Chapter 10
Major Technology 4: Computer Aided Engineering—CAE

Executive Summary

This chapter deals with the following topics:

- Basics and advanced techniques of Computer Aided Engineering (CAE)
- Providing insight into how engineers benefit from using CAE technologies
- Describing functioning, benefits, and limitations of CAE technologies in practice.

Quick Reader Orientation and Motivation

The intention of this chapter is:

- to give an overview of CAE technology in Virtual Product Creation as driver and enablers for Digital Transformation in engineering
- to present CAE technology as part of Virtual Product Creation from a practitioner's point of view to analyze the need and usefulness for day-to-day industrial work practice
- to give instructions on how to use CAE technology
- to explain models, frameworks, and
- representations that help to grasp the internal working modes of CAE technology.

In modern virtual product creation, functional and behavior simulations of products (and associated services) play a foundational and constantly increasing role. The need for not only creating a virtual model, but also "studying its behavior in reality" as well as "improving and streamlining its structure" [1] motivated scientists and engineers to develop advanced methods and software tools. Whenever a product has to be designed, there exist a long range of different kind of functional and behavioral requirements this product needs to fulfil. Furthermore, the entire manufacturing process is highly influenced by the shape and assembly structure of the product. Therefore, it is vital to predict if the product behaves as functionally intended, but also how it can be manufactured and assembled. Computer Aided Engineering (CAE) designates the state-of-the-art tool and methods which are available to conduct such analysis and prediction tasks.

© Springer-Verlag GmbH Germany, part of Springer Nature 2022

R. Stark, *Virtual Product Creation in Industry*,
https://doi.org/10.1007/978-3-662-64301-3_10

10.1 Background and Evolution of CAE

Managing and controlling execution efficiency and availability of resources as part of virtual product creation is a major task and delivery of PLM (Product Lifecycle Management) IT solutions. Hence, it was introduced broadly in industry and became indispensable for virtual product creation in industry during the last two decades. CAE can be considered as an integral part of PLM and Virtual Product Creation (compare Chap. 4 *"Virtual Product Creation—what is it"*) that provides methods for engineers to simulate a product's behavior under real conditions. Originally, *CAE was a term used to describe the procedure of the entire product engineering process, from design and virtual testing with sophisticated analytical algorithms to the planning of manufacturing*. However, it became clear after some years, that the nature and diversity of both, model and file management as well as analysis and simulation tasks, do differ significantly from those of the design and product structure, BOM (Bill of Material) and the manufacturing process. Consequently, the terms *PLM* and *Virtual Product Creation* were created to extend the original CAE procedure idea to the full landscape of the entire product lifecycle (compare Chap. 4 *"Virtual Product Creation—what is it"*).

With the rise of powerful computers in the late 70s, it became increasingly possible to calculate large numerical problems. The development of the Boeing 777 in the early 1990s can be seen as one of such corner milestones. This was the first extremely complex product entirely virtually designed, where also a digital mock-up (DMU) was developed (compare Chap. 12 "Digital Mock-Up"). In automotive industry, e.g., extensive CAE simulation was driven by highly increasing occupant safety requirements, which led to early technology, compute centers in the 70ties and 80ties and then to a major thrust in the 90ties by intruding powerful UNIX workstations and Cray supercomputing.

Today's CAE software landscape is partially traceable to the development of application-specific tools by specific corporations within the IT and PLM vendor market place. Unlike in the past, even large OEM corporations no longer develop their own CAE codes but rely on the tool competence of IT and PLM vendors. Interestingly enough, there exist still today many hidden CAE kernel applications which were programmed initially by universities institutes or by CAE expert teams of companies in the 70ties, 80ties and 90ties of the twentieth century: some of them remain within their originally code (e.g. Fortran) and would need major refactoring in order to be transferred to modern software code architectures.

Today there also exist highly specialized service providers, who concentrate on the development of CAE software for different technology branches, without actually being active in product development themselves.

10.2 Engineering Understanding of CAE

The use of computer-assisted simulation reduces financial risks, which are generally connected to product development. According to [2], product development from a corporate perspective is mainly centered on the action of making an investment, and the corresponding expectation of future profit. CAE promises an efficient and goal-oriented mode of working, as well as monitoring and increased control over the development process.

10.2.1 Why Does an Engineer Use CAE?

Unlike in the past, when engineers were dependent on building physical prototypes in order to test the future final product properties and behaviors, engineers nowadays can rely on the capabilities of CAE solutions to simulate upfront the relevant product functional performance capabilities. Such front loading of computational engineering capabilities with the help of CAE solutions helps engineers to avoid unnecessary and costly engineering iteration cycles and costly modifications to physical prototypes (see Figs. 10.1 and 10.4).

The process of product development begins with the idea and its drafting in CAD as depicted in Fig. 10.1. To reach the final goal of physical manufacturing, the product needs to be designed and evaluated in terms of all critical requirements such as static loads, durability, heat resistance etc.). Without the use of CAE, calculations were traditionally done manually (e.g. based on algebraic mathematics) incl. heuristics and experiences from respective literature or company knowledge. Once all preparation methods have been completed, a prototype is made, which is then tested under realistic conditions. The insights gained here then possibly result in the prototype being redesigned, in which case the iterative design process would start from the beginning until all conditions are satisfied before then serial production can begin. In such traditional process, all tests had to be done with the physical prototype, which has the following disadvantages:

- Significant time delay due to the lead-time to manufacture and assembly physical prototypes.
- Costly operations to procure and product all physical components.
- Significant efforts and time delays to loop back findings of the test results at the physical test stand into the digital master file (CAD).

The early use of CAE (see Fig. 10.1 on the right side) significantly improves the conception and embodiment phase prior to prototyping, as well as speeding up the entire process. The calculation and behavioral prediction in realistic conditions allows for optimization before the prototype is even created. The numerical methods used by CAE can make assertions that are more precise and that help to check and verify individual scenarios and circumstances early on. It is, however, necessary to be

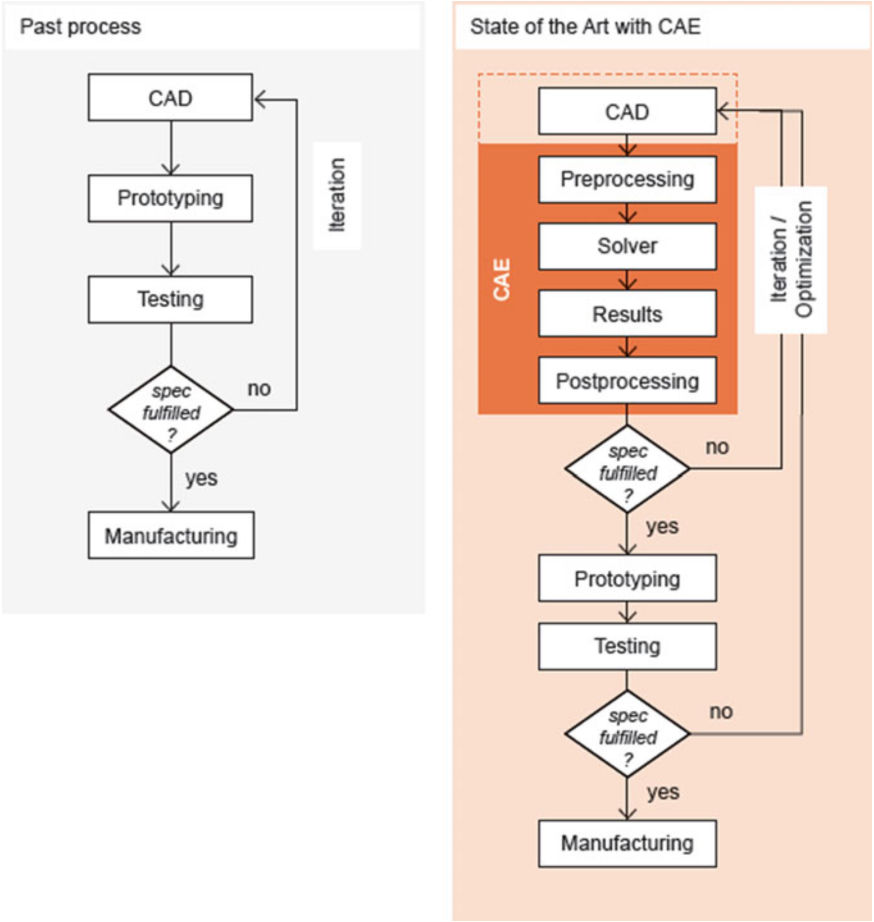


Fig. 10.1 Front-loading with upfront CAE to reduce iterative engineering cycles

proficient in all CAE methodological steps and procedures (such as all four steps in the orange box: pre-processing, solver and simulations control, results management and post-processing). The earlier insights can be generated how a component or specific feature will function within a construction unit, the more effective the development process can be (see the principle in picture in Fig. 10.2).

Timely knowledge prevents making last-minute stressful and costly changes in the end, which could even impact on other areas of the project, too. An analysis of the dependencies and interconnectedness of components as part of the CAE model is important to be determined early on.

Early decisions regarding the general direction or the type of problem-solving approach (compare [3]) affect more than 80% of the product life cycle costs. This is why it is important to be able to predict costs and technical feasibility (function,

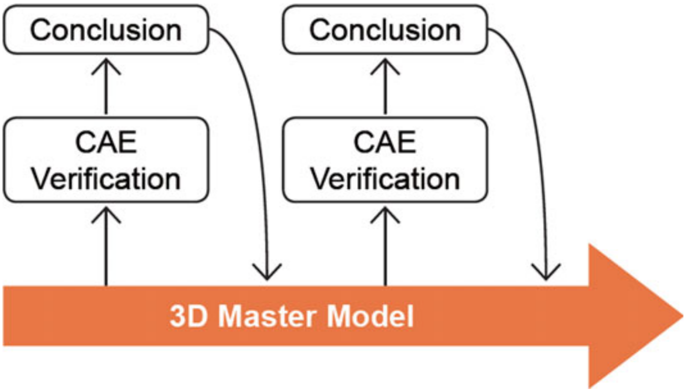


Fig. 10.2 Effect of CAE conclusions on the product development process

behavior, performance, manufacturability etc.) of a product concept early on. The growing complexity of technical systems, as well as the desire for higher levels of employee productivity, require increasingly more powerful digital and virtual solutions based on IT resources (compare Chap. 5 *"The technology history of Virtual Product Creation"*). The continuous advancement of information technology with faster and more robust digital processes, as well as higher degrees of virtual model details and network data rates, provide businesses with innovative solutions to build and use simulation models. The potential for numerical calculations of scientific or technical problems is increasingly being discovered and will continue to evolve. Simulations follow the idea that *a complex problem reduces to a series of greatly simplified problems* [4]. For this reason, the computer simulation of such processes and engineering problems is particularly well-suited and has meanwhile reached a full mature level. Therefore, CAE has been evolving from an exclusive expert skill set to a widely used engineering capability across different degrees of engineers, designers and analysts.

10.2.2 What is CAE Doing for an Engineer?

CAE is meanwhile used as a standard engineering validation and verification solution in different industries and technical applications: in classic machine-construction industries, in automotive, aviation, aerospace and maritime industries, a diverse set of products such as vehicles, aircrafts, ships, machine tools, pumps etc. a high variety of CAE analysis templates exist. CAE is used both, for the product development itself, as well as for production planning of the products. Production planning stretches to industrial areas such as material sourcing, storage, logistics and disposal. This is why product data management (PDM) and product lifecycle management (PLM) are becoming increasingly important for manufacturing enterprises (compare the

Sect. 5.3 *"Product Data Management"* and Chap. 11 *"Product Data Management and Bill of Materials"*). The result is a desire for more integrated software solutions that can deal with the wide spectrum of tasks [5]. A market overview for CAE software is for instance provided by [6].

The CAE software application landscape and its associated model fundamentals are diverse and offer different technology foundations. Due to the fact that several scientific disciplines and technical branches have developed their own solutions sets based on individual demands for numerical calculation methods, different CAE product and research prototypes have been developed and are still under new development. Figure 10.3 illustrates the different CAE disciplines and simulation types:

As shown in Fig. 10.3, structural analysis is one of the major CAE disciplines. This field of expertise is focused on the simulation of components or structure regarding specific physical phenomena. This can include analysis of components under static load, acoustic analyses, and questions regarding thermodynamics, fluid mechanics or electromagnetism. Here it is important to document the behavior of a material or a continuum in a particular state of aggregation. The structures are generally set in a three dimensional space, meaning that the calculations and models are based on a 3-D case.

