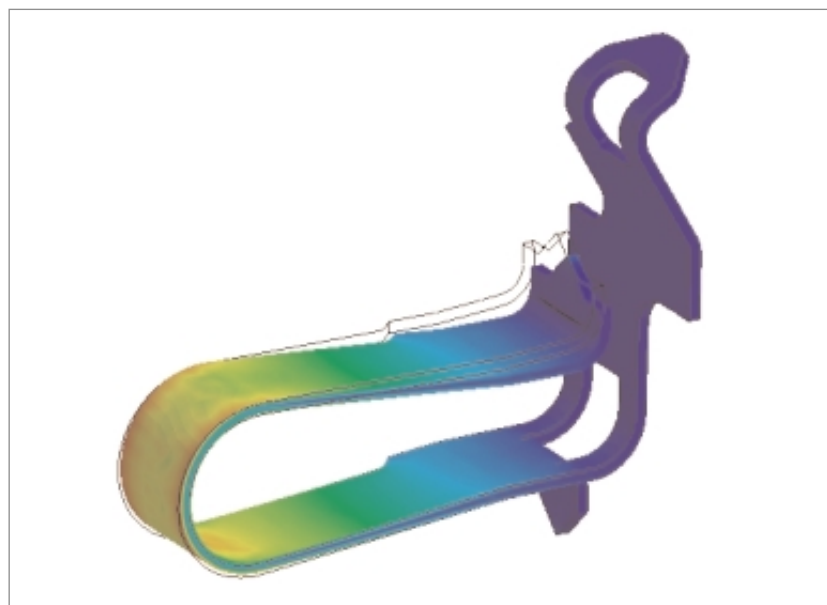


Figure 2
Stress distribution and deformations.

Turbine casing

R. Goodfellow, Siemens AG, Power Generation, Newcastle Upon Tyne, UK

At Siemens Power Generation, Parsons Works, in Newcastle, UK, DesignSpace is used to study the effects of pressure and temperature loading on turbine casings [25, 26]. The geometry is directly imported into DesignSpace from a Pro/Engineer database. Figure 1 shows the temperature distribution of a turbine casing.

The temperature distribution and an inner pressure was used as a load case to calculate the stresses. The result is shown in Figure 2.

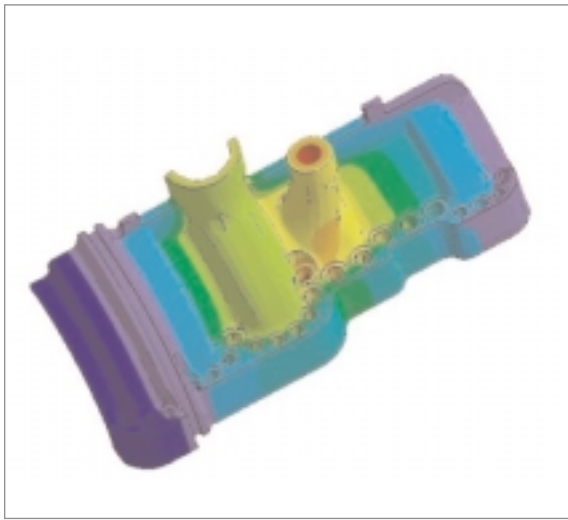


Figure 1
Temperature distribution in a turbine casing.

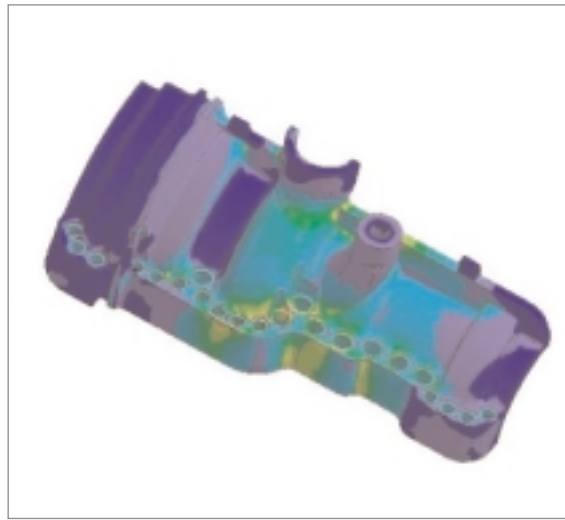


Figure 2
Stress distribution as a result of temperature load case.