In solid mechanics, materials are generally analyzed that correspond to the Hook's law. Distortions in the purely elastic area can be solved with linear numerical methods. Plastic distortions can be calculated within limits in a reasonable way. However, outside those limits non-linearity of material laws lead to a significant increase of the associated calculation efforts. In fracture mechanics other laws apply, which require their own specific methods. Variables to be calculated include tension, elongation and displacement. For solidity analysis, usually the *Finite Element Analysis* (FEA)—here the mathematical principles of virtual displacements of small finite elements are applied—is used, but the *Boundary Element Method* (BEM)—using the mathematical equations of integrals—can also be leveraged if the volume is rather thin.

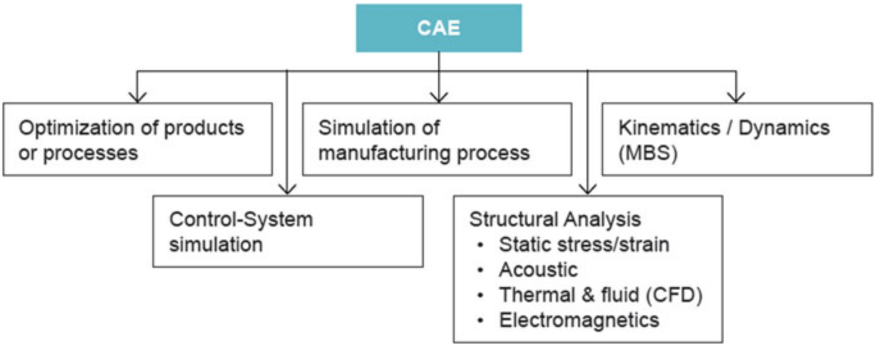


Fig. 10.3 The main disciplines of CAE

Closely combined with the solidity analysis, is the *eigenmode* analysis. The goal is to determine the *eigenfrequency* and the *eigenmode* of a structure. Resonance vibrations present a common problem in machine construction, as many systems are subject to the vibration generated by internal and external forces. This is where FEA or BEM are then employed.

In the numerical *Computational Fluid Dynamics* (CFD) fluids and gases are analyzed. State variables include pressure, flow velocity and density, among other things. The base equations (typically modelled after Navier–Stokes) are calculated or approximated using the *Finite Difference Method* (FDM), the *Finite Volume Method* (FVM)—both are based on the mathematical principle of differential equations—or the FEA approach.

In the area of thermodynamics, processes are examined where energy, in the form of heat, flows through a medium and is radiated off or transferred into a different form of energy. There are close correlations to computational fluid dynamics, as warming processes are often viewed in terms of fluid dynamics. One application is the combustion process in a gas motor, for example.

With numerical methods, electromagnetic effects are also studied. Calculated state variables may be electric field strength, eddy currents, etc.

In the subdomain of kinetics and dynamics the movement of components or assemblies and the dynamic forces and momentum being created are analyzed. The technical term for this area is called *Multi-Body Dynamics* or *Multi-Body Systems.* With the control systems simulation, technical systems are viewed on a global scale, and the flow of energy and materials is analyzed. With help of the physical and control technological dependencies between components of a system, a kind of circuit diagram is established (not a spatial diagram), which is why the field is also called a 1D simulation. The simulation of manufacturing processes and the optimization of products or processes helps itself to methods and technologies from the CAE subdomains, but can be viewed as separate branches due to their content and praxis (compare Chap. 9 "*CAPP, CAM and NC Technology"*). For example, the simulation of a welding robot may include the structural analysis (in the form of thermodynamic analyses) and kinetics (for the movement of the robot). Optimization processes can be applied to structural analyses as well as control systems.

In numerous technical applications the movement of a body in space, and/or the relative movement of components in relation to one-another one are subject of analysis. Such components can be connected via joints, springs or dampers. Force, momentum or acceleration occur which are dependent on the movement, or affect the multi-body system. Sketching out behavior of such kinetic or dynamic systems is the goal of the *Multi-body Simulation* (MBS). The objects can ideally be shown as rigid or flexible. In the kinetic simulation, there are globally no open degrees of freedom, but with a dynamic one, there are. The latter is numerically more difficult to solve.

By following the three-dimensional modeling of the system in the CAE software, the physical attributes and dependencies can be defined. The corresponding material attributes, connections, starting and border conditions (forces, momentum, bearings/support) are defined. As such, a system of equations can be established and calculated.

The *multi-domain and control systems simulation* is used when energy or material sizes need to be viewed on a global scale. In technical systems, the connections between individual components are shown as a circuit diagram. Cause and effect are clearly defined [7]. As the structure of a three-dimensional component is not directly considered unlike to the previous simulation varieties, only the physical relationships between the actors are considered, the method is also known as a 1D CAE simulation. When dependencies between different physical quantities can be generated, the simulation is called a multi-domain simulation.

With the help of the above described various CAE methods Engineers can create digital prototypes in order to reduce or totally replace physical prototypes. In the world of technical system development, physical prototypes serve traditionally as:

1. Selected models or artefacts to prove out (*validate*) constructional, functional and behavioral ideas and intentions with the help of their special physical realization (in non-production mode)—in automotive industry, e.g., the terms *working horse* or *mule* exist for early full vehicle prototypes.
2. A rather complete set of physical systems and components at a certain development gateway stage (mostly prior to and shortly after the production release gateway) under almost production ready circumstances in order to *verify* a determined set of constructional, functional and behavioral attributes of the product according to a sign-off list of requirements and engineering/ performance targets.

A physical prototype, therefore, might still be needed despite of powerful CAE simulations for a number of reasons: especially in early CAE capability ramp-up phases in enterprises "invisible" human errors in CAE model build and simulation set-up and execution might occur during engineering development. This can lead to late surprises such as component collisions, fatigue problems, thermal failure etc. Due to late notification until physical prototype testing cost intensive rectifications is then unavoidable and causes "unnecessary" churn, readiness delays and on-cost. Another source of error could be the simulation itself: if a mistake is made in the model assumption or the planned calculation procedures of the component or product environments, this fault can distort the entire findings of the CAE project. This is more often the case than expected, since realistic conditions can be best anticipated as outer boundary conditions for simulation, but they cannot be perfectly imitated due to limited knowledge about complex physical interrelations. Therefore, CAE needs careful verification of the used model types. Another common reason for still constructing a prototype is the desire to see and feel the actual product with all human senses. This can affect the tactility of the product, or the comfort of a product (like a seat). Current VR technologies do attempt to bridge this gap in a virtual space, but options are still limited regarding other human sensibilities such as force feedback and tactile experiences.

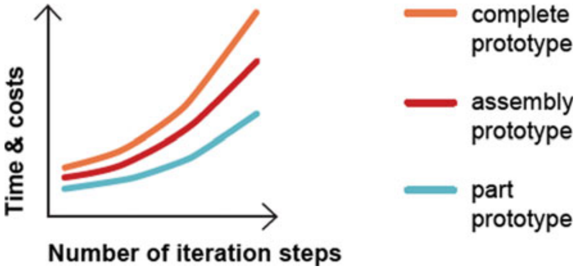


Fig. 10.4 CAE target to reduce development costs and expensive physical prototypes

Nevertheless, the better a CAE simulation of a component or product is, the less reliant engineers have to be on physical prototypes. This is desirable as a physical prototype can be expensive and time-consuming, as qualitatively shown in Fig. 10.4.

Each iteration of the prototype costs additional time and budget. For example, if a company considers the development of a new train generation, it is hardly financially possible to build more than one prototype of such train, before arriving at the final product for the customer. In aviation industry even the first full flight prototype is also finally sold to a specific customer after integrating all updated packages as part of the development completion. For substructures, such as seats or armrests, a higher number of physical prototypes might be possible. Thus, the CAE is in a position to replace physical prototypes by digital ones and consequently helps engineers to improve product development on multiple levels by realizing a systematic validation and verification of components, sub-systems products and complete technical systems consisting of multiple, interacting products.

10.3 How Does CAE Work?

With the help of CAE and its simulation techniques, the behavior of a (technical) system that either already exists or is under development can be analyzed with respect to certain system or product attributes/properties. The simulation can be understood as an experiment on a digital model: the results of such “simulation experiment” can be used to drive conclusions regarding the behavior of a real technical system (product, machine, production system etc.). In order to trust the outcome of a specific CAE simulation, it is necessary that the underlying digital model type and the simulation algorithms have been generically validated against tested behavior of an equivalent physical realized technical system or product.

Please note below the major ten steps of a successful CAE analysis project:

1. Detail scope analysis of the (real/physical) technical system set-up and clarification of the technical system performance targets.
2. Determination of the purpose of the CAE analysis (and simulation types) and clarification of the intended simulation type requirements.
3. Selection of the appropriate CAE discipline(s) (compare Fig. 10.3).

4. Digital model formulation (incl. usage of the correct model elements).
5. Digital model generation and implementation (model build).
6. Verification of the digital model according to certain model criteria.
7. Simulation run, i.e. conduction of the simulation on an appropriate computer or computer cluster.
8. Validation of the simulation results according to engineering and model theory knowledge.
9. Evaluation of the simulation results towards the technical system performance targets (with potential modification proposals to improve the original digital model, e.g. CAD model).
10. Closure and book shelfing: report out to and discussion with engineering partners and stakeholders, documentation of the results (incl. design modification proposals) and lessons learned conclusions with potential improvements of CAE analysis procedures.

Within the first step, the scope analysis of the technical system, the subdivision of the system into its sub-systems and components takes place. This step is often directly combined with or preceded by a formulation of the requirements and should determine what will be depicted for which purpose. Following the determination of the CAE analysis type and its underlying core method/procedure in steps 2 and 3 it will be decided which results are expected in which type of format. As a result, these decisions largely determine which objects receive black-box characters, as they may require special control algorithms, which the analyst and costumer of the CAE analysis project may not need or want to view.

In steps 4 and 5 the active choice needs to be made which model elements should be used within the model formulation and how exactly those model elements need to be linked to each other in order to achieve most realistic simulation behaviors of the digital model. In addition, the corresponding assembly model is determined as part of all connected component models. At times, there is academic differentiation between theoretical and experimental model building. The former describes characteristics of the system to be displayed by utilizing (physical) laws and hypotheses and corresponding mathematical formulation and equations. The experimental model development, however, concentrate on taking observations from physical test stand experiments into account, generally by using and incorporating measured test data into physical model assumptions. The test measurements are, however, generally approximated or fitted on a curve for simplified usage in order to enable the linking to the solving process of the theoretical models.

The implementation part of step 5 explains the conversion of a digital preprocessor model into a format that is machine-readable, i.e. this is the transition from a model preprocessor to the simulation model in a specific solver environment. Generally, specific for representational languages, procedures or even programs are used for it. This work step can efficiently decide which models are effective and feasible, using effective representation or programming methods. If models are not documented at all or are only poorly documented, the verification of the digital model in step 6 becomes tedious and time consuming.

Step 6, the verification of the digital model, is in fact the last stage where domain knowledge can and should be integrated into the digital model, these steps should, therefore, provide ease-of-use for the user and can help to avoid modeling errors. The more complicated and layered the data entry options are for the user, the more likely it is that errors can occur, which can lead to results that do not accurately reflect the behavior of the technical system to be represented. The split between usability and accessibility for less simulation-competent users, the desire for efficient programming, as well as shorter development times will always stay in competition to each other and should be clarified upfront during the requirements phase of the simulation project.

Step 7, the execution of the simulation, finally describes the experiment based on the implemented model. Generally, this involves the distribution of parameter values, which are subject to change in repeated instances. The solver environment steers the execution of solving all inherit mathematical equation tables in a timely fashion. Depending on the type of computer or computer clusters in usage it might take between seconds, minutes or hours (for complex problems with several hundreds of thousands or even millions of mathematical equations it might even take weeks). The results of a simulation are affected by the choice of the mathematical solution algorithm as well as by its parameters. These approaches should be documented in order to make the experiment reproducible. There exist multiple approaches to achieve simulation results. The simplest is to change the accuracy and the time step distance until no significant variations appear in the results. It is advisable to compare the results of different solution algorithms.

Step 8, the validation of the simulation results, represents one of the biggest challenges. It does offer, however, significant potential to improve the results of a CAE analysis project by validating the soundness and the correctness of the results. In the validation process, it is determined whether the results really accurately resemble the original system. The question is often difficult to answer with a simple yes or no. The simulation can never perfectly represent the behavior of the original systems, which is why deviations are often found. To assign limitations on deviations is usually difficult to prove. Often, a qualitative process for specific values over time with determined deviation limits can provide a good indicator for the validity of a model regarding specific characteristics even without performing a quantitative proof.

The evaluation of the simulation results (step 9) constitutes an extensive and oftentimes difficult task, since it is necessary to apply domain knowledge and CAE model build/simulation knowledge at the same time. Therefore, the CAE analyst needs to engage closely with the System, Component and Design Engineers in order to conclude meaningful results. Oftentimes it also remains invisible whether numerical particularities of the solving process itself (e.g. rounding occasions) might also have an influence on specific final results of structural analysis (such as stress and displacements), thermal analysis (such as temperatures or thermal flux) or acoustic and vibration analysis (*eigenvalues, eigenfrequency* etc.). In order to close the loop at the end of the CAE analysis (step 10) it is essential to include the final results of the CAE simulation run but also details from the preparation steps beforehand. The following result types are of high interest at the end:

- the official report and the associated oral explanation as part of the virtual product creation collaboration,
- the discussion with other experts and stakeholders,
- the creation of lessons learned on modeling practices and
- the final documentation of the entire simulation project.

This final part of the process is oftentimes neglected by CAE analysts in industrial practices as well as by researchers in science: leaving a gap-riddled documentation of the models and of the verification methods often leads to issues and impacts, when the models are to be reused or adapted for a different project.

The steps 3–6 are iterative in their execution (please also compare Fig. 10.5). The goal is to have the model describe the desired characteristics as accurately as possible, after all. As a result, the process should begin with simple models, which can be refined by comparing measured values, for example. This commonly requires detailed mapping of sub-components, but does not preclude the need for more specific parameter values.

10.4 CAE in Product Development

The calculation/simulation of technical systems takes place in one of the following three phases of product development, fulfilling a specific function within each one of those phases:

1. In the first phase, at the concept design phase, i.e. during the preliminary calculations to drive major physics of the design it is important to establish major requirements of a product (e.g. approximate number of components, required operating power, material alternatives etc.). As the design is generally not finalized during this stage, the calculation is often performed analytically along experiences or guidelines or with the help of concept CAE models. Figure 10.6 shows an example of a car body shell CAE model during the concept phase of the vehicle development. In order to support the target setting of the overall vehicle performance targets with respect to vehicle package, NVH behavior (noise, vibration, harshness) and crash, it is decisive to use *CAE concept models* in various degrees to determine major dimensions and topological design principles of the body structure. The upper part of Fig. 10.6 depicts a simple concept model whereas the lower part shows a refined one based on input from other different digital model sources.

2. When the product is (partially) constructed and it is necessary to make decisions about component separation as part of the embodiment development phase, the concept design is transformed into a proper system, product and component design. This second phase needs the help of specific CAE verification calculations. Within this context, the use of CAE software tools becomes mandatory,

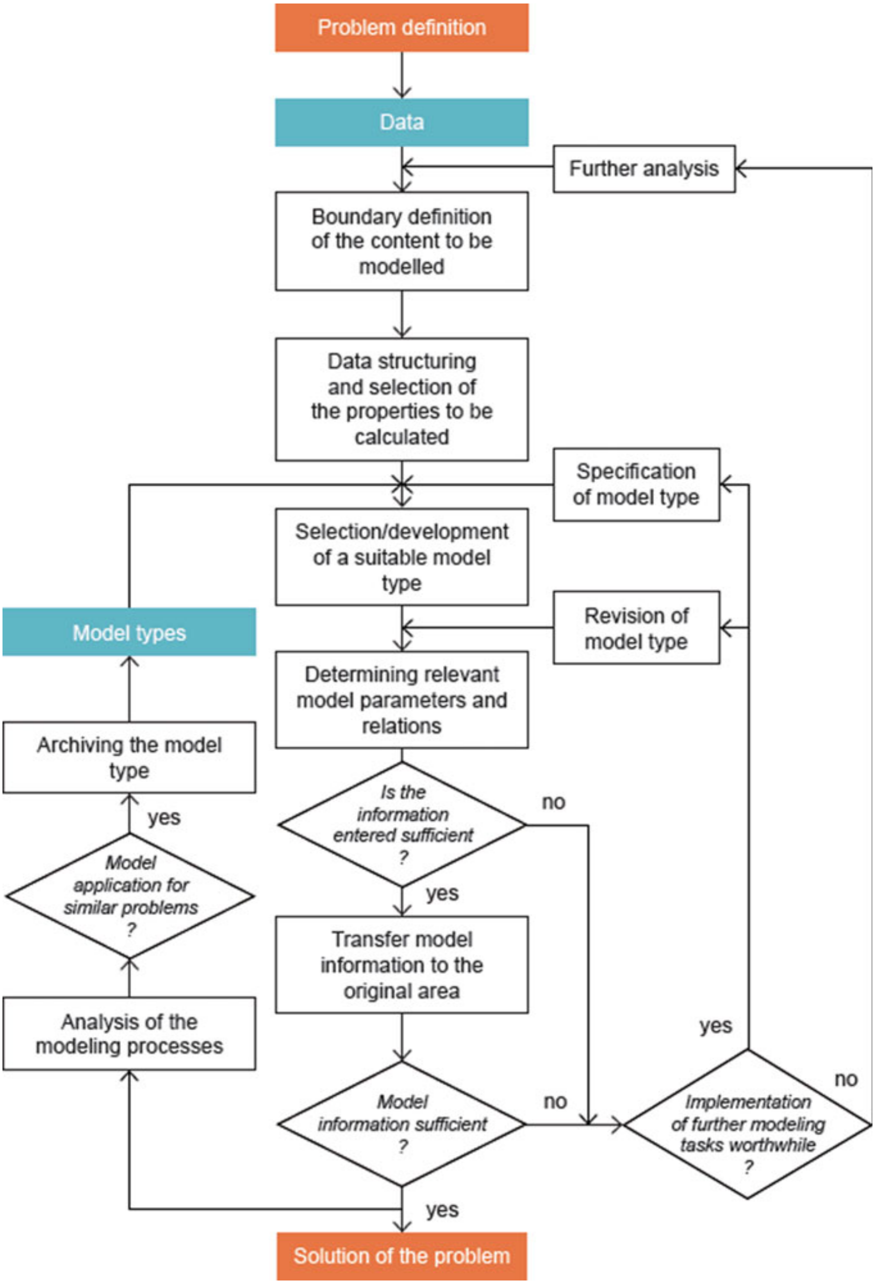
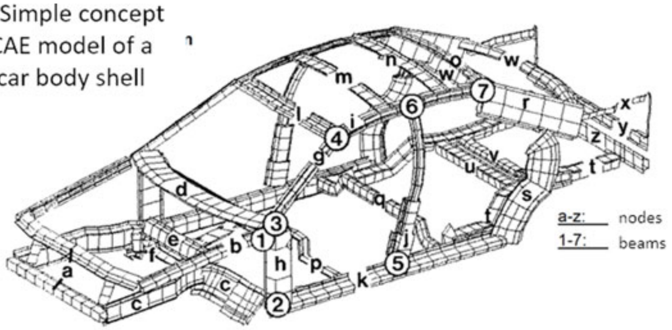


Fig. 10.5 Iterative flow of activities in order to achieve a robust CAE model build (formulation, generation and implementation)

a Simple concept CAE model of a car body shell



b Refined concept CAE model of a car body shell based on predecessor scan data & engineering knowledge data

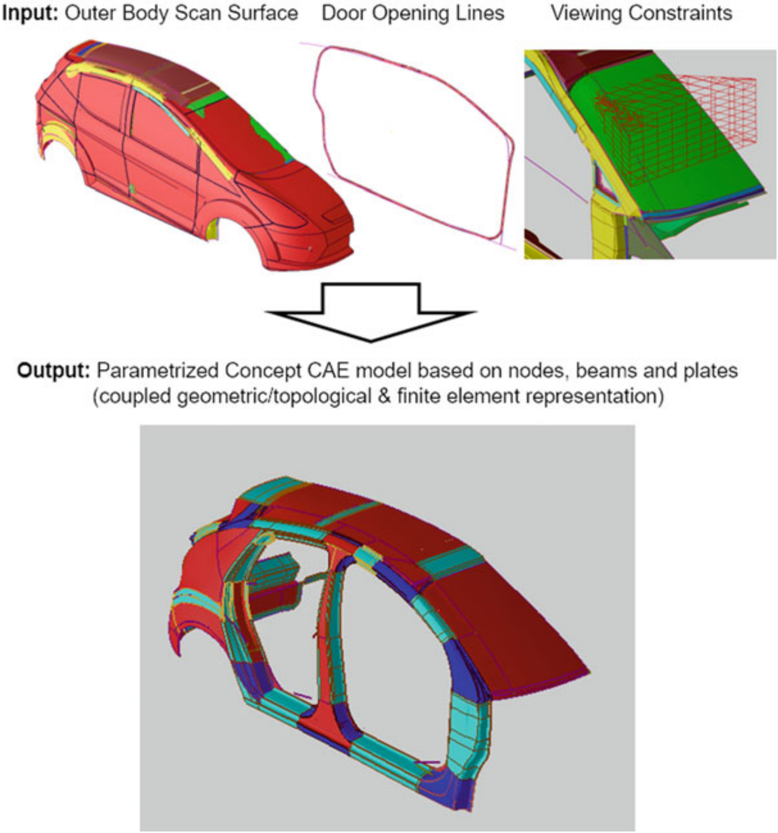


Fig. 10.6 Different types of concept models as part of car body engineering

as the overall technical and/or product system is already too complex for traditional analytical calculations. Oftentimes design details and crucial development decision scenarios need the extensive support of CAE analysis. Figure 10.7 shows examples of a detailed body shell FE (finite element) mesh model (a), the connecting FE assembly of an engine mount mesh model to a rail assembly mesh model (b)—both are used for vehicle crash simulation purposes)—and detailed FE mesh models for the durability analysis of a wheel hub (c).

The third development phase of CAE calculation is driven by optimization goals. If a product does not yet fulfill certain product performance targets, or if there exist ample potential for design characteristics improvement, specific CAE analysis support is desired (please compare example in Fig. 10.8).

10.4.1 From CAD to CAE—CAE Model Build

In classical virtual product creation process, the 3D design is developed with the help of CAD software. All relevant product, design, function and manufacturing information is stored in the CAD model or as associated meta data (as part of the data storage environment such as PDM). That includes the design (geometric information), normal or supply components (screws, glue etc.), manufacturing information as well as a material list. If CAE is integrated in the CAD application, the model generally can be immediately transferred to the simulation (see Fig. 10.9).

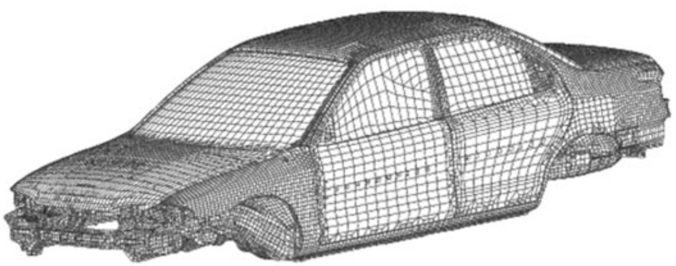
If the CAE software is in a separate application, the information needs to be transferred. If the CAE software cannot process the native CAD file format, instead only working with exchange formats, the 3D data has to be converted [5]. If appropriate interfaces are present between the CAE, CAD and database software, the files can be accessed simultaneously. The CAD model is then exported into a neutral format (like STEP or IGES) and subsequently imported into the CAE software.

Before components are transferred to the simulation software, first it is evaluated whether the construction details are relevant for the simulation. Often construction details (for example chamfers, drillings) are not important to a simulation, and can unnecessarily increase the complexity of the calculation grid, and the resulting calculations [8]. As safety factors are generally used for permissible values (e.g. max. stress or deformation), higher real stresses are covered by omitting features.

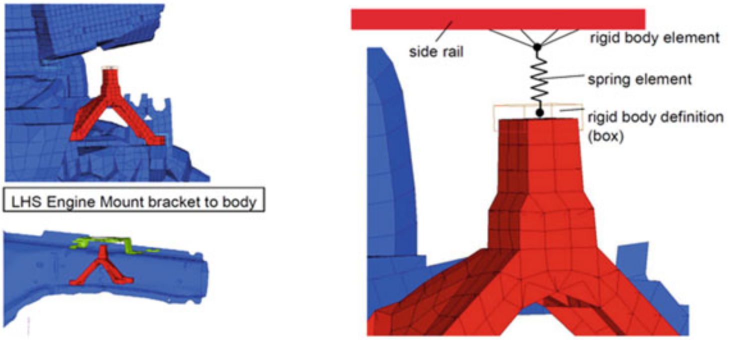
One possibility to evaluate the influence of suppressed features and to include them in the results of a simulation is the use of benchmarking. In this case, general cases are simulated with and without a feature, and results are compared afterwards.