Mode shape analysis of Hot Saws

P. Öllinger, Siemens AG, Industrial Solutions and Services, Erlangen, Germany

J. Lorek, Siemens AG, Power Generation, Erlangen, Germany

DesignSpace can be used to study vibration problems. In a pipe mill of Siemens', hot saws are in use. The vibration noise has to be reduced to a minimum level.

The eigenfrequencies should be out of the range of any resonance frequencies. A simple model was generated in the Autodesk Mechanical Desktop CAD system. Using the SAT plug-in of DesignSpace, the geometry was transferred. The simple boundary conditions are the clamped displacements at the inner ring. One of the results is shown in Figure 1.

This is the mode shape of a higher eigenfrequency.



Figure 1
One of the higher eigenmodes of the hot saw.

F. Acknowledgement

The author of this paper wants to thank all his colleagues for contribution of their materials. It would not have been possible to publish such a paper without the efforts of the following:

Dr. A. Chakir, Siemens AG, Power Transmission and Distribution, Berlin, Germany

P. Dietz, Siemens AG, Medical Solutions, Erlangen, Germany

Dr. A. Feinauer, Bosch–Siemens Hausgeräte GmbH, Giengen, Germany

Dr. W. Fricke, Siemens AG, Industrial Solutions and Services, Erlangen, Germany

R. Gansekow, Siemens AG, Transportation Systems, Krefeld, Germany

R. Goodfellow, Siemens AG, Power Generation, Newcastle Upon Tyne, UK

P. Janssen, Siemens Metering AG, Zug, Switzerland

Dr. F.M. Kamm, Infineon Technologies AG, Munich, Germany

K. Kullig, Framatome ANP, Erlangen, Germany

H. Lang, Framatome ANP, Erlangen, Germany

H.-J. Lomoth, Framatome ANP, now with Siemens AG, Transportation Systems, Erlangen, Germany

J. Lorek, Siemens AG, Power Generation, Erlangen, Germany

P. Öllinger, Siemens AG, Industrial Solutions and Services, Erlangen, Germany

S. Ramminger, Siemens AG, Corporate Technology, Munich, Germany

Dr. T. Reichmann, Siemens AG, Transportation Systems, Erlangen, Germany

J.M. Riedler, I. Gavela Perez and M. Stadler, EPCOS AG, Deutschlandsberg, Austria

R. Röcklein, Siemens AG, Corporate Technology, Erlangen, Germany

D.W. Röder, Siemens AG, Power Transmission and Distribution, Erlangen, Germany

R. Romatowski, Siemens AG, Information and Communication Mobile, Kamp-Lintfort, Germany

R. Scheday, Siemens AG, Power Transmission and Distribution, Nuremberg, Germany

Dr. H. Spanier, Siemens AG, Transportation Systems, Erlangen, Germany

K. Wirtz, Framatome ANP, Erlangen, Germany

Additionally, the author would like to thank the German distributor CADFEM for its continued cooperation. Finally, I would like to give special thanks my close colleagues Bernd Schröder and Joachim Lorek for all their help and Jayaweera Henry George Tissera for his consultancy.