To take drilling as an example, if it is known that the maximum stress on a bore increases by a factor of 2 compared to the same design without a bore at a certain type of load, this factor can be applied in similar applications. A possible discrepancy is covered by safety factors. It is to be ensured that the benchmark can be transferred to a specific case.

a Detailed body shell finite element model for structural analysis



b Verification CAE model of an engine mount bracket to body rail assembly to support vehicle crash analysis



c Verification CAE model of a suspension wheel hub: durability analysis with von Mises stress calculation for bending and torsional modes

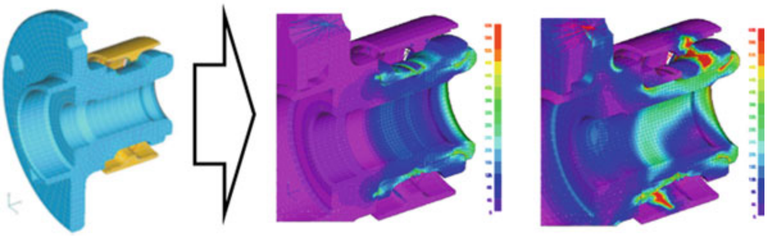


Fig. 10.7 Different types of verification CAE analysis in vehicle development

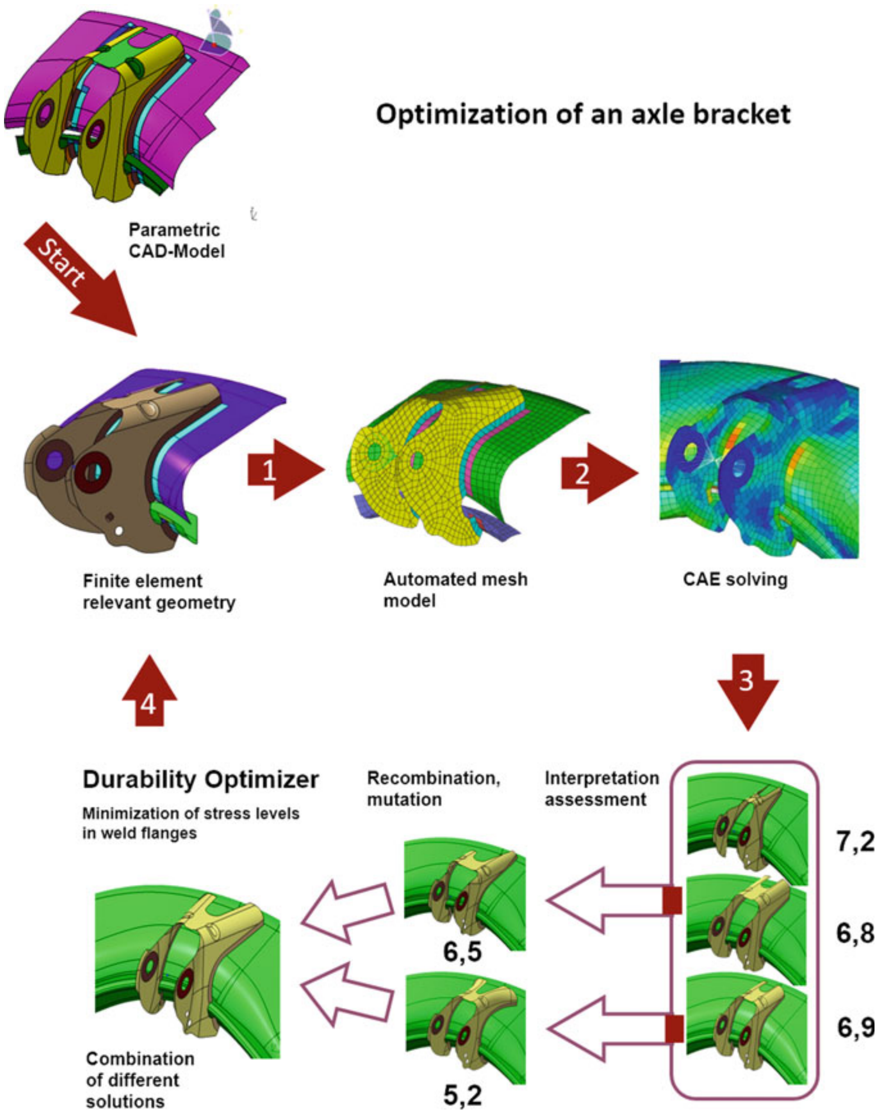


Fig. 10.8 Design topology optimization by using CAE analysis

This is where further CAE model build preprocessing takes place to set up the mathematical model (structure, boundary conditions, loads etc.) and resulting numerical equation system. After having performed the simulation run the post-processing to analyze the component or assembly also takes place within the CAE system environment. Analysis result files from the computing process are typically displayed in the same CAE software. The insights gained then flow back to the construction,

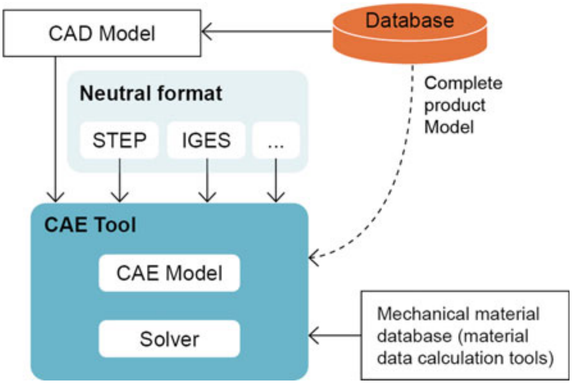


Fig. 10.9 Model transformation from CAD to CAE

which is how the component can be improved. This overall model build process for the Finite Element (FE) analysis is illustrated in Fig. 10.10.

Figure 10.11 shows the core relations between CAE software modules, simulation phases and result types. CAE software packages offer specific functions for specific tasks such as creating a model environment to pre-process the geometric model with all necessary engineering boundary conditions such as loads, forces, inertia, moments etc. and specific finite element connectivity conditions (compare e.g. illustration B in Fig. 10.7 with respect to the engine mount bracket integration into the rail structure).

As a result of the pre-processing stage, the simulation model is created. The solver then solves the mathematical model numerically and generates a result file. The post-processor then looks at the results, and creates a representation for analysis. After critical areas and zones at the design component or assembly are found, the

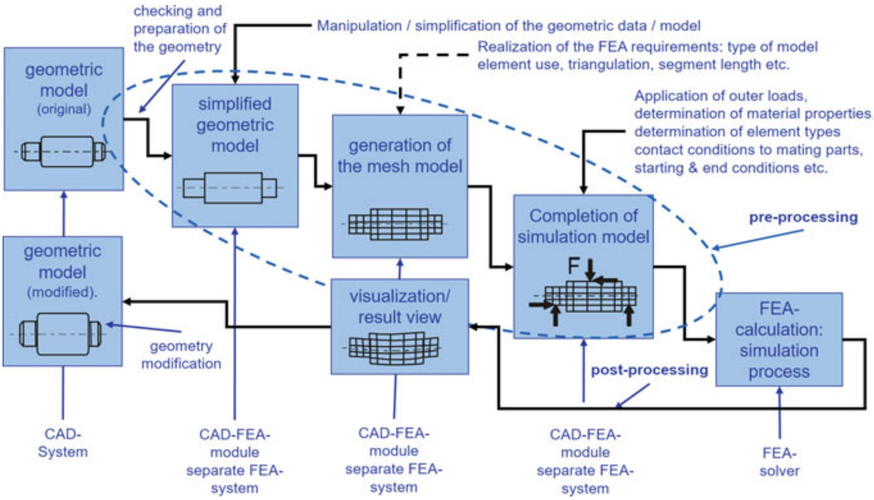


Fig. 10.10 General model build process for a finite element (FE) analysis

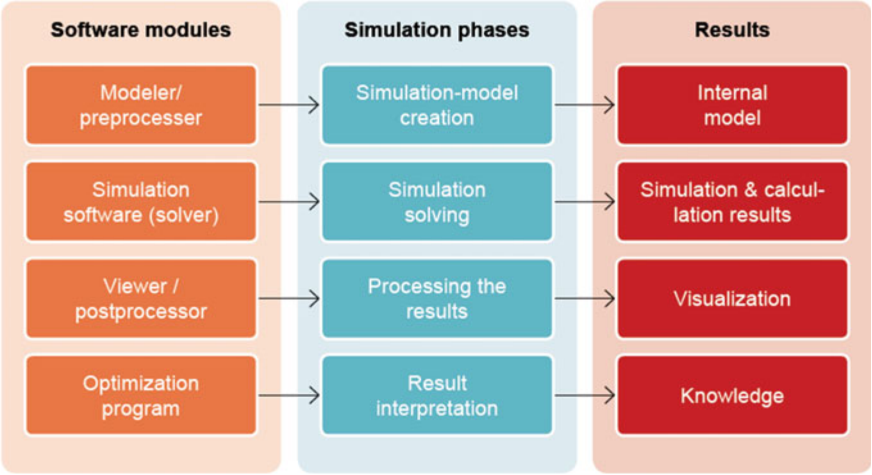


Fig. 10.11 Core relations between CAE software modules, simulation phases and result types

model undergoes an optimization process. For this, the results need to be correctly interpreted. Root causes for critical areas are also studied in order to improve them. Finally, the new insights and knowledge can be (re-) used in future projects as well.

The following explanations focus on the CAE Build process with respect to CAD-CAE transfer and necessary neutral exchange file formats, where a structural or kinematic simulation of 3D data is the aim—not a 1D simulation. This is why the use of neutral formats and its capabilities becomes important. In practical application, the creation of a CAD model is generally done by the engineering design department, while the preparation of the simulation model belongs to the calculation/CAE departments. This clear separation of responsibilities within a company is mainly due to competencies. The following steps need to be followed during the transition phase.

10.4.2 Interfaces/Formats to Transfer CAD Models to CAE

When converting CAD files into a neutral file format, two types are differentiated [5]: the first one represents geometrically exact systems, which can replicate data without loss in geometrical precision with the help of mathematical methods. Examples are STEP, IGES or JT. Currently, some formats also allow the storage or product information, which goes beyond pure geometry. That includes assembly structures, material data or different configurations.

The second neutral format type uses methods to approximate the original geometry. Most commonly, surfaces are represented with polyhedrons, which significantly reduces the amount of data. This does allow for some inaccuracies, particularly on freeform surfaces. Normally, only geometric information will be stored, no assembly structure or other metadata. That is why these file formats are often used for DMU-applications or for visualizing large assemblies rather than for CAE calculations.

CAD models, which represent three-dimensional volumes, can be created through two different modeling methods (compare Chap. 7 "*Computer-Aided Design—CAD*"): a surface model defines the volume only implicitly through its (volume) surrounding topological connectivity of the poly surfaces and through the mathematics of the individual surfaces. This type, however, cannot directly be leveraged to check whether a point is inside or outside of the implicitly defined volume body.

Depending on the CAE application, a surface or volume model might be required as part of the CAE model build. If, for example, the goal of a CAE analysis is a strength calculation for a tin construction, the surface model suffices, as the surface mesh is only set up with 2D shell elements anyways. If a complex casting is needed or the liquid flow in a pipe is to be simulated, the entire volume body needs to be meshed. Here it is very helpful if the information about the interior and surface of the object is known beforehand.

The neutral CAD format IGES (Initial Graphics Exchange Specification) enables to exchange geometric information as well as metadata like assembly structure or material. It was published by the U.S. National Bureau of Standards in 1980 (nowadays National Institute for Standards and Technology, NIST) and, therefore, the code is standardized. The rules how to convert the original geometry are not clearly defined, this is why the representation differ from software to software. With the conversion, the original shape does not lose its geometric accuracy [9]. IGES saves data as surface model without the information about volumes.

Similarly, to the IGES format, STEP (STandard for the Exchange of Product data) belongs to the geometrically exact exchange formats. It is standardized according to ISO 10303 and specialized with different application protocols for specific technology branches (for example AP203 for general mechanical engineering, AP224 for manufacturing purposes. The construction history as well as features and geometric constraints are lost. For this reason, it is difficult to edit the STEP model at later stages. Due to its versatility and performance capacity STEP is widely recognized. For visualization applications, the format is not considered the first choice due to its complexity [9].

The exchange format STL (Surface Tessellation Language) exists since 1988 meanwhile is a wide-spread geometric interface option. The original geometry is approximated with triangles, and the degree of accuracy can be influenced or set by the user individually. Information regarding geometric features, component assembly structure or construction history are also not saved in the STL format. The format can compress data sets efficiently and its application spans from simple geometries without free-form surfaces up to complex models or models with a high accuracy requirement producing large amounts of data. A clear disadvantage of the STL format represents the fact that the STL format lacks standardized data representation modes (numerical representation, header etc.). The facetted representation, however, is well suited for visualization applications (compare [10]).

With the JT format (Jupiter Tessellation), geometry is either approximated with triangles (like STL) or displayed geometrically exact (like STEP or IGES). Advantages of JT are the standardization according to ISO 14306 and the continuous functional updates to satisfy the requirements of different technological branches. Object and meta data as well as assembly structure and geometry features can be stored and safely exchanged. These characteristics make JT one of the versatile and sustainable exchange format.

10.4.3 Pre-processing of a FEA Model

The finite element analysis (FEA) is a standard method for the calculation of continuums, as already introduced in sub-chapter 10.2.2. As shown in Fig. 10.12 the structure of the CAE/FEA model is separated into spatial elements, in order to reduce a global problem to its basic physical calculations in each element. On the elementary level with defined properties, the state variables and the local behavior can be evaluated with approximating functions [4]. The process in an FEA is generally a good example also for the workflows of other CAE simulations. Similarly to the general CAE process, the FE analysis begins with the import a CAD model, some CAE applications, however, also offer a modeling environment of geometry within the CAE application itself. The structure is idealized, meaning components irrelevant to the simulation are eliminated from the calculation model and features that are less important are suppressed. The further preprocessing then contains the following steps:

- The material values are defined according to the construction and the features of the design and are then assigned to individual components.
- Afterwards, the element type is selected: 2D shell elements, volume elements or their subtypes and the corresponding mesh based on those elements gets generated. CAE engineers have to use substantial heuristics to refine the mesh according to geometric shape characteristics and/or areas of physical load applications (auto meshing algorithms might be limited in assuming the right engineering knowledge).
- The completed mesh is then assigned with boundary conditions in the form of load cases, which contain clamping and fixtures, the forces and momentum affecting the component and other conditions such as symmetry, contacts etc.
- The solver then generates the individual elements according to the discrete structure transfer functions, assigning them to the total stiffness matrix.
- The results, meaning the component stiffness and resistance to bending or yield, is then calculated in each element and displayed in color coding as part of the post-processing. The component behavior can then be predicted using the plots,

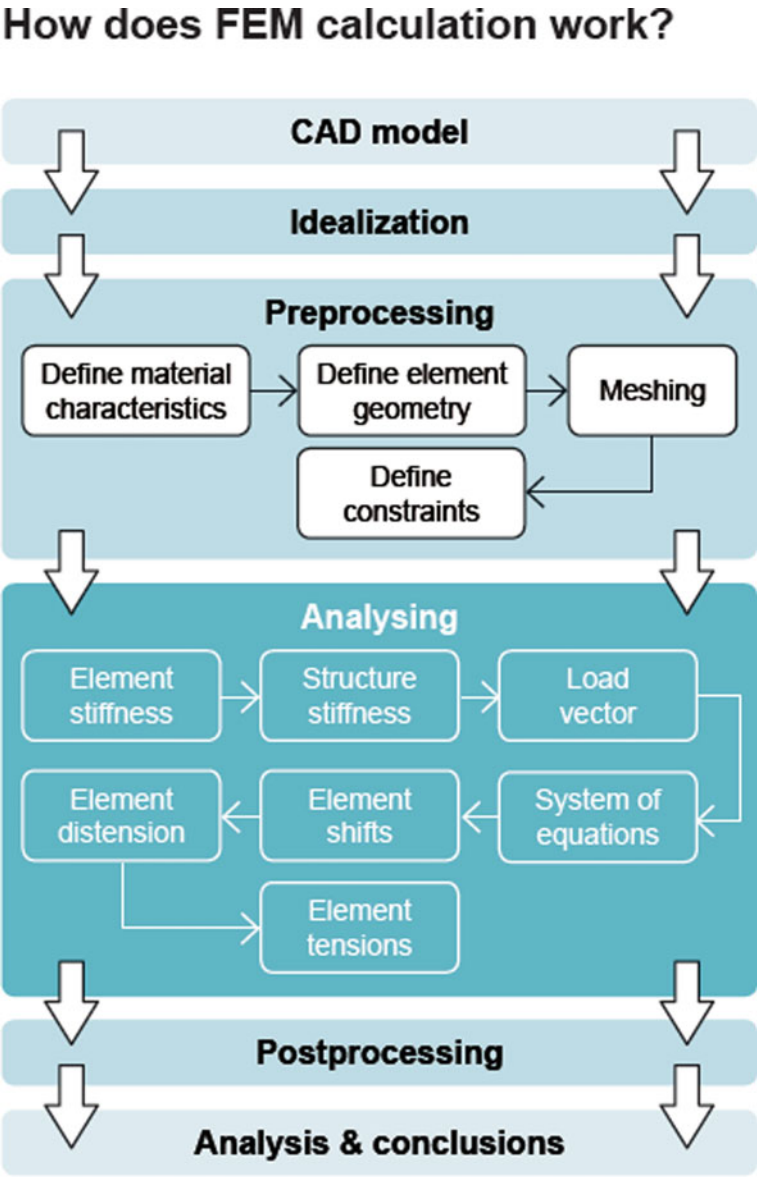


Fig. 10.12 How does FEA calculation work?

tables or graphs for analysis and evaluation (compare [10] and [4]) as well as Fig. 10.12).

The generation of the FE mesh, itself, has a strong influence on the quality and accuracy of the simulation. Depending on different quality markers (for example skewness, aspect ratio, Jacobian matrix and determinant structure, warpage etc., compare also [11]) it can be analyzed whether an element can be calculated numerically in a reproducible way, or if distortions, inaccuracies or singularities are to be expected. The mesh is generated manually with the help of the meshing tool of the preprocessor and the knowledge of the CAE analyst. Methods such as the “feature approach” allow for a clean mesh establishment and allow for optimized meshes under consideration of geometric and functional elements (features). The condition is that the basic CAD data set contains these geometric elements.

The general rule is that the more refined the mesh, the more precise the solution approach for differential equations can be expected. Areas in which critical component strain, high gradients or rigidity deflection is expected, need to be meshed more finely [10]. Since the manual adaption of the mesh takes time and is expensive, it is important to test how much detail needs to be evaluated beforehand, in order to achieve a sufficiently accurate calculation result [12].

A technique called *adaptive meshing* exists to automate CAE model build: where higher stresses are located the mesh is refined automatically by the solver to get more accurate results. Figure 10.13 explains this principle in more detail: in the example of a tool holder first a fixed or automated mesh is being applied by using standard meshing algorithms which take in to account the curvature related mesh element size rules. After having applied all outer constraints and loads the numerical solve of the finite element problem is executed and the high stress areas are detected. Based on such results the refinement of the mesh in the areas of high stress is applied in order to receive refined results which can differ from the initial results by 10–25%. The upper example a of Fig. 10.13 shows the example of the original tool holder design whereas the example b (lower areas) shows the same principle at the already beforehand *topology optimized* or *generatively designed* tool holder. The approaches of *topology optimization* and *generative design* are explained later in this section and represents a specific approach (compare also Figs. 10.14 and 10.15).

There is a wide variety of element types to display the complex structure of a technical application. In the following, the main types, their features and their uses are elaborated [12]. In principle, they are categorized into 1D, 2D and 3D elements, depending on the number of dimensions in which the principle force transfer takes place. Elements can have different amounts of integration points, depending on the degree of approximated transfer functions (linear, quadratic etc.).

The rod element belongs to the 1D-Elements. The main distension takes place via the longitudinal axis and is much larger than the measurements in perpendicular directions. It can only handle forces or line loads in direction of its longitudinal axis (tension and compression). A typical use would be the representation of steering linkage for wheel suspension.

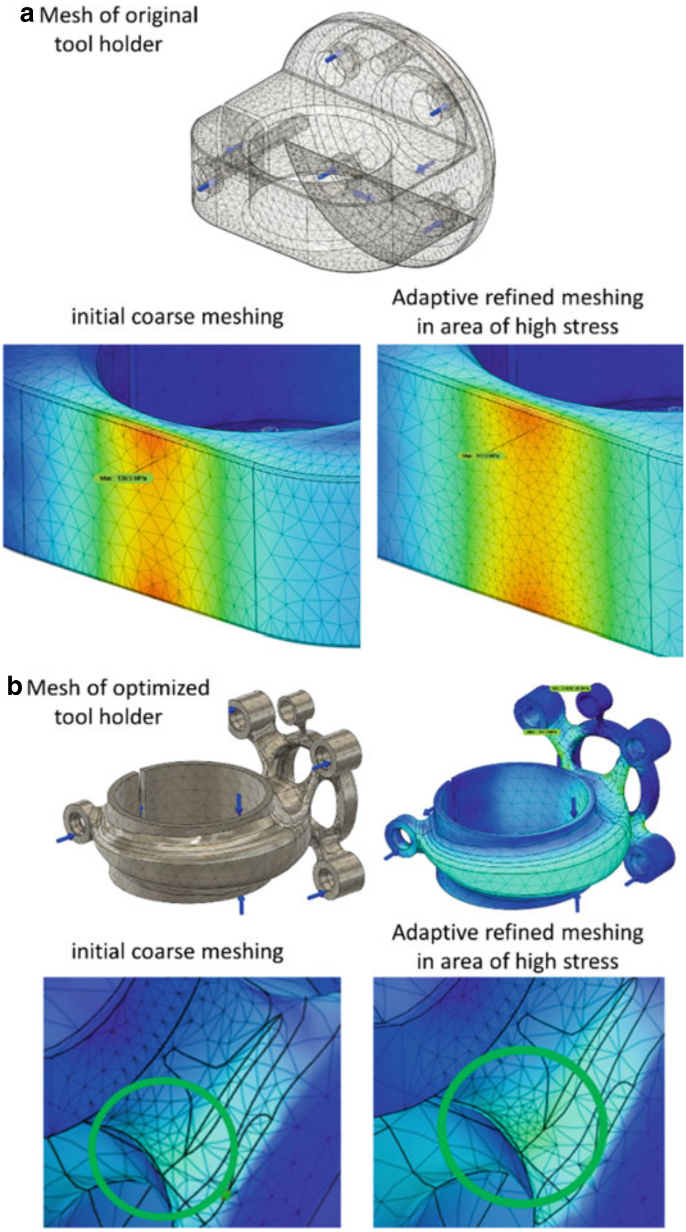


Fig. 10.13 Principle of adaptive meshing in areas of high stress (courtesy support by Autodesk Inc.)

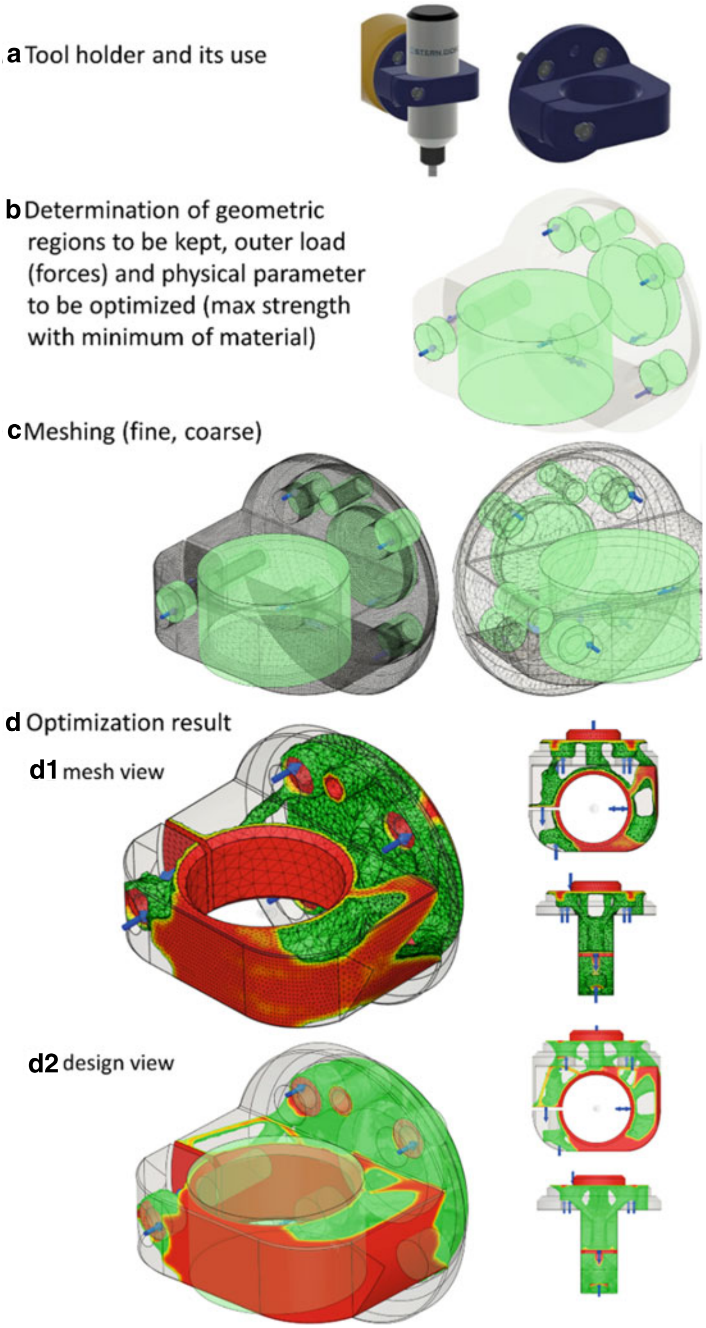


Fig. 10.14 Topology optimization (courtesy support by Autodesk Inc.)