G. Literature

- /1/ W. Oldekop, Hrsg., Druckwasserreaktoren für Kernkraftwerke, Thiemig, München, 1979
- /2/ FEN (Finite Element News) 1986, Issue 1 (February)
- /3/ Gerhard Müller, Finite Elemente – Einführung in das Arbeiten mit ANSYS anhand von nachvollziehbaren Beispielen, Hüthig-Verlag, Heidelberg, 1989
- /4/ Gabriel J. DeSalvo, Ph. D. and John A. Swanson, Ph. D., ANSYS ENGINEERING ANALYSIS SYSTEM USER'S Manual, March 1, 1975, Swanson Analysis Systems, Inc., Elizabeth, Pennsylvania
- /5/ ANSYS PC/ED User's Manual for Revision 4.2, September 1, 1985, by Robert W. Gorman, Swanson Analysis Systems, Inc.
- /6/ ANSYS-PC/ED/TH User's Manual for Revision 4.2B, Update 0, September 15, 1986, Edited by James I. Finkel, Krishna S. Raichur, Robert W. Gorman, Swanson Analysis Systems, Inc.
- /7/ W. Eifmann, U. Heeschen, G. Müller, Nonlinear, Dynamic, Plastic Component-Support Analysis 5th SMIRT Konferenz, Vol. L., 1979
- /8/ R. Engel, H.-J. Lomoth, K.-M. Pülschen, Anwendung kombinierter Finite/Finite-Differenzen Analysen bei Problemen mit Fluid-Struktur-Wechselwirkung, CADFEM Users Meeting, Luzern, 1990
- /9/ R. Engel, Ermittlung der Eigenfrequenzen von Fluid-Struktur Systemen und Vergleich mit Messungen, CADFEM Users Meeting, Bamberg, 1991
- /10/ Konrad Schramm u. Jose A. Amaral, FE-Analyse eines Rohrleitungsstützens bei transientser Temperaturschichtung, CADFEM Users Meeting, Arolsen/Kassel, 1992
- /11/ H. Lang, Beanspruchung von Rohrleitungen durch Schichtungsvorgänge, CADFEM Users Meeting, Arolsen/Kassel, 1992
- /12/ G. Schön, Ermüdungsberechnung auf der Basis von Bauteilmessungen der FEM mit dem Ermüdungsüberwachungssystem FAMOS, CADFEM Users Meeting, Miesbach/Bayern, 1994
- /13/ Robert Engel, Analysis of fluid-structure interaction problems in nuclear reactor engineering, Int. J. of Computer Application in Technology, Vol. 7, Nos. 3-6, 1994, pp. 193-205
- /14/ W. Kleinöder, Ermittlung der Temperaturverteilung im Abgaskanal einer GuD-Anlage unter Berücksichtigung von Strahlung. Anpassung des Rechenmodelles an Meßergebnisse, CADFEM Users Meeting, Bad Wildungen bei Kassel, 1995
- /15/ Hans-Jürgen Lomoth, Mechanische Analyse großer nichtlinearer Strukturen, CADFEM Users Meeting, Bad Aibling, 1996
- /16/ Th. Steinkopff, Finite-Element Modelling of Domain Process in Ferroelectrica: A nonlinear 3D Piezo-Element, CADFEM Users Meeting, Bad Neuenahr-Ahrweiler, 1998

- /17/ Peter-Christian Eccardt, Modellierung elektrischer Abschlußnetzwerke für Piezoelemente, CADFEM Users Meeting, Bad Neuenahr-Ahrweiler, 1998
- /18/ A. Chakir, H. Koch, Numerische Untersuchung der turbulenten, natürlichen Konvektion in horizontalen, koaxialen Zylindern mit CFD FLOTRAN-ANSYS, CADFEM Users Meeting, Bad Neuenahr-Ahrweiler, 1998
- /19/ J. Ulrich, Simulation mechanischer Bewegungen in Flüssigkeit mit Hilfe einer Fluid-Struktur-Kopplung, CADFEM Users Meeting, Bad Wildungen bei Kassel, 1995
- /20/ H. Moser, Dynamische FE-Analyse einer thermisch-elektrischen und thermo-piezoresistiv-elektrischen Kopplung zur Optimierung eines Hall IC's, CADFEM Users Meeting, Fulda, 1997
- /21/ J. Wibbeler + J. Otto, CADFEM, P.-C. Eccardt, Siemens AG, Simulation elastischer Komponenten im elektrostatischen Feld, CADFEM Users Meeting Friedrichshafen, 2000
- /22/ A. Chakir, Transient thermal behaviour of buried Gas Insulated Transmission Line (GIL) at variable loads, CADFEM Users Meeting, Friedrichshafen, 2000
- /23/ Barbara Schwind und Gerhard Müller, Dem kritischen Punkt auf der Spur, CAE Journal No. 4 and 5, 1988
- /24/ Gerhard Müller, Wärmeübertragung durch Strahlung, Möglichkeiten im Finite Elemente Programm ANSYS, CAD/CAM Report, Nr. 4 and 5, 1989
- /25/ R. Goodfellow, Siemens and the new power generation, Design Engineering Advertorial Supplement, June 2000
- /26/ Up-front Simulation at Siemens (R. Goodfellow), ANSYS Solutions, Fall 2000, Volume 2 No. 4, p. 11