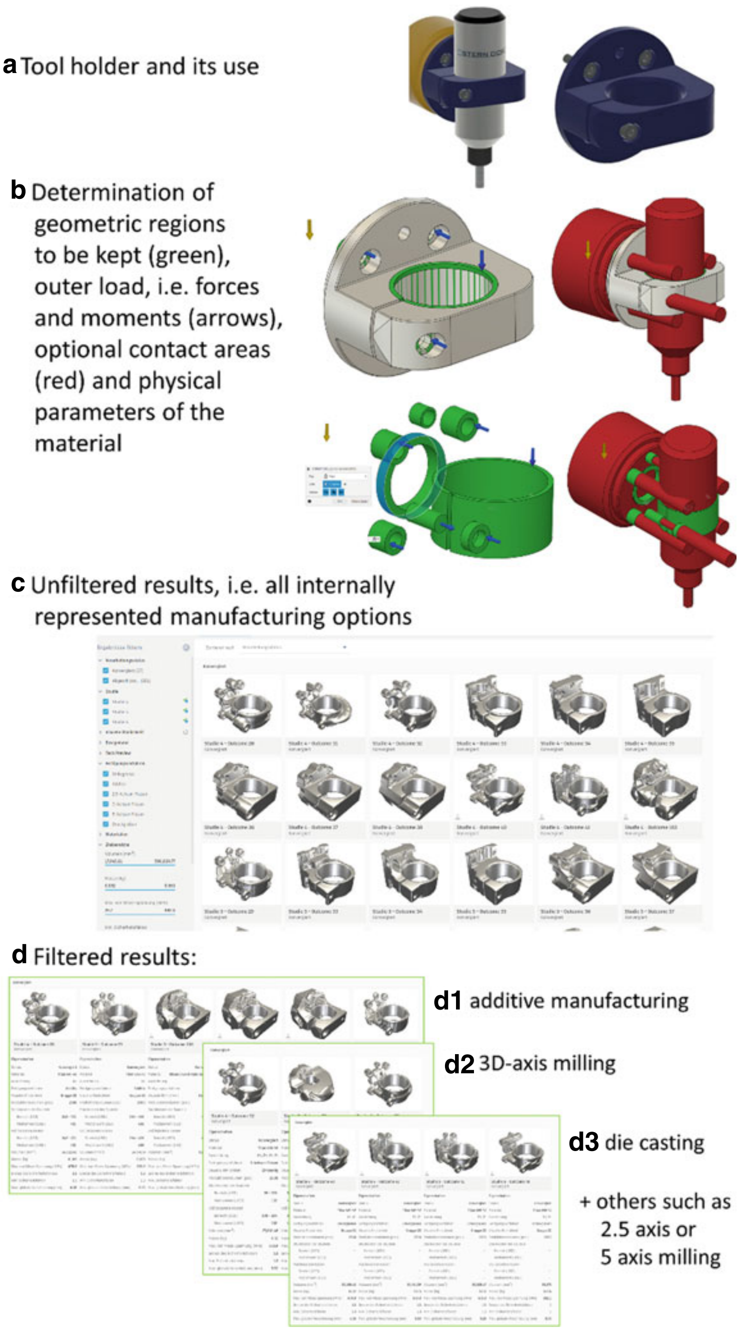


Fig. 10.15 Principle of generative design (courtesy support by Autodesk Inc.)

The beam element is a 3D-body. It has the same properties like the rod but it is also capable of transferring twisting and bending torque. The Bernoulli Hypothesis applies, that is why no shear deformation are taken into account. The bends of the beam are smaller than the height of the beam. Different cross-sectional areas can be defined. Because of its characteristics the beam element may be used for screws or even tubes.

Shell elements are created by layering disk and plate elements. This is why forces and moments in the plane of the element as well as perpendicular to the plane can be transferred. The combination of planar, shear and bending loads result in principal stress, which is to be considered for analysis. The thickness of the shell element is virtually added, for meshing only a middle surface is needed. It is used for sheet metal, in general thin structures with a defined thickness. In addition, composite or anisotropic materials may be represented. Shell elements can be triangles or squares. However, the theory of shell elements are rather complex and cannot be understood by engineers easily (recommended references are: [13, 14]).

With volume elements, 3D-structures are meshed. They are able to represent and reflect the behavior of continua, no matter which geometry. The shapes of the volume elements are hexahedrons, pentahedrons or tetrahedrons.

There exist several coupling elements, like distribution or rigid couplings, that are used to connect elements, for example to a single fixed point. Another application is the connection of a point mass to the structure. The operating degrees of freedom can be adjusted. Furthermore, spring or damper elements or contact definitions may be set up.

10.4.4 Utilizing FEA Models Within Optimization Problems

With the help of CAE tools, products and processes can be optimized for specific features and towards specific behaviors. Each type of optimization calculation is based on the principle that certain necessary constraints need to be kept and criteria for the optimal desired state are defined (via the *objective* or *target function*). In order to reach this optimal state, design variables need to be flexible (see [15]). The goal of the objective/target function is dependent on the design, converging on a local or global extremum, the desired state has been reached. Whether a global optimum can be reached is oftentimes also dependent on the starting conditions. Gradient based methods often lead to local extrema, which is why the influence of starting conditions on the optimization calculation should always be checked thoroughly (see [16]).

There is a basic distinction between the optimization of a control system and a 3D structural optimization. In the first case, a system with mathematical/physical dependencies is present, for example a combustion motor with a transmission. The goal of a system optimization can be the optimal working point, meaning the least consumptive state of the system. In the structural optimization, one optimizes a three-dimensional structure for specific features or towards specific behaviors. These would include factors such as minimal weight, reduction of component tension or minimal material use for the same constructive functionality. The three main techniques of structure optimization are topology optimization, parameter optimization and form optimization.

The outcome of the topology optimization (compare Figs. 10.8 and 10.14) is simply based on an available design space, including load introductions (for example, storage, forces, static analysis, thermic analysis etc.). Boundary conditions are defined, such as maximal tension or material usage, as well as symmetry conditions. The solver then calculates at which points of the design space material is required and where it is not necessarily needed. The result is an analysis of the ideal usage of the design space. The phenomena is also visible in nature: for example, tree branches or bone marrow only grow in places where material strength is needed. The result is a more or less defined design recommendation, which is then converted into a constructible format.

Complex structures with irregular geometries are particularly suited for additive-generative or casting procedures. Figure 10.14 shows an example of topology optimization of a tool holder: a given tool holder CAD design (see step a) is constraint in step b by determining design topology areas to be kept (due to the assembly design constraints), by applying loads and by setting the optimization criterion. After applying the mesh (step c), the algorithms can solve the constraint optimization problem, which result in the optimization outcomes (see step d).

In contrast to topology optimization, parameter optimization is based on an already existing design. Design parameters (e.g. wall thickness, length of a girder, thickness of an axle) are defined as alterable in the objective/target function. This often has the advantage that (in the case of an FEA) the FE mesh is retained and can be adapted in the frame of smaller geometric changes, without requiring the generation of a new mesh.

Shape optimization improves the local geometry of a component. A common use of FEA is to minimize tension on transfer points (radials, rigidity deflections on cross section modifications etc.). The requirement is—similar to the parameter optimization—that a certain design has already been established, and only details need to be adapted. The variable can be the position of nodes on the surface of the component, for example. If the mesh is not deformed too drastically during the shape optimization, it is sufficient to perform an automatic mesh smoothing after each step without re-meshing.

10.5 Advanced CAE Technologies

The simulation types named in this sub-chapter are meanwhile highly developed and matured. Therefore, applied with the right engineering and CAE method knowledge, they can achieve precise results in their specialized fields. A current challenge is the so-called “flexible body dynamics”, the combination of multi body dynamics and finite element analysis, oftentimes using two types of CAE simulation based on a co-simulation framework. This means that the bodies of a MBD model are no longer treated as rigid, but can be replaced by moldable FE models of the actual component. The technique of integrating an FE model is called floating frame of reference (FFR), representing one method for the co-simulation approach. For example, the movement and expansion of the piston rod in a combustion engine can be simulated. Due to the reciprocal influence of a MBS and a FEA model the problem is non-linear and analytically as well as numerically difficult to solve [17].

A further focus of study is the combination of multi domain simulation (MDS) and MDB/FEA. A possible application is the simulation of an electric window motor in a car: The control of motors or sensors is set in the field of mechatronics. The entire system can be displayed in a multi domain system. At the same time, the dynamic strain on the windowpane or the fixture for it on the body can be simulated via MDS. Furthermore, it is of interest to implement such a system as Hardware-in-the-Loop (HiL). This involves combining several real components with the virtual system. Another application is the digital twin: here a real system is digitally mapped and fed with measured data (such as movement, duration etc.). This can then be used to draw conclusions regarding the current or future state of the system.

The real-time simulation of a flexible thin-walled component and cables or hoses also presents a unique challenge. The difficulty lies in the numerous conditions: cables and hoses are sometimes comprised of multiple layers of different materials. In addition, a cross-section with twisted wire strands does not clearly show deformations, further complicating behavioral predictions. The assembly of thin-walled objects, for example covers in the automobile industry, often takes advantage of their flexibility, which is why it is manually warped to achieve its correct installation position. This deformation is difficult to simulate due to the human component and is generally analyzed in practical testing.

The field of research topology optimization is not fully developed and has much potential. For problems such as effectively used assembly space, highest degree of rigidity etc. there is generally not only one solution, but a variety of local optima with different specifications. As the algorithms work with evolutionary and partially heuristic principles, the question whether a problem was optimally solved is often not easy to answer.

Acoustic phenomena in systems are examined using NVH analyses. They are needed in the automobile industry, for example, where attention must be paid to motoring experience and comfort. The complex automobile system contains a number of oscillation sources (motor, transmission, chassis etc.) and resonance bodies (covers, body etc.). The system as a whole is impossible to simulate in a structural analysis due to its complexity. As a result, simulation procedures such as FEA, BEM and MBS are combined. With the transfer path analysis (TPA) the paths on which sound and vibrations are transferred from their originator to the recipient (generally the human) are analyzed [18].

Due to new compute power with the help of grid and cloud computing it is nowadays possible to combined FEA analysis with design synthesis. This new type of hybrid approach is called *Generative Design*. There does not yet exist an absolute clear scientific or normative industry wide standard definition for the term and field of *Generative Design*, however, the common understanding can be expressed by the current explanation by AUTODESK Inc. (https://www.autodesk.com/solutions/generative-design# visited in September 2020):

Generative design is a design exploration process. Designers or engineers input design goals into the generative design software, along with parameters such as performance or spatial requirements, materials, manufacturing methods, and cost constraints. The software explores all the possible permutations of a solution, quickly generating design alternatives. It tests and learns from each iteration what works and what doesn't.

In actual facts *Generative Design* combines engineering design and analysis tasks with cost estimation and manufacturing feasibility work which usually are handled separately from each other:

- 3D Design of individual components
- 3D Assembly and constraint product modeling
- CAE analysis and shape/topology optimization
- Design for Manufacturing (DFM) and manufacturing feasibility analysis as well as
- Cost engineering.

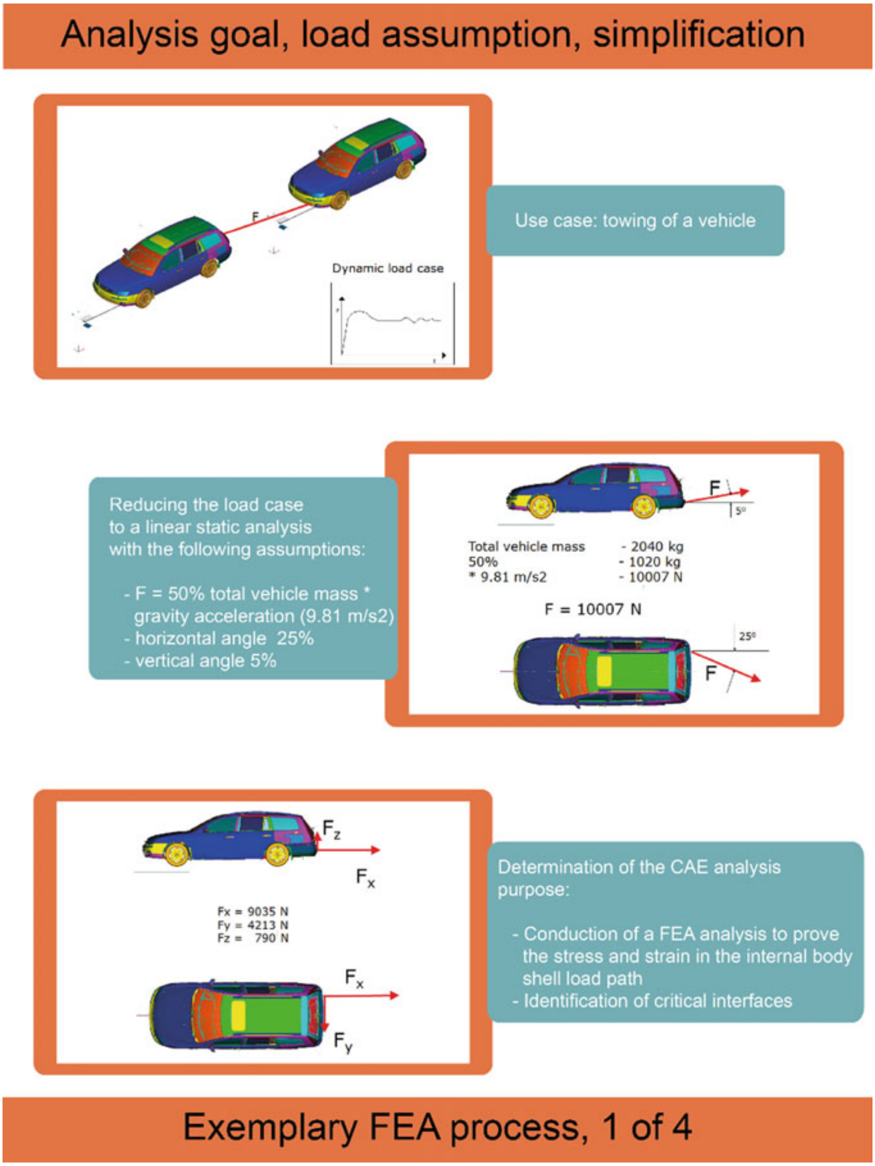
In order to achieve meaningful design exploration and associated design proposal offering with additional information sets the user is guided through a systematic approach. Figure 10.15 gives an insight into the generative design principle: starting from the overall assembly situation (see part a) and the interface design knowledge to the adjacent part (see part b) the design and analysis engineer receives a set of design alternatives (see part c).

In the following, the design and analysis engineer has to provide additional information sets for the algorithms to resolve the underconstraint mathematical problem according to the options at the pareto front (i.e. to achieve the improvement of one design factor without deteriorating another design factor). Hence, in order to reduce the high number of solution proposals offered by the compute algorithms it is necessary to further filter or select the manufacturing technology options (see part d) by the design and analysis engineer. Based on AI (artificial intelligence) heuristics cost estimations are provided to the use of such Generative Design environment in order to discuss and or decide on the option to take. This design proposal then needs to be further refined and executed by classical design (CAD) and analysis (CAE) methods and tools.

10.6 Exemplary Automotive FEA Project Cases

This section illustrates a typical FEA analysis as part of an automotive body shell use case. The example deals with the digital (CAE based) design verification of a body shell rear end in combination with a towing hook design. It shows the assumptions and simplification which are made based on engineering knowledge and heuristics in the automotive body shell development. The purpose of this example is to show the tight interaction between product/technical system know-how and specific CAE method and tool knowledge. Only if both viewpoints are aligned and supported by prior system validation test (using hardware) such CAE (FEA) based design verification can be applied successfully in industrial practice.

This industrial project case is divided into four parts: Fig. 10.16 illustrates the overall situation of the FEA analysis case by describing the load case (towing of



Analysis goal, load assumption, simplification

Use case: towing of a vehicle

Reducing the load case to a linear static analysis with the following assumptions:

- F = 50% total vehicle mass * gravity acceleration (9.81 m/s2)
- horizontal angle 25%
- vertical angle 5%

Total vehicle mass - 2040 kg
50% - 1020 kg
* 9.81 m/s2 - 10007 N

F = 10007 N

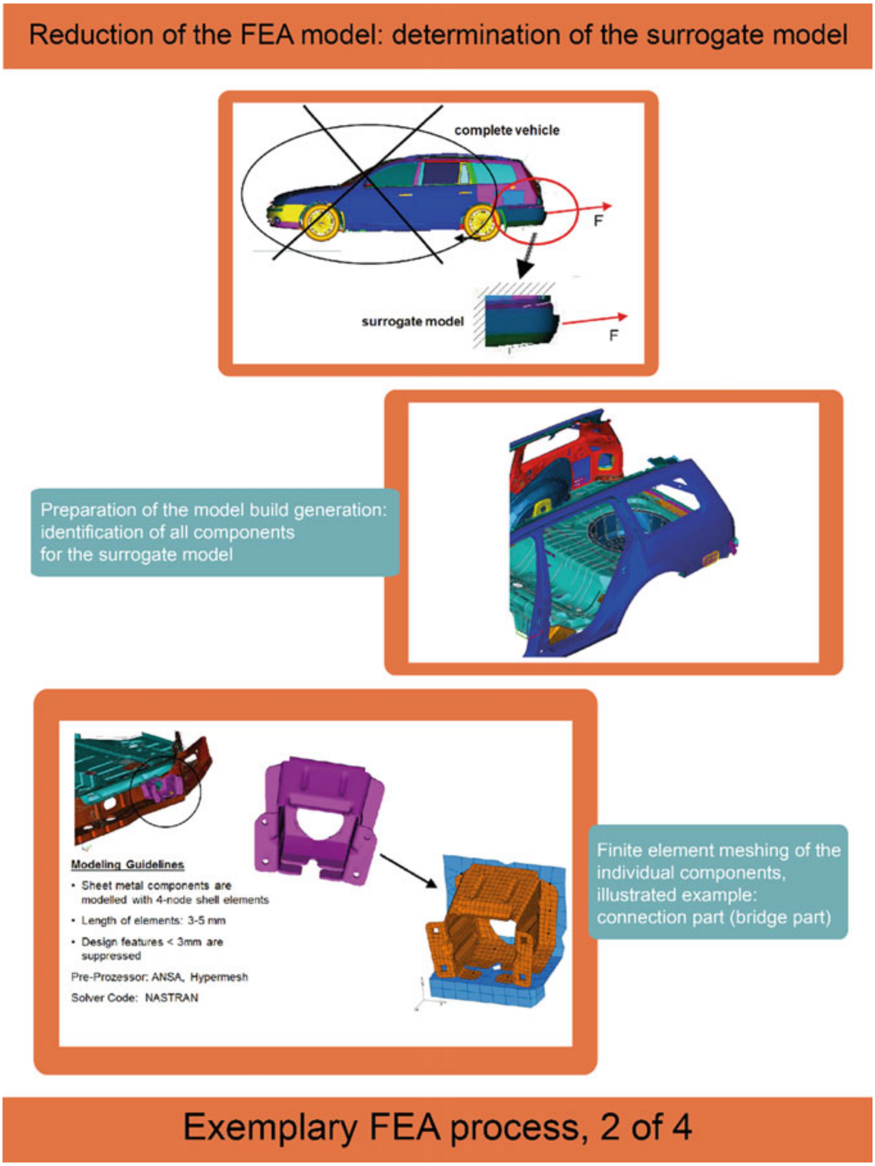
Fx = 9035 N
Fy = 4213 N
Fz = 790 N

Determination of the CAE analysis purpose:

- Conduction of a FEA analysis to prove the stress and strain in the internal body shell load path
- Identification of critical interfaces

Exemplary FEA process, 1 of 4

Fig. 10.16 Exemplary automotive FEA project case, part one another car), the first simplification (reducing the dynamic load case to an equivalent static one with the help of higher forces) and the clarification of the angle of direction of the force elements). Figure 10.17 shows how a CAE Engineer has to make the right decision to cut the overall body CAE mesh model and to reduce it to a (relevant) one for the vehicle rear end. The aim is to calculate the load path and stress analysis



Reduction of the FEA model: determination of the surrogate model

Preparation of the model build generation:
identification of all components
for the surrogate model

Modeling Guidelines

- Sheet metal components are modelled with 4-node shell elements
- Length of elements: 3-5 mm
- Design features < 3mm are suppressed

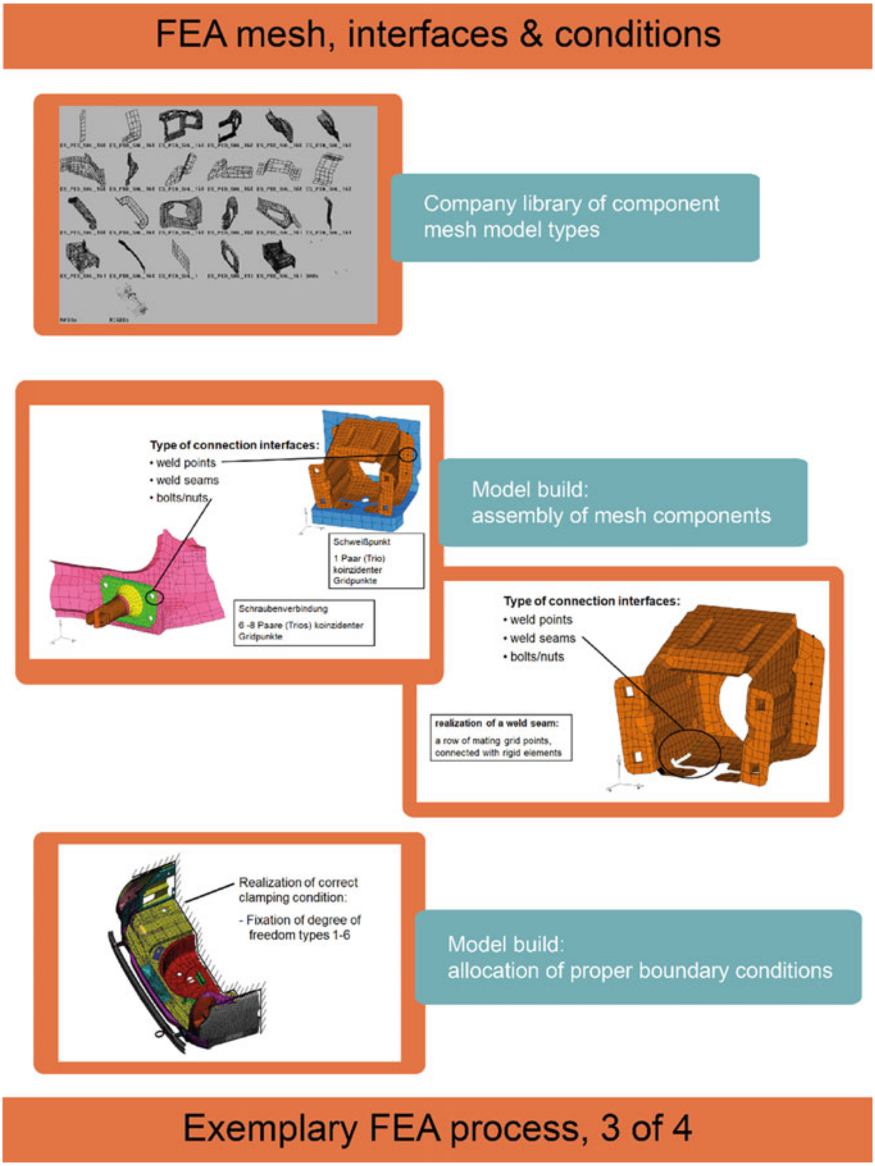
Pre-Prozessor: ANSA, Hypermesh

Solver Code: NASTRAN

Finite element meshing of the
individual components,
illustrated example:
connection part (bridge part)

Exemplary FEA process, 2 of 4

Fig. 10.17 Exemplary automotive FEA project case, part two based on the outer load assumptions (as already clarified in Fig. 10.16) and the correct boundary conditions. Figure 10.18 shows the library of mesh model types which are proven and recommended within the company's best CAE practice guidelines ("CAE cookbook") and how the element connections need to be modeled. Especially the interface between rigid machinery parts (like the screw in towing hook assembly



FEA mesh, interfaces & conditions

Company library of component mesh model types

Type of connection interfaces:

- weld points
- weld seams
- bolts/nuts

Schweißpunkt

1 Paar (Trio) koinzidenter Gridpunkte

Schraubenverbindung

6 -8 Paare (Trios) koinzidenter Gridpunkte

Model build:
assembly of mesh components

Type of connection interfaces:

- weld points
- weld seams
- bolts/nuts

realization of a weld seam:

a row of mating grid points, connected with rigid elements

Realization of correct clamping condition:

- Fixation of degree of freedom types 1-6

Model build:
allocation of proper boundary conditions

Exemplary FEA process, 3 of 4

Fig. 10.18 Exemplary automotive FEA project case, part three plate) and sheet metal components (of the body shell itself) need to be modeled on purpose with the help of specific grid points. The bottom picture illustrates the need to position and fix the body shell rear end assembly in the right way. Figure 10.19 shows the necessary final digital model verification before the simulation run can be started.