Other published books on ANSYS in German:

- /27/ Thomas Fritscher, Wolf-Udo Zammert, FEM-Praxis mit ANSYS: Grundkurs, Vieweg Verlag, Braunschweig, Wiesbaden, 1993
- /28/ G. Müller, C. Groth, FEM für Praktiker – Grundlagen, expert Verlag, Renningen, 5. Auflage, 2000
- /29/ Ulrich Stelzmann, Clemens Groth, Günter Müller, FEM für Praktiker – Band 2: Strukturdynamik, expert Verlag, Renningen, 2000
- /30/ Clemens Groth, Günter Müller, FEM für Praktiker – Band 3: Temperaturfelder, expert Verlag, Renningen, 2000

Additionally:

- /31/ Nicholas M. Baran, Finite Element Analysis on Microcomputers, McGraw Hill, New York 1988
- /32/ J. Adam, Festigkeitslehre und FEM-Anwendungen, Hüthig-Verlag, Heidelberg, 1991
- /33/ Peter Fröhlich, FEM-Leitfaden, Springer Verlag, Berlin, Heidelberg, New York, 1995

H. Biographical notes

The author and editor Gerhard Müller started using finite element software in 1976 as a design engineer. Since 1980 he has been working as a finite-element consultant in a soft-

ware consulting department of Siemens AG. He published a book on the use on ANSYS in 1989 and several papers about the application of different programs. He is a member and co-founder of the German ANSYS User Club.



ANSYS Inc.—founded in 1970 as Swanson Analysis Systems Inc.—develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries.

The Company focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-conscious product development, from design concept to final-stage testing and validation.

Headquartered in Canonsburg, Pennsylvania U.S.A with more than 18 strategic sales locations throughout the world, ANSYS Inc. employs more than 450 people and distributes its ANSYS®, DesignSpace®, AI*Solutions,™ and ICEM-CFD Engineering products as well as its CADOE technologies through a network of channel partners in 37 countries.



Siemens' ANSYS Service Distributor in Germany, Austria and Switzerland

In the early eighties, the time of incorporation of CAD-FEM GmbH, the finite element method was still in its infancy. As this new technology went through a growing and maturing process, CADFEM's mission was to take finite element methodology to German companies. Since the founding days, the CADFEM name has always stood for excellence in client service and product quality, and we take special pride in being the preeminent finite element service provider in Germany.

Software – No matter what your analysis needs look like, CADFEM's products like ANSYS, DesignSpace and others cover virtually the full spectrum of finite element application.

Service – Finite element analysis can be tough. But with CADFEM's education, documentation and support you have one less thing to worry about. Whoever wants to learn finite element methods, at CADFEM you can join a seminar. And when the numbers get tough, the support group is ready to assist via phone, fax, e-mail, internet or on site.

Consulting – Let the CADFEM project team do your jobs. Here's why: Experienced specialists are available to help when clients face bottlenecks or require specific know-how. If you can't wait to apply finite element technology in its full range, CADFEM offers relief. Let's join forces, there's safety in numbers.

Contact

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
USA

Phone +41-724-746-3304
Fax +41-724-514-9494
e-mail: ansysinfo@ansys.com
www.ansys.com

CADFEM GmbH
Marktplatz 2
D-85567 Grafing near Munich
Germany

Tel. +49(0)8092-7005-0
Fax +49(0)8092-7005-77
e-mail: info@cadfem.de
www.cadfem.de



Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
ansysinfo@ansys.com
Europe
eu.sales@ansys.com

Toll Free USA/Canada:
1.866.ANSYS.AI
Toll Free Mexico:
001.866.ANSYS.AI
Europe:
T 44.118.9880229



Publication subject to change without prior notice.
ANSYS and DesignSpace are registered in the U.S. Patent and
Trademark Office. All other trademarks and registered trademarks
are the property of their respective owners.
©2002 SAS IP, Inc. All Rights Reserved. Printed in U.S.A.
MWP0008MAR02