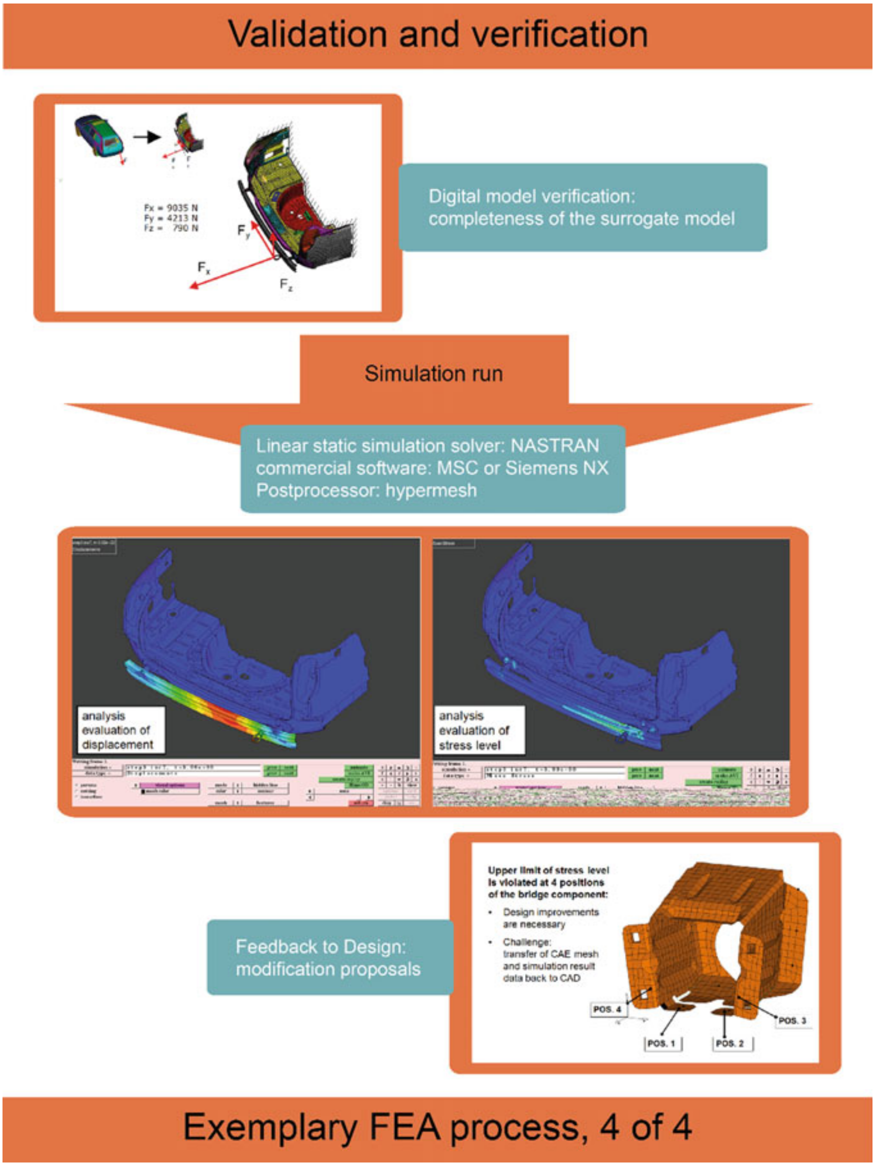


Fig. 10.19 Exemplary automotive FEA project case, part four

It also illustrates the post-processing results of the FEA analysis. The graphical visualization of the displacement (left side) and of the stress distribution within the sheet metal assembly (based on von Mises stress type) are indicators for the CAE analysts to finally judge whether the results are acceptable or do cause problems over time. CAE analysts then prepare an analysis report based on specific components and their embedment within the overall technical system or assembly.

Finally, improvement proposals are made and an overall assessment is discussed with the Design Engineers with respect to possible design modifications. As a consequence within this project case, the bridge component (bracket), shown in the bottom of Fig. 10.19 had to be redesigned since the stress levels at the folded flanges were too high. A deep drawn bracket would usually provide more stiffness but it does it would not usually require more efforts to provide an extra tool set for its manufacturing compared to a sheet metal folding part).

A second example deals with a modular CAE model build to support efficient CAE analysis for front crash investigations as shown in Fig. 10.20. There exist two different CAD models as design alternatives for the front bumper (part a). The CAE mesh assemblies (part b) are divided into different domains, which are individually meshed depending on specific strategies and accuracy requirements. The interface conditions for the individual connection types of the mesh elements can also be maintained if associate mesh model build is deployed: this included the connection type, the intelligence of determining the number of connection types (based on rule) and the time step settings for the simulation itself. The only difference between the CAE mesh assemblies is the mesh of the bumper itself, the other mesh parts of the assembly stay exactly the same. The lower part of Fig. 10.20 shows the simulation results for the full-frontal crash (c1) versus the one of the partial frontal crash (c2).

10.7 Final Remarks

Computer Aided Engineering (CAE) meanwhile constitutes a major digital engineering discipline which is indispensable to validate and verify early design concepts, functional layouts and final release ready design proposals before any physical prototype evaluations take place. Due to the fact that all associated skills in tools and technology, product and CAE models, modeling methods as well as in the underlying mechanical laws and mathematical formulations require substantial knowledge of engineers and analysts make it extremely challenging to distribute those skills widespread in the organizations. Therefore, especially Small and Medium Size Enterprises (SME) are still dependent on massive support by specialized engineering service providers instead of building up internal CAE skilled engineers.

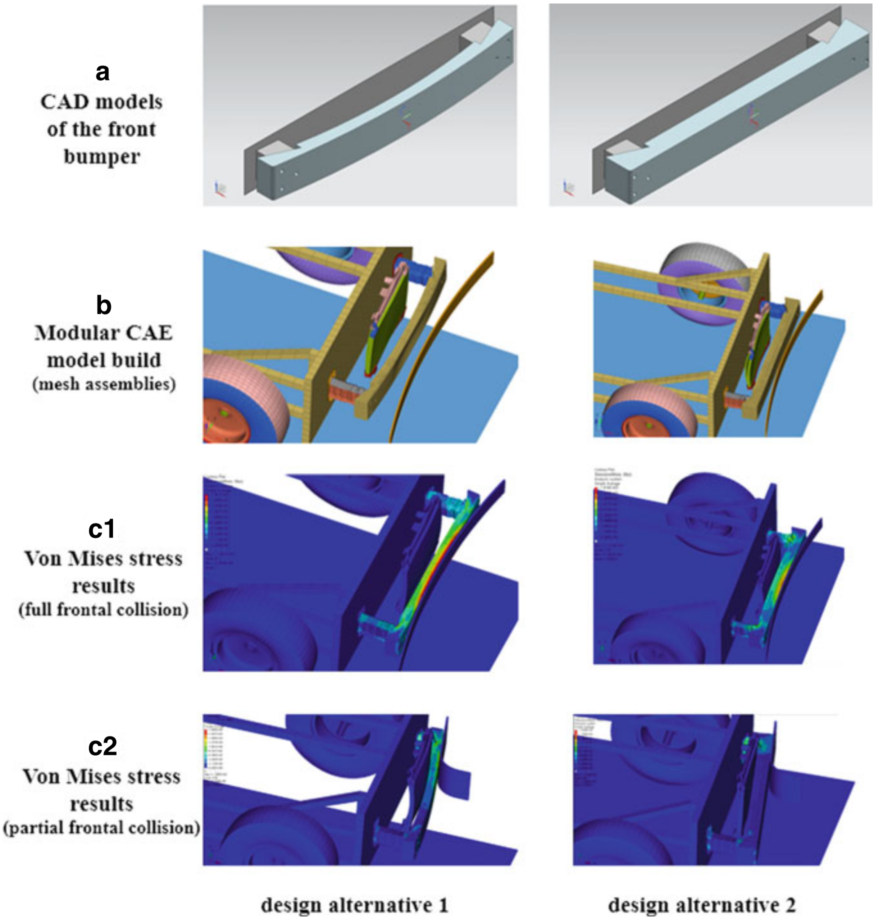


Fig. 10.20 Modular CAE model build as part of a conceptual front-end crash evaluation

Two trends are currently visible and evident:

- Bigger companies try to mix the skill set between design engineers and CAE analysts in a new way in order to start with CAE rather than using it as after-the-fact verification solution only
- The rising complexity and connectivity of technical systems under development do require new advanced CAE modeling and analytical skill sets of engineers and analysts as part of the comprehensive Advanced Systems Engineering (ASE) development capability. This does require, however, new assistance for engineers in system modelling, systems integration and system validation/verification.

As of today, CAE is still seen as an expert group development operation and not yet as an engineering skill set which needs to be support by every engineer. Therefore, middle and upper management need to be re-educated in virtual product creation capabilities in order to drive the critical new functional and behavior analytic skill sets within major product development processes and company organizational set-ups.

References

1. Kolbasin A, Husu O (2018) Computer-aided design and computer-aided engineering. MATEC web of conferences, vol 170. https://doi.org/10.1051/matecconf/201817001115
2. McHaney R (2009) Understanding computer simulation. Roger McHaney & Ventus Publishing ApS; ISBN 978-87-7681-505-9
3. Matsuura Y, Kaminishi K, Matsuura Y (n.d.) (2018) The impact of CAE advancement on product development process. In: International conference on innovation and management. http://www4.pucsp.br/icim/ingles/downloads/papers/TL_060.pdf. Accessed 27 Aug 2018
4. Jagota V, Sethi AP, Kumar K (2013) Finite element method: an overview. Walailak J Sci Tech 2013:1–8
5. Hirz M, Dietrich W, Gferrer, Lang J (2013) Overview of virtual product development. In: Hirz M, Dietrich W, Gferrer A, Lang J (eds.) Integrated computer-aided design in automotive development (p. Chapter 2). ISBN 978-3-642-11940-8
6. Orbis Research (2018) Global (North America, Europe and Asia-Pacific, South America, Middle East and Africa) CAE software market 2017 forecast to 2022. http://www.orbisresearch.com/reports/index/global-north-america-europe-and-asia-pacific-south-america-middle-east-and-africa-cae-software-market-2017-forecast-to-2022. Accessed on 27 Aug 2018
7. Brunberg A, Abel D, Autschbach R (2009) An object-oriented model of the cardiovascular system with a focus on physiological control loops. In; IFMBE proceedings, vol 22, pp 2600–2603. Springer, Berlin, Heidelberg
8. Suresh G (2007) A formal theory for estimating defeaturing-induced engineering analysis errors. Comput Aided Des 39:60–68. https://doi.org/10.1016/j.cad.2006.09.006
9. Pfouga A, Stjepandic J (2018) Leveraging 3D geometric knowledge in the product lifecycle based on industrial standards. J Comput Design Eng 5:54–67. https://doi.org/10.1016/j.jcde.2017.11.002
10. Erhunmwun I, Ikponmwosa U (2017) Review on finite element method. J Appl Sci Environ Manage 21:999–1002. eISSN: 2659-1502; print ISSN: 1119-8362
11. Altair Engineering Inc. (2012) Element quality and checks. http://altairuniversity.com/wp-content/uploads/2012/04/Student_Guide_211-233.pdf. Accessed on 23 Aug 2018
12. Steinke P (2007) Finite-Elemente-Methode - Rechnergestützte Einführung. Springer. ISBN 978-3-642-53937-4
13. Timoshenko S, Woinowsky-Krieger S (1959) Theory of plates and shells. Mc Graw-Hill Book Company, Singapore
14. Rieg F, Hackenschmidt R, Alber-Laukant B Finite Elemente Analysis für Ingenieure (Finite Element Analysis for Engineering). Hanser. ISBN 978-3-446-43469-1
15. Larsson R (2016) Methodology for topology and shape optimization: application to a rear lower control arm. Chalmers University of Technology, Göteborg, Department of Applied Mechanics, Sweden
16. de Weck O (2005) Design optimization—structural design optimization. Massachusetts Institute of Technology (MIT). http://citeseerx.ist.psu.edu/viewdoc/download?doi=10.1.1.207.9728&rep=rep1&type=pdf. Accessed on 17 Nov 2019
17. Shabana AA, Wang G, Kulkarni S (2018) Further investigation on the coupling between the reference and elastic displacements in flexible body dynamics. J Sound Vib 427:159–177
18. Siemens AG (2020) Transfer path analysis—a guideline to qualifying and quantifying vibro-acoustic transfer paths. https://www.plm.automation.siemens.com/en_us/Images/What_is_transfer_path_analysis_tcm1023-220904.pdf. Accessed on 27 Sept 2020